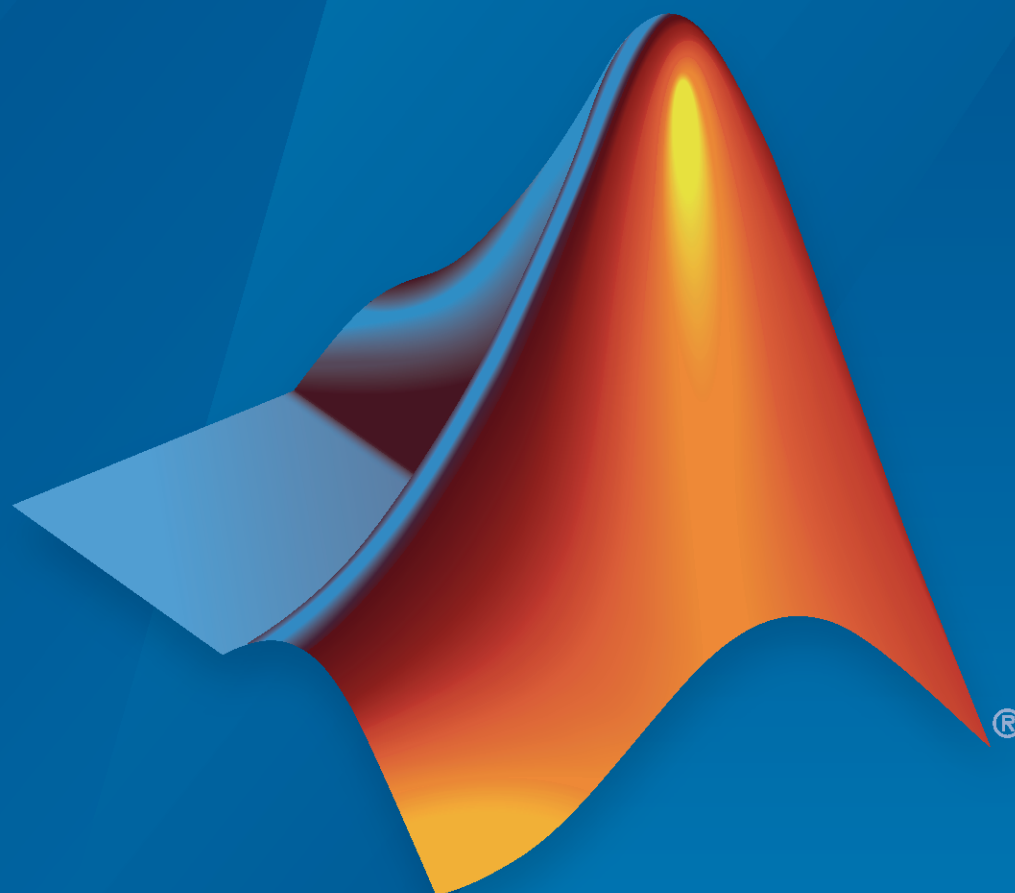


Simulink® Coder™

User's Guide



MATLAB® & SIMULINK®

R2023a



How to Contact MathWorks



Latest news: www.mathworks.com
Sales and services: www.mathworks.com/sales_and_services
User community: www.mathworks.com/matlabcentral
Technical support: www.mathworks.com/support/contact_us



Phone: 508-647-7000



The MathWorks, Inc.
1 Apple Hill Drive
Natick, MA 01760-2098

Simulink® Coder™ User's Guide

© COPYRIGHT 2011–2023 by The MathWorks, Inc.

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from The MathWorks, Inc.

FEDERAL ACQUISITION: This provision applies to all acquisitions of the Program and Documentation by, for, or through the federal government of the United States. By accepting delivery of the Program or Documentation, the government hereby agrees that this software or documentation qualifies as commercial computer software or commercial computer software documentation as such terms are used or defined in FAR 12.212, DFARS Part 227.72, and DFARS 252.227-7014. Accordingly, the terms and conditions of this Agreement and only those rights specified in this Agreement, shall pertain to and govern the use, modification, reproduction, release, performance, display, and disclosure of the Program and Documentation by the federal government (or other entity acquiring for or through the federal government) and shall supersede any conflicting contractual terms or conditions. If this License fails to meet the government's needs or is inconsistent in any respect with federal procurement law, the government agrees to return the Program and Documentation, unused, to The MathWorks, Inc.

Trademarks

MATLAB and Simulink are registered trademarks of The MathWorks, Inc. See www.mathworks.com/trademarks for a list of additional trademarks. Other product or brand names may be trademarks or registered trademarks of their respective holders.

Patents

MathWorks products are protected by one or more U.S. patents. Please see www.mathworks.com/patents for more information.

Revision History

April 2011	Online only	New for Version 8.0 (Release 2011a)
September 2011	Online only	Revised for Version 8.1 (Release 2011b)
March 2012	Online only	Revised for Version 8.2 (Release 2012a)
September 2012	Online only	Revised for Version 8.3 (Release 2012b)
March 2013	Online only	Revised for Version 8.4 (Release 2013a)
September 2013	Online only	Revised for Version 8.5 (Release 2013b)
March 2014	Online only	Revised for Version 8.6 (Release 2014a)
October 2014	Online only	Revised for Version 8.7 (Release 2014b)
March 2015	Online only	Revised for Version 8.8 (Release 2015a)
September 2015	Online only	Revised for Version 8.9 (Release 2015b)
October 2015	Online only	Rereleased for Version 8.8.1 (Release 2015aSP1)
March 2016	Online only	Revised for Version 8.10 (Release 2016a)
September 2016	Online only	Revised for Version 8.11 (Release 2016b)
March 2017	Online only	Revised for Version 8.12 (Release 2017a)
September 2017	Online only	Revised for Version 8.13 (Release 2017b)
March 2018	Online only	Revised for Version 8.14 (Release 2018a)
September 2018	Online only	Revised for Version 9.0 (Release 2018b)
March 2019	Online only	Revised for Version 9.1 (Release 2019a)
September 2019	Online only	Revised for Version 9.2 (Release 2019b)
March 2020	Online only	Revised for Version 9.3 (Release 2020a)
September 2020	Online only	Revised for Version 9.4 (Release 2020b)
March 2021	Online only	Revised for Version 9.5 (Release 2021a)
September 2021	Online only	Revised for Version 9.6 (Release 2021b)
March 2022	Online only	Revised for Version 9.7 (Release 2022a)
September 2022	Online only	Revised for Version 9.8 (Release 2022b)
March 2023	Online only	Revised for Version 9.9 (Release 2023a)

Check Bug Reports for Issues and Fixes

Software is inherently complex and is not free of errors. The output of a code generator might contain bugs, some of which are not detected by a compiler. MathWorks reports critical known bugs brought to its attention on its Bug Report system at www.mathworks.com/support/bugreports/. In the search bar, type the phrase "Incorrect Code Generation" to obtain a report of known bugs that produce code that might compile and execute, but still produce wrong answers. To save a search, click Save Search.

The bug reports are an integral part of the documentation for each release. Examine periodically all bug reports for a release, as such reports may identify inconsistencies between the actual behavior of a release you are using and the behavior described in this documentation.

In addition to reviewing bug reports, you should implement a verification and validation strategy to identify potential bugs in your design, code, and tools.

Model Architecture and Design

	Design Preparation
1	
	Divide Software Problem 1-2
	Map System Architecture to Simulink Modeling Environment 1-3
	Modeling Algorithms 1-3
	Modeling Interfaces 1-4
	Modeling Systems 1-5
	Modeling Run-Time Environments 1-6
	Application Interfaces
2	
	Model Code Interfaces 2-2
	Periodic and Aperiodic Function Interfaces 2-4
	Rate-Based Top Models 2-4
	Export-Function Top Models 2-5
	Periodic and Aperiodic Function Interface Configuration 2-7
	Startup, Reset, and Shutdown Function Interfaces 2-8
	Generate Code for Initialize and Terminate Events 2-9
	Generate Code for Reset Events 2-12
	Event Names and Code Aggregation 2-14
	Limitation 2-15
	Data Interfaces 2-17
	C Data Interfaces 2-17
	Message Communication Interfaces 2-18
	Generate C++ Messages to Communicate Data Between Simulink Components 2-18
	Generate C Messages to Communicate Data Between Simulink Components 2-22
	Client-Server Communication Interfaces 2-29
	Device Driver I/O Interfaces 2-30

3

Use Products and Blocks Supported for Code Generation	3-2
Related Products	3-2
Blocks Supported for Code Generation	3-4
Block Set Support for Code Generation	3-4
Consider Modeling Semantics when Generating Code	3-5
Data Propagation	3-5
Sample Time Propagation	3-6
Latches for Subsystem Blocks	3-7
Block Execution Order	3-8
Algebraic Loops	3-9
Modeling Guidelines for Blocks	3-11
Modeling Guidelines for Model Configuration	3-12

Timers in Simulink Coder

4

Absolute and Elapsed Time Computation	4-2
About Timers	4-2
Timers for Periodic and Asynchronous Tasks	4-2
Allocation of Timers	4-3
Integer Timers in Generated Code	4-3
Elapsed Time Counters in Triggered Subsystems	4-3
Access Timers Programmatically	4-4
About Timer APIs	4-4
C API for S-Functions	4-4
TLC API for Code Generation	4-6
Specify Clock Resolution Used by Target Environment Clock	4-7
Generate Code for an Elapsed Time Counter	4-11
Absolute Time Limitations	4-14

Time-Based Scheduling in Simulink Coder

5

Time-Based Scheduling and Code Generation	5-2
Sample Time Considerations	5-2
Tasking Modes	5-2
Model Execution and Rate Transitions	5-3
Execution During Simulink Model Simulation	5-4

Model Execution in Real Time	5-4
Single-Tasking Versus Multitasking Operation	5-4
Modeling for Single-Tasking Execution	5-6
Single-Tasking Mode	5-6
Build a Program for Single-Tasking Execution	5-6
Single-Tasking Execution	5-6
Modeling for Multitasking Execution	5-9
Multitasking and Pseudomultitasking Modes	5-9
Build a Program for Multitasking Execution	5-11
Execute Multitasking Models	5-11
Multitasking Execution	5-12
Handle Rate Transitions	5-16
Rate Transitions	5-16
Data Transfer Problems	5-17
Data Transfer Assumptions	5-17
Rate Transition Block Options	5-18
Automatic Rate Transition	5-19
Visualize Inserted Rate Transition Blocks	5-20
Periodic Sample Rate Transitions	5-22
Protect Data Integrity with volatile Keyword	5-26
Separate Rate Transition Block Code and Data from Algorithm Code and Data	5-26
Configure Time-Based Scheduling	5-33
Configure Start and Stop Times	5-33
Configure the Solver Type	5-33
Configure the Tasking Mode	5-33
Time-Based Scheduling Example Models	5-35
Optimize Memory Usage for Time Counters	5-35
Single-Rate Modeling (Bare Board, No OS)	5-40
Multirate Modeling in Single-Tasking Mode (Bare Board, No OS) ..	5-40
Multirate Modeling in Multitasking Mode (Bare Board, No OS)	5-41
Multirate Modeling in Multitasking Mode (VxWorks® OS)	5-42
Trade Determinism and Data Integrity to Improve System Performance	5-43

Event-Based Scheduling in Simulink Coder

6

Asynchronous Events	6-2
Asynchronous Support	6-2
Block Library for Calls to an Example Real-Time Operating System	6-2
Examples and Additional Information	6-3
Generate Interrupt Service Routines	6-4
Connecting the Async Interrupt Block	6-4
Requirements and Restrictions	6-4

Performance Considerations	6-5
Dual-Model Approach for Developing Real-Time Systems that Include ISRs	6-5
Dual-Model Approach: Simulation	6-6
Dual-Model Approach: Code Generation	6-6
Spawn and Synchronize Execution of RTOS Task	6-11
Pass Asynchronous Events in RTOS as Input to a Referenced Model	6-28
Rate Transitions and Asynchronous Blocks	6-34
About Rate Transitions and Asynchronous Blocks	6-34
Handle Rate Transitions for Asynchronous Tasks	6-35
Handle Multiple Asynchronous Interrupts	6-35
Protect Data Integrity with volatile Keyword	6-37
Timers in Asynchronous Tasks	6-38
Create a Customized Asynchronous Library	6-40
About Implementing Asynchronous Blocks	6-40
Async Interrupt Block Implementation	6-40
Task Sync Block Implementation	6-43
asynclib.tlc Support Library	6-45
Import Asynchronous Event Data for Simulation	6-47
Capabilities	6-47
Input Data Format	6-47
Example	6-47
Asynchronous Support Limitations	6-50
Asynchronous Task Priority	6-50
Convert an Asynchronous Subsystem into a Model Reference	6-50

Subsystems in Simulink Coder

7

Generate Code and Executables for Individual Subsystems	7-2
Subsystem Build Limitations	7-3
Generate Inlined Subsystem Code	7-5
Configure Subsystem to Inline Code	7-5
Exceptions to Inlining	7-6
See Also	7-6
Generate Subsystem Code as Separate Function and Files	7-7
Subsystem Function Dependence	7-7
Generate Subsystem as a Reusable Function	7-7
Generate Subsystem as a Nonreusable Function	7-8

Generate Code for Model Reference Hierarchy	8-2
Model Reference Technology	8-2
Generate Code for Model Reference Hierarchy	8-2
Working with Code Generation Folders	8-6
Build Model Reference Targets	8-7
Set Configuration Parameters for Code Generation of Model Hierarchies	
Configuration Parameter Requirements	8-8
Configuration Parameters Changed During Code Generation	8-10
Naming Requirements	8-11
Custom Target Requirements	8-11
Miscellaneous Considerations	8-11
Convert Subsystem to Referenced Model and Generate Code	
Configure Code Mappings	8-13
Convert Subsystem to Referenced Model	8-13
Copy Code Mappings After Subsystem Conversion	8-15
Review Generated Code	8-15
Generate Reusable Code from Model Reference Hierarchy	
Storage Classes for Parameters and Signals Used in Model Blocks	
Storage Classes for Parameters in Model Blocks	8-19
Storage Classes for Signals in Model Blocks	8-20
Signal Name Mismatches Across Model Reference Boundary	8-20
Code Generation Limitations for Model Reference	
Storage Class Limitations	8-22
Data Logging Limitations	8-22
State Initialization Limitations	8-22
Model Reuse Limitations	8-22
S-Functions Limitations	8-23
Subsystems Limitations	8-23
Targets Limitation	8-23
Conditional Models Limitation	8-23
Local Solver Limitation	8-23
Simulink Cache Files for Incremental Code Generation	
Contents of Simulink Cache Files	8-24
Share Code Generation Artifacts	8-24
Specify Instance-Specific Parameter Values for Reusable Referenced Model	
Pass Parameter Data to Referenced Model Entry-Point Functions as Arguments	8-27
Control Data Types of Model Arguments and Argument Values	8-49
Generated Functions in a Model Reference Hierarchy	
Generated Functions in a Top Model	8-51

Combined Models in Simulink Coder

9

Combine Code Generated for Multiple Models	9-2
Considerations for Combining Code for Different Models	9-2
Control Ownership of Data	9-3
Combine Code Generated for Multiple Models or Multiple Instances of a Model	9-3
Function Reuse in Generated Code	9-5
File Packaging for Models (Code and Data)	9-10

Code Reuse for Simulink Coder

10

What Is Code Reuse?	10-2
What Is Reentrant Code?	10-3
Choose a Componentization Technique for Code Reuse	10-4
Simulink Function Blocks and Code Generation	10-6
Why Generate Code from Simulink Function Blocks and Function Callers?	10-6
Implementation Options	10-6
Choose a Modeling Pattern	10-6
Specify Function Scope	10-8
Decide Whether to Generate Export Function Code	10-8
Configure Function Code Interfaces	10-8
Uncalled Simulink Function Blocks	10-8
Requirements	10-9
Limitation	10-9
Generate Code for Simulink Function and Function Caller	10-9
Generate Reentrant Code from Top Models	10-13
Generate Reentrant, Multi-Instance Code	10-13
Share Data Between Instances	10-16
Generate Reentrant Code from Simulink Function Blocks	10-18
Identify Requirements	10-18
Create Model	10-18
Configure Model and Model Elements	10-20
Generate and Inspect C Code	10-22
Generate and Inspect C++ Code	10-24
Limitations	10-26

Generate Reentrant Code from Subsystems	10-28
Function Reuse in Generated Code	10-29
Limitations	10-33
Generate Reusable Code from Library Subsystems Shared Across Models	10-36
What Is a Reusable Library Subsystem?	10-36
Reusable Library Subsystem Code Placement and Naming	10-36
Configure Models That Include Reusable Library Subsystems	10-38
Generate Reusable Code for Subsystems Shared Across Models ..	10-39
Generate Reusable Code from Stateflow Atomic Subcharts	10-46
Generate Reusable Code for Linked Atomic Subcharts	10-46
Generate Reusable Code for Unlinked Atomic Subcharts	10-46
Generate Reusable Code for Unit Testing	10-47
Generate Shared Utility Code	10-52
When to Generate Shared Utility Code	10-52
Configure Naming of Generated Functions	10-52
Control Placement of Shared Utility Code	10-53
Control Placement of rtwtypes.h for Shared Utility Code	10-54
Avoid Duplicate Header Files for Exported Data	10-54
Reduce Shared Utility Code Generation with Incremental Builds .	10-54
Manage the Shared Utility Code Checksum	10-55
Generate Shared Utility Code for Fixed-Point Functions	10-60
Generate Shared Utility Code for Custom Data Types	10-62
Shared Constant Parameters for Code Reuse	10-64
Suppress Shared Constants in the Generated Code	10-65
Determine Why Subsystem Code Is Not Reused	10-67
Review Subsystems Section of HTML Code Generation Report ...	10-67
Compare Subsystem Checksum Data	10-67
Generate Reusable Code from For Each Subsystems	10-70

Configure Model Parameters for Simulink Coder

11

Configure Run-Time Environment Options	11-2
Configure Production and Test Hardware	11-2
Production Hardware Considerations	11-8
Test Hardware Considerations	11-8
Example Production Hardware Setting That Affects Normal Mode Simulation	11-9
Using the Target Framework	11-11
Target Framework Overview	11-11
Target Framework Workflows	11-11

Register New Hardware Devices	11-13
Specify Hardware Implementation for New Device	11-13
Specify Hardware Implementation That Persists Over MATLAB Sessions	11-14
Create Hardware Implementation by Modifying Existing Implementation	11-14
Create Hardware Implementation by Reusing Existing Implementation	11-15
Validate Hardware Device Data	11-15
Export Hardware Device Data	11-16
Create Alternative Identifier for Target Object	11-16
Upgrade Data Definitions for Hardware Devices	11-17

Model Protection in Simulink Coder

12

Protect Models to Conceal Contents	12-2
Prepare the Parent Model	12-2
Protect the Referenced Model	12-3
Protected Model Requirements and Limitations	12-6
Code Generation Requirements and Limitations	12-7
Nested Protected Model Requirements and Limitations	12-8
 Create Protected Models with Multiple Targets	 12-9
 Test Protected Models	 12-10
 Package and Share Protected Models	 12-12
Harness Model	12-12
MAT-File with Base Workspace Definitions	12-12
Simulink Data Dictionary	12-13
Protected Model File Contents	12-13
 Specify Custom Obfuscators for Protected Models	 12-16
 Define Callbacks for Protected Models	 12-17
Creating Callbacks	12-17
Defining Callback Code	12-17
Create a Protected Model with Callbacks	12-18
 Sign a Protected Model	 12-19
Sign Your Protected Model	12-19
 Best Practices for Creating Protected Models	 12-21
Coordinate Settings with the User of Your Protected Model	12-21
Minimize the Model External Interface	12-21
Contents to Include in Your Protected Model	12-23

13

Code Generation of Stateflow Blocks	13-2
Comparison of Code Generation Methods	13-2
Generate Code from Atomic Subcharts	13-5
Generate Reusable Code for Unlinked Atomic Subcharts	13-5
Generate Reusable Code for Linked Atomic Subcharts	13-5
Inline State Functions in Generated Code	13-7
Inlined Generated Code for State Functions	13-7
How to Set the State Function Inline Option	13-8
Best Practices for Controlling State Function Inlining	13-8

Block Authoring and Code Generation for Simulink Coder

14

S-Functions and Code Generation	14-2
Types of S-Functions	14-2
Files Required for Implementing Noninlined and Inlined S-Functions	14-4
Guidelines for Writing S-Functions that Support Code Generation	14-4
Import Calls to External Code into Generated Code with Legacy Code Tool	14-6
Legacy Code Tool and Code Generation	14-6
Generate Inlined S-Function Files for Code Generation	14-6
Apply Code Style Settings to Legacy Functions	14-7
Address Dependencies on Files in Different Locations	14-8
Deploy S-Functions for Simulation and Code Generation	14-8
Integrate External C++ Objects	14-8
Legacy Code Tool Examples	14-11
Integrate External C Functions That Pass Input Arguments by Value Versus Address	14-11
Integrate External C Functions That Pass the Output Argument as a Return Argument	14-14
Integrate External C++ Object Methods	14-16
Integrate External C Functions That Pass Input and Output Arguments as Parameters with a Fixed-Point Data Type	14-18
Integrate External C Functions That Pass Input and Output Arguments as Signals with a Fixed-Point Data Type	14-21
Integrate External C Functions with Instance-Specific Persistent Memory	14-23
Integrate External C Functions That Use Structure Arguments ...	14-25
Integrate External C Functions That Pass Input and Output Arguments as Signals with Complex Data	14-28
Integrate External C Functions That Pass Arguments That Have Inherited Dimensions	14-30

Integrate External C Functions That Pass Arguments as Multi-Dimensional Signals	14-33
Integrate External C Functions That Implement Start and Terminate Actions	14-35
Integrate External C Functions That Implement N-Dimensional Table Lookups	14-37
Integrate External C Functions That Pass Arguments as Multi-Dimensional Signals	14-40
Integrate External C Functions with a Block Sample Time Specified, Inherited, and Parameterized	14-43
Integrate External C Functions That Pass Input and Output Arguments as Parameters with a Fixed-Point Data Type	14-47
Integrate External C Functions That Implement N-Dimensional Table Lookups	14-50
Integrate External C++ Object Methods	14-53
External Code Integration Examples	14-56
Integrate External C++ Code into a Model Using S-Functions ...	14-56
Insert External C and C++ Code into Stateflow Charts for Code Generation	14-57
Integrate External C Code into Generated Code by Using Custom Code Blocks and Model Configuration Parameters	14-59
Integrate External C Code into Generated Code by Using Custom Code Blocks and Model Configuration Parameters	14-60
Insert External C++ Code into Stateflow Charts for Code Generation	14-62
S-Functions That Support Expression Folding	14-64
Categories of Output Expressions	14-64
Acceptance or Denial of Requests for Input Expressions	14-67
Expression Folding in a TLC Block Implementation	14-69
S-Functions That Specify Port Scope and Reusability	14-73
S-Functions That Specify Sample Time Inheritance Rules	14-77
S-Functions for Code Reuse	14-81
S-Functions for Multirate Multitasking Environments	14-86
About S-Functions for Multirate Multitasking Environments	14-86
Rate Grouping Support in S-Functions	14-86
Create Multitasking, Multirate, Port-Based Sample Time S-Functions	14-86
Write Noninlined S-Function	14-91
Guidelines for Writing Noninlined S-Functions	14-91
Noninlined S-Function Parameter Type Limitations	14-92
Write Wrapper S-Function and TLC Files	14-93
MEX S-Function Wrapper	14-93
TLC S-Function Wrapper	14-96
Code Overhead for Noninlined S-Functions	14-96

Inline a Wrapper S-Function	14-97
The Inlined Code	14-99
Write Fully Inlined S-Functions	14-100
Multiport S-Function	14-100
Guidelines for Writing Inlined S-Functions	14-100
Write Fully Inlined S-Functions with mdlRTW Routine	14-102
S-Function RTWdata	14-102
Direct-Index Lookup Table Algorithm	14-103
Direct-Index Lookup Table Example	14-104
Error Handling	14-105
User Data Caching	14-105
mdlRTW Usage	14-105

Data, Function, and File Definition

15

Data Representation in Simulink Coder

Preserve Variables in Generated Code	15-2
Access Signal, State, and Parameter Data During Execution	15-2
Configure Data Accessibility	15-15
Limitations	15-16
Access Signal, State, and Parameter Data During Execution	15-17
Data Structures in the Generated Code	15-31
Standard Data Structures	15-31
Real-Time Model Data Structure	15-32
How Generated Code Exchanges Data with an Environment	15-35
Data Interfaces in the Generated Code	15-35
Configure Data Interface	15-42
Control Data and Function Interface in Generated Code	15-44
Control Type Names, Field Names, and Variable Names of Standard I/O Structures (Embedded Coder)	15-44
Control Names of Generated Entry-Point Functions (Embedded Coder)	15-44
Control Data Interface for Nonreentrant Code	15-44
Control Data Interface for Reentrant Code	15-46
Prevent Unintended Changes to the Interface	15-46
Reduce Number of Arguments by Using Structures	15-47
Control Data Types of Arguments	15-48
Promote Data Item to the Interface	15-48

How Generated Code Stores Internal Signal, State, and Parameter Data	15-49
Internal Data in the Generated Code	15-49
Local Variables in the Generated Code	15-65
Appearance of Test Points in the Generated Code	15-65
Appearance of Workspace Variables in the Generated Code	15-65
Promote Internal Data to the Interface	15-66
Control Default Representation of Internal Data (Embedded Coder)	15-67
Choose Data Configuration Approach	15-70
Data Element Categories	15-70
Where to Store Data Elements Used by Your Model	15-71
When to Configure a Data Element for Code Generation	15-71
Configure a Data Element for Code Generation	15-72
Choose Storage Class for Controlling Data Representation in Generated Code	15-75
Storage Class Properties	15-75
Default Storage Class	15-75
Built-in and Predefined Storage Classes	15-75
Auto	15-77
Default	15-77
Model Default	15-77
Dictionary Default	15-77
ExportedGlobal	15-78
ImportedExtern, ImportedExternPointer	15-78
Bitfield	15-79
CompilerFlag	15-79
Const, Volatile, and ConstVolatile	15-79
Define, ImportedDefine	15-80
ExportToFile	15-80
FileScope	15-81
GetSet	15-81
ImportFromFile	15-82
Localizable	15-82
Reusable	15-83
Struct	15-83
MultiInstance	15-83
ParamStruct, SignalStruct	15-84
Storage Class Limitations	15-84
Use Enumerated Data in Generated Code	15-86
Enumerated Data Types	15-86
Specify Integer Data Type for Enumeration	15-86
Customize Enumerated Data Type	15-87
Control Enumerated Type Implementation in Generated Code	15-91
Type Casting for Enumerations	15-92
Enumerated Type Limitations	15-92
Data Stores in Generated Code	15-94
About Data Stores	15-94
Generate Code for Data Store Memory Blocks	15-94
Storage Classes for Data Store Memory Blocks	15-95
Data Store Buffering in Generated Code	15-96
Data Stores Shared by Instances of a Reusable Model	15-99

Structures in Generated Code Using Data Stores	15-99
Specify Single-Precision Data Type for Embedded Application ..	15-104
Use single Data Type as Default for Underspecified Types	15-104
Tune Phase Parameter of Sine Wave Block During Code Execution	15-107
Switch Between Output Waveforms During Code Execution for Waveform Generator Block	15-109
Create Tunable Calibration Parameter in the Generated Code ..	15-112
Represent Block Parameter as Tunable Global Variable	15-112
Apply Storage Class When Block Parameter Refers to Numeric MATLAB Variable	15-115
Create Storage Class That Represents Calibration Parameters (Embedded Coder)	15-115
Initialize Parameter Value From System Constant or Other Macro (Embedded Coder)	15-118
Code Generation Impact of Storage Location for Parameter Objects	15-119
Configure Accessibility of Signal Data	15-120
Programmatic Interfaces for Tuning Parameters	15-120
Set Tunable Parameter Minimum and Maximum Values	15-120
Considerations for Other Modeling Goals	15-121
Limitations for Block Parameter Tunability in Generated Code	15-122
Tunable Expression Limitations	15-122
Linear Block Parameter Tunability	15-123
Parameter Structures	15-124
Code Generation of Parameter Objects With Expression Values	15-125
Considerations and Limitations	15-125
Expression Preservation	15-125
Configure Packaging of Parameter Arguments in Generated Code	15-127
Code Generation Behavior	15-127
Limitations	15-130
Parameter Data Types in the Generated Code	15-131
Significance of Parameter Data Types	15-131
Typecasts Due to Parameter Data Type Mismatches	15-131
Considerations for Other Modeling Patterns	15-132
Generate Efficient Code by Specifying Data Types for Block Parameters	15-135
Eliminate Unnecessary Typecasts and Shifts by Matching Data Types	15-135
Reduce Memory Consumption by Storing Parameter Value in Small Data Type	15-137
Reuse Parameter Data in Different Data Type Contexts	15-143

Organize Data into Structures in Generated Code	15-147
Techniques to Create Structures	15-150
Default Application of Structured Storage Class	15-151
Direct Application of Structured Storage Class	15-152
Nonvirtual Buses and Parameter Structures	15-152
Combine Techniques to Work Around Limitations	15-159
Arrays of Structures	15-160
Structure Padding	15-160
Limitations	15-160
Switch Between Sets of Parameter Values During Simulation and Code Execution	15-162
Design Data Interface by Configuring Inport and Outport Blocks	15-170
Generate Efficient Code for Bus Signals	15-174
Code Efficiency for Bus Signals	15-174
Set Bus Diagnostics	15-174
Optimize Virtual and Nonvirtual Buses	15-175
Control Signal and State Initialization in the Generated Code ..	15-177
Signal and State Initialization in the Generated Code	15-177
Generate Tunable Initial Conditions	15-178
Generate Tunable Initial Condition Structure for Bus Signal	15-182
Initialization of Signal, State, and Parameter Data in the Generated Code	15-188
Static Initialization and Dynamic Initialization	15-188
Real-World Ground Initialization Requiring Nonzero Bit Pattern .	15-188
Initialization of Signal and State Data	15-189
Initialization of Parameter Data	15-190
Data Initialization in the Generated Code	15-190
Modeling Goals	15-195
Optimize Generated Code Using Fixed-Point Data with Simulink, Stateflow, and MATLAB	15-197
Declare Workspace Variables as Tunable Parameters Using the Model Parameter Configuration Dialog Box	15-198
Declare Existing Workspace Variables as Tunable Parameters ..	15-198
Declare New Tunable Parameters	15-198
Set Tunable Parameter Code Generation Options	15-199
Programmatically Declare Workspace Variables as Tunable Parameters	15-199
Share Data Between Code Generated from Simulink, Stateflow, and MATLAB	15-201

C Code Generation Configuration for Model Interface Elements	16-2
Migration	16-2
Code Definitions	16-3
Data Default Configurations	16-3
Configurations for Individual Data Elements	16-6
Configure Data	16-6
Code Definition and Mapping Limitations and Considerations	16-8
Limitations	16-8
Default Code Mapping Constraints	16-8
Model Editing Operations and Individual Data Element Code Mappings	16-8
Storage Classes and Reentrant, Multi-Instance Models and Components	16-8
Copy Code Mappings When Converting Subsystems to Referenced Models	16-9
Migration of Model Data Configurations to Code Mappings	16-11
Effects of Migration	16-11
Considerations Before Migrating	16-12
Copy Blocks and Signal Lines Between Models	16-15
Switch Between GRT- and ERT-Based System Target Files	16-16
Workflow Changes	16-17
Configure Root-Level Inport Blocks for C Code Generation	16-18
Choose Customization Options for Inport Interface	16-18
Configure Default Code Generation Settings for Root-Level Inports	16-19
Configure Code Generation Settings for Individual Root-Level Inports	16-20
Configure Code Generation Settings for Root-Level Inports Programmatically	16-22
Choose Storage Class and Storage Class Properties for Root-Level Inports	16-25
Configure Root-Level Outport Blocks for C Code Generation	16-27
Choose Customization Options for Outport Interface	16-27
Configure Default Code Generation Settings for Root-Level Outports	16-28
Configure Code Generation Settings for Individual Root-Level Outports	16-30
Configure Code Generation Settings for Root-Level Outports Programmatically	16-31
Choose Storage Class and Storage Class Properties for Root-Level Outports	16-32
Configure Signal Data for C Code Generation	16-34
Choose Code Configuration Options for Signals	16-34
Add Signals to Model Code Mappings	16-35
Configure Default Code Generation Settings for Signals	16-37
Configure Code Generation Settings for Individual Signals	16-38

Remove Signals from Model Code Mappings	16-40
Configure Code Generation Settings for Signals Programmatically	16-40
Choose Storage Class and Storage Class Properties for Signals ..	16-42
Configure Parameters for C Code Generation	16-44
Choose Customization Options for Parameters	16-44
Configure Default Code Generation Settings for Parameters	16-46
Configure Code Generation for Individual Parameters	16-47
Configure Code Generation Settings for Parameters Programmatically	16-49
Choose Storage Class and Storage Class Properties for Model Parameters	16-51
Configure Block States for C Code Generation	16-53
Choose Code Configuration Options for States	16-53
Configure Default Code Generation Settings for States	16-54
Configure Code Generation Settings for Individual States	16-55
Configure Code Generation Settings for States Programmatically	16-57
Choose Storage Class and Storage Class Properties for States ...	16-58
Configure Data Stores for C Code Generation	16-60
Choose Customization Options for Data Stores	16-60
Configure Default Code Generation Settings for Data Stores	16-61
Configure Code Generation for Individual Data Stores	16-63
Configure Code Generation Settings for Data Stores Programmatically	16-64
Choose Storage Class and Storage Class Properties for Data Stores	16-66
Manage Multiple Code Mappings for a Model	16-68
Configure Alternative Code Mappings by Using Configuration Sets	16-68
Activate Code Mappings in the Simulink Editor	16-69
Create and Activate Code Mappings Programmatically	16-70
Code Mappings and Embedded Coder Dictionary	16-70
Configure Model Data Interface for C Code Generation	16-72
Configure Generated C Function Interface for Model Entry-Point Functions	16-79
What Is an Entry-Point Function?	16-79
Types of Generated Entry-Point Functions	16-79
Configure Whether Entry-Point Functions Are Reusable	16-82
How to Interface with Generated Entry-Point Functions	16-83
C Function Interface Customization Limitations	16-83

Configure Entry-Point Function Interfaces for Simulink Function and Function Caller Blocks	17-2
Configure Generated C Function Interface for Global Simulink Function Block	17-2
Configure Generated C Function Interface for Exported Scoped Simulink Function Block	17-6
Configure Generated C++ Function Interface for Exported Scoped Simulink Function Block	17-10
Simulink Function Code Interface Limitations	17-14
Generate C++ Class Interface to Model or Subsystem Code	17-15
Generate C++ Class Interface to Model Code	17-15
Generate C++ Class Interface to Nonvirtual Subsystem Code ...	17-16
C++ Class Interface Limitations	17-16
Execution of Code Generated from a Model	17-18
Program Execution	17-18
Program Timing	17-19
External Mode Communication	17-19
Data Logging in Single-Tasking and Multitasking Model Execution	17-20
Non-Real-Time Single-Tasking Systems	17-20
Non-Real-Time Multitasking Systems	17-21
Real-Time Single-Tasking Systems	17-22
Real-Time Multitasking Systems	17-23
Multitasking Systems Using Real-Time Tasking Primitives	17-24
Rapid Prototyping and Embedded Model Execution Differences ..	17-25
Rapid Prototyping Model Functions	17-27

Column-Major Layout to Row-Major Layout Conversion of Models with Lookup Table Blocks	18-2
Interpolation Algorithm for Row-Major Array Layout	18-10
Interpolation with Subtable Selection Algorithm for Row-Major Array Layout	18-16
Direct Lookup Table Algorithm for Row-Major Array Layout	18-22
Generate Row-Major Code for S-Functions	18-29
Row-Major Algorithms for Row-Major Array Layout	18-32

Generate Row-Major Code for Model That Contains a MATLAB Function Block	18-35
Row-Major Code Generation	18-35
Mixed-Majority Code Generation	18-36

Configuring Data in Generated Code

19

Configure Multi-Instance Code Generation	19-2
Configure a Top Model for Multi-Instance Code Generation	19-2
Configure a Referenced Model for Multi-Instance Code Generation	19-2
Configure Data Interfaces	19-3
Configure Internal Data	19-4

Code Generation

Configuration for Simulink Coder

20

Code Generation Configuration	20-2
Open the Model Configuration for Code Generation	20-2
Configuration Tools	20-2
Configure Code Generation Parameters for Model Programmatically	20-4
Modify Parameters to Support Execution efficiency	20-4
Configure Model from Command Line	20-6
Use Configuration Reference to Select Code Generation Target ..	20-11
Check Model and Configuration for Code Generation	20-16
Check Model for Code Efficiency with Model Advisor	20-16
Check Model During Code Generation by using the Code Generation Advisor	20-16
Application Objectives Using Code Generation Advisor	20-18
High-Level Code Generation Objectives	20-18
Configure Model for Code Generation Objectives Using Code Generation Advisor	20-19
Configure Model for Code Generation Objectives by Using Configuration Parameters Dialog Box	20-19

Simulink Coder Model Advisor Checks for Standards and Code Efficiency	20-21
Configure Code Comments	20-22
Include MATLAB Code as Comments in Generated Code	20-23
Include MATLAB Code as Comments in the Generated Code	20-23
Location of Comments in Generated Code	20-23
Include MATLAB User Comments in Generated Code	20-25
Limitations of MATLAB Source Code as Comments	20-26
Construction of Generated Identifiers	20-27
Identifier Name Collisions and Mangling	20-28
Identifier Name Collisions with Referenced Models	20-28
Specify Identifier Length to Avoid Naming Collisions	20-29
Specify Reserved Names for Generated Identifiers	20-30
Reserved Keywords	20-31
C Reserved Keywords	20-31
C++ Reserved Keywords	20-31
Other Common Reserved Keywords	20-32
Code Generation Code Replacement Library Keywords	20-32
Configure Model for Debugging	20-34

System Target File Configuration

21

Configure a System Target File	21-2
Select a Solver That Supports Code Generation	21-2
Select a System Target File from STF Browser	21-2
Select a System Target File Programmatically	21-3
Develop Custom System Target Files	21-4
Configure STF-Related Code Generation Parameters	21-5
Specify Generated Code Interfaces	21-5
Configure Numeric Data Support	21-8
Configure Time Value Support	21-8
Configure Noninlined S-Function Support	21-9
Configure Model Function Generation and Argument Passing	21-9
Configure Code Reuse Support	21-10
Configure Standard Math Library for Target System	21-12
Compare System Target File Support Across Products	21-17
Compare Product System Target Files	21-18
Compare Code Styles and STF Support	21-20
Compare Generated Code Features by Product	21-21
Compare Generated Code Features by STF	21-23

Internationalization Support in Simulink Coder

22

Internationalization and Code Generation	22-2
Locale Settings	22-2
Prepare to Generate Code for Mixed Languages and Locales	22-2
Character Set Limitations	22-2
XML Escape Sequence Replacements	22-2
Generate and Review Code with Mixed Languages and Mixed Locales	22-3

Basic Code Generation for Simulink Coder

23

Generate Code by Using the Quick Start Tool	23-2
Configuration Parameter Changes for Models That Have a Configuration Reference	23-2
Next Steps	23-2
Generate Code Using Simulink Coder	23-4
Aircraft Position Radar Model	23-10

Source Code Generation in Simulink Coder

24

Configure Model, Generate Code, and Simulate	24-2
About This Example	24-2
Functional Design of the Model	24-2
View the Top Model	24-2
View the Subsystems	24-3
Simulation Test Environment	24-4
Run Simulation Tests	24-8
Key Points	24-9
Learn More	24-10
Configure Model and Generate Code	24-11
About This Example	24-11
Configure the Model for Code Generation	24-11
Save Your Model Configuration as a MATLAB Function	24-12
Check Model Conditions and Configuration Settings	24-13
Generate Code for the Model	24-13
Review the Generated Code	24-13
Generate an Executable	24-14
Key Points	24-14
Configure Data Interface	24-16
About This Example	24-16

Declare Data	24-16
Use Data Objects	24-17
Add New Data Objects	24-19
Enable Data Objects for Generated Code	24-19
Effects of Simulation on Data Typing	24-20
Manage Data	24-21
Key Points	24-22
Call External C Functions	24-23
About This Example	24-23
Include External C Functions in a Model	24-23
Create a Block That Calls a C Function	24-23
Validate External Code in the Simulink Environment	24-25
Validate C Code as Part of a Model	24-26
Call a C Function from Generated Code	24-27
Key Points	24-27
Reload Generated Code	24-29
Manage Build Process Folders	24-30
File Generation Control Parameters	24-30
Build Process Folders	24-31
Manage Build Process Files	24-35
model.bat	24-39
model.h	24-39
rtwtypes.h	24-40
Manage Build Process File Dependencies	24-43
System Header Files	24-44
Code Generator Header Files	24-46
Add Build Process Dependencies	24-52
File Dependency Information for the Build Process	24-52
Folder Dependency Information for the Build Process	24-54
Build Process Support for File and Folder Names	24-57
Filenames with Spaces	24-57
Folder Names with Spaces	24-57
Troubleshooting Errors When Folder Names Have Spaces	24-60
Folder Names with Special Characters	24-61
Very Long Folder Paths	24-61
Code Generation of Matrices and Arrays	24-63
Array Storage in Computer Memory	24-63
Code Generator Matrix Parameters	24-65
Internal Data Storage for Complex Number Arrays	24-67
Unsupported Blocks for Row-Major Code Generation	24-67
Limitations for Row-Major Code Generation	24-68
Cross-Release Shared Utility Code Reuse	24-69
Workflow to Reuse Shared Utility Code	24-69
Required Editing for Shared Utility Code Reuse	24-70

Cross-Release Code Integration	24-71
Workflow	24-71
Limitations	24-73
Simulink.Bus Support	24-74
Root-Level I/O Through Global Variables in Generated Code	24-75
Communicate Between Current and Previous Release Components Through Global Data Stores	24-77
Parameter Tuning	24-78
Use Multiple Instances of Code Generated from Reusable Referenced Model	24-79
Compare Simulation Behavior of Model Component in Current Release and Generated Code from Previous Release	24-80
Import AUTOSAR Code from Previous Releases	24-81
Integration of Code from Multiple Folders	24-82
Integrate Generated Code by Using Cross-Release Workflow	24-85
Compile Code in Another Development Environment	24-93
Generate Code and Simulate Models in a Project	24-97
Generate Code and Simulate Models Programmatically with Project	24-100

Report Generation in Simulink Coder

25

Reports for Code Generation	25-2
HTML Code Generation Report Content	25-2
HTML Code Generation Report Location	25-2
HTML Code Generation Report for Referenced Models	25-2
HTML Code Generation Report Extensions	25-3
Generate Code Generation Report	25-5
Configure Model	25-5
Generate Report	25-5
Share Report	25-5
Limitation	25-6
Document Generated Code with Simulink Report Generator	25-7
Generate Code for the Model	25-7
Open the Report Generator	25-8
Set Report Name, Location, and Format	25-10
Include Models and Subsystems in a Report	25-11
Customize the Report	25-11
Generate the Report	25-12
Get Code Description of Generated Code	25-13
Get Generated Type and Name of Inport Structures	25-20

Analyze Generated Data Code Interface	25-23
Navigating Code Interface Report Subsections	25-23
Interpreting the Entry-Point Functions Subsection	25-24
Interpreting the Inports and Outports Subsections	25-27
Interpreting the Interface Parameters Subsection	25-28
Interpreting the Data Stores Subsection	25-29
Code Interface Report Limitations	25-29

Code Replacement in Simulink Coder

26

What Is Code Replacement?	26-2
Code Replacement Libraries	26-2
Code Replacement Terminology	26-4
Code Replacement Limitations	26-5
Choose a Code Replacement Library	26-6
About Choosing a Code Replacement Library	26-6
Explore Available Code Replacement Libraries	26-6
Explore Code Replacement Library Contents	26-6
Replace Code Generated from Simulink Models	26-8

Guidelines and Standards for Simulink Coder

27

Modeling Guidelines for Generated Code	27-2
Code Generation Modeling Guidelines	27-2
Industry Standards	27-4
IEC 61508: 2010	27-4
ISO 26262: 2018	27-4
IEC 62304: 2015	27-4
EN 50128 / EN 50128: 2011	27-5
ISO 25119 Standard	27-5
DO-178C Standard	27-5
AUTOSAR Standard	27-6
Modeling Guidelines and Model Advisor Checks for Verifying	
Compliance with Industry Standards	27-7
High-Integrity System Modeling Guidelines	27-7
MAB Modeling Guidelines	27-7
JMAAB Modeling Guidelines	27-7

Real-Time Systems in Simulink Coder

28

Deploy Algorithm Model for Real-Time Rapid Prototyping	28-2
About Real-Time Rapid Prototyping	28-2
Goals of Real-Time Rapid Prototyping	28-2
Refine Code With Real-Time Rapid Prototyping	28-2
Deploy Environment Model for Real-Time Hardware-In-the-Loop (HIL) Simulation	28-4
About Hardware-In-the-Loop Simulation	28-4
Set Up and Run HIL Simulations	28-4
Untrusted Custom Code, Custom Targets, and Callbacks	28-6

Desktops in Simulink Coder

29

Accelerate, Refine, and Test Hybrid Dynamic System on Host Computer by Using RSim System Target File	29-2
About Rapid Simulation	29-2
Rapid Simulation Advantage	29-2
General Rapid Simulation Workflow	29-2
Identify Rapid Simulation Requirements	29-3
Configure Inports to Provide Simulation Source Data	29-4
Configure and Build Model for Rapid Simulation	29-4
Set Up Rapid Simulation Input Data	29-6
Scripts for Batch and Monte Carlo Simulations	29-13
Run Rapid Simulations	29-13
Tune Parameters Interactively During Rapid Simulation	29-21
Rapid Simulation Target Limitations	29-22
Run Rapid Simulations over a Range of Parameter Values	29-24
Run Batch Simulations Without Recompiling Generated Code ..	29-30
Use MAT-Files to Feed Data to Inport Blocks for Rapid Simulations	29-35
Use S-Function Target for Model or Subsystem	29-40
Required Files for S-Function Deployment	29-40
Sample Time Propagation in Generated S-Functions	29-41
Solver Type for Top Models with Generated S-Functions	29-41
Tunable Parameters in Generated S-Functions	29-41
Macro Parameters	29-43
Checksums and the S-Function Target	29-43
Generated S-Function Compatibility	29-44

External Code Integration in Simulink Coder

Choose an External Code Integration Workflow 30-2

- Choose a Software Execution Framework for Scheduling Code Execution 30-3
- Evaluate Characteristics of External Code 30-4
- Identify Integration Requirements 30-6
- Choose a Workflow 30-7

Configure Target Hardware Characteristics 30-9

Check Code Generation Assumptions 30-12

- Check Code Generator Assumptions for Development Computer .. 30-12
- Coder Assumptions List 30-13

Call Reusable External Algorithm Code for Simulation and Code Generation 30-15

- Workflow 30-15
- Choose an Integration Approach 30-16
- Insert External Code into Stateflow Charts 30-21

Place External C/C++ Code in Generated Code 30-23

- Workflow 30-23
- Choose an Integration Approach 30-24
- Integrate External Code by Using Custom Code Blocks 30-25
- Integrate External Code by Using Model Configuration Parameters 30-27
- Integrate External C Code into Generated Code by Using Custom Code Blocks and Model Configuration Parameters 30-28

Call External Device Drivers 30-31

Apply Function and Operator Code Replacements 30-32

Build Integrated Code Within the Simulink Environment 30-33

- Workflow 30-33
- Configure Parameters for Integrated Code Build Process 30-33
- Preserve External Code Files in Build Folder 30-34
- Build Support for S-Functions 30-35

Generate Component Source Code for Export to External Code Base 30-40

- Modeling Options 30-40
- Requirements 30-40
- Limitations for Export-Function Subsystems 30-42
- Workflow 30-42
- Choose an Integration Approach 30-43
- Generate C Function Code for Export-Function Model 30-44

Generate C++ Function and Class Code for Export-Function Model	30-48
Generate Code for Export-Function Subsystems	30-52
Generate Shared Library for Export to External Code Base	30-54
About Generated Shared Libraries	30-54
Workflow	30-54
Generate Shared Libraries	30-55
Create Application Code That Uses Generated Shared Libraries ..	30-55
Limitations	30-58
Exchange Data Between External C/C++ Code and Simulink Model or Generated Code	30-59
Import External Code into Model	30-59
Export Generated Code to External Environment	30-60
Simulink Representations of C Data Types and Constructs	30-60
Considerations for Other Modeling Goals	30-65
Exchange Data Between External Calling Code and Generated Code	30-67
Data Exchange for Reentrant Generated Code	30-67
Data Exchange for Nonreentrant Generated Code	30-67
Generate Code That Matches Appearance of External Code	30-70

Program Building, Interaction, and Debugging in Simulink Coder

31

Select C or C++ Programming Language	31-2
Select and Configure C or C++ Compiler	31-3
Language Standards Compliance	31-3
Programming Language Considerations	31-4
C++ Language Support Limitations	31-4
Code Generator Assumes Wrap on Signed Integer Overflows	31-5
Choose and Configure Compiler	31-5
Include S-Function Source Code	31-6
Troubleshoot Compiler Issues	31-7
Compiler Version Mismatch Errors	31-7
Results for Model Simulation and Program Execution Differ	31-7
Generates Expected Code and Produces Unexpected Results	31-8
Compile-Time Issues	31-8
LCC Compiler Does Not Support Ampersands in Source Folder Paths	31-9
LCC Compiler Might Not Support Line Lengths of Rapid Accelerator Code	31-9
Approaches for Building Code Generated from Simulink Models	31-11

Configure CMake Build Process	31-13
Specify CMake Toolchain Definition	31-13
Available CMake Toolchain Definitions	31-14
Generate Only Code and CMake Configuration File for Algorithm Export	31-15
Create Custom CMake Toolchain Definition	31-17
Configure Toolchain (ToolchainInfo) or Template Makefile Build Process	31-20
Toolchain Approach	31-20
Upgrade Model to Use Toolchain Approach	31-22
Template Makefile Approach	31-24
Specify TLC for Code Generation	31-28
Use of Checksums in Recompilation of Generated Code	31-28
Define Custom Makefile-Based Toolchains Using Target Framework	31-30
Overview of Makefile-Based Toolchain Definition	31-30
Define GCC-Based Toolchain for Windows Development Computer	31-30
Define Microsoft Visual Studio Toolchain for Building 32-Bit Application	31-32
Template Makefiles and Make Options	31-36
Types of Template Makefiles	31-36
Specify Template Makefile Options	31-36
Template Makefiles for UNIX Platforms	31-37
Template Makefiles for the Microsoft Visual C++ Compiler	31-37
Template Makefiles for the LCC Compiler	31-38
Build Process Workflow for Real-Time Systems	31-39
Working Folder	31-39
Build Folder and Code Generation Folders	31-39
Set Model Parameters for Code Generation	31-39
Configure Build Process	31-40
Set Code Generation Parameters	31-41
Build and Run a Program	31-42
Contents of the Build Folder	31-43
Customized Makefile Generation	31-44
Build Models from a Windows Command Prompt Window	31-45
Rebuild a Model	31-48
Control Regeneration of Top Model Code	31-49
Regeneration of Top Model Code	31-49
Force Regeneration of Top Model Code	31-49
Reduce Build Time for Referenced Models by Using Parallel Builds	31-51
Build Referenced Models in Parallel	31-51
Monitor Parallel Building of Referenced Models	31-51

Relocate or Share Generated Code	31-56
Package and Relocate Generated Code	31-56
Package and Relocate Generated Code From Command Line	31-57
Limitations	31-59
Compile and Debug Generated C Code with Microsoft Visual Studio	31-61
Executable Program Generation	31-62

32 | Host/Target Communication in Simulink Coder

External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling	32-2
Communication Mechanisms for External Mode	32-3
Simulink Feature Support for External Mode	32-4
External Mode Simulations by Using the Run on Custom Hardware App	32-5
External Mode Control Panel	32-6
Security for External Mode Simulations	32-6
External Mode Simulation by Using XCP Communication	32-8
Run XCP External Mode Simulation on Development Computer ...	32-8
Triggered Signal Monitoring for XCP External Mode Simulations	32-13
Run XCP External Mode Simulation From Command Line	32-16
Memory Allocation for Communication Buffers During XCP External Mode Simulation	32-20
Target Application Arguments	32-20
XCP External Mode Limitations	32-21
Graphical Controls for XCP External Mode Simulations	32-27
Equivalent Controls on Hardware Tab and External Mode Control Panel	32-27
XCP External Mode Control Panel	32-28
XCP External Signal & Triggering Dialog Box	32-29
Set Up External Mode Connectivity Between Simulink and Target Hardware	32-35
Customise Connectivity for XCP External Mode Simulations	32-36
Customize Connectivity for TCP/IP or Serial External Mode Simulations	32-40
Execution Tool Template	32-41
Register Custom Launcher	32-43
Customize XCP Server Software	32-45
External Mode Abstraction Layer	32-46
XCP Server Protocol Layer	32-48
XCP Server Transport Layer	32-49
XCP Platform Abstraction Layer	32-49

External Mode Simulation with TCP/IP or Serial Communication	
.....	32-55
Create and Configure Model	32-55
Build Target Executable	32-57
Run Target Application	32-58
Tune Parameters	32-59
Stop Target Application	32-61
Control Memory Allocation for Communication Buffers in Target ..	32-61
TCP/IP or Serial External Mode Control Panel	32-61
Configure Host Monitoring of Target Application Signal Data	32-64
Configure Host Archiving of Target Application Signal Data	32-70
Summary of Graphical Controls for TCP/IP or Serial External Mode Simulation	32-72
Blocks and Subsystems Compatible with External Mode	32-73
External Mode Mechanism for Downloading Tunable Parameters	32-75
Choose Communication Protocol for Client and Server	32-78
Use External Mode Programmatically	32-84
Animate Stateflow Charts in External Mode	32-88
TCP/IP and Serial External Mode Limitations	32-90
Create a Transport Layer for TCP/IP or Serial External Mode Communication	32-94
Design of External Mode	32-94
External Mode Communications Overview	32-96
External Mode Source Files	32-97
Implement a Custom Transport Layer	32-100

Logging in Simulink Coder

33

Log Program Execution Results	33-2
Log Data for Analysis	33-2
Configure State, Time, and Output Logging	33-6
Log Data with Scope and To Workspace Blocks	33-8
Log Data with To File Blocks	33-8
Data Logging Differences Between Single- and Multitasking	33-8

Data Interchange Using the C API in Simulink Coder

34

Exchange Data Between Generated and External Code Using C API	34-2
Generated C API Files	34-2
Generate C API Files	34-4
Description of C API Files	34-5
Generate C API Data Definition File for Exchanging Data with a Target System	34-16
C API Limitations	34-17

Use C API to Access Model Signals and States	34-19
Use C API to Access Model Parameters	34-24

ASAP2 Data Measurement and Calibration in Simulink Coder

35

Export ASAP2 File for Data Measurement and Calibration	35-2
Generation of ASAP2 File Using Generate Calibration Files Tool ...	35-2
Generation of ASAP2 File Using ASAP2Interface Parameter	35-4
Calibrate Generated Code and Monitor Signals	35-11

Generate ASAP2 and CDF Calibration Files

36

Generate ASAP2 and CDF Calibration Files	36-2
Open Generate Calibration Files Tool	36-2
ASAP2 File Generation	36-3
CDF File Generation	36-4
Create a Host-Based ASAM-ASAP2 Data Definition File for Data Measurement and Calibration	36-4
Create a Host-Based ASAM-ASAP2 Data Definition File for Data Measurement and Calibration	36-6

Direct Memory Access to Generated Code for Simulink Coder

37

Access Memory in Generated Code Using Global Data Map	37-2
--	-------------

Customize Generated ASAP2 File

38

Customize Generated ASAP2 File	38-2
Customize Computation Method Names	38-2
Export Characteristics and Measurement Objects in Groups	38-3
Generate Record Layouts	38-4
Exclude Structures and 64-Bit Integers	38-4
Generate ASAP2 Containing Default Event List	38-5

Specify ECU Address Extension	38-6
Merge ASAP2 Files	38-6
Create and Add Custom Data Elements	38-8
Customize an ASAP2 File	38-9
Customize Using Base Files	38-15
Customize ASAP2 Fields by Using Base Class	38-15
Customize ASAP2 Sections by Deriving from Base Class	38-15
Customize an ASAP2 File	38-17

39 | **Configure Model Data Elements for ASAP2 File Generation**

Configure Model Data Elements for ASAP2 File Generation	39-2
View Calibration Attributes	39-2
Configure Inport for ASAP2 File Generation	39-2
Configure Parameter for ASAP2 File Generation	39-4

Performance

40 | **Optimizations for Generated Code in Simulink Coder**

Increase Code Generation Speed	40-3
Build a Model in Increments	40-3
Build Large Model Reference Hierarchies in Parallel	40-3
Minimize Memory Requirements During Code Generation	40-4
Generate Only Code	40-4
Suppress Creation of a Code Generation Report	40-4
Control Compiler Optimizations	40-6
Design Techniques to Optimize Models for Efficient Code Generation	40-7
Use the Model Advisor to Optimize a Model	40-7
Design Techniques to Optimize a Model for Code Generation	40-7
Design Techniques to Optimize Stateflow Objects for Code Generation	40-8
Control Memory Allocation for Time Counters	40-10
Execution Profiling for Generated Code	40-11

Optimize Generated Code by Combining Multiple for Constructs	40-13
.....	
Subnormal Number Execution Speed	40-16
Simulation Time with and Without Subnormal Numbers	40-17
Flush Subnormal Numbers to Zero	40-18
Remove Code for Out-of-Range Floating Point to Integer Conversions	40-20
.....	
Remove Code That Maps NaN to Integer Zero	40-23
Disable Nonfinite Checks or Inlining for Math Functions	40-26
Fold Expressions	40-31
Minimize Computations and Storage for Intermediate Results at Block Outputs	40-35
Expression Folding	40-35
Example Model	40-35
Generate Code	40-35
Enable Optimization	40-36
Generate Code with Optimization	40-36
Inline Invariant Signals	40-38
Optimize Generated Code Using Inline Invariant Signals	40-38
Inline Numeric Values of Block Parameters	40-41
Configure Loop Unrolling Threshold	40-46
Use memcpy Function to Optimize Generated Code for Vector Assignments	40-48
Example Model	40-48
Generate Code	40-49
Generate Code with Optimization	40-50
Generate Target Optimizations Within Algorithm Code	40-51
Remove Code for Blocks That Have No Effect on Computational Results	40-52
Eliminate Dead Code Paths in Generated Code	40-56
Floating-Point Multiplication to Handle a Net Slope Correction	40-59
Use Conditional Input Branch Execution	40-62
Optimize Generated Code for Complex Signals	40-66
Speed Up Linear Algebra in Code Generated from a MATLAB Function Block	40-68
Specify LAPACK Library	40-68
Write LAPACK Callback Class	40-68

Generate LAPACK Calls by Specifying a LAPACK Callback Class . . .	40-69
Locate LAPACK Library in Execution Environment	40-70
Speed Up Matrix Operations in Code Generated from a MATLAB	
Function Block	40-71
Specify BLAS Library	40-71
Write BLAS Callback Class	40-71
Generate BLAS Calls by Specifying a BLAS Callback Class	40-73
Locate BLAS Library in Execution Environment	40-73
Usage Notes and Limitations for OpenBLAS Library	40-74
Speed Up Fast Fourier Transforms in Code Generated from a	
MATLAB Function Block	40-75
Use Built-in FFTW Library Callback for FFTW Library Usage	40-75
Use Custom FFT Library Callback for FFTW Library Usage	40-75
Synchronize Multithreaded FFTW Planning in Code Generated from	
a MATLAB Function Block	40-78
Prerequisites	40-78
Create a Model with a MATLAB Function Block That Calls an FFT	
Function	40-78
Write Supporting C Code	40-79
Create an FFT Library Callback Class	40-79
Configure Code Generation Parameters and Build the Model	40-80
Control Memory Allocation for Variable-Size Arrays in a MATLAB	
Function Block	40-82
Provide Upper Bounds for Variable-Size Arrays	40-82
Disable Dynamic Memory Allocation for MATLAB Function Blocks	
.	40-82
Modify the Dynamic Memory Allocation Threshold	40-82
Optimize Memory Usage for Time Counters	40-84
Optimize Generated Code Using Boolean Data for Logical Signals	
.	40-89
Reduce Memory Usage for Boolean and State Configuration	
Variables	40-92
Customize Stack Space Allocation	40-93
Optimize Generated Code Using memset Function	40-95
Vector Operation Optimization	40-99
Enable and Reuse Local Block Outputs in Generated Code	40-102
Example Model	40-102
Generate Code Without Optimization	40-102
Enable Local Block Outputs and Generate Code	40-103
Reuse Local Block Outputs and Generate Code	40-104
Generate SIMD Code from Simulink Blocks	40-105
Blocks That Support SIMD Code Generation	40-105
Generate SIMD Code Compared to Plain C Code	40-107

Limitations	40-109
Optimize Code for Reduction Operations by Using SIMD	40-110

High-throughput Applications in Simulink Coder

41

Code Generation for Deep Learning Simulink Model That Performs Lane and Vehicle Detection	41-2
Generate Generic C/C++ for Sequence-to-Sequence Deep Learning Simulink Models	41-10
Generate Code for Deep Learning Networks Using MATLAB Function Block	41-22
Classify Images by Using GoogLeNet	41-22
Create GoogLeNet Model	41-23
Simulate Model	41-25
Generate Code	41-26
Generate Code for Blocks from Deep Neural Networks Library ..	41-27
Classify Images by Using GoogLeNet	41-27
Create GoogLeNet Model	41-28
Simulate Model	41-30
Generate Code	41-31
Workflow for Deep Learning C/C++ Code Generation for Simulink Models	41-32

Verification

Simulation and Code Comparison in Simulink Coder

42

Simulation and Code Comparison	42-2
Configure Signal Data for Logging	42-2
Log Simulation Data	42-3
Run Executable and Load Data	42-4
Visualize and Compare Results	42-5
Compare States for Simulation and Code Generation	42-7

43

Numerical Consistency of Model and Generated Code Simulation	
Results	43-2
Numerical Consistency	43-2
Numerical Consistency in Complex Systems	43-2
Reasons for Block-Level Numerical Differences	43-4

Customization

44

Control Build Process Compiling and Linking	44-2
Control Library Location and Naming During Build	44-3
Library Control Parameters	44-3
Specify the Location of Precompiled Libraries	44-4
Location of Model Reference Libraries	44-5
Control the Suffix Applied to Library File Names	44-5
Customize Post-Code-Generation Build Processing	44-7
Workflow for Setting Up Customizations	44-7
Build Information Object	44-7
Program a Post Code Generation Command	44-8
Define a Post Code Generation Command	44-9
Customize Build Process with PostCodeGenCommand and Relocate Generated Code to an External Environment	44-9
Suppress Makefile Generation	44-11
Configure Generated Code with TLC	44-12
Assigning Target Language Compiler Variables	44-12
Set Target Language Compiler Options	44-14
Use makecfg to Customize Generated Makefiles for S-Functions	44-15
Use rtwmakecfg.m API to Customize Generated Makefiles	44-17
About the rtwmakecfg Function	44-17
Create the rtwmakecfg Function	44-17
Modify the Template Makefile for rtwmakecfg	44-19
Register Custom Toolchain and Build Executable	44-21
Customize Build Process with STF_make_rtw_hook File	44-25
STF_make_rtw_hook File	44-25

Conventions for Using the STF_make_rtw_hook File	44-25
STF_make_rtw_hook.m Function Prototype and Arguments	44-25
Applications for STF_make_rtw_hook.m	44-27
Control Code Regeneration Using STF_make_rtw_hook.m	44-27
Use STF_make_rtw_hook.m for Your Build Procedure	44-28
Hook Method after_tlc	44-28
Customize Build Process with sl_customization.m	44-29
The sl_customization.m File	44-29
Register Build Process Hook Functions Using sl_customization.m	44-30
Variables Available for sl_customization.m Hook Functions	44-31
Example of Build Process Customization with sl_customization.m	44-31
Replace STF_rtw_info_hook Supplied Target Data	44-33

Run-Time Data Interface Extensions in Simulink Coder

45

Customize Generated ASAP2 File	45-2
About ASAP2 File Customization	45-2
ASAP2 File Structure on the MATLAB Path	45-2
Customize the Contents of the ASAP2 File	45-2
ASAP2 Templates	45-3
Customize Computation Method Names	45-5
Suppress Computation Methods for FIX_AXIS	45-6

Custom Target Development in Simulink Coder

46

Custom Targets	46-2
Sample Custom Targets	46-4
Target Development Folders, Files, and Builds	46-6
Folder and File Naming Conventions	46-6
Components of a Custom Target	46-6
Key Folders Under Target Root (mytarget)	46-10
Key Files in Target Folder (mytarget/mytarget)	46-12
Additional Files for Externally Developed Targets	46-14
Target Development and the Build Process	46-14
Customize System Target Files	46-19
Control Code Generation With the System Target File	46-19
System Target File Naming and Location Conventions	46-19
System Target File Structure	46-19
Define and Display Custom Target Options	46-25
Tips and Techniques for Customizing Your System Target File ...	46-30

Create a Custom Target Configuration	46-33
Customize Template Makefiles	46-44
Template Makefiles and Tokens	46-44
Invoke the make Utility	46-49
Structure of the Template Makefile	46-50
Customize and Create Template Makefiles	46-52
Custom Target Optional Features	46-57
Support Toolchain Approach with Custom Target	46-59
Support Model Referencing	46-61
Requirements for Model Referencing with a Custom System Target File	46-61
Declaring Model Referencing Compliance	46-61
Providing Model Referencing Support in the TMF	46-62
Controlling Configuration Option Value Agreement	46-63
Preventing Resource Conflicts (Optional)	46-64
Support Compiler Optimization Level Control	46-65
About Compiler Optimization Level Control and Custom Targets ..	46-65
Declaring Compiler Optimization Level Control Compliance	46-65
Providing Compiler Optimization Level Control Support in the Target Makefile	46-66
Support C Function Prototype Control	46-67
About C Function Prototype Control and Custom Targets	46-67
Declaring C Function Prototype Control Compliance	46-67
Providing C Function Prototype Control Support in the Custom Static Main Program	46-68
Support C++ Class Interface Control	46-69
About C++ Class Interface Control and Custom Targets	46-69
Declaring C++ Class Interface Control Compliance	46-69
Providing C++ Class Interface Control Support in the Custom Static Main Program	46-69
Support Concurrent Execution of Multiple Tasks	46-71
Device Drivers	46-72

Model Architecture and Design

Design Preparation

- “Divide Software Problem” on page 1-2
- “Map System Architecture to Simulink Modeling Environment” on page 1-3

Divide Software Problem

A step in project assessment process is to break down your software problem into smaller parts. You can represent each part as a separate application or component.

Smaller components:

- Facilitate separation of concerns.
- Are easier to work with.
- Enable parallel development.
- Promote reuse.

When assessing a component for reuse, consider whether the component is:

- Reusable as is.
- Reusable with slight variations.
- Expected to be volatile.

If you have access to System Composer™, you can use that software to graphically represent software components as you decompose the software problem and design a software architecture. For more information, see “Author Architecture Models” (System Composer).

As part of the decomposition process, identify data and execution dependencies, such as ports, function-call interfaces, and parameters that require tuning and signal data that must be measured during program execution.

See Also

Related Examples

- “Map System Architecture to Simulink Modeling Environment” on page 1-3

Map System Architecture to Simulink Modeling Environment

When designing models for rapid-prototyping deployment, think about these design factors.

“Modeling Algorithms” (Embedded Coder)	Given initial state and input, a set of tasks or instructions that efficiently produce a correct result that you want.
“Modeling Interfaces” (Embedded Coder)	Mechanisms that enable algorithm components to communicate and exchange information across component boundaries.
“Modeling Systems” (Embedded Coder)	Collection of algorithm components that achieve a higher-level, domain-specific goal or result. Components often share resources.
“Modeling Target Platform Environments” (Embedded Coder)	Framework that handles scheduling of system algorithm resources and execution.

Consider the following questions concerning modeling capabilities. Use the information listed with the questions as a guide as you design models. Designing a model with a specific run-time execution environment in mind can help you avoid rework and future conversion and maintenance costs.

Modeling Algorithms

Architecture Considerations	Modeling Considerations	Related Information
What is the system domain?	Product prerequisites (based on domains of components)	<ul style="list-style-type: none"> • “Use Products and Blocks Supported for Code Generation” on page 3-2 • “Simulink Control Design” • “Signal Processing” • “Signal Generation, Manipulation, and Analysis” (DSP System Toolbox)
Does the system involve physical domains, such as mechanical, electrical, or hydraulic domains?	Physical systems	<ul style="list-style-type: none"> • “Physical Modeling” • “Basic Principles of Modeling Physical Networks” (Simscape) • “Essential Physical Modeling Techniques” (Simscape)
What aspects of your algorithm can you represent with blocks provided by MathWorks® products? What blocks do you need to create?	Block usage, creation, and customization	“Use Products and Blocks Supported for Code Generation” on page 3-2
Does the architecture include state machine components?	Event-driven system	“Model Finite State Machines by Using Stateflow Charts” (Stateflow)

Modeling Interfaces

Architecture Considerations	Modeling Considerations	Related Information
<ul style="list-style-type: none"> • What data must you represent in the generated code? • How do you need to represent input and output—data type, dimension, complexity? • How will the data change? 	Data representation	<ul style="list-style-type: none"> • “Define Interfaces of Model Components” • “Data Representation in Generated Code”
Where and how is data pulled into the system and pulled within the system?	Input	“Comparison of Signal Loading Techniques”
<ul style="list-style-type: none"> • Where and how is data pushed within the system and out of the system? • What external triggers are needed? 	Output	<ul style="list-style-type: none"> • Simulation Data Inspector • “Organize Parameter Data into a Structure by Using Struct Storage Class” (Embedded Coder)
<ul style="list-style-type: none"> • What functions do you need to define for each component? • What is the prototype for each entry-point function? 	Functions and function calls	“Configure Generated C Function Interface for Model Entry-Point Functions” on page 16-79
Do you need to export functions that are invoked by controlling logic that is outside the model?	Function export	<ul style="list-style-type: none"> • “Export-Function Models Overview” • “Generate Component Source Code for Export to External Code Base” on page 30-40
Does the system monitor signals or log data (for example, for calibration)?	C API and ASAP2 data exchange interfaces	<ul style="list-style-type: none"> • “Exchange Data Between Generated and External Code Using C API” on page 34-2 • “Export ASAP2 File for Data Measurement and Calibration” on page 35-2
Do you need to replace code generated for functions or operators, for example, to optimize the code for specific hardware?	Code replacement	<ul style="list-style-type: none"> • “What Is Code Replacement?” on page 26-2 • “What Is Code Replacement Customization?” (Embedded Coder)
Is there a requirement for elaboration and future considerations?	Elaboration and future considerations	“Define Interfaces of Model Components”

Modeling Systems

Architecture Considerations	Modeling Considerations	Related Information
<ul style="list-style-type: none"> • What is the scope of the system? Controller? External environment or plant? Test harness? • How is the system partitioned into algorithm components (chunks of logic)? • Which components can you represent in Simulink? • Can you design components for reuse? What is the motivation for reuse (for example, division of labor or plug-n-play)? 	Componentization	<ul style="list-style-type: none"> • “Define Interfaces of Model Components” • “Component-Based Modeling Guidelines” • “Compare Capabilities of Model Components” • “Create Custom Library” • “Extend Modeling Functionality with Custom Blocks” • “Generate Subsystem Code as Separate Function and Files” on page 7-7 • “Generate Code for Model Reference Hierarchy” on page 8-2 • “Code Generation of Stateflow Blocks” on page 13-2
<ul style="list-style-type: none"> • Do aspects of the system require unit testing? • Is a team of people collaborating on the project? • Do you need to protect intellectual property? 	Model referencing	<ul style="list-style-type: none"> • “Model Reference Basics” • “Compare Capabilities of Model Components” • “Generate Code for Model Reference Hierarchy” on page 8-2 • “Generate Code from Atomic Subcharts” on page 13-5
Are you modeling a client-server architecture?	Simulink Function and Caller blocks	<ul style="list-style-type: none"> • “Client-Server Architecture” • “Simulink Functions Overview”
Is relevant legacy or custom code available?	External code integration	“Choose an External Code Integration Workflow” (Embedded Coder)
Can you apply a reference architecture or reference components?	Model and project templates	<ul style="list-style-type: none"> • “Create Template from Model” • “Create a New Project Using Templates”

Architecture Considerations	Modeling Considerations	Related Information
Can you reuse functions?	Function reuse	<ul style="list-style-type: none"> • “Generate Reusable Code for Subsystems Shared Across Models” on page 10-39 • “Generate Reusable Code from Library Subsystems Shared Across Models” on page 10-36 • “Generate Reentrant Code from Top Models” on page 10-13 • “Generate Reentrant Code from Subsystems” on page 10-28 • “Generate Reusable Code from Stateflow Atomic Subcharts” (Embedded Coder)
<ul style="list-style-type: none"> • Do components need to share access to global data? • Within the system, do state changes occur? In each case, how does the result get communicated? • Are there identifier (naming) issues to consider? 	Shared data	<ul style="list-style-type: none"> • “Local and Global Data Stores” • “Data Structures in the Generated Code” on page 15-31 • “Storage Classes for Parameters and Signals Used in Model Blocks” on page 8-19 • “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49 • “Data Stores in Generated Code” on page 15-94 • “Customize Generated Identifier Naming Rules” (Embedded Coder)

Modeling Run-Time Environments

Architecture Considerations	Modeling Considerations	Related Information
<ul style="list-style-type: none"> • What level of control over run-time interfacing does your application require? • How much of your system can you represent in a model? 	Runtime interfacing	<ul style="list-style-type: none"> • “Execution of Code Generated from a Model” on page 17-18 • See Modeling Interfaces.
Is the system partitioned into concurrent components to maximize parallelism? Which components?	Concurrency	“Multicore Processor Targets”

Architecture Considerations	Modeling Considerations	Related Information
<ul style="list-style-type: none"> • Are components driven by an external clock? • What clock rates do system components use? • Do components use a single rate or multiple rates? 	Clocks and clock rates	“Define Interfaces of Model Components”
<ul style="list-style-type: none"> • Are components in the system driven by clocks? • What clock rates do system components use? • Do components use a single rate or multiple rates? • What are the priorities of system tasks and functions? 	Time-based scheduling	<ul style="list-style-type: none"> • “Absolute and Elapsed Time Computation” on page 4-2 • “Time-Based Scheduling”
<ul style="list-style-type: none"> • Are components in the system driven by events (interrupts)? • What are the priorities of system tasks and functions? 	Event-based scheduling	<ul style="list-style-type: none"> • “Absolute and Elapsed Time Computation” on page 4-2 • “Event-Based Scheduling” • “Model Finite State Machines by Using Stateflow Charts” (Stateflow)
Does the system need to handle initialization, reset, or terminate events?	Initialization, reset, termination	<ul style="list-style-type: none"> • “Using Initialize, Reinitialize, Reset, and Terminate Functions” • “Startup, Reset, and Shutdown Function Interfaces” on page 2-8
<ul style="list-style-type: none"> • Is the system a single-tasking or multitasking system? • Are components required to execute in real time? • What are the execution order dependencies (sequencing) between components? • What are the time constraints for task and function execution? 	Task execution	<ul style="list-style-type: none"> • “Execution of Code Generated from a Model” on page 17-18 • “Modeling for Single-Tasking Execution” on page 5-6 • “Modeling for Multitasking Execution” on page 5-9

Architecture Considerations	Modeling Considerations	Related Information
<ul style="list-style-type: none">• If you know the processing platform, what is it?• Will the system run on a single-core or multicore processor?• Is the system a distributed system?• Is the processing platform hybrid or heterogeneous?• Does the architecture employ symmetric or asymmetric multiprocessing? If asymmetric, how is the platform software partitioned across CPUs?	Processing platforms	"Multicore Processor Targets"

Application Interfaces

Model Code Interfaces

To produce code that can interact between model elements, the code generator must be aware of interface requirements. The first step for doing that is to represent required interfaces in your top model by including specific model elements. For example, you can model an interface for reading data from a model element by using a root-level In Bus Element or Inport block.

This table lists typical code interfaces and ways to represent them in a top model.

Type of Code Interface	Model Element or Pattern
Callable rate-based entry point	Block in a rate-based model that is configured for single-tasking or multitasking execution or that is set up as a partition in the Simulink Schedule Editor
Callable initialize function entry point	Initialize Function block
Callable terminate function entry point	Terminate Function block
Call to target platform receiver service	In Bus Element or Inport block
Call to target platform sender service	Out Bus Element or Outport block
Access target platform nonvolatile memory	Initialize Function and Terminate Function blocks or Simulink Function and Function Caller blocks
Reuse memory between a state variable and root-level output variable	Function loopback pattern that models state variable as a signal or Delay block that models state variable
Message communication	Send block directly connected to a root-level Outport block and Receive block directly connected to a root-level Inport block
Client-server communication	Simulink Function and Function Caller blocks
Tunable parameters and parameter arguments	Parameter objects stored in the model workspace
Data for measurement	Signals (output port of a block), states (states on a block), and Datastore Memory blocks
Custom service interface	Custom blocks

These topics provide more information for the different types of interfaces:

- “Periodic and Aperiodic Function Interfaces” on page 2-4
- “Startup, Reset, and Shutdown Function Interfaces” on page 2-8
- “Data Interfaces” on page 2-17
- “Message Communication Interfaces” on page 2-18
- “Client-Server Communication Interfaces” on page 2-29
- “Device Driver I/O Interfaces” on page 2-30

Modeling guidelines can be helpful depending on your deployment objective and how you configure code interfaces for the code generator. In some cases, you might need to create custom representations that involve creating custom blocks.

For information on modeling guidelines, see “Modeling Guidelines and Block Usage”.

For information on creating custom blocks, see “Code Generation for Custom Blocks”.

Periodic and Aperiodic Function Interfaces

Target environment software initiates execution of algorithm code generated for a top model (application or component) by calling generated execution entry-point functions. The execution entry-point functions that implement the top model behavior can be periodic or aperiodic. The entry-point function code that the code generator produces for a top model depends on the modeling styles and periodic rates represented in the model. You can represent periodic and aperiodic entry-point functions in a model by using a combination of two modeling styles: rate-based and, if you are using Embedded Coder[®], export-function.

This table lists conditions that might prompt you to choose each modeling style.

Rate-Based Modeling Style Conditions	Export-Function Modeling Style Conditions
<ul style="list-style-type: none"> Assumes rate monotonic scheduling. Code generator produces a unique entry-point function for each rate represented in the model. Sample time is design data. It is not an implementation consideration. Can generate reusable code from multiple instances of a model. 	<ul style="list-style-type: none"> You are using Embedded Coder. A function represents the entry-point function for a server. A function exchanges data with multiple other functions. A function represents a periodic entry-point function that has a rate that is not a multiple of the fastest rate used by the top model. A function represents a periodic entry-point function that other events also trigger. Multiple functions represent periodic entry-point functions that are triggered at the same period.

If you want a top model to support multiple instantiation, you cannot model periodic functions as function-call subsystems. Use rate-based modeling.

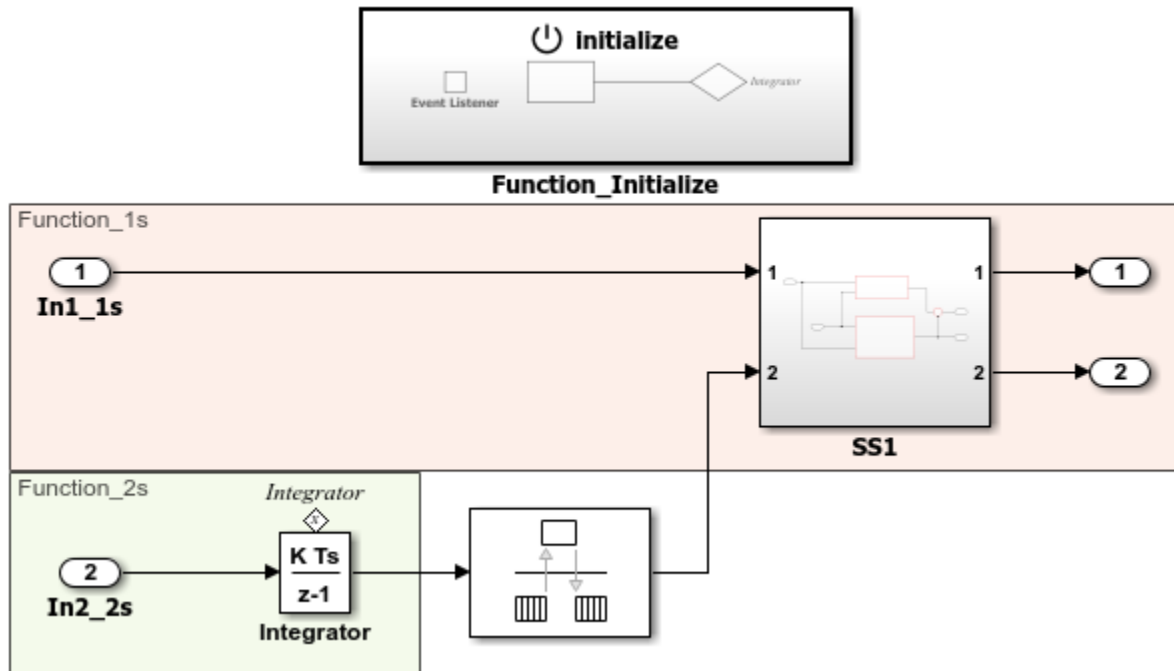
For more information about modeling execution behavior, see “Rate-Based Top Models” on page 2-4 and “Export-Function Top Models” on page 2-5.

For options to configure the function prototype for generated periodic and aperiodic execution entry-point functions, see “Periodic and Aperiodic Function Interface Configuration” on page 2-7.

Rate-Based Top Models

A rate-based top model represents periodic entry-point functions by using unique periodic rates.

This example model shows a rate-based implementation of a component top model. The model implements periodic entry-point functions by using the rates 1 second and 2 seconds. An Initialize Function block initializes the component.



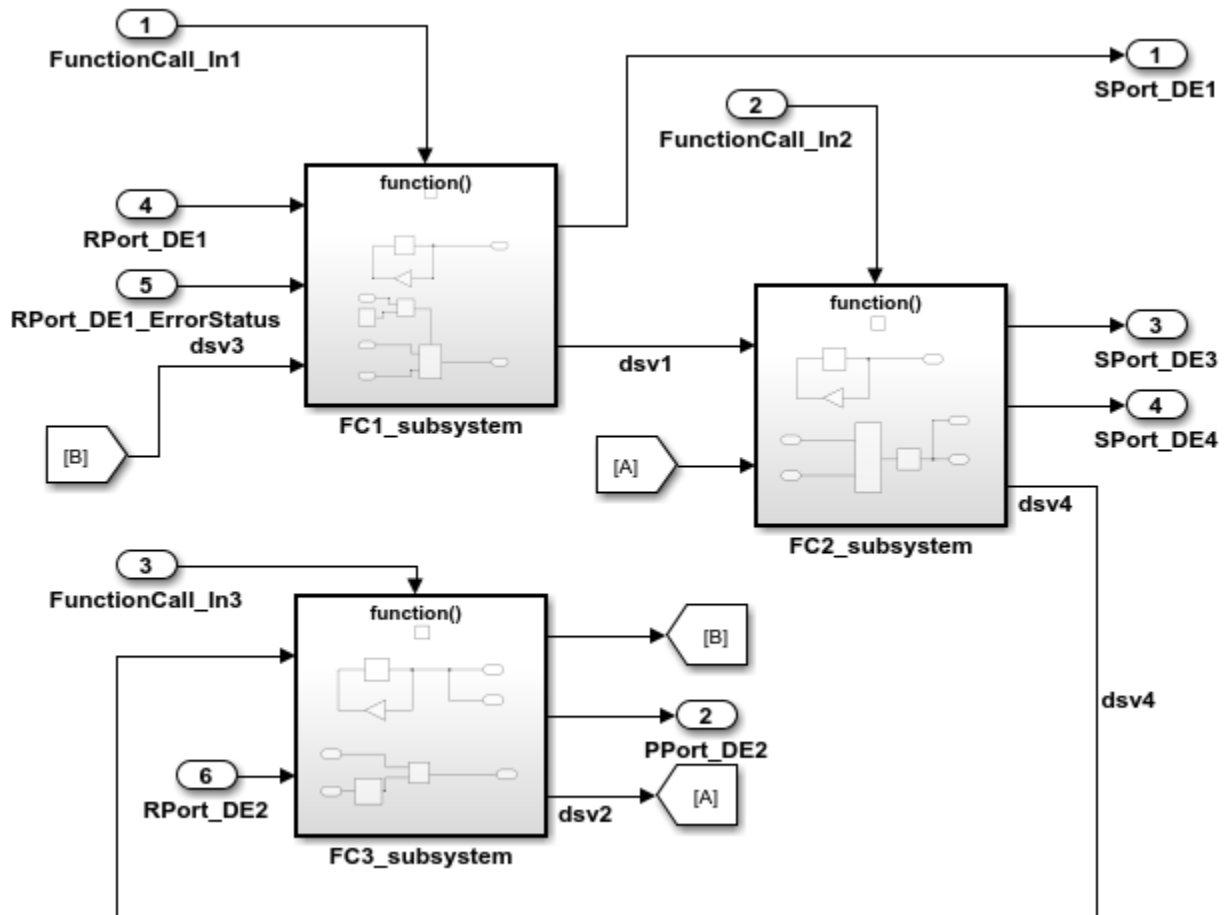
When you generate code for a rate-based top model, the C code contains rate-grouped step entry-point functions, one function for each discrete rate in the model. The target environment function scheduler calls the generated periodic step functions.

You can design a rate-based top model to represent periodic and aperiodic entry-point functions. For example, in the JMAAB-B type beta architecture, an asynchronous trigger entry-point function interacts with periodic rate-based entry-point functions.

Export-Function Top Models

If you are using Embedded Coder, for export-function models, the code generator produces code for independent functions that execute at a periodic rate or aperiodically in response to events. You can integrate the independent function code with other target environment software, including the function scheduler and other platform and custom services.

This example top model shows an export-function implementation of a component. The model uses three Function-Call Subsystem blocks, FC1_subsystem, FC1_subsystem, and FC3_subsystem, which have periodic rates to implement three periodic entry-point functions. Signal lines that connect the Function-Call Subsystem blocks, dsv1, dsv2, dsv3, and dsv4, represent data transfers.



When you generate code for an export-function implementation, the code includes a callable model entry-point function for each function represented in the top model. Unless you define and configure the interface otherwise, the code generator uses the name of the function-call subsystem input port as the name of the entry-point function. You can override the default naming by defining a function customization template in an Embedded Coder Dictionary and configuring the function name in the model code mappings.

You can control how the code generator packages the code for an export-function implementation. For example, by using subsystem block parameter **Function packaging**, you can specify whether the code generator inlines the function code for a subsystem or produces separate function code that passes I/O as global data. When inlined, the code generator uses the name of the function-call subsystem input port to name the entry-point function. When producing separate function code, the code generator names the function based on the settings of other parameters, such as **File name options**.

For more information, see “Generate Component Source Code for Export to External Code Base” (Embedded Coder), “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder), and “Interactively Configure C++ Interface” (Embedded Coder).

Periodic and Aperiodic Function Interface Configuration

The code generator produces several types of entry-point functions for model functions (see “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder)). As an example, for a component model that includes two periodic function-call subsystems and an aperiodic function-call subsystem, the code generator produces three callable entry-point functions for initiating component execution:

- *model_step0* - function for the first periodic rate
- *model_step1* - function for the second periodic rate
- *function* - aperiodic function, where *function* is the name of the Inport block for a function-call subsystem or label of the signal line connecting the Inport and subsystem

If you are using Embedded Coder, you can customize the entry-point function prototypes to align with function calls initiated by the target environment function scheduler by applying one of these approaches:

- For the model, in the Code Mappings editor or by using the code mappings programming interface, configure the function name and arguments for each model execution function individually.
- Define a function customization template for execution functions in the Embedded Coder Dictionary (see “Code Interface Definitions” (Embedded Coder)). Then, for the model, in the Code Mappings editor or by using the code mappings programming interface, configure the code generator to use that template for model execution functions.

For more information, see “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder).

See Also

Related Examples

- “Schedule Components to Avoid Data Dependency Issues”
- “Export-Function Models Overview”
- “Create Rate-Based Model”
- “Create an Export-Function Model”
- “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder)
- “Configure C++ Class Interface for Rate-Based Models” (Embedded Coder)
- “Configure C++ Class Interface for Export-Function Models” (Embedded Coder)
- “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder)

Startup, Reset, and Shutdown Function Interfaces

Applications sometimes require logic that executes during system initialization, reset, and termination sequences. To model startup, reset, and shutdown processing in a model, use the Initialize Function, Reset Function (data interface configurations only), and Terminate Function blocks. You can use the blocks anywhere in a model hierarchy.

The Initialize Function, Reset Function, and Terminate Function blocks can control execution of an application or component in response to initialize, reset, or terminate events. Examples of when to generate code that responds to these events include:

- Starting and stopping an application or component.
- Calculating initial conditions.
- Saving and restoring state from nonvolatile memory.
- Generating reset entry-point functions that respond to external events.

You can place the blocks at different levels of a model hierarchy. Each nonvirtual subsystem can have its own set of initialize, reset, and terminate functions. In a lower-level model, Simulink aggregates the content of the functions with corresponding instances in the parent model.

The Initialize Function and Terminate Function blocks contain an Event Listener block. To specify the event type of the function—*Initialize*, *Reset*, or *Terminate*—use the **Event type** parameter of the Event Listener block. The function block reads or writes the state of conditions for other blocks. By default, the Initialize Function block initializes block state with the State Writer block. The Terminate Function block saves the block state by using the State Reader block. When the function is triggered by an initialize or terminate event, the function writes the value of the state variable or reads the value from the specified block.

Application and component models can use the Initialize Function, Reset Function (data interface configurations only), and Terminate Function blocks to model complex startup, reset, and shutdown sequences. The subsystems work with various modeling styles. Software-in-the-loop (SIL) simulation of initialize, reset, or terminate functions works with export-function modeling only.

The code generator produces initialize and terminate code differently than reset code. For initialize and terminate code, the code generator includes your initialize and terminate code in the default entry-point functions, `model_initialize` and `model_terminate`. The code generator produces reset code only if you model reset behavior.

For example, consider a component model that includes these elements:

- An Initialize Function block
- Two Reset Function blocks that specify event names `tooHighReset` and `tooLowReset`
- A Terminate Function block

For this scenario, the code generator produces four entry-point functions that respond to target platform function calls triggered by events:

- `model_initialize` - initialization function
- `model_tooHighReset` - reset function for `tooHighReset` event
- `model_tooLowReset` - reset function, for `tooLowReset` event
- `model_terminate` - terminate function

If you are using Embedded Coder, you can customize the entry-point function names to align with function calls initiated by the target environment function scheduler by applying one of these approaches:

- For the model, in the Code Mappings editor or by using the code mappings programming interface, configure the function names for model initialize, reset, and terminate functions individually.
- Define function customization templates in the Embedded Coder Dictionary. Then, for the model, in the Code Mappings editor or by using the code mappings programming interface, configure the code generator to use the templates for the model event-triggered functions.

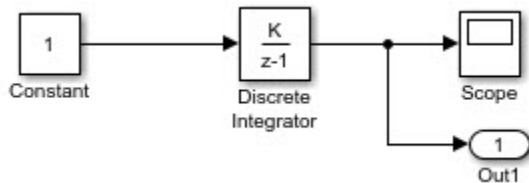
For more information about modeling startup, reset, and shutdown behavior, see “Using Initialize, Reinitialize, Reset, and Terminate Functions”. For guidelines on modeling startup and shutdown behavior for component models, see “cgs1_0404: Model startup and shutdown events by using Initialize Function and Terminate Function blocks for component deployment”.

Generate Code for Initialize and Terminate Events

When you generate code for a component that includes Initialize Function and Terminate Function blocks, the code generator:

- Includes initialize event code with default initialize code in entry-point function `model_initialize`.
- Includes terminate event code with default terminate code in entry-point function `model_terminate`.

Consider the model `rtwdemo_irt_base`.



For this model, the code generator produces initialize and terminate entry-point functions that other code can interface with.

```
void rtwdemo_irt_base_initialize(void)
void rtwdemo_irt_base_terminate(void)
```

This code appears in the generated file `rtwdemo_irt_base.c`. The initialize function, `rtwdemo_irt_base_initialize`:

- Initializes an error status.
- Allocates memory for block I/O and state parameters.
- Sets the output value
- Sets the initial condition for the discrete integrator.

The terminate function, `rtwdemo_irt_base_terminate`, requires no code.

This code assumes that support for nonfinite numbers and MAT-file logging is disabled.

```

void rtwdemo_irt_base_initialize(void)
{
    rtmSetErrorStatus(rtwdemo_irt_base_M, (NULL));

    (void) memset((void *)&rtwdemo_irt_base_DW, 0,
                 sizeof(DW_rtwdemo_irt_base_T));

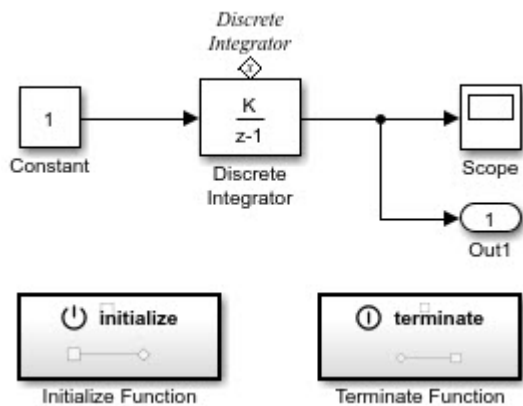
    rtwdemo_irt_base_Y.Out1 = 0.0;

    rtwdemo_irt_base_DW.DiscreteIntegrator_DSTATE = 0.0;
}

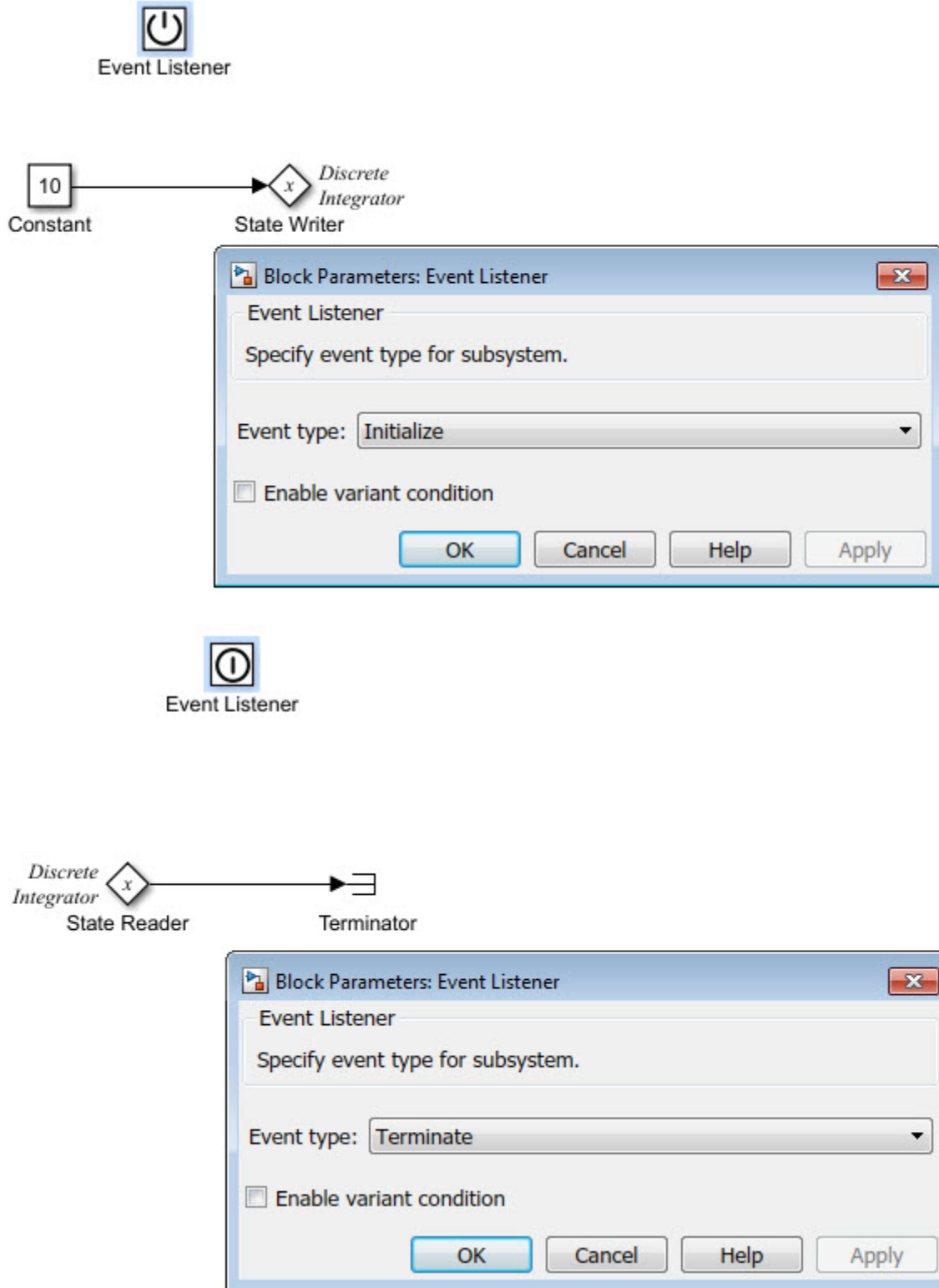
void rtwdemo_irt_base_terminate(void)
{
    /* (no terminate code required) */
}

```

Add Initialize Function and Terminate Function blocks to the model (see `rtwdemo_irt_initterm`). The Initialize Function block uses the State Writer block to set the initial condition of a Discrete Integrator block. The Terminate Function block includes a State Reader block, which reads the state of the Discrete Integrator block.



Parameter **Event type** for the Event Listener block for the initialize and terminate functions is set to Initialize and Terminate, respectively. The initialize function uses the State Writer block to initialize the input value for the Discrete Integrator block to 10. The terminate function uses the State Reader block to read the state of the Discrete Integrator block.



The code generator includes the event code that it produces for the Initialize Function and Terminate Function blocks with standard initialize and terminate code in entry-point functions

`rtwdemo_irt_initterm_initialize` and `rtwdemo_irt_initterm_terminate`. This code assumes that support for nonfinite numbers and MAT-file logging is disabled.

```
void rtwdemo_irt_initterm_initialize(void)
{
    rtmSetErrorStatus(rtwdemo_irt_initterm_M, (NULL));

    (void) memset((void *)&rtwdemo_irt__initterm_DW, 0,
                 sizeof(DW_rtwdemo_irt__initterm_T));

    rtwdemo_irt_initterm_Y.Out1 = 0.0;

    rtwdemo_irt_initterm_DW.DiscreteIntegrator_DSTATE = 10.0;
}

void rtwdemo_irt__initterm_terminate(void)
{
    /* (no terminate code required) */
}
```

For guidelines on modeling startup and shutdown behavior for component models, see “[cgsl_0404: Model startup and shutdown events by using Initialize Function and Terminate Function blocks for component deployment](#)”.

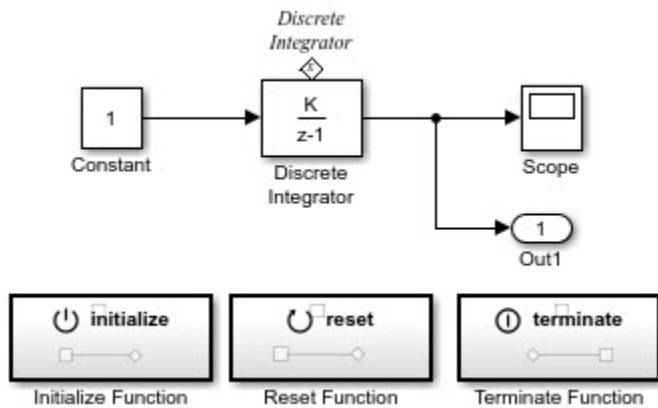
Generate Code for Reset Events

For application models and component models that are configured with a data interface, you can generate code that responds to a reset event by including an Initialize Function or Terminate Function block. Configure the block for a reset by setting the **Event type** parameter of its Event Listener block to **Reset**. Also set the **Event name** parameter. The default name is **reset**.

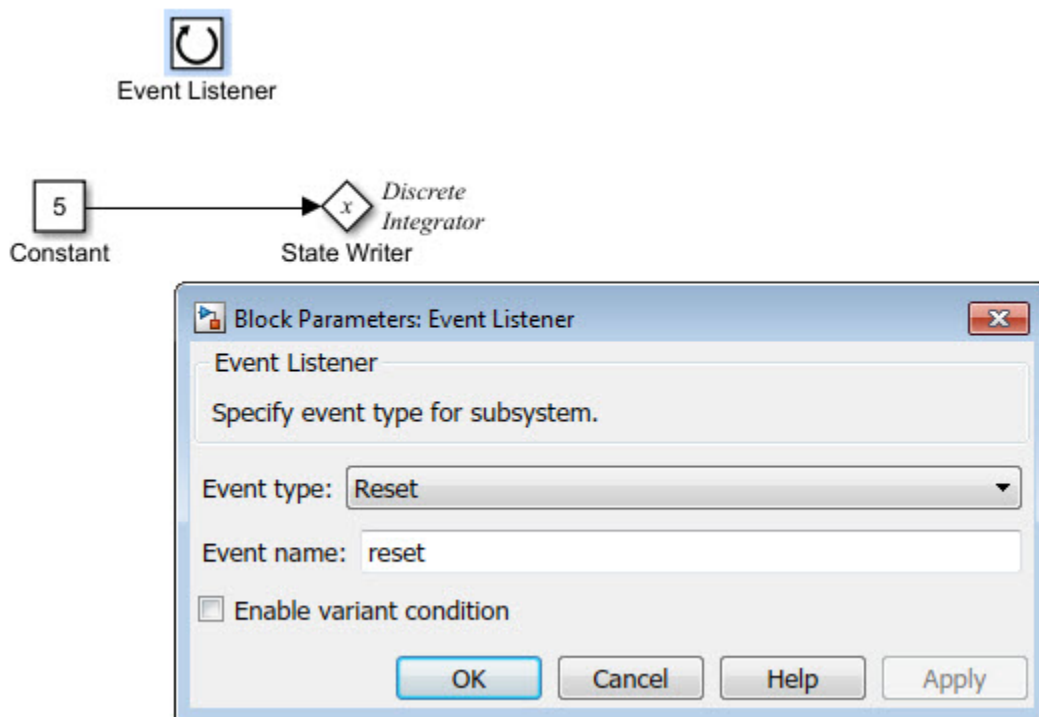
The code generator produces a reset entry-point function *only* if you model reset behavior. If a component contains multiple reset specifications, the code that the code generator produces depends on whether reset functions share an event name. For a given component hierarchy:

- For reset functions with unique event names, the code generator produces a separate entry-point function for each named event. The name of each function is the name of the corresponding event.
- For reset functions that share an event name, the code generator aggregates the reset code into one entry-point function. The code for the reset functions appears in order, starting with the lowest level (innermost) of the component hierarchy and ending with the root (outermost). The name of the function is `model_reset`. For more information, see “Event Names and Code Aggregation” on page 2-14.

Consider the model `rtwdemo_irt_reset`, which includes a Reset Function block derived from an Initialize Function block.



The **Event type** and **Event name** parameters of the Event Listener block are set to **Reset** and **reset**, respectively. The function uses the State Writer block to reset the input value for the Discrete Integrator block to 5.



The code generator produces reset function `rtwdemo_irt_reset_reset`.

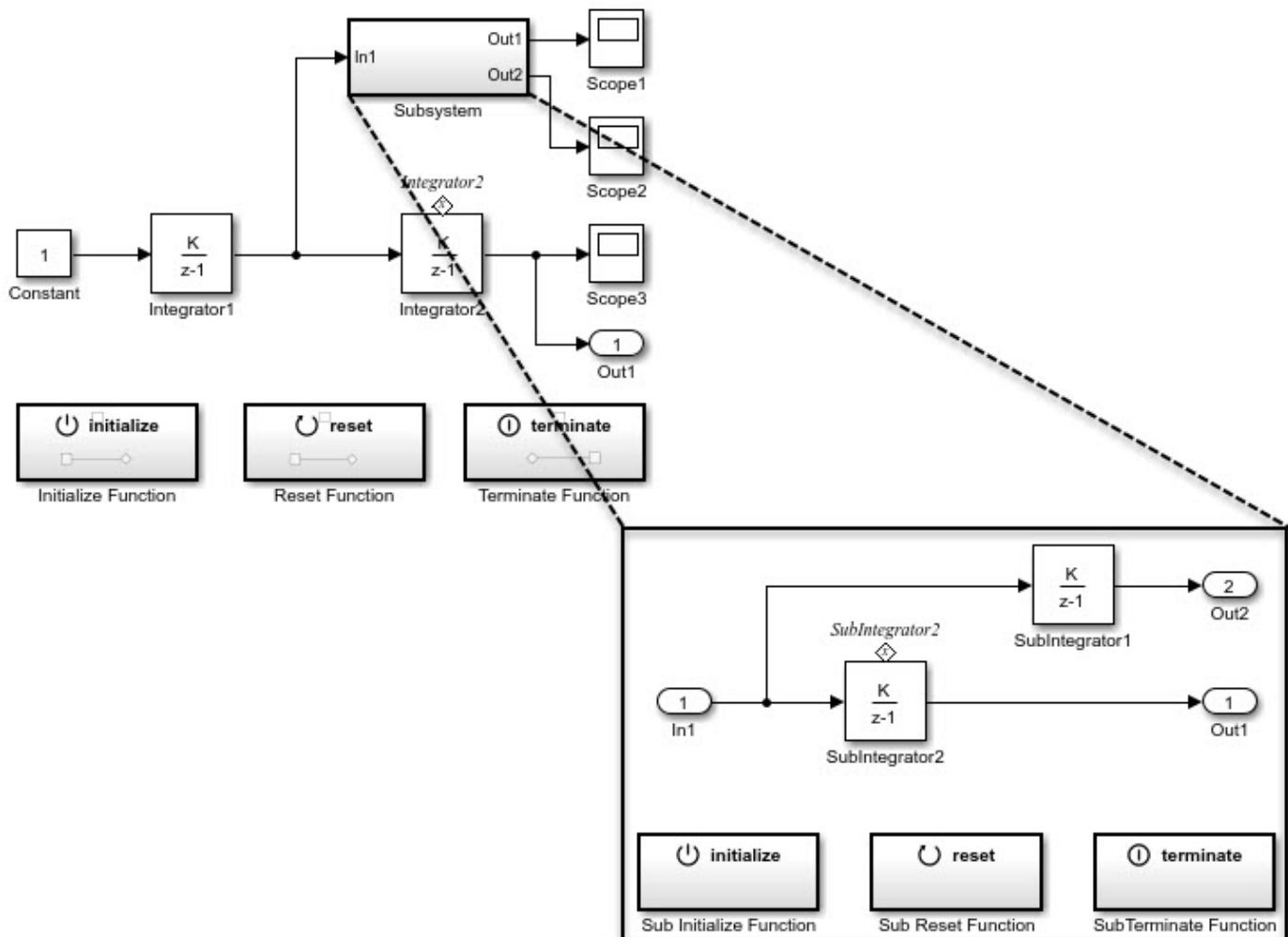
```
void rtwdemo_irt_reset_reset(void)
{
    rtwdemo_irt_reset_DW.DiscreteIntegrator_DSTATE = 5.0;
}
```

Event Names and Code Aggregation

Use the Initialize Function and Terminate Function blocks to define multiple initialize, reset, and terminate functions for a component hierarchy. Define only one initialize function and one terminate function per hierarchy level. You can define multiple reset functions for a hierarchy level. The event names that you configure for the functions at a given level must be unique.

When producing code, the code generator aggregates code for functions that have a given event name across the entire component hierarchy into one entry-point function. The code for reset functions appears in order, starting with the lowest level (innermost) of the component hierarchy and ending with the root (outermost). The code generator uses the event name to name the function.

For example, the model `rtwdemo_irt_shared` includes a subsystem that replicates the initialize, reset, and terminate functions that are in the parent model.



Although the model includes multiple copies of the initialize, reset, and terminate functions, the code generator produces one entry-point function for reset (`rtwdemo_irt_shared_reset`), one for initialize (`rtwdemo_irt_shared_initialize`), and one for terminate (`rtwdemo_irt_shared_terminate`). Within each entry-point function, after listing code for blocks

configured with an initial condition (*model_P.block_IC*), the code generator orders code for components, starting with the lowest level of the hierarchy and ending with the root.

```

.
.
.
void rtwdemo_irt_shared_reset(void)
{
    rtwdemo_irt_shared_DW.SubIntegrator2_DSTATE = 5.0;

    rtwdemo_irt_shared_DW.Integrator2_DSTATE = 5.0;
}
.
.
.
void rtwdemo_irt_shared_initialize(void)
{
    rtmSetErrorStatus(rtwdemo_irt_shared_M, (NULL));

    (void) memset(((void *)&rtwdemo_irt_shared_DW), 0,
        sizeof(DW_rtwdemo_irt_shared_T));

    rtwdemo_irt_shared_Y.Out1 = 0.0;

    rtwdemo_irt_shared_DW.Integrator1_DSTATE = 0.0;

    rtwdemo_irt_shared_DW.SubIntegrator2_DSTATE = 2.0;

    rtwdemo_irt_shared_DW.Integrator2_DSTATE = 10.0;
}
.
.
.
void rtwdemo_irt_shared_terminate(void)
{
    /* (no terminate code required) */
}

```

If you rename the event configured for the subsystem reset function to `reset_02`, the code generator produces two reset entry-point functions, `rtwdemo_irt_shared_reset` and `rtwdemo_irt_shared_reset_02`.

```

void rtwdemo_irt_shared_reset(void)
{
    rtwdemo_irt_shared_DW.SubIntegrator2_DSTATE = 5.0;
}

void rtwdemo_irt_shared_reset_02(void)
{
    rtwdemo_irt_shared_DW.Integrator2_DSTATE = 5.0;
}

```

Limitation

- You cannot generate code from a harness model—a root model that contains a Model block, which exposes initialize, reset, or terminate function ports.

- You cannot model reset processing in a component model that is configured with a service code interface.

See Also

Initialize Function | Reset Function | Terminate Function

Related Examples

- “Using Initialize, Reinitialize, Reset, and Terminate Functions”
- “Initialization of Signal, State, and Parameter Data in the Generated Code” on page 15-188

Data Interfaces

“Model Code Interfaces” on page 2-2 lists modeling elements that you can use to represent interfaces for interacting with target environment resources. To configure the modeling elements for code generation, in the **Code Mappings editor** or by using the code mappings programming interface, you map the interface elements represented in the model to data code interfaces.

C Data Interfaces

When using Simulink Coder to generate code, you map model data interface elements to storage classes. A storage class specifies the appearance and placement of data elements in the generated code.

To represent data interface elements in a model and configure them for code generation:

- 1 In the model, include data interface elements as recommended in “Model Code Interfaces” on page 2-2.
- 2 Open the **Simulink Coder** app.
- 3 If you want to set up default mappings for categories of model elements, select **Code Interface > Default Code Mappings**. In the **Code Mappings editor**, on the **Data Defaults** and **Function Defaults** tabs, configure default mappings.
- 4 Select **Code Interface > Individual Element Code Mappings**.
- 5 In the Code Mappings editor, on each element category tab, inspect the mappings and property settings for individual interface elements. If necessary, adjust the settings.
- 6 Save your interface configuration changes by saving the model.

If you prefer, you can map data elements to storage classes by using the code mappings programming interface (see `coder.mapping.api.CodeMapping`).

For more information, see **Code Mappings editor** and “Data and Function Interfaces”.

Message Communication Interfaces

Within a platform environment, code generated from a Simulink model can communicate with platform software or with other code generated from Simulink models by using message-based communication. Generate C++ component code for message-based communication with platform software. Generate C++ or C component code for message-based communication with other models.

Generate C++ Messages to Communicate Data Between Simulink Components

To generate C++ code that supports message-based communication between model components in the Simulink environment, use the Simulink Messages & Events Library blocks Send and Receive. To customize the communication, use the Queue block (from the same library) to set parameters for capacity, sorting policy (LIFO, FIFO, and priority), and overwriting policy (behavior when the queue exceeds capacity). You can generate C++ code for GRT-based system target files by using Simulink Coder or for ERT-based system target files by using Embedded Coder.

Messages are an effective communication technique for distributed and complex systems that you can model within Simulink.

How to Prepare Models in Simulink for Message-Based Communication

In Simulink, model message-based communication between model components:

- 1 Create a model that contains a Send block (referenced model).
- 2 Create a model that contains a Receive block (referenced model).
- 3 Create a model that has two Model blocks (top model).
 - Set the first Model block to the model that contains the Send block (model from step 1).
 - Set the second Model block to the model that contains the Receive block (model from step 2).
- 4 When you run the model, a queue generates in the top model above the message line. A queue, explicit or implicit, controls the message communication. You can use the generated queue or you can add a Queue block in the top model to explicitly specify communication parameters.

How Model Components Pass Messages and How the Code Implements the Behavior

Referenced models pass messages:

- 1 In the model that contains the Send block, the Send block converts signals into messages.
- 2 The top model that contains the queue manages messages according to parameters that define capacity, order of delivery, and other quality of service (QoS) metrics.
- 3 In the model that contains the Receive block, the Receive block converts messages back to signals.

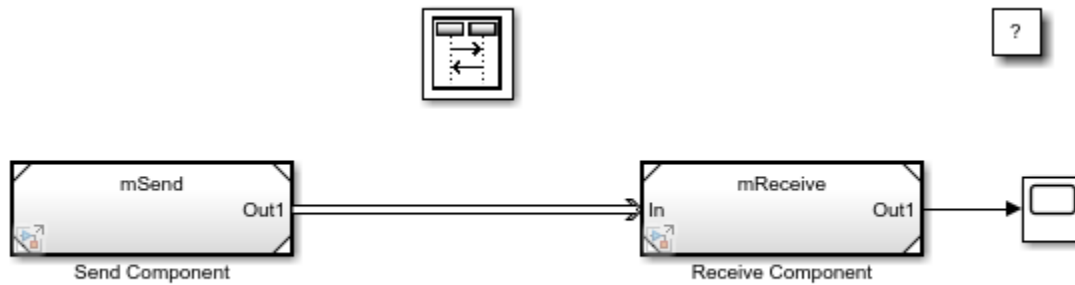
In the generated C++ code, the top model facilitates the connection between the send and receive referenced models by establishing a set interface that the referenced models can access independently of one another.

The generated C++ code implements message behavior:

- 1 A service is created at each model boundary. A service contains a reference to the top model and an entry point function, referred to as a service function, for referenced models to use to pass messages to the top model.
- 2 The top model initializes each service to create a connection to each referenced model.
- 3 The referenced models invoke service functions to pass messages to the top model.

C++ Code Generation Example

This example generates and examines C++ code from the model provided in “Establish Message Send and Receive Interfaces Between Software Components”.



Copyright 2019 The MathWorks, Inc.

Generate C++ Code:

- 1 Open the model.
- 2 In the Apps gallery, click **Embedded Coder**.
- 3 For each model (top and both referenced models), in the Configuration Parameters dialog box, set these parameters:
 - In the Code Generation pane, set **Language** to C++.
 - In the Interface pane, set **Code interface packaging** to C++ class.

Save the model.

- 4 Generate code. On the **C++ Code** tab, click **Build**.
- 5 View the generated code. On the **C++ Code** tab, click **View Code**.

Examine C++ Code:

- 1 A service is created at each model boundary. In C++, services are represented as objects that hold an instance of the top model and a service function that referenced models invoke to pass messages.
 - a To view the creation of the services, open the top model C++ file `MessageSendReceiveDefaultBufferModel.cpp`. View the constructor method.

```

// Constructor
MessageSendReceiveDefaultBufferModelModelClass::
MessageSendReceiveDefaultBufferModelModelClass():
MessageSendReceiveDefaultBuff_B()
,MessageSendReceiveDefaultBuf_DW()
,ReceiveComponentRecvData(*this)
,SendComponentSendData(*this)
,Receive_ComponentMDLOBJ0(get_ReceiveComponentRecvData())
,Send_ComponentMDLOBJ1(get_SendComponentSendData())
,MessageSendReceiveDefaultBuf_M()
{
// Currently there is no constructor body generated.
}

```

The constructor methods, `ReceiveComponentRecvData(*this)` and `SendComponentSendData(*this)`, create the receive and send service objects respectively by taking as an argument a reference to the instance of the top model. Each service object saves a reference to the top model and defines the message interface (the service functions `RecvData` and `SendData`).

- b To view the receive and send service classes, open the top model header file `MessageSendReceiveDefaultBufferModel.h`, and view the following section.

```

// Forward declaration
class MessageSendReceiveDefaultBufferModelModelClass;
class
MessageSendReceiveDefaultBufferModelModelClassMessa_ReceiveComponent_RecvDataT
: public RecvData_real_T
{
private:
MessageSendReceiveDefaultBufferModelModelClass & aProvider;
public:
MessageSendReceiveDefaultBufferModelModelClassMessa_ReceiveComponent_RecvDataT
(MessageSendReceiveDefaultBufferModelModelClass & aProvider);
virtual void RecvData(real_T *data, int32_T length, int32_T *status);
};

class
MessageSendReceiveDefaultBufferModelModelClassMessa_ReceiveComponent_SendDataT
: public SendData_real_T
{
private:
MessageSendReceiveDefaultBufferModelModelClass & aProvider;
public:
MessageSendReceiveDefaultBufferModelModelClassMessa_ReceiveComponent_SendDataT
(MessageSendReceiveDefaultBufferModelModelClass & aProvider);
virtual void SendData(const real_T *data, int32_T length, int32_T *status);
};

```

- 2 The top model initializes each service to create a connection to each referenced model. To view the initialization, open the top model C++ file `MessageSendReceiveDefaultBufferModel.cpp`. View the constructor method.

```

// Constructor
MessageSendReceiveDefaultBufferModelModelClass::
MessageSendReceiveDefaultBufferModelModelClass():
MessageSendReceiveDefaultBuff_B()
,MessageSendReceiveDefaultBuf_DW()
,ReceiveComponentRecvData(*this)
,SendComponentSendData(*this)
,Receive_ComponentMDL0BJ0(get_ReceiveComponentRecvData())
,Send_ComponentMDL0BJ1(get_SendComponentSendData())
,MessageSendReceiveDefaultBuf_M()
{
// Currently there is no constructor body generated.
}

```

The constructor method

`Receive_ComponentMDL0BJ0(get_ReceiveComponentRecvData())`, passes a reference to the receive service to the receive referenced model. The constructor method, `Send_ComponentMDL0BJ1(get_SendComponentSendData())`, passes a reference to the send service to the send referenced model.

- 3 The referenced models invoke service functions to pass messages to the top model. In C++, referenced models invoke the top model (common ancestor, if in a hierarchy) service functions to send or receive messages (specifically, a referenced model invokes the abstract service method (`RecvData` or `SendData`) from the interface created in step 2). The abstract interface classes are emitted to a shared folder. The implementation of the service functions in each service is defined in the top model C++ file.
 - a To view the abstract interface class to send messages, open from the shared folder the header file `SendData_real_T.h`.

```

class SendData_real_T
{
public:
virtual void SendData(const real_T *data, int32_T length, int32_T *status) = 0;
};

```

- b To view the implementation of the service function to send messages, open the top model C++ file `MessageSendReceiveDefaultBufferModel.cpp`.

```

// Model step function
void MessageSendReceiveDefaultBufferModelModelClass::
Messa_ReceiveComponent_SendData(const real_T *data, int32_T length, int32_T
*status_ptr) // Explicit Task: ExplicitTask0
{
UNUSED_PARAMETER(length);

```

- c To view how the send referenced model invokes the service function, open its C++ file `mSend.cpp`. In the step function, the model invokes the service function to send messages to the top model and receives back a return status.

```

// Send: '<Root>/Send'
Out1SendData.SendData(&rtb_Sinewave, sizeof(real_T), &status);

```

- d To view the abstract interface class to receive messages, open from the shared folder the header file `RecvData_real_T.h`.

```

class RecvData_real_T
{
public:
    virtual void RecvData(real_T *data, int32_T length, int32_T *status) = 0;
};

```

- e To view the implementation of the service function to receive messages, open top model C++ file `MessageSendReceiveDefaultBufferModel.cpp`.

```

// Model step function
void MessageSendReceiveDefaultBufferModelModelClass::
    Messa_ReceiveComponent_RecvData(real_T *data, int32_T length, int32_T
    *status_ptr) // Explicit Task: ExplicitTask0
{
    UNUSED_PARAMETER(length);
}

```

- f To view how the receive referenced model invokes the service function, open its C++ file `mRecieve.cpp`. In the step function, the model invokes the service function to receive messages and a status from the top model.

```

// Output and update for referenced model: 'mReceive'
void mReceiveModelClass::step(real_T *rty_Out1)
{
    int32_T status = 1;

    // Receive: '<Root>/Receive'
    status = 1;
    InRecvData.RecvData(rty_Out1, sizeof(real_T), &status);
}

```

Considerations and Limitations

- C++ code support is available for GRT-based system target files by using the Simulink Coder App.
- C++ code support is available for ERT-based system target files by using the Embedded Coder App.
- To generate code, for top and referenced models. select the same language (C++) and the same system target file.
- You cannot configure function prototype control (FPC) for a top model that has root message ports.
- External models and variant models are not supported.
- Software-in-the-loop (SIL) and processor-in-the-loop (PIL) simulations are not supported.

Generate C Messages to Communicate Data Between Simulink Components

To generate C code that supports message-based communication between model components in the Simulink environment, use the Simulink Messages & Events Library blocks `Send` and `Receive`. To customize the communication, use the `Queue` block (from the same library) to set parameters for capacity, sorting policy (LIFO, FIFO, and priority), and overwriting policy (behavior when the queue exceeds capacity). You can generate C code for GRT-based system target files by using Simulink Coder or for ERT-based system target files by using Embedded Coder.

Messages are an effective communication technique for distributed and complex systems that you can model within Simulink.

How to Prepare Models in Simulink for Message-Based Communication

In Simulink, you can model message-based communication between model components:

- 1** Create a model that contains a Send block (referenced model).
- 2** Create a model that contains a Receive block (referenced model).
- 3** Create a model that has two Model blocks (top model).
 - Set the first Model block to the model that contains the Send block (model from step 1).
 - Set the second Model block to the model that contains the Receive block (model from step 2).
- 4** When you run the model, a queue generates in the top model above the message line. A queue, explicit or implicit, controls the message communication. You can use the generated queue or you can add a Queue block in the top model to explicitly specify communication parameters.

How Model Components Pass Messages and How the Code Implements the Behavior

Referenced models pass messages:

- 1** In the model that contains the Send block, the Send block converts signals into messages.
- 2** The top model that contains the queue manages messages according to parameters that define capacity, order of delivery, and other quality of service (QoS) metrics.
- 3** In the model that contains the Receive block, the Receive block converts messages back to signals.

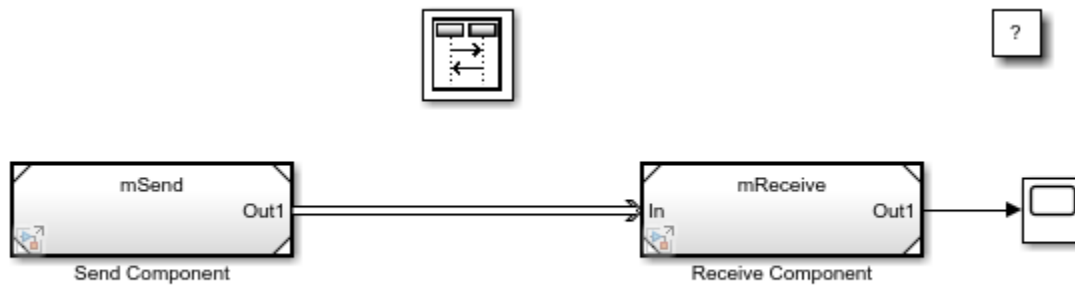
In the generated C code, the top model facilitates the connection between the send and receive referenced models by establishing a set interface that the referenced models can access independently of one another.

The generated C code implements message behavior:

- 1** A service is created at each model boundary. A service contains a reference to the top model and an entry point function, referred to as a service function, for referenced models to use to pass messages to the top model.
- 2** The top model initializes each service to create a connection to each referenced model.
- 3** The referenced models invoke service functions to pass messages to the top model.

C Code Generation Example

This example generates and examines C code from the model provided in “Establish Message Send and Receive Interfaces Between Software Components”.



Copyright 2019 The MathWorks, Inc.

Generate C Code:

- 1 Open the model.
- 2 In the Apps gallery, click **Embedded Coder**.
- 3 For each model (top and both referenced models), in the Configuration Parameters dialog box, in the Code Generation pane, set **Language** to C and save the model.
- 4 Generate code. On the **C Code** tab, click **Build**.
- 5 View the generated code. On the **C Code** tab, click **View Code**.

Examine C Code:

- 1 A service is created at each model boundary. In C, referenced models represent services as DWork. The service provides a pointer to an instance of the top model and a service function that referenced models invoke to pass messages.
 - a To view the service to send messages, open the send referenced model header file `mSend.h`. View the DWork allocation.

```

/* Block states (default storage) for model 'mSend' */
typedef struct {
    SendData_real_T Out1_SendData;    /* synthesized block */
    real_T lastSin;                   /* '<S1>/Sine Wave' */
    real_T lastCos;                   /* '<S1>/Sine Wave' */
    int32_T systemEnable;             /* '<S1>/Sine Wave' */
} DW_mSend_f_T;
    
```

- b To view the send message data type, if the data type is shareable (for example, built-in data types, imported bus types, or exported bus types that have a specified data type) the information is located in a shared header file. If the data type is not shareable, the information is located the model header file `mSend.h`. For this example, view the shareable data type by opening the shared header file `SendData_real_T.h`.

```

typedef struct {
    void *host;
    void (*SendData)(void *, const real_T *, int32_T, int32_T *);
} SendData_real_T;
    
```

- c To view the service to receive messages, open the receive referenced model header file, `mReceive.h`. View the DWork allocation.

```

/* Block states (default storage) for model 'mReceive' */
typedef struct {
    RecvData_real_T In_RecvData;      /* synthesized block */
} DW_mReceive_f_T;

```

- d** To view the received messages data type, if the data type is shareable (for example, built-in data types, imported bus types, or exported bus types that have a specified data type) the information is located in a shared header file. If the data type is not shareable the information is located the model header file, `mReceive.h`. For this example, view the shareable data type by opening the shared header file `RecvData_real_T.h`.

```

typedef struct {
    void *host;
    void (*RecvData)(void *, real_T *, int32_T, int32_T *);
} RecvData_real_T;

```

- 2** The top model initializes each service to create a connection to each referenced model. In C, the top model initializes each referenced model `DWork`. To view the initialization, open the top model C file `MessageSendReceiveDefaultBufferModel.c`.

```

/* Model initialize function */
void MessageSendReceiveDefaultBufferModel_initialize(void)
{
    /* Registration code */

    /* states (dwork) */
    MessageSendReceiveDefaultBuf_DW.SendComponent_InstanceData.rtdw.Out1_SendData.SendData
    = Messa_ReceiveComponent_SendData;
    MessageSendReceiveDefaultBuf_DW.SendComponent_InstanceData.rtdw.Out1_SendData.host
    = MessageSendReceiveDefaultBuf_M;
    MessageSendReceiveDefaultBuf_DW.ReceiveComponent_InstanceData.rtdw.In_RecvData.RecvData
    = Messa_ReceiveComponent_RecvData;
    MessageSendReceiveDefaultBuf_DW.ReceiveComponent_InstanceData.rtdw.In_RecvData.host
    = MessageSendReceiveDefaultBuf_M;
}

```

- 3** The referenced models invoke service functions to pass messages to the top model. In C, a referenced model invokes a service function by dereferencing the service function pointer and passing a pointer to the instance of the top model. You can view the prototypes of the service functions in the top model header file. You can view the referenced model invocations of those service functions in the referenced model C files.

- a** To view the prototype of the service function to send messages, open the top model header file, `MessageSendReceiveDefaultBufferModel.h`.

```

/* Exported entry point function */
extern void Messa_ReceiveComponent_SendData(void *host, const real_T *data,
    int32_T length, int32_T *status_ptr);

```

- b** To view the implementation of the service function to send messages, open the top model C file, `MessageSendReceiveDefaultBufferModel.c`.

```

/* Model step function */
void Messa_ReceiveComponent_SendData(void *host, const real_T *data, int32_T
length, int32_T *status_ptr)      /* Explicit Task: ExplicitTask0 */
{
    UNUSED_PARAMETER(host);
    UNUSED_PARAMETER(length);

    /* DiscreteEventSubgraph generated from: '<Root>/Receive Component' */
    *status_ptr = Messa_ReceiveComponent_SendDa_l(data);
}

static int32_T Messa_ReceiveComponent_SendDa_l(const real_T *data)
{
    int32_T status = 1;
    Msg_real_T msg;

    /* DiscreteEventSubgraph generated from: '<Root>/Receive Component' */
    msg = MessageSendReceiveDefaultBufferModel_createMsg(data);

    /* Queue generated from: '<Root>/Receive Component' incorporates:
     * DiscreteEventSubgraph generated from: '<Root>/Receive Component'
     */
    if (MessageSendReceiveDefaultBuff_l
        (&MessageSendReceiveDefaultBuf_DW.Queue_InsertedFor_ReceiveCompon, &msg))
    {
        status = 0;
    }

    /* End of Queue generated from: '<Root>/Receive Component' */
    return status;
}

```

- c To view how the send referenced model invokes the service function, open the C file for the model `mSend.c`. In the step function, the model invokes the service to send instance data and a message to the top model and receives back a return status.

```

/* Send: '<Root>/Send' */
if (localDW->Out1_SendData.SendData != NULL) {
    localDW->Out1_SendData.SendData(localDW->Out1_SendData.host, &rtb_Sinewave,
    sizeof(real_T), &status);
}

```

- d To view the prototype of the service function to receive messages, open the top model header file `MessageSendReceiveDefaultBufferModel.h`.

```

/* Exported entry point function */
extern void Messa_ReceiveComponent_RecvData(void *host, real_T *data, int32_T
length, int32_T *status_ptr);

```

- e To view the implementation of the service function to receive messages, open the top model C file `MessageSendReceiveDefaultBufferModel.c`.

```

/* Model step function */
void Messa_ReceiveComponent_RecvData(void *host, real_T *data, int32_T length,
int32_T *status_ptr) /* Explicit Task: ExplicitTask0 */
{
    UNUSED_PARAMETER(host);
    UNUSED_PARAMETER(length);

    /* DiscreteEventSubgraph generated from: '<Root>/Receive Component' */
    *status_ptr = Messa_ReceiveComponent_RecvDa_l(data);
}

static int32_T Messa_ReceiveComponent_RecvDa_l(real_T *data)
{
    int32_T status = 1;
    Msg_real_T msg;

    /* Queue generated from: '<Root>/Receive Component' incorporates:
    * DiscreteEventSubgraph generated from: '<Root>/Receive Component'
    */
    if (MessageSendReceiveDefaultBuf_lh
        (&MessageSendReceiveDefaultBuf_DW.Queue_InsertedFor_ReceiveCompon, &msg))
    {
        status = 0;
        *data = msg.fData;
    }

    /* End of Queue generated from: '<Root>/Receive Component' */
    return status;
}

```

- f To view how the receive referenced model invokes the service function, open the C file for the model `mReceive.c`. In the step function, the model invokes the service to receive a message payload and a return status.

```

/* Output and update for referenced model: 'mReceive' */
void mReceive(real_T *rty_Out1, DW_mReceive_f_T *localDW)
{
    int32_T status = 1;

    /* Receive: '<Root>/Receive' */
    status = 1;
    if (localDW->In_RecvData.RecvData != NULL) {
        localDW->In_RecvData.RecvData(localDW->In_RecvData.host, rty_Out1, sizeof
            (real_T), &status);
    }

    /* End of Receive: '<Root>/Receive' */
}

```

Considerations and Limitations

- C code support is available for GRT-based system target files by using the Simulink Coder App.
- C code support is available for ERT-based system target files by using the Embedded Coder App.
- To generate code, for top and referenced models, select the same language (C) and the same system target file.

- You cannot configure function prototype control (FPC) for a top model that has root message ports.
- External models and variant models are not supported.
- Software-in-the-loop (SIL) and processor-in-the-loop (PIL) simulations are not supported.

See Also

`coder.ceval` | `coder.ExternalDependency` |

`coder.ExternalDependency.updateBuildInfo` | `Parameter Writer` | `Send` | `Receive` | `Queue`

Related Examples

- “Simulink Messages Overview”
- “Establish Message Send and Receive Interfaces Between Software Components”
- “Wireless Tire Pressure Monitoring System with Fault Logging”
- “Prepare Sensor and Controller Models in a Distributed Monitoring System for Code Generation”
- “Develop Interface for External C/C++ Code”

Client-Server Communication Interfaces

Applications in a platform environment can communicate by using client-server communication. To model client-server communication, use the Simulink Function and Function Caller blocks. A Simulink Function block represents a shared resource. You define the logic as a resource in a Simulink Function block, which separates the function interface (name and arguments) from the logic implementation. Function callers (Function Caller blocks, MATLAB Function blocks, and Stateflow charts) can use the function logic at different levels of the model hierarchy.

The Simulink Function block shares states between function callers. The code generator produces one function. If the Simulink Function block contains blocks that have states, such as a delay or memory, the states persist between function callers.

For more information about using the Simulink Function and Function Caller blocks, see “Simulink Function Blocks and Code Generation” on page 10-6.

See Also

Simulink Function | Function Caller

Related Examples

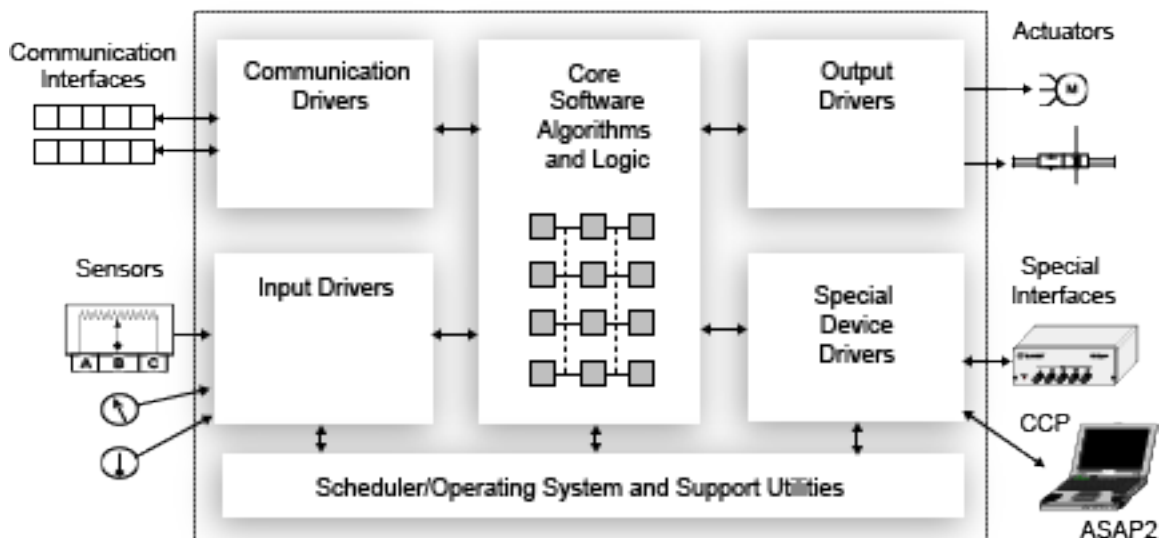
- “Simulink Functions Overview”
- “Simulink Function Blocks and Code Generation” on page 10-6
- “Generate Reentrant Code from Simulink Function Blocks” on page 10-18
- “Configure Entry-Point Function Interfaces for Simulink Function and Function Caller Blocks” on page 17-2

Device Driver I/O Interfaces

Device drivers interact with hardware directly. They depend on characteristics of the hardware. For example, a device driver for an analog-to-digital converter initializes, reads data from, and writes data to hardware registers. Hardware differences and dependencies concern data type size, endianness, shift operations, compiler directives, and optimized function and operator support. Other application code interfaces with device drivers by using a programming interface and data mapped to specific memory addresses. Typically, simulation on a development computer is not possible. Reading from and writing to a register during simulation on a development computer produces unexpected and unwanted results.

Device drivers for communication protocols and target hardware are essential to real-time development projects. For example, you can have a working device driver that you want to integrate with algorithmic code that reads data from and writes data to an I/O device that the driver supports. The code generator can produce a single set of application source files from an algorithm model and integrated driver code.

This figure shows an application that requires integration of existing driver code for hardware devices. The core software algorithms and logic can be a combination of code modules for external reusable algorithms integrated into the Simulink environment and code generated as part of an overall model design.



Integrate device driver code in a component model by using one of these methods:

- Import driver function code into a C Caller block.
- Generate an S-function, compile and link the generated S-function, and create a mask for calling the generated S-function by using the Legacy Code Tool.

See Also

Related Examples

- “Integrate C Code Using C Caller Blocks”
- “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6

Modeling in Simulink Coder

- “Use Products and Blocks Supported for Code Generation” on page 3-2
- “Consider Modeling Semantics when Generating Code” on page 3-5
- “Modeling Guidelines for Blocks” on page 3-11
- “Modeling Guidelines for Model Configuration” on page 3-12

Use Products and Blocks Supported for Code Generation

As you construct a model, to prevent issues later in the development process and potential rework, determine whether the code generator supports products and blocks that you want to use.


Related Products

The following table summarizes MathWorks products that extend and complement Simulink Coder Embedded Coder software. For information about these products and how they support code generation, see the product-specific documentation at www.mathworks.com.

Product	Examples of Code Generation Applications
Aerospace Blockset™	Aircraft, spacecraft, rocket, propulsion systems, and unmanned airborne vehicles
Audio Toolbox™	Audio processing systems
Automated Driving Toolbox™	Designing, simulating, and testing ADAS and autonomous driving systems
AUTOSAR Blockset	Modeling and simulation of AUTOSAR Classic and Adaptive ECU software
Communications Toolbox™	Physical layer of communication systems
Computer Vision Toolbox™	Video processing, image processing, and computer vision systems
Control System Toolbox™	Linear control systems
DSP System Toolbox™	Signal processing systems
Embedded Coder	Embedded systems, rapid prototyping boards, and microprocessors in mass production
Fixed-Point Designer™	Fixed-point systems
Fuzzy Logic Toolbox™	System designs based on fuzzy logic
HDL Verifier™	Direct programming interface (DPI) component and transaction-level model (TLM) generation from Simulink
IEC Certification Kit	ISO 26262 and IEC 61508 certification
Model-Based Calibration Toolbox™	Developing processes for systematically identifying optimal balance of engine performance, emissions, and fuel economy, and reusing statistical models for control design, hardware-in-the-loop (HIL) testing, or powertrain simulation
Model Predictive Control Toolbox™	Model predictive controllers
Deep Learning Toolbox™	Neural networks
Parallel Computing Toolbox™	Parallel builds for large Simulink models
Phased Array System Toolbox™	Sensor array systems in radar, sonar, wireless communications, and medical imaging applications
Polyspace® Bug Finder™	MISRA-C compliance and static analysis of generated code

Product	Examples of Code Generation Applications
Polyspace Code Prover™	Formal analysis of generated code
Powertrain Blockset™	Real-time testing of powertrain applications
Robotics System Toolbox™	Design, simulate, and test robotics applications
ROS Toolbox	Design, simulate, and deploy ROS-based applications
Simscape™	Systems spanning mechanical, electrical, hydraulic, and other physical domains as physical networks
Simscape Driveline™	Driveline (drivetrain) systems
Simscape Electrical™	Electronic, electromechanical, and electrical power systems
Simscape Fluids™	Hydraulic power and control systems
Simscape Multibody™	Three-dimensional mechanical systems
Simulink 3D Animation™	Systems with 3D visualizations
Simulink Check™	Model standards compliance checking and metrics
Simulink Code Inspector™	Automated reviews of generated code
Simulink Control Design™	Autotuning of PID controllers and online frequency response estimation
Simulink Coverage™	Model and code structural coverage analysis
Simulink Design Optimization™	Tuning adaptive lookup tables and checking signal characteristics
Simulink Desktop Real-Time™	Rapid prototyping or hardware-in-the-loop (HIL) simulation of control system and signal processing algorithms
Simulink Real-Time™	Rapid control prototyping, hardware-in-the-loop (HIL) simulation, and other real-time testing applications
Simulink Report Generator™	Automatically generating project documentation in a standard format
Requirements Toolbox™	Authoring and tracing requirements to design and code
Simulink Test™	Software-in-the-loop (SIL), processor-in-the-loop (PIL), and real-time hardware-in-the-loop (HIL) testing of generated code
SoC Blockset™	Designing, evaluating, and implementing SoC hardware and software architectures
Stateflow®	State machines and flow charts
System Identification Toolbox™	Systems identified from measured input-output data
Vehicle Dynamics Blockset™	Modeling and simulation of vehicle dynamics in 3D environment
Vehicle Network Toolbox™	CAN blocks for Accelerator and Rapid Accelerator simulations and code deployment on Windows®

Blocks Supported for Code Generation

You can generate efficient code for a subset of Simulink built-in blocks and product-specific blockset blocks. The following tables list these blocks. In the tables, an information  icon before the name of a block indicates that specific usage notes or limitations apply to code generation for that block. To view the usage notes and limitations, in the corresponding reference page, scroll down to the **Extended Capabilities** section at the bottom of the page and expand the **C/C++ Code Generation** section.

- Blocks Supported for C/C++ Code Generation (Category List)
- Blocks Supported for C/C++ Code Generation (Alphabetical List)

Simulink also provides a web page that summarizes characteristics of Simulink built-in blocks. The content, which is sourced from the Simulink block library, identifies blocks that are not recommended for production code generation and gives other usage guidelines. To view the page, in the MATLAB® Command Window, type `showblockdatatypetable`.

For detailed information about a block, see the block reference page.

For modeling guidelines regarding blocks, see “Modeling Guidelines for Blocks” on page 3-11.

Block Set Support for Code Generation

Several products that include blocks are available for you to consider for code generation. However, before using the blocks for one of these products, consult the documentation for that product to confirm which blocks support code generation.

Consider Modeling Semantics when Generating Code

In this section...
"Data Propagation" on page 3-5
"Sample Time Propagation" on page 3-6
"Latches for Subsystem Blocks" on page 3-7
"Block Execution Order" on page 3-8
"Algebraic Loops" on page 3-9

Data Propagation

The first stage of code generation is compilation of the block diagram. This stage is analogous to that of a C or C++ program. The compiler carries out type checking and preprocessing. Similarly, the Simulink engine verifies that input/output data types of block ports are consistent, line widths between blocks are of expected thickness, and the sample times of connecting blocks are consistent.

The Simulink engine propagates data from one block to the next along signal lines. The data propagated consists of

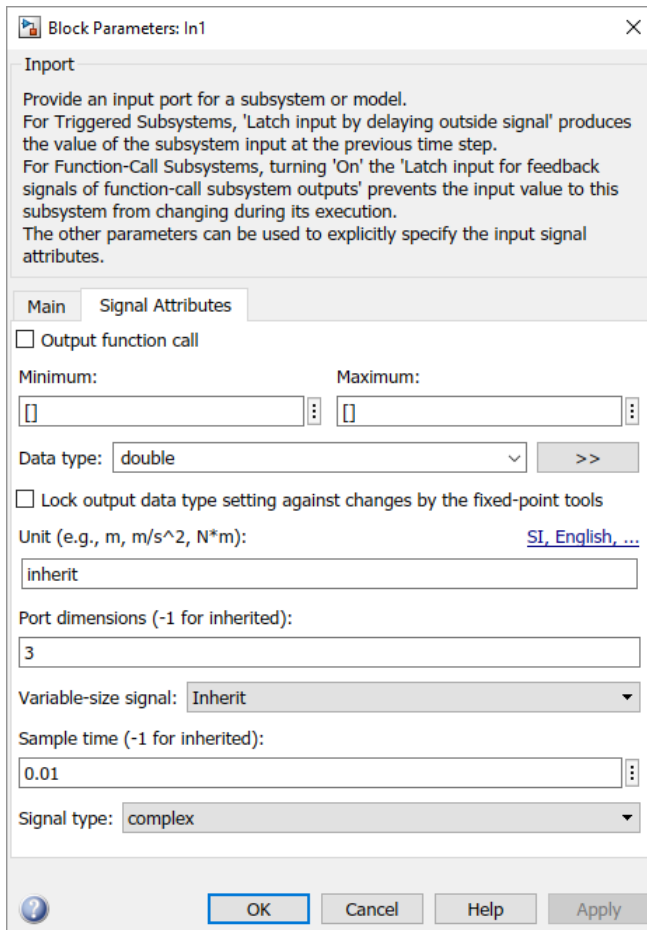
- Data type
- Line widths
- Sample times

You can verify what data types a Simulink block supports by typing

```
showblockdatatypetable
```

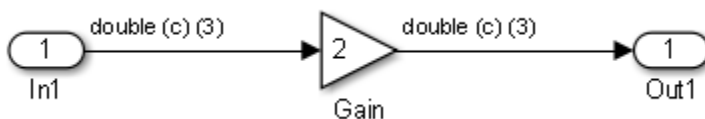
at the MATLAB prompt, or (from the Help browser) clicking the command above.

The Simulink engine typically derives signal attributes from a source block. For example, the Inport block's parameters dialog box specifies the signal attributes for the block.



In this example, the Inport block has a port width of 3, a sample time of .01 seconds, the data type is double, and the signal is complex.

This figure shows the propagation of the signal attributes associated with the Inport block through a simple block diagram.



In this example, the Gain and Outport blocks inherit the attributes specified for the Inport block.

Sample Time Propagation

Inherited sample times in source blocks (for example, a root inport) can sometimes lead to unexpected and unintended sample time assignments. Since a block may specify an inherited sample time, information available at the outset is often insufficient to compile a block diagram completely.

In such cases, the Simulink engine propagates the known or assigned sample times to those blocks that have inherited sample times but that have not yet been assigned a sample time. Thus, the engine continues to fill in the blanks (the unknown sample times) until sample times have been assigned to

as many blocks as possible. Blocks that still do not have a sample time are assigned a default sample time.

For a completely deterministic model (one where no sample times are set using the above rules), you should explicitly specify the sample times of your source blocks. Source blocks include root inport blocks and blocks without input ports. You do not have to set subsystem input port sample times. You might want to do so, however, when creating modular systems.

An unconnected input implicitly connects to ground. For ground blocks and ground connections, the sample time is always constant (`inf`).

All blocks have an inherited sample time ($T_s = -1$). They are assigned a sample time of $(T_f - T_i)/50$.

Blocks Whose Outputs Have Constant Values

When you display sample time colors, by default, Constant blocks appear magenta in color to indicate that the block outputs have constant values during simulation. Downstream blocks whose output values are also constant during simulation, such as Gain blocks, similarly appear magenta if they use an inherited sample time. The code generated for these blocks depends in part on the tunability of the block parameters.

If you set model configuration parameter **Default parameter behavior** to `Inlined`, the block parameters are not tunable in the generated code. Because the block outputs are constant, the code generator eliminates the block code due to constant folding. If the code generator cannot fold the code, or if you select settings to disable constant folding, the block code appears in the model initialization function. The generated code is more efficient because it does not compute the outputs of these blocks during execution.

However, if you configure a block or model so that the block parameters appear in the generated code as tunable variables, the code generator represents the blocks in a different way. Block parameters are tunable if, for example:

- You set parameter **Default parameter behavior** to `Tunable`. By default, numeric block parameters appear as tunable fields of a global parameter structure.
- You use a tunable parameter, such as a `Simulink.Parameter` object that uses a storage class other than `Auto`, as the value of one or more numeric block parameters. These block parameters are tunable regardless of the setting that you choose for model configuration parameter **Default parameter behavior**.

If a block parameter is tunable, the generated code must compute the block outputs during execution. Therefore, the block code appears in the model `step` function. If the model uses multiple discrete rates, the block code appears in the output function for the fastest downstream rate that uses the block outputs.

Latches for Subsystem Blocks

When an Inport block is the signal source for a triggered or function-call subsystem, you can use latch options to preserve input values while the subsystem executes. The Inport block latch options include:

For	Use
Triggered subsystems	Latch input by delaying outside signal

For	Use
Function-call subsystems	Latch input for feedback signals of function-call subsystem outputs

When you select subsystem block parameter **Latch input for feedback signals of function-call subsystem outputs** for a function-call subsystem, the code generator

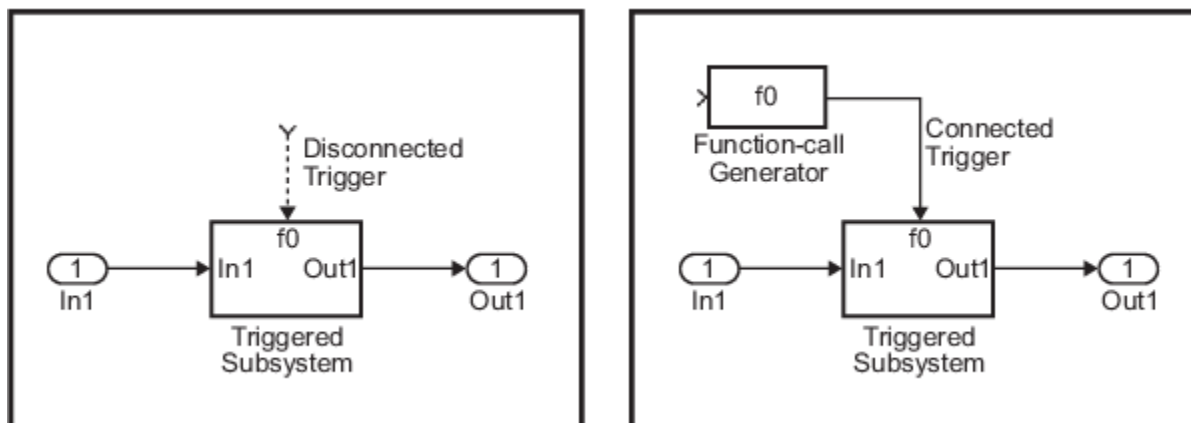
- Preserves latches in generated code regardless of optimizations that might be set
- Places the code for latches at the start of a subsystem's output/update function

For more information on these options, see the block description of Inport.

Block Execution Order

Once the Simulink engine compiles the block diagram, it creates a *model*.rtw file (analogous to an object file generated from a C or C++ file). The *model*.rtw file contains the connection information of the model, as well as the signal attributes. Thus, the timing engine in can determine when blocks with different rates should be executed.

You cannot override this execution order by directly calling a block (in handwritten code) in a model. For example, in the next figure the `disconnected_trigger` model on the left has its trigger port connected to ground, which can lead to the blocks inheriting a constant sample time. Calling the trigger function, `f()`, directly from user code does not work. Instead, you should use a function-call generator to specify the rate at which `f()` should be executed, as shown in the `connected_trigger` model on the right.



Instead of the function-call generator, you could use another block that can drive the trigger port. Then, you should call the model's main entry point to execute the trigger function.

For multirate models, a common use of the code generator is to generate code for individual models separately and then manually code the I/O between the generated code modules. This approach places the burden of data consistency between models on the developer of the models. Another approach is to let Simulink and the code generator maintain data consistency between rates and generate multirate code for use in a multitasking environment. The Rate Transition block is able to interface periodic and asynchronous signals. For a description of the Simulink Coder block libraries, see “Asynchronous Events” on page 6-2. For more information on multirate code generation, see “Modeling for Multitasking Execution” on page 5-9.

Algebraic Loops

Algebraic loops are circular dependencies between variables. This prevents the straightforward direct computation of their values. For example, in the case of a system of equations

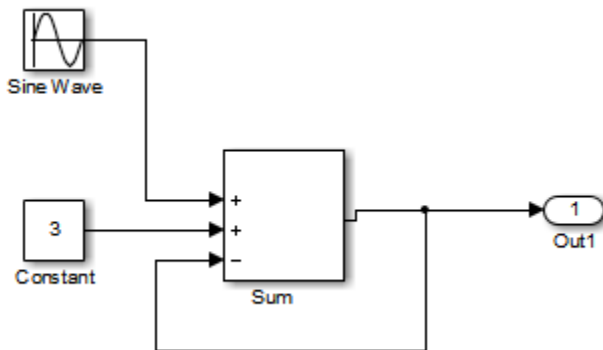
- $x = y + 2$
- $y = -x$

the values of x and y cannot be directly computed.

To solve this, either repeatedly try potential solutions for x and y (in an intelligent manner, for example, using gradient based search) or “solve” the system of equations. In the previous example, solving the system into an explicit form leads to

- $2x = 2$
- $y = -x$
- $x = 1$
- $y = -1$

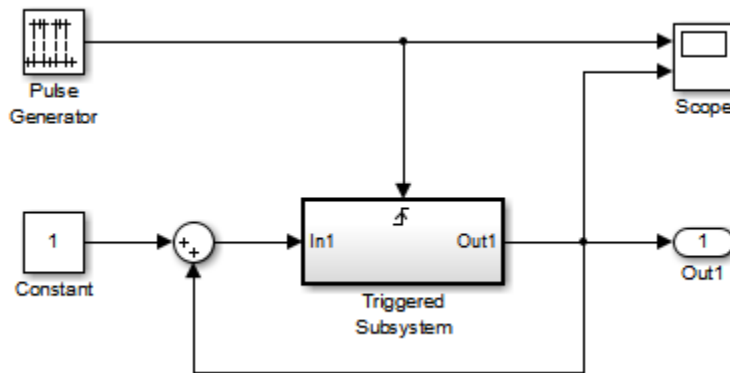
An algebraic loop exists whenever the output of a block having direct feedthrough (such as Gain, Sum, Product, and Transfer Fcn) is fed back as an input to the same block. The Simulink engine is often able to solve models that contain algebraic loops, such as the next diagram.



The code generator does not produce code that solves algebraic loops. This restriction includes models that use Algebraic Constraint blocks in feedback paths. However, the Simulink engine can often eliminate algebraic loops that arise, by grouping equations in certain ways in models that contain them. It does this by separating the update and output functions to avoid circular dependencies. For details, see “Algebraic Loop Concepts”.

Algebraic Loops in Triggered Subsystems

While the Simulink engine can minimize algebraic loops involving atomic and enabled subsystems, a special consideration applies to some triggered subsystems. An example for which code can be generated is shown in the following model and triggered subsystem.



The default Simulink behavior is to combine output and update methods for the subsystem, which creates an apparent algebraic loop, even though the Unit Delay block in the subsystem has no direct feedthrough.

You can allow the Simulink engine to solve the problem by splitting the output and update methods of triggered and enabled-triggered subsystems when feasible. If you want the code generator to take advantage of this feature, select subsystem block parameter **Minimize algebraic loop occurrences**. Select this parameter to avoid algebraic loop warnings in triggered subsystems involved in loops.

Note If you check this box, the generated code for the subsystem might contain split output and update methods, even if the subsystem is not actually involved in a loop. Also, if a direct feedthrough block (such as a Gain block) is connected to the inport in the above triggered subsystem, the Simulink engine cannot solve the problem, and the code generator is unable to generate code.

A model configuration parameter **Minimize algebraic loop occurrences** is available also. Selecting it enables the code generator to produce code for models containing Model blocks that are involved in algebraic loops.

Modeling Guidelines for Blocks

Code generation modeling guidelines include recommended model settings, block usage, and block parameters. When you develop models for code generation, use these guidelines.

Code Generation Modeling Guidelines	"cgsl_0101: Zero-based indexing" "cgsl_0102: Evenly spaced breakpoints in lookup tables" "cgsl_0103: Precalculated signals and parameters" "cgsl_0201: Redundant Unit Delay and Memory blocks"
---	---

See Also

"Modeling Guidelines for Model Configuration" on page 3-12

Modeling Guidelines for Model Configuration

When you develop models and generate code, use the modeling guideline configuration recommendations.

Code Generation Modeling Guidelines	"cgsl_0301: Prioritization of code generation objectives for code efficiency"
High-Integrity Systems Modeling Guidelines	<p>"hisl_0043: Configuration Parameters > Diagnostics > Solver"</p> <p>"hisl_0044: Configuration Parameters > Diagnostics > Sample Time"</p> <p>"hisl_0301: Configuration Parameters > Diagnostics > Compatibility"</p> <p>"hisl_0302: Configuration Parameters > Diagnostics > Data Validity > Parameters"</p> <p>"hisl_0303: Configuration Parameters > Diagnostics > Data Validity > Merge blocks"</p> <p>"hisl_0304: Configuration Parameters > Diagnostics > Data Validity > Model initialization"</p> <p>"hisl_0305: Configuration Parameters > Diagnostics > Data Validity > Debugging"</p> <p>"hisl_0306: Configuration Parameters > Diagnostics > Connectivity > Signals"</p> <p>"hisl_0307: Configuration Parameters > Diagnostics > Connectivity > Buses"</p> <p>"hisl_0308: Configuration Parameters > Diagnostics > Connectivity > Function calls"</p> <p>"hisl_0309: Configuration Parameters > Diagnostics > Type Conversion"</p> <p>"hisl_0310: Configuration Parameters > Diagnostics > Model Referencing"</p> <p>"hisl_0311: Configuration Parameters > Diagnostics > Stateflow"</p>

See Also

"Modeling Guidelines for Blocks" on page 3-11

Timers in Simulink Coder

- “Absolute and Elapsed Time Computation” on page 4-2
- “Access Timers Programmatically” on page 4-4
- “Specify Clock Resolution Used by Target Environment Clock” on page 4-7
- “Generate Code for an Elapsed Time Counter” on page 4-11
- “Absolute Time Limitations” on page 4-14

Absolute and Elapsed Time Computation

In this section...

“About Timers” on page 4-2

“Timers for Periodic and Asynchronous Tasks” on page 4-2

“Allocation of Timers” on page 4-3

“Integer Timers in Generated Code” on page 4-3

“Elapsed Time Counters in Triggered Subsystems” on page 4-3

About Timers

Certain blocks require the value of either *absolute* time (that is, the time from the start of program execution to the present time) or *elapsed* time (for example, the time elapsed between two trigger events). Targets that support the real-time model (`rtModel`) data structure provide efficient time computation services to blocks that request absolute or elapsed time. Absolute and elapsed timer features include

- Timers are implemented as unsigned integers in generated code.
- In multirate models, at most one timer is allocated per rate. If no blocks executing at a given rate require a timer, a timer is not allocated to that rate. This minimizes memory allocated for timers and significantly reduces overhead involved in maintaining timers.
- Allocation of elapsed time counters for use of blocks within triggered subsystems is minimized, further reducing memory usage and overhead.
- S-function and TLC APIs let your S-functions access timers, in simulation and code generation.
- The word size of the timers is determined by a user-specified maximum counter value that you set with model configuration parameter **Application lifespan (days)**. If you specify this value, timers do not overflow. For more information, see “Control Memory Allocation for Time Counters” on page 40-10.

See “Absolute Time Limitations” on page 4-14 for more information about absolute time and the restrictions that it imposes.

Timers for Periodic and Asynchronous Tasks

Timing services provided for blocks execute within *periodic* tasks (that is, tasks running at the model base rate or subrates).

The code generator also provides timer support for blocks whose execution is *asynchronous* with respect to the periodic timing source of the model. See the following topics:

- “Timers in Asynchronous Tasks” on page 6-38
- “Create a Customized Asynchronous Library” on page 6-40

Allocation of Timers

If you create or maintain an S-Function block that requires absolute or elapsed time data, it must register the requirement (see “Access Timers Programmatically” on page 4-4). In multirate models, timers are allocated on a per-rate basis. For example, consider a model structured as follows:

- There are three rates, A, B, and C, in the model.
- No blocks running at rate B require absolute or elapsed time.
- Two blocks running at rate C register a requirement for absolute time.
- One block running at rate A registers a requirement for absolute time.

In this case, two timers are generated, running at rates A and C respectively. The timing engine updates the timers as the tasks associated with rates A and C execute. Blocks executing at rates A and C obtain time data from the timers associated with rates A and C.

Integer Timers in Generated Code

In the generated code, timers for absolute and elapsed time are implemented as unsigned integers. The default size is 64 bits. This is the amount of memory allocated for a timer if you set model configuration parameter **Application lifespan (days)** to `inf`. For an application with a sample rate of 1000 MHz, a 64-bit counter will not overflow for more than 500 years. See “Timers in Asynchronous Tasks” on page 6-38 and “Control Memory Allocation for Time Counters” on page 40-10 for more information.

Elapsed Time Counters in Triggered Subsystems

Some blocks, such as the Discrete-Time Integrator block, perform computations requiring the elapsed time (ΔT) since the previous block execution. Blocks requiring elapsed time data must register the requirement (see “Access Timers Programmatically” on page 4-4). A triggered subsystem then allocates and maintains a single elapsed time counter if required. This timer functions at the subsystem level, not at the individual block level. The timer is generated if the triggered subsystem (or a unconditionally executed subsystem within the triggered subsystem) contains one or more blocks requiring elapsed time data.

Note If you are using simplified initialization mode, elapsed time is reset on first execution after becoming enabled, whether or not the subsystem is configured to reset on enable. For more information, see “Underspecified initialization detection”.

See Also

More About

- “Access Timers Programmatically” on page 4-4
- “Generate Code for an Elapsed Time Counter” on page 4-11
- “Optimize Memory Usage for Time Counters” on page 40-84
- “Absolute Time Limitations” on page 4-14

Access Timers Programmatically

In this section...

“About Timer APIs” on page 4-4

“C API for S-Functions” on page 4-4

“TLC API for Code Generation” on page 4-6

About Timer APIs

This topic describes APIs that let your S-functions take advantage of the efficiencies offered by absolute and elapsed timers. SimStruct macros are provided for use in simulation, and TLC functions are provided for inlined code generation. Note that

- To generate and use the new timers as described above, your S-functions must register the need to use an absolute or elapsed timer by calling `ssSetNeedAbsoluteTime` or `ssSetNeedElapseTime` in `mdlInitializeSampleTime`.
- Existing S-functions that read absolute time but do not register by using these macros continue to operate as expected, but generate less efficient code.

C API for S-Functions

The SimStruct macros described in this topic provide access to absolute and elapsed timers for S-functions during simulation.

In the functions below, the SimStruct `*S` argument is a pointer to the `simstruct` of the calling S-function.

- `void ssSetNeedAbsoluteTime(SimStruct *S, boolean b)`: if `b` is `TRUE`, registers that the calling S-function requires absolute time data, and allocates an absolute time counter for the rate at which the S-function executes (if such a counter has not already been allocated).
- `int ssGetNeedAbsoluteTime(SimStruct *S)`: returns 1 if the S-function has registered that it requires absolute time.
- `double ssGetTaskTime(SimStruct *S, tid)`: read absolute time for a given task with task identifier `tid`. `ssGetTaskTime` operates transparently, regardless of whether or not you use the new timer features. `ssGetTaskTime` is documented in the SimStruct Functions chapter of the Simulink documentation.
- `void ssSetNeedElapseTime(SimStruct *S, boolean b)`: if `b` is `TRUE`, registers that the calling S-function requires elapsed time data, and allocates an elapsed time counter for the triggered subsystem in which the S-function executes (if such a counter has not already been allocated). See also “Elapsed Time Counters in Triggered Subsystems” on page 4-3.
- `int ssGetNeedElapseTime(SimStruct *S)`: returns 1 if the S-function has registered that it requires elapsed time.
- `void ssGetElapseTime(SimStruct *S, (double *)elapseTime)`: returns, to the location pointed to by `elapseTime`, the value (as a double) of the elapsed time counter associated with the S-function.
- `void ssGetElapseTimeCounterDtype(SimStruct *S, (int *)dtype)`: returns the data type of the elapsed time counter associated with the S-function to the location pointed to by

`dtype`. This function is intended for use with the `ssGetElapseTimeCounter` function (see below).

- `void ssGetElapseResolution(SimStruct *S, (double *)resolution)`: returns the resolution (that is, the sample time) of the elapsed time counter associated with the S-function to the location pointed to by `resolution`. This function is intended for use with the `ssGetElapseTimeCounter` function (see below).
- `void ssGetElapseTimeCounter(SimStruct *S, (void *)elapseTime)`: This function is provided for the use of blocks that require the elapsed time values for fixed-point computations. `ssGetElapseTimeCounter` returns, to the location pointed to by `elapseTime`, the integer value of the elapsed time counter associated with the S-function. If the counter size is 64 bits, the value is returned as an array of two 32-bit words, with the low-order word stored at the lower address.

To determine how to access the returned counter value, obtain the data type of the counter by calling `ssGetElapseTimeCounterDtype`, as in the following code:

```
int    *y_dtype;
ssGetElapseTimeCounterDtype(S, y_dtype);

switch(*y_dtype) {
  case SS_DOUBLE_UINT32:
    {
      uint32_T dataPtr[2];
      ssGetElapseTimeCounter(S, dataPtr);
    }
    break;
  case SS_UINT32:
    {
      uint32_T dataPtr[1];
      ssGetElapseTimeCounter(S, dataPtr);
    }
    break;
  case SS_UINT16:
    {
      uint16_T dataPtr[1];
      ssGetElapseTimeCounter(S, dataPtr);
    }
    break;
  case SS_UINT8:
    {
      uint8_T dataPtr[1];
      ssGetElapseTimeCounter(S, dataPtr);
    }
    break;
  case SS_DOUBLE:
    {
      real_T dataPtr[1];
      ssGetElapseTimeCounter(S, dataPtr);
    }
    break;
  default:
    ssSetErrorStatus(S, "Invalid data type for elapse time
      counter");
    break;
}
```

If you want to use the actual elapsed time, issue a call to the `ssGetElapseTime` function to access the elapsed time directly. You do not need to get the counter value and then calculate the elapsed time.

```
double *y_elapseTime;  
.  
.  
.  
ssGetElapseTime(S, elapseTime)
```

TLC API for Code Generation

The following TLC functions support elapsed time counters in generated code when you inline S-functions by writing TLC scripts for them.

- `LibGetTaskTimeFromTID(block)`: Generates code to read the absolute time for the task in which `block` executes.

`LibGetTaskTimeFromTID` is documented with other sample time functions in the TLC Function Library Reference pages of the Target Language Compiler documentation.

Note Do not use `LibGetT` for this purpose. `LibGetT` always reads the base rate (`tid 0`) timer. If `LibGetT` is called for a block executing at a subrate, the wrong timer is read, causing serious errors.

- `LibGetElapseTime(system)`: Generates code to read the elapsed time counter for `system`. (`system` is the parent system of the calling block.) See “Generate Code for an Elapsed Time Counter” on page 4-11 for an example of code generated by this function.
- `LibGetElapseTimeCounter(system)`: Generates code to read the integer value of the elapsed time counter for `system`. (`system` is the parent system of the calling block.) This function should be used in conjunction with `LibGetElapseTimeCounterDtypeId` and `LibGetElapseTimeResolution`. (See the discussion of `ssGetElapseTimeCounter` above.)
- `LibGetElapseTimeCounterDtypeId(system)`: Generates code that returns the data type of the elapsed time counter for `system`. (`system` is the parent system of the calling block.)
- `LibGetElapseTimeResolution(system)`: Generates code that returns the resolution of the elapsed time counter for `system`. (`system` is the parent system of the calling block.)

See Also

More About

- “Absolute and Elapsed Time Computation” on page 4-2
- “Generate Code for an Elapsed Time Counter” on page 4-11
- “Absolute Time Limitations” on page 4-14

Specify Clock Resolution Used by Target Environment Clock

For model functions that include blocks that request absolute or elapsed time values, improve the entry-point code generated for those functions by configuring the model to use a specific clock resolution. Clock resolution is the smallest increment of a clock value. For example, if a clock increments its value once per second, the clock resolution is 1 second.

Benefits of specifying a clock resolution include:

- Clock resolution that aligns with target environment clock requirements. For models configured to use a service code interface, specifying a clock resolution results in code that reads time values that produce more accurate results.
- Decoupling of the clock resolution and the solver properties that Simulink uses during simulation, such as the fixed-step size and sample times. The decoupling enables you to generate code that aligns with the target environment clock resolution and produces more accurate results.
- Influencing the data type that Simulink and the code generator use to represent time values. For example, in normal, accelerator, and rapid accelerator modes, Simulink uses the specified clock resolution to deduce fixed-point data types, which produces fixed-point simulation and generated code execution output that match.
- Portability of models between code interface configurations. You can attach a model that has a specified clock resolution to a shared Embedded Coder Dictionary that defines a data or service code interface configuration.
- Generated code that is easier to read.

Specify Clock Resolution

To specify a clock resolution for the code generator to apply for a model, configure the model with these model configuration parameter settings:

- Set **System target file** (SystemTargetFile) to `ert.tlc`.
- Set solver parameter **Type** (SolverType) to `Fixed-step`.
- Set **Clock resolution** (ClockResolution) to a scalar value of type `double`, which represents the clock resolution in seconds.

If the model includes periodic functions, the clock resolution that you specify must divide the sample times used by the periodic functions with no remainder. For example, if a model includes periodic functions that use sample times `1`, `0.2`, and `0.1`, the `0.05` is a valid clock resolution setting but `0.03` is not valid. If the value that you specify for the clock resolution is not a divisor, you must change the clock resolution setting or the offending sample time.

You can specify a clock resolution for components and subcomponents in a model hierarchy. The clock resolution settings for models across the hierarchy must match. For models configured to use a service interface, the code generator checks for clock resolution consistency across entry-point functions generated from the same model function used across the model hierarchy. If the clock resolution settings are not consistent, the code generator returns an error.

How Code Generator Uses Specified Clock Resolution

How the code generator uses a specified clock resolution depends on the model code interface configuration. For models configured to use a data code interface, a specified clock resolution influences how the code generator represents the clock tick in the generated code. The clock tick is

the increment of a processor clock. For models that you configure with a service code interface, a specified clock resolution influences timer service interface calls in the code.

For a model configured to use a service code interface, the code generator checks the value of the related **Application lifespan (days)** parameter. Timer service interfaces support 32-bit clock tick values. If the **Application lifespan (days)** parameter setting is large enough to result in a clock tick that cannot be stored as a 32-bit value, then an overflow condition results and the code generator returns an error.

For aperiodic functions that are driven by an S-function that specifies the `SS_OPTION_ASYNCHRONOUS` option and a clock resolution, the clock resolution that the S-function specifies overrides the setting of the **Clock resolution** parameter.

For a model that uses a service code interface, when you specify a clock resolution, the code generator facilitates implementation of the timer service functions that run in the target environment and are called by the generated entry-point functions by including the clock resolution in two places:

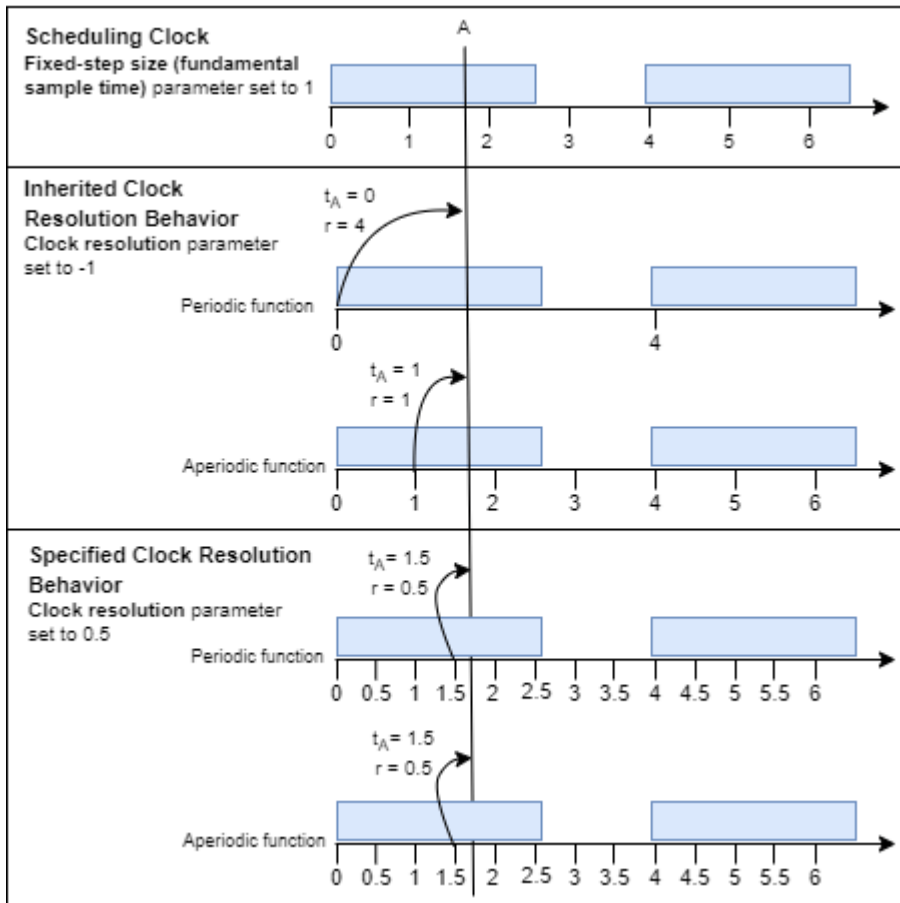
- In the Timer Service Interface section of the Code Interface Report. See “Analyze Generated Service Code Interface Report” (Embedded Coder).
- As metadata in the generated code descriptor file `codedescriptor.dmr`. You can access the metadata by using the code descriptor programming interface. See “Get Metadata About Service Interface” (Embedded Coder).

If you choose not to specify a clock resolution, the code generator uses these default values for the clock resolution:

- Model fixed-step size (fundamental sample time), for aperiodic functions
- Function sample time, for periodic functions

Suppose a model includes an aperiodic function and a periodic function, both of which use a block that requests absolute time from a target environment clock. The fixed-step size (fundamental sample time) for the model is 1 and the sample time configured for the periodic function is 4. The model is configured to use a service code interface that applies the during-execution data communication method.

This figure shows the functions requesting the absolute time value at time A. When using the inherited clock resolution behavior (with **Clock resolution** set to -1), you can get time values that are more accurate by using aperiodic functions rather than periodic functions. You can get time values that are even more accurate by specifying a clock resolution as a scalar value.



Improve Accuracy of Time Values Read by Generated Entry-Point Functions

This example shows that for a model that is configured to use a service code interface, you can improve the accuracy of the time values read by generated entry-point functions by configuring the model with a specific clock resolution. The aperiodic function `Integrator` in example model `ComponentDeploymentFcn` includes the `Discrete Time Integrator` block, which requests the clock tick. Model configuration parameter **Fixed-step size (fundamental sample time)** is set to `auto`.

Generate and inspect the code for the model with the **Clock resolution** parameter set to the default value `-1`. In this case, the code generator initializes the clock resolution based on scheduling properties for the model style and the type of entry-point function.


Open example model `ComponentDeploymentFcn`.

```
open_system("ComponentDeploymentFcn.slx");
```

Use the default clock resolution to generate code, then inspect the code.

1. Open the Embedded Coder app.

2. Configure the `Integrator` function to use a timer service interface that is configured for during-execution communication. In the Code Mappings Editor - C (Embedded Coder), on the **Functions**

tab, select the row for function `Exported Function:Aperiodic`. Click the pencil button . In the dialog box that appears, set the **Timer Service** property to `get_tick_during`. Save the model.

3. In the toolstrip, in the **Modeling** tab, click **Modeling Settings**.
4. In the Configuration Parameters dialog box, search for and select the **Clock resolution** parameter. The parameter is set to the default value -1. Close the dialog box.
5. Generate code.
6. In the code generation report, select the Code Interface Report and navigate to the Timer Service Interfaces section. The report shows that the clock resolution is set to 1 second.
7. In the generated file `ComponentDeploymentFcn.c`, inspect the code for entry-point function `cd_integrator`. The code generator uses the clock resolution to calculate the integer value in the assignment statement for variable `tmp`. The value 1.25 is the product of the clock resolution (1) and the gain value specified for the Discrete Time Integrator block (1.25) and is the frequency at which the function requests the elapsed time.

```
tmp = 1.25 * (real_T)Integrator_ELAPS_T;
```

Configure the model with a clock resolution of 0.5. Then, generate and reinspect the code.

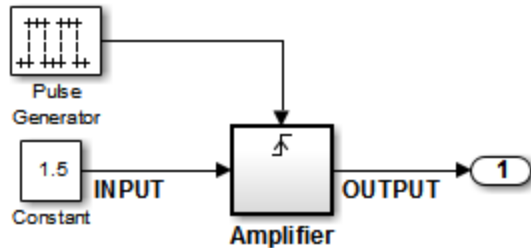
1. In the toolstrip, click **Settings**.
2. In the Configuration Parameters dialog box, search for and select the **Clock resolution** parameter. Set the parameter to 0.5. Apply the configuration change and close the dialog box.
3. Generate code.
4. In the code generation report, open the Code Interface Report and navigate to the Timer Service Interfaces section. The report shows that the clock resolution is set to 0.5 seconds.
5. In the generated file `ComponentDeploymentFcn.c`, reinspect the code for entry-point function `cd_integrator`. In this case, the code generator uses the specified clock resolution 0.5 to calculate the integer value in the assignment statement for variable `tmp`. The value 0.625 is the product of the clock resolution (0.5) and the gain value specified for the Discrete Time Integrator block (1.25). The generated code calls the timer service more frequently, which results in the function reading time values that are more accurate.

```
tmp = 0.625 * (real_T)Integrator_ELAPS_T;
```

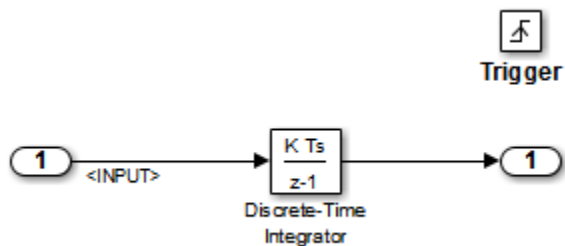
For more information about configuring timer service interfaces for models configured to use a service interface configuration, see “Timer Service Interfaces” (Embedded Coder). For information about configuring a model that uses a data code interface, see “Data Interfaces” (Embedded Coder).

Generate Code for an Elapsed Time Counter

This example shows a model that includes a triggered subsystem, `Amplifier`, consisting of a Discrete-Time Integrator block that uses an elapsed time counter. The model `ex_elapseTime` is in the folder `matlab/help/toolbox/rtw/examples`.



ex_elapseTime Model



Amplifier Subsystem

Code in the generated header file `ex_elapseTime.h` for the model uses 64 bits to implement the timer for the base rate (`clockTick0` and `clockTickH0`).

```

/*
 * Timing:
 * The following substructure contains information regarding
 * the timing information for the model.
 */
struct {
    time_T taskTime0;
    uint32_T clockTick0;
    uint32_T clockTickH0;
    time_T stepSize0;
    time_T tFinal;
    boolean_T stopRequestedFlag;
} Timing;

```

The code generator allocates storage for the previous-time value and elapsed-time value of the `Amplifier` subsystem (`Amplifier_PREV_T`) in the `D_Work(states)` structure in `ex_elapsedTime.h`.

```

/* Block states (auto storage) for system '<Root>' */
typedef struct {
    real_T DiscreteTimeIntegrator_DSTATE;

```

```

    /* '<S1>/Discrete-Time Integrator' */
    int32_T clockTickCounter;
    /* '<Root>/Pulse Generator' */
    uint32_T Amplifier_ELAPS_T[2];
    /* '<Root>/Amplifier' */
    uint32_T Amplifier_PREV_T[2];
    /* '<Root>/Amplifier' */
    } DW_ex_elapseTime_T;

```

The elapsed time computation is performed as follows within the `ex_elapseTime_step` function:

```

/* ---
   Outputs for Triggered SubSystem:
   '<Root>/Amplifier' incorporates:
   TriggerPort: '<S1>/Trigger'
--- */
zcEvent = rt_ZCFcn(RISING_ZERO_CROSSING,
    &ex_elapseTime_PrevZCX.Amplifier_Trig_ZCE,
    ((real_T)rtb_PulseGenerator));
if (zcEvent != NO_ZCEVENT) {
    elapseT_H = ex_elapseTime_M->Timing.clockTickH0 -
        ex_elapseTime_DW.Amplifier_PREV_T[1];
    if (ex_elapseTime_DW.Amplifier_PREV_T[0] >
        ex_elapseTime_M->Timing.clockTick0) {
        elapseT_H--;
    }

    ex_elapseTime_DW.Amplifier_ELAPS_T[0] =
        ex_elapseTime_M->Timing.clockTick0
        - ex_elapseTime_DW.Amplifier_PREV_T[0];
    ex_elapseTime_DW.Amplifier_PREV_T[0] =
        ex_elapseTime_M->Timing.clockTick0;
    ex_elapseTime_DW.Amplifier_ELAPS_T[1] =
        elapseT_H;
    ex_elapseTime_DW.Amplifier_PREV_T[1] =
        ex_elapseTime_M->Timing.clockTickH0;

```

As shown above, the elapsed time is maintained as a state of the triggered subsystem. The Discrete-Time Integrator block finally performs its output and update computations using the elapsed time.

```

/* ---
DiscreteIntegrator: '<S1>/Discrete-Time Integrator'
--- */
OUTPUT = ex_elapseTime_DW.DiscreteTimeIntegrator_DSTATE;

/* ---
Update for DiscreteIntegrator:
   '<S1>/Discrete-Time Integrator' incorporates:
   Constant: '<Root>/Constant'
--- */
ex_elapseTime_DW.DiscreteTimeIntegrator_DSTATE += 0.3 * (real_T)
    ex_elapseTime_DW.Amplifier_ELAPS_T[0] * 1.5;

```

See Also

More About

- “Absolute and Elapsed Time Computation” on page 4-2
- “Absolute Time Limitations” on page 4-14

Absolute Time Limitations

Absolute time is the time that has elapsed from the beginning of program execution to the present time, as distinct from *elapsed time*, the interval between two events. See “Absolute and Elapsed Time Computation” on page 4-2 for more information.

When you design an application that is intended to run indefinitely, you must take care when logging time values, or using charts or blocks that depend on absolute time. If the value of time reaches the largest value that can be represented by the data type used by the timer to store time, the timer overflows and the logged time or block output is incorrect.

If your target uses `rtModel`, you can avoid timer overflow by specifying a value for model configuration parameter **Application lifespan**. See “Integer Timers in Generated Code” on page 4-3 for more information.

The following limitations apply to absolute time:

- If you log time values by selecting model configuration parameter **Time**, your model uses absolute time.
- Every Stateflow chart that uses time is dependent on absolute time. The only way to eliminate the dependency is to change the Stateflow chart to not use time.
- The following Simulink blocks depend on absolute time:
 - Backlash
 - Chirp Signal
 - Clock
 - Derivative
 - Digital Clock
 - Discrete-Time Integrator (only when used in triggered subsystems)
 - From File
 - From Workspace
 - Pulse Generator
 - Ramp
 - Rate Limiter
 - Repeating Sequence
 - Signal Generator
 - Sine Wave (only when the **Sine type** parameter is set to Time-based)
 - Step
 - To File
 - To Workspace (only when logging to `StructureWithTime` format)
 - Transport Delay
 - Variable Transport Delay

In addition to the Simulink blocks above, blocks in other blocksets may depend on absolute time. See the documentation for the blocksets that you use.

See Also

More About

- “Absolute and Elapsed Time Computation” on page 4-2

Time-Based Scheduling in Simulink Coder

Time-Based Scheduling and Code Generation

In this section...

“Sample Time Considerations” on page 5-2

“Tasking Modes” on page 5-2

“Model Execution and Rate Transitions” on page 5-3

“Execution During Simulink Model Simulation” on page 5-4

“Model Execution in Real Time” on page 5-4

“Single-Tasking Versus Multitasking Operation” on page 5-4

Sample Time Considerations

Simulink models run at one or more sample times. The Simulink product provides considerable flexibility in building multirate systems, that is, systems with more than one sample time. However, this same flexibility also allows you to construct models for which the code generator cannot generate real-time code for execution in a multitasking environment. To make multirate models operate as expected in real time (that is, to give the right answers), you sometimes must modify your model or instruct the Simulink engine to modify the model for you. In general, the modifications involve placing Rate Transition blocks between blocks that have unequal sample times. The following sections discuss issues you must address to use a multirate model in a multitasking environment. For a comprehensive discussion of sample times, including rate transitions, see “What Is Sample Time?”, “Sample Times in Subsystems”, “Sample Times in Systems”, “Resolve Rate Transitions”, and associated topics.

Tasking Modes

There are two execution modes for a fixed-step model: single-tasking and multitasking. These modes are available only for fixed-step solvers. To select an execution mode, select model configuration parameter **Treat each discrete rate as a separate task**. When you select this parameter, multitasking execution is applied for a multirate model. When you clear this parameter, single-tasking execution is applied.

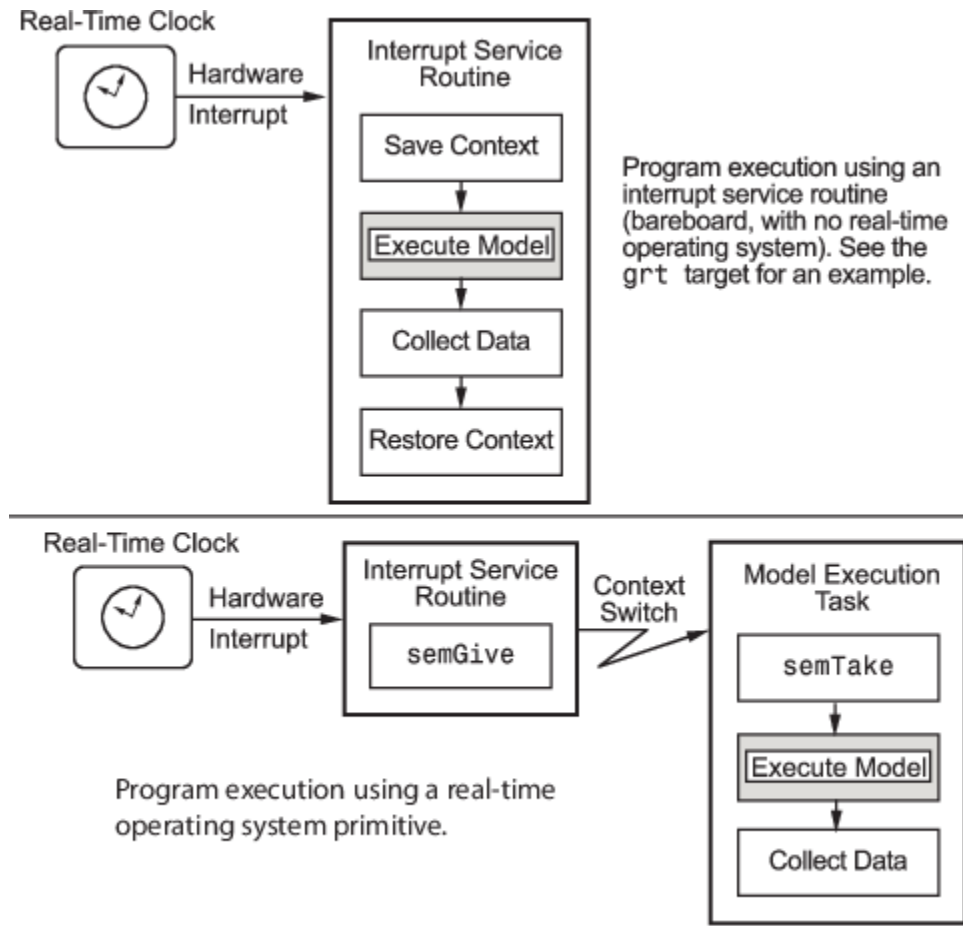
Note A model that is multirate and uses multitasking cannot reference a multirate model that uses single-tasking.

Execution of models in a real-time system can be done with the aid of a real-time operating system, or it can be done on *bare-metal* target hardware, where the model runs in the context of an interrupt service routine (ISR).

The fact that a system (such as The Open Group UNIX® or Microsoft® Windows systems) is multitasking does not imply that your program can execute in real time. This is because the program might not preempt other processes when required.

In operating systems (such as PC-DOS) where only one process can exist at a given time, an interrupt service routine (ISR) must perform the steps of saving the processor context, executing the model code, collecting data, and restoring the processor context.

Other operating systems, such as POSIX-compliant ones, provide automatic context switching and task scheduling. This simplifies the operations performed by the ISR. In this case, the ISR simply enables the model execution task, which is normally blocked. The next figure illustrates this difference.



Model Execution and Rate Transitions

To generate code that executes as expected in real time, you (or the Simulink engine) might need to identify and handle sample rate transitions within the model. In multitasking mode, by default the Simulink engine flags errors during simulation if the model contains invalid rate transitions. You can use the model configuration parameter **Multitask data transfer** to alter this behavior. Parameter **Single task data transfer** is available for the same purpose for single-tasking mode.

To avoid raising rate transition errors, insert Rate Transition blocks between tasks. You can request that the Simulink engine handle rate transitions automatically by inserting hidden Rate Transition blocks. See “Automatic Rate Transition” on page 5-19 for an explanation of this option.

To understand such problems, first consider how Simulink simulations differ from real-time programs.

Execution During Simulink Model Simulation

Before the Simulink engine simulates a model, it orders the blocks based upon their topological dependencies. This includes expanding virtual subsystems into the individual blocks they contain and flattening the entire model into a single list. Once this step is complete, each block is executed in order.

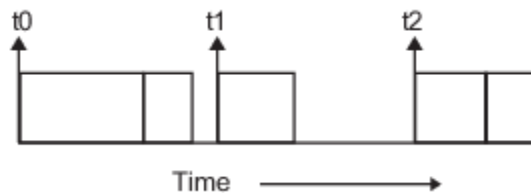
The key to this process is the ordering of blocks. A block whose output is directly dependent on its input (that is, a block with direct feedthrough) cannot execute until the block driving its input executes.

Some blocks set their outputs based on values acquired in a previous time step or from initial conditions specified as a block parameter. The output of such a block is determined by a value stored in memory, which can be updated independently of its input. During simulation, computations are performed prior to advancing the variable corresponding to time. This results in computations occurring instantaneously (that is, no computational delay).

Model Execution in Real Time

A real-time program differs from a Simulink simulation in that the program must execute the model code synchronously with real time. Every calculation results in some computational delay. This means the sample intervals cannot be shortened or lengthened (as they can be in a Simulink simulation), which leads to less efficient execution.

Consider the following timing figure.



Note the processing inefficiency in the sample interval t_1 . That interval cannot be compressed to increase execution speed because, by definition, sample times are clocked in real time.

You can circumvent this potential inefficiency by using the multitasking mode. The multitasking mode defines tasks with different priorities to execute parts of the model code that have different sample rates.

See “Multitasking and Pseudomultitasking Modes” on page 5-9 for a description of how this works. It is important to understand that section before proceeding here.

Single-Tasking Versus Multitasking Operation

Single-tasking programs require longer sample intervals, because all computations must be executed within each clock period. This can result in inefficient use of available CPU time, as shown in the previous figure.

Multitasking mode can improve the efficiency of your program if the model is large and has many blocks executing at each rate.

However, if your model is dominated by a single rate, and only a few blocks execute at a slower rate, multitasking can actually degrade performance. In such a model, the overhead incurred in task switching can be greater than the time required to execute the slower blocks. In this case, it is more efficient to execute all blocks at the dominant rate.

If you have a model that can benefit from multitasking execution, you might need to modify your model by adding Rate Transition blocks (or instruct the Simulink engine to do so) to generate expected results.

For more information about the two modes of execution and examples, see “Modeling for Single-Tasking Execution” on page 5-6 and “Modeling for Multitasking Execution” on page 5-9.

See Also

More About

- “What Is Sample Time?”
- “Sample Times in Subsystems”
- “Sample Times in Systems”
- “Configure Time-Based Scheduling” on page 5-33
- “Sample Times in Subsystems”
- “Sample Times in Systems”
- “Resolve Rate Transitions”
- “Handle Rate Transitions” on page 5-16
- “Modeling for Single-Tasking Execution” on page 5-6
- “Modeling for Multitasking Execution” on page 5-9
- “Time-Based Scheduling Example Models” on page 5-35

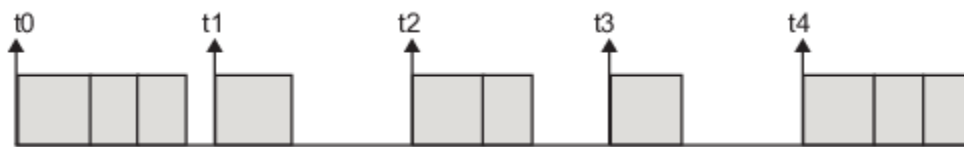
Modeling for Single-Tasking Execution

Single-Tasking Mode

You can execute model code in a strictly single-tasking manner. While this mode is less efficient with regard to execution speed, in certain situations, it can simplify your model.

In single-tasking mode, the base sample rate must define a time interval that is long enough to allow the execution of all blocks within that interval.

The next figure illustrates the inefficiency inherent in single-tasking execution.



Single-tasking system execution requires a base sample rate that is long enough to execute one step through the entire model.

Build a Program for Single-Tasking Execution

To use single-tasking execution, clear model configuration parameter **Treat each discrete rate as a separate task**. If you select the parameter, single-tasking mode is used in the following cases:

- If your model contains one sample time
- If your model contains a continuous and a discrete sample time and the fixed step size is equal to the discrete sample time

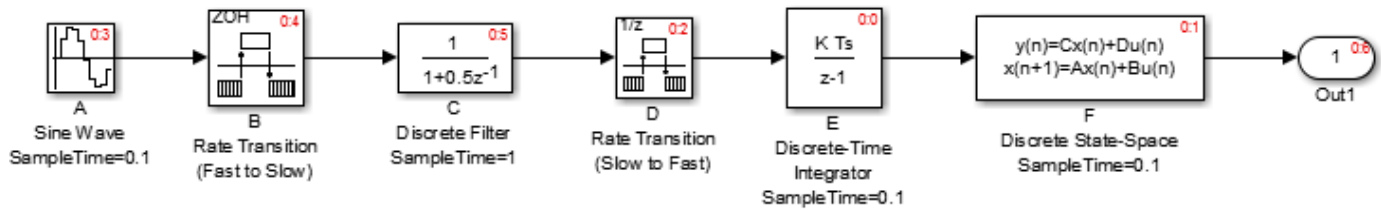
Single-Tasking Execution

This example examines how a simple multirate model executes in both real time and simulation, using a fixed-step solver. It considers operation in both single-tasking and multitasking modes, as determined by setting model configuration parameter **Treat each discrete rate as a separate task**.

The example model is shown in the next figure. The discussion refers to the six blocks of the model as A through F, as labeled in the block diagram.

The execution order of the blocks (indicated in the upper right of each block) has been forced into the order shown by assigning higher priorities to blocks F, E, and D. The ordering shown is one possible valid execution ordering for this model. For more information, see “Simulation Phases in Dynamic Systems”.

The execution order is determined by data dependencies between blocks. In a real-time system, the execution order determines the order in which blocks execute within a given time interval or task. This discussion treats the model's execution order as a given, because it is concerned with the allocation of block computations to tasks, and the scheduling of task execution.



Note The discussion and timing diagrams in this section are based on the assumption that the Rate Transition blocks are used in the default (protected) mode, with block parameters **Ensure data integrity during data transfer** and **Ensure deterministic data transfer (maximum delay)** selected

This example considers the execution of the above model when model configuration parameter **Treat each discrete rate as a separate task** is cleared, which indicates the single-tasking mode.

In a single-tasking system, if you select model configuration parameter **Block reduction**, fast-to-slow Rate Transition blocks are optimized out of the model. The default case is shown (parameter **Block reduction** selected), so block B does not appear in the timing diagrams in this section. For more information, see “Block reduction”.

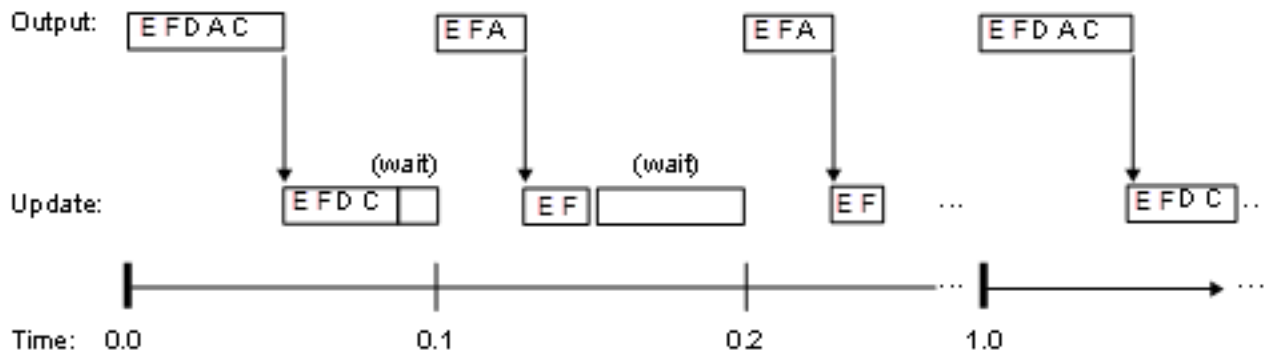
The following table shows, for each block in the model, the execution order, sample time, and whether the block has an output or update computation. Block A does not have discrete states, and accordingly does not have an update computation.

Execution Order and Sample Times (Single-Tasking)

Blocks (in Execution Order)	Sample Time (in Seconds)	Output	Update
E	0.1	Y	Y
F	0.1	Y	Y
D	1	Y	Y
A	0.1	Y	N
C	1	Y	Y

Real-Time Single-Tasking Execution

The next figure shows the scheduling of computations when the generated code is deployed in a real-time system. The generated program is shown running in real time, under control of interrupts from a 10 Hz timer.



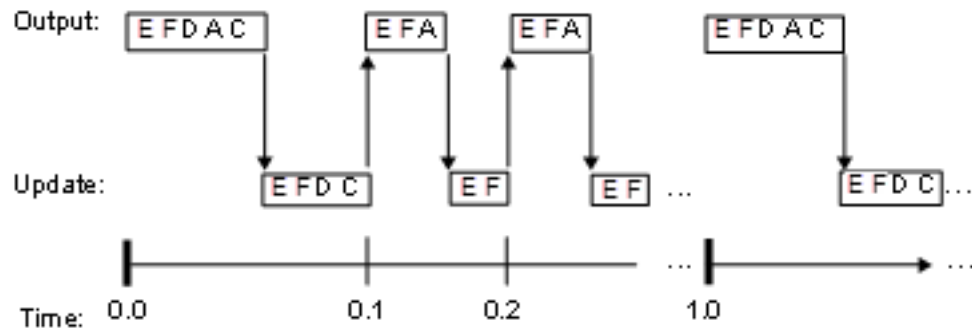
At time 0.0, 1.0, and every second thereafter, both the slow and fast blocks execute their output computations; this is followed by update computations for blocks that have states. Within a given time interval, output and update computations are sequenced in block execution order.

The fast blocks execute on every tick, at intervals of 0.1 second. Output computations are followed by update computations.

The system spends some portion of each time interval (labeled “wait”) idling. During the intervals when only the fast blocks execute, a larger portion of the interval is spent idling. This illustrates an inherent inefficiency of single-tasking mode.

Simulated Single-Tasking Execution

The next figure shows the execution of the model during the Simulink simulation loop.



Because time is simulated, the placement of ticks represents the iterations of the simulation loop. Blocks execute in exactly the same order as in the previous figure, but without the constraint of a real-time clock. Therefore there is no idle time between simulated sample periods.

See Also

More About

- “Time-Based Scheduling and Code Generation” on page 5-2
- “Configure Time-Based Scheduling” on page 5-33
- “Time-Based Scheduling Example Models” on page 5-35

Modeling for Multitasking Execution

Multitasking and Pseudomultitasking Modes

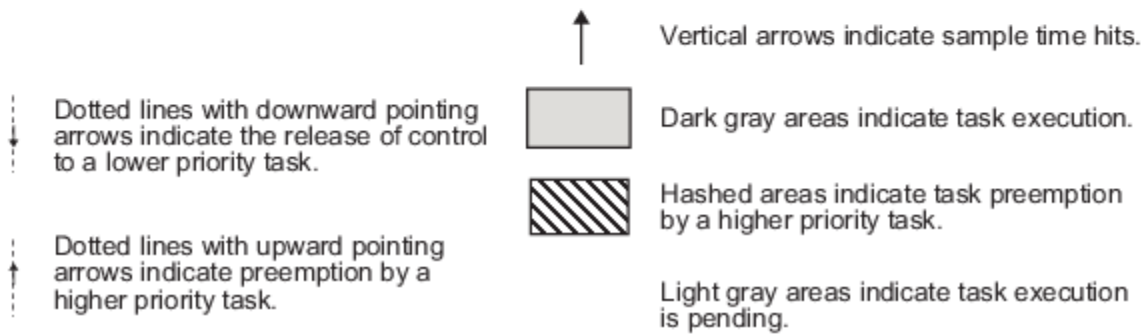
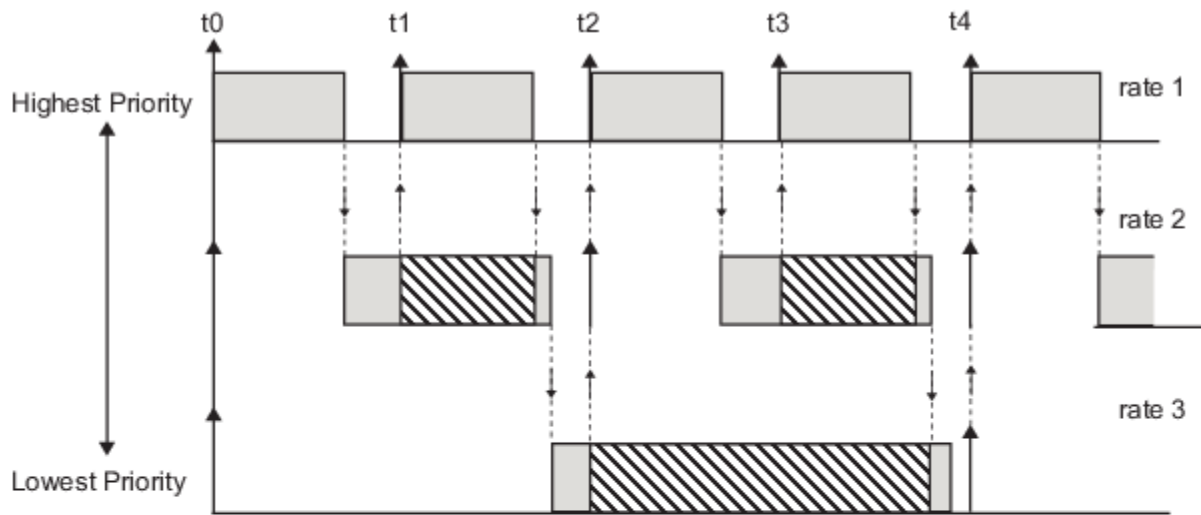
When periodic tasks execute in a multitasking mode, by default the blocks with the fastest sample rates are executed by the task with the highest priority, the next fastest blocks are executed by a task with the next higher priority, and so on. Time available in between the processing of high-priority tasks is used for processing lower priority tasks. This results in efficient program execution.

Where tasks are asynchronous rather than periodic, there may not necessarily be a relationship between sample rates and task priorities; the task with the highest priority need not have the fastest sample rate. You specify asynchronous task priorities using Async Interrupt and Task Sync blocks. You can switch the sense of what priority numbers mean by selecting or clearing model configuration parameter **Higher priority value indicates higher task priority**.

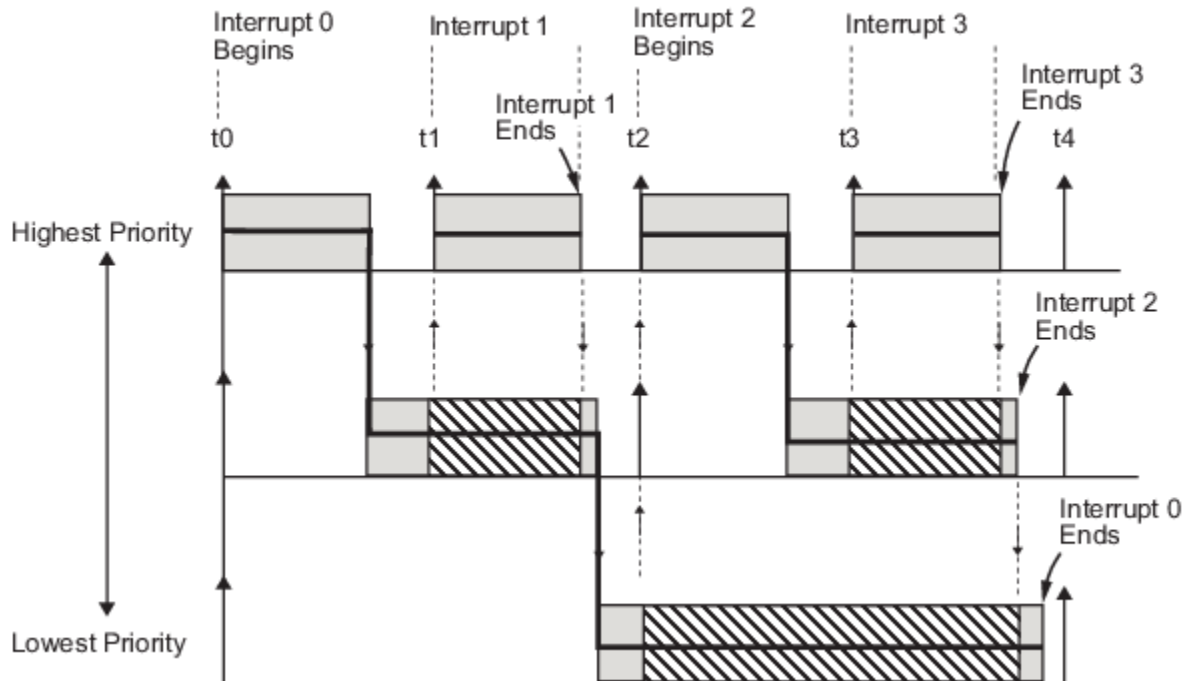
In multitasking environments (that is, under a real-time operating system), you can define separate tasks and assign them priorities. For bare-metal target hardware (that is, no real-time operating system present), you cannot create separate tasks. However, generated application modules implement what is effectively a multitasking execution scheme using overlapped interrupts, accompanied by programmatic context switching.

This means an interrupt can occur while another interrupt is currently in progress. When this happens, the current interrupt is preempted, the floating-point unit (FPU) context is saved, and the higher priority interrupt executes its higher priority (that is, faster sample rate) code. Once complete, control is returned to the preempted ISR.

The next figures illustrate how timing of tasks in multirate systems are handled by the code generator in multitasking, pseudomultitasking, and single-tasking environments.



The next figure shows how overlapped interrupts are used to implement pseudomultitasking. In this case, Interrupt 0 does not return until after Interrupts 1, 2, and 3.



Build a Program for Multitasking Execution

To use multitasking execution, select model configuration parameter **Treat each discrete rate as a separate task**. This parameter is active only if you set **Solver type** to **Fixed-step**. Auto mode results in a multitasking environment if your model has two or more different sample times. A model with a continuous and a discrete sample time runs in single-tasking mode if the fixed-step size is equal to the discrete sample time.

Execute Multitasking Models

In cases where the continuous part of a model executes at a rate that is different from the discrete part, or a model has blocks with different sample rates, the Simulink engine assigns each block a *task identifier* (*tid*) to associate the block with the task that executes at the block's sample rate.

You set sample rates and their constraints on the **Solver** pane of the Model Configuration Parameters dialog box. To generate code, select **Fixed-step** for the solver type. Certain restrictions apply to the sample rates that you can use:

- The sample rate of a block must be an integer multiple of the base (that is, the fastest) sample period.
- When you set model configuration parameter **Periodic sample time constraint** to **unconstrained**, the base sample period is determined by the setting of parameter **Fixed step size**.
- When you set parameter **Periodic sample time constraint** to **Specified**, the base rate fixed-step size is the first element of the sample time matrix that you specify for parameter **Sample time properties**. The **Solver** pane from the example model `rtwdemo_mrmtbb` shows an example.

Simulation time

Start time: Stop time:

Solver selection

Type: Solver:

▼ Solver details

Fixed-step size (fundamental sample time):

Tasking and sample time options

Periodic sample time constraint:

Sample time properties:

Treat each discrete rate as a separate task

Automatically handle rate transition for data transfer

Higher priority value indicates higher task priority

- Continuous blocks execute by using an integration algorithm that runs at the base sample rate. The base sample period is the greatest common denominator of rates in the model only when you set model configuration parameter **Periodic sample time constraint** to Unconstrained and parameter **Fixed step size** to Auto.
- The continuous and discrete parts of the model can execute at different rates only if the discrete part is executed at the same or a slower rate than the continuous part and is an integer multiple of the base sample rate.

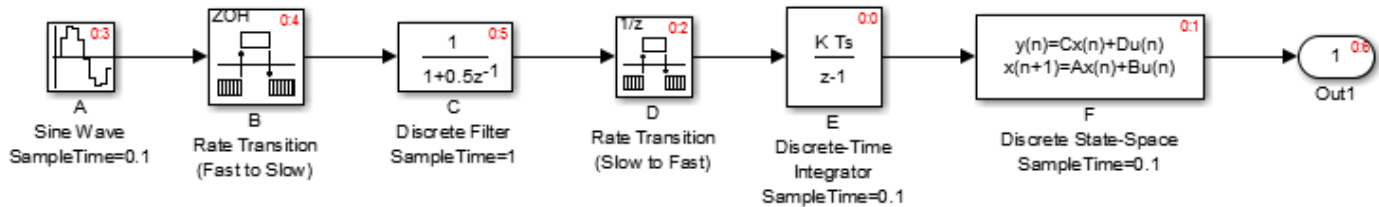
Multitasking Execution

This example examines how a simple multirate model executes in both real time and simulation, using a fixed-step solver. It considers operation in both single-tasking and multitasking modes, as determined by setting model configuration parameter **Treat each discrete rate as a separate task**.

The example model is shown in the next figure. The discussion refers to the six blocks of the model as A through F, as labeled in the block diagram.

The execution order of the blocks (indicated in the upper right of each block) has been forced into the order shown by assigning higher priorities to blocks F, E, and D. The ordering shown is one possible valid execution ordering for this model. For more information, see “Simulation Phases in Dynamic Systems”.

The execution order is determined by data dependencies between blocks. In a real-time system, the execution order determines the order in which blocks execute within a given time interval or task. This discussion treats the model's execution order as a given, because it is concerned with the allocation of block computations to tasks, and the scheduling of task execution.



Note The discussion and timing diagrams in this section are based on the assumption that the Rate Transition blocks are used in the default (protected) mode, with block parameters **Ensure data integrity during data transfer** and **Ensure deterministic data transfer (maximum delay)** selected

This example considers the execution of the preceding model when model configuration parameter **Tasking mode** is set to **MultiTasking**. Block computations are executed under two tasks, prioritized by rate:

- The slower task, which gets the lower priority, is scheduled to run every second. This is called the *1 second task*.
- The faster task, which gets higher priority, is scheduled to run 10 times per second. This is called the *0.1 second task*. The 0.1 second task can preempt the 1 second task.

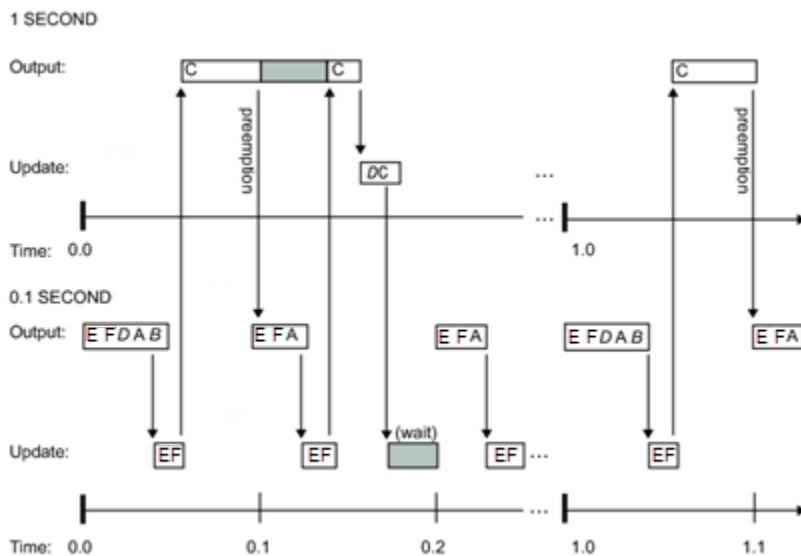
The following table shows, for each block in the model, the execution order, the task under which the block runs, and whether the block has an output or update computation. Blocks A and B do not have discrete states, and accordingly do not have an update computation.

Task Allocation of Blocks in Multitasking Execution

Blocks (in Execution Order)	Task	Output	Update
E	0.1 second task	Y	Y
F	0.1 second task	Y	Y
D	The Rate Transition block uses port-based sample times. Output runs at the output port sample time under 0.1 second task. Update runs at input port sample time under 1 second task. For more information on port-based sample times, see "Referenced Model Sample Times".	Y	Y
A	0.1 second task	Y	N
B	The Rate Transition block uses port-based sample times. Output runs at the output port sample time under 0.1 second task. For more information on port-based sample times, see "Referenced Model Sample Times".	Y	N
C	1 second task	Y	Y

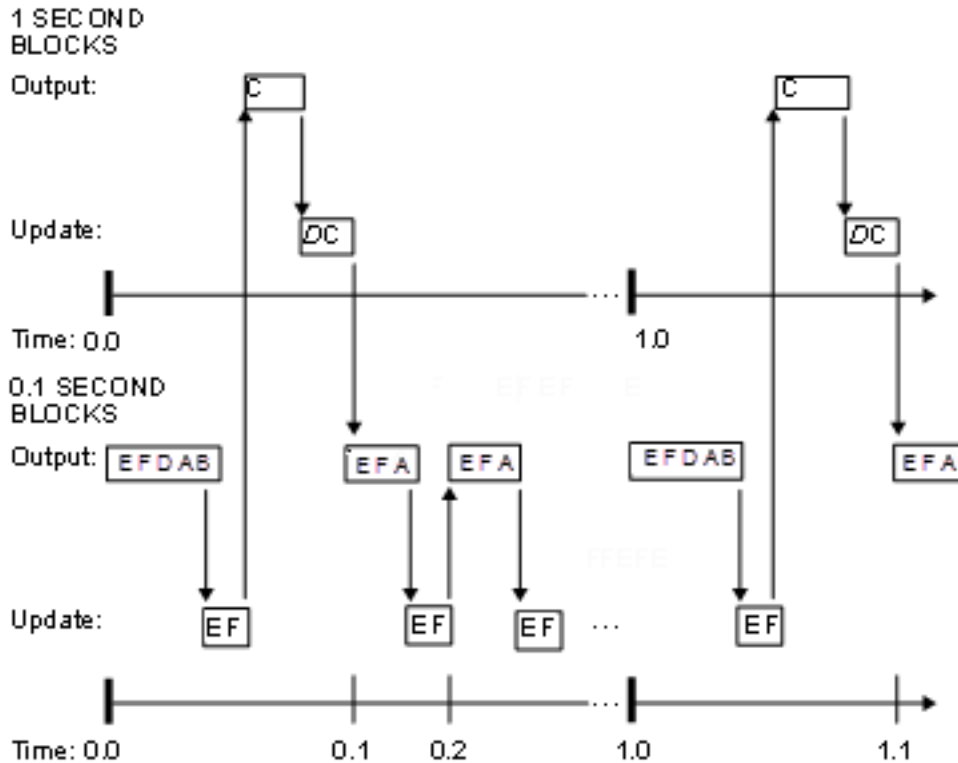
Real-Time Multitasking Execution

The next figure shows the scheduling of computations in MultiTasking solver mode when the generated code is deployed in a real-time system. The generated program is shown running in real time, as two tasks under control of interrupts from a 10 Hz timer.



Simulated Multitasking Execution

The next figure shows the Simulink execution of the same model, in MultiTasking solver mode. In this case, the Simulink engine runs the blocks in one thread of execution, simulating multitasking. No preemption occurs.



See Also

More About

- "Time-Based Scheduling and Code Generation" on page 5-2
- "Sample Times in Subsystems"
- "Sample Times in Systems"
- "Configure Time-Based Scheduling" on page 5-33
- "Resolve Rate Transitions"
- "Handle Rate Transitions" on page 5-16
- "Time-Based Scheduling Example Models" on page 5-35

Handle Rate Transitions

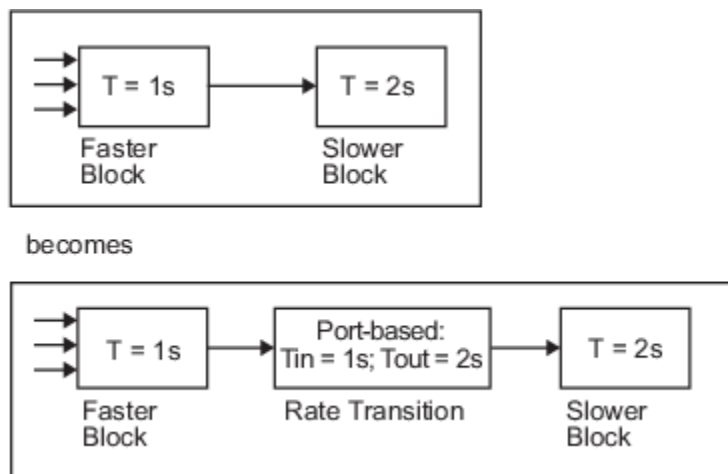
Rate Transitions

Two periodic sample rate transitions can exist within a model:

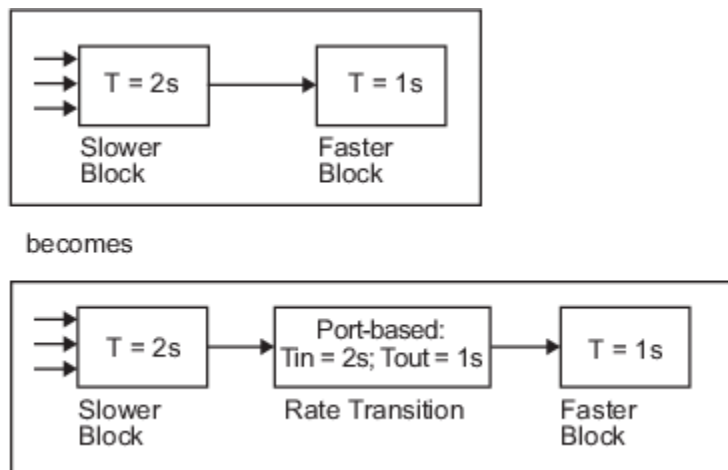
- A faster block driving a slower block
- A slower block driving a faster block

The following sections concern models with periodic sample times with zero offset only. Other considerations apply to multirate models that involve asynchronous tasks. For details on how to generate code for asynchronous multitasking, see “Asynchronous Support” on page 6-2.

In multitasking and pseudomultitasking systems, differing sample rates can cause blocks to be executed in the wrong order. To prevent possible errors in calculated data, you must control model execution at these transitions. When connecting faster and slower blocks, you or the Simulink engine must add Rate Transition blocks between them. Fast-to-slow transitions are illustrated in the next figure.



Slow-to-fast transitions are illustrated in the next figure.



Note Although the Rate Transition block offers a superset of the capabilities of the Unit Delay block (for slow-to-fast transitions) and the Zero-Order Hold block (for fast-to-slow transitions), you should use the Rate Transition block instead of these blocks.

Data Transfer Problems

Rate Transition blocks deal with issues of data integrity and determinism associated with data transfer between blocks running at different rates.

- *Data integrity*: A problem of data integrity exists when the input to a block changes during the execution of that block. Data integrity problems can be caused by preemption.

Consider the following scenario:

- A faster block supplies the input to a slower block.
- The slower block reads an input value V_1 from the faster block and begins computations using that value.
- The computations are preempted by another execution of the faster block, which computes a new output value V_2 .
- A data integrity problem now arises: when the slower block resumes execution, it continues its computations, now using the “new” input value V_2 .

Such a data transfer is called *unprotected*. “Faster to Slower Transitions in Real Time” on page 5-22 shows an unprotected data transfer.

In a *protected* data transfer, the output V_1 of the faster block is held until the slower block finishes executing.

- *Deterministic* versus *nondeterministic* data transfer: In a *deterministic* data transfer, the timing of the data transfer is completely predictable, as determined by the sample rates of the blocks.

The timing of a *nondeterministic* data transfer depends on the availability of data, the sample rates of the blocks, and the time at which the receiving block begins to execute relative to the driving block.

You can use the Rate Transition block to protect data transfers in your application and make them deterministic. These characteristics are considered desirable in most applications. However, the Rate Transition block supports flexible options that allow you to compromise data integrity and determinism in favor of lower latency. The next section summarizes these options.

Data Transfer Assumptions

When processing data transfers between tasks, the code generator makes these assumptions:

- Data transitions occur between a single reading task and a single writing task.
- A read or write of a byte-sized variable is atomic.
- When two tasks interact through a data transition, only one of them can preempt the other.
- For periodic tasks, the faster rate task has higher priority than the slower rate task; the faster rate task preempts the slower rate task.
- All tasks run on a single processor. Time slicing is not allowed.

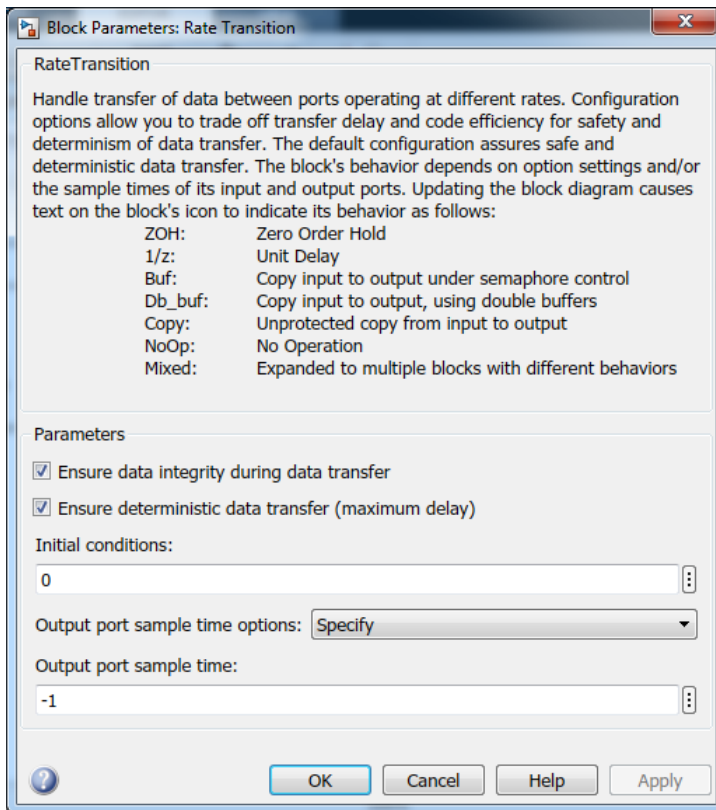
- Processes do not crash or restart (especially while data is transferred between tasks).

Rate Transition Block Options

Several parameters of the Rate Transition block are relevant to its use in code generation for real-time execution, as discussed below. For a complete block description, see Rate Transition.

The Rate Transition block handles periodic (fast to slow and slow to fast) and asynchronous transitions. When inserted between two blocks of differing sample rates, the Rate Transition block automatically configures its input and output sample rates for the type of transition; you do not need to specify whether a transition is slow-to-fast or fast-to-slow (low-to-high or high-to-low priorities for asynchronous tasks).

The critical decision you must make in configuring a Rate Transition block is the choice of data transfer mechanism to be used between the two rates. Your choice is dictated by considerations of safety, memory usage, and performance. As the Rate Transition block parameter dialog box in the next figure shows, the data transfer mechanism is controlled by two options.



- **Ensure data integrity during data transfer:** When you select this parameter, data transferred between rates maintains its integrity (the data transfer is protected). When you clear this parameter, data might not maintain its integrity (the data transfer is unprotected). By default, parameter **Ensure data integrity during data transfer** is selected.
- **Ensure deterministic data transfer (maximum delay):** This parameter is supported for periodic tasks with an offset of zero and fast and slow rates that are multiples of each other. When this parameter is selected, the Rate Transition block behaves like a Zero-Order Hold block (for fast to slow transitions) or a Unit Delay block (for slow to fast transitions). The Rate Transition

block controls the timing of data transfer in a completely predictable way. When this parameter is cleared, the data transfer is nondeterministic. By default, **Ensure deterministic data transfer (maximum delay)** is selected for transitions between periodic rates with an offset of zero; for asynchronous transitions, it cannot be selected.

The Rate Transition block offers three modes of operation with respect to data transfer. In order of level of safety:

- **Protected/Deterministic (default):** This is the safest mode. The drawback of this mode is that it introduces deterministic latency into the system for the case of slow-to-fast periodic rate transitions. For that case, the latency introduced by the Rate Transition block is one sample period of the slower task. For the case of fast-to-slow periodic rate transitions, the Rate Transition block does not introduce additional latency.
- **Protected/NonDeterministic:** In this mode, for slow-to-fast periodic rate transitions, data integrity is protected by double-buffering data transferred between rates. For fast-to-slow periodic rate transitions, a semaphore flag is used. The blocks downstream from the Rate Transition block use the latest available data from the block that drives the Rate Transition block. Maximum latency is less than or equal to one sample period of the faster task.

The drawbacks of this mode are its nondeterministic timing. The advantage of this mode is its low latency.

- **Unprotected/NonDeterministic:** This mode is not recommended for mission-critical applications. The latency of this mode is the same as for Protected/NonDeterministic mode, but memory requirements are reduced since neither double-buffering nor semaphores are required. That is, the Rate Transition block does nothing in this mode other than to pass signals through; it simply exists to notify you that a rate transition exists (and can cause generated code to compute incorrect answers). Selecting this mode, however, generates the least amount of code.

Note In unprotected mode (**Ensure data integrity during data transfer** is selected), the Rate Transition block does nothing other than allow the rate transition to exist in the model.

Rate Transition Blocks and Continuous Time

The sample time at the output port of a Rate Transition block can only be discrete or fixed in minor time step. This means that when a Rate Transition block inherits continuous sample time from its destination block, it treats the inherited sample time as Fixed in Minor Time Step. Therefore, the output function of the Rate Transition block runs only at major time steps. If the destination block sample time is continuous, Rate Transition block output sample time is the base rate sample time (if solver is fixed-step), or zero-order-hold-continuous sample time (if solver is variable-step).

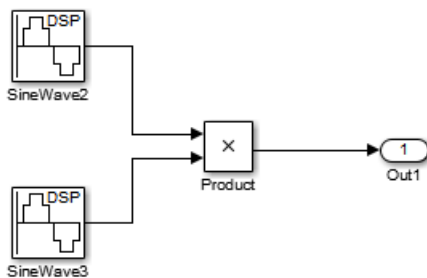
Automatic Rate Transition

The Simulink engine can detect mismatched rate transitions in a multitasking model during an update diagram and automatically insert Rate Transition blocks to handle them. To enable this, select model configuration parameter **Automatically handle rate transition for data transfer**. By default, this parameter is cleared. When you select this parameter:

- Simulink handles transitions between periodic sample times and asynchronous tasks.
- Simulink inserts hidden Rate Transition blocks in the block diagram.

- The code generator produces code for the Rate Transition blocks that were automatically inserted. This code is identical to the code generated for Rate Transition blocks that were inserted manually.
- Automatically inserted Rate Transition blocks operate in protected mode for periodic tasks and asynchronous tasks. You cannot alter this behavior. For periodic tasks, automatically inserted Rate Transition blocks operate with the level of determinism specified by model configuration parameter **Deterministic data transfer**. The default setting is **Whenever possible**, which enables determinism for data transfers between periodic sample-times that are related by an integer multiple. For more information, see Deterministic data transfer. To use other modes, you must insert Rate Transition blocks and set their modes manually.

For example, in this model, SineWave2 has a sample time of 2, and SineWave3 has a sample time of 3.

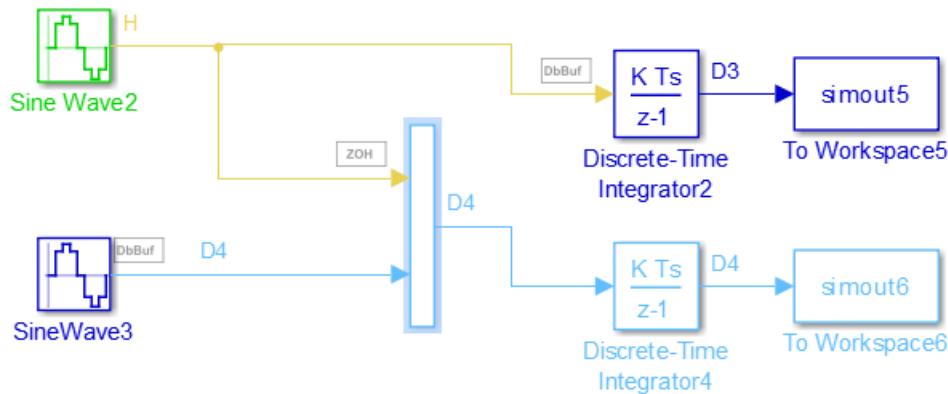


When you select model configuration parameter **Automatically handle rate transition for data transfer**, Simulink inserts a Rate Transition block between each Sine Wave block and the Product block. The inserted blocks have the parameter values to reconcile the Sine Wave block sample times.

If the input port and output port data sample rates in a model are not multiples of each other, Simulink inserts a Rate Transition block whose sample rate is the greatest common divisor (GCD) of the two rates. If no other block in the model contains this new rate, an error occurs during simulation. In this case, you must insert a Rate Transition block manually.

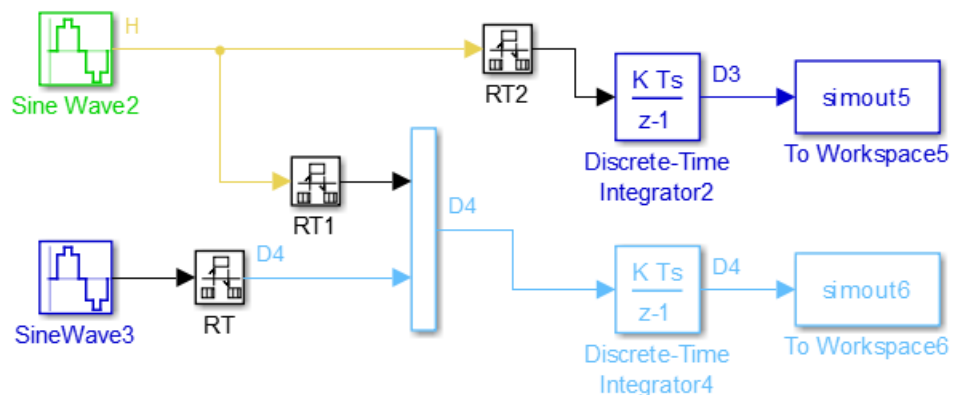
Visualize Inserted Rate Transition Blocks

When you select model configuration parameter **Automatically handle rate transition for data transfer**, Simulink inserts Rate Transition blocks in the paths that have mismatched transition rates. These blocks are hidden by default. To visualize the inserted blocks, update the diagram. Badge labels appear in the model and indicate where Simulink inserted Rate Transition blocks during the compilation phase. For example, in this model, three Rate Transition blocks were inserted between the two Sine Wave blocks and the Multiplexer and Integrator when the model compiled. The ZOH and DbBuf badge labels indicate these blocks.



You can show or hide badge labels. Open the **Debug** tab. In the **Diagnostics** section of the **Information Overlays/Sample Time** gallery select or clear **Automatic Rate Transitions**.

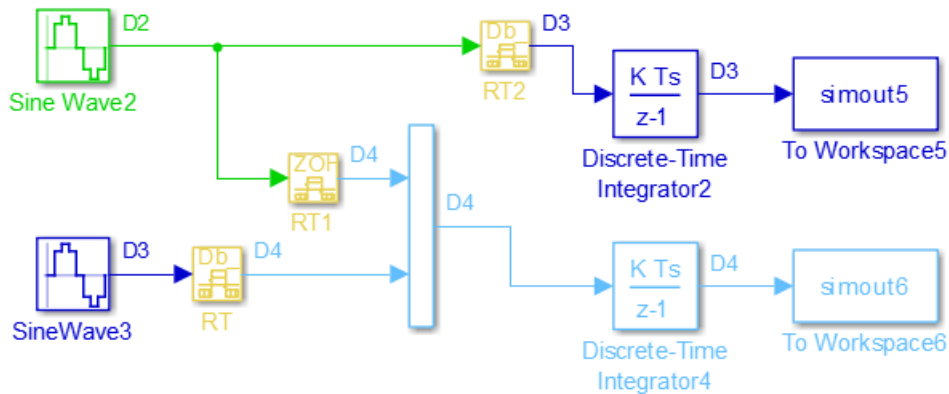
To configure the hidden Rate Transition blocks so that the blocks are visible, right-click a badge label and click **Insert rate transition block**.



When you make hidden Rate Transition blocks visible:

- You can see the type of Rate Transition block inserted and the location of the block in the model.
- You can set the block parameter **Initial Conditions**.
- You can change block parameter settings for rate transfer.

Validate the changes to your model by updating your diagram.



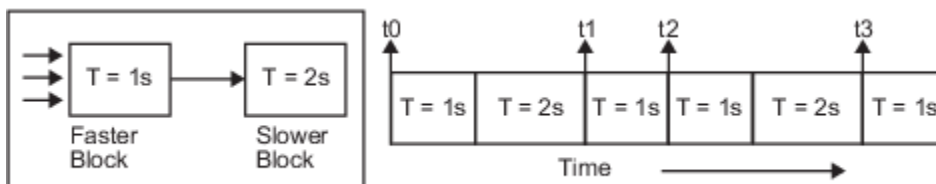
Displaying inserted Rate Transition blocks is not compatible with export-function models. To learn more about the types of Rate Transition blocks, see Rate Transition.

Periodic Sample Rate Transitions

These sections describe cases in which Rate Transition blocks are required for periodic sample rate transitions. The discussion and timing diagrams in these sections are based on the assumption that the Rate Transition block is used in its default (protected/deterministic) mode. Model configuration parameters **Ensure data integrity during data transfer** and **Ensure deterministic data transfer (maximum delay)** are selected. These are the settings used for automatically inserted Rate Transition blocks.

Faster to Slower Transitions in a Simulink Model

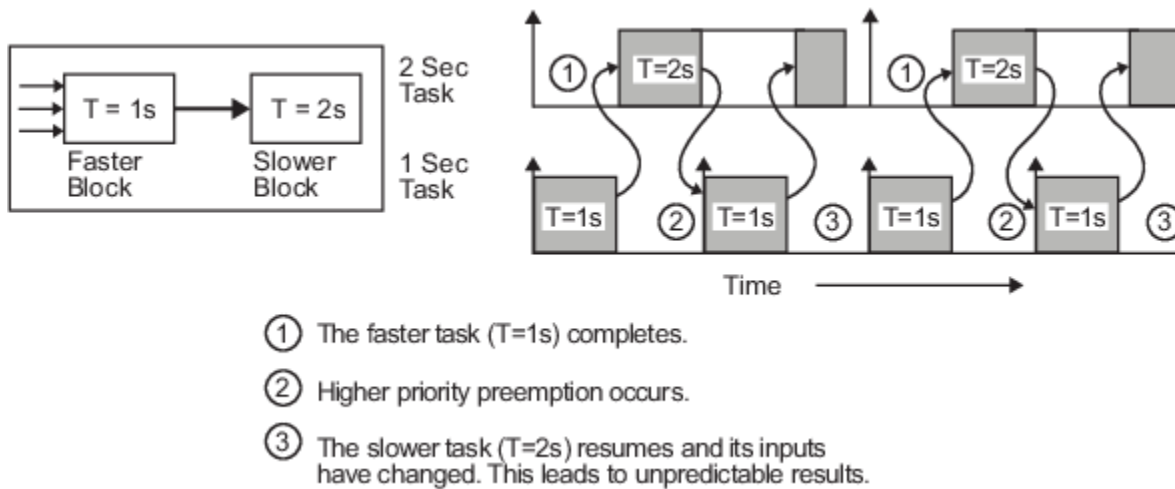
In a model where a faster block drives a slower block having direct feedthrough, the outputs of the faster block are computed first. In simulation intervals where the slower block does not execute, the simulation progresses more rapidly because there are fewer blocks to execute. The next figure illustrates this situation.



A Simulink simulation does not execute in real time, which means that it is not bound by real-time constraints. The simulation waits for, or moves ahead to, whatever tasks are required to complete simulation flow. The actual time interval between sample time steps can vary.

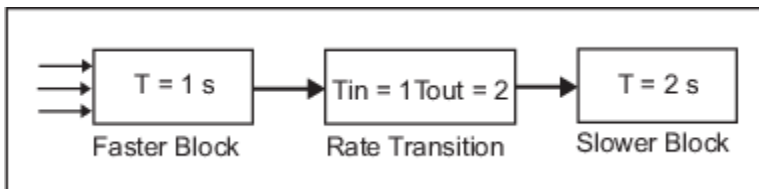
Faster to Slower Transitions in Real Time

In models where a faster block drives a slower block, you must compensate for the fact that execution of the slower block might span more than one execution period of the faster block. This means that the outputs of the faster block can change before the slower block has finished computing its outputs. The next figure shows a situation in which this problem arises ($T = \text{sample time}$). Note that lower priority tasks are preempted by higher priority tasks before completion.



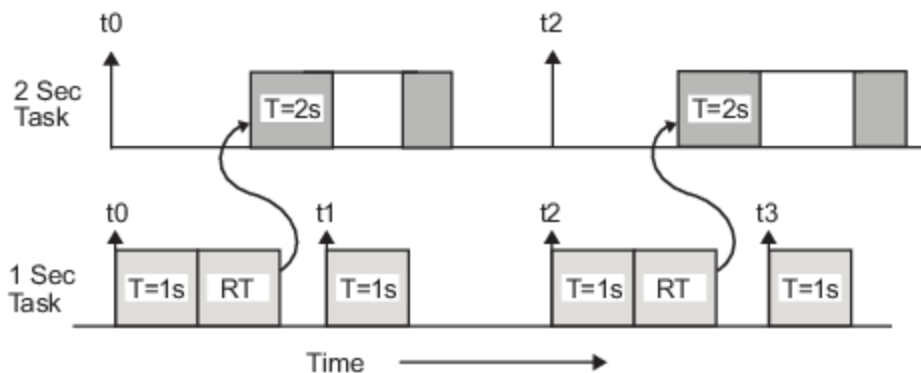
In the above figure, the faster block executes a second time before the slower block has completed execution. This can cause unpredictable results because the input data to the slow task is changing. Data might not maintain its integrity in this situation.

To avoid this situation, the Simulink engine must hold the outputs of the 1 second (faster) block until the 2 second (slower) block finishes executing. The way to accomplish this is by inserting a Rate Transition block between the 1 second and 2 second blocks. The input to the slower block does not change during its execution, maintaining data integrity.



It is assumed that the Rate Transition block is used in its default (protected/deterministic) mode.

The Rate Transition block executes at the sample rate of the slower block, but with the priority of the faster block.

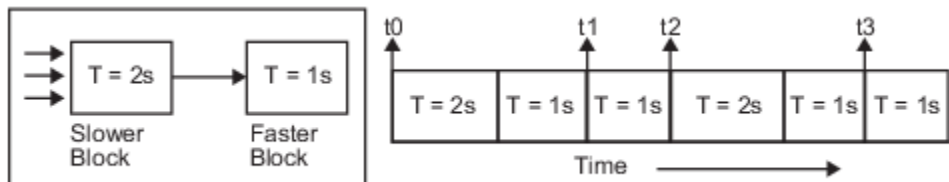


When you add a Rate Transition block, the block executes before the 2 second block (its priority is higher) and its output value is held constant while the 2 second block executes (it executes at the slower sample rate).

Slower to Faster Transitions in a Simulink Model

In a model where a slower block drives a faster block, the Simulink engine again computes the output of the driving block first. During sample intervals where only the faster block executes, the simulation progresses more rapidly.

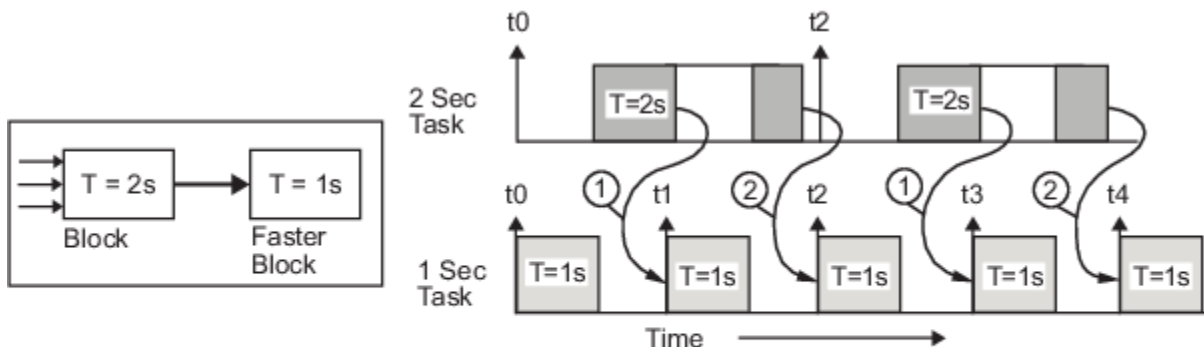
The next figure shows the execution sequence.



As you can see from the preceding figures, the Simulink engine can simulate models with multiple sample rates in an efficient manner. However, a Simulink simulation does not operate in real time.

Slower to Faster Transitions in Real Time

In models where a slower block drives a faster block, the generated code assigns the faster block a higher priority than the slower block. This means the faster block is executed before the slower block, which requires special care to avoid incorrect results.

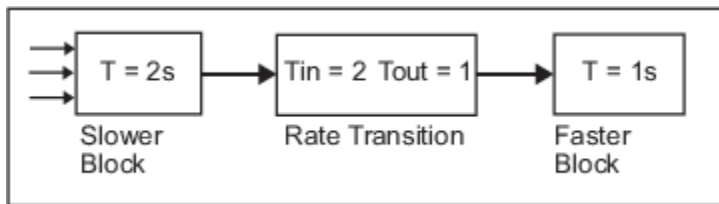


- ① The faster block executes a second time prior to the completion of the slower block.
- ② The faster block executes before the slower block.

This timing diagram illustrates two problems:

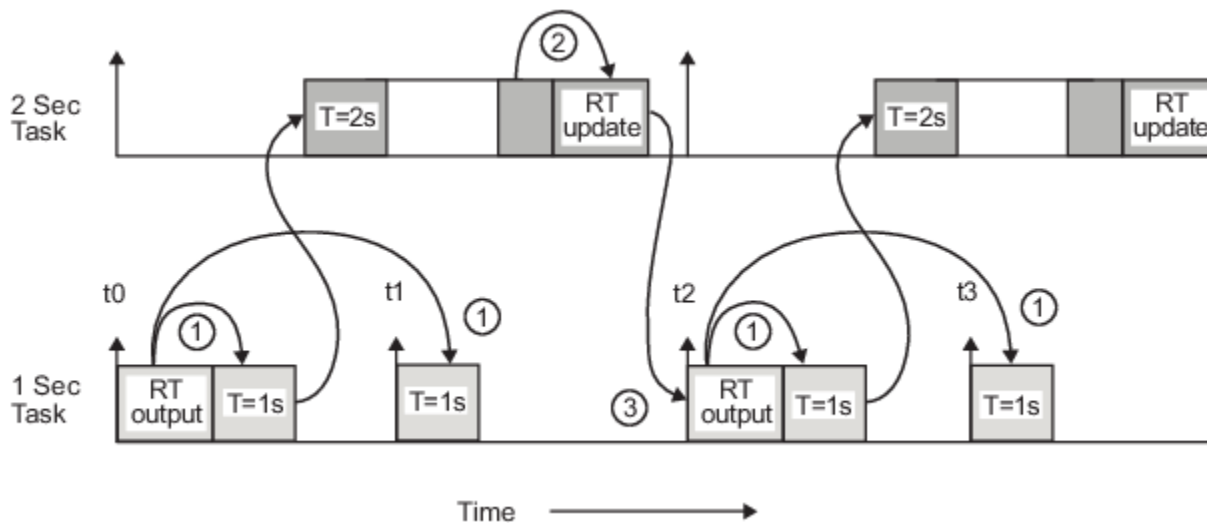
- Execution of the slower block is split over more than one faster block interval. In this case the faster task executes a second time before the slower task has completed execution. This means the inputs to the faster task can have incorrect values some of the time.
- The faster block executes before the slower block (which is backward from the way a Simulink simulation operates). In this case, the 1 second block executes first; but the inputs to the faster task have not been computed. This can cause unpredictable results.

To eliminate these problems, you must insert a Rate Transition block between the slower and faster blocks.



It is assumed that the Rate Transition block is used in its default (protected/deterministic) mode.

The next figure shows the timing sequence that results with the added Rate Transition block.



Three key points about transitions in this diagram (refer to circled numbers):

- 1 The Rate Transition block output runs in the 1 second task, but at a slower rate (2 seconds). The output of the Rate Transition block feeds the 1 second task blocks.
- 2 The Rate Transition update uses the output of the 2 second task to update its internal state.
- 3 The Rate Transition output in the 1 second task uses the state of the Rate Transition that was updated in the 2 second task.

The first problem is addressed because the Rate Transition block is updating at a slower rate and at the priority of the slower block. The input to the Rate Transition block (which is the output of the slower block) is read after the slower block completes executing.

The second problem is addressed because the Rate Transition block executes at a slower rate and its output does not change during the computation of the faster block it is driving. The output portion of a Rate Transition block is executed at the sample rate of the slower block, but with the priority of the faster block. Since the Rate Transition block drives the faster block and has effectively the same priority, it is executed before the faster block.

Note This use of the Rate Transition block changes the model. The output of the slower block is now delayed by one time step compared to the output without a Rate Transition block.

Protect Data Integrity with volatile Keyword

When you select model configuration parameter **Ensure data integrity during data transfer**, the code generated for a Rate Transition block defines global buffers and semaphores and uses them to protect the integrity of the transferred data.

Particularly for a multitasking application, the tasks (rates) involved in a data transfer can write to the transferred data, buffers, and semaphores at times that your compiler cannot anticipate. To prevent your compiler from optimizing the assembly code in a manner that compromises the integrity of the transferred data, the code generator applies the keyword `volatile` to the buffers and semaphores. The code generator does not apply `volatile` to the global variable that represents the transferred data because the `volatile` buffers and semaphores typically offer enough protection.

With Embedded Coder, you can explicitly apply `volatile` to the transferred data by applying the built-in custom storage class `Volatile` to the input of the Rate Transition block. For example, you can use this technique to help protect the integrity of data that your external code shares with the generated code.

Alternatively, to protect data that your external code shares with the generated code, you can write your own C functions that read and write the data in a protected manner. Then, you can apply the custom storage class `GetSet` to the data in the model, which causes the generated code to call your functions instead of directly accessing the data.

For more information about applying `volatile`, see “Protect Global Data with `const` and `volatile` Type Qualifiers” (Embedded Coder). For more information about `GetSet`, see “Access Data Through Functions with Storage Class `GetSet`” (Embedded Coder).

Separate Rate Transition Block Code and Data from Algorithm Code and Data

You can specify whether the code generator inlines code and data it produces for Rate Transition blocks with model code or places the code and data in separate functions that the model code calls. You control this by selecting the **Rate Transition block code** parameter. Separating Rate Transition block code and data from algorithm code and data enables you to independently analyze, optimize, and test Rate Transition block and algorithm code. By default, Rate Transition block code is set inline with algorithm code and data. You can separate the code and data so that the generated code contains separate `get` and `set` functions that the `model_step` functions call and a dedicated structure for state data. The generated code also contains separate `start` and `initialize` functions that the `model_initialize` function calls.

Example Model

Open the example model `rtwdemo_ratetrans`. This multirate, multitasking model contains several Rate Transition blocks operating at different modes.

```
open_system('rtwdemo_ratetrans');
set_param('rtwdemo_ratetrans','SystemTargetFile','ert.tlc');
set_param('rtwdemo_ratetrans','GenerateComments','Off');
```


Separate Code for Rate Transition blocks

In the Configuration Parameters dialog box, the **Rate Transition block code** parameter is set to **Function**. Generate code for the model. The code is in the files `rtwdemo_ratetrans.c` and `rtwdemo_ratetrans.h`.

```
currentDir = pwd;
[~,cgDir] = rtwdemodir();
slbuild('rtwdemo_ratetrans');

### Starting build procedure for: rtwdemo_ratetrans
### Successful completion of build procedure for: rtwdemo_ratetrans

Build Summary

Top model targets built:

Model                Action                Rebuild Reason
=====
rtwdemo_ratetrans    Code generated and compiled. Code generation information file does not exist

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 21.251s

hfile=fullfile(cgDir, 'rtwdemo_ratetrans_ert_rtw','rtwdemo_ratetrans.h');
rtwdemodbtype(hfile,'typedef struct {' DW;', 1, 1);

typedef struct {
    real_T OutportBufferForOut3[20];
    real_T Integrator1_DSTATE[20];
    real_T Integrator2_DSTATE[20];
    real_T Integrator3_DSTATE[20];
    real_T Integrator1_PREV_U[20];
    real_T Integrator2_PREV_U[20];
    real_T Integrator3_PREV_U[20];
    uint32_T Algorithm_PREV_T;
    struct {
        uint_T Algorithm_RESET_ELAPS_T:1;
    } bitsForTID1;

    uint8_T Integrator1_SYSTEM_ENABLE;
    uint8_T Integrator2_SYSTEM_ENABLE;
    uint8_T Integrator3_SYSTEM_ENABLE;
} DW;
```

For Rate Transition blocks, the state data is not in the global state structure, `DW_rtwdemo_ratetrans_T`. This data is in its own structure in the file `rtwdemo_ratetrans_rtb.h`.

This code is in the file `rtwdemo_ratetrans.c`.

```
cfile=fullfile(cgDir, 'rtwdemo_ratetrans_ert_rtw','rtwdemo_ratetrans.c');
rtwdemodbtype(cfile,'void rtwdemo_ratetrans_step0','void rtwdemo_ratetrans_terminate(void)', 1, 0);

void rtwdemo_ratetrans_step0(void)
{
    (rtM->Timing.RateInteraction.TID0_1)++;
    if ((rtM->Timing.RateInteraction.TID0_1) > 1) {
```

```

    rtM->Timing.RateInteraction.TID0_1 = 0;
}

rtwdemo_rate_DetAndIntegS2F_get(rtY.Out1);
rtwdemo_rate_r_IntegOnlyS2F_get(rtY.Out2);
memcpy(&rtY.Out3[0], &rtDW.OutputBufferForOut3[0], 20U * sizeof(real_T));
rtwdemo_rate_DetAndIntegF2S_set(rtU.In1);
rtwdemo_rate_r_IntegOnlyF2S_set(rtU.In2);
}

void rtwdemo_ratetrans_step1(void)
{
    real_T rtb_DetAndIntegF2S[20];
    real_T rtb_IntegOnlyF2S[20];
    real_T Integrator3_DSTATE;
    real_T tmp;
    int32_T i;
    uint32_T Algorithm_ELAPS_T;
    rtwdemo_rate_DetAndIntegF2S_get(rtb_DetAndIntegF2S);
    rtwdemo_rate_r_IntegOnlyF2S_get(rtb_IntegOnlyF2S);
    if (rtDW.bitsForTID1.Algorithm_RESET_ELAPS_T) {
        Algorithm_ELAPS_T = 0U;
    } else {
        Algorithm_ELAPS_T = rtM->Timing.clockTick1 - rtDW.Algorithm_PREV_T;
    }

    rtDW.Algorithm_PREV_T = rtM->Timing.clockTick1;
    rtDW.bitsForTID1.Algorithm_RESET_ELAPS_T = false;
    tmp = 0.001 * (real_T)Algorithm_ELAPS_T;
    for (i = 0; i < 20; i++) {
        if (rtDW.Integrator1_SYSTEM_ENABLE == 0) {
            rtDW.Integrator1_DSTATE[i] += tmp * rtDW.Integrator1_PREV_U[i];
        }

        if (rtDW.Integrator2_SYSTEM_ENABLE == 0) {
            rtDW.Integrator2_DSTATE[i] += tmp * rtDW.Integrator2_PREV_U[i];
        }

        Integrator3_DSTATE = rtDW.Integrator3_DSTATE[i];
        if (rtDW.Integrator3_SYSTEM_ENABLE == 0) {
            Integrator3_DSTATE += tmp * rtDW.Integrator3_PREV_U[i];
        }

        rtDW.Integrator3_DSTATE[i] = Integrator3_DSTATE;
        rtDW.OutputBufferForOut3[i] = Integrator3_DSTATE;
        rtDW.Integrator1_PREV_U[i] = rtb_DetAndIntegF2S[i];
        rtDW.Integrator2_PREV_U[i] = rtb_IntegOnlyF2S[i];
        rtDW.Integrator3_PREV_U[i] = rtU.In3[i];
    }

    rtDW.Integrator1_SYSTEM_ENABLE = 0U;
    rtDW.Integrator2_SYSTEM_ENABLE = 0U;
    rtDW.Integrator3_SYSTEM_ENABLE = 0U;
    rtwdemo_rate_DetAndIntegS2F_set(rtDW.Integrator1_DSTATE);
    rtwdemo_rate_r_IntegOnlyS2F_set(rtDW.Integrator2_DSTATE);
    rtM->Timing.clockTick1++;
}

```

```

void rtwdemo_ratetrans_initialize(void)
{
    rtDW.bitsForTID1.Algorithm_RESET_ELAPS_T = true;
    rtDW.Integrator1_SYSTEM_ENABLE = 1U;
    rtDW.Integrator2_SYSTEM_ENABLE = 1U;
    rtDW.Integrator3_SYSTEM_ENABLE = 1U;
}

```

The `rtwdemo_ratetrans_step0` and `rtwdemo_ratetrans_step1` functions contain calls to `get` and `set` functions. These functions contain the Rate Transition block code. These function definitions are in the file `rtwdemo_ratetrans_rtb.c`.

Generate Inlined Code for Rate Transition blocks

In the Configuration Parameters dialog box, set the **Rate Transition block code** parameter to **Inline**. Generate code for the model. The code is in the files `rtwdemo_ratetrans.c` and `rtwdemo_ratetrans.h`.

```

set_param('rtwdemo_ratetrans','RateTransitionBlockCode','Inline');
slbuild('rtwdemo_ratetrans')

### Starting build procedure for: rtwdemo_ratetrans
### Successful completion of build procedure for: rtwdemo_ratetrans

```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_ratetrans	Code generated and compiled.	Generated code was out of date.

1 of 1 models built (0 models already up to date)
 Build duration: 0h 0m 13.388s

This code is now in the file `rtwdemo_ratetrans.h`.

```

hfile=fullfile(cgDir, 'rtwdemo_ratetrans_ert_rtw','rtwdemo_ratetrans.h');
rtwdemodbtype(hfile, 'typedef struct {', '} DW;', 1, 1);

```

```

typedef struct {
    real_T Integrator1_DSTATE[20];
    real_T Integrator2_DSTATE[20];
    real_T Integrator3_DSTATE[20];
    real_T DetAndIntegS2F_Buffer0[20];
    volatile real_T IntegOnlyS2F_Buffer[40];
    real_T DetAndIntegF2S_Buffer[20];
    volatile real_T IntegOnlyF2S_Buffer0[20];
    real_T Integrator1_PREV_U[20];
    real_T Integrator2_PREV_U[20];
    real_T Integrator3_PREV_U[20];
    uint32_T Algorithm_PREV_T;
    struct {
        uint_T Algorithm_RESET_ELAPS_T:1;
    } bitsForTID1;

    volatile int8_T IntegOnlyS2F_ActiveBufIdx;
}

```

```

volatile int8_T IntegOnlyF2S_semaphoreTaken;
uint8_T Integrator1_SYSTEM_ENABLE;
uint8_T Integrator2_SYSTEM_ENABLE;
uint8_T Integrator3_SYSTEM_ENABLE;
} DW;

```

For Rate Transition blocks, the state data is not in the global state structure, `DW_rtwdemo_ratetrans_T`. This data is in its own structure in the file `rtwdemo_ratetrans_rtb.h`.

This code is now in the file `rtwdemo_ratetrans_rtb.c`.

```

cfile=fullfile(cgDir, 'rtwdemo_ratetrans_ert_rtw', 'rtwdemo_ratetrans.c');
rtwdemodbtype(cfile, 'void rtwdemo_ratetrans_step0', 'void rtwdemo_ratetrans_terminate(void)', 1, 0);

void rtwdemo_ratetrans_step0(void)
{
    int32_T i;
    int32_T i_0;
    (rtM->Timing.RateInteraction.TID0_1)++;
    if ((rtM->Timing.RateInteraction.TID0_1) > 1) {
        rtM->Timing.RateInteraction.TID0_1 = 0;
    }

    if (rtM->Timing.RateInteraction.TID0_1 == 1) {
        memcpy(&rtY.Out1[0], &rtDW.DetAndIntegS2F_Buffer0[0], 20U * sizeof(real_T));
    }

    i = rtDW.IntegOnlyS2F_ActiveBufIdx * 20;
    for (i_0 = 0; i_0 < 20; i_0++) {
        rtY.Out2[i_0] = rtDW.IntegOnlyS2F_Buffer[i_0 + i];
    }

    if (rtM->Timing.RateInteraction.TID0_1 == 1) {
        memcpy(&rtDW.DetAndIntegF2S_Buffer[0], &rtU.In1[0], 20U * sizeof(real_T));
    }

    if (rtDW.IntegOnlyF2S_semaphoreTaken == 0) {
        for (i = 0; i < 20; i++) {
            rtDW.IntegOnlyF2S_Buffer0[i] = rtU.In2[i];
        }
    }
}

void rtwdemo_ratetrans_step1(void)
{
    real_T rtb_IntegOnlyF2S[20];
    real_T Integrator1_DSTATE;
    real_T Out3;
    real_T tmp;
    int32_T i;
    uint32_T Algorithm_ELAPS_T;
    rtDW.IntegOnlyF2S_semaphoreTaken = 1;
    for (i = 0; i < 20; i++) {
        rtb_IntegOnlyF2S[i] = rtDW.IntegOnlyF2S_Buffer0[i];
    }

    rtDW.IntegOnlyF2S_semaphoreTaken = 0;
}

```

```

if (rtDW.bitsForTID1.Algorithm_RESET_ELAPS_T) {
    Algorithm_ELAPS_T = 0U;
} else {
    Algorithm_ELAPS_T = rtM->Timing.clockTick1 - rtDW.Algorithm_PREV_T;
}

rtDW.Algorithm_PREV_T = rtM->Timing.clockTick1;
rtDW.bitsForTID1.Algorithm_RESET_ELAPS_T = false;
tmp = 0.001 * (real_T)Algorithm_ELAPS_T;
for (i = 0; i < 20; i++) {
    Integrator1_DSTATE = rtDW.Integrator1_DSTATE[i];
    if (rtDW.Integrator1_SYSTEM_ENABLE == 0) {
        Integrator1_DSTATE += tmp * rtDW.Integrator1_PREV_U[i];
    }

    rtDW.Integrator1_DSTATE[i] = Integrator1_DSTATE;
    if (rtDW.Integrator2_SYSTEM_ENABLE == 0) {
        rtDW.Integrator2_DSTATE[i] += tmp * rtDW.Integrator2_PREV_U[i];
    }

    if (rtDW.Integrator3_SYSTEM_ENABLE != 0) {
        Out3 = rtDW.Integrator3_DSTATE[i];
    } else {
        Out3 = tmp * rtDW.Integrator3_PREV_U[i] + rtDW.Integrator3_DSTATE[i];
    }

    rtY.Out3[i] = Out3;
    rtDW.Integrator1_PREV_U[i] = rtDW.DetAndIntegF2S_Buffer[i];
    rtDW.Integrator2_PREV_U[i] = rtb_IntegOnlyF2S[i];
    rtDW.Integrator3_DSTATE[i] = Out3;
    rtDW.Integrator3_PREV_U[i] = rtU.In3[i];
    rtDW.DetAndIntegS2F_Buffer0[i] = Integrator1_DSTATE;
}

rtDW.Integrator1_SYSTEM_ENABLE = 0U;
rtDW.Integrator2_SYSTEM_ENABLE = 0U;
rtDW.Integrator3_SYSTEM_ENABLE = 0U;
for (i = 0; i < 20; i++) {
    rtDW.IntegOnlyS2F_Buffer[i + (rtDW.IntegOnlyS2F_ActiveBufIdx == 0) * 20] =
        rtDW.Integrator2_DSTATE[i];
}

rtDW.IntegOnlyS2F_ActiveBufIdx = (int8_T)(rtDW.IntegOnlyS2F_ActiveBufIdx == 0);
rtM->Timing.clockTick1++;
}

void rtwdemo_ratetrans_initialize(void)
{
    rtDW.bitsForTID1.Algorithm_RESET_ELAPS_T = true;
    rtDW.Integrator1_SYSTEM_ENABLE = 1U;
    rtDW.Integrator2_SYSTEM_ENABLE = 1U;
    rtDW.Integrator3_SYSTEM_ENABLE = 1U;
}

```

The code is inlined in functions `rtwdemo_ratetrans_step0` and `rtwdemo_ratetrans_step1`.

Limitation

The code generator does not separate the code and data for Rate Transition blocks that have variable-size signals or are inside of a For Each Subsystem block.

See Also

- Rate Transition block code (Embedded Coder)

```
bdclose('rtwdemo_ratetrans');  
rtwdemoclean;  
cd(currentDir)
```

See Also

More About

- “Time-Based Scheduling and Code Generation” on page 5-2
- “Sample Times in Subsystems”
- “Sample Times in Systems”
- “Modeling for Multitasking Execution” on page 5-9
- “Configure Time-Based Scheduling” on page 5-33
- “Resolve Rate Transitions”
- “Time-Based Scheduling Example Models” on page 5-35

Configure Time-Based Scheduling

For details about solver options, see “Solver Pane”.

Configure Start and Stop Times

The value of model configuration parameter **Stop time** must be greater than or equal to the value of parameter **Start time**. If the stop time is zero, or if the total simulation time (Stop minus Start) is less than zero, the generated program runs for one step. If the stop time is set to `inf`, the generated program runs indefinitely.

When using the GRT or ERT targets, you can override the stop time when running a generated program from the Microsoft Windows command prompt or UNIX¹ command line. To override the stop time that was set during code generation, use the `-tf` switch.

```
model -tf n
```

The program runs for `n` seconds. If `n = inf`, the program runs indefinitely.

Certain blocks have a dependency on absolute time. If you are designing a program that is intended to run indefinitely (model configuration parameter **Stop time** = `inf`), and your generated code does not use the `rtModel` data structure (that is, it uses `simstructs` instead), you must not use these blocks. See “Absolute Time Limitations” on page 4-14 for a list of blocks that can potentially overflow timers.

If you know how long an application that depends on absolute time needs to run, you can prevent the timers from overflowing and force the use of optimal word sizes by selecting model configuration parameter **Application lifespan (days)**. See “Control Memory Allocation for Time Counters” on page 40-10 for details.

Configure the Solver Type

For code generation, you must configure a model to use a fixed-step solver for system target files except for S-function and RSim system target files. You can configure the S-function and RSim targets with a fixed-step or variable-step solver.

Configure the Tasking Mode

The code generator supports both single-tasking and multitasking modes for periodic sample times. See “Time-Based Scheduling and Code Generation” on page 5-2 for details.

See Also

More About

- “Time-Based Scheduling and Code Generation” on page 5-2
- “Sample Times in Subsystems”
- “Sample Times in Systems”

¹ UNIX is a registered trademark of The Open Group in the United States and other countries.

- “Time-Based Scheduling Example Models” on page 5-35

Time-Based Scheduling Example Models

Optimize Memory Usage for Time Counters

This example shows how to optimize the amount of memory that the code generator allocates for time counters. The example optimizes the memory that stores elapsed time, the interval of time between two events.

The code generator represents time counters as unsigned integers. The word size of time counters is based on the setting of the model configuration parameter **Application lifespan (days)**, which specifies the expected maximum duration of time the application runs. You can use this parameter to prevent time counter overflows. The default size is 64 bits.

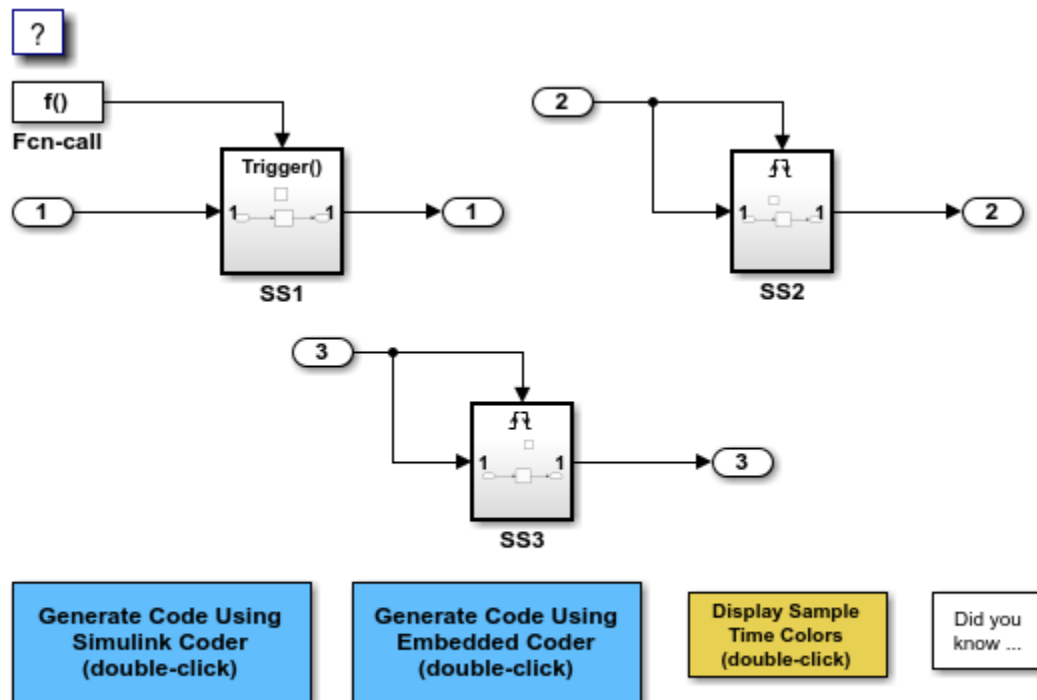
The number of bits that a time counter uses depends on the setting of the **Application lifespan (days)** parameter. For example, if a time counter increments at a rate of 1 kHz, to avoid an overflow, the counter has the following number of bits:

- Lifespan < 0.25 sec: 8 bits
- Lifespan < 1 min: 16 bits
- Lifespan < 49 days: 32 bits
- Lifespan > 50 days: 64 bits

A 64-bit time counter does not overflow for 590 million years.

Open Example Model

Open the example model `rtwdemo_abstime`.



Copyright 1994-2020 The MathWorks, Inc.

The model consists of three subsystems SS1, SS2, and SS3. Open the Model Configuration Parameters dialog box. On the **Math and Data Types** pane, the **Application lifespan (days)** parameter is set to the default, which is `inf`.

The three subsystems contain a discrete-time integrator that requires elapsed time as input to compute its output value. The subsystems vary as follows:

- SS1 - Clocked at 1 kHz. Does not require a time counter. **Sample time type** parameter for trigger port is set to `periodic`. Elapsed time is inlined as 0.001.
- SS2 - Clocked at 100 Hz. Requires a time counter. Based on a lifespan of 1 day, a 32-bit counter stores the elapsed time.
- SS3 - Clocked at 0.5 Hz. Requires a time counter. Based on a lifespan of 1 day, a 16-bit counter stores the elapsed time.

Simulate the Model

Simulate the model. By default, the model is configured to show sample times in different colors. Discrete sample times for the three subsystems appear red, green, and blue. Triggered subsystems are blue-green.

Generate Code and Report

1. Create a temporary folder for the build and inspection process.
2. Configure the model for the code generator to use the GRT system target file and a lifespan of `inf` days.

3. Build the model.

```
### Starting build procedure for: rtwdemo_abstime
### Successful completion of build procedure for: rtwdemo_abstime
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_abstime	Code generated and compiled.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)

Build duration: 0h 0m 11.921s

Review Generated Code

Open the generated source file `rtwdemo_abstime.h`.

```
struct tag_RTM_rtwdemo_abstime_T {
    const char_T *errorStatus;

    /*
     * Timing:
     * The following substructure contains information regarding
     * the timing information for the model.
     */
    struct {
        uint32_T clockTick1;
        uint32_T clockTickH1;
        uint32_T clockTick2;
        uint32_T clockTickH2;
        struct {
            uint16_T TID[3];
            uint16_T cLimit[3];
        } TaskCounters;
    } Timing;
};

/* Block states (default storage) */
extern DW_rtwdemo_abstime_T rtwdemo_abstime_DW;

/* Zero-crossing (trigger) state */
extern PrevZCX_rtwdemo_abstime_T rtwdemo_abstime_PrevZCX;

/* External inputs (root inport signals with default storage) */
extern ExtU_rtwdemo_abstime_T rtwdemo_abstime_U;

/* External outputs (root outports fed by signals with default storage) */
extern ExtY_rtwdemo_abstime_T rtwdemo_abstime_Y;

/* Model entry point functions */
extern void rtwdemo_abstime_initialize(void);
extern void rtwdemo_abstime_step0(void);
extern void rtwdemo_abstime_step1(void);
extern void rtwdemo_abstime_step2(void);
```

```
extern void rtwdemo_abstime_terminate(void);

/* Real-time Model object */
extern RT_MODEL_rtwdemo_abstime_T *const rtwdemo_abstime_M;

/*-
 * The generated code includes comments that allow you to trace directly
 * back to the appropriate location in the model. The basic format
 * is <system>/block_name, where system is the system number (uniquely
 * assigned by Simulink) and block_name is the name of the block.
 *
 * Use the MATLAB hilite_system command to trace the generated code back
 * to the model. For example,
 *
 * hilite_system('<S3>') - opens system 3
 * hilite_system('<S3>/Kp') - opens and selects block Kp which resides in S3
 *
 * Here is the system hierarchy for this model
 *
 * '<Root>' : 'rtwdemo_abstime'
 * '<S1>' : 'rtwdemo_abstime/SS1'
 * '<S2>' : 'rtwdemo_abstime/SS2'
 * '<S3>' : 'rtwdemo_abstime/SS3'
 */
#endif /* RTW_HEADER_rtwdemo_abstime_h_ */
```

Four 32-bit unsigned integers, `clockTick1`, `clockTickH1`, `clockTick2`, and `clockTickH2` are counters for storing the elapsed time of subsystems SS2 and SS3.

Enable Optimization and Regenerate Code

1. Reconfigure the model to set the lifespan to 1 day.
2. Build the model.

```
### Starting build procedure for: rtwdemo_abstime
### Successful completion of build procedure for: rtwdemo_abstime
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_abstime	Code generated and compiled.	Incremental checksum changed.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 10.449s
```

Review the Regenerated Code

```
struct tag_RTM_rtwdemo_abstime_T {
    const char_T *errorStatus;

    /*
     * Timing:
     * The following substructure contains information regarding
```

```

    * the timing information for the model.
    */
    struct {
        uint32_T clockTick1;
        uint16_T clockTick2;
        struct {
            uint16_T TID[3];
            uint16_T cLimit[3];
        } TaskCounters;
    } Timing;
};

/* Block states (default storage) */
extern DW_rtdemo_abstime_T rtdemo_abstime_DW;

/* Zero-crossing (trigger) state */
extern PrevZCX_rtdemo_abstime_T rtdemo_abstime_PrevZCX;

/* External inputs (root inport signals with default storage) */
extern ExtU_rtdemo_abstime_T rtdemo_abstime_U;

/* External outputs (root outports fed by signals with default storage) */
extern ExtY_rtdemo_abstime_T rtdemo_abstime_Y;

/* Model entry point functions */
extern void rtdemo_abstime_initialize(void);
extern void rtdemo_abstime_step0(void);
extern void rtdemo_abstime_step1(void);
extern void rtdemo_abstime_step2(void);
extern void rtdemo_abstime_terminate(void);

/* Real-time Model object */
extern RT_MODEL_rtdemo_abstime_T *const rtdemo_abstime_M;

/*-
 * The generated code includes comments that allow you to trace directly
 * back to the appropriate location in the model. The basic format
 * is <system>/block_name, where system is the system number (uniquely
 * assigned by Simulink) and block_name is the name of the block.
 *
 * Use the MATLAB hilite_system command to trace the generated code back
 * to the model. For example,
 *
 * hilite_system('<S3>') - opens system 3
 * hilite_system('<S3>/Kp') - opens and selects block Kp which resides in S3
 *
 * Here is the system hierarchy for this model
 *
 * '<Root>' : 'rtdemo_abstime'
 * '<S1>' : 'rtdemo_abstime/SS1'
 * '<S2>' : 'rtdemo_abstime/SS2'
 * '<S3>' : 'rtdemo_abstime/SS3'
 */
#endif
/* RTW_HEADER_rtdemo_abstime_h_ */

```

The new setting for the **Application lifespan (days)** parameter instructs the code generator to set aside less memory for the time counters. The regenerated code includes:

- 32-bit unsigned integer, `clockTick1`, for storing the elapsed time of the task for SS2
- 16-bit unsigned integer, `clockTick2`, for storing the elapsed time of the task for SS3

Related Information

- “Model Configuration Parameters: Code Generation Optimization”
- “Timers in Asynchronous Tasks” on page 6-38
- “Time-Based Scheduling and Code Generation” on page 5-2

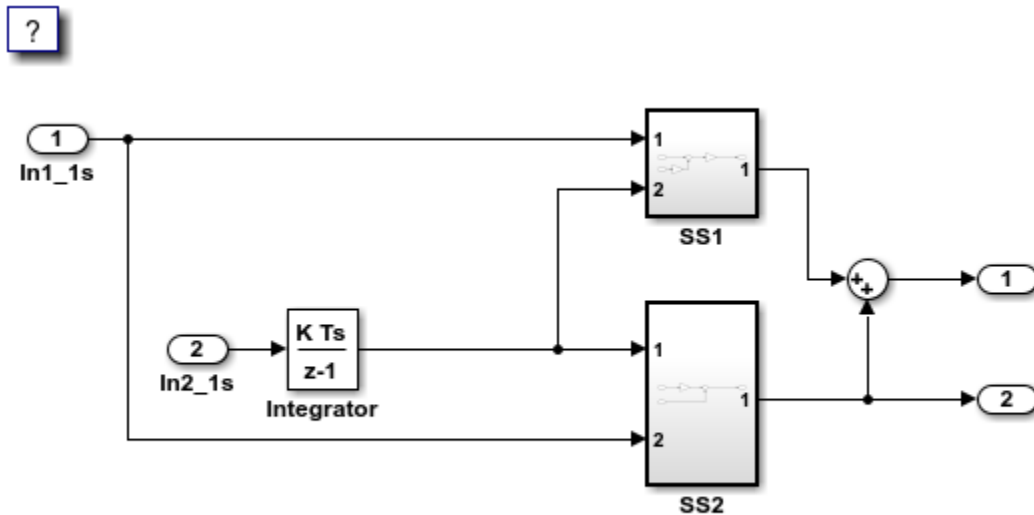
Single-Rate Modeling (Bare Board, No OS)

This model shows code generation for a single-rate discrete-time model configured for a bare-board target (one with no operating system).

Open Example Model

Open the example model `rtwdemo_srb`.

```
open_system('rtwdemo_srb')
```



Copyright 1994-2020 The MathWorks, Inc.

The model uses one sample time and is configured to display sample-time colors when an update diagram occurs. Inport blocks `In1_1s` and `In2_1s` specify a 1-second sample time, which is enforced by the setting of model configuration parameter **Periodic sample time constraint**.

The model is configured to display color-coded sample times with annotations. To see them, after opening the model, update the diagram by pressing **Ctrl+D**. To display the legend, press **Ctrl+J**. Because the model is configured with one sample time, the model appears red, which is the color that represents the fastest discrete sample time in the model.

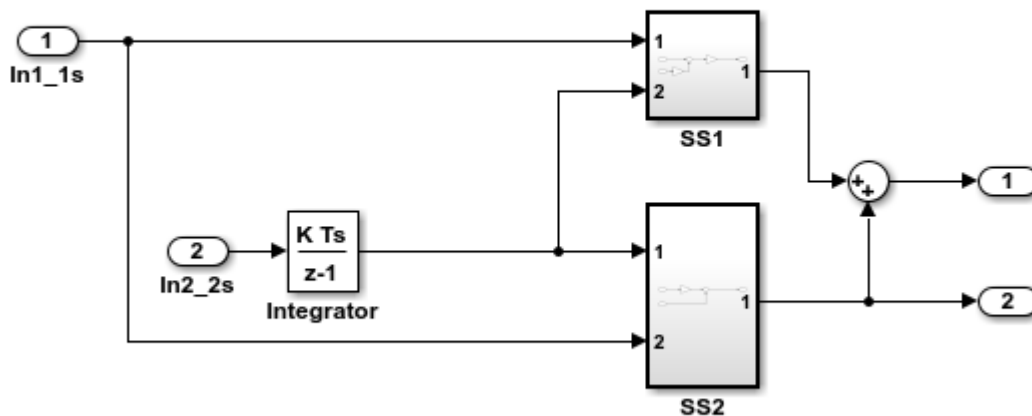
Multirate Modeling in Single-Tasking Mode (Bare Board, No OS)

This model shows the code generated for a multirate discrete-time model configured for single-tasking on a bare-board target (one with no operating system).

Open Example Model

Open the example model `rtwdemo_mrstbb`.

```
open_system('rtwdemo_mrstbb')
```



Copyright 1994-2020 The MathWorks, Inc.

The model contains two sample times. Inport block 1 and Inport block 2 specify 1-second and 2-second sample times, respectively, which are enforced by the setting of model configuration parameter **Periodic sample time constraint**. The solver is set for single-tasking operation. Rate transition blocks are, therefore, not included between blocks executing at different sample times because preemption will not occur.

The model is configured to display color-coded sample times with annotations. To see them, after opening the model, update the diagram by pressing **Ctrl+D**. To display the legend, press **Ctrl+J**. Red represents the fastest discrete sample time in the model, green represents the second fastest, and yellow represents mixed sample times.

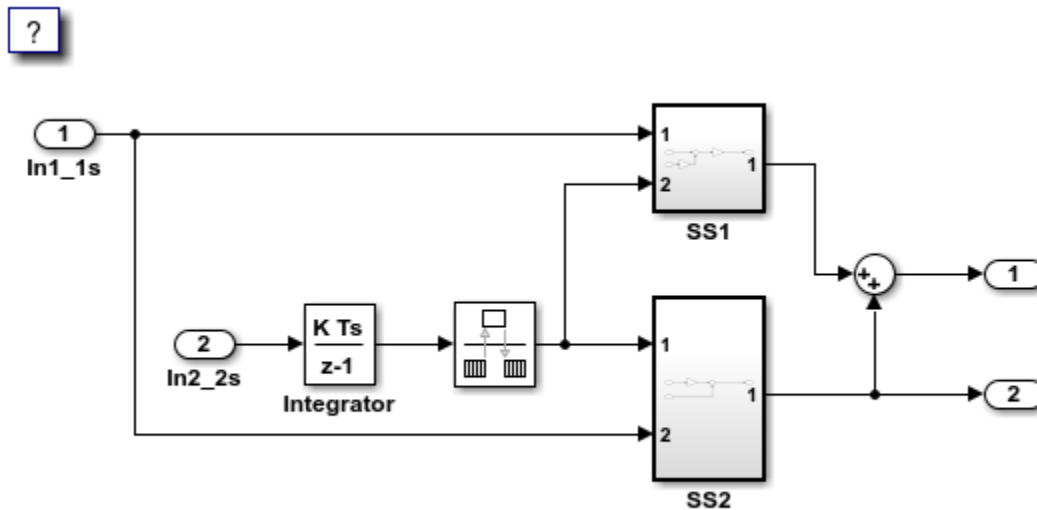
Multirate Modeling in Multitasking Mode (Bare Board, No OS)

This model shows the code generated for a multirate discrete-time model configured for a multitasking bare-board target (one with no operating system).

Open Example Model

Open the example model `rtwdemo_mrmtbb`.

```
open_system('rtwdemo_mrmtbb')
```



Copyright 1994-2020 The MathWorks, Inc.

Explore Example Model

The model contains two sample times. Inport block 1 and Inport block 2 specify 1-second and 2-second sample times, respectively, which are enforced by the setting of model configuration parameter **Periodic sample time constraint**. The solver is set for multitasking operation, which means a rate transition block is required to ensure that data integrity is enforced when the 1-second task preempts the 2-second task. Proper rate transitions are always enforced by Simulink® and Simulink Coder™. This model specifies an explicit rate transition block. Alternatively, this block could be automatically inserted by Simulink setting model configuration parameter **Automatically handle rate transition for data transfer**.

The model is configured to display sample-time colors upon diagram update. To see them, after opening the model, update the diagram by pressing **Ctrl+D**. To display the legend, press **Ctrl+J**. Red represents the fastest discrete sample time in the model, green represents the second fastest, and yellow represents mixed sample times.

Data Transfer Assumptions

Basis of operation for data transfers between tasks:

- Data transitions occur between a single reading task and a single writing task.
- A read or write of a byte sized variable is atomic.
- When two tasks interact through a data transition, only one of them can preempt the other.
- For periodic tasks, the faster rate task has higher priority than the slower rate task; the faster rate task always preempts the slower rate task.
- Tasks run on a single processor. Time slicing is not allowed.
- Processes do not crash and restart (especially while data is being transferred between tasks)

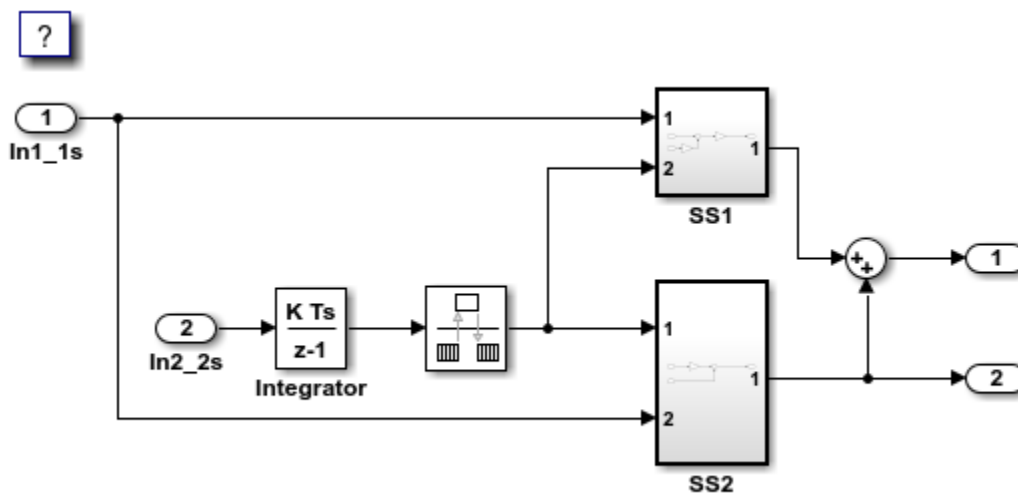
Multirate Modeling in Multitasking Mode (VxWorks® OS)

This example generates code for a multirate discrete-time model configured for a multitasking operating system target (VxWorks®). The model contains two sample times. Inport block 1 and Inport block 2 specify 1-second and 2-second sample times, respectively, which are enforced by the setting of model configuration parameter **Periodic sample time constraint**. The solver is set for multitasking operation, which means a Rate Transition block is required to ensure that data integrity is enforced when the 1-second task preempts the 2-second task. Simulink® and the code generator enforce proper rate transitions. This model specifies an explicit Rate Transition block. Alternatively, you can instruct Simulink® to insert this block for you by setting model configuration parameter **Automatically handle rate transition for data transfer**.

The model is configured to display color-coded sample times with annotations. To see them, after opening the model, update the diagram by pressing Ctrl+D. To display the legend, press Ctrl+J. Red represents the fastest discrete sample time in the model, green represents the second fastest, and yellow represents mixed sample times.

Example Model

```
model = 'rtwdemo_mrmtos';
open_system(model);
```



Trade Determinism and Data Integrity to Improve System Performance

This model shows the differences in the operation modes of the Rate Transition block when used in a multirate, multitasking model. The flexible options for the Rate Transition block allow you to select the mode that is best suited for your application. You can trade levels of determinism and data integrity to improve system performance.

Rate Transition Block Modes of Operation

Ensure data integrity and determinism (DetAndInteg): Data is transferred such that all data bytes for the signal (including all elements of a wide signal) are from the same time step. Additionally, it is ensured that the relative sample time (delay) from which the data is transferred from one rate to another is always the same. Only ANSI® C code is used, no target specific 'critical section' protection is needed.

Ensure integrity (IntegOnly): Data is transferred such that all data bytes for the signal (including all elements of a wide signal) are from the same time step. However, from one transfer of data to the next, the relative sample time (delay) for which the data is transferred can vary. In this mode, the code to read/write the data is run more often than in the DetandInt mode. In the worst case, the delay is equivalent to the DetandInt mode, but the delay can be less which is important in some applications. Also, this mode supports data transfers to/from asynchronous rates which the DetandInt mode cannot support. Only ANSI-C code is used, no target specific 'critical section' protection is needed.

No data consistency operations are performed (None): For this case, the Rate Transition block does not generate code. This mode is acceptable in some applications where atomic access of scalar data types is guaranteed and when the relative temporal values of the data is not important. This mode does not introduce any delay.

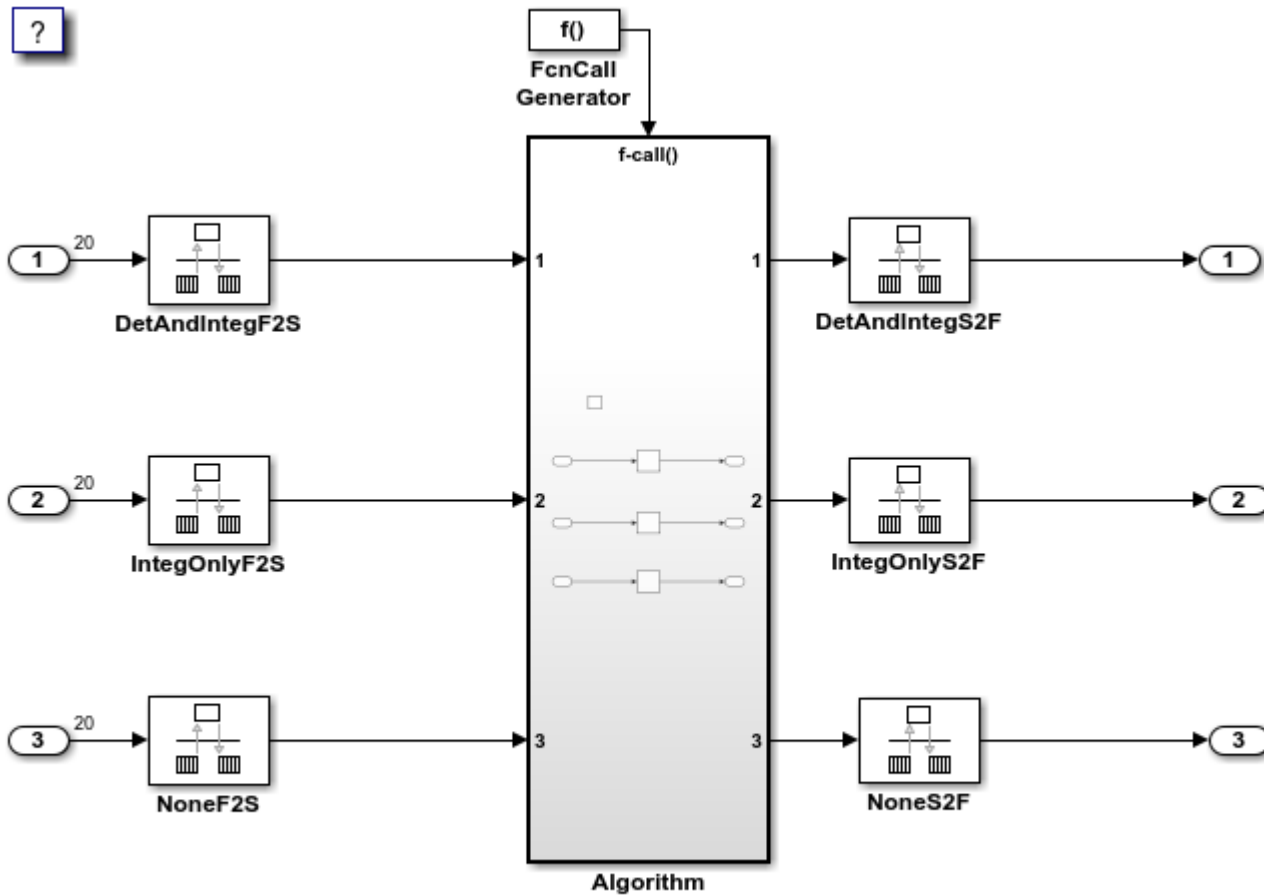
Data Transfer Assumptions

Basis of operation for data transfers between tasks:

- Data transitions occur between a single reading task and a single writing task.
- A read or write of a byte sized variable is atomic.
- When two tasks interact through a data transition, only one of them can preempt the other.
- For periodic tasks, the faster rate task has higher priority than the slower rate task; the faster rate task always preempts the slower rate task.
- All tasks run on a single processor. Time slicing is not allowed.
- Processes do not crash/restart (especially while data is being transferred between tasks).

Model `rtwdemo_ratetrans`

```
open_system('rtwdemo_ratetrans')
```



Copyright 1994-2020 The MathWorks, Inc.

Model `rtwdemo_ratetrans` shows the differences in the operation modes of the following Rate Transition blocks.

Rate Transition Block `DetAndIntegF2S`

Determinism and data integrity (fast to slow transition):

- The block output is used as a persistent data buffer.
- Data is written to output at slower rate but done during the faster rate context.
- Data as seen by the slower rate is always the value when both the faster and slower rate last executed. Any subsequent steps by the faster rate (and associated data updates) while the slower rate is running are not seen by the slower rate.

Rate Transition Block `DetAndIntegS2F`

Determinism and data integrity (slow to fast transition):

- Uses two persistent data buffers, an internal buffer and the blocks output.
- The internal buffer is copied to the output at the slower rate but done during the faster rate context.

- The internal buffer is written at the slower rate and during the slower rate context.
- The data that Fast rate sees is always delayed, i.e. data is from the previous step of the slow rate code.

Rate Transition Block IntegOnlyF2S

Data integrity only (fast to slow transition):

- The block output is used as a persistent data buffer.
- Data is written to buffer during the faster rate context if a flag indicates it not in the process of being read.
- The flag is set and data is copied from the buffer to output at the slow rate, the flag is then cleared. This is an additional copy as compared to the deterministic case.
- Data as seen by the slower rate can be from a more recent step of the faster rate than from when the slower rate and faster rate both executed.

Rate Transition Block IntegOnlyS2F

Data integrity only (slow to fast transition):

- Uses two persistent data buffers, both are internal buffers.
- One of the 2 buffers is always copied to the output at faster rate.
- One of the 2 buffers is written at the slower rate and during the slower rate context, then the active buffer is switched.
- The data as seen by the faster rate can be more recent than for the deterministic case. Specifically, when both the slower and faster rate have their hits, the faster rate will see a previous value from the slower rate. But, subsequent steps for the faster rate may see an updated value (when the slower rate updates the non-active buffer and switches the active buffer flag).

Rate Transition Block NoneF2S

No code is generated for the Rate Transition block when determinism and data integrity is waived.

Rate Transition Block NoneS2F

No code is generated for the Rate Transition block when determinism and data integrity is waived.

```
bdclose('rtwdemo_ratetrans');
```

See Also

More About

- “Time-Based Scheduling and Code Generation” on page 5-2
- “Modeling for Single-Tasking Execution” on page 5-6
- “Modeling for Multitasking Execution” on page 5-9

Event-Based Scheduling in Simulink Coder

- “Asynchronous Events” on page 6-2
- “Generate Interrupt Service Routines” on page 6-4
- “Spawn and Synchronize Execution of RTOS Task” on page 6-11
- “Pass Asynchronous Events in RTOS as Input to a Referenced Model” on page 6-28
- “Rate Transitions and Asynchronous Blocks” on page 6-34
- “Timers in Asynchronous Tasks” on page 6-38
- “Create a Customized Asynchronous Library” on page 6-40
- “Import Asynchronous Event Data for Simulation” on page 6-47
- “Asynchronous Support Limitations” on page 6-50

Asynchronous Events

Asynchronous Support

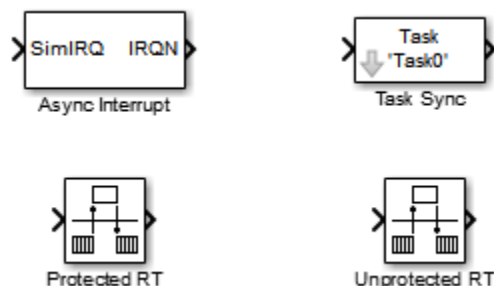
Normally, you time models from which you plan to generate code from a *periodic* interrupt source (for example, a hardware timer). Blocks in a periodically clocked single-rate model run at a timer interrupt rate (the base rate of the model). Blocks in a periodically clocked multirate model run at the base rate or at multiples of that rate.

Many systems must also support execution of blocks in response to events that are *asynchronous* with respect to the periodic timing source of the system. For example, a peripheral device might signal completion of an input operation by generating an interrupt. The system must service such interrupts, for example, by acquiring data from the interrupting device.

This topic explains how to use blocks to model and generate code for asynchronous event handling, including servicing of hardware-generated interrupts, maintenance of timers, asynchronous read and write operations, and spawning of asynchronous tasks under a real-time operating system (RTOS). This block library demonstrates integration with an example RTOS (VxWorks®). Although the blocks target an example RTOS, this chapter provides source code analysis and other information you can use to develop blocks that support asynchronous event handling for an alternative target RTOS.²

Block Library for Calls to an Example Real-Time Operating System

The next figure shows the blocks in the vxlib1 block library.



The key blocks in the library are the Async Interrupt and Task Sync blocks. These blocks are targeted for an example RTOS (VxWorks). You can use them, with modification, to support your RTOS applications.

Note Use the blocks in the Interrupt Templates block library (Async Interrupt and Task Sync) for simulation and code generation. These blocks provide starting point examples to help you develop custom blocks for a target environment.

To implement asynchronous support for an RTOS other than the example RTOS, use these guidelines and the example code in “Create a Customized Asynchronous Library” on page 6-40 to adapt the Interrupt Template library blocks for your RTOS.

² VxWorks is a registered trademark of Wind River® Systems, Inc.

The Interrupt Template library includes blocks that you can use to:

- Generate interrupt-level code — Async Interrupt block
- Spawn an RTOS task that calls a function call subsystem — Task Sync block

The use of protected and unprotected Rate Transition blocks in asynchronous contexts is discussed in “Rate Transitions and Asynchronous Blocks” on page 6-34. For general information on rate transitions, see “Time-Based Scheduling and Code Generation” on page 5-2.

Examples and Additional Information

- Example model `rtwdemo_async` uses the `tornado.tlc` system target file and Interrupt Template block library. To open the model, type `rtwdemo_async` at the MATLAB command prompt.
- Example model `rtwdemo_async_mdltreftop` uses the `tornado.tlc` system target file and Interrupt Template block library. To open the model, type `rtwdemo_async_mdltreftop` at the MATLAB command prompt.
- “Time-Based Scheduling and Code Generation” on page 5-2 discusses general multitasking and rate transition issues for periodic models.
- The Embedded Coder documentation discusses the `ert.tlc` system target file, including task execution and scheduling.
- For information about the system calls to the example RTOS (VxWorks) mentioned in this topic, see VxWorks system documentation on the Wind River website.

See Also

More About

- “Time-Based Scheduling and Code Generation” on page 5-2
- “Generate Interrupt Service Routines” on page 6-4
- “Spawn and Synchronize Execution of RTOS Task” on page 6-11
- “Pass Asynchronous Events in RTOS as Input to a Referenced Model” on page 6-28
- “Timers in Asynchronous Tasks” on page 6-38
- “Import Asynchronous Event Data for Simulation” on page 6-47
- “Rate Transitions and Asynchronous Blocks” on page 6-34
- “Create a Customized Asynchronous Library” on page 6-40
- “Asynchronous Support Limitations” on page 6-50

Generate Interrupt Service Routines

To generate an interrupt service routine (ISR) associated with a specific VME interrupt level for an example RTOS (for example, VxWorks), use the Async Interrupt block. The Async Interrupt block enables the specified interrupt level and installs an ISR that calls a connected function call subsystem.

You can also use the Async Interrupt block in a simulation. It provides an input port that you can enable and connect to a simulated interrupt source.

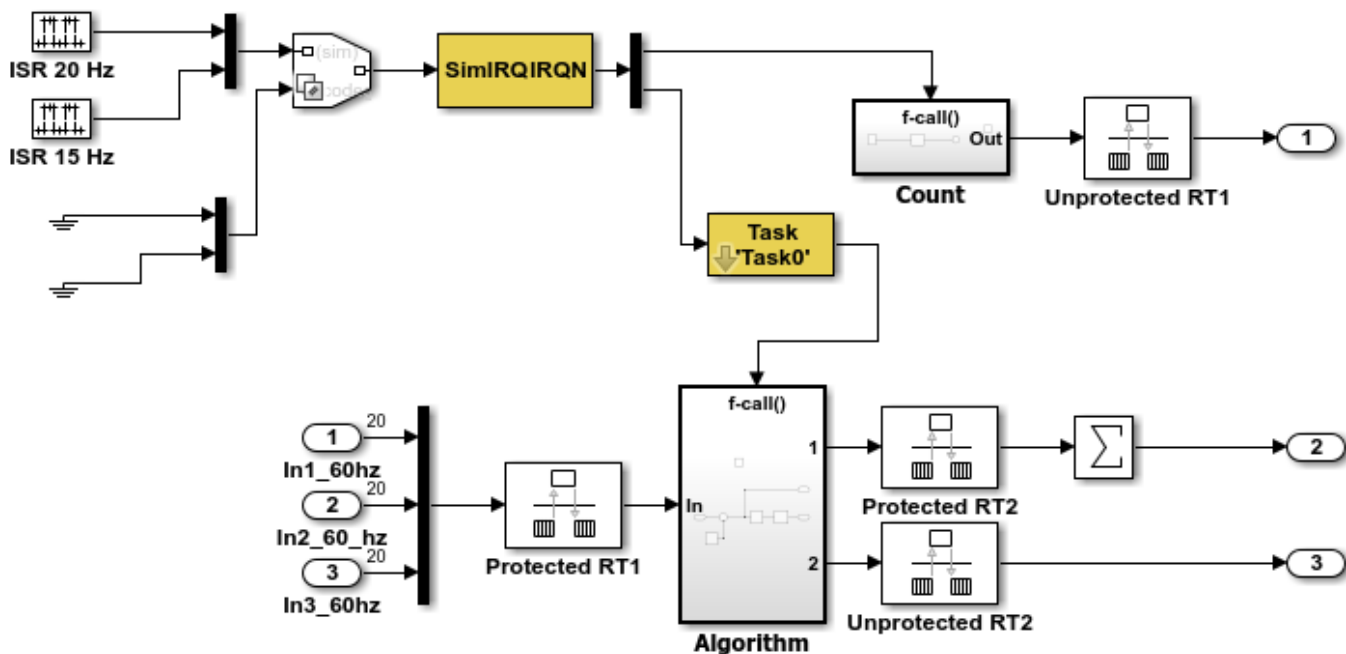
Note The operating system integration techniques that are demonstrated in this section use blocks that are in the Interrupt Templates block library. The blocks in that library provide starting point examples to help you develop custom blocks for a target environment.

Connecting the Async Interrupt Block

To generate an ISR, connect an output of the Async Interrupt block to the control input of:

- A function call subsystem
- The input of a Task Sync block
- The input to a Stateflow chart configured for a function call input event

Example model `rtwdemo_async` shows an Async Interrupt block that is configured to service two interrupt sources. Each of the two output signals is connected to a function call subsystem.



Requirements and Restrictions

- The Async Interrupt block supports VME interrupts 1 through 7.

- The Async Interrupt block uses these system calls to the example RTOS (VxWorks):
 - `sysIntEnable`
 - `sysIntDisable`
 - `intConnect`
 - `intLock`
 - `intUnlock`
 - `tickGet`

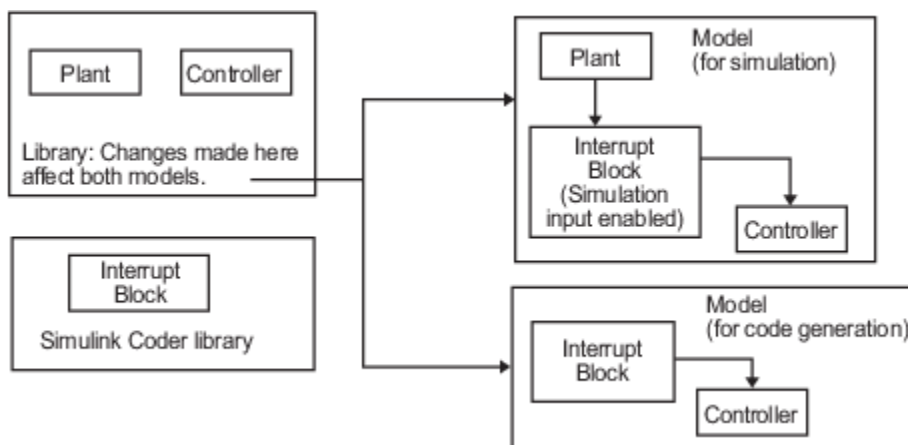
Performance Considerations

Execution of large subsystems at interrupt level can have a significant impact on interrupt response time for interrupts of equal and lower priority in the system. It is best to keep ISRs as short as possible. Connect only function call subsystems that contain a small number of blocks to an Async Interrupt block.

A better solution for large subsystems is to use the Task Sync block to synchronize the execution of the function call subsystem to a RTOS task. Place the Task Sync block between the Async Interrupt block and the function call subsystem. The Async Interrupt block installs the Task Sync block as the ISR. The ISR releases a synchronization semaphore (performs a `semGive`) to the task, and returns immediately from interrupt level. The task is then scheduled and run by the RTOS (VxWorks). See “Spawn and Synchronize Execution of RTOS Task” on page 6-11 for more information.

Dual-Model Approach for Developing Real-Time Systems that Include ISRs

When developing a real-time system that includes ISRs, consider using a *dual-model* approach. Develop one model that includes a plant and a controller for simulation, and another model that includes only the controller for code generation. By using a Simulink library, you can implement changes to both models simultaneously. This figure shows how changes made to the plant or controller, both of which are in a library, are propagated to the models.

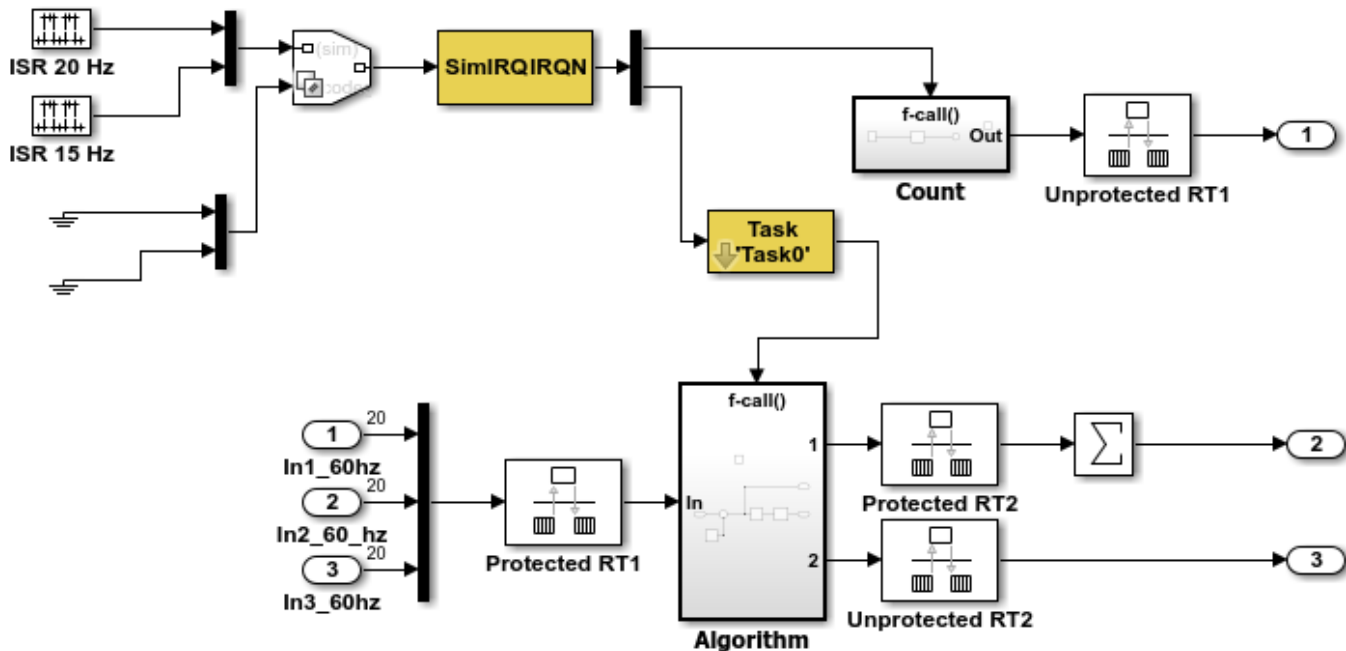


Dual-Model Use of Async Interrupt Block for Simulation and Code Generation

A *single-model* approach is also possible. The Plant component of the model is active only during simulation. For code generation, you switch out the Plant components out of the system and you generate code only for the interrupt block and controller parts of the model.

Dual-Model Approach: Simulation

Example model `rtwdemo_async` shows the dual-model approach to modeling. During simulation, the Pulse Generator blocks provide simulated interrupt signals.



During simulation:

- When the value of the input port is nonzero, Simulink imitates the behavior of each interrupt by calling the destination function unconditionally where the interrupt occurs.
- Subsystems connected to Async Interrupt block output ports execute in order of their priority in the RTOS (in this example, VxWorks). If two or more interrupt signals occur simultaneously, the Async Interrupt block executes the downstream systems in the order specified by their interrupt levels (level 7 gets the highest priority). The first input element maps to the first output element.

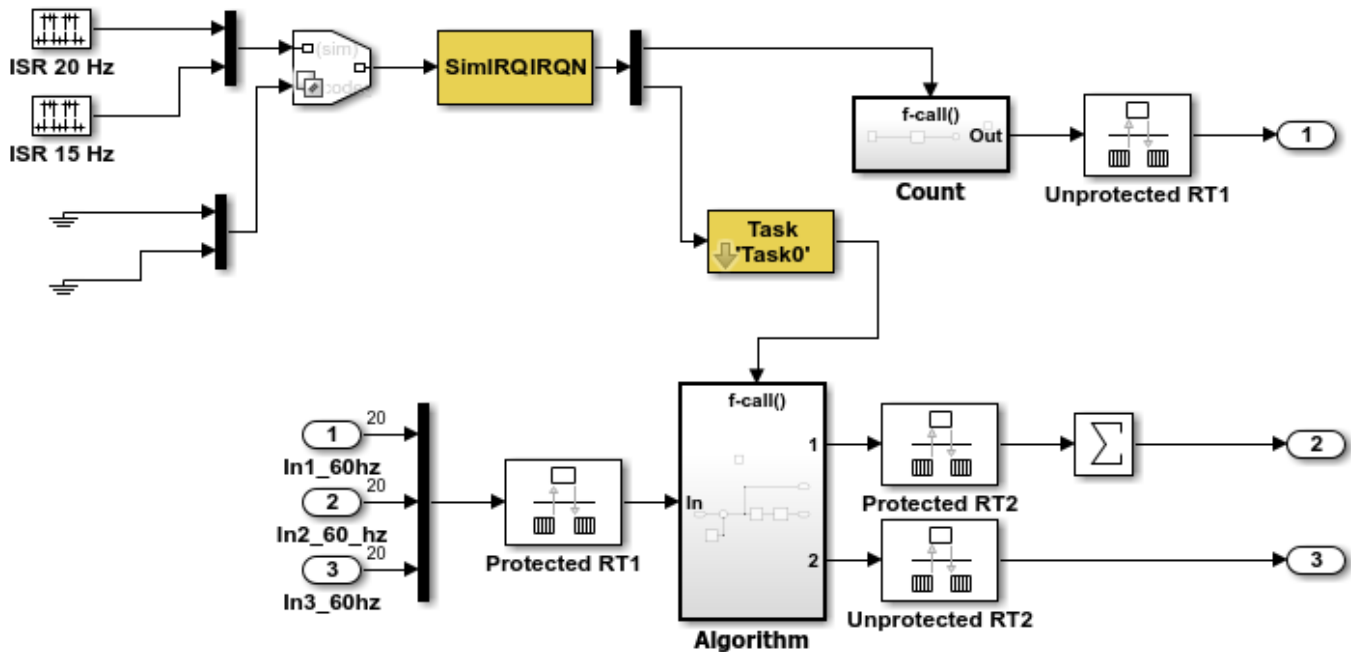
You can use the Async Interrupt block in a simulation without enabling the simulation input. The Async Interrupt block inherits the base rate of the model and calls the connected subsystems in order of their priorities in the RTOS. (In this case, the Async Interrupt block behaves as if inputs receive a 1 simultaneously.)

Dual-Model Approach: Code Generation

In the generated code for the example model:

- Ground blocks provide input signals to the Variant Source block
- The Async Interrupt block does not use its simulation input

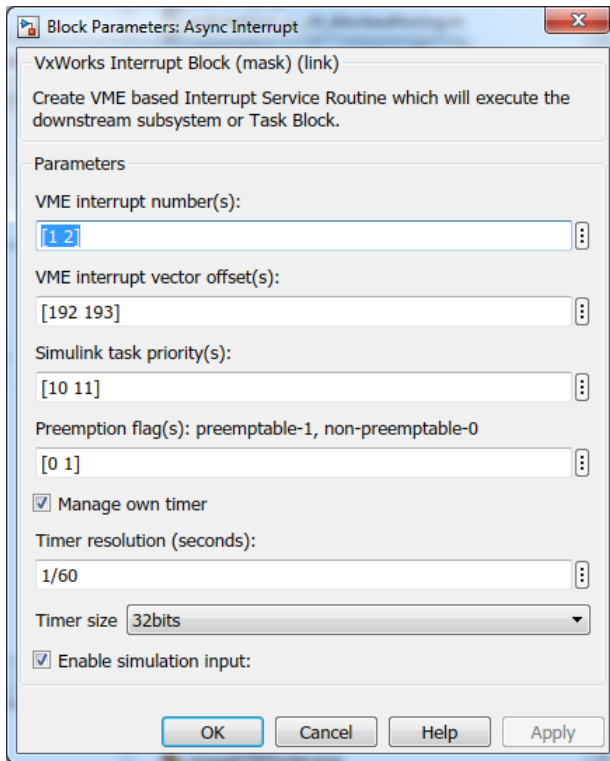
The Ground blocks drive control input of the Variant Source block, so the code generator does not produce code for that signal path. The code generator does not produce code for blocks that drive the simulation control input to the Variant Source block because that path is not selected during code generation. However, the sample times of driving blocks for the simulation input to the Variant Source block contribute to the sample times supported in the generated code. To avoid including unnecessary sample times in the generated code, use the sample times of the blocks driving the simulation input in the model where generated code is intended.



The code generator produces standalone functions for the ISRs and the interrupt vector table is as follows:

Offset	
192	<code>&isr_num1_vec192()</code>
193	<code>&isr_num2_vec193()</code>

Consider the code generated from this model, assuming that the Async Interrupt block parameters are configured as shown in this figure.



Initialization Code

During code generation, the Async Interrupt block installs interrupt code in the code generated for subsystem blocks as interrupt service routines. The interrupt vectors for IRQ1 and IRQ2 are stored at locations 192 and 193, relative to the base of the interrupt vector table, as specified by the **VME interrupt vector offset(s)** parameter.

Installing an ISR requires two RTOS (VxWorks) calls, `int_connect` and `sysInt_Enable`. The Async Interrupt block inserts these calls in the `mdl_initialize` function, as shown in this code fragment.

```
/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
/* Connect and enable ISR function: isr_num1_vec192 */
if( intConnect(INUM_TO_IVEC(192), isr_num1_vec192, 0) != OK) {
    printf("intConnect failed for ISR 1.\n");
}
sysIntEnable(1);

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
/* Connect and enable ISR function: isr_num2_vec193 */
if( intConnect(INUM_TO_IVEC(193), isr_num2_vec193, 0) != OK)
{
    printf("intConnect failed for ISR 2.\n");
}
sysIntEnable(2);
```

The Async Interrupt block does not configure the hardware that generates the interrupt. Typically, the interrupt source is a VME I/O board, which generates interrupts for specific events (for example, at the end of an analog-to-digital conversion). You set up the VME interrupt level and vector in registers or by using jumpers on the board. You can use the `mdlStart` routine of a user-written device driver (S-function) to set up the registers and enable interrupt generation on the board. You

must match the interrupt level and vector specified in the Async Interrupt block parameter dialog box to the level and vector set up on the I/O board.

Generated ISR Code

The actual ISR generated for IRQ1 in the RTOS (VxWorks) is listed below.

```
/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */

void isr_num1_vec192(void)
{
    int_T lock;
    FP_CONTEXT context;

    /* Use tickGet() as a portable tick counter example.
       A much higher resolution can be achieved with a
       hardware counter */
    Async_Code_M->Timing.clockTick2 = tickGet();

    /* disable interrupts (system is configured as non-ive) */
    lock = intLock();

    /* save floating point context */
    fppSave(&context);

    /* Call the system: <Root>/Subsystem A */
    Count(0, 0);

    /* restore floating point context */
    fppRestore(&context);

    /* re-enable interrupts */
    intUnlock(lock);
}

```

Note these features of the ISR:

- Because of the setting of the **Preemption Flag(s)** parameter, the ISR is locked; that is, it cannot be preempted by a higher priority interrupt. The ISR is locked and unlocked in the example RTOS (VxWorks) by calls to the functions `int_lock` and `int_unlock`.
- The connected subsystem, `Count`, is called from within the ISR.
- The `Count` function executes algorithmic (model) code. Therefore, the floating-point context is saved and restored across the call to `Count`.
- The ISR maintains its own absolute time counter, which is distinct from other periodic base rate or subrate counters in the system. Timing data is maintained for use by code generated for blocks executed within the ISR that require absolute or elapsed time.

See “Timers in Asynchronous Tasks” on page 6-38 for details.

Model Termination Code

The model termination function disables the interrupts in the RTOS (VxWorks):

```
/* Model terminate function */
void Async_Code_terminate(void)
{

```

```
/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
/* Disable interrupt for ISR system: isr_num1_vec192 */
sysIntDisable(1);

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
/* Disable interrupt for ISR system: isr_num2_vec193 */
sysIntDisable(2);
}
```

See Also

More About

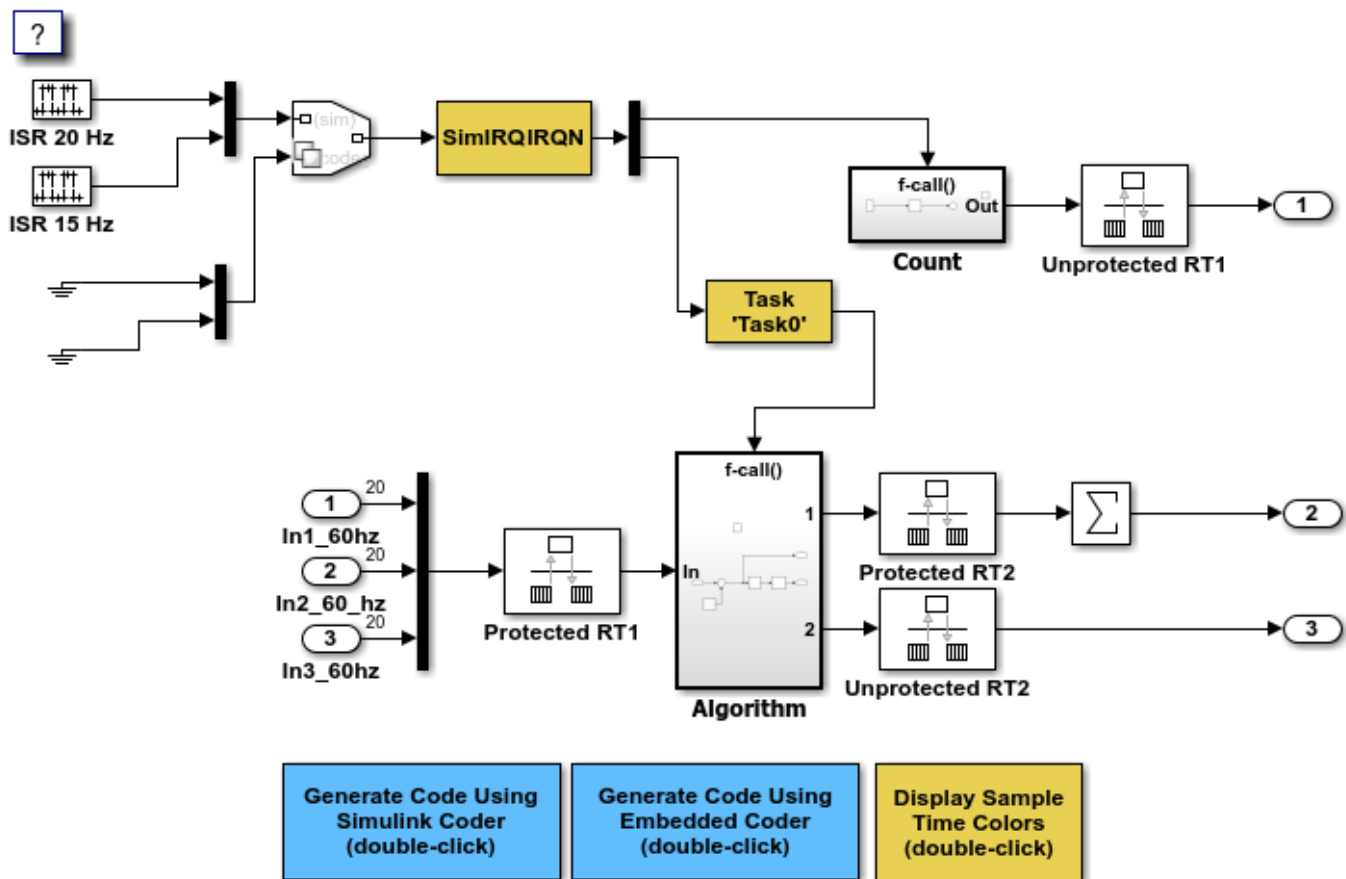
- “Spawn and Synchronize Execution of RTOS Task” on page 6-11
- “Pass Asynchronous Events in RTOS as Input to a Referenced Model” on page 6-28
- “Import Asynchronous Event Data for Simulation” on page 6-47
- “Rate Transitions and Asynchronous Blocks” on page 6-34

Spawn and Synchronize Execution of RTOS Task

This example shows how to simulate and generate code for asynchronous events on a real-time multitasking system. The model contains an Async Interrupt block, a Task Sync block, asynchronously executed subsystems Count and Algorithm, and Rate Transition blocks. The Async Interrupt block creates two Versa Module Eurocard (VME) interrupt service routines (ISRs) that pass interrupt signals to subsystem Count and the Task Sync block. Count executes at interrupt level. Algorithm executes as an asynchronous task. The code that the code generator produces for subsystems is tailored for the VxWorks® operating system.

About the Example Model

Open the example model `rtwdemo_async`.



Copyright 1994-2020 The MathWorks, Inc.

You can place an Async Interrupt block between a simulated interrupt source and one of these model elements:

- Function call subsystem
- Task Sync block

- A Stateflow® chart configured for a function call input event
- A referenced model with an Inport block that connects to one of the preceding model elements

The Async Interrupt and Task Sync blocks enable the subsystems to execute asynchronously. Reconfigure the Async Interrupt and Task Sync blocks to generate code for an alternative run-time environment.

Count represents a simple interrupt service routine (ISR) that executes at interrupt level. It is best to keep ISRs as simple as possible. This subsystem includes only a Discrete-Time Integrator block.

Algorithm includes more substance. It includes multiple blocks and produces two output values. Execution of larger subsystems at interrupt level can significantly impact response time for interrupts of equal and lower priority in the system. A better solution for larger subsystems is to use the Task Sync block to represent the ISR for the function-call subsystem.

The Async Interrupt block generates calls to ISRs. Place the block between a simulated interrupt source and one of the following:

- Function call subsystem
- Task Sync block
- A Stateflow chart configured for a function call input event

For each specified interrupt level, the block generates a Versa Module Eurocard (VME) ISR that executes the connected subsystem, Task Sync block, or chart.

In the example model, the Async Interrupt block is configured for VME interrupts 1 and 2, by using interrupt vector offsets 192 and 193. Interrupt 1 connects directly to subsystem **Count**. Interrupt 2 connects to a Task Sync block, which serves as the ISR for **Algorithm**. Place a Task Sync block in one of the following locations:

- Between an Async Interrupt block and a function-call subsystem or Stateflow chart.
- At the output port of a Stateflow chart that has an event, **Output to Simulink**, that you configure as a function call.

In the example model, the Task Sync block is between the Async Interrupt block and function-call subsystem **Algorithm**. The Task Sync block is configured with the task name **Task()**, a priority of 50, a stack size of 8192, and data transfers of the task synchronized with the caller task. The spawned task uses a semaphore to synchronize task execution. The Async Interrupt block triggers a release of the task semaphore.

Four Rate Transition blocks handle data transfers between ports that operate at different rates. In two instances, protected Rate Transition blocks protect data transfers (prevent them from being preempted and corrupted). In the other two instances, unprotected Rate Transition blocks introduce no special behavior. Their presence informs Simulink® of a rate transition.

The code generated for the Async Interrupt and Task Sync blocks is tailored for the example RTOS (VxWorks). However, you can modify the blocks to generate code specific to your run-time environment.

Data Transfer Assumptions

- Data transfers occur between one reading task and one writing task.
- A read or write operation on a byte-size variable is atomic.

- When two tasks interact, only one can preempt the other.
- For periodic tasks, the task with the faster rate has higher priority than the task with the slower rate. The task with the faster rate preempts the tasks with slower rates.
- Tasks run on a single processor. Time slicing is not allowed.
- Processes do not stop and restart, especially while data is being transferred between tasks.

Simulate the Model

Simulate the model. By default, the model is configured to show sample times in different colors. Discrete sample times for input and output appear red and green, respectively. Constants are reddish-blue. Asynchronous interrupts and tasks are purple. The Rate Transition Blocks, which are a hybrid rate (their input and output sample times can differ), are yellow.

Generate Code and Report

Generate code and a code generation report for the model. Generated code for the Async Interrupt and Task Sync blocks is for the example RTOS (VxWorks). However, you can modify the blocks to generate code for another run-time environment.

1. Create a temporary folder for the build and inspection process.
2. Build the model.

```
### Starting build procedure for: rtwdemo_async
Warning: Simulink Coder: The tornado.tlc target will be removed in a future
release.
### Successful completion of code generation for: rtwdemo_async
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_async	Code generated.	Code generation information file does not exist.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 16.539s
```

Review Initialization Code

Open the generated source file `rtwdemo_async.c`. The initialization code:

1. Creates and initializes the synchronization semaphore `Task0_semaphore`.

```
*(SEM_ID *)rtwdemo_async_DW.SFunction_PWORK.SemID = semBCreate(SEM_Q_PRIORITY,
SEM_EMPTY);
if (rtwdemo_async_DW.SFunction_PWORK.SemID == NULL) {
    printf("semBCreate call failed for block Task0.\n");
}
```

2. Spawns task `task0` and assigns the task priority 50.

```
rtwdemo_async_DW.SFunction_IWORK.TaskID = taskSpawn("Task0",
```

```

    50.0,
    VX_FP_TASK,
    8192.0,
    (FUNCPTR)Task0,
    0, 0, 0, 0, 0, 0, 0, 0, 0, 0);
if (rtwdemo_async_DW.SFunction_IWORK.TaskID == ERROR) {
    printf("taskSpawn call failed for block Task0.\n");
}

/* End of Start for S-Function (vxtask1): '<S4>/S-Function' */

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
/* Connect and enable ISR function: isr_num1_vec192 */
if (intConnect(INUM_TO_IVEC(192), isr_num1_vec192, 0) != OK) {
    printf("intConnect failed for ISR 1.\n");
}

sysIntEnable(1);

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
/* Connect and enable ISR function: isr_num2_vec193 */
if (intConnect(INUM_TO_IVEC(193), isr_num2_vec193, 0) != OK) {
    printf("intConnect failed for ISR 2.\n");
}

sysIntEnable(2);

```

3. Connects and enables ISR `isr_num1_vec192` for interrupt 1 and ISR `isr_num2_vec193` for interrupt 2.

```

{
    int32_T i;

    /* InitializeConditions for RateTransition: '<Root>/Protected RT1' */
    for (i = 0; i < 60; i++) {
        rtwdemo_async_DW.ProtectedRT1_Buffer[i] = 0.0;
    }

    /* End of InitializeConditions for RateTransition: '<Root>/Protected RT1' */

    /* InitializeConditions for RateTransition: '<Root>/Protected RT2' */
    for (i = 0; i < 60; i++) {
        rtwdemo_async_DW.ProtectedRT2_Buffer[i] = 0.0;
    }

    /* End of InitializeConditions for RateTransition: '<Root>/Protected RT2' */

    /* SystemInitialize for S-Function (vxinterrupt1): '<Root>/Async Interrupt' incorporates:
     * SubSystem: '<Root>/Count'
     */
    /* System initialize for function-call system: '<Root>/Count' */

    /* InitializeConditions for DiscreteIntegrator: '<S2>/Integrator' */
    rtwdemo_async_DW.Integrator_DSTATE_l = 0.0;
    rtwdemo_async_DW.Integrator_PREV_U_o = 0.0;

    /* SystemInitialize for Output: '<Root>/Out1' incorporates:

```

```

    *   Output: '<S2>/Out'
    */
    rtdemo_async_Y.Out1 = 0.0;

    /* SystemInitialize for S-Function (vxinterrupt1): '<Root>/Async Interrupt' incorporates:
    *   SubSystem: '<S3>/Subsystem'
    */
    /* System initialize for function-call system: '<S3>/Subsystem' */

    /* SystemInitialize for S-Function (vxtask1): '<S4>/S-Function' incorporates:
    *   SubSystem: '<Root>/Algorithm'
    */

    /* System initialize for function-call system: '<Root>/Algorithm' */
    rtdemo_async_M->Timing.clockTick2 = rtdemo_async_M->Timing.clockTick4;

    /* InitializeConditions for DiscreteIntegrator: '<S1>/Integrator' */
    rtdemo_async_DW.Integrator_DSTATE = 0.0;

    /* InitializeConditions for Sum: '<S1>/Sum1' incorporates:
    *   DiscreteIntegrator: '<S1>/Integrator'
    */
    rtdemo_async_DW.Integrator_PREV_U = 0.0;

    /* SystemInitialize for Sum: '<S1>/Sum' incorporates:
    *   Output: '<S1>/Out1'
    */
    memset(&rtdemo_async_B.Sum[0], 0, 60U * sizeof(real_T));

    /* SystemInitialize for Output: '<Root>/Out3' incorporates:
    *   Output: '<S1>/Out2'
    */
    rtdemo_async_Y.Out3 = 0.0;

    /* End of SystemInitialize for S-Function (vxtask1): '<S4>/S-Function' */

    /* End of SystemInitialize for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */
}

/* Enable for S-Function (vxinterrupt1): '<Root>/Async Interrupt' incorporates:
*   SubSystem: '<Root>/Count'
*/
/* Enable for function-call system: '<Root>/Count' */
rtdemo_async_DW.Count_RESET_ELAPS_T = true;

/* Enable for DiscreteIntegrator: '<S2>/Integrator' */
rtdemo_async_DW.Integrator_SYSTEM_ENABLE_h = 1U;

/* Enable for S-Function (vxinterrupt1): '<Root>/Async Interrupt' incorporates:
*   SubSystem: '<S3>/Subsystem'
*/

/* Enable for function-call system: '<S3>/Subsystem' */

/* Enable for S-Function (vxtask1): '<S4>/S-Function' incorporates:
*   SubSystem: '<Root>/Algorithm'
*/

```

```

/* Enable for function-call system: '<Root>/Algorithm' */
rtwdemo_async_M->Timing.clockTick2 = rtwdemo_async_M->Timing.clockTick4;
rtwdemo_async_DW.Algorithm_RESET_ELAPS_T = true;

/* Enable for DiscreteIntegrator: '<S1>/Integrator' */
rtwdemo_async_DW.Integrator_SYSTEM_ENABLE = 1U;

/* End of Enable for S-Function (vxtask1): '<S4>/S-Function' */

/* End of Enable for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */
}

/* Model terminate function */
static void rtwdemo_async_terminate(void)
{
    /* Terminate for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

    /* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
    /* Disable interrupt for ISR system: isr_num1_vec192 */
    sysIntDisable(1);

    /* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
    /* Disable interrupt for ISR system: isr_num2_vec193 */
    sysIntDisable(2);

    /* End of Terminate for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

    /* Terminate for S-Function (vxinterrupt1): '<Root>/Async Interrupt' incorporates:
    * SubSystem: '<S3>/Subsystem'
    */

    /* Termination for function-call system: '<S3>/Subsystem' */

    /* Terminate for S-Function (vxtask1): '<S4>/S-Function' */

    /* VxWorks Task Block: '<S4>/S-Function' (vxtask1) */
    /* Destroy task: Task0 */
    taskDelete(rtwdemo_async_DW.SFunction_IWORK.TaskID);

    /* End of Terminate for S-Function (vxtask1): '<S4>/S-Function' */

    /* End of Terminate for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */
}

/*=====
 * Start of Classic call interface
 *=====*/
void MdlOutputs(int_T tid)
{
    rtwdemo_async_output(tid);
}

void MdlUpdate(int_T tid)
{
    rtwdemo_async_update(tid);
}

void MdlInitializeSizes(void)

```

```

{
}

void MdlInitializeSampleTimes(void)
{
}

void MdlInitialize(void)
{
}

void MdlStart(void)
{
    rtwdemo_async_initialize();
}

void MdlTerminate(void)
{
    rtwdemo_async_terminate();
}

/* Registration function */
RT_MODEL_rtwdemo_async_T *rtwdemo_async(void)
{
    /* Registration code */

    /* initialize non-finites */
    rt_InitInfAndNaN(sizeof(real_T));

    /* initialize real-time model */
    (void) memset((void *)rtwdemo_async_M, 0,
                 sizeof(RT_MODEL_rtwdemo_async_T));

    /* Initialize timing info */
    {
        int_T *mdlTsMap = rtwdemo_async_M->Timing.sampleTimeTaskIDArray;
        mdlTsMap[0] = 0;
        mdlTsMap[1] = 1;

        /* polyspace +2 MISRA2012:D4.1 [Justified:Low] "rtwdemo_async_M points to
           static memory which is guaranteed to be non-NULL" */
        rtwdemo_async_M->Timing.sampleTimeTaskIDPtr = (&mdlTsMap[0]);
        rtwdemo_async_M->Timing.sampleTimes =
            (&rtwdemo_async_M->Timing.sampleTimesArray[0]);
        rtwdemo_async_M->Timing.offsetTimes =
            (&rtwdemo_async_M->Timing.offsetTimesArray[0]);

        /* task periods */
        rtwdemo_async_M->Timing.sampleTimes[0] = (0.016666666666666666);
        rtwdemo_async_M->Timing.sampleTimes[1] = (0.05);

        /* task offsets */
        rtwdemo_async_M->Timing.offsetTimes[0] = (0.0);
        rtwdemo_async_M->Timing.offsetTimes[1] = (0.0);
    }

    rtmSetTPtr(rtwdemo_async_M, &rtwdemo_async_M->Timing.tArray[0]);
}

```

```

{
  int_T *mdlSampleHits = rtwdemo_async_M->Timing.sampleHitArray;
  int_T *mdlPerTaskSampleHits = rtwdemo_async_M->Timing.perTaskSampleHitsArray;
  rtwdemo_async_M->Timing.perTaskSampleHits = (&mdlPerTaskSampleHits[0]);
  mdlSampleHits[0] = 1;
  rtwdemo_async_M->Timing.sampleHits = (&mdlSampleHits[0]);
}

rtmSetTFinal(rtwdemo_async_M, 0.5);
rtwdemo_async_M->Timing.stepSize0 = 0.016666666666666666;
rtwdemo_async_M->Timing.stepSize1 = 0.05;
rtwdemo_async_M->solverInfoPtr = (&rtwdemo_async_M->solverInfo);
rtwdemo_async_M->Timing.stepSize = (0.016666666666666666);
rtsiSetFixedStepSize(&rtwdemo_async_M->solverInfo, 0.016666666666666666);
rtsiSetSolverMode(&rtwdemo_async_M->solverInfo, SOLVER_MODE_MULTITASKING);

/* block I/O */
rtwdemo_async_M->blockIO = ((void *) &rtwdemo_async_B);
(void) memset(((void *) &rtwdemo_async_B), 0,
              sizeof(B_rtwdemo_async_T));

/* states (dwork) */
rtwdemo_async_M->dwork = ((void *) &rtwdemo_async_DW);
(void) memset((void *)&rtwdemo_async_DW, 0,
              sizeof(DW_rtwdemo_async_T));

/* external inputs */
rtwdemo_async_M->inputs = (((void*)&rtwdemo_async_U));
(void)memset(&rtwdemo_async_U, 0, sizeof(ExtU_rtwdemo_async_T));

/* external outputs */
rtwdemo_async_M->outputs = (&rtwdemo_async_Y);
(void)memset(&rtwdemo_async_Y, 0, sizeof(ExtY_rtwdemo_async_T));

/* Initialize Sizes */
rtwdemo_async_M->Sizes.numContStates = (0); /* Number of continuous states */
rtwdemo_async_M->Sizes.numY = (3); /* Number of model outputs */
rtwdemo_async_M->Sizes.numU = (60); /* Number of model inputs */
rtwdemo_async_M->Sizes.sysDirFeedThru = (1); /* The model is direct feedthrough */
rtwdemo_async_M->Sizes.numSampTimes = (2); /* Number of sample times */
rtwdemo_async_M->Sizes.numBlocks = (19); /* Number of blocks */
rtwdemo_async_M->Sizes.numBlockIO = (6); /* Number of block outputs */
return rtwdemo_async_M;
}

/*=====
 * End of Classic call interface
 *=====*/

```

The order of these operations is important. Before the code generator enables the interrupt that activates the task, it must spawn the task.

Review Task and Task Synchronization Code

In the generated source file `rtwdemo_async.c`, review the task and task synchronization code.

The code generator produces the code for function `Task0` from the Task Sync block. That function includes a small amount of interrupt-level code and runs as an RTOS task.

The task waits in an infinite for loop until the system releases a synchronization semaphore. If the system releases the semaphore, the function updates its task timer and calls the code generated for the Algorithm subsystem.

In the example model, the **Synchronize the data transfer of this task with the caller task** parameter for the Task Sync block is set. This parameter setting updates the timer associated with the Task Sync block (rtM->Timing.clockTick2) with the value of the timer that the Async Interrupt block (rtM->Timing.clockTick3) maintains. As a result, code for blocks within the Algorithm subsystem use timer values that are based on the time of the most recent interrupt, rather than the most recent activation of Task0.

```
{
/* Wait for semaphore to be released by system: rtdemo_async/Task Sync */
for (;;) {
    if (semTake(*(SEM_ID *)rtdemo_async_DW.SFunction_PWORK.SemID,NO_WAIT) !=
        ERROR) {
        logMsg("Rate for Task Task0() too fast.\n",0,0,0,0,0,0);

#ifdef STOPONOVERRUN

        logMsg("Aborting real-time simulation.\n",0,0,0,0,0,0);
        semGive(stopSem);
        return(ERROR);

#endif

    } else {
        semTake(*(SEM_ID *)rtdemo_async_DW.SFunction_PWORK.SemID, WAIT_FOREVER);
    }

/* Use the upstream clock tick counter for this Task. */
rtdemo_async_M->Timing.clockTick2 = rtdemo_async_M->Timing.clockTick4;

/* Call the system: '<Root>/Algorithm' */
{
    int32_T i;
    int32_T i_0;

/* RateTransition: '<Root>/Protected RT1' */
i = rtdemo_async_DW.ProtectedRT1_ActiveBufIdx * 60;
for (i_0 = 0; i_0 < 60; i_0++) {
    rtdemo_async_B.ProtectedRT1[i_0] =
        rtdemo_async_DW.ProtectedRT1_Buffer[i_0 + i];
}

/* End of RateTransition: '<Root>/Protected RT1' */

/* S-Function (vxtask1): '<S4>/S-Function' */

/* Output and update for function-call system: '<Root>/Algorithm' */
{
    real_T Sum;
    int32_T i;
    rtdemo_async_M->Timing.clockTick2 = rtdemo_async_M->Timing.clockTick4;
    if (rtdemo_async_DW.Algorithm_RESET_ELAPS_T) {
        rtdemo_async_DW.Algorithm_ELAPS_T = 0U;
    }
}
}
}
```

```

} else {
    rtwdemo_async_DW.Algorithm_ELAPS_T =
        rtwdemo_async_M->Timing.clockTick2 -
        rtwdemo_async_DW.Algorithm_PREV_T;
}

rtwdemo_async_DW.Algorithm_PREV_T = rtwdemo_async_M->Timing.clockTick2;
rtwdemo_async_DW.Algorithm_RESET_ELAPS_T = false;

/* DiscreteIntegrator: '<S1>/Integrator' */
if (rtwdemo_async_DW.Integrator_SYSTEM_ENABLE == 0) {
    /* DiscreteIntegrator: '<S1>/Integrator' */
    rtwdemo_async_DW.Integrator_DSTATE += 0.016666666666666666 * (real_T)
        rtwdemo_async_DW.Algorithm_ELAPS_T
        * rtwdemo_async_DW.Integrator_PREV_U;
}

/* End of DiscreteIntegrator: '<S1>/Integrator' */

/* Output: '<Root>/Out3' incorporates:
 * SignalConversion generated from: '<S1>/Out2'
 */
rtwdemo_async_Y.Out3 = rtwdemo_async_DW.Integrator_DSTATE;

/* Sum: '<S1>/Sum1' */
rtwdemo_async_DW.Integrator_PREV_U = -0.0;
for (i = 0; i < 60; i++) {
    /* Sum: '<S1>/Sum' incorporates:
     * Constant: '<S1>/Offset'
     */
    Sum = rtwdemo_async_B.ProtectedRT1[i] + 1.25;
    rtwdemo_async_B.Sum[i] = Sum;

    /* Sum: '<S1>/Sum1' */
    rtwdemo_async_DW.Integrator_PREV_U += Sum;
}

/* Update for DiscreteIntegrator: '<S1>/Integrator' */
rtwdemo_async_DW.Integrator_SYSTEM_ENABLE = 0U;
}

/* End of Outputs for S-Function (vxtask1): '<S4>/S-Function' */

/* RateTransition: '<Root>/Protected RT2' */
for (i = 0; i < 60; i++) {
    rtwdemo_async_DW.ProtectedRT2_Buffer[i +
        (rtwdemo_async_DW.ProtectedRT2_ActiveBufIdx == 0) * 60] =
        rtwdemo_async_B.Sum[i];
}

rtwdemo_async_DW.ProtectedRT2_ActiveBufIdx = (int8_T)
    (rtwdemo_async_DW.ProtectedRT2_ActiveBufIdx == 0);

/* End of RateTransition: '<Root>/Protected RT2' */
}
}
}

```



```

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
void isr_num1_vec192(void)
{
    int_T lock;
    FP_CONTEXT context;

    /* Use tickGet() as a portable tick
       counter example. A much higher resolution can
       be achieved with a hardware counter */
    rtwdemo_async_M->Timing.clockTick3 = tickGet();

    /* disable interrupts (system is configured as non-preemptive) */
    lock = intLock();

    /* save floating point context */
    fppSave(&context);

    /* Call the system: '<Root>/Count' */
    {
        /* S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

        /* Output and update for function-call system: '<Root>/Count' */
        if (rtwdemo_async_DW.Count_RESET_ELAPS_T) {
            rtwdemo_async_DW.Count_ELAPS_T = 0U;
        } else {
            rtwdemo_async_DW.Count_ELAPS_T = rtwdemo_async_M->Timing.clockTick3 -
                rtwdemo_async_DW.Count_PREV_T;
        }

        rtwdemo_async_DW.Count_PREV_T = rtwdemo_async_M->Timing.clockTick3;
        rtwdemo_async_DW.Count_RESET_ELAPS_T = false;

        /* DiscreteIntegrator: '<S2>/Integrator' */
        if (rtwdemo_async_DW.Integrator_SYSTEM_ENABLE_h == 0) {
            /* DiscreteIntegrator: '<S2>/Integrator' */
            rtwdemo_async_DW.Integrator_DSTATE_l += 0.016666666666666666 * (real_T)
                rtwdemo_async_DW.Count_ELAPS_T * rtwdemo_async_DW.Integrator_PREV_U_o;
        }

        /* End of DiscreteIntegrator: '<S2>/Integrator' */

        /* Output: '<Root>/Out1' incorporates:
           * SignalConversion generated from: '<S2>/Out'
           */
        rtwdemo_async_Y.Out1 = rtwdemo_async_DW.Integrator_DSTATE_l;

        /* Update for DiscreteIntegrator: '<S2>/Integrator' incorporates:
           * Constant: '<S2>/Constant'
           */
        rtwdemo_async_DW.Integrator_SYSTEM_ENABLE_h = 0U;
        rtwdemo_async_DW.Integrator_PREV_U_o = 1.0;

        /* End of Outputs for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */
    }

    /* restore floating point context */
    fppRestore(&context);
}

```

```

    /* re-enable interrupts */
    intUnlock(lock);
}

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
void isr_num2_vec193(void)
{
    /* Use tickGet() as a portable tick
       counter example. A much higher resolution can
       be achieved with a hardware counter */
    rtdemo_async_M->Timing.clockTick4 = tickGet();

    /* Call the system: '<S3>/Subsystem' */
    {
        /* S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

        /* Output and update for function-call system: '<S3>/Subsystem' */

        /* S-Function (vxtask1): '<S4>/S-Function' */

        /* VxWorks Task Block: '<S4>/S-Function' (vxtask1) */
        /* Release semaphore for system task: Task0 */
        semGive(*(SEM_ID *)rtdemo_async_DW.SFunction_PWORK.SemID);

        /* End of Outputs for S-Function (vxtask1): '<S4>/S-Function' */

        /* End of Outputs for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */
    }
}

```

The code generator produces code for ISRs `isr_num1_vec192` and `isr_num2_vec293`. ISR `isr_num2_vec192`:

- Disables interrupts.
- Saves floating-point context.
- Calls the code generated for the subsystem that connects to the referenced model Inport block, which receives the interrupt.
- Restores floating-point context.
- Reenables interrupts.

```

void isr_num1_vec192(void)
{
    int_T lock;
    FP_CONTEXT context;

    /* Use tickGet() as a portable tick
       counter example. A much higher resolution can
       be achieved with a hardware counter */
    rtdemo_async_M->Timing.clockTick3 = tickGet();

    /* disable interrupts (system is configured as non-preemptive) */
    lock = intLock();

    /* save floating point context */
}

```

```

fppSave(&context);

/* Call the system: '<Root>/Count' */
{
  /* S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

  /* Output and update for function-call system: '<Root>/Count' */
  if (rtwdemo_async_DW.Count_RESET_ELAPS_T) {
    rtwdemo_async_DW.Count_ELAPS_T = 0U;
  } else {
    rtwdemo_async_DW.Count_ELAPS_T = rtwdemo_async_M->Timing.clockTick3 -
      rtwdemo_async_DW.Count_PREV_T;
  }

  rtwdemo_async_DW.Count_PREV_T = rtwdemo_async_M->Timing.clockTick3;
  rtwdemo_async_DW.Count_RESET_ELAPS_T = false;

  /* DiscreteIntegrator: '<S2>/Integrator' */
  if (rtwdemo_async_DW.Integrator_SYSTEM_ENABLE_h == 0) {
    /* DiscreteIntegrator: '<S2>/Integrator' */
    rtwdemo_async_DW.Integrator_DSTATE_l += 0.01666666666666666 * (real_T)
      rtwdemo_async_DW.Count_ELAPS_T * rtwdemo_async_DW.Integrator_PREV_U_o;
  }

  /* End of DiscreteIntegrator: '<S2>/Integrator' */

  /* Outport: '<Root>/Out1' incorporates:
   * SignalConversion generated from: '<S2>/Out'
   */
  rtwdemo_async_Y.Out1 = rtwdemo_async_DW.Integrator_DSTATE_l;

  /* Update for DiscreteIntegrator: '<S2>/Integrator' incorporates:
   * Constant: '<S2>/Constant'
   */
  rtwdemo_async_DW.Integrator_SYSTEM_ENABLE_h = 0U;
  rtwdemo_async_DW.Integrator_PREV_U_o = 1.0;

  /* End of Outputs for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */
}

/* restore floating point context */
fppRestore(&context);

/* re-enable interrupts */
intUnlock(lock);
}

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */

```

ISR `isr_num2_vec293` maintains a timer that stores the tick count at the time that the interrupt occurs. After updating the timer, the ISR releases the semaphore that activates Task0.

```

/* Spawned with priority: 50 */
void Task0(void)
{
  /* Wait for semaphore to be released by system: rtwdemo_async/Task Sync */
  for (;;) {

```

```

if (semTake(*(SEM_ID *)rtwdemo_async_DW.SFunction_PWORK.SemID,NO_WAIT) !=
    ERROR) {
    logMsg("Rate for Task Task0() too fast.\n",0,0,0,0,0,0);
}

#if STOPONOVERRUN

    logMsg("Aborting real-time simulation.\n",0,0,0,0,0,0);
    semGive(stopSem);
    return(ERROR);

#endif

} else {
    semTake(*(SEM_ID *)rtwdemo_async_DW.SFunction_PWORK.SemID, WAIT_FOREVER);
}

/* Use the upstream clock tick counter for this Task. */
rtwdemo_async_M->Timing.clockTick2 = rtwdemo_async_M->Timing.clockTick4;

/* Call the system: '<Root>/Algorithm' */
{
    int32_T i;
    int32_T i_0;

    /* RateTransition: '<Root>/Protected RT1' */
    i = rtwdemo_async_DW.ProtectedRT1_ActiveBufIdx * 60;
    for (i_0 = 0; i_0 < 60; i_0++) {
        rtwdemo_async_B.ProtectedRT1[i_0] =
            rtwdemo_async_DW.ProtectedRT1_Buffer[i_0 + i];
    }

    /* End of RateTransition: '<Root>/Protected RT1' */

    /* S-Function (vxtask1): '<S4>/S-Function' */

    /* Output and update for function-call system: '<Root>/Algorithm' */
    {
        real_T Sum;
        int32_T i;
        rtwdemo_async_M->Timing.clockTick2 = rtwdemo_async_M->Timing.clockTick4;
        if (rtwdemo_async_DW.Algorithm_RESET_ELAPS_T) {
            rtwdemo_async_DW.Algorithm_ELAPS_T = 0U;
        } else {
            rtwdemo_async_DW.Algorithm_ELAPS_T =
                rtwdemo_async_M->Timing.clockTick2 -
                rtwdemo_async_DW.Algorithm_PREV_T;
        }

        rtwdemo_async_DW.Algorithm_PREV_T = rtwdemo_async_M->Timing.clockTick2;
        rtwdemo_async_DW.Algorithm_RESET_ELAPS_T = false;

        /* DiscreteIntegrator: '<S1>/Integrator' */
        if (rtwdemo_async_DW.Integrator_SYSTEM_ENABLE == 0) {
            /* DiscreteIntegrator: '<S1>/Integrator' */
            rtwdemo_async_DW.Integrator_DSTATE += 0.016666666666666666 * (real_T)
                rtwdemo_async_DW.Algorithm_ELAPS_T
                * rtwdemo_async_DW.Integrator_PREV_U;
        }
    }
}

```

```

    /* End of DiscreteIntegrator: '<S1>/Integrator' */

    /* Outport: '<Root>/Out3' incorporates:
     * SignalConversion generated from: '<S1>/Out2'
     */
    rtwdemo_async_Y.Out3 = rtwdemo_async_DW.Integrator_DSTATE;

    /* Sum: '<S1>/Sum1' */
    rtwdemo_async_DW.Integrator_PREV_U = -0.0;
    for (i = 0; i < 60; i++) {
        /* Sum: '<S1>/Sum' incorporates:
         * Constant: '<S1>/Offset'
         */
        Sum = rtwdemo_async_B.ProtectedRT1[i] + 1.25;
        rtwdemo_async_B.Sum[i] = Sum;

        /* Sum: '<S1>/Sum1' */
        rtwdemo_async_DW.Integrator_PREV_U += Sum;
    }

    /* Update for DiscreteIntegrator: '<S1>/Integrator' */
    rtwdemo_async_DW.Integrator_SYSTEM_ENABLE = 0U;
}

/* End of Outputs for S-Function (vxtask1): '<S4>/S-Function' */

/* RateTransition: '<Root>/Protected RT2' */
for (i = 0; i < 60; i++) {
    rtwdemo_async_DW.ProtectedRT2_Buffer[i +
        (rtwdemo_async_DW.ProtectedRT2_ActiveBufIdx == 0) * 60] =
        rtwdemo_async_B.Sum[i];
}

rtwdemo_async_DW.ProtectedRT2_ActiveBufIdx = (int8_T)
    (rtwdemo_async_DW.ProtectedRT2_ActiveBufIdx == 0);

/* End of RateTransition: '<Root>/Protected RT2' */
}
}
}

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
void isr_num1_vec192(void)
{
    int_T lock;
    FP_CONTEXT context;

    /* Use tickGet() as a portable tick
     counter example. A much higher resolution can
     be achieved with a hardware counter */
    rtwdemo_async_M->Timing.clockTick3 = tickGet();

    /* disable interrupts (system is configured as non-preemptive) */
    lock = intLock();

    /* save floating point context */
    fppSave(&context);
}

```

```

/* Call the system: '<Root>/Count' */
{
  /* S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

  /* Output and update for function-call system: '<Root>/Count' */
  if (rtwdemo_async_DW.Count_RESET_ELAPS_T) {
    rtwdemo_async_DW.Count_ELAPS_T = 0U;
  } else {
    rtwdemo_async_DW.Count_ELAPS_T = rtwdemo_async_M->Timing.clockTick3 -
      rtwdemo_async_DW.Count_PREV_T;
  }

  rtwdemo_async_DW.Count_PREV_T = rtwdemo_async_M->Timing.clockTick3;
  rtwdemo_async_DW.Count_RESET_ELAPS_T = false;

  /* DiscreteIntegrator: '<S2>/Integrator' */
  if (rtwdemo_async_DW.Integrator_SYSTEM_ENABLE_h == 0) {
    /* DiscreteIntegrator: '<S2>/Integrator' */
    rtwdemo_async_DW.Integrator_DSTATE_l += 0.016666666666666666 * (real_T)
      rtwdemo_async_DW.Count_ELAPS_T * rtwdemo_async_DW.Integrator_PREV_U_o;
  }

  /* End of DiscreteIntegrator: '<S2>/Integrator' */

  /* Output: '<Root>/Out1' incorporates:
   * SignalConversion generated from: '<S2>/Out'
   */
  rtwdemo_async_Y.Out1 = rtwdemo_async_DW.Integrator_DSTATE_l;

  /* Update for DiscreteIntegrator: '<S2>/Integrator' incorporates:
   * Constant: '<S2>/Constant'
   */
  rtwdemo_async_DW.Integrator_SYSTEM_ENABLE_h = 0U;
  rtwdemo_async_DW.Integrator_PREV_U_o = 1.0;

  /* End of Outputs for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */
}

/* restore floating point context */
fppRestore(&context);

/* re-enable interrupts */
intUnlock(lock);
}

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */

```

Review Task Termination Code

The Task Sync block generates the following termination code.

```

static void rtwdemo_async_terminate(void)
{
  /* Terminate for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

  /* VxWorks Interrupt Block: '<Root>/Async Interrupt' */

```

```

/* Disable interrupt for ISR system: isr_num1_vec192 */
sysIntDisable(1);

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
/* Disable interrupt for ISR system: isr_num2_vec193 */
sysIntDisable(2);

/* End of Terminate for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

/* Terminate for S-Function (vxinterrupt1): '<Root>/Async Interrupt' incorporates:
 * SubSystem: '<S3>/Subsystem'
 */

/* Termination for function-call system: '<S3>/Subsystem' */

/* Terminate for S-Function (vxtask1): '<S4>/S-Function' */

/* VxWorks Task Block: '<S4>/S-Function' (vxtask1) */
/* Destroy task: Task0 */
taskDelete(rtwdemo_async_DW.SFunction_IWORK.TaskID);

/* End of Terminate for S-Function (vxtask1): '<S4>/S-Function' */

/* End of Terminate for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */
}

```

Related Information

- Async Interrupt
- Task Sync
- “Generate Interrupt Service Routines” on page 6-4
- “Timers in Asynchronous Tasks” on page 6-38
- “Create a Customized Asynchronous Library” on page 6-40
- “Import Asynchronous Event Data for Simulation” on page 6-47
- “Load Data to Root-Level Input Ports”
- “Asynchronous Events” on page 6-2
- “Rate Transitions and Asynchronous Blocks” on page 6-34
- “Asynchronous Support Limitations” on page 6-50

See Also

More About

- “Generate Interrupt Service Routines” on page 6-4
- “Pass Asynchronous Events in RTOS as Input to a Referenced Model” on page 6-28
- “Timers in Asynchronous Tasks” on page 6-38
- “Import Asynchronous Event Data for Simulation” on page 6-47
- “Rate Transitions and Asynchronous Blocks” on page 6-34

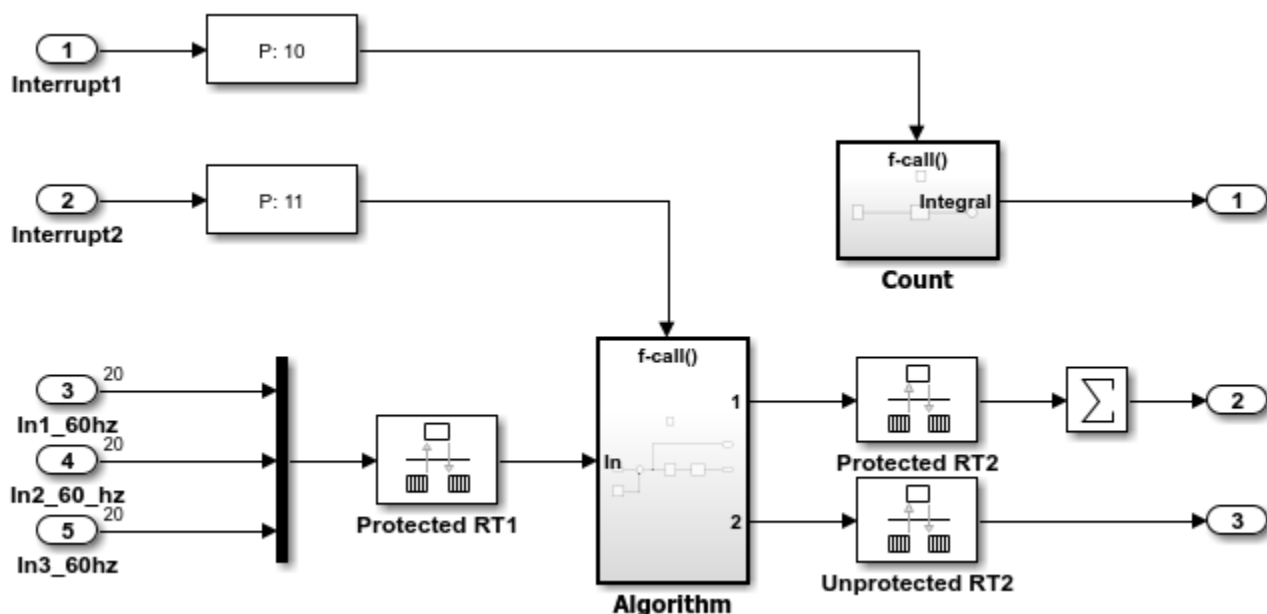
Pass Asynchronous Events in RTOS as Input to a Referenced Model

This example shows how to simulate and generate code for asynchronous events on a real-time multitasking system.

About the Example Model

Open the example model `rtwdemo_async_mdltreftop`.

```
open_system('rtwdemo_async_mdltreftop');
```



The model simulates an interrupt source and includes an Async Interrupt block and referenced model. The Async Interrupt block creates two Versa Module Eurocard (VME) interrupt service routines (ISRs) that pass interrupt signals to Inport blocks 1 and 2 of the referenced model.

You can place an Async Interrupt block between a simulated interrupt source and one of these model elements:

- Function call subsystem
- Task Sync block
- A Stateflow® chart configured for a function call input event
- A referenced model with an Inport block that connects to one of the preceding model elements

In this example model, the Async Interrupt block passes asynchronous events (function-call trigger signals), Interrupt1 and Interrupt2, to the referenced model through Inport blocks 1 and 2.

The code generator produces code that is tailored for the VxWorks operating system. Reconfigure the Async Interrupt block to generate code for an alternative application run-time environment.

Open the referenced model. The referenced model includes the two `Inport` blocks that receive the interrupts, each connected to an `Asynchronous Task Specification` block, function-call subsystems `Count` and `Algorithm`, and `Rate Transition` blocks. The `Asynchronous Task Specification` block, in combination with a root-level `Inport` block, allows a reference model to receive asynchronous function-call input. To use the block:

- 1 Connect the `Asynchronous Task Specification` block to the output port of the root-level `Inport` block that outputs a function-call trigger.
- 2 Select the **Output function call** parameter of the `Inport` block to specify that it accepts function-call signals.
- 3 On the parameters dialog box for the `Asynchronous Task Specification` block, set the task priority for the asynchronous task associated with an `Inport` block. Specify an integer or []. If you specify an integer, it must match the priority of the interrupt initiated by the `Async Interrupt` block in the parent model. If you specify [], the priorities do not have to match.

The `Asynchronous Task Specification` block for the higher priority interrupt, `interrupt1`, connect to function-call subsystem `Count`. `Count` represents a simple interrupt service routine (ISR). The second `Asynchronous Task Specification` block connect to the subsystem `Algorithm`, which includes more substance. It includes multiple blocks and produces two output values. Both subsystems execute at interrupt level.

For each interrupt level specified for the `Async Interrupt` block in the parent model, the block generates a VME ISR that executes the connected subsystem, `Task Sync` block, or chart.

In the example top model, the `Async Interrupt` block is configured for VME interrupts 1 and 2, using interrupt vector offsets 192 and 193. Interrupt 1 is wired to trigger subsystem `Count`. Interrupt 2 is wired to trigger subsystem `Algorithm`.

The `Rate Transition` blocks handle data transfers between ports that operate at different rates. In two instances, the blocks protect data transfers (prevent them from being preempted and corrupted). In the other instance, no special behavior occurs.

Data Transfer Assumptions

- Data transfers occur between one reading task and one writing task.
- A read or write operation on a byte-sized variable is atomic.
- When two tasks interact, only one can preempt the other.
- For periodic tasks, the task with the faster rate has higher priority than the task with the slower rate. The task with the faster rate preempts the tasks with slower rates.
- Tasks run on a single processor. Time slicing is not allowed.
- Processes do not crash and restart, especially while data is being transferred between tasks.

Simulate the Model

Simulate the model. By default, the model is configured to show sample times in different colors. Discrete sample times for input and output appear red and green, respectively. Constants are magenta. Asynchronous interrupts are purple. The `Rate Transition` blocks, which are hybrid (input and output sample times can differ), appear yellow.

Generate Code and Report

Generate code and a code generation report for the model. Async Interrupt block and Task Sync block generated code is for the example RTOS (VxWorks). However, you can modify the blocks to generate code for another run-time environment.

1. Create a temporary folder for the build and inspection process.

```
currentDir = pwd;
[~,cgDir] = rtwdemodir();
```

2. Build the model.

```
slbuild('rtwdemo_async_mdltreftop');
### Starting serial model reference code generation build.
Warning: Simulink Coder: The tornado.tlc target will be removed in a future release.
### Successfully updated the model reference code generation target for: rtwdemo_async_mdltreftop
### Starting build procedure for: rtwdemo_async_mdltreftop
Warning: Simulink Coder: The tornado.tlc target will be removed in a future release.
### Successful completion of code generation for: rtwdemo_async_mdltreftop

Build Summary

Code generation targets built:

Model          Action          Rebuild Reason
=====
rtwdemo_async_mdltreftop  Code generated.  rtwdemo_async_mdltreftop.c does not exist.

Top model targets built:

Model          Action          Rebuild Reason
=====
rtwdemo_async_mdltreftop  Code generated.  Code generation information file does not exist.

2 of 2 models built (0 models already up to date)
Build duration: 0h 1m 10.11s
```

Review Initialization Code

Open the generated source file `rtwdemo_async_mdltreftop.c`. The initialization code connects and enables ISR `isr_num1_vec192` for interrupt 1 and ISR `isr_num2_vec193` for interrupt 2.

```
cfile = fullfile(cgDir, 'rtwdemo_async_mdltreftop_tornado_rtw', 'rtwdemo_async_mdltreftop.c');
rtwdemodbtype(cfile, ...
    'static void rtwdemo_async_mdltreftop_initialize(void)', ...
    '/* Model terminate function */', ...
    1, 0);

static void rtwdemo_async_mdltreftop_initialize(void)
{
    /* Start for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

    /* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
```

```

/* Connect and enable ISR function: isr_num1_vec192 */
if (intConnect(INUM_TO_IVEC(192), isr_num1_vec192, 0) != OK) {
    printf("intConnect failed for ISR 1.\n");
}

sysIntEnable(1);

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
/* Connect and enable ISR function: isr_num2_vec193 */
if (intConnect(INUM_TO_IVEC(193), isr_num2_vec193, 0) != OK) {
    printf("intConnect failed for ISR 2.\n");
}

sysIntEnable(2);

/* End of Start for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

/* SystemInitialize for ModelReference: '<Root>/Model' incorporates:
 * Inport: '<Root>/In1_60hz'
 * Inport: '<Root>/In2_60_hz'
 * Inport: '<Root>/In3_60hz'
 * Outport: '<Root>/Out1'
 * Outport: '<Root>/Out2'
 * Outport: '<Root>/Out3'
 */
rtwdemo_async_mdrefbot_Init(&rtwdemo_async_mdleftop_Y.Out1);
rtwdemo_async_mdrefbot_Enable();
}

```

Review ISR Code

In the generated source file `rtwdemo_async_mdleftop.c`, review the code for ISRs `isr_num1_vec192` and `isr_num2_vec293`. Each ISR:

- Disables interrupts.
- Saves floating-point context.
- Calls the code generated for the subsystem connected to the referenced model Inport block that receives the interrupt.
- Restores floating-point context.
- Reenables interrupts.

```

cfile = fullfile(cgDir, 'rtwdemo_async_mdleftop_tornado_rtw', 'rtwdemo_async_mdleftop.c');
rtwdemodbtype(cfile, ...
    'void isr_num1_vec192(void)', ...
    'time_T rt_SimUpdateDiscreteEvents', ...
    1, 0);

void isr_num1_vec192(void)
{
    int_T lock;
    FP_CONTEXT context;

    /* disable interrupts (system is configured as non-preemptive) */
    lock = intLock();

    /* save floating point context */

```

```

fppSave(&context);

/* Call the system: '<Root>/Model' */
{
  /* S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

  /* ModelReference: '<Root>/Model' incorporates:
  * Inport: '<Root>/In1_60hz'
  * Inport: '<Root>/In2_60_hz'
  * Inport: '<Root>/In3_60hz'
  * Outport: '<Root>/Out1'
  * Outport: '<Root>/Out2'
  * Outport: '<Root>/Out3'
  */
  rtwdemo_async_mdrefbot_Interrupt1(&rtwdemo_async_mdleftop_Y.Out1);

  /* End of Outputs for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */
}

/* restore floating point context */
fppRestore(&context);

/* re-enable interrupts */
intUnlock(lock);
}

/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
void isr_num2_vec193(void)
{
  FP_CONTEXT context;

  /* save floating point context */
  fppSave(&context);

  /* Call the system: '<Root>/Model' */
  {
    /* S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

    /* ModelReference: '<Root>/Model' incorporates:
    * Inport: '<Root>/In1_60hz'
    * Inport: '<Root>/In2_60_hz'
    * Inport: '<Root>/In3_60hz'
    * Outport: '<Root>/Out1'
    * Outport: '<Root>/Out2'
    * Outport: '<Root>/Out3'
    */
    rtwdemo_async_mdrefbot_Interrupt2();

    /* End of Outputs for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */
  }

  /* restore floating point context */
  fppRestore(&context);
}

```

Review Task Termination Code

The Task Sync block generates the following termination code.

```

cfile = fullfile(cgDir, 'rtwdemo_async_mdltreftop_tornado_rtw', 'rtwdemo_async_mdltreftop.c');
rtwdemodbtype(cfile, ...
    'static void rtwdemo_async_mdltreftop_terminate(void)', ...
    '/*=====*/', ...
    1, 0);

static void rtwdemo_async_mdltreftop_terminate(void)
{
    /* Terminate for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */

    /* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
    /* Disable interrupt for ISR system: isr_num1_vec192 */
    sysIntDisable(1);

    /* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
    /* Disable interrupt for ISR system: isr_num2_vec193 */
    sysIntDisable(2);

    /* End of Terminate for S-Function (vxinterrupt1): '<Root>/Async Interrupt' */
}

```

Related Information

- Async Interrupt
- Asynchronous Task Specification
- “Generate Interrupt Service Routines” on page 6-4
- “Timers in Asynchronous Tasks” on page 6-38
- “Create a Customized Asynchronous Library” on page 6-40
- “Import Asynchronous Event Data for Simulation” on page 6-47
- “Load Data to Root-Level Input Ports”
- “Asynchronous Events” on page 6-2
- “Rate Transitions and Asynchronous Blocks” on page 6-34
- “Asynchronous Support Limitations” on page 6-50

```

bdclose('rtwdemo_async_mdltreftop');
rtwdemoclean;
cd(currentDir)

```

See Also

More About

- “Generate Interrupt Service Routines” on page 6-4
- “Spawn and Synchronize Execution of RTOS Task” on page 6-11
- “Timers in Asynchronous Tasks” on page 6-38
- “Import Asynchronous Event Data for Simulation” on page 6-47
- “Rate Transitions and Asynchronous Blocks” on page 6-34

Rate Transitions and Asynchronous Blocks

Because an asynchronous function call subsystem can preempt or be preempted by other model code, an inconsistency arises when more than one signal element is connected to an asynchronous block. The issue is that signals passed to and from the function call subsystem can be in the process of being written to or read from when the preemption occurs. Thus, some old and some new data is used. This situation can also occur with scalar signals in some cases. For example, if a signal is a double (8 bytes), the read or write operation might require two machine instructions.

Note The operating system integration techniques that are demonstrated in this section use blocks that are in the Interrupt Templates block library. The blocks in that library provide starting point examples to help you develop custom blocks for a target environment.

About Rate Transitions and Asynchronous Blocks

The Simulink Rate Transition block is designed to deal with preemption problems that occur in data transfer between blocks running at different rates. These issues are discussed in “Time-Based Scheduling and Code Generation” on page 5-2.

You can handle rate transition issues automatically by selecting model configuration parameter **Automatically handle rate transition for data transfer**. This saves you from having to manually insert Rate Transition blocks to avoid invalid rate transitions, including invalid *asynchronous-to-periodic* and *asynchronous-to-asynchronous* rate transitions, in multirate models. For asynchronous tasks, the Simulink engine configures inserted blocks for data integrity but not determinism during data transfers.

For asynchronous rate transitions, the Rate Transition block provides data integrity, but cannot provide determinism. Therefore, when you insert Rate Transition blocks explicitly, you must clear the block parameter **Ensure deterministic data transfer (maximum delay)**.

When you insert a Rate Transition block between two blocks to maintain data integrity and priorities are assigned to the tasks associated with the blocks, the code generator assumes that the higher priority task can preempt the lower priority task and the lower priority task cannot preempt the higher priority task. If the priority associated with task for either block is not assigned or the priorities of the tasks for both blocks are the same, the code generator assumes that either task can preempt the other task.

Priorities of periodic tasks are assigned by the Simulink engine, in accordance with the parameters specified in the **Solver selection** section of the **Solver** pane of the Configuration Parameters dialog box. When the **Periodic sample time constraint** parameter is set to Unconstrained, the model base rate priority is set to 40. Priorities for subrates then increment or decrement by 1 from the base rate priority, depending on the setting of parameter **Higher priority value indicates higher task priority**.

You can assign priorities manually by using parameter **Periodic sample time properties**. The Simulink engine does not assign a priority to asynchronous blocks. For example, the priority of a function call subsystem that connects back to an Async Interrupt block is assigned by the Async Interrupt block.

Block parameter **Simulink task priority** of the Async Interrupt block specifies a priority level (required) for every interrupt number entered for parameter **VME interrupt number(s)**. The priority array sets the priorities of the subsystems connected to each interrupt.

For the Task Sync block, if the example RTOS (VxWorks) is the target, clear block parameter **Higher priority value indicates higher task priority**. Parameter **Simulink task priority** specifies the block priority relative to connected blocks (in addition to assigning an RTOS priority to the generated task code).

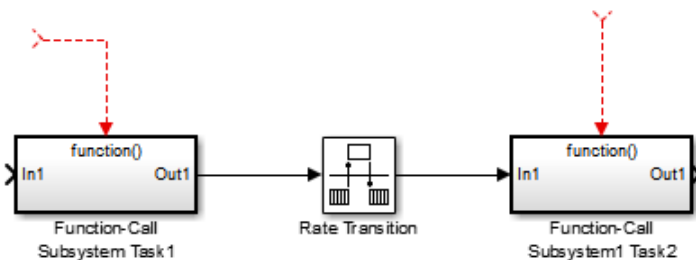
The `vxlib1` library provides two types of rate transition blocks as a convenience. These are simply preconfigured instances of the Rate Transition block:

- Protected Rate Transition block: Rate Transition block that is configured with block parameter **Ensure data integrity during data transfers** selected and **Ensure deterministic data transfer** cleared.
- Unprotected Rate Transition block: Rate Transition block that is configured with the **Ensure data integrity during data transfers** cleared.

Handle Rate Transitions for Asynchronous Tasks

For rate transitions that involve asynchronous tasks, you can maintain data integrity. However, you cannot achieve determinism. You have the option of using the Rate Transition block or target-specific rate transition blocks.

Consider the following model, which includes a Rate Transition block.



You can use the Rate Transition block in either of the following modes:

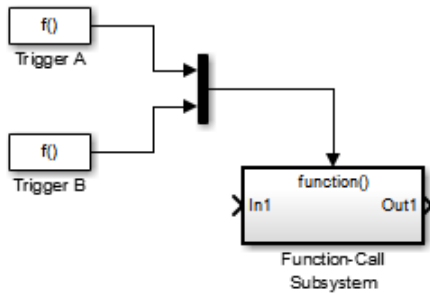
- Maintain data integrity, no determinism
- Unprotected

Alternatively, you can use target-specific rate transition blocks. The following blocks are available for the example RTOS (VxWorks):

- Protected Rate Transition block (reader)
- Protected Rate Transition block (writer)
- Unprotected Rate Transition block

Handle Multiple Asynchronous Interrupts

Consider the following model, in which two functions trigger the same subsystem.



The two tasks must have equal priorities. When priorities are the same, the outcome depends on whether they are firing periodically or asynchronously, and also on a diagnostic setting. The following table and notes describe these outcomes:

Supported Sample Time and Priority for Function Call Subsystem with Multiple Triggers

	Async Priority = 1	Async Priority = 2	Async Priority Unspecified	Periodic Priority = 1	Periodic Priority = 2
Async Priority = 1	Supported (1)				
Async Priority = 2		Supported (1)			
Async Priority Unspecified			Supported (2)		
Periodic Priority = 1				Supported	
Periodic Priority = 2					Supported

- Control these outcomes by using model configuration parameter **Tasks with equal priority**; set this diagnostic to **none** if tasks of equal priority cannot preempt each other in the target system.
- For this case, the following warning message is issued unconditionally:

The function call subsystem <name> has multiple asynchronous triggers that do not specify priority. Data integrity will not be maintained if these triggers can preempt one another.

Empty cells in the above table represent multiple triggers with differing priorities, which are unsupported.

The code generator provides absolute time management for a function call subsystem connected to multiple interrupts in the case where timer settings for TriggerA and TriggerB (time source, resolution) are the same.

Assume that all of the following conditions are true for the model shown above:

- A function call subsystem is triggered by two asynchronous triggers (TriggerA and TriggerB) having identical priority settings.
- Each trigger sets the source of time and timer attributes by calling the functions `ssSetTimeSource` and `ssSetAsyncTimerAttributes`.

- The triggered subsystem contains a block that needs elapsed or absolute time (for example, a Discrete Time Integrator).

The asynchronous function call subsystem has one global variable, `clockTick#` (where `#` is the task ID associated with the subsystem). This variable stores absolute time for the asynchronous task. There are two ways timing can be handled:

- If the time source is set to `SS_TIMESOURCE_BASERATE`, the code generator produces timer code in the function call subsystem, updating the clock tick variable from the base rate clock tick. Data integrity is maintained if the same priority is assigned to TriggerA and TriggerB.
- If the time source is `SS_TIMESOURCE_SELF`, generated code for both TriggerA and TriggerB updates the same clock tick variable from the hardware clock.

The word size of the clock tick variable can be set directly or be established according to setting of model configuration parameter **Application lifespan (days)** and the timer resolution set by the TriggerA and TriggerB S-functions (which must be the same). See “Timers in Asynchronous Tasks” on page 6-38 and “Control Memory Allocation for Time Counters” on page 40-10 for more information.

Protect Data Integrity with volatile Keyword

When you select block parameter **Ensure data integrity during data transfer**, the code generated for a Rate Transition block defines `volatile` global buffers and semaphores and uses them to protect the integrity of the transferred data. For additional protection, or for protection without a Rate Transition block, you can explicitly apply `volatile` to the transferred data. For more information, see “Protect Global Data with const and volatile Type Qualifiers” (Embedded Coder).

See Also

More About

- “Handle Rate Transitions” on page 5-16
- “Time-Based Scheduling and Code Generation” on page 5-2
- “Asynchronous Support Limitations” on page 6-50

Timers in Asynchronous Tasks

An ISR can set a source for absolute time. This is done with the function `ssSetTimeSource`. The function `ssSetTimeSource` cannot be called before `ssSetOutputPortWidth` is called. If this occurs, the program will come to a halt and generate an error message. `ssSetTimeSource` has the following three options:

- `SS_TIMESOURCE_SELF`: Each generated ISR maintains its own absolute time counter, which is distinct from a periodic base rate or subrate counters in the system. The counter value and the timer resolution value (specified with the block parameter **Timer resolution (seconds)** of the Async Interrupt block) are used by downstream blocks to determine absolute time values required by block computations.
- `SS_TIMESOURCE_CALLER`: The ISR reads time from a counter maintained by its caller. Time resolution is thus the same as its caller's resolution.
- `SS_TIMESOURCE_BASERATE`: The ISR can read absolute time from the model's periodic base rate. Time resolution is thus the same as its base rate resolution.

Note The operating system integration techniques that are demonstrated in this section use blocks that are in the Interrupt Templates block library. The blocks in that library provide starting point examples to help you develop custom blocks for a target environment.

By default, the counter is implemented as a 32-bit unsigned integer member of the `Timing` substructure of the real-time model structure. For a target that supports the `rtModel` data structure, when the time data type is not set by using `ssSetAsyncTimeDataType`, the counter word size is determined by the setting for model configuration parameter **Application lifespan (days)**. As an example (from ERT target code),

```
/* Real-time Model Data Structure */
struct _RT_MODEL_elapseTime_exp_Tag {
    const char *errorStatus;

    /*
     * Timing:
     * The following substructure contains information regarding
     * the timing information for the model.
     */
    struct {
        uint32_T clockTick1;
        uint32_T clockTick2;
    } Timing;
};
```

The example omits unused fields in the `Timing` data structure (a feature of ERT target code not found in GRT). For a target that supports the `rtModel` data structure, the counter word size is determined by the setting of model configuration parameter **Application lifespan (days)**.

By default, the `vxlib1` library blocks for the example RTOS (VxWorks) set the timer source to `SS_TIMESOURCE_SELF` and update their counters by using the system call `tickGet`. `tickGet` returns a timer value maintained by the RTOS kernel. The maximum word size for the timer is `UINT32`. The following example shows a generated call to `tickGet`.

```
/* VxWorks Interrupt Block: '<Root>/Async Interrupt' */
void isr_num2_vec193(void)
```

```

{
    /* Use tickGet() as a portable tick counter example. A much
       higher resolution can be achieved with a hardware counter */
    rtM->Timing.clockTick2 = tickGet();
    . . .
}

```

The `tickGet` call is supplied only as an example. It can (and in many instances should) be replaced by a timing source that has better resolution. If you are implementing a custom asynchronous block for an RTOS other than the example RTOS (VxWorks), you should either generate an equivalent call to the target RTOS, or generate code to read a timer register on the target hardware.

Change the setting for the block parameter **Timer resolution (seconds)** for your Async Interrupt block to match the resolution of timing source for you target computer.

The counter is updated at interrupt level. Its value represents the tick value of the timing source at the most recent execution of the ISR. The rate of this timing source is unrelated to sample rates in the model. In fact, typically it is faster than the model's base rate. Select the timer source and set its rate and resolution based on the expected rate of interrupts to be serviced by the Async Interrupt block.

For an example of timer code generation, see “Async Interrupt Block Implementation” on page 6-40.

See Also

Related Examples

- “Generate Interrupt Service Routines” on page 6-4
- “Spawn and Synchronize Execution of RTOS Task” on page 6-11
- “Create a Customized Asynchronous Library” on page 6-40
- “Import Asynchronous Event Data for Simulation” on page 6-47

More About

- “Absolute and Elapsed Time Computation” on page 4-2
- “Time-Based Scheduling and Code Generation” on page 5-2
- “Asynchronous Events” on page 6-2
- “Asynchronous Support Limitations” on page 6-50

Create a Customized Asynchronous Library

This topic describes how to implement asynchronous blocks for use with your target RTOS, using the Async Interrupt and Task Sync blocks as a starting point. Rate Transition blocks are target-independent, so you do not need to develop customized rate transition blocks.

Note The operating system integration techniques that are demonstrated in this section use blocks that are in the Interrupt Templates block library. The blocks in that library provide starting point examples to help you develop custom blocks for a target environment.

About Implementing Asynchronous Blocks

You can customize the asynchronous library blocks by modifying the block implementation. These files are

- The block's underlying S-function MEX-file
- The TLC files that control code generation of the block

In addition, you need to modify the block masks to remove references specific to the example RTOS (VxWorks) and to incorporate parameters required by your target RTOS.

Custom block implementation is an advanced topic, requiring familiarity with the Simulink MEX S-function format and API, and with the Target Language Compiler (TLC). These topics are covered in the following documents:

- Simulink topics “What Is an S-Function?”, “Use S-Functions in Models”, “How S-Functions Work”, and “Implementing S-Functions” describe MEX S-functions and the S-function API in general.
- The “Inlining S-Functions”, “Inline C MEX S-Functions”, and “S-Functions and Code Generation” on page 14-2 describe how to create a TLC block implementation for use in code generation.

The following sections discuss the C/C++ and TLC implementations of the asynchronous library blocks, including required `SimStruct` macros and functions in the TLC asynchronous support library (`asynclib.tlc`).

Async Interrupt Block Implementation

The source files for the Async Interrupt block are located in `matlabroot/rtw/c/tornado/devices` (open):

- `vxinterrupt1.c`: C MEX-file source code, for use in configuration and simulation
- `vxinterrupt1.tlc`: TLC implementation, for use in code generation
- `asynclib.tlc`: library of TLC support functions, called by the TLC implementation of the block. The library calls are summarized in “asynclib.tlc Support Library” on page 6-45.

C MEX Block Implementation

Most of the code in `vxinterrupt1.c` performs ordinary functions that are not related to asynchronous support (for example, obtaining and validating parameters from the block mask, marking parameters nontunable, and passing parameter data to the `model.rtw` file).

The `mdlInitializeSizes` function uses special `SimStruct` macros and `SS_OPTIONS` settings that are required for asynchronous blocks, as described below.

Note that the following macros cannot be called before `ssSetOutputPortWidth` is called:

- `ssSetTimeSource`
- `ssSetAsyncTimerAttributes`
- `ssSetAsyncTimerResolutionEl`
- `ssSetAsyncTimerDataType`
- `ssSetAsyncTimerDataTypeEl`
- `ssSetAsyncTaskPriorities`
- `ssSetAsyncTaskPrioritiesEl`

If one of the above macros is called before `ssSetOutputPortWidth`, the following error message appears:

```
SL_SfcnMustSpecifyPortWidthBfCallSomeMacro {
S-function '%s' in '%<BLOCKFULLPATH>'
must set output port %d width using
ssSetOutputPortWidth before calling macro %s
}
```

ssSetAsyncTimerAttributes

`ssSetAsyncTimerAttributes` declares that the block requires a timer, and sets the resolution of the timer as specified for block parameter **Timer resolution (seconds)**.

The function prototype is

```
ssSetAsyncTimerAttributes(SimStruct *S, double res)
```

where

- `S` is a `SimStruct` pointer.
- Parameter **Timer resolution (seconds)** is set to `res`.

The following code excerpt shows the call to `ssSetAsyncTimerAttributes`.

```
/* Setup Async Timer attributes */
ssSetAsyncTimerAttributes(S,mxGetPr(TICK_RES)[0]);
```

ssSetAsyncTaskPriorities

`ssSetAsyncTaskPriorities` sets the Simulink task priority for blocks executing at each interrupt level, as specified for block parameter **Simulink task priority**.

The function prototype is

```
ssSetAsyncTaskPriorities(SimStruct *S, int numISRs,
                        int *priorityArray)
```

where

- `S` is a `SimStruct` pointer.

- `numISRs` is the number of interrupts specified for block parameter **VME interrupt number(s)** .
- `priorityarray` is an integer array containing the interrupt numbers specified for block parameter **VME interrupt number(s)**.

The following code excerpt shows the call to `ssSetAsyncTaskPriorities`:

```
/* Setup Async Task Priorities */
priorityArray = malloc(numISRs*sizeof(int_T));
for (i=0; i<numISRs; i++) {
    priorityArray[i] = (int_T)(mxGetPr(ISR_PRIORITIES)[i]);
}
ssSetAsyncTaskPriorities(S, numISRs, priorityArray);
free(priorityArray);
priorityArray = NULL;
}
```

SS_OPTION Settings

The code excerpt below shows the `SS_OPTION` settings for `vxinterrupt1.c`. `SS_OPTION_ASYNCHRONOUS_INTERRUPT` should be used when a function call subsystem is attached to an interrupt. For more information, see the documentation for `SS_OPTION` and `SS_OPTION_ASYNCHRONOUS` in `matlabroot/simulink/include/simstruc.h`.

```
ssSetOptions( S, (SS_OPTION_EXCEPTION_FREE_CODE |
                 SS_OPTION_DISALLOW_CONSTANT_SAMPLE_TIME |
                 SS_OPTION_ASYNCHRONOUS_INTERRUPT));
```

If an S-function specifies the `SS_OPTION_DISALLOW_CONSTANT_SAMPLE_TIME` option and inherits a sample time of `Inf`, the code generator determines how to produce code for the block based on whether the block is invariant. A block is invariant if its port signals are invariant. A signal is invariant if it has a constant value during the entire simulation. If you specify a Constant block sample time, do not assume that the port signals are invariant. For more information, see “Inline Invariant Signals” on page 40-38. If the block is not invariant, the code generator produces code only in the initialize entry-point function. If the block is invariant, the code generator does not produce code for the block.

TLC Implementation

This section discusses each function of `vxinterrupt1.tlc`, with an emphasis on target-specific features that you will need to change to generate code for your target RTOS.

Generate #include Directives

`vxinterrupt1.tlc` begins with the statement

```
%include "vxlib.tlc"
```

`vxlib.tlc` is a target-specific file that generates directives to include header files for the example RTOS (VxWorks). You should replace this with a file that generates includes for your target RTOS.

BlockInstanceSetup Function

For each connected output of the Async Interrupt block, `BlockInstanceSetup` defines a function name for the corresponding ISR in the generated code. The functions names are of the form

```
isr_num_vec_offset
```

where *num* is the ISR number defined for block parameter **VME interrupt number(s)**, and *offset* is an interrupt table offset defined by block parameter **VME interrupt vector offset(s)**.

In a custom implementation, this naming convention is optional.

The function names are cached for use by the `Outputs` function, which generates the actual ISR code.

Outputs Function

`Outputs` iterates over the connected outputs of the Async Interrupt block. An ISR is generated for each such output.

The ISR code is cached in the "Functions" section of the generated code. Before generating the ISR, `Outputs` does the following:

- Generates a call to the downstream block (cached in a temporary buffer).
- Determines whether the ISR should be locked or not (as specified by block parameter **Preemption Flag(s)**).
- Determines whether the block connected to the Async Interrupt block is a Task Sync block. (This information is obtained by using the `asynclib` calls `LibGetFcnCallBlock` and `LibGetBlockAttribute`.) If so,
 - The preemption flag for the ISR must be set to 1. An error results otherwise.
 - The RTOS (VxWorks) calls to save and restore floating-point context are generated, unless the user has configured the model for integer-only code generation.

When generating the ISR code, `Outputs` calls the `asynclib` function `LibNeedAsyncCounter` to determine whether a timer is required by the connected subsystem. If so, and if the time source is set to be `SS_TIMESOURCE_SELF` by `ssSetTimeSource`, `LibSetAsyncCounter` is called to generate an RTOS (VxWorks) `tickGet` function call and update the counter. In your implementation, you should generate either an equivalent call to the target RTOS, or generate code to read the a timer register on the target hardware.

Start Function

The `Start` function generates the required RTOS (VxWorks) calls (`int_connect` and `sysInt_Enable`) to connect and enable each ISR. You should replace this with calls to your target RTOS.

Terminate Function

The `Terminate` function generates the call `sysIntDisable` to disable each ISR. You should replace this with calls to your target RTOS.

Task Sync Block Implementation

The source files for the Task Sync block are located in `matlabroot/rtw/c/tornado/devices` (open). They are

- `vxtask1.cpp`: MEX-file source code, for use in configuration and simulation.
- `vxtask1.tlc`: TLC implementation, for use in code generation.
- `asynclib.tlc`: library of TLC support functions, called by the TLC implementation of the block. The library calls are summarized in "asynclib.tlc Support Library" on page 6-45.

C MEX Block Implementation

Like the Async Interrupt block, the Task Sync block sets up a timer, in this case with a fixed resolution. The priority of the task associated with the block is obtained from block parameter **Simulink task priority**. The SS_OPTION settings are the same as those used for the Async Interrupt block.

```
ssSetAsyncTimerAttributes(S, 0.01);

priority = (int_T) (*(mxGetPr(PRIORITY)));
ssSetAsyncTaskPriorities(S,1,&priority);

ssSetOptions(S, (SS_OPTION_EXCEPTION_FREE_CODE |
                 SS_OPTION_ASYNCHRONOUS |
                 SS_OPTION_DISALLOW_CONSTANT_SAMPLE_TIME));
}
```

TLC Implementation

Generate #include Directives

`vxtask1.tlc` begins with the statement

```
%include "vxlib.tlc"
```

`vxlib.tlc` is a target-specific file that generates directives to include header files for the example RTOS (VxWorks). You should replace this with a file that generates includes for your target RTOS.

BlockInstanceSetup Function

The `BlockInstanceSetup` function derives the task name, block name, and other identifiers used later in code generation. It also checks for and warns about unconnected block conditions, and generates a storage declaration for a semaphore (`stopSem`) that is used in case of interrupt overflow conditions.

Start Function

The `Start` function generates the required RTOS (VxWorks) calls to define storage for the semaphore that is used in management of the task spawned by the Task Sync block. Depending on the value of the `CodeFormat` TLC variable of the target, either a static storage declaration or a dynamic memory allocation call is generated. This function also creates a semaphore (`semBCreate`) and spawns an RTOS task (`taskSpawn`). You should replace these with calls to your target RTOS.

Outputs Function

The `Outputs` function generates an example RTOS (VxWorks) task that waits for a semaphore. When it obtains the semaphore, it updates the block's tick timer and calls the downstream subsystem code, as described in "Spawn and Synchronize Execution of RTOS Task" on page 6-11. `Outputs` also generates code (called from interrupt level) that grants the semaphore.

Terminate Function

The `Terminate` function generates the example RTOS (VxWorks) call `taskDelete` to end execution of the task spawned by the block. You should replace this with calls to your target RTOS.

Note also that if the target RTOS has dynamically allocated memory associated with the task, the `Terminate` function should deallocate the memory.

asynclib.tlc Support Library

`asynclib.tlc` is a library of TLC functions that support the implementation of asynchronous blocks. Some functions are specifically designed for use in asynchronous blocks. For example, `LibSetAsyncCounter` generates a call to update a timer for an asynchronous block. Other functions are utilities that return information required by asynchronous blocks (for example, information about connected function call subsystems).

The following table summarizes the public calls in the library. For details, see the library source code and the `vxinterrupt1.tlc` and `vxtask1.tlc` files, which call the library functions.

Summary of asynclib.tlc Library Functions

Function	Description
<code>LibBlockExecuteFcnCall</code>	For use by inlined S-functions with function call outputs. Generates code to execute a function call subsystem.
<code>LibGetBlockAttribute</code>	Returns a field value from a block record.
<code>LibGetFcnCallBlock</code>	Given an S-Function block and call index, returns the block record for the downstream function call subsystem block.
<code>LibGetCallerClockTickCounter</code>	Provides access to the time counter of an upstream asynchronous task.
<code>LibGetCallerClockTickCounter-HighWord</code>	Provides access to the high word of the time counter of an upstream asynchronous task.
<code>LibManageAsyncCounter</code>	Determines whether an asynchronous task needs a counter and manages its own timer.
<code>LibNeedAsyncCounter</code>	If the calling block requires an asynchronous counter, returns <code>TLC_TRUE</code> , otherwise returns <code>TLC_FALSE</code> .
<code>LibSetAsyncClockTicks</code>	Returns code that sets <code>clockTick</code> counters that are to be maintained by the asynchronous task.
<code>LibSetAsyncCounter</code>	Generates code to set the tick value of the block's asynchronous counter.
<code>LibSetAsyncCounterHighWord</code>	Generates code to set the tick value of the high word of the block's asynchronous counter

See Also

More About

- “Asynchronous Events” on page 6-2
- “Generate Interrupt Service Routines” on page 6-4
- “Spawn and Synchronize Execution of RTOS Task” on page 6-11
- “Pass Asynchronous Events in RTOS as Input to a Referenced Model” on page 6-28
- “Timers in Asynchronous Tasks” on page 6-38
- “Import Asynchronous Event Data for Simulation” on page 6-47
- “Rate Transitions and Asynchronous Blocks” on page 6-34

- “Asynchronous Support Limitations” on page 6-50

Import Asynchronous Event Data for Simulation

Capabilities

You can import asynchronous event data into a function-call subsystem via an Inport block. For standalone fixed-step simulations, you can specify:

- The time points at which each asynchronous event occurs
- The number of asynchronous events at each time point

Input Data Format

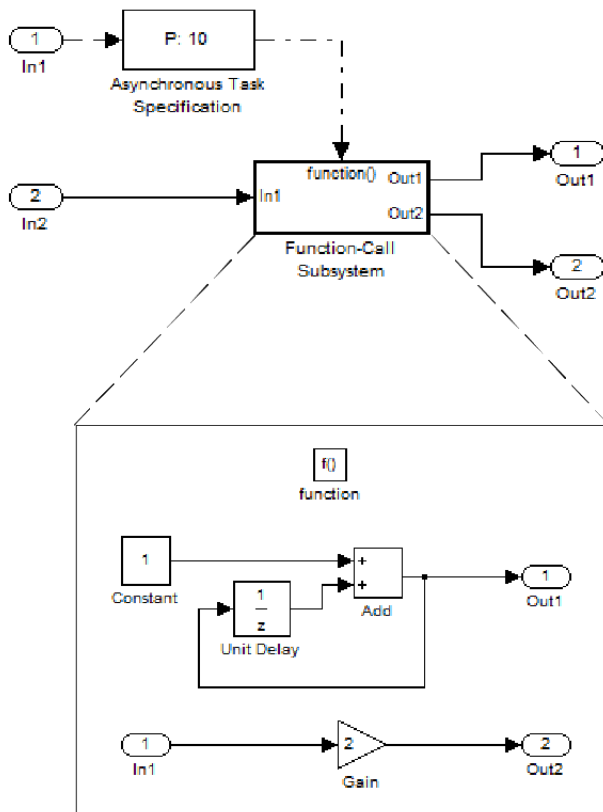
You can enter your asynchronous data at the MATLAB command line or on the **Data Import/Export** pane of the Configuration Parameters dialog box. In either case, a number of restrictions apply to the data format.

- The expression for parameter model configuration parameter **Input** must be a comma-separated list of tables.
- The table corresponding to the input port outputting asynchronous events must be a column vector containing time values for the asynchronous events.
 - The time vector of the asynchronous events must be of double data type and monotonically increasing.
 - Time data must be integer multiples of the model step size.
 - To specify multiple function calls at a given time step, you must repeat the time value accordingly. In other words, if you wish to specify three asynchronous events at $t = 1$ and two events at $t = 9$, then you must list 1 three times and 9 twice in your time vector. (`t = [1 1 1 9 9]'`)
- The table corresponding to normal data input port can be of another supported format.

See “Load Data to Root-Level Input Ports” for more information.

Example

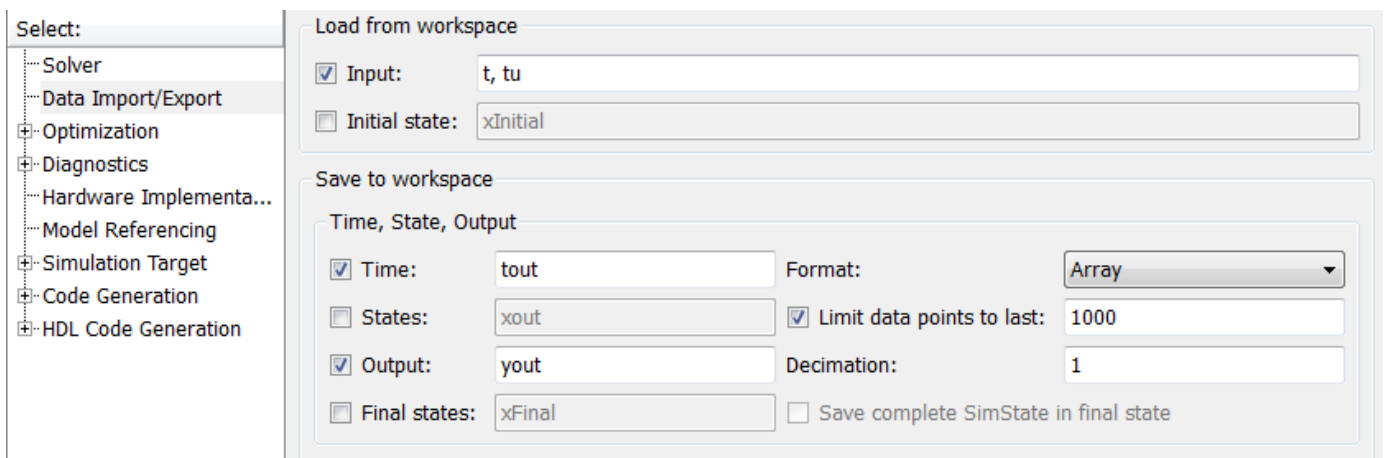
In this model, a function-call subsystem is used to track the total number of asynchronous events and to multiply a set of input values by 2.



- 1 To input data by using the Model Configuration Parameters dialog box,
 - a Select **Input**.
 - b For this example, enter the following command in the MATLAB window:

```
>> t = [1 1 5 9 9 9]', tu = [[0:10]' [0:10]']
```

Alternatively, you can enter the data as t , tu in the Data Import/Export pane:



Here, t is a column vector containing the times of asynchronous events for Inport block In1 while tu is a table of input values versus time for Inport block In2.

- 2 By default, the **Time** and **Output** parameters are selected and the output variables are named *tout* and *yout*.
- 3 Simulate the model.
- 4 Display the output by entering [*tout* *yout*] at the MATLAB command line and obtain:

```
ans =
     0     0    -1
     1     2     2
     2     2     2
     3     2     2
     4     2     2
     5     3    10
     6     3    10
     7     3    10
     8     3    10
     9     6    18
    10     6    18
```

Here the first column contains the simulation times.

The second column represents the output of Out1 — the total number of asynchronous events. Since the function-call subsystem is triggered twice at $t = 1$, the output is 2. It is not called again until $t = 5$, and so does not increase to 3 until then. Finally, it is called three times at 9, so it increases to 6.

The third column contains the output of Out2 obtained by multiplying the input value at each asynchronous event time by 2. At any other time, the output is held at its previous value

See Also

More About

- “Asynchronous Events” on page 6-2
- “Pass Asynchronous Events in RTOS as Input to a Referenced Model” on page 6-28
- “Load Data to Root-Level Input Ports”
- “Rate Transitions and Asynchronous Blocks” on page 6-34

Asynchronous Support Limitations

In this section...

“Asynchronous Task Priority” on page 6-50

“Convert an Asynchronous Subsystem into a Model Reference” on page 6-50

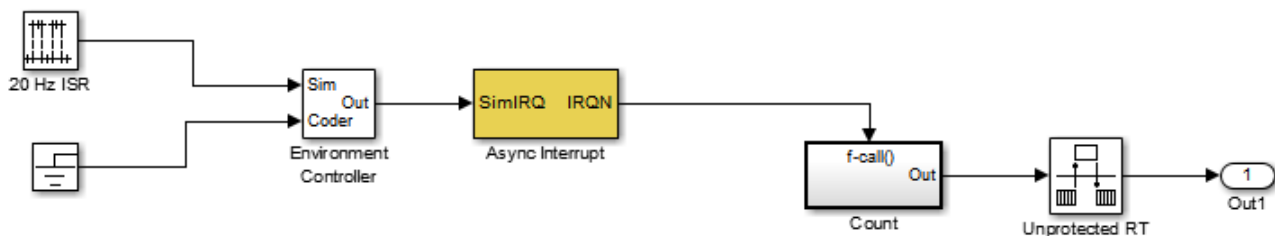
Asynchronous Task Priority

The Simulink product does not simulate asynchronous task behavior. Although you can specify a task priority for an asynchronous task represented in a model with the Task Sync block, the priority setting is for code generation purposes only and is not honored during simulation.

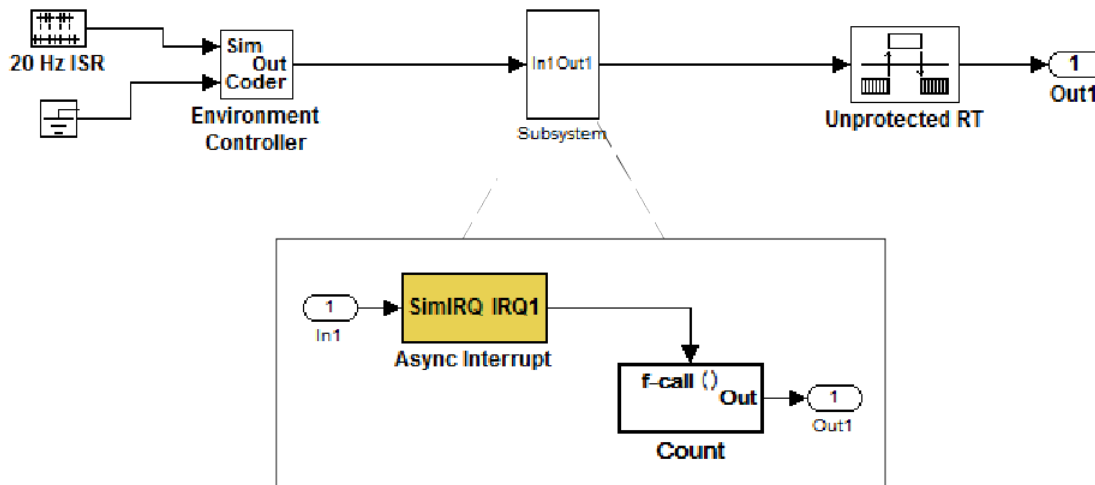
Convert an Asynchronous Subsystem into a Model Reference

You can use the Asynchronous Task Specification block to specify an asynchronous function-call input to a referenced model. However, you must convert the Async Interrupt and Function-Call blocks into a subsystem and then convert the subsystem into a referenced model.

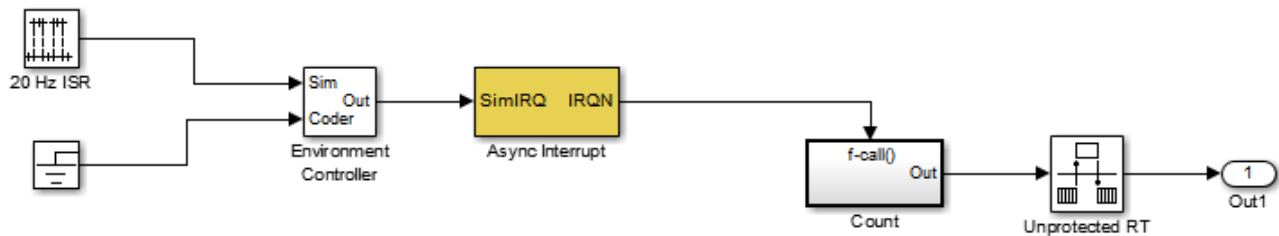
This example shows how to complete a conversion.

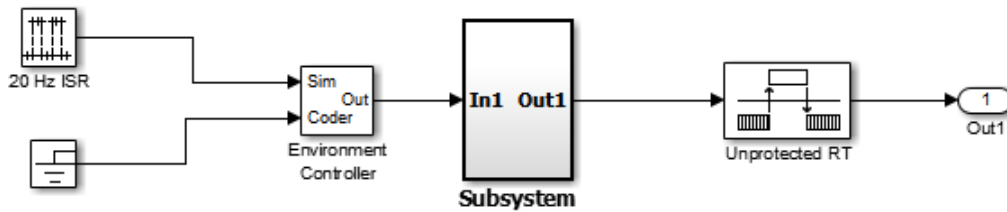


- 1 Convert the Async Interrupt and Count blocks into a subsystem. Select both blocks and right-click Count. From the menu, select **Subsystem & Model Reference > Create Subsystem from Selection**.



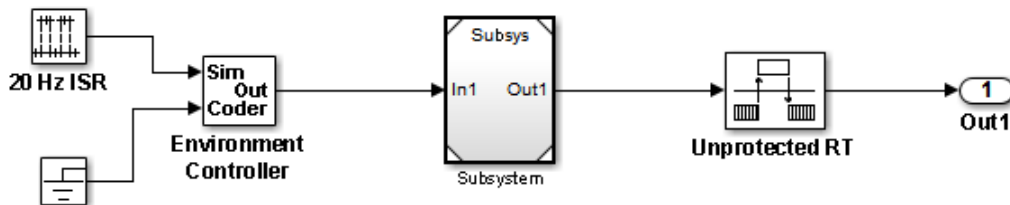
- 2 To prepare for converting the new subsystem to a Model block, set these model configuration parameters in the top model.
 - Set **Multitask data transfer** to error.
 - Set **Multitask conditionally executed subsystem** to error.
 - Set **Bus signal treated as vector** to error.
 - Set **Context-dependent inputs** to error.
 - Set **Multitask data store** to error.
 - Set **Underspecified initialization detection** to Simplified.
 - If your model is large or complex, run the Model Advisor checks in the folder “Migrating to Simplified Initialization Mode Overview” and make the suggested changes.
- 3 Convert the subsystem to an atomic subsystem. Select the subsystem block parameter **Treat as atomic unit**.



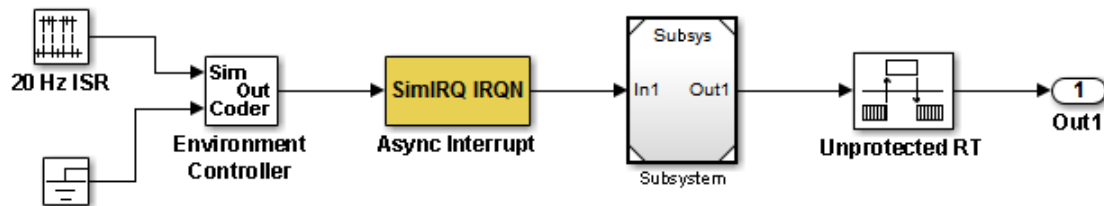


4 Convert the subsystem to a Model block. Right-click the subsystem and select **Subsystem & Model Reference > Convert Subsystem to > Referenced Model**. The Model Reference Conversion Advisor opens.

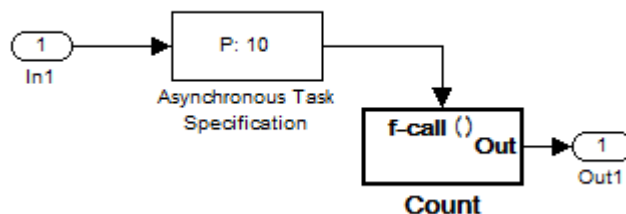
5 Replace the subsystem in the top model with the new Model block.



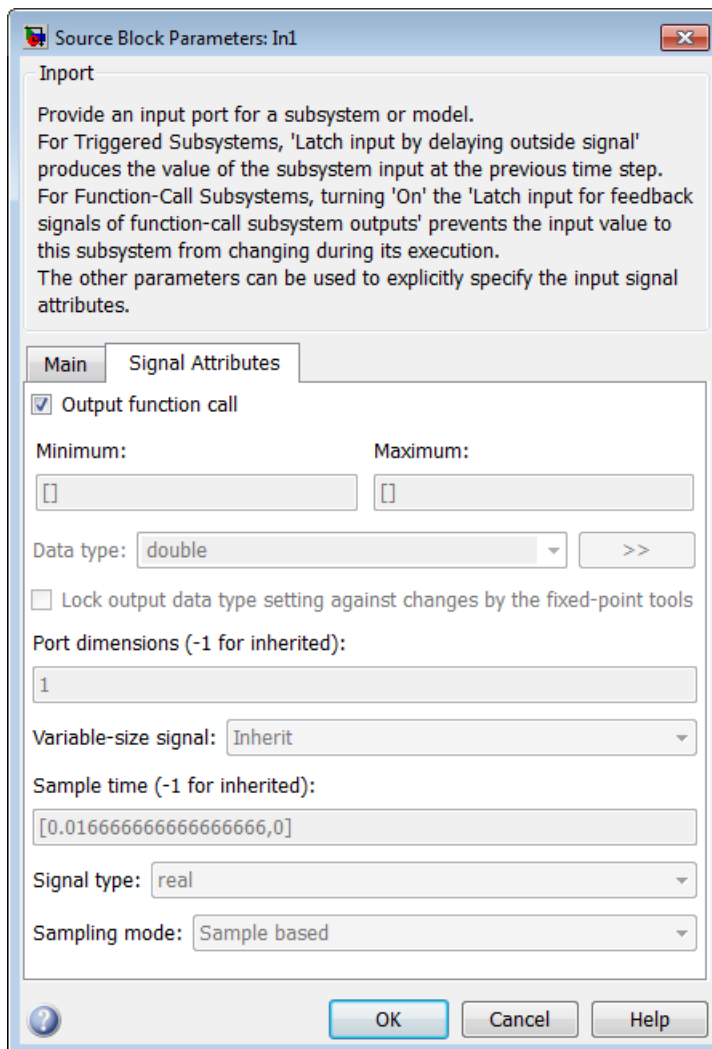
6 Move the Async Interrupt block from the referenced model to the top model, before the Model block.



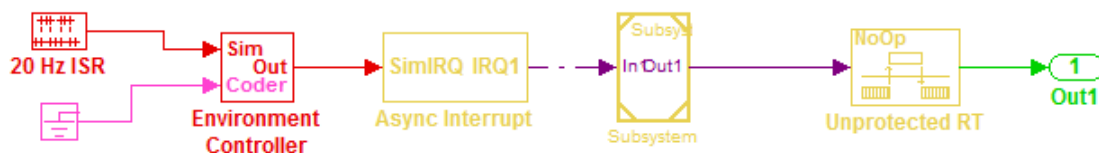
7 Insert an Asynchronous Task Specification block in the referenced model. Set the priority of the Asynchronous Task Specification block. (For more information on setting the priority, see Asynchronous Task Specification.)



8 In the referenced model, double-click the input port to open its Source Block Parameters dialog box. In the **Signal Attributes** tab, select the **Output function call** parameter. Click **OK**.



- 9 Save your model and then update the diagram to verify your settings.



See Also

More About

- “Asynchronous Events” on page 6-2

Subsystems in Simulink Coder

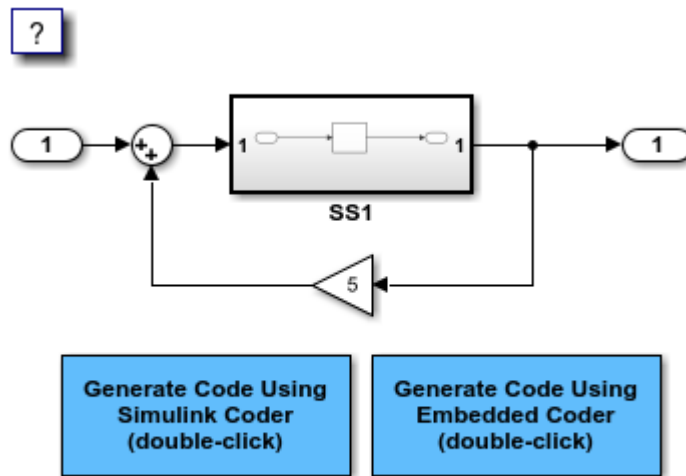
- “Generate Code and Executables for Individual Subsystems” on page 7-2
- “Generate Inlined Subsystem Code” on page 7-5
- “Generate Subsystem Code as Separate Function and Files” on page 7-7

Generate Code and Executables for Individual Subsystems

You can generate code and build a standalone executable for a subsystem within a model. When you generate code for an individual subsystem, the code generation and build process uses the code generation and build parameters of the root model.

- 1 Open a model with a subsystem, such as `rtwdemo_atomic`.

```
open_system('rtwdemo_atomic')
```



Copyright 1994-2020 The MathWorks, Inc.

- 2 Open Simulink Coder or Embedded Coder.

If you are using Embedded Coder, from the **C Code** tab, click **View Code** to open the **Code View** editor.
- 3 Right-click the Subsystem block. From the context menu, click **Block Parameters (Subsystem)**. From the **Block Parameters: SS1** window, open the **Code Generation** tab.
- 4 Use the **Function name options** and **File name options** parameters to control the names of the generated functions and files. Set the **Function name options** parameter to **Auto** so that the code generator uses the name of the subsystem in the generated file names.
- 5 Right-click the Subsystem block. From the context menu, select **C/C++ Code > Build This Subsystem**.
- 6 To configure properties such as the name, class, and storage class of variables (or data objects) that are referenced as block parameters in the subsystem, convert the subsystem to a Model block. Configure the properties through the Code Mappings editor.

For more information on tunable and inlined parameters and storage classes, see “Create Tunable Calibration Parameter in the Generated Code” on page 15-112.

The build process displays status messages in the MATLAB Command Window. When the build is complete, the generated executable is in your working folder. The name of the generated executable is `subsystem.exe` (on PC platforms) or `subsystem` (on The Open Group UNIX

platforms). *subsystem* is the name of the source Subsystem block. In this example, the Subsystem block is named SS1.

The generated code is in a build subfolder named *subsystem_target_rtw*. *subsystem* is the name of the source subsystem block and *target* is the name of the target configuration.

Subsystem Build Limitations

These limitations apply to building subsystems:

- You cannot use a subsystem build for a model configured with an ERT-based system target file and service code interface.
- A subsystem build does not support a subsystem that has a function-call trigger input or a function-call output.
- When you build a subsystem that includes an Outport block for which the **Data type** parameter specifies a bus object, you must address errors that result from setting signal labels. To configure the software to display these errors, in the Configuration Parameters dialog box for the parent model, on the **Diagnostics > Connectivity** pane, set the **Signal label mismatch** parameter to error.
- When a subsystem is in a triggered or function-call subsystem, the right-click build process might fail if the subsystem code is not sample-time independent. To determine whether a subsystem is sample-time independent:
 - 1 Copy all blocks in the subsystem to an empty model.
 - 2 In the Configuration Parameters dialog box, on the **Solver** pane, set:
 - a **Type** to Fixed-step.
 - b **Periodic sample time constraint** to Ensure sample time independent.
 - c Click **Apply**.
 - 3 Update the model. If the model is sample-time dependent, Simulink generates an error in the process of updating the diagram.
- When you use the right-click build process for a subsystem, the code generator attempts to use the subsystem name for generated code files. In some cases, there can be a conflict with the name that you specify when you set, for example, **File name options** to Use function name or **Function name options** to Use subsystem name. You see an error:

```
The subsystem 'model/subsys'
is trying to generate code to an reserved file (subsys) for
the model 'subsys'...
```

To resolve the error, modify one of the conflicting file names so that the names are unique.

- In a subsystem build warning, the subsystem block path hyperlink that is created references a temporary model block path instead of the actual model block path. In the Diagnostic Viewer, clicking the subsystem hyperlink does not take you to the block. In the Command Window, you see a message:

```
...
No system or file called 'subsystemName' found.
...
```

See Also

Related Examples

- “Generate Predictable Function Interface to Match Graphical Block Interface” (Embedded Coder)
- “Generate Subsystem Code as Separate Function and Files” on page 7-7

More About

- “Modeling Guidelines for Subsystems”

Generate Inlined Subsystem Code

To inline subsystem code with your model code, configure a nonvirtual subsystem. In the Subsystem Block Parameters dialog box, the **Function packaging** parameter specifies the format of the subsystem generated code. This parameter has these settings:

- Auto
- Inline
- Nonreusable function
- Reusable function

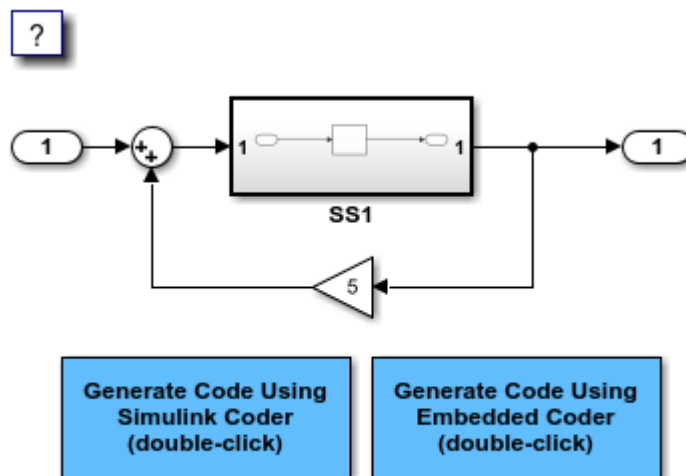
The code generator can generate inlined code for the **Auto** and **Inline** settings. The inline setting directs the code generator to inline the subsystem code unconditionally.

The default **Auto** setting directs the code generator to generate the most efficient code for the subsystem based on the type and number of instances of the subsystem that exist in the model. When there is only one instance of a subsystem, the **Auto** setting inlines the subsystem code. In Embedded Coder, when there are multiple instances of a subsystem that are not too complex, the **Auto** setting inlines the code for each subsystem. Otherwise, the **Auto** setting generates a single copy of the function (as a reusable function). For a function-call subsystem that has multiple callers, the **Auto** setting generates subsystem code that is consistent with the **Nonreusable function** setting.

Configure Subsystem to Inline Code

- 1 Open a model with a subsystem, such as `rtwdemo_atomic`.

```
open_system('rtwdemo_atomic')
```



Copyright 1994-2020 The MathWorks, Inc.

- 2 Open Simulink Coder or Embedded Coder.

If you are using Embedded Coder, from the **C Code** tab, click **View Code** to open the **Code View** editor.

- 3 Right-click the Subsystem block. From the context menu, select **Block Parameters (Subsystem)**.
- 4 In the Subsystem Parameters dialog box, verify that **Treat as atomic unit** is selected. This option makes the subsystem nonvirtual. When the subsystem is nonvirtual, the **Function packaging** option is available on the **Code Generation** tab.
- 5 Click the **Code Generation** tab and select Auto or Inline from the **Function packaging** parameter.
- 6 Click **Apply** and close the dialog box.

The border of the subsystem thickens, indicating that it is nonvirtual.

- 7 Generate code from the model. The code generator inlines subsystem code within `rtwdemo_atomic.c`. You can identify this code by system and block identification tags, such as:

```
/* Atomic SubSystem Block: <Root>/AtomicSubsys1 */
```

Exceptions to Inlining

There are certain cases in which the code generator does not inline a nonvirtual subsystem, even though you select the **Inline** setting.

- If a noninlined S-function calls a function-call subsystem, the code generator ignores the **Inline** setting. Because noninlined S-functions use function pointers to make function calls, the code generator must generate a function with all arguments present.
- In a feedback loop involving function-call subsystems, the code generator generates a function instead of inlined code for one of the subsystems. Based on the internal, sorted order of the subsystems, the code generator selects which subsystem to generate a function.
- If an S-function, an Async Interrupt, or a Task Sync block that has the option `SS_OPTION_FORCE_NONINLINED_FCNCALL` set to `TRUE` calls a subsystem, the code generator generates a function instead of inlined code for the subsystem. The VxWorks block library (`vxlib1`) contains the user-defined Async Interrupt and Task Sync blocks.³

The **Function packaging** parameter is disabled in Simulink Function blocks.

See Also

- “Generate Subsystem Code as Separate Function and Files” on page 7-7
- “Generate Reentrant Code from Subsystems” on page 10-28
- “Remove Algebraic Loops”

³ VxWorks is a registered trademark of Wind River Systems, Inc.

Generate Subsystem Code as Separate Function and Files

You can configure an atomic subsystem to generate code to a separate function and file. This results in code that is more modular and enables you to unit test code for individual subsystems.

The **Function packaging** Subsystem block parameter has two settings that cause a subsystem to generate code as a separate function. The **Reusable** function option generates a function that passes I/O, states, and parameters as function arguments. The **Nonreusable** function option generates a function that passes I/O, states, and parameters as a combination of function arguments and global data structures.

Subsystem Function Dependence

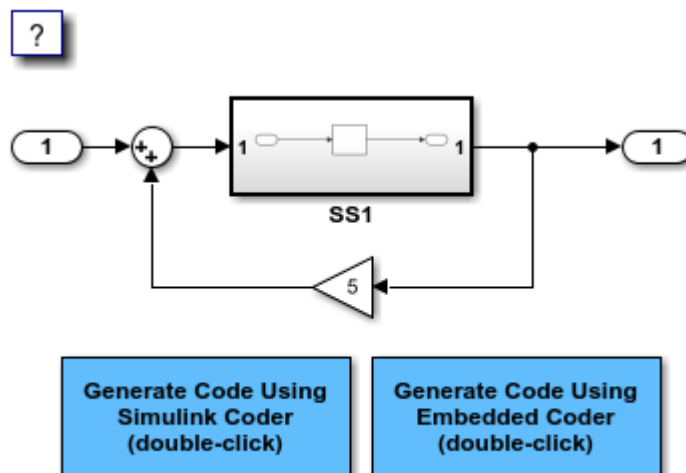
When generating code for a subsystem, the code can reference global data structures of the model, even if the subsystem function code is in a separate file. Each subsystem code file contains `include` directives and comments describing the dependencies. The code generator checks for cyclic file dependencies and produces warnings about them at build time. For descriptions of how the code generator packages code, see “Manage Build Process File Dependencies” on page 24-43.

To generate subsystem function code that is independent of the code generated for the parent model, place the subsystem in a library and configure it as a reusable subsystem, as described in “Generate Reusable Code from Library Subsystems Shared Across Models” on page 10-36.

If you have Embedded Coder, you can generate code for library consisting of reusable subsystems that have different function interfaces. For more information, see “Library-Based Code Generation for Reusable Library Subsystems” (Embedded Coder).

Generate Subsystem as a Reusable Function

- 1 Open a model with a subsystem, such as `rtwdemo_atomic`.



Copyright 1994-2020 The MathWorks, Inc.

If you are using Embedded Coder, from the **C Code** tab, click **View Code** to open the **Code View** editor.

- 2 Right-click the Subsystem block. From the context menu, select **Block Parameters (Subsystem)**.
- 3 In the Subsystem Parameters dialog box, verify that **Treat as atomic unit** is selected. With that parameter selected, on the **Code Generation** tab, the **Function packaging** parameter is available.
- 4 Click the **Code Generation** tab and select Reusable function from the **Function packaging** parameter. This enables two parameters:
 - The **Function name options** parameter controls the naming of the generated function.
 - The **File name options** parameter controls the naming of the generated file.
- 5 Set the **File name options** parameter to Use subsystem name.
- 6 Click **Apply** and close the dialog box.
- 7 If you are using Embedded Coder, open the Configuration Parameters dialog box. Verify that the model configuration parameter **File packaging format** (Embedded Coder) is set to Modular.
- 8 Generate the code.

```
#include "SS1.h"

/* Include model header file for global data */
#include "rtwdemo_atomic.h"
#include "rtwdemo_atomic_private.h"

/* Outputs for atomic system: '<Root>/SS1' */
real_T myfun(DW_myfun_T *localDW)
{
    /* DiscreteIntegrator: '<S1>/Integrator' */
    return localDW->Integrator_DSTATE;
}

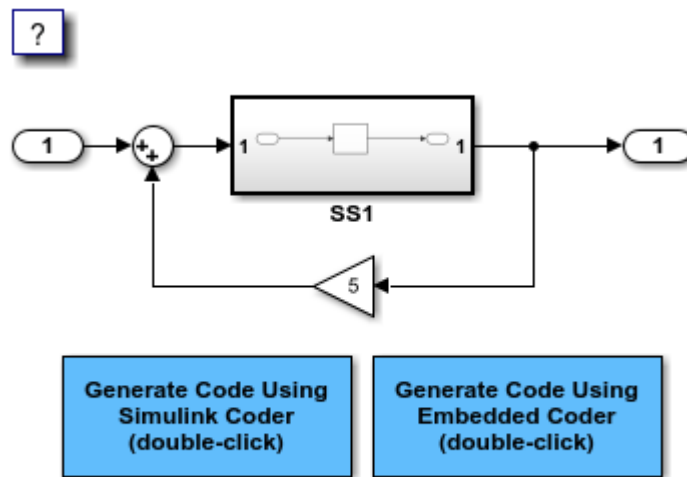
/* Update for atomic system: '<Root>/SS1' */
void myfun_Update(real_T rtu_In1, DW_myfun_T *localDW)
{
    /* Update for DiscreteIntegrator: '<S1>/Integrator' */
    localDW->Integrator_DSTATE += rtu_In1;
}
```

The reusable function `myfun` passes in inputs and states as arguments to the subsystem function.

For more information, see “Generate Reentrant Code from Subsystems” on page 10-28 and “Generate Reusable Code from Library Subsystems Shared Across Models” on page 10-36.

Generate Subsystem as a Nonreusable Function

- 1 Open a model with a subsystem, such as `rtwdemo_atomic`.



Copyright 1994-2020 The MathWorks, Inc.

If you are using Embedded Coder, from the **C Code** tab, click **View Code** to open the **Code View** editor.

- 2 Right-click the Subsystem block. From the context menu, select **Block Parameters (Subsystem)**.
- 3 In the Subsystem Parameters dialog box, verify that **Treat as atomic unit** is selected. With that parameter selected, on the **Code Generation** tab, the **Function packaging** parameter is available.
- 4 Click the **Code Generation** tab and select **Nonreusable** function from the **Function packaging** parameter. This enables two parameters:
 - The **Function name options** parameter controls the naming of the generated function.
 - The **File name options** parameter controls the naming of the generated file.
- 5 Set the **File name options** parameter to **Use subsystem name**.

If you are using Embedded Coder, verify that the **Function interface** parameter is set to **void_void**. For more information, see "Generate Predictable Function Interface to Match Graphical Block Interface" (Embedded Coder).

- 6 Click **Apply** and close the dialog box.
- 7 If you are using Embedded Coder, open the Configuration Parameters dialog box. Verify that the model configuration parameter **File packaging format** (Embedded Coder) is set to **Modular**.
- 8 Generate the code.

```
#include "SS1.h"

/* Include model header file for global data */
#include "rtwdemo_atomic.h"
#include "rtwdemo_atomic_private.h"

/* Outputs for atomic system: '<Root>/SS1' */
void myfun(void)
{
    /* Output: '<Root>/Out1' incorporates:
```

```
    * DiscreteIntegrator: '<S1>/Integrator'  
    */  
    rtwdemo_atomic_Y.Out1 = rtwdemo_atomic_DW.Integrator_DSTATE;  
}  
  
/* Update for atomic system: '<Root>/SS1' */  
void myfun_Update(void)  
{  
    /* Update for DiscreteIntegrator: '<S1>/Integrator' */  
    rtwdemo_atomic_DW.Integrator_DSTATE += rtwdemo_atomic_B.Sum;  
}
```

The nonreusable function `myfun` passes in inputs and states through the global data structures `rtwdemo_atomic_Y` and `rtwdemo_atomic_DW`.

See Also

More About

- “Generate Reentrant Code from Subsystems” on page 10-28
- “Generate Reusable Code from Library Subsystems Shared Across Models” on page 10-36
- “Generate Code and Executables for Individual Subsystems” on page 7-2
- “Generate Predictable Function Interface to Match Graphical Block Interface” (Embedded Coder)
- “Remove Algebraic Loops”

Referenced Models in Simulink Coder

- “Generate Code for Model Reference Hierarchy” on page 8-2
- “Set Configuration Parameters for Code Generation of Model Hierarchies” on page 8-8
- “Convert Subsystem to Referenced Model and Generate Code” on page 8-13
- “Generate Reusable Code from Model Reference Hierarchy” on page 8-16
- “Storage Classes for Parameters and Signals Used in Model Blocks” on page 8-19
- “Code Generation Limitations for Model Reference” on page 8-22
- “Simulink Cache Files for Incremental Code Generation” on page 8-24
- “Specify Instance-Specific Parameter Values for Reusable Referenced Model” on page 8-27
- “Generated Functions in a Model Reference Hierarchy” on page 8-51

Generate Code for Model Reference Hierarchy

To achieve incremental loading of models, incremental code generation, modular development of referenced models, and code reuse, you can generate code for the model reference hierarchy.

You can include one model inside another model by using a Model block. Each instance of a Model block is a model reference. For simulation and code generation, blocks within a referenced model execute together as a unit. The model that contains a referenced model is a parent model. A collection of parent and referenced models constitutes a model hierarchy.

Model Reference Technology

When you generate code for a referenced model hierarchy, the code generator produces a standalone executable for the top model and a library module called a model reference target for each referenced model. When the code executes, the top model executable invokes the model reference targets to compute the referenced model output. Model reference targets are sometimes called *Simulink Coder targets*.

Do not confuse a model reference target (Simulink Coder target) with these other types of targets:

- Target hardware — A platform for which the Simulink Coder software generates code.
- System target file — A file that instructs the Simulink Coder software how to generate code for particular purpose.
- Rapid Simulation target (RSim) — A system target file that is part of the Simulink Coder product.
- Simulation target — A MEX file that implements a referenced model, which is executed by using Simulink Accelerator™ software.

The code generator places the code for the top model in a hierarchy in the code generation folder and places the code for referenced models in an `slprj` folder. Subfolders in `slprj` are separate places for different types of files. For folder information, see “Manage Build Process Folders” on page 24-30.

By default, the Simulink Coder uses *incremental code generation*. When generating code, the software compares structural checksums of referenced model files to the generated code files to determine whether to regenerate model reference targets. To control when rebuilds occur, use the configuration parameter **Model Referencing > Rebuild**. For details, see Rebuild.

The Simulink Coder software also uses *incremental loading*. The code for a referenced model is not loaded into memory until the code for its parent model executes and requires the outputs of the referenced model. Simulink Coder then loads the referenced model target and executes the code. Once loaded, the target remains in memory until it is no longer used.

If you have Embedded Coder, custom targets must declare themselves as compliant with the model reference if they support Model blocks. For more information, see “Support Model Referencing” on page 46-61.

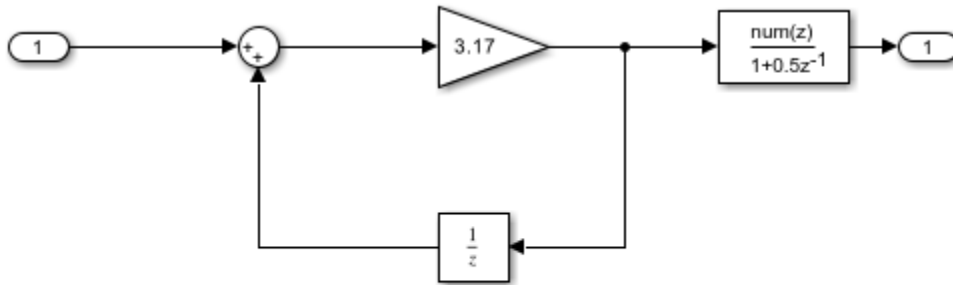
Generate Code for Model Reference Hierarchy

This example shows how to reference an existing model and generate code for the model reference hierarchy.

Create Model Reference Hierarchy

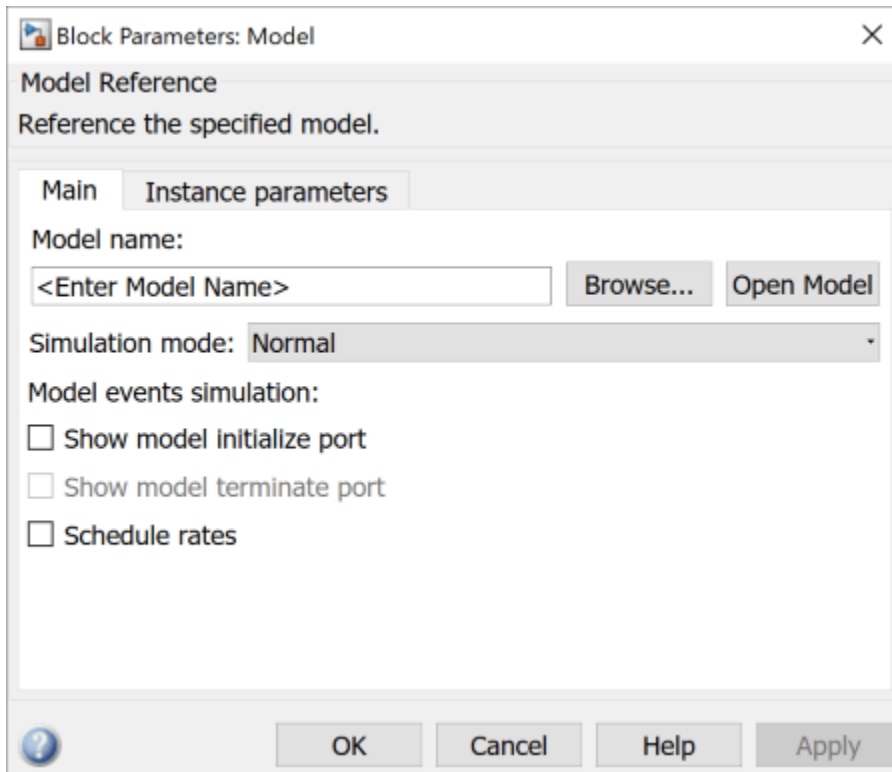
1. Open an existing model, for example the `ex_model_arg_ref` model. This model represents a reusable algorithm. This is a GRT model. If you have Embedded Coder, change the model configuration parameter **System target file** to `ert.tlc`.

```
open_system('ex_model_arg_ref');
```



Copyright 2021 The MathWorks, Inc.

2. A referenced model must specify that it is available to be referenced and whether it can be referenced just once or multiple times. Specify the model configuration parameter Total number of instances allowed per top model as **One** or **Multiple**. By default, **Total number of instances allowed per top model** is set to **Multiple**.
3. Create a new model that you want as a top model in the model reference hierarchy. The top model refers to other models as referenced models or multiple instances of a referenced model.
4. Add Inports and Outports to the top model as required. To add a referenced model, use the Model block.
5. The new block is initially unresolved because it does not specify a referenced model. To open the block parameters dialog box, double-click the Model block or use the context menu of the Model block.

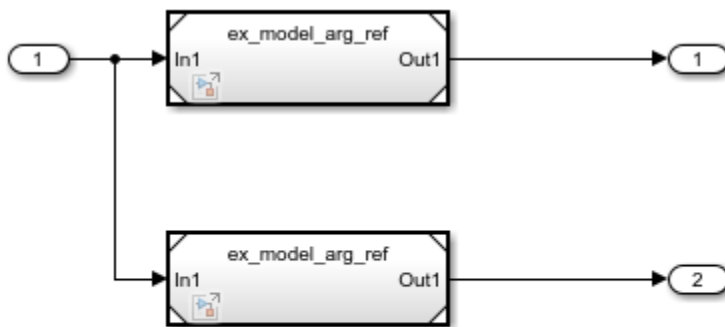


6. Enter the name of the referenced model in the **Model name** field. This name must contain fewer than 60 characters, exclusive of the `.slx` suffix.

7. Click **OK**. If the referenced model contains root-level inputs or outputs, the Model block displays corresponding input and output ports.

8. Create another instance of the Model block so that the top model looks like model:

```
open_system('ex_model_arg');
```



Copyright 2021 The MathWorks, Inc.

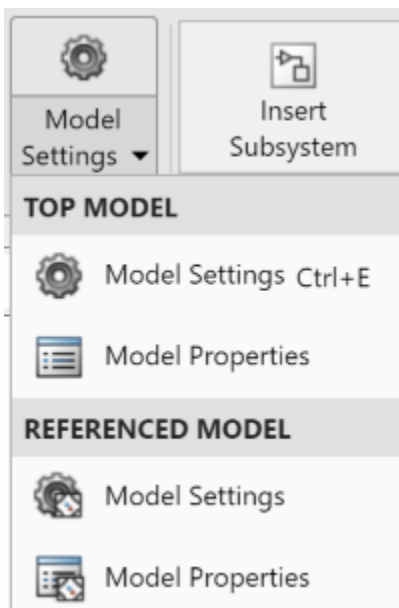
9. Name the model `ex_model_arg` and save the model.

You can also convert a subsystem to a referenced model. For more details, see “Convert Subsystems to Referenced Models”.

Generate Code for Model Reference Hierarchy

1. Verify that the model configuration parameter Solver selection **Type** is set to **Fixed-step** for the top model and the referenced model. To open the respective Configuration Parameters dialog boxes, on the **Modeling** tab, select **Model Settings** for the respective models.

For a model reference hierarchy that has multiple referenced models, you can access the **Model Settings** of the referenced model that is open and the top model through this menu. To access the **Model Settings** of another referenced model in the model hierarchy, open the referenced model to access this menu.



2. For the top model, open the **C Code** tab by selecting either **Simulink Coder** app or **Embedded Coder** app. To generate code for the model reference hierarchy, click the **Build** button.

The Simulink Coder build process generates and compiles code. The current folder now contains new files and a new folder.

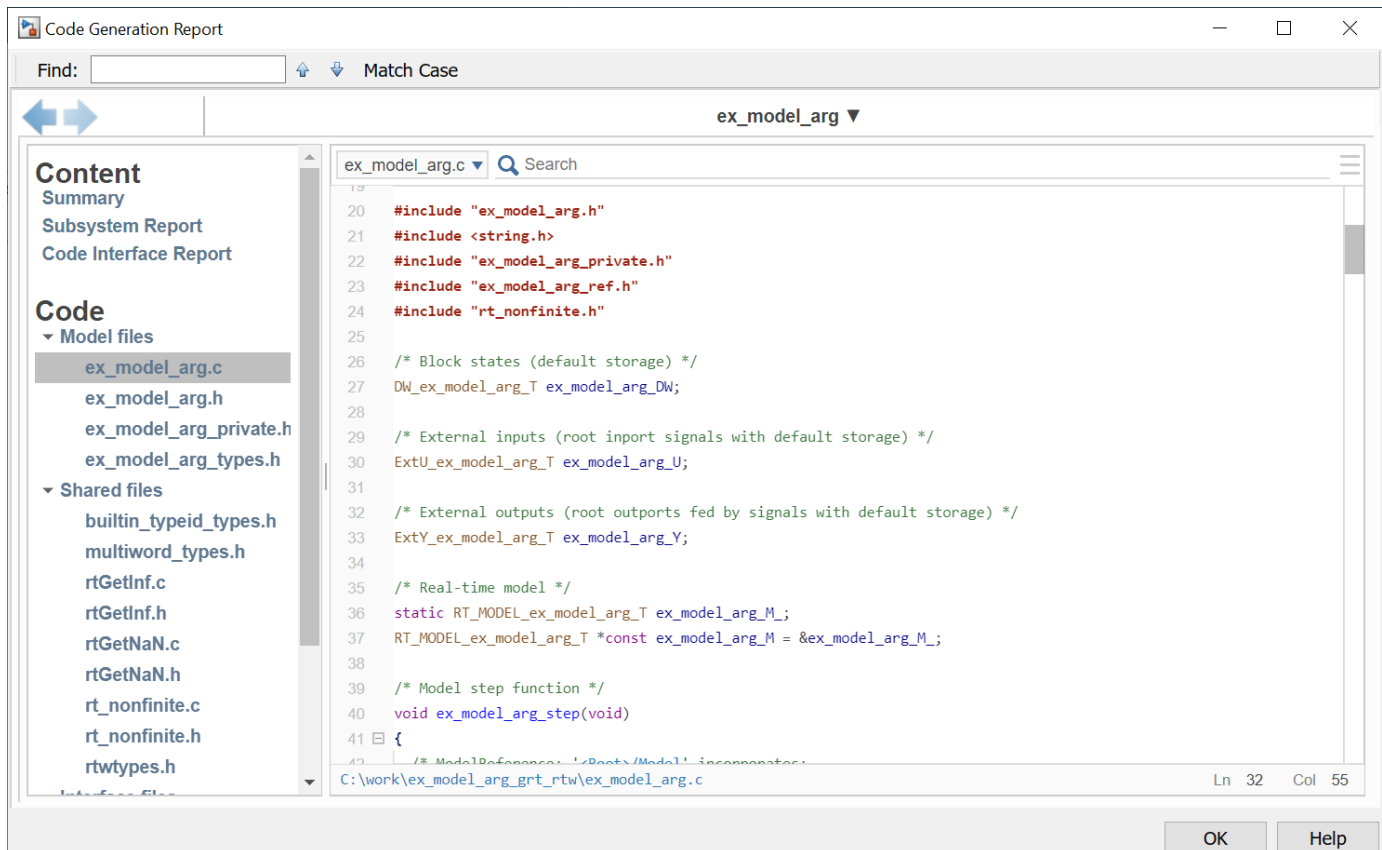
- `ex_model_arg_grt_rtw` — The build folder containing the generated code for the top model.
- `slprj > grt > ex_model_arg_ref` — The folder containing the generated code for the referenced model.
- `ex_model_arg.exe` — The executable created by the build process.
- `ex_model_arg.slxc` — The Simulink cache file for the top model.
- `ex_model_arg_ref.slxc` — The Simulink cache file for the referenced model.

For more information about the configuration parameter requirements, see “Set Configuration Parameters for Code Generation of Model Hierarchies” on page 8-8.

View Generated Code in Code Generation Report

To view the generated code, in the **C Code** tab, select **Open Report**. The code generation report contains hyperlinks to all the files generated for the model reference hierarchy.

To open the code generation report for a referenced model, at the top of the report, click `ex_model_arg`. In the drop-down, click `ex_model_arg_ref`. The source files for the referenced model are displayed along with the generated code for the referenced model.



To navigate back to the top model, click the **Go Backward** button on the top-left corner of the code generation report window.

Working with Code Generation Folders

Model reference code is generated in your code generation folder. Simulation target code is generated in your simulation cache folder. Due to the two different folders, there are constraints on:

- When and where you build model reference targets.
- How you access the model reference targets.

You can store the models referenced by Model blocks anywhere on the MATLAB path. A top model can include models stored on different file systems or in different folders. The simulation targets and generated code derived from these models cannot be stored on different file systems or in different folders. To allow code reuse, you must set up models referenced by a top model to simulate and generate model reference target code in a single code generation folder.

If you reference the same model from several top models, each stored in a different folder, you must choose one of these approaches:

- Work with the same code generation folder and be sure that the models are on your path.
- Allow separate code generation folders, simulation targets, and Simulink Coder targets to be generated in each folder in which you work.

The second approach requires maintenance of several instances of the model reference code. It is possible for generated code to become redundant, for example, when you make changes to the referenced model. To minimize code regeneration of referenced models, choose a specific code generation folder for all sessions.

Build Model Reference Targets

The code generator produces a model reference target directly from the Simulink model. The code generator generates or regenerates model reference targets, for example, when they require an update.

When you execute the `slbuild` command by using arguments in the MATLAB Command Window, Simulink and Simulink Coder can generate a simulation target for an accelerator mode referenced model and a model reference target for a referenced model.

The code generator produces only one model reference target for multiple instances of a referenced model. For more information, see “Generate Reentrant Code from Subsystems” on page 10-28.

Reduce Change Checking Time

You can reduce the time that Simulink and Simulink Coder spend checking whether model reference targets need to be rebuilt by setting configuration parameter values:

- In the top model, consider setting the model configuration parameter **Model Referencing** > **Rebuild** to `If changes in known dependencies detected`.
- In the referenced models throughout the hierarchy, set the configuration parameter **Diagnostics** > **Data Validity** > **Signal resolution** to `Explicit only` or `None`.

These parameter values exist in a referenced model configuration set, not in the individual Model block. Setting either value for an instance of a referenced model sets it for the instances of that model.

See Also

Related Examples

- “Specify Instance-Specific Parameter Values for Reusable Referenced Model” on page 8-27
- “Establish Data Ownership in a Model Hierarchy” (Embedded Coder)
- “Use Configuration Reference to Select Code Generation Target” (Embedded Coder)
- “Manage Simulation Targets for Referenced Models”
- “Generate Reentrant Code from Top Models” on page 10-13

Set Configuration Parameters for Code Generation of Model Hierarchies

A model reference hierarchy must satisfy various Simulink Coder requirements, as described in this topic. A model referencing hierarchy to be processed by the Simulink Coder software must also satisfy:

- The Simulink requirements listed in:
 - “Set Configuration Parameters for Model Hierarchies”
 - “Model Reference Interface and Boundary”
- The Simulink limitations listed in “Signal Requirements and Limitations”
- The Simulink Coder limitations listed in “Code Generation Limitations for Model Reference” on page 8-22

Configuration Parameter Requirements

A referenced model uses a configuration set in the same way a top model does. By default, every model in a hierarchy has its own configuration set, which it uses in the same way that it would if the model executed independently.

Because each model can have its own configuration set, configuration parameter values can be different in different models. Furthermore, some parameter values are intrinsically incompatible with model referencing. The response of the code generator to an inconsistent or unusable configuration parameter depends on the parameter:

- Where a parameter inconsistency has no significance, the code generator ignores or resolves the inconsistency without posting a warning.
- Where a nontrivial and possibly acceptable solution exists to an inconsistent parameter, the code generator resolves the conflict silently; resolves it with a warning; or generates an error.
- If an acceptable resolution is not possible, the code generator produces an error. You must then change parameter values to eliminate the issue.

When a model reference hierarchy contains many referenced models that have incompatible parameter values, or a changed parameter value must propagate to many referenced models, manually eliminating configuration parameter incompatibilities can be time-consuming. You can control or eliminate such overhead by using configuration references to assign an externally stored configuration set to multiple models. For more information, see “Share a Configuration with Multiple Models”.

To select a code generation target for a model reference hierarchy without modifying the individual models, use a configuration reference in the Simulink data dictionary. For more information, see “Use Configuration Reference to Select Code Generation Target” (Embedded Coder).

The following tables list configuration parameters that can cause problems if set in certain ways, or if set differently in a referenced model than in a parent model. Where possible, the code generator resolves violations of these requirements, but most cases require you to make changes to parameters in your models.

Configuration Requirements for Model Referencing with System Target Files

Dialog Box Pane	Option	Requirement
Solver	Start time	Some system target files require the start time of models to be zero.
Hardware Implementation	All options	Values must be the same for top and referenced models.
Code Generation	System target file	Must be the same for top and referenced models.
	Language	Must be the same for top and referenced models.
	Generate code only	Must be the same for top and referenced models.
Identifiers	Maximum identifier length	Cannot be longer for a referenced model than for its parent model.
Interface	Code replacement library	Must be the same for top and referenced models.
	ASAP2 interface	Can be on or off in a top model, but must be off in a referenced model. If it is not off, the code generator temporarily sets it to off during code generation.

Configuration Requirements for Model Referencing with ERT System Target Files (Requires Embedded Coder)

Dialog Box Pane	Option	Requirement
Identifiers	Global variables Global types Subsystem methods Local temporary variables Constant macros	\$R token must be included.
	Signal naming	Must be the same for top and referenced models.
	M-function	If specified, must be the same for top and referenced models.
	Parameter naming	Must be the same for top and referenced models.
	#define naming	Must be the same for top and referenced models.
Interface	Ignore custom storage classes	Must be the same for top and referenced models.
	Support floating-point numbers	Must be the same for top and referenced models.
	Support non-finite numbers	If off for top model, must be off for referenced models.
	Support complex numbers	If off for top model, must be off for referenced models.
	Remove error status field in real-time model data structure	If on for top model, must be on for referenced models.
Code Placement	Use owner from data object for data definition placement	Must be the same for top and referenced models.
	Signal display level	Must be the same for top and referenced models.
	Parameter tune level	Must be the same for top and referenced models.

Configuration Parameters Changed During Code Generation

For referenced models, if these **Configuration Parameters > Code Generation > Identifiers** parameters have settings that do not contain a \$R token (which represents the name of the referenced model), code generation prepends the \$R token to the identifier format:

- **Global variables** (CustomSymbolStrGlobalVar)
- **Global types** (CustomSymbolStrType)
- **Subsystem methods** (CustomSymbolStrFcn)

- **Constant macros** (CustomSymbolStrMacro)

You can use the Model Advisor to identify models in a model referencing hierarchy for which code generation changes configuration parameter settings.

- 1 In the Simulink Editor, from the **Modeling** tab, select **Model Advisor**.
- 2 Expand **By Task > Model Referencing**.
- 3 Run the **Check code generation identifier formats used for model reference** check.

If a script that operates on generated code uses identifier formats that code generation changes, then update the script to use the updated identifier format (which includes an appended \$R token).

For more information about identifiers, see “Identifier Format Control” (Embedded Coder).

Naming Requirements

Within a model that uses model referencing, names of the constituent models cannot collide. When you generate code from a model that uses model referencing, the **Maximum identifier length** parameter must be large enough to accommodate the root model name and the name-mangling text. A code generation error occurs if **Maximum identifier length** is not large enough.

When a name conflict occurs between a symbol within the scope of a higher-level model and a symbol within the scope of a referenced model, the symbol from the referenced model is preserved. The code generator performs name mangling on the symbol from the higher-level model.

Embedded Coder Naming Requirements

The Embedded Coder product lets you control the formatting of generated symbols in much greater detail. When generating code with an ERT target from a model that uses model referencing:

- The \$R token (in addition to the \$M token) must be included in the **Identifier format control** parameter specifications except for **Shared utilities identifier format**.
- The **Maximum identifier length** must be large enough to accommodate full expansions of the \$R and \$M tokens.

For more information, see “Model Configuration Parameters: Code Generation Identifiers”.

Custom Target Requirements

If you have Embedded Coder, a custom target must meet various requirements to support model referencing. For details, see “Support Model Referencing” on page 46-61.

Miscellaneous Considerations

Minimize occurrences of algebraic loops by selecting the **Minimize algebraic loop occurrences** parameter on the **Model Referencing** pane. The setting of this option affects only generation of code from the model. For more information about direct feed through, see “Algebraic Loop Concepts”.

Use the **Integer rounding mode** parameter on your model's blocks to simulate the rounding behavior of the C compiler that you intend to use to compile code generated from the model. This setting appears on the **Signal Attributes** pane of the parameter dialog boxes of blocks that can perform signed integer arithmetic, such as the Product, Matrix Multiply and n-D Lookup Table blocks.

For most blocks, the value of **Integer rounding mode** completely defines rounding behavior. For blocks that support fixed-point data and the Simplest rounding mode, the value of **Signed integer division rounds to** also affects rounding. For details, see “Rounding” (Fixed-Point Designer).

See Also

Related Examples

- “Manage Configuration Sets for a Model”
- “Set Configuration Parameters for Model Hierarchies”

Convert Subsystem to Referenced Model and Generate Code

Converting subsystems to referenced models enhances incremental code generation, modular development of referenced models, and code reuse. Many large models use a combination of referenced models and subsystems.

To convert a Subsystem block to a Model block, use any of these methods:

- In the Simulink Editor, select a Subsystem block. On the Simulink Toolstrip, the **Subsystem Block** tab, select **Convert > Model Block**.
- In the MATLAB Command Window, use the `Simulink.SubSystem.convertToModelReference` function with the `'UseConversionAdvisor'` argument set to `true`.

Before you convert a subsystem to a referenced model:

- 1 Configure the Subsystem block as an atomic subsystem. In the Subsystem Block Parameters, select the **Treat as atomic unit** property.

An atomic subsystem executes as a unit relative to the parent model. Subsystem execution does not interleave with parent block execution. This property makes it possible to extract subsystems for use as standalone models and as functions in the generated code.

- 2 Set the model configuration parameter **Signal resolution** to `Explicit only` or `None`.

You can map a storage class to a category of model data elements or to an individual element by using the Code Mappings editor or the Code Mappings API (`coder.mapping.api.CodeMapping`). Then, use the Code Mappings editor or the API to specify code configurations settings specific for that storage class, such as header files, definition files, and memory sections. If your model contains code mappings, you can migrate the code mappings when you convert a subsystem to a referenced model. For more information, see **Code Mappings Editor - C**.

Configure Code Mappings

- 1 Open the example model `RollAxisAutopilot`.


```
openExample('RollAxisAutopilot')
```
- 2 On the Simulink toolstrip, the **Apps** tab, select the **Embedded Coder** or **Simulink Coder** app.
- 3 In the **C Code** tab, click **Code Interface > Default Code Mappings**. This option pins the Code Mappings editor.
- 4 Map a storage class to a category of model data element. In the Code Mappings editor, select the **Data Defaults** tab. For the **Inports** category, specify the **Storage Class** as `ExportedGlobal`.

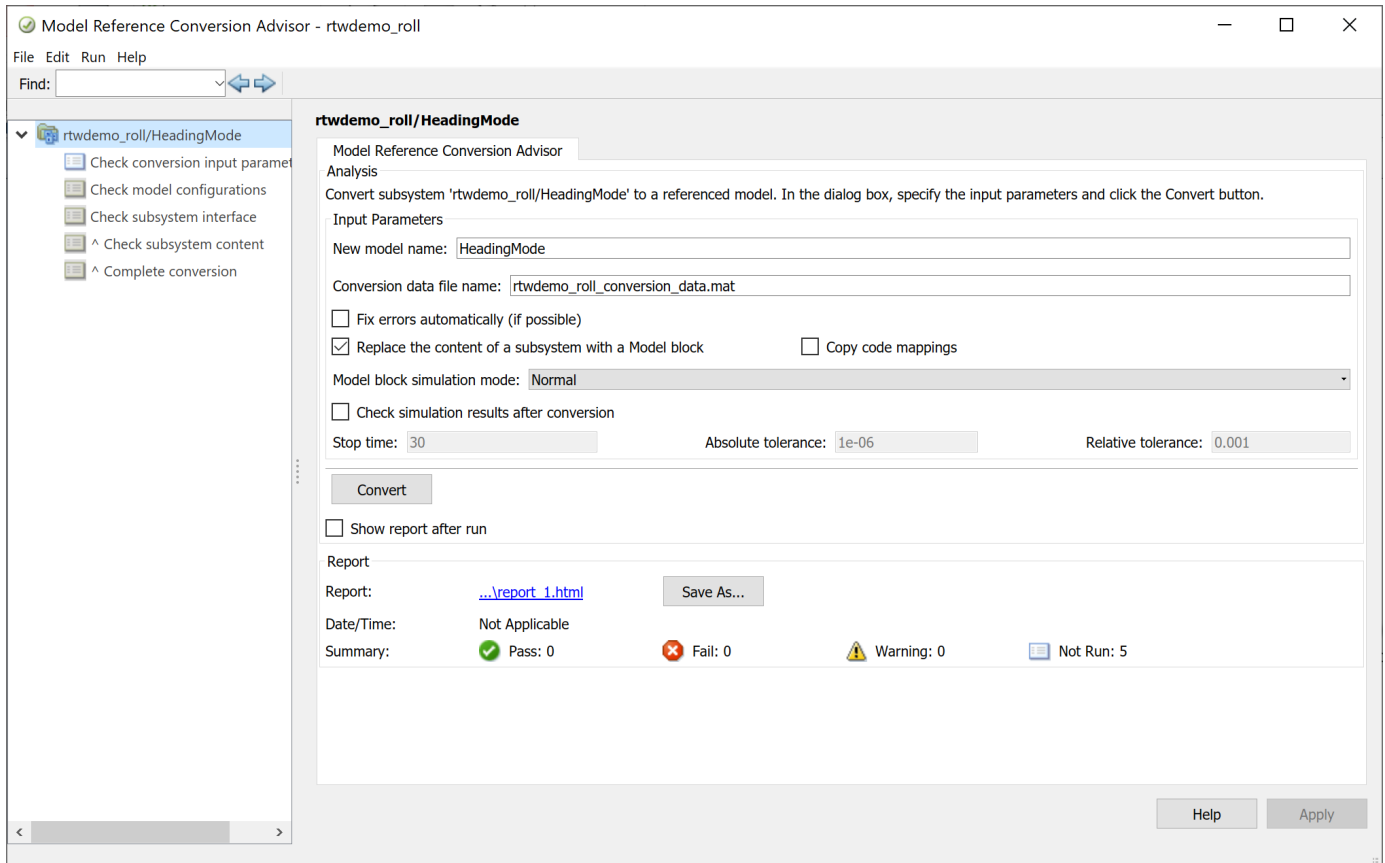
Alternatively, to configure the default mapping for Inports, in the MATLAB Command Window, enter:

```
roll_mapping = coder.mapping.api.get('RollAxisAutopilot');
setDataDefault(roll_mapping, 'Inports', 'StorageClass', 'ExportedGlobal');
```

Convert Subsystem to Referenced Model

- 1 To convert a subsystem to a referenced model, open the **Model Reference Conversion Advisor**. Select the subsystem `HeadingMode`. On the **Subsystem Block** tab, select **Convert > Model Block**.

The Model Reference Conversion Advisor enables you to interactively specify conversion parameters and fix issues that the advisor finds.



- 2 Review the autogenerated model name. If needed, you can update this name.
- 3 To make the conversion process faster, select the **Fix errors automatically (if possible)**. This option fixes some conversion issues. You do not control the fixes.
- 4 To copy the code mappings specified for the model, select the **Copy code mappings** option. This option migrates the code mappings from the parent model to the newly converted referenced model. By default, this option is cleared. This option is not supported for models configured with an ERT-based system target file and service code interface.
- 5 Click **Convert**.

The Model Reference Conversion Advisor steps through each of the checks. Address any issues that the advisor reports. After the Model Reference Conversion Advisor successfully completes the conversion, it replaces the Subsystem block with a Model block that references the new model.

Alternatively, to convert a subsystem to a referenced model, in the MATLAB Command Window, enter:

```
Simulink.SubSystem.convertToModelReference(...
    'RollAxisAutopilot/HeadingMode', ...
    'HeadingMode', ...
    'AutoFix',true,...
    'ReplaceSubsystem',true,...
    'CopyCodeMappings',true);
```

Copy Code Mappings After Subsystem Conversion

If you want to copy the code mappings from the parent model to the referenced model and you did not previously select the **Copy code mappings** option in the Model Reference Conversion Advisor, in the MATLAB Command Window, enter:

```
roll_mapping = coder.mapping.api.get('RollAxisAutopilot');
roll_sc = getDataDefault(roll_mapping, 'Inports', 'StorageClass');

mode_mapping = coder.mapping.api.get('HeadingMode');
setDataDefault(mode_mapping, 'Inports', 'StorageClass', roll_sc);
```

Review Generated Code

For the RollAxisAutopilot, model which contains the referenced model HeadingMode, open the **C Code** tab. To generate code for the model reference hierarchy, click **Build**.

The Simulink Coder build process generates and compiles code. The current folder now contains new files and a new folder.

Folder or File	Purpose
RollAxisAutopilot_grt_rtw	The build folder containing the generated code for the top model.
slprj > grt > Heading Mode	The build folder containing the generated code for the referenced model.
RollAxisAutopilot.exe	The executable created by the build process.
RollAxisAutopilot.slxc	The Simulink cache file for the top model.
HeadingMode.slxc	The Simulink cache file for the referenced model.

See Also

Related Examples

- “Generate Code for Model Reference Hierarchy” on page 8-2
- “Convert Subsystems to Referenced Models”

Generate Reusable Code from Model Reference Hierarchy

A reusable subsystem function that has input or output connected to the root Inport or Outport block of a referenced model can impact code reuse. It can prevent you from reusing atomic subsystems in a reference model context in the same way you might reuse the subsystems in a standalone model.

This example shows how to generate reusable code when you have a reusable subsystem referenced inside Model block.

Generate Reusable Code for Subsystem

1. Convert these blocks into a subsystem. Drag a box to outline the subsystem that you want to create. From the **Multiple** tab on the Simulink Editor toolstrip, select **Create Subsystem**.

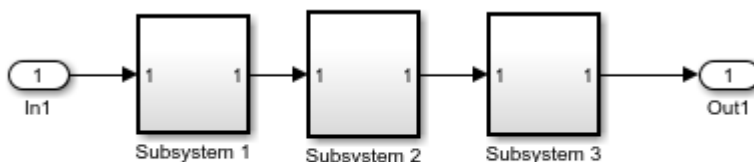


Copyright 2021 The MathWorks, Inc.

2. Make these changes to the Subsystem block parameters:

- Select “Treat as atomic unit”.
- On the **Code Generation** tab, set “Function packaging” to Reusable function.

3. Create a model that includes three instances of the preceding subsystem.



Copyright 2021 The MathWorks, Inc.

4. Set the model configuration parameter Default parameter behavior to Inlined.

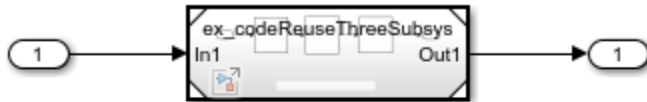
5. To build the model and generate code, press **Ctrl+B**.

The code generator optimizes the code by generating one copy of the function for the reused subsystem in the `model.c` file.

```
void ex_codeReuseThreeSubsys_Subsystem1(real_T rtu_In1,
    B_Subsystem1_ex_codeReuseThreeSubsys_T *localB)
{
    /* Gain: '<S1>/Gain' */
    localB->Gain = 3.0 * rtu_In1;
}
```

Generate Reusable Code for Subsystem in Referenced Model

1. Create a top model by using a Model block that references the preceding model that has three instances of the reusable subsystem.



Copyright 2021 The MathWorks, Inc.

2. Generate code for the top model. The code generator generates three different function signatures in the slprj folder.

```

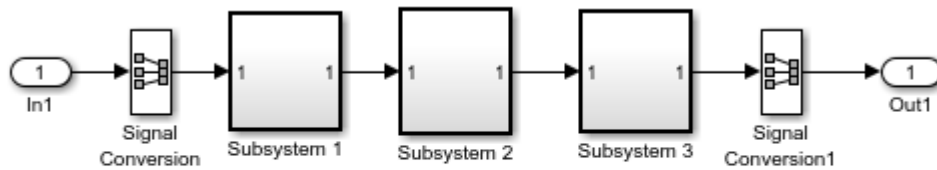
/* Output and update for atomic system: '<Root>/Subsystem 1' */
void ex_codeReuseThreeSubsys_Subsystem1(const real_T *rtu_In1,
    B_Subsystem1_ex_codeReuseThreeSubsys_T *localB)
{
    /* Gain: '<S1>/Gain' */
    localB->Gain = 3.0 * *rtu_In1;
}

/* Output and update for atomic system: '<Root>/Subsystem 2' */
void ex_codeReuseThreeSubsys_Subsystem2(real_T rtu_In1,
    B_Subsystem2_ex_codeReuseThreeSubsys_T *localB)
{
    /* Gain: '<S2>/Gain' */
    localB->Gain = 3.0 * rtu_In1;
}

/* Output and update for atomic system: '<Root>/Subsystem 3' */
void ex_codeReuseThreeSubsys_Subsystem3(real_T rtu_In1, real_T *rty_Out1)
{
    /* Gain: '<S3>/Gain' */
    *rty_Out1 = 3.0 * rtu_In1;
}

```

3. To enable the subsystem functions for code reuse, insert Signal Conversion blocks into the referenced model. Place one block between the Inport and Subsystem 1 and another block between Subsystem 3 and the Outport.



Copyright 2021 The MathWorks, Inc.

If the subsystem has a Merge block with initial conditions, do not add a Signal Conversion block. Instead, add a Bias block to obtain a reusable function.

4. Select the model configuration parameter Pass fixed-size scalar root inputs by value for code generation.

5. Generate code for the top model. The code generator generates a single reusable function in the `slprj` folder.

```
void ex_codeReuseSubsysSigConvert_Subsystem1(real_T rtu_In1,
      B_Subsystem1_ex_codeReuseSubsysSigConvert_T *localB,
      P_Subsystem1_ex_codeReuseSubsysSigConvert_T *localP)
{
  /* Gain: '<S1>/Gain' */
  localB->Gain = localP->Gain_Gain * rtu_In1;
}
```

See Also

Related Examples

- “Convert Subsystems to Referenced Models”
- “Reference Existing Models”
- “Generate Reusable Code from Library Subsystems Shared Across Models” on page 10-36

Storage Classes for Parameters and Signals Used in Model Blocks

To transfer data between your referenced model and top model, use signals and parameters. You can also use parameters to integrate external data into the model reference hierarchy. To include one model inside another model and create a model reference hierarchy, use a Model block.

You can apply a storage class to a data item in a model, such as signal or parameter, to:

- Make the data item appear in the generated code as a global symbol, typically a global variable.
- Prevent optimizations such as **Default parameter behavior** and **Signal storage reuse** from eliminating the data item from the generated code.

For examples, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.

You can map a storage class to a category of model data elements or to an individual element by using the Code Mappings editor or the Code Mappings API (`coder.mapping.api.CodeMapping`). You can specify code configurations settings specific to that storage class, such as header files, definition files, and memory sections using the Code Mappings editor or the API. For more information, see **Code Mappings Editor - C**.

Storage Classes for Parameters in Model Blocks

Models containing Model blocks can use storage classes for parameters to interface with external code. Storage classes, with the exception of `Auto`, are tunable. The supported storage classes include:

- Predefined storage classes in Simulink Coder and Embedded Coder
- A new storage class created by using the Embedded Coder Dictionary

For limitations on Embedded Coder storage classes, see “Storage Class Limitations” on page 15-84.

The restrictions on parameters in referenced models are:

- Tunable parameters are not supported for noninlined S-functions.
- Tunable parameters set through the Model Parameter Configuration dialog box are ignored.

Concerning how global tunable parameters are declared, defined, and used in code generated for targets, consider:

- A global tunable parameter is a parameter in the base workspace that has a storage class other than `Auto`.
- An `extern` declaration is generated for models that use a given parameter.
- If a parameter is exported, the top model defines (allocates memory for) the parameter whether the top model uses the parameter or not.
- Global parameters are accessed directly as global memory. They are not passed as arguments to the functions that are generated for the referenced models.
- In a referenced model that sets the default storage class for a category of parameter data to `Default` in the Code Mapping editor, symbols for `Model default` parameters are generated by using unstructured variables (`rtP_XXX`) instead of being written into the `model_P` structure. You can then compile each referenced model independently.

Parameters that you use as Model block arguments must be defined in the referenced model workspace. For details, see “Parameterize Instances of a Reusable Referenced Model”.

Storage Classes for Signals in Model Blocks

Models containing Model blocks can use signals of storage class `Auto`.

A global signal is a signal that has a storage class other than `Auto`. The supported storage classes include:

- Predefined storage classes in Simulink Coder and Embedded Coder
- A new storage class created by using Embedded Coder Dictionary

If you have Embedded Coder, you can create a global signal by using a storage class defined in the Embedded Coder Dictionary that has **Storage Type** set as `Unstructured` or by creating a storage in the Embedded Coder Dictionary that has **Storage Type** set as `Unstructured`.

These signals are distinct from individual signals that use the storage class `Model default`. When you set the default storage class of the corresponding data category to `Default` in the Code Mapping editor, these signals are treated as test points that have the `Auto` storage class.

When you declare signals as global, be aware of how the signal data is handled. Global signals are declared, defined, and used as follows:

- An `extern` declaration is generated for models that use a global signal.
As a result, if a signal crosses a Model block boundary, the top model and the referenced model both generate `extern` declarations for the signal.
- For an exported signal, the top model defines (allocates memory for) the signal, whether or not the top model itself uses the signal.
- Global signals used by a referenced model are accessed directly as global memory. They are not passed as arguments to the functions that are generated for the referenced models.

Storage classes that you create by using the Embedded Coder Dictionary also follow the preceding rules for the top model.

Note In a referenced model, you cannot use the storage class that you create by using the Embedded Coder Dictionary on root-level Inports and Outports.

Certain storage classes have limited support for use with model reference. See “Storage Class Limitations” on page 15-84.

Signal Name Mismatches Across Model Reference Boundary

Within a parent model, the name and storage class for a signal entering or leaving a Model block might not match those of the signal attached to the root inport or outport within that referenced model. Because referenced models are compiled independently without regard to a parent model, they cannot adapt to the possible variations in how parent models label and store signals.

The code generator accepts cases where input and output signals in a referenced model have the `Auto` storage class. When such signals are test pointed or are global, certain restrictions apply. This

table describes how mismatches in signal labels and storage classes between parent and referenced models are handled by the software.

Relationships of Signals and Storage Classes Across Model Reference Boundary

Storage Class in Referenced Model	Storage Class in Parent Model	Signal Passing Method	Signal Mismatch Checking
Auto	Non-Auto	Function argument	Signal label mismatch
Model default (when the Code Mapping editor specifies Default storage class) or resolved to Signal Object	Non-Auto	Function argument	Signal label mismatch
Non-Auto	Auto	Global variable	None
Non-Auto	Non-Auto	Global variable	One of the storage classes must be Auto (otherwise there is an error)

These signal resolution rules apply to code generation:

- If the storage class of a root input or output signal in a referenced model is **Auto** or **Model default** (when you set the storage class of the corresponding data category to **Default** in the Code Mapping editor), the signal is passed as a function argument. When such a signal is **Non-Auto** or resolves to a **Simulink.Signal** object in the parent model, the **Signal label mismatch** diagnostic is applied.
- If a root input or output signal in a referenced model is global, the signal is communicated by using direct memory access (global variable). The corresponding signal in the parent model must be **Auto**.
- You cannot use a storage class that you create by using the Embedded Coder Dictionary in a referenced model.

You can set the model configuration parameter **Signal label mismatch** diagnostic to error, warning, or none.

See Also

“Signal label mismatch”

Related Examples

- “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75
- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Establish Data Ownership in a Model Hierarchy” (Embedded Coder)
- “Flexible Storage Class for Different Model Hierarchy Contexts” (Embedded Coder)

Code Generation Limitations for Model Reference

Code generation limitations for model references influence how you properly set up the model reference hierarchy for code generation.

In addition to these limitations, a model reference hierarchy used for code generation must satisfy:

- The Simulink requirements and limitations listed in:
 - “Configuration Requirements for All Referenced Model Simulation”
 - “Model Reference Requirements and Limitations”
 - “Model Reference Interface and Boundary”
- The Simulink Coder requirements applicable to the code generation target, as listed in “Configuration Parameter Requirements” on page 8-8.

Storage Class Limitations

If you have Embedded Coder, some restrictions exist on grouped storage classes in referenced models. For details, see “Storage Class Limitations” on page 15-84.

Data Logging Limitations

- To Workspace blocks and Scope blocks are ignored when Simulink Coder generates code for a referenced model. The resulting code is the same as if the blocks did not exist. In some cases for models with Scope block and MAT-file logging enabled, the code generator produces an extra PWork in the model's DWork structure.
- Code generated for referenced models cannot log data to MAT-files. If you enable data logging for a referenced model, Simulink Coder disables the option before code generation and re-enables it afterward.
- If you log states for a model that contains referenced models, the ordering of the states in the output is determined by block sorted order, and might not match between simulation output and generated code MAT-file logging output.

State Initialization Limitations

When a top model uses the model configuration parameter **Initial state** to specify initial conditions, Simulink Coder does not initialize the discrete states of the referenced models during code generation.

Model Reuse Limitations

If a referenced model used for code generation has at least one of the following characteristics, the model must specify the model configuration parameter **Total number of instances allowed per top model** as One. Other instances of the model can exist in the hierarchy. If you do not set the parameter to One, or more than one instance of the model exists in the hierarchy, an error occurs. The characteristics are:

- The model references another model that has been set to a single instance.
- The model contains an internal signal or state that has a storage class that is not supported for multi-instance models. You must set the storage class to `Auto` or `Model default` for internal

signals and states. The default storage class for internal data must be a multi-instance storage class.

- The model uses exported Stateflow graphical functions.
- The model contains an S-function that is:
 - Inlined but you have not set the option `SS_OPTION_WORKS_WITH_CODE_REUSE`
 - Not inlined
- The model contains a function-call subsystem that:
 - The Simulink engine forces to be a function
 - Is called by a wide signal

S-Functions Limitations

- A referenced model cannot use noninlined S-functions generated by Simulink Coder.
- The Simulink Coder S-function target does not support model referencing.

For additional information, see “S-Functions in Referenced Models”.

Subsystems Limitations

If you generate code for an atomic subsystem as a reusable function, inputs or outputs that connect the subsystem to a referenced model might prevent code reuse, as described in “Generate Reentrant Code from Subsystems” on page 10-28.

Targets Limitation

The Simulink Coder S-function target does not support model referencing.

Conditional Models Limitation

When a model contains a trigger or enable port, you cannot generate standalone Simulink Coder code.

Local Solver Limitation

You can not generate code for a model that uses a local solver. For more information on local solvers, see “Use Local Solvers in Referenced Models”.

See Also

Related Examples

- “Set Configuration Parameters for Code Generation of Model Hierarchies” on page 8-8
- “Set Configuration Parameters for Model Hierarchies”

Simulink Cache Files for Incremental Code Generation

Share the generated code for various target types between team members inside an organization by using Simulink cache files. When you generate code from a Simulink model hierarchy, each referenced model has an associated Simulink cache file. These files have an `.slxc` file extension. The cache files contain artifacts from the build that can reduce the time required for successive model simulation and code generation. Simulink cache files are generated when you simulate models in accelerator or rapid accelerator mode or generate code for models. For more information, see “Share Simulink Cache Files for Faster Simulation”.

Contents of Simulink Cache Files

At the end of a build, the code generator packages the artifacts and generated code for the model hierarchy in the Simulink cache files. The cached artifacts include:

- Shared utilities
- All files under the `slprj\system_target_file\model_name`
- `model_name_rtw_system_target_file` folder for the top model
- Any binaries generated for the top model

You cannot modify the contents of the Simulink cache files. If you update the build artifacts inside the `slprj` folder, the Simulink cache file picks up these updates during the next build cycle.

Share Code Generation Artifacts

If you are involved in team development of large models that reference other models, use Simulink cache files to synchronize builds and share generated code artifacts to enhance incremental code generation. You can leverage Simulink cache files to avoid unnecessary code generation.

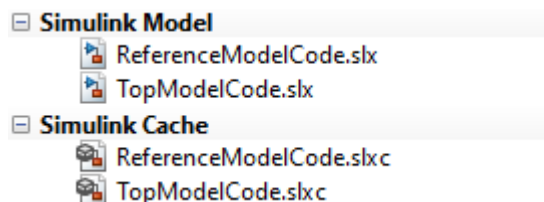
For example, assume that you collaborate with your team members by using a source control system and your models are built every night in a continuous integration system.

- 1 Open your model. For example, `TopModelCode`.

```
model = 'TopModelCode';
openExample('simulinkcoder/FilePackagingModelsCodeAndDataExample', ...
    'supportingFile', model);
```

- 2 Build the model. On the **C Code** tab, click **Build**.

Observe the Simulink cache files generated for all referenced models. The `ReferenceModelCode` model is a referenced model in `TopModelCode`.



Simulink stores cache files in the folder specified by the “Simulation cache folder” preference.

To see all supported configurations, double-click the cache file. The Simulink cache report contains information related to each release, platform, and list of packaged targets.

Simulink cache for TopModelCode

This Simulink cache contains derived files for the following releases and platforms:

R2022b : win64

Code generation

Target	Build Type	Folder Structure
ert	Top model	Model specific
grt	Top model	Model specific

Alternatively, to get the contents of the Simulink cache file in a MATLAB table, use the `slxcinfo` function.

- 3 Submit your models to the source control system. The continuous integration system builds the models overnight and updates the existing Simulink cache files.

Note Because Simulink cache files are derived files, do not submit them to source control. If you share Simulink cache files by storing them in a source control system, you cannot diff or merge different versions of these files.

- 4 Assume your team member made some changes to the model and submitted the changes to the source control system. When you want to work with the model hierarchy, fetch these files:
 - Design files (model and associated files) from the source control system.
 - Simulink cache files from the continuous integration system.
- 5 Make your changes to the model.
- 6 Build the model. The code generator extracts code generation artifacts stored in the Simulink cache files from the overnight build and avoids unnecessary rebuilds and code regeneration.

The time to build the model and generate code is less than the first time that you built the model.

Note To unpack the simulation and code generation targets from the Simulink cache files without updating, simulating, or generating code for the model hierarchy, use the `slxcunpack` function.

- 7 Submit your updated model files to the source control system.

See Also

`slxcinfo` | `slxcunpack`

Related Examples

- “Share Simulink Cache Files for Faster Simulation”
- “Reduce Build Time for Referenced Models by Using Parallel Builds” on page 31-51

More About

- “Pull, Push, and Fetch Files with Git”

External Websites

- Simulink Cache (1 min, 27 sec)

Specify Instance-Specific Parameter Values for Reusable Referenced Model

When you use model referencing to break a large system into components, each component is a separate model. You can reuse a component by referring to it with multiple Model blocks. Each Model block is an instance of the component. You can then configure a block parameter (such as the **Gain** parameter of a Gain block) to use either the same value or a different value for each instance of the component. To use different values, create and use a model argument to set the value of the block parameter.

When you generate code from a model hierarchy that uses model arguments, the arguments appear in the code as formal parameters of the referenced model entry-point functions, such as the output (step) function. The generated code then passes the instance-specific parameter values, which you specify in each Model block, to the corresponding function calls.

Whether you use or do not use model arguments, you can use storage classes to configure block parameters to appear in the generated code as tunable global variables. You can also use storage classes to generate tunable model argument values, which the generated code stores in memory and passes to the function calls. You can then change the values during execution.

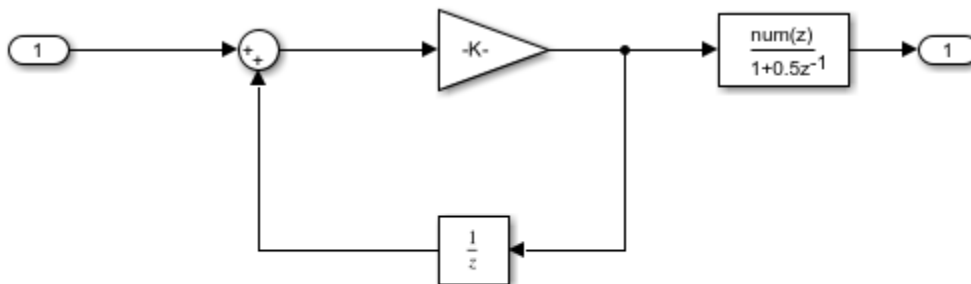
Pass Parameter Data to Referenced Model Entry-Point Functions as Arguments

Configure a referenced model to accept parameter data through formal parameters of the generated model entry-point function. This technique enables you to specify a different parameter value for each instance (Model block) of the referenced model.

Configure Referenced Model to Use Model Arguments

Open the model `ex_arg_code_ref`. This model represents a reusable algorithm.

```
open_system('ex_arg_code_ref')
```



Copyright 2021 The MathWorks, Inc.

On the **Modeling** tab, click **Model Data Editor**.

In the Model Data editor, in the data table, use the **Data Type** column to set the data type of the Inport block to `single`. Due to data type inheritance, the other signals in the model use the same data type.

Select the **Parameters** tab.

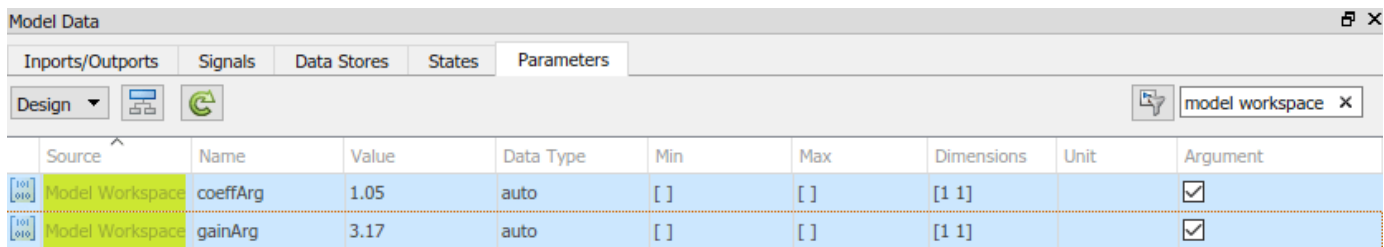
In the model, select the Gain block.

In the Model Data Editor, use the **Value** column to set the value of the **Gain** parameter to `gainArg`. `gainArg` is a Simulink.Parameter object in the model workspace whose value is 3.17.

Use the Model Data Editor to set the **Numerator** parameter to the Simulink.Parameter object `coeffArg` whose value is 1.05.

In the Model Data Editor, click the **Show/refresh additional information** button.

Use the **Filter contents** box to find each parameter object (`gainArg` and `coeffArg`). For each object, select the check box in the **Argument** column.



Source	Name	Value	Data Type	Min	Max	Dimensions	Unit	Argument
Model Workspace	coeffArg	1.05	auto	[]	[]	[1 1]		<input checked="" type="checkbox"/>
Model Workspace	gainArg	3.17	auto	[]	[]	[1 1]		<input checked="" type="checkbox"/>

Save the `ex_arg_code_ref` model.

Alternatively, at the command prompt, you can use these commands to configure the blocks and the parameter objects:

```
set_param('ex_arg_code_ref/In1', 'OutDataTypeStr', 'single')

set_param('ex_arg_code_ref/Gain', 'Gain', 'gainArg')
modelWorkspace = get_param('ex_arg_code_ref', 'ModelWorkspace');
assignin(modelWorkspace, 'gainArg', Simulink.Parameter(3.17));

set_param('ex_arg_code_ref/Discrete Filter', 'Numerator', 'coeffArg')
assignin(modelWorkspace, 'coeffArg', Simulink.Parameter(1.05));

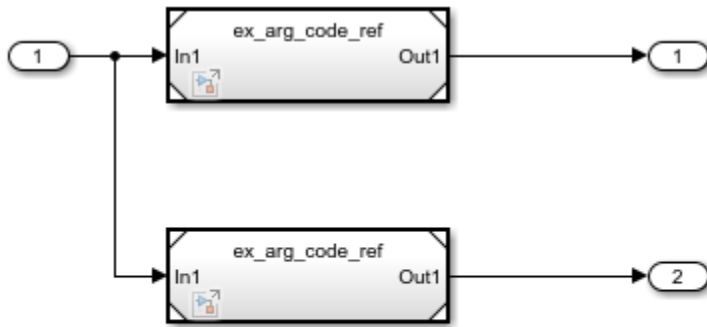
set_param('ex_arg_code_ref', 'ParameterArgumentNames', 'coeffArg, gainArg')

save_system('ex_arg_code_ref')
```

Specify Instance-Specific Parameter Values in Model Blocks

Open the model `ex_arg_code`. This model uses multiple instances (Model blocks) of the reusable algorithm.

```
open_system('ex_arg_code')
```

Copyright 2021 The MathWorks, Inc.

In the model, open the Model Data Editor **Parameters** tab (**Modeling > Model Data Editor**). The Model Data Editor shows four rows that correspond to the model arguments, `coeffArg` and `gainArg`, that you can specify for the two Model blocks.

Use the Model Data Editor to set values for the model arguments in Model. For example, use the values in the figure. For Model1, do not specify a value for the model arguments. By default, a model argument uses the last value specified below it in the model hierarchy (indicated by the value from below).

Source	Name	Value	Data Type	Min	Max	Dimensions	Unit	Argument
Model	<code>coeffArg</code>	0.98						<input type="checkbox"/>
Model	<code>gainArg</code>	2.98						<input type="checkbox"/>
Model1	<code>coeffArg</code>	<from below>						<input type="checkbox"/>
Model1	<code>gainArg</code>	<from below>						<input type="checkbox"/>

Alternatively, at the command prompt, you can use this command to set the values for Model:

```
instSpecParams = get_param('ex_arg_code/Model','InstanceParameters');
instSpecParams(1).Value = '.98';
instSpecParams(2).Value = '2.98';
set_param('ex_arg_code/Model','InstanceParameters',instSpecParams);
```

Generate code from the top model.

```
slbuild('ex_arg_code')

### Starting serial model reference code generation build.
### Successfully updated the model reference code generation target for: ex_arg_code_ref
### Starting build procedure for: ex_arg_code
### Successful completion of build procedure for: ex_arg_code
```

Build Summary

Code generation targets built:

Model	Action	Rebuild Reason
ex_arg_code_ref	Code generated and compiled.	ex_arg_code_ref.c does not exist.

Top model targets built:

Model	Action	Rebuild Reason
ex_arg_code	Code generated and compiled.	Code generation information file does not exist.

2 of 2 models built (0 models already up to date)

Build duration: 0h 0m 23.148s

The file `ex_arg_code_ref.c` defines the referenced model entry-point function, `ex_arg_code_ref`. The function has two formal parameters, `rtp_coeffArg` and `rtp_gainArg`, that correspond to the model arguments, `coeffArg` and `gainArg`. The formal parameters use the data type `real32_T`, which corresponds to the data type `single` in Simulink®.

```
file = fullfile('slprj','grt','ex_arg_code_ref','ex_arg_code_ref.c');
rtwdemodbtype(file,'/* Output and update for referenced model:',...
    'real32_T rtp_gainArg',1,1)

/* Output and update for referenced model: 'ex_arg_code_ref' */
void ex_arg_code_ref(const real32_T *rtu_In1, real32_T *rty_Out1, real32_T
    rtp_coeffArg, real32_T rtp_gainArg, DW_ex_arg_code_ref_f_T *
    localDW)
{
    real32_T rtb_Sum;

    /* Sum: '<Root>/Sum' incorporates:
     * UnitDelay: '<Root>/Unit Delay'
     */
    rtb_Sum = *rtu_In1 + localDW->UnitDelay_DSTATE;

    /* Gain: '<Root>/Gain' */
    rtb_Sum *= rtp_gainArg;

    /* DiscreteFilter: '<Root>/Discrete Filter' */
    localDW->DiscreteFilter_tmp = rtb_Sum - 0.5F * localDW->DiscreteFilter_states;
    *rty_Out1 = rtp_coeffArg * localDW->DiscreteFilter_tmp;

    /* Update for UnitDelay: '<Root>/Unit Delay' */
    localDW->UnitDelay_DSTATE = rtb_Sum;

    /* Update for DiscreteFilter: '<Root>/Discrete Filter' */
    localDW->DiscreteFilter_states = localDW->DiscreteFilter_tmp;
}

/* Model initialize function */
void ex_arg_code_ref_initialize(const char_T **rt_errorStatus,
    RT_MODEL_ex_arg_code_ref_T *const ex_arg_code_ref_M, DW_ex_arg_code_ref_f_T
    *localDW)
{
    /* Registration code */
}
```

```

/* initialize error status */
rtmSetErrorStatusPointer(ex_arg_code_ref_M, rt_errorStatus);

/* states (dwork) */
(void) memset((void *)localDW, 0,
              sizeof(DW_ex_arg_code_ref_f_T));
}

```

The file `ex_arg_code.c` contains the definition of the top model entry-point function, `ex_arg_code`. This function calls the referenced model entry-point function, `ex_arg_code_ref`, and uses the model argument values that you specified (such as 1.11 and 3.34) as the values of `rtp_coeffArg` and `rtp_gainArg`.

```

file = fullfile('ex_arg_code_grt_rtw', 'ex_arg_code.c');
rtwdemodbtype(file, '/* ModelReference: '<Root>/Model' incorporates:', ...
              '1.05F, 3.17F)', 1, 1)

```

```

/* ModelReference: '<Root>/Model' incorporates:
 * Inport: '<Root>/In1'
 * Outport: '<Root>/Out1'
 */
ex_arg_code_ref(&ex_arg_code_U.In1, &ex_arg_code_Y.Out1, 0.98F, 2.98F,
               &(ex_arg_code_DW.Model_InstanceData.rtdw));

/* ModelReference: '<Root>/Model1' incorporates:
 * Inport: '<Root>/In1'
 * Outport: '<Root>/Out2'
 */
ex_arg_code_ref(&ex_arg_code_U.In1, &ex_arg_code_Y.Out2, 1.05F, 3.17F,
               &(ex_arg_code_DW.Model1_InstanceData.rtdw));

/* Matfile logging */
rt_UpdateTXYLogVars(ex_arg_code_M->rtwLogInfo,
                   (&ex_arg_code_M->Timing.taskTime0));

/* signal main to stop simulation */
{
    /* Sample time: [0.2s, 0.0s] */
    if ((rtmGetTFinal(ex_arg_code_M) != -1) &&
        !((rtmGetTFinal(ex_arg_code_M) - ex_arg_code_M->Timing.taskTime0) >
          ex_arg_code_M->Timing.taskTime0 * (DBL_EPSILON))) {
        rtmSetErrorStatus(ex_arg_code_M, "Simulation finished");
    }
}

/* Update absolute time for base rate */
/* The "clockTick0" counts the number of times the code of this task has
 * been executed. The absolute time is the multiplication of "clockTick0"
 * and "Timing.stepSize0". Size of "clockTick0" ensures timer will not
 * overflow during the application lifespan selected.
 * Timer of this task consists of two 32 bit unsigned integers.
 * The two integers represent the low bits Timing.clockTick0 and the high bits
 * Timing.clockTickH0. When the low bit overflows to 0, the high bits increment.
 */
if (!(++ex_arg_code_M->Timing.clockTick0)) {
    ++ex_arg_code_M->Timing.clockTickH0;
}

```

```

    ex_arg_code_M->Timing.taskTime0 = ex_arg_code_M->Timing.clockTick0 *
        ex_arg_code_M->Timing.stepSize0 + ex_arg_code_M->Timing.clockTickH0 *
        ex_arg_code_M->Timing.stepSize0 * 4294967296.0;
}

/* Model initialize function */
void ex_arg_code_initialize(void)
{
    /* Registration code */

    /* initialize non-finites */
    rt_InitInfAndNaN(sizeof(real_T));

    /* initialize real-time model */
    (void) memset((void *)ex_arg_code_M, 0,
        sizeof(RT_MODEL_ex_arg_code_T));
    rtmSetTFinal(ex_arg_code_M, 10.0);
    ex_arg_code_M->Timing.stepSize0 = 0.2;

    /* Setup for data logging */
    {
        static RTWLogInfo rt_DataLoggingInfo;
        rt_DataLoggingInfo.loggingInterval = (NULL);
        ex_arg_code_M->rtwLogInfo = &rt_DataLoggingInfo;
    }

    /* Setup for data logging */
    {
        rtwLogXSignalInfo(ex_arg_code_M->rtwLogInfo, (NULL));
        rtwLogXSignalPtrs(ex_arg_code_M->rtwLogInfo, (NULL));
        rtwLogT(ex_arg_code_M->rtwLogInfo, "tout");
        rtwLogX(ex_arg_code_M->rtwLogInfo, "");
        rtwLogXFinal(ex_arg_code_M->rtwLogInfo, "");
        rtwLogVarNameModifier(ex_arg_code_M->rtwLogInfo, "rt_");
        rtwLogFormat(ex_arg_code_M->rtwLogInfo, 0);
        rtwLogMaxRows(ex_arg_code_M->rtwLogInfo, 0);
        rtwLogDecimation(ex_arg_code_M->rtwLogInfo, 1);

        /*
         * Set pointers to the data and signal info for each output
         */
        {
            static void * rt_LoggedOutputSignalPtrs[] = {
                &ex_arg_code_Y.Out1,
                &ex_arg_code_Y.Out2
            };

            rtwLogYSignalPtrs(ex_arg_code_M->rtwLogInfo, ((LogSignalPtrsType)
                rt_LoggedOutputSignalPtrs));
        }

        {
            static int_T rt_LoggedOutputWidths[] = {
                1,
                1
            };
        }
    }
}

```

```
static int_T rt_LoggedOutputNumDimensions[] = {
    1,
    1
};

static int_T rt_LoggedOutputDimensions[] = {
    1,
    1
};

static boolean_T rt_LoggedOutputIsVarDims[] = {
    0,
    0
};

static void* rt_LoggedCurrentSignalDimensions[] = {
    (NULL),
    (NULL)
};

static int_T rt_LoggedCurrentSignalDimensionsSize[] = {
    4,
    4
};

static BuiltInDTypeId rt_LoggedOutputDataTypeIds[] = {
    SS_SINGLE,
    SS_SINGLE
};

static int_T rt_LoggedOutputComplexSignals[] = {
    0,
    0
};

static RTWPreprocessingFcnPtr rt_LoggingPreprocessingFcnPtrs[] = {
    (NULL),
    (NULL)
};

static const char_T *rt_LoggedOutputLabels[] = {
    "",
    "" };

static const char_T *rt_LoggedOutputBlockNames[] = {
    "ex_arg_code/Out1",
    "ex_arg_code/Out2" };

static RTWLogDataTypeConvert rt_RTWLogDataTypeConvert[] = {
    { 0, SS_SINGLE, SS_SINGLE, 0, 0, 0, 1.0, 0, 0.0 },
    { 0, SS_SINGLE, SS_SINGLE, 0, 0, 0, 1.0, 0, 0.0 }
};

static RTWLogSignalInfo rt_LoggedOutputSignalInfo[] = {
    {
        2,
        rt_LoggedOutputWidths,
    }
};
```

```

        rt_LoggedOutputNumDimensions,
        rt_LoggedOutputDimensions,
        rt_LoggedOutputIsVarDims,
        rt_LoggedCurrentSignalDimensions,
        rt_LoggedCurrentSignalDimensionsSize,
        rt_LoggedOutputDataTypeIds,
        rt_LoggedOutputComplexSignals,
        (NULL),
        rt_LoggingPreprocessingFcnPtrs,

        { rt_LoggedOutputLabels },
        (NULL),
        (NULL),
        (NULL),

        { rt_LoggedOutputBlockNames },

        { (NULL) },
        (NULL),
        rt_RTWLogDataTypeConvert
    }
};

    rtwliSetLogYSignalInfo(ex_arg_code_M->rtwLogInfo, rt_LoggedOutputSignalInfo);

    /* set currSigDims field */
    rt_LoggedCurrentSignalDimensions[0] = &rt_LoggedOutputWidths[0];
    rt_LoggedCurrentSignalDimensions[1] = &rt_LoggedOutputWidths[1];
}

    rtwliSetLogY(ex_arg_code_M->rtwLogInfo, "yout");
}

/* states (dwork) */
(void) memset((void *)&ex_arg_code_DW, 0,
             sizeof(DW_ex_arg_code_T));

/* external inputs */
ex_arg_code_U.In1 = 0.0F;

/* external outputs */
(void)memset(&ex_arg_code_Y, 0, sizeof(ExtY_ex_arg_code_T));

/* Model Initialize function for ModelReference Block: '<Root>/Model' */
ex_arg_code_ref_initialize(rtmGetErrorStatusPointer(ex_arg_code_M),
    &(ex_arg_code_DW.Model_InstanceData.rtm),
    &(ex_arg_code_DW.Model_InstanceData.rtdw));

/* Model Initialize function for ModelReference Block: '<Root>/Model1' */
ex_arg_code_ref_initialize(rtmGetErrorStatusPointer(ex_arg_code_M),
    &(ex_arg_code_DW.Model1_InstanceData.rtm),
    &(ex_arg_code_DW.Model1_InstanceData.rtdw));

/* Matfile logging */
rt_StartDataLoggingWithStartTime(ex_arg_code_M->rtwLogInfo, 0.0, rtmGetTFinal
    (ex_arg_code_M), ex_arg_code_M->Timing.stepSize0, (&rtmGetErrorStatus
    (ex_arg_code_M)));

```

```

/* SystemInitialize for ModelReference: '<Root>/Model' */
ex_arg_code_ref_Init(&(ex_arg_code_DW.Model_InstanceData.rtdw));

/* SystemInitialize for ModelReference: '<Root>/Model1' */
ex_arg_code_ref_Init(&(ex_arg_code_DW.Model1_InstanceData.rtdw));
}

/* Model terminate function */
void ex_arg_code_terminate(void)
{
/* (no terminate code required) */
}

```

The formal parameters use the data type `real32_T` (single) because:





- 1 The block parameters in `ex_arg_code_ref` determine their data types through internal rules. For example, in the Gain block dialog box, on the **Parameter Attributes** tab, **Parameter data type** is set to **Inherit: Inherit via internal rule** (the default). In this case, the internal rule chooses the same data type as the input and output signals, `single`.
- 2 The model arguments in the model workspace use context-sensitive data typing because the value of the `DataType` property is set to `auto` (the default). With this setting, the model arguments use the same data type as the block parameters, `single`.
- 3 The formal parameters in the generated code use the same data type as the model arguments, `single`.

Generate Tunable Argument Values

You can configure the instance-specific values in the Model blocks to appear in the generated code as tunable global variables. This technique enables you to store the parameter values for each instance in memory and to tune the values during code execution.

In the top model `ex_arg_code`, select the Model Data Editor **Parameters** tab.

Use the Model Data Editor to set the values of the model arguments according to this figure.

	Source ^	Name	Value
	Model	coeffArg	coeffForInst1
	Model	gainArg	gainForInst1
	Model1	coeffArg	coeffForInst2
	Model1	gainArg	gainForInst2

View the contents of the `ex_arg_code_ref` model workspace in Model Explorer.

Copy `gainArg` and `coeffArg` from the `ex_arg_code_ref` model workspace to the base workspace.

Rename `gainArg` as `gainForInst1`. Rename `coeffArg` as `coeffForInst1`.

```

gainForInst1 = getVariable(modelWorkspace, 'gainArg');
gainForInst1 = copy(gainForInst1);
coeffForInst1 = getVariable(modelWorkspace, 'coeffArg');
coeffForInst1 = copy(coeffForInst1);

```

Copy `gainForInst1` and `coeffForInst1` as `gainForInst2` and `coeffForInst2`.

```
gainForInst2 = copy(gainForInst1);
coeffForInst2 = copy(coeffForInst1);
```

Set the instance-specific parameter values by using the `Value` property of the parameter objects in the base workspace.

```
gainForInst1.Value = 2.98;
coeffForInst1.Value = 0.98;
```

```
gainForInst2.Value = 3.34;
coeffForInst2.Value = 1.11;
```

For each new parameter object, on the **Code Generation** tab, set `StorageClass` to `ExportedGlobal`. This setting causes the parameter objects to appear in the generated code as tunable global variables.

```
gainForInst1.StorageClass = 'ExportedGlobal';
coeffForInst1.StorageClass = 'ExportedGlobal';
gainForInst2.StorageClass = 'ExportedGlobal';
coeffForInst2.StorageClass = 'ExportedGlobal';
```

```
instSpecParams = get_param('ex_arg_code/Model', 'InstanceParameters');
instSpecParams(1).Value = 'coeffForInst1';
instSpecParams(2).Value = 'gainForInst1';
instSpecParams1 = get_param('ex_arg_code/Model1', 'InstanceParameters');
instSpecParams1(1).Value = 'coeffForInst2';
instSpecParams1(2).Value = 'gainForInst2';
set_param('ex_arg_code/Model', 'InstanceParameters', instSpecParams);
set_param('ex_arg_code/Model1', 'InstanceParameters', instSpecParams1);
```

Generate code from the top model.

```
slbuild('ex_arg_code')
```

```
### Starting serial model reference code generation build.
### Model reference code generation target for ex_arg_code_ref is up to date.
### Starting build procedure for: ex_arg_code
### Successful completion of build procedure for: ex_arg_code
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
ex_arg_code	Code generated and compiled.	Generated code was out of date.

```
1 of 2 models built (1 models already up to date)
Build duration: 0h 0m 15.865s
```

The file `ex_arg_code.c` defines the global variables that correspond to the parameter objects in the base workspace.

```
file = fullfile('ex_arg_code_grt_rtw', 'ex_arg_code.c');
rtwdemodbtype(file, '/* Exported block parameters */', ...
    '/* Block states (default storage) */', 1, 0)
```

```
/* Exported block parameters */
```



```

real32_T coeffForInst1 = 0.98F;      /* Variable: coeffForInst1
                                     * Referenced by: '<Root>/Model'
                                     */
real32_T coeffForInst2 = 1.11F;     /* Variable: coeffForInst2
                                     * Referenced by: '<Root>/Model1'
                                     */
real32_T gainForInst1 = 2.98F;      /* Variable: gainForInst1
                                     * Referenced by: '<Root>/Model'
                                     */
real32_T gainForInst2 = 3.34F;     /* Variable: gainForInst2
                                     * Referenced by: '<Root>/Model1'
                                     */

```

In each call to `ex_arg_code_ref`, the top model algorithm uses the global variables to set the values of the formal parameters.

```

rtwdemodbtype(file, /* ModelReference: '<Root>/Model' incorporates:',...
                  'gainForInst2);',1,1)

/* ModelReference: '<Root>/Model' incorporates:
 * Inport: '<Root>/In1'
 * Outport: '<Root>/Out1'
 */
ex_arg_code_ref(&ex_arg_code_U.In1, &ex_arg_code_Y.Out1, coeffForInst1,
                gainForInst1, &(ex_arg_code_DW.Model_InstanceData.rtdw));

/* ModelReference: '<Root>/Model1' incorporates:
 * Inport: '<Root>/In1'
 * Outport: '<Root>/Out2'
 */
ex_arg_code_ref(&ex_arg_code_U.In1, &ex_arg_code_Y.Out2, coeffForInst2,
                gainForInst2, &(ex_arg_code_DW.Model1_InstanceData.rtdw));

/* Matfile logging */
rt_UpdateTXYLogVars(ex_arg_code_M->rtwLogInfo,
                    (&ex_arg_code_M->Timing.taskTime0));

/* signal main to stop simulation */
{
    /* Sample time: [0.2s, 0.0s] */
    if ((rtmGetTFinal(ex_arg_code_M)!=-1) &&
        !((rtmGetTFinal(ex_arg_code_M)-ex_arg_code_M->Timing.taskTime0) >
          ex_arg_code_M->Timing.taskTime0 * (DBL_EPSILON))) {
        rtmSetErrorStatus(ex_arg_code_M, "Simulation finished");
    }
}

/* Update absolute time for base rate */
/* The "clockTick0" counts the number of times the code of this task has
 * been executed. The absolute time is the multiplication of "clockTick0"
 * and "Timing.stepSize0". Size of "clockTick0" ensures timer will not
 * overflow during the application lifespan selected.
 * Timer of this task consists of two 32 bit unsigned integers.
 * The two integers represent the low bits Timing.clockTick0 and the high bits
 * Timing.clockTickH0. When the low bit overflows to 0, the high bits increment.
 */
if (!(++ex_arg_code_M->Timing.clockTick0)) {
    ++ex_arg_code_M->Timing.clockTickH0;
}

```

```

    }

    ex_arg_code_M->Timing.taskTime0 = ex_arg_code_M->Timing.clockTick0 *
        ex_arg_code_M->Timing.stepSize0 + ex_arg_code_M->Timing.clockTickH0 *
        ex_arg_code_M->Timing.stepSize0 * 4294967296.0;
}

/* Model initialize function */
void ex_arg_code_initialize(void)
{
    /* Registration code */

    /* initialize non-finites */
    rt_InitInfAndNaN(sizeof(real_T));

    /* initialize real-time model */
    (void) memset((void *)ex_arg_code_M, 0,
        sizeof(RT_MODEL_ex_arg_code_T));
    rtmSetTFinal(ex_arg_code_M, 10.0);
    ex_arg_code_M->Timing.stepSize0 = 0.2;

    /* Setup for data logging */
    {
        static RTWLogInfo rt_DataLoggingInfo;
        rt_DataLoggingInfo.loggingInterval = (NULL);
        ex_arg_code_M->rtwLogInfo = &rt_DataLoggingInfo;
    }

    /* Setup for data logging */
    {
        rtwliSetLogXSignalInfo(ex_arg_code_M->rtwLogInfo, (NULL));
        rtwliSetLogXSignalPtrs(ex_arg_code_M->rtwLogInfo, (NULL));
        rtwliSetLogT(ex_arg_code_M->rtwLogInfo, "tout");
        rtwliSetLogX(ex_arg_code_M->rtwLogInfo, "");
        rtwliSetLogXFinal(ex_arg_code_M->rtwLogInfo, "");
        rtwliSetLogVarNameModifier(ex_arg_code_M->rtwLogInfo, "rt_");
        rtwliSetLogFormat(ex_arg_code_M->rtwLogInfo, 0);
        rtwliSetLogMaxRows(ex_arg_code_M->rtwLogInfo, 0);
        rtwliSetLogDecimation(ex_arg_code_M->rtwLogInfo, 1);

        /*
         * Set pointers to the data and signal info for each output
         */
        {
            static void * rt_LoggedOutputSignalPtrs[] = {
                &ex_arg_code_Y.Out1,
                &ex_arg_code_Y.Out2
            };

            rtwliSetLogYSignalPtrs(ex_arg_code_M->rtwLogInfo, ((LogSignalPtrsType)
                rt_LoggedOutputSignalPtrs));
        }

        {
            static int_T rt_LoggedOutputWidths[] = {
                1,
                1
            };
        }
    }
}

```

```
static int_T rt_LoggedOutputNumDimensions[] = {
    1,
    1
};

static int_T rt_LoggedOutputDimensions[] = {
    1,
    1
};

static boolean_T rt_LoggedOutputIsVarDims[] = {
    0,
    0
};

static void* rt_LoggedCurrentSignalDimensions[] = {
    (NULL),
    (NULL)
};

static int_T rt_LoggedCurrentSignalDimensionsSize[] = {
    4,
    4
};

static BuiltInDTypeId rt_LoggedOutputDataTypeIds[] = {
    SS_SINGLE,
    SS_SINGLE
};

static int_T rt_LoggedOutputComplexSignals[] = {
    0,
    0
};

static RTWPreprocessingFcnPtr rt_LoggingPreprocessingFcnPtrs[] = {
    (NULL),
    (NULL)
};

static const char_T *rt_LoggedOutputLabels[] = {
    "",
    "" };

static const char_T *rt_LoggedOutputBlockNames[] = {
    "ex_arg_code/Out1",
    "ex_arg_code/Out2" };

static RTWLogDataTypeConvert rt_RTWLogDataTypeConvert[] = {
    { 0, SS_SINGLE, SS_SINGLE, 0, 0, 0, 1.0, 0, 0.0 },
    { 0, SS_SINGLE, SS_SINGLE, 0, 0, 0, 1.0, 0, 0.0 }
};

static RTWLogSignalInfo rt_LoggedOutputSignalInfo[] = {
    {
        2,
```

```

    rt_LoggedOutputWidths,
    rt_LoggedOutputNumDimensions,
    rt_LoggedOutputDimensions,
    rt_LoggedOutputIsVarDims,
    rt_LoggedCurrentSignalDimensions,
    rt_LoggedCurrentSignalDimensionsSize,
    rt_LoggedOutputDataTypeIds,
    rt_LoggedOutputComplexSignals,
    (NULL),
    rt_LoggingPreprocessingFcnPtrs,

    { rt_LoggedOutputLabels },
    (NULL),
    (NULL),
    (NULL),

    { rt_LoggedOutputBlockNames },

    { (NULL) },
    (NULL),
    rt_RTWLogDataTypeConvert
}
};

    rttiSetLogYSignalInfo(ex_arg_code_M->rtwLogInfo, rt_LoggedOutputSignalInfo);

    /* set currSigDims field */
    rt_LoggedCurrentSignalDimensions[0] = &rt_LoggedOutputWidths[0];
    rt_LoggedCurrentSignalDimensions[1] = &rt_LoggedOutputWidths[1];
}

    rttiSetLogY(ex_arg_code_M->rtwLogInfo, "yout");
}

/* states (dwork) */
(void) memset((void *)&ex_arg_code_DW, 0,
              sizeof(DW_ex_arg_code_T));

/* external inputs */
ex_arg_code_U.In1 = 0.0F;

/* external outputs */
(void) memset(&ex_arg_code_Y, 0, sizeof(ExtY_ex_arg_code_T));

/* Model Initialize function for ModelReference Block: '<Root>/Model' */
ex_arg_code_ref_initialize(rtmGetErrorStatusPointer(ex_arg_code_M),
                          &(ex_arg_code_DW.Model_InstanceData.rtm),
                          &(ex_arg_code_DW.Model_InstanceData.rtdw));

/* Model Initialize function for ModelReference Block: '<Root>/Model1' */
ex_arg_code_ref_initialize(rtmGetErrorStatusPointer(ex_arg_code_M),
                          &(ex_arg_code_DW.Model1_InstanceData.rtm),
                          &(ex_arg_code_DW.Model1_InstanceData.rtdw));

/* Matfile logging */
rt_StartDataLoggingWithStartTime(ex_arg_code_M->rtwLogInfo, 0.0, rtmGetTFinal
    (ex_arg_code_M), ex_arg_code_M->Timing.stepSize0, (&rtmGetErrorStatus
    (ex_arg_code_M)));

```

```

/* SystemInitialize for ModelReference: '<Root>/Model' */
ex_arg_code_ref_Init(&(ex_arg_code_DW.Model_InstanceData.rtdw));

/* SystemInitialize for ModelReference: '<Root>/Model1' */
ex_arg_code_ref_Init(&(ex_arg_code_DW.Model1_InstanceData.rtdw));
}

/* Model terminate function */
void ex_arg_code_terminate(void)
{
/* (no terminate code required) */
}

```

The global variables in the generated code use the data type `real32_T` (single) because:

- 1 The parameter objects in the base workspace use context-sensitive data typing because the `DataType` property is set to `auto` (the default). With this setting, the parameter objects in the base workspace use the same data type as the model arguments, `single`.
- 2 The global variables in the generated code use the same data type as the parameter objects in the base workspace.

Group Multiple Model Arguments into Single Structure

Use the Model Explorer to copy `gainArg` and `coeffArg` from the `ex_arg_code_ref` model workspace into the base workspace.

```

temp = getVariable(modelWorkspace, 'gainArg');
gainArg = copy(temp);
temp = getVariable(modelWorkspace, 'coeffArg');
coeffArg = copy(temp);

```

At the command prompt, combine these two parameter objects into a structure, `structArg`.

```

structArg = Simulink.Parameter(struct('gain', gainArg.Value, ...
    'coeff', coeffArg.Value));

```

Use the Model Explorer to move `structArg` into the model workspace.

```

assignin(modelWorkspace, 'structArg', copy(structArg));
clear structArg gainArg coeffArg

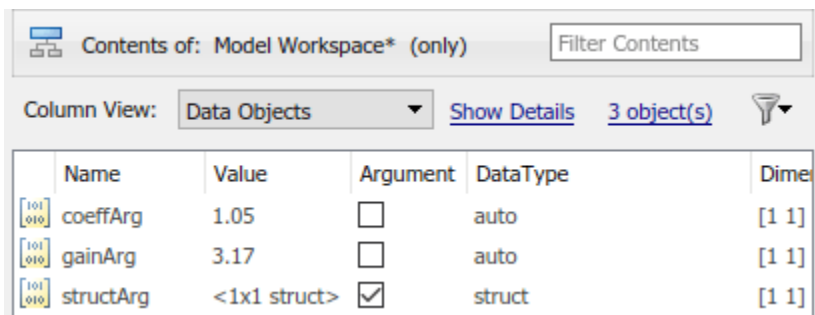
```

In the **Contents** pane, configure `structArg` as the only model argument.

```

set_param('ex_arg_code_ref', 'ParameterArgumentNames', 'structArg')

```



The screenshot shows the 'Contents of: Model Workspace* (only)' pane. The 'Column View' is set to 'Data Objects'. A table lists three objects: `coeffArg`, `gainArg`, and `structArg`. The `structArg` object is selected, and its 'Argument' checkbox is checked, indicating it is the only model argument.

Name	Value	Argument	DataType	Dimension
coeffArg	1.05	<input type="checkbox"/>	auto	[1 1]
gainArg	3.17	<input type="checkbox"/>	auto	[1 1]
structArg	<1x1 struct>	<input checked="" type="checkbox"/>	struct	[1 1]

In the `ex_arg_code_ref` model, select the Model Data Editor **Parameters** tab.

Use the Model Data Editor to set the value of the **Gain** parameter to `structArg.gain` and the value of the **Numerator** parameter to `structArg.coeff`. Save the model.

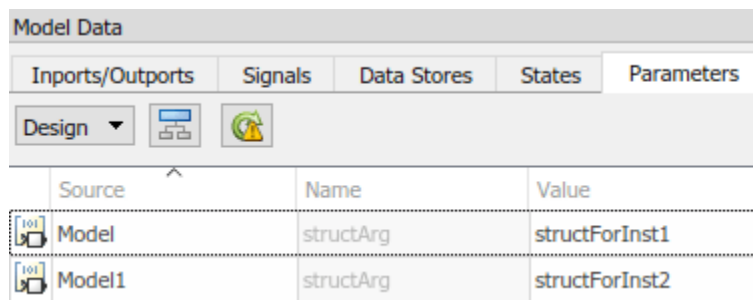
```
set_param('ex_arg_code_ref/Gain', 'Gain', 'structArg.gain')
set_param('ex_arg_code_ref/Discrete Filter', ...
    'Numerator', 'structArg.coeff')
save_system('ex_arg_code_ref')
```

At the command prompt, combine the four parameter objects in the base workspace into two structures. Each structure stores the parameter values for one instance of `ex_arg_code_ref`.

```
structForInst1 = Simulink.Parameter(struct('gain', gainForInst1.Value, ...
    'coeff', coeffForInst1.Value));

structForInst2 = Simulink.Parameter(struct('gain', gainForInst2.Value, ...
    'coeff', coeffForInst2.Value));
```

In the top model `ex_arg_code`, use the Model Data Editor to set the argument values according to this figure.



The screenshot shows the Model Data Editor interface with the Parameters tab selected. Below the tabs, there are icons for Design, a refresh button, and a warning icon. The main table has columns for Source, Name, and Value. Two rows are visible, both selected with a dashed border:

Source	Name	Value
Model	structArg	structForInst1
Model1	structArg	structForInst2

```
instSpecParams = get_param('ex_arg_code/Model', 'InstanceParameters');
instSpecParams(1).Value = 'structForInst1';
instSpecParams1 = get_param('ex_arg_code/Model1', 'InstanceParameters');
instSpecParams1(1).Value = 'structForInst2';
set_param('ex_arg_code/Model', 'InstanceParameters', instSpecParams);
set_param('ex_arg_code/Model1', 'InstanceParameters', instSpecParams1);
```

Click the **Show/refresh additional information** button.

For the new parameter objects `structForInst1` and `structForInst2`, use the Model Explorer to apply the storage class `ExportedGlobal`.

```
structForInst1.StorageClass = 'ExportedGlobal';
structForInst2.StorageClass = 'ExportedGlobal';
```

At the command prompt, use the function `Simulink.Bus.createObject` to create a `Simulink.Bus` object. The hierarchy of elements in the object matches the hierarchy of the structure fields. The default name of the object is `s1Bus1`.

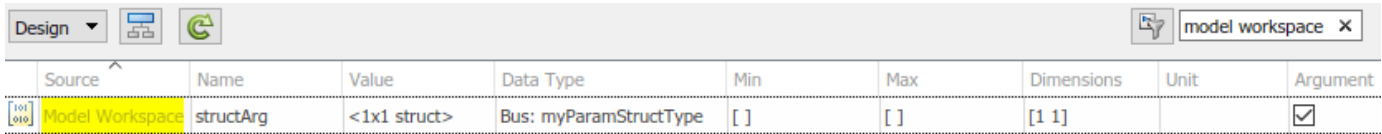
```
Simulink.Bus.createObject(structForInst1.Value);
```

Rename the bus object as `myParamStructType` by copying it.

```
myParamStructType = copy(s1Bus1);
```

In the Model Data Editor for `ex_arg_code`, use the Data Type column to set the data type of `structForInst1` and `structForInst2` to `Bus: myParamStructType`.

```
structForInst1.DataType = 'Bus: myParamStructType';
structForInst2.DataType = 'Bus: myParamStructType';
```



Source	Name	Value	Data Type	Min	Max	Dimensions	Unit	Argument
Model Workspace	structArg	<1x1 struct>	Bus: myParamStructType	[]	[]	[1 1]		<input checked="" type="checkbox"/>

```
temp = getVariable(modelWorkspace, 'structArg');
temp = copy(temp);
temp.DataType = 'Bus: myParamStructType';
assignin(modelWorkspace, 'structArg', copy(temp));
```

Save the `ex_arg_code_ref` model.

```
save_system('ex_arg_code_ref')
```

When you use structures to group parameter values, you cannot take advantage of context-sensitive data typing to control the data types of the fields of the structures (for example, the fields of `structForInst1`). However, you can use the properties of the bus object to control the field data types.

At the command prompt, set the data type of the elements in the bus object to `single`. The corresponding fields in the structures (such as `structForInst1` and `structArg`) use the same data type.

```
myParamStructType.Elements(1).DataType = 'single';
myParamStructType.Elements(2).DataType = 'single';
```

Generate code from the top model, `ex_arg_code`.

```
slbuild('ex_arg_code')
```

```
### Starting serial model reference code generation build.
### Successfully updated the model reference code generation target for: ex_arg_code_ref
### Starting build procedure for: ex_arg_code
### Successful completion of build procedure for: ex_arg_code
```

Build Summary

Code generation targets built:

Model	Action	Rebuild Reason
ex_arg_code_ref	Code generated and compiled.	Model or library ex_arg_code_ref changed.

Top model targets built:

Model	Action	Rebuild Reason
ex_arg_code	Code generated and compiled.	Referenced models were updated.

```
2 of 2 models built (0 models already up to date)
Build duration: 0h 0m 20.112s
```

The file `ex_arg_code_types.h` defines the structure type `myParamStructType`, which corresponds to the `Simulink.Bus` object.

```
file = fullfile('ex_arg_code_grt_rtw','ex_arg_code_types.h');
rtwdemodbtype(file,'typedef struct {' myParamStructType;',1,1)
```

```
typedef struct {
    real32_T gain;
    real32_T coeff;
} myParamStructType;
```

In the file `ex_arg_code_ref.c`, the referenced model entry-point function has a formal parameter, `rtp_structArg`, that correspond to the model argument `structArg`.

```
file = fullfile('slprj','grt','ex_arg_code_ref','ex_arg_code_ref.c');
rtwdemodbtype(file,'/* Output and update for referenced model:',...
    '*rtp_structArg)',1,1)

/* Output and update for referenced model: 'ex_arg_code_ref' */
void ex_arg_code_ref(const real32_T *rtu_In1, real32_T *rty_Out1, const
    myParamStructType *rtp_structArg, DW_ex_arg_code_ref_f_T
    *localDW)
{
    real32_T rtb_Sum;

    /* Sum: '<Root>/Sum' incorporates:
     * UnitDelay: '<Root>/Unit Delay'
     */
    rtb_Sum = *rtu_In1 + localDW->UnitDelay_DSTATE;

    /* Gain: '<Root>/Gain' */
    rtb_Sum *= rtp_structArg->gain;

    /* DiscreteFilter: '<Root>/Discrete Filter' */
    localDW->DiscreteFilter_tmp = rtb_Sum - 0.5F * localDW->DiscreteFilter_states;
    *rty_Out1 = rtp_structArg->coeff * localDW->DiscreteFilter_tmp;

    /* Update for UnitDelay: '<Root>/Unit Delay' */
    localDW->UnitDelay_DSTATE = rtb_Sum;

    /* Update for DiscreteFilter: '<Root>/Discrete Filter' */
    localDW->DiscreteFilter_states = localDW->DiscreteFilter_tmp;
}

/* Model initialize function */
void ex_arg_code_ref_initialize(const char_T **rt_errorStatus,
    RT_MODEL_ex_arg_code_ref_T *const ex_arg_code_ref_M, DW_ex_arg_code_ref_f_T
    *localDW)
{
    /* Registration code */

    /* initialize error status */
    rtmSetErrorStatusPointer(ex_arg_code_ref_M, rt_errorStatus);

    /* states (dwork) */
    (void) memset((void *)localDW, 0,
```



```

        sizeof(DW_ex_arg_code_ref_f_T));
}

```

The file `ex_arg_code.c` defines the global structure variables that correspond to the parameter objects in the base workspace.

```

file = fullfile('ex_arg_code_grt_rtw','ex_arg_code.c');
rtwdemodbtype(file,'/* Exported block parameters */',...
    '/* Block states (default storage) */',1,0)

/* Exported block parameters */
myParamStructType structForInst1 = {
    2.98F,
    0.98F
} ;
/* Variable: structForInst1
 * Referenced by: '<Root>/Model1'
 */

myParamStructType structForInst2 = {
    3.34F,
    1.11F
} ;
/* Variable: structForInst2
 * Referenced by: '<Root>/Model1'
 */

```

The top model algorithm in the file `ex_arg_code.c` passes the addresses of the structure variables to the referenced model entry-point function.

```

file = fullfile('ex_arg_code_grt_rtw','ex_arg_code.c');
rtwdemodbtype(file,'/* ModelReference: '<Root>/Model1' incorporates:',...
    '&structForInst2);',1,1)

/* ModelReference: '<Root>/Model1' incorporates:
 * Inport: '<Root>/In1'
 * Outport: '<Root>/Out1'
 */
ex_arg_code_ref(&ex_arg_code_U.In1, &ex_arg_code_Y.Out1, &structForInst1,
    &(ex_arg_code_DW.Model_InstanceData.rtdw));

/* ModelReference: '<Root>/Model1' incorporates:
 * Inport: '<Root>/In1'
 * Outport: '<Root>/Out2'
 */
ex_arg_code_ref(&ex_arg_code_U.In1, &ex_arg_code_Y.Out2, &structForInst2,
    &(ex_arg_code_DW.Model1_InstanceData.rtdw));

/* Matfile logging */
rt_UpdateTXYLogVars(ex_arg_code_M->rtwLogInfo,
    (&ex_arg_code_M->Timing.taskTime0));

/* signal main to stop simulation */
{
    /* Sample time: [0.2s, 0.0s] */
    if ((rtmGetTFinal(ex_arg_code_M)!=-1) &&
        !((rtmGetTFinal(ex_arg_code_M)-ex_arg_code_M->Timing.taskTime0) >
            ex_arg_code_M->Timing.taskTime0 * (DBL_EPSILON))) {
        rtmSetErrorStatus(ex_arg_code_M, "Simulation finished");
    }
}

```

```

    }
}

/* Update absolute time for base rate */
/* The "clockTick0" counts the number of times the code of this task has
 * been executed. The absolute time is the multiplication of "clockTick0"
 * and "Timing.stepSize0". Size of "clockTick0" ensures timer will not
 * overflow during the application lifespan selected.
 * Timer of this task consists of two 32 bit unsigned integers.
 * The two integers represent the low bits Timing.clockTick0 and the high bits
 * Timing.clockTickH0. When the low bit overflows to 0, the high bits increment.
 */
if (!(++ex_arg_code_M->Timing.clockTick0)) {
    ++ex_arg_code_M->Timing.clockTickH0;
}

ex_arg_code_M->Timing.taskTime0 = ex_arg_code_M->Timing.clockTick0 *
    ex_arg_code_M->Timing.stepSize0 + ex_arg_code_M->Timing.clockTickH0 *
    ex_arg_code_M->Timing.stepSize0 * 4294967296.0;
}

/* Model initialize function */
void ex_arg_code_initialize(void)
{
    /* Registration code */

    /* initialize non-finites */
    rt_InitInfAndNaN(sizeof(real_T));

    /* initialize real-time model */
    (void) memset((void *)ex_arg_code_M, 0,
        sizeof(RT_MODEL_ex_arg_code_T));
    rtmSetTFinal(ex_arg_code_M, 10.0);
    ex_arg_code_M->Timing.stepSize0 = 0.2;

    /* Setup for data logging */
    {
        static RTWLogInfo rt_DataLoggingInfo;
        rt_DataLoggingInfo.loggingInterval = (NULL);
        ex_arg_code_M->rtwLogInfo = &rt_DataLoggingInfo;
    }

    /* Setup for data logging */
    {
        rtliSetLogXSignalInfo(ex_arg_code_M->rtwLogInfo, (NULL));
        rtliSetLogXSignalPtrs(ex_arg_code_M->rtwLogInfo, (NULL));
        rtliSetLogT(ex_arg_code_M->rtwLogInfo, "tout");
        rtliSetLogX(ex_arg_code_M->rtwLogInfo, "");
        rtliSetLogXFinal(ex_arg_code_M->rtwLogInfo, "");
        rtliSetLogVarNameModifier(ex_arg_code_M->rtwLogInfo, "rt_");
        rtliSetLogFormat(ex_arg_code_M->rtwLogInfo, 0);
        rtliSetLogMaxRows(ex_arg_code_M->rtwLogInfo, 0);
        rtliSetLogDecimation(ex_arg_code_M->rtwLogInfo, 1);

        /*
         * Set pointers to the data and signal info for each output
         */
    }
}

```

```

static void * rt_LoggedOutputSignalPtrs[] = {
    &ex_arg_code_Y.Out1,
    &ex_arg_code_Y.Out2
};

rtliSetLogYSignalPtrs(ex_arg_code_M->rtwLogInfo, ((LogSignalPtrsType)
    rt_LoggedOutputSignalPtrs));
}

{
static int_T rt_LoggedOutputWidths[] = {
    1,
    1
};

static int_T rt_LoggedOutputNumDimensions[] = {
    1,
    1
};

static int_T rt_LoggedOutputDimensions[] = {
    1,
    1
};

static boolean_T rt_LoggedOutputIsVarDims[] = {
    0,
    0
};

static void* rt_LoggedCurrentSignalDimensions[] = {
    (NULL),
    (NULL)
};

static int_T rt_LoggedCurrentSignalDimensionsSize[] = {
    4,
    4
};

static BuiltInDTypeId rt_LoggedOutputDataTypeIds[] = {
    SS_SINGLE,
    SS_SINGLE
};

static int_T rt_LoggedOutputComplexSignals[] = {
    0,
    0
};

static RTWPreprocessingFcnPtr rt_LoggingPreprocessingFcnPtrs[] = {
    (NULL),
    (NULL)
};

static const char_T *rt_LoggedOutputLabels[] = {
    "",
    "" };

```

```

static const char_T *rt_LoggedOutputBlockNames[] = {
    "ex_arg_code/Out1",
    "ex_arg_code/Out2" };

static RTWLogDataTypeConvert rt_RTWLogDataTypeConvert[] = {
    { 0, SS_SINGLE, SS_SINGLE, 0, 0, 0, 1.0, 0, 0.0 },

    { 0, SS_SINGLE, SS_SINGLE, 0, 0, 0, 1.0, 0, 0.0 }
};

static RTWLogSignalInfo rt_LoggedOutputSignalInfo[] = {
    {
        2,
        rt_LoggedOutputWidths,
        rt_LoggedOutputNumDimensions,
        rt_LoggedOutputDimensions,
        rt_LoggedOutputIsVarDims,
        rt_LoggedCurrentSignalDimensions,
        rt_LoggedCurrentSignalDimensionsSize,
        rt_LoggedOutputDataTypeIds,
        rt_LoggedOutputComplexSignals,
        (NULL),
        rt_LoggingPreprocessingFcnPtrs,

        { rt_LoggedOutputLabels },
        (NULL),
        (NULL),
        (NULL),

        { rt_LoggedOutputBlockNames },

        { (NULL) },
        (NULL),
        rt_RTWLogDataTypeConvert
    }
};

rtliSetLogYSignalInfo(ex_arg_code_M->rtwLogInfo, rt_LoggedOutputSignalInfo);

/* set currSigDims field */
rt_LoggedCurrentSignalDimensions[0] = &rt_LoggedOutputWidths[0];
rt_LoggedCurrentSignalDimensions[1] = &rt_LoggedOutputWidths[1];
}

rtliSetLogY(ex_arg_code_M->rtwLogInfo, "yout");
}

/* states (dwork) */
(void) memset((void *)&ex_arg_code_DW, 0,
    sizeof(DW_ex_arg_code_T));

/* external inputs */
ex_arg_code_U.In1 = 0.0F;

/* external outputs */
(void)memset(&ex_arg_code_Y, 0, sizeof(ExtY_ex_arg_code_T));

```

```

/* Model Initialize function for ModelReference Block: '<Root>/Model' */
ex_arg_code_ref_initialize(rtmGetErrorStatusPointer(ex_arg_code_M),
    &(ex_arg_code_DW.Model_InstanceData.rtm),
    &(ex_arg_code_DW.Model_InstanceData.rtdw));

/* Model Initialize function for ModelReference Block: '<Root>/Model1' */
ex_arg_code_ref_initialize(rtmGetErrorStatusPointer(ex_arg_code_M),
    &(ex_arg_code_DW.Model1_InstanceData.rtm),
    &(ex_arg_code_DW.Model1_InstanceData.rtdw));

/* Matfile logging */
rt_StartDataLoggingWithStartTime(ex_arg_code_M->rtwLogInfo, 0.0, rtmGetTFinal
    (ex_arg_code_M), ex_arg_code_M->Timing.stepSize0, (&rtmGetErrorStatus
    (ex_arg_code_M)));

/* SystemInitialize for ModelReference: '<Root>/Model' */
ex_arg_code_ref_Init(&(ex_arg_code_DW.Model_InstanceData.rtdw));

/* SystemInitialize for ModelReference: '<Root>/Model1' */
ex_arg_code_ref_Init(&(ex_arg_code_DW.Model1_InstanceData.rtdw));
}

/* Model terminate function */
void ex_arg_code_terminate(void)
{
    /* (no terminate code required) */
}

```

Control Data Types of Model Arguments and Argument Values

When you use model arguments, you can apply a data type to:

- The block parameters that use the arguments (for certain blocks, such as those in the Discrete library).
- The arguments in the referenced model workspace.
- The argument values that you specify in Model blocks.

To generate efficient code by eliminating unnecessary typecasts and C shifts, consider using inherited and context-sensitive data typing to match the data types.

- In the model workspace, use a MATLAB variable whose data type is `double` or a parameter object whose `DataType` property is set to `auto`. In this case, the variable or object uses the same data type as the block parameter.
- When you set the argument values in Model blocks, take advantage of context-sensitive data typing. To set an argument value, use an untyped value.
 - A literal number such as `15.23`. Do not use a typed expression such as `single(15.23)`.
 - A MATLAB variable whose data type is `double`.
 - A `Simulink.Parameter` object whose `DataType` property is set to `auto`.

In these cases, the number, variable, or object uses the same data type as the model argument in the referenced model workspace. If you also configure the model argument to use context-sensitive data typing, you can control the data types of the block parameter, the argument, and the argument value by specifying the type only for the block parameter.

For basic information about controlling parameter data types, see “Parameter Data Types in the Generated Code” on page 15-131.

Use Model Argument in Different Data Type Contexts

If you use a model argument to set multiple block parameter values, and the data types of the block parameters differ, you cannot use context-sensitive data typing (`double` or `auto`) for the argument in the model workspace. You must explicitly specify a data type for the argument. For example, if the argument in the model workspace is a parameter object (such as `Simulink.Parameter`), set the `DataType` property to a value other than `auto`. For more information about this situation, see “Reuse Parameter Data in Different Data Type Contexts” on page 15-143.

In this case, you can continue to take advantage of context-sensitive data typing to control the data type of the argument values that you specify in Model blocks. Each argument value uses the data type that you specify for the corresponding argument in the model workspace.

See Also

Related Examples

- “Generate Code for Model Reference Hierarchy” on page 8-2
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49
- “Parameterize Instances of a Reusable Referenced Model”
- “Organize Data into Structures in Generated Code” on page 15-147
- “Parameter Data Types in the Generated Code” on page 15-131

Generated Functions in a Model Reference Hierarchy

When you generate code for a model, the code generator creates a subfolder `<model>_grt_rtw` (Simulink Coder) or a subfolder `<model>_ert_rtw` (Embedded Coder) in your current working folder and places source code files in this folder. The generated code is in these two primary files:

- `<model>.c` — This file contains the algorithm code. The generated code is mainly comprised of model data and model entry-point functions that are accessible to a caller by including the `<model>.h` file. You can call the generated model entry-point functions from external code or from a version of the generated main function that you modify.
- `<model>.h` — This file contains the extern declarations for block outputs, continuous states, model output, entry points, and timing data. For more information about data structures, see “Data Structures in the Generated Code” on page 15-31.

When you generate code for a referenced model hierarchy, the code generator places the code for the top model in the subfolder `<model>_grt_rtw` (Simulink Coder) or the subfolder `<model>_ert_rtw` (Embedded Coder) in your current working folder and places the code for referenced models in an `s\prj` folder.

Generated Functions in a Top Model

In a model reference hierarchy, the code generator commonly produces these functions for a top model.

Function Name	Description
<code><model>_step</code>	Execution function that contains the output and update code for the blocks in a Simulink model. This function calls the referenced model function. The code generator produces this function by default. To generate separate functions for output and update code, clear the model configuration parameter Single output/update function . For more information, see <code>model_step</code> .
<code><model>_initialize</code>	Initialization function that executes the initialization code for a Simulink model. The code generator calls this function once at the start of the application code. This function might call the <code><ref_model>_Init</code> function of a referenced model to initialize states and <code><ref_model>_initialize</code> for zero initialization. For more information, see <code>model_initialize</code> .
<code><model>_terminate</code>	Termination function that contains the code to turn off a system. For ERT-based models, you can suppress generation of this function by clearing the model configuration parameter Terminate function required . For more information, see <code>model_terminate</code> .

Generated Functions in a Referenced Model

The functions generated for a referenced model are internal functions used by the code generator. These functions are not intended to interface with external code. In a model reference hierarchy, the code generator commonly produces these functions for a referenced model.

Function Name	Description
<code><ref_model>_Init</code>	Initialization function in a referenced model that is called from a top model entry-point function <code><model>_initialize</code> . The <code><model>_initialize</code> of the top model initializes the code for the model reference hierarchy and calls the <code><ref_model>_Init</code> of the child model to initialize the states to their initial condition.
<code><ref_model></code>	Execution function that serves a similar purpose as <code><model>_step</code> of a top model but is not an entry-point function for the external code. The two functions differ in terms of the scope. The <code><ref_model></code> function computes the outputs from internal state and updates that state. The <code><model>_step</code> does the same but calls the individual <code><ref_model></code> function of the referenced models.
<code><ref_model>_initialize</code>	Initialization function that executes the initialization code for a referenced model. The code generator calls this function from a top model entry-point function <code><model>_initialize</code> . The <code><model>_initialize</code> of the top model initializes the code for the model reference hierarchy and calls the <code><ref_model>_initialize</code> of the child model to zero-initialize the referenced model code.

See Also

Related Examples

- “Configure Generated C Function Interface for Model Entry-Point Functions” on page 16-79

Combined Models in Simulink Coder

- “Combine Code Generated for Multiple Models” on page 9-2
- “Function Reuse in Generated Code” on page 9-5
- “File Packaging for Models (Code and Data)” on page 9-10

Combine Code Generated for Multiple Models

To combine code, which the code generator produces for multiple models or multiple instances of a model, into one executable program, use these techniques:

- Model reference hierarchy. See “Model Reference Basics” and “Generate Code for Model Reference Hierarchy” on page 8-2.
- If you have Embedded Coder software, interface the code for multiple models to a common harness program. From the harness program, call the entry-point functions generated for each model. The `ert.tlc` system target file has restrictions, related to embedded processing that might be incompatible with your application.
- Generate reusable, multi-instance code that is reentrant. See “Combine Code Generated for Multiple Models or Multiple Instances of a Model” on page 9-3.

Note The S-function system target (`rtwsfcn.tlc`) does not support combining code generated for multiple models.

To combine models for simulation and code generation, consider using model referencing. Model referencing helps with:

- Symbol naming consistency
- Required scheduling of the overall algorithm
- Model configuration consistency

Considerations for Combining Code for Different Models

If you combine code generated for different models that is, without using referenced models, consider the following:

- Data is global. Symbol (name) clashes can result.
- Configuration parameter settings for the models must match, including settings such as hardware word sizes.
- Reuse and sharing of code can be suboptimal (for example, duplicate code for shared utility functions, scheduling, and solvers).
- Scheduling can be more complex (for example, models can have periodic sample times that are not multiples of each other, making scheduling from a common timer interrupt more complicated)
- For plant models that use continuous time and state, the continuous time signals connecting models are not handled by a single solver like continuous time signals within a model, which can lead to subtle, numeric differences.

Share Data Across Models

Use unidirectional signal connections between models. These connections affect the order in which models are called. For example, if you use an output signal from `modelA` as input to `modelB`, the `modelA` output computation is called first.

Timing Issues

When combining code generated for multiple models or multiple instances of a model:

- Configure the models by using the same solver mode (single-tasking or multitasking).
- If the models use continuous states, configure the models by using the same solver.

If the base rates for the models differ, the main program (such as `rt_malloc_main.c`) must set up the timer interrupt to occur at the greatest common divisor rate of the models. The main program calls each model at a time interval.

Data Logging and External Mode Support

A multiple-model program can log data to separate MAT-files for each model.

Only one of the models in a multiple-model program can use external mode.

Control Ownership of Data

If you have Embedded Coder software, you can specify an owner for individual data items such as signals, parameters, and states. The owner of a data item generates the definition, the memory allocation and initialization, for the data item. For example, if you apply a storage class to a `Simulink.Signal` object so that it appears as a global variable in the generated code, specify one of the combined models as the owner of the object. The code generated for that model defines the variable.

If you use model referencing, you can modularize the generated code and establish clear ownership of data when you work in a team.

If you do not use model referencing, you can prevent generation of duplicate definitions for a data item. For example, suppose you store a `Simulink.Parameter` object in the base workspace and apply the storage class `ExportedGlobal`. If you generate code from two separate models that use the object, each model generates a definition for the corresponding global variable. Instead, you can specify an owner for the object so that only the owner generates a definition.

To specify an owner for a data item:

- 1 Apply a storage class to the data item. See “Organize Parameter Data into a Structure by Using Struct Storage Class” (Embedded Coder).
- 2 Configure the owner of the data item by specifying a value for the **Owner** property.
- 3 Select the model configuration parameter **Use owner from data object for data definition placement**.

For more information about controlling ownership and file placement of data definitions and declarations, see “Control Placement of Global Data Definitions and Declarations in Generated Files” (Embedded Coder).

Combine Code Generated for Multiple Models or Multiple Instances of a Model

For each model that you are combining code for, generate the code.

- 1 Set the system target file to a GRT- or ERT-based system target file. The system target file for the models that you combine must be the same.
- 2 If you intend to have multiple instances of that model in the application, set the model configuration parameter **Code interface packaging** to `Reusable` function. If you specified

an ERT-based system target file, optionally, you can set the model configuration parameter **Use dynamic memory allocation for model initialization**, depending on whether you want to statically or dynamically allocate the memory for each instance of the model.

- 3** Generate source code. The code generator includes an allocation function in the generated file *model.c*. The allocation function dynamically allocates model data for each instance of the model.

After generating source code for each model:

- 1** Compile the code for each model that you are combining.
- 2** Combine the makefiles generated for the models into one makefile.
- 3** Create a combined simulation engine by modifying a main program, such as *rt_malloc_main.c*. The main program initializes and calls the code generated for each model.
- 4** Run the makefile. The makefile links the object files and the main program to an executable program.

See Also

Related Examples

- “Generate Code for Model Reference Hierarchy” on page 8-2
- “Generate Reusable Code from Model Reference Hierarchy” on page 8-16

Function Reuse in Generated Code

This example shows how to configure an atomic subsystem for generating reusable code. To specify that the code generated for a subsystem execute as an atomic unit, on the Block Parameters dialog box, select the **Treat as atomic unit** parameter. That parameter enables the **Function Packaging** parameter on the **Code Generation** tab. The **Function Packaging** parameter has these four settings:

- **Inline**: Inline the subsystem code
- **Nonreusable function**: Function with I/O passed as global data
- **Reusable function**: Function with I/O passed as function arguments
- **Auto**: Let Simulink Coder optimize based on context

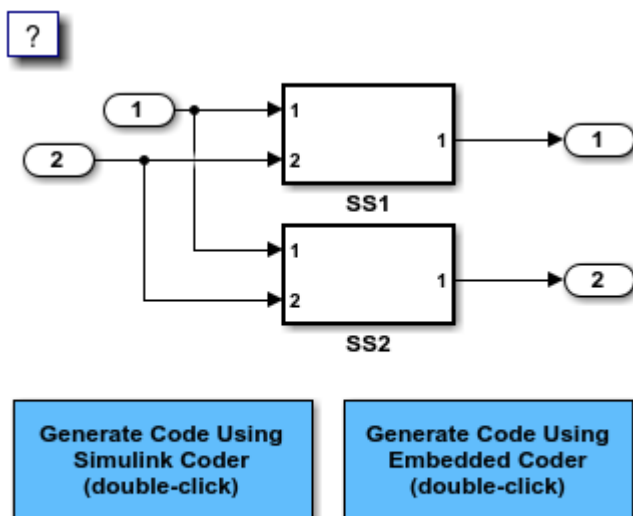
The **Reusable function** and **Auto** settings permit the code generator to reuse subsystem code. The **Reusable function** and **Nonreusable function** settings enable the **Function name options**, **Function name**, and **File name options** parameters.

If you have an Embedded Coder® license, you can configure a nonreusable subsystem to accept arguments.

Example Model

The `rtwdemo_ssreuse` model contains two identical subsystems, SS1 and SS2. For these subsystems, the **Function packaging** parameter is set to **Reusable function**, and the **Function name** parameter is `myfun`. The subsystems are parameterized masked subsystems. To see the contents of the masked subsystems, right-click the subsystem blocks and select **Mask > Look Under Mask**.

```
model = 'rtwdemo_ssreuse';
open_system(model);
```



Copyright 1994-2020 The MathWorks, Inc.

Generate and Inspect Code

Create a temporary folder for the build and inspection process. Open the **Simulink Coder** or **Embedded Coder** app. Then, generate and inspect the code.

```
currentDir=pwd;
[~,cgDir]=rtwdemodir();
slbuild(model)

### Starting build procedure for: rtwdemo_ssreuse
### Successful completion of build procedure for: rtwdemo_ssreuse

Build Summary

Top model targets built:

Model                Action                Rebuild Reason
=====
rtwdemo_ssreuse     Code generated and compiled.  Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 12.727s

cfile=fullfile(cgDir, 'rtwdemo_ssreuse_grt_rtw', 'rtwdemo_ssreuse.c');
rtwdemodbtype(cfile, '/* Model step', '/* Model initialize', 1, 0);

/* Model step function */
void rtwdemo_ssreuse_step(void)
{
    /* Outputs for Atomic SubSystem: '<Root>/SS1' */

    /* Inport: '<Root>/In1' incorporates:
     * Inport: '<Root>/In2'
     */
    myfun(rtwdemo_ssreuse_U.In1, rtwdemo_ssreuse_U.In2, rtwdemo_ssreuse_P.T1Data,
          rtwdemo_ssreuse_P.T1Break, &rtwdemo_ssreuse_B.SS1);

    /* End of Outputs for SubSystem: '<Root>/SS1' */

    /* Output: '<Root>/Out1' */
    rtwdemo_ssreuse_Y.Out1 = rtwdemo_ssreuse_B.SS1.LookupTable;

    /* Outputs for Atomic SubSystem: '<Root>/SS2' */

    /* Inport: '<Root>/In1' incorporates:
     * Inport: '<Root>/In2'
     */
    myfun(rtwdemo_ssreuse_U.In1, rtwdemo_ssreuse_U.In2, rtwdemo_ssreuse_P.T2Data,
          rtwdemo_ssreuse_P.T2Break, &rtwdemo_ssreuse_B.SS2);

    /* End of Outputs for SubSystem: '<Root>/SS2' */

    /* Output: '<Root>/Out2' */
    rtwdemo_ssreuse_Y.Out2 = rtwdemo_ssreuse_B.SS2.LookupTable;
}

```

In the model step function, there are two calls to the reusable function, `myfun`. The mask parameters, `T1Break`, `T1Data`, `T2Break`, and `T2Data`, are function arguments.

Change the **Function Packaging** parameter to Inline.

```
set_param('rtwdemo_ssreuse/SS1','RTWSystemCode','Inline')
set_param('rtwdemo_ssreuse/SS2','RTWSystemCode','Inline')
```

Generate and inspect the code.

```
slbuild(model)
```

```
### Starting build procedure for: rtwdemo_ssreuse
### Successful completion of build procedure for: rtwdemo_ssreuse
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_ssreuse	Code generated and compiled.	Generated code was out of date.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 10.379s
```

```
cfile=fullfile(cgDir, 'rtwdemo_ssreuse_grt_rtw', 'rtwdemo_ssreuse.c');
rtwdemodbtype(cfile, '/* Model step', '/* Model initialize', 1, 0);
```

```
/* Model step function */
void rtwdemo_ssreuse_step(void)
{
    real_T Out1_tmp;

    /* Outputs for Atomic SubSystem: '<Root>/SS2' */
    /* Outputs for Atomic SubSystem: '<Root>/SS1' */
    /* Sum: '<S1>/Sum' incorporates:
     * Inport: '<Root>/In1'
     * Inport: '<Root>/In2'
     * Sum: '<S2>/Sum'
     */
    Out1_tmp = rtwdemo_ssreuse_U.In1 + rtwdemo_ssreuse_U.In2;

    /* End of Outputs for SubSystem: '<Root>/SS2' */

    /* Output: '<Root>/Out1' incorporates:
     * Lookup_n-D: '<S1>/Lookup Table'
     * Sum: '<S1>/Sum'
     */
    rtwdemo_ssreuse_Y.Out1 = look1_binlx(Out1_tmp, rtwdemo_ssreuse_P.T1Break,
        rtwdemo_ssreuse_P.T1Data, 10U);

    /* End of Outputs for SubSystem: '<Root>/SS1' */

    /* Outputs for Atomic SubSystem: '<Root>/SS2' */
    /* Output: '<Root>/Out2' incorporates:
     * Lookup_n-D: '<S2>/Lookup Table'
     */
    rtwdemo_ssreuse_Y.Out2 = look1_binlx(Out1_tmp, rtwdemo_ssreuse_P.T2Break,
        rtwdemo_ssreuse_P.T2Data, 10U);
```

```

    /* End of Outputs for SubSystem: '<Root>/SS2' */
}

```

In the model step function, the subsystem code is inlined.

Change the **Function Packaging** parameter to Nonreusable function. For SS2, change the **Function name** parameter to myfun2.

```

set_param('rtwdemo_ssreuse/SS1','RTWSystemCode','Nonreusable function')
set_param('rtwdemo_ssreuse/SS2','RTWSystemCode','Nonreusable function')
set_param('rtwdemo_ssreuse/SS2','RTWFcnName','myfun2')

```

Generate and inspect the code.

```

slbuild(model)

### Starting build procedure for: rtwdemo_ssreuse
### Successful completion of build procedure for: rtwdemo_ssreuse

Build Summary

Top model targets built:

Model          Action          Rebuild Reason
=====
rtwdemo_ssreuse Code generated and compiled. Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.175s

cfile=fullfile(cgDir, 'rtwdemo_ssreuse_grt_rtw', 'rtwdemo_ssreuse.c');
rtwdemodbtype(cfile, '/* Model step', '/* Model initialize', 1, 0);

/* Model step function */
void rtwdemo_ssreuse_step(void)
{
    /* Outputs for Atomic SubSystem: '<Root>/SS1' */
    myfun();

    /* End of Outputs for SubSystem: '<Root>/SS1' */

    /* Outputs for Atomic SubSystem: '<Root>/SS2' */
    myfun2();

    /* End of Outputs for SubSystem: '<Root>/SS2' */
}

```

The model step function contains calls to the functions myfun and myfun2. These functions have a void-void interface.

Change the **Function Packaging** parameter to Auto.

```

set_param('rtwdemo_ssreuse/SS1','RTWSystemCode','Auto')
set_param('rtwdemo_ssreuse/SS2','RTWSystemCode','Auto')

```

For the auto setting, Simulink Coder chooses the optimal format. For this model, the optimal format is a reusable function.

Close the model and clean up.

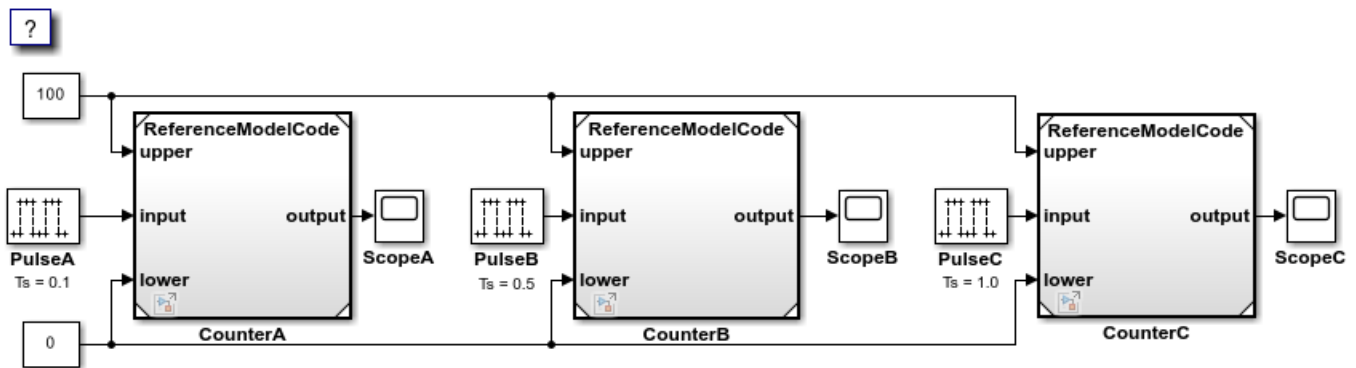
```
bdclose(model)
rtwdemoclean;
cd(currentDir)
```

File Packaging for Models (Code and Data)

This example shows how referenced models provide system interface encapsulation and incremental code generation. You can reference one model from another model (one or more times), and aspects of the referenced model are fixed: input/output signal types, parameter types, and sample times. Therefore, you can modularize your design and perform incremental code generation.

The code generator partitions data and functions of a referenced model into its own set of files, independent of its parent model. In this example, the referenced model `ReferenceModelCode` is referenced three times. For simulation and code generation, the model is incrementally generated, which means `ReferenceModelCode` builds the first time, but not on subsequent builds, unless you change `ReferenceModelCode`.

```
open_system('TopModelCode')
```



Copyright 1994-2022 The MathWorks, Inc.

- 1 Generate and inspect the code.
- 2 Change the value of one of the constant inputs in `TopModelCode`.
- 3 Generate code. The code generator does not reproduce code for reference model `TopModelCode`.

```
bdclose('TopModelCode');
```

Code Reuse for Simulink Coder

What Is Code Reuse?

Code reuse is a programming technique that reduces time and resources to develop software. When you develop code for reuse, the code serves multiple purposes. The technique involves modularization, which enables multiple individuals to develop code for various system components independently and in parallel. The technique also simplifies software distribution.

A software library is an example of code reuse. A library can contain code for several functions, each having specific behavior. To use a library, you need to know only the interface, which is the specification for calling the library functions.

Reusable code can be reentrant. You can invoke, interrupt, and reinvoke reentrant code. Reentrant code resides in shared memory. Instances of data associated with each use of the code are unique and preserved. Multiple calls to a reusable, reentrant function, for example, can access the function code with each call maintaining a unique data set.

Applications of code reuse include:

- Team-based development.
- Switching between implementations of a function.
- Intellectual property protection.
- Overriding library behavior with a custom implementation.
- Implementing a library that is based on standard library functions.
- Unit testing.
- Execution speed improvement.

The code generator supports code reuse and reentrancy. You can use key Simulink componentization techniques to partition a model into design components that you simulate, generate code for, and verify independently. You can save individual components as subsystems, libraries, referenced models, or combinations of these elements from which you can generate code. For example, the code generator produces reusable function code from library subsystems and export-function models.

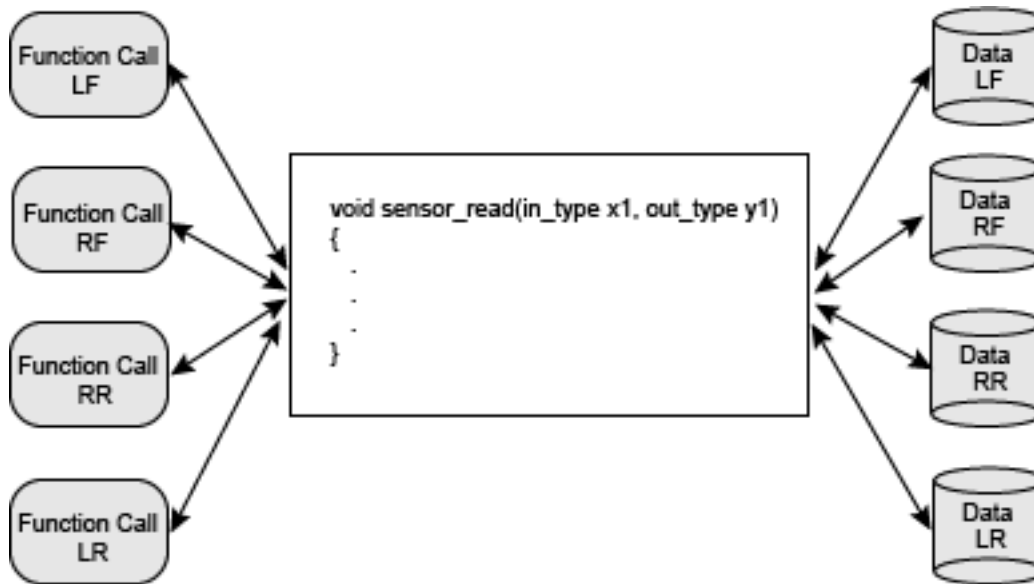
See Also

More About

- “What Is Reentrant Code?” (Embedded Coder)
- “Choose a Componentization Technique for Code Reuse” (Embedded Coder)

What Is Reentrant Code?

Reentrant (multi-instance) code is a reusable routine that multiple programs can invoke, interrupt, and reinvoked simultaneously. When you want to reuse code, but associate each instance of the shared code with unique, preserved data, use reentrant code. For example, consider this figure that shows four instances of function `sensor_read`. Although the function algorithm is the same in each case, the data associated with each instance varies and is relative to the sensor position.



To make generated code for a model component reentrant, you configure the component such that model entry-point functions receive root-level input and output data as arguments. How you configure the component depends on the modeling style or technique that you apply.

See Also

More About

- “What Is Code Reuse?” (Embedded Coder)
- “Choose a Componentization Technique for Code Reuse” (Embedded Coder)

Choose a Componentization Technique for Code Reuse

Key componentization techniques that you can use with Simulink and the code generator to produce reusable code include:

- Referenced models
- Subsystems
- Library subsystems
- Combinations of models, subsystems, and library subsystems

Choose componentization techniques based on your code reuse goals.

Goal	Referenced Model	Subsystem in Model	Subsystem in Library
Design for explicit code reuse.	√	√	√
Facilitate parallel team development.	√	√	√
Reuse function within a model or across models.	√		
Reduce build time by generating reusable code incrementally.	√		
Verify reusable code with SIL or PIL simulation.	√		
Optimize generated code by configuring code generator to detect opportunities for code reuse.	√	√	√
Maximize reuse with context-dependent behavior.		√	√
Develop a frequently used, and infrequently changed, utility function.			√

See Also

More About

- “Compare Capabilities of Model Components”
- “Generate Reentrant Code from Top Models” (Embedded Coder)

- “Generate Reentrant Code from Subsystems” (Embedded Coder)
- “Generate Reentrant Code from Simulink Function Blocks” (Embedded Coder)
- “Generate Reusable Code from Library Subsystems Shared Across Models” (Embedded Coder)
- “Library-Based Code Generation for Reusable Library Subsystems” (Embedded Coder)

Simulink Function Blocks and Code Generation

Why Generate Code from Simulink Function Blocks and Function Callers?

Simulink Function blocks provide a mechanism for generating C or C++ code for modeling components that represent shared resources. You define the logic as a resource in a Simulink Function block, which separates the function interface (name and arguments) from the implementation of the logic. Function callers (Function Caller blocks, MATLAB Function blocks, and Stateflow charts) can then reuse the function logic at different levels of the model hierarchy.

Simulink Function blocks provide an alternative to reusable subsystems. For example, a consideration for using a Simulink Function block instead of a subsystem block is that a Simulink Function block shares states between function callers. The code generator produces one function. If the Simulink Function block contains blocks that have states, such as a delay or memory, the states persist between function callers. The order of the function calls is an important consideration.

Reusable functions that the code generator produces from subsystems do not share states. The code generator produces one function for multiple instances of the subsystem as an optimization. If that subsystem contains blocks that have states, the code generator produces one function, but passes a different state variable to each instance. The instances do not share states.

Other uses of Simulink Function blocks and callers include:

- Nesting calls to a function.
- Calling a function defined in one modeling component from another modeling component.
- Generating functions that are globally accessible or scoped.
- Producing code for a client and server application.


Implementation Options

Choose how to implement Simulink functions and function callers based on your code generation requirements. Considerations include:

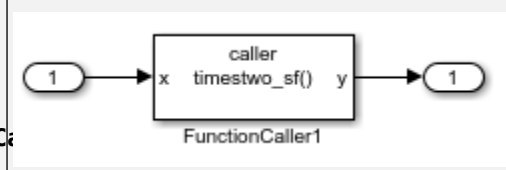


- How you represent a function and function callers in a model
- Scope of a function
- Whether to export a function from a model (requires Embedded Coder)
- Function code interface customizations (requires Embedded Coder)
- Code generation requirements
- Code generation limitations

Choose a Modeling Pattern

This table shows C code for a function that multiplies an input value times two and a Simulink Function block that can represent that function in a Simulink model.

Function Definition	Modeling Element
<pre> codegen-folder/subsystem.c #include "timestwo_sf.h" #include "ex_slfunc_comp_sf.h" #include "ex_slfunc_comp_sf_private.h" void timestwo_sf(real_T rtu_x, real_T *rty_y) { *rty_y = 2.0 * rtu_x; } </pre>	Simulink Function block 

A Simulink function caller invokes a function defined with a Simulink Function block. From anywhere in a model or chart hierarchy, you can call a function defined with a Simulink Function block by using one of these modeling elements:

Function Call	Modeling Element
<pre> codegen-folder/model.c void ex_slfunc_comp_sf_step(void) { real_T rtb_FunctionCaller1; timestwo_sf(ex_slfunc_comp_sf_U.In1, &rtb_FunctionCaller1); ex_slfunc_comp_sf_Y.Out1 = rtb_FunctionCaller1; . . } </pre>	Function Caller block 
<pre> codegen-folder/model.c void ex_slfunc_comp_gf_step(void) { real_T rtb_y1_l; timestwo_gf(ex_slfunc_comp_gf_U.In4, &rtb_y1_l); ex_slfunc_comp_gf_Y.Out4 = rtb_y1_l; . . } </pre>	Stateflow chart transition 
<pre> codegen-folder/model.c void ex_slfunc_comp_mf_step(void) { real_T rtb_y; timestwo_mf(ex_slfunc_comp_mf_U.In3, &rtb_y); ex_slfunc_comp_mf_Y.Out3 = rtb_y; . . } </pre>	MATLAB Function block 

For more information about modeling choices, see “Simulink Functions Overview”.

Specify Function Scope

A function that you define with a Simulink Function block can be global or scoped.

- Global--The code generator places code for a global function in source and header files that are separate from model code files (for example, *function.c* and *function.h*). The separate files make the function code available for sharing between function callers.
- Scoped--The code generator places code for a scoped function in model code files (*model.c* and *model.h*). To call a scoped function in the context of a model, the function caller must be at the same level as the function in the model hierarchy, or one or more levels below.

To create a library of functions that are accessible from anywhere in the generated model code, set up each function as a scoped Simulink Function block. Place each scoped function within a virtual subsystem at the root level of a model.

For more information, see and “Scoped, Global, and Port-Scoped Simulink Function Blocks Overview”.

Decide Whether to Generate Export Function Code

Although you can use Simulink Function blocks in a single top-model design, function code is more reusable when you generate it as standalone, atomic components. If you are using Embedded Coder, you can design functions in the context of export-function models.

For information, see “Generate Component Source Code for Export to External Code Base” (Embedded Coder) and “Export-Function Models Overview”.

Configure Function Code Interfaces

With Embedded Coder, simplify integration of generated code with external code by configuring generated function code interfaces for Simulink Function and Function Caller blocks. You can configure the function interfaces for:

- Global Simulink Function blocks
- Scoped Simulink Function blocks that are at the root level of a model

For more information, see “Configure Entry-Point Function Interfaces for Simulink Function and Function Caller Blocks” (Embedded Coder).

Uncalled Simulink Function Blocks

If you use a Simulink Function block in a rate-based model and do not call that function, the code generator treats the Simulink Function block as a constant and does not produce function code. For example, this can occur during model development when you are ready to define a function, but are not ready to identify a caller.

To identify such blocks in a rate-based model, display sample time colors during simulation (see “View Sample Time Information”). By default, Constant blocks appear magenta. Because the code generator considers uncalled Simulink Function blocks constants during simulation, they appear magenta.

Requirements

- Within a model hierarchy, function names are unique. If the code generator finds multiple functions with the same name, it issues an error. Change the name of one of the functions and delete the `s\prj` folder.
- The signature (for example, the arguments and argument data types) for a function and function callers must match.
 - If the code generator finds the function first and the signature of a function caller does not match, the code generator issues an error. Change the function caller signature to match the signature of the Simulink Function block or delete the `s\prj` folder.
 - If the code generator finds a function caller first and the signature of the function does not match, the code generator issues a warning message. Change the signature of the function or function caller so that the signatures match.
- In a Simulink Function block definition, do not define input and output signals for Argument Inport and Argument Outport blocks with a storage class.
- Do not specify Argument Inport and Argument Outport blocks as test points.
- If you specify the data type of input and output signals for Argument Inport and Argument Outport blocks as a `Simulink.IntEnumType`, `Simulink.AliasType`, or `Simulink.Bus`, set the `DataScope` property to `Imported` or `Exported`.
- A function interface and function callers must agree in data type, complexity, dimension, and number of arguments.

Limitation

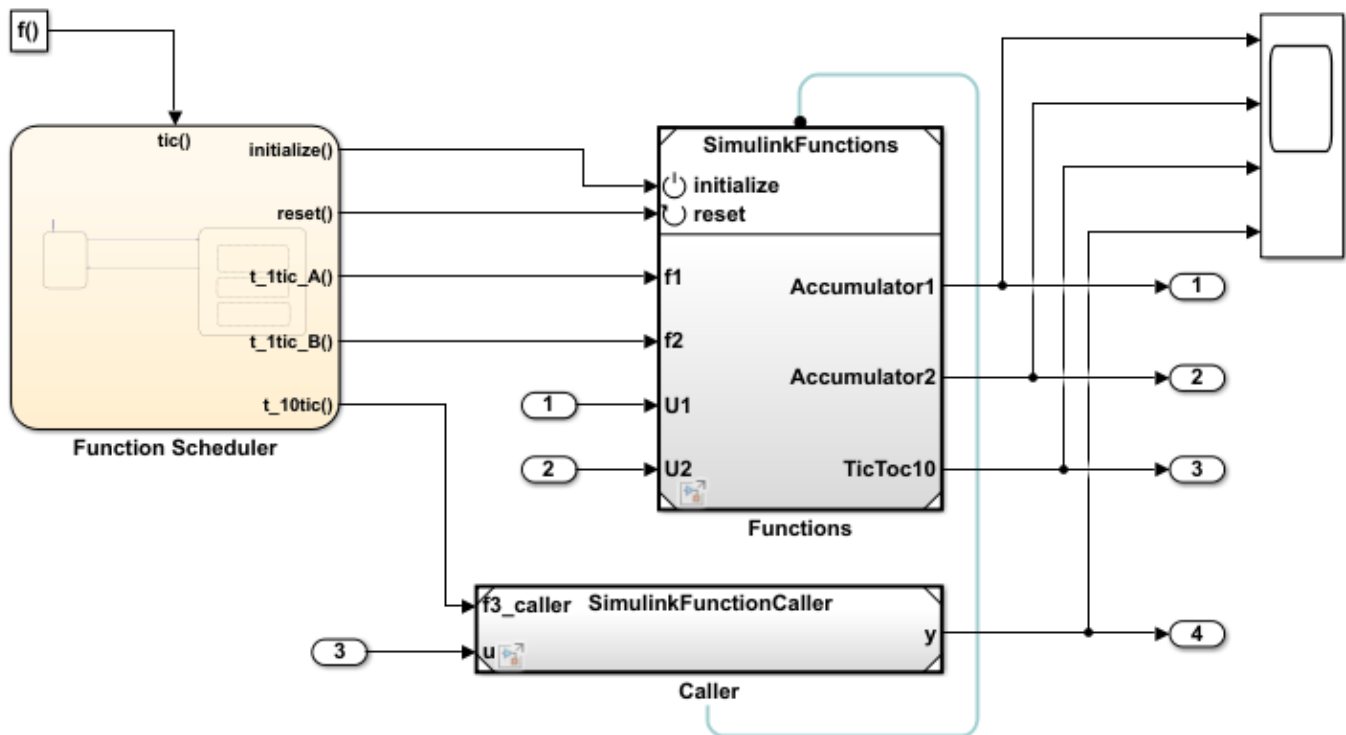
Simulink functions and function callers do not honor the `MaxStackSize` parameter.

Generate Code for Simulink Function and Function Caller

This example shows how to generate C code for Simulink Function and Function Caller blocks and displays the relevant generated code.

Open the example model `SimulinkFunctionsTestHarness`. The model uses Stateflow software, but this example reviews only the code generated from the referenced models.

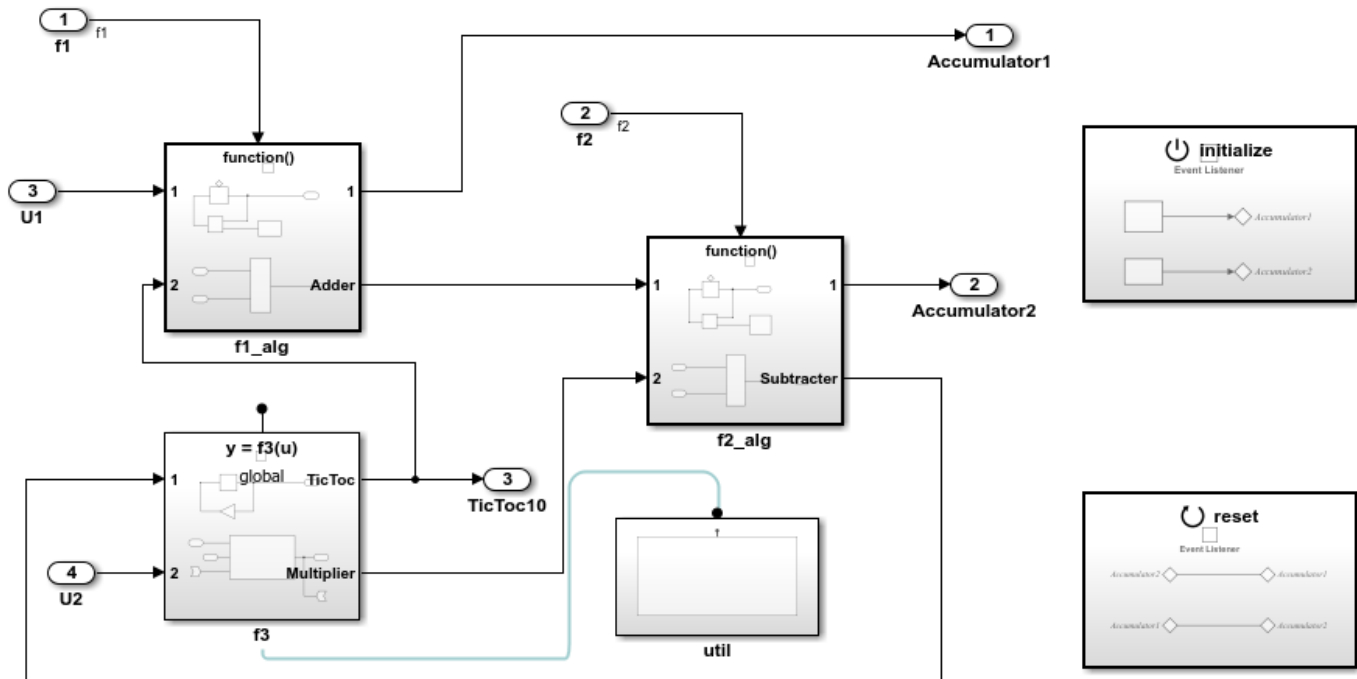
```
open_system('SimulinkFunctionsTestHarness')
```



Generate Code for Function Definition

Open model SimulinkFunctions. The Simulink Function block is the f3 subsystem defined as $y = f3(u)$.

```
open_system('SimulinkFunctions')
```



Generate code. The code generator creates `SimulinkFunctions.c`. This file contains the function initialization code and function definition.

Initialization code for function f3:

```
void SimulinkFunctions_initialize(void)
{
    rtDWork.Delay_DSTATE = 1;
}
```

Code for function f3:

```
void f3(real_T rtu_u, real_T *rty_y)
{
    int8_T = rtb_Gain;

    rtY.TicToc10 = rtDWork.Delay_DSTATE;

    rtb_Gain = (int8_T)(int32_T)-(int32_T)rtY.TicToc10;

    adder(rtB.Subtract, rtU.U2, rtu_u, &rtB.FunctionCaller);

    *rty_y = rtB.FunctionCaller;

    rtDWork.Delay_DSTATE = rtb_Gain;
}

static void adder(real_T rtu_u1, real_T rtu_u2, real_T rtu_u3, real_T *rty_y)
{
    *rty_y = (rtu_u1 + rtu_u2) + rtu_u3;
}
```

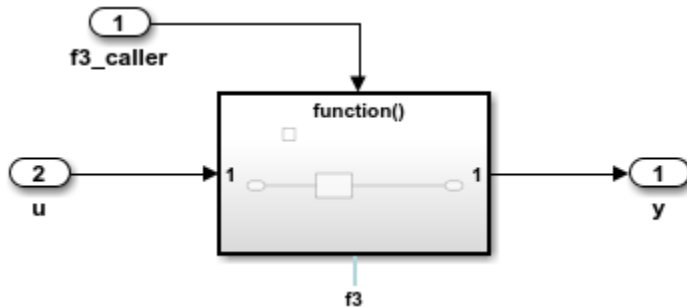
The shared header file `f3.h` contains the entry-point declaration for function f3.

```
#include "rtwtypes.h"
extern void f3(real_T rtu_u, real_T *rtty_y);
```

Generate Code For Function Caller

Open model `SimulinkFunctionCaller` to view the contents of the caller subsystem.

```
open_system('SimulinkFunctionCaller')
```



Generate code. The code generator creates the files `SimulinkFunctionCaller.h` and `SimulinkFunctionCaller.c` in the folder `SimulinkFunctionCaller_ert_rtw`.

`SimulinkFunctionCaller.h` includes the shared header file, `f3.h`, which contains the function entry-point declaration.

`SimulinkFunctionCaller.c` calls function `f3`.

```
void f3_caller(void)
{
    rtY.y = f3(rtu_u);
}
```

More About

- “Generate Reentrant Code from Simulink Function Blocks” (Embedded Coder)
- “Simulink Functions Overview”
- “Scoped Simulink Function Blocks in Models”
- “Simulink Function Blocks in Referenced Models”

See Also

Simulink Function | Function Caller

Related Examples

- “Simulink Functions Overview”
- “Generate Reentrant Code from Simulink Function Blocks” (Embedded Coder)
- “Configure Entry-Point Function Interfaces for Simulink Function and Function Caller Blocks” (Embedded Coder)
- “Client-Server Communication Interfaces” (Embedded Coder)

Generate Reentrant Code from Top Models

By default, for top models, the code generator produces code that is not reentrant. Entry-point functions have a void-void interface. Code communicates with other code by sharing access to global data structures that reside in shared memory.

For applications that can benefit from reuse and require that each use or instance of the code maintains its own unique data, configure a model such that the code generator produces reentrant code. To generate reentrant code, set the model configuration parameter `Code interface packaging` to `Reusable function`. If you are using Embedded Coder and generating C++ code, alternatively, you can set the parameter to `C++ class`. In both cases, the code generator:

- Packages model data, such as block I/O, DWork vectors, and parameters, in the real-time model data structure (`rtModel`).
- Passes the real-time model data structure as an input argument, by reference, to generated model entry-point functions.
- Passes root-level input and output arguments to generated model entry-point functions as individual arguments.
- Allocates memory for model data structures statically.
- Exports the real-time model data structure in the generated header file `model.h`.

Apply additional diagnostic and code generation control by setting these model configuration parameters:

- To select the severity level for diagnostic messages that the code generator displays when a model does not meet requirements for multi-instance code, set parameter `Multi-instance code error diagnostic` to `None`, `Warning`, or `Error`. Set the parameter to `Error` unless you need to alter the severity level for diagnostics displayed when a model violates requirements for generating multi-instance code.
- To control how the generated code passes root-level model input and output to the reusable execution (step) function (requires Embedded Coder), set parameter `Pass root-level I/O as (Embedded Coder)` to `Individual arguments`, `Structure reference`, or `Part of model data structure`.

When you set `Code interface packaging` to `Reusable function`, the code generator packages model data (such as block I/O, Dwork, and parameters) into the real-time model data structure, and passes the model structure to generated model entry-point functions. If you set `Pass root-level I/O as (Embedded Coder)` to `Part of model data structure`, the code generator packages root-level model input and output into the real-time model data structure also.

- To reduce memory usage by omitting the error status field from the real-time model data structure (requires Embedded Coder), select model configuration parameter **Remove error status field in real-time model data structure**.
- To include a function in the generated file `model.c` that uses `malloc` to dynamically allocate memory for model instance data (requires Embedded Coder), select model configuration parameter **Use dynamic memory allocation for model initialization**. If you do not select this parameter, the generated code statically allocates memory for model data structures.

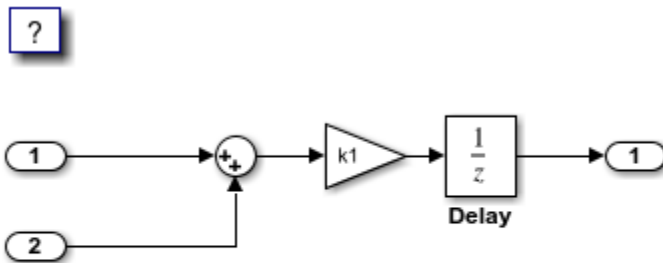
Generate Reentrant, Multi-Instance Code

This example shows you how to configure a model for reentrant, multi-instance code generation. Multiple programs can use reentrant code simultaneously. When you configure a model for reentrancy, the execution (step) entry-point function uses root-level input and output arguments instead of global data structures. After examining the configuration settings, generate and review the generated code.

Open the Model

Open the model `Reusable`. The model contains two root Inport blocks and a root Outport block.

```
model='Reusable';
open_system(model);
```



Copyright 1994-2022 The MathWorks, Inc.

Examine Relevant Model Configuration Parameter Settings

1. Open the **Embedded Coder** app.
2. Open the Model Configuration Parameters dialog box.
3. Model configuration parameter **System target file** is set to `ert.tlc`. Although you can generate reentrant code for a model configured with **System target file** set to `grt.tlc`, ERT and ERT-based system target files provide more control over how the code passes root-level I/O.
4. Open the **Code Generation > Interface** pane and explore relevant model configuration parameter settings.
 - **Code interface packaging** is set to `Reusable` function. This parameter setting instructs the code generator to produce reusable, multi-instance code.
 - Setting of `Reusable` function displays parameter **Multi-instance code error diagnostic**. That parameter is set to `Error`, indicating that the code generator abort if the model violates requirements for generating multi-instance code.
 - **Pass root-level I/O as** is set to `Part of model data structure`. This setting packages root-level model input and output into the real-time model data structure (`rtModel`), which is an optimized data structure that replaces `SimStruct` as the top-level data structure for a model.
 - **Remove error status field in real-time model data structure** is selected. This parameter setting reduces memory usage by omitting the error status field from the generated real-time model data structure.

Generate and Review Code

```
slbuild(model);

### Starting build procedure for: Reusable
### Successful completion of build procedure for: Reusable

Build Summary

Top model targets built:

Model      Action                                     Rebuild Reason
=====
Reusable   Code generated and compiled.             Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.936s
```

Review the Generated Code

- `ert_main.c` is an example main program (execution framework) for the model. This code controls model code execution by calling the entry-point function `Reusable_step`. Use this file as a starting point for coding your execution framework.
- `Reusable.c` contains entry points for the code that implements the model algorithm. This file includes the rate scheduling code.
- `Reusable.h` declare model data structures and a public interface to the model entry points and data structures.
- `rtwtypes.h` defines data types, structures, and macros that the generated code requires.

Code Interface

Open and review the Code Interface Report. Use the information in that report to write the interface code for your execution framework.

1. Include the generated header file by adding directive `#include Reusable.h`.
2. Write input data to the generated code for model Inport blocks.
3. Call the generated entry-point functions.
4. Read data from the generated code for the model Outport block.

Input ports:

- `<Root>/In1` of data `real_T` with dimension of 1
- `<Root>/In2` of data `real_T` with dimension of 1

Entry-point functions:

- Initialization entry-point function, `void Reusable_initialize(RT_MODEL *const rtM)`. At startup, call this function once.
- Output and update (step) entry-point function, `void Reusable_step(RT_MODEL *const rtM)`. Call this function periodically at the fastest rate in the model. For this model, call the function every second. To achieve real-time execution, attach this function to a timer.

Output port:

- <Root>/Out1 of data type `real_T` with dimension of 1

Examine the Step Function

Examine the `|Reusable_step|` function code in `|Reusable.c|`.

```
cfile = fullfile('Reusable_ert_rtw','Reusable.c');
rtwdemodtype(cfile,'/* Model step function', '/* Model initialize function ', 1, 0);

/* Model step function */
void Reusable_step(RT_MODEL *const rtM)
{
    D_Work *rtDWork = rtM->dwork;
    ExternalInputs *rtU = (ExternalInputs *) rtM->inputs;
    ExternalOutputs *rtY = (ExternalOutputs *) rtM->outputs;

    /* Outport: '<Root>/Out1' incorporates:
     *   UnitDelay: '<Root>/Delay'
     */
    rtY->Out1 = rtDWork->Delay_DSTATE;

    /* Gain: '<Root>/Gain' incorporates:
     *   Sum: '<Root>/Sum'
     *   UnitDelay: '<Root>/Delay'
     */
    rtDWork->Delay_DSTATE = (rtU->In1 + rtU->In2) * rtP.k1;
}
```

The code generator passes model data to the `Reusable_step` function as part of the real-time model data structure. Try different settings for model configuration parameters **Code interface packaging** and **Pass root-level I/O** and regenerate code. Observe how the function prototype changes.

Close the Model and Report

Close the model and the code generation report.

```
bdclose(model)
```

Share Data Between Instances

When your code calls a reentrant model entry-point function multiple times, each call represents an instance of the model. By default, the code generator generates code that assumes each instance reads from and writes to a separate copy of the signals, block states, and parameters in the model.

- To share a piece of parameter data between the instances (for example, to share a setpoint for a reusable PID control algorithm), use a parameter object, such as `Simulink.Parameter`. Then, configure the parameter with a storage class other than `Auto` or in the Code Mappings editor, set the default storage class for the corresponding category of parameter data `Default` (the default setting) to `Model default`. The parameter object appears in the code as a global symbol, such as a global variable, that the function accesses directly. For more information, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.
- To share a piece of nonparameter data between the instances (for example, to share a fault indication or an accumulator), use a data store. You can configure the data store to appear in the

code as a global symbol, such as a global variable, that the function accesses directly. Create a global data store by using a `Simulink.Signal` object or use a Data Store Memory block and select block parameter **Share across model instances**. For more information, see “Model Global Data by Creating Data Stores” and Data Store Memory.

See Also

More About

- “What Is Reentrant Code?” (Embedded Coder)
- “Storage Classes and Reentrant, Multi-Instance Models and Components” on page 16-8

Generate Reentrant Code from Simulink Function Blocks

If you are using Embedded Coder, you can generate reusable, reentrant code by representing an algorithm as a scoped Simulink Function block. Examples of when to generate reentrant code from Simulink Function blocks are when a function shares state between function callers within a model or for client/server applications. You can generate code that is highly modularized by using multiple instances of a shared Simulink Function block in an export-function model. The code generator produces function code and associates each use or call to the function with instance-specific data. The scope of the function depends on whether you place the function at the root level of model or in a subsystem.

The code generator produces reentrant function code when you configure a:

- Top model with model configuration parameter Code interface packaging set to **Reusable function** or **C++ class** (C++ only).
- Referenced model with Total number of instances allowed per top model set to **Multiple**.

Call a function represented by a scoped Simulink Function block from one level above, at the same level, or from a level below the level of the function definition. You can scope a function in an atomic or nonvirtual subsystem, but function call accessibility is limited to the same level or below of the hierarchy. The function name does not have to be unique.

Identify Requirements

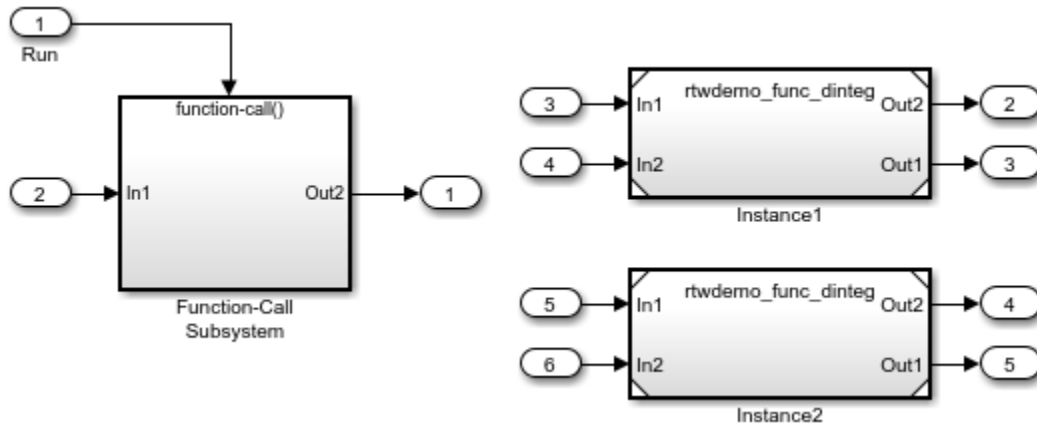
Before and while you design your model, consider:

- How many instances of each function are required?
- Do you need to restrict call sites for a function to the model containing the function definition?
- Do you need for a function to interface with signals in the local environment, but keep those signals hidden from callers?
- Do functions need to communicate directly with each other?
- Do functions need to connect to external I/O?
- Do you need to log function output?

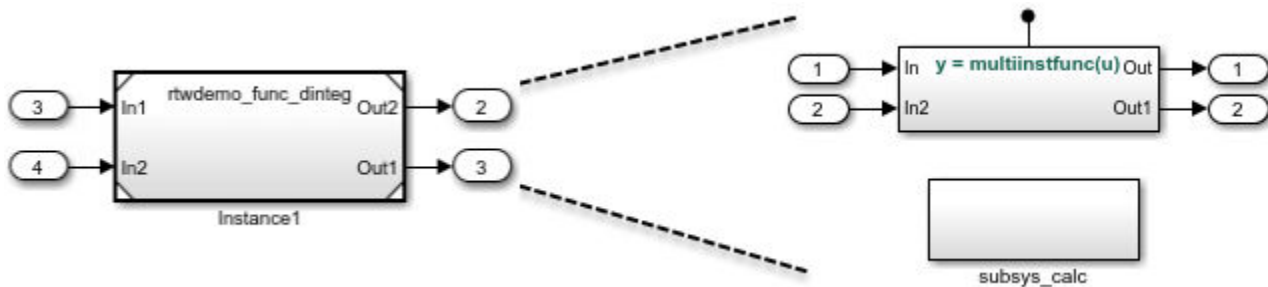
Create Model

Use the example model component `rtwdemo_comp` to see how to use Simulink Function blocks to generate reentrant C code. Use example model `rtwdemo_comp_cpp` if you prefer to generate C++ code. Open the model and examine the model hierarchy.

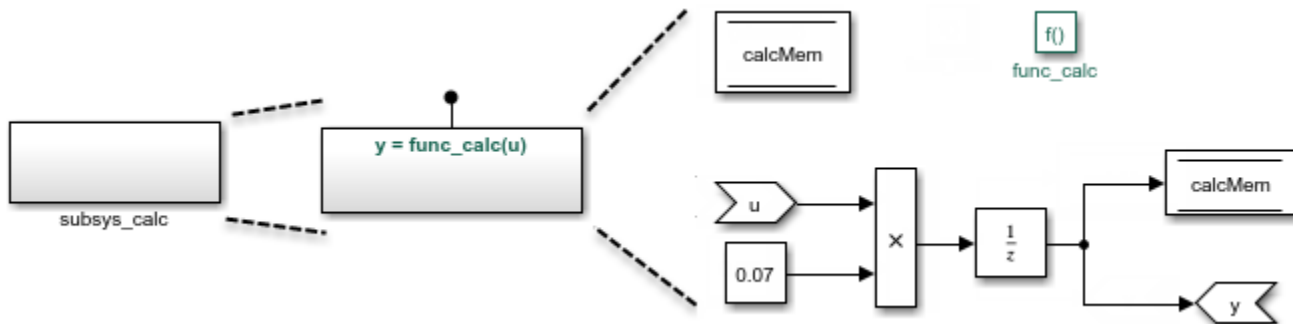
The top level of the model includes a function-call subsystem and two instances of a referenced model.



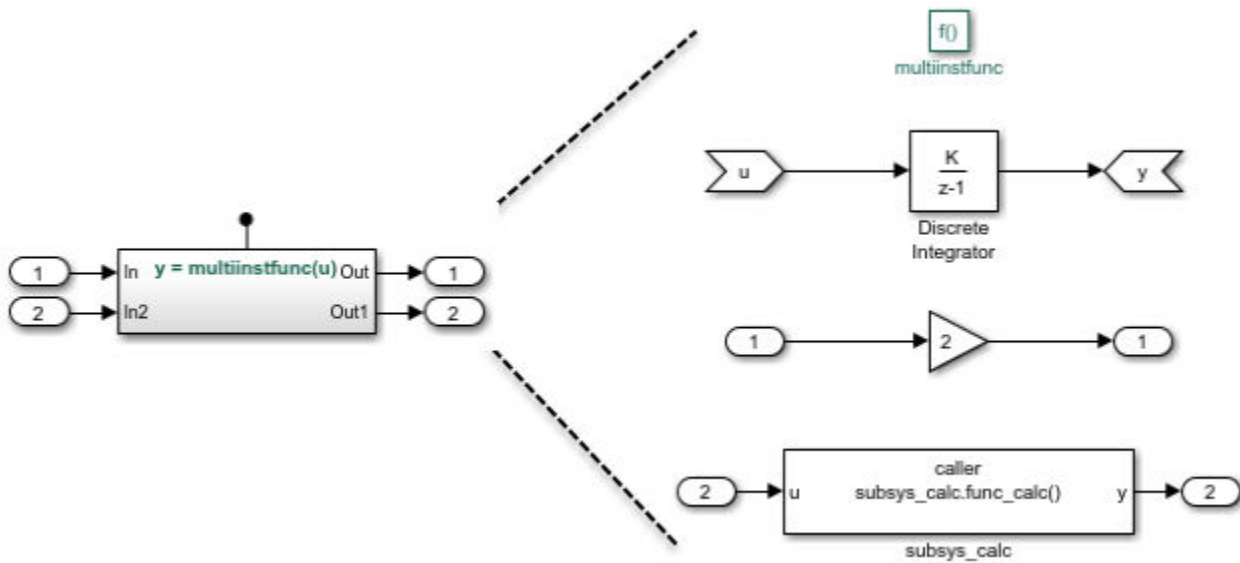
The referenced model, `rtwdemo_func_dinteg`, consists of a Simulink Function block that defines function `multiinstfunc` and subsystem `subsys_calc`.



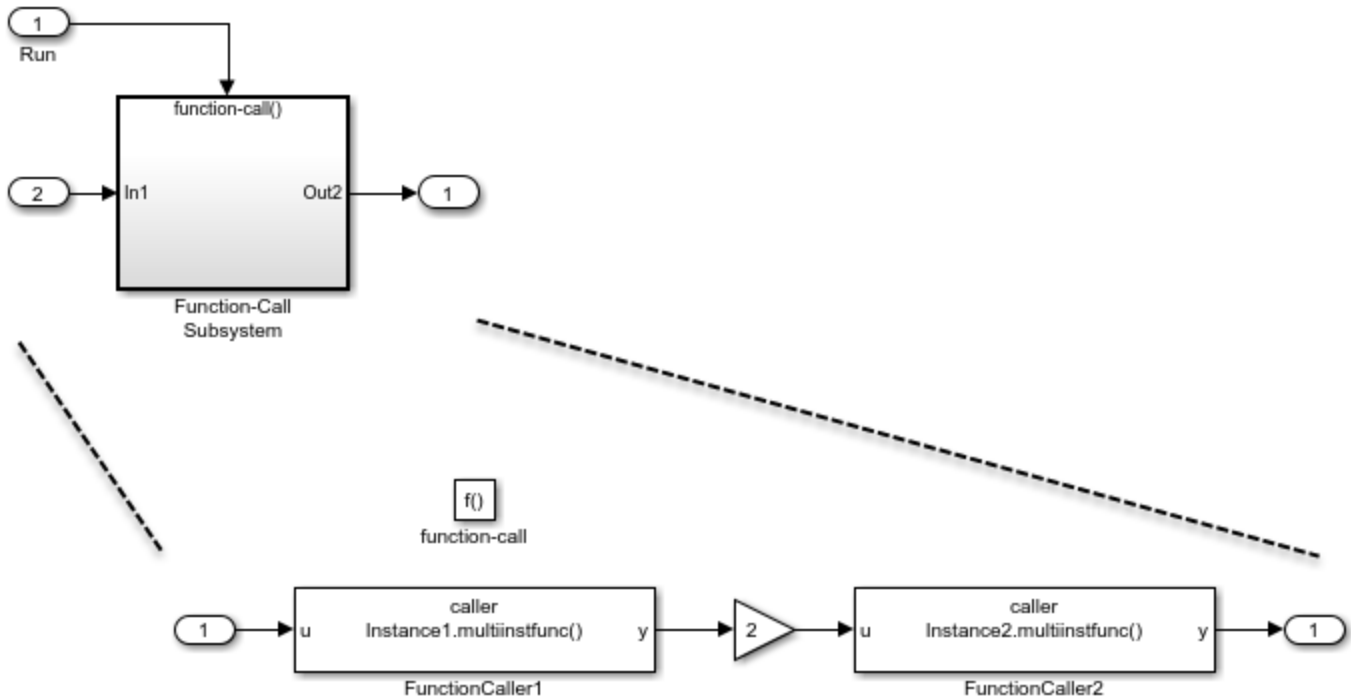
The subsystem consists of a Simulink Function block. This use of a Simulink Function block shows how you can limit the scope of the function that the block defines to the model that contains the subsystem. The code generator produces function code for `func_calc` and associates each call to the function with instance-specific data. The data includes states, such as data stored in memory.



The Simulink Function block that defines function `multiinstfunc` uses a Function Caller block to invoke function `func_calc`. That Simulink Function block also shows that it can interface to signals in the local environment of the block through Inport and Outport blocks.



At the top level of model `rtwdemo_comp`, the function-call subsystem uses Function Caller blocks to invoke the two instances of function `multiinstfunc`. The code generator produces function code and associates each call with instance-specific data.



Configure Model and Model Elements

Configure Simulink Function Blocks

Configure the Simulink Function blocks by setting parameters for the function Trigger Port block. For the code generator to produce reentrant code from Simulink Function blocks:

- Configure block instances with the same function name.
- Set block parameter **Function visibility** to `scoped`.

In the example, the function name for the Simulink Function blocks in the two instances of the referenced model `rtwdemo_func_dinteg` is specified as `multiinstfunc`.

Configure Function Callers

Configure blocks to call the Simulink Function block. You can call a Simulink Function block from a Function Caller block, Stateflow Chart block, or MATLAB Function block. For this example, use Function Caller blocks. For each of the blocks, set a value for the block parameter **Function prototype**. Start typing a prototype. For example, type `y`. Prototype options based on function definitions in the model appear in a selection list. Select the prototype that corresponds to each function call.

For this example, the prototypes are configured as follows:

- In the function-call subsystem, the prototypes for the function callers are configured as `y = Instance1.multiinstfunc(u)` and `y = Instance2.multiinstfunc(u)`. The `Instance` prefix identifies each function invocation uniquely and associates the invocation with its own data set.
- The function caller in function `multiinstfunc` is configured with the prototype `y = subsys_calc.func_calc(u)`. The prefix `subsys_calc` identifies the subsystem that contains the function definition.

For the example, the input and output argument specifications and sample time retain default settings.

Subsystem Configuration

Configuration changes are not required for the subsystem in the example model. When you include a Simulink Function block in a subsystem, the code generator

- Scopes the function to the model that includes the subsystem.
- Treats the subsystem as an atomic unit.

Configure Referenced Model

Configure the referenced model that includes a Simulink Function block:

- In the Block Parameters dialog box, set parameter **Model name** to the referenced model file name. For this example, the model name is `rtwdemo_func_dinteg.slx`.
- Set model configuration parameter **Total number of instances allowed per top model** to `Multiple`.
- To generate a C++ class interface for the reference model, set model configuration parameter **Language** to `C++` and parameter **Code interface packaging** to `C++ class`.

Optionally, you can customize model entry-point function interfaces. You can specify entry-point function names. For the execution (step) entry-point function you can configure the function name and arguments. Custom entry-point function interfaces can minimize changes to existing external code that you integrate with the generated code. The example uses the default function interfaces. See “Configure Entry-Point Function Interfaces for Simulink Function and Function Caller Blocks” (Embedded Coder).

Configure Top Model

Configure the top model for a model component. If you want the model component (that is, the top model) to be reusable, set model configuration parameter **Code interface packaging** to **Reusable function**. If you are generating C++ code, you can set this parameter to **C++ class**. In either case, also:

- Set model configuration parameter **Multi-instance code error diagnostic** to **Error**.
- Set model configuration parameter **Pass root-level I/O as** to **Part of model data structure**.

Optionally, you can customize model entry-point function interfaces. You can specify entry-point function names. For the execution (step) entry-point function you can configure the function name and arguments. Custom entry-point function interfaces can minimize changes to existing external code that you integrate with the generated code. The example uses the default function interfaces. See “Configure Entry-Point Function Interfaces for Simulink Function and Function Caller Blocks” (Embedded Coder).

Generate and Inspect C Code

Generate C code for the model.

- Function code for multi-instance Simulink Function block

When you place a scoped Simulink Function block in a referenced model that you use multiple times in another model, the code generator places the function code in the *model.c* file for the referenced model. For this example, the code generator places function code for `multiinstfunc` in `slprj/ert/rtwdemo_func_dinteg/rtwdemo_func_dinteg.c`.

```
real_T rtwdemo_func_dinteg_multiinstfunc(RT_MODEL_rtwdemo_func_dinteg_T * const
    rtwdemo_func_dinteg_M, const real_T rtu_u)
{
    real_T rtb_TmpLatchAtInOutputport1;
    real_T rtb_TmpLatchAtIn2Outputport1;
    real_T rty_y_0;

    rtb_TmpLatchAtInOutputport1 =
        *rtwdemo_func_dinteg_M->rtwdemo_func_dintegrtxtInport.rtu_In1;

    rtb_TmpLatchAtIn2Outputport1 =
        *rtwdemo_func_dinteg_M->rtwdemo_func_dintegrtxtInport.rtu_In2;

    *rtwdemo_func_dinteg_M->rtwdemo_func_dintegrtxtOutputport.rty_Out2 = 2.0 *
        rtb_TmpLatchAtInOutputport1;

    rtwdemo_func_dinteg_func_calc(rtwdemo_func_dinteg_M, rtb_TmpLatchAtIn2Outputport1,
        rtwdemo_func_dinteg_M->rtwdemo_func_dintegrtxtOutputport.rty_Out1);

    rty_y_0 = rtwdemo_func_dinteg_M->dwork.DiscreteIntegrator_DSTATE;

    rtwdemo_func_dinteg_M->dwork.DiscreteIntegrator_DSTATE += 0.1 * rtu_u;
    return rty_y_0;
}
```

- Function code for a Simulink Function block defined in a subsystem

The code generator places the function code for a Simulink Function block that you define in a subsystem in the *model.c* file for the model that contains the subsystem. For this example, the code generator places the function code for `func_calc` in `slprj/ert/rtwdemo_func_dinteg/rtwdemo_func_dinteg.c`.


```

void rtwdemo_func_dinteg_func_calc(RT_MODEL_rtwdemo_func_dinteg_T * const
    rtwdemo_func_dinteg_M, real_T rtu_u, real_T *rty_y)
{
    rtwdemo_func_dinteg_M->dwork.calcMem =
        rtwdemo_func_dinteg_M->dwork.UnitDelay_DSTATE;

    *rty_y = rtwdemo_func_dinteg_M->dwork.UnitDelay_DSTATE;

    rtwdemo_func_dinteg_M->dwork.UnitDelay_DSTATE = rtu_u * 0.07;
}

```

- Structure that stores multi-instance data for reusable functions

The code generator uses a structure similar to the real-time model (RT_MODEL) data structure to store the multi-instance data associated with a reusable function. The code generator defines the structure in `slprj/ert/rtwdemo_func_dinteg/rtwdemo_func_dinteg.h`.

```
typedef struct rtwdemo_func_dinteg_tag_RTM RT_MODEL_rtwdemo_func_dinteg_T;
```

- Initialization code for multi-instance referenced model

For each instance of a referenced model that includes the same scoped Simulink Function block, the code generator produces initialization and startup function code. A single copy of the initialization code is defined in `slprj/ert/rtwdemo_func_dinteg/rtwdemo_func_dinteg.c`.

```

void rtwdemo_func_dinteg_Start(RT_MODEL_rtwdemo_func_dinteg_T *const
    rtwdemo_func_dinteg_M, const real_T *rtu_In1, const real_T *rtu_In2, real_T
    *rty_Out2, real_T *rty_Out1)
{
    rtwdemo_func_dinteg_M->rtwdemo_func_dintegrtxtInport.rtu_In1 = rtu_In1;
    rtwdemo_func_dinteg_M->rtwdemo_func_dintegrtxtInport.rtu_In2 = rtu_In2;
    rtwdemo_func_dinteg_M->rtwdemo_func_dintegrtxtOutport.rty_Out2 = rty_Out2;
    rtwdemo_func_dinteg_M->rtwdemo_func_dintegrtxtOutport.rty_Out1 = rty_Out1;
}

void rtwdemo_func_dinteg_initialize(const char_T **rt_errorStatus,
    RT_MODEL_rtwdemo_func_dinteg_T *const rtwdemo_func_dinteg_M)
{
    {
        rtmSetErrorStatusPointer(rtwdemo_func_dinteg_M, rt_errorStatus);
    }
}

```

The initialization code is called for each instance of a referenced model that contains the same Simulink Function block. That code is in file `rtwdemo_comp.c` in the build folder.

```

.
.
.
rtwdemo_func_dinteg_initialize(rtmGetErrorStatusPointer(rtwdemo_comp_M),
    (&(rtwdemo_comp_M->Instance1)));

rtwdemo_func_dinteg_initialize(rtmGetErrorStatusPointer(rtwdemo_comp_M),
    (&(rtwdemo_comp_M->Instance2)));

rtwdemo_func_dinteg_Start((&(rtwdemo_comp_M->Instance1)), &rtU->In2, &rtU->In3,
    &rtY->Out2, &rtY->Out3);

rtwdemo_func_dinteg_Start((&(rtwdemo_comp_M->Instance2)), &rtU->In4, &rtU->In5,
    &rtY->Out4, &rtY->Out5);
}

```

- Top model entry-point function declarations

The model header file `rtwdemo_comp.h` includes extern declarations for top model initialize, terminate, and execution (run) entry-point functions.

```
extern void rtwdemo_comp_initialize(RT_MODEL_rtwdemo_comp_T *const
    rtwdemo_comp_M);
```

```
extern void Run(RT_MODEL_rtwdemo_comp_T *const rtwdemo_comp_M);
```

- Reference model entry-point function declarations

Header file `slprj/ert/rtwdemo_func_dinteg/rtwdemo_func_dinteg.h` includes extern declarations for reference model entry-point functions.

```
extern void rtwdemo_func_dinteg_initialize(const char_T **rt_errorStatus,
    RT_MODEL_rtwdemo_func_dinteg_T *const rtwdemo_func_dinteg_M);
extern real_T rtwdemo_func_dinteg_multiinstfunc(RT_MODEL_rtwdemo_func_dinteg_T *
    const rtwdemo_func_dinteg_M, const real_T rtu_u);
extern void rtwdemo_func_dinteg_Start(RT_MODEL_rtwdemo_func_dinteg_T *const
    rtwdemo_func_dinteg_M, const real_T *rtu_In1, const real_T *rtu_In2, real_T
    *rty_Out2, real_T *rty_Out1);
```

Generate and Inspect C++ Code

Generate C++ code for the model.

- Function code for multi-instance Simulink Function block

When you place a scoped Simulink Function block in a referenced model that you use multiple times in another model, the code generator places the function code in the `model.cpp` file for the referenced model. For this example, the code generator places function code for `multiinstfunc` in `slprj/ert/rtwdemo_func_dinteg/rtwdemo_func_dinteg_cpp.cpp`.

```
real_T rtwdemo_func_dintegModelClass::multiinstfunc(const real_T rtu_u)
{
    real_T rtb_TmpLatchAtInOutputport1;
    real_T rtb_TmpLatchAtIn2Outputport1;
    real_T rty_y_0;

    rtb_TmpLatchAtInOutputport1 = *rtwdemo_func_dinteg_cprtrtu_In1;

    rtb_TmpLatchAtIn2Outputport1 = *rtwdemo_func_dinteg_cprtrtu_In2;

    *rtwdemo_func_dinteg_crtrty_Out2 = 2.0 * rtb_TmpLatchAtInOutputport1;

    rtwdemo_func_dinteg_c_func_calc(rtb_TmpLatchAtIn2Outputport1,
        rtwdemo_func_dinteg_crtrty_Out1);

    rty_y_0 = rtwdemo_func_dinteg_cpprtDW.DiscreteIntegrator_DSTATE;

    rtwdemo_func_dinteg_cpprtDW.DiscreteIntegrator_DSTATE += 0.1 * rtu_u;
    return rty_y_0;
}
```

- Function code for a Simulink Function block defined in a subsystem

The code generator places the function code for a Simulink Function block that you define in a subsystem in `model.cpp` for the model that contains the subsystem. For this example, the code generator places the function code for `func_calc` in `slprj/ert/rtwdemo_func_dinteg/rtwdemo_func_dinteg_cpp.cpp`.

```
void rtwdemo_func_dintegModelClass::rtwdemo_func_dinteg_c_func_calc(real_T rtu_u,
    real_T *rty_y)
{
    rtwdemo_func_dinteg_cpprtDW.calcMem =
        rtwdemo_func_dinteg_cpprtDW.UnitDelay_DSTATE;

    *rty_y = rtwdemo_func_dinteg_cpprtDW.UnitDelay_DSTATE;

    rtwdemo_func_dinteg_cpprtDW.UnitDelay_DSTATE = rtu_u * 0.07;
}
```

- Structure that stores multi-instance data for reusable functions

The code generator uses a structure similar to the real-time model (RT_MODEL) data structure to store the multi-instance data associated with a Reusable function. The code generator defines the structure in `slprj/ert/rtwdemo_func_dinteg/rtwdemo_func_dinteg_cpp.h`.

```
typedef struct rtwdemo_func_dinteg_cpp_tag_RTM rtwdemo_func_dinteg_cp_RTM_MODEL;
```

- Initialization code for multi-instance referenced model

For each instance of a referenced model that includes the same scoped Simulink Function block, the code generator produces initialization and startup function code. A single copy of the initialization code is defined in `slprj/ert/rtwdemo_func_dinteg/rtwdemo_func_dinteg_cpp.cpp`.

```
void rtwdemo_func_dintegModelClass::start(real_T *rtu_In1, real_T *rtu_In2,
    real_T *rty_Out2, real_T *rty_Out1)
{
    rtwdemo_func_dinteg_cp_rtrtu_In1 = rtu_In1;
    rtwdemo_func_dinteg_cp_rtrtu_In2 = rtu_In2;
    rtwdemo_func_dinteg_cr_rty_Out2 = rty_Out2;
    rtwdemo_func_dinteg_cr_rty_Out1 = rty_Out1;
}
.
.
.
void rtwdemo_func_dintegModelClass::initializeRTM()
{
    (void) memset((void *)(&rtwdemo_func_dinteg_cp_rtm), 0,
        sizeof(rtwdemo_func_dinteg_cp_RTM_MODEL));
}
}
```

The initialization code is called for each instance of a referenced model that contains the same Simulink Function block. That code is in file `rtwdemo_comp_cpp.cpp` in the build folder.

```
.
.
.
```

```
Instance1MDL0BJ0.initializeRTM();

Instance1MDL0BJ0.setErrorStatusPointer(rtmGetErrorStatusPointer((&rtM)));
Instance1MDL0BJ0.initialize();

Instance2MDL0BJ1.initializeRTM();

Instance2MDL0BJ1.setErrorStatusPointer(rtmGetErrorStatusPointer((&rtM)));

Instance1MDL0BJ0.start(&rtU.In2, &rtU.In3, &rtY.Out2, &rtY.Out3);

Instance2MDL0BJ1.start(&rtwU.In4, &rtwU.In5, &rtY.Out4, &rtY.Out5);
}
```

- Top model class declaration

The model header file `rtwdemo_comp_cpp.h` includes the class declaration for the top model.

```
class rtwdemo_compModelClass {
public:
    ExtU rtU;

    ExtY rtY;

    void initialize();

    void Run();

    rtwdemo_compModelClass();
}
```

```

~rtwdemo_compModelClass();

RT_MODEL * getRTM();

private:
RT_MODEL rtM;

rtwdemo_func_dintegModelClass Instance1MDL0BJ0;

rtwdemo_func_dintegModelClass Instance2MDL0BJ1;
};

```

- Reference model class declaration

Header file `s\prj\ert\rtwdemo_func_dinteg\rtwdemo_func_dinteg_cpp.h` includes the class declaration for the reference model.

```

class rtwdemo_func_dintegModelClass {
public:
real_T multiinstfunc(const real_T rtu_u);

void start(real_T *rtu_In1, real_T *rtu_In2, real_T *rty_Out2, real_T
*rty_Out1);

rtwdemo_func_dintegModelClass();
~rtwdemo_func_dintegModelClass();

rtwdemo_func_dinteg_cp_RT_MODEL * getRTM();

void initializeRTM();

void setErrorStatusPointer(const char_T **rt_errorStatus);

private:
rtwdemo_func_dinteg_cpp_DW rtwdemo_func_dinteg_cpprtDW;

const real_T *rtwdemo_func_dinteg_cprrtu_In1;
const real_T *rtwdemo_func_dinteg_cprrtu_In2;
real_T *rtwdemo_func_dinteg_crrty_Out2;
real_T *rtwdemo_func_dinteg_crrty_Out1;

rtwdemo_func_dinteg_cp_RT_MODEL rtwdemo_func_dinteg_cpprtM;

void rtwdemo_func_dinteg_c_func_calc(real_T rtu_u, real_T *rty_y);
};

```

Limitations

These code generation limitations apply to export-function models that include multiple instances of a Simulink Function block.

- You must set the model configuration parameter **Pass root-level I/O as** to **Part of model data structure**.
- Generated code is compatible with single-threaded execution only. To avoid race conditions for shared signal data, invoke instances of the function code from the same execution thread.
- You cannot:
 - Generate code if the export-function model includes model variants.
 - Enable the external mode data exchange interface.

See Also

More About

- “Simulink Functions Overview”
- “Simulink Functions”
- “Simulink Function Blocks in Referenced Models”
- “Storage Classes and Reentrant, Multi-Instance Models and Components” on page 16-8
- “Simulink Function Blocks and Code Generation” on page 10-6

Generate Reentrant Code from Subsystems

You can reduce the amount of code that the code generator produces for identical atomic subsystems by configuring the subsystems as reusable functions that pass data as arguments (for example, `rtB_*` for block input and output, `rtDW_*` for continuous states, and `rtP_*` for parameters). By default, the code generator produces subsystem code that communicates with other code by sharing access to global data structures that reside in shared memory. By passing data as arguments, the code can be reentrant. Each instance of the code maintains its own unique data.

To configure subsystems for reusability and reentrancy, configure mask and block parameters of the identical subsystems the same way. The code generator performs a checksum to determine whether subsystems are identical and whether the code is reusable. If the code is reusable, the code generator produces a single instance of reusable, reentrant function code.

To configure the subsystem block parameters:

- 1 If a subsystem is virtual, select block parameter “Treat as atomic unit” to enable function packaging parameters.
- 2 Set block parameter “Function packaging” to `Reusable function`. The code generator produces a separate function with arguments for each subsystem. Selecting `Reusable function` also enables additional parameters that you can use to control the names of the function and file that the code generator produces for the subsystem code.
- 3 Select a value for block parameter “Function name options”. To generate reusable, reentrant code, specify the same setting for identical subsystems.
 - To let the code generator determine the function name, select `Auto`.
 - To use the subsystem name or, for a library block, the name of the library block, select `Use subsystem name`.
 - To display block parameter “Function name” and enter a unique, valid C/C++ function name, select `User specified`.

If multiple instances of an identical subsystem exist within a model reference hierarchy, select `Auto`.

If you are using Embedded Coder, you can use identifier format controls. See “Identifier Format Control” (Embedded Coder).

- 4 Set “File name options”. To generate reusable, reentrant code, specify the same setting for identical subsystems.
 - To let the code generator determine file naming select `Auto`.
 - To use the subsystem name or, for a library block, the name of the library block select `Use subsystem name`.
 - To use the function name, as specified by block parameter **Function name options** select `Use function name`.
 - To display block parameter “File name (no extension)” and enter a file name, excluding the extension (for example, `.c` or `.cpp`) select `User specified`.

Other considerations:

- If multiple instances of an identical subsystem exist within a model reference hierarchy, select `Auto`.

- If the code generator does not generate a separate file for a subsystem, the function code is placed in the file generated from the subsystem's parent system. If the parent is the model itself, the code generator places the function code in *model.c* or *model.cpp*.
- If your generated code is under source control, specify a value other than **Auto**. This specification prevents the generated file name from changing when you modify and rebuild the model.
- If you select **Use subsystem name**, the code generator mangles the subsystem file name if the model contains Model blocks, or if a model reference target is being generated for the model. In these situations, the code generator uses the file name *modelsubsystem.c*.

If subsystem A has mask parameter b and K and subsystem B has mask parameters c and K, code reuse is not possible. The code generator produces separate functions for subsystems A and B. If you set block parameters for subsystems A and B differently, code reuse is not possible.

Function Reuse in Generated Code

This example shows how to configure an atomic subsystem for generating reusable code. To specify that the code generated for a subsystem execute as an atomic unit, on the Block Parameters dialog box, select the **Treat as atomic unit** parameter. That parameter enables the **Function Packaging** parameter on the **Code Generation** tab. The **Function Packaging** parameter has these four settings:

- **Inline**: Inline the subsystem code
- **Nonreusable function**: Function with I/O passed as global data
- **Reusable function**: Function with I/O passed as function arguments
- **Auto**: Let Simulink Coder optimize based on context

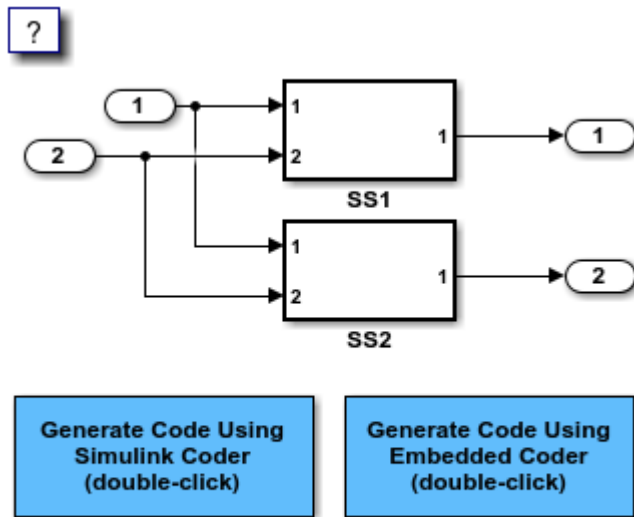
The **Reusable function** and **Auto** settings permit the code generator to reuse subsystem code. The **Reusable function** and **Nonreusable function** settings enable the **Function name options**, **Function name**, and **File name options** parameters.

If you have an Embedded Coder® license, you can configure a nonreusable subsystem to accept arguments.

Example Model

The `rtwdemo_ssreuse` model contains two identical subsystems, SS1 and SS2. For these subsystems, the **Function packaging** parameter is set to **Reusable function**, and the **Function name** parameter is `myfun`. The subsystems are parameterized masked subsystems. To see the contents of the masked subsystems, right-click the subsystem blocks and select **Mask > Look Under Mask**.

```
model = 'rtwdemo_ssreuse';
open_system(model);
```



Copyright 1994-2020 The MathWorks, Inc.

Generate and Inspect Code

Create a temporary folder for the build and inspection process. Open the **Simulink Coder** or **Embedded Coder** app. Then, generate and inspect the code.

```
currentDir=pwd;
[~,cgDir]=rtwdemodir();
slbuild(model)

### Starting build procedure for: rtwdemo_ssreuse
### Successful completion of build procedure for: rtwdemo_ssreuse

Build Summary

Top model targets built:

Model          Action          Rebuild Reason
=====
rtwdemo_ssreuse Code generated and compiled. Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 12.727s

cfile=fullfile(cgDir, 'rtwdemo_ssreuse_grt_rtw', 'rtwdemo_ssreuse.c');
rtwdemodbtype(cfile, '/* Model step', '/* Model initialize', 1, 0);

/* Model step function */
void rtwdemo_ssreuse_step(void)
{
    /* Outputs for Atomic SubSystem: '<Root>/SS1' */

    /* Inport: '<Root>/In1' incorporates:
     * Inport: '<Root>/In2'
     */
}
```



```

myfun(rtwdemo_ssreuse_U.In1, rtwdemo_ssreuse_U.In2, rtwdemo_ssreuse_P.T1Data,
      rtwdemo_ssreuse_P.T1Break, &rtwdemo_ssreuse_B.SS1);

/* End of Outputs for SubSystem: '<Root>/SS1' */

/* Output: '<Root>/Out1' */
rtwdemo_ssreuse_Y.Out1 = rtwdemo_ssreuse_B.SS1.LookupTable;

/* Outputs for Atomic SubSystem: '<Root>/SS2' */

/* Inport: '<Root>/In1' incorporates:
 * Inport: '<Root>/In2'
 */
myfun(rtwdemo_ssreuse_U.In1, rtwdemo_ssreuse_U.In2, rtwdemo_ssreuse_P.T2Data,
      rtwdemo_ssreuse_P.T2Break, &rtwdemo_ssreuse_B.SS2);

/* End of Outputs for SubSystem: '<Root>/SS2' */

/* Output: '<Root>/Out2' */
rtwdemo_ssreuse_Y.Out2 = rtwdemo_ssreuse_B.SS2.LookupTable;
}

```

In the model step function, there are two calls to the reusable function, `myfun`. The mask parameters, `T1Break`, `T1Data`, `T2Break`, and `T2Data`, are function arguments.

Change the **Function Packaging** parameter to **Inline**.

```

set_param('rtwdemo_ssreuse/SS1','RTWSystemCode','Inline')
set_param('rtwdemo_ssreuse/SS2','RTWSystemCode','Inline')

```

Generate and inspect the code.

```

slbuild(model)

### Starting build procedure for: rtwdemo_ssreuse
### Successful completion of build procedure for: rtwdemo_ssreuse

Build Summary

Top model targets built:

Model          Action          Rebuild Reason
=====
rtwdemo_ssreuse Code generated and compiled.  Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 10.379s

cfile=fullfile(cgDir, 'rtwdemo_ssreuse_grt_rtw', 'rtwdemo_ssreuse.c');
rtwdemodbtype(cfile, '/* Model step', '/* Model initialize', 1, 0);

/* Model step function */
void rtwdemo_ssreuse_step(void)
{
    real_T Out1_tmp;

    /* Outputs for Atomic SubSystem: '<Root>/SS2' */
    /* Outputs for Atomic SubSystem: '<Root>/SS1' */

```

```

/* Sum: '<S1>/Sum' incorporates:
 * Inport: '<Root>/In1'
 * Inport: '<Root>/In2'
 * Sum: '<S2>/Sum'
 */
Out1_tmp = rtwdemo_ssreuse_U.In1 + rtwdemo_ssreuse_U.In2;

/* End of Outputs for SubSystem: '<Root>/SS2' */

/* Output: '<Root>/Out1' incorporates:
 * Lookup_n-D: '<S1>/Lookup Table'
 * Sum: '<S1>/Sum'
 */
rtwdemo_ssreuse_Y.Out1 = look1_binlx(Out1_tmp, rtwdemo_ssreuse_P.T1Break,
    rtwdemo_ssreuse_P.T1Data, 10U);

/* End of Outputs for SubSystem: '<Root>/SS1' */

/* Outputs for Atomic SubSystem: '<Root>/SS2' */
/* Output: '<Root>/Out2' incorporates:
 * Lookup_n-D: '<S2>/Lookup Table'
 */
rtwdemo_ssreuse_Y.Out2 = look1_binlx(Out1_tmp, rtwdemo_ssreuse_P.T2Break,
    rtwdemo_ssreuse_P.T2Data, 10U);

/* End of Outputs for SubSystem: '<Root>/SS2' */
}

```

In the model step function, the subsystem code is inlined.

Change the **Function Packaging** parameter to **Nonreusable function**. For SS2, change the **Function name** parameter to **myfun2**.

```

set_param('rtwdemo_ssreuse/SS1','RTWSystemCode','Nonreusable function')
set_param('rtwdemo_ssreuse/SS2','RTWSystemCode','Nonreusable function')
set_param('rtwdemo_ssreuse/SS2','RTWFcnName','myfun2')

```

Generate and inspect the code.

```

slbuild(model)

### Starting build procedure for: rtwdemo_ssreuse
### Successful completion of build procedure for: rtwdemo_ssreuse

Build Summary

Top model targets built:

Model                Action                Rebuild Reason
=====
rtwdemo_ssreuse     Code generated and compiled.  Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.175s

cfile=fullfile(cgDir, 'rtwdemo_ssreuse_grt_rtw', 'rtwdemo_ssreuse.c');
rtwdemodbtype(cfile, '/* Model step', '/* Model initialize', 1, 0);

/* Model step function */

```

```

void rtwdemo_ssreuse_step(void)
{
    /* Outputs for Atomic SubSystem: '<Root>/SS1' */
    myfun();

    /* End of Outputs for SubSystem: '<Root>/SS1' */

    /* Outputs for Atomic SubSystem: '<Root>/SS2' */
    myfun2();

    /* End of Outputs for SubSystem: '<Root>/SS2' */
}

```

The model step function contains calls to the functions `myfun` and `myfun2`. These functions have a void-void interface.

Change the **Function Packaging** parameter to **Auto**.

```

set_param('rtwdemo_ssreuse/SS1', 'RTWSystemCode', 'Auto')
set_param('rtwdemo_ssreuse/SS2', 'RTWSystemCode', 'Auto')

```

For the `auto` setting, Simulink Coder chooses the optimal format. For this model, the optimal format is a reusable function.

Close the model and clean up.

```

bdclose(model)
rtwdemoclean;
cd(currentDir)

```

Limitations

Code reuse for subsystems is an optimization feature and is not guaranteed. The following list of limitations on code reuse in subsystems is nonexhaustive.

- The code generator uses a checksum to determine whether subsystems are identical and reusable. Subsystem code is not reused if:
 - In blocks and data objects, you use symbols to specify dimensions.
 - A port has different sample times, data types, complexity, frame status, or dimensions across subsystems.
 - The output of a subsystem is marked as a global signal.
 - Subsystems contain identical blocks with different names or parameter settings.
 - The output of a subsystem is connected to a Merge block, and the output of the Merge block is configured with a custom storage class that is implemented in the C code as nonaddressable memory (for example, `BitField`).
 - The input of a subsystem is nonscalar and is configured with a custom storage class that is implemented in the C code as nonaddressable memory.
 - A masked subsystem has a parameter that is nonscalar and is configured with a custom storage class that is implemented in the C code as nonaddressable memory.
 - A function-call subsystem uses mask parameters when you set the model configuration parameter `Default parameter behavior` to `Tunable`. To reuse the masked function-call

subsystem, place it inside a new atomic subsystem without a mask. Then move the Trigger block from the masked subsystem into the atomic subsystem.

- A block in a subsystem uses a partially tunable expression. Some partially tunable expressions disable code reuse.

Partially tunable expressions are expressions that contain one or more tunable variables and an expression that is not tunable. For example, suppose that you create the tunable variable *K* with value 15.23 and the tunable variable *P* with value [5;7;9]. The expression *K+P'* is a partially tunable expression because the expression *P'* is not tunable. For more information about tunable expression limitations, see “Tunable Expression Limitations” on page 15-122.

- For subsystems that contain S-function blocks that are reusable, the blocks must meet the requirements listed in “S-Functions for Code Reuse” on page 14-81.
- If you select **Reusable function**, and the code generator determines that you cannot reuse the code for a subsystem, it generates a separate function that is not reused. The code generation report might show that the separate function is reusable, even if only one subsystem uses it. If you prefer that the code generator inline subsystem code in such cases (rather than deployed as functions), set “Function packaging” to **Auto**.
- If a reusable subsystem uses a shared local data store and you configure default mapping for model data elements, leave the default storage class mapping for category **Shared local data stores** set to **Default**.
- Use of these blocks in a subsystem can prevent the subsystem code from being reused:
 - Scope blocks (with data logging enabled)
 - S-Function blocks that fail to meet certain criteria (see “S-Functions for Code Reuse” on page 14-81)
 - To File blocks (with data logging enabled)
 - To Workspace blocks (with data logging enabled)
- For reusable library subsystems (subsystems shared across reference models), the code generator uses a checksum to determine whether subsystems are identical. The code generator places the reusable library subsystem code in the shared utilities folder, and the reusable code is independent of the generated code of the top model or the reference model. For example, the reusable library subsystem code does not include *model.h* or *model_types.h*.

Reusable code that the code generator places in the shared utilities folder and is dependent on the model code does not compile. If the code generator determines that the reusable library subsystem code is dependent on the model code, the reusable subsystem code is not placed in the shared utilities folder. The code generator produces code that is dependent on the model code when the reusable library subsystem:

- Contains a block that uses time-related functionality, such as a Step block, or continuous time or multirate blocks.
- Contains one or more Model blocks.
- Contains a subsystem that is not inlined or a reusable library subsystem.
- Contains a signal or parameter with a storage class that places the data declaration in a file under *model_ert_rtw* (For example, **ExportedGlobal** storage class places data in *model.c*)
- Contains a user-defined storage class like **Enumeration**, **Simulink.Signal**, **Simulink.Parameter**, and so on where **Data Scope** is not set to **Exported**. The code generator might place the type definition in *model_types.h*.

- Is a variant subsystem that generates preprocessor conditionals. The code generator places preprocessor directives that define the variant objects in *model_types.h*
- You cannot reuse a subsystem if there are multiple instances of the same subsystem in the model and one subsystem is fed by a Constant block but the other subsystem is not. To work around this limitation, insert a Signal Conversion block before the subsystems.
- Two reusable subsystems that have different checksums cannot share the same function in the generated code. When you specify the same function name for both subsystems in **Block Parameters (Subsystem) > Code Generation > Function name:**
 - If the subsystems are library-linked, the code generator mangles the function name to generate unique function names. The code generator uses the subsystem naming rules to generate the function names.
 - If the subsystems are not library-linked, the code generator issues an error.

The preceding limitations also apply for library-based code generation. For more information, see “Library-Based Code Generation for Reusable Library Subsystems” (Embedded Coder).

See Also

More About

- “What Is Reentrant Code?” (Embedded Coder)
- “Generate Reusable Code from Library Subsystems Shared Across Models” (Embedded Coder)
- “Generate Subsystem Code as Separate Function and Files” on page 7-7
- “Storage Classes and Reentrant, Multi-Instance Models and Components” on page 16-8

Generate Reusable Code from Library Subsystems Shared Across Models

What Is a Reusable Library Subsystem?

A reusable library subsystem is a subsystem included in a library that is configured for reuse. You must define a subsystem in a library and configure it for reuse to reuse the subsystem across models.

To reuse common functionality, you can include multiple instances of a subsystem:

- Within a single model, which is a top model or part of model reference hierarchy
- Across multiple referenced models in a model reference hierarchy
- Across multiple top models that contain Model blocks
- Across multiple top models that do not include Model blocks

The code generator uses checksums to determine reusability. There are cases when the code generator cannot reuse subsystem code.

For incremental code generation, if the reusable library subsystem changes, a rebuild of itself and its parents occurs. During the build, if a matching function is not found, a new instance of the reusable function is generated into the shared utilities folder. If a different matching function is found from previous builds, that function is used, and a new reusable function is not emitted.

For subsequent builds, unused files are not replaced or deleted from your folder. During development of a model, when many obsolete shared functions exist in the shared utilities folder, you can delete the folder and regenerate the code. If all instances of a reusable library subsystem are removed from a model reference hierarchy and you regenerate the code, the obsolete shared functions remain in the shared utilities folder until you delete them.

If a model changes such that the change might cause different generated code for the subsystem, a new reusable function is generated. For example, model configuration parameters that modify code comments might cause different generated code for the subsystem even if the reusable library subsystem did not change.

Embedded Coder users can generate code from a library that contains subsystems are configured for reuse. For more information, see “Library-Based Code Generation for Reusable Library Subsystems” (Embedded Coder).

Reusable Library Subsystem Code Placement and Naming

The generated code of a reusable library subsystem is independent of the generated code of the model. Code for the reusable library subsystem is generated to the shared utility folder, `s\prj/target/_sharedutils`, instead of the model reference hierarchy folders. The generated code for the supporting types, which are generated to the `.h` file, are also in the shared utilities folder.

For unique naming, reusable function names have a checksum appended to the reusable library subsystem name. For example, the code and files for a subsystem, `SS1`, which links to a reusable library subsystem, `RLS`, might be:

- Function name: `RLS_mgdjlngd`

- File name: RLS_mgdj_lnd.c and RLS_mgdj_lnd.h

To configure a reusable library subsystem, set the Subsystem parameters as listed here:

- Select **Treat as an atomic unit**.
- On the **Code Generation** tab:
 - Set **Function packaging** to Reusable function.
 - Set the **Function name options** and **File name options** parameters based on this table. This table summarizes how the code generator produces code and highlights the **File name options** in bold that enable you to generate code from reusable library subsystems. For the purpose of this table, assume that the Subsystem block is named mySubsystem and the File name value for the User specified option is myFunction.

Function Name Options	File Name Options	Generated Folder Location	Generated Function Name	Generated File Names
Auto	Auto	/slprj/ target/ _sharedutils	mySubsystem_ checksum	mySubsystem_ checksum.c, mySubsystem_ checksum.h
	Use subsystem name	/slprj/ target/ _sharedutils	mySubsystem_ checksum	mySubsystem_ checksum.c, mySubsystem_ checksum.h
	Use function name	/slprj/ target/ _sharedutils	mySubsystem_ checksum	mySubsystem_ checksum.c, mySubsystem_ checksum.h
	User specified	model_target _rtw	model_Subsys tem	myFile.h
Use subsystem name	Auto	model_target _rtw	model_Subsys tem	Generated code included in model.h
	Use subsystem name	/slprj/ target/ _sharedutils	mySubsystem_ checksum	mySubsystem_ checksum.c, mySubsystem_ checksum.h
	Use function name	/slprj/ target/ _sharedutils	mySubsystem_ checksum	mySubsystem_ checksum.c, mySubsystem_ checksum.h
	User specified	model_target _rtw	model_Subsys tem	myFile.h

Function Name Options	File Name Options	Generated Folder Location	Generated Function Name	Generated File Names
User specified	Auto (This specification is supported for one unique instance of the subsystem in the model.)	/slprj/ target/ _sharedutils	myFunction	myFunction.c , myFunction.h
	Use subsystem name	model_target _rtw	myFunction	mySubsystem.c, mySubsystem.h
	Use function name (This specification is supported for one unique instance of the subsystem in the model.)	/slprj/ target/ _sharedutils	myFunction	myFunction.c , myFunction.h
	User specified (This specification is supported if the user-specified function name and file name are same and only for one unique instance of the subsystem in the model.)	/slprj/ target/ _sharedutils	myFunction	myFunction.c , myFunction.h

For more information about the Subsystem block parameters, see Subsystem.

Configure Models That Include Reusable Library Subsystems

For a model to use a reusable library subsystem, you must configure the model differently depending on whether the model is a reference model or top model. If the subsystem is in a referenced model hierarchy, set the configuration parameter, “Shared code placement” to **Auto**. Otherwise, for each model that uses the subsystem, set the model configuration parameter **Shared code placement** to **Shared location**.

If a reusable library subsystem uses a shared local data store and you configure default mapping for model data elements, leave the default storage class mapping for category **Shared local data stores** set to **Default**.

In a model reference hierarchy, if an instance of the reusable library subsystem is in the top model, then on the **Model Referencing** pane of the Configuration Parameters dialog box, you must select the **Pass fixed-size scalar root input by value for code generation** parameter. If you do not select the parameter, a separate shared function is generated for the reusable library subsystem instance in the top model, and a reusable function is generated for instances in the referenced models.

If a reusable library subsystem is connected to the root Output, reuse does not happen with identical subsystems that are not connected to the root Output. However, you can set **Pass reusable system outputs as** to `Individual` arguments on the **Optimizations** pane to make sure that reuse occurs between these subsystems. This parameter requires an Embedded Coder license.

For more information on creating a library, see “Custom Libraries”. For an example of creating a reusable library subsystem, see “Generate Reusable Code for Subsystems Shared Across Models” on page 10-39.

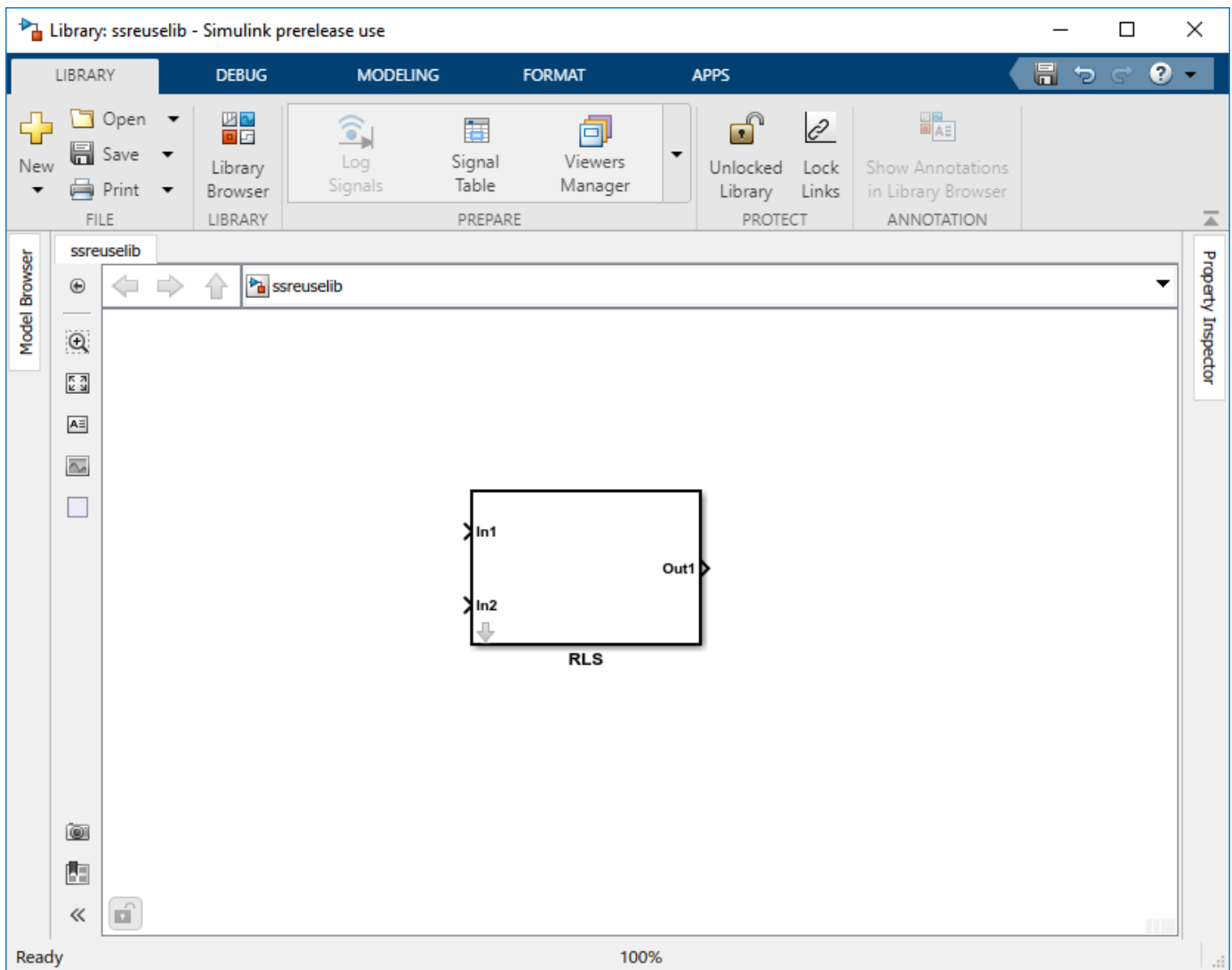
Generate Reusable Code for Subsystems Shared Across Models

This example shows how to configure a reusable library subsystem and generate a reusable function for a subsystem shared across referenced models. The result is reusable code for the subsystem, which is generated to the shared utility folder (`slprj/target/_sharedutils`).

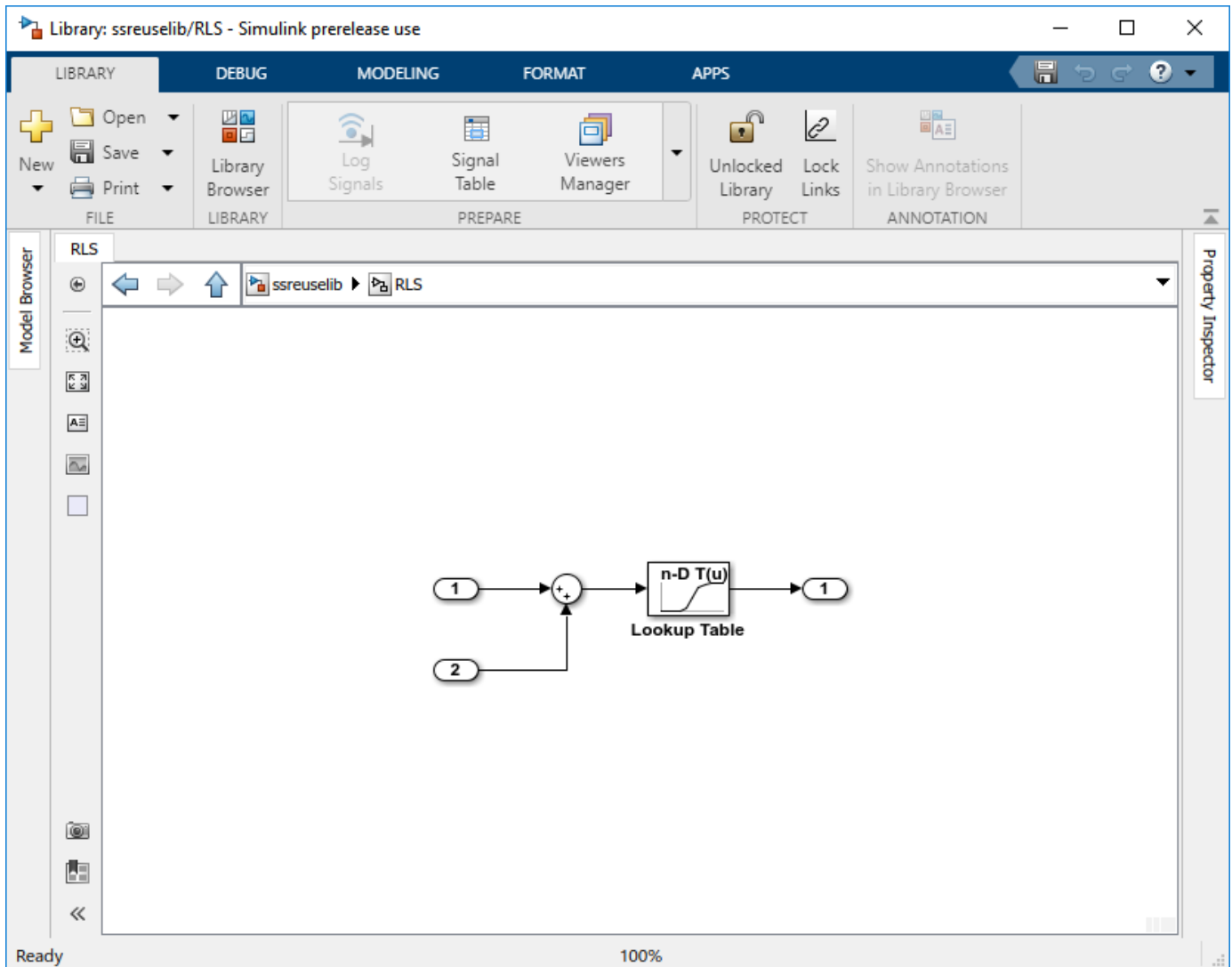
- “Create a reusable library subsystem.” on page 10-39
- “Create the example model.” on page 10-42
- “Set configuration parameters of the top model.” on page 10-43
- “Create and propagate a configuration reference.” on page 10-43
- “Generate and view the code.” on page 10-44

Create a reusable library subsystem.

- 1 In the Simulink Editor, on the **Simulation** tab, select **New > Library**. Select **Blank Library** from the Simulink Start Page window. Open `rtwdemo_ssreuse` to copy and paste subsystem `SS1` into the Library Editor. This action loads the variables for `SS1` into the base workspace. Rename the subsystem block to `RLS`.



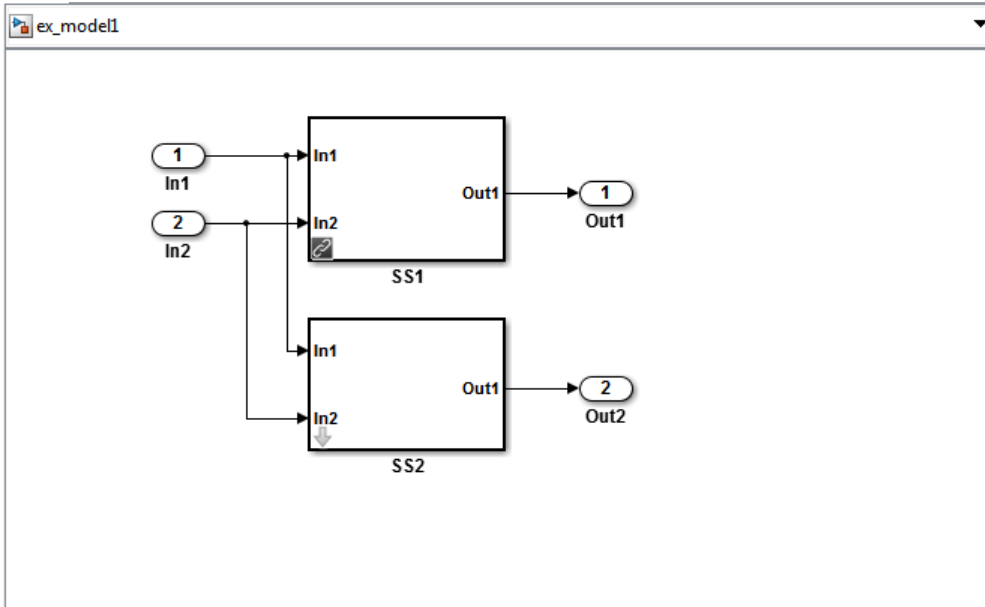
- 2 Click the Subsystem block and press **Ctrl+U** to view the contents of subsystem RLS.



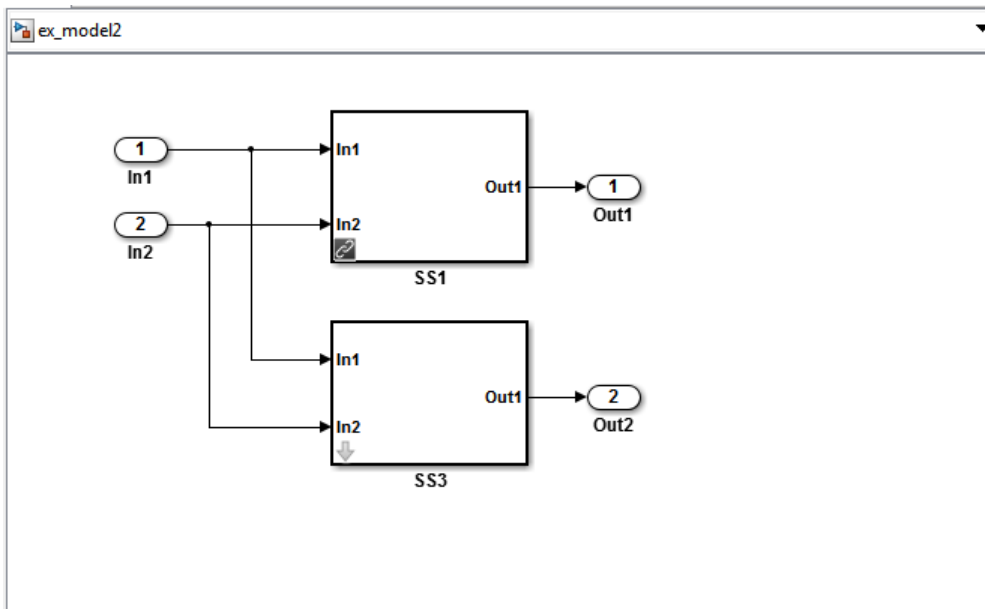
- 3 To configure the subsystem, in the Library editor, right-click RLS. In the context menu, select **Block Parameters (Subsystem)**. In the Subsystem Parameters dialog box, choose the following options:
 - Select **Treat as an atomic unit**.
 - On the **Code Generation** tab:
 - Set **Function packaging** to Reusable function.
 - Set **Function name options** to User specified and verify that the **Function name** is set to myfun.
 - Set **File name options** to Auto.
- 4 Click **Apply** and **OK**.
- 5 Save the reusable library as ssreuselib, which creates a file, ssreuselib.slx.

Create the example model.

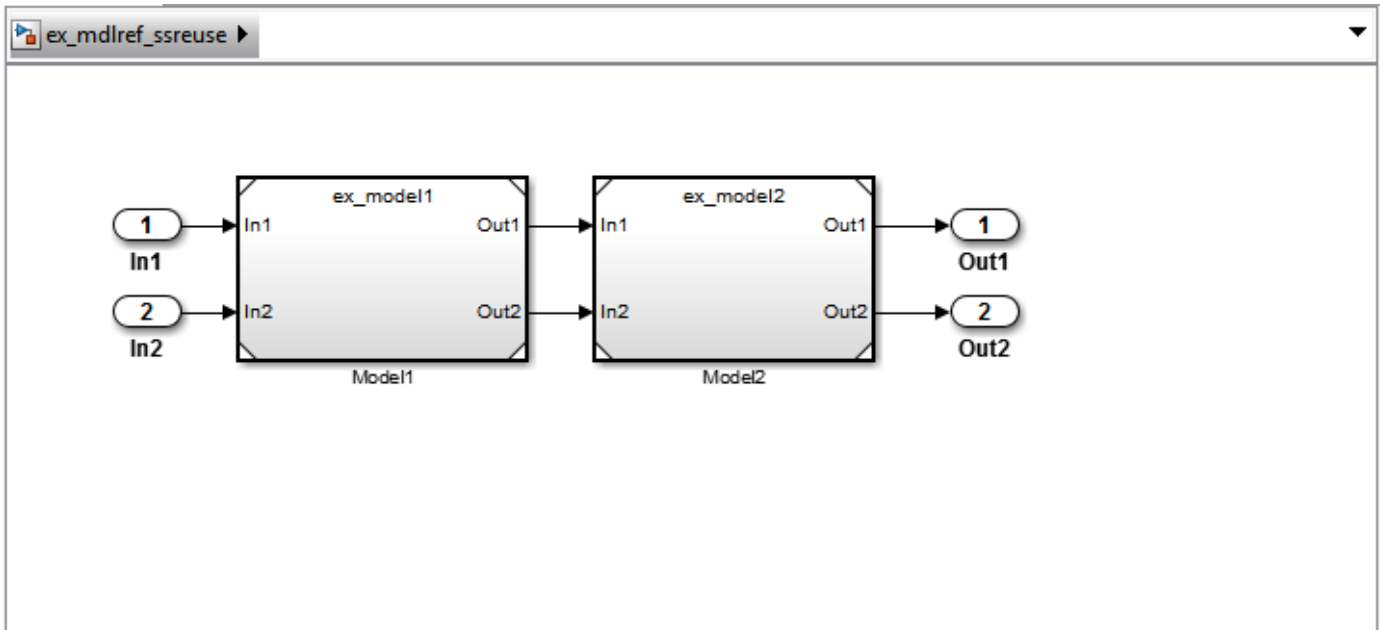
- 1 Create a model which includes one instance of RLS from `ssreuselib`. Name this subsystem SS1. Add another subsystem and name it SS2. Name the model `ex_model1`.



- 2 Create another model which includes one instance of RLS from `ssreuselib`. Name this subsystem SS1. Add another subsystem and name it SS3. Name the model `ex_model2`.



- 3 Create a top model with two model blocks that reference `ex_model1` and `ex_model2`. Save the top model as `ex_md1ref_ssreuse`.



Set configuration parameters of the top model.

- 1 With model `ex_mdhref_ssreuse` open in the Simulink Editor, on the **C Code** tab, click **Settings** to open the Configuration Parameters dialog box.
- 2 On the **Solver** pane, specify the **Type** as Fixed-step.
- 3 On the **Model Referencing** pane, click the ellipsis to view the advanced parameters. Then, select **Pass fixed-size scalar root inputs by value for code generation**.
- 4 On the **Code Generation > Report** pane, select **Create code generation report** and **Open report automatically**.
- 5 On the **Code Generation > Interface** pane, set the “Shared code placement” to Shared location.
- 6 On the **Code Generation > Identifiers** pane, set the **Maximum identifier length** to 256. This step is optional.
- 7 Click **Apply** and **OK**.

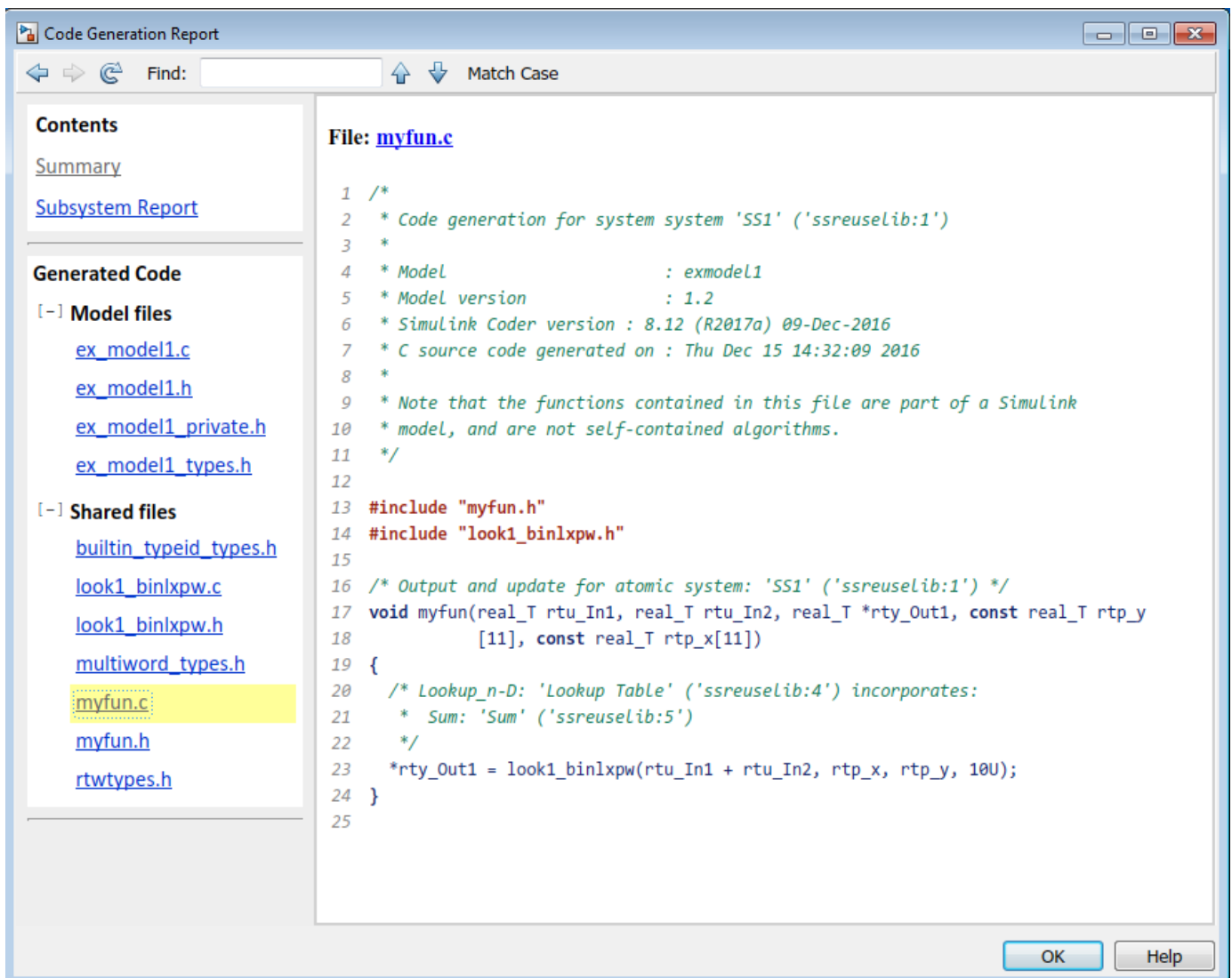
Create and propagate a configuration reference.

- 1 In the Simulink Editor, on the **Modeling** tab, click **Model Explorer** to open the Model Explorer. In the left navigation column of the Model Explorer, expand the `ex_mdhref_ssreuse` node.
- 2 Select the Configurations node below the `ex_mdhref_ssreuse` node. In the **Contents** pane, right-click **Configuration** and select **Convert to Configuration Reference**.
- 3 In the Convert Active Configuration to Reference dialog box, click **OK**. This action converts the model configuration set to a configuration reference, `Simulink.ConfigSetRef`, and creates the configuration reference object, `configSetObj`, in the base workspace.
- 4 In the left navigation column, right-click **Reference (Active)** and select **Propagate to Referenced Models**.
- 5 In the Configuration Reference Propagation to Referenced Models dialog box, select the referenced models in the list. Click **Propagate**.

Now, the top model and referenced models use the same configuration reference, Reference (Active), which points to a model configuration reference object, configSetObj, in the base workspace. When you save your model, you also need to export the configSetObj to a MAT-file.

Generate and view the code.

- 1 To generate code, in the Simulink Editor, press **Ctrl+B**. After the code is generated, the code generation report opens.
- 2 To view the code generation report for a referenced model, in the left navigation pane, in the **Referenced Models** section, select `ex_model1`. The code generation report displays the generated files for `ex_model1`.
- 3 In the left navigation pane, expand the **Shared files**. The code generator uses the reusable library subsystem name. The code for subsystem SS1 is in `myfun.c` and `myfun.h`.



- 4 Click **Back** and navigate to the `ex_model2` code generation report. `ex_model2` uses the same source code, `myfun.c` and `myfun.h`, as the code for `ex_model1`. Your subsystem function and file names will be different.

See Also

More About

- “Custom Libraries”
- “Generate Reentrant Code from Subsystems” (Embedded Coder)
- “Generate Subsystem Code as Separate Function and Files” on page 7-7

Generate Reusable Code from Stateflow Atomic Subcharts

Generate Reusable Code for Linked Atomic Subcharts

To specify code generation parameters for linked atomic subcharts from the same library:

- 1 Open the library model that contains your atomic subchart.
- 2 Unlock the library.
- 3 Right-click the library chart and select **Block Parameters**.
- 4 In the dialog box, specify the following parameters:
 - a On the **Main** tab, select parameter **Treat as atomic unit**.
 - b On the **Code Generation** tab, set parameter **Function packaging** to Reusable function.
 - c Set **File name options** to User specified.
 - d For **File name**, enter the name of the file without an extension.
 - e Click **OK** to apply the changes.
- 5 (OPTIONAL) Customize the generated function names for atomic subcharts:
 - a Set model configuration parameter **System target file** to `ert.tlc`.
 - b For model configuration parameter **Subsystem methods**, specify the format of the function names using a combination of the following naming rule tokens:
 - \$R — root model name
 - \$F — type of interface function for the atomic subchart
 - \$N — block name
 - \$H — subsystem index
 - \$M — name-mangling text
 - c Click **OK** to apply the changes.

When you generate code for your model, a separate file stores the code for linked atomic subcharts from the same library.

Generate Reusable Code for Unlinked Atomic Subcharts

To specify code generation parameters for an unlinked atomic subchart:

- 1 In your chart, right-click the atomic subchart and select **Properties**.
- 2 In the dialog box, specify the following properties:
 - a Set property **Code generation function packaging** to Reusable function.
 - b Set **Code generation file name options** to User specified.
 - c For **Code generation file name**, enter the name of the file without extension.
 - d Click **OK** to apply the changes.
- 3 (OPTIONAL) Customize the generated function names for atomic subcharts:

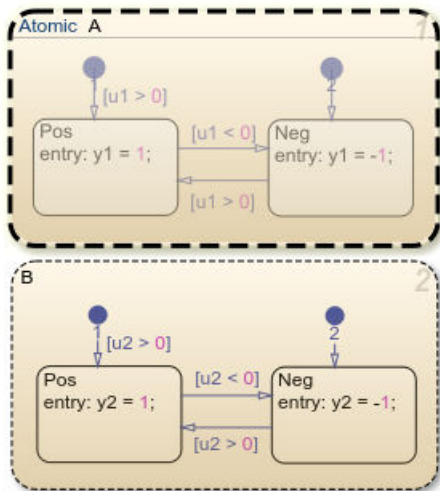
- a Set model configuration parameter **System target file** to `ert.tlc`.
- b For model configuration parameter **Subsystem methods**, specify the format of the function names using a combination of the following naming rule tokens:
 - `$R` — root model name
 - `$F` — type of interface function for the atomic subchart
 - `$N` — block name
 - `$H` — subsystem index
 - `$M` — name-mangling text
- c Click **OK** to apply the changes.

When you generate code for your model, a separate file stores the code for the atomic subchart. For more information, see “Generate Code from Atomic Subcharts” on page 13-5.

Generate Reusable Code for Unit Testing

Convert a State to an Atomic Subchart

To convert state A to an atomic subchart, right-click the state and select **Group & Subchart > Atomic Subchart**. State A changes to an atomic subchart:



Set Up a Standalone C File for the Atomic Subchart

- 1 Open the properties dialog box for A.
- 2 Set property **Code generation function packaging** to Reusable function.
- 3 Set **Code generation file name options** to User specified.
- 4 For **Code generation file name**, enter saturator as the name of the file.
- 5 Click **OK**.

Set Up the Code Generation Report

- 1 Set model configuration parameter **System target file** to `ert.tlc`.

- 2 Select model configuration parameters **Create code generation report** and **Open report automatically**.
- 3 Select parameters **Code-to-model** and **Model-to-code**.
- 4 Click **Apply**.

Customize the Generated Function Names

- 1 Set model configuration parameter **Subsystem methods** to the format scheme $\$R\$N\$M\F , where:
 - $\$R$ is the root model name.
 - $\$N$ is the block name.
 - $\$M$ is the mangle token.
 - $\$F$ is the type of interface function for the atomic subchart.

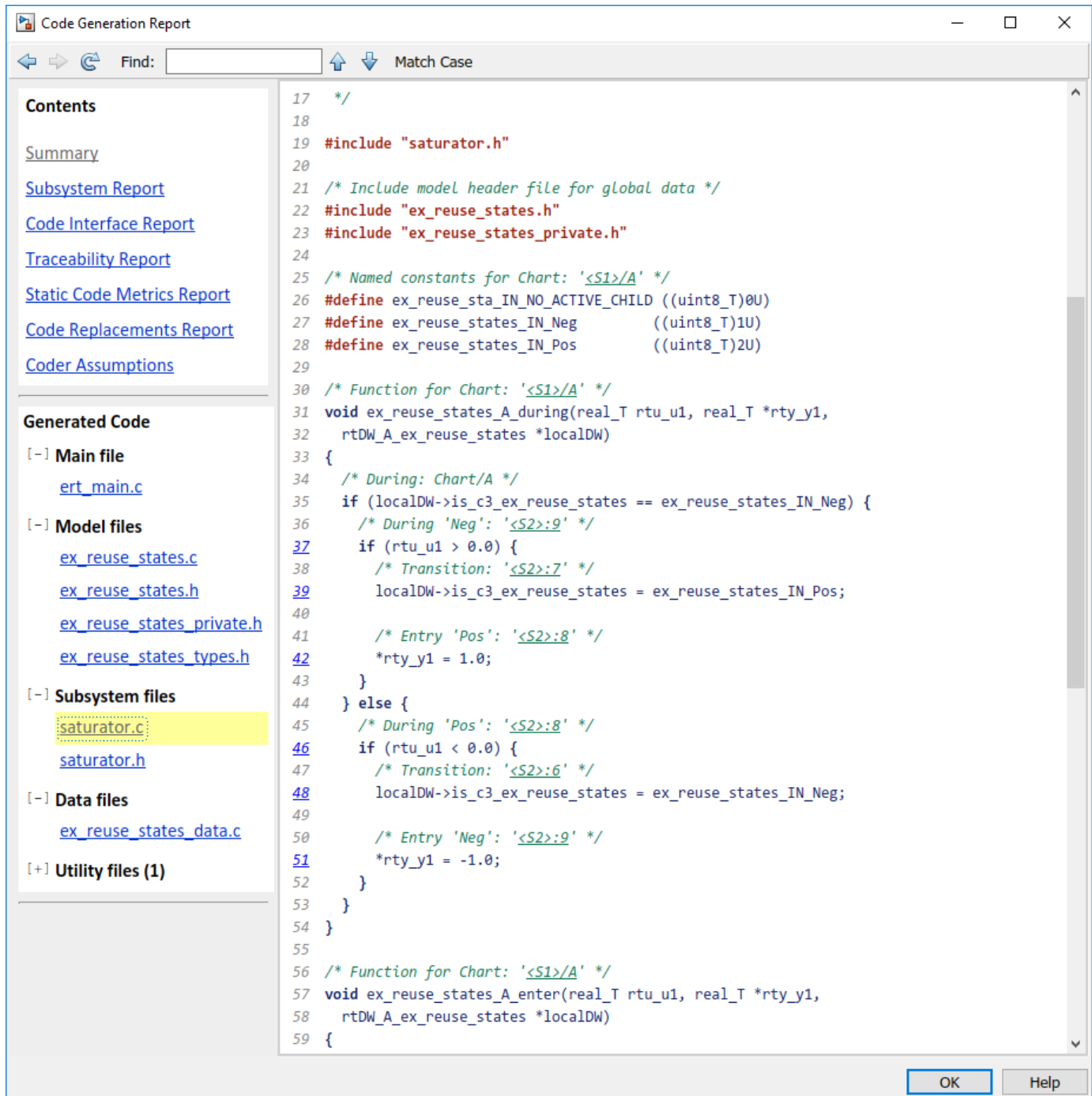
For more information, see Subsystem methods (Embedded Coder).

- 2 Click **Apply**.

Generate Code for Only the Atomic Subchart

To generate code for your model, press **Ctrl+B**. In the code generation report that appears, you see a separate file that contains the generated code for the atomic subchart.

To inspect the code for `saturation.c`, click the hyperlink in the report to see the following code:



Line 28 shows that the during function generated for the atomic subchart has the name `ex_reuse_states_A_during`. This name follows the format scheme `RNMF` specified for **Subsystem methods**:

- `$R` is the root model name, `ex_reuse_states`.
- `$N` is the block name, `A`.
- `$M` is the mangle token, which is empty.

- \$F is the type of interface function for the atomic subchart, during.

Note The line numbers shown can differ from the numbers that appear in your code generation report.

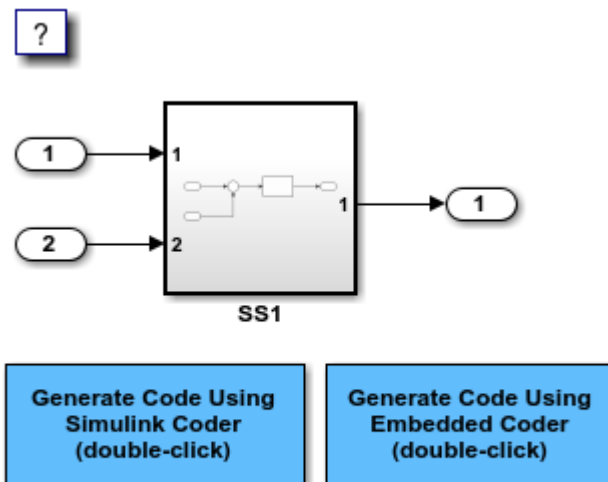
File Packaging for Atomic Subsystems (Code Only)

This model shows how to partition an atomic subsystem into its own source file. Once a subsystem is specified as atomic, on the Subsystem Block Parameter dialog, on the **Code Generation** tab, you can specify how the subsystem is represented in the generated code using the **Function packaging** parameter:

- **Inline**: an Inlined function
- **Function**: function with I/O passed as global data
- **Reusable function**: function with I/O passed as function arguments
- **Auto**: Simulink® Coder™ optimizes based on context

With the **Function** and **Reusable function** options, you can specify a function and file name with **Function name** and **File name**, respectively. In the model, `rtwdemo_filepart`, `SS1` subsystem is configured as **Reusable function** with function name `myfun` and file name `myfile`.

```
open_system('rtwdemo_filepart')
```



Copyright 1994-2020 The MathWorks, Inc.

This capability only partitions the code for a subsystem. The data is declared and owned by the parent model. For full system encapsulation (i.e., data and functions), use Model Reference.

```
bdclose('rtwdemo_filepart');
```

See Also

More About

- “Model Finite State Machines by Using Stateflow Charts” (Stateflow)
- “Encapsulate Modal Logic by Using Subcharts” (Stateflow)

Generate Shared Utility Code

When to Generate Shared Utility Code

Blocks in a model can require common functionality to implement their algorithms. Consider modularizing this functionality into standalone support or helper functions. This approach can be more efficient than inlining the code for the functionality for each block instance. To decide about using a library or a shared utility, consider:

- Packaging functions that can have multiple callers into a library when the functions are defined statically. That is, before you use the code generator to produce code for your model, the function source code exists in a file.
- Packaging functions that can have multiple callers as shared utilities (produced during code generation) when the functions cannot be defined statically. For example, several model- and block-specific properties specify which functions are used and their behavior. Also, these properties determine type definitions (for example, `typedef`) in shared utility header files. The number of possible combinations of properties that determine unique behavior make it impractical to define statically the possible function files before code generation.

Configure Naming of Generated Functions

Configure a default naming rule for generated shared utility functions by using one of these methods.

Configuration	Action
A new model or an existing model for which you have mapped default customization settings by using the Code Mappings Editor - C or code mappings programming interface	In the Embedded Coder Dictionary , define a function customization template that specifies a custom function naming rule. Then, in the Code Mappings editor or by using the code mappings programming interface, map the Shared utility function category to that template. For more information, see “Configure Default C Code Generation for Categories of Data Elements and Functions” (Embedded Coder).
A model created in a release earlier than R2018a and that has not been configured with the Code Mappings Editor - C or code mappings programming interface	Set the model configuration parameter Shared utilities identifier format (<code>CustomSymbolsStrUtil</code>) to a custom function naming rule. For more information, see “Identifier Format Control” (Embedded Coder).

Once you use the **Code Mappings Editor - C** or code mappings programming interface to configure a model, setting model configuration parameter **Shared utilities identifier format** does not impact code generation. This setting is not applicable for models configured with these specifications.

System Target File	Programming Language
<code>ert.tlc</code>	C++
<code>autosar.tlc</code>	C
<code>autosar_adaptive.tlc</code>	C++

The naming rule for shared utility functions must include the conditional checksum token `$.C`. To customize the length of the checksum that the code generator produces, use parameter **Shared**

checksum length (SharedChecksumLength) . Increasing the length of the checksum reduces the probability of clashes.

This example shows the customization of shared utility function names for C code generation.

- 1 Open the example model `rtwdemo_configdefaults` and the **Embedded Coder** app.
- 2 Set model configuration parameter **Shared code placement** to `Shared location`.
- 3 Open the Embedded Coder Dictionary. On the **C Code** tab, select **Code Interface > Embedded Coder Dictionary**.
- 4 On the **Function Customization Template** tab, below the table, click **Create** to add a new function customization template.
- 5 For the function customization template, specify these configurations:
 - **Name:** `exSharedUtility`
 - **Function Name:** `mymodel_$$N$$C`

You can customize this format by using a string. You must include token `$$C`.
- 6 Open the Code Mappings editor. On the **C Code** tab, select **Code Interface > Default Code Mappings**.
- 7 On the **Function Defaults** tab, for the **Shared Utility** category, configure **Function Customization Template** as `exSharedUtility`.
- 8 Generate code and a code generation report.
- 9 In the left navigation pane of the report, review the list of generated files for shared utilities under **Shared files**.

Control Placement of Shared Utility Code

Control placement of shared utility code with the model configuration parameter **Shared code placement**. The default option value is `Auto`. For this setting, the code generator places required code for fixed-point and other utilities in the `model.c` file, the `model.cpp` file, or a separate file in the build folder (for example, `vdp_grt_rtw`) if a model does not contain existing shared utility code or one of these blocks:

- Model blocks
- Simulink Function blocks
- Function Caller blocks
- Calls to Simulink Functions from Stateflow or MATLAB Function blocks
- Stateflow graphical functions when you select parameter **Export Chart Level Functions**

If a model contains one or more of the preceding blocks, the code generator creates and uses a shared utilities folder within `slprj`. The naming convention for shared utility folders is `slprj/target/_sharedutils`. `target` is `sim` for simulations with Model blocks or the name of the system target file for code generation.

```
slprj/sim/_sharedutils      % folder used with simulation
slprj/grt/_sharedutils     % folder used with grt.tlc STF
slprj/ert/_sharedutils     % folder used with ert.tlc STF
slprj/mytarget/_sharedutils % folder used with mytarget.tlc STF
```

To force a model build to use the `slprj` folder for shared utility generation, even when the current model does not contain existing shared utility code or one of the preceding blocks, set parameter **Shared code placement** to `Shared location`. The code generator places utilities under the `slprj` folder rather than in the normal build folder. This setting is useful when you are manually combining code from several models, because it prevents symbol collisions between the models.

Control Placement of `rtwtypes.h` for Shared Utility Code

When the configuration parameter **Data type replacement** is set to Use `coder` typedefs, the generated `rtwtypes.h` header file provides fundamental type definitions, `#define` statements, and enumerations. For more information, see “`rtwtypes.h`” on page 24-40.

Control placement of `rtwtypes.h` file by selecting whether the build process uses the shared utilities folder. If the model build uses a shared utilities folder, the build process places `rtwtypes.h` in `s\prj\target_sharedutils`. Otherwise, the software places `rtwtypes.h` in `model_target_rtw`.

Adding a model to a model hierarchy or changing an existing model in the hierarchy can result in updates to the shared `rtwtypes.h` file during code generation. If updates occur, recompile and, depending on your development process, reverify previously generated code. To minimize updates to the `rtwtypes.h` file, make the following changes in the Configuration Parameters dialog box:

- Select model configuration parameter **Support: complex numbers** even if the model does not currently use complex data types. Selecting this parameter protects against a future requirement to add support for complex data types when integrating code.
- Clear parameter **Support non-inlined S-functions**. If you use noninlined S-functions in the model, this option generates an error.
- Clear parameter **Classic call interface**. This parameter setting disables use of the GRT system target file.

Avoid Duplicate Header Files for Exported Data

Exported header files appear in the shared utility folder when:

- You control the file placement of declarations for signals, parameters, and states by defining and applying storage classes.
- The code generator places utility code in a shared location.

For example, you can specify a header file for data by using:

- A storage class that you define in the **Embedded Coder Dictionary** and map to categories of model data in the **Code Mappings Editor - C**.
- The **Code Generation** tab in a Signal Properties dialog box.
- The `HeaderFile` property of a data object. Data objects are objects of the classes `Simulink.Signal` and `Simulink.Parameter`.

If you want the declaration to appear in the file `model.h`, it is a best practice to leave the header file name unspecified. By default, the code generator places data declarations in `model.h`.

If you specify `model.h` as the header file name, and if the code generator places utility code in a shared location, you cannot generate code from the model. The code generator cannot create the file `model.h` in both the model build folder and the shared utility folder.

Reduce Shared Utility Code Generation with Incremental Builds

You can specify that the model build generates C source files in a shared utilities folder. See “Control Placement of Shared Utility Code” on page 10-53. These files include C source files that contain

function definitions and header files that contain macro definitions. For this discussion, the term functions means functions and macros.

Blocks within the same model and blocks in different models can use a shared function when you use model reference or when you build multiple standalone models from the same start build folder. The code generator produces the code for a given function only once for the block that first triggers code generation. As the product determines the requirement to generate function code for subsequent blocks, it performs a file existence check. If the file exists, the model build does not regenerate the function. The shared utility code mechanism requires that a given function and file name represent the same functional behavior regardless of which block or model generates the function. To satisfy this requirement:

- Model properties that determine function behavior contribute to the shared utility checksum or determine the function and file name.
- Block properties that determine the function behavior also determine the function and file name.

During compilation, makefile rules for the shared utilities folder select compilation of only new C files and incrementally archive the object file into the shared utility library, `rtwshared.lib`, or `rtwshared.a`. Incremental compilation also occurs.

Manage the Shared Utility Code Checksum

When you set the model configuration parameter **Shared code placement** to `Shared location` or when the model contains Model blocks, the code generator places shared code in the shared utilities folder. The build process generates a shared utilities checksum of the code generation configuration for the shared code.

During subsequent code generation, if the checksum file `s\prj\target_sharedutils/checksummap.mat` exists relative to the current folder, the code generator reads that file. The code generator verifies that the current model that you are building has configuration properties that match the checksum of properties from the shared utility model. If mismatches occur between the properties defined in `checksummap.mat` and the current model properties, you see an error. Use the error message to manage the checksum (for example, diagnose and resolve the configuration issues with the current model).

View the Shared Utility Checksum Hash Table

It is helpful to view the property values that contribute to the checksum. This example uses the `rtwdemo_lct_start_term.slx` model. To load the `checksum.mat` file into MATLAB and view the `targetInfoStruct` that defines the checksum-related properties:

- 1 Open the `rtwdemo_lct_start_term.slx` model. In the Command Window, type:

```
rtwdemo_lct_start_term
```

- 2 Create and move to a new working folder.

```
mkdir C:\Temp\demo
cd C:\Temp\demo
```

- 3 Save a copy of the model in the folder.

- 4 Build the model, which is set up to produce shared utilities.

- a On the **Apps** tab, under **Code Generation**, click **Simulink Coder**.
- b On the **C Code** tab, in the **Generate Code** section, click the **Build** button.

- 5 Move to the `_sharedutils` folder created by the build process.

```
cd C:\Temp\demo\slprj\grt\_sharedutils
```

- 6 Load the `checksummap.mat` file into MATLAB.

```
load checksummap
```

- 7 Display the contents of `hashTbl.targetInfoStruct` and examine the checksum-related property values.

```
hashTbl.targetInfoStruct
```

For this example, the Command Window displays `hashTbl.targetInfoStruct` for the shared utilities that you generated from the model:

```
ans =
```

```
struct with fields:
```

```

    ShiftRightIntArith: 'on'
    ProdShiftRightIntArith: 'on'
        Endianess: 'LittleEndian'
        ProdEndianess: 'LittleEndian'
        wordlengths: '8,16,32,32,64,32,64,64,64,64'
        Prodwordlengths: '8,16,32,32,64,32,64,64,64,64'
        TargetWordSize: '64'
        ProdWordSize: '64'
    TargetHWDeviceType: 'Custom Processor->MATLAB Host Computer'
    ProdHWDeviceType: 'Intel->x86-64 (Windows64)'
    TargetIntDivRoundTo: 'Zero'
    ProdIntDivRoundTo: 'Zero'
    UseDivisionForNetSlopeComputation: 'off'
    PurelyIntegerCode: 'off'
    PortableWordSizes: 'off'
    SupportNonInlinedSFcns: ''
    RTWReplacementTypes: ''
        MaxIdInt8: 'MAX_int8_T'
        MinIdInt8: 'MIN_int8_T'
        MaxIdUInt8: 'MAX_uint8_T'
        MaxIdInt16: 'MAX_int16_T'
        MinIdInt16: 'MIN_int16_T'
        MaxIdUInt16: 'MAX_uint16_T'
        MaxIdInt32: 'MAX_int32_T'
        MinIdInt32: 'MIN_int32_T'
        MaxIdUInt32: 'MAX_uint32_T'
        MaxIdInt64: 'MAX_int64_T'
        MinIdInt64: 'MIN_int64_T'
        MaxIdUInt64: 'MAX_uint64_T'
        BooleanTrueId: 'true'
        BooleanFalseId: 'false'
    TypeLimitIdReplacementHeaderFile: ''
    SharedCodeRepository: ''
        TargetLang: 'C'
        TargetLangStd: 'C89/C90 (ANSI)'
    InstructionSetExtions: 'None'
    PreserveExternInFcnDecls: 'on'
    ImplementImageWithCVMat: 'off'
    EnableSignedRightShifts: 'on'
    EnableSignedLeftShifts: 'on'

```

```

        TflName: 'None'
        TflChecksum: [4.2298e+09 3.9634e+09
                    2.9213e+09 173860989]
        UtilMemSecPreStatement: ''
        UtilMemSecPostStatement: ''
        UtilMemSecComment: ''
        CodeCoverageChecksum: [3.6498e+09 78774415
                              2.5508e+09 2.1183e+09]
        TargetLargestAtomicInteger: 'Char'
        TargetLargestAtomicFloat: 'None'
        ProdLargestAtomicInteger: 'Char'
        ProdLargestAtomicFloat: 'Float'
        LongLongMode: 'on'
        ProdLongLongMode: 'off'
        HardwareBoard: 'None'
        toolchainOrTMF: 'Microsoft Visual C++ 2017 v15.0 |
                       nmake (64-bit Windows)'

```

Relate the Shared Utility Checksum to Configuration Parameters

Examine the `targetInfoStruct` hash table from the shared utility model. Some key-value pairs relate directly to a model property. Other pairs relate to groups of properties.

The following table describes the key-value pairs.

Key Names	Model Properties
ShiftRightIntArith	TargetShiftRightIntArith
ProdShiftRightIntArith	ProdShiftRightIntArith
Endianness	TargetEndianness
ProdEndianness	ProdEndianness
wordlengths	TargetBitPerChar, TargetBitPerShort, TargetBitPerInt, TargetBitPerLong, TargetBitPerLongLong, TargetBitPerFloat, TargetBitPerDouble, TargetWordSize, TargetBitPerPointer, TargetBitPerSizeT, TargetBitPerPtrDiffT
Prodwordlengths	ProdBitPerChar, ProdBitPerShort, ProdBitPerInt, ProdBitPerLong, ProdBitPerLongLong, ProdBitPerFloat, ProdBitPerDouble, ProdWordSize, ProdBitPerPointer, ProdBitPerSizeT, ProdBitPerPtrDiffT
TargetWordSize	TargetWordSize
ProdWordSize	ProdWordSize
TargetHWDeviceType	TargetHWDeviceType
ProdHWDeviceType	ProdHWDeviceType
TargetIntDivRoundTo	TargetIntDivRoundTo
ProdIntDivRoundTo	ProdIntDivRoundTo

Key Names	Model Properties
UseDivisionForNetSlopeComputation	UseDivisionForNetSlopeComputation
PurelyIntegerCode	PurelyIntegerCode
PortableWordSizes	PortableWordSizes
SupportNonInlinedSFcns	SupportNonInlinedSFcns
RTWReplacementTypes	EnableUserReplacementTypes, ReplacementTypes
MaxIdInt8	MaxIdInt8
MinIdInt8	MinIdInt8
MaxIdUInt8	MaxIdUInt8
MaxIdInt16	MaxIdInt16
MinIdInt16	MinIdInt16
MaxIdUInt16	MaxIdUInt16
MaxIdInt32	MaxIdInt32
MinIdInt32	MinIdInt32
MaxIdUInt32	MaxIdUInt32
MaxIdInt64	MaxIdInt64
MinIdInt64	MinIdInt64
MaxIdUInt64	MaxIdUInt64
BooleanTrueId	BooleanTrueId
BooleanFalseId	BooleanFalseId
TypeLimitIdReplacementHeaderFile	TypeLimitIdReplacementHeaderFile
SharedCodeRepository	Reserved (internal use only)
TargetLang	TargetLang
TargetLangStd	TargetLangStandard
InstructionSetExtions	InstructionSetExtensions
PreserveExternInFcnDecls	PreserveExternInFcnDecls
ImplementImageWithCVMat	ImplementImageWithCVMat
EnableSignedRightShifts	EnableSignedRightShifts
EnableSignedLeftShifts	EnableSignedLeftShifts
TflName	CodeReplacementLibrary
TflChecksum	Reserved (internal use only)
UtilMemSecPreStatement	MemSecFuncSharedUtil, MemSecPackage
UtilMemSecPostStatement	MemSecFuncSharedUtil, MemSecPackage
UtilMemSecComment	MemSecFuncSharedUtil, MemSecPackage
CodeCoverageChecksum	Reserved (internal use only)
TargetLargestAtomicInteger	TargetLargestAtomicInteger
TargetLargestAtomicFloat	TargetLargestAtomicFloat

Key Names	Model Properties
ProdLargestAtomicInteger	ProdLargestAtomicInteger
ProdLargestAtomicFloat	ProdLargestAtomicFloat
LongLongMode	TargetLongLongMode
ProdLongLongMode	ProdLongLongMode
HardwareBoard	HardwareBoard
toolchainOrTMF	Name of: <ul style="list-style-type: none"> • Toolchain if build process uses either a toolchain (<code>Toolchain</code>) or a template makefile that is associated with a toolchain. • Template makefile if build process uses template makefile (<code>TemplateMakefile</code>).

See Also

More About

- “Generate Shared Utility Code for Fixed-Point Functions” on page 10-60
- “Generate Shared Utility Code for Custom Data Types” on page 10-62
- “Cross-Release Shared Utility Code Reuse” on page 24-69
- “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder)

Generate Shared Utility Code for Fixed-Point Functions

An important set of generated functions that the model build places in the shared utility folder are the fixed-point support functions. Based on model and block properties, there are many possible versions of fixed-point utilities functions that make it impractical to provide a complete set as static files. Generating only the required fixed-point utility functions during the code generation process is an efficient alternative.

The shared utility checksum mechanism makes sure that several critical properties are identical for models that use the shared utilities. For the fixed-point functions, there are additional properties that determine function behavior. The mechanism codes these properties into the functions and file names to maintain requirements. The additional properties include:

Category	Function/Property
Block properties	<ul style="list-style-type: none"> Fixed-point operation that the block performs Fixed-point data type and scaling (Slope, Bias) of function inputs and outputs Overflow handling mode (Saturation, Wrap) Rounding Mode (Floor, Ceil, Zero)
Model properties	<code>get_param(bdroot, 'NoFixptDivByZeroProtection')</code>

The property-based naming convention for the fixed-point utilities is as follows:

```
operation + [zero protection] + output data type + output bits +
[input1 data] + input1 bits + [input2 data type + input2 bits] +
[shift direction] + [saturate mode] + [round mode]
```

The file names shown are examples of generated fixed-point utility files. The function or macro names in the file are identical to the file name without the extension.

```
FIX2FIX_U12_U16.c
FIX2FIX_S9_S9_SR99.c
ACCUM_POS_S30_S30.h
MUL_S30_S30_S16.h
div_nzp_s16s32_floor.c
div_s32_sat_floor.c
```

For these examples, the table shows how the respective fields correspond.

The ACCUM_POS example uses the output variable as one of the input variables. So, the file and macro name only contain the output and second input.

The second div example has identical data type and bits for both inputs and the output. So, the file and function name only include the output.

Operation	FIX2FIX	FIX2FIX	ACCUM_POS	MUL	div	div
Zero protection	NULL	NULL	NULL	NULL	_nzp	NULL
Output data type	_U	_S	_S	_S	_s	_s
Output bits	12	9	30	30	16	32
Input data type	_U	_S	_S	_S [and _S]	s	NULL
Input bits	16	9	30	30 [and 16]	32	NULL

Operation	FIX2FIX	FIX2FIX	ACCUM_POS	MUL	div	div
Shift direction	NULL	SR99	NULL	NULL	NULL	NULL
Saturate mode	NULL	NULL	NULL	NULL	NULL	_sat
Round mode	NULL	NULL	NULL	NULL	_floor	_floor

See Also

More About

- “Generate Shared Utility Code” on page 10-52
- “Generate Shared Utility Code for Custom Data Types” on page 10-62
- “Cross-Release Shared Utility Code Reuse” on page 24-69

Generate Shared Utility Code for Custom Data Types

By default, if a model employs a custom data type (such as a `Simulink.AliasType` object or an enumeration class), the code generator places the corresponding type definition (`typedef`) in the `model_types.h` file. When you generate code from multiple models, each model duplicates the type definition. These duplicate definitions can prevent you from compiling the bodies of generated code together.

However, you can configure the code generator to place a single type definition in a header file in the `_sharedutils` folder. Then, when you generate code from a model, if the type definition already exists in the `_sharedutils` folder, the code generator does not duplicate the definition, but instead reuses it through inclusion (`#include`).

Through this mechanism, you can share:

- Simulink data type objects that you instantiate from the classes `Simulink.AliasType`, `Simulink.Bus`, and `Simulink.NumericType`. For basic information about creating and using these objects, see “Manage Replacement of Simulink Data Types in Generated Code” (Embedded Coder) and `Simulink.Bus`.
- Enumerations that you define, for example, by authoring an `enum` class in a script file or by using the function `Simulink.defineIntEnumType`. For basic information about defining enumerations in Simulink, see “Use Enumerated Data in Simulink Models”.

To share a custom data type across multiple models:

- 1 Define the data type. For example, create the `Simulink.AliasType` object.
- 2 Set data scope and header file properties to specific values that enable sharing.

For a data type object, set the `DataScope` property to `'Exported'` and, optionally, specify the header file name through the `HeaderFile` property.

For an enumeration that you define as an `enum` class in a script file, implement the `getDataScope` method (with return value `'Exported'`) and, optionally, implement the `getHeaderFile` method.

For an enumeration that you define by using the `Simulink.defineIntEnumType` function, set the `'DataScope'` pair argument to `'Exported'` and, optionally, specify the `'HeaderFile'` pair argument

- 3 Use the data type in the models.
- 4 Before generating code from each model, set the model configuration parameter **Shared code placement** to `Shared location`.
- 5 Generate code from the models.

Note You can configure the definition of the custom data type to appear in a header file in the `_sharedutils` folder. The shared utility functions that the model build generates into the `_sharedutils` folder do not use the custom data type name. Only model code located in code folders for each model uses the custom data type name.

See Also

More About

- “Generate Shared Utility Code” on page 10-52
- “Generate Shared Utility Code for Fixed-Point Functions” on page 10-60
- “Cross-Release Shared Utility Code Reuse” on page 24-69
- “Control File Placement of Custom Data Types” (Embedded Coder)

Shared Constant Parameters for Code Reuse

You can share the generated code for constant parameters across models if either of the following conditions applies:

- Constant parameters are shared in a model reference hierarchy.
- The model configuration parameter “Shared code placement” is set to Shared location.

If you do not want to generate shared constants, and model configuration parameter **Shared code placement** is set to Shared location, set parameter `GenerateSharedConstants` to `off`. For example, to turn off shared constants for the current model, in the Command Window, type the following.

```
set_param(gcs, 'GenerateSharedConstants', 'off');
```

When the model configuration parameter `GenerateSharedConstants` is set to `off`, the code generator generates the constant parameters shared across the model reference hierarchy in a shared location, in the `slprj/target/_sharedutils` folder but outside the `const_params.c` file. The code generator defines the individual constant parameters in the reusable library subsystem files individually.

When the model configuration parameter `GenerateSharedConstants` is set to `on`, the code generator produces shared constant parameters individually and places them in the file `const_params.c`. The code generator places that file in the shared utilities folder `slprj/target/_sharedutils`.

For example, if a constant has multiple uses within a model reference hierarchy where the top model is named `topmod`, the code for the shared constant is as follows:

- In the shared utility folder, `slprj/grt/_sharedutils`, the constant parameters are defined in `const_params.c` and named `rtCP_pooled_` appended to a unique checksum:

```
extern const real_T rtCP_pooled_lfcjjmohiecj[7];
const real_T rtCP_pooled_lfcjjmohiecj[7] = { 1.0, 2.0, 3.0, 4.0, 5.0, 6.0, 7.0 };

extern const real_T rtCP_pooled_ppphohdbfcba[7];
const real_T rtCP_pooled_ppphohdbfcba[7] = { 7.0, 6.0, 5.0, 4.0, 3.0, 2.0, 1.0 };
```

- In `top_model_private.h` or in a referenced model, `ref_model_private.h`, for better readability, the constants are renamed as follows:

```
extern const real_T rtCP_pooled_lfcjjmohiecj[7];
extern const real_T rtCP_pooled_ppphohdbfcba[7];

#define rtCP_Constant_Value    rtCP_pooled_lfcjjmohiecj /* Expression: [1 2 3 4 5 6 7]
 * Referred by: '<Root>/Constant' */
#define rtCP_Gain_Gain        rtCP_pooled_ppphohdbfcba /* Expression: [7 6 5 4 3 2 1]
 * Referred by: '<Root>/Gain' */
```

- In `topmod.c` or `refmod.c`, the call site might be:

```
for (i = 0; i < 7; i++) {
    topmod_Y.Out1[i] = (topmod_U.In1 + rtCP_Constant_Value[i]) * rtCP_Gain_Gain[i];
}
```

The code generator attempts to generate constants as individual constants to the `const_params.c` file in the shared utilities folder. If a model contains a parameter that the code generator cannot inline directly (for example, an array parameter), the code defines a structure type that represents the data. This structure uses the `const` storage type qualifier, so some build toolchains can optimize the assembly code further.

The declaration of the structure, `ConstParam_model`, is in `model.h`.

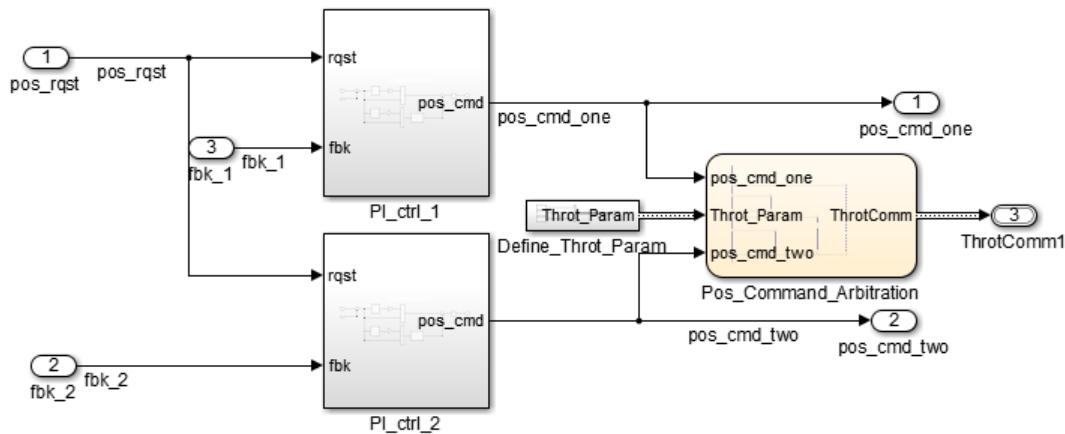
Suppress Shared Constants in the Generated Code

You can choose whether the code generator produces shared constants and shared functions. You might want to be able to keep the code and data separate between subsystems, or you might find that sharing constants results in a memory shortage during code generation.

You can change this setting programmatically by using the parameter `GenerateSharedConstants` with `set_param` and `get_param`.

In the following example, when `GenerateSharedConstants` is set to on, the code generator defines the constant values in the `_sharedutils` folder in the `const_params.c` file. When `GenerateSharedConstants` is set to off, the code generator defines the constant values in a nonshared area, in the `model_ert_rtw` file in the `model_data.c` file.

Open the model `rtwdemo_throttlecntrl`:



Verify that model configuration parameter “Shared code placement” is set to Shared location. If the parameter is set to Auto, the code generator ignores setting `GenerateSharedConstants`. If you try to set the parameter value, an error message appears. By default, value `GenerateSharedConstants` is selected.

In the Command Window, set `GenerateSharedConstants` to on:

```
>> set_param('rtwdemo_throttlecntrl', 'GenerateSharedConstants', 'on')
```

You see the shared constant definitions in the folder `slprj/grt/_sharedutils`, in the file `const_params.c`:

```
extern const real_T rtCP_pooled_H4eTKtECwveN[9];
const real_T rtCP_pooled_H4eTKtECwveN[9] = { 1.0, 0.75, 0.6, 0.0, 0.0, 0.0, 0.6,
    0.75, 1.0 };

extern const real_T rtCP_pooled_SghuHxKVKGHD[9];
const real_T rtCP_pooled_SghuHxKVKGHD[9] = { -1.0, -0.5, -0.25, -0.05, 0.0, 0.05,
    0.25, 0.5, 1.0 };

extern const real_T rtCP_pooled_WqWb2t17NA2R[7];
const real_T rtCP_pooled_WqWb2t17NA2R[7] = { -1.0, -0.25, -0.01, 0.0, 0.01, 0.25,
    1.0 };

extern const real_T rtCP_pooled_Ygnal0wM3c14[7];
const real_T rtCP_pooled_Ygnal0wM3c14[7] = { 1.0, 0.25, 0.0, 0.0, 0.0, 0.25, 1.0
};
```

In the Command Window, set `GenerateSharedConstants` to off:

```
>> set_param('rtwdemo_throttletcntrl','GenerateSharedConstants','off')
```

You can see the unshared constants in the folder `rtwdemo_throttletcntrl_grt_rtw`, in the file `rtwdemo_throttletcntrl_data.c`:

```
/* Constant parameters (auto storage) */
const ConstP_rtwdemo_throttletcntrl_T rtwdemo_throttletcntrl_ConstP = {
  /* Pooled Parameter (Expression: P_OutMap)
   * Referenced by:
   *   '<S2>/Proportional Gain Shape'
   *   '<S3>/Proportional Gain Shape'
   */
  { 1.0, 0.25, 0.0, 0.0, 0.0, 0.25, 1.0 },

  /* Pooled Parameter (Expression: P_InErrMap)
   * Referenced by:
   *   '<S2>/Proportional Gain Shape'
   *   '<S3>/Proportional Gain Shape'
   */
  { -1.0, -0.25, -0.01, 0.0, 0.01, 0.25, 1.0 },

  /* Pooled Parameter (Expression: I_OutMap)
   * Referenced by:
   *   '<S2>/Integral Gain Shape'
   *   '<S3>/Integral Gain Shape'
   */
  { 1.0, 0.75, 0.6, 0.0, 0.0, 0.0, 0.6, 0.75, 1.0 },

  /* Pooled Parameter (Expression: I_InErrMap)
   * Referenced by:
   *   '<S2>/Integral Gain Shape'
   *   '<S3>/Integral Gain Shape'
   */
  { -1.0, -0.5, -0.25, -0.05, 0.0, 0.05, 0.25, 0.5, 1.0 }
};
```

Shared constant Parameters Limitations

The code generator does not produce shared constants for a model when:

- The model has a code replacement library (CRL) that is specified for data alignment.
- The **Memory Section** for constants is `MemVolatile` or `MemConstVolatile`.
- The parameter `GenerateSharedConstants` is set to off.

Individual constants are not shared, if:

- A constant is referenced by a non-inlined S-function.
- A constant has a user-defined type where **Data Scope** is not set to `Exported`.

See Also

More About

- “Generate Reentrant Code from Subsystems” (Embedded Coder)
- “Generate Reusable Code from Library Subsystems Shared Across Models” on page 10-36

Determine Why Subsystem Code Is Not Reused

Due to the limitations described in “Generate Reentrant Code from Subsystems” on page 10-28, the code generator might not reuse generated code as you expect. To determine why code generated for a subsystem is not reused:

- 1 Review the Subsystems section of the code generation report.
- 2 Compare subsystem checksum data.

Review Subsystems Section of HTML Code Generation Report

If the code generator does not generate code for a subsystem as reusable code, and you configured the subsystem as reusable, examine the Subsystems section of the code generation report (see “Generate Code Generation Report” on page 25-5). The Subsystems section contains:

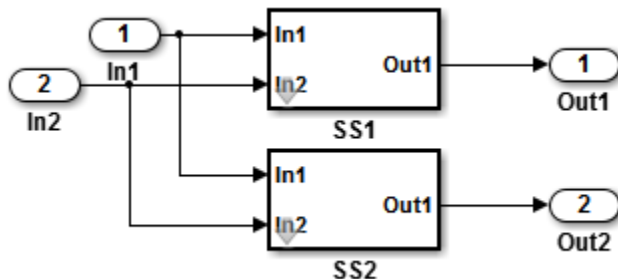
- A table that summarizes how nonvirtual subsystems were converted to generated code.
- Diagnostic information that describes why subsystems were not generated as reusable code.

The Subsystems section also maps noninlined subsystems in the model to functions or reused functions in the generated code. For an example, open and build the `rtwdemo_atomic` model.

Compare Subsystem Checksum Data

You can determine why subsystem code is not reused by comparing subsystem checksum data. The code generator determines whether subsystems are identical by comparing subsystem checksums, as noted in “Limitations” on page 10-33. For subsystem reuse across referenced models, this procedure might not flag every difference.

Consider the model, `rtwdemo_ssreuse`. `SS1` and `SS2` are instances of the same subsystem. In both instances the subsystem block parameter **Function packaging** is set to `Reusable function`.



Use the method `Simulink.SubSystem.getChecksum` to get the checksum for a subsystem. Review the results to determine why code is not reused.

- 1 Open the model `rtwdemo_ssreuse`. Save a copy of the model in a folder where you have write access.
- 2 Associate the subsystems `SS1` and `SS2` with `gcb`. For each of the subsystems, in the model window, select the subsystem. While the subsystem is selected, in the Command Window, enter:

```
SS1 = gcb;
```

```
SS2 = gcb;
```

- 3 Use the method `Simulink.SubSystem.getChecksum` to get the checksum for each subsystem. This method returns two output values: the checksum value and details on the input used to compute the checksum.

```
[chksum1, chksum1_details] = ...
Simulink.SubSystem.getChecksum(SS1);
[chksum2, chksum2_details] = ...
Simulink.SubSystem.getChecksum(SS2);
```

- 4 Compare the two checksum values. The values should be equal based on the subsystem configurations.

```
isequal(chksum1, chksum2)
ans =
     1
```

- 5 To use `Simulink.SubSystem.getChecksum` to determine why the checksums of two subsystems differ, change the data type mode of the output port of SS1 so that it differs from that of SS2.

- a Look under the mask of SS1. Right-click the subsystem. In the context menu, select **Mask > Look Under Mask**.
- b In the block diagram of the subsystem, double-click the Lookup Table block to open the Subsystem Parameters dialog box.
- c Click **Data Types**.
- d Select block parameter **Saturate on integer overflow** and click **OK**.

- 6 Get the checksum for SS1. Compare the checksums for the two subsystems. This time, the checksums are not equal.

```
[chksum1, chksum1_details] = ...
Simulink.SubSystem.getChecksum(SS1);
isequal(chksum1, chksum2)
ans =
     0
```

- 7 After you determine that the checksums are different, find out why. The Simulink engine uses information, such as signal data types, some block parameter values, and block connectivity information, to compute the checksums. To determine why checksums are different, compare the data that computes the checksum values. You can get this information from the second value returned by `Simulink.SubSystem.getChecksum`, which is a structure array with four fields.

Look at the structure `chksum1_details`.

```
chksum1_details

chksum1_details =
    ContentsChecksum: [1x1 struct]
    InterfaceChecksum: [1x1 struct]
    ContentsChecksumItems: [287x1 struct]
    InterfaceChecksumItems: [53x1 struct]
```

`ContentsChecksum` and `InterfaceChecksum` are component checksums of the subsystem checksum. The remaining two fields, `ContentsChecksumItems` and `InterfaceChecksumItems`, contain the checksum details.

- 8 Determine whether a difference exists in the subsystem contents, interface, or both. For example:

```

isequal(chksum1_details.ContentsChecksum.Value,...
        chksum2_details.ContentsChecksum.Value)
ans =
    0
isequal(chksum1_details.InterfaceChecksum.Value,...
        chksum2_details.InterfaceChecksum.Value)
ans =
    1

```

In this case, differences exist in the content.

- 9** Write a script like this script to find the differences.

```

idxForCDiffs=[];
for idx = 1:length(chksum1_details.ContentsChecksumItems)
    if (~strcmp(chksum1_details.ContentsChecksumItems(idx).Identifier, ...
               chksum2_details.ContentsChecksumItems(idx).Identifier))
        disp(['Identifiers different for contents item ', num2str(idx)]);
        idxForCDiffs=[idxForCDiffs, idx];
    end
    if (ischar(chksum1_details.ContentsChecksumItems(idx).Value))
        if (~strcmp(chksum1_details.ContentsChecksumItems(idx).Value, ...
                   chksum2_details.ContentsChecksumItems(idx).Value))
            disp(['Character vector values different for contents item ', num2str(idx)]);
            idxForCDiffs=[idxForCDiffs, idx];
        end
    end
    if (isnumeric(chksum1_details.ContentsChecksumItems(idx).Value))
        if (chksum1_details.ContentsChecksumItems(idx).Value ~= ...
           chksum2_details.ContentsChecksumItems(idx).Value)
            disp(['Numeric values different for contents item ', num2str(idx)]);
            idxForCDiffs=[idxForCDiffs, idx];
        end
    end
end
end

```

- 10** Run the script. The example assumes that you named the script `check_details`.

```

check_details
Character vector values different for contents item 202

```

The results indicate that differences exist for index item 202 in the subsystem contents.

- 11** Use the returned index values to get the handle, identifier, and value details for each difference found.

```

chksum1_details.ContentsChecksumItems(202)

ans =

    Handle: 'rtwdemo_ssreuse/SS1/Lookup Table'
    Identifier: 'SaturateOnIntegerOverflow'
    Value: 'on'

```

The details identify the Lookup Table block parameter **Saturate on integer overflow** as the focus for debugging a subsystem reuse issue.

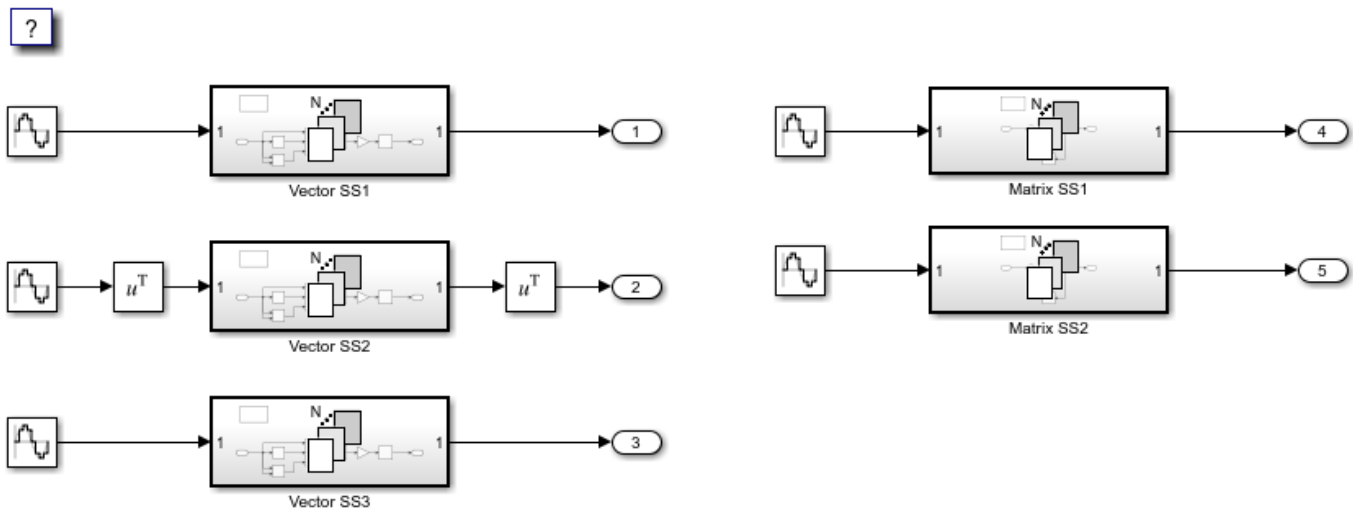
Generate Reusable Code from For Each Subsystems

In the model-based design, to convert inefficient repetitive algorithms into compact forms, use For Each subsystems. For Each subsystems repeat execution during a simulation time step on each element or subarray of an input signal or mask parameter array. In addition, to reduce model complexity, you can also generate reusable code for identical For Each subsystems by configuring them as reusable functions. You must configure mask and block parameters of the identical subsystems the same way. The code generator performs a checksum to determine whether subsystems are identical and the code is reusable. If the code is reusable, the code generator produces a single instance of reusable code which is shared by the identical subsystems.

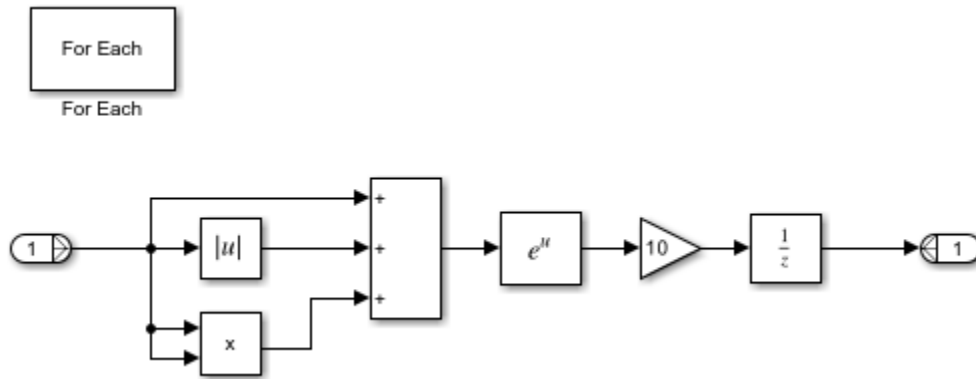
This example shows how you can generate reusable code for For Each subsystems that perform the same processing on their respective inputs.

Example Model

This example generates code from the ForEachReuse model provided in “Improved Code Reuse Using For-Each Subsystems”.



The model contains three For Each subsystems Vector SS1, Vector SS2, and Vector SS3 that apply the same processing to each scalar element of the vector signal at their respective inputs. For example, the Vector SS3 subsystem contains these blocks.



To generate a single shared function for the three subsystems, the input signal dimension of the subsystems must be the same. For `Vector SS1` and `Vector SS3`, set the partition dimension and width to 1. For `Vector SS2` to also set the input signal dimension 1, insert a Math Function block to transpose the 1-by-8 row vector into an 8-by-1 column vector. To convert the output of the subsystem back to a 1-by-8 row vector, use a second Math Function block.

The remaining `For Each` subsystem `Matrix SS1` and `Matrix SS2` process matrix signals using the same algorithm and have the same input signal dimension setting.

Generate and Inspect Code

- 1 To configure the subsystems as reusable functions, open the Block Parameters dialog box. On the **Code Generation** tab, set **Function packaging** to `Reusable` function.
- 2 To build the model and generate code, press **Ctrl+B**.

```
### Starting build procedure for: ForEachReuse
### Successful completion of build procedure for: ForEachReuse
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
ForEachReuse	Code generated and compiled.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)

Build duration: 0h 0m 16.637s

The code generator generates a single output function `VectorProcessing` in `VectorProcessing.c`.

```
* Output and update for iterator system:
*   '<Root>/Vector SS1'
*   '<Root>/Vector SS2'
*   '<Root>/Vector SS3'
*/
void VectorProcessing(int32_T NumIters, const real_T rtu_In1[10], real_T
                    rty_Out1[10], DW_VectorProcessing_T localDW[10])
{
```

```
/* local scratch DWork variables */
int32_T ForEach_itr;
real_T rtb_Out1_CoreSubsysCanOut;
```

The function has an input parameter `NumIters` that indicates the number of independent scalars that each For Each Subsystem block processes. This function is called three times in the `ForEachReuse.c` with the parameter `NumIters` set to 10, 8, and 7 respectively.

The code generator also generates a single output function for `Matrix SS1` and `Matrix SS2` subsystem in `MatrixProcessing.c` which is called twice in the `ForEachReuse.c`.

```
* Output and update for iterator system:
*   '<Root>/Matrix SS1'
*   '<Root>/Matrix SS2'
*/
void MatrixProcessing(int32_T NumIters, const real_T rtu_In1[32], real_T
                    rty_Out1[32], DW_MatrixProcessing_T localDW[2])
{
  /* local scratch DWork variables */
  int32_T ForEach_itr;
  real_T rtb_MathFunction_h_0;
  int32_T i;
  int32_T rtb_MathFunction_h_tmp;
  int32_T tmp;
```

See Also

For Each Subsystem

Related Examples

- “Generate Reusable Code from Library Subsystems Shared Across Models” on page 10-36
- “Repeat an Algorithm Using a For-Each Subsystem”

Configure Model Parameters for Simulink Coder

- “Configure Run-Time Environment Options” on page 11-2
- “Using the Target Framework” on page 11-11
- “Register New Hardware Devices” on page 11-13

Configure Run-Time Environment Options

When you use Simulink software to create and execute a model and use the code generator to produce C or C++ code, consider your configuration for up to three run-time environments:

- The MATLAB development computer run-time environment that runs MathWorks software during application development.
- The production hardware run-time environment in which you deploy an application when it is put into production.
- The test hardware run-time environment in which you test an application under development before deployment.

One run-time environment can serve in multiple capacities, but the run-time environments remain conceptually distinct. Often, the MATLAB development computer is the test hardware. Typically, the production hardware is different from, and less powerful than, the MATLAB development or the test hardware. Many types of production hardware can do little more than run a downloaded executable file.

Provide information about the production hardware board and the compiler that you use with it when:

- You use Simulink software to simulate a model for which you later generate code
- You use the code generator to produce code for deployment on *production* hardware

The software uses the board and compiler information to get bit-true agreement for the results of integer and fixed-point operations performed in simulation and in code generated for the production hardware. The code generator uses the information to create code that executes with maximum efficiency.

When you generate code for testing on *test* hardware, provide information about the test hardware board and the compiler that you use. The code generator uses this information to create code that provides bit-true agreement between results from:

- Integer and fixed-point operations performed in simulation
- Generated code run on the production hardware
- Generated code run on the test hardware

You can achieve bit-true agreement for results even if the production and test hardware are different. Where the C standard does not completely define behavior, the compilers for the two types of hardware can use different defaults.

Configure Production and Test Hardware

You can specify model simulation or code generation for a specific hardware board and its device type. For example, you can set the data size, byte ordering, and compiler behavior, such as integer rounding. You can configure:

- The production hardware and the compiler that you use with it. This information affects simulation and code generation. See “Example Production Hardware Setting That Affects Normal Mode Simulation” on page 11-9.

- The test hardware and the compiler that you use with it. This information affects only code generation.

Configure production hardware by selecting **Configuration Parameters > Hardware Implementation**. By default, the Hardware Implementation pane lists **Hardware board**, **Device vendor**, and **Device type** parameter fields only. Unless you have installed hardware support packages, **Hardware board** lists values None or Determine by Code Generation system target file, and Get Hardware Support Packages. After installing a hardware support package, the list also includes the corresponding hardware board name. If you select a hardware board name, parameters for that board appear. To set device details, such as data size and byte ordering, click **Device details**.

Configure test hardware on the **Configuration Parameters > Hardware Implementation > Advanced parameters** pane. To enable parameters for configuring test hardware details, disable the **Configuration Parameters > Hardware Implementation > Advanced parameters > Test hardware is the same as production hardware** parameter. Code generated for test hardware executes in the environment specified by the test hardware parameters. The code behaves as if it were executing in the environment specified for the production hardware. For more information, see “Test Hardware Considerations” on page 11-8.

Default values and properties appear as initial values in the **Hardware Implementation** pane when:

- You specify a **System target file** in the **Code Generation** pane.
- The system target file specifies a default microprocessor and its hardware properties.

You cannot change parameters that have only one possible value. Parameters that have more than one possible value provide a list of valid values. If you specify hardware properties manually in **Hardware Implementation** pane, verify that these values are consistent with the system target file. Otherwise, the generated code can fail to compile or execute, or can execute but produce incorrect results.

Hardware implementation parameters describe hardware and compiler properties to MATLAB software. The code generator uses the information to produce code for the run-time environment that runs as efficiently as possible. The generated code gives bit-true agreement for the results of integer and fixed-point operations in simulation, production code, and test code.

For details about specific parameters, see “Hardware Implementation Pane”. To see an example of **Hardware Implementation** pane capabilities, see the `rtwdemo_targetsettings` example model. For details related to configuring a hardware implementation, see:

- “Specify Hardware Board” on page 11-4
- “Specify Device Vendor” on page 11-4
- “Specify the Device Type” on page 11-4
- “Set Bit Lengths for Device Data Types” on page 11-5
- “Set Byte Ordering for Device” on page 11-6
- “Set Quotient Rounding Behavior for Signed Integer Division” on page 11-6
- “Set Arithmetic Right Shift Behavior for Signed Integers” on page 11-7
- “Update Release 14 Hardware Configuration” on page 11-7

Specify Hardware Board

Specify the hardware board that runs the code generated from your model. Select a value for **Configuration Parameters > Hardware Implementation > Hardware board**.

The Hardware Implementation pane identifies the system target file selected on **Configuration Parameters > Code Generation**.

To configure test hardware, use the **Configuration Parameters > Hardware Implementation > Advanced parameters** pane.

To enable parameters for configuring test hardware details, set ProdEqTarget to off.

Ways to Specify the Hardware Board

If	Select
The menu <i>includes</i> the name of the hardware board that you want to use.	The name of that hardware board. If you select a hardware board name, parameters for that board appear.
The menu <i>does not include</i> the name of the hardware board that you want to use.	Get Hardware Support Packages. That value opens the Support Package Installer. Install the support package that you want. After you install the support package, the menu includes relevant hardware board names.
The model configuration <i>uses</i> system target file <code>ert.tlc</code> , <code>realtime.tlc</code> , or <code>autosar.tlc</code> .	None. No hardware board is specified for the hardware implementation.
The model configuration <i>does not use</i> system target file <code>ert.tlc</code> , <code>realtime.tlc</code> , or <code>autosar.tlc</code> .	Determine by Code Generation system target file. The code generator uses the specified system target file to determine the hardware implementation.

Specify Device Vendor

To specify the vendor of the microprocessor of the hardware device, use the **Device vendor** parameter. Your selection determines the available microprocessors in the **Device type** menu. If the vendor name does not appear, select Custom Processor. Then, use the **Device type** parameter to specify the microprocessor.

- For complete lists of **Device vendor** and **Device type** values, see “Device vendor” and “Device type”.
- To add **Device vendor** and **Device type** values to the default set that is displayed on the **Hardware Implementation** pane, see “Register New Hardware Devices” on page 11-13.

Specify the Device Type

To specify the microprocessor name from the supported devices listed for your **Device vendor** selection, use the **Device type** parameter. If the microprocessor does not appear in the menu, change **Device vendor** to Custom Processor. Then, specify device details for your custom device.

If you select a device type for which the system target file specifies default hardware properties, the properties appear as initial values. You cannot change the value of parameters with only one possible selection. Parameters that have more than one possible value provide a menu. Select values for your hardware.

Set Bit Lengths for Device Data Types

The **Number of bits** parameters describe the **native word size** of the microprocessor and the bit lengths of **char**, **short**, **int**, and **long** data. For code generation to succeed:

- The bit lengths must be such that **char** <= **short** <= **int** <= **long**.
- Bit lengths must be multiples of 8, with a maximum of 32.
- The bit length for **long** data must not be less than 32.

The `rtwtypes.h` file defines integer type names. The values that you provide must be consistent with the word sizes as defined in the compiler `limits.h` header file. The code generator maps its integer type names to the corresponding Simulink integer type names.

If no ANSI® C type with a matching word size is available, but a larger ANSI C type is available, the code generator uses the larger type for `int8_T`, `uint8_T`, `int16_T`, `uint16_T`, `int32_T`, and `uint32_T`. When the code generator uses a larger type, the resulting logged values (for example, MAT-file logging) can have different data types than logged values for simulation.

An application can use an integer data of length from 1 (unsigned) or 2 (signed) bits up to 32 bits. If the integer length matches the length of an available type, the code generator uses that type. If a matching type is not available, the code generator uses the smallest available type that can hold the data, generating code that does not use unnecessary higher-order bits. For example, on hardware that supports 8-bit, 16-bit, and 32-bit integers, for a signal specified as 24 bits, the code generator implements the data as an `int32_T` or `uint32_T`.

Code that uses emulated integer data is not maximally efficient. This code can be useful during application development for emulating integer lengths that are available only on production hardware. Emulation does not affect the results of execution.

During code generation, the software checks the compatibility of model data types with the data types that you specify for production hardware.

- If none of the lengths that you specify for production hardware integers is 32 bits, the software generates an error.
- If the lengths of data types that the model uses are smaller than the available production hardware integer lengths, the software generates a warning.

Mapping of Integer Types from Code Generator to Simulink

Code Generator Integer Type	Simulink Integer Type
boolean_T	boolean
int8_T	int8
uint8_T	uint8
int16_T	int16
uint16_T	uint16
int32_T	int32
uint32_T	uint32

Set Byte Ordering for Device

The **Byte ordering** parameter specifies whether the hardware uses **Big Endian** (most significant byte first) or **Little Endian** (least significant byte first) byte ordering. If left as **Unspecified**, the code generator produces code that determines the endianness of the hardware. This setting is the least efficient.

Set Quotient Rounding Behavior for Signed Integer Division

ANSI C does not completely define the quotient rounding technique for compilers to use when dividing one signed integer by another. So, the behavior is implementation-dependent. If both integers are positive, or both are negative, the quotient must round down. If either integer is positive and the other is negative, the quotient can round up or down.

The **Signed integer division rounds to** parameter instructs the code generator about how the compiler rounds the result of signed integer division. Providing this information does not change the operation of the compiler. It only describes that behavior to the code generator, which uses the information to optimize code generated for signed integer division. The parameter values are:

- **Zero** — If the quotient is between two integers, the compiler chooses the integer that is closer to zero as the result.
- **Floor** — If the quotient is between two integers, the compiler chooses the integer that is closer to negative infinity.
- **Undefined** — If **Zero** or **Floor** do not describe the compiler behavior or if that behavior is unknown, choose this value.

Avoid selecting **Undefined**. When the code generator does not know the signed integer division rounding behavior of the compiler, the model build generates extra code.

The compiler quotient rounding behavior varies according to these values.

You can obtain the compiler implementation for signed integer division rounding from the compiler documentation. If documentation is not available, you can determine this behavior by experiment.

Example Quotient Rounding for Zero, Floor, and Undefined

N	D	Ideal N/D	Zero	Floor	Undefined
33	4	8.25	8	8	8
-33	4	-8.25	-8	-9	-8 or -9
33	-4	-8.25	-8	-9	-8 or -9
-33	-4	8.25	8	8	8 or 9

Set Arithmetic Right Shift Behavior for Signed Integers

ANSI C does not define the behavior of right shifts on negative integers for compilers. So, the behavior is implementation-dependent. The **Shift right on a signed integer as arithmetic shift** option instructs the code generator about how the compiler implements right shifts on negative integers. Providing this information does not change the operation of the compiler. It only describes that behavior to the code generator, which uses the information to optimize the code generated for arithmetic right shifts.

If the C compiler implements a signed integer right shift as an arithmetic right shift, select the option. Otherwise, clear the option. An arithmetic right shift fills bits vacated by the right shift with the value of the most significant bit, which indicates the sign of the number in two's-complement notation. The option is selected by default. If your compiler handles right shifts as arithmetic shifts, this setting is preferred.

- When you select the option, the code generator produces efficient code whenever the Simulink model performs arithmetic shifts on signed integers.
- When the option is cleared, the code generator produces fully portable but less efficient code to implement right arithmetic shifts.

You can obtain the compiler implementation for arithmetic right shifts from the compiler documentation. If documentation is not available, you can determine this behavior by experiment.

Update Release 14 Hardware Configuration

If your model was created before Release 14 and you have not updated the model, the **Configure current execution hardware device** parameter (TargetUnknown) value is 'on' by default.

To update your model, clear the box for **Configuration Parameters > Hardware Implementation > Advanced parameters > Test hardware > Configure test hardware**. Or in the Command Window, type:

```
cs = getActiveConfigSet('your_model_name');
set_param(cs, 'TargetUnknown', 'off');
```

This update to your model:

- Enables the **Test Hardware is the same as production hardware** parameter (ProdEqTarget), setting the parameter to 'on'.
- Copies the **Production device vendor and type** parameter (ProdHWDeviceType) value to the **Test device vendor and type** parameter (TargetHWDeviceType).

To complete the update:

- 1 Clear the box for **Configuration Parameters > Hardware Implementation > Advanced parameters > Test hardware > Test Hardware is the same as production hardware**. Apply this step only if your production and test hardware are different.
- 2 Set the parameters in **Configuration Parameters > Hardware implementation > Advanced parameters** to match your production and test systems.
- 3 Save the model.

Production Hardware Considerations

When you configure production hardware, consider these points:

- Production hardware can have word sizes and other hardware characteristics that differ from the MATLAB development computer. You can prototype code on hardware that is different from the production hardware or the MATLAB development computer. When producing code, the code generator accounts for these differences.
- The Simulink product uses some of the information in the production hardware configuration. That information enables simulations without code generation to give the same results as executing generated code. For example, the results can detect error conditions that arise on the production hardware, such as hardware overflow.
- The code generator produces code that provides bit-true agreement with Simulink results for integer and fixed-point operations. Generated code that emulates unavailable data lengths runs less efficiently than without emulation. The emulation does not affect bit-true agreement with Simulink for integer and fixed-point results.
- If you change run-time environments during application development, before generating or regenerating code, reconfigure the hardware implementation parameters for the new run-time environment. When code executes on hardware for which it was not generated, bit-true agreement is not always achieved for results of integer and fixed-point operations in simulation, production code, and test code.
- To compile code generated from the model, use the **Integer rounding mode** parameter on model blocks to simulate the rounding behavior of the C compiler that you intend. This setting appears on the **Signal Attributes** pane in the parameter dialog boxes of blocks that can perform signed integer arithmetic, such as the Product, Matrix Multiply and n-D Lookup Table blocks.
- For most blocks, the value of **Integer rounding mode** completely defines rounding behavior. For blocks that support fixed-point data and the simplest rounding mode, the value of **Signed integer division rounds to** also affects rounding. For details, see “Precision” (Fixed-Point Designer).
- When models contain Model blocks, configure models that they reference to use identical hardware settings.

Test Hardware Considerations

By default, the test hardware configuration is the same as the configuration for the production hardware. You can use the generated code for testing in an environment that is identical to the production environment.

If the test and production environments differ, you can generate code that runs on test hardware as if it were running on production hardware:

- 1 To enable test hardware parameters, clear the box for **Configuration Parameters > Hardware Implementation > Advanced parameters > Test hardware > Test hardware is the same as production hardware**. Or, in the Command Window type:

```
cs = getActiveConfigSet('your_model_name');
set_param(cs, 'ProdEqTarget', 'off');
```

- 2 Specify device type details through the test hardware (Target*) parameters.

If you select a system target file that specifies a default microprocessor and its hardware properties, these default values and properties appear as initial values.

Parameters with only one possible value cannot be changed. If you modify hardware properties, check that their values are consistent with the system target file. Otherwise, the generated code can fail to compile or execute, or can execute but produce incorrect results.

Example Production Hardware Setting That Affects Normal Mode Simulation

Changing some production hardware settings, for example, `ProdLongLongMode` and `ProdIntDivRoundTo`, can affect normal mode simulation results. The following example simulates an adder with four inputs. In the first simulation, `ProdLongLongMode` is disabled. In the second simulation, `ProdLongLongMode` is enabled. In the plot of simulation outputs, you observe small differences between output values in the time step range 125–175.

```
model = 'hwSettingEffect';
new_system(model)
open_system(model)

% Create adder
pos = [140 140 200 340];
add_block('simulink/Math Operations/Add', ...
    [model '/sum_int32'], ...
    'Inputs','+++',' ...
    'SaturateOnIntegerOverflow', ...
    'on', ...
    'Position', ...
    pos)

pos = [75 155 105 175];
add_block('built-in/Inport',[model '/In1'],'Position',pos)
set_param([model '/In1'], 'OutDataTypeStr', ...
    'int32','PortDimensions','1','SampleTime','1');
add_line(model, 'In1/1','sum_int32/1')

pos = [75 205 105 225];
add_block('built-in/Inport',[model '/In2'],'Position',pos)
set_param([model '/In2'], 'OutDataTypeStr', ...
    'int32','PortDimensions','1','SampleTime','1');
add_line(model, 'In2/1','sum_int32/2')

pos = [75 255 105 275];
add_block('built-in/Inport',[model '/In3'],'Position',pos)
set_param([model '/In3'], 'OutDataTypeStr', ...
    'int32','PortDimensions','1','SampleTime','1');
add_line(model, 'In3/1','sum_int32/3')

pos = [75 305 105 325];
add_block('built-in/Inport',[model '/In4'],'Position',pos)
set_param([model '/In4'], 'OutDataTypeStr', ...
    'int32','PortDimensions','1','SampleTime','1');
add_line(model, 'In4/1','sum_int32/4')

pos = [275 230 305 250];
add_block('built-in/Outport',[model '/Out1'],'Position',pos)
```

```
add_line(model, 'sum_int32/1','Out1/1')

% Specify input data
t = 0:200;
peakValue = 1.5e9;
in1 = peakValue * sin(t*2*pi/100);
in2 = peakValue * cos(t*2*pi/70);
in3 = -peakValue * sin(t*2*pi/40);
in4 = -peakValue * cos(t*2*pi/30);
set = Simulink.SimulationData.Dataset;
set = set.addElement(1, timeseries(int32(in1),t,'Name','sig1'));
set = set.addElement(2, timeseries(int32(in2),t,'Name','sig2'));
set = set.addElement(3, timeseries(int32(in3),t,'Name','sig3'));
set = set.addElement(4, timeseries(int32(in4),t,'Name','sig4'));

set_param(model, 'LoadExternalInput', 'on');
set_param(model, 'ExternalInput', 'set');

set_param(model, 'StopTime', '50');

% Disable production hardware setting and run first simulation
set_param(model, 'ProdLongLongMode', 'off');
[~,~, y1] = sim(model, 200);

% Enable production hardware setting and run second simulation
set_param(model, 'ProdLongLongMode', 'on');
[~,~, y2] = sim(model, 200);

plot([y1 y2]);
figure(gcf);
```

The difference in behavior is due to the accumulator data type in the Sum block. The **Accumulator data type** block parameter is set to `Inherit`: `Inherit` via `internal` rule. For this example, the resulting accumulator data type is 64 bits wide if the use of the C `long long` data type is enabled. Otherwise, it is 32 bits wide. Depending on the input values for the sum block, the 32-bit accumulator can saturate when the 64-bit accumulator does not. Therefore, normal mode behavior can depend on the `ProdLongLongMode` setting. In both cases, the normal mode behavior and production hardware behavior matches bitwise.

See Also

More About

- “Hardware Implementation Pane”
- “Device vendor”
- “Device type”
- “Precision” (Fixed-Point Designer)

Using the Target Framework

The Target Framework enables you to define target hardware characteristics and integrate tools with MATLAB and Simulink. Using classes from the `target` package, capture target hardware or software platform information for MathWorks software. Perform tasks to optimize, prototype, verify, and deploy design components.

Target Framework Overview

Using target objects, you can capture information for MathWorks software. For example, to register a new hardware device for code generation, use this workflow:

- 1 Create a `target.Processor` object for the new hardware device.
- 2 Create a `target.LanguageImplementation` object for language implementation details.
- 3 Specify language implementation details.
- 4 Associate the language implementation with the hardware device.
- 5 Add the `target.Processor` object to an internal database.
- 6 On the **Hardware Implementation** pane, use the **Device vendor** and **Device type** settings to specify the hardware device as the target device for code generation.

For more information, see “Specify Hardware Implementation for New Device” on page 11-13.

To create and manage the objects, use these functions:

- `target.create` -- Create an object that captures component information for MathWorks software, for example, target processor and compiler properties.
- `target.add` -- Add the object to an internal database.
- `target.update` -- Update the target object in the internal database.
- `target.get` -- Retrieve the object from the internal database.
- `target.export` -- Export the object data for sharing.
- `target.upgrade` -- Upgrade existing definitions of hardware devices.
- `target.remove` -- Remove the object from the internal database.
- `target.clear` -- Clear all target objects from internal database.

Target Framework Workflows

This table lists tasks that you can perform by using `target` classes and functions.

Task	See
Register new hardware devices and hardware boards for Hardware Implementation settings.	“Register New Hardware Devices” on page 11-13
Build a target application using a CMake-based toolchain.	“Create Custom CMake Toolchain Definition” on page 31-17
Deploy, download, and run an application on target hardware.	“Set Up PIL Connectivity by Using target Package” (Embedded Coder)

Task	See
Tune parameters and monitor target application signals during an external mode simulation.	“Set Up External Mode Connectivity Between Simulink and Target Hardware” on page 32-35 “External Mode Simulation by Using XCP Communication” on page 32-8 “Customize XCP Server Software” on page 32-45
Test numerical equivalence between your MATLAB code or Simulink model and the generated code by running a processor-in-the-loop (PIL) simulation.	“Set Up PIL Connectivity by Using target Package” (Embedded Coder) “Configure and Run PIL Simulation” (Embedded Coder)
Perform code execution profiling through a PIL simulation.	“Set Up PIL Connectivity by Using target Package” (Embedded Coder) “Create Execution-Time Profile for Generated Code” (Embedded Coder)

See Also

target

Related Examples

- “Custom Targets” on page 46-2
- “Register New Hardware Devices” on page 11-13
- “Create Custom CMake Toolchain Definition” on page 31-17
- “Set Up External Mode Connectivity Between Simulink and Target Hardware” on page 32-35
- “Set Up PIL Connectivity by Using target Package” (Embedded Coder)

Register New Hardware Devices

On the **Hardware Implementation** pane, you can specify parameters that describe target hardware and compiler properties for MATLAB software, which enables you to:

- Observe target hardware during model simulations.
- Generate optimized code for production or test hardware.
- Directly test or deploy generated code on target hardware.

The **Hardware Implementation** pane supports a range of target hardware. To extend the range, register new hardware devices by using the `target.Processor` and `target.LanguageImplementation` classes.

Specify Hardware Implementation for New Device

To register a new hardware device:

- 1 Create a `target.Processor` object for the new hardware device.

```
myProc = target.create('Processor', ...
    'Name', 'MyProcessor', ...
    'Manufacturer', 'MyManufacturer');
```

- 2 Create a `target.LanguageImplementation` object for language implementation details.

```
myLanguageImplementation = target.create('LanguageImplementation', ...
    'Name', 'MyProcessorImplementation');
```

- 3 Specify language implementation details.

```
myLanguageImplementation.Endianess = target.Endianess.Little;

myLanguageImplementation.AtomicIntegerSize = 64;
myLanguageImplementation.AtomicFloatSize = 64;
myLanguageImplementation.WordSize = 64;

myLanguageImplementation.DataTypes.Char.Size = 8;
myLanguageImplementation.DataTypes.Short.Size = 16;
myLanguageImplementation.DataTypes.Int.Size = 32;
myLanguageImplementation.DataTypes.Long.Size = 64;
myLanguageImplementation.DataTypes.LongLong.IsSupported = true;
myLanguageImplementation.DataTypes.LongLong.Size = 64;
myLanguageImplementation.DataTypes.Float.Size = 32;
myLanguageImplementation.DataTypes.Double.Size = 64;

myLanguageImplementation.DataTypes.Pointer.Size = 32;

myLanguageImplementation.DataTypes.SizeT.Size = 64;
myLanguageImplementation.DataTypes.PtrDiffT.Size = 64;
```

- 4 Associate the language implementation with the hardware device.

```
myProc.LanguageImplementations = myLanguageImplementation;
```

- 5 Add the `target.Processor` object to an internal database.

```
objectsAdded = target.add(myProc);
```

On the **Hardware Implementation** pane, you can now set **Device vendor** and **Device type** to `MyManufacturer` and `MyProcessor` respectively.

Specify Hardware Implementation That Persists Over MATLAB Sessions

By default, when you add the target object to the internal database, the target data is available only for the current MATLAB session. You can specify target data persistence over MATLAB sessions.

- 1 Create a `target.Processor` object for a new hardware device.

```
myProc = target.create('Processor', ...
                      'Name', 'MyProcessor', ...
                      'Manufacturer', 'MyManufacturer');

existingImplementation = target.get('LanguageImplementation', ...
                                  'ARM Compatible-ARM Cortex');
myProc.LanguageImplementations = existingImplementation;
```

- 2 Add the `target.Processor` object to an internal database and specify persistence of target data over MATLAB sessions.

```
objectsAdded = target.add(myProc, 'UserInstall', true);
```

If you subsequently modify the object in the MATLAB workspace and then want to update the object in the internal database, you can use the `target.update` function.

- 3 You can remove the object from the internal database.

```
target.remove(objectsAdded);
```

To remove multiple persistent objects from the internal database, use the `target.clear` function.

Create Hardware Implementation by Modifying Existing Implementation

If an existing hardware implementation contains most of the values that you want in a new hardware implementation, you can quickly create the new implementation by creating and modifying a copy of the existing implementation.

- 1 Create a `target.Processor` object for the new hardware device.

```
myProc = target.create('Processor', ...
                      'Name', 'MyProcessor', ...
                      'Manufacturer', 'MyManufacturer');
```

- 2 Create a `target.LanguageImplementation` object that copies an existing language implementation.

```
myCopiedImplementation = target.create('LanguageImplementation', ...
                                      'Name', 'MyCopiedImplementation', ...
                                      'Copy', 'Atmel-AVR');
```

- 3 Specify the required language implementation details. For example, byte ordering.

```
myCopiedImplementation.Endianness = target.Endianness.Big;
```

- 4 Associate the language implementation with the hardware device.

```
myProc.LanguageImplementations = myCopiedImplementation;
```

- 5 Add the `target.Processor` object to an internal database.

```
objectsAdded = target.add(myProc);
```


Create Hardware Implementation by Reusing Existing Implementation

If your hardware device requires the same hardware implementation as an existing implementation, you can reuse the existing implementation.

- 1 Create a `target.Processor` object for the new hardware device.

```
myProc = target.create('Processor', ...
                      'Name', 'MyProcessor', ...
                      'Manufacturer', 'MyManufacturer');
```

- 2 Retrieve the existing implementation by using the identifier for the device vendor and type, for example, 'ARM Compatible-ARM Cortex'.

```
existingImplementation = target.get('LanguageImplementation', ...
                                  'ARM Compatible-ARM Cortex');
```

- 3 Associate the language implementation with the hardware device.

```
myProc.LanguageImplementations = existingImplementation;
```

- 4 Add the `target.Processor` object to an internal database.

```
objectsAdded = target.add(myProc);
```

Validate Hardware Device Data

To validate the data integrity of target objects, use the `IsValid` property or `validate` method of the `target.Object` base class.

Consider an example where you create a `target.Processor` object and associate an existing language implementation with the object.

```
myProcessor = target.create('Processor');
myProcessor.LanguageImplementations = target.get('LanguageImplementation', ...
                                                'ARM Compatible-ARM Cortex');
```

To validate the created object, run `myProcessor.IsValid` or `myProcessor.validate()`.

```
myProcessor.IsValid
```

```
ans =
    logical
     0
```

```
myProcessor.validate()
```

```
Error using target.Processor/validate
Target data validation failed.
* Undefined property "Name" in "Processor" object.
* Undefined identifier in "Processor" object.
```

The validation fails because these `target.Processor` properties are not specified:

- `Name` — Processor name
- `Id` — Object identifier

You can specify a processor name, which also specifies the object identifier.

```
myProcessor.Name = 'MyProcessor';
```

Check the validity of `myProcessor`.

```
myProcessor.IsValid
```

```
ans =
    logical
     1
```

The validity of the object is established.

Note When you use the `target.add` function to register a target object, the software also checks the validity of the object.

Export Hardware Device Data

You can share previously created hardware device data across computers and users.

For this example, specify a hardware device and add it to an internal database.

```
myProc = target.create('Processor', ...
    'Name', 'MyProcessor', ...
    'Manufacturer', 'MyManufacturer');
existingImplementation = target.get('LanguageImplementation', ...
    'ARM Compatible-ARM Cortex');
myProc.LanguageImplementations = existingImplementation;

objectsAdded = target.add(myProc);
```

To create a function for sharing the hardware device data, run:

```
target.export(myProc, 'FileName', 'exportMyProcFunction')
```

The `target.export` function creates `exportMyProcFunction.m` in the current working folder.

```
function registeredObjects = exportMyProcFunction(varargin)
% This function was generated using target data export.

% Create target.Processor "MyManufacturer-MyProcessor"
processor = target.create("Processor");
processor.LanguageImplementations(1) = ...
    target.get("LanguageImplementation", "ARM Compatible-ARM Cortex");
processor.Manufacturer = "MyManufacturer";
processor.Name = "MyProcessor";

% Add the target objects to MATLAB memory
registeredObjects = target.add(processor, varargin{:});
```

Now, you can use the generated function to share the hardware device data in your database across computers and users. For example, on another computer, run this command.

```
objectsAdded = exportMyProcFunction;
```

The generated function recreates the `target.Processor` object, `MyManufacturer-MyProcessor`, and adds it to an internal database.

Create Alternative Identifier for Target Object

To create alternative identifiers for target objects, use the `target.Alias` class.

For example, if a `target.Processor` object has a long class identifier, you can create a `target.Alias` object that provides a short identifier for the `target.Processor` object.

- 1 Retrieve the `target.Processor` object.

```
processorObj = target.get('Processor', ...
    'Analog Devices-ADSP-CM40x (ARM Cortex-M)');
```

- 2 Use the `target.create` function to create a `target.Alias` object.

```
aliasProcessorObj = target.create('Alias');
```

- 3 Use `target.Alias` object properties to specify the alternative identifier and original target object.

```
aliasProcessorObj.Name = 'myShortName';
aliasProcessorObj.For = processorObj;
```

- 4 Add the `target.Alias` object to an internal database.

```
target.add(aliasProcessorObj);
```

- 5 To retrieve the original `target.Processor` object, run:

```
target.get('Processor', 'myShortName');
```

Upgrade Data Definitions for Hardware Devices

To upgrade existing hardware device definitions that are specified through `rtwTargetInfo.m` or `sl_customization.m` files, use the `target.upgrade` function.

rtwTargetInfo.m File

Suppose you have the hardware device definition in an `rtwTargetInfo.m` file:

```
function rtwTargetInfo(tr)

    % Add registration function handle to the Target Registry
    tr.registerTargetInfo(@loc_register_hardware);
end

function hw = loc_register_hardware
    hw = RTW.HWDeviceRegistry;
    hw.Vendor = 'MyManufacturer';
    hw.Type = 'MyDevice';
    hw.Alias = {};
    hw.Platform = {'Prod', 'Target'};
    hw.setWordSizes([8 16 32 64 64 64 64 64 64 64]);
    hw.Endianess = 'Little';
    hw.IntDivRoundTo = 'Zero';
    hw.ShiftRightIntArith = true;
    hw.LargestAtomicInteger = 'Long';
    hw.LargestAtomicFloat = 'Double';
end
```

To upgrade the data definitions contained in the file, run:

```
target.upgrade('rtwTargetInfo', 'myPathTo/rtwTargetInfo.m');
```

In the current folder, the function creates this `registerUpgradedTargets.m` file:

```
function processor = registerUpgradedTargets(varargin)
% This function was generated using target data export.

    % Create target.LanguageImplementation 'MyManufacturer-MyDevice'
    languageimplementation = target.create('LanguageImplementation');
    languageimplementation.AtomicFloatSize = 64;
    languageimplementation.AtomicIntegerSize = 64;
    languageimplementation.DataTypes.Char.Size = 8;
    languageimplementation.DataTypes.Double.Size = 64;
    languageimplementation.DataTypes.Float.Size = 64;
    languageimplementation.DataTypes.Half.IsSupported = false;
    languageimplementation.DataTypes.Half.Size = 16;
    languageimplementation.DataTypes.Int.Size = 32;
    languageimplementation.DataTypes.Long.Size = 64;
```

```

languageimplementation.DataTypes.LongLong.IsSupported = false;
languageimplementation.DataTypes.LongLong.Size = 64;
languageimplementation.DataTypes.Pointer.Size = 64;
languageimplementation.DataTypes.PtrDiffT.Size = 64;
languageimplementation.DataTypes.Short.Size = 16;
languageimplementation.DataTypes.SizeT.Size = 64;
languageimplementation.Name = 'MyManufacturer-MyDevice';
languageimplementation.WordSize = 64;

% Create target.Processor 'MyManufacturer-MyDevice'
processor = target.create('Processor');
processor.LanguageImplementations(1) = languageimplementation;
processor.Manufacturer = 'MyManufacturer';
processor.Name = 'MyDevice';

% Add the target objects to MATLAB memory
target.add(processor, varargin{:});
end

```

To register the hardware device with MATLAB, run:

```
registerUpgradedTargets()
```

If you want the registration to persist across MATLAB sessions, run:

```
registerUpgradedTargets('UserInstall', true)
```

sl_customization.m File

Suppose you have multiple hardware device definitions in an `sl_customization.m` file:

```

function sl_customization(cm)
% sl_customization function to register a device
% vendor and type with Simulink.
% Copy or rename this file to sl_customization.m.
cm.registerTargetInfo(@loc_register_device);
cm.registerTargetInfo(@loc_register_device2);
cm.registerTargetInfo(@loc_createConfig);

cm.registerTargetInfo(@locRegisterTfl);
cm.CodeCoverageTools.add('DummyCoverageToolForTesting',...
    'HDummyCovTool',...
    'A Coverage Tool Vendor');
end

function thisDev = loc_register_device
thisDev = RTW.HWDeviceRegistry;
thisDev.Vendor = 'MyDevVendor';
thisDev.Type = 'MyDevType';
thisDev.Alias = {};
thisDev.Platform = {'Prod', 'Target'};
thisDev.setWordSizes([8 16 32 32 32]);
thisDev.LargestAtomicInteger = 'Char';
thisDev.LargestAtomicFloat = 'None';
thisDev.Endianess = 'Unspecified';
thisDev.IntDivRoundTo = 'Undefined';
thisDev.ShiftRightIntArith = true;
thisDev.setEnabled({'IntDivRoundTo'});
end

function thisDev = loc_register_device2
thisDev = RTW.HWDeviceRegistry;
thisDev.Vendor = 'MyDevVendor';
thisDev.Type = 'MyDevType2';
thisDev.Alias = {};
thisDev.Platform = {'Prod', 'Target'};
thisDev.setWordSizes([8 16 32 32 32]);
thisDev.LargestAtomicInteger = 'Char';
thisDev.LargestAtomicFloat = 'None';
thisDev.Endianess = 'Unspecified';
thisDev.IntDivRoundTo = 'Undefined';

```

```

    thisDev.ShiftRightIntArith = true;
    thisDev.setEnabled({'IntDivRoundTo'});
end

% local function
function config = loc_createConfig
    config = rtw.connectivity.ConfigRegistry;
    config.ConfigName = 'Infineon->C16x, XC16x';
    config.ConfigClass = 'pil_slcust.HostDemoConfig1';
    config.SystemTargetFile = {'custom_target.tlc'};
    config.TemplateMakefile = {'custom_target.tmf'};
    config.TargetHWDeviceType = {'Infineon->C16x, XC16x'};
end

function thisTfl = locRegisterTfl
    thisTfl(1) = RTW.TflRegistry;
    thisTfl(1).Name = 'myTFL1';
    thisTfl(1).Description = 'Test';
    thisTfl(1).TableList = {'tfl_table_Sum',...
                           'tfl_table_Product',...
                           }; % Sum includes Add and Subtract
    thisTfl(1).BaseTfl = 'ANSI_C';
    thisTfl(1).TargetHWDeviceType = {'*'};
end

```

To upgrade the RTW.HWDeviceRegistry data definitions in the file, run:

```
target.upgrade('sl_customization', 'myPathTo/sl_customization.m')
```

In the current folder, the function creates this registerUpgradedTargets.m file:

```

function targetObjects = registerUpgradedTargets(varargin)
% This function was generated using target data export.

% Create target.LanguageImplementation 'MyDevVendor-MyDevType'
languageimplementation = target.create('LanguageImplementation');
languageimplementation.AtomicIntegerSize = 8;
languageimplementation.DataTypes.Char.Size = 8;
languageimplementation.DataTypes.Double.Size = 64;
languageimplementation.DataTypes.Float.Size = 32;
languageimplementation.DataTypes.Half.IsSupported = false;
languageimplementation.DataTypes.Half.Size = 16;
languageimplementation.DataTypes.Int.Size = 32;
languageimplementation.DataTypes.Long.Size = 32;
languageimplementation.DataTypes.LongLong.IsSupported = false;
languageimplementation.DataTypes.LongLong.Size = 64;
languageimplementation.DataTypes.Pointer.Size = 32;
languageimplementation.DataTypes.PtrDiffT.Size = 32;
languageimplementation.DataTypes.Short.Size = 16;
languageimplementation.DataTypes.SizeT.Size = 32;
languageimplementation.Endianess = target.Endianess.Unspecified;
languageimplementation.Name = 'MyDevVendor-MyDevType';
languageimplementation.WordSize = 32;

% Create target.Processor 'MyDevVendor-MyDevType'
processor = target.create('Processor');
processor.LanguageImplementations(1) = languageimplementation;
processor.Manufacturer = 'MyDevVendor';
processor.Name = 'MyDevType';

% Create target.LanguageImplementation 'MyDevVendor-MyDevType2'
languageimplementation2 = target.create('LanguageImplementation');
languageimplementation2.AtomicIntegerSize = 8;
languageimplementation2.DataTypes.Char.Size = 8;
languageimplementation2.DataTypes.Double.Size = 64;
languageimplementation2.DataTypes.Float.Size = 32;
languageimplementation2.DataTypes.Half.IsSupported = false;
languageimplementation2.DataTypes.Half.Size = 16;
languageimplementation2.DataTypes.Int.Size = 32;
languageimplementation2.DataTypes.Long.Size = 32;
languageimplementation2.DataTypes.LongLong.IsSupported = false;
languageimplementation2.DataTypes.LongLong.Size = 64;

```

```
languageimplementation2.DataTypes.Pointer.Size = 32;
languageimplementation2.DataTypes.PtrDiffT.Size = 32;
languageimplementation2.DataTypes.Short.Size = 16;
languageimplementation2.DataTypes.SizeT.Size = 32;
languageimplementation2.Endianess = target.Endianess.Unspecified;
languageimplementation2.Name = 'MyDevVendor-MyDevType2';
languageimplementation2.WordSize = 32;

% Create target.Processor 'MyDevVendor-MyDevType2'
processor2 = target.create('Processor');
processor2.LanguageImplementations(1) = languageimplementation2;
processor2.Manufacturer = 'MyDevVendor';
processor2.Name = 'MyDevType2';

targetObjects = [processor, processor2];

% Add the target objects to MATLAB memory
target.add(targetObjects, varargin{:});
end
```

To register the hardware device definitions with MATLAB, run:

```
registerUpgradedTargets()
```

If you want the registration to persist across MATLAB sessions, run:

```
registerUpgradedTargets('UserInstall', true)
```

See Also

[target.LanguageImplementation](#) | [target.Processor](#)

More About

- “Configure Run-Time Environment Options” on page 11-2

Model Protection in Simulink Coder

- “Protect Models to Conceal Contents” on page 12-2
- “Create Protected Models with Multiple Targets” on page 12-9
- “Test Protected Models” on page 12-10
- “Package and Share Protected Models” on page 12-12
- “Specify Custom Obfuscators for Protected Models” on page 12-16
- “Define Callbacks for Protected Models” on page 12-17
- “Sign a Protected Model” on page 12-19
- “Best Practices for Creating Protected Models” on page 12-21

Protect Models to Conceal Contents

When you want to share a model with a third party without revealing intellectual property, protect the model. Protecting a model conceals the implementation details of the original model by compiling it into a referenced model. The protected model includes derived files to support the optional functionalities that you specify.

When you protect a model, you can allow the user of the protected model to:

- Simulate a model that includes the protected model in normal, accelerator, or rapid accelerator mode.
- Open a read-only web view of the model, including model contents and block parameters. Creating a read-only web view of the model requires Simulink Report Generator.
- Generate code for a model that includes the protected model.
- Generate HDL code for a model that includes the protected model.
- Generate code for the protected model through the standalone interface, if you have Embedded Coder and specify an ERT-based system target file for the model.

You can optionally password-protect each option. If you choose password protection for one of these options, the software protects the supporting files by using AES-256 encryption.

When you create a protected model:

- Simulink creates and stores a protected version of the model in a file that has the same name as the source model, with an `.slxp` extension.
- The original model file, with the `.slx` extension, does not change. If you protect the model through a Model block, that Model block does not change.
- Optionally, Simulink creates a project archive (`.mlproj`) that contains the protected model, a harness model for the protected model, and additional supporting files.

This example shows how to create a protected model from a referenced model for read-only viewing, simulation, or code generation.

Prepare the Parent Model

Configure the Model blocks in the parent model to refer to the original referenced model. This step prevents the Model blocks from becoming protected references when you create the protected model.

- 1 Open the parent model that references the model you want to protect. For this example, open the model `sldemo_mdhref_basic`.
- 2 To run the workflow, create a local copy of the model `sldemo_mdhref_counter` that you want to protect. You can then create a local copy of the parent model `sldemo_mdhref_basic`. You must save the parent model in the same folder as the referenced model.
- 3 Open the `sldemo_mdhref_basic` model that you saved locally. Make sure that the Model blocks `CounterA`, `CounterB`, and `CounterC` reference the `sldemo_mdhref_counter` model that you saved locally.
- 4 For each Model block, open the Block Parameters dialog box and specify the extension `.slx` in the **Model name** field. When both the model and the protected model exist in the same

folder, `.slxp` takes precedence over `.slx`. If you do not specify an extension, then the original Model block in the model refers to the protected model instead of the original model. Click **OK**.

Protect the Referenced Model

- 1 In the `sldemo_mdhref_basic` model, click any of the three Model blocks. On the Simulink Toolstrip **Model Block** tab, click **Protect**.

The screenshot shows the 'Create Protected Model' dialog box for the model 'sldemo_mdhref_counter'. The dialog is titled 'Create Protected Model: sldemo_mdhref_counter' and has a close button (X) in the top right corner. The 'Description' section states: 'Create a protected model (.slxp) that allows read-only view, simulation, and code generation of the model with optional password protection.' The 'Allow user of protected model to' section contains several options with password input fields:

- Open read-only view of model: Enter password (optional) and Re-enter password (option...)
- Simulate: Enter password (optional) and Re-enter password (option...)
- Use generated code: Enter password (optional) and Re-enter password (option...)
- Content type: Obfuscated source code (dropdown menu)
- Use generated HDL code: Enter password (optional) and Re-enter password (option...)

The 'Tunable parameters for simulation' section is expanded, showing 'Options for saving protected model':

- Destination folder: H:\my_models (with a 'Browse...' button)
- Contents: Protected model (.slxp) and dependencies in a project (dropdown menu)
- Create harness model for protected model
- Name of project archive (.mlproj): sldemo_mdhref_counter_protected

At the bottom, there is a help icon (question mark) and three buttons: 'Create', 'Cancel', and 'Help'.

- 2 In the Create Protected Model dialog box, select the **Simulate** and **Use generated code** check boxes. These options allow the protected model user to simulate and generate code for a model that references the protected model. If you want to password-protect the functionality of the protected model, enter a password with a minimum of eight characters. Each option can have a unique password.
- 3 If you have Embedded Coder and specify an ERT-based system target file (for example, `ert.tlc`) for the model, the **Code interface** field is visible.

In this example, `sldemo_mdhref_basic` does not specify an ERT-based system target file, therefore the **Code interface** options are not available in the Create Protected Model dialog box.

From the **Code interface** drop-down list, select one of these options:

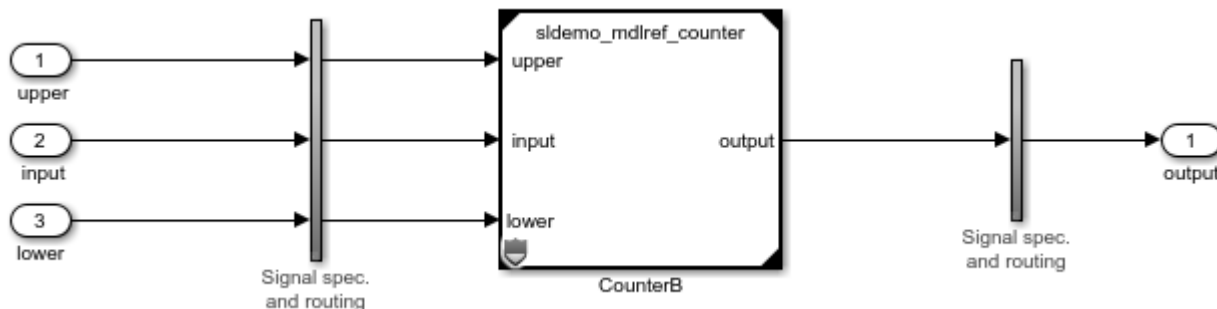
- **Model reference** — Specifies code access through the model reference code interface, which allows use of the protected model within a model reference hierarchy. Users of the protected model can generate code from a parent model that contains the protected model. Users can run Model block SIL/PIL simulations with the protected model.
 - **Top model** — Specifies code access through the standalone interface. Users of the protected model can run Model block SIL/PIL simulations with the protected model.
- 4 From the **Content type** list, select **Obfuscated source code** to conceal the source code purpose and logic of the protected model. For more information on the model protection options, see “Content type”.
 - 5 Expand the **Tunable parameters for simulation** section and select the parameters that you want the protected model user to be able to tune during simulation.

For code generation, parameters with non-auto storage classes are tunable. For more information, see “Storage Classes for Parameters and Signals Used in Model Blocks” on page 8-19.

- 6 In the **Destination folder** box, specify the folder path for the protected model. The default value is the current working folder.
- 7 To automatically collect, create, and package supporting files with the protected model, set **Contents** to **Protected Model (.slxp)** and **dependencies in a project**.

Note Before sharing the project, check whether the project contains the necessary supporting files. If supporting files are missing, simulating or generating code for the related harness model can help identify them. Add the missing dependencies to the project and update the harness model as needed.

- 8 Setting **Contents** to **Protected Model (.slxp)** and **dependencies in a project** selects the **Create harness model for protected model** check box. The harness model is included in the project and provides an isolated environment for the Model block that references the protected model.



- 9 To further customize your protected model, you can:
 - Specify a custom obfuscator for code obfuscation. See “Specify Custom Obfuscators for Protected Models” on page 12-16.
 - Specify multiple code generation targets. See “Create Protected Models with Multiple Targets” on page 12-9.
 - Define callbacks for the protected model. See “Define Callbacks for Protected Models” on page 12-17.
- 10 Click **Create**.

A project archive (.mlproj) that contains the protected model, harness model, and supporting files appears in the specified destination folder. To test the protected model, extract the contents of the project archive by double-clicking it in the Current Folder browser and specifying the destination folder. Alternatively, right-click on the project archive and click **Extract Here**. The project opens, and the extracted files are in a new folder named after the project archive.

- 11 When you create the protected model from the Simulink Editor, a protected model report is generated and included as part of the protected model. For this example, to view the protected model report, double-click the protected model or right-click the protected-model badge icon on the block in the harness model and select **Display Report**.

The screenshot shows a window titled "Protected Model report" with a search bar and "Match Case" option. The left sidebar shows a "Contents" pane with "Summary" selected. The main area displays the report for "sldemo_mdref_counter".

Summary for sldemo_mdref_counter

Environment

Environment information for protected model "sldemo_mdref_counter"

Model Version	1.78
Simulink version	10.1
Simulink Coder Version	9.3 (R2020a) 18-Nov-2019
Protected model generated on	Tue Jan 7 14:33:52 2020
Platform	win64

Configuration settings at the time of protected model creation: [click to open](#)

Supported functionality

Supported functionality for protected model "sldemo_mdref_counter"

Read-only view support	Off
Simulation support	On
Code generation support	On
HDL Code generation support	Off
Concurrent tasking support	Off
Code interface	Model reference
Target	grt
Obfuscation	On
Generated code content type	Obfuscated source code

Buttons: OK, Help

The report contains:

- A **Summary**, including the following tables:
 - **Environment**, providing the Simulink version and other product versions and the platform used to create the protected model.
 - **Supported functionality**, reporting On, Off, or On with password protection for each possible functionality that the protected model supports. If you configure your protected model for multiple targets, this table includes a list of supported targets.

- An **Interface Report**, including model interface information such as input and output specifications, exported function information, interface parameters, and data stores.

The report does not contain other sections that appear in a code generation report for non-protected models, such as a code metrics report and traceability report, because these reports include more information about the model's design.

- 12** You can test the protected model to compare it to the original model. For more information, see “Test Protected Models” on page 12-10.
- 13** To use the protected model in a model hierarchy, reference it through a Model block such as the one included in the harness model. The **Simulation mode** for Model blocks that reference a protected model is set to **Accelerator**. You cannot change the mode. For more information, see “Reference Protected Models from Third Parties”.

Other Methods of Protecting Models

To create a protected model from the current model, use one of these options:

- On the Simulink Toolstrip **Simulation** tab, click the **Save** button arrow, then select **Protected Model**.
- On the Simulink Toolstrip **C Code** tab, click the **Share** button arrow, then select **Generate Protected Model**.

To programmatically create a protected model, use the `Simulink.ModelReference.protect` function.

Protected Model Requirements and Limitations

When you create a protected model, consider these following requirements:

- You must have a Simulink Coder or HDL Coder™ license to create a protected model.
- The model must be available on the MATLAB path.
- The model cannot have unsaved changes.
- The model uses the configuration that is active during protection. You cannot change the configuration of a protected model.
- If the model contains variants, the protected model includes only the variant that is active during protection.
- The model protection process does not preserve callbacks. For more information on creating callbacks for use with a protected model, see “Define Callbacks for Protected Models” on page 12-17
- Do not rename the protected model or change its suffix. If you do so, the model is unusable until you restore its original name and suffix.
- Use a unique name for the model and for models that it references. If a protected model references a model that shares a name with either a different protected model or a different model within the hierarchy of another protected model, there are limitations for using the protected models. If a top model references two protected models that have such a naming conflict, you cannot protect the top model, generate code for the top model, or simulate the top model in software-in-the-loop (SIL), processor-in-the-loop (PIL), or rapid accelerator modes.
- If your model includes a non-inlined S-function, use a unique name for the .MEX file. If there is a name conflict with a different .MEX file on the MATLAB path, you cannot simulate the protected model.

The model must also meet the requirements listed in “Model Reference Requirements and Limitations”.

Code Generation Requirements and Limitations

To create a protected model that supports code generation, the model being protected must meet these requirements:

- The model must use normal, accelerator, software-in-the-loop (SIL), or processor-in-the-loop (PIL) modes and a single target.
- The model must not include noninlined S-functions.
- The model must not have the **Code Generation > Verification > Measure function execution time** configuration parameter selected. If this option is selected when you protect a model, the software turns off the parameter and displays a warning.
- Protected referenced models must support code generation without password protection.
- The model must be compatible with the **Content type** of each protected referenced model. This table provides compatibility information.

Code Generation Content Type Compatibility

Protected Parent Model Content Type	Compatible Protected Referenced Model Content Types
Binaries	<ul style="list-style-type: none"> • Binaries • Obfuscated source code
Binaries with 'ObfuscateCode' set to 'false'	<ul style="list-style-type: none"> • Binaries • Binaries with 'ObfuscateCode' set to 'false' • Obfuscated source code • Readable source code
Obfuscated source code	<ul style="list-style-type: none"> • Obfuscated source code
Readable source code	<ul style="list-style-type: none"> • Obfuscated source code • Readable source code

When the **Content type** of the protected parent model and protected referenced models do not match, code generation applies the **Content type** that provides the higher level of protection. For example, a protected parent model set for **Binaries** generates **Binaries** for protected referenced models that are set to **Obfuscated source code**. A protected parent model set for **Readable source code** generates **Obfuscated source code** for protected referenced models that are set to **Obfuscated source code**.

To avoid an error during code generation of a model that includes a protected model:

- The protected model name must be unique from other model names in the same model reference hierarchy.
- The interfaces must match.
- The parameters must be compatible.

Nested Protected Model Requirements and Limitations

To enable a recipient of your protected model to protect a model that references it, your protected model must:

- Support accelerator mode. You must select **Simulate** in the Create Protected Model dialog box or set 'Mode' to 'Accelerator' or 'CodeGeneration' using the `Simulink.ModelReference.protect` function.
- Use the default 'CodeInterface' setting: 'Model reference'.
- Not use password protection for simulation.
- Not have callbacks.
- Support without password protection the operations that the protected parent model is intended to support.
- Use a **Content type** that is compatible with the intended **Content type** of the protected parent model if the protected parent model is intended to support code generation. For a list of compatible **Content type** options, see the table in the preceding section.
- Use the same compiler as the protected parent model is intended to use.
- Not reference a model that shares a name with either a different protected model or a different model within the hierarchy of another protected model.

If the model that you want to protect references a protected model, the referenced protected model must meet the preceding requirements.

When you protect a model that references a protected model, you must specify the tunable parameters of the referenced protected model as tunable for the model you are protecting.

Note A protected model that you create with the **Use generated HDL code** option selected allows encryption and support for simulation and HDL code generation from a model that references the protected model. You cannot obfuscate the HDL source code, have callbacks, or use nested protected models with this option. To learn more about HDL code generation limitations, see “Protected Model Restrictions for HDL Code Generation” (HDL Coder).

See Also

More About

- “Package and Share Protected Models” on page 12-12
- “Specify Custom Obfuscators for Protected Models” on page 12-16
- “Create Protected Models with Multiple Targets” on page 12-9
- “Define Callbacks for Protected Models” on page 12-17

Create Protected Models with Multiple Targets

You can create a protected model that supports multiple code generation targets. This example shows how to use command-line functions to create a protected model that supports code generation for GRT and ERT targets.

- 1 Load a model and save a local copy. This model is configured for a GRT target.

```
openExample('sldemo_mdhref_counter');
save_system('sldemo_mdhref_counter','mdhref_counter.slx');
```

- 2 Add a required password for modifying a protected model. If you do not add a password, you are prompted to set a password when you create a modifiable, protected model.

```
Simulink.ModelReference.ProtectedModel.setPasswordForModify(...
'mdhref_counter','password');
```

- 3 Create a modifiable, protected model with support for code generation.

```
Simulink.ModelReference.protect('mdhref_counter','Mode',...
'CodeGeneration','Modifiable',true,'Report',true);
```

- 4 Get a list of targets that the protected model supports.

```
st = Simulink.ProtectedModel.getSupportedTargets('mdhref_counter')

st =

    'grt'    'sim'
```

- 5 Configure the unprotected model to support an ERT target.

```
set_param('mdhref_counter','SystemTargetFile','ert.tlc');
save_system('mdhref_counter');
```

- 6 Add support to the protected model for the ERT target. You are prompted for the modification password.

```
Simulink.ProtectedModel.addTarget('mdhref_counter');
```

- 7 Verify that the list of supported targets now includes the ERT target.

```
st = Simulink.ProtectedModel.getSupportedTargets('mdhref_counter')

st =

    'ert'    'grt'    'sim'
```

See Also

More About

- “Protect Models to Conceal Contents” on page 12-2

Test Protected Models

To test a protected model that you created, compare the output of the protected model to the output of the original model. Because you supply the protected model from the original model, both the original and the protected model might exist on the MATLAB path. In the parent model, if the Model block **Model name** parameter names the model without providing a suffix, the protected model takes precedence over the unprotected model. To override this default when testing the output, in the Model block **Model name** parameter, specify the file name with the extension of the unprotected model, `.slx`.

To compare the unprotected and protected versions of a Model block, use the Simulation Data Inspector. This example uses `sldemo_mdhref_basic` and the protected model, `sldemo_mdhref_counter.slxp`, which is created in “Protect Models to Conceal Contents” on page 12-2.

- 1 If it is not already open, open `sldemo_mdhref_basic`.

```
openExample('sldemo_mdhref_basic');
```
- 2 Enable logging for the output signal of the Model block, CounterA. In the Configuration Parameters dialog box, in the **Data Import/Export** pane, select the **Signal logging** parameter. Click **Apply** and **OK**.
- 3 Right-click the output signal. From the context menu, select **Properties**. In the Signal Properties dialog box, select **Log signal data**. Click **Apply** and **OK**. For more information, see “Save Signal Data Using Signal Logging”.
- 4 Right-click the CounterA block. From the context menu, select **Block Parameters (ModelReference)**. In the Block Parameters dialog box, specify the **Model name** parameter with the name of the unprotected model and the extension, `sldemo_mdhref_counter.slx`. Click **Apply** and **OK**. Repeat this step for CounterB block and CounterC block.
- 5 Simulate the model. When the simulation is complete, click the **Data Inspector** button to open the Simulation Data Inspector.
- 6 In the Simulation Data Inspector, rename the run to indicate that it is for the unprotected model.
- 7 In the Simulink Editor, right-click the CounterA block. From the context menu, select **Block Parameters (ModelReference)**. In the Block Parameters dialog box, specify the **Model name** parameter with the name of the protected model, `sldemo_mdhref_counter.slxp`. A badge icon appears on the Model block. Repeat this step for CounterB block and CounterC block.
- 8 Simulate the model, which now refers to the protected model. When the simulation is complete, a new run appears in the Simulation Data Inspector.
- 9 In the Simulation Data Inspector, rename the new run to indicate that it is for the protected model.
- 10 In the Simulation Data Inspector, click the **Compare** tab. From the **Baseline** and **Compare To** lists, select the runs from the unprotected and protected model, respectively. To compare the runs, click **Compare Runs**. For more information about comparing runs, see “Compare Simulation Data”.

See Also

More About

- “Protect Models to Conceal Contents” on page 12-2
- “Package and Share Protected Models” on page 12-12

Package and Share Protected Models

When you protect a model, you can automatically create and package the following contents in a project archive (.mlproj) for easy sharing:

- Protected model file (.slxp)
- Harness model file
- MAT-file with base workspace definitions
- Data dictionary pruned to relevant definitions
- Other supporting files

In the Create Protected Model dialog box, set **Contents** to Protected model (.slxp) and dependencies in a project.

Note Before sharing the project, check whether the project contains the necessary supporting files. If supporting files are missing, simulating or generating code for the related harness model can help identify them. Add the missing dependencies to the project and update the harness model as needed.

Alternatively, you can use one of these options to deliver the protected model package:

- Create a project archive to share a project that contains the protected model file and supporting files. For more information, see “Create a Project from a Model” and “Share Projects”.
- Provide the protected model file and supporting files as separate files.
- Combine the files into a ZIP or other container file.
- Provide the files in some other standard or proprietary format specified by the receiver.

Whichever approach you use to deliver a protected model, include information on how to retrieve the original files.

Harness Model

You can create a harness model when you create your protected model. The harness model contains a Model block that references the protected model. A third party can use the Model block to reference your protected model. The harness model is set up for simulation of the protected model.

MAT-File with Base Workspace Definitions

Referenced models can use object definitions or tunable parameters that are defined in the MATLAB base workspace. These variables are not saved with the model. When you protect a model, you must obtain the definitions of required base workspace entities and ship them with the model.

For example, if the model uses the following base workspace variables, they must be saved to a MAT-file:

- Global tunable parameter
- Global data store
- The following objects used by a signal that connects to a root-level model Inport or Outport:

- `Simulink.Signal`
- `Simulink.Bus`
- `Simulink.Alias`
- `Simulink.NumericType` that is an alias

To determine the required base workspace definitions and save them to a MAT-file, see “Explore Protected Model Capabilities”. Before executing the protected model as a part of a third-party model, the receiver of the protected model must load the MAT-file.

Simulink Data Dictionary

Referenced models can use data definitions from a data dictionary, which are not saved with the model. When you protect a model that uses a data dictionary, package and ship the data dictionary with the protected model.

Protected Model File Contents

A protected model file (`.slxp`) contains derived files that support the options that you select when you create the protected model. The derived files are unpacked when you or a third party use the protected model in simulation or code generation.

The derived files that are unpacked depend on the support that you enable when creating the protected model. The `slprj/sim/model/*` files are deleted after they are used.

This table provides the files that are unpacked depending on the options that you specify.

Supported Functionality	Derived Files
Simulation only and the referencing model simulates in normal mode	<p>The <code>model_msp.mexext</code> file is placed in the build folder. If the model uses a noninlined S-function directly or indirectly, the S-function MEX-files are placed in the build folder.</p> <p>The <code>slprj/sim/model/tmwinternal/*</code> files are unpacked in the <code>slprj/sim/</code> folder.</p>
Simulation only and the referencing model simulates in accelerator or rapid accelerator mode	<p>In addition to the preceding files in the build folder, these files are unpacked in the <code>slprj/sim/</code> folder:</p> <ul style="list-style-type: none"> • <code>slprj/sim/model/*.h</code> • <code>slprj/sim/model/modellib.a</code> (or <code>modellib.lib</code>) • <code>slprj/sim/model/tmwinternal/*</code> • <code>slprj/sim/_sharedutils/*</code> <p>For the protected model report, these additional files are unpacked (but not in the build folder):</p> <ul style="list-style-type: none"> • <code>slprj/sim/model/html/*</code> • <code>slprj/sim/model/buildinfo.mat</code>

Supported Functionality	Derived Files
Code generation	<p>These files are unpacked in the slprj folder after building your model (in addition to the preceding files):</p> <ul style="list-style-type: none"> • slprj/sim/model/*.h • slprj/sim/model/modellib.a (or <i>modellib.lib</i>) • slprj/sim/model/tmwinternal/* • slprj/sim/_sharedutils/* • slprj/target/model/*.h • slprj/target/model/model_rtwlib.a (or <i>model_rtwlib.lib</i>) • slprj/target/model/buildinfo.mat • slprj/target/model/codeinfo.mat • slprj/target/_sharedutils/* • slprj/target/model/tmwinternal/*
Code generation with Top model code interface (requires Embedded Coder license)	<p>These files are unpacked in the slprj folder after building your model (in addition to the preceding files):</p> <ul style="list-style-type: none"> • slprj/sim/model/*.h • slprj/sim/model/modellib.a (or <i>modellib.lib</i>) • slprj/sim/model/tmwinternal/* • slprj/sim/_sharedutils/* • <i>model_target_rtw</i>/*.h • <i>model_target_rtw</i>/*.objExt • <i>model_target_rtw</i>/buildinfo.mat • <i>model_target_rtw</i>/codeinfo.mat • slprj/target/_sharedutils/* • slprj/target/model/tmwinternal/* <p>For the protected model report, after building your model these files are unpacked (in addition to the preceding files):</p> <ul style="list-style-type: none"> • slprj/target/model/html/* • slprj/target/model/buildinfo.mat • slprj/target/_sharedutils/html/*

Supported Functionality	Derived Files
HDL code generation (requires HDL Coder license)	<p>The files are unpacked in the <code>hdlsrc</code> folder:</p> <ul style="list-style-type: none"> • <code>hdlsrc/model/model.vhd</code> (<i>model.v</i> if you specified Verilog as the Target language.) • <code>hdlsrc/model/Subsystem.vhd</code> (<i>Subsystem.v</i> if you specified Verilog as the Target language of the model that you protected. The additional HDL files depend on how hierarchically the referenced model was designed.) • <code>hdlsrc/model/model_pkg.vhd</code> (This file is not generated if you specified Verilog as the Target language of the model that you protected.) • <code>hdlsrc/model/model_report.html</code> • <code>hdlsrc/model/gm_model.slxp</code> (This file is a generated protected model.)

See Also

More About

- “Protect Models to Conceal Contents” on page 12-2

Specify Custom Obfuscators for Protected Models

When creating a protected model, you can specify your own postprocessing function for files that the protected model creation process generates. Before packaging the protected model files, this function is called by the `Simulink.ModelReference.protect` function. You can use this functionality to run your own custom obfuscator on the generated files by following these steps:

- 1 Create your postprocessing function. Use this function to call your custom obfuscator. The function must be on the MATLAB path and accept a `Simulink.ModelReference.ProtectedModel.HookInfo` object as an input variable.
- 2 In your function, get the files and exported symbol information that your custom obfuscator requires to process the protected model files. To get the files and information, access the properties of your function input variable. The variable is a `Simulink.ModelReference.ProtectedModel.HookInfo` object with the following properties:
 - `SourceFiles`
 - `NonSourceFiles`
 - `ExportedSymbols`
- 3 Pass the protected model file information to your custom obfuscator. The following is an example of a postprocessing function for custom obfuscation:

```
function myHook(protectedModelInfo)

    % Get source file list information.
    srcFileList = protectedModelInfo.SourceFiles;
    disp('### Obfuscating...');
    for i=1:length(srcFileList)
        disp(['### Obfuscator: Processing ' srcFileList{i} '...']);
        % call to custom obfuscator
        customObfuscator(srcFileList{i});
    end
end
```

- 4 Specify your postprocessing function when creating the protected model:

```
Simulink.ModelReference.protect('myModel, ...
    'Mode', ...
    'CodeGeneration', ...
    'CustomPostProcessingHook', ...
    @(protectedModelInfo)myHook(protectedModelInfo))
```

The protected model creator can also enable obfuscation of simulation target code and generated code through the `'ObfuscateCode'` option of the `Simulink.ModelReference.protect` function. Your custom obfuscator runs only on the generated code and not on the simulation target code or the generated HDL code. If both obfuscators are in use, the custom obfuscator is the last to run on the generated code before the files are packaged.

See Also

More About

- “Protect Models to Conceal Contents” on page 12-2

Define Callbacks for Protected Models

When you create a protected model, you can customize its behavior by defining callbacks. Callbacks specify code that executes when you view, simulate, or generate code for the protected model. You cannot have protected model callbacks with HDL code generation support enabled for a protected model. To learn more about HDL code generation limitations, see “Protected Model Restrictions for HDL Code Generation” (HDL Coder).

A protected model user cannot view or modify a callback. If a model references a protected model with callbacks, you cannot protect the model.

To create a protected model with callbacks:

- 1 Define `Simulink.ProtectedModel.Callback` objects for each callback.
- 2 To create your protected model, call the `Simulink.ModelReference.protect` function. Use the `'Callbacks'` option to specify a cell array of callbacks to include in the protected model.

Creating Callbacks

To create and define a protected model callback, create a `Simulink.ProtectedModel.Callback` object. Callback objects specify:

- The code to execute for the callback. The code can be a character vector of MATLAB commands or a script on the MATLAB path.
- The event that triggers the callback. The event can be `'PreAccess'` or `'Build'`.
- The protected model functionality that the event applies to. The functionality can be `'CODEGEN'`, `'SIM'`, `'VIEW'`, or `'AUTO'`. If you select `'AUTO'`, and the event is `'PreAccess'`, the callback applies to each functionality. If you select `'AUTO'`, and the event is `'Build'`, the callback applies only to `'CODEGEN'` functionality. If you do not select a functionality, the default behavior is `'AUTO'`.
- The option to override the protected model build process. This option applies only to `'CODEGEN'` functionality.

You can create only one callback per event and per functionality.

Defining Callback Code

You can define the code for a callback by using either a character vector of MATLAB commands or a script on the MATLAB path. When you write callback code, follow these guidelines:

- Callbacks must use MATLAB code (`.m` or `.p`).
- The code can include protected model functions or a MATLAB command that does not require loading the model.
- Callback code must not call out to external utilities unless those utilities are available in the environment where the protected model is used.
- Callback code cannot reference the source protected model unless you are using protected model functions.

You can use the `Simulink.ProtectedModel.getCallbackInfo` function in callback code to get information on the protected model. The function returns a

`Simulink.ProtectedModel.CallbackInfo` object that provides the protected model name and the names of submodels. If the callback is specified for 'CODEGEN' functionality and 'Build' event, the object provides the target identifier and model code interface type ('Top model' or 'Model reference').

Create a Protected Model with Callbacks

This example creates a protected model with a callback for code generation.

- 1 On the MATLAB path, create a callback script, `pm_callback.m`, containing:

```
s1 = 'Code interface is: ';  
cbinfoobj = Simulink.ProtectedModel.getCallbackInfo(...  
    'sldemo_mdref_counter', 'Build', 'CODEGEN');  
disp([s1 cbinfoobj.CodeInterface]);
```

- 2 Create a callback that uses the script.

```
pmCallback = Simulink.ProtectedModel.Callback('Build',...  
    'CODEGEN', 'pm_callback.m');
```

- 3 Create the protected model and specify the code generation callback.

```
Simulink.ModelReference.protect('sldemo_mdref_counter',...  
    'Mode', 'CodeGeneration', 'Callbacks', {pmCallback})
```

- 4 Build the protected model. Before the build, the callback displays the code interface.

```
slbuild('sldemo_mdref_basic')
```

See Also

[Simulink.ProtectedModel.Callback](#) | [Simulink.ProtectedModel.getCallbackInfo](#) | [Simulink.ModelReference.protect](#)

More About

- “Protect Models to Conceal Contents” on page 12-2

Sign a Protected Model

A *digital signature* is a method of verifying information about the origin of a digital message or file. You can digitally sign a protected model to allow the user of the model to verify:

- That the model was created by you, the publisher that signed the model.
- That the model was not changed after you signed it.

By using digital signature validation, the recipient can detect issues, such as an unfamiliar publisher or a tampered model, before opening or executing a protected model.

These parties interact in the digital signing workflow:

- The creator of the protected model
- The recipient who uses the protected model
- An external certificate authority (CA) that the creator and the recipient trust

To exchange a protected model with certificate validation, they use this workflow:

- 1** The certificate authority provides a digital certificate, which proves that a public key belongs to the creator of the model. The certificate authority also provides the corresponding public certificate to the model recipient.
- 2** The creator signs the protected model by using the private key that corresponds to the public key in the certificate. The creator sends the signed protected model to the recipient.
- 3** The recipient verifies the signature on the protected model. Verification compares the signature on the model to the certificate attached to the model and to the certificate of the CA. Verification fails if:
 - The protected model was changed after it was signed.
 - The protected model was not signed.
 - The protected model was signed with an expired certificate.
 - The protected model was self-signed with a certificate issued by the author.
 - The protected model was signed with a certificate for which the key usage does not include digital signing.
 - The protected model was signed with a missing or invalid certificate.
 - The certificate of the CA is missing in your system or is invalid.

As the model creator, you generate a public-private key pair and obtain a certificate for the public key from the CA. Then, you create a PKCS #12 file (.pfx file or .p12 file) that includes the private key and the certificate. Because the PKCS #12 file contains the private key, it is typically password-protected. Follow this workflow to sign your protected model before you send it to the recipient.

Sign Your Protected Model

To sign a protected model, use a certificate that is issued to you by a certificate authority (CA) that you and the user of the model both trust. If you sign a protected model using a self-signed certificate or a certificate issued by an internal CA, the recipient's system must trust the certificate for the signature validation to succeed. Using a self-signed certificate to sign protected models is not recommended.

- 1 Locate the PKCS #12 file that you or your organization created from the private key and the certificate. The file must have a .pfx or .p12 extension.
- 2 Protect the model that you want to sign. The protected version of the model has the extension .slxp.
- 3 Provide the password for the PKCS #12 file.

```
Simulink.ModelReference.ProtectedModel.setPasswordForCertificate('certificate_file.pfx', ...  
'password');
```

- 4 Sign the model by using the PKCS #12 file.

```
Simulink.ProtectedModel.sign('protected_model.slxp', 'certificate_file.pfx');
```

Your protected model is now signed with the PKCS #12 file. Do not change the protected model after you sign it. If you or another party changes the signed protected model, verification of the signature fails and the protected model cannot be used.

The recipient can verify the protected model by using the function `Simulink.ProtectedModel.verifySignature` or by selecting the Simulink preference Verify digital signature of protected model before opening. For more information, see “Reference Protected Models from Third Parties”.

See Also

`Simulink.ProtectedModel.sign`

More About

- “Protect Models to Conceal Contents” on page 12-2
- “Reference Protected Models from Third Parties”

Best Practices for Creating Protected Models

When you want to conceal the implementation details of a model that you share with a third party, create a protected model. Instead of including your original model files, the protected model includes derived files to support the functionalities that you specify. To create a protected model that supports functionalities that your user requires and that includes minimal information about your model, follow these best practices. The settings that you choose for the model before and during protection specify what the user of the protected model can do with it. Consider the trade-offs of different design options. To support additional functionalities, the protected model must include more content and therefore, more elements of your design might be visible. Because the protected model uses derived artifacts from a compiled model reference, you cannot change the model after you protect it.

Coordinate Settings with the User of Your Protected Model

When you develop a model that you will protect, coordinate with the user of the protected model to choose settings that allow their system to reference your protected model. After you protect your model, you cannot modify options such as configuration parameter settings to make your protected model compatible with a parent model. Before you protect your model, configure it to use compatible options.

Choose a Unique Model Name

You cannot change the name of your protected model after you protect it. To avoid name conflicts, choose a model name that is unique from other models in the system. Choose a model name that does not conflict with other models within your user's model hierarchy. Use the name to save your source (unprotected) model before you protect it.

Configure Model Parameters

Before you protect your model, choose configuration parameter settings by coordinating with the user of your protected model. When the parent model references your protected model, the configuration parameters of the models must be compatible. For example, if you set the parameter **Solver** to **Variable-step** for your protected model, the model that references your protected model must have **Solver** set to **Variable-step**. If your model contains multiple configuration sets, the protected model uses the configuration set that is active when you protect the model. To minimize parameter incompatibilities, send your model configuration set to the user of your protected model so that they can use the configuration for their parent model. For more information, see “Share a Configuration with Multiple Models”.

Minimize the Model External Interface

Except for input and output ports, minimize the number of design elements that provide an interface into your model. To allow interfacing with external components, the protected model does not protect those design elements. For the following items, consider these alternative options that allow protection.

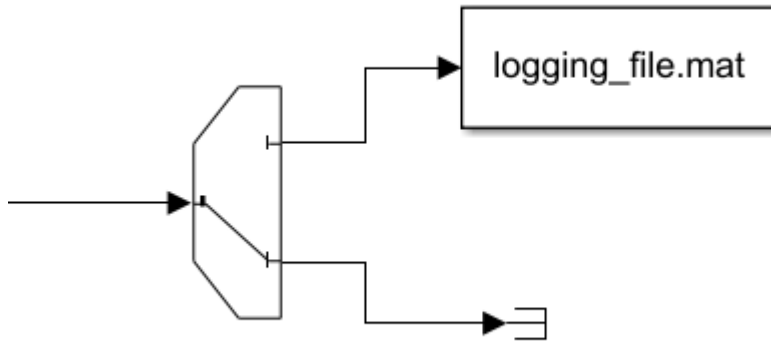
- **Tunable Parameters** — Minimize the number of tunable parameters in the code that use a storage class other than `Auto` or `Localizable`. Parameters that use a global storage class in your model's generated code are visible in the code for the protected model. If the user of your protected model requires access to tunable parameters, agree on a list of the parameters that the protected model provides. For more information, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.

- **Data Objects** — Data objects, including parameter and signal data objects, data type objects, and bus objects, that you store externally are not protected. Include only data objects that the user of the protected model requires access to. Use the data objects to monitor signals and calibrate parameters while you develop your model. After calibration, specify values for the parameters, and then inline them. For more information, see “Data Objects”.
- **Data Stores** — Do not use global data stores on signal objects. Global data stores can make data visible across referenced models. Instead, use inports and outports to communicate data across model references. You can use Data Store Memory blocks as data stores that are local to your model. For more information, see “Local and Global Data Stores”.
- **Model Arguments** — Minimize the number of arguments that your model uses. When your protected model uses arguments, the user of your protected model must specify values for the arguments in the parent model. These data items are visible to the user of your protected model. For each model argument that your model uses, communicate the argument specifications to the user of your protected model.
- **Simulink Functions** — Consider the scope of Simulink functions that your model uses. When a Simulink function has a global scope or is scoped at the root level of your model, it provides a visible interface with your protected model. If you do not want a Simulink function to interface into your protected model, consider specifying a scope within a subsystem of your model. For more information, see “Scoped Simulink Function Blocks in Subsystems”.
- **Logging Blocks** — Do not use block-based logging in your protected model. To File blocks and To Workspace blocks that use the `Timeseries` format function in protected models, and therefore make the logged data visible to the protected model user. Signal logging, and other options that use the signal selector, do not work for signals inside a protected model.
- **Noninlined S-functions** — Consider the content of noninlined S-functions that your model uses. When you protect a model that contains a noninlined S-function, the MEX files that you write for the S-function are placed in the protected model build folder and are visible to the user of your protected model.

Design Options in Your Protected Model

If you want to use one of the preceding design elements while you develop your model, but the element is not required by the user of the model, you can use variants to exclude the design element from your protected model. An example is using a To File block during testing to log data that the user of the protected model does not require access to. To exclude the block by using variants:

- 1** Insert a variant block, such as a Manual Variant Sink block, on the signal that leads to the To File block.
- 2** Connect the first output of the variant block to the To File block.
- 3** Connect the second output of the variant block to a Terminator block to terminate the signal.
- 4** To log the signal while testing the model, activate the first variant output.
- 5** To protect the model without using the logging block, activate the second variant output. The protected model uses only the active variant, so the signal is terminated without being logged.



Contents to Include in Your Protected Model

Model Protection Options

When you specify functionalities for your protected model, the protected model includes artifacts to support those functionalities. Limiting the included artifacts enables you to control what contents are visible to the user of your model or to an external attacker that wants to view or change your model without permission. Consider the trade-offs of different design options. To support additional functionalities, the protected model must include more content and therefore, more elements of your design might be visible. Coordinate with the user of your model to agree on what functionalities they require. Specify support for only those functionalities.

- **Obfuscated source code** — Use this option when the user of your protected model must compile the generated source code. The protected model includes an obfuscated version of the generated source code. It behaves the same as your source code but is more difficult to understand for the model user and potential attackers.
- **Compiled code** — Use this option when the user of your protected model can use the compiled library from your protected model. The protected model includes the binary files and header files compiled from your generated source code. It does not include the source code. This option prevents the user of your model and potential attackers from accessing the model's generated source code.
- **Password protection** — Use this option to support some functionality for your protected model's user and protect the supporting files from external attackers. For each option that you specify a password, the derived files that support that option are protected by using AES-256 encryption. Before executing each functionality, the protected model prompts the user for a password. Coordinate with the user of your protected model to choose the password for each option. If your protected model is referenced by a parent model that will also be protected, do not password-protect functionalities that the parent protected model will support.
- **Digital signature** — Sign your protected model by using a certificate to protect the recipient from using the model if it has been changed by an external attacker. When you sign your model, the user of the model can verify that the protected model is published by you and did not change after you signed it. After you sign the model, if it is changed, a user cannot use it. Make sure that the user's computer storage of trusted certificates has the certificate from the certificate authority that certified your certificate.

Interface Report

When you create a protected model, the generated protected model report includes an interface report. The interface report includes information about input and output specifications, exported functions, interface parameters, and data stores. Use the interface report as a reference to communicate with the user of your protected model.

Create a Harness Model

When you create a protected model, you have the option to create a harness model that references your protected model. Use the harness model to define information that the user requires to interface your protected model with their system. The harness contains the same configuration set that your protected model uses. In the harness model, you can include data definitions that your protected model uses.

See Also

More About

- “Protect Models to Conceal Contents” on page 12-2

Stateflow Blocks in Simulink Coder

- “Code Generation of Stateflow Blocks” on page 13-2
- “Generate Code from Atomic Subcharts” on page 13-5
- “Inline State Functions in Generated Code” on page 13-7

Code Generation of Stateflow Blocks

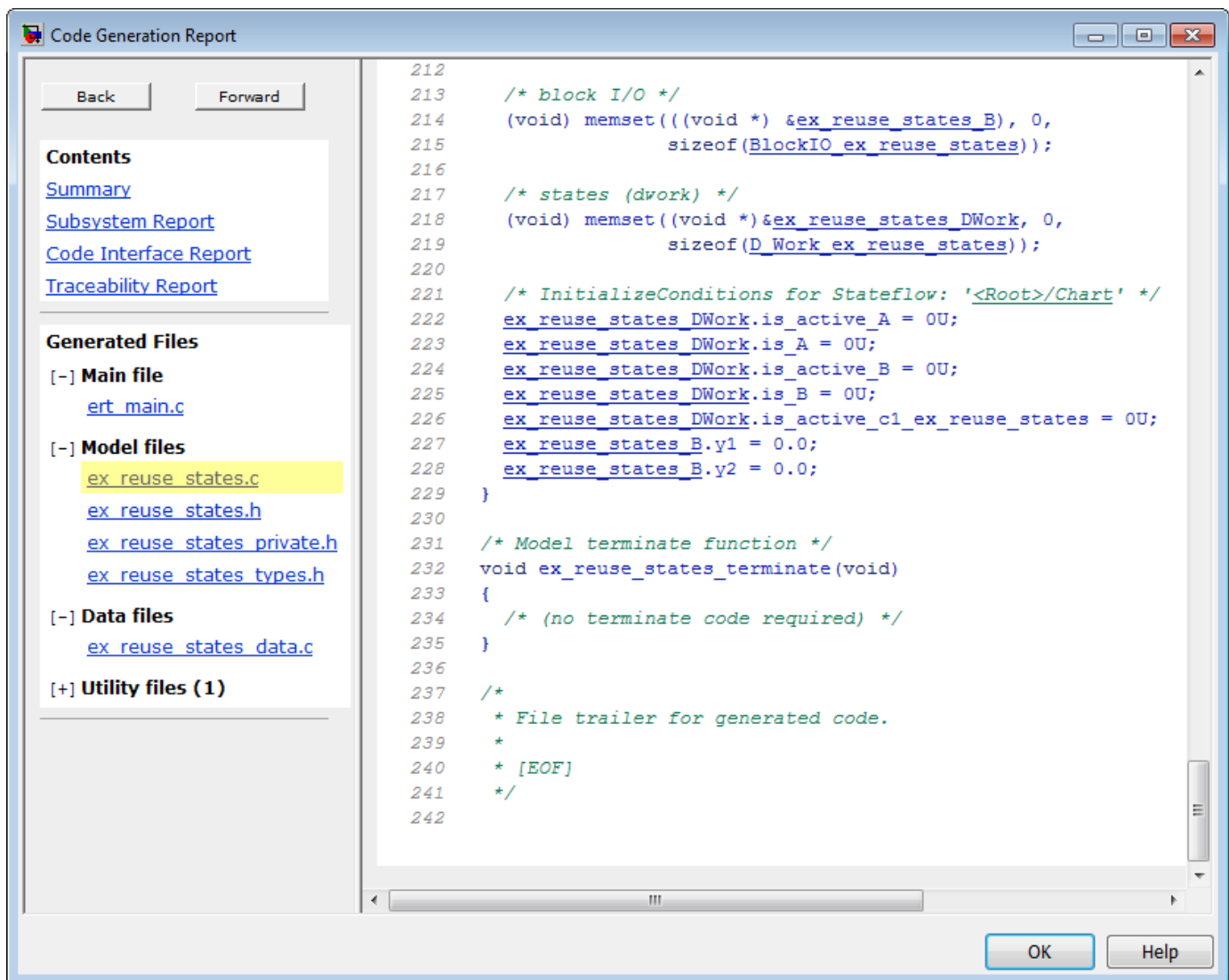
The code generator produces code for Stateflow blocks for rapid prototyping. If you have an Embedded Coder license, you can generate production code for Stateflow blocks.

Comparison of Code Generation Methods

The following sections compare two ways of generating code.

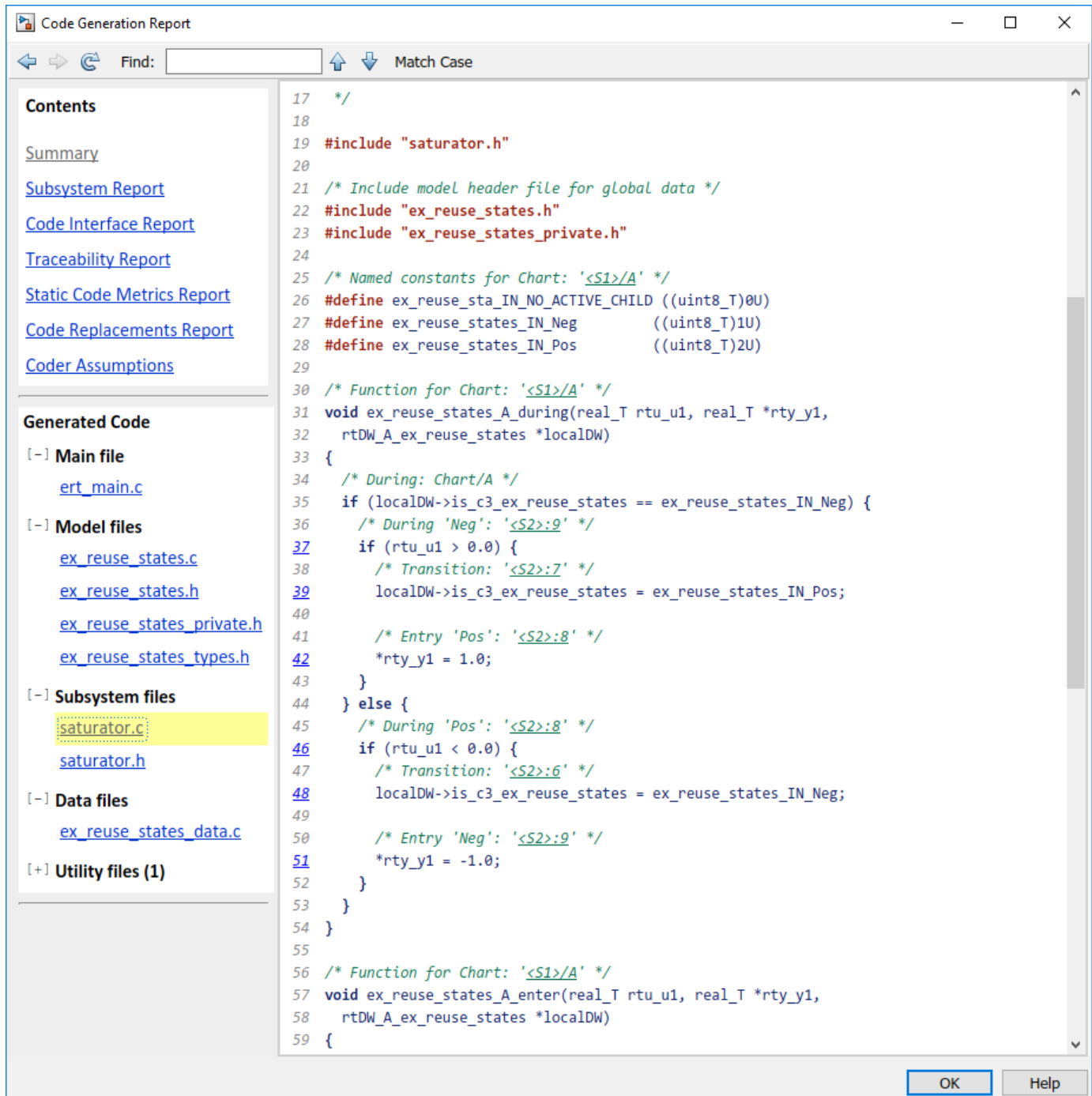
Code Generation Without Atomic Subcharts

You generate code for the entire model in one file and look through that entire file to find code for a specific part of the chart.



Code Generation With Atomic Subcharts

You specify code generation parameters so that code for an atomic subchart appears in a separate file. This method of code generation enables unit testing for a specific part of a chart. You can avoid searching through unrelated code and focus only on the part that interests you.



Note Unreachable Stateflow states are optimized out and are not included in the generated code.

See Also

Related Examples

- “Generate Code from Atomic Subcharts” on page 13-5
- “Generate C or C++ Code from Stateflow Blocks” (Stateflow)

Generate Code from Atomic Subcharts

To unit test a Stateflow chart in a Simulink model, first break the chart into smaller, independent components by using atomic subcharts. When you generate code for your chart, a separate file stores the code for the atomic subchart. Generating reusable code from atomic subcharts is useful for testing individual parts of your Stateflow chart. For more information, see “Create Reusable Subcomponents by Using Atomic Subcharts” (Stateflow).

Generate Reusable Code for Unlinked Atomic Subcharts

To specify code generation parameters for an unlinked atomic subchart:

- 1 In your chart, right-click the atomic subchart and select **Properties**.
- 2 In the dialog box, specify these parameters:
 - a Set **Code generation function packaging** to Reusable function.
 - b Set **Code generation file name options** to User specified.
 - c For **Code generation file name**, enter the name of the file with no extension.
- 3 Open the Configuration Parameters dialog box.
- 4 In the **Code Generation** pane, set the **System target file** parameter to `ert.tlc`.
- 5 (OPTIONAL) Customize the generated function names for atomic subcharts. In the **Code Generation > Identifiers** pane, set the **Subsystem methods** parameter. Specify the format of the function names by using a combination of these tokens:
 - \$R — root model name
 - \$F — type of interface function for the atomic subchart
 - \$N — block name
 - \$H — subsystem index
 - \$M — name-mangling text

For more information, see “Generate Separate Code for an Atomic Subchart” (Stateflow).

Generate Reusable Code for Linked Atomic Subcharts

To specify code generation parameters for linked atomic subcharts from the same library:

- 1 Open the library model that contains your atomic subchart.
- 2 Unlock the library.
- 3 Right-click the library chart and select **Block Parameters**.
- 4 In the dialog box, specify these parameters:
 - a On the **Main** tab, select **Treat as atomic unit**.
 - b On the **Code Generation** tab, set **Function packaging** to Reusable function.
 - c Set **File name options** to User specified.
 - d For **File name**, enter the name of the file with no extension.
- 5 In the **Code Generation** tab, set the **System target file** parameter to `ert.tlc`.

- 6** (OPTIONAL) Customize the generated function names for atomic subcharts. In the **Code Generation > Identifiers** tab, set the **Subsystem methods** parameter. Specify the format of the function names by using a combination of these tokens:
- \$R — root model name
 - \$F — type of interface function for the atomic subchart
 - \$N — block name
 - \$H — subsystem index
 - \$M — name-mangling text

When you generate code for your model, a separate file stores the code for linked atomic subcharts from the same library.

See Also

More About

- “Create Reusable Subcomponents by Using Atomic Subcharts” (Stateflow)
- “Generate C or C++ Code from Stateflow Blocks” (Stateflow)
- “Generate Separate Code for an Atomic Subchart” (Stateflow)

Inline State Functions in Generated Code

In this section...
“Inlined Generated Code for State Functions” on page 13-7
“How to Set the State Function Inline Option” on page 13-8
“Best Practices for Controlling State Function Inlining” on page 13-8

Inlined Generated Code for State Functions

By default, the code generator uses an internal heuristic to determine whether to inline generated code for state functions. The heuristic takes into consideration an inlining threshold. As code grows and shrinks in size, generated code for state functions can be unpredictable.

If your model includes Stateflow objects and you have rigorous requirements for traceability between generated code and the corresponding state functions, you can override the default behavior. Use the state property `Function Inline Option` to explicitly force or prevent inlining of state functions.

What Happens When You Force Inlining

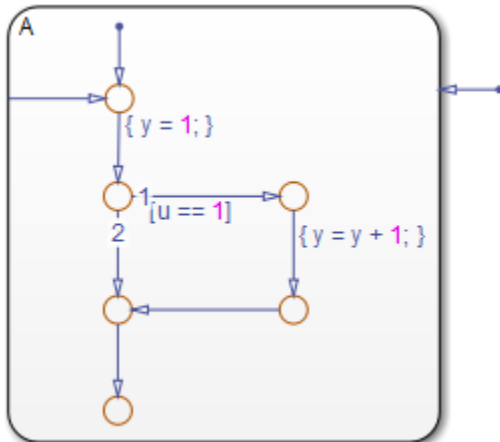
If you force inlining for a state, the code generator inlines code for state actions into the parent function. The parent function contains code for executing the state actions, outer transitions, and flow charts. It does not include code for empty state actions.

What Happens When You Prevent Inlining

If you prevent inlining for a state, the code generator produces these static functions for state *foo*.

Function	Description
<code>enter_atomic_foo</code>	Marks <i>foo</i> active and performs entry actions.
<code>enter_internal_foo</code>	Calls default paths.
<code>inner_default_foo</code>	Executes flow charts that originate when an inner transition and default transition reach the same junction inside a state. The code generator produces this function only when the flow chart is complex enough to exceed the inlining threshold. In generated code, Stateflow software calls this function from both the <code>enter_internal_foo</code> and <code>foo</code> functions.
<code>foo</code>	Checks for valid outer transitions and if none, performs during actions.
<code>exit_atomic_foo</code>	Performs exit actions and marks <i>foo</i> inactive.
<code>exit_internal_foo</code>	Performs exit actions of the child substates and then exits <i>foo</i> .

Suppose the following chart is in model M.



If you prevent inlining for state A, the code generator produces this code.

```
static void M_inner_default_A(void);
static void M_exit_atomic_A(void);
static void M_A(void);
static void M_enter_atomic_A(void);
static void M_enter_internal_A(void);
```

How to Set the State Function Inline Option

To set the function inlining property for a state:

- 1 Right-click inside the state and, from the context menu, select **Properties**.

The State properties dialog box opens.

- 2 For property **Function Inline Option**, select one of these values.

Value	Behavior
Inline	Forces inlining of state functions into the parent function, as long as the function is not part of a recursion. See “What Happens When You Force Inlining” on page 13-7.
Function	Prevents inlining of state functions. Generates up to six static functions for the state. See “What Happens When You Prevent Inlining” on page 13-7.
Auto	Uses internal heuristics to determine whether or not to inline the state functions.

- 3 Click **Apply**.

Best Practices for Controlling State Function Inlining

To	Set Function Inline Option Property To
Generate a separate function for each action of a state and a separate function for each action of its substates	Function for the state and each substate

To	Set Function Inline Option Property To
Generate a separate function for each action of a state, but include code for the associated action of its substates	Function for the state and Inline for each substate

Block Authoring and Code Generation for Simulink Coder

- “S-Functions and Code Generation” on page 14-2
- “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6
- “Legacy Code Tool Examples” on page 14-11
- “Integrate External C Functions That Pass Input and Output Arguments as Parameters with a Fixed-Point Data Type” on page 14-47
- “Integrate External C Functions That Implement N-Dimensional Table Lookups” on page 14-50
- “Integrate External C++ Object Methods” on page 14-53
- “External Code Integration Examples” on page 14-56
- “S-Functions That Support Expression Folding” on page 14-64
- “S-Functions That Specify Port Scope and Reusability” on page 14-73
- “S-Functions That Specify Sample Time Inheritance Rules” on page 14-77
- “S-Functions for Code Reuse” on page 14-81
- “S-Functions for Multirate Multitasking Environments” on page 14-86
- “Write Noninlined S-Function” on page 14-91
- “Write Wrapper S-Function and TLC Files” on page 14-93
- “Write Fully Inlined S-Functions” on page 14-100
- “Write Fully Inlined S-Functions with mdlRTW Routine” on page 14-102

S-Functions and Code Generation

You use S-functions to extend Simulink support for simulation and code generation. For example, you can use them to:

- Represent custom algorithms
- Interface existing external code with Simulink model and the generated code
- Represent device drivers for interfacing with hardware
- Generate highly optimized code for embedded systems
- Verify code generated for a subsystem as part of a Simulink simulation

The application program interface (API) for writing S-functions allows you to implement generic algorithms in the Simulink environment with a great deal of flexibility. If you intend to use S-functions in a model for code generation, the level of flexibility can vary. For example, it is not possible to access the MATLAB workspace from an S-function that you use with the code generator. This topic explains conditions to be aware of for using S-functions. However, using the techniques presented in this topic, you can create S-functions for most applications that work with the generated code.

Although S-functions provide a generic and flexible solution for implementing complex algorithms in a model, the underlying API incurs overhead in terms of memory and computation resources. Most often the additional resources are acceptable for real-time rapid prototyping systems. Often, though, additional resources are unavailable in real-time embedded applications. You can minimize memory and computational requirements by using the Target Language Compiler technology provided with the code generator to inline your S-functions. If you are producing an S-function for existing external code, consider using the Legacy Code Tool to generate your S-function and relevant TLC file.

This topic assumes that you understand the following concepts:

- Level-2 S-functions
- Target Language Compiler (TLC) scripting
- How the code generator produces and builds C/C++ code

Notes This information is for code generator users. Even if you do not currently use the code generator, follow these practices when writing S-functions, especially if you are creating general-purpose S-functions.

Types of S-Functions

Examples for which you might choose to implement an S-function for simulation and code generation include:

- 1 “I’m not concerned with efficiency. I want to write one version of my algorithm and have it work in the Simulink and code generator products automatically.”
- 2 “I want to implement a highly optimized algorithm in the Simulink and code generator products that looks like a built-in block and generates efficient code.”
- 3 “I have a lot of hand-written code that I need to interface. I want to call my function from the Simulink and code generator products efficiently.”

Respectively, the preceding situations map to the following MathWorks terminology:

- 1 Noninlined S-function
- 2 Inlined S-function
- 3 Auto-generated S-function for external code

Noninlined S-Functions

A noninlined S-function is a C or C++ MEX S-function that is treated identically by the Simulink engine and generated code. In general, you implement your algorithm once according to the S-function API. The Simulink engine and generated code call the S-function routines (for example, `mdlOutputs`) during model execution.

Additional memory and computation resources are required for each instance of a noninlined S-Function block. However, this routine of incorporating algorithms into models and code generation applications is typical during the prototyping phase of a project where efficiency is not important. The advantage gained by forgoing efficiency is the ability to change model parameters and structures rapidly.

Writing a noninlined S-function does not involve TLC coding. Noninlined S-functions are the default case for the build process in the sense that once you build a MEX S-function in your model, there is no additional preparation before pressing **Ctrl+B** to build your model.

Some restrictions exist concerning the names and locations of noninlined S-function files when generating makefiles. See “Write Noninlined S-Function” on page 14-91.

Inlined S-Functions

For S-functions to work in the Simulink environment, some overhead code is generated. When the code generator produces code from models that contain S-functions (without `sfunction.tlc` files), it embeds some of this overhead code in the generated code. If you want to optimize your real-time code and eliminate some of the overhead code, you must *inline* (or embed) your S-functions. This involves writing a TLC (`sfunction.tlc`) file that eliminates overhead code from the generated code. The Target Language Compiler processes `sfunction.tlc` files to define how to inline your S-function algorithm in the generated code.

Note Do not confuse the term *inline* with the C++ *inline* keyword. Inline means to specify text in place of the call to the general S-function API routines (for example, `mdlOutputs`). For example, when a TLC file is used to inline an S-function, the generated code contains the C/ C++ code that normally appears within the S-function routines and the S-function itself that has been removed from the build process.

A fully inlined S-function builds your algorithm (block) into generated code in a manner that is indistinguishable from a built-in block. Typically, a fully inlined S-function requires you to implement your algorithm twice: once for the Simulink model (C/C++ MEX S-function) and once for code generation (TLC file). The complexity of the TLC file depends on the complexity of your algorithm and the level of efficiency you try to achieve in the generated code. TLC files vary from simple to complex in structure. See “Inlining S-Functions”.

Auto-generated S-Functions for Legacy or Custom Code

If you need to invoke hand-written C/C++ code in your model, consider using the Simulink Legacy Code Tool. The Legacy Code Tool can automate the generation of a fully inlined S-function and a corresponding TLC file based on information that you register in a Legacy Code Tool data structure.

For more information, see “Integrate C Functions Using Legacy Code Tool” and see “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6.

Files Required for Implementing Noninlined and Inlined S-Functions

This topic briefly describes what files and functions you need to create noninlined and inlined S-functions.

- Noninlined S-functions require the C or C++ MEX S-function source code (*sfunction.c* or *sfunction.cpp*).
- Fully inlined S-functions require an *sfunction.tlc* file. Fully inlined S-functions produce the optimal code for a parameterized S-function. This is an S-function that operates in a specific mode dependent upon fixed S-function parameters that do not change during model execution. For a given operating mode, the *sfunction.tlc* file specifies the exact code that is generated to implement the algorithm for that mode. For example, the direct-index lookup table S-function in “Write Fully Inlined S-Functions with mdlRTW Routine” on page 14-102 contains two operating modes — one for evenly spaced *x-data* and one for unevenly spaced *x-data*.

Note Fully inlined S-functions that are generated to invoke legacy or custom C/C++ code also require an *sfunction.tlc* file, which is generated by Legacy Code Tool.

Fully inlined S-functions might require the placement of the mdlRTW routine in your S-function MEX-file *sfunction.c* or *sfunction.cpp*. The mdlRTW routine lets you place information in *model.rtw*. *model.rtw* is the record file that specifies a model, and which the code generator invokes the Target Language Compiler to process before executing *sfunction.tlc* when generating code.

Including a mdlRTW routine is useful when you want to introduce non-tunable parameters into your TLC file. Such parameters are used to determine which operating mode is active in a given instance of the S-function. Based on this information, the TLC file for the S-function can generate highly efficient, optimal code for that operating mode.

Guidelines for Writing S-Functions that Support Code Generation

- You can use C/C++ MEX, MATLAB language, and Fortran MEX S-functions with code generation.
- You can inline S-functions for code generation by providing an inlining TLC file. See S-Function Inlining in “Target Language Compiler”. MATLAB and Fortran MEX S-functions must be inlined. C/C++ MEX S-functions can be inlined for code efficiency, or noninlined.
- To automatically generate a fully inlined C MEX S-function for invoking legacy or custom code, use the Legacy Code Tool. For more information, see “Integrate C Functions Using Legacy Code Tool” and see “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6.
- If code efficiency is not an overriding consideration, for example, if you are rapid prototyping, you can choose not to inline a C/C++ MEX S-function. For more information, see “Write Noninlined S-Function” on page 14-91.

See Also

More About

- “Import Calls to External Code into Generated Code with Legacy Code Tool” (Embedded Coder)

Import Calls to External Code into Generated Code with Legacy Code Tool

In this section...

“Legacy Code Tool and Code Generation” on page 14-6
 “Generate Inlined S-Function Files for Code Generation” on page 14-6
 “Apply Code Style Settings to Legacy Functions” on page 14-7
 “Address Dependencies on Files in Different Locations” on page 14-8
 “Deploy S-Functions for Simulation and Code Generation” on page 14-8
 “Integrate External C++ Objects” on page 14-8

Legacy Code Tool and Code Generation

You can use the Simulink Legacy Code Tool to generate fully inlined C MEX S-functions for legacy or custom code. The S-functions are optimized for embedded components, such as device drivers and lookup tables, and they call existing C or C++ functions.

Note The Legacy Code Tool can interface with C++ functions, but not C++ objects. To work around this issue so that the tool can interface with C++ objects, see “Legacy Code Tool Limitations”.

You can use the tool to:

- Compile and build the generated S-function for simulation.
- Generate a masked S-Function block that is configured to call the existing external code.

If you want to include these types of S-functions in models for which you intend to generate code, use the tool to generate a TLC block file. The TLC block file specifies how the generated code for a model calls the existing C or C++ function.

If the S-function depends on files in folders other than the folder containing the S-function dynamically loadable executable file, use the tool to generate an *sFunction_makecfg.m* or *rtwmakecfg.m* file for the S-function. Generating the file maintains those dependencies when you build a model that includes the S-function. For example, for some applications, such as custom targets, you might want to locate files in a target-specific location. The build process looks for *sFunction_makecfg.m* or *rtwmakecfg.m* in the same folder as the S-function dynamically loadable executable and calls the function in the file.

For more information, see “Integrate C Functions Using Legacy Code Tool”.

Generate Inlined S-Function Files for Code Generation

Depending on the code generation requirements of your application, to generate code for a model that uses the S-function, do either of the following:

- Generate one *.cpp* file for the inlined S-function. In the Legacy Code Tool data structure, set the value of the `Options.singleCPPMexFile` field to `true` before generating the S-function source file from your existing C function. For example:

```
def.Options.singleCPPMexFile = true;
legacy_code('sfcn_cmex_generate', def);
```

- Generate a source file and a TLC block file for the inlined S-function. For example:

```
def.Options.singleCPPMexFile = false;
legacy_code('sfcn_cmex_generate', def);
legacy_code('sfcn_tlc_generate', def);
```

singleCPPMexFile Limitations

You cannot set the `singleCPPMexFile` field to `true` if

- `Options.language='C++'`
- You use one of the following Simulink objects with the `IsAlias` property set to `true`:
 - `Simulink.Bus`
 - `Simulink.AliasType`
 - `Simulink.NumericType`
- The Legacy Code Tool function specification includes a `void*` or `void**` to represent scalar work data for a state argument
- `HeaderFiles` field of the Legacy Code Tool structure specifies multiple header files

Apply Code Style Settings to Legacy Functions

To apply the model configuration parameters for code style to a legacy function:

- 1 Initialize the Legacy Code Tool data structure. For example:

```
def = legacy_code('initialize');
```

- 2 In the data structure, set the value of the `Options.singleCPPMexFile` field to `true`. For example:

```
def.Options.singleCPPMexFile = true;
```

To check the setting, enter:

```
def.Options.singleCPPMexFile
```

singleCPPMexFile Limitations

You cannot set the `singleCPPMexFile` field to `true` if

- `Options.language='C++'`
- You use one of the following Simulink objects with the `IsAlias` property set to `true`:
 - `Simulink.Bus`
 - `Simulink.AliasType`
 - `Simulink.NumericType`
- The Legacy Code Tool function specification includes a `void*` or `void**` to represent scalar work data for a state argument
- `HeaderFiles` field of the Legacy Code Tool structure specifies multiple header files

Address Dependencies on Files in Different Locations

By default, the Legacy Code Tool assumes that files on which an S-function depends reside in the same folder as the dynamically loadable executable file for the S-function. If your S-function depends on files that reside elsewhere and you are using the template makefile build process, generate an *sFunction_makecfg.m* or *rtwmakecfg.m* file for the S-function. For example, you might generate this file if your Legacy Code Tool data structure defines compilation resources as path names.

To generate the *sFunction_makecfg.m* or *rtwmakecfg.m* file, call the `legacy_code` function with `'sfcn_makecfg_generate'` or `'rtwmakecfg_generate'` as the first argument, and the name of the Legacy Code Tool data structure as the second argument. For example:

```
legacy_code('sfcn_makecfg_generate', lct_spec);
```

If you use multiple registration files in the same folder and generate an S-function for each file with a single call to `legacy_code`, the call to `legacy_code` that specifies `'sfcn_makecfg_generate'` or `'rtwmakecfg_generate'` must be common to all registration files. For more information, see “Handling Multiple Registration Files”.

For example, if you define `defs` as an array of Legacy Code Tool structures, you call `legacy_code` with `'sfcn_makecfg_generate'` once.

```
defs = [defs1(:);defs2(:);defs3(:)];
legacy_code('sfcn_makecfg_generate', defs);
```

For more information, see “Build Support for S-Functions” on page 30-35.

Deploy S-Functions for Simulation and Code Generation

You can deploy the S-functions that you generate with the Legacy Code Tool so that other people can use them. To deploy an S-function for simulation and code generation, share the following files:

- Registration file
- Compiled dynamically loadable executable
- TLC block file
- *sFunction_makecfg.m* or *rtwmakecfg.m* file
- Header, source, and include files on which the generated S-function depends

When you use these deployed files:

- Before using the deployed files in a Simulink model, add the folder that contains the S-function files to the MATLAB path.
- If the Legacy Code Tool data structure registers required files as absolute paths and the location of the files changes, regenerate the *sFunction_makecfg.m* or *rtwmakecfg.m* file.

Integrate External C++ Objects

The Legacy Code Tool can interface with C++ functions, but not C++ objects. Using the previous example as a starting point, here is an example of how you can work around this limitation.

- Modify the class definition for `adder` in a new file `adder_cpp.hpp`. Add three new macros that dynamically allocate a new `adder` object, invoke the method `add_one()`, and free the memory

allocated. Each macro takes a pointer to an `adder` object. Because each function called by the Legacy Code Tool must have a C-like signature, the pointer is cached and passed as a `void*`. Then you must explicitly cast to `adder*` in the macro. The new class definition for `adder`:

```
#ifndef _ADDER_CPP_
#define _ADDER_CPP_

class adder {
private:
    int int_state;
public:
    adder(): int_state(0) {};
    int add_one(int increment);
    int get_val() {return int_state;};
};

// Method wrappers implemented as macros
#define createAdder(work1) \
    *(work1) = new adder

#define deleteAdder(work1) \
    delete(static_cast<adder*>(*(work1)))

#define adderOutput(work1, u1) \
    (static_cast<adder*> ((work1)))->add_one(u1)

#endif /* _ADDER_CPP_ */
```

- Update `adder_cpp.cpp`. With the class modification, instead of one global instance, each generated S-function manages its own `adder` object.

```
#include "adder_cpp.hpp"

int adder::add_one(int increment)
{
    int_state += increment;
    return int_state;
}
```

- Update `rtwdemo_sfund_adder_cpp.cpp` with the following changes:
 - `StartFcnSpec` calls the macro that allocates a new `adder` object and caches the pointer.


```
def.StartFcnSpec = 'createAdder(void **work1)';
```
 - `OutputFcnSpec` calls the macro that invokes the method `add_one()` and provides the S-function specific `adder` pointer object.


```
def.OutputFcnSpec = 'int32 y1 = adderOutput(void *work1, int32 u1)';
```
 - `TerminateFcnSpec` calls the macro that frees the memory.


```
def.TerminateFcnSpec = 'deleteAdder(void **work1)';
```

See Also

`legacy_code`

Related Examples

- “Integrate C Functions Using Legacy Code Tool”
- “Legacy Code Tool Examples” on page 14-11
- “Call External C Code from Model and Generated Code” (Embedded Coder)

Legacy Code Tool Examples

You can use the Simulink Legacy Code Tool to generate fully inlined C MEX S-functions for legacy or custom code. The S-functions are optimized for embedded components, such as device drivers and lookup tables, and they call existing C or C++ functions. These examples show how to use the Legacy Code Tool to integrate a variety of external functions.

Integrate External C Functions That Pass Input Arguments by Value Versus Address

This example shows how to use the Legacy Code Tool to integrate legacy C functions that pass their input arguments by value versus address.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototypes of the legacy functions being called in this example are:

- `FLT filterV1(const FLT signal, const FLT prevSignal, const FLT gain)`
- `FLT filterV2(const FLT* signal, const FLT prevSignal, const FLT gain)`

FLT is a typedef to float. The legacy source code is in the files `your_types.h`, `myfilter.h`, `filterV1.c`, and `filterV2.c`.

Note the difference in the `OutputFcnSpec` defined in the two structures; the first case specifies that the first input argument is passed by value, while the second case specifies pass by pointer.

```

defs = [];

% rtwdemo_sfunk_filterV1
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfunk_filterV1';
def.OutputFcnSpec = 'single y1 = filterV1(single u1, single u2, single p1)';
def.HeaderFiles = {'myfilter.h'};
def.SourceFiles = {'filterV1.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
defs = [defs; def];

% rtwdemo_sfunk_filterV2
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfunk_filterV2';

```

```
def.OutputFcnSpec = 'single y1 = filterV2(single u1[1], single u2, single p1)';
def.HeaderFiles   = {'myfilter.h'};
def.SourceFiles   = {'filterV2.c'};
def.IncPaths      = {'rtwdemo_lct_src'};
def.SrcPaths      = {'rtwdemo_lct_src'};
defs = [defs; def];
```

Generate S-Functions for Simulation

To generate C-MEX S-functions according to the description provided by the input argument 'defs', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-functions call the legacy functions in simulation. The source code for the S-functions is in the files `rtwdemo_sfun_filterV1.c` and `rtwdemo_sfun_filterV2.c`.

```
legacy_code('sfcn_cmex_generate', defs);
```

Compile the Generated S-Functions for Simulation

After you generate the C-MEX S-function source files, to compile the S-functions for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', defs);
```

```
### Start Compiling rtwdemo_sfun_filterV1
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570')
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('rtwdemo_sfun_filterV1.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570')
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_filterV1
### Exit

### Start Compiling rtwdemo_sfun_filterV2
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570')
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('rtwdemo_sfun_filterV2.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570')
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_filterV2
### Exit
```

Generate TLC Block Files for Code Generation

After you compile the S-functions and use them in simulation, you can call the function `legacy_code()` again with the first input set to 'sfcn_tlc_generate' to generate TLC block files. Block files specify how the generated code for a model calls the legacy code. If you do not generate TLC block files and you try to generate code for a model that includes the S-functions, code generation fails. The TLC block files for the S-functions are `rtwdemo_sfun_filterV1.tlc` and `rtwdemo_sfun_filterV2.tlc`.

```
legacy_code('sfcn_tlc_generate', defs);
```

Generate an `rtwmakecfg.m` File for Code Generation

After you create the TLC block files, you can call the function `legacy_code()` again with the first input set to 'rtwmakecfg_generate' to generate an `rtwmakecfg.m` file to support code generation. If the

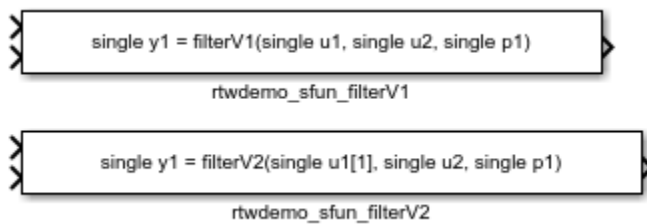
required source and header files for the S-functions are not in the same folder as the S-functions, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', defs);
```

Generate Masked S-Function Blocks for Calling the Generated S-Functions

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again with the first input set to `'slblock_generate'` to generate masked S-function blocks that call the S-functions. The software places the blocks in a new model. From there you can copy them to an existing model.

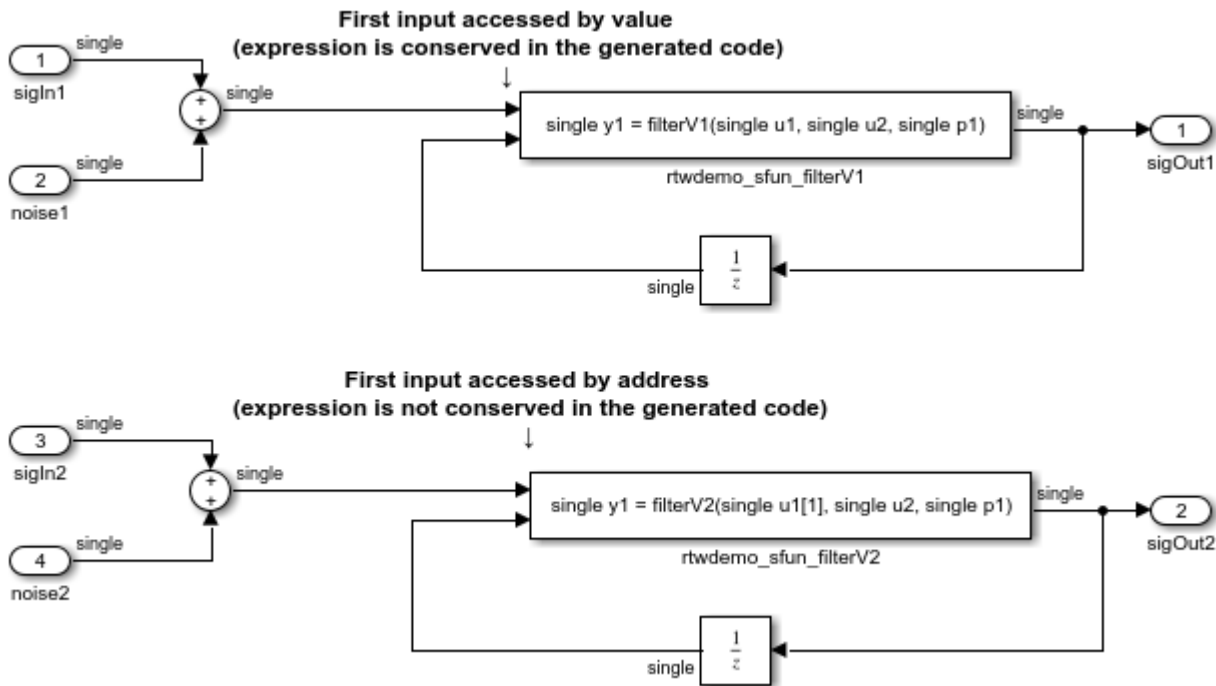
```
legacy_code('slblock_generate', defs);
```



Show the Generated Integration with Legacy Code

The model `rtwdemo_lct_filter` shows integration of the model with the legacy code. The subsystem `TestFilter` serves as a harness for the calls to the legacy C functions via the generate S-functions, with unit delays serving to store the previous output values.

```
open_system('rtwdemo_lct_filter')
open_system('rtwdemo_lct_filter/TestFilter')
sim('rtwdemo_lct_filter')
```



Integrate External C Functions That Pass the Output Argument as a Return Argument

This example shows how to use the Legacy Code Tool to integrate legacy C functions that pass their output as a return argument.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototype of the legacy functions being called in this example is:

```
FLT gainScalar(const FLT in, const FLT gain)
```

FLT is a typedef to float. The legacy source code is in the files `your_types.h`, `gain.h`, and `gainScalar.c`.

```
% rtwdemo_sfun_gain_scalar
def = legacy_code('initialize');
```

```
def.SFunctionName = 'rtwdemo_sfun_gain_scalar';
def.OutputFcnSpec = 'single y1 = gainScalar(single u1, single p1)';
def.HeaderFiles   = {'gain.h'};
def.SourceFiles   = {'gainScalar.c'};
def.IncPaths      = {'rtwdemo_lct_src'};
def.SrcPaths      = {'rtwdemo_lct_src'};
```

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfun_gain_scalar.c`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```
### Start Compiling rtwdemo_sfun_gain_scalar
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('rtwdemo_sfun_gain_scalar.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_gain_scalar
### Exit
```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again with the first input set to 'sfcn_tlc_generate' to generate a TLC block file. The block file specifies how the generated code for a model calls the legacy code. If you do not generate a TLC block file and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_gain_scalar.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an `rtwmakecfg.m` File for Code Generation

After you create the TLC block file, you can call the function `legacy_code()` again with the first input set to 'rtwmakecfg_generate' to generate an `rtwmakecfg.m` file to support code generation. If the required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

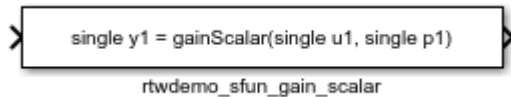
```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again with the first input set to 'slblock_generate' to generate a masked S-function block that calls that S-

function. The software places the block in a new model. From there you can copy it to an existing model.

```
legacy_code('slblock_generate', def);
```



Show the Generated Integration with Legacy Code

The model `rtwdemo_lct_gain` shows integration of the model with the legacy code. The subsystem `TestGain` serves as a harness for the call to the legacy C function via the generate S-function.

```
open_system('rtwdemo_lct_gain')
open_system('rtwdemo_lct_gain/TestGain')
sim('rtwdemo_lct_gain')
```



Integrate External C++ Object Methods

Integrate legacy C++ object methods by using the Legacy Code Tool.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C++ MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a block TLC file and optional `rtwmakecfg.m` file that calls the legacy code during code generation.

Provide the Legacy Function Specification

Functions provided with the Legacy Code Tool take a specific data structure or array of structures as the argument. The data structure is initialized by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The definition of the legacy C++ class in this example is:

```
class adder {
private:
    int int_state;
public:
    adder();
    int add_one(int increment);
    int get_val();
};
```

The legacy source code is in the files `adder_cpp.h` and `adder_cpp.cpp`.


```

% rtwdemo_sfun_adder_cpp
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_adder_cpp';
def.StartFcnSpec = 'createAdder()';
def.OutputFcnSpec = 'int32 y1 = adderOutput(int32 u1)';
def.TerminateFcnSpec = 'deleteAdder()';
def.HeaderFiles = {'adder_cpp.h'};
def.SourceFiles = {'adder_cpp.cpp'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
def.Options.language = 'C++';
def.Options.useTlcWithAccel = false;

```

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfun_adder_cpp.cpp`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```

### Start Compiling rtwdemo_sfun_adder_cpp
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019'.
MEX completed successfully.
mex('rtwdemo_sfun_adder_cpp.cpp', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\
Building with 'Microsoft Visual C++ 2019'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_adder_cpp
### Exit

```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again. Set the first input to 'sfcn_tlc_generate' to generate a TLC block file that supports code generation through Simulink® Coder™. If the TLC block file is not created and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_adder_cpp.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an rtwmakecfg.m File for Code Generation

After you create the TLC block file, you can call the function `legacy_code()` again. Set the first input to 'rtwmakecfg_generate' to generate an `rtwmakecfg.m` file that supports code generation through Simulink® Coder™. If the required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

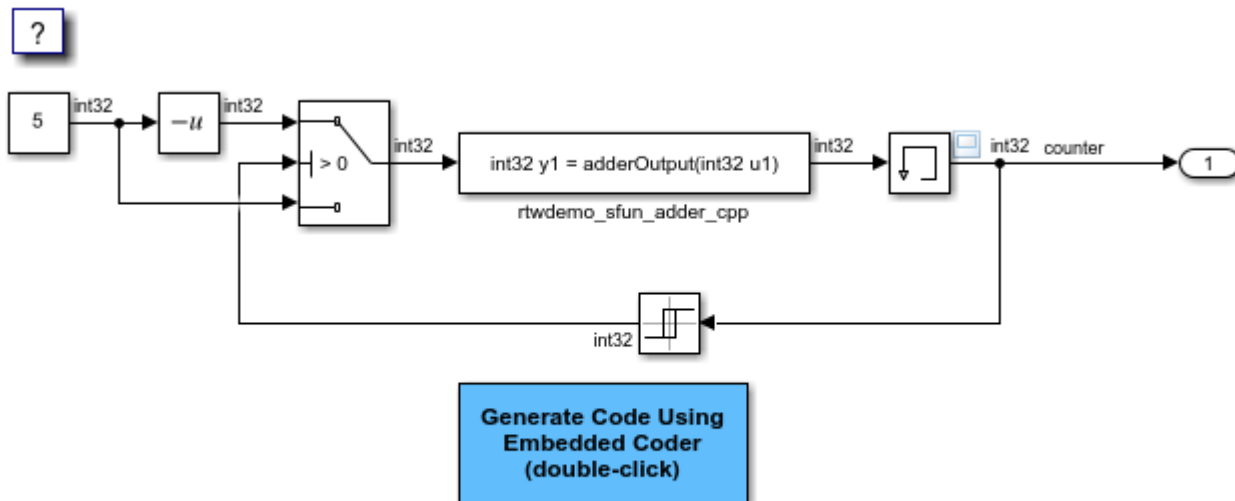
After you compile the C-MEX S-function source, you can call the function `legacy_code()` again. Set the first input to `'slblock_generate'` to generate a masked S-function block that is configured to call that S-function. The software places the block in a new model. You can copy the block to an existing model.

```
% legacy_code('slblock_generate', def);
```

Show the Generated Integration with Legacy Code

The model `rtwdemo_lct_cpp` shows integration with the legacy code.

```
open_system('rtwdemo_lct_cpp')
sim('rtwdemo_lct_cpp')
```



Copyright 1990-2020 The MathWorks, Inc.

Integrate External C Functions That Pass Input and Output Arguments as Parameters with a Fixed-Point Data Type

Integrate legacy C functions that pass their inputs and outputs by using parameters of a fixed-point data type with the Legacy Code Tool.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototype of the legacy functions being called in this example is:

```
myFixpt timesS16(const myFixpt in1, const myFixpt in2, const uint8_T fracLength)
```

`myFixpt` is logically a fixed point data type, which is physically a typedef to a 16-bit integer:

```
myFixpt = Simulink.NumericType;
myFixpt.DataTypeMode = 'Fixed-point: binary point scaling';
myFixpt.Signed = true;
myFixpt.WordLength = 16;
myFixpt.FractionLength = 10;
myFixpt.IsAlias = true;
myFixpt.HeaderFile = 'timesFixpt.h';
```

The legacy source code is in the files `timesFixpt.h`, and `timesS16.c`.

```
% rtwdemo_sfun_gain_fixpt
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_gain_fixpt';
def.OutputFcnSpec = 'myFixpt y1 = timesS16(myFixpt u1, myFixpt p1, uint8 p2)';
def.HeaderFiles = {'timesFixpt.h'};
def.SourceFiles = {'timesS16.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
```

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfun_gain_fixpt.c`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```
### Start Compiling rtwdemo_sfun_gain_fixpt
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib577');
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('rtwdemo_sfun_gain_fixpt.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib577');
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_gain_fixpt
### Exit
```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again. Set the first input to `'sfcn_tlc_generate'` to generate a TLC block file. The block file specifies how the generated code for a model calls the legacy code. If you do not generate a TLC block file and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_gain_fixpt.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an `rtwmakecfg.m` File for Code Generation

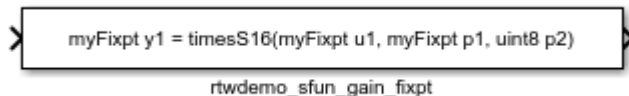
After you create the TLC block file, you can call the function `legacy_code()` again. Set the first input to `'rtwmakecfg_generate'` to generate an `rtwmakecfg.m` file that supports code generation. If the required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again. Set the first input to `'slblock_generate'` to generate a masked S-function block that calls that S-function. The software places the block in a new model. You can copy the block to an existing model.

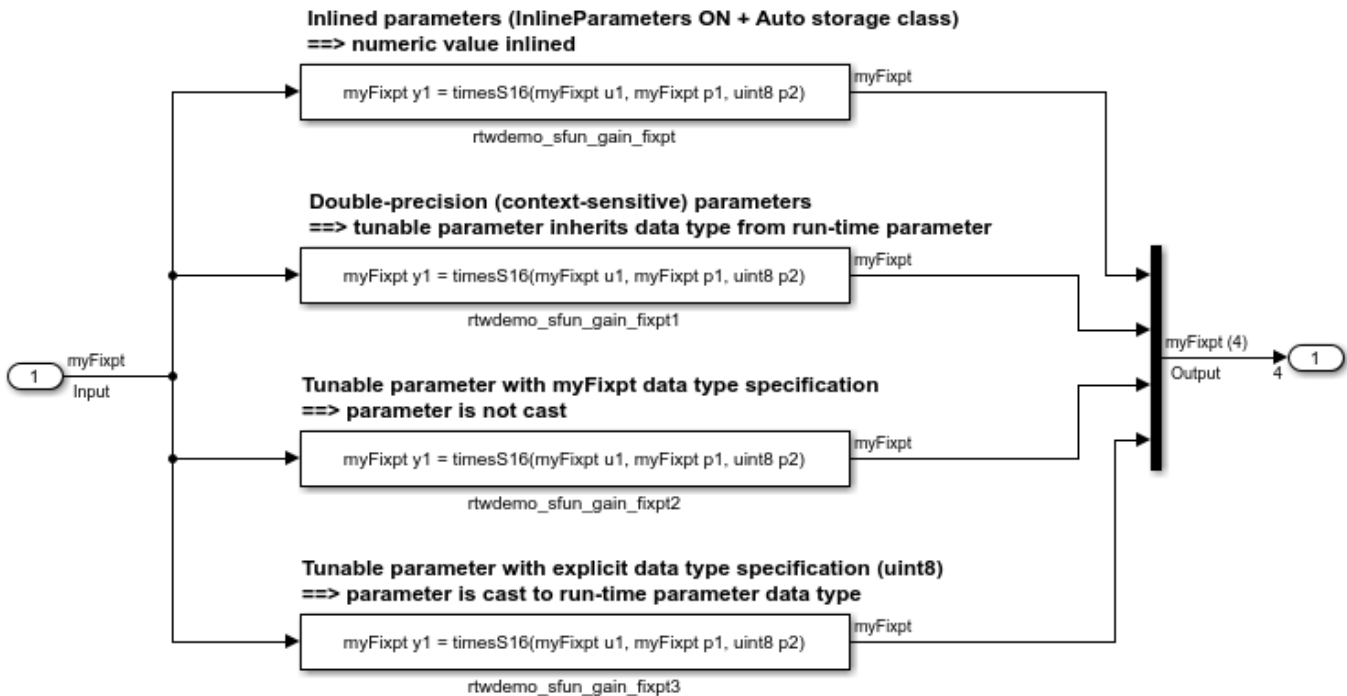
```
legacy_code('slblock_generate', def);
```



Show the Generated Integration with Legacy Code

The model `rtwdemo_lct_fixpt_params` shows integration of the model with the legacy code. The subsystem `TestFixpt` serves as a harness for the call to the legacy C function via the generated S-function.

```
open_system('rtwdemo_lct_fixpt_params')
open_system('rtwdemo_lct_fixpt_params/TestFixpt')
sim('rtwdemo_lct_fixpt_params')
```



Integrate External C Functions That Pass Input and Output Arguments as Signals with a Fixed-Point Data Type

This example shows how to use the Legacy Code Tool to integrate legacy C functions that pass their inputs and outputs by using parameters of fixed-point data type.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototype of the legacy functions being called in this example is:

```
myFixpt timesS16(const myFixpt in1, const myFixpt in2, const uint8_T fracLength)
```

`myFixpt` is logically a fixed-point data type, which is physically a typedef to a 16-bit integer:

```
myFixpt = Simulink.NumericType;
myFixpt.DataTypeMode = 'Fixed-point: binary point scaling';
```

```
myFixpt.Signed = true;
myFixpt.WordLength = 16;
myFixpt.FractionLength = 10;
myFixpt.IsAlias = true;
myFixpt.HeaderFile = 'timesFixpt.h';
```

The legacy source code is in the files `timesFixpt.h`, and `timesS16.c`.

```
% rtwdemo_sfun_times_s16
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_times_s16';
def.OutputFcnSpec = 'myFixpt y1 = timesS16(myFixpt u1, myFixpt u2, uint8 p1)';
def.HeaderFiles = {'timesFixpt.h'};
def.SourceFiles = {'timesS16.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
```

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfun_times_s16.c`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```
### Start Compiling rtwdemo_sfun_times_s16
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570');
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('rtwdemo_sfun_times_s16.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570');
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_times_s16
### Exit
```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again with the first input set to 'sfcn_tlc_generate' to generate a TLC block file. The block file specifies how the generated code for a model calls the legacy code. If you do not generate a TLC block file and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_times_s16.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an `rtwmakecfg.m` File for Code Generation

After you create the TLC block file, you can call the function `legacy_code()` again with the first input set to 'rtwmakecfg_generate' to generate an `rtwmakecfg.m` file to support code generation. If the

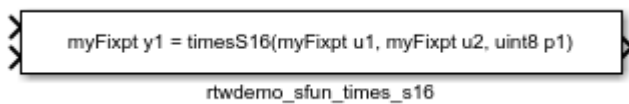
required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again with the first input set to `'slblock_generate'` to generate a masked S-function block that calls that S-function. The software places the block in a new model. From there you can copy it to an existing model.

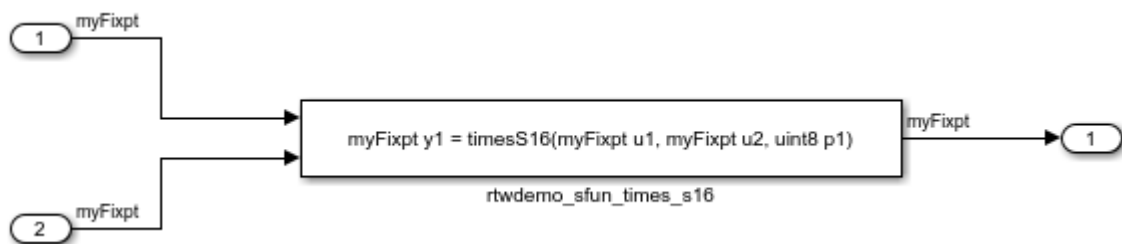
```
legacy_code('slblock_generate', def);
```



Show the Integration of the Model with Legacy Code

The model `rtwdemo_lct_fixpt_signals` shows integration of the model with the legacy code. The subsystem `TestFixpt` serves as a harness for the call to the legacy C function via the generated S-function.

```
open_system('rtwdemo_lct_fixpt_signals')
open_system('rtwdemo_lct_fixpt_signals/TestFixpt')
sim('rtwdemo_lct_fixpt_signals')
```



Integrate External C Functions with Instance-Specific Persistent Memory

Integrate legacy C functions that use instance-specific persistent memory by using the Legacy Code Tool.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.

- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototypes of the legacy functions being called in this example are:

```
void memory_bus_init(COUNTERBUS *mem, int32_T upper_sat, int32_T lower_sat);
```

```
void memory_bus_step(COUNTERBUS *input, COUNTERBUS *mem, COUNTERBUS *output);
```

`mem` is an instance-specific persistent memory for applying a one integration step delay. `COUNTERBUS` is a struct typedef defined in `counterbus.h` and implemented with a `Simulink.Bus` object in the base workspace. The legacy source code is in the files `memory_bus.h`, and `memory_bus.c`.

```
evalin('base','load rtwdemo_lct_data.mat')

% rtwdemo_sfun_work
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_work';
def.InitializeConditionsFcnSpec = ...
    'void memory_bus_init(COUNTERBUS work1[1], int32 p1, int32 p2)';
def.OutputFcnSpec = ...
    'void memory_bus_step(COUNTERBUS u1[1], COUNTERBUS work1[1], COUNTERBUS y1[1])';
def.HeaderFiles = {'memory_bus.h'};
def.SourceFiles = {'memory_bus.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
```

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfun_work.c`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```
### Start Compiling rtwdemo_sfun_work
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570');
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('rtwdemo_sfun_work.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570');
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
```



```
### Finish Compiling rtwdemo_sfun_work
### Exit
```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again with the first input set to `'sfcn_tlc_generate'` to generate a TLC block file. The block file specifies how the generated code for a model calls the legacy code. If you do not generate a TLC block file and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_work.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an `rtwmakecfg.m` File for Code Generation

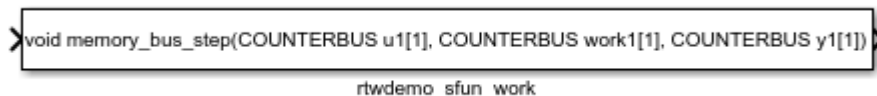
After you create the TLC block file, you can call the function `legacy_code()` again with the first input set to `'rtwmakecfg_generate'` to generate an `rtwmakecfg.m` file to support code generation. If the required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again with the first input set to `'slblock_generate'` to generate a masked S-function block that calls that S-function. The software places the block in a new model. From there you can copy it to an existing model.

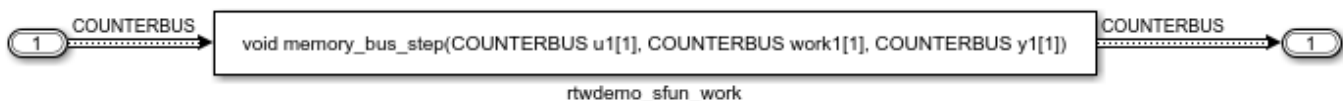
```
legacy_code('slblock_generate', def);
```



Show the Integration of the Model with Legacy Code

The model `rtwdemo_lct_work` shows integration of the model with the legacy code. The subsystem `memory_bus` serves as a harness for the call to the legacy C function.

```
open_system('rtwdemo_lct_work')
open_system('rtwdemo_lct_work/memory_bus')
sim('rtwdemo_lct_work')
```



Integrate External C Functions That Use Structure Arguments

Integrate legacy C functions with structure arguments that use Simulink® buses with the Legacy Code Tool.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional rtwmakecfg.m file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototype of the legacy functions being called in this example is:

```
counterbusFcn(COUNTERBUS *u1, int32_T u2, COUNTERBUS *y1, int32_T *y2)
```

COUNTERBUS is a struct typedef defined in `counterbus.h` and implemented with a `Simulink.Bus` object in the base workspace. The legacy source code is in the files `counterbus.h`, and `counterbus.c`.

```
evalin('base','load rtwdemo_lct_data.mat')

% rtwdemo_sfun_counterbus
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_counterbus';
def.OutputFcnSpec = ...
    'void counterbusFcn(COUNTERBUS u1[1], int32 u2, COUNTERBUS y1[1], int32 y2[1])';
def.HeaderFiles = {'counterbus.h'};
def.SourceFiles = {'counterbus.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
```

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfun_counterbus.c`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```
### Start Compiling rtwdemo_sfun_counterbus
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
```

```

    mex('rtwdemo_sfun_counterbus.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\T
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_counterbus
### Exit

```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again with the first input set to `'sfcn_tlc_generate'` to generate a TLC block file. The block file specifies how the generated code for a model calls the legacy code. If you do not generate a TLC block file and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_counterbus.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an `rtwmakecfg.m` File for Code Generation

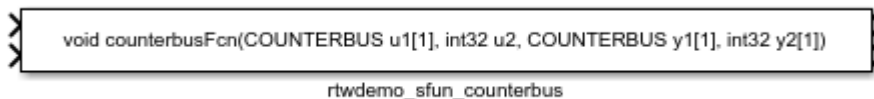
After you create the TLC block file, you can call the function `legacy_code()` again with the first input set to `'rtwmakecfg_generate'` to generate an `rtwmakecfg.m` file to support code generation. If the required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again with the first input set to `'slblock_generate'` to generate a masked S-function block that calls that S-function. The software places the block in a new model. From there you can copy it to an existing model.

```
legacy_code('slblock_generate', def);
```



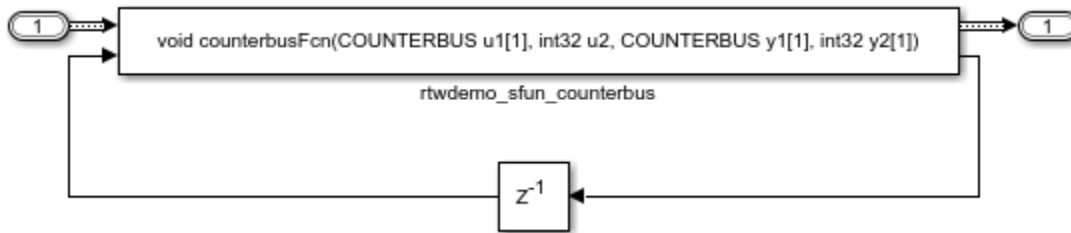
Show the Integration of the Model with Legacy Code

The model `rtwdemo_lct_bus` shows integration of the model with the legacy code. The subsystem `TestCounter` serves as a harness for the call to the legacy C function.

```

open_system('rtwdemo_lct_bus')
open_system('rtwdemo_lct_bus/TestCounter')
sim('rtwdemo_lct_bus')

```



Integrate External C Functions That Pass Input and Output Arguments as Signals with Complex Data

Integrate legacy C functions using complex signals with the Legacy Code Tool.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmkecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototype of the legacy functions being called in this example is:

```
void cplx_gain(creal_T *input, creal_T *gain, creal_T *output);
```

`creal_T` is the complex representation of a double. The legacy source code is in the files `cplxgain.h`, and `cplxgain.c`.

```
% rtwdemo_sfuns_gain_scalar
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfuns_cplx_gain';
def.OutputFcnSpec = ...
    ['void cplx_gain(complex<double> u1[1], '...
    'complex<double> p1[1], complex<double> y1[1])'];
def.HeaderFiles = {'cplxgain.h'};
def.SourceFiles = {'cplxgain.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
```

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfuns_cplx_gain.c`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```
### Start Compiling rtwdemo_sfun_cplx_gain
    mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
    mex('rtwdemo_sfun_cplx_gain.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TE
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_cplx_gain
### Exit
```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again with the first input set to 'sfcn_tlc_generate' to generate a TLC block file. The block file specifies how the generated code for a model calls the legacy code. If you do not generate a TLC block file and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_cplx_gain.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an `rtwmakecfg.m` File for Code Generation

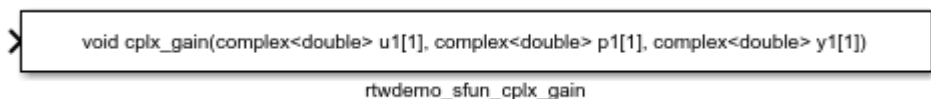
After you create the TLC block file, you can call the function `legacy_code()` again with the first input set to 'rtwmakecfg_generate' to generate an `rtwmakecfg.m` file to support code generation. If the required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again with the first input set to 'slblock_generate' to generate a masked S-function block that calls that S-function. The software places the block in a new model. From there you can copy it to an existing model.

```
legacy_code('slblock_generate', def);
```

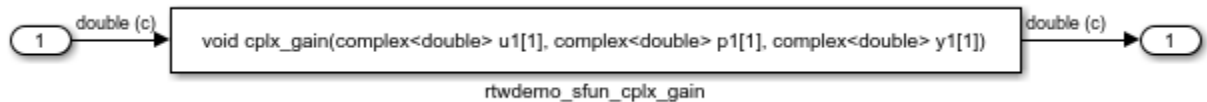


```
void cplx_gain(complex<double> u1[1], complex<double> p1[1], complex<double> y1[1])
rtwdemo_sfun_cplx_gain
```

Show the Integration of the Model with Legacy Code

The model `rtwdemo_lct_cplxgain` shows integration of the model with the legacy code. The subsystem `complex_gain` serves as a harness for the call to the legacy C function via the generate S-function.

```
if isempty(find_system('SearchDepth',0,'Name','rtwdemo_lct_cplxgain'))
    open_system('rtwdemo_lct_cplxgain')
    open_system('rtwdemo_lct_cplxgain/complex_gain')
    sim('rtwdemo_lct_cplxgain')
end
```



Integrate External C Functions That Pass Arguments That Have Inherited Dimensions

This example shows how to use the Legacy Code Tool to integrate legacy C functions whose arguments have inherited dimensions.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using `'initialize'` as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototypes of the legacy functions being called in this example are:

- `void mat_add(real_T *u1, real_T *u2, int32_T nbRows, int32_T nbCols, real_T *y1)`
- `void mat_mult(real_T *u1, real_T *u2, int32_T nbRows1, int32_T nbCols1, int32_T nbCols2, real_T *y1)`

`real_T` is a typedef to `double`, and `int32_T` is a typedef to a 32-bit integer. The legacy source code is in the files `mat_ops.h`, and `mat_ops.c`.

```
defs = [];
```

```
% rtwdemo_sfuns_mat_add
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfuns_mat_add';
def.OutputFcnSpec = ['void mat_add(double u1[][], double u2[][], ' ...
                    'int32 u3, int32 u4, double y1[size(u1,1)][size(u1,2)]]'];
def.HeaderFiles = {'mat_ops.h'};
```

```

def.SourceFiles = {'mat_ops.c'};
def.IncPaths    = {'rtwdemo_lct_src'};
def.SrcPaths    = {'rtwdemo_lct_src'};
defs = [defs; def];

% rtwdemo_sfun_mat_mult
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_mat_mult';
def.OutputFcnSpec = ['void mat_mult(double u1[p1][p2], double u2[p2][p3], '...
                    'int32 p1, int32 p2, int32 p3, double y1[p1][p3)'];
def.HeaderFiles  = {'mat_ops.h'};
def.SourceFiles  = {'mat_ops.c'};
def.IncPaths     = {'rtwdemo_lct_src'};
def.SrcPaths     = {'rtwdemo_lct_src'};
defs = [defs; def];

```

Generate S-Functions for Simulation

To generate C-MEX S-functions according to the description provided by the input argument 'defs', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-functions call the legacy functions during simulation. The source code for the S-function is in the files `rtwdemo_sfun_mat_add.c` and `rtwdemo_sfun_mat_mult.c`.

```
legacy_code('sfcn_cmex_generate', defs);
```

Compile the Generated S-Functions for Simulation

After you generate the C-MEX S-function source files, to compile the S-functions for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', defs);
```

```

### Start Compiling rtwdemo_sfun_mat_add
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('rtwdemo_sfun_mat_add.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_mat_add
### Exit

### Start Compiling rtwdemo_sfun_mat_mult
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('rtwdemo_sfun_mat_mult.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_mat_mult
### Exit

```

Generate TLC Block Files for Code Generation

After you compile the S-functions and use them in simulation, you can call the function `legacy_code()` again with the first input set to 'sfcn_tlc_generate' to generate TLC block files. Block files specify how the generated code for a model calls the legacy code. If you do not generate TLC block files and you

try to generate code for a model that includes the S-functions, code generation fails. The TLC block files for the S-functions are `rtwdemo_sfun_mat_add.tlc` and `rtwdemo_sfun_mat_mult.tlc`.

```
legacy_code('sfcn_tlc_generate', defs);
```

Generate an `rtwmakecfg.m` File for Code Generation

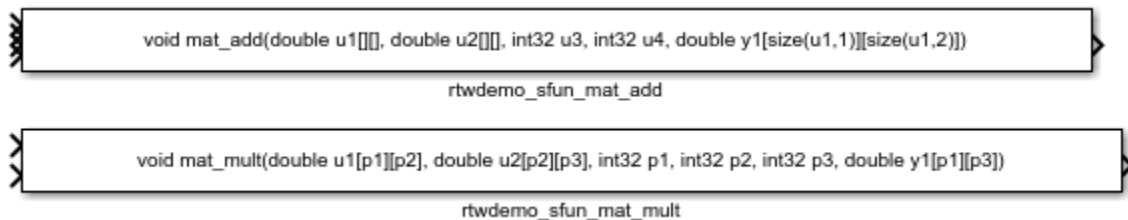
After you create the TLC block files, you can call the function `legacy_code()` again with the first input set to `'rtwmakecfg_generate'` to generate an `rtwmakecfg.m` file to support code generation. If the required source and header files for the S-functions are not in the same folder as the S-functions, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', defs);
```

Generate Masked S-Function Blocks for Calling the Generated S-Functions

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again with the first input set to `'slblock_generate'` to generate masked S-function blocks that call the S-functions. The software places the blocks in a new model. From there you can copy them to an existing model

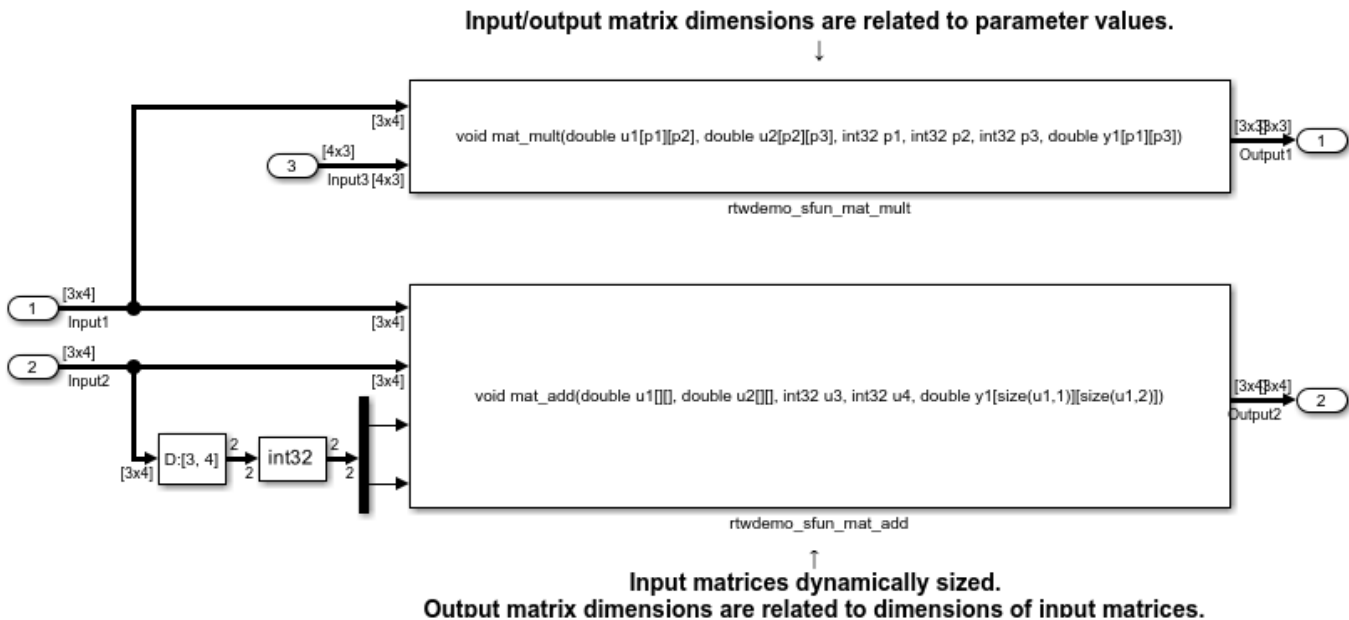
```
legacy_code('slblock_generate', defs);
```



Show the Generated Integration with Legacy Code

The model `rtwdemo_lct_inherit_dims` shows integration of the model with the legacy code. The subsystem `TestMatOps` serves as a harness for the calls to the legacy C functions, with unit delays serving to store the previous output values.

```
open_system('rtwdemo_lct_inherit_dims')
open_system('rtwdemo_lct_inherit_dims/TestMatOps')
sim('rtwdemo_lct_inherit_dims')
```

Integrate External C Functions That Pass Arguments as Multi-Dimensional Signals

This example shows how to use the Legacy Code Tool to integrate legacy C functions with multi-dimensional Signals.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional rtwmakecfg.m file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototype of the legacy functions being called in this example is:

```
void array3d_add(real_T *y1, real_T *u1, real_T *u2, int32_T nbRows, int32_T nbCols, int32_T nbPages);
```

`real_T` is a typedef to `double`, and `int32_T` is a typedef to a 32-bit integer. The legacy source code is in the files `ndarray_ops.h`, and `ndarray_ops.c`.

```
% rtwdemo_sfun_ndarray_add
def = legacy_code('initialize');
```

```

def.SFunctionName = 'rtwdemo_sfun_ndarray_add';
def.OutputFcnSpec =...
    ['void array3d_add(double y1[size(u1,1)][size(u1,2)][size(u1,3)], ',...
     'double u1[][][], double u2[][][], ' ...
     'int32 size(u1,1), int32 size(u1,2), int32 size(u1,3))'];
def.HeaderFiles   = {'ndarray_ops.h'};
def.SourceFiles   = {'ndarray_ops.c'};
def.IncPaths      = {'rtwdemo_lct_src'};
def.SrcPaths      = {'rtwdemo_lct_src'};

```

y1 is a 3-D output signal of same dimensions as the 3-D input signal u1. Note that the last 3 arguments passed to the legacy function correspond to the number of element in each dimension of the 3-D input signal u1.

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfun_ndarray_add.c`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```

### Start Compiling rtwdemo_sfun_ndarray_add
    mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
    mex('rtwdemo_sfun_ndarray_add.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_ndarray_add
### Exit

```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again with the first input set to 'sfcn_tlc_generate' to generate a TLC block file. The block file specifies how the generated code for a model calls the legacy code. If you do not generate a TLC block file and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_ndarray_add.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an `rtwmakecfg.m` File for Code Generation

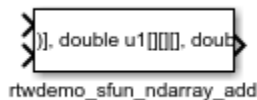
After you create the TLC block file, you can call the function `legacy_code()` again with the first input set to 'rtwmakecfg_generate' to generate an `rtwmakecfg.m` file to support code generation. If the required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again with the first input set to `'slblock_generate'` to generate a masked S-function block that calls that S-function. The software places the block in a new model. From there you can copy it to an existing model.

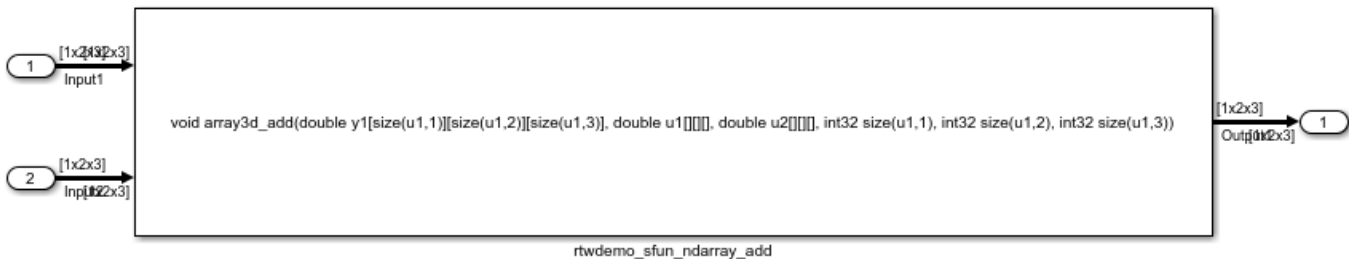
```
legacy_code('slblock_generate', def);
```



Showing the Generated Integration with Legacy Code

The model `rtwdemo_lct_ndarray` shows integration of the model with the legacy code. The subsystem `ndarray_add` serves as a harness for the call to the legacy C function.

```
open_system('rtwdemo_lct_ndarray')
open_system('rtwdemo_lct_ndarray/ndarray_add')
sim('rtwdemo_lct_ndarray')
```



Integrate External C Functions That Implement Start and Terminate Actions

Integrate legacy C functions that have start and terminate actions by using the Legacy Code Tool.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using `'initialize'` as the first

input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototypes of the legacy functions being called in this example are:

- void initFaultCounter(unsigned int *counter)
- void openLogFile(void **fid)
- void incAndLogFaultCounter(void *fid, unsigned int *counter, double time)
- void closeLogFile(void **fid)

The legacy source code is in the files `your_types.h`, `fault.h`, and `fault.c`.

```
% rtwdemo_sfun_fault
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_fault';
def.InitializeConditionsFcnSpec = 'initFaultCounter(uint32 work2[1])';
def.StartFcnSpec = 'openLogFile(void **work1)';
def.OutputFcnSpec = ...
    'incAndLogFaultCounter(void *work1, uint32 work2[1], double ul)';
def.TerminateFcnSpec = 'closeLogFile(void **work1)';
def.HeaderFiles = {'fault.h'};
def.SourceFiles = {'fault.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
def.Options.useTlcWithAccel = false;
```

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfun_fault.c`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```
### Start Compiling rtwdemo_sfun_fault
    mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570');
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
    mex('rtwdemo_sfun_fault.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570');
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_fault
### Exit
```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again with the first input set to 'sfcn_tlc_generate' to generate a TLC block file. The block file specifies how the generated code for a model calls the legacy code. If you do not generate a TLC

block file and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_fault.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an `rtwmakecfg.m` File for Code Generation

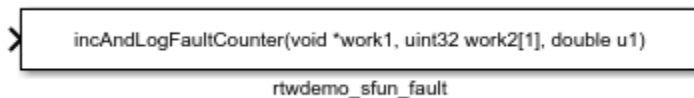
After you create the TLC block file, you can call the function `legacy_code()` again with the first input set to `'rtwmakecfg_generate'` to generate an `rtwmakecfg.m` file to support code generation. If the required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again with the first input set to `'slblock_generate'` to generate a masked S-function block that calls that S-function. The software places the block in a new model. From there you can copy it to an existing model.

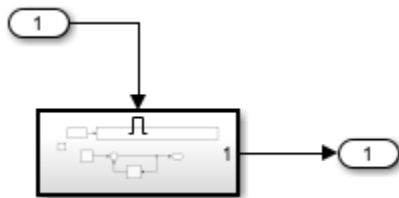
```
legacy_code('slblock_generate', def);
```



Showing the Generated Integration with Legacy Code

The model `rtwdemo_lct_start_term` shows integration of the model with the legacy code. The subsystem `TestFixpt` serves as a harness for the call to the legacy C function, and the scope compares the output of the function with the output of the built-in Simulink® product block; the results should be identical.

```
open_system('rtwdemo_lct_start_term')
open_system('rtwdemo_lct_start_term/TestFault')
sim('rtwdemo_lct_start_term')
```



Integrate External C Functions That Implement N-Dimensional Table Lookups

Integrate legacy C functions that implement N-dimensional table lookups by using the Legacy Code Tool.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototype of the legacy functions being called in this example is:

```
FLT directLookupTableND(const FLT *tableND, const UINT32 nbDims, const UINT32 *tableDims,
const UINT32 *tableIdx)
```

FLT is a typedef to float, and UINT32 is a typedef to unsigned int32. The legacy source code is in the files `your_types.h`, `lookupTable.h`, and `directLookupTableND.c`.

```
defs = [];
evalin('base', 'load rtwdemo_lct_data.mat')
```

```
% rtwdemo_sfun_dlut3D
```

```
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_dlut3D';
def.OutputFcnSpec = 'single y1 = DirectLookupTable3D(single p1[][][], uint32 p2[3], uint32 u1[3]);';
def.HeaderFiles = {'lookupTable.h'};
def.SourceFiles = {'directLookupTableND.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
defs = [defs; def];
```

```
% rtwdemo_sfun_dlut4D
```

```
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_dlut4D';
def.OutputFcnSpec = 'single y1 = DirectLookupTable4D(single p1[][][][], uint32 p2[4], uint32 u1[4]);';
def.HeaderFiles = {'lookupTable.h'};
def.SourceFiles = {'directLookupTableND.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
defs = [defs; def];
```

Generate S-Functions for Simulation

To generate C-MEX S-functions according to the description provided by the input argument 'defs', call the function `legacy_code()` again. Set the first input to 'sfcn_cmex_generate'. The S-functions call the legacy functions during simulation. The source code for the S-functions is in the files `rtwdemo_sfun_dlut3D.c` and `rtwdemo_sfun_dlut4D.c`.

```
legacy_code('sfcn_cmex_generate', defs);
```

Compile the Generated S-Functions for Simulation

After you generate the C-MEX S-function source files, to compile the S-functions for simulation with Simulink®, call the function `legacy_code()` again. Set the first input to 'compile'.

```
legacy_code('compile', defs);

### Start Compiling rtwdemo_sfun_dlut3D
    mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
    mex('rtwdemo_sfun_dlut3D.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_dlut3D
### Exit

### Start Compiling rtwdemo_sfun_dlut4D
    mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
    mex('rtwdemo_sfun_dlut4D.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_dlut4D
### Exit
```

Generate TLC Block Files for Code Generation

After you compile the S-functions and use them in simulation, you can call the function `legacy_code()` again. Set the first input to 'sfcn_tlc_generate' to generate TLC block files. Block files specify how the generated code for a model calls the legacy code. If you do not generate TLC block files and you try to generate code for a model that includes the S-functions, code generation fails. The TLC block files for the S-functions are `rtwdemo_sfun_dlut3D.tlc` and `rtwdemo_sfun_dlut4D.tlc`.

```
legacy_code('sfcn_tlc_generate', defs);
```

Generate an `rtwmakecfg.m` File for Code Generation

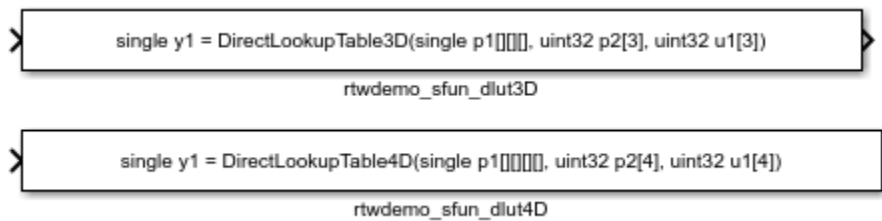
After you create the TLC block files, you can call the function `legacy_code()` again. Set the first input to 'rtwmakecfg_generate' to generate an `rtwmakecfg.m` file to support code generation. If the required source and header files for the S-functions are not in the same folder as the S-functions, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', defs);
```

Generate Masked S-Function Blocks for Calling the Generated S-Functions

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again. Set the first input to 'slblock_generate' to generate masked S-function blocks that call the S-functions. The software places the blocks in a new model. You can copy the blocks to an existing model.

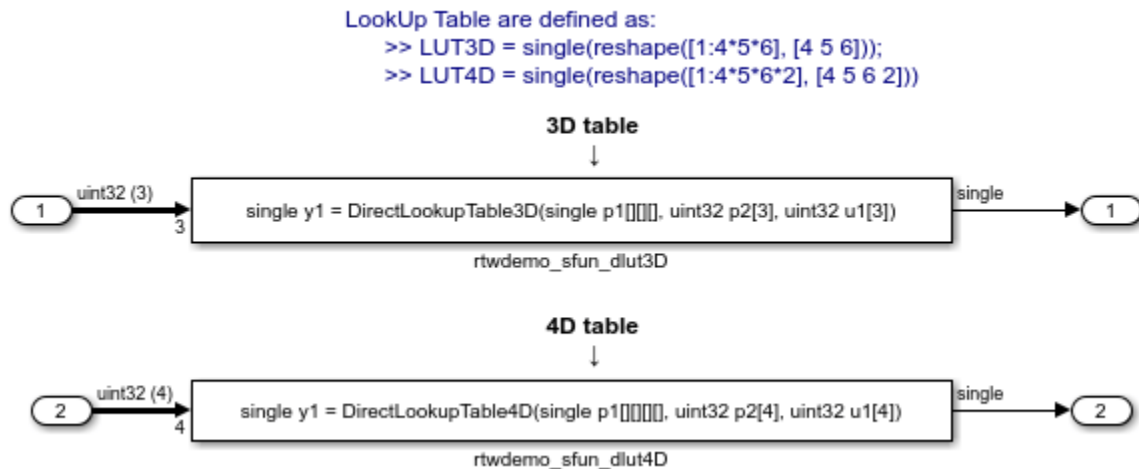
```
legacy_code('slblock_generate', defs);
```



Show the Generated Integration with Legacy Code

The model `rtwdemo_lct_lut` shows integration of the model with the legacy code. The subsystem `TestFixpt` serves as a harness for the call to the legacy C function, and the Display blocks compare the output of the function with the output of the built-in Simulink® lookup blocks. The results are identical.

```
open_system('rtwdemo_lct_lut')
open_system('rtwdemo_lct_lut/TestLut1')
sim('rtwdemo_lct_lut')
```



Integrate External C Functions That Pass Arguments as Multi-Dimensional Signals

This example shows how to use the Legacy Code Tool to integrate legacy C functions with multi-dimensional Signals.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototype of the legacy functions being called in this example is:

```
void array3d_add(real_T *y1, real_T *u1, real_T *u2, int32_T nbRows, int32_T nbCols, int32_T nbPages);
```

`real_T` is a typedef to `double`, and `int32_T` is a typedef to a 32-bit integer. The legacy source code is in the files `ndarray_ops.h`, and `ndarray_ops.c`.

```
% rtwdemo_sfun_ndarray_add
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_ndarray_add';
def.OutputFcnSpec =...
    ['void array3d_add(double y1[size(u1,1)][size(u1,2)][size(u1,3)], ',...
    'double u1[][][], double u2[][][], ' ...
    'int32 size(u1,1), int32 size(u1,2), int32 size(u1,3)'];
def.HeaderFiles = {'ndarray_ops.h'};
def.SourceFiles = {'ndarray_ops.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
```

`y1` is a 3-D output signal of same dimensions as the 3-D input signal `u1`. Note that the last 3 arguments passed to the legacy function correspond to the number of element in each dimension of the 3-D input signal `u1`.

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfun_ndarray_add.c`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```
### Start Compiling rtwdemo_sfun_ndarray_add
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570');
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('rtwdemo_sfun_ndarray_add.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570');
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_ndarray_add
### Exit
```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again with the first input set to `'sfcn_tlc_generate'` to generate a TLC block file. The block file specifies how the generated code for a model calls the legacy code. If you do not generate a TLC block file and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_ndarray_add.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an `rtwmakecfg.m` File for Code Generation

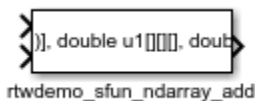
After you create the TLC block file, you can call the function `legacy_code()` again with the first input set to `'rtwmakecfg_generate'` to generate an `rtwmakecfg.m` file to support code generation. If the required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again with the first input set to `'slblock_generate'` to generate a masked S-function block that calls that S-function. The software places the block in a new model. From there you can copy it to an existing model.

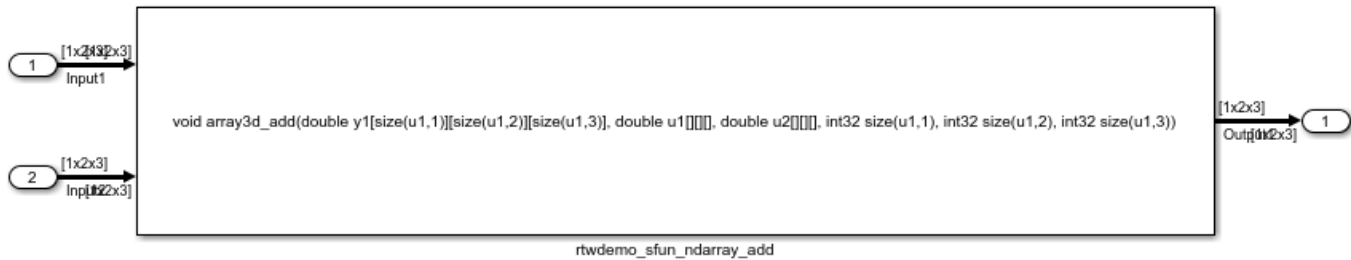
```
legacy_code('slblock_generate', def);
```



Showing the Generated Integration with Legacy Code

The model `rtwdemo_lct_ndarray` shows integration of the model with the legacy code. The subsystem `ndarray_add` serves as a harness for the call to the legacy C function.

```
open_system('rtwdemo_lct_ndarray')
open_system('rtwdemo_lct_ndarray/ndarray_add')
sim('rtwdemo_lct_ndarray')
```



Integrate External C Functions with a Block Sample Time Specified, Inherited, and Parameterized

This example shows how to use the Legacy Code Tool to integrate legacy C functions with the block's sample time specified, inherited and parameterized.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototype of the legacy functions being called in this example is:

```
FLT gainScalar(const FLT in, const FLT gain)
```

FLT is a typedef to float. The legacy source code is in the files `your_types.h`, `gain.h`, and `gainScalar.c`.

```
defs = [];
```

```
% rtwdemo_sfun_st_inherited
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_st_inherited';
def.OutputFcnSpec = 'single y1 = gainScalar(single u1, single p1)';
def.HeaderFiles = {'gain.h'};
def.SourceFiles = {'gainScalar.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
defs = [defs; def];
```

```
% rtwdemo_sfun_st_fixed
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_st_fixed';
def.OutputFcnSpec = 'single y1 = gainScalar(single u1, single p1)';
def.HeaderFiles = {'gain.h'};
```

```

def.SourceFiles = {'gainScalar.c'};
def.IncPaths    = {'rtwdemo_lct_src'};
def.SrcPaths    = {'rtwdemo_lct_src'};
def.SampleTime  = [2 1];
defs = [defs; def];

% rtwdemo_sfun_st_parameterized
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_st_parameterized';
def.OutputFcnSpec = 'single y1 = gainScalar(single u1, single p1)';
def.HeaderFiles   = {'gain.h'};
def.SourceFiles   = {'gainScalar.c'};
def.IncPaths      = {'rtwdemo_lct_src'};
def.SrcPaths      = {'rtwdemo_lct_src'};
def.SampleTime    = 'parameterized';
defs = [defs; def];

```

Generate S-Functions for Simulation

To generate C-MEX S-functions according to the description provided by the input argument 'defs', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-functions call the legacy functions during simulation. The source code for the S-functions is in the files `rtwdemo_sfun_st_inherited.c` and `rtwdemo_sfun_st_fixed.c`. `rtwdemo_sfun_st_parameterized.c`.

```
legacy_code('sfcn_cmex_generate', defs);
```

Compile the Generated S-Functions for Simulation

After you generate the C-MEX S-function source files, to compile the S-functions for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', defs);
```

```

### Start Compiling rtwdemo_sfun_st_inherited
    mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
    mex('rtwdemo_sfun_st_inherited.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_st_inherited
### Exit

### Start Compiling rtwdemo_sfun_st_fixed
    mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
    mex('rtwdemo_sfun_st_fixed.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_st_fixed
### Exit

### Start Compiling rtwdemo_sfun_st_parameterized
    mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.

```

```

    mex('rtwdemo_sfun_st_parameterized.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_st_parameterized
### Exit

```

Generate TLC Block Files for Code Generation

After you compile the S-functions and use them in simulation, you can call the function `legacy_code()` again with the first input set to `'sfcn_tlc_generate'` to generate TLC block files. Block files specify how the generated code for a model calls the legacy code. If you do not generate TLC block files and you try to generate code for a model that includes the S-functions, code generation fails. The TLC block files for the S-functions are `rtwdemo_sfun_st_inherited.tlc` and `rtwdemo_sfun_st_fixed.tlc`. `rtwdemo_sfun_st_parameterized.tlc`.

```
legacy_code('sfcn_tlc_generate', defs);
```

Generate an `rtwmakecfg.m` File for Code Generation

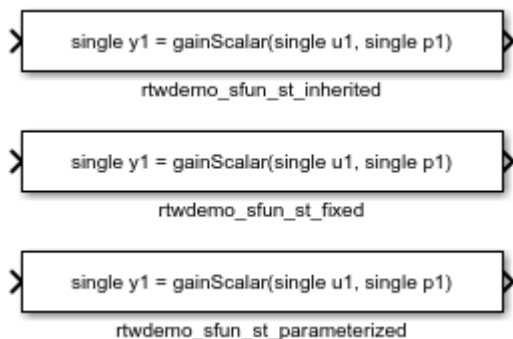
After you create the TLC block files, you can call the function `legacy_code()` again with the first input set to `'rtwmakecfg_generate'` to generate an `rtwmakecfg.m` file to support code generation. If the required source and header files for the S-functions are not in the same folder as the S-functions, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', defs);
```

Generate Masked S-Function Blocks for Calling the Generated S-Functions

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again with the first input set to `'slblock_generate'` to generate masked S-function blocks that call the S-functions. The software places the blocks in a new model. From there you can copy them to an existing model.

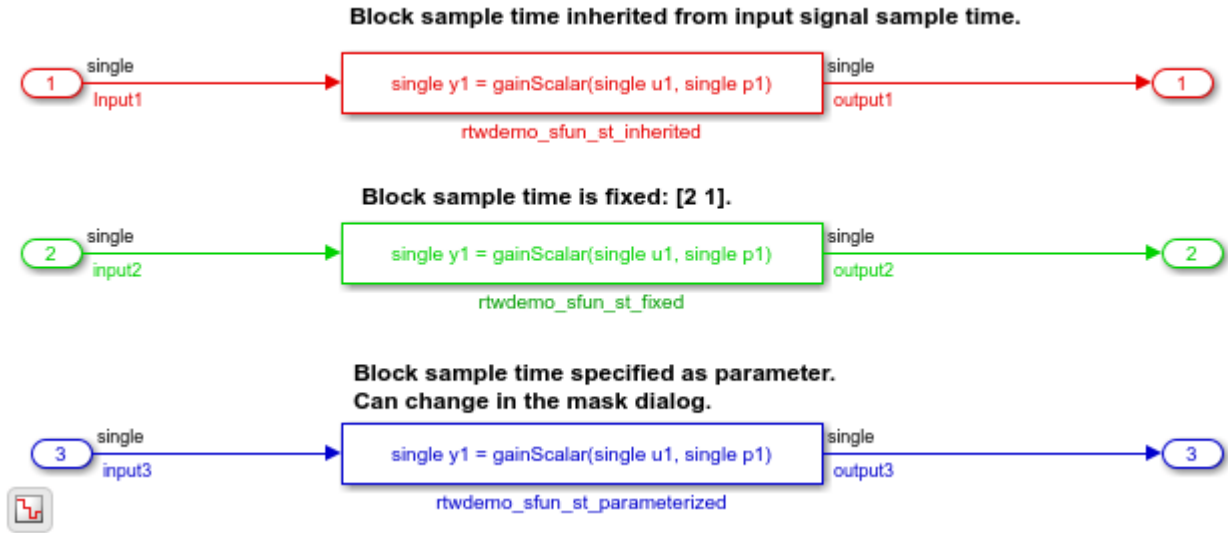
```
legacy_code('slblock_generate', defs);
```



Show the Generated Integration with Legacy Code

The model `rtwdemo_lct_sampletime` shows integration of the model with the legacy code. The subsystem `sample_time` serves as a harness for the calls to the legacy C functions, with unit delays serving to store the previous output values.

```
open_system('rtwdemo_lct_sampletime')
open_system('rtwdemo_lct_sampletime/sample_time')
sim('rtwdemo_lct_sampletime')
```



See Also

legacy_code

Related Examples

- “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6
- “Integrate C Functions Using Legacy Code Tool”
- “Call External C Code from Model and Generated Code” (Embedded Coder)

Integrate External C Functions That Pass Input and Output Arguments as Parameters with a Fixed-Point Data Type

Integrate legacy C functions that pass their inputs and outputs by using parameters of a fixed-point data type with the Legacy Code Tool.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototype of the legacy functions being called in this example is:

```
myFixpt timesS16(const myFixpt in1, const myFixpt in2, const uint8_T fracLength)
```

`myFixpt` is logically a fixed point data type, which is physically a typedef to a 16-bit integer:

```
myFixpt = Simulink.NumericType;
myFixpt.DataTypeMode = 'Fixed-point: binary point scaling';
myFixpt.Signed = true;
myFixpt.WordLength = 16;
myFixpt.FractionLength = 10;
myFixpt.IsAlias = true;
myFixpt.HeaderFile = 'timesFixpt.h';
```

The legacy source code is in the files `timesFixpt.h`, and `timesS16.c`.

```
% rtwdemo_sfundemo_gain_fixpt
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfundemo_gain_fixpt';
def.OutputFcnSpec = 'myFixpt y1 = timesS16(myFixpt u1, myFixpt p1, uint8 p2)';
def.HeaderFiles = {'timesFixpt.h'};
def.SourceFiles = {'timesS16.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
```

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfundemo_gain_fixpt.c`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```
### Start Compiling rtwdemo_sfun_gain_fixpt
    mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
    mex('rtwdemo_sfun_gain_fixpt.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\T
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_gain_fixpt
### Exit
```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again. Set the first input to 'sfcn_tlc_generate' to generate a TLC block file. The block file specifies how the generated code for a model calls the legacy code. If you do not generate a TLC block file and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_gain_fixpt.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an `rtwmakecfg.m` File for Code Generation

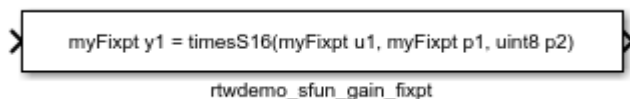
After you create the TLC block file, you can call the function `legacy_code()` again. Set the first input to 'rtwmakecfg_generate' to generate an `rtwmakecfg.m` file that supports code generation. If the required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again. Set the first input to 'slblock_generate' to generate a masked S-function block that calls that S-function. The software places the block in a new model. You can copy the block to an existing model.

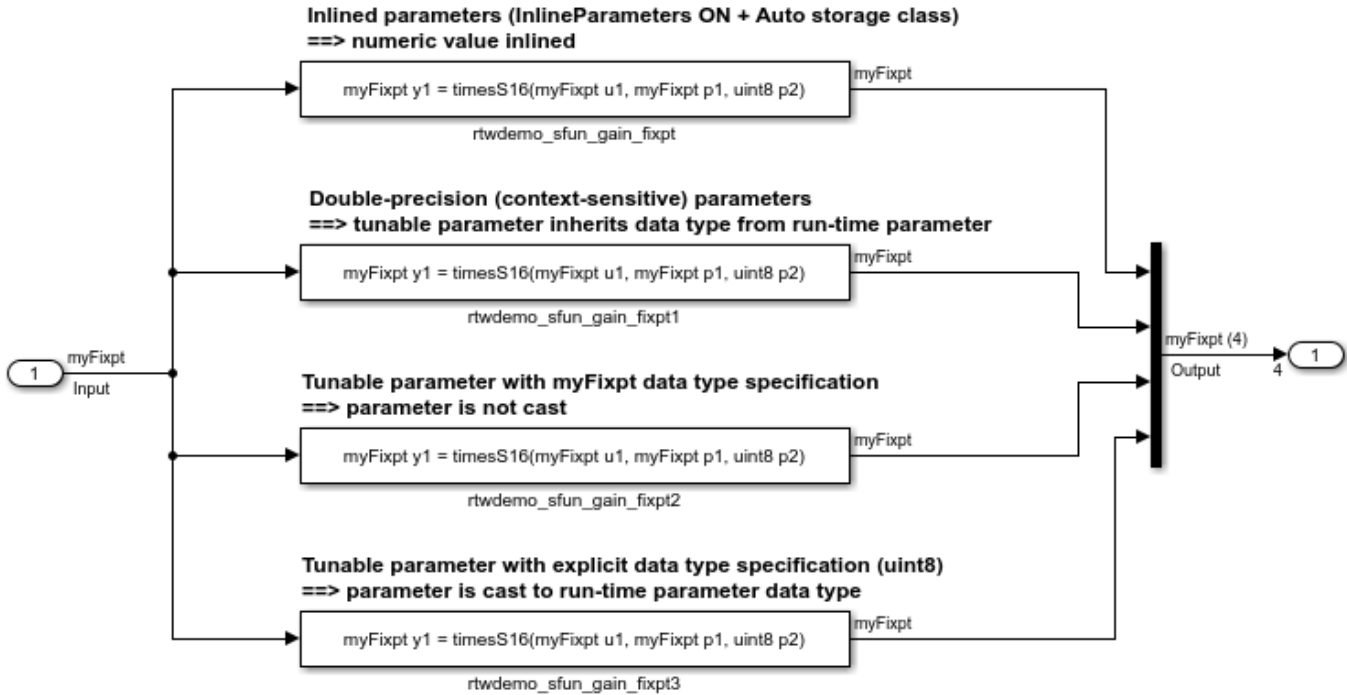
```
legacy_code('slblock_generate', def);
```



Show the Generated Integration with Legacy Code

The model `rtwdemo_lct_fixpt_params` shows integration of the model with the legacy code. The subsystem `TestFixpt` serves as a harness for the call to the legacy C function via the generated S-function.


```
open_system('rtwdemo_lct_fixpt_params')
open_system('rtwdemo_lct_fixpt_params/TestFixpt')
sim('rtwdemo_lct_fixpt_params')
```



See Also

legacy_code

Related Examples

- “Integrate C Functions Using Legacy Code Tool”
- “Call External C Code from Model and Generated Code” (Embedded Coder)

Integrate External C Functions That Implement N-Dimensional Table Lookups

Integrate legacy C functions that implement N-dimensional table lookups by using the Legacy Code Tool.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C-MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a TLC block file and optional `rtwmakecfg.m` file that specifies how the generated code for a model calls the legacy code.

Provide the Legacy Function Specification

Legacy Code Tool functions take a specific data structure or array of structures as the argument. You can initialize the data structure by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The prototype of the legacy functions being called in this example is:

```
FLT directLookupTableND(const FLT *tableND, const UINT32 nbDims, const UINT32 *tableDims,
const UINT32 *tableIdx)
```

FLT is a typedef to float, and UINT32 is a typedef to unsigned int32. The legacy source code is in the files `your_types.h`, `lookupTable.h`, and `directLookupTableND.c`.

```
defs = [];
evalin('base','load rtwdemo_lct_data.mat')
```

```
% rtwdemo_sfundlut3D
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfundlut3D';
def.OutputFcnSpec = 'single y1 = DirectLookupTable3D(single p1[][][], uint32 p2[3], uint32 u1[3]);
def.HeaderFiles = {'lookupTable.h'};
def.SourceFiles = {'directLookupTableND.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
defs = [defs; def];
```

```
% rtwdemo_sfundlut4D
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfundlut4D';
def.OutputFcnSpec = 'single y1 = DirectLookupTable4D(single p1[][][][][], uint32 p2[4], uint32 u1[4]);
def.HeaderFiles = {'lookupTable.h'};
def.SourceFiles = {'directLookupTableND.c'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
defs = [defs; def];
```

Generate S-Functions for Simulation

To generate C-MEX S-functions according to the description provided by the input argument 'defs', call the function `legacy_code()` again. Set the first input to 'sfcn_cmex_generate'. The S-functions call

the legacy functions during simulation. The source code for the S-functions is in the files `rtwdemo_sfun_dlut3D.c` and `rtwdemo_sfun_dlut4D.c`.

```
legacy_code('sfcn_cmex_generate', defs);
```

Compile the Generated S-Functions for Simulation

After you generate the C-MEX S-function source files, to compile the S-functions for simulation with Simulink®, call the function `legacy_code()` again. Set the first input to `'compile'`.

```
legacy_code('compile', defs);
```

```
### Start Compiling rtwdemo_sfun_dlut3D
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570')
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('rtwdemo_sfun_dlut3D.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570')
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_dlut3D
### Exit

### Start Compiling rtwdemo_sfun_dlut4D
mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570')
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('rtwdemo_sfun_dlut4D.c', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570')
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_dlut4D
### Exit
```

Generate TLC Block Files for Code Generation

After you compile the S-functions and use them in simulation, you can call the function `legacy_code()` again. Set the first input to `'sfcn_tlc_generate'` to generate TLC block files. Block files specify how the generated code for a model calls the legacy code. If you do not generate TLC block files and you try to generate code for a model that includes the S-functions, code generation fails. The TLC block files for the S-functions are `rtwdemo_sfun_dlut3D.tlc` and `rtwdemo_sfun_dlut4D.tlc`.

```
legacy_code('sfcn_tlc_generate', defs);
```

Generate an `rtwmakecfg.m` File for Code Generation

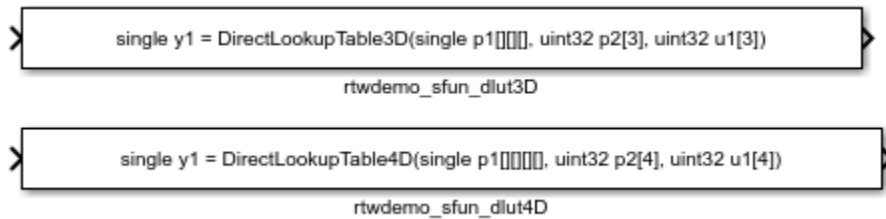
After you create the TLC block files, you can call the function `legacy_code()` again. Set the first input to `'rtwmakecfg_generate'` to generate an `rtwmakecfg.m` file to support code generation. If the required source and header files for the S-functions are not in the same folder as the S-functions, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', defs);
```

Generate Masked S-Function Blocks for Calling the Generated S-Functions

After you compile the C-MEX S-function source, you can call the function `legacy_code()` again. Set the first input to `'slblock_generate'` to generate masked S-function blocks that call the S-functions. The software places the blocks in a new model. You can copy the blocks to an existing model.

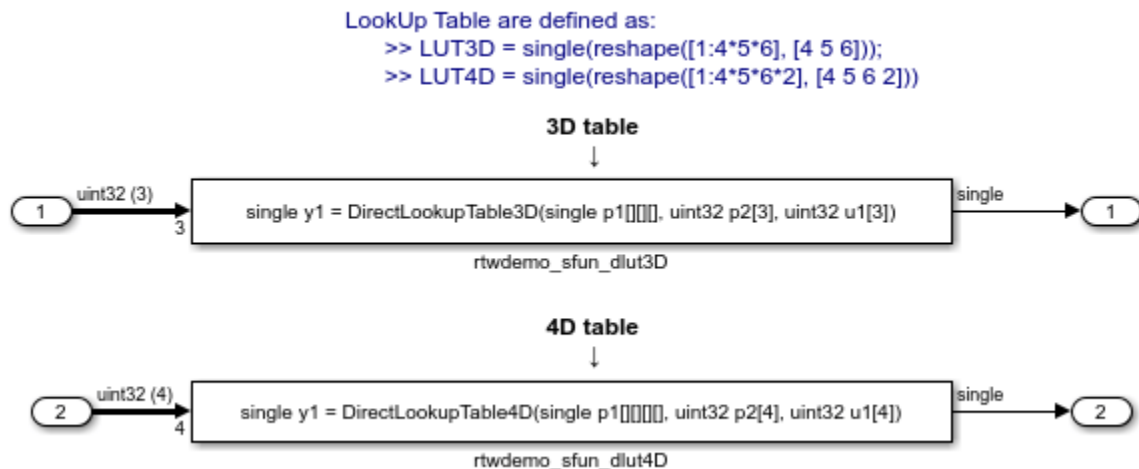
```
legacy_code('slblock_generate', defs);
```



Show the Generated Integration with Legacy Code

The model `rtwdemo_lct_lut` shows integration of the model with the legacy code. The subsystem `TestFixpt` serves as a harness for the call to the legacy C function, and the Display blocks compare the output of the function with the output of the built-in Simulink® lookup blocks. The results are identical.

```
open_system('rtwdemo_lct_lut')
open_system('rtwdemo_lct_lut/TestLut1')
sim('rtwdemo_lct_lut')
```



See Also

`legacy_code`

Related Examples

- “Integrate C Functions Using Legacy Code Tool”
- “Call External C Code from Model and Generated Code” (Embedded Coder)

Integrate External C++ Object Methods

Integrate legacy C++ object methods by using the Legacy Code Tool.

With the Legacy Code Tool, you can:

- Provide the legacy function specification.
- Generate a C++ MEX S-function that calls the legacy code during simulation.
- Compile and build the generated S-function for simulation.
- Generate a block TLC file and optional rtwmakecfg.m file that calls the legacy code during code generation.

Provide the Legacy Function Specification

Functions provided with the Legacy Code Tool take a specific data structure or array of structures as the argument. The data structure is initialized by calling the function `legacy_code()` using 'initialize' as the first input. After initializing the structure, assign its properties to values corresponding to the legacy code being integrated. The definition of the legacy C++ class in this example is:

```
class adder {
private:
    int int_state;
public:
    adder();
    int add_one(int increment);
    int get_val();
};
```

The legacy source code is in the files `adder_cpp.h` and `adder_cpp.cpp`.

```
% rtwdemo_sfun_adder_cpp
def = legacy_code('initialize');
def.SFunctionName = 'rtwdemo_sfun_adder_cpp';
def.StartFcnSpec = 'createAdder()';
def.OutputFcnSpec = 'int32 y1 = adderOutput(int32 u1)';
def.TerminateFcnSpec = 'deleteAdder()';
def.HeaderFiles = {'adder_cpp.h'};
def.SourceFiles = {'adder_cpp.cpp'};
def.IncPaths = {'rtwdemo_lct_src'};
def.SrcPaths = {'rtwdemo_lct_src'};
def.Options.language = 'C++';
def.Options.useTlcWithAccel = false;
```

Generate an S-Function for Simulation

To generate a C-MEX S-function according to the description provided by the input argument 'def', call the function `legacy_code()` again with the first input set to 'sfcn_cmex_generate'. The S-function calls the legacy functions during simulation. The source code for the S-function is in the file `rtwdemo_sfun_adder_cpp.cpp`.

```
legacy_code('sfcn_cmex_generate', def);
```

Compile the Generated S-Function for Simulation

After you generate the C-MEX S-function source file, to compile the S-function for simulation with Simulink®, call the function `legacy_code()` again with the first input set to 'compile'.

```
legacy_code('compile', def);
```

```
### Start Compiling rtwdemo_sfun_adder_cpp
    mex('-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\TEMP\Bdoc23a_2213998_3568\ib57
Building with 'Microsoft Visual C++ 2019'.
MEX completed successfully.
    mex('rtwdemo_sfun_adder_cpp.cpp', '-IB:\matlab\toolbox\rtw\rtwdemos\rtwdemo_lct_src', '-IC:\
Building with 'Microsoft Visual C++ 2019'.
MEX completed successfully.
### Finish Compiling rtwdemo_sfun_adder_cpp
### Exit
```

Generate a TLC Block File for Code Generation

After you compile the S-function and use it in simulation, you can call the function `legacy_code()` again. Set the first input to 'sfcn_tlc_generate' to generate a TLC block file that supports code generation through Simulink® Coder™. If the TLC block file is not created and you try to generate code for a model that includes the S-function, code generation fails. The TLC block file for the S-function is: `rtwdemo_sfun_adder_cpp.tlc`.

```
legacy_code('sfcn_tlc_generate', def);
```

Generate an rtwmakecfg.m File for Code Generation

After you create the TLC block file, you can call the function `legacy_code()` again. Set the first input to 'rtwmakecfg_generate' to generate an `rtwmakecfg.m` file that supports code generation through Simulink® Coder™. If the required source and header files for the S-function are not in the same folder as the S-function, and you want to add these dependencies in the makefile produced during code generation, generate the `rtwmakecfg.m` file.

```
legacy_code('rtwmakecfg_generate', def);
```

Generate a Masked S-Function Block for Calling the Generated S-Function

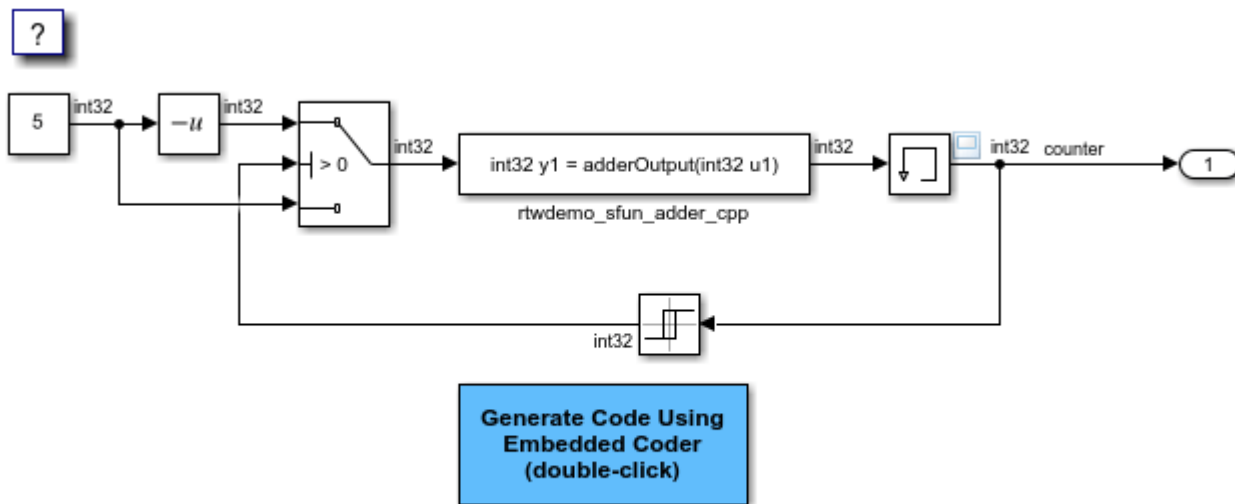
After you compile the C-MEX S-function source, you can call the function `legacy_code()` again. Set the first input to 'slblock_generate' to generate a masked S-function block that is configured to call that S-function. The software places the block in a new model. You can copy the block to an existing model.

```
% legacy_code('slblock_generate', def);
```

Show the Generated Integration with Legacy Code

The model `rtwdemo_lct_cpp` shows integration with the legacy code.

```
open_system('rtwdemo_lct_cpp')
sim('rtwdemo_lct_cpp')
```



Copyright 1990-2020 The MathWorks, Inc.

See Also

legacy_code

Related Examples

- "Integrate C Functions Using Legacy Code Tool"
- "Call External C Code from Model and Generated Code" (Embedded Coder)

External Code Integration Examples

This topic shows various scenarios of external code integration.

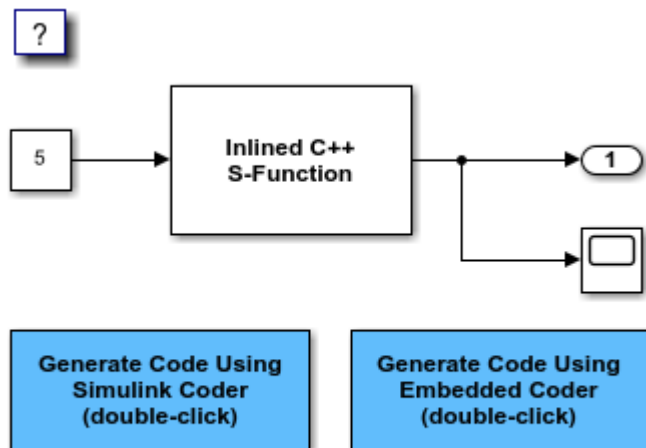
Integrate External C++ Code into a Model Using S-Functions

This example shows you how to integrate C++ code into a model by using S-functions.

Open Example Model

Open the example model `rtwdemo_cpp`.

```
open_system('rtwdemo_cpp');
```



Copyright 1994-2020 The MathWorks, Inc.

Configure C++ Compiler

1. In the MATLAB® Command Window, enter this command:

```
mex -setup
```

2. When prompted, select a C++ compiler that is installed on your development computer.

Integrate C++ Code

You can integrate C++ code into a model by using S-functions.

1. Prepare the S-function source.

- Open `sfun_cppcount_cpp.cpp`
- Open `sfun_cppcount_cpp.h`

2. Open the **Simulink Coder** or **Embedded Coder** app.

3. Configure S-function source files and folders. Open the Model Configuration Parameter dialog box, navigate to the **Code Generation > Custom Code** pane, and add the S-function source files and include folders.

4. Inline the S-function to eliminate overhead of S-functions in the generated code.

Open `sfun_cppcount_cpp.tlc`

Configure Model for C++ Code Generation

Set model configuration parameter **Language** to C++.

Generate and review code.

Generate and review the C++ code.

Related Topics

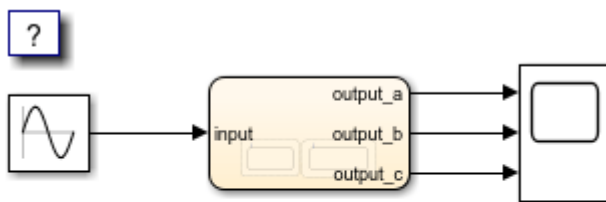
- “Programming Language Considerations” on page 31-4
- “C++ Language Support Limitations” on page 31-4

Insert External C and C++ Code into Stateflow Charts for Code Generation

This example shows how to use Stateflow® to integrate external code into a model.

Open Model

```
model='rtwdemo_sfcustom';
open_system(model);
```



[▶ Build model using Simulink Coder](#)

Copyright 1994-2020 The MathWorks, Inc.

Integrate Code

1. The example includes the custom header file `my_header.c` and the custom source file `my_function.c`.

```
%Open files my_header.h and my_function.c
eval('edit my_header.h')
eval('edit my_function.c')
```

2. On the Model Configuration Parameters dialog box **Simulation Target** pane, enter the custom source file and header file. Also enter additional include directories and source files.

In this example, the custom header file `my_header.c` and source file `my_function.c` are entered on the **Simulation Target** pane.

```
%Open Configuration Parameters dialog box
configSet = getActiveConfigSet(model);
openDialog(configSet);
```

3. To generate code, on the Model Configuration Parameters dialog box **Code Generation > Custom Code** pane, enter the same custom source file and header file. Also enter the same additional include directories and source files.

In this example, the custom header file `my_header.c` and source file `my_function.c` are entered on the **Code Generation > Custom Code** pane.

Open the **Code Generation > Custom Code** pane of the Model Configuration Parameters dialog box.

```
configSet = getActiveConfigSet(model);
openDialog(configSet);
```

Generate Code

```
slbuild('rtwdemo_sfcustom')

### Starting build procedure for: rtwdemo_sfcustom
### Successful completion of build procedure for: rtwdemo_sfcustom
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_sfcustom	Code generated and compiled.	Code generation information file does not exist.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 8.4901s
```

Call C Code from Stateflow

To call custom C code functions from Stateflow, use the same syntax as graphical function calls:

```
result = my_custom_function(in_args);
```

To call variables of structure type, use the dot notation: `result = my_var.my_field;`

See Also

- “Configure Custom Code Settings for Simulation” (Stateflow)
- “Access Custom C++ Code in Stateflow Charts” (Stateflow)

Close Model

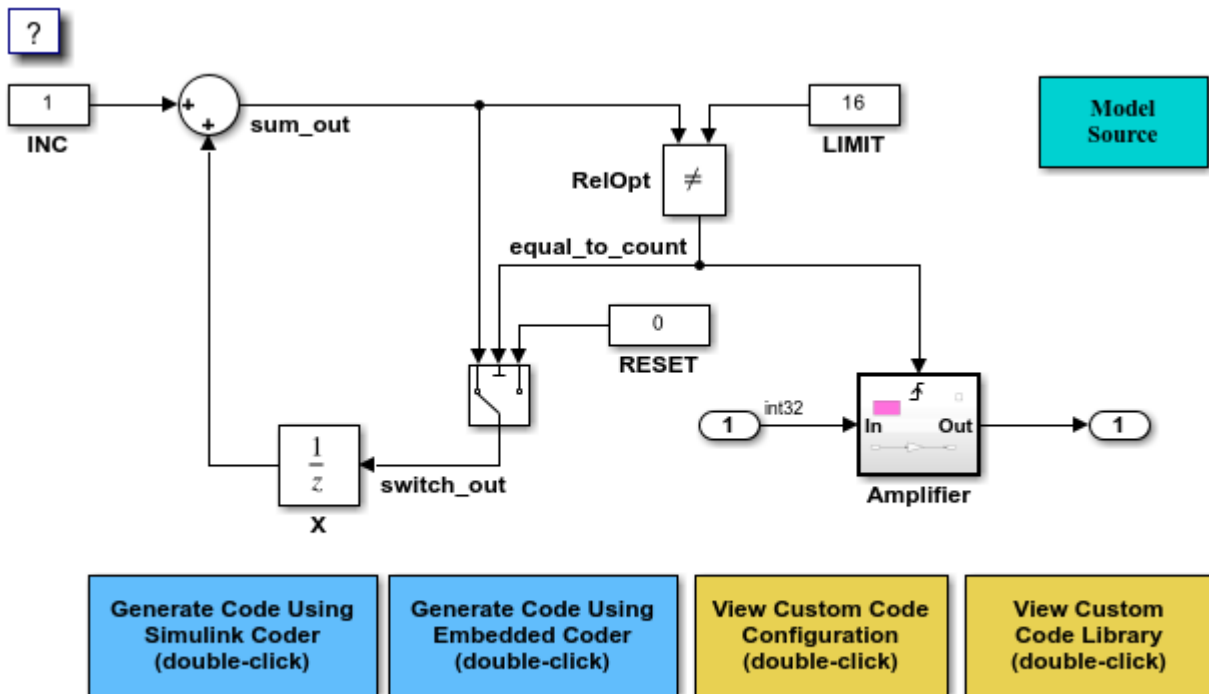
```
rtwdemoclean;
close_system('rtwdemo_sfcustom',0);
```

Integrate External C Code into Generated Code by Using Custom Code Blocks and Model Configuration Parameters

This example shows how to place external code in generated code by using custom code blocks and model configuration parameters.

1. Open the model `rtwdemo_slcustcode`.

```
open_system('rtwdemo_slcustcode')
```



Copyright 1994-2022 The MathWorks, Inc.

2. Open the **Simulink Coder** or **Embedded Coder** app.
3. Open the Model Configuration Parameters dialog box and navigate to the **Custom Code** pane.
4. On the **Additional source code** tab, examine the settings for model configuration parameters **Additional code** and **Initialize code**.
 - **Additional code** specifies a comment and sets the variable `GLOBAL_INT2` to `-1`.
 - **Initialize code** initializes the variable `GLOBAL_INT2` to `1`.
5. Close the dialog box.

6. Double-click the Model Source block. Block parameter **Top of Model Source** specifies that the code generator declare the variable GLOBAL_INT1 and set it to 0 at the top of the generated file rtwdemo_slcustcode.c.

7. Open the triggered subsystem `Amplifier`. The subsystem includes the System Outputs block. The code generator places code that you specify in that block in the generated code for the nearest parent atomic subsystem. In this case, the code generator places the external code in the generated code for the `Amplifier` subsystem. The external code:

- Declares the pointer variable `*intPtr` and initializes it with the value of variable GLOBAL_INT1.
- Sets the pointer variable to -1 during execution.
- Resets the pointer variable to 0 before exiting.

8. Generate code and a code generation report.

9. Examine the code in the generated source file rtwdemo_slcustcode.c. At the top of the file, after the `#include` statements, you find the following declaration code. The example specifies the first declaration with the **Additional code** configuration parameter and the second declaration with the Model Source block.

```
int_T GLOBAL_INT2 = -1;
```

```
int_T GLOBAL_INT1 = 0;
```

The Output function for the `Amplifier` subsystem includes the following code, which shows the external code integrated with generated code that applies the gain. The example specifies the three lines of code for the pointer variable with the System Outputs block in the `Amplifier` subsystem.

```
int_T *intPtr = &GLOBAL_INT1;
```

```
*intPtr = -1;
```

```
rtwdemo_slcustcode_Y.Output = rtwdemo_slcustcode_U.Input << 1;
```

```
*intPtr = 0;
```

The following assignment appears in the model initialize entry-point function. The example specifies this assignment with the **Initialize code** configuration parameter.

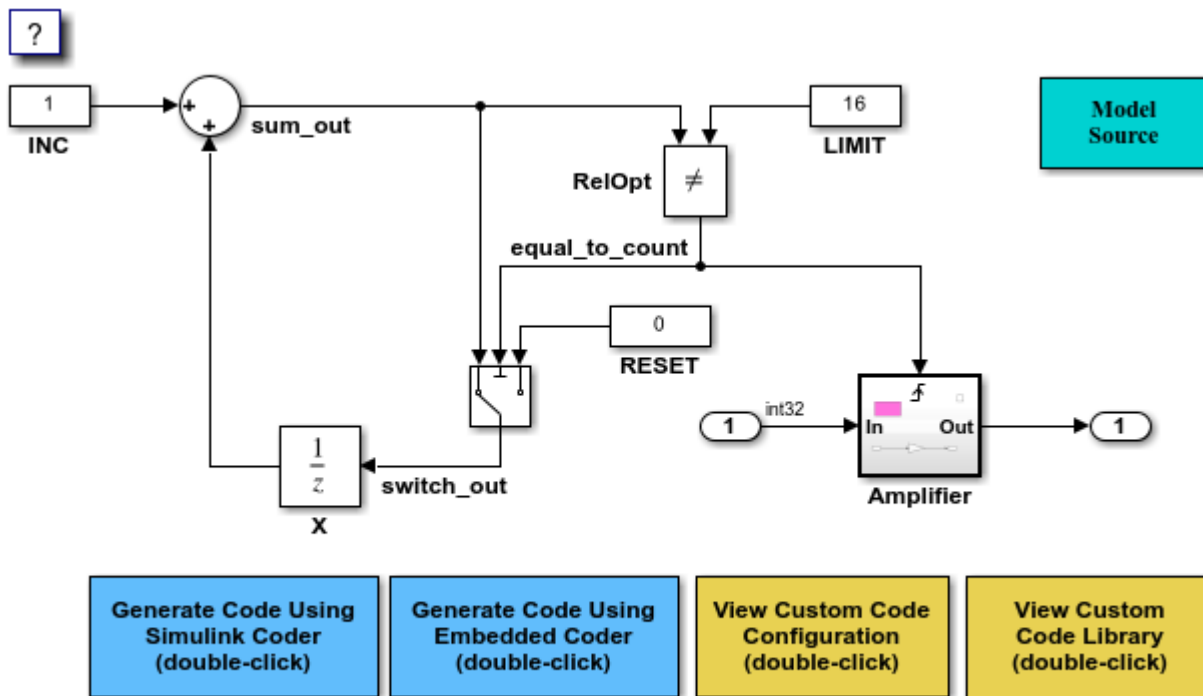
```
GLOBAL_INT2 = 1;
```

Integrate External C Code into Generated Code by Using Custom Code Blocks and Model Configuration Parameters

This example shows how to place external code in generated code by using custom code blocks and model configuration parameters.

1. Open the model rtwdemo_slcustcode.

```
open_system('rtwdemo_slcustcode')
```



Copyright 1994-2022 The MathWorks, Inc.

2. Open the **Simulink Coder** or **Embedded Coder** app.
3. Open the Model Configuration Parameters dialog box and navigate to the **Custom Code** pane.
4. On the **Additional source code** tab, examine the settings for model configuration parameters **Additional code** and **Initialize code**.
 - **Additional code** specifies a comment and sets the variable `GLOBAL_INT2` to -1.
 - **Initialize code** initializes the variable `GLOBAL_INT2` to 1.
5. Close the dialog box.
6. Double-click the Model Source block. Block parameter **Top of Model Source** specifies that the code generator declare the variable `GLOBAL_INT1` and set it to 0 at the top of the generated file `rtwdemo_slcustcode.c`.
7. Open the triggered subsystem **Amplifier**. The subsystem includes the System Outputs block. The code generator places code that you specify in that block in the generated code for the nearest parent atomic subsystem. In this case, the code generator places the external code in the generated code for the **Amplifier** subsystem. The external code:
 - Declares the pointer variable `*intPtr` and initializes it with the value of variable `GLOBAL_INT1`.
 - Sets the pointer variable to -1 during execution.
 - Resets the pointer variable to 0 before exiting.
8. Generate code and a code generation report.

9. Examine the code in the generated source file `rtwdemo_slcustcode.c`. At the top of the file, after the `#include` statements, you find the following declaration code. The example specifies the first declaration with the **Additional code** configuration parameter and the second declaration with the Model Source block.

```
int_T GLOBAL_INT2 = -1;
```

```
int_T GLOBAL_INT1 = 0;
```

The Output function for the Amplifier subsystem includes the following code, which shows the external code integrated with generated code that applies the gain. The example specifies the three lines of code for the pointer variable with the System Outputs block in the Amplifier subsystem.

```
int_T *intPtr = &GLOBAL_INT1;
```

```
*intPtr = -1;
```

```
rtwdemo_slcustcode_Y.Output = rtwdemo_slcustcode_U.Input << 1;
```

```
*intPtr = 0;
```

The following assignment appears in the model initialize entry-point function. The example specifies this assignment with the **Initialize code** configuration parameter.

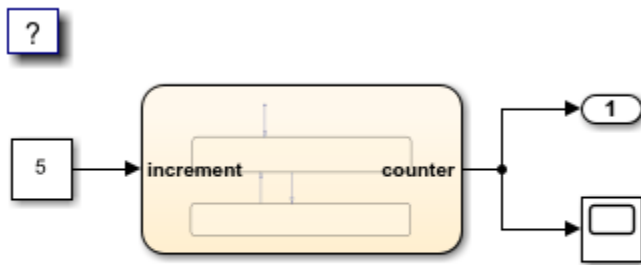
```
GLOBAL_INT2 = 1;
```

Insert External C++ Code into Stateflow Charts for Code Generation

This example shows how to use Stateflow® to integrate external C++ code into a model.

Open Model

```
model='rtwdemo_sfcpp';  
open_system(model);
```



Copyright 1994-2020 The MathWorks, Inc.

To setup a C++ compiler, on the command line, enter `mex -setup`.

Integrate C++ Code

1. Create C-style wrapper functions or access macros for each method called from the C action language.

```
%Open example wrapper functions.
filename=fullfile(matlabroot, 'toolbox', 'rtw', 'rtwdemos', 'custom_cpp_src', 'adder_cpp.cpp');
eval('edit(filename)');
clear filename;

filename=fullfile(matlabroot, 'toolbox', 'rtw', 'rtwdemos', 'custom_cpp_src', 'adder_cpp.h');
eval('edit(filename)');
clear filename;
```

2. On the Configuration Parameters dialog box **Code Generation > Custom Code** pane, enter the S-Function source files and include directories.

```
%Open Configuration Parameters dialog box
configSet = getActiveConfigSet(model);
openDialog(configSet);
```

3. On the Configuration Parameters dialog box **Simulation Target > Custom Code** pane, enter the S-Function source files and include directories.

```
%Open Configuration Parameters dialog box
configSet = getActiveConfigSet(model);
openDialog(configSet);
```

Generate C++ Code

On the Configuration Parameters dialog box **Code Generation** pane, set **Language** to C++. To generate code, select **Build**.

```
%Open Configuration Parameters dialog box
configSet = getActiveConfigSet(model);
openDialog(configSet);
clear model;
```

See Also

- “Programming Language Considerations” on page 31-4
- “C++ Language Support Limitations” on page 31-4

Close Model

```
rtwdemoclean;
close_system('rtwdemo_sfcpp',0);
```

S-Functions That Support Expression Folding

Use expression folding to increase the efficiency of code generated by your own inlined S-Function blocks, by calling macros provided in the S-Function API.

The S-Function API lets you specify whether a given S-Function block should nominally accept expressions at a given input port. A block should not always accept expressions. For example, if the address of the signal at the input is used, expressions should not be accepted at that input, because it is not possible to take the address of an expression.

The S-Function API also lets you specify whether an expression can represent the computations associated with a given output port. When you request an expression at a block's input or output port, the Simulink engine determines whether or not it can honor that request, given the block's context. For example, the engine might deny a block's request to output an expression if the destination block does not accept expressions at its input, if the destination block has an update function, or if multiple output destinations exist.

The decision to honor or deny a request to output an expression can also depend on the category of output expression the block uses.

To take advantage of expression folding in your S-functions, you should understand when to request acceptance and generation of expressions for specific blocks. You do not have to understand the algorithm by which the Simulink engine chooses to accept or deny these requests. However, if you want to trace between the model and the generated code, it is helpful to understand some of the more common situations that lead to denial of a request.

Categories of Output Expressions

When you implement a C MEX S-function, you can specify whether the code corresponding to a block's output is to be generated as an expression. If the block generates an expression, you must specify that the expression is *constant*, *trivial*, or *generic*.

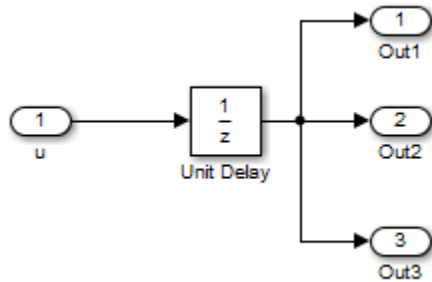
A *constant* output expression is a direct access to one of the block's parameters. For example, the output of a Constant block is defined as a constant expression because the output expression is simply a direct access to the block's `Value` parameter.

A *trivial* output expression is an expression that can be repeated, without a performance penalty, when the output port has multiple output destinations. For example, the output of a Unit Delay block is defined as a trivial expression because the output expression is simply a direct access to the block's state. Because the output expression does not have computations, it can be repeated more than once without degrading the performance of the generated code.

A *generic* output expression is an expression that should be assumed to have a performance penalty if repeated. As such, a generic output expression is not suitable for repeating when the output port has multiple output destinations. For instance, the output of a Sum block is a generic rather than a trivial expression because it is costly to recompute a Sum block output expression as an input to multiple blocks.

Examples of Trivial and Generic Output Expressions

Consider this block diagram. The Delay block has multiple destinations, yet its output is designated as a trivial output expression, so that it can be used more than once without degrading the efficiency of the code.



This code excerpt shows the code generated from the Unit Delay block in this block diagram. The three root outputs are directly assigned from the state of the Unit Delay block, which is stored in a field of the global data structure `rtDWork`. Since the assignment is direct, without expressions, there is no performance penalty associated with using the trivial expression for multiple destinations.

```
void MdlOutputs(int_T tid)
{
    ...
    /* Output: <Root>/Out1 incorporates:
     *   UnitDelay: <Root>/Unit Delay */
    rtY.Out1 = rtDWork.Unit_Delay_DSTATE;

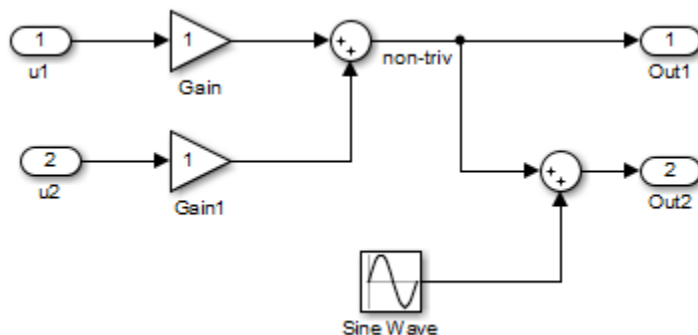
    /* Output: <Root>/Out2 incorporates:
     *   UnitDelay: <Root>/Unit Delay */
    rtY.Out2 = rtDWork.Unit_Delay_DSTATE;

    /* Output: <Root>/Out3 incorporates:
     *   UnitDelay: <Root>/Unit Delay */
    rtY.Out3 = rtDWork.Unit_Delay_DSTATE;

    ...
}
```

The code generated shows how code is generated for Sum blocks with single and multiple destinations.

On the other hand, consider the Sum blocks in this model:



The upper Sum block in the model generates the signal labeled `non_triv`. Computation of this output signal involves two multiplications and an addition. If the Sum block's output were permitted to generate an expression even when the block had multiple destinations, the block's operations would be duplicated in the generated code. In this case, the generated expressions would proliferate to four multiplications and two additions. This would degrade the efficiency of the program. Accordingly the output of the Sum block is not allowed to be an expression because it has multiple destinations.

The Simulink engine does not permit the output of the upper Sum block to be an expression because the signal `non_triv` is routed to two output destinations. Instead, the result of the multiplication and addition operations is stored in a temporary variable (`rtb_non_triv`) that is referenced twice in the statements that follow, as seen in the code excerpt below.

In contrast, the lower Sum block, which has only a single output destination (`Out2`), does generate an expression.

```
void MdlOutputs(int_T tid)
{
    /* local block i/o variables */
    real_T rtb_non_triv;
    real_T rtb_Sine_Wave;

    /* Sum: <Root>/Sum incorporates:
    *   Gain: <Root>/Gain
    *   Inport: <Root>/u1
    *   Gain: <Root>/Gain1
    *   Inport: <Root>/u2
    *
    * Regarding <Root>/Gain:
    *   Gain value: rtP.Gain_Gain
    *
    * Regarding <Root>/Gain1:
    *   Gain value: rtP.Gain1_Gain
    */
    rtb_non_triv = (rtP.Gain_Gain * rtU.u1) + (rtP.Gain1_Gain *
rtU.u2);

    /* Output: <Root>/Out1 */
    rtY.Out1 = rtb_non_triv;

    /* Sin Block: <Root>/Sine Wave */

    rtb_Sine_Wave = rtP.Sine_Wave_Amp *
        sin(rtP.Sine_Wave_Freq * rtmGetT(rtM_model) +
            rtP.Sine_Wave_Phase) + rtP.Sine_Wave_Bias;

    /* Output: <Root>/Out2 incorporates:
    *   Sum: <Root>/Sum1
    */
    rtY.Out2 = (rtb_non_triv + rtb_Sine_Wave);
}
```

Specify the Category of an Output Expression

The S-Function API provides macros that let you declare whether an output of a block should be an expression, and if so, to specify the category of the expression. This table specifies when to declare a block output to be a constant, trivial, or generic output expression.

Types of Output Expressions

Category of Expression	When to Use
Constant	Use only if block output is a direct memory access to a block parameter.
Trivial	Use only if block output is an expression that can appear multiple times in the code without reducing efficiency (for example, a direct memory access to a field of the DWork vector, or a literal).
Generic	Use if output is an expression, but not constant or trivial.

You must declare outputs as expressions in the `mdlSetWorkWidths` function using macros defined in the S-Function API. The macros have the following arguments:

- `SimStruct *S`: pointer to the block's `SimStruct`.
- `int idx`: zero-based index of the output port.
- `bool value`: pass in `TRUE` if the port generates output expressions.

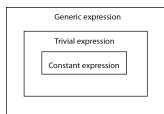
The following macros are available for setting an output to be a constant, trivial, or generic expression:

- `void ssSetOutputPortConstOutputExprInRTW(SimStruct *S, int idx, bool value)`
- `void ssSetOutputPortTrivialOutputExprInRTW(SimStruct *S, int idx, bool value)`
- `void ssSetOutputPortOutputExprInRTW(SimStruct *S, int idx, bool value)`

The following macros are available for querying the status set by prior calls to the macros above:

- `bool ssGetOutputPortConstOutputExprInRTW(SimStruct *S, int idx)`
- `bool ssGetOutputPortTrivialOutputExprInRTW(SimStruct *S, int idx)`
- `bool ssGetOutputPortOutputExprInRTW(SimStruct *S, int idx)`

The set of generic expressions is a superset of the set of trivial expressions, and the set of trivial expressions is a superset of the set of constant expressions.



Therefore, when you query an output that has been set to be a constant expression with `ssGetOutputPortTrivialOutputExprInRTW`, it returns `True`. A constant expression is considered a trivial expression because it is a direct memory access that can be repeated without degrading the efficiency of the generated code.

Similarly, an output that has been configured to be a constant or trivial expression returns `True` when queried for its status as a generic expression.

Acceptance or Denial of Requests for Input Expressions

A block can request that its output be represented in code as an expression. Such a request can be denied if the destination block cannot accept expressions at its input port. Furthermore, conditions independent of the requesting block and its destination blocks can prevent acceptance of expressions.

A block should not be configured to accept expressions at its input port under the following conditions:

- The block must take the address of its input data. It is not possible to take the address of most types of input expressions.
- The code generated for the block references the input more than once (for example, the Abs or Max blocks). This would lead to duplication of a potentially complex expression and a subsequent degradation of code efficiency.

If a block refuses to accept expressions at an input port, then a block that is connected to that input port is not permitted to output a generic or trivial expression.

A request to output a constant expression is not denied, because there is no performance penalty for a constant expression, and the software can take the parameter's address.

S-Function API to Specify Input Expression Acceptance

The S-Function API provides macros that let you:

- Specify whether a block input should accept nonconstant expressions (that is, trivial or generic expressions).
- Query whether a block input accepts nonconstant expressions.

By default, block inputs do not accept nonconstant expressions.

You should call the macros in your `mdlSetWorkWidths` function. The macros have these arguments:

- `SimStruct *S`: pointer to the block's `SimStruct`.
- `int idx`: zero-based index of the input port.
- `bool value`: pass in `TRUE` if the port accepts input expressions; otherwise pass in `FALSE`.

The macro available for specifying whether or not a block input should accept a nonconstant expression is:

```
void ssSetInputPortAcceptExprInRTW(SimStruct *S, int portIdx, bool value)
```

The corresponding macro available for querying the status set by any prior calls to `ssSetInputPortAcceptExprInRTW` is:

```
bool ssGetInputPortAcceptExprInRTW(SimStruct *S, int portIdx)
```

Denial of Block Requests to Output Expressions

Even after a specific block requests that it be allowed to generate an output expression, that request can be denied for generic reasons. These reasons include, but are not limited to:

- The output expression is nontrivial, and the output has multiple destinations.
- The output expression is nonconstant, and the output is connected to at least one destination that does not accept expressions at its input port.
- The output is a test point.
- The output has been assigned an external storage class.
- The output must be stored using global data (for example is an input to a merge block or a block with states).

- The output signal is complex.

You do not need to consider these generic factors when deciding whether or not to utilize expression folding for a particular block. However, these rules can be helpful when you are examining generated code and analyzing cases where the expression folding optimization is suppressed.

Expression Folding in a TLC Block Implementation

To take advantage of expression folding, modify the TLC block implementation of an inlined S-Function such that it informs the Simulink engine whether it generates or accepts expressions at its

- Input ports, as explained in “S-Function API to Specify Input Expression Acceptance” on page 14-68.
- Output ports, as explained in “Categories of Output Expressions” on page 14-64.

This topic discusses required modifications to the TLC implementation.

Expression Folding Compliance

In the `BlockInstanceSetup` function of your S-function, register your block to be compliant with expression folding. Otherwise, expression folding requested or allowed at the block's outputs or inputs will be disabled, and temporary variables will be used.

To register expression folding compliance, call the TLC library function `LibBlockSetIsExpressionCompliant(block)`, which is defined in `matlabroot/rtw/c/tlc/lib/utillib.tlc`. For example:

```
%% Function: BlockInstanceSetup =====
%%
%%function BlockInstanceSetup(block, system) void
%%
%%    %<LibBlockSetIsExpressionCompliant(block)>
%%
%%endfunction
```

You can conditionally disable expression folding at the inputs and outputs of a block by making the call to this function conditionally.

If you override one of the TLC block implementations provided by the code generator with your own implementation, you should not make the preceding call until you have updated your implementation.

Output Expressions

The `BlockOutputSignal` function is used to generate code for a scalar output expression or one element of a nonscalar output expression. If your block outputs an expression, you should add a `BlockOutputSignal` function. The prototype of the `BlockOutputSignal` is

```
%function BlockOutputSignal(block,system,portIdx,ucv,lc,lc,lc,idx,retType) void
```

The arguments to `BlockOutputSignal` are these:

- `block`: the record for the block for which an output expression is being generated
- `system`: the record for the system containing the block
- `portIdx`: zero-based index of the output port for which an expression is being generated
- `ucv`: user control variable defining the output element for which code is being generated

- `lcv`: loop control variable defining the output element for which code is being generated
- `idx`: signal index defining the output element for which code is being generated
- `retType`: character vector defining the type of signal access desired:

"Signal" specifies the contents or address of the output signal

"SignalAddr" specifies the address of the output signal

The `BlockOutputSignal` function returns a character vector for the output signal or address. The character vector should enforce the precedence of the expression by using opening and terminating parentheses, unless the expression consists of a function call. The address of an expression can only be returned for a constant expression; it is the address of the parameter whose memory is being accessed. The code implementing the `BlockOutputSignal` function for the Constant block is:

```
%% Function: BlockOutputSignal =====
%% Abstract:
%%     Return the reference to the parameter. This function *may*
%%     be used by Simulink when optimizing the Block IO data structure.
%%
%function BlockOutputSignal(block,system,portIdx,ucv,lcv,idx,retType) void
    %switch retType
        %case "Signal"
            %return LibBlockParameter(Value,ucv,lcv,idx)
        %case "SignalAddr"
            %return LibBlockParameterAddr(Value,ucv,lcv,idx)
        %default
            %assign errTxt = "Unsupported return type: %<retType>"
            %<LibBlockReportError(block,errTxt)>
    %endswitch
%endfunction
```

The code implementing the `BlockOutputSignal` function for the Relational Operator block is:

```
%% Function: BlockOutputSignal =====
%% Abstract:
%%     Return an output expression. This function *may*
%%     be used by Simulink when optimizing the Block IO data structure.
%%
%function BlockOutputSignal(block,system,portIdx,ucv,lcv,idx,retType) void
%switch retType
%case "Signal"
%assign logicOperator = ParamSettings.Operator
    %if ISEQUAL(logicOperator, "~=")
        %assign op = "!="
    elseif ISEQUAL(logicOperator, "==") %assign op = "=="
        %else
%assign op = logicOperator
    %endif
    %assign u0 = LibBlockInputSignal(0, ucv, lcv, idx)
    %assign u1 = LibBlockInputSignal(1, ucv, lcv, idx)
    %return "(%<u0> %<op> %<u1>)"
%default
    %assign errTxt = "Unsupported return type: %<retType>"
    %<LibBlockReportError(block,errTxt)>
%endswitch
%endfunction
```

Expression Folding for Blocks with Multiple Outputs

When a block has a single output, the `Outputs` function in the block's TLC file is called only if the output port is not an expression. Otherwise, the `BlockOutputSignal` function is called.

If a block has multiple outputs, the `Outputs` function is called if any output port is not an expression. The `Outputs` function should guard against generating code for output ports that are expressions.

This is achieved by guarding sections of code corresponding to individual output ports with calls to `LibBlockOutputSignalIsExpr()`.

For example, consider an S-Function with two inputs and two outputs, where

- The first output, `y0`, is equal to two times the first input.
- The second output, `y1`, is equal to four times the second input.

The `Outputs` and `BlockOutputSignal` functions for the S-function are shown in this code excerpt:

```
%% Function: BlockOutputSignal =====
%% Abstract:
%%     Return an output expression. This function *may*
%%     be used by Simulink when optimizing the Block IO data structure.
%%
%function BlockOutputSignal(block,system,portIdx,ucv,lcx,idx,retType) void
%switch retType
%case "Signal"
    %assign u = LibBlockInputSignal(portIdx, ucv, lcv, idx)
    %case "Signal"
    %if portIdx == 0
        %return "(2 * %<u>)"
    %elseif portIdx == 1
        %return "(4 * %<u>)"
    %endif
%default
%assign errTxt = "Unsupported return type: %<retType>"
    %<LibBlockReportError(block,errTxt)>
%endswitch
%endfunction
%%
%% Function: Outputs =====
%% Abstract:
%%     Compute output signals of block
%%
%function Outputs(block,system) Output
%assign rollVars = ["U", "Y"]
%roll sigIdx = RollRegions, lcv = RollThreshold, block, "Roller", rollVars
%assign u0 = LibBlockInputSignal(0, "", lcv, sigIdx)
%assign u1 = LibBlockInputSignal(1, "", lcv, sigIdx)
%assign y0 = LibBlockOutputSignal(0, "", lcv, sigIdx)
%assign y1 = LibBlockOutputSignal(1, "", lcv, sigIdx)
%if !LibBlockOutputSignalIsExpr(0)
    %<y0> = 2 * %<u0>;
%endif
%if !LibBlockOutputSignalIsExpr(1)
    %<y1> = 4 * %<u1>;
%endif
%endroll
%endfunction
```

Comments for Blocks That Are Expression-Folding-Compliant

In the past, blocks preceded their outputs code with comments of the form

```
/* %<Type> Block: %<Name> */
```

When a block is expression-folding-compliant, the initial line shown above is generated automatically. Do not include the comment as part of the block's TLC implementation. Register additional information by using the `LibCacheBlockComment` function.

The `LibCacheBlockComment` function takes a character vector as an input, defining the body of the comment, except for the opening header, the final newline of a single or multiline comment, and the closing trailer.

The following TLC code illustrates registering a block comment. Note the use of the function `LibBlockParameterForComment`, which returns a character vector, suitable for a block comment, specifying the value of the block parameter.

```
%openfile commentBuf
$c(*) Gain value: %<LibBlockParameterForComment(Gain)>
%closefile commentBuf
%<LibCacheBlockComment(block, commentBuf)>
```

See Also

Related Examples

- “S-Functions That Specify Port Scope and Reusability” on page 14-73
- “S-Functions That Specify Sample Time Inheritance Rules” on page 14-77
- “S-Functions for Code Reuse” on page 14-81

S-Functions That Specify Port Scope and Reusability

You can use the following `SimStruct` macros in the `mdlInitializeSizes` method to specify the scope and reusability of the memory used for your S-function's input and output ports:

- `ssSetInputPortOptimOpts`: Specify the scope and reusability of the memory allocated to an S-function input port.
- `ssSetOutputPortOptimOpts`: Specify the scope and reusability of the memory allocated to an S-function output port.
- `ssSetInputPortOverWritable`: Specify whether one of your S-function's input ports can be overwritten by one of its output ports.
- `ssSetOutputPortOverwritesInputPort`: Specify whether an output port can share its memory buffer with an input port.

You declare an input or output as local or global, and indicate its reusability, by passing one of the following four options to the `ssSetInputPortOptimOpts` and `ssSetOutputPortOptimOpts` macros:

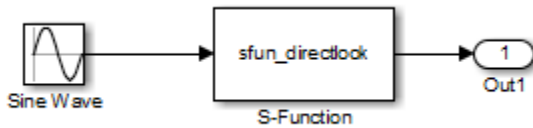
- `SS_NOT_REUSABLE_AND_GLOBAL`: Indicates that the input and output ports are stored in separate memory locations in the global block input and output structure.
- `SS_NOT_REUSABLE_AND_LOCAL`: Indicates that the code generator can declare individual local variables for the input and output ports.
- `SS_REUSABLE_AND_LOCAL`: Indicates that the code generator can reuse a single local variable for these input and output ports.
- `SS_REUSABLE_AND_GLOBAL`: Indicates that these input and output ports are stored in a single element in the global block input and output structure.

Note Marking an input or output port as a local variable does not imply that the code generator uses a local variable in the generated code. If your S-function accesses the inputs and outputs only in its `mdlOutputs` routine, the code generator declares the inputs and outputs as local variables. However, if the inputs and outputs are used elsewhere in the S-function, the code generator includes them in the global block input and output structure.

The reusability setting indicates if the memory associated with an input or output port can be overwritten. To reuse input and output port memory:

- 1 Indicate the ports are reusable using either the `SS_REUSABLE_AND_LOCAL` or `SS_REUSABLE_AND_GLOBAL` option in the `ssSetInputPortOptimOpts` and `ssSetOutputPortOptimOpts` macros.
- 2 Indicate the input port memory is overwritable using `ssSetInputPortOverWritable`.
- 3 If your S-function has multiple input and output ports, use `ssSetOutputPortOverwritesInputPort` to indicate which output and input ports share memory.

The following example shows how different scope and reusability settings affect the generated code. The following model contains an S-function block pointing to the C MEX S-function `matlabroot/toolbox/simulink/simdemos/simfeatures/src/sfun_directlook.c`, which models a direct 1-D lookup table.



The S-function's `mdlInitializeSizes` method declares the input port as reusable, local, and overwritable and the output port as reusable and local, as follows:

```
static void mdlInitializeSizes(SimStruct *S)
{
/* snip */
  ssSetInputPortOptimOpts(S, 0, SS_REUSABLE_AND_LOCAL);
  ssSetInputPortOverWritable(S, 0, TRUE);

/* snip */
  ssSetOutputPortOptimOpts(S, 0, SS_REUSABLE_AND_LOCAL);

/* snip */
}

```

The generated code for this model stores the input and output signals in a single local variable `rtb_SFunction`, as shown in the following output function:

```
static void sl_directlook_output(int_T tid)
{
/* local block i/o variables */
  real_T rtb_SFunction[2];

/* Sin: '<Root>/Sine Wave' */
  rtb_SFunction[0] = sin(((real_T)sl_directlook_DWork.counter[0] +
  sl_directlook_P.SineWave_Offset) * 2.0 * 3.1415926535897931E+000 /
  sl_directlook_P.SineWave_NumSamp) * sl_directlook_P.SineWave_Amp[0] +
  sl_directlook_P.SineWave_Bias;
  rtb_SFunction[1] = sin(((real_T)sl_directlook_DWork.counter[1] +
  sl_directlook_P.SineWave_Offset) * 2.0 * 3.1415926535897931E+000 /
  sl_directlook_P.SineWave_NumSamp) * sl_directlook_P.SineWave_Amp[1] +
  sl_directlook_P.SineWave_Bias;

/* S-Function Block: '<Root>/S-Function' */
{
  const real_T *xData = &sl_directlook_P.SFunction_XData[0];
  const real_T *yData = &sl_directlook_P.SFunction_YData [0];
  real_T spacing = xData[1] - xData[0];
  if (rtb_SFunction[0] <= xData[0] ) {
    rtb_SFunction[0] = yData[0];
  } else if (rtb_SFunction[0] >= yData[20] ) {
    rtb_SFunction[0] = yData[20];
  } else {
    int_T idx = (int_T)( ( rtb_SFunction[0] - xData[0] ) / spacing );
    rtb_SFunction[0] = yData[idx];
  }

  if (rtb_SFunction[1] <= xData[0] ) {
    rtb_SFunction[1] = yData[0];
  } else if (rtb_SFunction[1] >= yData[20] ) {
    rtb_SFunction[1] = yData[20];
  } else {
    int_T idx = (int_T)( ( rtb_SFunction[1] - xData[0] ) / spacing );
    rtb_SFunction[1] = yData[idx];
  }
}

/* Outport: '<Root>/Out1' */
  sl_directlook_Y.Out1[0] = rtb_SFunction[0];
  sl_directlook_Y.Out1[1] = rtb_SFunction[1];
  UNUSED_PARAMETER(tid);
}

```

This table shows variations of the code generated for this model when using the generic real-time target (GRT). Each row explains a different setting for the scope and reusability of the S-function's input and output ports.

Scope and reusability	S-function mdlInitializeSizes code	Generated code
Inputs: Local, reusable, overwriteable Outputs: Local, reusable	<pre>ssSetInputPortOptimOpts(S, 0, SS_REUSABLE_AND_LOCAL); ssSetInputPortOverWritable(S, 0, TRUE); ssSetOutputPortOptimOpts(S, 0, SS_REUSABLE_AND_LOCAL);</pre>	<p>The <i>model.c</i> file declares a local variable in the output function.</p> <pre>/* local block i/o variables */ real_T rtb_SFunction[2];</pre>
Inputs: Global, reusable, overwriteable Outputs: Global, reusable	<pre>ssSetInputPortOptimOpts(S, 0, SS_REUSABLE_AND_GLOBAL); ssSetInputPortOverWritable(S, 0, TRUE); ssSetOutputPortOptimOpts(S, 0, SS_REUSABLE_AND_GLOBAL);</pre>	<p>The <i>model.h</i> file defines a block signals structure with a single element to store the S-function's input and output.</p> <pre>/* Block signals (auto storage) */ typedef struct { real_T SFunction[2]; } BlockIO_sl_directlook;</pre> <p>The <i>model.c</i> file uses this element of the structure in calculations of the S-function's input and output signals.</p> <pre>/* Sin: '<Root>/Sine Wave' */ sl_directlook_B.SFunction[0] = sin ... /* snip */ /*S-Function Block:<Root>/S-Function*/ { const real_T *xData = &sl_directlook_P.SFunction_XData[0]</pre>
Inputs: Local, not reusable Outputs: Local, not reusable	<pre>ssSetInputPortOptimOpts(S, 0, SS_NOT_REUSABLE_AND_LOCAL); ssSetInputPortOverWritable(S, 0, FALSE); ssSetOutputPortOptimOpts(S, 0, SS_NOT_REUSABLE_AND_LOCAL);</pre>	<p>The <i>model.c</i> file declares local variables for the S-function's input and output in the output function</p> <pre>/* local block i/o variables */ real_T rtb_SineWave[2]; real_T rtb_SFunction[2];</pre>
Inputs: Global, not reusable Outputs: Global, not reusable	<pre>ssSetInputPortOptimOpts(S, 0, SS_NOT_REUSABLE_AND_GLOBAL); ssSetInputPortOverWritable(S, 0, FALSE); ssSetOutputPortOptimOpts(S, 0, SS_NOT_REUSABLE_AND_GLOBAL);</pre>	<p>The <i>model.h</i> file defines a block signal structure with individual elements to store the S-function's input and output.</p> <pre>/* Block signals (auto storage) */ typedef struct { real_T SineWave[2]; real_T SFunction[2]; } BlockIO_sl_directlook;</pre> <p>The <i>model.c</i> file uses the different elements in this structure when calculating the S-function's input and output.</p> <pre>/* Sin: '<Root>/Sine Wave' */ sl_directlook_B.SineWave[0] = sin ... /* snip */ /*S-Function Block:<Root>/S-Function*/ { const real_T *xData = &sl_directlook_P.SFunction_XData[0]</pre>

To check if the S-function input or output buffer reuse takes place, you can use `LibBlockInputSignalBufferDstPort` function.

See Also

Related Examples

- “S-Functions That Support Expression Folding” on page 14-64
- “S-Functions That Specify Sample Time Inheritance Rules” on page 14-77
- “S-Functions for Code Reuse” on page 14-81

S-Functions That Specify Sample Time Inheritance Rules

For the Simulink engine to determine whether a model can inherit a sample time from a parent model, the S-functions in the model need to specify how they use sample times. You can specify this information by calling the macro `ssSetModelReferenceSampleTimeInheritanceRule` from `mdlInitializeSizes` or `mdlSetWorkWidths`. To use this macro:

- 1 Check whether the S-function calls any of these macros:
 - `ssGetSampleTime`
 - `ssGetInputPortSampleTime`
 - `ssGetOutputPortSampleTime`
 - `ssGetInputPortOffsetTime`
 - `ssGetOutputPortOffsetTime`
 - `ssGetInputPortSampleTimeIndex`
 - `ssGetOutputPortSampleTimeIndex`
- 2 Check for these in your S-function TLC code:
 - `LibBlockSampleTime`
 - `CompiledModel.SampleTime`
 - `LibBlockInputSignalSampleTime`
 - `LibBlockInputSignalOffsetTime`
 - `LibBlockOutputSignalSampleTime`
 - `LibBlockOutputSignalOffsetTime`
- 3 Depending on your search results, use `ssSetModelReferenceSampleTimeInheritanceRule` as indicated in this table.

If...	Use...	Example
None of the macros or functions are present, the S-function does not preclude the model from inheriting a sample time.	<code>ssSetModelReferenceSampleTimeInheritanceRule</code> (S, USE_DEFAULT_FOR_DISCRETE_INHERITANCE)	

If...	Use...	Example
<p>Any of the macros or functions are used for:</p> <ul style="list-style-type: none"> • Throwing errors if sample time is inherited, continuous, or constant • Checking <code>ssIsSampleHit</code> • Checking whether sample time is inherited in either <code>mdlSetInputPortSampleTime</code> or <code>mdlSetOutputPortSampleTime</code> before setting 	<p><code>ssSetModelReferenceSampleTimeInheritance(S, USE_DEFAULT_FOR_DISCRETE_INHERITANCE)</code></p>	<p>Consider this <code>mdlOutputs</code> code:</p> <pre>static void mdlOutputs(SimStruct *S, int tid) { const real_T *u = (const real_T*) ssGetInputPortSignal(S,0); real_T *y = ssGetOutputPortSignal(S,0); y[0] = ssGetSampleTime(S,tid) * u[0]; }</pre>

If...	Use...	Example
<p>The S-function uses its sample time for computing parameters, outputs, and so on.</p>	<p><code>ssSetModelReferenceSampleTimeInheritanceRule(S, DISALLOW_SAMPLE_TIME_INHERITANCE)</code></p>	<p>Consider the <code>mdlOutputs</code> code from the S-function example <code>sfun_multirate.c</code>:</p> <pre> static void mdlOutputs(SimStruct *S, int tid) { InputRealPtrsType enablePtrs; int *enabled = ssGetInputPortSampleTime(S, tid); int enableTid = ssGetInputPortSampleTime(S, tid); int signalTid = ssGetInputPortSampleTime(S, tid); real_T enableTs = ssGetInputPortSampleTime(S, tid); real_T enableTsOffset = ssGetInputPortSampleTime(S, tid); if (enableTs == CONTINUOUS_SAMPLE_TIME) if (ssIsMajorTimeStep(S) && ssIsSampleHit(S, enableTid, tid) ssIsSpecialSampleHit(S, enableTid, tid)) enablePtrs = ssGetInputPortRealPtrs(S, tid); *enabled = (*enablePtrs); } else { int enableTid = ssGetInputPortSampleTime(S, tid); if (ssIsSampleHit(S, enableTid, tid) (enableTid == signalTid && ssIsSpecialSampleHit(S, signalTid, tid))) enablePtrs = ssGetInputPortRealPtrs(S, tid); *enabled = (*enablePtrs); } } if (ssIsSampleHit(S, signalTid, tid) ssIsSpecialSampleHit(S, signalTid, tid)) InputRealPtrsType uPtrs = ssGetInputPortRealPtrs(S, tid); real_T signal = *uPtrs; int i; for (i = 0; i < NOUTPUTS; i++) { int outTid = ssGetOutputPortSampleTime(S, i); if (outTid == signalTid ssIsSpecialSampleHit(S, signalTid, tid)) real_T *y = ssGetOutputPortRealPtrs(S, i); *y = signal; } } } /* end mdlOutputs */ </pre>

Note If an S-function does not set the `ssSetModelReferenceSampleTimeInheritanceRule` macro, by default the Simulink engine assumes that the S-function does not preclude the model containing that S-function from inheriting a sample time. However, the engine issues a warning indicating that the model includes S-functions for which this macro is not set.

You can use settings in the Configuration Parameters on the **Diagnostics > Sample Time** pane to control how the Simulink engine responds when it encounters S-functions that have unspecified sample time inheritance rules. Toggle the **Unspecified inheritability of sample time** diagnostic to none, warning, or error. The default is warning.

For information about Model block sample time inheritance, see “Referenced Model Sample Times”.

See Also

Related Examples

- “Specify S-Function Sample Times”
- “S-Functions That Support Expression Folding” on page 14-64
- “S-Functions That Specify Port Scope and Reusability” on page 14-73
- “S-Functions for Code Reuse” on page 14-81

S-Functions for Code Reuse

You can reuse the generated code for identical subsystems that occur in multiple instances within a model and across referenced models. For more information about code generation of subsystems for code reuse, see “Generate Subsystem Code as Separate Function and Files” on page 7-7.

S-Function Code Reuse Requirements

To use S-function for code reuse for a subsystem, the S-function must meet these requirements:

- The S-function must be inlined.
- Code generated from the S-function must not use static variables.
- The S-function must initialize its pointer work vector in only `mdlStart` and not before.
- The S-function must not be a sink that logs data to the workspace.
- The S-function must register its parameters as run-time parameters in `mdlSetWorkWidths`. For this purpose, the S-function must not use `ssWriteRTWParameters` in its `mdlRTW` function.
- The S-function must not be a device driver.

Additionally, your S-function must set the `SS_OPTION_WORKS_WITH_CODE_REUSE` flag in the `ssSetOptions` function. This flag indicates that your S-function meets the requirements for subsystem code reuse. If the flag is set and your S-function does not meet the requirements, the code generator does not generate a reusable function and you see a warning.

S-Functions Reused Across Models

You can place S-functions inside reusable library subsystems. In the S-function file, set both of these flags:

- `SS_OPTION_WORKS_WITH_CODE_REUSE` flag in the `ssSetOptions` function
- `ssSetSupportedForCodeReuseAcrossModels` set to 1 or `true`

If you use the `legacy_code` tool to generate your S-function, the S-function option `supportCodeReuseAcrossModels` specifies code reuse across models for the subsystem that contains the S-function.

Configure the S-function for code reuse across models only if the S-function does not access any model-specific data structures. This configuration is necessary because the generated code in `_sharedutils` folder is compiled separately from the generated code in `model_ert_rtw` folder. The reusable library subsystem code generated in the `_sharedutils` folder does not have access to types and macros that are declared in `model.h`. The reusable library subsystem code must be independent. For more information, see “Generate Reusable Code from Library Subsystems Shared Across Models” on page 10-36.

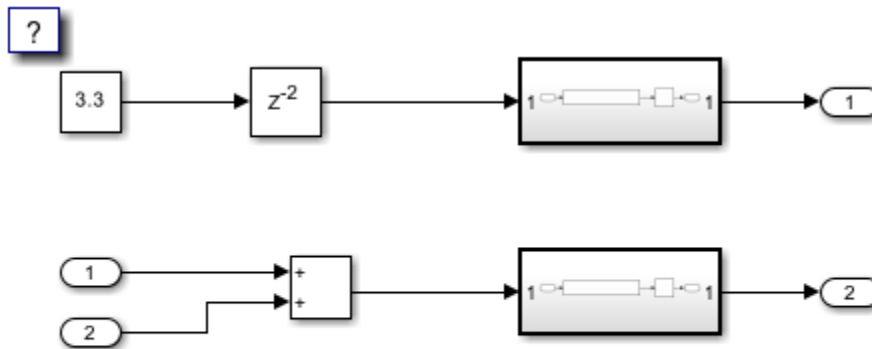
If your S-function uses custom functions defined in your external header files, add the `LibAddtoSystemCustomIncludes(system, incFileName)` function in your S-function's TLC file. For example, when you add `LibAddtoSystemCustomIncludes("company_math.h")` in your `.tlc` file, the reusable library subsystem code in the `_sharedutils` folder includes the `#company_math.h` header file.

1. Create a new MATLAB® working folder. In the MATLAB Command Window, copy these external code files to your current MATLAB working folder:

```
currentDir = pwd;
[~,cgDir] = rtwdemodir();
copyfile(fullfile(matlabroot,'toolbox','rtw','rtwdemos','doubleIt.c'));
copyfile(fullfile(matlabroot,'toolbox','rtw','rtwdemos','doubleIt.h'));
```

2. Open example model `rtwdemo_sfcn_rls`. This example model uses two instances of the reusable library subsystem `rtwdemo_sfcn_rls_lib`. The S-function `sfun_mySrc` inside the reusable library subsystem uses external files `doubleIt.c` and `doubleIt.h`.

```
rtwdemo_sfcn_rls;
```



Copyright 2020 The MathWorks, Inc.

3. Generate the S-function code by using legacy code tool. Use these commands in your MATLAB Command window:

```
def = legacy_code('initialize');
def.SFunctionName = 'sfun_mySrc';
def.SourceFiles = {'doubleIt.c'};
def.HeaderFiles = {'doubleIt.h'};
def.OutputFcnSpec = 'double y1 = doubleIt(double u1)';
```

4. Specify the S-function for code reuse:

```
def.Options.supportCodeReuseAcrossModels = true;
```

5. Generate the S-function code TLC block file. Then compile the S-function.

```
legacy_code('sfcn_tlc_generate', def);
legacy_code('sfcn_cmex_generate', def);
legacy_code('compile', def);
```

```
### Start Compiling sfun_mySrc
mex('-IC:\TEMP\Bdoc23a_2213998_3568\ib570499\6\tp03c82afd_8d2d_45df_8671_53b0234eef4\coder\
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('sfun_mySrc.c', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570499\6\tp03c82afd_8d2d_45df_8671_53b
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling sfun_mySrc
### Exit
```

6. The code generator creates the S-function file `sfun_mySrc.c` and its TLC file `sfun_mySrc.tlc` in your MATLAB working folder. In the `sfun_mySrc.c` file, the code generator adds these specifications:

```
file = fullfile('sfun_mySrc.c');
rtwdemodbtype(file, '* All options have the form SS_OPTION_<name>', ...
    'ssSetSupportedForCodeReuseAcrossModels(S, 1);', 1, 1);

    * All options have the form SS_OPTION_<name> and are documented in
    * matlabroot/simulink/include/simstruc.h. The options should be
    * bitwise or'd together as in
    * ssSetOptions(S, (SS_OPTION_name1 | SS_OPTION_name2))
    */
ssSetOptions(S,
    SS_OPTION_USE_TLC_WITH_ACCELERATOR |
    SS_OPTION_CAN_BE_CALLED_CONDITIONALLY |
    SS_OPTION_EXCEPTION_FREE_CODE |
    SS_OPTION_WORKS_WITH_CODE_REUSE |
    SS_OPTION_SFUNCTION_INLINED_FOR_RTW |
    SS_OPTION_DISALLOW_CONSTANT_SAMPLE_TIME
);
ssSetSupportedForCodeReuseAcrossModels(S, 1);
```

In the `sfun_mySrc.c`, the `SS_OPTION_WORKS_WITH_CODE_REUSE` flag is set in the `ssSetOptions` function and `ssSetSupportedForCodeReuseAcrossModels` is set to true.

7. On the Simulink® toolstrip, open the **Embedded Coder** or **Simulink Coder** app.

8. Generate code for the model. On the **C Code** tab, click **Build**.

```
evalc('slbuild(''rtwdemo_sfnc_rls'')');
```

The code generator produces a reusable function that can be reused across models in the `_sharedutils` folder. Models in a model reference hierarchy share this `_sharedutils` folder to reuse code. For this example, the reusable function is generated in the `slprj/grt/_sharedutils/sfunc.c` file:

```
file = fullfile('slprj', 'grt', '_sharedutils', 'sfunc.c');
rtwdemodbtype(file, 'Output and update for atomic system', ...
    'rtb_sfnc_mySrc', 1, 1);

    * Output and update for atomic system:
    *   'ReusableSubsystem' ('rtwdemo_sfnc_rls_lib:60')
    *   'ReusableSubsystem' ('rtwdemo_sfnc_rls_lib:60')
    */
void sfnc(real_T rtu_In1, B_sfnc_T *localB, DW_sfnc_T *localDW)
{
    /* local block i/o variables */
    real_T rtb_sfnc_mySrc;
```

S-Functions Reused Within a Model

By default, S-functions inside a subsystem are not configured for code reuse within a model. You can specify the S-functions inside a subsystem for code reuse within a model. In the S-function file, set the `SS_OPTION_WORKS_WITH_CODE_REUSE` flag in the `ssSetOptions` function. This flag indicates that your S-function meets the requirements for subsystem code reuse and is configured for code

reuse. If the flag is set and your S-function does not meet the requirements, the code generator does not generate a reusable function and you see a warning.

1. Create a new MATLAB working folder and copy these external code files to your current MATLAB working folder:

```
currentDir = pwd;
[~,cgDir] = rtwdemodir();
copyfile(fullfile(matlabroot, 'toolbox', 'rtw', 'rtwdemos', 'doubleIt.c'));
copyfile(fullfile(matlabroot, 'toolbox', 'rtw', 'rtwdemos', 'doubleIt.h'));
```

2. For the example model `rtwdemo_sfcn_rls`, generate the S-function code by using legacy code tool. Do not include the S-function option `supportCodeReuseAcrossModels`. Use these commands in your MATLAB Command window:

```
def = legacy_code('initialize');
def.SFunctionName = 'sfun_mySrc';
def.SourceFiles = {'doubleIt.c'};
def.HeaderFiles = {'doubleIt.h'};
def.OutputFcnSpec = 'double y1 = doubleIt(double u1)';
```

3. Generate the S-function code TLC block file. Then compile the S-function.

```
legacy_code('sfcn_tlc_generate', def);
legacy_code('sfcn_cmex_generate', def);
legacy_code('compile', def);
```

```
### Start Compiling sfun_mySrc
mex('-IC:\TEMP\Bdoc23a_2213998_3568\ib570499\6\tpe8dae8e1_b5b0_4655_9e33_b26f18c1e739\coderd
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
mex('sfun_mySrc.c', '-IC:\TEMP\Bdoc23a_2213998_3568\ib570499\6\tpe8dae8e1_b5b0_4655_9e33_b26
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
### Finish Compiling sfun_mySrc
### Exit
```

4. The code generator creates the S-function file `sfun_mySrc.c` and its TLC file `sfun_mySrc.tlc` in your MATLAB working folder.

```
file = fullfile('sfun_mySrc.c');
rtwdemodbtype(file, '* All options have the form SS_OPTION_<name>', ...
    'SS_OPTION_DISALLOW_CONSTANT_SAMPLE_TIME', 1, 1);

* All options have the form SS_OPTION_<name> and are documented in
* matlabroot/simulink/include/simstruc.h. The options should be
* bitwise or'd together as in
* ssSetOptions(S, (SS_OPTION_name1 | SS_OPTION_name2))
*/
ssSetOptions(S,
    SS_OPTION_USE_TLC_WITH_ACCELERATOR |
    SS_OPTION_CAN_BE_CALLED_CONDITIONALLY |
    SS_OPTION_EXCEPTION_FREE_CODE |
    SS_OPTION_WORKS_WITH_CODE_REUSE |
    SS_OPTION_SFUNCTION_INLINED_FOR_RTW |
    SS_OPTION_DISALLOW_CONSTANT_SAMPLE_TIME
```

In the `sfun_mySrc.c` file, the code generator adds `SS_OPTION_WORKS_WITH_CODE_REUSE` flag in the `ssSetOptions` function by default but does not include the `ssSetSupportedForCodeReuseAcrossModels` specification that prevents the S-function from being reused across models.

5. On the Simulink toolstrip, open the **Embedded Coder** or **Simulink Coder** app.

6. Generate code for the model. On the **C Code** tab, click **Build**.

```
evalc('slbuild(''rtwdemo_sfcn_rls'')');
```

The code generator produces a reusable function for the subsystem in the `rtwdemo_sfcn_rls.c` file, which is outside the `_sharedutils` folder. This code can be reused within the model but not across the model reference hierarchy:

```
file = fullfile('rtwdemo_sfcn_rls_grt_rtw','rtwdemo_sfcn_rls.c');
rtwdemodbtype(file,'void sfunc', ...
    'localDW->UnitDelay_DSTATE = rtb_sfun_mySrc;',1,1);

void sfunc_Init(DW_sfunc_T *localDW)
{
    /* InitializeConditions for UnitDelay: '<S1>/Unit Delay' */
    localDW->UnitDelay_DSTATE = 0.0;
}

/*
 * Output and update for atomic system:
 *   '<Root>/ReusableSubsystem1'
 *   '<Root>/ReusableSubsystem2'
 */
void sfunc(real_T rtu_In1, B_sfunc_T *localB, DW_sfunc_T *localDW)
{
    /* local block i/o variables */
    real_T rtb_sfun_mySrc;

    /* UnitDelay: '<S1>/Unit Delay' */
    localB->UnitDelay = localDW->UnitDelay_DSTATE;

    /* S-Function (sfun_mySrc): '<S1>/sfun_mySrc' */
    rtb_sfun_mySrc = doubleIt(rtu_In1);

    /* Update for UnitDelay: '<S1>/Unit Delay' */
    localDW->UnitDelay_DSTATE = rtb_sfun_mySrc;
}
```

See Also

Related Examples

- “S-Functions That Support Expression Folding” on page 14-64
- “S-Functions That Specify Port Scope and Reusability” on page 14-73
- “S-Functions That Specify Sample Time Inheritance Rules” on page 14-77

S-Functions for Multirate Multitasking Environments

In this section...

“About S-Functions for Multirate Multitasking Environments” on page 14-86

“Rate Grouping Support in S-Functions” on page 14-86

“Create Multitasking, Multirate, Port-Based Sample Time S-Functions” on page 14-86

About S-Functions for Multirate Multitasking Environments

S-functions can be used in models with multiple sample rates and deployed in multitasking target environments. Likewise, S-functions themselves can have multiple rates at which they operate. The code generator produces code for multirate multitasking models using an approach called *rate grouping*. In code generated for ERT-based targets, rate grouping generates separate *model_step* functions for the base rate task and each subrate task in the model. Although rate grouping is a code generation feature found in ERT targets only, your S-functions can use it in other contexts when you code them as explained below.

Rate Grouping Support in S-Functions

To take advantage of rate grouping, you must inline your multirate S-functions if you have not done so. You need to follow certain Target Language Compiler protocols to exploit rate grouping. Coding TLC to exploit rate grouping does not prevent your inlined S-functions from functioning properly in GRT. Likewise, your inlined S-functions will still generate valid ERT code even if you do not make them rate-grouping-compliant. If you do so, however, they will generate more efficient code for multirate models.

For instructions and examples of Target Language Compiler code illustrating how to create and upgrade S-functions to generate rate-grouping-compliant code, see “Rate Grouping Compliance and Compatibility Issues” (Embedded Coder).

For each multirate S-function that is not rate grouping-compliant, the code generator issues the following warning when you build:

```
Warning: Simulink Coder: Code of output function for multirate block
'<Root>/S-Function' is guarded by sample hit checks rather than being rate
grouped. This will generate the same code for all rates used by the block,
possibly generating dead code. To avoid dead code, you must update the TLC
file for the block.
```

You will also find a comment such as the following in code generated for each noncompliant S-function:

```
/* Because the output function of multirate block
<Root>/S-Function is not rate grouped,
the following code might contain unreachable blocks of code.
To avoid this, you must update your block TLC file. */
```

The words “update function” are substituted for “output function” in these warnings.

Create Multitasking, Multirate, Port-Based Sample Time S-Functions

The following instructions show how to support both data determinism and data integrity in multirate S-functions. They do not cover cases where there is no determinism nor integrity. Support for frame-based processing does not affect the requirements.

Note The slow rates must be multiples of the fastest rate. The instructions do not apply when two rates being interfaced are not multiples or when the rates are not periodic.

Rules for Properly Handling Fast-to-Slow Transitions

The rules that multirate S-functions should observe for inputs are

- The input should only be read at the rate that is associated with the input port sample time.
- Generally, the input data is written to DWork, and the DWork can then be accessed at the slower (downstream) rate.

The input can be read at every sample hit of the input rate and written into DWork memory, but this DWork memory cannot then be directly accessed by the slower rate. DWork memory that will be read by the slow rate must only be written by the fast rate when there is a *special sample hit*. A special sample hit occurs when both this input port rate and rate to which it is interfacing have a hit. Depending on their requirements and design, algorithms can process the data in several locations.

The rules that multirate S-functions should observe for outputs are

- The output should not be written by a rate other than the rate assigned to the output port, except in the optimized case described below.
- The output should always be written when the sample rate of the output port has a hit.

If these conditions are met, the S-Function block can specify that the input port and output port can both be made local and reusable.

You can include an optimization when little or no processing needs to be done on the data. In such cases, the input rate code can directly write to the output (instead of by using DWork) when there is a special sample hit. If you do this, however, you must declare the output port to be *global* and *not reusable*. This optimization results in one less memcpy but does introduce nonuniform processing requirements on the faster rate.

Whether you use this optimization or not, the most recent input data, as seen by the slower rate, is the value when both the faster and slower rate had their hits (and possible earlier input data as well, depending on the algorithm). Subsequent steps by the faster rate and the associated input data updates are not seen by the slower rate until the next hit for the slow rate occurs.

Pseudocode Examples of Fast-to-Slow Rate Transition

The pseudocode below abstracts how you should write your C MEX code to handle fast-to-slow transitions, illustrating with an input rate of 0.1 second driving an output rate of one second. A similar approach can be taken when inlining the code. The block has following characteristics:

- File: `sfun_multirate_zoh.c`, Equation: $y = u(t_{\text{slow}})$
- Input: local and reusable
- Output: local and reusable
- DirectFeedthrough: yes

```
OutputFcn
if (ssIsSampleHit("1")) {
    if (ssIsSpecialSampleHit("1")) {
        DWork = u;
    }
}
```

```

if (ssIsSampleHit("1")) {
    y = DWork;
}

```

An alternative, slightly optimized approach for simple algorithms:

- Input: local and reusable
- Output: global and not reusable because it needs to persist between special sample hits
- DirectFeedthrough: yes

```

OutputFcn
if (ssIsSampleHit(".1")) {
    if (ssIsSpecialSampleHit("1")) {
        y = u;
    }
}

```

Example adding a simple algorithm:

- File: `sfun_multirate_avg.c`; Equation: $y = \text{average}(u)$
- Input: local and reusable
- Output: local and reusable
- DirectFeedthrough: yes

(Assume `DWork[0:10]` and `DWork[mycounter]` are initialized to zero)

```

OutputFcn
if (ssIsSampleHit(".1")) {
    /* In general, processing on 'u' could be done here,
       it runs on every hit of the fast rate. */
    DWork[DWork[mycounter]++] = u;
    if (ssIsSpecialSampleHit("1")) {
        /* In general, processing on DWork[0:10] can be done
           here, but it does cause the faster rate to have
           nonuniform processing requirements (every 10th hit,
           more code needs to be run).*/
        DWork[10] = sum(DWork[0:9])/10;
        DWork[mycounter] = 0;
    }
}
if (ssIsSampleHit("1")) {
    /* Processing on DWork[10] can be done here before
       outputting. This code runs on every hit of the
       slower task. */
    y = DWork[10];
}

```

Rules for Properly Handling Slow-to-Fast Transitions

When output rates are faster than input rates, input should only be read at the rate that is associated with the input port sample time, observing the following rules:

- Always read input from the update function.
- Use no special sample hit checks when reading input.
- Write the input to a `DWork`.
- When there is a special sample hit between the rates, copy the `DWork` into a second `DWork` in the output function.
- Write the second `DWork` to the output at every hit of the output sample rate.

The block can request that the input port be made local but it cannot be set to reusable. The output port can be set to local and reusable.

As in the fast-to-slow transition case, the input should not be read by a rate other than the one assigned to the input port. Similarly, the output should not be written to at a rate other than the rate assigned to the output port.

An optimization can be made when the algorithm being implemented is only required to run at the slow rate. In such cases, you use only one DWork. The input still writes to the DWork in the update function. When there is a special sample hit between the rates, the output function copies the same DWork directly to the output. You must set the output port to be global and not reusable in this case. This optimization results in one less memcpy operation per special sample hit.

In either case, the data that the fast rate computations operate on is always delayed, that is, the data is from the previous step of the slow rate code.

Pseudocode Examples of Slow-to-Fast Rate Transition

The pseudocode below abstracts what your S-function needs to do to handle slow-to-fast transitions, illustrating with an input rate of one second driving an output rate of 0.1 second. The block has following characteristics:

- File: `sfun_multirate_delay.c`, Equation: $y = u(\text{tslow}-1)$
- Input: Set to local, will be local if output/update are combined (ERT) otherwise will be global. Set to not reusable because input needs to be preserved until the update function runs.
- Output: local and reusable
- DirectFeedthrough: no

```
OutputFcn
if (ssIsSampleHit(".1")) {
    if (ssIsSpecialSampleHit("1")) {
        DWork[1] = DWork[0];
    }
    y = DWork[1];
}
UpdateFcn
if (ssIsSampleHit("1")) {
    DWork[0] = u;
}
```

An alternative, optimized approach can be used by some algorithms:

- Input: Set to local, will be local if output/update are combined (ERT) otherwise will be global. Set to not reusable because input needs to be preserved until the update function runs.
- Output: global and not reusable because it needs to persist between special sample hits.
- DirectFeedthrough: no

```
OutputFcn
if (ssIsSampleHit(".1")) {
    if (ssIsSpecialSampleHit("1")) {
        y = DWork;
    }
}
UpdateFcn
if (ssIsSampleHit("1")) {
    DWork = u;
}
```

Example adding a simple algorithm:

- File: `sfun_multirate_modulate.c`, Equation: $y = \sin(t_{\text{fast}}) + u(t_{\text{slow}}-1)$
- Input: Set to local, will be local if output/update are combined (an ERT feature) otherwise will be global. Set to not reusable because input needs to be preserved until the update function runs.
- Output: local and reusable
- DirectFeedthrough: no

```
OutputFcn
if (ssIsSampleHit(".1") {
    if (ssIsSpecialSampleHit("1") {
        /* Processing not likely to be done here. It causes
        * the faster rate to have nonuniform processing
        * requirements (every 10th hit, more code needs to
        * be run).*/
        DWork[1] = DWork[0];
    }
    /* Processing done at fast rate */
    y = sin(ssGetTaskTime(".1")) + DWork[1];
}
UpdateFcn
if (ssIsSampleHit("1")) {
    /* Processing on 'u' can be done here. There is a delay of
    one slow rate period before the fast rate sees it.*/
    DWork[0] = u;}
```

Write Noninlined S-Function

A noninlined S-function is a C or C++ MEX S-function that is treated identically by the Simulink engine and by the generated code. You implement your algorithm once according to the S-function API. The Simulink engine and generated code call the S-function routines (for example, `mdlOutputs`) during model execution.

Noninlined S-functions are identified by the absence of an `sfunction.tlc` file for your S-function. The file name varies depending on your platform. For example, on a 64-bit Microsoft Windows system, the file name is `sfunction.mexw64`. In the MATLAB Command Window, type `mexext` to see which extension your system uses.

Guidelines for Writing Noninlined S-Functions

- The MEX-file cannot call MATLAB functions.
- If the MEX-file uses functions in the MATLAB External Interface libraries, include the header file `cg_sfun.h` instead of `mex.h` or `simulink.c`. For the header file `cg_sfun.h`, at the end of your S-function, include these lines:

```
#ifndef MATLAB_MEX_FILE /* Is this file being compiled as a MEX-file? */
#include "simulink.c" /* MEX-file interface mechanism */
#else
#include "cg_sfun.h" /* Code generation registration function */
#endif
```

- Use only the MATLAB API function that the code generator supports. The supported API functions are:
 - `mxGetDoubles`
 - `mxGetEps`
 - `mxGetInf`
 - `mxGetM`
 - `mxGetN`
 - `mxGetNaN`
 - `mxGetPr` (Not recommended. Use `mxGetDoubles` instead.)
 - `mxGetScalar`
 - `mxGetString`
 - `mxIsEmpty`
 - `mxIsFinite`
 - `mxIsInf`
- MEX library calls are not supported in generated code. To use such calls in the MEX-file and not in the generated code, add the following condition:

```
#ifndef MATLAB_MEX_FILE
#endif
```

- Use only full matrices that contain only real data.
- Do not specify a return value for calls to `mxGetString`. If you do specify a return value, the MEX-file does not compile. Instead, use the second input argument of the function, which returns a pointer to a character vector.

- Use the correct `#define s-function_name` statement. The S-function name that you specify must match the S-function file name.
- If possible, use the data types `real_T` and `int_T` instead of `double` and `int`. The data types `real_T` and `int_T` are more generic and can be used in multiple environments.
- Provide the build process with the names of the modules used to build the S-function. Use a template make file, the `set_param` function, or the `S-function modules` field of the S-Function block parameters dialog box. For example, suppose that you build your S-function with this command:

```
mex sfun_main.c sfun_module1.c sfun_module2.c
```

You can then use the following call to `set_param` to include the required modules:

```
set_param(sfun_block, "SFunctionModules", "sfun_module1 sfun_module2")
```

When you are ready to generate code, force the code generator to rebuild the top model. For more information, see “Control Regeneration of Top Model Code” on page 31-49.

Noninlined S-Function Parameter Type Limitations

Parameters to noninlined S-functions can be of the following types only:

- Double precision
- Characters in scalars, vectors, or 2-D matrices

For more flexibility in the type of parameters that you can supply to S-functions or the operations in the S-function, inline your S-function and consider using an mdlRTW S-function routine.

Use of other functions from the MATLAB `matrix.h` API or other MATLAB APIs, such as `mex.h` and `mat.h`, are not supported. If you call unsupported APIs from an S-function source file, compiler errors occur. For details on supported MATLAB API functions, see the files `matlabroot/rtw/c/src/rt_matrix.h` and `matlabroot/rtw/c/src/rt_matrix.c`

If you use `mxGetPr` on an empty matrix, the function does not return `NULL`. It returns a random value. Therefore, you must protect calls to `mxGetPr` by using `mxIsEmpty`.

See Also

Related Examples

- “Inlined and Noninlined S-Function Code”
- “S-Functions and Code Generation” on page 14-2

Write Wrapper S-Function and TLC Files

Create S-functions that work seamlessly with the Simulink and code generator products by using the wrapper concept. You can:

- Interface your algorithms in Simulink models by writing MEX S-function wrappers (*sfunction.mex*).
- Direct the code generator to insert your algorithm into the generated code by creating a TLC S-function wrapper (*sfunction.tlc*).

MEX S-Function Wrapper

Creating S-functions by using an S-function wrapper enables you to insert C/C++ code algorithms in Simulink models and the generated code with little or no change to your original C/C++ function. A MEX S-function wrapper is an S-function that calls code, which resides in another module.

Note Use a MEX S-function wrapper only in the MATLAB version in which you created the wrapper.

Suppose that you have an algorithm (that is, a C function) called `my_alg` that resides in the file `my_alg.c`. You can integrate `my_alg` into a Simulink model by creating a MEX S-function wrapper (for example, `wrapsfcn.c`). A Simulink model can then call `my_alg` from an S-Function block. The Simulink S-function contains a set of empty functions that the Simulink engine requires for various API related purposes. For example, although only `mdlOutputs` calls `my_alg`, the engine calls `mdlTerminate`, even though this S-function routine performs no action.

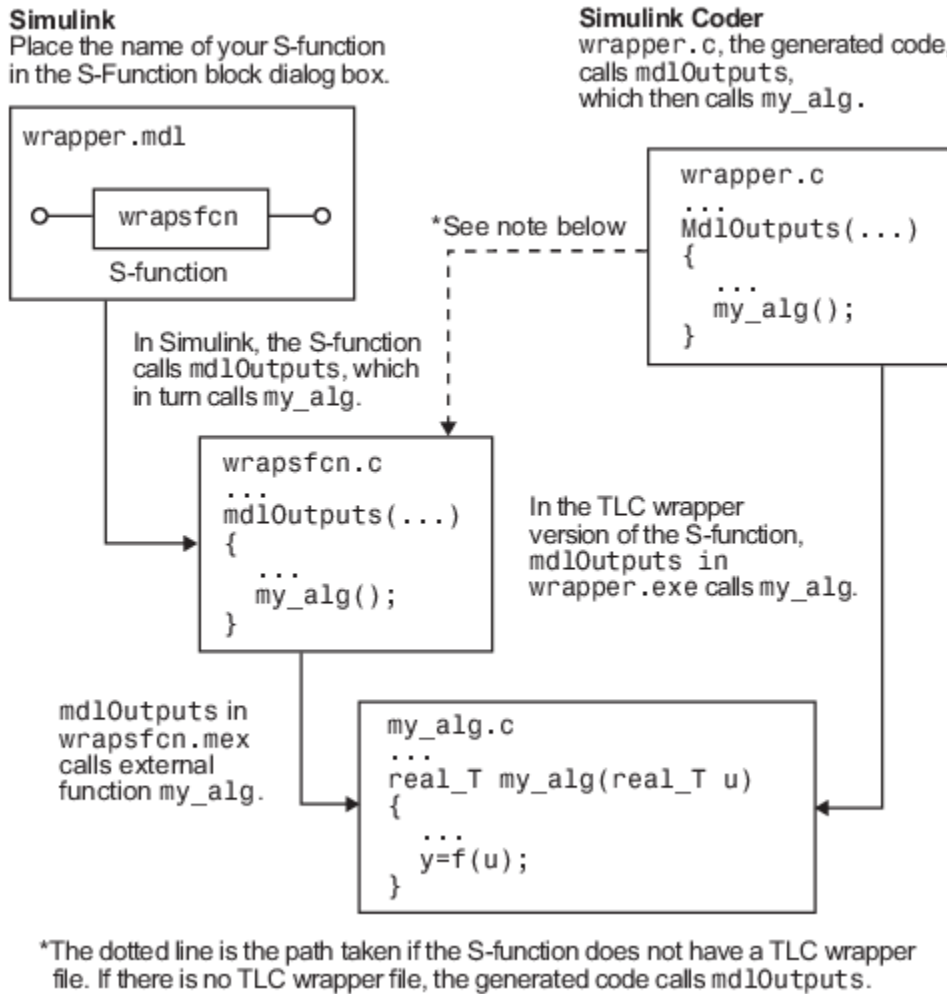
You can embed the call to `my_alg` in the generated code by creating a TLC S-function wrapper (for example, `wrapsfcn.tlc`). You can eliminate the empty function calls. You can avoid the overhead of executing the `mdlOutputs` function and you can then eliminate the `my_alg` function.

Wrapper S-functions are useful when you are creating algorithms that are procedural or when you are integrating legacy code into a Simulink model. If you want to create code that is:

- Interpretive in nature (that is, highly parameterized by operating modes)
- Heavily optimized (that is, no extra tests to decide what mode the code is operating in)

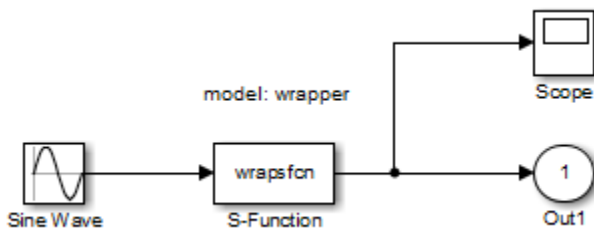
then you must create a fully inlined TLC file for your S-function.

The next figure shows the wrapper S-function concept.



Using an S-function wrapper to import algorithms into your Simulink model means that the S-function serves as an interface that calls your C/C++ algorithms from mdlOutputs. You can quickly integrate large standalone C/C++ programs into your model without having to change the code.

This sample model includes an S-function wrapper.



Two files are associated with the wrapsfcn block: the S-function wrapper and the C/C++ code that contains the algorithm. The first three statements:

- 1 Define the name of the S-function (what you enter in the Simulink S-Function block dialog box).

- 2 Specify that the S-function is using the level 2 format.
- 3 Provide access to the `SimStruct` data structure. `SimStruct` contains pointers to data used during simulation and code generation and defines macros that store data in and retrieve data from the `SimStruct`.

```
#define S_FUNCTION_NAME wrapsfcn
#define S_FUNCTION_LEVEL 2
#include "simstruc.h"

extern real_T my_alg(real_T u); /* Declare my_alg as extern */

/*
 * mdlInitializeSizes - initialize the sizes array
 */
static void mdlInitializeSizes(SimStruct *S)
{
    ssSetNumSFcnParams( S, 0); /*number of input arguments*/

    if (!ssSetNumInputPorts(S, 1)) return;
    ssSetInputPortWidth(S, 0, 1);
    ssSetInputPortDirectFeedThrough(S, 0, 1);

    if (!ssSetNumOutputPorts(S,1)) return;
    ssSetOutputPortWidth(S, 0, 1);

    ssSetNumSampleTimes( S, 1);
}

/*
 * mdlInitializeSampleTimes - indicate that this S-function runs
 * at the rate of the source (driving block)
 */
static void mdlInitializeSampleTimes(SimStruct *S)
{
    ssSetSampleTime(S, 0, INHERITED_SAMPLE_TIME);
    ssSetOffsetTime(S, 0, 0.0);
}

/*
 * mdlOutputs - compute the outputs by calling my_alg, which
 * resides in another module, my_alg.c
 */
static void mdlOutputs(SimStruct *S, int_T tid)
{
    InputRealPtrsType uPtrs = ssGetInputPortRealSignalPtrs(S,0);
    real_T *y = ssGetOutputPortRealSignal(S,0);
    *y = my_alg(*uPtrs[0]); /* Call my_alg in mdlOutputs */
}

/*
 * mdlTerminate - called when the simulation is terminated.
 */
static void mdlTerminate(SimStruct *S)
{
}

#ifdef MATLAB_MEX_FILE /* Is this file being compiled as a MEX-file? */
#include "simulink.c" /* MEX-file interface mechanism */
#else
#include "cg_sfun.h" /* Code generation registration function */
#endif
#endif
```

For more information, see “Templates for C S-Functions”.

The S-function routine `mdlOutputs` contains a function call to `my_alg`, which is the C function containing the algorithm that the S-function performs. For `my_alg.c`, the code is:

```

#ifdef MATLAB_MEX_FILE
#include "tmwtypes.h"
#else
#include "rtwtypes.h"
#endif
real_T my_alg(real_T u)
{
return(u * 2.0);
}

```

For more information, see “Manage Build Process File Dependencies” on page 24-43.

The wrapper S-function `wrapsfcn` calls `my_alg`, which computes $u * 2.0$. To build `wrapsfcn.mex`, use this command:

```
mex wrapsfcn.c my_alg.c
```

TLC S-Function Wrapper

A TLC S-function wrapper is a TLC file that specifies how the code generator calls your code. For example, you can inline the call to `my_alg` in the `mdlOutputs` section of the generated code. In the “MEX S-Function Wrapper” on page 14-93 example, the call to `my_alg` is embedded in the `mdlOutputs` section as:

```
*y = my_alg(*uPtrs[0]);
```

When you are creating a TLC S-function wrapper, the goal is to embed the same type of call in the generated code.

Look at how the code generator executes S-functions that are not inlined. A noninlined S-function is identified by the absence of the file `sfunction.tlc` and the existence of `sfunction.mex`. When generating code for a noninlined S-function, the code generator produces a call to `mdlOutputs` through a function pointer that, in this example, then calls `my_alg`.

The wrapper example contains one S-function, `wrapsfcn.mex`. You must compile and link an additional module, `my_alg`, with the generated code. At the MATLAB command prompt, enter:

```
set_param('wrapper/S-Function', 'SFunctionModules', 'my_alg')
```

Code Overhead for Noninlined S-Functions

The code generated when using `grt.tlc` as the system target file *without* `wrapsfcn.tlc` is:

```
<Generated code comments for wrapper model with noninlined wrapsfcn S-function>
```

```

#include <math.h>
#include <string.h>
#include "wrapper.h"
#include "wrapper.prm"

/* Start the model */
void mdlStart(void)
{
/* (start code not required) */
}

/* Compute block outputs */
void mdlOutputs(int_T tid)
{
/* Sin Block: <Root>/Sin */
rtB.Sin = rtP.Sin.Amplitude *
sin(rtP.Sin.Frequency * ssGetT(rtS) + rtP.Sin.Phase);
}

```



```

/* Level2 S-Function Block: <Root>/S-Function (wrapsfcn) */
{
  /* Noninlined S-functions create a SimStruct object and
   * generate a call to S-function routine mdlOutputs
   */
  SimStruct *rts = ssGetSFunction(rtS, 0);
  sfcnOutputs(rts, tid);
}

/* Output Block: <Root>/Out */
rtY.Out = rtB.S_Function;
}

/* Perform model update */
void mdlUpdate(int_T tid)
{
  /* (update code not required) */
}

/* Terminate function */
void mdlTerminate(void)
{
  /* Level2 S-Function Block: <Root>/S-Function (wrapsfcn) */
  {
    /* Noninlined S-functions require a SimStruct object and
     * the call to S-function routine mdlTerminate
     */
    SimStruct *rts = ssGetSFunction(rtS, 0);
    sfcnTerminate(rts);
  }
}

#include "wrapper.reg"

/* [EOF] wrapper.c */

```

The `wrapper.reg` generated file contains the initialization of the `SimStruct` for the wrapper S-Function block. There is one child `SimStruct` for each S-Function block in your model. You can significantly reduce this overhead by creating a TLC wrapper for the S-function.

Inline a Wrapper S-Function

The generated code makes the call to your S-function, `wrapsfcn.c`, in `mdlOutputs` by using this code:

```

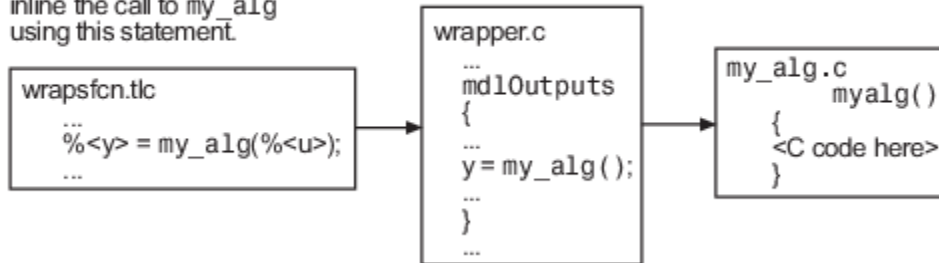
SimStruct *rts = ssGetSFunction(rtS, 0);
sfcnOutputs(rts, tid);

```

This call has computational overhead associated with it. The Simulink engine creates a `SimStruct` data structure for the S-Function block. The code generator constructs a call through a function pointer to execute `mdlOutputs`, then `mdlOutputs` calls `my_alg`. By inlining the call to your C/C++ algorithm, `my_alg`, you can eliminate both the `SimStruct` and the extra function call, thereby improving the efficiency and reducing the size of the generated code.

Inlining a wrapper S-function requires an `sfunction.tlc` file for the S-function. The TLC file must contain the function call to `my_alg`. The figure shows the relationships between the algorithm, the wrapper S-function, and the `sfunction.tlc` file.

The `wrapsfcn.tlc` file tells Simulink Coder how to inline the call to `my_alg` using this statement.



To inline the call to `my_alg`, place your function call in an `sfunction.tlc` file with the same name as the S-function (in this example, `wrapsfcn.tlc`). The Target Language Compiler overrides the default method of placing calls to your S-function in the generated code.

This code is the TLC file `wrapsfcn.tlc` that inlines `wrapsfcn.c`:

```
%% File      : wrapsfcn.tlc
%% Abstract:
%%      Example inlined tlc file for S-function wrapsfcn.c
%%

implements "wrapsfcn" "C"

%% Function: BlockTypeSetup =====
%% Abstract:
%%      Create function prototype in model.h as:
%%      "extern real_T my_alg(real_T u);"
%%
%function BlockTypeSetup(block, system) void
    %openfile buffer
    extern real_T my_alg(real_T u); /* This line is placed in wrapper.h */
    %closefile buffer
    %<LibCacheFunctionPrototype(buffer)>
%endfunction %% BlockTypeSetup

%% Function: Outputs =====
%% Abstract:
%%      y = my_alg( u );
%%
%function Outputs(block, system) Output
    /* %<Type> Block: %<Name> */
    %assign u = LibBlockInputSignal(0, "", "", 0)
    %assign y = LibBlockOutputSignal(0, "", "", 0)
    %% PROVIDE THE CALLING STATEMENT FOR "algorithm"
    %% The following line is expanded and placed in mdlOutputs within wrapper.c
    %<y> = my_alg(%<u>);

%endfunction %% Outputs
```

The first section of this code inlines the `wrapsfcn` S-Function block and generates the code in C:

```
implements "wrapsfcn" "C"
```

The next task is to inform the code generator that the routine `my_alg` must be declared as external in the generated `wrapper.h` file for any `wrapsfcn` S-Function blocks in the model. Do this declaration once for all `wrapsfcn` S-Function blocks by using the `BlockTypeSetup` function. In this function, you direct the Target Language Compiler to create a buffer and cache the `my_alg` as `extern` in the `wrapper.h` generated header file.

The final step is the inlining of the call to the function `my_alg`. The `Outputs` function inlines the call. In this function, you access the block input and output and place a direct call to `my_alg`. The call is embedded in `wrapper.c`.

The Inlined Code

The code generated when you inline your wrapper S-function is similar to the default generated code. The `mdlTerminate` function does not contain a call to an empty function and the `mdlOutputs` function now directly calls `my_alg`.

```
void mdlOutputs(int_T tid)
{
    /* Sin Block: <Root>/Sin */
    rtB.Sin = rtP.Sin.Amplitude *
        sin(rtP.Sin.Frequency * ssGetT(rtS) + rtP.Sin.Phase);

    /* S-Function Block: <Root>/S-Function */
    rtB.S_Function = my_alg(rtB.Sin); /* Inlined call to my_alg */

    /* Output Block: <Root>/Out */
    rtY.Out = rtB.S_Function;
}
```

`wrapper.reg` does not create a child `SimStruct` for the S-function because the generated code is calling `my_alg` directly, eliminating over 1 KB of memory usage.

See Also

Related Examples

- “Write Noninlined S-Function” on page 14-91
- “Write Fully Inlined S-Functions” on page 14-100

Write Fully Inlined S-Functions

A fully inlined S-function builds your algorithm (block) into generated code that you cannot distinguish from a built-in block. Typically, a fully inlined S-function requires you to implement your algorithm twice: once for the Simulink model (C/C++ MEX S-function) and once for code generation (TLC file).

Using the example in “Write Wrapper S-Function and TLC Files” on page 14-93, you can eliminate the call to `my_alg` entirely by specifying the explicit code (that is, `2.0 * u`) in `wrapsfcn.tlc`. While this can improve performance, if you are working with a large amount of C/C++ code, the task can be lengthy. You also have to maintain your algorithm in two places, the C/C++ S-function itself and the corresponding TLC file. Consider whether the performance gains might outweigh the disadvantages. To inline the algorithm used in this example, in the `Outputs` section of your `wrapsfcn.tlc` file, instead of writing:

```
%<y> = my_alg(%<u>);
```

Use:

```
%<y> = 2.0 * %<u>;
```

This code is the code produced in `mdlOutputs`:

```
void mdlOutputs(int_T tid)
{
    /* Sin Block: <Root>/Sin */
    rtB.Sin = rtP.Sin.Amplitude *
        sin(rtP.Sin.Frequency * ssGetT(rtS) + rtP.Sin.Phase);

    /* S-Function Block: <Root>/S-Function */
    rtB.S_Function = 2.0 * rtB.Sin; /* Explicit embedding of algorithm */

    /* Output Block: <Root>/Out */
    rtY.Out = rtB.S_Function;
}
```

The Target Language Compiler replaces the call to `my_alg` with the algorithm itself.

Multiport S-Function

A more advanced multiport inlined S-function example is `sfun_multiport.c` and `sfun_multiport.tlc`. This S-function illustrates how to create a fully inlined TLC file for an S-function that contains multiple ports.

Guidelines for Writing Inlined S-Functions

- Consider using the block property `RTWdata` (see “S-Function `RTWdata`” on page 14-102). This property is a structure of character vectors that you can associate with a block. The code generator saves the structure with the model in the `model.rtw` file and makes the `.rtw` file more readable. For example in the MATLAB Command Window, suppose you enter these commands:

```
mydata.field1 = 'information for field1';
mydata.field2 = 'information for field2';
set_param(sfun_block, 'RTWdata', mydata);
```

The `.rtw` file that the code generator produces for the block includes the comments specified in the structure `mydata`.

- Consider using the `mdlRTW` function to inline your C MEX S-function in the generated code for:
 - Renaming tunable parameters in the generated code.
 - Introducing non-tunable parameters into a TLC file.

See Also

Related Examples

- “Write Fully Inlined S-Functions with `mdlRTW` Routine” on page 14-102

Write Fully Inlined S-Functions with mdlRTW Routine

You can inline more complex S-functions by using the S-function `mdlRTW` routine. The `mdlRTW` routine provides the code generation process with more information about how the S-function is to be inlined by creating a parameter record of a non-tunable parameter for use with a TLC file. The `mdlRTW` routine places information in the `model.rtw` file. The `mdlRTW` function is described in the text file `matlabroot/simulink/src/sfuntmpl_doc.c`.

To use the `mdlRTW` function, take steps to create a direct-index lookup S-function. Lookup tables are collections of ordered data points of a function. Typically, these tables use some interpolation scheme to approximate values of the associated function between known data points. To incorporate the example lookup table algorithm into a Simulink model, the first step is to write an S-function that executes the algorithm in `mdlOutputs`. To produce the most efficient code, the next step is to create a corresponding TLC file to eliminate computational overhead and improve the speed of the lookup computations.

The Simulink product provides support for general-purpose lookup 1-D, 2-D, and n-D algorithms. You can use these algorithms as they are or create a custom lookup table S-function to fit your requirements. You can create a 1-D lookup S-function, `sfun_directlook.c`, and its corresponding inlined `sfun_directlook.tlc` file (see “Target Language Compiler” for more details). You can:

- Error check of S-function parameters.
- Cache information for the S-function that does not change during model execution.
- Use the `mdlRTW` function to customize the code generator to produce the optimal code for a given set of block parameters.
- Create a TLC file for an S-function that either fully inlines the lookup table code or calls a wrapper function for the lookup table algorithm.

S-Function RTWdata

`RTWdata` is a property of blocks, which can be used by the Target Language Compiler when inlining an S-function. `RTWdata` is a structure of character vectors that you can attach to a block. `RTWdata` is saved with the model and placed in the `model.rtw` file when you generate code. For example, this set of MATLAB commands:

```
mydata.field1 = 'information for field1';
mydata.field2 = 'information for field2';
set_param(gcf, 'RTWdata', mydata)
get_param(gcf, 'RTWdata')
```

produces this result:

```
ans =
    field1: 'information for field1'
    field2: 'information for field2'
```

The information for the associated S-Function block inside the `model.rtw` file is:

```
Block {
  Type                "S-Function"
  RTWdata {
    field1             "information for field1"
    field2             "information for field2"
  }
}
```

Note RTWdata is saved in the model file for S-functions that are not linked to a library. However, RTWdata is **not persistent** for S-Function blocks that are linked to a library.

Direct-Index Lookup Table Algorithm

The 1-D lookup table block provided in the Simulink library uses interpolation or extrapolation when computing outputs. In this example, you create a lookup table that directly indexes the output vector (y-data vector) based on the current input (x-data) point.

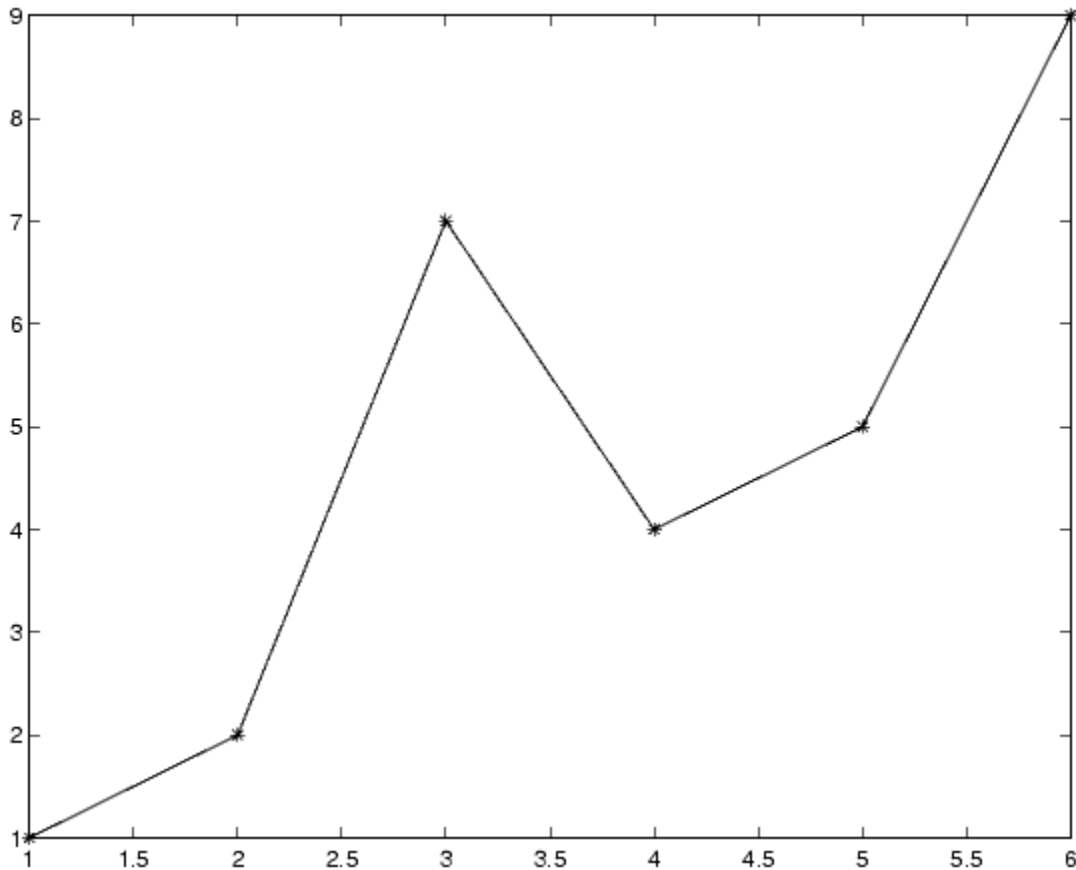
This direct 1-D lookup example computes an approximate solution $p(x)$ to a partially known function $f(x)$ at $x=x_0$, given data point pairs (x,y) in the form of an x-data vector and a y-data vector. For a given data pair (for example, the i 'th pair), $y_i = f(x_i)$. It is assumed that the x-data values are monotonically increasing. If x_0 is outside the range of the x-data vector, the first or last point is returned.

The parameters to the S-function are:

XData, YData, XEvenlySpaced

XData and YData are double vectors of equal length representing the values of the unknown function. XDataEvenlySpaced is a scalar, 0.0 for false and 1.0 for true. If the XData vector is evenly spaced, XDataEvenlySpaced is 1.0 and more efficient code is generated.

The graph shows how the parameters XData=[1:6] and YData=[1,2,7,4,5,9] are handled. For example, if the input (x-value) to the S-Function block is 3, the output (y-value) is 7.



Direct-Index Lookup Table Example

Improve the lookup table by inlining a direct-index S-function with a TLC file. This direct-index lookup table S-function does not require a TLC file. The example uses a TLC file for the direct-index lookup table S-function to reduce the code size and increase efficiency of the generated code.

Implementation of the direct-index algorithm with an inlined TLC file requires the S-function main module, `sfun_directlook.c` and a corresponding `lookup_index.c` module. The `lookup_index.c` module contains the `GetDirectLookupIndex` function that is used to locate the index in the XData for the current `x` input value when the XData is unevenly spaced. The `GetDirectLookupIndex` routine is called from the S-function and the generated code. The example uses the wrapper concept for sharing C/C++ code between Simulink MEX-files and the generated code.

If the XData is evenly spaced, then both the S-function main module and the generated code contain the lookup algorithm to compute the `y`-value of a given `x`-value because the algorithm is short.

The inlined TLC file is `sfun_directlook.tlc`, which is used to either perform a wrapper call or embed the optimal C/C++ code for the S-function. (See the example in “mdlRTW Usage” on page 14-105).

Error Handling

In `sfun_directlook.tlc`, the `mdlCheckParameters` routine verifies that:

- The new parameter settings are valid.
- `XData` and `YData` are vectors of the same length containing real, finite numbers.
- `XDataEvenlySpaced` is a scalar.
- The `XData` vector is a monotonically increasing vector and evenly spaced.

The `mdlInitializeSizes` function explicitly calls `mdlCheckParameters` after it verifies the number of parameters passed to the S-function. After the Simulink engine calls `mdlInitializeSizes`, it then calls `mdlCheckParameters` whenever you change the parameters or reevaluate them.

User Data Caching

In `sfun_directlook.tlc`, the `mdlStart` routine shows how to cache information that does not change during the simulation or while the generated code is executing. The example caches the value of the `XDataEvenlySpaced` parameter in `UserData`, a field of the `SimStruct`. The following line in `mdlInitializeSizes` instructs the Simulink engine to disallow changes to `XDataEvenlySpaced`.

```
ssSetSFcnParamTunable(S, iParam, SS_PRM_NOT_TUNABLE);
```

During execution, `mdlOutputs` accesses the value of `XDataEvenlySpaced` from `UserData` rather than calling the `mxGetPr` MATLAB API function.

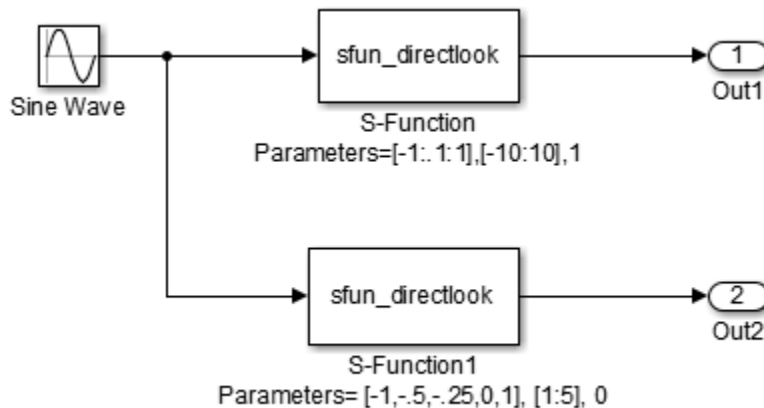
mdlRTW Usage

The code generator calls the `mdlRTW` routine while generating the `model.rtw` file. To produce optimal code for your Simulink model, you can add information to the `model.rtw` file about the mode in which your S-Function block is operating.

The example adds parameter settings to the `model.rtw` file. The parameter settings do not change during execution. In this case, the `XDataEvenlySpaced` S-function parameter cannot change during execution (`ssSetSFcnParamTunable` was specified as false (0) for it in `mdlInitializeSizes`). The parameter setting (`XSpacing`) uses the function `ssWriteRTWParamSettings`.

Because `xData` and `yData` are registered as run-time parameters in `mdlSetWorkWidths`, the code generator writes to the `model.rtw` file automatically.

Before examining the S-function and the inlined TLC file, consider the generated code for this model.



The model uses evenly spaced XData in the top S-Function block and unevenly spaced XData in the bottom S-Function block. When creating this model, specify the following commands for each S-Function block.

```
set_param('sfun_directlook_ex/S-Function','SFunctionModules','lookup_index')
set_param('sfun_directlook_ex/S-Function1','SFunctionModules','lookup_index')
```

The build process uses the module `lookup_index.c` when creating the executable.

When generating code for this model, the code generator uses the S-function `mdlRTW` method to generate a `model.rtw` file with the value `EvenlySpaced` for the `XSpacing` parameter for the top S-Function block and the value `UnEvenlySpaced` for the `XSpacing` parameter for the bottom S-Function block. The TLC-file uses the value of `XSpacing` to determine what algorithm to include in the generated code. The generated code contains the lookup algorithm when the XData is evenly spaced, but calls the `GetDirectLookupIndex` routine when the XData is unevenly spaced. The generated `model.c` or `model.cpp` code for the lookup table example model is similar to this code:

```
/*
 * sfun_directlook_ex.c
 *
 * Code generation for Simulink model
 * "sfun_directlook_ex.slx".
 *
 * ...
 */

#include "sfun_directlook_ex.h"
#include "sfun_directlook_ex_private.h"

/* External outputs (root outputs fed by signals with auto storage) */
ExtY_sfundirectlook_ex_T sfundirectlook_ex_Y;

/* Real-time model */
RT_MODEL_sfundirectlook_ex_T sfundirectlook_ex_M;
RT_MODEL_sfundirectlook_ex_T *const sfundirectlook_ex_M =
    &sfundirectlook_ex_M;

/* Model output function */
void sfundirectlook_ex_output(void)
{
    /* local block i/o variables */
    real_T rtb_SFunction;
    real_T rtb_SFunction1;

    /* Sin: '<Root>/Sine Wave' */
    rtb_SFunction1 = sin(sfundirectlook_ex_M->Timing.t[0]);
}
```

```

/* Code that is inlined for the top S-function block in the
 * sfun_directlook_ex model
 */
/* S-Function (sfun_directlook): '<Root>/S-Function' */
{
  const real_T *xData = sfun_directlook_ex_ConstP.SFunction_XData;
  const real_T *yData = sfun_directlook_ex_ConstP.SFunction_YData;
  real_T spacing = xData[1] - xData[0];
  if (rtb_SFunction1 <= xData[0] ) {
    rtb_SFunction = yData[0];
  } else if (rtb_SFunction1 >= yData[20] ) {
    rtb_SFunction = yData[20];
  } else {
    int_T idx = (int_T)( ( rtb_SFunction1 - xData[0] ) / spacing );
    rtb_SFunction = yData[idx];
  }
}

/* Outport: '<Root>/Out1' */
sfun_directlook_ex_Y.Out1 = rtb_SFunction;

/* Code that is inlined for the bottom S-function block in the
 * sfun_directlook_ex model
 */
/* S-Function (sfun_directlook): '<Root>/S-Function1' */
{
  const real_T *xData = sfun_directlook_ex_ConstP.SFunction1_XData;
  const real_T *yData = sfun_directlook_ex_ConstP.SFunction1_YData;
  int_T idx;
  idx = GetDirectLookupIndex(xData, 5, rtb_SFunction1);
  rtb_SFunction1 = yData[idx];
}

/* Outport: '<Root>/Out2' */
sfun_directlook_ex_Y.Out2 = rtb_SFunction1;
}

/* Model update function */
void sfun_directlook_ex_update(void)
{
  /* signal main to stop simulation */
  {
    /* Sample time: [0.0s, 0.0s] */
    if ((rtmGetTFinal(sfun_directlook_ex_M)!=-1) &&
        !((rtmGetTFinal(sfun_directlook_ex_M)-sfun_directlook_ex_M->Timing.t[0])
          > sfun_directlook_ex_M->Timing.t[0] * (DBL_EPSILON))) {
      rtmSetErrorStatus(sfun_directlook_ex_M, "Simulation finished");
    }
  }
}

/* Update absolute time for base rate */
/* The "clockTick0" counts the number of times the code of this task has
 * been executed. The absolute time is the multiplication of "clockTick0"
 * and "Timing.stepSize0". Size of "clockTick0" ensures timer will not
 * overflow during the application lifespan selected.
 * Timer of this task consists of two 32 bit unsigned integers.
 * The two integers represent the low bits Timing.clockTick0 and the high bits
 * Timing.clockTickH0. When the low bit overflows to 0, the high bits increment.
 */
if (!(++sfun_directlook_ex_M->Timing.clockTick0)) {
  ++sfun_directlook_ex_M->Timing.clockTickH0;
}

sfun_directlook_ex_M->Timing.t[0] = sfun_directlook_ex_M->Timing.clockTick0 *
sfun_directlook_ex_M->Timing.stepSize0 +
sfun_directlook_ex_M->Timing.clockTickH0 *
sfun_directlook_ex_M->Timing.stepSize0 * 4294967296.0;
}
...

```

See Also

Related Examples

- “Write Fully Inlined S-Functions” on page 14-100

Data, Function, and File Definition

Data Representation in Simulink Coder

- “Preserve Variables in Generated Code” on page 15-2
- “Access Signal, State, and Parameter Data During Execution” on page 15-17
- “Data Structures in the Generated Code” on page 15-31
- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “Control Data and Function Interface in Generated Code” on page 15-44
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49
- “Choose Data Configuration Approach” on page 15-70
- “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75
- “Use Enumerated Data in Generated Code” on page 15-86
- “Data Stores in Generated Code” on page 15-94
- “Specify Single-Precision Data Type for Embedded Application” on page 15-104
- “Tune Phase Parameter of Sine Wave Block During Code Execution” on page 15-107
- “Switch Between Output Waveforms During Code Execution for Waveform Generator Block” on page 15-109
- “Create Tunable Calibration Parameter in the Generated Code” on page 15-112
- “Limitations for Block Parameter Tunability in Generated Code” on page 15-122
- “Code Generation of Parameter Objects With Expression Values” on page 15-125
- “Configure Packaging of Parameter Arguments in Generated Code” on page 15-127
- “Parameter Data Types in the Generated Code” on page 15-131
- “Generate Efficient Code by Specifying Data Types for Block Parameters” on page 15-135
- “Reuse Parameter Data in Different Data Type Contexts” on page 15-143
- “Organize Data into Structures in Generated Code” on page 15-147
- “Switch Between Sets of Parameter Values During Simulation and Code Execution” on page 15-162
- “Design Data Interface by Configuring Inport and Outport Blocks” on page 15-170
- “Generate Efficient Code for Bus Signals” on page 15-174
- “Control Signal and State Initialization in the Generated Code” on page 15-177
- “Initialization of Signal, State, and Parameter Data in the Generated Code” on page 15-188
- “Optimize Generated Code Using Fixed-Point Data with Simulink, Stateflow, and MATLAB” on page 15-197
- “Declare Workspace Variables as Tunable Parameters Using the Model Parameter Configuration Dialog Box” on page 15-198
- “Share Data Between Code Generated from Simulink, Stateflow, and MATLAB” on page 15-201

Preserve Variables in Generated Code

As you iteratively develop a model, you capture the values of signals and states that model execution generates. You also tune parameter values during execution to observe results on the signals and states. You can then base your design decisions upon analysis of these outputs. To access signal, state, and parameter data in a rapid prototyping environment, you can configure the generated code to store the data in addressable memory.

Depending on the system target file that you use (such as `grt.tlc`), by default, optimization settings can make the generated code more efficient by:

- Eliminating unnecessary global and local storage for signals and some states.

However, the optimizations do not eliminate storage for root-level Inport and Outport blocks, which represent the primary inputs and outputs of a system. To access this data, you do not need to take optimizations into consideration.

- Inlining the numeric values of block parameters. The code does not store the parameters in memory, so you cannot interact with the parameters during code execution.

To generate code that instead allocates addressable memory for this data, you can disable the optimizations, or override the optimizations by specifying code generation settings for individual data items.

Access Signal, State, and Parameter Data During Execution

As you iteratively develop a model, you capture output signal and state data that model execution generates. You also tune parameter values during execution to observe results on the outputs. You can then base your design decisions upon analysis of these outputs. To access this signal, state, and parameter data in a rapid prototyping environment, you can configure the generated code to store the data in addressable memory.

By default, optimization settings make the generated code more efficient by eliminating unnecessary signal storage and inlining the numeric values of block parameters. To generate code that instead allocates addressable memory for this data, you can disable the optimizations or specify code generation settings for individual data items.

Explore Example Model

Run a script that prepares the model `rtwdemo_configrpinterface` for this example.

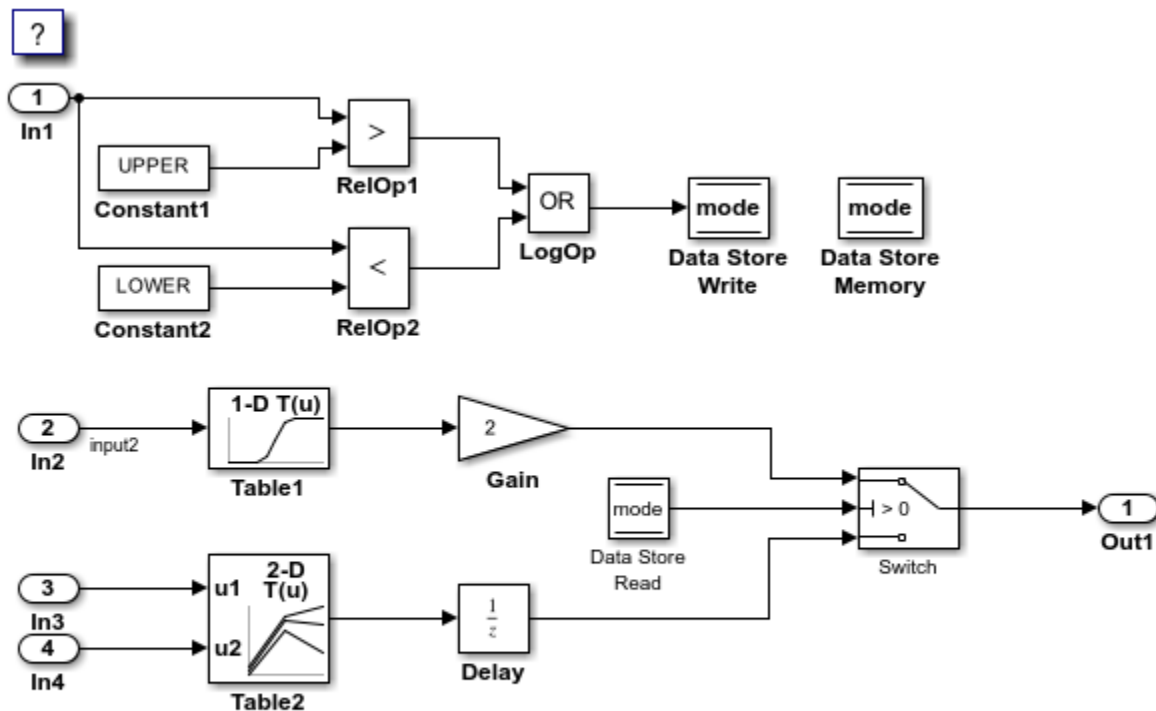
```
prepare_rtwdemo_configrpinterface
```

This data is configured for code generation:

- Parameters `UPPER`, `LOWER`, `K1`, `Table1`, and `Table2`
- Signals `In1`, `In2`, `In3`, `In4`, and `Out1`
- States `X` (delay) and `mode` (data store memory write)

Open the example model, `rtwdemo_configrpinterface`.

```
rtwdemo_configrpinterface
```

Copyright 1994-2020 The MathWorks, Inc.

The model loads numeric MATLAB® variables into the model workspace. The workspace variables set some block parameters in the model. However, the Gain block in the model uses the literal value 2.

Disable Optimizations

1. In the model, clear the model configuration parameter **Signal storage reuse**. When you clear this optimization and other optimizations such as **Eliminate superfluous local variables (expression folding)**, the generated code allocates memory for signal lines. Clearing **Signal storage reuse** disables most of the other optimizations.

```
set_param('rtwdemo_configrpinterface','OptimizeBlockIOStorage','off')
```

2. Set the model configuration parameter **Default parameter behavior** to Tunable. When set to Tunable, this configuration parameter causes the generated code to allocate memory for block parameters and workspace variables.

```
set_param('rtwdemo_configrpinterface','DefaultParameterBehavior','Tunable')
```

3. Generate code.

```
slbuild('rtwdemo_configrpinterface')
```

```
### Starting build procedure for: rtwdemo_configrpinterface
### Successful completion of build procedure for: rtwdemo_configrpinterface
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_configrinterface	Code generated and compiled.	Code generation information file does not exist

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 37.816s

4. In the code generation report, view the file `rtwdemo_configrinterface.h`. This header file defines a structure type that contains signal data. The structure contains fields that each represent a signal line in the model. For example, the output signal of the Gain block, whose name is `Gain`, appears as the field `Gain`.

```
file = fullfile('rtwdemo_configrinterface_grt_rtw','rtwdemo_configrinterface.h');
rtwdemodbtype(file,'/* Block signals (default storage) */',...
    'B_rtwdemo_configrinterface_T;',1,1)
```

```
/* Block signals (default storage) */
typedef struct {
    real_T Delay; /* '<Root>/Delay' */
    real_T Table2; /* '<Root>/Table2' */
    real_T Table1; /* '<Root>/Table1' */
    real_T Gain; /* '<Root>/Gain' */
    boolean_T RelOp1; /* '<Root>/RelOp1' */
    boolean_T RelOp2; /* '<Root>/RelOp2' */
    boolean_T LogOp; /* '<Root>/LogOp' */
    boolean_T DataStoreRead; /* '<Root>/Data Store Read' */
} B_rtwdemo_configrinterface_T;
```

The file defines a structure type that contains block parameter data. For example, the **Gain** parameter of the Gain block appears as the field `Gain_Gain`. The other fields of the structure represent other block parameters and workspace variables from the model, including initial conditions for signals and states.

```
rtwdemodbtype(file,'/* Parameters (default storage) */',...
    '/* Real-time Model Data Structure */',1,0)
```

```
/* Parameters (default storage) */
struct P_rtwdemo_configrinterface_T_ {
    Table1_Type Table1; /* Variable: Table1
    * Referenced by: '<Root>/Table1'
    */
    Table2_Type Table2; /* Variable: Table2
    * Referenced by: '<Root>/Table2'
    */
    real_T LOWER; /* Variable: LOWER
    * Referenced by: '<Root>/Constant2'
    */
    real_T UPPER; /* Variable: UPPER
    * Referenced by: '<Root>/Constant1'
    */
    real_T Gain_Gain; /* Expression: 2
    * Referenced by: '<Root>/Gain'
    */
};
```

```

real_T Delay_InitialCondition;      /* Expression: 0
                                   * Referenced by: '<Root>/Delay'
                                   */
uint32_T Table2_maxIndex[2];      /* Computed Parameter: Table2_maxIndex
                                   * Referenced by: '<Root>/Table2'
                                   */
boolean_T DataStoreMemory_InitialValue;
                                   /* Computed Parameter: DataStoreMemory_InitialValue
                                   * Referenced by: '<Root>/Data Store Memory'
                                   */
};

```

5. View the file `rtwdemo_configrpinterface_data.c`. This source file allocates global memory for a parameter structure and initializes the field values based on the parameter values in the model.

6. View the source file `rtwdemo_configrpinterface.c`. The code allocates global memory for a structure variable that contains signal data.

```

file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface.c');
rtwdemodbtype(file, /* Block signals (default storage) */,...
              'B_rtwdemo_configrpinterface_T rtwdemo_configrpinterface_B;',1,1)

/* Block signals (default storage) */
B_rtwdemo_configrpinterface_T rtwdemo_configrpinterface_B;

```

The code algorithm in the model `step` function calculates the signal values. It then assigns these values to the fields of the signal structure. To perform the calculations, the algorithm uses the parameter values from the fields of the parameter structure.

Exclude Data Items from Optimizations

When you want to select code generation optimizations such as **Signal storage reuse**, you can preserve individual data items from the optimizations. The generated code then allocates addressable memory for the items.

Select the optimizations that you previously cleared.

```

set_param('rtwdemo_configrpinterface','OptimizeBlockIOStorage','on')
set_param('rtwdemo_configrpinterface','LocalBlockOutputs','on')
set_param('rtwdemo_configrpinterface','DefaultParameterBehavior','Inlined')

```

1. In the **C Code** tab, select **Code Interface > Individual Element Code Mappings**.
2. In the Code Mappings editor, inspect the **Signals/States** tab.
3. In the model, select the output signal of the Gain block.
4. Pause on the ellipsis that appears above or below the signal line to open the action bar. Click the **Add Signal** button. In the Code Mappings editor, the **Signals** node expands and lists the signal that you added.

```

cm = coder.mapping.api.get('rtwdemo_configrpinterface');
gain_ports = get_param('rtwdemo_configrpinterface/Gain','PortHandles');
gain_outPort = gain_ports.Outport;
addSignal(cm,gain_outPort);

```

5. Apply the default storage class for Outports to the signal by setting **Storage Class** to Model default.

```
setSignal(cm,gain_outPort,'StorageClass','Model default','Identifier','Gain');
```

6. In the model, select the Gain block.

7. In the Property Inspector, set the gain value to the model workspace parameter object K1.

8. In the Code Mappings editor, on the **Parameters** tab, expand **Model Parameters**.

9. Select parameter K1. Apply a storage class other than Auto by setting **Storage class**. For example, use the storage class Model default. By using Model default, K1 acquires the default storage class specified for model parameters, Default, and appears as a field of the global parameters structure.

```
set_param('rtwdemo_configrpinterface/Gain','Gain','K1');
setModelParameter(cm,'K1','StorageClass','Model default');
```

10. Generate code.

```
slbuild('rtwdemo_configrpinterface')

### Starting build procedure for: rtwdemo_configrpinterface
### Successful completion of build procedure for: rtwdemo_configrpinterface

Build Summary

Top model targets built:

Model                Action                Rebuild Reason
=====
rtwdemo_configrpinterface  Code generated and compiled.  Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 15.563s
```

11. In the code generation report, view the file `rtwdemo_configrpinterface.h`. The structure that contains signal data now defines only one field, Gain, which represents the test-pointed output of the Gain block.

```
file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface.h');
rtwdemodbtype(file,'/* Block signals (default storage) */',...
    'B_rtwdemo_configrpinterface_T;',1,1)
```

```
/* Block signals (default storage) */
typedef struct {
    real_T Gain;                /* '<Root>/Gain' */
} B_rtwdemo_configrpinterface_T;
```

The structure that contains block parameter data defines one field, K1, which represents the parameter object K1.

```
rtwdemodbtype(file,'/* Parameters (default storage) */',...
    '/* Real-time Model Data Structure */',1,0)
```

```
/* Parameters (default storage) */
```

```

struct P_rtwdemo_configrpininterface_T_ {
    int8_T K1; /* Variable: K1
               * Referenced by: '<Root>/Gain'
               */
};

```

Access Data Through Generated Interfaces

You can configure the generated code to contain extra code and files so that you can access model data through standardized interfaces. For example, use the C API to log signal data and tune parameters during execution.

Copy this custom source code into a file named `ex_myHandCode.c` in your current folder.

```

#include "ex_myHandHdr.h"

#define paramIdx 0 /* Index of the target parameter,
determined by inspecting the array of structures generated by the C API. */
#define sigIdx 0 /* Index of the target signal,
determined by inspecting the array of structures generated by the C API. */

void tuneFcn(rtwCAPI_ModelMappingInfo *mmi, time_T *tPtr)
{
    /* Take action with the parameter value only at
    the beginning of simulation and at the 5-second mark. */
    if (*tPtr == 0 || *tPtr == 5) {

        /* Local variables to store information extracted from
        the model mapping information (mmi). */
        void** dataAddrMap;
        const rtwCAPI_DataTypeMap *dataTypeMap;
        const rtwCAPI_ModelParameters *params;
        int_T addrIdx;
        uint16_T dTypeIdx;
        uint8_T slDataType;

        /* Use built-in C API macros to extract information. */
        dataAddrMap = rtwCAPI_GetDataAddressMap(mmi);
        dataTypeMap = rtwCAPI_GetDataTypeMap(mmi);
        params = rtwCAPI_GetModelParameters(mmi);
        addrIdx = rtwCAPI_GetModelParameterAddrIdx(params,paramIdx);
        dTypeIdx = rtwCAPI_GetModelParameterDataTypeIdx(params,paramIdx);
        slDataType = rtwCAPI_GetDataTypeSLId(dataTypeMap, dTypeIdx);

        /* Handle data types 'double' and 'int8'. */
        switch (slDataType) {

            case SS_DOUBLE: {
                real_T* dataAddress;
                dataAddress = dataAddrMap[addrIdx];
                /* At the 5-second mark, increment the parameter value by 1. */
                if (*tPtr == 5) {
                    (*dataAddress)++;
                }
                printf("Parameter value is %f\n", *dataAddress);
                break;
            }
        }
    }
}

```

```

        case SS_INT8: {
            int8_T* dataAddress;
            dataAddress = dataAddrMap[addrIdx];
            if (*tPtr == 5) {
                (*dataAddress)++;
            }
            printf("Parameter value is %i\n", *dataAddress);
            break;
        }
    }
}

void logFcn(rtwCAPI_ModelMappingInfo *mmi, time_T *tPtr)
{
    /* Take action with the signal value only when
       the simulation time is an integer value. */
    if (*tPtr-(int_T)*tPtr == 0) {

        /* Local variables to store information extracted from
           the model mapping information (mmi). */
        void** dataAddrMap;
        const rtwCAPI_DataTypeMap *dataTypeMap;
        const rtwCAPI_Signals *sigs;
        int_T addrIdx;
        uint16_T dTypeIdx;
        uint8_T slDataType;

        /* Use built-in C API macros to extract information. */
        dataAddrMap = rtwCAPI_GetDataAddressMap(mmi);
        dataTypeMap = rtwCAPI_GetDataTypeMap(mmi);
        sigs = rtwCAPI_GetSignals(mmi);
        addrIdx = rtwCAPI_GetSignalAddrIdx(sigs, sigIdx);
        dTypeIdx = rtwCAPI_GetSignalDataTypeIdx(sigs, sigIdx);
        slDataType = rtwCAPI_GetDataTypeSLId(dataTypeMap, dTypeIdx);

        /* Handle data types 'double' and 'single'. */
        switch (slDataType) {

            case SS_DOUBLE: {
                real_T* dataAddress;
                dataAddress = dataAddrMap[addrIdx];
                printf("Signal value is %f\n", *dataAddress);
                break;
            }

            case SS_SINGLE: {
                real32_T* dataAddress;
                dataAddress = dataAddrMap[addrIdx];
                printf("Signal value is %f\n", *dataAddress);
                break;
            }
        }
    }
}

```

2. Copy this custom header code into a file named `ex_myHandHdr.h` in your current folder.

```
#include <stdio.h>
#include <string.h>
#include <math.h>
/* Include rtw_modelmap.h for definitions of C API macros. */
#include "rtw_modelmap.h"
#include "builtin_typeid_types.h"
#include "rtwtypes.h"
void tuneFcn(rtwCAPI_ModelMappingInfo *mmi, time_T *tPtr);
void logFcn(rtwCAPI_ModelMappingInfo *mmi, time_T *tPtr);
```

These files use the C API to access signal and parameter data in the code that you generate from the example model.

3. Set model configuration parameters **Include headers** and **Source files** to #include "ex_myHandHdr.h" and ex_myHandCode.c, respectively.

```
set_param('rtwdemo_configrpinterface', 'CustomHeaderCode', '#include "ex_myHandHdr.h"')
set_param('rtwdemo_configrpinterface', 'CustomSource', 'ex_myHandCode.c')
```

4. Select model configuration parameter **MAT-file Logging**. The generated executable runs only until the simulation stop time (which you set in the model configuration parameters).

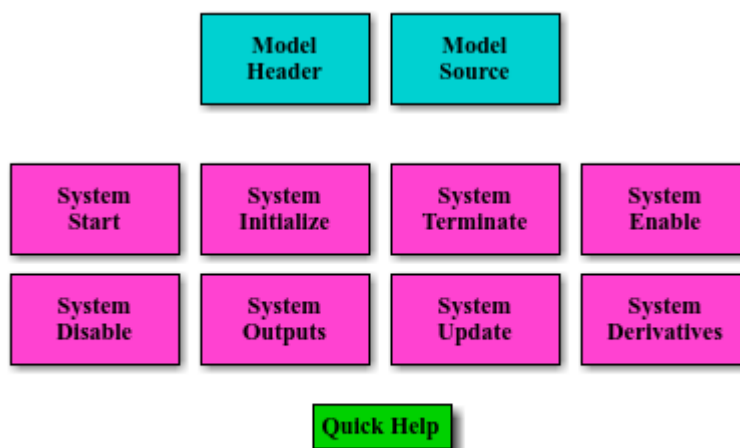
```
set_param('rtwdemo_configrpinterface', 'MatFileLogging', 'on')
```

5. Select C API model configuration parameters **parameters**, **signals**, **states**, and **root-level I/O**.

```
set_param('rtwdemo_configrpinterface', 'RTWCAPIParams', 'on')
set_param('rtwdemo_configrpinterface', 'RTWCAPISignals', 'on')
set_param('rtwdemo_configrpinterface', 'RTWCAPIStates', 'on')
set_param('rtwdemo_configrpinterface', 'RTWCAPIRootIO', 'on')
```

6. Load the Custom Code block library.

custcode



These blocks allow you to insert custom code into specific files and functions.

7. Add a System Outputs block to the model.

```
add_block('custcode/System Outputs','rtwdemo_configrpinterface/System Outputs')
```

8. In the System Outputs block dialog box, set **System Outputs Function Execution Code** to this code:

```
{
rtwdemo_configrpinterface_U.input2++;
rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo(rtwdemo_configrpinterface_M).mmi);
tuneFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));
}
```

9. In the block dialog box, set **System Outputs Function Exit Code** to this code:

```
{
rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo(rtwdemo_configrpinterface_M).mmi);
logFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));
}
```

Alternatively, to configure the System Outputs block, at the command prompt, use these commands:

```
temp.TLCFile = 'custcode';
temp.Location = 'System Outputs Function';
temp.Middle = sprintf(['\nrtwdemo_configrpinterface_U.input2++;'...
'\nrtwCAPI_ModelMappingInfo *MMI = '...
'\nrtmGetDataMapInfo(rtwdemo_configrpinterface_M).mmi);'...
'\ntuneFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));\n}']);
temp.Bottom = sprintf(['\nrtwCAPI_ModelMappingInfo *MMI = '...
'\nrtmGetDataMapInfo(rtwdemo_configrpinterface_M).mmi);'...
'\nlogFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));\n}']);
set_param('rtwdemo_configrpinterface/System Outputs','RTWdata',temp)
```

10. Generate code.

```
slbuild('rtwdemo_configrpinterface')

### Starting build procedure for: rtwdemo_configrpinterface
### Successful completion of build procedure for: rtwdemo_configrpinterface

Build Summary

Top model targets built:

Model                Action                Rebuild Reason
=====
rtwdemo_configrpinterface  Code generated and compiled.  Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 16.705s
```

11. In the code generation report, view the interface file `rtwdemo_configrpinterface_capi.c`. This file initializes the arrays of structures that you can use to interact with data items through the C API. For example, in the array of structures `rtBlockSignals`, the first structure (index 0) describes the test-pointed output signal of the Gain block in the model.

```
file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface_capi.c');
rtwdemodbtype(file,'/* Block output signal information */',...
'\n/* Individual block tuning',1,0)

/* Block output signal information */
```



```

static const rtwCAPI_Signals rtBlockSignals[] = {
  /* addrMapIndex, sysNum, blockPath,
   * signalName, portNumber, dataTypeIndex, dimIndex, fxpIndex, sTimeIndex
   */
  { 0, 0, TARGET_STRING("rtwdemo_configrpinterface/Gain"),
    TARGET_STRING(""), 0, 0, 0, 0, 0 },

  {
    0, 0, (NULL), (NULL), 0, 0, 0, 0, 0
  }
};

```

The fields of the structure, such as `addrMapIndex`, indicate indices into other arrays of structures, such as `rtDataAddrMap`, that describe the characteristics of the signal. These characteristics include the address of the signal data (a pointer to the data), the numeric data type, and the dimensions of the signal.

12. In the file `rtwdemo_configrpinterface.c`, view the code algorithm in the model step function. The algorithm first executes the code that you specified in the System Outputs block.

```

file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface.c');
rtwdemodbtype(file,'/* user code (Output function Body) */',...
  '/* Logic: '<Root>/LogOp' incorporates:',1,0)

/* user code (Output function Body) */

/* System '<Root>' */
{
  rtwdemo_configrpinterface_U.input2++;
  rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo
    (rtwdemo_configrpinterface_M).mmi);
  tuneFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));
}

/* DataStoreWrite: '<Root>/Data Store Write' incorporates:
 * Constant: '<Root>/Constant1'
 * Constant: '<Root>/Constant2'
 * Inport: '<Root>/In1'
 * Logic: '<Root>/LogOp'
 * RelationalOperator: '<Root>/RelOp1'
 * RelationalOperator: '<Root>/RelOp2'
 */
mode = ((rtwdemo_configrpinterface_U.In1 > 10.0) ||
  (rtwdemo_configrpinterface_U.In1 < -10.0));

/* Gain: '<Root>/Gain' incorporates:
 * Inport: '<Root>/In2'
 * Lookup_n-D: '<Root>/Table1'
 */
rtwdemo_configrpinterface_B.Gain = (real_T)rtwdemo_configrpinterface_P.K1 *
  look1_binlc(rtwdemo_configrpinterface_U.input2, rtCP_Table1_bp01Data,
    rtCP_Table1_tableData, 10U);

/* Switch: '<Root>/Switch' incorporates:
 * DataStoreRead: '<Root>/Data Store Read'
 */
if (mode) {

```

```

    /* Output: '<Root>/Out1' */
    output = rtwdemo_configrpinterface_B.Gain;
} else {
    /* Output: '<Root>/Out1' incorporates:
    * UnitDelay: '<Root>/Delay'
    */
    output = rtwdemo_configrpinterface_DW.X;
}

/* End of Switch: '<Root>/Switch' */

/* Lookup_n-D: '<Root>/Table2' incorporates:
* Inport: '<Root>/In3'
* Inport: '<Root>/In4'
*/
rtwdemo_configrpinterface_DW.X = look2_binlc(rtwdemo_configrpinterface_U.In3,
    rtwdemo_configrpinterface_U.In4, rtCP_Table2_bp01Data, rtCP_Table2_bp02Data,
    rtCP_Table2_tableData, rtCP_Table2_maxIndex, 3U);

/* user code (Output function Trailer) */

/* System '<Root>' */
{
    rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo
        (rtwdemo_configrpinterface_M).mmi);
    logFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));
}

/* Matfile logging */
rt_UpdateTXYLogVars(rtwdemo_configrpinterface_M->rtwLogInfo,
    (&rtwdemo_configrpinterface_M->Timing.taskTime0));

/* signal main to stop simulation */
{
    /* Sample time: [1.0s, 0.0s] */
    if ((rtmGetTFinal(rtwdemo_configrpinterface_M)!=-1) &&
        !((rtmGetTFinal(rtwdemo_configrpinterface_M)-
            rtwdemo_configrpinterface_M->Timing.taskTime0) >
            rtwdemo_configrpinterface_M->Timing.taskTime0 * (DBL_EPSILON))) {
        rtmSetErrorStatus(rtwdemo_configrpinterface_M, "Simulation finished");
    }
}

/* Update absolute time for base rate */
/* The "clockTick0" counts the number of times the code of this task has
* been executed. The absolute time is the multiplication of "clockTick0"
* and "Timing.stepSize0". Size of "clockTick0" ensures timer will not
* overflow during the application lifespan selected.
* Timer of this task consists of two 32 bit unsigned integers.
* The two integers represent the low bits Timing.clockTick0 and the high bits
* Timing.clockTickH0. When the low bit overflows to 0, the high bits increment.
*/
if (!(++rtwdemo_configrpinterface_M->Timing.clockTick0)) {
    ++rtwdemo_configrpinterface_M->Timing.clockTickH0;
}

rtwdemo_configrpinterface_M->Timing.taskTime0 =
    rtwdemo_configrpinterface_M->Timing.clockTick0 *
    rtwdemo_configrpinterface_M->Timing.stepSize0 +

```

```

    rtwdemo_configrpinterface_M->Timing.clockTickH0 *
    rtwdemo_configrpinterface_M->Timing.stepSize0 * 4294967296.0;
}

/* Model initialize function */
void rtwdemo_configrpinterface_initialize(void)
{
    /* Registration code */

    /* initialize non-finites */
    rt_InitInfAndNaN(sizeof(real_T));

    /* initialize real-time model */
    (void) memset((void *)rtwdemo_configrpinterface_M, 0,
        sizeof(RT_MODEL_rtwdemo_configrpinte_T));
    rtmSetTFinal(rtwdemo_configrpinterface_M, 10.0);
    rtwdemo_configrpinterface_M->Timing.stepSize0 = 1.0;

    /* Setup for data logging */
    {
        static RTWLogInfo rt_DataLoggingInfo;
        rt_DataLoggingInfo.loggingInterval = (NULL);
        rtwdemo_configrpinterface_M->rtwLogInfo = &rt_DataLoggingInfo;
    }

    /* Setup for data logging */
    {
        rtwliSetLogXSignalInfo(rtwdemo_configrpinterface_M->rtwLogInfo, (NULL));
        rtwliSetLogXSignalPtrs(rtwdemo_configrpinterface_M->rtwLogInfo, (NULL));
        rtwliSetLogT(rtwdemo_configrpinterface_M->rtwLogInfo, "");
        rtwliSetLogX(rtwdemo_configrpinterface_M->rtwLogInfo, "");
        rtwliSetLogXFinal(rtwdemo_configrpinterface_M->rtwLogInfo, "");
        rtwliSetLogVarNameModifier(rtwdemo_configrpinterface_M->rtwLogInfo, "rt_");
        rtwliSetLogFormat(rtwdemo_configrpinterface_M->rtwLogInfo, 0);
        rtwliSetLogMaxRows(rtwdemo_configrpinterface_M->rtwLogInfo, 1000);
        rtwliSetLogDecimation(rtwdemo_configrpinterface_M->rtwLogInfo, 1);
        rtwliSetLogY(rtwdemo_configrpinterface_M->rtwLogInfo, "");
        rtwliSetLogYSignalInfo(rtwdemo_configrpinterface_M->rtwLogInfo, (NULL));
        rtwliSetLogYSignalPtrs(rtwdemo_configrpinterface_M->rtwLogInfo, (NULL));
    }

    /* block I/O */
    (void) memset(((void *) &rtwdemo_configrpinterface_B), 0,
        sizeof(B_rtwdemo_configrpinterface_T));

    /* Storage classes */
    output = 0.0;

    /* states (dwork) */
    (void) memset((void *)&rtwdemo_configrpinterface_DW, 0,
        sizeof(DW_rtwdemo_configrpinterface_T));

    /* exported global states */
    mode = false;

    /* external inputs */
    (void)memset(&rtwdemo_configrpinterface_U, 0, sizeof
        (ExtU_rtwdemo_configrpinterfac_T));
}

```

```

/* Initialize DataMapInfo substructure containing ModelMap for C API */
rtwdemo_configrpininterface_InitializeDataMapInfo();

/* Matfile logging */
rt_StartDataLoggingWithStartTime(rtwdemo_configrpininterface_M->rtwLogInfo, 0.0,
    rtmGetTFinal(rtwdemo_configrpininterface_M),
    rtwdemo_configrpininterface_M->Timing.stepSize0, (&rtmGetErrorStatus
    (rtwdemo_configrpininterface_M)));

/* Start for DataStoreMemory: '<Root>/Data Store Memory' */
mode = false;

/* InitializeConditions for Lookup_n-D: '<Root>/Table2' incorporates:
 * UnitDelay: '<Root>/Delay'
 */
rtwdemo_configrpininterface_DW.X = 0.0;
}

/* Model terminate function */
void rtwdemo_configrpininterface_terminate(void)
{
    /* (no terminate code required) */
}

```

This code first perturbs the input signal `input2` by incrementing the value of the signal each time the `step` function executes. The code then uses the built-in macro `rtmGetDataMapInfo` to extract model mapping information from the model data structure `rtwdemo_configrpininterface_M`. The pointer `MMI` points to the extracted mapping information, which allows the functions `tuneFcn` and `logFcn` to access the information contained in the arrays of structures that the C API file `rtwdemo_configrpininterface_capi.c` defines.

13. View the function `tuneFcn` in the file `ex_myHandCode.c`. This function uses the C API (through the model mapping information `mmi`) and a pointer to the simulation time to print the value of the parameter `K1` at specific times during code execution. When the simulation time reaches 5 seconds, the function changes the parameter value in memory. By using a `switch case` block, the function can access the parameter data whether the data type is `int8` or `double`.

14. View the code algorithm in the model `step` function again. Near the end of the function, the algorithm executes the code that you specified in the System Outputs block. This code calls the function `logFcn`.

```

rtwdemodbtype(file, '/* user code (Output function Trailer) */', ...
    '/* Matfile logging */', 1, 0)

/* user code (Output function Trailer) */

/* System '<Root>' */
{
    rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo
    (rtwdemo_configrpininterface_M).mmi);
    logFcn(MMI, rtmGetTPtr(rtwdemo_configrpininterface_M));
}

```

15. View the function `logFcn` in the file `ex_myHandCode.c`. The function uses the C API to print the value of the test-pointed signal. The function can access the signal data whether the data type is `single` or `double`.

16. At the command prompt, run the generated executable `rtwdemo_configrpinterface.exe`.

```
system('rtwdemo_configrpinterface')
```

The parameter and signal values appear in the Command Window output.

For more information about data interfaces, including the C API, see “Generated Code Interfacing”.

Configure Data Accessibility

Goal	Considerations and More Information
Configure signals as accessible and parameters as tunable by default	<p>Clear the model configuration parameter Signal storage reuse and set the configuration parameter Default parameter behavior to <code>Tunable</code>. These settings prevent the elimination of storage for signals and prevent parameter inlining. Each block parameter and signal line appears in the generated code as a field of a structure. For more information about these data structures, see “How Generated Code Exchanges Data with an Environment” on page 15-35 and “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49.</p> <p>For more information about Default parameter behavior, see <code>Default parameter behavior</code>. For more information about optimizations that eliminate storage for signals, see “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49 and “Minimize Computations and Storage for Intermediate Results at Block Outputs” on page 40-35.</p>
Preserve accessibility and tunability of a data item after you select optimizations	<p>You can generate more efficient code by selecting optimizations such as Signal storage reuse, but the code generator eliminates storage for as many data items as possible. To exclude individual data items from the optimizations:</p> <ul style="list-style-type: none"> • Configure signals in the Code Mappings editor. In the editor you can set the default configuration for signals, add signals of interest to the code mappings, and then set the storage class for individual signals to the default storage class or another storage class. Signals can be named by setting the <code>Identifier</code> storage class property. • Apply a storage class, such as <code>ExportedGlobal</code>, to states and parameters. See “C Code Generation Configuration for Model Interface Elements” on page 16-2.
Represent a data item as a separate global variable in the generated code	<p>When you disable optimizations, signal lines, block states, and parameters appear in the generated code as fields of structures. You cannot control the names of the structures without Embedded Coder. To instead store a data item in a separate global variable whose name, file placement, and other characteristics you can control, apply a storage class to a signal, state, or <code>Simulink.Parameter</code> object. See “C Code Generation Configuration for Model Interface Elements” on page 16-2.</p>

Goal	Considerations and More Information
Generate a standardized C-code interface for accessing data	You can configure the generated code to contain extra code and files so that you can access model data through standardized interfaces. For more information, see “Exchange Data Between Generated and External Code Using C API” on page 34-2.
Tune parameters and monitor signals during external mode simulation	<p>When you generate code and an external executable from a model, you can simulate the model in external mode to communicate with the running executable. You can tune parameters and monitor signals during the simulation. However, in this simulation mode, parameter tunability limitations that apply to code generation also apply to the simulation. For information about the code generation limitations, see “Limitations for Block Parameter Tunability in Generated Code” on page 15-122.</p> <p>For information about external mode, see “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” on page 32-2.</p>
Tune parameters and monitor signals with Simulink Real-Time	If you have Simulink Real-Time, you can tune parameters and monitor signals during execution of your real-time application. Make signals accessible and parameters tunable by clearing optimizations and applying test points and storage classes. See “Parameter Tuning with Simulink Real-Time Explorer” (Simulink Real-Time).

Limitations

For information about limitations that apply to the tunability of parameters in the generated code, see “Limitations for Block Parameter Tunability in Generated Code” on page 15-122.

See Also

Related Examples

- “Deploy Algorithm Model for Real-Time Rapid Prototyping” on page 28-2
- “Access Signal, State, and Parameter Data During Execution” on page 15-17
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49
- “Create Tunable Calibration Parameter in the Generated Code” on page 15-112
- “Switch Between Sets of Parameter Values During Simulation and Code Execution” on page 15-162

Access Signal, State, and Parameter Data During Execution

As you iteratively develop a model, you capture output signal and state data that model execution generates. You also tune parameter values during execution to observe results on the outputs. You can then base your design decisions upon analysis of these outputs. To access this signal, state, and parameter data in a rapid prototyping environment, you can configure the generated code to store the data in addressable memory.

By default, optimization settings make the generated code more efficient by eliminating unnecessary signal storage and inlining the numeric values of block parameters. To generate code that instead allocates addressable memory for this data, you can disable the optimizations or specify code generation settings for individual data items.

Explore Example Model

Run a script that prepares the model `rtwdemo_configrpinterface` for this example.

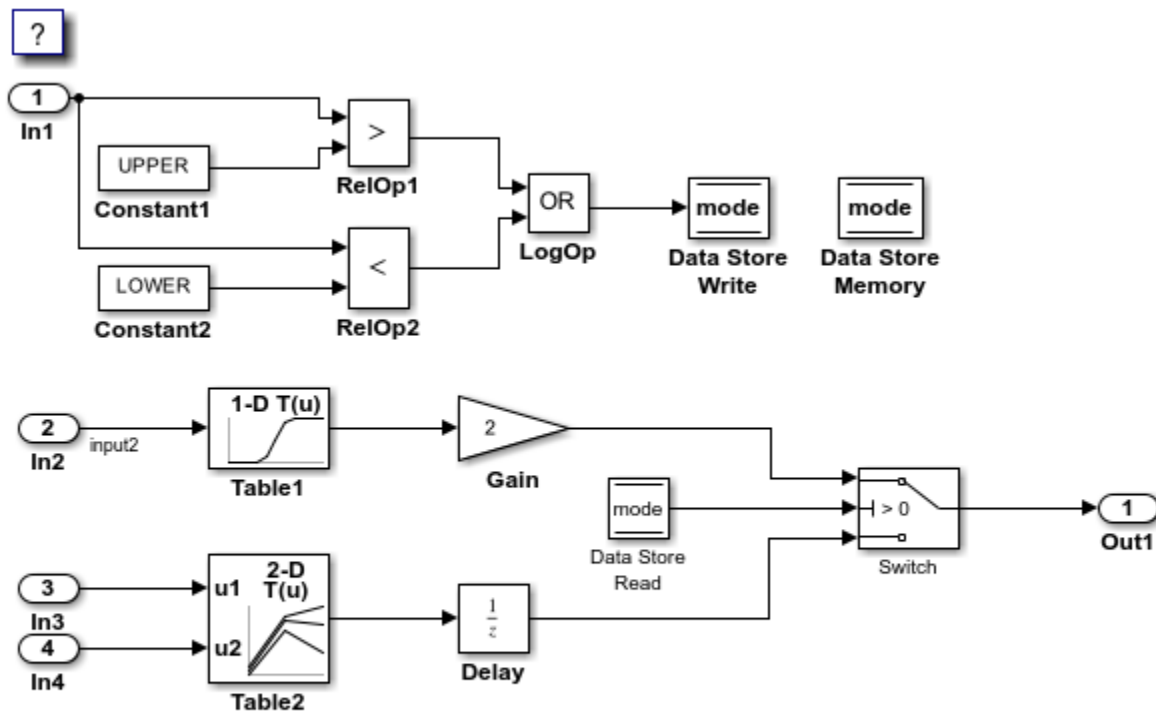
```
prepare_rtwdemo_configrpinterface
```

This data is configured for code generation:

- Parameters UPPER, LOWER, K1, Table1, and Table2
- Signals In1, In2, In3, In4, and Out1
- States X (delay) and mode (data store memory write)

Open the example model, `rtwdemo_configrpinterface`.

```
rtwdemo_configrpinterface
```



Copyright 1994-2020 The MathWorks, Inc.

The model loads numeric MATLAB® variables into the model workspace. The workspace variables set some block parameters in the model. However, the Gain block in the model uses the literal value 2.

Disable Optimizations

1. In the model, clear the model configuration parameter **Signal storage reuse**. When you clear this optimization and other optimizations such as **Eliminate superfluous local variables (expression folding)**, the generated code allocates memory for signal lines. Clearing **Signal storage reuse** disables most of the other optimizations.

```
set_param('rtwdemo_configrpinterface', 'OptimizeBlockIOStorage', 'off')
```

2. Set the model configuration parameter **Default parameter behavior** to Tunable. When set to Tunable, this configuration parameter causes the generated code to allocate memory for block parameters and workspace variables.

```
set_param('rtwdemo_configrpinterface', 'DefaultParameterBehavior', 'Tunable')
```

3. Generate code.

```
slbuild('rtwdemo_configrpinterface')
```

```
### Starting build procedure for: rtwdemo_configrpinterface
### Successful completion of build procedure for: rtwdemo_configrpinterface
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_configrpinterface	Code generated and compiled.	Code generation information file does not exist

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 37.816s

4. In the code generation report, view the file `rtwdemo_configrpinterface.h`. This header file defines a structure type that contains signal data. The structure contains fields that each represent a signal line in the model. For example, the output signal of the Gain block, whose name is Gain, appears as the field `Gain`.

```
file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface.h');
rtwdemodbtype(file,'/* Block signals (default storage) */',...
    'B_rtwdemo_configrpinterface_T;',1,1)
```

```
/* Block signals (default storage) */
typedef struct {
    real_T Delay; /* '<Root>/Delay' */
    real_T Table2; /* '<Root>/Table2' */
    real_T Table1; /* '<Root>/Table1' */
    real_T Gain; /* '<Root>/Gain' */
    boolean_T RelOp1; /* '<Root>/RelOp1' */
    boolean_T RelOp2; /* '<Root>/RelOp2' */
    boolean_T LogOp; /* '<Root>/LogOp' */
    boolean_T DataStoreRead; /* '<Root>/Data Store Read' */
} B_rtwdemo_configrpinterface_T;
```

The file defines a structure type that contains block parameter data. For example, the **Gain** parameter of the Gain block appears as the field `Gain_Gain`. The other fields of the structure represent other block parameters and workspace variables from the model, including initial conditions for signals and states.

```
rtwdemodbtype(file,'/* Parameters (default storage) */',...
    '/* Real-time Model Data Structure */',1,0)

/* Parameters (default storage) */
struct P_rtwdemo_configrpinterface_T_ {
    Table1_Type Table1; /* Variable: Table1
                       * Referenced by: '<Root>/Table1'
                       */
    Table2_Type Table2; /* Variable: Table2
                       * Referenced by: '<Root>/Table2'
                       */
    real_T LOWER; /* Variable: LOWER
                 * Referenced by: '<Root>/Constant2'
                 */
    real_T UPPER; /* Variable: UPPER
                 * Referenced by: '<Root>/Constant1'
                 */
    real_T Gain_Gain; /* Expression: 2
                    * Referenced by: '<Root>/Gain'
                    */
};
```

```

real_T Delay_InitialCondition;      /* Expression: 0
                                   * Referenced by: '<Root>/Delay'
                                   */
uint32_T Table2_maxIndex[2];      /* Computed Parameter: Table2_maxIndex
                                   * Referenced by: '<Root>/Table2'
                                   */
boolean_T DataStoreMemory_InitialValue;
                                   /* Computed Parameter: DataStoreMemory_InitialValue
                                   * Referenced by: '<Root>/Data Store Memory'
                                   */
};

```

5. View the file `rtwdemo_configrpinterface_data.c`. This source file allocates global memory for a parameter structure and initializes the field values based on the parameter values in the model.

6. View the source file `rtwdemo_configrpinterface.c`. The code allocates global memory for a structure variable that contains signal data.

```

file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface.c');
rtwdemodbtype(file, /* Block signals (default storage) */ , ...
    'B_rtwdemo_configrpinterface_T rtwdemo_configrpinterface_B;', 1, 1)

/* Block signals (default storage) */
B_rtwdemo_configrpinterface_T rtwdemo_configrpinterface_B;

```

The code algorithm in the model `step` function calculates the signal values. It then assigns these values to the fields of the signal structure. To perform the calculations, the algorithm uses the parameter values from the fields of the parameter structure.

Exclude Data Items from Optimizations

When you want to select code generation optimizations such as **Signal storage reuse**, you can preserve individual data items from the optimizations. The generated code then allocates addressable memory for the items.

Select the optimizations that you previously cleared.

```

set_param('rtwdemo_configrpinterface', 'OptimizeBlockIOStorage', 'on')
set_param('rtwdemo_configrpinterface', 'LocalBlockOutputs', 'on')
set_param('rtwdemo_configrpinterface', 'DefaultParameterBehavior', 'Inlined')

```

1. In the **C Code** tab, select **Code Interface > Individual Element Code Mappings**.
2. In the Code Mappings editor, inspect the **Signals/States** tab.
3. In the model, select the output signal of the Gain block.
4. Pause on the ellipsis that appears above or below the signal line to open the action bar. Click the **Add Signal** button. In the Code Mappings editor, the **Signals** node expands and lists the signal that you added.

```

cm = coder.mapping.api.get('rtwdemo_configrpinterface');
gain_ports = get_param('rtwdemo_configrpinterface/Gain', 'PortHandles');
gain_outPort = gain_ports.Outputport;
addSignal(cm, gain_outPort);

```

5. Apply the default storage class for Outports to the signal by setting **Storage Class** to Model default.

```
setSignal(cm,gain_outPort, 'StorageClass', 'Model default', 'Identifier', 'Gain');
```

6. In the model, select the Gain block.

7. In the Property Inspector, set the gain value to the model workspace parameter object K1.

8. In the Code Mappings editor, on the **Parameters** tab, expand **Model Parameters**.

9. Select parameter K1. Apply a storage class other than Auto by setting **Storage class**. For example, use the storage class Model default. By using Model default, K1 acquires the default storage class specified for model parameters, Default, and appears as a field of the global parameters structure.

```
set_param('rtwdemo_configrpinterface/Gain', 'Gain', 'K1');
setModelParameter(cm, 'K1', 'StorageClass', 'Model default');
```

10. Generate code.

```
slbuild('rtwdemo_configrpinterface')

### Starting build procedure for: rtwdemo_configrpinterface
### Successful completion of build procedure for: rtwdemo_configrpinterface
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_configrpinterface	Code generated and compiled.	Generated code was out of date.

1 of 1 models built (0 models already up to date)

Build duration: 0h 0m 15.563s

11. In the code generation report, view the file `rtwdemo_configrpinterface.h`. The structure that contains signal data now defines only one field, Gain, which represents the test-pointed output of the Gain block.

```
file = fullfile('rtwdemo_configrpinterface_grt_rtw', 'rtwdemo_configrpinterface.h');
rtwdemodbtype(file, /* Block signals (default storage) */', ...
    'B_rtwdemo_configrpinterface_T;', 1, 1)
```

```
/* Block signals (default storage) */
typedef struct {
    real_T Gain; /* '<Root>/Gain' */
} B_rtwdemo_configrpinterface_T;
```

The structure that contains block parameter data defines one field, K1, which represents the parameter object K1.

```
rtwdemodbtype(file, /* Parameters (default storage) */', ...
    /* Real-time Model Data Structure */', 1, 0)
```

```
/* Parameters (default storage) */
```

```

struct P_rtwdemo_configrpininterface_T_ {
    int8_T K1; /* Variable: K1
               * Referenced by: '<Root>/Gain'
               */
};

```

Access Data Through Generated Interfaces

You can configure the generated code to contain extra code and files so that you can access model data through standardized interfaces. For example, use the C API to log signal data and tune parameters during execution.

Copy this custom source code into a file named `ex_myHandCode.c` in your current folder.

```

#include "ex_myHandHdr.h"

#define paramIdx 0 /* Index of the target parameter,
determined by inspecting the array of structures generated by the C API. */
#define sigIdx 0 /* Index of the target signal,
determined by inspecting the array of structures generated by the C API. */

void tuneFcn(rtwCAPI_ModelMappingInfo *mmi, time_T *tPtr)
{
    /* Take action with the parameter value only at
    the beginning of simulation and at the 5-second mark. */
    if (*tPtr == 0 || *tPtr == 5) {

        /* Local variables to store information extracted from
        the model mapping information (mmi). */
        void** dataAddrMap;
        const rtwCAPI_DataTypeMap *dataTypeMap;
        const rtwCAPI_ModelParameters *params;
        int_T addrIdx;
        uint16_T dTypeIdx;
        uint8_T slDataType;

        /* Use built-in C API macros to extract information. */
        dataAddrMap = rtwCAPI_GetDataAddressMap(mmi);
        dataTypeMap = rtwCAPI_GetDataTypeMap(mmi);
        params = rtwCAPI_GetModelParameters(mmi);
        addrIdx = rtwCAPI_GetModelParameterAddrIdx(params,paramIdx);
        dTypeIdx = rtwCAPI_GetModelParameterDataTypeIdx(params,paramIdx);
        slDataType = rtwCAPI_GetDataTypeSLId(dataTypeMap, dTypeIdx);

        /* Handle data types 'double' and 'int8'. */
        switch (slDataType) {

            case SS_DOUBLE: {
                real_T* dataAddress;
                dataAddress = dataAddrMap[addrIdx];
                /* At the 5-second mark, increment the parameter value by 1. */
                if (*tPtr == 5) {
                    (*dataAddress)++;
                }
                printf("Parameter value is %f\n", *dataAddress);
                break;
            }
        }
    }
}

```

```

        case SS_INT8: {
            int8_T* dataAddress;
            dataAddress = dataAddrMap[addrIdx];
            if (*tPtr == 5) {
                (*dataAddress)++;
            }
            printf("Parameter value is %i\n", *dataAddress);
            break;
        }
    }
}

void logFcn(rtwCAPI_ModelMappingInfo *mmi, time_T *tPtr)
{
    /* Take action with the signal value only when
       the simulation time is an integer value. */
    if (*tPtr-(int_T)*tPtr == 0) {

        /* Local variables to store information extracted from
           the model mapping information (mmi). */
        void** dataAddrMap;
        const rtwCAPI_DataTypeMap *dataTypeMap;
        const rtwCAPI_Signals *sigs;
        int_T addrIdx;
        uint16_T dTypeIdx;
        uint8_T slDataType;

        /* Use built-in C API macros to extract information. */
        dataAddrMap = rtwCAPI_GetDataAddressMap(mmi);
        dataTypeMap = rtwCAPI_GetDataTypeMap(mmi);
        sigs = rtwCAPI_GetSignals(mmi);
        addrIdx = rtwCAPI_GetSignalAddrIdx(sigs, sigIdx);
        dTypeIdx = rtwCAPI_GetSignalDataTypeIdx(sigs, sigIdx);
        slDataType = rtwCAPI_GetDataTypeSLId(dataTypeMap, dTypeIdx);

        /* Handle data types 'double' and 'single'. */
        switch (slDataType) {

            case SS_DOUBLE: {
                real_T* dataAddress;
                dataAddress = dataAddrMap[addrIdx];
                printf("Signal value is %f\n", *dataAddress);
                break;
            }

            case SS_SINGLE: {
                real32_T* dataAddress;
                dataAddress = dataAddrMap[addrIdx];
                printf("Signal value is %f\n", *dataAddress);
                break;
            }
        }
    }
}

```

2. Copy this custom header code into a file named `ex_myHandHdr.h` in your current folder.

```

#include <stdio.h>
#include <string.h>
#include <math.h>
/* Include rtw_modelmap.h for definitions of C API macros. */
#include "rtw_modelmap.h"
#include "builtin_typeid_types.h"
#include "rtwtypes.h"
void tuneFcn(rtwCAPI_ModelMappingInfo *mmi, time_T *tPtr);
void logFcn(rtwCAPI_ModelMappingInfo *mmi, time_T *tPtr);

```

These files use the C API to access signal and parameter data in the code that you generate from the example model.

3. Set model configuration parameters **Include headers** and **Source files** to #include "ex_myHandHdr.h" and ex_myHeaderCode.c, respectively.

```

set_param('rtwdemo_configrpinterface', 'CustomHeaderCode', '#include "ex_myHandHdr.h"')
set_param('rtwdemo_configrpinterface', 'CustomSource', 'ex_myHeaderCode.c')

```

4. Select model configuration parameter **MAT-file Logging**. The generated executable runs only until the simulation stop time (which you set in the model configuration parameters).

```

set_param('rtwdemo_configrpinterface', 'MatFileLogging', 'on')

```

5. Select C API model configuration parameters **parameters**, **signals**, **states**, and **root-level I/O**.

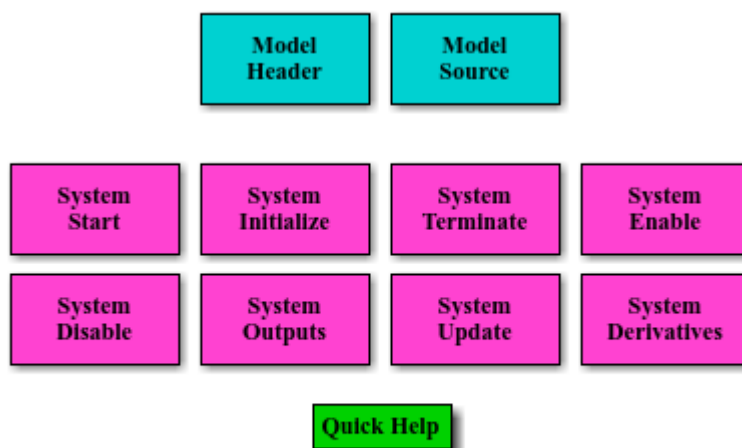
```

set_param('rtwdemo_configrpinterface', 'RTWCAPIParams', 'on')
set_param('rtwdemo_configrpinterface', 'RTWCAPISignals', 'on')
set_param('rtwdemo_configrpinterface', 'RTWCAPIStates', 'on')
set_param('rtwdemo_configrpinterface', 'RTWCAPIRootIO', 'on')

```

6. Load the Custom Code block library.

custcode



These blocks allow you to insert custom code into specific files and functions.



7. Add a System Outputs block to the model.

```
add_block('custcode/System Outputs','rtwdemo_configrpinterface/System Outputs')
```

8. In the System Outputs block dialog box, set **System Outputs Function Execution Code** to this code:

```
{
rtwdemo_configrpinterface_U.input2++;
rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo(rtwdemo_configrpinterface_M).mmi);
tuneFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));
}
```

9. In the block dialog box, set **System Outputs Function Exit Code** to this code:

```
{
rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo(rtwdemo_configrpinterface_M).mmi);
logFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));
}
```

Alternatively, to configure the System Outputs block, at the command prompt, use these commands:

```
temp.TLCFile = 'custcode';
temp.Location = 'System Outputs Function';
temp.Middle = sprintf(['\nrtwdemo_configrpinterface_U.input2++;'...
'\nrtwCAPI_ModelMappingInfo *MMI = '...
'\nrtmGetDataMapInfo(rtwdemo_configrpinterface_M).mmi);'...
'\ntuneFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));\n']);
temp.Bottom = sprintf(['\nrtwCAPI_ModelMappingInfo *MMI = '...
'\nrtmGetDataMapInfo(rtwdemo_configrpinterface_M).mmi);'...
'\nlogFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));\n']);
set_param('rtwdemo_configrpinterface/System Outputs','RTWdata',temp)
```

10. Generate code.

```
slbuild('rtwdemo_configrpinterface')

### Starting build procedure for: rtwdemo_configrpinterface
### Successful completion of build procedure for: rtwdemo_configrpinterface

Build Summary

Top model targets built:

Model                Action                Rebuild Reason
=====
rtwdemo_configrpinterface  Code generated and compiled.  Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 16.705s
```

11. In the code generation report, view the interface file `rtwdemo_configrpinterface_capi.c`. This file initializes the arrays of structures that you can use to interact with data items through the C API. For example, in the array of structures `rtBlockSignals`, the first structure (index 0) describes the test-pointed output signal of the Gain block in the model.

```
file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface_capi.c');
rtwdemodbtype(file,'/* Block output signal information */',...
'\n/* Individual block tuning',1,0)

/* Block output signal information */
```

```

static const rtwCAPI_Signals rtBlockSignals[] = {
  /* addrMapIndex, sysNum, blockPath,
   * signalName, portNumber, dataTypeIndex, dimIndex, fxpIndex, sTimeIndex
   */
  { 0, 0, TARGET_STRING("rtwdemo_configrpinterface/Gain"),
    TARGET_STRING(""), 0, 0, 0, 0, 0 },

  {
    0, 0, (NULL), (NULL), 0, 0, 0, 0, 0
  }
};

```

The fields of the structure, such as `addrMapIndex`, indicate indices into other arrays of structures, such as `rtDataAddrMap`, that describe the characteristics of the signal. These characteristics include the address of the signal data (a pointer to the data), the numeric data type, and the dimensions of the signal.

12. In the file `rtwdemo_configrpinterface.c`, view the code algorithm in the model step function. The algorithm first executes the code that you specified in the System Outputs block.

```

file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface.c');
rtwdemodbtype(file,'/* user code (Output function Body) */',...
  '/* Logic: '<Root>/LogOp' incorporates:',1,0)

/* user code (Output function Body) */

/* System '<Root>' */
{
  rtwdemo_configrpinterface_U.input2++;
  rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo
    (rtwdemo_configrpinterface_M).mmi);
  tuneFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));
}

/* DataStoreWrite: '<Root>/Data Store Write' incorporates:
 * Constant: '<Root>/Constant1'
 * Constant: '<Root>/Constant2'
 * Inport: '<Root>/In1'
 * Logic: '<Root>/LogOp'
 * RelationalOperator: '<Root>/RelOp1'
 * RelationalOperator: '<Root>/RelOp2'
 */
mode = ((rtwdemo_configrpinterface_U.In1 > 10.0) ||
  (rtwdemo_configrpinterface_U.In1 < -10.0));

/* Gain: '<Root>/Gain' incorporates:
 * Inport: '<Root>/In2'
 * Lookup_n-D: '<Root>/Table1'
 */
rtwdemo_configrpinterface_B.Gain = (real_T)rtwdemo_configrpinterface_P.K1 *
  look1_binlc(rtwdemo_configrpinterface_U.input2, rtCP_Table1_bp01Data,
    rtCP_Table1_tableData, 10U);

/* Switch: '<Root>/Switch' incorporates:
 * DataStoreRead: '<Root>/Data Store Read'
 */
if (mode) {

```



```

    /* Output: '<Root>/Out1' */
    output = rtwdemo_configrpinterface_B.Gain;
} else {
    /* Output: '<Root>/Out1' incorporates:
    * UnitDelay: '<Root>/Delay'
    */
    output = rtwdemo_configrpinterface_DW.X;
}

/* End of Switch: '<Root>/Switch' */

/* Lookup_n-D: '<Root>/Table2' incorporates:
* Inport: '<Root>/In3'
* Inport: '<Root>/In4'
*/
rtwdemo_configrpinterface_DW.X = look2_binlc(rtwdemo_configrpinterface_U.In3,
    rtwdemo_configrpinterface_U.In4, rtCP_Table2_bp01Data, rtCP_Table2_bp02Data,
    rtCP_Table2_tableData, rtCP_Table2_maxIndex, 3U);

/* user code (Output function Trailer) */

/* System '<Root>' */
{
    rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo
        (rtwdemo_configrpinterface_M).mmi);
    logFcn(MMI, rtmGetTPtr(rtwdemo_configrpinterface_M));
}

/* Matfile logging */
rt_UpdateTXYLogVars(rtwdemo_configrpinterface_M->rtwLogInfo,
    (&rtwdemo_configrpinterface_M->Timing.taskTime0));

/* signal main to stop simulation */
{
    /* Sample time: [1.0s, 0.0s] */
    if ((rtmGetTFinal(rtwdemo_configrpinterface_M)!=-1) &&
        !((rtmGetTFinal(rtwdemo_configrpinterface_M)-
            rtwdemo_configrpinterface_M->Timing.taskTime0) >
            rtwdemo_configrpinterface_M->Timing.taskTime0 * (DBL_EPSILON))) {
        rtmSetErrorStatus(rtwdemo_configrpinterface_M, "Simulation finished");
    }
}

/* Update absolute time for base rate */
/* The "clockTick0" counts the number of times the code of this task has
* been executed. The absolute time is the multiplication of "clockTick0"
* and "Timing.stepSize0". Size of "clockTick0" ensures timer will not
* overflow during the application lifespan selected.
* Timer of this task consists of two 32 bit unsigned integers.
* The two integers represent the low bits Timing.clockTick0 and the high bits
* Timing.clockTickH0. When the low bit overflows to 0, the high bits increment.
*/
if (!(++rtwdemo_configrpinterface_M->Timing.clockTick0)) {
    ++rtwdemo_configrpinterface_M->Timing.clockTickH0;
}

rtwdemo_configrpinterface_M->Timing.taskTime0 =
    rtwdemo_configrpinterface_M->Timing.clockTick0 *
    rtwdemo_configrpinterface_M->Timing.stepSize0 +

```

```

    rtwdemo_configrpinterface_M->Timing.clockTickH0 *
    rtwdemo_configrpinterface_M->Timing.stepSize0 * 4294967296.0;
}

/* Model initialize function */
void rtwdemo_configrpinterface_initialize(void)
{
    /* Registration code */

    /* initialize non-finites */
    rt_InitInfAndNaN(sizeof(real_T));

    /* initialize real-time model */
    (void) memset((void *)rtwdemo_configrpinterface_M, 0,
        sizeof(RT_MODEL_rtwdemo_configrpinte_T));
    rtmSetTFinal(rtwdemo_configrpinterface_M, 10.0);
    rtwdemo_configrpinterface_M->Timing.stepSize0 = 1.0;

    /* Setup for data logging */
    {
        static RTWLogInfo rt_DataLoggingInfo;
        rt_DataLoggingInfo.loggingInterval = (NULL);
        rtwdemo_configrpinterface_M->rtwLogInfo = &rt_DataLoggingInfo;
    }

    /* Setup for data logging */
    {
        rtliSetLogXSignalInfo(rtwdemo_configrpinterface_M->rtwLogInfo, (NULL));
        rtliSetLogXSignalPtrs(rtwdemo_configrpinterface_M->rtwLogInfo, (NULL));
        rtliSetLogT(rtwdemo_configrpinterface_M->rtwLogInfo, "");
        rtliSetLogX(rtwdemo_configrpinterface_M->rtwLogInfo, "");
        rtliSetLogXFinal(rtwdemo_configrpinterface_M->rtwLogInfo, "");
        rtliSetLogVarNameModifier(rtwdemo_configrpinterface_M->rtwLogInfo, "rt_");
        rtliSetLogFormat(rtwdemo_configrpinterface_M->rtwLogInfo, 0);
        rtliSetLogMaxRows(rtwdemo_configrpinterface_M->rtwLogInfo, 1000);
        rtliSetLogDecimation(rtwdemo_configrpinterface_M->rtwLogInfo, 1);
        rtliSetLogY(rtwdemo_configrpinterface_M->rtwLogInfo, "");
        rtliSetLogYSignalInfo(rtwdemo_configrpinterface_M->rtwLogInfo, (NULL));
        rtliSetLogYSignalPtrs(rtwdemo_configrpinterface_M->rtwLogInfo, (NULL));
    }

    /* block I/O */
    (void) memset(((void *) &rtwdemo_configrpinterface_B), 0,
        sizeof(B_rtwdemo_configrpinterface_T));

    /* Storage classes */
    output = 0.0;

    /* states (dwork) */
    (void) memset((void *)&rtwdemo_configrpinterface_DW, 0,
        sizeof(DW_rtwdemo_configrpinterface_T));

    /* exported global states */
    mode = false;

    /* external inputs */
    (void)memset(&rtwdemo_configrpinterface_U, 0, sizeof
        (ExtU_rtwdemo_configrpinterfac_T));
}

```

```

/* Initialize DataMapInfo substructure containing ModelMap for C API */
rtwdemo_configrpininterface_InitializeDataMapInfo();

/* Matfile logging */
rt_StartDataLoggingWithStartTime(rtwdemo_configrpininterface_M->rtwLogInfo, 0.0,
    rtmGetTFinal(rtwdemo_configrpininterface_M),
    rtwdemo_configrpininterface_M->Timing.stepSize0, (&rtmGetErrorStatus
    (rtwdemo_configrpininterface_M)));

/* Start for DataStoreMemory: '<Root>/Data Store Memory' */
mode = false;

/* InitializeConditions for Lookup_n-D: '<Root>/Table2' incorporates:
 * UnitDelay: '<Root>/Delay'
 */
rtwdemo_configrpininterface_DW.X = 0.0;
}

/* Model terminate function */
void rtwdemo_configrpininterface_terminate(void)
{
    /* (no terminate code required) */
}

```

This code first perturbs the input signal `input2` by incrementing the value of the signal each time the `step` function executes. The code then uses the built-in macro `rtmGetDataMapInfo` to extract model mapping information from the model data structure `rtwdemo_configrpininterface_M`. The pointer `MMI` points to the extracted mapping information, which allows the functions `tuneFcn` and `logFcn` to access the information contained in the arrays of structures that the C API file `rtwdemo_configrpininterface_capi.c` defines.

13. View the function `tuneFcn` in the file `ex_myHandCode.c`. This function uses the C API (through the model mapping information `mmi`) and a pointer to the simulation time to print the value of the parameter `K1` at specific times during code execution. When the simulation time reaches 5 seconds, the function changes the parameter value in memory. By using a `switch case` block, the function can access the parameter data whether the data type is `int8` or `double`.

14. View the code algorithm in the model `step` function again. Near the end of the function, the algorithm executes the code that you specified in the System Outputs block. This code calls the function `logFcn`.

```

rtwdemodbtype(file, '/* user code (Output function Trailer) */', ...
    '/* Matfile logging */', 1, 0)

/* user code (Output function Trailer) */

/* System '<Root>' */
{
    rtwCAPI_ModelMappingInfo *MMI = (&rtmGetDataMapInfo
    (rtwdemo_configrpininterface_M).mmi);
    logFcn(MMI, rtmGetTPtr(rtwdemo_configrpininterface_M));
}

```

15. View the function `logFcn` in the file `ex_myHandCode.c`. The function uses the C API to print the value of the test-pointed signal. The function can access the signal data whether the data type is `single` or `double`.

16. At the command prompt, run the generated executable `rtwdemo_configrpinterface.exe`.

```
system('rtwdemo_configrpinterface')
```

The parameter and signal values appear in the Command Window output.

For more information about data interfaces, including the C API, see “Generated Code Interfacing”.

See Also

[Simulink.Parameter](#) | [Simulink.Signal](#) | [Simulink.Signal](#) | [System Outputs](#)

Related Examples

- “Data Structures in the Generated Code” on page 15-31
- “Preserve Variables in Generated Code” on page 15-2
- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Exchange Data Between Generated and External Code Using C API” on page 34-2

Data Structures in the Generated Code

The code generator organizes the data used by the model in data structures. For each model, the code generator stores data, such as signals and states, in global structures that are also referred to as standard data structures. Use these data structures to access the data in the generated code.

Real-time model (`rtModel`) data structure is a top level data structure that contains instance-specific data, which is used by the model for reusable functions. The code generator uses this data structure for only multi-instance code generation.

Standard Data Structures

In a model, root-level Inport and Outport blocks represent the primary inputs and outputs of the block algorithm. By default, the code generator aggregates these blocks into standard structures that store input and output data.

Signal lines, block parameters, states, and other model data appear in the generated code as fields of standard structures. For general information, see “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49.

The table shows some common data structures in the generated code. The name of the structure type and, if applicable, structure variable in the code that you generate from a model depends on the model settings. Regardless of the settings, the name of the structure type in the code contains the short name from the **Short Name of Structure Type** column in the table.

Data Structures Generated for a Model

Short Name of Structure Type	Data Represented in the Structure
ExtU	Data from root Inport blocks
ExtY	Data from root Outport blocks
B	Block output signals
ConstB	Block outputs that have constant values
P	Block parameters
ConstP	Constant parameters
DW	Discrete block states
XDis	Status of enabled subsystems
InstP	Parameter arguments
SharedDSM	Shared local data stores

You can exclude data from appearing in these structures by using:

- Direct application of storage classes. For example, you can use storage classes to represent signals, tunable parameters, and states as individual global variables. For more information, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.
- Configuration parameters, such as those on the **Optimization** pane in the Configuration Parameters dialog box. You can adjust these configuration parameters to control the default representation of data. For more information, see “Model Configuration Parameters: Code Generation Optimization”.

Real-Time Model Data Structure

To access the fields of an S-function's simulation data structure (`SimStruct`), the Simulink software provides a set of functions. The code generator replaces `SimStruct` with an optimized real-time model (`rtModel`) data structure.

Code generation by using the GRT and ERT system target files uses the real-time model data structure (`rtModel`). This structure encapsulates model-specific information in a much more compact form than the simulation structure `SimStruct`. Depending on the model, the model-specific information includes data such as internal data for reusable functions and error state.

When you set **Code interface packaging** to `Reusable function`, the code generator packages model data (such as block I/O, Dwork, and parameters) into the real-time model data structure. The code generator passes the real-time model structure by reference to generated model entry-point functions. If you set **Pass root-level I/O as** to `Part of model data structure`, the code generator packages root-level model input and output into the real-time model data structure.

Efficient capabilities of the generated code depend on generation of `rtModel` rather than `SimStruct`, including:

- Integer absolute and elapsed timing services
- Independent timers for asynchronous tasks
- Generation of improved C API code for signal, state, and parameter monitoring
- Pruning the data structure to minimize its size (ERT-derived system target files only)

The real-time model data structure is defined in the `main.c` or `main.cpp` file that is generated when you build your model. For ERT models, if you clear the model configuration parameter **Generate an example main program**, `rtmodel.h` file is generated. The `rtmodel.h` file contains `#include` directives required by the `rt_main.c` or `rt_cppclass_main.cpp` static main program module. The `rt_main.c` or `rt_cppclass_main.cpp` includes `rtmodel.h` to access model-specific data structures and entry points because the static main program module is not created at code generation time.

The code generator provides a set of macros for accessing the `rtModel` data structure. These macros are analogous to the `ssSetxxx` and `ssGetxxx` macros that S-functions use to access `SimStruct` data, including noninlined S-functions compiled by the code generator.

Use the set of macros `rtmGetxxx` and `rtmSetxxx` to access the real-time model data structure. These macros are used in the generated code as well as in the `main.c` or `main.cpp` module of your external code. If you are customizing `main.c` or `main.cpp` (either a static or a generated file), use `rtmGetxxx` and `rtmSetxxx` instead of the `ssSetxxx` and `ssGetxxx` macros.

Usage of `rtmGetxxx` and `rtmSetxxx` macros is the same as for the `ssSetxxx` and `ssGetxxx` versions, except that you replace `SimStruct S` by real-time model data structure `rtM`. This table lists `rtmGetxxx` and `rtmSetxxx` macros that are used in `rt_main.c` and `rt_main.cpp`.

Macros for Accessing the Real-Time Model Data Structure

rtm Macro Syntax	Description
<code>rtmGetdX(rtm)</code>	Get the derivatives of block continuous states
<code>rtmGetOffsetTimePtr(RT_MDL rtm)</code>	Return the pointer to vector that stores sample time offsets of the model associated with rtm
<code>rtmGetNumSampleTimes(RT_MDL rtm)</code>	Get the number of sample times of a block
<code>rtmGetPerTaskSampleHitsPtr(RT_MDL)</code>	Return a pointer to NumSampleTime × NumSampleTime matrix
<code>rtmGetRTWExtModeInfo(RT_MDL rtm)</code>	Return an external mode information data structure of the model (used by code generator only for external mode)
<code>rtmGetRTWLogInfo(RT_MDL)</code>	Return a data structure used by code generator logging (used by code generator only)
<code>rtmGetRTWRTModelMethodsInfo(RT_MDL)</code>	Return a data structure of real-time model methods information (used by code generator only)
<code>rtmGetRTWSolverInfo(RT_MDL)</code>	Return data structure containing model solver information (used by code generator only)
<code>rtmGetSampleHitPtr(RT_MDL)</code>	Return a pointer to Sample Hit flag vector
<code>rtmGetSampleTime(RT_MDL rtm, int TID)</code>	Get task sample time
<code>rtmGetSampleTimePtr(RT_MDL rtm)</code>	Get pointer to a task sample time
<code>rtmGetSampleTimeTaskIDPtr(RT_MDL rtm)</code>	Get pointer to a task ID
<code>rtmGetSimTimeStep(RT_MDL)</code>	Return simulation step type ID (MINOR_TIME_STEP, MAJOR_TIME_STEP)
<code>rtmGetStepSize(RT_MDL)</code>	Return the fundamental step size of the model
<code>rtmGetT(RT_MDL, t)</code>	Get the current simulation time
<code>rtmSetT(RT_MDL, t)</code>	Set the time of the next sample hit
<code>rtmGetTaskTime(RT_MDL, tid)</code>	Get the current time for the current task
<code>rtmGetTFinal(RT_MDL)</code>	Get the simulation stop time
<code>rtmSetTFinal(RT_MDL, finalT)</code>	Set the simulation stop time
<code>rtmGetTimingData(RT_MDL)</code>	Return a data structure used by timing engine of the model (used by code generator only)
<code>rtmGetTPtr(RT_MDL)</code>	Return a pointer to the current time
<code>rtmGetTStart(RT_MDL)</code>	Get the simulation start time
<code>rtmIsContinuousTask(rtm)</code>	Determine whether a task is continuous
<code>rtmIsMajorTimeStep(rtm)</code>	Determine whether the simulation is in a major step
<code>rtmIsSampleHit(RT_MDL, tid)</code>	Determine whether the sample time is hit
<code>rtmGetErrorStatus(rtm)</code>	Get the current error status
<code>rtmSetErrorStatus(rtm, val)</code>	Set the current error status
<code>rtmGetErrorStatusPointer(rtm)</code>	Return a pointer to the current error status
<code>rtmGetStopRequested(rtm)</code>	Return whether a stop is requested

rtm Macro Syntax	Description
<code>rtmGetBlockIO(rtm)</code>	Get the block I/O data structure
<code>rtmSetBlockIO(rtm, val)</code>	Set the block I/O data structure
<code>rtmGetContStates(rtm)</code>	Get the continuous states data structure
<code>rtmSetContStates(rtm, val)</code>	Set the continuous states data structure
<code>rtmGetDefaultParam(rtm)</code>	Get the default parameters data structure
<code>rtmSetDefaultParam(rtm, val)</code>	Set the default parameters data structure
<code>rtmGetPrevZCSigState(rtm)</code>	Get the previous zero-crossing signal state data structure
<code>rtmSetPrevZCSigState(rtm, val)</code>	Set the previous zero-crossing signal state data structure
<code>rtmGetRootDWork(rtm)</code>	Get the DWork data structure
<code>rtmSetRootDWork(rtm, val)</code>	Set the DWork data structure
<code>rtmGetU(rtm)</code>	Get the root inputs data structure (when root inputs are passed as part of the model data structure)
<code>rtmSetU(rtm, val)</code>	Set the root inputs data structure (when root inputs are passed as part of the model data structure)
<code>rtmGetY(rtm)</code>	Get the root outputs data structure (when root outputs are passed as part of the model data structure)
<code>rtmSetY(rtm, val)</code>	Set the root outputs data structure (when root outputs are passed as part of the model data structure)

For more information, see “SimStruct Macros and Functions Listed by Usage”.

See Also

Related Examples

- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49
- “Access Signal, State, and Parameter Data During Execution” on page 15-17
- “C Code Generation Configuration for Model Interface Elements” (Embedded Coder)

How Generated Code Exchanges Data with an Environment

To use the code that you generate from a model, you call the generated entry-point functions such as `step` and `initialize`. The environment in which you call these functions must provide input signal data and, depending on your application, scheduling information. The generated algorithm then calculates output data that the calling environment can use. The environment and the algorithm can exchange this data through global variables or through formal parameters (arguments).

The set of input and output data and the exchange mechanisms constitute the interfaces of the entry-point functions. When you understand the default interfaces and how to control them, you can:

- Write code that calls the generated code.
- Generate reusable (reentrant) code that you can call multiple times in a single application.
- Integrate the generated code with other, external code in your application.

In a model, root-level Inport and Outport blocks represent the primary inputs and outputs of the block algorithm. By default, the code generator aggregates these blocks into standard structures that store input and output data.

For information about how the generated code stores internal data, which does not participate in the model interface, see “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49.

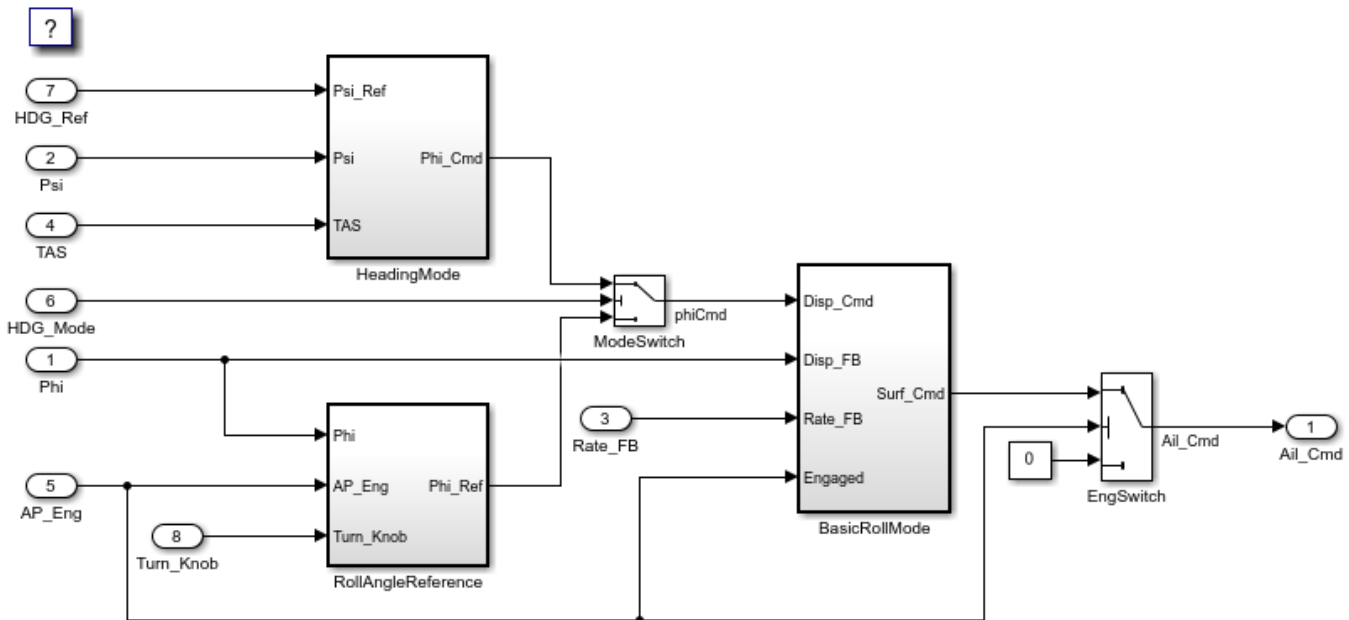
Data Interfaces in the Generated Code

This example shows how the generated code exchanges data with an environment.

Explore Example Model

Open the example model `RollAxisAutopilot`.

```
open_system('RollAxisAutopilot')
```



Copyright 1990-2020 The MathWorks, Inc.

At the root level, the model has a number of Inport blocks and one Outport block.

In the model, set **Configuration Parameters > Code Generation > System target file** to `grt.tlc`.

```
set_param('RollAxisAutopilot','SystemTargetFile','grt.tlc')
```

Inspect the setting for **Configuration Parameters > Code Generation > Interface > Code interface packaging**. The setting `Nonreusable` function means that the generated code is not reusable (not reentrant).

For this example, to generate simpler code, clear the **Configuration Parameters > Code Generation > Interface > Advanced parameters > Mat-file logging**.

```
set_param('RollAxisAutopilot','MatFileLogging','off')
```

Generate Nonreusable Code

Generate code from the model.

```
slbuild('RollAxisAutopilot')
```

```
### Starting build procedure for: RollAxisAutopilot
### Successful completion of build procedure for: RollAxisAutopilot
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
=====		

```
RollAxisAutopilot Code generated and compiled. Code generation information file does not exist
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 12.135s
```

Inspect the file `RollAxisAutopilot.h`. The file defines a structure type that represents input data and a type that represents output data.

```
file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot.h');
rtwdemodbtype(file,...
    /* External inputs (root inport signals with default storage) */,...
    /* ExtY_RollAxisAutopilot_T; ',1,1)

/* External inputs (root inport signals with default storage) */
typedef struct {
    real32_T Phi; /* '<Root>/Phi' */
    real32_T Psi; /* '<Root>/Psi' */
    real32_T Rate_FB; /* '<Root>/Rate_FB' */
    real32_T TAS; /* '<Root>/TAS' */
    boolean_T AP_Eng; /* '<Root>/AP_Eng' */
    boolean_T HDG_Mode; /* '<Root>/HDG_Mode' */
    real32_T HDG_Ref; /* '<Root>/HDG_Ref' */
    real32_T Turn_Knob; /* '<Root>/Turn_Knob' */
} ExtU_RollAxisAutopilot_T;

/* External outputs (root outports fed by signals with default storage) */
typedef struct {
    real32_T Ail_Cmd; /* '<Root>/Ail_Cmd' */
} ExtY_RollAxisAutopilot_T;
```

Each field corresponds to an Inport or Outport block at the root level of the model. The name of each field derives from the name of the block.

The file also defines a structure type, the *real-time model data structure*, whose single field stores a run-time indication of whether the generated code has encountered an error during execution.

```
rtwdemodbtype(file,'/* Real-time Model Data Structure */',...
    /* Block states (default storage) */',1,0)

/* Real-time Model Data Structure */
struct tag_RTM_RollAxisAutopilot_T {
    const char_T *errorStatus;
};
```

When you modify model configuration parameters to suit your application, this structure can also contain other data that pertain to the entire model, such as scheduling information.

For the structure type that represents the real-time model data structure, the file `RollAxisAutopilot_types.h` creates an alias (typedef) that the generated code later uses to allocate memory for the structure.

```
file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot_types.h');
rtwdemodbtype(file,'/* Forward declaration for rtModel */',...
    'RT_MODEL_RollAxisAutopilot_T; ',1,1)
```

```
/* Forward declaration for rtModel */
typedef struct tag_RTM_RollAxisAutopilot_T RT_MODEL_RollAxisAutopilot_T;
```

Using these structure types, the file `RollAxisAutopilot.c` defines (allocates memory for) global structure variables that store input and output data for the generated algorithm. The file also defines variables that represent the real-time model data structure and a pointer to the structure.

```
file = fullfile('RollAxisAutopilot_grt_rtw', 'RollAxisAutopilot.c');
rtwdemodbtype(file,...
    /* External inputs (root inport signals with default storage) */,...
    '= &RollAxisAutopilot_M;',1,1)

/* External inputs (root inport signals with default storage) */
ExtU_RollAxisAutopilot_T RollAxisAutopilot_U;

/* External outputs (root outports fed by signals with default storage) */
ExtY_RollAxisAutopilot_T RollAxisAutopilot_Y;

/* Real-time model */
static RT_MODEL_RollAxisAutopilot_T RollAxisAutopilot_M;
RT_MODEL_RollAxisAutopilot_T *const RollAxisAutopilot_M = &RollAxisAutopilot_M;
```

The file `RollAxisAutopilot.h` declares these structure variables. Your external code can include (`#include`) this file, whose general name is `model.h` where `model` is the name of the model, to access data that participate in the model interface.

In `RollAxisAutopilot.c`, the model `step` function, which represents the primary model algorithm, uses a `void void` interface (without arguments).

```
rtwdemodbtype(file,...
    /* Model step function */,'void RollAxisAutopilot_step(void)',1,1)

/* Model step function */
void RollAxisAutopilot_step(void)
```

In the function definition, the algorithm reads input data and writes output data by directly accessing the global structure variables. For example, near the end of the algorithm, the code for the `EngSwitch` block reads the value of the `AP_Eng` field (which represents an input) and, based on that value, conditionally writes a constant value to the `Ail_Cmd` field (which represents an output).

```
rtwdemodbtype(file,'if (RollAxisAutopilot_U.AP_Eng) {' , '      }',1,1)

if (RollAxisAutopilot_U.AP_Eng) {
    /* Outputs for Atomic SubSystem: '<Root>/BasicRollMode' */
    /* Saturate: '<S1>/CmdLimit' */
    if (RollAxisAutopilot_Y.Ail_Cmd > 15.0F) {
        /* Sum: '<S1>/Sum2' incorporates:
         * Output: '<Root>/Ail_Cmd'
         */
        RollAxisAutopilot_Y.Ail_Cmd = 15.0F;
    } else if (RollAxisAutopilot_Y.Ail_Cmd < -15.0F) {
        /* Sum: '<S1>/Sum2' incorporates:
         * Output: '<Root>/Ail_Cmd'
         */
    }
}
```

```

    RollAxisAutopilot_Y.Ail_Cmd = -15.0F;
}

/* End of Saturate: '<S1>/CmdLimit' */
/* End of Outputs for SubSystem: '<Root>/BasicRollMode' */
} else {
/* Sum: '<S1>/Sum2' incorporates:
 * Constant: '<Root>/Zero'
 * Outport: '<Root>/Ail_Cmd'
 */
RollAxisAutopilot_Y.Ail_Cmd = 0.0F;
}

/* End of Switch: '<Root>/EngSwitch' */
}

/* Model initialize function */
void RollAxisAutopilot_initialize(void)
{
/* Registration code */

/* initialize error status */
rtmSetErrorStatus(RollAxisAutopilot_M, (NULL));

/* states (dwork) */
(void) memset((void *)&RollAxisAutopilot_DW, 0,
              sizeof(DW_RollAxisAutopilot_T));

/* external inputs */
(void) memset(&RollAxisAutopilot_U, 0, sizeof(ExtU_RollAxisAutopilot_T));

/* external outputs */
RollAxisAutopilot_Y.Ail_Cmd = 0.0F;

/* SystemInitialize for Atomic SubSystem: '<Root>/RollAngleReference' */
/* InitializeConditions for UnitDelay: '<S4>/FixPt Unit Delay1' */
RollAxisAutopilot_DW.FixPtUnitDelay1_DSTATE = 0.0F;

/* End of SystemInitialize for SubSystem: '<Root>/RollAngleReference' */

/* SystemInitialize for Atomic SubSystem: '<Root>/BasicRollMode' */
/* InitializeConditions for DiscreteIntegrator: '<S1>/Integrator' */
RollAxisAutopilot_DW.Integrator_DSTATE = 0.0F;
RollAxisAutopilot_DW.Integrator_PrevResetState = 0;

/* End of SystemInitialize for SubSystem: '<Root>/BasicRollMode' */
}

/* Model terminate function */
void RollAxisAutopilot_terminate(void)
{
/* (no terminate code required) */
}

```

Due to the `void void` interface and the direct data access, the function is not reentrant. If you call the function multiple times in an application, each call reads and writes input and output data to the

same global structure variables, resulting in data corruption and unintentional interaction between the calls.

The model initialization function, `RollAxisAutopilot_initialize`, initializes input and output data to zero. The function also initializes the error status. The function directly accesses the global variables, which means the function is not reentrant.

```
rtwdemodbtype(file, '/* Model initialize function */', ...
    'rtmSetErrorStatus(RollAxisAutopilot_M, (NULL));', 1, 1)

/* Model initialize function */
void RollAxisAutopilot_initialize(void)
{
    /* Registration code */

    /* initialize error status */
    rtmSetErrorStatus(RollAxisAutopilot_M, (NULL));

rtwdemodbtype(file, ' /* external inputs */', ...
    '/* SystemInitialize for Atomic SubSystem:', 1, 0)

    /* external inputs */
    (void)memset(&RollAxisAutopilot_U, 0, sizeof(ExtU_RollAxisAutopilot_T));

    /* external outputs */
    RollAxisAutopilot_Y.Ail_Cmd = 0.0F;
}
```

Generate Reusable Code

You can configure the generated code as reentrant, which means that you can call the entry-point functions multiple times in an application. With this configuration, instead of directly accessing global variables, the entry-point functions exchange input, output, and other model data through formal parameters (pointer arguments). With these pointer arguments, each call can read inputs and write outputs to a set of separate global variables, preventing unintentional interaction between the calls.

In the model, set **Configuration Parameters > Code Generation > Interface > Code interface packaging** to Reusable function.

```
set_param('RollAxisAutopilot', 'CodeInterfacePackaging', 'Reusable function')
```

Generate code from the model.

```
slbuild('RollAxisAutopilot')

### Starting build procedure for: RollAxisAutopilot
### Successful completion of build procedure for: RollAxisAutopilot
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
RollAxisAutopilot	Code generated and compiled.	Generated code was out of date.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 14.175s
```

Now, in `RollAxisAutopilot.h`, the real-time model data structure contains pointers to the error indication, the input data, the output data, and additional data in the form of a `DWork` substructure (which stores, for example, block states such as the state of a Discrete-Time Integrator block).

```
file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot.h');
rtwdemodbtype(file,'/* Real-time Model Data Structure */',...
    '/* External data declarations for dependent source files */',1,0)
```

```
/* Real-time Model Data Structure */
struct tag_RTM_RollAxisAutopilot_T {
    const char_T *errorStatus;
    ExtU_RollAxisAutopilot_T *inputs;
    ExtY_RollAxisAutopilot_T *outputs;
    DW_RollAxisAutopilot_T *dwork;
};
```

To call the generated code multiple times in an application, your code must allocate memory for a real-time model data structure per call. The file `RollAxisAutopilot.c` defines a specialized function that allocates memory for a new real-time model data structure and returns a pointer to the structure. The function also allocates memory for the substructures that the fields in the model data structure point to, such as the input and output structures.

```
file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot.c');
rtwdemodbtype(file,'/* Model data allocation function */',...
    'RT_MODEL_RollAxisAutopilot_T *RollAxisAutopilot(void)',1,1)
```

```
/* Model data allocation function */
RT_MODEL_RollAxisAutopilot_T *RollAxisAutopilot(void)
```

In `RollAxisAutopilot.c`, the model step function accepts an argument that represents the real-time model data structure.

```
file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot.c');
rtwdemodbtype(file,...
    '/* Model step function */','void RollAxisAutopilot_step',1,1)
```

```
/* Model step function */
void RollAxisAutopilot_step(RT_MODEL_RollAxisAutopilot_T *const
```

In the function definition, the algorithm first extracts each pointer from the real-time model data structure into a local variable.

```
rtwdemodbtype(file,...
    '*RollAxisAutopilot_DW =','RollAxisAutopilot_M->outputs;',1,1)
```

```
DW_RollAxisAutopilot_T *RollAxisAutopilot_DW = RollAxisAutopilot_M->dwork;
ExtU_RollAxisAutopilot_T *RollAxisAutopilot_U = (ExtU_RollAxisAutopilot_T *)
    RollAxisAutopilot_M->inputs;
ExtY_RollAxisAutopilot_T *RollAxisAutopilot_Y = (ExtY_RollAxisAutopilot_T *)
    RollAxisAutopilot_M->outputs;
```

Then, to access the input and output data stored in global memory, the algorithm interacts with these local variables.

```

rtwdemodbtype(file, ' if (RollAxisAutopilot_U->AP_Eng) {',...
    ' /* End of Switch: '<Root>/EngSwitch' */',1,1)

if (RollAxisAutopilot_U->AP_Eng) {
    /* Outputs for Atomic SubSystem: '<Root>/BasicRollMode' */
    /* Saturate: '<S1>/CmdLimit' */
    if (RollAxisAutopilot_Y->Ail_Cmd > 15.0F) {
        /* Sum: '<S1>/Sum2' incorporates:
        *   Output: '<Root>/Ail_Cmd'
        */
        RollAxisAutopilot_Y->Ail_Cmd = 15.0F;
    } else if (RollAxisAutopilot_Y->Ail_Cmd < -15.0F) {
        /* Sum: '<S1>/Sum2' incorporates:
        *   Output: '<Root>/Ail_Cmd'
        */
        RollAxisAutopilot_Y->Ail_Cmd = -15.0F;
    }

    /* End of Saturate: '<S1>/CmdLimit' */
    /* End of Outputs for SubSystem: '<Root>/BasicRollMode' */
} else {
    /* Sum: '<S1>/Sum2' incorporates:
    *   Constant: '<Root>/Zero'
    *   Output: '<Root>/Ail_Cmd'
    */
    RollAxisAutopilot_Y->Ail_Cmd = 0.0F;
}

```

Similarly, the model initialization function accepts the real-time model data structure as an argument.

```

rtwdemodbtype(file,...
    '/* Model initialize function */','void RollAxisAutopilot_initialize',1,1)

/* Model initialize function */
void RollAxisAutopilot_initialize(RT_MODEL_RollAxisAutopilot_T *const

```

Because each call that you make to an entry-point function interacts with a separate real-time model data structure, you avoid unintentional interaction between the calls.

Configure Data Interface

To control the characteristics of the data interface in the generated code, see “Control Data and Function Interface in Generated Code” on page 15-44.

See Also

Related Examples

- “Data Structures in the Generated Code” on page 15-31
- “Choose an External Code Integration Workflow” (Embedded Coder)
- “Analyze Generated Data Code Interface” (Embedded Coder)

- “Control Data and Function Interface in Generated Code” (Embedded Coder)
- “How Generated Code Stores Internal Signal, State, and Parameter Data” (Embedded Coder)

Control Data and Function Interface in Generated Code

To use the code that you generate from a model, you call generated entry-point functions such as `step` and `initialize`. The calling environment and the generated functions exchange input and output data through global variables or through formal parameters (arguments). This data and the exchange mechanisms constitute the interfaces of the entry-point functions. For information about the default interfaces for reentrant and nonreentrant models in the generated code, see “How Generated Code Exchanges Data with an Environment” on page 15-35.

By controlling the interfaces that appear in the generated code, you can:

- Minimize the modifications that you must make to existing code.
- Generate stable interfaces that do not change or minimally change when you make changes to the model.
- Generate code that exchanges data more efficiently (for example, by using pointers and pass-by-reference arguments for nonscalar data).

Control Type Names, Field Names, and Variable Names of Standard I/O Structures (Embedded Coder)

By default, for nonreentrant code, Inport blocks at the root level of the model appear in the generated code as fields of a global structure variable. Similarly, Outport blocks appear in a different structure. For reentrant code, depending on your setting for model configuration parameter **Pass root-level I/O**, the code generator can also package input and output data into standard structures.

With Embedded Coder, you can control these names. See “Manage Replacement of Simulink Data Types in Generated Code” (Embedded Coder).

Control Names of Generated Entry-Point Functions (Embedded Coder)

To use the generated code, you write code that calls generated entry-point functions. For example, entry-point functions include `model_step`, `model_initialize`, and top-level functions generated from an export-function model. To control the names of the model entry-point functions, use the Code Mappings editor (requires Embedded Coder) to apply a combination of these techniques:

- On the **Function Defaults** tab, specify default naming rules for categories of entry-point functions by applying function customization templates in the Code Mappings editor. With this technique, a naming rule applies to functions in a category. For more information, see “Configure Default Code Generation for Functions” (Embedded Coder).
- On the **Functions** tab, specify names for individual entry-point functions by either directly editing the **Function Name** column or through a configuration dialog box opened from the **Function Preview** column. Names that you specify override default naming rules specified by function customization templates. For more information, see “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder)

Control Data Interface for Nonreentrant Code

When you set the model configuration parameter **Code interface packaging** to **Nonreusable function** (the default), generated entry-point functions are not reentrant. Typically, the functions exchange data with the calling environment through direct access to global variables.

Configure Inport or Output Block as Separate Global Variable

To remove the block from the standard I/O structures by creating a separate global variable, apply a storage class, such as `ExportedGlobal` or `ExportToFile`, to the signal that the block represents. You can configure a default storage class for categories of data elements, such as Inport blocks. As you add such blocks to the model, they acquire the storage class that you specify. You can use the Code Mappings editor also to configure individual blocks. You might do this if a model has just a few data elements of a specific category or override the default configuration settings.

For an example, see “Design Data Interface by Configuring Inport and Outport Blocks” on page 15-170. For general information about configuring data for code generation, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.

Configure Generated Code to Read or Write to Global Variables Defined by External Code

If your calling code already defines a global variable that you want the generated code to use as input data or use to store output data, you can reuse the variable by preventing the code generator from duplicating the definition. Apply a storage class to the corresponding Inport or Outport block in the model. Choose a storage class that specifies an imported data scope, such as `ImportedExtern` or `ImportFromFile`. For information about applying storage classes, see “C Code Generation Configuration for Model Interface Elements” on page 16-2 and “Organize Parameter Data into a Structure by Using Struct Storage Class” (Embedded Coder).

Package Multiple Inputs or Outputs into Custom Structure

You can configure a single Inport or Outport block to appear in the generated code as a custom structure that contains multiple input or output signals. You can also configure the block to appear as a substructure of the default I/O structures or as a separate structure variable.

Configure the block as a nonvirtual bus by using a `Simulink.Bus` object as the data type of the block. If your external code defines the structure type, consider using the `Simulink.importExternalCTypes` function to generate the bus object.

- To generate the bus signal as a substructure in the standard I/O structures, leave the block storage class at the default setting, `Auto`. If you have Embedded Coder, in the Code Mappings editor, on the **Data Defaults** tab, set the storage class for categories **Inports** and **Outports** to `Default`.
- To generate the bus signal as a separate global structure variable, apply a storage class such as `ExportedGlobal` or `ExportToFile`.

For more information about grouping signals into custom structures in the generated code, see “Organize Data into Structures in Generated Code” on page 15-147.

Configure Inport or Output Block as Function Call (Embedded Coder)

If your external code defines a function that returns input data for the generated code or accepts output data that the generated code produces, you can configure an Inport or Outport block so that the generated code calls the function instead of accessing a global variable. Apply the Embedded Coder storage class `GetSet`. For more information, see “Access Data Through Functions with Storage Class `GetSet`” (Embedded Coder).

Pass Inputs and Outputs Through Function Arguments (Embedded Coder)

With Embedded Coder, you can optionally configure the model step (execution) function to access root-level input and output through arguments instead of directly reading and writing to global

variables. Completely control the argument characteristics such as name, order, and passing mechanism (by reference or by value). This level of configuration can help to integrate generated code with your external code.

To pass inputs and outputs through arguments, in the Configure C Step Function interface dialog box, select **Configure arguments for Step function prototype**. Each Inport and Outport block at the root level of the model appears in the code as an argument of the execution function. For more information, see “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder).

Configure Referenced Model Inputs and Outputs as Global Variables (void-void)

By default, for a nonreentrant referenced model, the generated code passes root-level input and output through function arguments. A nonreentrant referenced model is one in which you set model configuration parameter **Total number of instances allowed per top model** to One.

To pass this data through global variables instead (for a void-void interface), in the referenced model, apply storage classes such as `ExportedGlobal` and `ExportToFile` to root-level Inport and Outport blocks.

- To apply a default storage class to such blocks, use the Code Mapping Editor. With this technique, as you add such blocks to the model, the blocks acquire the default storage class, preserving the void-void interface. For more information, see “Configure Default Code Generation for Data” (Embedded Coder).
- To override the default storage class, apply storage classes directly to individual blocks by using the Code Mapping editor. See “C Code Generation Configuration for Model Interface Elements” on page 16-2.
- For an example that requires Embedded Coder, see “Establish Data Ownership in a Model Hierarchy” (Embedded Coder).

Control Data Interface for Reentrant Code

When you set **Code interface packaging** to `Reusable` function, generated entry-point functions are reentrant. The functions exchange data with the calling environment through formal parameters (arguments). By default, each root-level Inport and Outport block appears in the generated code as a separate argument instead of a field of the standard I/O structures.

- To reduce the number of arguments, see “Reduce Number of Arguments by Using Structures” on page 15-47.
- To control the data types of the arguments, see “Control Data Types of Arguments” on page 15-48.

Prevent Unintended Changes to the Interface

Some changes that you make to a model change the entry-point function interfaces in the generated code. For example, if you change the name of the model, the names of the functions can change. If you configure the model code to exchange data through arguments, when you add or remove Inport or Outport blocks or change the names of the blocks, the corresponding arguments can change.

For easier maintenance of your calling code, prevent changes to the interfaces of the entry-point functions.

- If you exchange input and output data through arguments, configure the generated code to package Inport and Outport blocks into structures instead of allowing each block to appear as a separate argument (the default). Then, when you add or remove Inport or Outport blocks, or change their properties such as name and data type, the fields of the structures change, but the function interfaces do not change. See “Reduce Number of Arguments by Using Structures” on page 15-47.
- Set the data types of Inport and Outport blocks explicitly instead of using an inherited data type setting (which these blocks use by default). Inherited data type settings can cause the blocks to use different data types depending on the data types of upstream and downstream signals. For more information about configuring data types, see “Control Data Types of Signals”.
- With Embedded Coder, specify function names that do not depend on the model name. If you specify a naming rule with a function customization template, do not use the token \$R in the rule. See “Control Names of Generated Entry-Point Functions (Embedded Coder)” on page 15-44.

Reduce Number of Arguments by Using Structures

Reducing the number of arguments of a function can improve the readability of the code and reduce consumption of stack memory. To create a structure argument that can pass multiple pieces of data at one time, use these techniques:

- Manually combine multiple Inport or Outport blocks so that they appear in the generated code as fields of a structure or substructure of a standard data structure. The generated entry-point function or functions accept the address of the structure as a separate argument or as a substructure (field) of the standard I/O structures.

Replace the Inport or Outport blocks with a single block, and configure the new block as a nonvirtual bus by using a `Simulink.Bus` object as the data type of the block. If your external code defines the structure type, consider using the `Simulink.importExternalCTypes` function to generate the bus object. See “Organize Data into Structures in Generated Code” on page 15-147 and `Simulink.importExternalCTypes`.

- When you generate reentrant code with Embedded Coder, configure Inport and Outport blocks to appear in aggregated structures by default. Set model configuration parameter **Pass root-level I/O as** to a setting other than `Individual arguments`.
 - To package Inport and Outport blocks into the real-time model data structure, select `Part of model data structure`. The code generator aggregates the blocks into the default I/O structures to which the real-time model data structure points. The generated entry-point function or functions accept the real-time model data structure as a single argument. If you choose this setting, the function has the smallest number of arguments.
 - To aggregate Inport blocks into a structure and Outport blocks into a different structure, select `Structure reference`. The generated entry-point function or functions accept the address of each structure as an argument. Another argument accepts the real-time model data structure. If you choose this setting, the inputs and outputs are more identifiable in the function interfaces.

With this setting, if you remove the root-level Inport blocks or the root-level Outport blocks, the function signature can change. Similarly, the signature can change if you add a root-level Inport or Outport block to a model that does not include such blocks.

- For more control over the characteristics of the structures, set **Pass root-level I/O as** to `Part of model data structure`. Then, set the default storage class for Inport and Outport blocks to a structured storage class that you create. With this technique:

- You can create a single structure for the blocks or create two separate structures.
- You can control the names of the structure types.
- The structures appear as substructures of the real-time model data structure. You must set **Pass root-level I/O as** to **Part of model data structure**.

Create the storage class by using the Embedded Coder Dictionary (see “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder)). Apply the storage class by using the Code Mappings editor (see “Configure Default Code Generation for Data” (Embedded Coder)).

For more information, see [Pass root-level I/O as](#) (Embedded Coder).

Control Data Types of Arguments

You can configure the generated entry-point functions to exchange data through arguments. For a scalar or array argument, to control the name of the primitive data type, use a `Simulink.AliasType` object to either set the data type of the corresponding block or to configure data type replacements for the entire model. These techniques require Embedded Coder. For more information, see “Manage Replacement of Simulink Data Types in Generated Code” (Embedded Coder).

Promote Data Item to the Interface

By default, the code generator assumes that Inport and Outport blocks at the root level of the model constitute the data interface of the model. You can promote an arbitrary signal, block parameter, or block state to the interface so that other systems and components can access it. See “Promote Internal Data to the Interface” on page 15-66.

See Also

Related Examples

- “Design Data Interface by Configuring Inport and Outport Blocks” on page 15-170
- “Define Interfaces of Model Components”
- “Analyze Generated Data Code Interface” (Embedded Coder)
- “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder)
- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “Exchange Data Between External C/C++ Code and Simulink Model or Generated Code” on page 30-59
- “Configure Generated Code According to Interface Control Document Specifications” (Embedded Coder)
- “C Code Generation Configuration for Model Interface Elements” (Embedded Coder)

How Generated Code Stores Internal Signal, State, and Parameter Data

To calculate outputs from inputs, the generated code stores some internal data in global memory. A signal that does not connect to a root-level input or output (Inport or Outport block) is internal data.

Internal data can also include:

- A block state such as the state of a Unit Delay block. The algorithm must retain the state value between execution cycles, so the generated code typically stores states in global memory (for example, as a global variable or a field of a global structure variable).
- A block parameter, such as the **Gain** parameter of a Gain block, whose value the code generator cannot inline in the code. For example, the code generator cannot inline the value of a nonscalar parameter.
- The status indicator of a conditionally executed subsystem such as an enabled subsystem.

For more efficient code, you can configure optimizations such as **Configuration Parameters > Default parameter behavior** and **Configuration Parameters > Signal storage reuse** that attempt to eliminate storage for internal data. However, the optimizations cannot eliminate storage for some data, which consume memory in the generated code.

When you understand the default format in which the generated code stores internal data, you can:

- Make signals accessible and parameters tunable by default. You can then interact with and monitor the code during execution.
- Generate efficient production code by eliminating storage for internal data and, depending on your hardware and build toolchain, controlling the placement in memory of data that the optimizations cannot eliminate.
- Promote pieces of internal data to the model interface so that other components and systems can access that data.

For information about how the generated code exchanges data with a calling environment through interfaces, see “How Generated Code Exchanges Data with an Environment” on page 15-35.

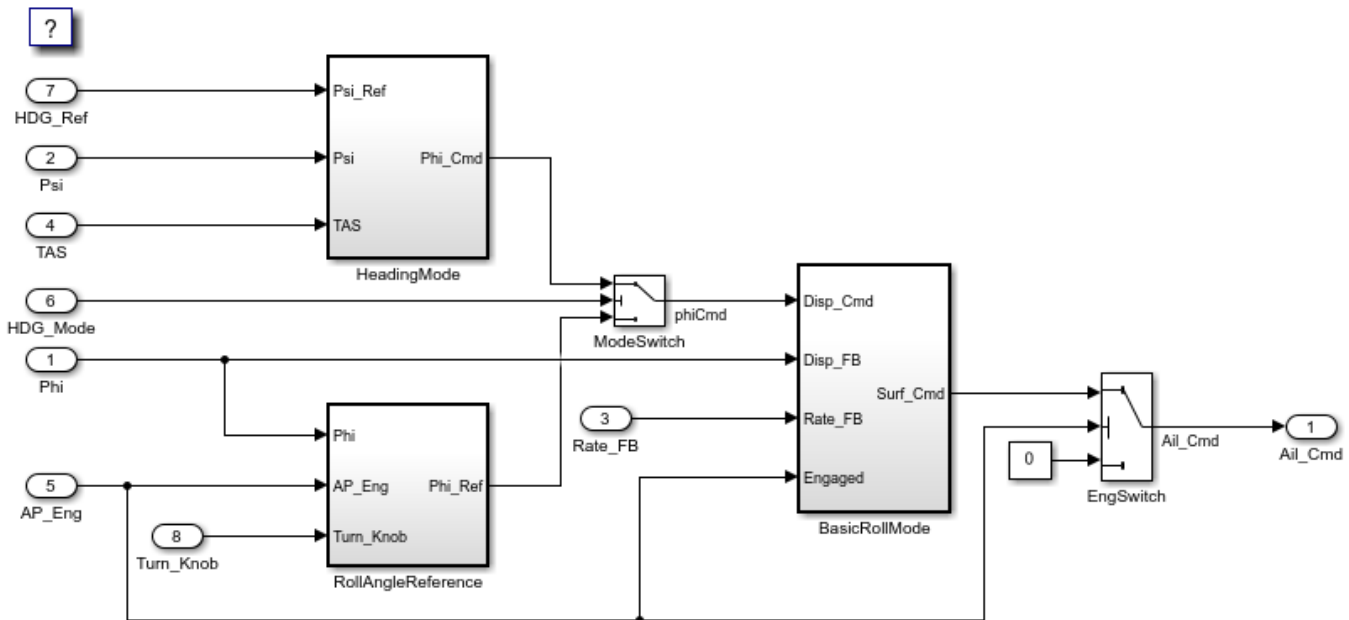
Internal Data in the Generated Code

This example shows how the generated code stores internal data such as block states.

Explore Example Model

Open the example model RollAxisAutopilot.

```
open_system('RollAxisAutopilot')
```



Copyright 1990-2020 The MathWorks, Inc.

The model contains internal signals that do not connect to root-level Inport or Outport blocks. Some of the signals have a name, such as the `phiCmd` signal.

The model also contains some blocks that maintain state data. For example, in the `BasicRollMode` subsystem, a Discrete-Time Integrator block labeled `Integrator` maintains a state.

In the model, set **Configuration Parameters > Code Generation > System target file** to `grt.tlc`.

```
set_param('RollAxisAutopilot','SystemTargetFile','grt.tlc')
```

Inspect the setting for **Configuration Parameters > Code Generation > Interface > Code interface packaging**. The setting `Nonreusable` function means that the generated code is not reusable (reentrant).

For this example, generate simpler code by clearing **Configuration Parameters > Code Generation > Interface > Advanced parameters > Mat-file logging**.

```
set_param('RollAxisAutopilot','MatFileLogging','off')
```

Generate Nonreusable Code

Set these configuration parameters:

- Set **Default parameter behavior** to `Tunable`.
- Clear **Signal storage reuse**.

```
set_param('RollAxisAutopilot','DefaultParameterBehavior','Tunable',...
         'OptimizeBlockIOStorage','off')
```

Generate code from the model.


```
slbuild('RollAxisAutopilot')
```

```
### Starting build procedure for: RollAxisAutopilot
### Successful completion of build procedure for: RollAxisAutopilot
```

```
Build Summary
```

```
Top model targets built:
```

Model	Action	Rebuild Reason
RollAxisAutopilot	Code generated and compiled.	Code generation information file does not exist

```
1 of 1 models built (0 models already up to date)
```

```
Build duration: 0h 0m 33.39s
```

The file `RollAxisAutopilot.h` defines several structure types that represent internal data. For example, the block input and output structure defines one field for each internal signal in the model. Each field name derives from the name of the block that generates the signal or, if you specify a name for the signal, from the name of the signal.

```
file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot.h');
rtwdemodbtype(file,...
    /* Block signals (default storage) */,'} B_RollAxisAutopilot_T;',1,1)
```

```
/* Block signals (default storage) */
typedef struct {
    real32_T phiCmd; /* '<Root>/ModeSwitch' */
    real32_T hdgError; /* '<S2>/Sum' */
    real32_T DispGain; /* '<S2>/DispGain' */
    real32_T Product; /* '<S2>/Product' */
    real32_T Abs; /* '<S3>/Abs' */
    real32_T FixPtUnitDelay1; /* '<S4>/FixPt Unit Delay1' */
    real32_T Xnew; /* '<S4>/Enable' */
    real32_T TKSwitch; /* '<S3>/TKSwitch' */
    real32_T RefSwitch; /* '<S3>/RefSwitch' */
    real32_T Integrator; /* '<S1>/Integrator' */
    real32_T DispLimit; /* '<S1>/DispLimit' */
    real32_T Sum; /* '<S1>/Sum' */
    real32_T DispGain_f; /* '<S1>/DispGain' */
    real32_T RateLimit; /* '<S1>/RateLimit' */
    real32_T Sum1; /* '<S1>/Sum1' */
    real32_T RateGain; /* '<S1>/RateGain' */
    real32_T Sum2; /* '<S1>/Sum2' */
    real32_T CmdLimit; /* '<S1>/CmdLimit' */
    real32_T IntGain; /* '<S1>/IntGain' */
    boolean_T NotEngaged; /* '<S3>/NotEngaged' */
    boolean_T TKThreshold; /* '<S3>/TKThreshold' */
    boolean_T RefThreshold2; /* '<S3>/RefThreshold2' */
    boolean_T RefThreshold1; /* '<S3>/RefThreshold1' */
    boolean_T Or; /* '<S3>/Or' */
    boolean_T NotEngaged_e; /* '<S1>/NotEngaged' */
} B_RollAxisAutopilot_T;
```

The file defines a structure type, the `DWork` structure, to represent block states such as the state of the Discrete-Time Integrator block.

```

rtwdemodbtype(file,...
    /* Block states (default storage) for system','} DW_RollAxisAutopilot_T;',1,1)

/* Block states (default storage) for system '<Root>' */
typedef struct {
    real32_T FixPtUnitDelay1_DSTATE;    /* '<S4>/FixPt Unit Delay1' */
    real32_T Integrator_DSTATE;        /* '<S1>/Integrator' */
    int8_T Integrator_PrevResetState;   /* '<S1>/Integrator' */
} DW_RollAxisAutopilot_T;

```

The file defines a structure type to represent parameter data. Each tunable block parameter in the model, such as the **Gain** parameter of a Gain block, appears as a field of this structure. If a block parameter acquires its value from a MATLAB® variable or a `Simulink.Parameter` object, the variable or object appears as a field, not the block parameter.

The file also defines a structure type, the *real-time model data structure*, whose single field represents a run-time indication of whether the generated code has encountered an error during execution.

```

rtwdemodbtype(file, /* Real-time Model Data Structure */',...
    /* Block parameters (default storage) */',1,0)

/* Real-time Model Data Structure */
struct tag_RTM_RollAxisAutopilot_T {
    const char_T *errorStatus;
};

```

For the structure type that represents the real-time model data structure, the file `RollAxisAutopilot_types.h` creates an alias that the generated code later uses to allocate memory for the structure.

```

file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot_types.h');
rtwdemodbtype(file, /* Forward declaration for rtModel */',...
    'RT_MODEL_RollAxisAutopilot_T;',1,1)

/* Forward declaration for rtModel */
typedef struct tag_RTM_RollAxisAutopilot_T RT_MODEL_RollAxisAutopilot_T;

```

Using these structure types, the file `RollAxisAutopilot.c` defines (allocates memory for) global structure variables that store internal data for the generated algorithm. The file also defines variables that represent the real-time model data structure and a pointer to the structure.

```

file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot.c');
rtwdemodbtype(file, /* Block signals (default storage) */',...
    '= &RollAxisAutopilot_M;',1,1)

/* Block signals (default storage) */
B_RollAxisAutopilot_T RollAxisAutopilot_B;

/* Block states (default storage) */
DW_RollAxisAutopilot_T RollAxisAutopilot_DW;

/* External inputs (root inport signals with default storage) */
ExtU_RollAxisAutopilot_T RollAxisAutopilot_U;

```

```

/* External outputs (root outports fed by signals with default storage) */
ExtY_RollAxisAutopilot_T RollAxisAutopilot_Y;

/* Real-time model */
static RT_MODEL_RollAxisAutopilot_T RollAxisAutopilot_M_;
RT_MODEL_RollAxisAutopilot_T *const RollAxisAutopilot_M = &RollAxisAutopilot_M_;

```

The model `step` function, which represents the primary model algorithm, uses a `void void` interface (with no arguments).

```

rtwdemodbtype(file,...
    /* Model step function */,'void RollAxisAutopilot_step(void)',1,1)

```

```

/* Model step function */
void RollAxisAutopilot_step(void)

```

In the function definition, the algorithm performs calculations and stores intermediate results in the signal and state structures by directly accessing the global variables. The algorithm also reads parameter data from the corresponding global variable. For example, in the `BasicRollMode` subsystem, the code generated for the `Integrator` block reads and writes signal, state, and parameter data from the structures.

```

rtwdemodbtype(file,'/* DiscreteIntegrator: '<S1>/Integrator' */,...
    /* End of DiscreteIntegrator: '<S1>/Integrator' */',1,1)

/* DiscreteIntegrator: '<S1>/Integrator' */
if (RollAxisAutopilot_B.NotEngaged_e ||
    (RollAxisAutopilot_DW.Integrator_PrevResetState != 0)) {
    RollAxisAutopilot_DW.Integrator_DSTATE = RollAxisAutopilot_P.Integrator_IC;
}

if (RollAxisAutopilot_DW.Integrator_DSTATE >= RollAxisAutopilot_P.intLim) {
    RollAxisAutopilot_DW.Integrator_DSTATE = RollAxisAutopilot_P.intLim;
} else if (RollAxisAutopilot_DW.Integrator_DSTATE <=
    RollAxisAutopilot_P.Integrator_LowerSat) {
    RollAxisAutopilot_DW.Integrator_DSTATE =
        RollAxisAutopilot_P.Integrator_LowerSat;
}

/* DiscreteIntegrator: '<S1>/Integrator' */
RollAxisAutopilot_B.Integrator = RollAxisAutopilot_DW.Integrator_DSTATE;

/* Saturate: '<S1>/DispLimit' */
u0 = RollAxisAutopilot_B.phiCmd;
u1 = RollAxisAutopilot_P.DispLimit_LowerSat;
u2 = RollAxisAutopilot_P.dispLim;
if (u0 > u2) {
    /* Saturate: '<S1>/DispLimit' */
    RollAxisAutopilot_B.DispLimit = u2;
} else if (u0 < u1) {
    /* Saturate: '<S1>/DispLimit' */
    RollAxisAutopilot_B.DispLimit = u1;
} else {
    /* Saturate: '<S1>/DispLimit' */
    RollAxisAutopilot_B.DispLimit = u0;
}

```

```

}

/* End of Saturate: '<S1>/DispLimit' */

/* Sum: '<S1>/Sum' incorporates:
 * Inport: '<Root>/Phi'
 */
RollAxisAutopilot_B.Sum = RollAxisAutopilot_B.DispLimit -
    RollAxisAutopilot_U.Phi;

/* Gain: '<S1>/DispGain' */
RollAxisAutopilot_B.DispGain_f = RollAxisAutopilot_P.dispGain *
    RollAxisAutopilot_B.Sum;

/* Saturate: '<S1>/RateLimit' */
u0 = RollAxisAutopilot_B.DispGain_f;
u1 = RollAxisAutopilot_P.RateLimit_LowerSat;
u2 = RollAxisAutopilot_P.rateLim;
if (u0 > u2) {
    /* Saturate: '<S1>/RateLimit' */
    RollAxisAutopilot_B.RateLimit = u2;
} else if (u0 < u1) {
    /* Saturate: '<S1>/RateLimit' */
    RollAxisAutopilot_B.RateLimit = u1;
} else {
    /* Saturate: '<S1>/RateLimit' */
    RollAxisAutopilot_B.RateLimit = u0;
}

/* End of Saturate: '<S1>/RateLimit' */

/* Sum: '<S1>/Sum1' incorporates:
 * Inport: '<Root>/Rate_FB'
 */
RollAxisAutopilot_B.Sum1 = RollAxisAutopilot_B.RateLimit -
    RollAxisAutopilot_U.Rate_FB;

/* Gain: '<S1>/RateGain' */
RollAxisAutopilot_B.RateGain = RollAxisAutopilot_P.rateGain *
    RollAxisAutopilot_B.Sum1;

/* Sum: '<S1>/Sum2' */
RollAxisAutopilot_B.Sum2 = RollAxisAutopilot_B.Integrator +
    RollAxisAutopilot_B.RateGain;

/* Saturate: '<S1>/CmdLimit' */
u0 = RollAxisAutopilot_B.Sum2;
u1 = RollAxisAutopilot_P.CmdLimit_LowerSat;
u2 = RollAxisAutopilot_P.cmdLim;
if (u0 > u2) {
    /* Saturate: '<S1>/CmdLimit' */
    RollAxisAutopilot_B.CmdLimit = u2;
} else if (u0 < u1) {
    /* Saturate: '<S1>/CmdLimit' */
    RollAxisAutopilot_B.CmdLimit = u1;
} else {
    /* Saturate: '<S1>/CmdLimit' */
    RollAxisAutopilot_B.CmdLimit = u0;
}

```

```

}

/* End of Saturate: '<S1>/CmdLimit' */

/* Gain: '<S1>/IntGain' */
RollAxisAutopilot_B.IntGain = RollAxisAutopilot_P.intGain *
    RollAxisAutopilot_B.Sum1;

/* Update for DiscreteIntegrator: '<S1>/Integrator' */
RollAxisAutopilot_DW.Integrator_DSTATE +=
    RollAxisAutopilot_P.Integrator_gainval * RollAxisAutopilot_B.IntGain;
if (RollAxisAutopilot_DW.Integrator_DSTATE >= RollAxisAutopilot_P.intLim) {
    RollAxisAutopilot_DW.Integrator_DSTATE = RollAxisAutopilot_P.intLim;
} else if (RollAxisAutopilot_DW.Integrator_DSTATE <=
    RollAxisAutopilot_P.Integrator_LowerSat) {
    RollAxisAutopilot_DW.Integrator_DSTATE =
        RollAxisAutopilot_P.Integrator_LowerSat;
}

RollAxisAutopilot_DW.Integrator_PrevResetState = (int8_T)
    RollAxisAutopilot_B.NotEngaged_e;

/* End of Update for DiscreteIntegrator: '<S1>/Integrator' */
/* End of Outputs for SubSystem: '<Root>/BasicRollMode' */

/* Switch: '<Root>/EngSwitch' incorporates:
 * Inport: '<Root>/AP_Eng'
 */
if (RollAxisAutopilot_U.AP_Eng) {
    /* Outport: '<Root>/Ail_Cmd' */
    RollAxisAutopilot_Y.Ail_Cmd = RollAxisAutopilot_B.CmdLimit;
} else {
    /* Outport: '<Root>/Ail_Cmd' incorporates:
     * Constant: '<Root>/Zero'
     */
    RollAxisAutopilot_Y.Ail_Cmd = RollAxisAutopilot_P.Zero_Value_c;
}

/* End of Switch: '<Root>/EngSwitch' */
}

/* Model initialize function */
void RollAxisAutopilot_initialize(void)
{
    /* Registration code */

    /* initialize error status */
    rtmSetErrorStatus(RollAxisAutopilot_M, (NULL));

    /* block I/O */
    (void) memset(((void *) &RollAxisAutopilot_B), 0,
        sizeof(B_RollAxisAutopilot_T));

    /* states (dwork) */
    (void) memset(((void *)&RollAxisAutopilot_DW, 0,
        sizeof(DW_RollAxisAutopilot_T));

    /* external inputs */

```

```

(void)memset(&RollAxisAutopilot_U, 0, sizeof(ExtU_RollAxisAutopilot_T));

/* external outputs */
RollAxisAutopilot_Y.Ail_Cmd = 0.0F;

/* SystemInitialize for Atomic SubSystem: '<Root>/RollAngleReference' */
/* InitializeConditions for UnitDelay: '<S4>/FixPt Unit Delay1' */
RollAxisAutopilot_DW.FixPtUnitDelay1_DSTATE =
    RollAxisAutopilot_P.LatchPhi_vinit;

/* End of SystemInitialize for SubSystem: '<Root>/RollAngleReference' */

/* SystemInitialize for Atomic SubSystem: '<Root>/BasicRollMode' */
/* InitializeConditions for DiscreteIntegrator: '<S1>/Integrator' */
RollAxisAutopilot_DW.Integrator_DSTATE = RollAxisAutopilot_P.Integrator_IC;
RollAxisAutopilot_DW.Integrator_PrevResetState = 0;

/* End of SystemInitialize for SubSystem: '<Root>/BasicRollMode' */
}

/* Model terminate function */
void RollAxisAutopilot_terminate(void)
{
    /* (no terminate code required) */
}

```

Due to the `void` `void` interface and the direct data access, the function is not reentrant. If you call the function multiple times in an application, each call writes data to the global structure variables and the subsequent call can read that data, resulting in unintentional interference between the calls.

The model initialization function `RollAxisAutopilot_initialize` initializes all of the internal data to zero. The function also initializes the error status by calling a specialized macro function. The initialization function directly accesses the global variables, which means that the function is not reentrant.

```

rtwdemodbtype(file, '/* Model initialize function */', ...
    'sizeof(DW_RollAxisAutopilot_T));', 1, 1)

```

```

/* Model initialize function */
void RollAxisAutopilot_initialize(void)
{
    /* Registration code */

    /* initialize error status */
    rtmSetErrorStatus(RollAxisAutopilot_M, (NULL));

    /* block I/O */
    (void) memset(((void *) &RollAxisAutopilot_B), 0,
        sizeof(B_RollAxisAutopilot_T));

    /* states (dwork) */
    (void) memset(((void *)&RollAxisAutopilot_DW), 0,
        sizeof(DW_RollAxisAutopilot_T));
}

```

The function then initializes the block states in the `DWork` structure to the initial values that the block parameters in the model specify. Two of the three states in the model have tunable initial values, so the code initializes them by reading data from the parameters structure.

```

rtwdemodbtype(file,...
    /* SystemInitialize for Atomic SubSystem: '<Root>/RollAngleReference' */,...
    /* Model terminate function */ ,1,0)

/* SystemInitialize for Atomic SubSystem: '<Root>/RollAngleReference' */
/* InitializeConditions for UnitDelay: '<S4>/FixPt Unit Delay1' */
RollAxisAutopilot_DW.FixPtUnitDelay1_DSTATE =
    RollAxisAutopilot_P.LatchPhi_vinit;

/* End of SystemInitialize for SubSystem: '<Root>/RollAngleReference' */

/* SystemInitialize for Atomic SubSystem: '<Root>/BasicRollMode' */
/* InitializeConditions for DiscreteIntegrator: '<S1>/Integrator' */
RollAxisAutopilot_DW.Integrator_DSTATE = RollAxisAutopilot_P.Integrator_IC;
RollAxisAutopilot_DW.Integrator_PrevResetState = 0;

/* End of SystemInitialize for SubSystem: '<Root>/BasicRollMode' */
}

```

Generate Reusable Code

You can configure the generated code as reentrant, which means you can call the entry-point functions multiple times in an application. With this configuration, instead of directly accessing global variables, the entry-point functions accept internal data through formal parameters (pointer arguments). With these pointer arguments, each call can maintain internal data in a set of separate global variables, preventing unintentional interaction between the calls.

In the model, set **Configuration Parameters > Code Generation > Interface > Code interface packaging** to Reusable function.

```
set_param('RollAxisAutopilot','CodeInterfacePackaging','Reusable function')
```

Generate code from the model.

```
slbuild('RollAxisAutopilot')

### Starting build procedure for: RollAxisAutopilot
### Successful completion of build procedure for: RollAxisAutopilot
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
RollAxisAutopilot	Code generated and compiled.	Generated code was out of date.

1 of 1 models built (0 models already up to date)

Build duration: 0h 0m 22.83s

Now, in `RollAxisAutopilot.h`, the real-time model data structure contains pointers to the error indication, the internal data, and primary input and output data in the form of `ExtU` and `ExtY` substructures (the fields of which represent Inport and Outport blocks at the root level of the model).

```

file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot.h');
rtwdemodbtype(file,'/* Real-time Model Data Structure */',...
    /* External data declarations for dependent source files */ ,1,0)

```

```

/* Real-time Model Data Structure */
struct tag_RTM_RollAxisAutopilot_T {
    const char_T *errorStatus;
    B_RollAxisAutopilot_T *blockIO;
    ExtU_RollAxisAutopilot_T *inputs;
    ExtY_RollAxisAutopilot_T *outputs;
    DW_RollAxisAutopilot_T *dwork;
};

/* Block parameters (default storage) */
extern P_RollAxisAutopilot_T RollAxisAutopilot_P;

```

To call the generated code multiple times in an application, your code must allocate memory for a real-time model data structure per call. The file `RollAxisAutopilot.c` defines a specialized function that allocates memory for a new real-time model data structure and returns a pointer to the structure. The function also allocates memory for the substructures that the fields in the model data structure point to, such as the `DWork` structure.

```

file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot.c');
rtwdemodbtype(file,'/* Model data allocation function */',...
    'RT_MODEL_RollAxisAutopilot_T *RollAxisAutopilot(void)',1,1)

```

```

/* Model data allocation function */
RT_MODEL_RollAxisAutopilot_T *RollAxisAutopilot(void)

```

The model `step` function accepts an argument that represents the real-time model data structure.

```

rtwdemodbtype(file,'/* Model step function */','void RollAxisAutopilot_step',1,1)

```

```

/* Model step function */
void RollAxisAutopilot_step(RT_MODEL_RollAxisAutopilot_T *const

```

In the function definition, the algorithm first extracts each pointer from the real-time model data structure into a local variable.

```

rtwdemodbtype(file,'*RollAxisAutopilot_B =','RollAxisAutopilot_M->outputs;',1,1)

```

```

    B_RollAxisAutopilot_T *RollAxisAutopilot_B = RollAxisAutopilot_M->blockIO;
    DW_RollAxisAutopilot_T *RollAxisAutopilot_DW = RollAxisAutopilot_M->dwork;
    ExtU_RollAxisAutopilot_T *RollAxisAutopilot_U = (ExtU_RollAxisAutopilot_T *)
        RollAxisAutopilot_M->inputs;
    ExtY_RollAxisAutopilot_T *RollAxisAutopilot_Y = (ExtY_RollAxisAutopilot_T *)
        RollAxisAutopilot_M->outputs;

```

Then, to access the internal data stored in global memory, the algorithm interacts with these local variables.

```

rtwdemodbtype(file,'/* DiscreteIntegrator: '<S1>/Integrator' */',...
    '/* End of DiscreteIntegrator: '<S1>/Integrator' */',1,1)

```

```

/* DiscreteIntegrator: '<S1>/Integrator' */
if (RollAxisAutopilot_B->NotEngaged_e ||
    (RollAxisAutopilot_DW->Integrator_PrevResetState != 0)) {
    RollAxisAutopilot_DW->Integrator_DSTATE = RollAxisAutopilot_P.Integrator_IC;
}

```



```

}

if (RollAxisAutopilot_DW->Integrator_DSTATE >= RollAxisAutopilot_P.intLim) {
    RollAxisAutopilot_DW->Integrator_DSTATE = RollAxisAutopilot_P.intLim;
} else if (RollAxisAutopilot_DW->Integrator_DSTATE <=
    RollAxisAutopilot_P.Integrator_LowerSat) {
    RollAxisAutopilot_DW->Integrator_DSTATE =
        RollAxisAutopilot_P.Integrator_LowerSat;
}

/* DiscreteIntegrator: '<S1>/Integrator' */
RollAxisAutopilot_B->Integrator = RollAxisAutopilot_DW->Integrator_DSTATE;

/* Saturate: '<S1>/DispLimit' */
u0 = RollAxisAutopilot_B->phiCmd;
u1 = RollAxisAutopilot_P.DispLimit_LowerSat;
u2 = RollAxisAutopilot_P.dispLim;
if (u0 > u2) {
    /* Saturate: '<S1>/DispLimit' */
    RollAxisAutopilot_B->DispLimit = u2;
} else if (u0 < u1) {
    /* Saturate: '<S1>/DispLimit' */
    RollAxisAutopilot_B->DispLimit = u1;
} else {
    /* Saturate: '<S1>/DispLimit' */
    RollAxisAutopilot_B->DispLimit = u0;
}

/* End of Saturate: '<S1>/DispLimit' */

/* Sum: '<S1>/Sum' */
RollAxisAutopilot_B->Sum = RollAxisAutopilot_B->DispLimit -
    RollAxisAutopilot_U->Phi;

/* Gain: '<S1>/DispGain' */
RollAxisAutopilot_B->DispGain_f = RollAxisAutopilot_P.dispGain *
    RollAxisAutopilot_B->Sum;

/* Saturate: '<S1>/RateLimit' */
u0 = RollAxisAutopilot_B->DispGain_f;
u1 = RollAxisAutopilot_P.RateLimit_LowerSat;
u2 = RollAxisAutopilot_P.rateLim;
if (u0 > u2) {
    /* Saturate: '<S1>/RateLimit' */
    RollAxisAutopilot_B->RateLimit = u2;
} else if (u0 < u1) {
    /* Saturate: '<S1>/RateLimit' */
    RollAxisAutopilot_B->RateLimit = u1;
} else {
    /* Saturate: '<S1>/RateLimit' */
    RollAxisAutopilot_B->RateLimit = u0;
}

/* End of Saturate: '<S1>/RateLimit' */

/* Sum: '<S1>/Sum1' */
RollAxisAutopilot_B->Sum1 = RollAxisAutopilot_B->RateLimit -
    RollAxisAutopilot_U->Rate_FB;

```

```

/* Gain: '<S1>/RateGain' */
RollAxisAutopilot_B->RateGain = RollAxisAutopilot_P.rateGain *
    RollAxisAutopilot_B->Sum1;

/* Sum: '<S1>/Sum2' */
RollAxisAutopilot_B->Sum2 = RollAxisAutopilot_B->Integrator +
    RollAxisAutopilot_B->RateGain;

/* Saturate: '<S1>/CmdLimit' */
u0 = RollAxisAutopilot_B->Sum2;
u1 = RollAxisAutopilot_P.CmdLimit_LowerSat;
u2 = RollAxisAutopilot_P.cmdLim;
if (u0 > u2) {
    /* Saturate: '<S1>/CmdLimit' */
    RollAxisAutopilot_B->CmdLimit = u2;
} else if (u0 < u1) {
    /* Saturate: '<S1>/CmdLimit' */
    RollAxisAutopilot_B->CmdLimit = u1;
} else {
    /* Saturate: '<S1>/CmdLimit' */
    RollAxisAutopilot_B->CmdLimit = u0;
}

/* End of Saturate: '<S1>/CmdLimit' */

/* Gain: '<S1>/IntGain' */
RollAxisAutopilot_B->IntGain = RollAxisAutopilot_P.intGain *
    RollAxisAutopilot_B->Sum1;

/* Update for DiscreteIntegrator: '<S1>/Integrator' */
RollAxisAutopilot_DW->Integrator_DSTATE +=
    RollAxisAutopilot_P.Integrator_gainval * RollAxisAutopilot_B->IntGain;
if (RollAxisAutopilot_DW->Integrator_DSTATE >= RollAxisAutopilot_P.intLim) {
    RollAxisAutopilot_DW->Integrator_DSTATE = RollAxisAutopilot_P.intLim;
} else if (RollAxisAutopilot_DW->Integrator_DSTATE <=
    RollAxisAutopilot_P.Integrator_LowerSat) {
    RollAxisAutopilot_DW->Integrator_DSTATE =
        RollAxisAutopilot_P.Integrator_LowerSat;
}

RollAxisAutopilot_DW->Integrator_PrevResetState = (int8_T)
    RollAxisAutopilot_B->NotEngaged_e;

/* End of Update for DiscreteIntegrator: '<S1>/Integrator' */
/* End of Outputs for SubSystem: '<Root>/BasicRollMode' */

/* Switch: '<Root>/EngSwitch' */
if (RollAxisAutopilot_U->AP_Eng) {
    /* Outport: '<Root>/Ail_Cmd' */
    RollAxisAutopilot_Y->Ail_Cmd = RollAxisAutopilot_B->CmdLimit;
} else {
    /* Outport: '<Root>/Ail_Cmd' incorporates:
    * Constant: '<Root>/Zero'
    */
    RollAxisAutopilot_Y->Ail_Cmd = RollAxisAutopilot_P.Zero_Value_c;
}

```

```

    /* End of Switch: '<Root>/EngSwitch' */
}

/* Model initialize function */
void RollAxisAutopilot_initialize(RT_MODEL_RollAxisAutopilot_T *const
    RollAxisAutopilot_M)
{
    DW_RollAxisAutopilot_T *RollAxisAutopilot_DW = RollAxisAutopilot_M->dwork;

    /* SystemInitialize for Atomic SubSystem: '<Root>/RollAngleReference' */
    /* InitializeConditions for UnitDelay: '<S4>/FixPt Unit Delay1' */
    RollAxisAutopilot_DW->FixPtUnitDelay1_DSTATE =
        RollAxisAutopilot_P.LatchPhi_vinit;

    /* End of SystemInitialize for SubSystem: '<Root>/RollAngleReference' */

    /* SystemInitialize for Atomic SubSystem: '<Root>/BasicRollMode' */
    /* InitializeConditions for DiscreteIntegrator: '<S1>/Integrator' */
    RollAxisAutopilot_DW->Integrator_DSTATE = RollAxisAutopilot_P.Integrator_IC;
    RollAxisAutopilot_DW->Integrator_PrevResetState = 0;

    /* End of SystemInitialize for SubSystem: '<Root>/BasicRollMode' */
}

/* Model terminate function */
void RollAxisAutopilot_terminate(RT_MODEL_RollAxisAutopilot_T
    * RollAxisAutopilot_M)
{
    /* model code */
    rt_FREE(RollAxisAutopilot_M->blockIO);
    rt_FREE(RollAxisAutopilot_M->inputs);
    rt_FREE(RollAxisAutopilot_M->outputs);
    rt_FREE(RollAxisAutopilot_M->dwork);
    rt_FREE(RollAxisAutopilot_M);
}

/* Model data allocation function */
RT_MODEL_RollAxisAutopilot_T *RollAxisAutopilot(void)
{
    RT_MODEL_RollAxisAutopilot_T *RollAxisAutopilot_M;
    RollAxisAutopilot_M = (RT_MODEL_RollAxisAutopilot_T *) malloc(sizeof
        (RT_MODEL_RollAxisAutopilot_T));
    if (RollAxisAutopilot_M == (NULL)) {
        return (NULL);
    }

    (void) memset((char *)RollAxisAutopilot_M, 0,
        sizeof(RT_MODEL_RollAxisAutopilot_T));

    /* block I/O */
    {
        B_RollAxisAutopilot_T *b = (B_RollAxisAutopilot_T *) malloc(sizeof
            (B_RollAxisAutopilot_T));
        rt_VALIDATE_MEMORY(RollAxisAutopilot_M,b);
        RollAxisAutopilot_M->blockIO = (b);
    }

    /* states (dwork) */

```

```

{
    DW_RollAxisAutopilot_T *dwork = (DW_RollAxisAutopilot_T *) malloc(sizeof
        (DW_RollAxisAutopilot_T));
    rt_VALIDATE_MEMORY(RollAxisAutopilot_M,dwork);
    RollAxisAutopilot_M->dwork = (dwork);
}

/* external inputs */
{
    ExtU_RollAxisAutopilot_T *RollAxisAutopilot_U = (ExtU_RollAxisAutopilot_T *)
        malloc(sizeof(ExtU_RollAxisAutopilot_T));
    rt_VALIDATE_MEMORY(RollAxisAutopilot_M,RollAxisAutopilot_U);
    RollAxisAutopilot_M->inputs = (((ExtU_RollAxisAutopilot_T *)
        RollAxisAutopilot_U));
}

/* external outputs */
{
    ExtY_RollAxisAutopilot_T *RollAxisAutopilot_Y = (ExtY_RollAxisAutopilot_T *)
        malloc(sizeof(ExtY_RollAxisAutopilot_T));
    rt_VALIDATE_MEMORY(RollAxisAutopilot_M,RollAxisAutopilot_Y);
    RollAxisAutopilot_M->outputs = (RollAxisAutopilot_Y);
}

{
    B_RollAxisAutopilot_T *RollAxisAutopilot_B = RollAxisAutopilot_M->blockIO;
    DW_RollAxisAutopilot_T *RollAxisAutopilot_DW = RollAxisAutopilot_M->dwork;
    ExtU_RollAxisAutopilot_T *RollAxisAutopilot_U = (ExtU_RollAxisAutopilot_T *)
        RollAxisAutopilot_M->inputs;
    ExtY_RollAxisAutopilot_T *RollAxisAutopilot_Y = (ExtY_RollAxisAutopilot_T *)
        RollAxisAutopilot_M->outputs;

    /* block I/O */
    (void) memset(((void *) RollAxisAutopilot_B), 0,
        sizeof(B_RollAxisAutopilot_T));

    /* states (dwork) */
    (void) memset((void *)RollAxisAutopilot_DW, 0,
        sizeof(DW_RollAxisAutopilot_T));

    /* external inputs */
    (void)memset(RollAxisAutopilot_U, 0, sizeof(ExtU_RollAxisAutopilot_T));

    /* external outputs */
    RollAxisAutopilot_Y->Ail_Cmd = 0.0F;
}

return RollAxisAutopilot_M;
}

```

Similarly, the model initialization function accepts the real-time model data structure as an argument.

```

rtwdemodbtype(file,...
    /* Model initialize function */,'void RollAxisAutopilot_initialize',1,1)

/* Model initialize function */
void RollAxisAutopilot_initialize(RT_MODEL_RollAxisAutopilot_T *const

```

Because each call that you make to an entry-point function interacts with a separate real-time model data structure, you avoid unintentional interaction between the calls.

Eliminate Internal Data with Code Generation Optimizations

For more efficient code that consumes less memory, select the optimizations, such as **Default parameter behavior**, that you cleared earlier.

```
set_param('RollAxisAutopilot','DefaultParameterBehavior','Inlined',...
         'OptimizeBlockIOStorage','on',...
         'LocalBlockOutputs','on')
```

In this example, for simpler code, set **Code interface packaging** to `Nonreusable function`.

```
set_param('RollAxisAutopilot','CodeInterfacePackaging','Nonreusable function')
```

Generate code from the model.

```
slbuild('RollAxisAutopilot')

### Starting build procedure for: RollAxisAutopilot
### Successful completion of build procedure for: RollAxisAutopilot
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
RollAxisAutopilot	Code generated and compiled.	Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 18.309s

Now, `RollAxisAutopilot.h` does not define a structure for block inputs and outputs. For all of the internal signals in the model, the optimizations either eliminated storage or created local function variables instead of global structure fields.

The optimizations were not able to eliminate storage for the three block states, so the file continues to define the `DWork` structure type.

```
file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot.h');
rtwdemodbtype(file,...
    /* Block states (default storage) for system','} DW_RollAxisAutopilot_T;',1,1)

/* Block states (default storage) for system '<Root>' */
typedef struct {
    real32_T FixPtUnitDelay1_DSTATE;    /* '<S4>/FixPt Unit Delay1' */
    real32_T Integrator_DSTATE;        /* '<S1>/Integrator' */
    int8_T Integrator_PrevResetState;  /* '<S1>/Integrator' */
} DW_RollAxisAutopilot_T;
```

The code generated for the Discrete-Time Integrator block now stores state and output data only in the `DWork` structure.

```
file = fullfile('RollAxisAutopilot_grt_rtw','RollAxisAutopilot.c');
rtwdemodbtype(file,'/* Update for DiscreteIntegrator: '<S1>/Integrator''',...
    /* End of Update for DiscreteIntegrator: '<S1>/Integrator' */',1,1)
```

```

/* Update for DiscreteIntegrator: '<S1>/Integrator' incorporates:
 * Gain: '<S1>/IntGain'
 */
RollAxisAutopilot_DW.Integrator_DSTATE += 0.5F * rtb_TKSwitch * 0.025F;
if (RollAxisAutopilot_DW.Integrator_DSTATE >= 5.0F) {
    RollAxisAutopilot_DW.Integrator_DSTATE = 5.0F;
} else if (RollAxisAutopilot_DW.Integrator_DSTATE <= -5.0F) {
    RollAxisAutopilot_DW.Integrator_DSTATE = -5.0F;
}

RollAxisAutopilot_DW.Integrator_PrevResetState = (int8_T)tmp;

```

The optimizations also eliminated storage for the block parameters in the model. For example, in the Discrete-Time Integrator block, the **Upper saturation limit** and **Lower saturation limit** parameters are set to `intLim` and `-intLim`. `intLim` is a `Simulink.Parameter` object that stores the value 5. In the code generated for the Discrete-Time Integrator, these block parameters and `intLim` appear as inlined literal numbers `5.0F` and `-5.0F`.

If a model contains a parameter that the code generator cannot inline directly (for example, an array parameter), the code defines a structure type that represents the data. This *constant parameters* structure uses the `const` storage type qualifier, so some build toolchains can optimize the assembly code further.

The declaration of the structure, `ConstParam__model_`, is in `model.h`.

```

/* Constant parameters (auto storage) */
typedef struct {
    /* Expression: [1 2 3 4 5 6 7]
     * Referenced by: '<Root>/Constant'
     */
    real_T Constant_Value[7];

    /* Expression: [7 6 5 4 3 2 1]
     * Referenced by: '<Root>/Gain'
     */
    real_T Gain_Gain[7];
} ConstParam_model;

```

The declaration of the constant parameters, `model_ConstP`, is in `model_data.c` by default.

```

/* Constant parameters (auto storage) */
const ConstParam_model model_ConstP = {
    /* Expression: [1 2 3 4 5 6 7]
     * Referenced by: '<Root>/Constant'
     */
    { 1.0, 2.0, 3.0, 4.0, 5.0, 6.0, 7.0 },

    /* Expression: [7 6 5 4 3 2 1]
     * Referenced by: '<Root>/Gain'
     */
    { 7.0, 6.0, 5.0, 4.0, 3.0, 2.0, 1.0 }
};

```

`model_ConstP` is passed as an argument to referenced models.

Local Variables in the Generated Code

When you select the optimization **Configuration Parameters > Enable local block outputs**, the code generator attempts to yield more efficient code by representing internal signals as local variables instead of fields of a global structure. If the memory consumed by local variables risks exceeding the stack space available on your target hardware, consider indicating the maximum stack size by setting **Configuration Parameters > Maximum stack size (bytes)**. For more information, see [Maximum stack size \(bytes\)](#).

Appearance of Test Points in the Generated Code

A *test point* is a signal that is stored in a unique memory location. For information about including test points in your model, see [“Configure Signals as Test Points”](#).

When you generate code for models that include test points, the build process allocates a separate memory buffer for each test point. By default, test points are stored as members of a standard data structure such as `model_B`.

If you have Embedded Coder:

- You can control the default representation of test points by specifying code generation settings for the **Internal data** category of data in the Code Mapping Editor (see [“Configure Default Code Generation for Data”](#) (Embedded Coder)).
- You can specify that the build process ignore test points in the model, allowing optimal buffer allocation, by using the Ignore test point signals (Embedded Coder) parameter. Ignoring test points facilitates transitioning from prototyping to deployment and avoids accidental degradation of generated code due to workflow artifacts. See [Ignore test point signals \(Embedded Coder\)](#).

Virtual buses do not appear in generated code, even when associated with a test point. To display a bus in generated code, use a nonvirtual bus or a virtual bus converted to a nonvirtual bus by a Signal Conversion block.

Appearance of Workspace Variables in the Generated Code

Workspace variables are variables that you use to specify block parameter values in a model. Workspace variables include numeric MATLAB variables and Simulink.Parameter objects that you store in a workspace, such as the base workspace, or in a data dictionary.

When you set **Default parameter behavior** to `Tunable`, by default, workspace variables appear in the generated code as tunable fields of the global parameters structure. If you use such a variable to specify multiple block parameter values, the variable appears as a single field of the global parameters structure. The code does not create multiple fields to represent the block parameters. Therefore, tuning the field value during code execution changes the mathematical behavior of the model in the same way as tuning the value of the MATLAB variable or parameter object during simulation.

If you have Embedded Coder, you can control the default representation of workspace variables by specifying code generation settings for categories of parameter data in the Code Mapping Editor (see [“Configure Default Code Generation for Data”](#) (Embedded Coder)).

- The **Model parameters** category applies to variables that you store in a model workspace.
- The **External parameters** category applies to variables that you store in the base workspace or a data dictionary.

Promote Internal Data to the Interface

By default, the code generator assumes that other systems and components in your application do not need to access internal data. For example, internal data are subject to optimizations that can eliminate them from the generated code. For prototyping and testing purposes, you can access internal data by clearing the optimizations or by configuring test points and applying storage classes (see “Preserve Variables in Generated Code” on page 15-2). For optimized production code, configure individual data items to appear in the generated code as part of the model interface.

Data That You Can Promote

Depending on the reentrancy of the generated code, that is, the setting that you choose for **Code interface packaging**, you can configure each data item in a model to participate in the interface by appearing in the code as one of these entities:

- A global symbol, such as a global variable or a call to a specialized function
- A formal parameter (argument) of the generated entry-point functions

The table shows the mechanisms that each category of data can use to participate in the interface.

Category of Data	Appear as Global Symbol	Appear as Argument of Entry-Point Function
Root-level Inport or Outport block	Only for a nonreentrant model.	Yes.
Signal connecting two blocks	Only for a nonreentrant model.	Only for a reentrant model and only as a field of a structure. Alternatively, connect the signal to a root-level Outport block.
Block state	Only for a nonreentrant model.	Only for a reentrant model and only as a field of a structure.
Data store such as a Data Store Memory block	Yes.	Only for a reentrant model and only as a field of a structure.
Block parameter or parameter object such as <code>Simulink.Parameter</code>	Yes.	Only as a field of a structure.

Single-Instance Algorithm

For a single-instance algorithm (you set **Code interface packaging** to `Nonreusable function`), apply storage classes directly to individual data items by using the Model Data Editor or the Property Inspector. With a directly applied storage class, a data item appears in the code as a global symbol such as a global variable. The storage class also prevents optimizations from eliminating storage for the data item.

You can apply storage classes to signals, block states, and block parameters. (For block parameters, you apply storage classes indirectly through parameter objects such as `Simulink.Parameter`).

However, for a signal, consider connecting the signal to an Outport block at the root level of the model. Then, optionally, you can apply a storage class to the block. In the block diagram, the Outport block shows that the signal represents a system output.

For more information about storage classes, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.

Reentrant Algorithm

For a reentrant algorithm (you set **Code interface packaging** to `Reusable` function), use different techniques to configure data items to appear in the code as formal parameters (arguments) of the generated entry-point functions.

- For an internal signal, directly apply the storage class `Model default` (see “C Code Generation Configuration for Model Interface Elements” on page 16-2). If you have Embedded Coder, in the Code Mapping Editor, for the **Internal data** category, set the default storage class to `Default` or to a structured storage class that you define in an Embedded Coder Dictionary (see “Create Code Definitions for Use in the Code Mappings Editor” (Embedded Coder)). Alternatively, configure the signal as a test point (see “Appearance of Test Points in the Generated Code” on page 15-65). By default, the signal appears as a field of one of the standard data structures (see “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49). If you do not want the signal to appear in production code, use a test point so that you can later select the model configuration parameter **Ignore test point signals**.

Alternatively, connect the signal to an Outport block at the root level of the model. Connecting the signal to a root-level Outport block prevents optimizations from eliminating the signal from the code. To help with signal routing in a large model, use Goto and From blocks.

- For a block parameter, create a parameter object such as `Simulink.Parameter` and directly apply a storage class other than `Auto` to the object. The storage class prevents optimizations from inlining the parameter value in the code.

With the storage class, the parameter is shared between instances of the model, which are calls to the entry-point functions. The functions access the parameter data directly, as a global symbol, not an argument. To enable each instance of the model to use a different value for the parameter, configure the parameter as a model argument. For more information, see “Specify Instance-Specific Parameter Values for Reusable Referenced Model” on page 8-27.

For information about applying storage classes, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.

Control Default Representation of Internal Data (Embedded Coder)

By default, the code generator aggregates internal data that optimizations cannot eliminate, such as most state data, into standard structures such as the `DWork` structure. With Embedded Coder, you can control how the generated code stores this data.

Control Placement of Data in Memory by Inserting Pragmas

Use the Code Mapping Editor to specify a default memory section for each category of data such as states and signals (**Internal data**). In the generated code, your custom pragmas or other decorations surround the data definitions and declarations.

You can also partition the structures according to atomic subsystems in your model so that you can specify different default memory sections for the data of subroutines and other algorithmic subcomponents.

For more information, see “Control Data and Function Placement in Memory by Inserting Pragmas” (Embedded Coder).

Control Names of Types, Fields, and Global Variables for Standard Data Structures

You can control some characteristics of the standard data structures. For more information, see “Manage Replacement of Simulink Data Types in Generated Code” (Embedded Coder).

For additional control over structure characteristics, such as placement in generated code files, create your own structured storage class by using the Embedded Coder Dictionary. Then, apply the storage class to categories of data by using the Code Mapping Editor. The storage class removes the data from the standard structures, creating other structures that you can more finely control. For more information about applying default storage classes to categories of data, see “Configure Default Code Generation for Data” (Embedded Coder). For more information about creating a storage class, see “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder).

Organize Data into Structures According to Subcomponents

- In the standard data structures, to create substructures that contain the data for a single-instance (nonreentrant) subroutine or subcomponent, use an atomic subsystem to encapsulate the corresponding blocks. In the subsystem parameters, set **Function packaging** to Reusable function. For more information, see “Generate Modular Function Code for Nonvirtual Subsystems” (Embedded Coder).

Alternatively, encapsulate the blocks in a model and use a Model block. In the referenced model, set **Configuration Parameters > Model Referencing > Total number of instances allowed per top model** to Multiple. For more information, see “Generate Code for Model Reference Hierarchy” on page 8-2.

- To create separate, standalone structures that contain the data for a multi-instance (reentrant) subroutine or subcomponent, in the model, use an atomic subsystem to encapsulate the corresponding blocks. In the subsystem parameters, set **Function packaging** to Nonreusable function and select **Function with separate data**. For more information, see “Generate Modular Function Code for Nonvirtual Subsystems” (Embedded Coder).

Alternatively, encapsulate the blocks in a model and use a Model block. In the referenced model, choose one of these techniques:

- Set **Configuration Parameters > Model Referencing > Total number of instances allowed per top model** to One. For more information, see “Generate Code for Model Reference Hierarchy” on page 8-2.
- Set **Total number of instances allowed per top model** to Multiple and create a structured storage class by using an Embedded Coder Dictionary. Then, apply the storage class to categories of data by using the Code Mapping Editor. To create a storage class, see “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder). To apply the storage class to categories of data, see “Configure Default Code Generation for Data” (Embedded Coder).

Organize Signal and Parameter Data into Meaningful, Custom Structures and Substructures

To organize arbitrary signals and parameters into custom structures and substructures, create nonvirtual bus signals and parameter structures. Optionally, to prevent optimizations from eliminating the data from the code, set the storage class of a bus signal or parameter structure to a value other than `Auto` (the default setting).

As you add blocks to the model, you must explicitly place each new signal and parameter into a bus or a structure.

For more information, see “Organize Data into Structures in Generated Code” on page 15-147.

Create Separate Global Variables Instead of Structure Fields

To make a category of internal data appear in the generated code as separate, unstructured global variables instead of fields of a standard data structure, apply an unstructured storage class to the data category by using the Code Mapping Editor. For example, apply the storage class `ExportedGlobal`. However, if you generate multi-instance, reentrant code by setting the configuration parameter **Code interface packaging** to a value other than `Nonreusable function`, you cannot use this technique for some categories of data (see “Storage Classes and Reentrant, Multi-Instance Models and Components” on page 16-8).

To apply default storage classes to categories of data by using the Code Mapping Editor, see “Configure Default Code Generation for Data” (Embedded Coder). To choose a storage class, see “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75.

See Also

Related Examples

- “Data Structures in the Generated Code” on page 15-31
- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “C Code Generation Configuration for Model Interface Elements” (Embedded Coder)

Choose Data Configuration Approach

When you generate code from a model, the code includes design data such as signals, states, and parameters, which your application code can read from and write to. When deciding how you want this data represented in the generated code, consider:

- What types of data elements do I want to use in my model?
- Where do I want to store the data?
- What are my code generation objectives?
- How can I configure the data to meet my code generation objectives?

The answers to these questions depend on your model architecture (standalone model, standalone hierarchy of referenced models, or a system of components), your application type (reentrant or single-instance), and whether you are integrating your code with existing external code.

Data Element Categories

The possible data elements in your model fall under the data element types listed in this table.

Data Element Type	Description
Inports	Root-level input ports of a model, such as Inport and Bus Element In blocks.
Outports	Root-level output ports of a model, such as Outport and Bus Element Out blocks.
Model-scoped parameters	<ul style="list-style-type: none"> • Model parameter arguments — Variables in the model workspace that you configure as model arguments. These parameters are exposed at the model block to enable the parent model to provide different values for each model instance. • Model parameters — Parameters that are defined in the model workspace. Unlike model arguments, these parameters are shared by each instance of the model.
External parameters	Parameters that you define as objects in the base workspace or in a data dictionary. Multiple models in an application can use these parameters.

Data Element Type	Description
Data stores	<ul style="list-style-type: none"> Internal data (local data stores) — Data stores that are accessible from anywhere in a subsystem hierarchy that is at or below the level at which you define the data store. Defined graphically in a model by including a Data Store Memory block or by creating a signal object in the model workspace. Global data stores — Data stores that are defined by a signal object in the base workspace or in a data dictionary. Multiple models in an application can use these data stores. Shared local data stores — Data Store Memory blocks that have the block parameter Share across model instances set. These data stores are accessible only in the system where the Data Store Memory block is located. The data store value is shared across instances of the model.
Internal data	Local data, such as internal signals and block states. Includes local data stores.
Constants	Constant parameters and constant-value block output in a model.

The code configuration options available for your data are dependent on the category of data that you are configuring. Where you store the data determines if you configure it by using a data object or by using a code mapping that is saved with the model.

Where to Store Data Elements Used by Your Model

Where you store data for your model can depend on the type of model architecture that you are using and your modeling goals:

- MATLAB base workspace — Used for the temporary storage of data while experimenting with modeling techniques.
- Model workspace — Used to permanently store data that is local to a single model. Because the data is stored with the model, using this workspace can improve model portability. The model workspace is predominantly used to store parameters, arguments, and numeric variables. These elements are not typically stored in the model workspace:
 - Signal objects — You cannot configure a signal object in the model workspace for code generation. You do not need to define a signal object to configure a signal for code generation.
 - Data type objects.
- Simulink data dictionary — Used to permanently store global data, data types, and data that is shared between models.

For more information, see “Determine Where to Store Variables and Objects for Simulink Models”.

When to Configure a Data Element for Code Generation

Whether you customize the generation of your data for code generation depends on your objectives for the generated code. If you choose not to configure customizations, the code generator determines whether to eliminate or change the representation of the data in generated code for optimization purposes.

There are many reasons you might choose to configure a data item. Configuring data prevents optimizations that might eliminate the data item from the generated code and enables you to configure how the generated code stores and defines the data item. For example, if you configure a data item, you can:

- Tune parameters.
- Monitor signals and states during execution.
- Configure data placement in memory.
- Exchange data with external code.
- Improve readability and traceability of generated code.

Configure a Data Element for Code Generation

To configure your data for code generation, apply a storage class. A storage class is a code generation setting that enables you to control the appearance and placement of a data element in the generated code and prevents optimizations that might eliminate storage for that data element. You can use storage classes available with Simulink Coder and Embedded Coder. If you have special requirements that are not met by these storage classes and you are generating code for an ERT-based target, you can define and use a new storage class. For more information on storage classes and storage class properties, see “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75.

When to Configure Data by Using a Data Object

For most model-owned data, you can map a storage class directly to a data item by using the Code Mappings editor or code mapping API. You must configure some data that your model uses for code generation by using a data object, including:

- Parameters — Includes parameters in the base workspace, model workspace, or data dictionary. Configure parameters by using a `Simulink.Parameter` object.
- Global data stores — Configure by using a signal object.

Before configuring numeric variables for code generation, you must convert them to `Simulink.Parameter` objects.

Configure Model-Owned Data by Using Code Mapping

Model-owned data consists of output blocks, signal lines, block states, data stores, and parameter objects in the model workspace. For this data, you can map a storage class to a data element in the coder app by using the Code Mappings editor or by using the code mappings API. For convenience, you can also configure external data objects in the Code Mappings editor, though for external data objects, the storage class is stored on the data object and not in the model. To see these objects, in the editor, click the **Refresh** link to the right of the category name.

To configure model-owned data, you first configure default code generation settings for categories of data elements. Using a default mapping saves time and reduces the risk of introducing errors in the code, especially when your model contains a significant number of elements in a particular category.

- In the **C Code** tab, select **Code Interface > Default Code Mappings**.
- On the **Data Defaults** tab, select a data category, then select a default storage class for that category. If you do not specify a storage class for a category, the `Default` storage class applies. In

this case, the code generator places the code for the category of data elements in standard structures. Other storage class options include:

- Relevant built-in storage classes, such as `ExportedGlobal`.
- Relevant storage classes in an available package, such as `ImportFromFile` (requires Embedded Coder).
- Storage classes defined in an Embedded Coder Dictionary (requires Embedded Coder). For more information on available storage classes, see “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75.
- In the Property Inspector, configure the properties for that storage class. (For example, header and definition files).

For more information, see “C Code Generation Configuration for Model Interface Elements” on page 16-2 “C Code Generation Configuration for Model Interface Elements” (Embedded Coder).

For certain categories of data, including inports, outports, data stores, and block states, the default mapping is likely the only configuration you need to perform. But for data elements such as parameters and internal signals, you might need to specify an individual mapping to prevent the data from being optimized away. To apply a storage class to an individual data element:

- In the Code Mappings editor, find the data element on its corresponding category tab (for example, to configure an individual parameter, select the **Parameters** tab).
- Select a storage class for the data element. To apply the default configuration, select the storage class `Model default`.
- In the Property Inspector, configure the properties for that storage class. (For example, header and definition files).

For more information, see “C Code Generation Configuration for Model Interface Elements” on page 16-2 “C Code Generation Configuration for Model Interface Elements” (Embedded Coder).

Configure External Data

Configure external data objects for code generation by using the Code Mappings editor or the Model Explorer. Select a data object in the **Contents** pane. Then, in the **Dialog** pane, on the **Code Generation** tab, select a storage class and set properties for that storage class. For convenience, you can also configure external data objects in the Code Mappings editor. To view and configure these objects, in the editor, click the **Refresh** link to the right of the category name.

You can configure external data objects programmatically. For example, to configure a `Simulink.Parameter` object for code generation by applying the storage class `ExportedGlobal`, you can use code similar to:

```
P = Simulink.Parameter;
P.Value = 5;
P.DataType = 'int32';
P.CoderInfo.StorageClass = 'ExportedGlobal';
```

Load Storage Class Packages into Embedded Coder Dictionary

To apply a storage class from a storage class package other than Simulink (such as a package that you create):

- 1 Open the **Embedded Coder** app.

- 2** Open the Embedded Coder Dictionary. On the **C Code** tab, select **Code Interface > Embedded Coder Dictionary**.
- 3** In the Embedded Coder Dictionary, click **Storage Class**.
- 4** Below the **Storage Classes** table, click **Manage Packages**.
- 5** In the Manage Packages dialog box, select a package. If the package of interest is not in the list, click **Refresh**. Then, select your package.
- 6** Click **Load**.
- 7** Close the Embedded Coder Dictionary.
- 8** Use the Code Mappings editor or code mappings API to configure the model's interface elements.

For more information, see **Embedded Coder Dictionary**, **Code Mappings Editor - C**, and `coder.mapping.api.CodeMapping`.

To apply the storage class by using a data object (required parameters), instead of creating a `Simulink.Signal` or `Simulink.Parameter` object, create a `myPackage.Signal` or `myPackage.Parameter` object. To create data objects from your package, see “Create Data Objects from Another Data Class Package”. For an example that shows how to create and apply your own storage class, see “Create and Apply a Storage Class” (Embedded Coder).

See Also

Code Mappings Editor - C | `coder.mapping.api.CodeMapping`

More About

- “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75
- “Configure Default C Code Generation for Categories of Data Elements and Functions” (Embedded Coder)
- “C Code Generation Configuration for Model Interface Elements” on page 16-2

Choose Storage Class for Controlling Data Representation in Generated Code

A storage class is a code generation setting that you apply to data, such as parameters, signals, and states. During code configuration, use a storage class to control the appearance and placement of a data element in the generated code and to prevent optimizations from eliminating storage for that data element.

For model data, you can apply a storage class directly to a model data element by using the Code Mappings editor or code mappings API. Using either the editor or the API, apply a default storage class for categories of data, and then override that setting, as needed, for individual data elements.

For external data, you apply a storage class to a data object by using the Model Explorer or programmatically by using functions such as `get_param` and `set_param`.

Once you have specified a storage class for an element or category of elements, you can set properties for that storage class, such as header files, definition files, and memory sections.

Storage Class Properties

The property settings for a storage class define how and where the code generator represents data in the generated code. The properties that the user of a storage class can configure vary depending on the storage class. For most individual elements, the user can configure an identifier for naming the data element in the code. To identify properties that the user can configure for predefined storage classes, see the following information on specific storage classes.

When you create a storage class by using the Custom Storage Class Designer, you can choose whether the user of the storage class can specify additional settings. See “Allow Users of Storage Class to Specify Property Value” (Embedded Coder).

Default Storage Class

By default, the storage class for an individual data element is `Auto`. By using this default value, you are choosing to have the code generator handle storage class selection and application. In this case, the data element is subject to code generation optimizations, which can eliminate the element from the code or change the representation of the element. If optimizations do not eliminate the data element, the element appears as a field of a standard data structure in the generated code.

When using the Code Mappings editor or the code mappings API, a data element configured with storage class `Auto` that is not eliminated by optimizations acquires the default code generation settings for the corresponding data category. If you do not specify a code configuration for a category of data elements, the storage class is `Default`. When a data element category is using this storage class, data elements in that category appear as a field of a standard data structure in the generated code.

Built-in and Predefined Storage Classes

To prevent optimizations from eliminating storage for a data element, you can choose a storage class for the element based on your code generation requirements. Available storage classes include built-in storage classes, predefined storage classes in the `Simulink` package, and might also include other project-specific storage classes defined in an Embedded Coder Dictionary. If you have special

requirements that are not met by the listed storage classes and you are generating code for an ERT-based target, you can define and use a new storage class. See “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder) and “Create Storage Classes by Using the Custom Storage Class Designer” (Embedded Coder).

For Simulink Coder, you can choose from these built-in and predefined storage classes.

Requirements	Storage Class
Enable optimizations, potentially generating more efficient code.	“Auto” on page 15-77 (Individual data elements)
For data elements that cannot be optimized away, represent data as a field of a standard data structure.	“Default” on page 15-77 (Default mapping)
Prevent optimizations from eliminating storage for a data element.	“Model Default” on page 15-77 (Individual mapping)
Access data from a standalone global variable. Generated code contains the variable declaration and definition.	“ExportedGlobal” on page 15-78
Access data from a standalone global variable. Generated code contains the variable declaration. Your external code provides the definition.	“ImportedExtern, ImportedExternPointer” on page 15-78

If you have Embedded Coder, you can choose from these additional predefined storage classes available in the Simulink package.

Requirements	Storage Class
Prevent optimizations from eliminating storage for a data element.	“Dictionary Default” on page 15-77 (Default mapping)
Generate standalone variables.	<ul style="list-style-type: none"> • “ExportToFile” on page 15-80 • “ImportFromFile” on page 15-82 • “FileScope” on page 15-81 • “Localizable” on page 15-82 • “Reusable” on page 15-83 • “Const, Volatile, and ConstVolatile” on page 15-79
Generate variables as fields of a structure.	<ul style="list-style-type: none"> • “Struct” on page 15-83 • “Bitfield” on page 15-79
Generate variables that have a value that is fixed during code compilation.	<ul style="list-style-type: none"> • “Define, ImportedDefine” on page 15-80 • “CompilerFlag” on page 15-79
Generate variables that you access through a function call.	“GetSet” on page 15-81
Generate variables for single-instance data and generate structures for multi-instance data.	“MultiInstance” (Embedded Coder)

These storage classes are examples of storage classes that you can add to an Embedded Coder Dictionary. These examples are defined and made available when you prepare a model for code generation by using the Quick Start tool.

Requirements	Storage Class
Generate global structures that contain parameter data and signal or state data.	"ParamStruct, SignalStruct" on page 15-84

Note Signal objects defined in the base workspace or a data dictionary with a non-auto storage class are not supported for models that are configured with an ERT-based system target file and service code interface.

Auto

When you select this storage class, the code generator handles how to represent the data in the generated code and can enable optimizations to operate on the data element, potentially generating more efficient code.

Auto is the default storage class setting for each data element in a model. The data element is subject to code generation optimizations, which can eliminate the element from the code or change the representation of the element. For information about these optimizations, such as those on the **Configuration Parameters > Code Generation > Optimization** pane, see "How Generated Code Stores Internal Signal, State, and Parameter Data" on page 15-49.

Optimizations cannot eliminate some data, such as most block states, from the code. The remaining data acquires a default storage class that you specify on the **Data Defaults** tab of the Code Mappings editor (see "Configure Default Code Generation for Data" (Embedded Coder)). If a data element cannot be eliminated, the name of the element in the code is based on naming rules that you specify by using model configuration parameters. See "Identifier Format Control" (Embedded Coder)—requires Embedded Coder.

Default

On the **Data Defaults** tab of the Code Mappings editor, **Default** is the default storage class setting for each data element category. If you leave the storage class setting for a category at this value, data elements that are not subject to code generation optimizations appear as a field of a standard data structure. See "How Generated Code Stores Internal Signal, State, and Parameter Data" on page 15-49.

Model Default

When you configure an individual data element for code generation, use the **Model Default** storage class to prevent optimizations from eliminating storage for a data element. See "How Generated Code Stores Internal Signal, State, and Parameter Data" on page 15-49. With this storage class setting, the data element acquires the default storage class that you specify for the corresponding data category on the **Data Defaults** tab of the Code Mappings editor. The name of the data element in the code is the same as the name in the model.

Dictionary Default

In Embedded Coder, if you link a model to a data dictionary, which includes a coder dictionary that configures default code definitions for categories of data, you can use the Code Mappings editor to apply the dictionary defaults. On the **Data Defaults** tab, select a category, then set the storage class

to `Dictionary Default`. For more information on configuring a default code mapping in a coder dictionary, see “Configure Default Code Mapping in a Shared Dictionary” (Embedded Coder).

ExportedGlobal

Use this built-in storage class to generate a global variable definition and declaration. By default, the name of the variable is the name of the data element. The code declares the variable in the generated file `model.h`, which you can include (`#include`) in your external code. The code generator does not optimize this global variable from the generated code.

When you use this storage class, you can also configure this property.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.

For an example that uses this storage class, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.

ImportedExtern, ImportedExternPointer

To make a data element in a model represent a global variable that your external code defines, use the built-in storage class `ImportedExtern`. The generated algorithmic code uses the variable without defining it.

Generate code that reads from and writes to a global variable defined by your external code. The generated code declares the variable in the generated file `model_private.h` so that the model entry-point functions can read and write to the variable.

For example, you can apply the storage class `ImportedExtern` to a signal line, block state, or parameter object. For imported data:

- The generated code does not initialize parameter data. Your code must initialize imported parameter data.
- The generated initialization functions dynamically initialize some signal and state data. Unlike data that the generated code allocates, the code does not initialize imported signal or state data to a stored value of zero. Instead, the code immediately initializes the data to the real-world nonzero value that you specify in Simulink.

Note The code generator initializes zero values that you specify in an Initialize Function block, even if those values have imported scope.

When your external code defines a data element and provides a pointer for accessing that data, use `ImportedExternPointer`. The generated code reads from and writes to that pointer. The generated code declares the variable in the generated file `model_private.h` and reads and writes to the data by dereferencing the pointer.

When you use these storage classes, you can also configure this property.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.

Bitfield

Use this storage class to generate a structure that stores Boolean data in named bit fields.

When you use this storage class, you can also configure these properties.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.
StructName	Name of the structure for the data element in the generated code.

For an example that uses this storage class, see “Bitfields” (Embedded Coder).

CompilerFlag

Use this storage class to support preprocessor conditionals defined by using a compiler flag or option.

If you build the generated code by using Embedded Coder, to specify the compiler option, you can use the model configuration parameter **Configuration Parameters > Code Generation > Custom Code > Code information > Defines**. See Code Generation Pane: Custom Code: Additional Build Information: Defines.

When you use this storage class, you can also configure this property.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.

For an example that uses this storage class, see “Generate Preprocessor Conditionals for Variant Systems” (Embedded Coder).

Const, Volatile, and ConstVolatile

Use these storage classes to generate a global variable definition and declaration with the `const`, `volatile`, or `const` and `volatile` type qualifiers.

When you use these storage classes, you can also configure these properties.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.
HeaderFile	Source header file that contains declarations for global data read by the data element and external code.

Property	Description
DefinitionFile	Source definition file that contains definitions for global data read by the data element and external code.
Owner	Owner of global data that is defined with the code generated for one of multiple components that use that data.
PreserveDimensions	When model configuration parameter Array layout is set to Row-major, a flag that indicates whether to preserve dimensions of a data element that is represented in generated code as a multidimensional array.

For an example that uses this storage class, see “Type Qualifiers” (Embedded Coder).

Define, ImportedDefine

Use the Define storage class to generate a macro (`#define` directive) such as `#define myParam 5`.

Use ImportedDefine to generate code that uses a macro (`#define` directive) defined in a header file in your external code. For an example, see “Macro Definitions (`#define`)” (Embedded Coder).

When you use these storage classes, you can also configure these properties.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.
HeaderFile	Source header file that contains declarations for global data that is read by the data element and external code.

For an example that uses this storage class, see “Macro Definitions (`#define`)” (Embedded Coder).

ExportToFile

Generate a global variable definition and declaration to an external file. You can specify the name and placement of the files that define and declare the variable. See “Control Placement of Global Data Definitions and Declarations in Generated Files” (Embedded Coder).

When you use this storage class, you can also configure these properties.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.
HeaderFile	Source header file where the code generator places the declarations for global data read by the data element and external code.

Property	Description
DefinitionFile	Source definition file where the code generator places the definitions for global data read by the data element and external code. The code generator honors this setting only for single-instance models. The setting is ignored for reusable multi-instance models because the data definition is handled at the parent level.
Owner	A component in the model hierarchy where the code generator places a global data definition instead of placing it in the top component of the hierarchy. Requires that you set the model configuration parameter Use owner from data object for data definition placement .
PreserveDimensions	When model configuration parameter Array layout is set to Row-major, a flag that indicates whether to preserve dimensions of a data element that is represented in generated code as a multidimensional array.

For an example that uses this storage class, see “Definition, Initialization, and Declaration of Parameter Data” (Embedded Coder).

FileScope

Use this storage class to generate a global variable definition and declaration that has the `static` type qualifier. In the generated code, the scope of the variable is limited to the current file, which is typically `model.c`.

In a model reference hierarchy, if a referenced model uses a parameter object (such as `Simulink.Parameter`) that you create in the base workspace or a data dictionary, you cannot apply `FileScope` to the object. As a workaround, move the parameter object into the model workspace of the referenced model. Then, you can use `FileScope`.

When you use this storage class, you can also configure these properties.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.
PreserveDimensions	When model configuration parameter Array layout is set to Row-major, a flag that indicates whether to preserve dimensions of a data element that is represented in generated code as a multidimensional array.

GetSet

Use this storage class to generate code that interacts with data by calling custom accessor functions. Your external code defines the data and provides the function definitions.

When you use this storage class, you can also configure these properties.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.
HeaderFile	Source header file that contains declarations for the <code>get</code> and <code>set</code> functions.
GetFunction	Data element that appears in the generated code as a call to a specified <code>get</code> function.
SetFunction	Data element that appears in the generated code as a call to a specified <code>set</code> function.
PreserveDimensions	When model configuration parameter Array layout is set to <code>Row-major</code> , a flag that indicates whether to preserve dimensions of a data element that is represented in generated code as a multidimensional array.

For an example that uses this storage class, see “Access Data Through Functions with Storage Class `GetSet`” (Embedded Coder).

ImportFromFile

Use this storage class to generate code that reads from and writes to a global variable defined by your external code. `ImportFromFile` is similar to `ExportToFile`, but the generated code does not define the variable.

When you use this storage class, you can also configure these properties.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.
HeaderFile	Source header file that contains declarations for global data that is read by the data element and external code. To avoid linker errors, you must add include guards to the beginning of the header file.
PreserveDimensions	When model configuration parameter Array layout is set to <code>Row-major</code> , a flag that indicates whether to preserve dimensions of a data element that is represented in generated code as a multidimensional array.

For an example that uses this storage class, see “Integrate External Application Code with Code Generated from PID Controller” (Embedded Coder).

Localizable

For signals, if possible, minimize the use of global storage by using local variables.

- If the variable is used in a single function, the variable is local to that function.
- If the variable is used in a single file, the variable is local to that file.
- If the variable is used in more than one function or file, the variable is a global.

Generating local variables prevents the code generator from removing the variables from the generated code.

When you use this storage class, you can also configure these properties.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.
PreserveDimensions	When model configuration parameter Array layout is set to Row-major, a flag that indicates whether to preserve dimensions of a data element that is represented in generated code as a multidimensional array.

For an example that uses this storage class, see “Generate Local Variables with Localizable Storage Class” (Embedded Coder).

Reusable

Use this storage class to reuse the same variable for multiple independent signals in a model. The code generator stores intermediate calculations of a data path (a series of connected blocks) in a single, reused global variable.

When you use this storage class, you can also configure this property.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.

For an example that uses this storage class, see “Specify Buffer Reuse for Signals in a Path” (Embedded Coder).

Struct

Use this storage class to generate a global structure with a name that you can specify.

When you use this storage class, you can also configure these properties.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.
StructName	Name of the structure for the data element in the generated code.

For an example that uses this storage class, see “Organize Parameter Data into a Structure by Using Struct Storage Class” (Embedded Coder) and “Structures of Signals” (Embedded Coder).

Multinstance

To generate unstructured variables for single-instance data and structures for multi-instance data, use this storage class. When you apply this storage class to a data item, the Embedded Coder Dictionary determines if it is a single-instance storage class or a multi-instance storage class. The dictionary determines the type of class by the type of data and by the context of the model within the model reference hierarchy. You can duplicate this storage class to create a copy that you can edit.

When you apply this storage class to an individual data element, you can also configure the **Identifier** property. The **Identifier** property is an identifier string that the code generator uses to name the data element in the generated code.

For an example that uses this storage class, see “Flexible Storage Class for Different Model Hierarchy Contexts” (Embedded Coder).

ParamStruct, SignalStruct

Use these storage classes to generate global structures that contain parameter and signal or state data, respectively. In a hierarchy of components (referenced models or atomic subsystems), you can use these storage classes to create a corresponding hierarchy of structures. These storage classes appear in the Code Mappings editor only after preparing a model for code generation by using the Quick Start tool.

These storage classes are available only for model-owned data.

When you use these storage classes, you can also configure this property.

Property	Description
Identifier	Identifier string that the code generator uses to name the data element in the generated code.

Storage Class Limitations

- When you use storage classes in the Code Mappings editor (Embedded Coder) or code mappings API, some limitations apply. See “Limitations” (Embedded Coder).
- Data objects cannot use an Embedded Coder storage class and a multiword data type.
- For Embedded Coder storage classes in models that use referenced models:
 - If you apply a grouped storage class, such as `Struct` or `Bitfield`, to multiple data items, you must set the storage class **Data scope** property to `Imported` and you must provide the data declaration in an external header file. Grouped storage classes use a single variable in the generated code to represent multiple data objects.
 - If a parameter object exists in the base workspace or a data dictionary, and a referenced model uses the object, you cannot apply the storage class `FileScope`. To apply this storage class to the parameter object, move the object into the model workspace of the referenced model.
- You cannot apply the storage class `FileScope` to data items used by a data exchange interface (C API, external mode, or ASAP2) or MAT-file logging. File-scoped data is not externally accessible.
- You cannot apply the storage class `FileScope` to data that is used in multiple files.

See Also

Related Examples

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Organize Parameter Data into a Structure by Using Struct Storage Class” (Embedded Coder)
- “Configure Default C Code Generation for Categories of Data Elements and Functions” (Embedded Coder)

- “Control Data and Function Interface in Generated Code” on page 15-44
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Use Enumerated Data in Generated Code

In this section...

“Enumerated Data Types” on page 15-86

“Specify Integer Data Type for Enumeration” on page 15-86

“Customize Enumerated Data Type” on page 15-87

“Control Enumerated Type Implementation in Generated Code” on page 15-91

“Type Casting for Enumerations” on page 15-92

“Enumerated Type Limitations” on page 15-92

Enumerated Data Types

Enumerated data is data that is restricted to a finite set of values. An *enumerated data type* is a MATLAB class that defines a set of *enumerated values*. Each enumerated value consists of an *enumerated name* and an *underlying integer* which the software uses internally and in generated code. The following is a MATLAB class definition for an enumerated data type named `BasicColors`, which is used in the examples in this section.

```
classdef BasicColors < Simulink.IntEnumType
    enumeration
        Red(0)
        Yellow(1)
        Blue(2)
    end
end
```

For basic information about enumerated data types and their use in Simulink models, see “Use Enumerated Data in Simulink Models”. For information about enumerated data types in Stateflow charts, see “Define Enumerated Data Types” (Stateflow).

Specify Integer Data Type for Enumeration

When you specify a data type for your enumeration, you can:

- Control the size of enumerated data types in the generated code by specifying a superclass.
- Reduce RAM/ROM usage.
- Improve code portability.
- Improve integration with legacy code.

You can specify these integer data types:

- `int8`
- `uint8`
- `int16`
- `uint16`
- `int32`
- `Simulink.IntEnumType`. Specify values in the range of the signed integer for your hardware platform.

Use a Class Definition in a MATLAB File

To specify an integer data type size, derive your enumeration class from the integer data type.

```
classdef Colors < int8
    enumeration
        Red(0)
        Green(1)
        Blue(2)
    end
end
```

The code generator generates this code:

```
typedef int8_T Colors;

#define Red      ((Colors)0)
#define Green   ((Colors)1)
#define Blue     ((Colors)2)
```

Use the Function Simulink.defineIntEnumType

To specify an integer data type size, specify the name-value pair `StorageType` as the integer data type.

```
Simulink.defineIntEnumType('Colors',{'Red','Green','Blue'},...
[0;1;2],'StorageType','int8')
```

The code generator generates this code:

```
typedef int8_T Colors;

#define Red      ((Colors)0)
#define Green   ((Colors)1)
#define Blue     ((Colors)2)
```

Customize Enumerated Data Type

When you generate code from a model that uses enumerated data, you can implement these static methods to customize the behavior of the type during simulation and in generated code:

- `getDefaultValue` — Specifies the default value of the enumerated data type.
- `getDescription` — Specifies a description of the enumerated data type.
- `getHeaderFile` — Specifies a header file where the type is defined for generated code.
- `getDataScope` — Specifies whether generated code exports or imports the enumerated data type definition to or from a separate header file.
- `addClassNameToEnumNames` — Specifies whether the class name becomes a prefix in generated code.

The first of these methods, `getDefaultValue`, is relevant to both simulation and code generation, and is described in “Specify a Default Enumerated Value”. The other methods are relevant only to code generation. To customize the behavior of an enumerated type, include a version of the method in the `methods(Static)` section of the enumeration class definition. If you do not want to customize the type, omit the `methods(Static)` section. The table summarizes the methods and the data to supply for each one.

Static Method	Purpose	Default Value Without Implementing Method	Custom Return Value
getDefaultValue	Specifies the default enumeration member for the class.	First member specified in the enumeration definition	A character vector containing the name of an enumeration member in the class (see “Instantiate Enumerations”).
getDescription	Specifies a description of the enumeration class.	' '	A character vector containing the description of the type.
getHeaderFile	Specifies the name of a header file. The method <code>getDataScope</code> determines the significance of the file.	' '	A character vector containing the name of the header file that defines the enumerated type. By default, the generated <code>#include</code> directive uses the preprocessor delimiter " instead of < and >. To generate the directive <code>#include <myTypes.h></code> , specify the custom return value as ' <code><myTypes.h></code> '.
getDataScope	Specifies whether generated code exports or imports the definition of the enumerated data type. Use the method <code>getHeaderFile</code> to specify the generated or included header file that defines the type.	'Auto'	One of: 'Auto', 'Exported', or 'Imported'.
addClassNameToEnumNames	Specifies whether to prefix the class name in generated code.	false	true or false.

Specify a Description

To specify a description for an enumerated data type, include this method in the `methods(Static)` section of the enumeration class:

```
function retVal = getDescription()
% GETDESCRIPTION Optional description of the data type.
    retVal = 'description';
end
```

Substitute a MATLAB character vector for *description*. The generated code that defines the enumerated type includes the specified description.

Import Type Definition in Generated Code

To prevent generated code from defining an enumerated data type, which allows you to provide the definition in an external file, include these methods in the `methods(Static)` section of the enumeration class:

```
function retVal = getHeaderFile()
    % GETHEADERFILE Specifies the file that defines this type in generated code.
    % The method getDataScope determines the significance of the specified file.
    retVal = 'imported_enum_type.h';
end

function retVal = getDataScope()
    % GETDATASCOPE Specifies whether generated code imports or exports this type.
    % Return one of:
    % 'Auto':    define type in model_types.h, or import if header file specified
    % 'Exported': define type in a generated header file
    % 'Imported': import type definition from specified header file
    % If you do not define this method, DataScope is 'Auto' by default.
    retVal = 'Imported';
end
```

Instead of defining the type in `model_types.h`, which is the default behavior, generated code imports the definition from the specified header file using a `#include` statement like:

```
#include "imported_enum_type.h"
```

Generating code does not create the imported header file. You must provide the header file, using the file name specified by the method `getHeaderFile`, that defines the enumerated data type.

To create a Simulink enumeration that corresponds to your existing C-code enumeration, use the `Simulink.importExternalCTypes` function.

Export Type Definition in Generated Code

To generate a separate header file that defines an enumerated data type, include these methods in the `methods(Static)` section of the enumeration class:

```
function retVal = getDataScope()
    % GETDATASCOPE Specifies whether generated code imports or exports this type.
    % Return one of:
    % 'Auto':    define type in model_types.h, or import if header file specified
    % 'Exported': define type in a generated header file
    % 'Imported': import type definition from specified header file
    % If you do not define this method, DataScope is 'Auto' by default.
    retVal = 'Exported';
end

function retVal = getHeaderFile()
    % GETHEADERFILE Specifies the file that defines this type in generated code.
    % The method getDataScope determines the significance of the specified file.
    retVal = 'exported_enum_type.h';
end
```

Generated code exports the enumerated type definition to the generated header file `exported_enum_type.h`.

Add Prefixes To Class Names

By default, enumerated values in generated code have the same names that they have in the enumeration class definition. Alternatively, your code can prefix every enumerated value in an enumeration class with the name of the class. You can use this technique to prevent identifier conflicts or to improve the readability of the code. To specify class name prefixing, include this method in the `methods(Static)` section of an enumeration class:

```

function retVal = addClassNameToEnumNames()
% ADDCLASSNAMETOENUMNAMES Specifies whether to add the class name
% as a prefix to enumeration member names in generated code.
% Return true or false.
% If you do not define this method, no prefix is added.
retVal = true;
end

```

Specify the return value as `true` to enable class name prefixing or as `false` to suppress prefixing. If you specify `true`, each enumerated value in the class appears in generated code as *EnumTypeName_EnumName*. For the example enumeration class `BasicColors` in “Enumerated Data Types” on page 15-86, the data type definition in generated code might look like this:

```

#ifndef _DEFINED_TYPEDEF_FOR_BasicColors_
#define _DEFINED_TYPEDEF_FOR_BasicColors_

typedef enum {
    BasicColors_Red = 0,          /* Default value */
    BasicColors_Yellow = 1,
    BasicColors_Blue = 2,
} BasicColors;

#endif

```

The enumeration class name `BasicColors` appears as a prefix for each of the enumerated names.

Control Use of Duplicate Enumeration Member Names

When you import the enumeration data from a header file, you can control the use of duplicate enumeration member names during code generation. Duplicate enumeration member names improve the readability of the code. Use the model configuration parameter **Duplicate enumeration member names** to allow duplicate enumeration member names in different enumeration types during code generation or to generate an error or a warning message. You can use duplicate enumeration member names only if two enumerations have the same `StorageType` and have these specifications:

- `DataScope` set to 'Imported'
- `StorageType` set to 'int8', 'int16', 'int32', 'uint8', 'uint16', or 'uint32'
- `Value` is the same

For example:

```

typedef int32_T enum {
    Red = 0,
    Yellow = 1,
    Blue = 2,
}A;

typedef int32_T enum {
    Black = 0,
    Yellow = 1,
    White = 2,
}B;

```

You can have a `Yellow` enumeration member in enumeration A and B without prefixing the member name with a class name to improve the readability of your code.

Control Enumerated Type Implementation in Generated Code

Suppose that you define an enumerated type `BasicColors`. You can specify that the generated code implement the type definition by using:

- An `enum` block. The native integer type of your hardware is the underlying integer type for the enumeration members.
- A `typedef` statement and a series of `#define` macros. The `typedef` statement bases the enumerated type name on a specific integer data type, such as `int8`. The macros associate the enumeration members with the underlying integer values.

Implement Enumerated Type by Using `enum` Block

To implement the type definition by using an `enum` block:

- In Simulink, define the enumerated type by using a `classdef` block in a script file. Derive the enumeration from the type `Simulink.IntEnumType`.
- Alternatively, use the function `Simulink.defineIntEnumType`. Do not specify the property `StorageType`.

When you generate code, the type definition appears in an `enum` block.

```
#ifndef _DEFINED_TYPEDEF_FOR_BasicColors_
#define _DEFINED_TYPEDEF_FOR_BasicColors_

typedef enum {
    Red = 0,           /* Default value */
    Yellow,
    Blue,
} BasicColors;

#endif
```

Implement Enumerated Type Using a Specific Integer Type

To implement the type definition using a `typedef` statement and `#define` macros:

- In Simulink, define the enumerated type using a `classdef` block in a script file. Derive the enumeration from a specific integer type such as `int8`.
- Alternatively, use the function `Simulink.defineIntEnumType`. Specify the property `StorageType` using a specific integer type such as `int8`.

When you generate code, the type definition appears as a `typedef` statement and a series of `#define` macros.

```
#ifndef _DEFINED_TYPEDEF_FOR_BasicColors_
#define _DEFINED_TYPEDEF_FOR_BasicColors_

typedef int8_T BasicColors;

#define Red ((BasicColors)0)           /* Default value */
#define Yellow ((BasicColors)1)
#define Blue ((BasicColors)2)

#endif
```

By default, the generated file `model_types.h` contains enumerated type definitions.

Type Casting for Enumerations

Safe Casting

A Simulink Data Type Conversion block accepts a signal of integer type. The block converts the input to one of the underlying values of an enumerated type.

If the input value does not match an underlying value of an enumerated type value, Simulink inserts a safe cast to replace the input value with the enumerated type default value.

Enable and Disable Safe Casting

You can enable or disable safe casting for enumerations during code generation for a Simulink Data Type Conversion block or a Stateflow block.

To control safe casting, enable or disable the **Saturate on integer overflow** block parameter. The parameter works as follows:

- **Enabled:** Simulink replaces a nonmatching input value with the default value of the enumerated values during simulation. The software generates a safe cast function during code generation.
- **Disabled:** For a nonmatching input value, Simulink generates an error during simulation. The software omits the safe cast function during code generation. In this case, the code is more efficient. However, the code may be more vulnerable to run-time errors.

Safe Cast Function in Generated Code

This example shows how the safe cast function `int32_T ET08_safe_cast_to_BasicColors` for the enumeration `BasicColors` appears in generated code when generated for 32-bit hardware.

```
static int32_T ET08_safe_cast_to_BasicColors(int32_T input)
{
    int32_T output;
    /* Initialize output value to default value for BasicColors (Red) */
    output = 0;
    if ((input >= 0) && (input <= 2)) {
        /* Set output value to input value if it is a member of BasicColors */
        output = input;
    }
    return output;
}
```

Through this function, the enumerated type's default value is used if the input value does not match one of underlying values of the enumerated type's values.

If the block's **Saturate on integer overflow** parameter is disabled, this function does not appear in generated code.

Enumerated Type Limitations

- Generated code does not support logging enumerated data.
- For `uint32`-based enumerations, enumeration values must be less than or equal to `intmax('int32')`.

See Also

[enumeration](#) | [Simulink.defineIntEnumType](#) | [Simulink.data.getEnumTypeInfo](#)

Related Examples

- “Use Enumerated Data in Simulink Models”
- “Simulink Enumerations”
- “Exchange Structured and Enumerated Data Between Generated and External Code” (Embedded Coder)

Data Stores in Generated Code

In this section...


- “About Data Stores” on page 15-94
- “Generate Code for Data Store Memory Blocks” on page 15-94
- “Storage Classes for Data Store Memory Blocks” on page 15-95
- “Data Store Buffering in Generated Code” on page 15-96
- “Data Stores Shared by Instances of a Reusable Model” on page 15-99
- “Structures in Generated Code Using Data Stores” on page 15-99

About Data Stores

A data store contains data that is accessible in a model hierarchy at or below the level in which the data store is defined. Data stores can allow subsystems and referenced models to share data without having to use I/O ports to pass the data from level to level. See “Data Stores with Data Store Memory Blocks” for information about data stores in Simulink. This section provides additional information about data store code generation.

Generate Code for Data Store Memory Blocks

To control the code generated for a Data Store Memory block, apply a storage class to the data store. You can associate a Data Store Memory block with a signal object that you store in a workspace or data dictionary, and control code generation for the block by applying the storage class to the object:

- 1 On the **Modeling** tab, click **Model Data Editor**.
- 2 In the Model Data Editor, select the **Data Stores** tab.
- 3 Begin editing the name of the target Data Store Memory block by clicking the corresponding row in the **Name** column.
- 4 Next to the name, click the button  and select **Create and Resolve**.
- 5 In the Create New Data dialog box, set **Value** to `Simulink.Signal`. Optionally, use the **Location** drop-down list to choose a workspace for storing the resulting `Simulink.Signal` object.
- 6 Click **Create**. The `Simulink.Signal` object, which has the same name as the data store, appears in the target workspace. Simulink selects the block parameter **Data store name must resolve to Simulink signal object**.

When the property dialog box for the object opens, click **OK**.

- 7 In the **Simulink Coder** or **Embedded Coder** app, open the Code Mappings editor. In the **C Code** tab, select **Code Interface > Individual Element Code Mappings**.
- 8 On the **Data Stores** tab, apply the target storage class.

Note When a Data Store Memory block is associated with a signal object, the mapping between the **Data store name** and the signal object name must be one-to-one. If two or more identically named entities map to the same signal object, the name conflict is flagged as an error at code generation time. See “Resolve Conflicts in Configuration of Signal Objects for Code Generation” for more information.

Storage Classes for Data Store Memory Blocks

You can control how Data Store Memory blocks in your model are stored and represented in the generated code by assigning storage classes and type qualifiers. You do this in almost exactly the same way you assign storage classes and type qualifiers for block states.

Data Store Memory blocks, like block states, have `Auto` storage class by default, and their memory is stored within the `DWork` vector. The symbolic name of the storage location is based on the data store name.

You can generate code from multiple Data Store Memory blocks that have the same data store name, subject to the following restriction: *at most one* of the identically named blocks can have a storage class other than `Auto`. An error is reported if this condition is not met.

For blocks with `Auto` storage class, the code generator produces a unique symbolic name for each block to avoid name clashes. For Data Store Memory blocks with storage classes other than `Auto`, the generated code uses the data store name as the symbol.

In the following model, a Data Store Write block writes to memory declared by the Data Store Memory block `myData`:



To control the storage declaration for a Data Store Memory block, in the coder app, use the Code Mapping editor. On the **Data Stores** tab, select a **Storage Class** for the block.

Data Store Memory blocks are nonvirtual because code is generated for their initialization in `.c` and `.cpp` files and their declarations in header files. The following table shows how the code generated for the Data Store Memory block in the preceding model differs for different storage classes. The table gives the variable declarations and `MdlOutputs` code generated for the `myData` block.

Storage Class	Declaration	Code
Auto or Model default (when Code Mapping Editor specifies Default storage class)	<p>In <code>model.h</code></p> <pre>typedef struct D_Work_tag { real_T myData; } D_Work;</pre> <p>In <code>model.c</code> or <code>model.cpp</code></p> <pre>/* Block states (auto storage) */ D_Work model_DWork;</pre>	<pre>model_DWork.myData = rtb_SineWave;</pre>

Storage Class	Declaration	Code
ExportedGlobal	In <i>model.c</i> or <i>model.cpp</i> /* Exported block states */ real_T myData; In <i>model.h</i> extern real_T myData;	myData = rtb_SineWave;
ImportedExtern	In <i>model_private.h</i> extern real_T myData;	myData = rtb_SineWave;
ImportedExternPointer	In <i>model_private.h</i> extern real_T *myData;	(*myData) = rtb_SineWave;

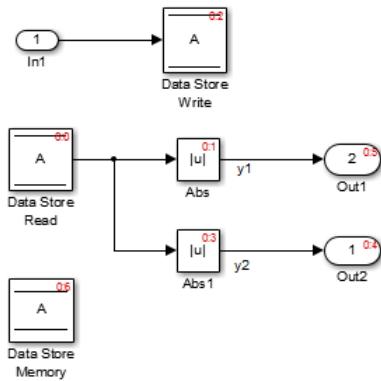
For information about applying storage classes, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.

For ERT models, you can preserve dimensions of multidimensional arrays in data stores. For more information, see “Preserve Dimensions of Multidimensional Arrays in Generated Code” (Embedded Coder).

Data Store Buffering in Generated Code

A Data Store Read block is a nonvirtual block that copies the value of the data store to its output buffer when it executes. Since the value is buffered, downstream blocks connected to the output of the data store read utilize the same value, even if a Data Store Write block updates the data store in between execution of two of the downstream blocks.

The next figure shows a model that uses blocks whose priorities have been modified to achieve a particular order of execution:



```

/* local block i/o variables */
real_T rtb_DataStoreRead;

/* DataStoreRead: '<Root>/Data Store Read' */
rtb_DataStoreRead = A; Buffer the value of A

/* Abs: '<Root>/Abs1' incorporates:
 * DataStoreRead: '<Root>/Data Store Read'
 */
y1 = fabs(A); Use A (whose value equals
the buffered value at this point

/* DataStoreWrite: '<Root>/Data Store Write' incorporates:
 * Import: '<Root>/In1'
 */
A = u1; Update the value of A

/* Abs: '<Root>/Abs' */
y2 = fabs(rtb_DataStoreRead); Consistently use the same buffered
value as before the update to A

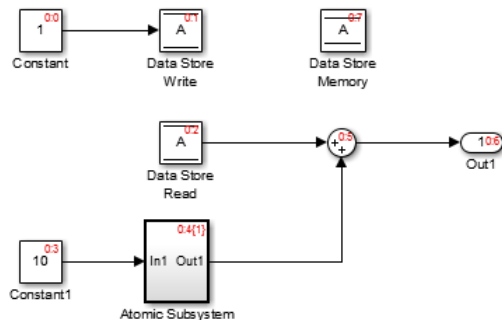
```

The following execution order applies:

- 1 The block Data Store Read buffers the current value of the data store A at its output.
- 2 The block Abs1 uses the buffered output of Data Store Read.
- 3 The block Data Store Write updates the data store.
- 4 The block Abs uses the buffered output of Data Store Read.

Because the output of Data Store Read is a buffer, both Abs and Abs1 use the same value: the value of the data store at the time that Data Store Read executes.

The next figure shows another example:



```

real_T rtb_DataStoreRead;

/* DataStoreWrite: '<Root>/Data Store Write' incorporates:
 * Constant: '<Root>/Constant'
 */
A = DoBufferDSMRead2_P.Constant_Value;

/* DataStoreRead: '<Root>/Data Store Read' */
rtb_DataStoreRead = A; Buffer the value of A

/* Outputs for atomic SubSystem: '<Root>/Atomic Subsystem' */
DoBufferDSMRead_AtomicSubsystem(); We don't do a global analysis to
detect if this function writes to A

/* end of Outputs for SubSystem: '<Root>/Atomic Subsystem' */

/* Output: '<Root>/Out1' incorporates:
 * Sum: '<Root>/Sum'
 */
Use the buffered value of A
DoBufferDSMRead2_Y.Out1 = rtb_DataStoreRead + DoBufferDSMRead2_B.Abs;

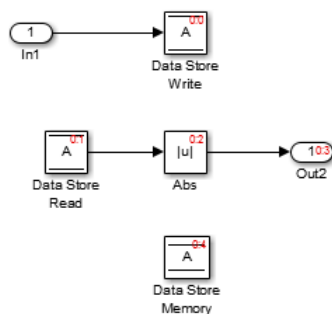
```

In this example, the following execution order applies:

- 1 The block Data Store Read buffers the current value of the data store A at its output.
- 2 Atomic Subsystem executes.
- 3 The Sum block adds the output of Atomic Subsystem to the output of Data Store Read.

Simulink assumes that Atomic Subsystem might update the data store, so Simulink buffers the data store. Atomic Subsystem executes after Data Store Read buffers its output, and the buffer provides a way for the Sum block to use the value of the data store as it was when Data Store Read executed.

In some cases, the code generator determines that it can optimize away the output buffer for a Data Store Read block, and the generated code refers to the data store directly, rather than a buffered value of it. The next figure shows an example:



```

/* Model step function */
void DONTbufferDSMRead_step(void)
{
/* DataStoreWrite: '<Root>/Data Store Write' incorporates:
 * Inport: '<Root>/In1'
 */
A = u1;

/* Abs: '<Root>/Abs' incorporates:
 * DataStoreRead: '<Root>/Data Store Read'
 */
y2 = fabs(A);
}

```


In the generated code, the argument of the `fabs()` function is the data store `A` rather than a buffered value.

Data Stores Shared by Instances of a Reusable Model

You can use a data store to share a piece of data between the instances of a reusable referenced model (see “Share Data Among Referenced Model Instances”) or a model that you configure to generate reentrant code (by setting the configuration parameter **Code interface packaging** to `Reusable` function). If you implement the data store as a Data Store Memory block and select the **Share across model instances** parameter:

- By default, the data store appears in the generated code as a separate global symbol.
- If you have Embedded Coder, to restrict access such that only the code generated from the model can use the data store, configure the data store to appear as `static` by applying the storage class `FileScope`. For more information about `FileScope` and other storage classes, see “Choose Storage Class for Controlling Data Representation in Generated Code” (Embedded Coder).

Structures in Generated Code Using Data Stores

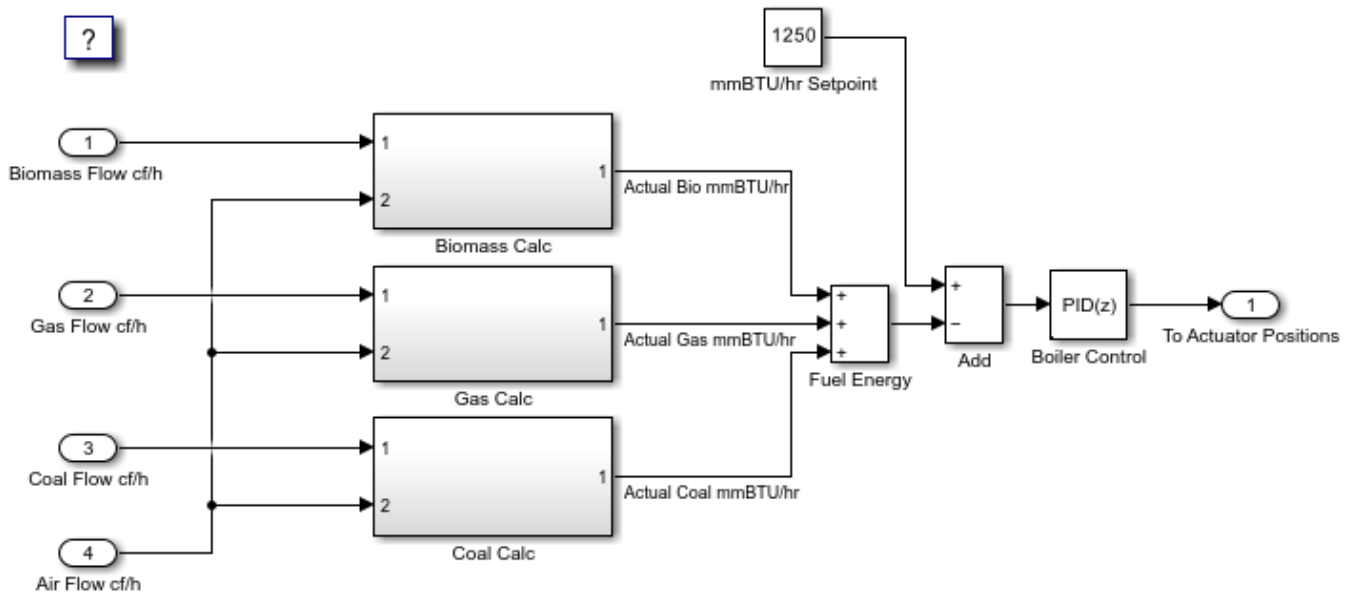
If you use more than one data store to provide global access to multiple signals in generated code, you can combine the signals into a single structure variable by using one data store. This combination of signal data can help you integrate the code generated from a model with other existing code that requires the data in a structure format.

This example shows how to store several model signals in a structure in generated code using a single data store. To store multiple signals in a data store, you configure the data store to accept a composite signal, such as a nonvirtual bus signal or an array of nonvirtual bus signals.

Explore Example Model

Open the example model `BusStructInCode`.

```
open_system('BusStructInCode')
```



Copyright 2023 The MathWorks, Inc.

The model contains three subsystems that perform calculations on the inputs from the top level of the model. In each subsystem, a Data Store Memory block stores an intermediate calculated signal.

Generate code with the model. In the code generation report, view the file `BusStructInCode.c`. The code defines a global variable for each data store.

```
real_T BioBTURate;
real_T CoalBTURate;
real_T GasBTURate;
```

Suppose that you want to integrate code generated from the example model with other existing code. Suppose also that the existing code requires access to data from the three data stores in a single structure variable. You can use a data store to assemble the target data in a structure in generated code.

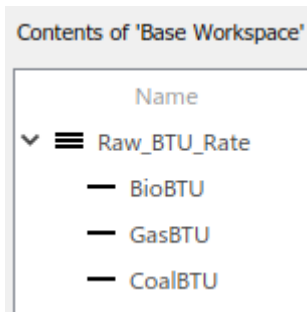
Configure Data Store

Configure a data store to contain multiple signals by creating a bus type to use as the data type of the data store. Define the bus type using the same hierarchy of elements as the structure that you want to appear in generated code.

- 1 Open the Type Editor tool.

```
typeditor
```

- 2 Define a new bus type `Raw_BTU_Rate` with one element for each of the three target signals. Name the elements `BioBTU`, `GasBTU`, and `CoalBTU`.

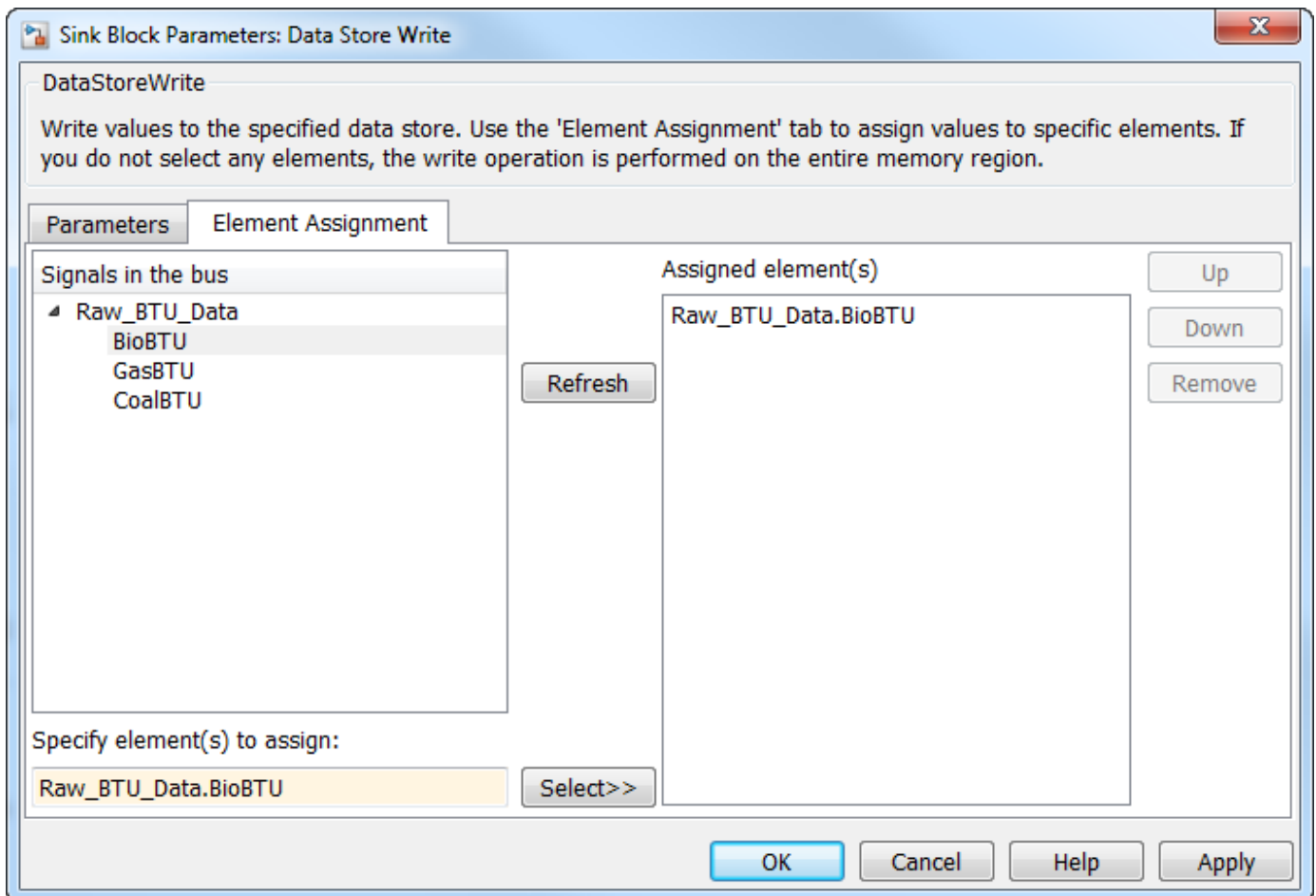


- 3 At the top level of the example model, add a Data Store Memory block.
- 4 On the **Modeling** tab, click **Model Data Editor**.
- 5 In the Model Data Editor, inspect the **Data Stores** tab.
- 6 For the new Data Store Memory block, use the **Name** column to set the data store name to Raw_BTU_Data.
- 7 Use the **Data Type** column to set the data type of the data store to Bus: Raw_BTU_Rate.
- 8 In the Code Mappings editor, on the **Data Stores** tab, apply the storage class ExportedGlobal to Raw_BTU_Data.

Write to Data Store Elements

To write to a specific element of a data store, use a Data Store Write block. On the **Element Assignment** tab in the dialog box, you can specify to write to a single element, a collection of elements, or the entire contents of a data store.

- 1 Open the **Biomass Calc** subsystem.
- 2 Delete the Data Store Memory block BioBTURate.
- 3 In the block dialog box for the Data Store Write block, set **Data store name** to Raw_BTU_Data.
- 4 On the **Element Assignment** tab, under **Signals in the bus**, expand the contents of the data store Raw_BTU_Data. Click the element BioBTU, and then click **Select**. Click **OK**.



- 5 Modify the **Gas Calc** and **Coal Calc** subsystems similarly.
 - Delete the Data Store Memory block in each subsystem.
 - In each Data Store Write block dialog box, set **Data store name** to `Raw_BTU_Data`.
 - In the **Gas Calc** subsystem, use the Data Store Write block to write to the data store element `GasBTU`. In the **Coal Calc** subsystem, write to the element `CoalBTU`.

Generate Code with Data Store Structure

- 1 Generate code for the example model.
- 2 In the code generation report, view the file `BusStructInCode.h`. The code defines a structure that corresponds to the bus type `Raw_BTU_Rate`.

```
typedef struct {
    real_T BioBTU;
    real_T GasBTU;
    real_T CoalBTU;
} Raw_BTU_Rate;
```

- 3 View the file `BusStructInCode.c`. The code represents the data store with a global variable `Raw_BTU_Data` of the structure type `Raw_BTU_Rate`. In the model step function, the code assigns the data from the calculated signals to the fields of the global variable `Raw_BTU_Data`.

See Also

Related Examples

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Structures in Generated Code Using Data Stores” on page 15-99
- “When to Use a Data Store”
- “Generate Code That Dereferences Data from a Literal Memory Address” (Embedded Coder)

Specify Single-Precision Data Type for Embedded Application

When you want code that uses only single precision, such as when you are targeting a single-precision processor, you can use model configuration parameters and block parameters to prevent the introduction of `double` in the model.

To design and validate a single-precision model, see “Validate a Floating-Point Embedded Model”. If you have Fixed-Point Designer, you can use the Single Precision Converter app (see **Single Precision Converter**).

Use single Data Type as Default for Underspecified Types

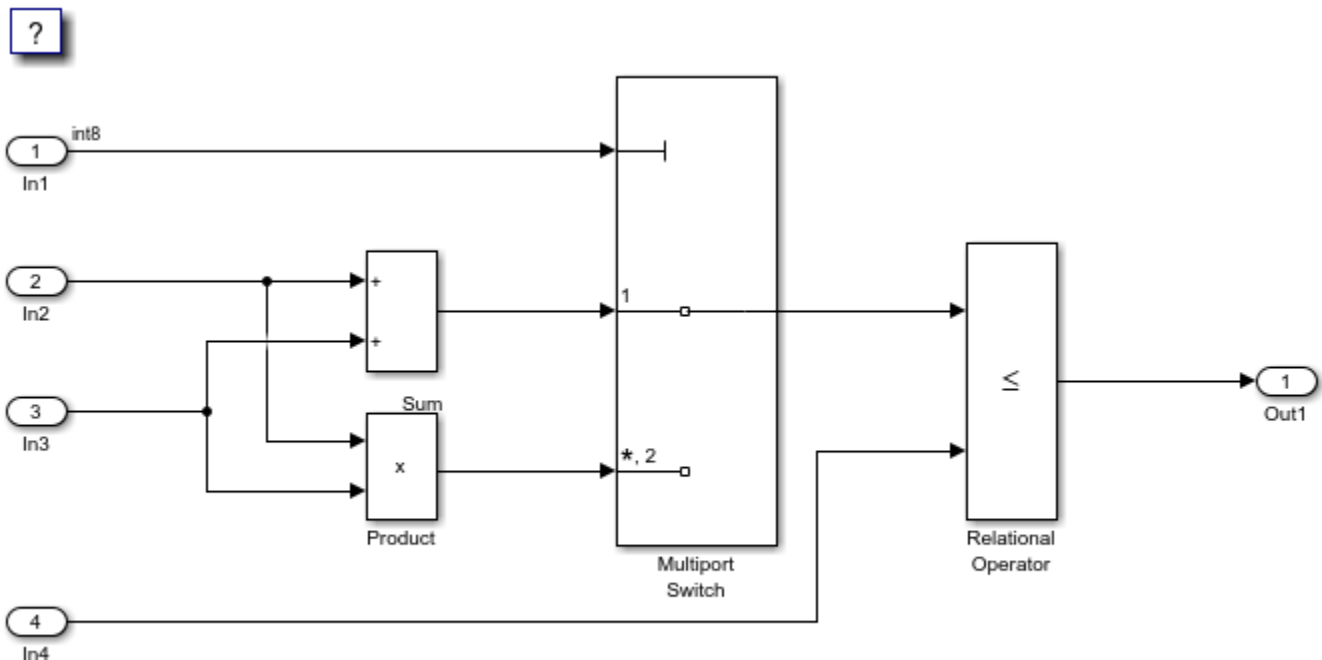
This example shows how to avoid introducing a double-precision data type in code generated for a single-precision hardware target.

If you specify an inherited data type for signals, but data type propagation rules cannot determine data types for the signals, the signal data types default to `double`. You can use a model configuration parameter to specify the default data type as `single`.

Explore Example Model

Open the example model `rtwdemo_underspecified_datatype` and configure it to show the generated names of blocks.

```
model = 'rtwdemo_underspecified_datatype';
load_system(model)
set_param(model, 'HideAutomaticNames', 'off')
open_system(model);
```



Copyright 2014-2020 The MathWorks, Inc.

The root inports In2, In3, and In4 specify `Inherit: Auto` for the **Data type** block parameter. The downstream blocks also use inherited data types.

Generate Code with double as Default Data Type

The model starts with the configuration parameter **System target file** set to `ert.tlc`, which requires Embedded Coder®. Set **System target file** to `grt.tlc` instead.

```
set_param(model, 'SystemTargetFile', 'grt.tlc')
```

Generate code from the model.

```
slbuild(model)
```

```
### Starting build procedure for: rtwdemo_underspecified_datatype
### Successful completion of build procedure for: rtwdemo_underspecified_datatype
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_underspecified_datatype	Code generated and compiled.	Code generation information file

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.896s
```

In the code generation report, view the file `rtwdemo_underspecified_datatype.h`. The code uses the `double` data type to define the variables In2, In3, and In4 because the Inport data types are underspecified in the model.

```
cfile = fullfile('rtwdemo_underspecified_datatype_grt_rtw', ...
    'rtwdemo_underspecified_datatype.h');
rtwdemodbtype(cfile, ...
    /* External inputs (root inport signals with default storage) */ , ...
    /* External outputs (root outports fed by signals with default storage) */ , ...
    1, 0);
```

```
/* External inputs (root inport signals with default storage) */
typedef struct {
    int8_T In1;           /* '<Root>/In1' */
    real_T In2;          /* '<Root>/In2' */
    real_T In3;          /* '<Root>/In3' */
    real_T In4;          /* '<Root>/In4' */
} ExtU_rtwdemo_underspecified_d_T;
```

Generate Code with single as Default Data Type

Open the Configuration Parameters dialog box. On the **Math and Data Types** pane, select `single` in the **Default for underspecified data type** drop-down list.

Alternatively, enable the optimization at the command prompt.

```
set_param(model, 'DefaultUnderspecifiedDataType', 'single');
```

Generate code from the model.

```
slbuild(model)
```

```
### Starting build procedure for: rtwdemo_underspecified_datatype
### Successful completion of build procedure for: rtwdemo_underspecified_datatype
```

```
Build Summary
```

```
Top model targets built:
```

Model	Action	Rebuild Reason
rtwdemo_underspecified_datatype	Code generated and compiled.	Generated code was out of date.

```
1 of 1 models built (0 models already up to date)
```

```
Build duration: 0h 0m 10.621s
```

In the code generation report, view the file `rtwdemo_underspecified_datatype.h`. The code uses the single data type to define the variables `In2`, `In3`, and `In4`.

```
rtwdemodbtype(cfile,...
    /* External inputs (root inport signals with default storage) */,...
    /* External outputs (root outports fed by signals with default storage) */,...
    1, 0);
```

```
/* External inputs (root inport signals with default storage) */
typedef struct {
    int8_T In1;           /* '<Root>/In1' */
    real32_T In2;        /* '<Root>/In2' */
    real32_T In3;        /* '<Root>/In3' */
    real32_T In4;        /* '<Root>/In4' */
} ExtU_rtwdemo_underspecified_d_T;
```

See Also

Related Examples

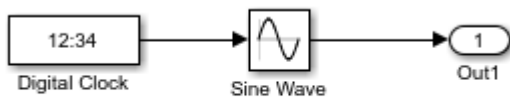
- “Default for underspecified data type”
- “Subnormal Number Execution Speed” on page 40-16
- Language standard
- “Manage Replacement of Simulink Data Types in Generated Code” (Embedded Coder)
- “About Data Types in Simulink”

Tune Phase Parameter of Sine Wave Block During Code Execution

Under certain conditions, you cannot configure the **Phase** parameter of a Sine Wave block to appear in the generated code as a tunable global variable (for more information, see the block reference page). This example shows how to generate code so that you can tune the phase during execution.

Create the model `ex_phase_tunable` by using a Digital Clock block.

```
open_system('ex_phase_tunable')
```



Copyright 2021 The MathWorks, Inc.

Set **Default parameter behavior** to **Tunable** so that the parameters of the Sine Wave block appear in the generated code as tunable fields of the global parameter structure.

```
set_param('ex_phase_tunable','DefaultParameterBehavior','Tunable')
```

Generate code from the model.

```
slbuild('ex_phase_tunable')
```

```
### Starting build procedure for: ex_phase_tunable
### Successful completion of code generation for: ex_phase_tunable
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
ex_phase_tunable	Code generated.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)

Build duration: 0h 0m 11.09s

In the code generation report, view the file `ex_phase_tunable.c`. The code algorithm in the model `step` function calculates the Sine Wave block output. The parameters of the block, including **Phase**, appear in the code as tunable structure fields.

```
file = fullfile('ex_phase_tunable_grt_rtw','ex_phase_tunable.c');
rtwdemodbtype(file,'/* Outport: '<Root>/Out1' incorporates:',...
    'ex_phase_tunable_P.SineWave_Bias;',1,1)
```

```
/* Outport: '<Root>/Out1' incorporates:
 * DigitalClock: '<Root>/Digital Clock'
 * Sin: '<Root>/Sine Wave'
 */
ex_phase_tunable_Y.Out1 = sin(ex_phase_tunable_P.SineWave_Freq *
```

```
((ex_phase_tunable_M->Timing.clockTick1+  
  ex_phase_tunable_M->Timing.clockTickH1* 4294967296.0)) * 1.0) +  
ex_phase_tunable_P.SineWave_Phase) * ex_phase_tunable_P.SineWave_Amp +  
ex_phase_tunable_P.SineWave_Bias;
```

During code execution, you can assign new values to the structure field that corresponds to the **Phase** parameter.

See Also

Related Examples

- “Preserve Variables in Generated Code” on page 15-2

Switch Between Output Waveforms During Code Execution for Waveform Generator Block

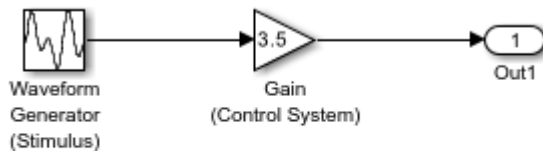
This example shows how to generate code that enables you to switch between stimulus waveforms during code execution.

For a Waveform Generator block, you cannot make the parameters of a waveform, such as amplitude and phase shift, tunable in the generated code. Instead, you can generate code that enables you to choose an active waveform from a set of waveform variants that you specify in the block. During execution of the code, you activate a variant by adjusting the value of a global structure field.

You must set the model configuration parameter **Default parameter behavior** to `Tunable`. Then, by default, block parameters in the model appear tunable in the generated code. These parameters can consume large amounts of memory for a large model.

Create the model `ex_switch_waveform`.

```
open_system('ex_switch_waveform')
```



Copyright 2021 The MathWorks, Inc.

In the Waveform Generator block, configure this waveform:

```
square(amp,10,0,dutyCycle)
```

In the base workspace, create the variables `amp` and `dutyCycle`.

```
amp = 2.71;
dutyCycle = 50;
```

This waveform represents the baseline stimulus that you want the application to use at the start of execution.

Suppose that, during execution, you want to observe the effects of changing the waveform frequency from 10 to 15 and the phase shift from 0 to 0.5. In the Waveform Generator block, add these waveform variants:

```
square(amp,10,0.5,dutyCycle)
square(amp,15,0,dutyCycle)
square(amp,15,0.5,dutyCycle)
```

Set the model configuration parameter **Default parameter behavior** to `Tunable`.

```
set_param('ex_switch_waveform','DefaultParameterBehavior','Tunable')
```

Generate code from the model.

```

slbuild('ex_switch_waveform')

### Starting build procedure for: ex_switch_waveform
### Successful completion of code generation for: ex_switch_waveform

Build Summary

Top model targets built:

Model          Action          Rebuild Reason
=====
ex_switch_waveform  Code generated.  Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 10.635s

The generated file ex_switch_waveform.h defines the standard structure type that stores tunable
parameter data for the model. The structure contains a field whose value represents the active
waveform.

file = fullfile('ex_switch_waveform_grt_rtw','ex_switch_waveform.h');
rtwdemodbtype(file,'/* Parameters (default storage) */',...
    '/* Real-time Model Data Structure */',1,0)

/* Parameters (default storage) */
struct P_ex_switch_waveform_T_ {
    real_T WaveformGeneratorStimulus_Select;
                                /* Mask Parameter: WaveformGeneratorStimulus_Select
                                 * Referenced by: '<S1>/Switch'
                                 */
    real_T GainControlSystem_Gain; /* Expression: 3.5
                                    * Referenced by: '<Root>/Gain (Control System)'
                                    */
};

```

The file `ex_switch_waveform_data.c` defines a global structure variable and initializes the field value to 1. This value represents the baseline waveform.

```

file = fullfile('ex_switch_waveform_grt_rtw','ex_switch_waveform_data.c');
rtwdemodbtype(file,'/* Block parameters (default storage) */',';',1,1)

/* Block parameters (default storage) */
P_ex_switch_waveform_T ex_switch_waveform_P = {
    /* Mask Parameter: WaveformGeneratorStimulus_Select
     * Referenced by: '<S1>/Switch'
     */
    1.0,

    /* Expression: 3.5
     * Referenced by: '<Root>/Gain (Control System)'
     */
    3.5
};

```

The file `ex_switch_waveform.c` defines the model execution function. The function uses a `switch` statement to determine the value of the active waveform, and then calculates the value of the root-level Output block, `Out1`.

```

file = fullfile('ex_switch_waveform_grt_rtw','ex_switch_waveform.c');
rtwdemodbtype(file,'switch ((int32_T)',...
    'ex_switch_waveform_Y.Out1 =',1,1)

switch ((int32_T)ex_switch_waveform_P.WaveformGeneratorStimulus_Select) {
    case 1:
        temp = temp_tmp - floor(temp_tmp) <= 0.5 ? 2.71 : -2.71;
        break;

    case 2:
        temp = (temp_tmp - 0.79577471545947676) - floor(temp_tmp -
            0.79577471545947676) <= 0.5 ? 2.71 : -2.71;
        break;

    case 3:
        temp = temp - floor(temp) <= 0.5 ? 2.71 : -2.71;
        break;

    default:
        temp = (temp - 1.1936620731892151) - floor(temp - 1.1936620731892151) <= 0.5
            ? 2.71 : -2.71;
        break;
}

/* End of MultiPortSwitch: '<S1>/MultiportSwitch' */

/* Output: '<Root>/Out1' incorporates:
 * Gain: '<Root>/Gain (Control System)'
 */
ex_switch_waveform_Y.Out1 = ex_switch_waveform_P.GainControlSystem_Gain * temp;

```

During code execution, to change the active waveform, adjust the value of the structure field in the global parameters structure.

See Also

Related Examples

- “Preserve Variables in Generated Code” on page 15-2

Create Tunable Calibration Parameter in the Generated Code

A calibration parameter is a value stored in global memory that an algorithm reads for use in calculations but does not write to. Calibration parameters are tunable because you can change the stored value during algorithm execution. You create calibration parameters so that you can:

- Determine an optimal parameter value by tuning the parameter and monitoring signal values during execution.
- Efficiently adapt an algorithm to different execution conditions by overwriting the parameter value stored in memory. For example, you can use the same control algorithm for multiple vehicles of different masses by storing different parameter values in each vehicle's engine control unit.

In Simulink, create a `Simulink.Parameter` object to represent a calibration parameter. You use the parameter object to set block parameter values, such as the **Gain** parameter of a Gain block. To control the representation of the parameter object in the generated code, you apply a storage class to the object.

To make block parameters accessible in the generated code by default, for example for rapid prototyping, set **Default parameter behavior** (see Default parameter behavior) to `Tunable`. For more information, see "Preserve Variables in Generated Code" on page 15-2.

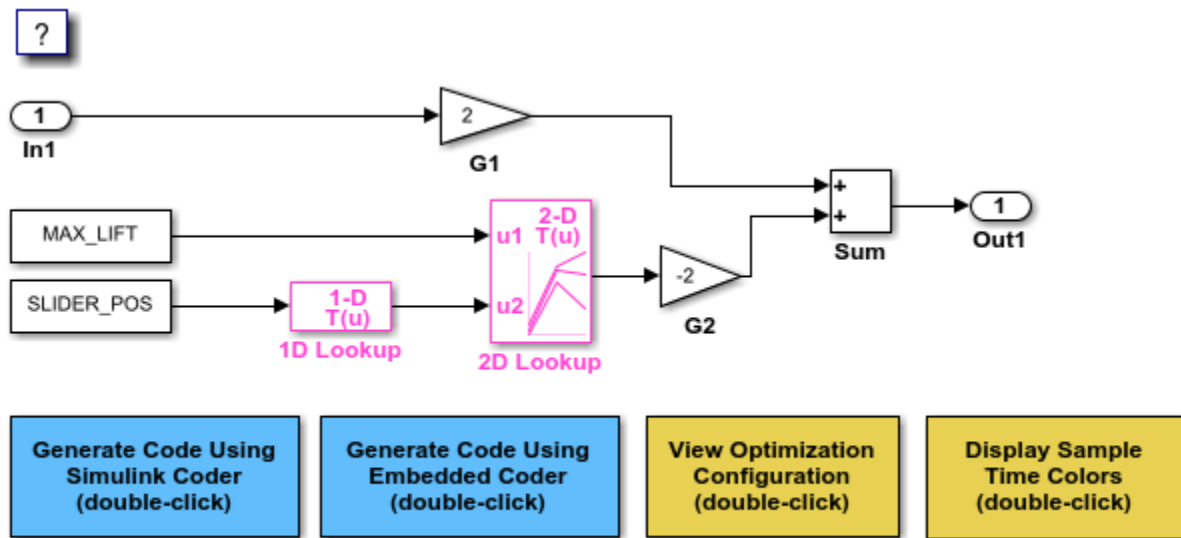
Represent Block Parameter as Tunable Global Variable

This example shows how to create tunable parameter data by representing block parameters as global variables in the generated code.

Configure Block Parameter by Using Parameter Object

Open the example model `rtwdemo_paraminline` and configure it to show the generated names of blocks.

```
load_system('rtwdemo_paraminline')
set_param('rtwdemo_paraminline','HideAutomaticNames','off')
open_system('rtwdemo_paraminline')
```



Copyright 1994-2020 The MathWorks, Inc.

On the **Modeling** tab, click **Model Data Editor**.

In the Model Data Editor, inspect the **Parameters** tab.

In the model, click the G1 Gain block. The Model Data Editor highlights the row that corresponds to the **Gain** parameter of the block.

In the Model Data Editor **Value** column, change the gain value from 2 to myGainParam.

Next to myGainParam, click the action button (with three vertical dots) and select **Create**.

In the Create New Data block dialog box, set **Value** to Simulink.Parameter(2). Click **Create**. A Simulink.Parameter object myGainParam stores the parameter value, 2, in the model workspace.

In the myGainParam dialog box, on the **Code Generation** tab, click **Configure in Coder App**.

In the Code Mappings editor, set **Storage Class** of myGainParam to ExportedGlobal. This storage class causes the parameter object to appear in the generated code as a tunable global variable.

Alternatively, to create the parameter object and configure the model, use these commands at the command prompt:

```
set_param('rtwdemo_paraminline/G1','Gain','myGainParam')
mws = get_param('rtwdemo_paraminline','modelworkspace');
mws.assignin('myGainParam',Simulink.Parameter(2));
cm = coder.mapping.utils.create('rtwdemo_paraminline');
setModelParameter(cm,'myGainParam','StorageClass','ExportedGlobal');
```

Use the Model Data Editor to create a parameter object, myOtherGain, for the G2 Gain block. Apply the storage class ExportedGlobal.

Alternatively, use these commands at the command prompt:

```
set_param('rtwdemo_paraminline/G2', 'Gain', 'myOtherGain')
mws.assignin('myOtherGain', Simulink.Parameter(-2));
setModelParameter(cm, 'myOtherGain', 'StorageClass', 'ExportedGlobal');
```

Generate and Inspect Code

Generate code from the model.

```
slbuild('rtwdemo_paraminline')

### Starting build procedure for: rtwdemo_paraminline
### Successful completion of build procedure for: rtwdemo_paraminline
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_paraminline	Code generated and compiled	Code generation information file does not exist

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 9.502s

The generated file `rtwdemo_paraminline.h` contains extern declarations of the global variables `myGainParam` and `myOtherGain`. You can include (`#include`) this header file so that your code can read and write the value of the variable during execution.

```
file = fullfile('rtwdemo_paraminline_grt_rtw', 'rtwdemo_paraminline.h');
rtwdemodbtype(file, ...
    'extern real_T myGainParam;', 'Referenced by: '<Root>/G2'', 1, 1)
```

```
extern real_T myGainParam;          /* Variable: myGainParam
                                   * Referenced by: '<Root>/G1'
                                   */
extern real_T myOtherGain;         /* Variable: myOtherGain
                                   * Referenced by: '<Root>/G2'
```

The file `rtwdemo_paraminline.c` allocates memory for and initializes `myGainParam` and `myOtherGain`.

```
file = fullfile('rtwdemo_paraminline_grt_rtw', 'rtwdemo_paraminline.c');
rtwdemodbtype(file, ...
    '/* Exported block parameters */', 'Referenced by: '<Root>/G2'', 1, 1)
```

```
/* Exported block parameters */
real_T myGainParam = 2.0;          /* Variable: myGainParam
                                   * Referenced by: '<Root>/G1'
                                   */
real_T myOtherGain = -2.0;        /* Variable: myOtherGain
                                   * Referenced by: '<Root>/G2'
```

The generated code algorithm in the model step function uses `myGainParam` and `myOtherGain` for calculations.

```
rtwdemodbtype(file, ...
    '/* Model step function */', '/* Model initialize function */', 1, 0)
```



```

/* Model step function */
void rtwdemo_paraminline_step(void)
{
    /* Outport: '<Root>/Out1' incorporates:
    * Gain: '<Root>/G1'
    * Gain: '<Root>/G2'
    * Inport: '<Root>/In1'
    * Sum: '<Root>/Sum'
    */
    rtwdemo_paraminline_Y.Out1 = myGainParam * rtwdemo_paraminline_U.In1 +
    myOtherGain * -75.0;
}

```

Apply Storage Class When Block Parameter Refers to Numeric MATLAB Variable

If you use a numeric variable to set the value of a block parameter, you cannot apply a storage class to the variable. As a workaround, you can convert the variable to a parameter object, and then apply a storage class to the object. To convert the variable to a parameter object, choose one of these techniques:

- On the Model Data Editor **Parameters** tab, with **Change view** set to Code, find the row that corresponds to the variable. In the **Storage Class** column, from the drop-down list, select **Convert to parameter object**. The Model Data Editor converts the variable to a parameter object. Then, use the **Storage Class** column to apply a storage class to the object.

You can also use this technique in the Model Explorer.

- Use the Data Object Wizard (see “Create Data Objects for a Model Using Data Object Wizard”). In the Wizard, select the **Parameters** check box. The Wizard converts variables to objects. Then, apply storage classes to the objects, for example, by using the Model Data Editor or the Model Explorer.

Create Storage Class That Represents Calibration Parameters (Embedded Coder)

This example shows how to create a storage class that yields a calibration parameter in the generated code. The storage class causes each parameter object (`Simulink.Parameter`) to appear as a global variable with special decorations such as keywords and pragmas.

In the generated code, the calibration parameters must appear as global variables defined in a file named `calPrms.c` and declared in `calPrms.h`. The variable definitions must look like these definitions:

```

#pragma SEC(CALPRM)

const volatile float param1 = 3.0F;
const volatile float param2 = 5.0F;
const volatile float param3 = 7.0F;

#pragma SEC()

```

The variables use the keywords `const` and `volatile`. The pragma `#pragma SEC(CALPRM)` controls the placement of the variables in memory. To implement the pragma, variable definitions must appear in a contiguous block of code.

Also, the generated code must include an ASAP2 (a2l) description of each parameter.

Create Package for Storing Storage Class and Memory Section Definitions

Now, create a package in your current folder by copying the example package `+SimulinkDemos`. The package stores the definitions of `Parameter` and `Signal` classes that you later use to apply the storage class to data elements in models. Later, the package also stores the definitions of the storage class and an associated memory section.

- 1 Set your current MATLAB folder to a writable location.
- 2 Copy the `+SimulinkDemos` package folder into your current folder. Name the copy `+myPackage`.

```
copyfile(fullfile(matlabroot,...
    'toolbox','simulink','simdemos','dataclasses','+SimulinkDemos'),...
    '+myPackage','f')
```

- 3 Navigate inside the `+myPackage` folder to the file `Signal.m` to edit the definition of the `Signal` class.
- 4 Uncomment the methods section that defines the method `setupCoderInfo`. In the call to the function `useLocalCustomStorageClasses`, replace `'packageName'` with `'myPackage'`. When you finish, the section looks like this:

```
methods
    function setupCoderInfo(h)
        % Use custom storage classes from this package
        useLocalCustomStorageClasses(h, 'myPackage');
    end
end % methods
```

- 5 Save and close the file.
- 6 Navigate inside the `+myPackage` folder to the file `Parameter.m` to edit the definition of the `Parameter` class. Uncomment the methods section that defines the method `setupCoderInfo` and replace `'packageName'` with `'myPackage'`.
- 7 Save and close the file.

Create Storage Class and Memory Section

- 1 Set your current folder to the folder that contains the package folder `+myPackage`.
- 2 Open the Custom Storage Class Designer.

```
cscdesigner('myPackage')
```

- 3 In the Custom Storage Class Designer, on the **Memory Sections** tab, click **New**.
- 4 For the new memory section, set properties according to the table.

Property	Value
Name	CalMem
Statements Surround	Group of variables
Pre Statement	<code>#pragma SEC(CALPRM)</code>
Post Statement	<code>#pragma SEC()</code>

Property	Value
Is const	Select
Is volatile	Select

- 5 Click **Apply**.
- 6 On the **Custom Storage Class** tab, click **New**.
- 7 For the new storage class, set properties according to the table.

Property	Value
Name	CalParam
For signals	Clear
Data scope	Exported
Header File	calPrms.h
Definition File	calPrms.c
Memory Section	CalMem

- 8 Click **OK**. In response to the message about saving changes, click **Yes**.

Set Default Parameter Object to myPackage.Parameter

To make applying the storage class easier, use the Model Explorer to change the default parameter object from `Simulink.Parameter` to `myPackage.Parameter`.

- 1 At the command prompt, open the Model Explorer.


```
daexplr
```
- 2 In the Model Explorer **Model Hierarchy** pane, select **Base Workspace**.
- 3 In the Model Explorer toolbar, click the arrow next to the **Add Simulink Parameter** button. In the drop-down list, select **Customize class lists**.
- 4 In the **Customize class lists** dialog box, under **Parameter classes**, select the check box next to **myPackage.Parameter**. Click **OK**.
- 5 In the Model Explorer toolbar, click the arrow next to the **Add Simulink Parameter** button. In the drop-down list, select **myPackage Parameter**.

A `myPackage.Parameter` object appears in the base workspace. You can delete this object.

Now, when you use tools such as the Model Data Editor to create parameter objects, Simulink creates `myPackage.Parameter` objects instead of `Simulink.Parameter` objects.

Apply Storage Class

In the example model `RollAxisAutopilot`, the `BasicRollMode` subsystem represents a PID controller. Configure the P, I, and D parameters as calibration parameters.

- 1 Open the model.


```
openExample('RollAxisAutopilot');
```
- 2 In the model, navigate into the `BasicRollMode` subsystem.
- 3 In the Apps gallery, click **Embedded Coder**.
- 4 Underneath the block diagram, open the Model Data Editor by selecting the **Model Data Editor** tab.
- 5 In the Model Data Editor, select the **Parameters** tab and update the block diagram.

Now, the data table contains rows that correspond to workspace variables used by the Gain blocks (which represent the P, I, and D parameters of the controller).

- 6 In the Model Data Editor, next to the **Filter contents** box, activate the **Filter using selection** button.
- 7 In the model, select the three Gain blocks.
- 8 In the **Filter contents** box, enter `model workspace`.

The variables that the Gain blocks use are in the model workspace.

- 9 In the data table, select the three rows and, in the **Storage Class** column for a row, select **Convert to parameter object**.

The Model Data Editor converts the workspace variables to `myPackage.Parameter` objects. Now, you can apply a storage class to the objects.

- 10 In the **Storage Class** column for a row, select **CalParam**.

Generate and Inspect Code

- 1 Generate code from the model.
- 2 In the code generation report, inspect the `calPrms.c` file. The file defines the calibration parameters.

```
/* Exported data definition */
#pragma SEC(CALPRM)

/* Definition for custom storage class: CalParam */
const volatile real32_T dispGain = 0.75F;
const volatile real32_T intGain = 0.5F;
const volatile real32_T rateGain = 2.0F;

#pragma SEC()
```

The file `calPrms.h` declares the parameters.

- 3 Inspect the interface file `RollAxisAutopilot.a2l`. The file contains information about each parameter, for example, for `dispGain`.

```
/begin CHARACTERISTIC
/* Name */ dispGain
/* Long Identifier */ ""
/* Type */ VALUE
/* ECU Address */ 0x0000 /* @ECU_Address@dispGain@ */
/* Record Layout */ Scalar_FLOAT32_IEEE
/* Maximum Difference */ 0
/* Conversion Method */ RollAxisAutopilot_CM_single
/* Lower Limit */ -3.4E+38
/* Upper Limit */ 3.4E+38
/end CHARACTERISTIC
```

Initialize Parameter Value From System Constant or Other Macro (Embedded Coder)

You can generate code that initializes a tunable parameter with a value calculated from some system constants (macros). For example, you can generate this code, which initializes a tunable parameter `totalVol` with a value calculated from macros `numVessels` and `vesInitVol`:

```
#define numVessels 16
#define vesInitVol 18.2
```

```
double totalVol = numVessels * vesInitVol;
```

This initialization technique preserves the mathematical relationship between the tunable parameter and the system constants, which can make the generated code more readable and easier to maintain. To generate this code:

- 1 Create parameter objects that represent the system constants.

```
numVessels = Simulink.Parameter(16);
vesInitVol = Simulink.Parameter(18.2);
```

- 2 Configure the objects to use the storage class `Define`, which yields a macro in the generated code.

```
numVessels.CoderInfo.StorageClass = 'Custom';
numVessels.CoderInfo.CustomStorageClass = 'Define';
```

```
vesInitVol.CoderInfo.StorageClass = 'Custom';
vesInitVol.CoderInfo.CustomStorageClass = 'Define';
```

- 3 Create another parameter object that represents the tunable parameter. Configure the object to use the storage class `ExportedGlobal`, which yields a global variable in the generated code.

```
totalVol = Simulink.Parameter;
totalVol.CoderInfo.StorageClass = 'ExportedGlobal';
```

- 4 Set the value of `totalVol` by using the expression `numVessels * vesInitVol`. To specify that the generated code preserve the expression, use the `slexpr` function.

```
totalVol.Value = slexpr('numVessels * vesInitVol');
```

- 5 Use `totalVol` to set block parameter values in your model. The code that you generate from the model initializes the tunable parameter with a value based on the system constants.

For more information and limitations about using an expression to set the value of a `Simulink.Parameter` object, see “Set Variable Value by Using a Mathematical Expression”.

Code Generation Impact of Storage Location for Parameter Objects

You can create a parameter object in the base workspace, a model workspace, or a data dictionary. However, when you end your MATLAB session, variables in the base workspace are deleted. To determine where to store parameter objects and other variables that a model uses, see “Determine Where to Store Variables and Objects for Simulink Models”.

The location of a parameter object can impact the file placement of the corresponding data definition in the generated code.

- If you place a parameter object in the base workspace or a data dictionary, the code generator assumes that the corresponding parameter data (for example, a global variable) belongs to the system from which you generate code, not to a specific component in the system. For example, if a model in a model reference hierarchy uses a parameter object with a storage class other than `Auto`, the data definition appears in the code generated for the top model in the hierarchy, not in the code generated for the model that uses the object.

However, if you have Embedded Coder, some storage classes enable you to specify the name of the model that owns a piece of data. When you specify an owner model, the code generated for that

model defines the data. For more information about data ownership, see “Control Placement of Global Data Definitions and Declarations in Generated Files” (Embedded Coder).

- If you place a parameter object in a model workspace, the code generator assumes that the model owns the data. If you generate code from a reference hierarchy that includes the containing model, the data definition appears in the code generated for the containing model.

For more information about data ownership, see “Control Placement of Global Data Definitions and Declarations in Generated Files” (Embedded Coder).

- If you apply a storage class other than `Auto` to a parameter object, the object appears in the generated code as a global symbol. Therefore, in a model reference hierarchy, two such objects in different model workspaces or dictionaries cannot have the same name. The name of each object must be unique throughout the model hierarchy.

However, if you have Embedded Coder, you can use the storage class `FileScope` to prevent name clashes between parameter objects in different model workspaces. See “Organize Parameter Data into a Structure by Using Struct Storage Class” (Embedded Coder).

If you store an `AUTOSAR.Parameter` object in a model workspace, the code generator ignores the storage class that you specify for the object.

Configure Accessibility of Signal Data

When you tune the value of a parameter during algorithm execution, you monitor or capture output signal values to analyze the results of the tuning. To represent signals in the generated code as accessible data, you can use techniques such as test points and storage classes. See “Preserve Variables in Generated Code” on page 15-2.

Programmatic Interfaces for Tuning Parameters

You can configure the generated code to include:

- A C application programming interface (API) for tuning parameters independent of external mode. The generated code includes extra code so that you can write your own code to access parameter values. See “Exchange Data Between Generated and External Code Using C API” on page 34-2.
- A Target Language Compiler API for tuning parameters independently of external mode. See “Parameter Functions”.

Set Tunable Parameter Minimum and Maximum Values

It is a best practice to specify minimum and maximum values for tunable parameters.

You can specify these minimum and maximum values:

- In the block dialog box that uses the parameter object. Use this technique to store the minimum and maximum information in the model.
- By using the properties of a `Simulink.Parameter` object that you use to set the parameter value. Use this technique to store the minimum and maximum information outside the model.

For more information, see “Specify Minimum and Maximum Values for Block Parameters”.

Considerations for Other Modeling Goals

Goal	Considerations and More Information
Apply storage type qualifiers <code>const</code> and <code>volatile</code>	If you have Embedded Coder, to generate the storage type qualifiers, see “Protect Global Data with <code>const</code> and <code>volatile</code> Type Qualifiers” (Embedded Coder).
Prevent name clashes between parameters in different components by applying the keyword <code>static</code>	If you have Embedded Coder, use the storage class <code>FileScope</code> or a similar storage class that you create. See “Choose Storage Class for Controlling Data Representation in Generated Code” (Embedded Coder).
Generate ASAP2 (a2l) description	You can generate an a2l file that uses the ASAP2 standard to describe your calibration parameters. For more information, see “Export ASAP2 File for Data Measurement and Calibration” on page 35-2.
Generate AUTOSAR XML (ARXML) description	If you have Embedded Coder, you can generate an ARXML file that describes calibration parameters used by models that you configure for the AUTOSAR standard. See “Model AUTOSAR Calibration Parameters and Lookup Tables” (AUTOSAR Blockset).
Store lookup table data for calibration	<p>To store lookup table data for calibration according to the ASAP2 or AUTOSAR standards (for example, <code>STD_AXIS</code>, <code>COM_AXIS</code>, or <code>CURVE</code>), you can use <code>Simulink.LookupTable</code> and <code>Simulink.Breakpoint</code> objects in lookup table blocks.</p> <p>However, some limitations apply. See <code>Simulink.LookupTable</code>. To work around the limitations of <code>Simulink.LookupTable</code> and <code>Simulink.Breakpoint</code> objects, use <code>Simulink.Parameter</code> objects instead.</p> <p>For more information, see “Define ASAP2 Information for Lookup Tables” on page 35-7 and “Configure Lookup Tables for AUTOSAR Calibration and Measurement” (AUTOSAR Blockset).</p>
Use pragmas to store parameter data in specific memory locations	If you have an Embedded Coder license, to generate code that includes custom pragmas, use storage classes and memory sections. See “Control Data and Function Placement in Memory by Inserting Pragmas” (Embedded Coder).

See Also

`Simulink.Parameter` | `Simulink.LookupTable` | `Simulink.Breakpoint`

Related Examples

- “Exchange Data Between External C/C++ Code and Simulink Model or Generated Code” on page 30-59
- “Reuse Parameter Data in Different Data Type Contexts” on page 15-143
- “Limitations for Block Parameter Tunability in Generated Code” on page 15-122
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49
- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Access Structured Data Through a Pointer That External Code Defines” (Embedded Coder)
- “Configure Lookup Tables for AUTOSAR Calibration and Measurement” (AUTOSAR Blockset)

Limitations for Block Parameter Tunability in Generated Code

A block parameter, MATLAB variable, or `Simulink.Parameter` object is tunable if it appears in the generated code as data stored in memory, such as a global variable. For example, when you apply the storage class `ExportedGlobal` to a parameter object, the parameter object appears tunable in the generated code. When you set **Default parameter behavior** to `Tunable`, MATLAB variables and parameter objects appear tunable in the generated code. By definition, model arguments also appear tunable.

Under certain conditions, the code generator cannot maintain tunability of a parameter, variable, object, or expression. In this case, the code generator inlines the numeric value, preventing you from changing the value during code execution.

To detect these conditions in your model, set the model configuration parameter **Detect loss of tunability** (see “Detect loss of tunability”) to `warning` or `error`.

Tunable Expression Limitations

You can specify block parameter values as expressions that use `Simulink.Parameter` objects or workspace variables. For example, you can use the expression `5 * gainParam`. For general information about using expressions to set block parameter values, see “Use Mathematical Expressions, MATLAB Functions, and Custom Functions”. For limitations with respect to expressions that you use to set the values of `Simulink.Parameter` objects, see “Code Generation of Parameter Objects With Expression Values” on page 15-125.

A tunable workspace variable is a `Simulink.Parameter` object or workspace variable that appears tunable in the generated code. For example, an object or variable is tunable if you apply a storage class other than `Auto` or if you set **Default parameter behavior** to `Tunable`.

An expression that contains one or more tunable workspace variables, model arguments, or tunable mask parameters is called a tunable expression. The expression is tunable because the code generator attempts to preserve the expression in the code. Because the code generator preserves the expression, you can change the values of the parameter data during code execution.

The code generator reduces certain expressions to an inlined numeric value in the generated code. The inlining renders workspace variables in the expression nontunable. To avoid loss of tunability due to unsupported expressions, observe these guidelines:

- Expressions involving complex (`i`) workspace variables or parameter objects are not supported.
- Certain operators and functions cause the code generator to reduce expressions and remove tunability. To determine whether an operator or function causes loss of tunability, use the information in this table.

Category	Operators or Functions
1	<code>+ - .* ./ < > <= >= == ~= & </code>
2	<code>* /</code>
3	<code>abs, acos, asin, atan, atan2, boolean, ceil, cos, cosh, exp, floor, log, log10, sign, sin, sinh, sqrt, tan, tanh, single, int8, int16, int32, uint8, uint16, uint32</code>
4	<code>: .^ ^ [] {} . \ .\ ' .' , ;</code>

- Use operators from category 1 without loss of tunability.
- Use operators from category 2 in expressions as long as at least one operand is a scalar. For example, scalar/scalar and scalar/matrix operand combinations are supported, but matrix/matrix combinations are not supported.
- You can use tunable workspace variables as arguments for the functions in category 3. If you use other functions, the code generator removes the tunability of the arguments.
- The operators in category 4 are not supported.
- The Fcn and If blocks do not support tunable expressions for code generation or in referenced models.
- You can write mask initialization code that creates and modifies variables. If you use those variables in an expression, the expression is not tunable in the generated code. This limitation also applies to Simulink library blocks with mask initialization code. For example, block parameters that specify bit operations for the Bitwise, Bit Set, and Bit Clear blocks are not tunable because the parameter is used in the mask initialization code for the block.
- You can specify a data type for the Simulink.Parameter objects or workspace variables that make up expressions. As long as the data type of these variables and objects and the data type of the corresponding block parameters are the same or `double`, the code generator can preserve tunability.

If the code generator preserves tunability of a parameter expression that includes at least one element of an integer type, simulation and execution of generated code can produce results that are numerically inconsistent. Evaluation of the expression in the generated code results in an overflow while the expression saturates during simulation. For more information, see “Numerical Consistency of Model and Generated Code Simulation Results” on page 43-2.

Linear Block Parameter Tunability

These blocks have a `Realization` parameter that affects the tunability of their numeric parameters:

- Transfer Fcn
- State-Space
- Discrete State-Space

To set the `Realization` parameter, you must use the command prompt:

```
set_param(gcf, 'Realization', 'auto')
```

For the `Realization` parameter, you can choose these options:

- `general`: The block's numeric parameters appear tunable in the generated code.
- `sparse`: The generated code represents the block's parameters as transformed values that increase efficiency. The parameters are not tunable.
- `auto`: The default. If one or more of the block's parameters are tunable (for example, because you use a tunable parameter object to set a parameter value), then the block uses the `general` realization. Otherwise, the block uses the `sparse` realization.

To tune the parameter values of one of these blocks during an external mode simulation, the block must use the `general` realization.

Parameter Structures

As described in “Organize Data into Structures in Generated Code” on page 15-147, you can create structures of parameter data in the generated code.

- If a parameter structure is tunable, the numeric fields of the structure are tunable in the generated code. However, if a field contains a nontunable entity, such as a multidimensional array, the structure fields are not tunable.
- You cannot declare individual substructures or fields within a parameter structure as tunable. You cannot use a `Simulink.Parameter` object as the value of a structure field. Instead, you must store the entire structure in the parameter object.

See Also

Related Examples

- “Create Tunable Calibration Parameter in the Generated Code” on page 15-112
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Code Generation of Parameter Objects With Expression Values

As described in “Set Variable Value by Using a Mathematical Expression”, you can set the value of a `Simulink.Parameter` object to an expression involving other MATLAB variables or parameter objects. When you generate code from a model that uses such a parameter object, to achieve your goal, use the information in the table.

Goal	Technique and More Information
Generate code without Embedded Coder.	<p>For the parameter object that uses the expression, apply an available storage class (see “C Code Generation Configuration for Model Interface Elements” on page 16-2).</p> <p>For the parameter objects used in the expression, apply only the storage class <code>Auto</code>.</p>
Generate code that defines a global variable and initializes it by using an expression involving system constants and other macros (requires Embedded Coder).	See “Initialize Parameter Value From System Constant or Other Macro (Embedded Coder)” (Embedded Coder).
Generate code that defines a macro whose value is an expression involving other macros (requires Embedded Coder).	<p>For the parameter object that uses the expression, apply a storage class that yields a macro in the generated code, such as <code>Define</code> (see “Macro Definitions (#define)” (Embedded Coder)).</p> <p>For the parameter objects used in the expression, apply storage classes that yield imported macros. For example, use the storage class <code>ImportedDefine</code>. Your external code must define these macros.</p>

Considerations and Limitations

- To avoid errors that prevent code generation, if you apply a storage class other than `Auto` to the dependent parameter object (which uses the expression as its value), the parameter objects used in the expression must use either `Auto` or a storage class that yields a macro in the generated code.
- You cannot set the value of a parameter object that represents a symbolic dimension (see “Implement Symbolic Dimensions for Array Sizes in Generated Code” (Embedded Coder)) to an expression.
- With Embedded Coder, you can generate code that preserves the expression, but only for the purpose of statically initializing the value of a global variable or macro that corresponds to the dependent parameter object. Follow the guidelines in “Expression Preservation” on page 15-125.

Expression Preservation

If you want Embedded Coder to preserve expressions in the generated code, adhere to these restrictions and guidelines.

- The dependent parameter object must use a storage class other than `Auto`. For example, to generate a global variable and initialize it by using the expression, use `ExportedGlobal` or `ExportToFile`.
- The parameter objects used in the expression must:
 - Use a storage class that yields a macro in the generated code, such as `Define`.
 - Have scalar, real values if the expression uses operators.
- The expression can use only these operators (in MATLAB syntax):
 - Mathematical: `+`, `-`, `*`
 - Relational: `==`, `~=`, `<`, `>`, `<=`, `>=`
- These data typing guidelines apply:
 - If possible, for a parameter object used in an expression, leave the `DataType` property at the default value, `auto`.

To use a value other than `auto`, you must set the `DataType` property of the object that uses the expression to the same value. Otherwise, the code generator does not preserve the expression.
 - If you want to use a parameter object in multiple different expressions, and the dependent parameter objects have different data types, leave the `DataType` property of the independent object at the default value, `auto`. In the generated code, the value of the macro is expressed as a floating-point number (with effective data type `double`), and the code initializes the dependent parameters by typecasting the result of each expression.

See Also

`Simulink.Parameter`

Related Examples

- “Create Tunable Calibration Parameter in the Generated Code” on page 15-112
- “Share and Reuse Block Parameter Values by Creating Variables”

Configure Packaging of Parameter Arguments in Generated Code

When you create a reusable component as a referenced model, configure block parameters to use the same value for instances of the component or a different value for each instance.

To customize the implementation of parameter arguments in the generated code, you can configure the packaging of these parameters in the model where they are defined.

Configure the default mapping for model parameter arguments by using one of these storage classes:

- **Default** — Each instance of the parameter is allocated a piece of memory in an exported structure whose attributes are defined by identifier naming rules.
- A structured storage class — Each instance of the parameter is allocated a piece of memory as defined by the code definition of that storage class.

Individually specify the storage class for model parameter arguments by using one of these storage classes:

- **Auto** — The parameter appears as an individual model function argument in the generated code. If a block in the model does not reference the parameter, it can be optimized away by the code generator.
- **Model default** — Each instance of the parameter is allocated a piece of memory as defined by the code definition of the default storage class. Specify this default storage class through the model default mapping.

If you use edit-time checking for code generation, you get an error for an invalid storage class.

For a parameter argument with a non-**Auto** storage class:

- On the **Model** block, you can specify the per-instance value as either a literal, a numeric MATLAB variable, or a `Simulink.Parameter` object.
- If the per-instance value is a `Simulink.Parameter` object, the object must have storage class **Auto**.
- If the per-instance value is a value expression, the evaluated value of the expression is used to statically initialize the parameter.

If you do not specify a default or individual storage class for a parameter argument, the code generator applies an internal storage class, `InstP`, to the parameter. The `InstP` storage class has these properties:

- Exported scope
- Type naming rule is `R_InstPM`
- Instance name is `InstPRM`

Code Generation Behavior

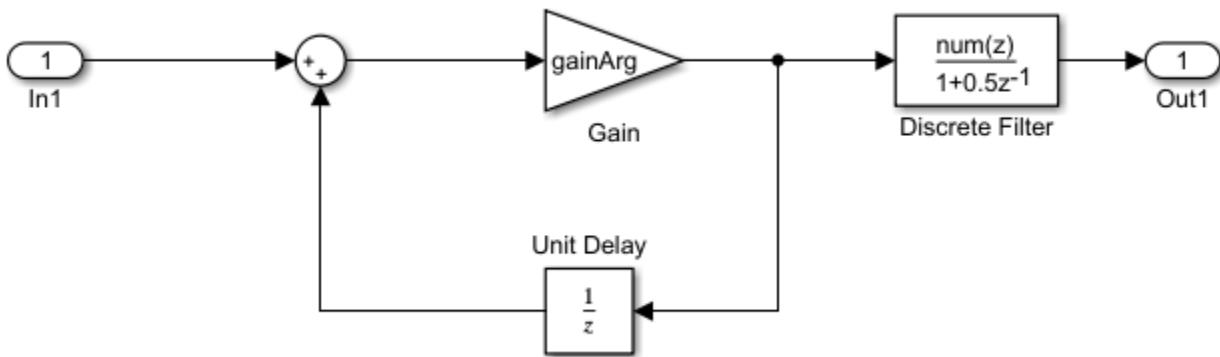
For single-instance referenced models, the parameter arguments reside in a standalone global structure. The top model defines the structure. The referenced model declares and uses the structure.

For multi-instance referenced models, the parameter arguments are part of a hierarchically nested structure that the top model declares and defines. The referenced model has access to its parameter values through a pointer in its self structure. The self structure points to a substructure of the structure that the top model defines.

For parameter arguments defined in the top model, if the argument does not have a storage class, it is inlined. If it has a storage class, the resulting code depends on the value of the **Code interface packaging** parameter:

- **Nonreusable function** — The parameter is generated in a structure defined in the top model.
- **Reusable function** — The parameter is generated in a structure passed as part of the first argument of the function.

To understand the behavior of instance-specific parameters in the generated code, consider this referenced model that defines two parameter arguments, `gainArg` and `coeffArg`.



Model Data									
Parameters									
Source	Name	Value	Data Type	Min	Max	Dimensions	Unit	Argument	
Model Workspace	coeffArg	1.05	auto	[]	[]	[1 1]		<input checked="" type="checkbox"/>	
Model Workspace	gainArg	3.17	auto	[]	[]	[1 1]		<input checked="" type="checkbox"/>	

First consider the case where this model is configured to support only a single instance and the default mapping for model parameter arguments does not have a storage class. In the `ex_arg_code_ref.h` file, there is a type definition and a declaration for the instance-specific parameters.

```
/* instance parameters */
extern ex_arg_code_ref_InstP ex_arg_code_refrtInstP;
```

```

/* instance parameters, for model 'ex_arg_code_ref' */
typedef struct {
    real32_T coeffArg;          /* Variable: coeffArg
                               * Referenced by: synthesized block
                               */
    real32_T gainArg;          /* Variable: gainArg
                               * Referenced by: synthesized block
                               */
} ex_arg_code_ref_InstP;

```

When the top model references this model and supplies values for the instance-specific parameters, the code for the top model defines the instance-specific parameters by using the values provided by the top model.

```

/* Define buffer for instance parameters, Block: '<Root>/Model' */
ex_arg_code_ref_InstP ex_arg_code_refrtInstP = {
    1.98F,
    3.98F
};

```

Now consider if the referenced model is configured to support multiple instances. `ex_arg_code_ref.h` contains the same type definition as the single-instance, but the self for the model also contains a pointer to an instance of that data structure. There is no pointer declaration to a standalone global variable of that structure.

```

/* instance parameters, for model 'ex_arg_code_ref' */
typedef struct {
    real32_T coeffArg;          /* Variable: coeffArg
                               * Referenced by: synthesized block
                               */
    real32_T gainArg;          /* Variable: gainArg
                               * Referenced by: synthesized block
                               */
} ex_arg_code_ref_InstP;

/* Real-time Model Data Structure */
struct ex_arg_code_ref_tag_RTM {
    ex_arg_code_ref_InstP *ex_arg_code_ref_InstP_ref;
};

```

The top model declares a parent structure, including an instance of this substructure, and initializes the pointer in the self structure for the referenced model.

```
/* Assign pointer for instance parameters, Block: '<Root>/Model' */
ex_arg_code_DW->Model_InstanceData.rtm.ex_arg_code_ref_InstP_ref =
    &ex_arg_code_M->ex_arg_code_InstP_ref->InstP_Model;

/* Assign pointer for instance parameters, Block: '<Root>/Model1' */
ex_arg_code_DW->Model1_InstanceData.rtm.ex_arg_code_ref_InstP_ref =
    &ex_arg_code_M->ex_arg_code_InstP_ref->InstP_Model1;
}
```

Limitations

When you configure an instance-specific parameter for code generation, these limitations apply:

- The default mapping for model parameter arguments cannot be reused for model parameters or external parameter objects.
- Instance-specific parameter values must be finite.
- Parameter arguments with a non-`Auto` storage class are not supported for C++ code generation with a C++ class code interface.

See Also

Related Examples

- “Generate Code for Model Reference Hierarchy” on page 8-2
- “Parameterize Instances of a Reusable Referenced Model”

Parameter Data Types in the Generated Code

The data type of a block parameter (such as the **Gain** parameter of a Gain block), numeric MATLAB variable, or `Simulink.Parameter` object determines the data type that the corresponding entity in the generated code uses (for example, a global variable or an argument of a function). To generate more efficient code, you can match parameter data types with signal data types or store parameters in smaller data types.

For basic information about setting block parameter data types in a model, see “Control Block Parameter Data Types”.

Significance of Parameter Data Types

The data type that a block parameter, MATLAB variable, or parameter object uses determines the data type that the generated code uses to store the parameter value in memory. For example:

- If you set the model configuration parameter **Default parameter behavior** (see Default parameter behavior) to `Tunable`, the **Gain** parameter of a Gain block appears in the generated code as a field of a global structure that stores parameter data. If you apply the data type `single` to the block parameter in the model, the structure field in the code uses the corresponding data type, `real32_T`.
- If you apply the storage class `ExportedGlobal` to a `Simulink.Parameter` object, the object appears in the generated code as a separate global variable. If you set the `DataType` property of the object to `int8`, the global variable in the code uses the corresponding data type, `int8_T`.
- If you configure a `Simulink.Parameter` object in a model workspace as a model argument, the object appears in the generated code as a formal parameter (argument) of a model entry-point function, such as the `step` function. The `DataType` property of the object determines the data type of the formal parameter.

Other than determining the data type that the generated code uses to store parameter values in memory, the data type of a parameter, variable, or object can also:

- Cause the block to cast the value of the parameter prior to code generation. The cast can result in overflow, underflow, or quantization.
- Cause the generated code to include extra code, for example saturation code.

Typecasts Due to Parameter Data Type Mismatches

When the data types of block parameters, workspace variables, and signals differ, blocks can use typecasts to reconcile the data type mismatches. These typecasts can cause the generated code algorithm, including the model `step` function, to include explicit casts to reconcile mismatches in storage data type and C bit shifts to reconcile mismatches in fixed-point scaling.

Parameter data type mismatches can occur when:

- The data type that you specify for a MATLAB variable or parameter object (`Simulink.Parameter`) differs from the data type of a block parameter. The block parameter typecasts the value of the variable or object.
- The data type that you specify for an initial value differs from the data type of the initialized signal or state.

- The data type that you specify for a block parameter differs from the data type of the signal or signals that the parameter operates on. Some blocks typecast the parameter to perform the operation. For example, the Gain block performs this typecast.

If you configure a variable or object to use bias or fractional fixed-point slope, the block parameter cannot perform the typecast. In this case, you must match the data type of the variable or parameter object with the data type of the block parameter. Use one of these techniques:

- Use context-sensitive data typing for the variable or parameter object. For a MATLAB variable, use a `double` number to set the value of the variable. For a parameter object, set the `DataType` property to `auto`.
- Use a `Simulink.AliasType` or `Simulink.NumericType` object to set the data type of the block parameter and the data type of a parameter object.

Use this technique when you cannot rely on context-sensitive data typing, for example, when you use the field of a structure to set the value of the block parameter.

- Manually specify the same data type for the block parameter and the variable or parameter object.

Use this technique to reduce the dependence of the model on inherited and context-sensitive data types and on external variables and objects.

For blocks that access parameter data through pointer or reference in the generated code, if you specify different data types for the workspace variable and the block parameter, the generated code implicitly casts the data type of the variable to the data type of the block parameter. Note, that an implicit cast requires a data copy which could significantly increase RAM consumption and slow down code execution speed for large data sets. For example, Lookup Table blocks often access large vectors or matrices through pointer or reference in the generated code.

For information about matching parameter data types when you use model arguments, see “Control Data Types of Model Arguments and Argument Values” on page 8-49.

Detect Downcast and Loss of Precision due to Data Type Mismatches

You can configure diagnostic configuration parameters to detect unintentional data type mismatches that result in quantization and loss of parameter precision. See “Model Configuration Parameters: Data Validity Diagnostics”.

Considerations for Other Modeling Patterns

When you use specific modeling patterns and constructs such as fixed-point data types, parameter structures, and lookup table objects, use different techniques to control parameter data types.

- “Tunable Parameters and Best-Precision Fixed-Point Scaling” on page 15-132
- “Control Data Types of Structure Fields” on page 15-133
- “Control Data Types of Lookup Table Objects” on page 15-133
- “Data Typing for Expressions in Parameter Objects” on page 15-134

Tunable Parameters and Best-Precision Fixed-Point Scaling

To apply best-precision fixed-point scaling to a tunable block parameter or parameter object, you can use the Fixed-Point Tool to autoscale an entire system or use the Data Type Assistant to configure individual parameters or objects. See “Calculate Best-Precision Fixed-Point Scaling for Tunable Block Parameters”.

If a tunable parameter uses best-precision fixed-point scaling, Simulink chooses a data type based on the minimum and maximum values that you specify for the parameter (see “Specify Minimum and Maximum Values for Block Parameters”). You can specify these values in the block dialog box that uses the parameter or in the properties of a `Simulink.Parameter` object.

If you do not specify a minimum or maximum, Simulink chooses a data type based on the value of the parameter. The chosen scaling might restrict the range of possible tuning values. Therefore, it is a best practice to specify minimum and maximum values for each tunable parameter.

A tunable parameter can use best-precision scaling even if you do not specify it in the parameter data type. For example, the Gain block can choose a best-precision scaling if the **Parameter data type** in the block dialog box is set to `Inherit: Inherit via internal rule`. This setting is the default for the block.

Control Data Types of Structure Fields

When you use a structure as the value of a block parameter (for example to initialize a bus signal), or when you organize multiple block parameter values into a single structure, you can create a `Simulink.Bus` object to use as the data type of a `Simulink.Parameter` object. You can then control the data types of individual fields in the structure. See “Control Field Data Types and Characteristics by Creating Parameter Object” and “Control Data Types of Initial Condition Structure Fields”.

Control Data Types of Lookup Table Objects

When you use `Simulink.LookupTable` and `Simulink.Breakpoint` objects to store table and breakpoint data for a lookup table block, to control the data types of the table and breakpoint data, use one of these techniques:

- Set the `Value` property of the embedded `Simulink.lookuptable.Table` and `Simulink.lookuptable.Breakpoint` objects by using untyped expressions such as `[1 2 3]`, which returns a `double` vector. If **Breakpoint specification** is set to `Explicit values`, use the MATLAB expression text box and tabular interface at the bottom of the property dialog box. To control the data type, set the `DataType` property to a value other than `auto`.

Use this technique to separate the value of the table or breakpoint data from the data type, which can improve readability and understanding of your design. You can then use a `Simulink.NumericType` or `Simulink.AliasType` object to:

- Customize the name of the data type in the generated code.
- Match the data type of the table or breakpoint data with the data type of a signal in the model.
- Set the `Value` property of the embedded objects by using typed expressions such as `single([1 2 3])`. To use a fixed-point data type, set the `Value` property with an `fi` object. If **Breakpoint specification** is set to `Explicit values`, use the MATLAB expression text box and tabular interface at the bottom of the property dialog box.

Set the `DataType` property of the embedded objects to the default value, `auto`. The table and breakpoint data then acquire the data type that you use to set the `Value` property.

Use this technique to store the data type information in the `Value` property, which can simplify the way you interact with the `Simulink.LookupTable` and `Simulink.Breakpoint` objects. You can leave the `DataType` property at the default value.

When you later change the breakpoint or table data in the `Value` property, preserve the data type information by using a typed expression. Alternatively, if you use a command at the command

prompt or a script to change the data, to avoid using a typed expression, use subscripted assignment, (:).

```
myLUTObject.Table.Value(:) = [4 5 6];
```

When you change the data stored in the `Value` property, if you do not use a typed expression or subscripted assignment, you lose the data type information.

When blocks in a subsystem use `Simulink.LookupTable` or `Simulink.Breakpoint` objects, you cannot set data type override (see “Control Fixed-Point Instrumentation and Data Type Override”) only on the subsystem. Instead, set data type override on the entire model.

Data Typing for Expressions in Parameter Objects

You can use an expression to set the value of a parameter object (such as `Simulink.Parameter`). The expression encodes mathematical relationships between different objects. When you use this technique, different data typing rules apply. See “Set Variable Value by Using a Mathematical Expression”.

See Also

Related Examples

- “Generate Efficient Code by Specifying Data Types for Block Parameters” on page 15-135
- “Reuse Parameter Data in Different Data Type Contexts” on page 15-143
- “Data Types Supported by Simulink”
- “Control Data Types of Signals”
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Generate Efficient Code by Specifying Data Types for Block Parameters

To generate more efficient code, match the data types of block parameters (such as the **Gain** parameter of a Gain block) with signal data types. Alternatively, you can store parameters in smaller data types.

Eliminate Unnecessary Typecasts and Shifts by Matching Data Types

These examples show how to generate efficient code by configuring a block parameter to use the same data type as a signal that the block operates on.

Store Data Type Information in Model

Open the example model `rtwdemo_configrpinterface` and configure it to show the generated names of blocks.

```
open_system('rtwdemo_configrpinterface')
set_param('rtwdemo_configrpinterface','HideAutomaticNames','off')
```

On the **Modeling** tab, click **Model Data Editor**.

In the Model Data Editor, select the **Parameters** tab.

In the model, select the Gain block. In the Model Data Editor, the **Data Type** column shows that the data type of the **Gain** parameter of the block is set to **Inherit: Same as input**. With this setting, the **Gain** parameter of this block uses the same data type as the input signal.

In the Code Mappings editor, under **Parameters**, set the storage class for K1 to `ExportedGlobal`.

In the Model Data Editor, set **Change view** to **Design**.

In the data table, for the row that represents K1, in the **Data Type** column, select **Auto**. With this setting, the parameter object acquires its data type from the block parameters that use the object (in this case, the **Gain** block parameter).

Alternatively, to create and configure the object, use these commands at the command prompt:

```
cm = coder.mapping.api.get('rtwdemo_configrpinterface');
setModelParameter(cm,'K1','StorageClass','ExportedGlobal');
hws = get_param(bdroot,'modelworkspace');
hws.setVariablePart('K1.DataType','Auto');
```

Generate code from the model.

```
slbuild('rtwdemo_configrpinterface')

### Starting build procedure for: rtwdemo_configrpinterface
### Successful completion of code generation for: rtwdemo_configrpinterface
```

Build Summary

Top model targets built:

```

Model                Action                Rebuild Reason
=====
rtwdemo_configrpinterface  Code generated.  Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 18.782s

```

The generated file `rtwdemo_configrpinterface.c` defines the global variable `K1` by using the data type `real_T`.

```

file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface.c');
rtwdemodbtype(file, /* Exported block parameters */,'real_T K1 = 2.0;',1,1)

/* Exported block parameters */
real_T K1 = 2.0;                /* Variable: K1

```

The generated code algorithm in the model `step` function uses `K1` directly without typecasting.

```

rtwdemodbtype(file,'output =',...
    '10U',1,1)

    output = K1 * look1_binlc((*input2), rtCP_Table1_bp01Data,
        rtCP_Table1_tableData, 10U);

```

In the Code Mappings editor, under **Parameters**, you can optionally set the storage class for `K1` to `Inherit: Inherit via internal rule` (the default).

In this case, the block parameter chooses the same data type as the input signal, `double`. However, when you use `Inherit: Inherit via internal rule`, under other circumstances (for example, when you use fixed-point data types), the block parameter might choose a different data type.

Store Data Type Information in Parameter Object

When you use a `Simulink.Parameter` object to export or import parameter data from the generated code to your external code, for example by applying the storage class `ImportedExtern`, you can specify data type information in the parameter object. To match the data type of the parameter object with a signal data type, create a `Simulink.NumericType` or `Simulink.AliasType` object. You can strictly control the data type that the parameter object uses in the generated code, eliminating the risk that Simulink® chooses a different data type when you make changes to the model.

At the command prompt, create a `Simulink.NumericType` object that represents the data type `double`.

```

myType = Simulink.NumericType;
myType.DataTypeMode = 'Double';

```

Use the Model Data Editor **Data Type** column to set these data types to `myType`:

- The parameter object. Use the **Parameters** tab.
- The Inport block named `In2`. Use the **Inports/Outports** tab. Due to data type propagation, the input signal of the Gain block also uses `myType`.

Use the Model Data Editor **Parameters** tab to set the data type of the **Gain** block parameter to `Inherit: Inherit from 'Gain'`.

Use this data type object as the data type of the parameter object.

Alternatively, to configure the object and the blocks, use these commands at the command prompt:

```
K1.DataType = 'myType';
set_param('rtwdemo_configrpinterface/In2','OutDataTypeStr','myType')
set_param('rtwdemo_configrpinterface/Gain','ParamDataTypeStr','Inherit: Inherit from ''Gain''')
```

Generate code from the model.

```
slbuild('rtwdemo_configrpinterface')

### Starting build procedure for: rtwdemo_configrpinterface
### Successful completion of code generation for: rtwdemo_configrpinterface
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_configrpinterface	Code generated.	Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 18.38s

The global variable K1 continues to use the data type real_T.

```
file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface.c');
rtwdemodbtype(file,'/* Exported block parameters */','real_T K1 = 2.0;',1,1)

/* Exported block parameters */
real_T K1 = 2.0;                                /* Variable: K1
```

Close the code generation report.

```
rtwdemoclean;
```

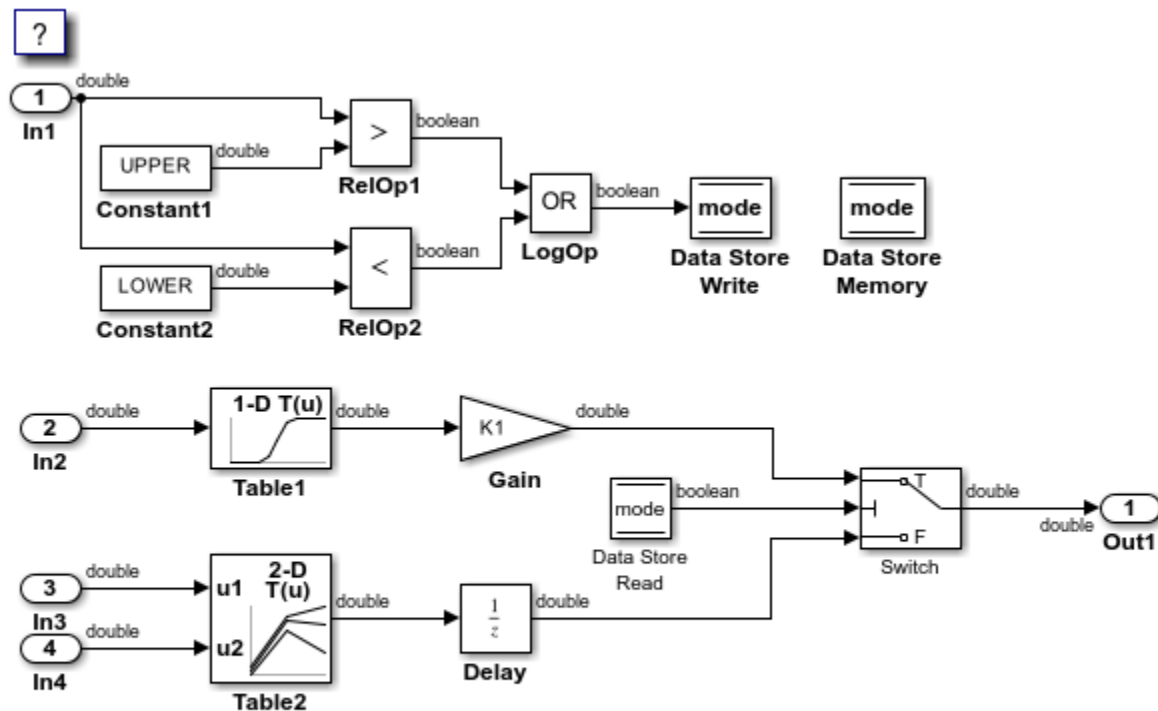
Reduce Memory Consumption by Storing Parameter Value in Small Data Type

When you use a parameter object (for example, `Simulink.Parameter`) to set block parameter values, you can configure the object to appear in the generated code as a tunable global variable. By default, the parameter object and the corresponding global variable typically uses the same data type as the signal or signals on which the block operates. For example, if the input signal of a Gain block uses the data type `int16`, the parameter object typically use the same data type. To reduce the amount of memory that this variable consumes, specify that the variable use a smaller integer data type such as `int8`.

Store Parameter Value in Integer Data Type

Open the model `rtwdemo_configrpinterface` and set the properties of the model.

```
open_system('rtwdemo_configrpinterface')
set_param('rtwdemo_configrpinterface','HideAutomaticNames','off')
set_param('rtwdemo_configrpinterface','ShowPortDataTypes','on')
set_param('rtwdemo_configrpinterface','SimulationCommand','Update')
```



Copyright 1994-2020 The MathWorks, Inc.

Many of the signals in the model use the data type `double`.

On the **Modeling** tab, click **Model Data Editor**.

In the Model Data Editor, inspect the **Parameters** tab.

Click the **Show/refresh additional information** button.

Next to the **Filter contents** box, activate the **Filter using selection** button.

In the model, click the Gain block. The Model Data Editor shows one row that corresponds to the **Gain** parameter of the block and one row that corresponds to the MATLAB® variable `K1`, which sets the parameter value to 2. `K1` resides in the model workspace.

In the C Code tab, select **Code Interface > Default Code Mappings**. In the Code Mappings editor, under **Parameters**, set the storage class for `K1` to `ExportedGlobal`.

```
cm = coder.mapping.api.get('rtwdemo_configrinterface');
setModelParameter(cm, 'K1', 'StorageClass', 'ExportedGlobal');
```

In the C Code tab, select **Code Interface > Default Code Mappings**. In the Model Data Editor, set **Change view** to **Design** and ensure that the **Data Type** column for `K1` is `int8`.

```
hws = get_param(bdroot, 'modelworkspace');
hws.setVariablePart('K1.DataType', 'int8');
```


For the row that represents the **Gain** block parameter, in the **Data Type** column, set the drop-down list to **Inherit: Inherit from 'Gain'**. With this setting, the **Gain** parameter of the block inherits the `int8` data type from the parameter object.

```
set_param('rtwdemo_configrpinterface/Gain','ParamDataTypeStr',...
    'Inherit: Inherit from ''Gain''')
```

Generate code from the model.

```
slbuild('rtwdemo_configrpinterface')

### Starting build procedure for: rtwdemo_configrpinterface
### Successful completion of code generation for: rtwdemo_configrpinterface
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_configrpinterface	Code generated.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 12.264s

The generated file `rtwdemo_configrpinterface.c` defines the global variable `K1` by using the data type `int8_T`, which corresponds to the data type `int8` in Simulink®.

```
file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface.c');
rtwdemodbtype(file,'/* Exported block parameters */','int8_T K1 = 2;',1,1)
```

```
/* Exported block parameters */
int8_T K1 = 2;                                /* Variable: K1
```

The code algorithm in the model `step` function uses `K1` to calculate the output of the **Gain** block. The algorithm casts `K1` to the data type `real_T` (double) because the signals involved in the calculation use the data type `real_T`.

```
rtwdemodbtype(file,'output =',...
    '10U',1,1)

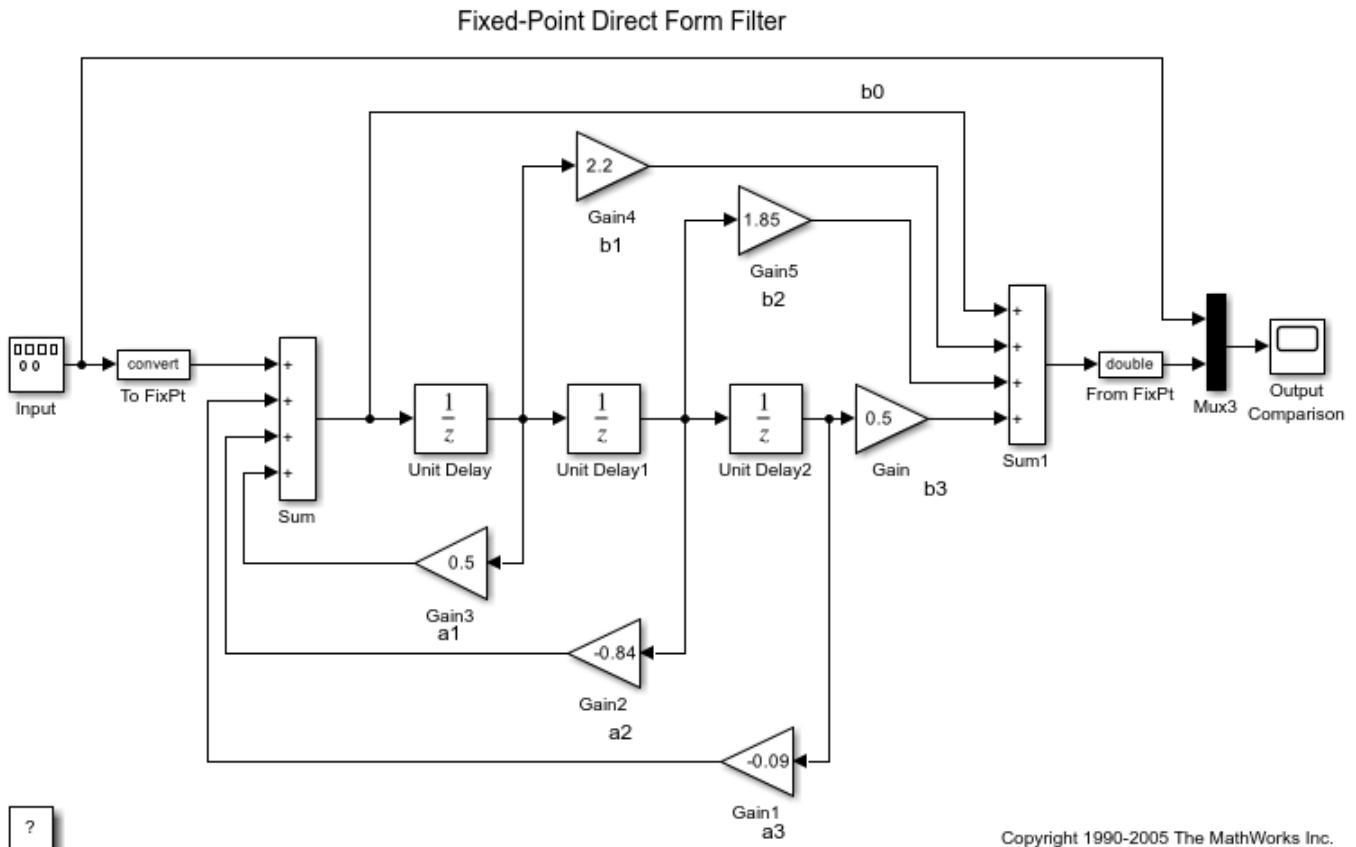
    output = (real_T)K1 * look1_binlc((*input2), rtCP_Table1_bp01Data,
        rtCP_Table1_tableData, 10U);
```

Store Fixed-Point Parameter Value in Smaller Integer Data Type

Suppose you configure the signals in your model to use fixed-point data types. You want a gain parameter to appear in the generated code as a tunable global variable. You know the range of real-world values that you expect the parameter to assume (for example, between 0 and 4). If you can meet your application requirements despite reduced precision, you can reduce memory consumption by configuring the parameter to use a different data type than the input and output signals of the block.

Open the example model `fxpdemo_direct_form2` and configure it to show the generated names of blocks.

```
load_system('fxpdemo_direct_form2')
set_param('fxpdemo_direct_form2','HideAutomaticNames','off')
open_system('fxpdemo_direct_form2')
```



Update the block diagram. Signals in this model use signed fixed-point data types with a word length of 16 and binary-point-only scaling.

Open the Gain5 block dialog box. The **Gain** parameter is set to 1.85. Suppose you want to configure this parameter.

Set **Gain** to myGainParam and click **Apply**.

Click the action button (with three vertical dots) next to the parameter value. Select **Create**.

In the Create New Data dialog box, set **Value** to `Simulink.Parameter(1.85)` and click **Create**. The `Simulink.Parameter` object myGainParam appears in the base workspace.

Open the **Simulink Coder** app. In the C Code tab, select **Code Interface > Default Code Mappings**. In the Code Mappings editor, under **Parameters**, set the storage class for myGainParam to `ExportedGlobal`. With this setting, myGainParam appears in the generated code as a global variable.

In the block dialog box, on the **Parameter Attributes** tab, set **Parameter minimum** to 0 and **Parameter maximum** to 4.

Set **Parameter data type** to `fixdt(0,8)` and click **Apply**.

Click the **Show Data Type Assistant** button. The Data Type Assistant shows that the expression `fixdt(0,8)` specifies an unsigned fixed-point data type with a word length of 8 and best-precision scaling. When you simulate or generate code, the block parameter chooses a fraction length (scaling) that enables the data type to represent values between the parameter minimum and maximum (0 and 4) with the best possible precision.

In the Data Type Assistant, set **Scaling** to **Binary point**. Click **Calculate Best-Precision Scaling**, **Fixed-point details**, and **Refresh Details**. The information under **Fixed-point details** shows that a fraction length of 5 can represent the parameter values with a precision of 0.03125.

Set **Scaling** back to **Best precision** and click **OK**. In this example, when you simulate or generate code, the block parameter chooses a fraction length of 5.

Alternatively, you can use these commands at the command prompt to create the object and configure the block:

```
myGainParam = Simulink.Parameter(1.85);
myGainParam.CoderInfo.StorageClass = 'ExportedGlobal';
set_param('fxpdemo_direct_form2/Gain5','Gain','myGainParam')
set_param('fxpdemo_direct_form2/Gain5','ParamMin','0','ParamMax','4')
set_param('fxpdemo_direct_form2/Gain5','ParamDataTypeStr','fixdt(0,8)')
```

Configure the model to produce a code generation report. To reduce clutter in the Command Window, clear the configuration parameter **Verbose build**.

```
set_param('fxpdemo_direct_form2','GenerateReport','on',...
    'LaunchReport','on','RTWVerbose','off')
```

Set hardware settings for the model

```
set_param('fxpdemo_direct_form2','TargetUnknown','off');
set_param('fxpdemo_direct_form2','PortableWordSizes','off');
set_param('fxpdemo_direct_form2','ProdEqTarget','off');
set_param('fxpdemo_direct_form2','TargetHWDeviceType','Custom Processor->MATLAB Host Computer');
```

Generate code from the model.

```
evalc('slbuild(''fxpdemo_direct_form2'')');
```

The generated file `fxpdemo_direct_form2.c` defines the global variable `myGainParam` by using the data type `uint8_T`, which corresponds to the specified word length, 8. The code initializes the variable by using an integer value that, given the fraction length of 5, represents the real-world parameter value 1.85.

```
file = fullfile('fxpdemo_direct_form2_grt_rtw','fxpdemo_direct_form2.c');
rtwdemodbtype(file,'/* Exported block parameters */','uint8_T myGainParam = 59U;',1,1)
```

```
/* Exported block parameters */
uint8_T myGainParam = 59U;          /* Variable: myGainParam
```

The code algorithm uses `myGainParam` to calculate the output of the Gain5 block. The algorithm uses a C shift to scale the result of the calculation.

```
rtwdemodbtype(file,'/* Gain: '<Root>/Gain5' incorporates:',...
    '/* Gain: '<Root>/Gain' incorporates:',1,0)
```

```
/* Gain: '<Root>/Gain5' incorporates:
```

```
*   UnitDelay: '<Root>/Unit Delay1'  
*/  
fxpdemo_direct_form2_B.Gain5 = (int16_T)((myGainParam *  
    fxdemo_direct_form2_B.UnitDelay1) >> 5);  
  
rtwdemoclean;
```

See Also

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Parameter Data Types in the Generated Code” on page 15-131
- “Reuse Parameter Data in Different Data Type Contexts” on page 15-143

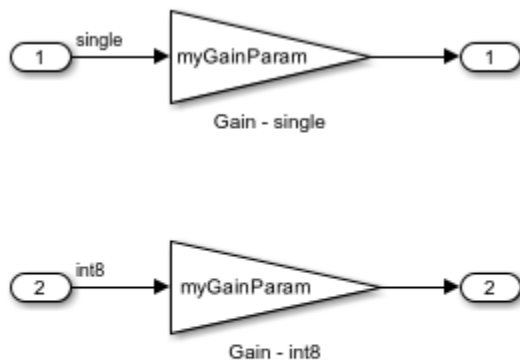
Reuse Parameter Data in Different Data Type Contexts

When you use a `Simulink.Parameter` object or a numeric MATLAB® variable to set two or more block parameter values, if the block parameters have different data types, you must explicitly specify the data type of the object or variable. For example, you cannot leave the data type of the parameter object at the default value, `auto`.

Create and Configure Example Model

Create the model `ex_paramdt_contexts`.

`ex_paramdt_contexts`



Copyright 2021 The MathWorks, Inc.

On the **Modeling** tab, click **Model Data Editor**.

In the Model Data Editor, on the **Imports/Outputs** tab, use the **Data Type** column to set the data type of the In1 Inport block to `single` and the data type of the In2 block to `int8`.

On the **Signals** tab, set the data types of the Gain block outputs to **Inherit: Same as input**.

On the **Parameters** tab, for the **Gain** parameters of the Gain blocks, set **Data Type** to **Inherit: Same as input**.

For the **Gain** parameters, set **Value** to `myGainParam`.

Alternatively, to configure the blocks, use these commands at the command prompt:

```
set_param('ex_paramdt_contexts/In1','OutDataTypeStr','single')
set_param('ex_paramdt_contexts/In2','OutDataTypeStr','int8')
set_param('ex_paramdt_contexts/Gain - single','Gain','myGainParam',...
    'OutDataTypeStr','Inherit: Same as input',...
    'ParamDataTypeStr','Inherit: Same as input')
set_param('ex_paramdt_contexts/Gain - int8','Gain','myGainParam',...
    'OutDataTypeStr','Inherit: Same as input',...
    'ParamDataTypeStr','Inherit: Same as input')
```

In the Model Data Editor, for either **Gain** block parameter, click the cell in the **Value** column. Next to myGainParam, click the action button (with three vertical dots) and select **Create**.

In the **Create New Data** dialog box, set **Value** to Simulink.Parameter(3) and click **Create**. A Simulink.Parameter object with value 3 appears in the model workspace.

In the **myGainParam** property dialog box, set **Data type** to int8.

On the Code Generation tab, click **Configure in Coder App**.

In the Code Mappings editor, set the storage class of myGainParam to ExportedGlobal. With the storage class ExportedGlobal, the object appears in the generated code as a global variable.

Alternatively, to create and configure the parameter object, use these commands at the command prompt:

```
mws = get_param('ex_paramdt_contexts', 'modelworkspace');
mws.assignin('myGainParam', Simulink.Parameter(3));
setVariablePart(mws, 'myGainParam.DataType', 'int8');
cm = coder.mapping.utils.create('ex_paramdt_contexts');
setModelParameter(cm, 'myGainParam', 'StorageClass', 'ExportedGlobal');
```

In this model, you use the parameter object myGainParam to set two block parameter values. The block parameters inherit different data types from the block input signals (single or int8). To use myGainParam in these different data type contexts, you explicitly specify the data type of the parameter object by setting the DataType property to int8.

Match Parameter Object Data Type with Signal Data Type

Optionally, use a Simulink.NumericType or Simulink.AliasType object to set the parameter object data type and one of the signal data types. This technique eliminates unnecessary typecasts and shifts in the generated code due to a mismatch between the parameter object data type and the signal data type.

At the command prompt, create a Simulink.NumericType object to represent the data type int8.

```
sharedType_int8 = fixdt('int8');
```

In the Model Data Editor, on the **Inports/Outports** tab, set the data type of the In2 Inport block to sharedType_int8.

On the **Parameters** tab, update the block diagram. The data table now contains a row that represents the parameter object, myGainParam.

Use the **Data Type** column to set the data type of the parameter object to sharedType_int8.

Alternatively, to configure the block and the object, use these commands at the command prompt:

```
myGainParam.DataType = 'sharedType_int8';
set_param('ex_paramdt_contexts/In2', 'OutDataTypeStr', 'sharedType_int8')
```

The parameter object and the signal use the data type int8. To change this data type, adjust the properties of the data type object sharedType_int8.

Generate and Inspect Code

Generate code from the model.

```
slbuild('ex_paramdt_contexts')

### Starting build procedure for: ex_paramdt_contexts
### Successful completion of build procedure for: ex_paramdt_contexts

Build Summary

Top model targets built:

Model                Action                Rebuild Reason
=====
ex_paramdt_contexts  Code generated and compiled.  Code generation information file does not exist

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.763s
```

The generated file `ex_paramdt_contexts.c` defines the global variable `myGainParam` by using the data type `int8_T`, which corresponds to the data type `int8` in Simulink®.

```
file = fullfile('ex_paramdt_contexts_grt_rtw','ex_paramdt_contexts.c');
rtwdemodbtype(file,'/* Exported block parameters */','int8_T myGainParam = 3;',1,1)
```

```
/* Exported block parameters */
int8_T myGainParam = 3;                /* Variable: myGainParam
```

The generated code algorithm in the model `step` function uses `myGainParam` to calculate the outputs of the two Gain blocks. In the case of the Gain block whose input signal uses the data type `single`, the code algorithm casts `myGainParam` to the data type `real32_T`, which corresponds to the data type `single` in Simulink.

```
rtwdemodbtype(file,'/* Model step function */',...
    '/* Model initialize function */',1,0)

/* Model step function */
void ex_paramdt_contexts_step(void)
{
    /* Output: '<Root>/Out1' incorporates:
     * Gain: '<Root>/Gain - single'
     * Inport: '<Root>/In1'
     */
    ex_paramdt_contexts_Y.Out1 = (real32_T)myGainParam * ex_paramdt_contexts_U.In1;

    /* Output: '<Root>/Out2' incorporates:
     * Gain: '<Root>/Gain - int8'
     * Inport: '<Root>/In2'
     */
    ex_paramdt_contexts_Y.Out2 = (int8_T)(myGainParam * ex_paramdt_contexts_U.In2);
}
```

See Also

Related Examples

- “Parameter Data Types in the Generated Code” on page 15-131

- “Generate Efficient Code by Specifying Data Types for Block Parameters” on page 15-135
- “Create Tunable Calibration Parameter in the Generated Code” on page 15-112

Organize Data into Structures in Generated Code

In C code, you use structures (`struct`) to store data in contiguous locations in memory. You can organize data, such as signals and states, by using meaningful names. Each structure acts as a namespace, so you can reuse a name to designate multiple data items. Like arrays, with structures, you can write code that efficiently transfers and operates on large amounts of data by using pointers.

By default, the code generator aggregates data into standard structures (see “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49). To control the characteristics of these structures or to override this default behavior by creating different structures, use the information in the table.

Goal	Technique
Control the characteristics of the standard data structures. For example, specify the names of the structure types, structure variables, and field names.	With Embedded Coder, see “Manage Replacement of Simulink Data Types in Generated Code” (Embedded Coder).
Control the placement of the structures in memory, for example, by inserting pragmas or other code decorations.	With Embedded Coder, see “Control Data and Function Placement in Memory by Inserting Pragmas” (Embedded Coder).

Goal	Technique
<p>Aggregate data into structures whose names and other basic characteristics you can control. As you add blocks and signals to the model, the resulting new data appears in these structures by default.</p>	<p>With Embedded Coder, apply a structured storage class to a category of data by using the Code Mapping Editor.</p> <ul style="list-style-type: none"> • When you generate nonreentrant, single-instance code by setting the model configuration parameter Code interface packaging to Nonreusable function, create flat, global structure variables by applying the built-in storage class Struct. Alternatively, use example storage classes created by Quick Start tool, ParamStruct and SignalStruct, or create and apply your own structured storage class. • When you generate multi-instance (reentrant) code from a model or a component, for example by setting Code interface packaging to a value other than Nonreusable function, in the Code Mappings editor, you cannot use the built-in storage class Struct or a structured storage class that you create in a package. Instead, create your own structured storage class by using an Embedded Coder Dictionary. If you use the Embedded Coder Quick Start tool to prepare the model for code generation (see “Generate Code by Using the Quick Start Tool” (Embedded Coder)), you can use example storage classes ParamStruct and SignalStruct. • You cannot use this technique to aggregate global data stores or global parameters (which include parameter objects that you store in the base workspace or a data dictionary). For these kinds of data, use other techniques to create structures. <p>For more information about the Code Mapping Editor, see “Configure Default C Code Generation for Categories of Data Elements and Functions” (Embedded Coder). To create your own storage class, see “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder).</p>

Goal	Technique
Improve readability of the generated code by organizing individual data items into a custom structure whose characteristics you can finely control.	<p>Create a <code>Simulink.Bus</code> object that represents the structure type that you want. Use the bus object to set the data types of nonvirtual bus signals and parameter structures in your model.</p> <ul style="list-style-type: none">• For an example that shows how to organize parameter data, see “Structures of Parameters” on page 15-152.• For an example that shows how to organize signal data, see “Structures of Signals” on page 15-154. <p>To create an array of structures, see “Arrays of Structures” on page 15-160.</p>
Exchange structured data between generated code and your external code, for example, when the external code already defines a custom structure type and a corresponding global variable.	<p>Create a <code>Simulink.Bus</code> object that represents the structure type that you want. If your external code already defines the type, use the <code>Simulink.importExternalCTypes</code> function to generate the bus object. Use the bus object to set the data types of nonvirtual bus signals and parameter structures in your model. For an example, see “Exchange Structured and Enumerated Data Between Generated and External Code” (Embedded Coder).</p>

Goal	Technique
Reduce the number of arguments (formal parameters) for generated functions.	<ul style="list-style-type: none"> • To reduce the number of arguments for the entry-point functions of the model from which you generate code, such as <code>model_step</code>, see “Reduce Number of Arguments by Using Structures” on page 15-47. • To reduce the number of arguments for the entry-point functions of a referenced model, in the referenced model: <ul style="list-style-type: none"> • Configure root-level Inport and Outport blocks to appear in the code as structure fields. Replace multiple such blocks with a single block that you configure as a nonvirtual bus signal. With this technique, you have the greatest control over the structure characteristics. As you add blocks to the model, you must manually modify the bus definitions. See “Nonvirtual Buses and Parameter Structures” on page 15-152. • Organize model arguments into a custom structure (see “Specify Instance-Specific Parameter Values for Reusable Referenced Model” on page 8-27). • To reduce the number of arguments for the entry-point functions of a masked, reentrant subsystem, aggregate mask parameters into a custom structure.
Organize lookup table data into a structure.	Use <code>Simulink.LookupTable</code> and <code>Simulink.Breakpoint</code> objects. See <code>Simulink.LookupTable</code> .
Generate bit fields.	See “Bitfields” (Embedded Coder) and “Optimize Generated Code by Packing Boolean Data into Bitfields” (Embedded Coder).

Techniques to Create Structures

To create structures in the generated code, you can use these techniques:

- Apply a structured storage class to categories of data by using the Code Mapping Editor. As you add blocks and signals to a model, new data elements acquire this storage class by default.
- Apply a structured storage class, such as the built-in storage class `Struct`, directly to individual data items by using the Model Data Editor.
- Create custom nonvirtual buses and parameter structures.

To decide which techniques to use, use the information in the table.

Capability	Default Application of Structured Storage Class	Direct Application of Structured Storage Class	Nonvirtual Buses and Parameter Structures
Aggregate new data items into a structure by default	Yes	No	No
Prevent elimination of the target data by optimizations (specify that the data appear in the generated code)	No	Yes	Only if you directly apply a storage class such as <code>ExportedGlobal</code> to the bus or structure
Aggregate data into a structure without changing the appearance of the block diagram	Yes	Yes	No
Place signal, state, and parameter data in the same structure	No	Yes	No
Include state data in a structure	Yes	Yes	No
Organize structures into nested structures	No	No	Yes
Organize structures into an array of structures	No	No	Yes
Control name of structure type	Yes	Yes, but the type name derives from the variable name, which you specify	Yes
Create a structure to reduce the number of arguments in a generated function	Yes, but you must create your own storage class by using an Embedded Coder Dictionary	No	Yes
Requires Embedded Coder	Yes	Yes	No

Default Application of Structured Storage Class

You can apply a default storage class to a category of model data. As you add blocks and signals to the model, the associated data acquire the default storage class that you specify. To aggregate new data into structures by default, you can apply structured storage classes. You must use example storage classes `ParamStruct` and `SignalStruct` created by the Quick Start tool or create your own storage class by using an Embedded Coder Dictionary.

You cannot use default application to aggregate global data stores or global parameters (parameter objects that you store in the base workspace or a data dictionary).

To apply default storage classes, use the Code Mapping Editor. In a model window, in the Apps gallery, click **Embedded Coder**. Then, under **Code Mappings > Data Defaults**, apply storage classes by using the **Storage Class** column.

For more information about the Code Mapping Editor, see “Configure Default C Code Generation for Categories of Data Elements and Functions” (Embedded Coder). To create your own storage class by using an Embedded Coder Dictionary, see “Create Code Definitions for Use in the Code Mappings Editor” (Embedded Coder).

Direct Application of Structured Storage Class

You can apply a structured storage class to individual data items. The direct application prevents code generation optimizations, such as **Default parameter behavior** and **Signal storage reuse**, from eliminating each data item from the generated code. The direct application also overrides the default storage classes that you specify with **Code Mappings > Data Defaults**.

To directly apply a storage class, use the Model Data Editor (on the **Modeling** tab, click **Model Data Editor**). Set **Change view** to Code and apply the storage class by using the **Storage Class** column.

For an example that shows how to use `Struct`, see “Configure Parameters for C Code Generation” (Embedded Coder). For more information about applying storage classes, see “Organize Parameter Data into a Structure by Using Struct Storage Class” (Embedded Coder).

Nonvirtual Buses and Parameter Structures

- To create a nonvirtual bus, use a Bus Creator block to organize multiple signal lines into a single bus, or configure an Inport block or Outport block as a nonvirtual bus. You must create a `Simulink.Bus` object that represents the structure type. For an example, see “Structures of Signals” on page 15-154. For information about nonvirtual buses, see “Create Nonvirtual Buses”.
- To create a parameter structure, use MATLAB commands or the Variable Editor to organize the values of multiple block parameters into a MATLAB structure. Optionally, create a `Simulink.Bus` object so that you can control the name of the structure type and other characteristics such as the data type and dimensions of each field. For an example, see “Structures of Parameters” on page 15-152. For general information about parameter structures, see “Organize Related Block Parameter Definitions in Structures”.

Structures of Parameters

Create a structure in the generated code. The structure stores parameter data.

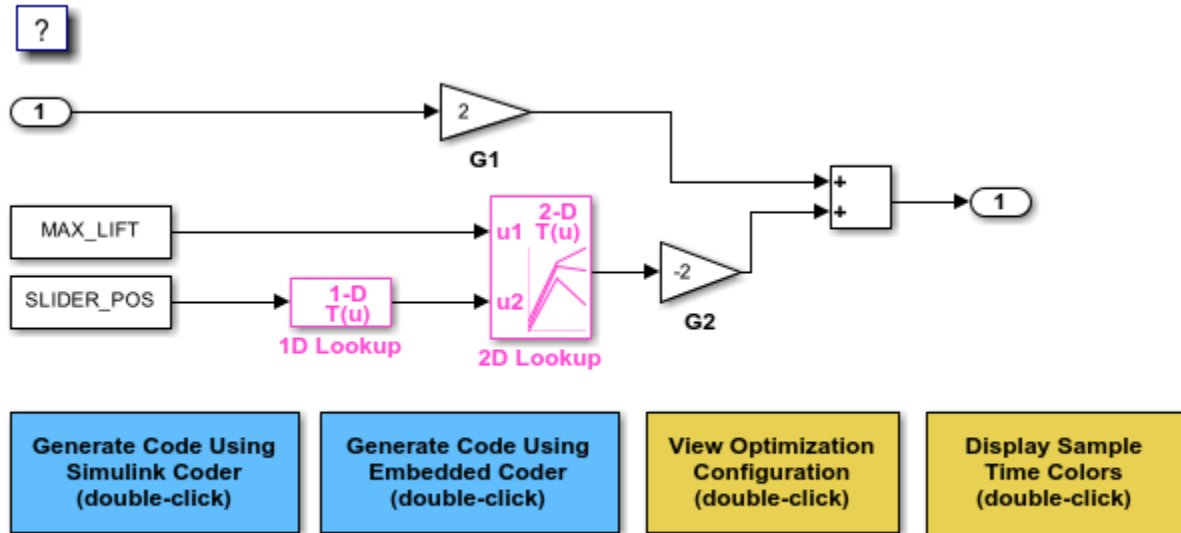
C Construct

```
typedef struct {
    double G1;
    double G2;
} myStructType;

myStructType myStruct = {
    2.0,
    -2.0
} ;
```

Procedure

1. Open the example model `rtwdemo_paraminline`.



Copyright 1994-2020 The MathWorks, Inc.

2. On the **Modeling** tab, click **Model Data Editor**. In the Model Data Editor, select the **Parameters** tab.
3. In the model, click the Gain block labeled G1. In the Model Data Editor, use the **Value** column to set the value of the **Gain** parameter to `myStruct.G1`.
4. Set the value of the **Gain** parameter in the G2 block to `myStruct.G2`.
5. Next to `myStruct.G2`, click the action button (with three vertical dots) and select **Create**.
6. In the Create New Data dialog box, set **Value** to `Simulink.Parameter(struct)` and click **Create**. A `Simulink.Parameter` object named `myStruct` appears in the base workspace.
7. In the `Simulink.Parameter` property dialog box, next to the **Value** property, click the action button and select **Open Variable Editor**.
8. Right-click the white space under the **Field** column and select **New**. Name the new structure field G1. Use the **Value** column to set the value of the field to 2.
9. Add a field G2 whose value is -2, and then close the Variable Editor.
10. In the `Simulink.Parameter` property dialog box, on the **Code Generation** tab, set **Storage class** to `ExportedGlobal`. The structure `myStruct` appears in the generated code as a global variable.
11. Generate code from the model.

Results

The generated header file `rtwdemo_paraminline_types.h` defines a structure type that has a random name.

```
typedef struct {
    real_T G1;
    real_T G2;
} struct_6h72eH5WFuEIyQr5YrdGuB;
```

The source file `rtwdemo_paraminline.c` defines and initializes the structure variable `myStruct`.

```
/* Exported block parameters */
struct_6h72eH5WFuEIyQr5YrdGuB myStruct = {
    2.0,
    -2.0
};

/* Variable: myStruct
 * Referenced by:
 *   '<Root>/G1'
 *   '<Root>/G2'
 */
```

Specify Name of Structure Type

1. Optionally, specify a name to use for the structure type definition (`struct`). At the command prompt, use the function `Simulink.Bus.createObject` to create a `Simulink.Bus` object that represents the structure type.
2. The default name of the object is `slBus1`. Change the name by copying the object into a new MATLAB® variable.
3. In the Model Data Editor, click the **Show/refresh additional information** button.
4. In the data table, find the row that corresponds to `myStruct`. Use the **Data Type** column to set the data type of `myStruct` to `Bus: myStructType`.
5. Generate code from the model.

The code generates the definition of the structure type `myStructType` and uses this type to define the global variable `myStruct`.

```
myStructType myStruct = {
    2.0,
    -2.0
};

/* Variable: myStruct
```

Structures of Signals

This example shows how to create a structure of signal data in the generated code.

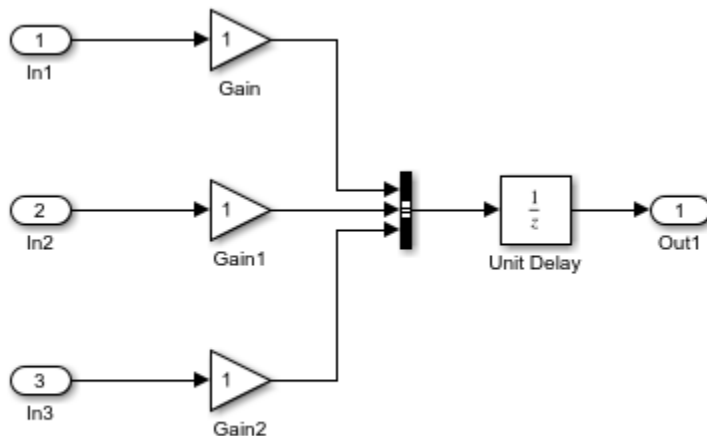
C Construct

```
typedef struct {
    double signal1;
    double signal2;
    double signal3;
} my_signals_type;
```


Procedure

To represent a structure type in a model, create a `Simulink.Bus` object. Use the object as the data type of buses in your model.

1. Create the `ex_signal_struct` model by using Gain blocks, a Bus Creator block, and a Unit Delay block. The Gain and Unit Delay blocks make the structure more identifiable in the generated code.



Copyright 2021 The MathWorks, Inc.

2. To configure the Bus Creator block to accept three inputs, in the block dialog box, set **Number of inputs** to 3.

3. In the toolstrip, on the **Modeling** tab, under **Design**, click **Type Editor**.

4. In the Type Editor toolstrip, in the **Add** gallery, click **Bus** to create a `Simulink.Bus` object. Then, name it `my_signals_type`. With `my_signals_type` selected, in the **Add** gallery, click **Bus Element** three times to add three elements. Name them `signal1`, `signal2`, and `signal3`. For more information, see Type Editor.

The screenshot shows the MATLAB/Simulink environment. On the left, the 'Contents of Base Workspace' table lists the following:

Name	Type	Complexity	Dimensions	DimensionsMode	Min	Max	Unit	Description
my_signals_type	Simulink.Bus	real	3	Fixed				
signal1	double	real	1	Fixed				
signal2	double	real	1	Fixed				
signal3	double	real	1	Fixed				

On the right, the 'Property Inspector' for the selected `my_signals_type` bus object is shown. The 'Name' property is set to `my_signals_type`.

5. Save the bus objects in your current folder as `ex_signal_struct_data.mat`.

This bus object represents the structure type that you want the generated code to use.

6. In the Bus Creator block dialog box, set **Output data type** to `Bus: my_signals_type`.

7. Select **Output as nonvirtual bus**. Click **OK**. A nonvirtual bus appears in the generated code as a structure.

8. Open the **Simulink Coder** app. In the **C Code** tab, select **Code Interface > Individual Element Code Mappings**.

9. Open the **Signals/States** tab. In the model, select the output signal of the Bus Creator block and click the **Add selected signals to code mappings** button in the Code Mappings editor.

11. For the added signal, set **Storage Class** to `ExportedGlobal`.

12. In the Property Inspector, set the **Code > Identifier** property to `sig_struct_var`. The output of the Bus Creator block appears in the generated code as a separate global structure variable named `sig_struct_var`.

13. Generate code from the model.

Results

The generated header file `ex_signal_struct_types.h` defines the structure type `my_signals_type`.

```
typedef struct {
    real_T signal1;
    real_T signal2;
    real_T signal3;
} my_signals_type;
```

The source file `ex_signal_struct.c` allocates memory for the global variable `sig_struct_var`, which represents the output of the Bus Creator block.

```
/* Exported block signals */
my_signals_type sig_struct_var;          /* '<Root>/Bus Creator' */
```

In the same file, in the model `step` function, the algorithm accesses `sig_struct_var` and the fields of `sig_struct_var`.

Nested Structures of Signals

You can create nested structures of signal data in the generated code.

C Construct

```
typedef struct {
    double signal1;
    double signal2;
    double signal3;
} B_struct_type;

typedef struct {
    double signal1;
    double signal2;
} C_struct_type;

typedef struct {
    B_struct_type subStruct_B;
```

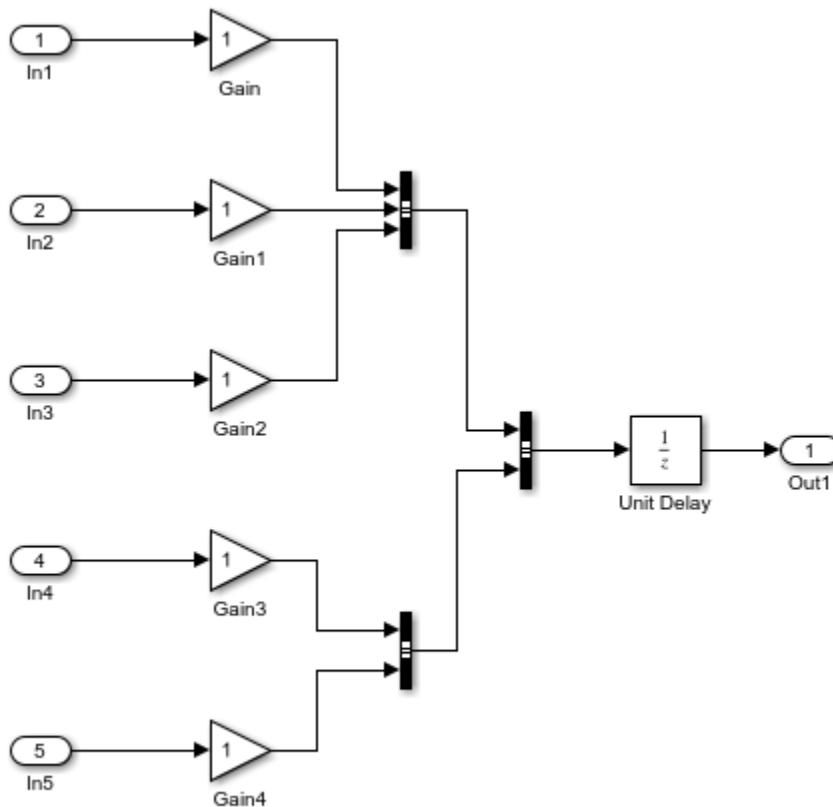
```
C_struct_type subStruct_C;
} A_struct_type;
```

Procedure

To represent a structure type in a model, create a `Simulink.Bus` object. Use the object as the data type of buses in your model.

To nest a structure inside another structure, use a bus object as the data type of an element in another bus object.

1. Create the `ex_signal_nested_struct` model with Gain blocks, Bus Creator blocks, and a Unit Delay block. The Gain and Unit Delay blocks make the structure more identifiable in the generated code.



Copyright 2021 The MathWorks, Inc.

2. To configure a Bus Creator block to accept three inputs, in the block dialog box, set **Number of inputs** to 3.

3. In the toolstrip, on the **Modeling** tab, under **Design**, click **Type Editor**.

In the Type Editor toolstrip, in the **Add** gallery, click **Bus** to create a `Simulink.Bus` object. Then, name it `A_struct_type`. With `A_struct_type` selected, in the **Add** gallery, click **Bus Element** twice to add two elements. Name the elements `subStruct_B` and `subStruct_C`. For more

information, see Type Editor. This bus object represents the top-level structure type that you want the generated code to use.

4. Similarly, create `Simulink.Bus` objects `B_struct_type` (with three elements) and `C_struct_type` (with two elements).

5. In the `A_struct_type` object, for the `subStruct_B` element, set **Data Type** to `Bus: B_struct_type`. Similarly, for the `subStruct_C` element, set **Data Type** to `Bus: C_struct_type`.

The screenshot shows the 'Contents of Base Workspace' window on the left and the 'Property Inspector' on the right. The workspace window displays a table of objects:

Name	Type	Complexity	Dimensions
▼ A_struct_type			
▼ subStruct_B	Bus: B_struct_type	real	1
— signal1	double	real	1
— signal2	double	real	1
— signal3	double	real	1
▼ subStruct_C	Bus: C_struct_type	real	1
— signal1	double	real	1
— signal2	double	real	1
▼ B_struct_type			
— signal1	double	real	1
— signal2	double	real	1
— signal3	double	real	1
▼ C_struct_type			
— signal1	double	real	1
— signal2	double	real	1

The Property Inspector on the right shows the selected object: `Simulink.Bus: A_struct_type`. Under the **Properties** section, the **Name** is set to `A_struct_type`. The **Description** field is empty. The **Code Generation** section is also visible but empty.

Each element in `A_struct_type` uses another bus object as a data type. These elements represent substructures.

6. In the dialog box of the Bus Creator block that collects the three Gain signals, set **Output data type** to `Bus: B_struct_type`. Click **Apply**.

7. Select **Output as nonvirtual bus** and click **OK**.

8. In the dialog box of the other subordinate Bus Creator block, set **Output data type** to `Bus: C_struct_type` and select **Output as nonvirtual bus**. Click **OK**.

9. In the last Bus Creator block dialog box, set **Output data type** to `Bus: A_struct_type` and select **Output as nonvirtual bus**. Click **OK**.

10. Open the **Simulink Coder** app. In the **C Code** tab, select **Code Interface > Individual Element Code Mappings**.

11. Open the **Signals/States** tab. In the model, select the output signal of the A_struct_type Bus Creator block, which feeds the Unit Delay block. Click the **Add selected signals to code mappings** button in the Code Mappings editor.
12. For the added signal, set **Storage Class** to ExportedGlobal.
13. In the Property Inspector, set the **Code > Identifier** property to sig_struct_var. The output of the Bus Creator block appears in the generated code as a separate global structure variable named sig_struct_var.
14. Generate code from the model.

Results

The generated header file `ex_signal_nested_struct_types.h` defines the structure types. Each structure type corresponds to a `Simulink.Bus` object.

```
typedef struct {
    real_T signal1;
    real_T signal2;
    real_T signal3;
} B_struct_type;

typedef struct {
    real_T signal1;
    real_T signal2;
} C_struct_type;

typedef struct {
    B_struct_type subStruct_B;
    C_struct_type subStruct_C;
} A_struct_type;
```

The generated source file `ex_signal_nested_struct.c` allocates memory for the global structure variable `sig_struct_var`. By default, the name of the A_struct_type Bus Creator block is Bus Creator2.

```
/* Exported block signals */
A_struct_type sig_struct_var;          /* '<Root>/Bus Creator2' */
```

In the same file, in the model step function, the algorithm accesses `sig_struct_var` and the fields of `sig_struct_var`.

Combine Techniques to Work Around Limitations

To work around some of the limitations of each technique, you can combine structured storage classes with nonvirtual buses and parameter structures. For example, you can:

- Include a structure of signal data and a structure of parameter data in the same parent structure.
- Nest structures while aggregating new data by default.

In the generated code, a flat parent structure, which corresponds to the structured storage class, contains substructures, which correspond to each bus and parameter structure. Choose one of these combined techniques:

- Apply the structured storage class directly to each bus and parameter structure. For example, set the storage class of two nonvirtual bus signals to `Struct`. Each bus appears in the generated code as a field (substructure) of a single structure.
- Leave the storage class of each bus and parameter structure at the default setting, `Auto`, or at `Model default`, which prevents code generation optimizations from eliminating the bus or parameter structure. Then, configure default storage classes such that signal data and parameter data use the structured storage class by default.

Arrays of Structures

You can further package multiple consistent bus signals or parameter structures into an array. The array of buses or parameter structures appears in the generated code as an array of structures. To create arrays of buses, see “Group Nonvirtual Buses in Arrays of Buses”. To create arrays of parameter structures, see “Group Multiple Parameter Structures into an Array”.

Structure Padding

By default, the code generator does not explicitly add padding fields to structure types. Structure types can appear in the generated code through, for example, the standard data structures (see “Data Structures in the Generated Code” on page 15-31), `Simulink.Bus` objects, and parameter structures that you use in a model.

However, when you use a code replacement library with Embedded Coder, you can specify data alignment (including structure padding) as part of the replacement library. For more information, see “Provide Data Alignment Specifications for Compilers” (Embedded Coder).

Limitations

- With built-in Simulink Coder and Embedded Coder features, you cannot generate or use a custom structure that contains a field whose value is a pointer. You can do so manually by creating an advanced storage class and writing accompanying TLC code (see “Finely Control Data Representation by Writing TLC Code for a Storage Class” (Embedded Coder)).
- You cannot use the built-in storage class `Struct`, or a structured storage class that you create with the Custom Storage Class Designer (you set the storage class property **Type** to `FlatStructure`), to set data defaults in the Code Mapping Editor.

See Also

`Simulink.Bus`

Related Examples

- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49
- “Exchange Data Between External C/C++ Code and Simulink Model or Generated Code” on page 30-59
- “C Code Generation Configuration for Model Interface Elements” (Embedded Coder)
- “Control Data and Function Placement in Memory by Inserting Pragmas” (Embedded Coder)
- “Flexible Storage Class for Different Model Hierarchy Contexts” (Embedded Coder)

- “Composite Interface Guidelines”

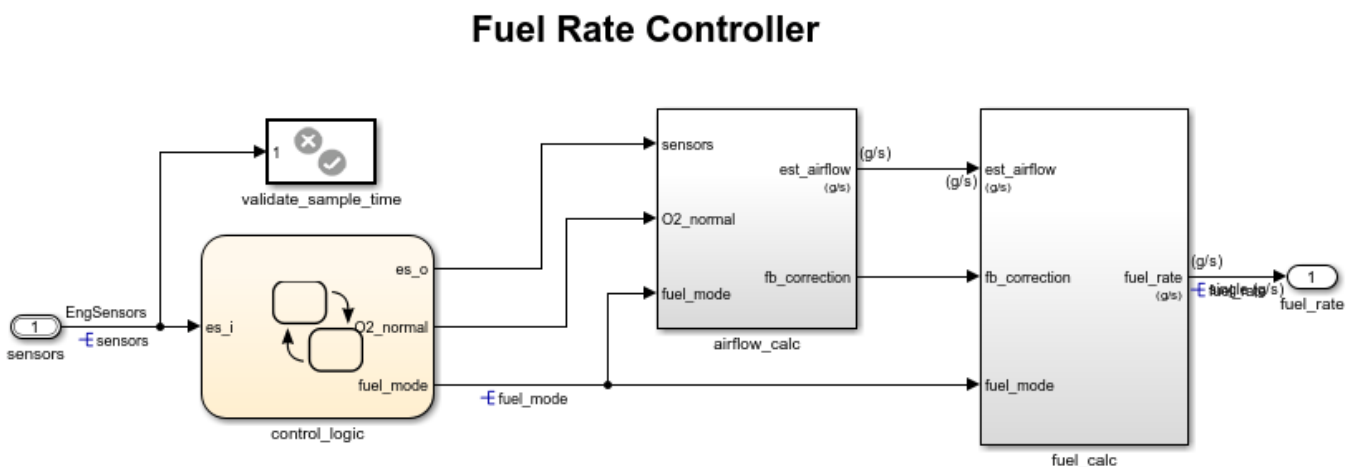
Switch Between Sets of Parameter Values During Simulation and Code Execution

To store multiple independent sets of values for the same block parameters, you can use an array of structures. To switch between the parameter sets, create a variable that acts as an index into the array, and change the value of the variable. You can change the value of the variable during simulation and, if the variable is tunable, during execution of the generated code.

Explore Example Model

Open the example model.

```
open_system('sldemo_fuelsys_dd_controller')
```



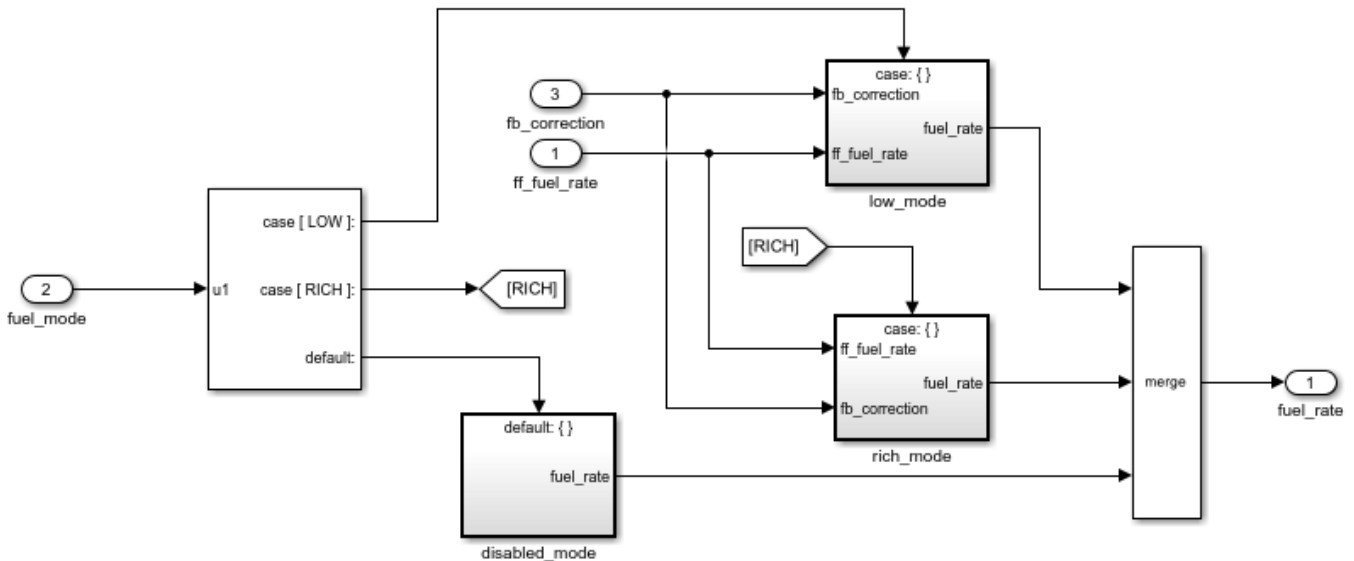
Copyright 1990-2022 The MathWorks, Inc.

This model represents the fueling system of a gasoline engine. The output of the model is the rate of fuel flow to the engine.

Navigate to the `switchable_compensation` nested subsystem.

```
open_system(['sldemo_fuelsys_dd_controller/fuel_calc/', ...
            'switchable_compensation'])
```

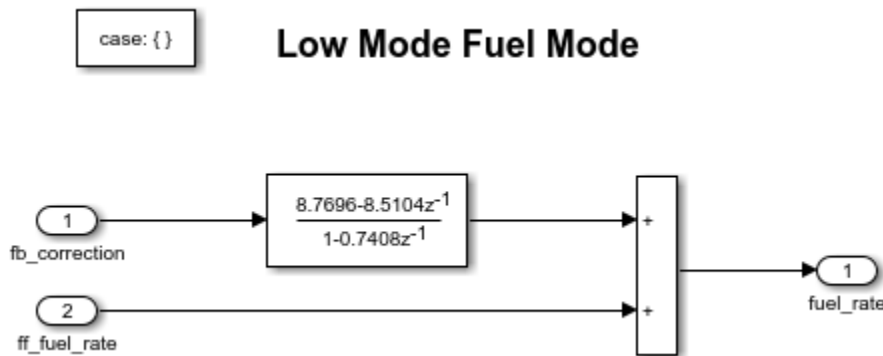

Loop Compensation and Filtering



This subsystem corrects and filters noise out of the fuel rate signal. The subsystem uses different filter coefficients based on the fueling mode, which the control logic changes based on sensor failures in the engine. For example, the control algorithm activates the `low_mode` subsystem during normal operation. It activates the `rich_mode` subsystem in response to sensor failure.

Open the `low_mode` subsystem.

```
open_system(['sldemo_fuelsys_dd_controller/fuel_calc/', ...
            'switchable_compensation/low_mode'])
```



The Discrete Filter block filters the fuel rate signal. In the block dialog box, the **Numerator** parameter sets the numerator coefficients of the filter.

The sibling subsystem `rich_mode` also contains a Discrete Filter block, which uses different coefficients.

Update the model diagram to display the signal data types. The input and output signals of the block use the single-precision, floating-point data type `single`.

In the lower-left corner of the model, click the model data badge, then click the **Data Dictionary** link. The data dictionary for this model, `sldemo_fuelsys_dd_controller.slidd`, opens in the Model Explorer.

In the Model Explorer **Model Hierarchy** pane, select the **Design Data** node.

In the **Contents** pane, view the properties of the Simulink.NumericType objects, such as `s16En15`. All of these objects currently represent the single-precision, floating-point data type `single`. The model uses these objects to set signal data types, including the input and output signals of the Discrete Filter blocks.

Suppose that during simulation and execution of the generated code, you want each of these subsystems to switch between different numerator coefficients based on a variable whose value you control.

Store Parameter Values in Array of Structures

Store the existing set of numerator coefficients in a Simulink.Parameter object whose value is a structure. Each field of the structure stores the coefficients for one of the Discrete Filter blocks.

```
lowBlock = ['sldemo_fuelsys_dd_controller/fuel_calc/'...
            'switchable_compensation/low_mode/Discrete Filter'];
richBlock = ['sldemo_fuelsys_dd_controller/fuel_calc/'...
             'switchable_compensation/rich_mode/Discrete Filter'];
params.lowNumerator = eval(get_param(lowBlock, 'Numerator'));
params.richNumerator = eval(get_param(richBlock, 'Numerator'));
params = Simulink.Parameter(params);
```

Copy the value of `params` into a temporary variable. Modify the field values in this temporary structure, and assign the modified structure as the second element of `params`.

```
temp = params.Value;
temp.lowNumerator = params.Value.lowNumerator * 2;
temp.richNumerator = params.Value.richNumerator * 2;
params.Value(2) = temp;
clear temp
```

The value of `params` is an array of two structures. Each structure stores one set of filter coefficients.

Create Variable to Switch Between Parameter Sets

Create a Simulink.Parameter object named `Ctrl`.

```
Ctrl = Simulink.Parameter(2);
Ctrl.DataType = 'uint8';
```

In the `low_mode` subsystem, in the Discrete Filter block dialog box, set the **Numerator** parameter to the expression `params(Ctrl).lowNumerator`.

```
set_param(lowBlock, 'Numerator', 'params(Ctrl).lowNumerator');
```

In the Discrete Filter block in the `rich_mode` subsystem, set the value of the **Numerator** parameter to `params(Ctrl).richNumerator`.

```
set_param(richBlock, 'Numerator', 'params(Ctrl).richNumerator');
```

The expressions select one of the structures in `params` by using the variable `Ctrl`. The expressions then dereference one of the fields in the structure. The field value sets the values of the numerator coefficients.

To switch between the sets of coefficients, you change the value of `Ctrl` to the corresponding index in the array of structures.

Use Bus Object as Data Type of Array of Structures

Optionally, create a `Simulink.Bus` object to use as the data type of the array of structures. You can:

- Control the shape of the structures.
- For each field, control characteristics such as data type and physical units.
- Control the name of the `struct` type in the generated code.

Use the function `Simulink.Bus.createObject` to create the object and rename the object as `paramsType`.

```
Simulink.Bus.createObject(params.Value)
paramsType = slBus1;
clear slBus1
```

You can use the `Simulink.NumericType` objects from the data dictionary to control the data types of the structure fields. In the bus object, use the name of a data type object to set the `DataType` property of each element.

```
paramsType.Elements(1).DataType = 's16En15';
paramsType.Elements(2).DataType = 's16En7';
```

Use the bus object as the data type of the array of structures.

```
params.DataType = 'Bus: paramsType';
```

Use Enumerated Type for Switching Variable

Optionally, use an enumerated type as the data type of the switching variable. You can associate each of the parameter sets with a meaningful name and restrict the allowed values of the switching variable.

Create an enumerated type named `FilterCoeffs`. Create an enumeration member for each of the structures in `params`. Set the underlying integer value of each enumeration member to the corresponding index in `params`.

```
Simulink.defineIntEnumType('FilterCoeffs',{'Weak','Aggressive'},[1 2])
```

Use the enumerated type as the data type of the switching variable. Set the value of the variable to `Aggressive`, which corresponds to the index 2.

```
Ctrl.Value = FilterCoeffs.Aggressive;
```

Add New Objects to Data Dictionary

Add the objects that you created to the data dictionary `sldemo_fuelsys_dd_controller.slidd`.

```
dictObj = Simulink.data.dictionary.open('sldemo_fuelsys_dd_controller.slidd');
sectObj = getSection(dictObj,'Design Data');
addEntry(sectObj,'Ctrl',Ctrl)
```

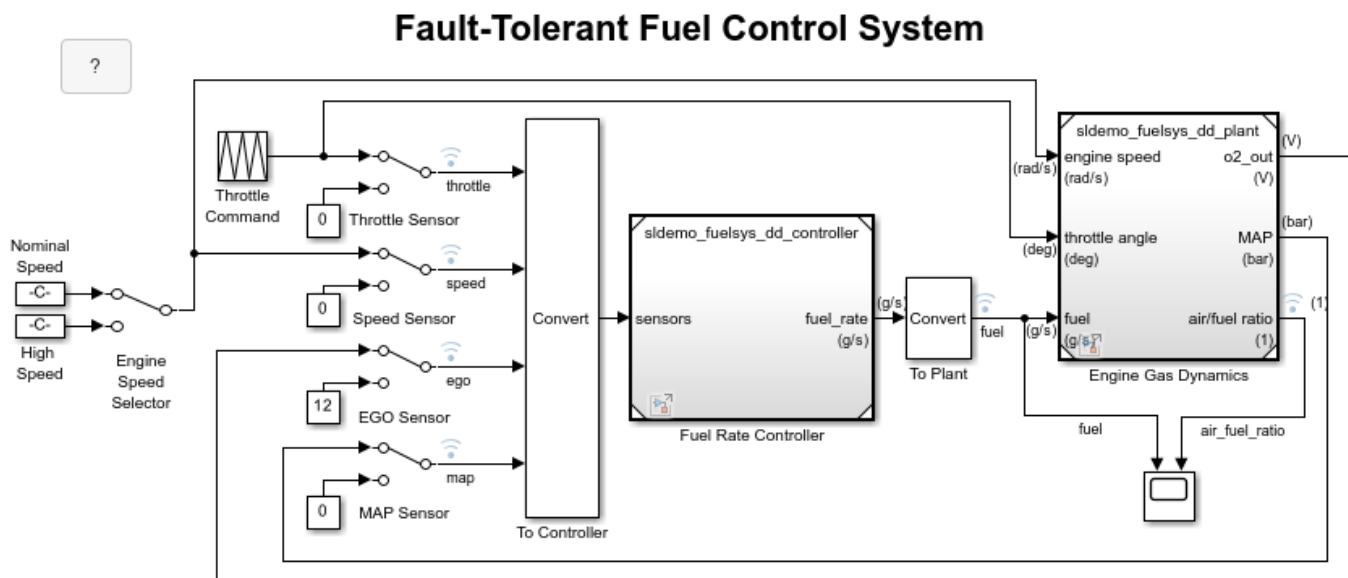
```
addEntry(sectObj, 'params', params)
addEntry(sectObj, 'paramsType', paramsType)
```

You can also store enumerated types in data dictionaries. However, you cannot import the enumerated type in this case because you cannot save changes to `sldemo_fuelsys_dd_controller.sldd`. For more information about storing enumerated types in data dictionaries, see “Enumerations in Data Dictionary”.

Switch Between Parameter Sets During Simulation

Open the example model `sldemo_fuelsys_dd`, which references the controller model `sldemo_fuelsys_dd_controller`.

```
open_system('sldemo_fuelsys_dd')
```



The sensor switches simulate any combination of sensor failures. The Engine Speed Selector switch simulates different engine speeds (rad/s).

Copyright 1990-2022 The MathWorks, Inc.

Set the simulation stop time to `Inf` so that you can interact with the model during simulation.

Begin a simulation run and open the Scope block dialog box. The scope shows that the fuel flow rate (the `fuel` signal) oscillates with significant amplitude during normal operation of the engine.

In the Model Explorer, view the contents of the data dictionary `sldemo_fuelsys_dd_controller.sldd`. Set the value of `Ctrl` to `FilterCoeffs.Weak`.

Update the `sldemo_fuelsys_dd` model diagram. The scope shows that the amplitude of the fuel rate oscillations decreases due to the less aggressive filter coefficients.

Stop the simulation.

Generate and Inspect Code

If you have Simulink Coder software, you can generate code that enables you to switch between the parameter sets during code execution.

In the Model Explorer, view the contents of the data dictionary `sldemo_fuelsys_dd_controller.sldd`. In the **Contents** pane, set **Column View** to **Storage Class**.

Use the **StorageClass** column to apply the storage class `ExportedGlobal` to `params` so that the array of structures appears as a tunable global variable in the generated code. Apply the same storage class to `Ctrl` so that you can change the value of the switching variable during code execution.

Alternatively, to configure the objects, use these commands:

```
tempEntryObj = getEntry(sectObj, 'params');
params = getValue(tempEntryObj);
params.StorageClass = 'ExportedGlobal';
setValue(tempEntryObj, params);

tempEntryObj = getEntry(sectObj, 'Ctrl');
Ctrl = getValue(tempEntryObj);
Ctrl.StorageClass = 'ExportedGlobal';
setValue(tempEntryObj, Ctrl);
```

Generate code from the controller model.

```
slbuild('sldemo_fuelsys_dd_controller')

### Starting build procedure for: sldemo_fuelsys_dd_controller
### Successful completion of code generation for: sldemo_fuelsys_dd_controller
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
sldemo_fuelsys_dd_controller	Code generated.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)

Build duration: 0h 1m 6.3938s

In the code generation report, view the header file `sldemo_fuelsys_dd_controller_types.h`. The code defines the enumerated data type `FilterCoeffs`.

```
file = fullfile('sldemo_fuelsys_dd_controller_ert_rtw', ...
    'sldemo_fuelsys_dd_controller_types.h');
rtwdemodbtype(file, '#ifndef DEFINED_TYPEDEF_FOR_FilterCoeffs_', ...
    '/* Forward declaration for rtModel */', 1, 0)
```

```
#ifndef DEFINED_TYPEDEF_FOR_FilterCoeffs_
#define DEFINED_TYPEDEF_FOR_FilterCoeffs_
```

```
typedef enum {
    Weak = 1, /* Default value */
```

```

    Aggressive
} FilterCoeffs;

#endif

```

The code also defines the structure type `paramsType`, which corresponds to the `Simulink.Bus` object. The fields use the single-precision, floating-point data type from the model.

```

rtwdemodbtype(file, '#ifndef DEFINED_TYPEDEF_FOR_paramsType_', ...
              '#ifndef DEFINED_TYPEDEF_FOR_FilterCoeffs_', 1, 0)

#ifndef DEFINED_TYPEDEF_FOR_paramsType_
#define DEFINED_TYPEDEF_FOR_paramsType_

typedef struct {
    real32_T lowNumerator[2];
    real32_T richNumerator[2];
} paramsType;

#endif

```

View the source file `sldemo_fuelsys_dd_controller.c`. The code uses the enumerated type to define the switching variable `Ctrl`.

```

file = fullfile('sldemo_fuelsys_dd_controller_ert_rtw', ...
              'sldemo_fuelsys_dd_controller.c');
rtwdemodbtype(file, 'FilterCoeffs Ctrl = Aggressive;', ...
              '/* Block signals (default storage) */', 1, 0)

FilterCoeffs Ctrl = Aggressive;          /* Variable: Ctrl
                                         * Referenced by:
                                         *   '<S12>/Discrete Filter'
                                         *   '<S13>/Discrete Filter'
                                         */

```

The code also defines the array of structures `params`.

```

rtwdemodbtype(file, '/* Exported block parameters */', ...
              '/* Variable: params', 1, 1)

/* Exported block parameters */
paramsType params[2] = { {
    { 8.7696F, -8.5104F },

    { 0.0F, 0.2592F }
}, { { 17.5392F, -17.0208F },

    { 0.0F, 0.5184F }
} } ;                                     /* Variable: params

```

The code algorithm in the model `step` function uses the switching variable to index into the array of structures.

To switch between the parameter sets stored in the array of structures, change the value of `Ctrl` during code execution.

Close the connections to the data dictionary. This example discards unsaved changes. To save the changes, use the '-save' option.

```
Simulink.data.dictionary.closeAll('sldemo_fuelsys_dd_controller.sldd','-discard')
```

See Also

Related Examples

- “Tune and Experiment with Block Parameter Values”
- “Create Tunable Calibration Parameter in the Generated Code” on page 15-112
- “Organize Related Block Parameter Definitions in Structures”
- “Access Structured Data Through a Pointer That External Code Defines” (Embedded Coder)

Design Data Interface by Configuring Inport and Output Blocks

The data interface of a model is the means by which the model exchanges data (for example, signal values) with other, external models or systems. Customize the data interface of a model to:

- Enable integration of the generated code with your own code.
- Improve traceability and readability of the code.

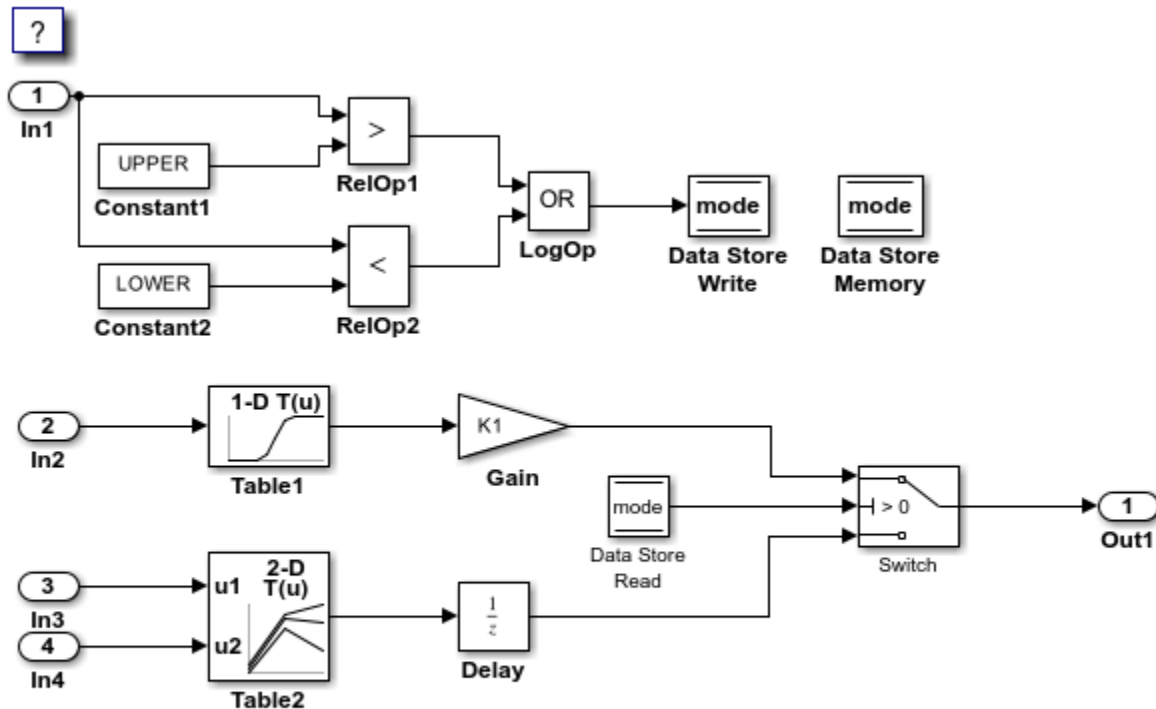
At the top level of a model, Inport and Output blocks represent the input and output signals of the model. To customize the data interface in the generated code, configure these blocks. Early in the design process, when a model can contain unconnected Inport and Output blocks, use this technique to specify the interface before developing the internal algorithm.

When you apply storage classes to Inport and Output blocks, each block appears in the generated code as a field of a global structure or as a separate global variable that the generated algorithm references directly. If you have Embedded Coder®, you can use function prototype control instead of storage classes to pass data into and out of the model `step` function as formal parameters. For more information, see “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder).

Design Data Interface

Open the example model `rtwdemo_configrpinterface`.

```
open_system('rtwdemo_configrpinterface')
```

Copyright 1994-2020 The MathWorks, Inc.

Configure the model to show the generated names of blocks.

```
set_param('rtwdemo_configrpinterface', 'HideAutomaticNames', 'off')
```

On the **Modeling** tab, click **Model Data Editor**.

In the Model Data Editor, select the **Inports/Outports** tab. Each row in the table represents an Output block or a signal that exits an Inport block.

For each of the signals that exit the Inport blocks, set **Data Type** to `single` or to a different data type. Due to the data type inheritance settings that the other blocks in the model use by default, downstream signals in the rest of the model use the same or a similar data type.

Optionally, configure other design attributes such as **Min** and **Max** (minimum and maximum values).

Configure Default Storage Class

You can configure a default storage class for Inport blocks and Outport blocks then select the blocks that should use the specified default.

- 1 Open the **Simulink Coder** app.
- 2 In the C Code tab, select **Code Interface > Default Code Mappings**.
- 3 In the Code Mappings editor, under **Inports and Outports**, set the default storage class for both **Inports** and **Outports** to `ExportedGlobal`.

- 4 Next, on the **Inports** tab, select the data items that should use the default storage class. Press **Ctrl** and click each inport.
- 5 For the selected inports, set the storage class to `Model default: ExportedGlobal`.
- 6 Similarly, on the **Outports** tab, set the storage class for the output to `Model default: ExportedGlobal`.

Alternatively, to configure the data defaults at the command prompt, use these commands:

```
cm = coder.mapping.api.get('rtwdemo_configrpinterface');
setDataDefault(cm,'Inports','StorageClass','ExportedGlobal');
setDataDefault(cm,'Outports','StorageClass','ExportedGlobal');

setInport(cm,'In1','StorageClass','Model default');
setInport(cm,'In2','StorageClass','Model default');
setInport(cm,'In3','StorageClass','Model default');
setInport(cm,'In4','StorageClass','Model default');

setOutput(cm,'Out1','StorageClass','Model default','Identifier','output_sig');
```

Generate code from the model.

```
slbuild('rtwdemo_configrpinterface');

### Starting build procedure for: rtwdemo_configrpinterface
### Successful completion of code generation for: rtwdemo_configrpinterface
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_configrpinterface	Code generated.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 13.312s

View the generated file `rtwdemo_configrpinterface.c`. Because you applied the storage class `ExportedGlobal` to the Inport and Output blocks, the code creates separate global variables that represent the inputs and the output.

```
file = fullfile('rtwdemo_configrpinterface_grt_rtw','rtwdemo_configrpinterface.c');
rtwdemodbtype(file,'/* Data with Exported storage */','real_T output_sig',1,1)

/* Data with Exported storage */
real_T input1;          /* '<Root>/In1' */
real_T input2;          /* '<Root>/In2' */
real_T input3;          /* '<Root>/In3' */
real_T input4;          /* '<Root>/In4' */
real_T output_sig;     /* '<Root>/Out1' */
```

The generated algorithm in the model `step` function directly references these global variables to calculate and store the output signal value, `output_sig`.

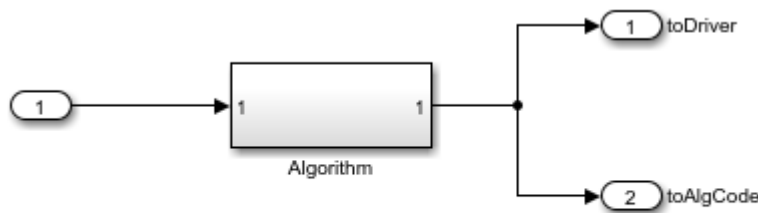
Route Signal Data to Multiple Outputs

Once you have configured the default storage classes for Inports and Outports, you can choose to override this default and set the storage class for some data elements individually. For example, you

can route a single signal to multiple Outport blocks and apply a different storage class to each Outport. For example, use this technique to send signal data to a custom function (such as a device driver) and to a global variable that your custom algorithmic code can use:

- 1 Branch the target signal line to each Outport block.
- 2 For more efficient code, leave the storage class of the target signal line configured to Auto (the default). Optimizations can then eliminate the signal line from the generated code.
- 3 In the Code Mappings editor, on the **Outports** tab, apply the storage class `GetSet` to one Outport block and `ExportToFile` to the other Outport block.
- 4 In the Property Inspector, configure the properties associated with each applied storage class.

```
open_system('ex_route_sig')
```



Copyright 2021 The MathWorks, Inc.

Limitations

You cannot apply a storage class to an Outport block if the input to the block has a variable size. Instead, apply the storage class to the signal line.

See Also

Related Examples

- “Configure Root-Level Inport Blocks for C Code Generation” on page 16-18
- “Configure Root-Level Outport Blocks for C Code Generation” on page 16-27
- “Analyze Generated Data Code Interface” (Embedded Coder)
- “Define Interfaces of Model Components”
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49
- “Configure Generated Code According to Interface Control Document Specifications” (Embedded Coder)
- “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder)

See Also

Code Mappings Editor - C | `coder.mapping.api.CodeMapping`

Generate Efficient Code for Bus Signals

In this section...

“Code Efficiency for Bus Signals” on page 15-174

“Set Bus Diagnostics” on page 15-174

“Optimize Virtual and Nonvirtual Buses” on page 15-175

In a model, you use bus signals to package multiple signals together into a single signal line. You can create virtual or nonvirtual bus signals. The representation in the generated code depends on:

- For a virtual bus, the generated code appears as if the bus did not exist.
- Generated code for a nonvirtual bus represents the bus data with a structure. When you want to trace the correspondence between the model and the code, the use of a structure in the generated code can be helpful. To generate structures using nonvirtual bus signals, see “Organize Data into Structures in Generated Code” on page 15-147.

For general information about buses, see and “Composite Interface Guidelines”.

To generate efficient code from models that contain bus signals, eliminate unnecessary data copies by following best practices as you construct the model.

Code Efficiency for Bus Signals

When you use buses in a model for which you intend to generate code:

- Setting bus diagnostic configuration parameters can make model development easier.
- The bus implementation techniques, and the choice of a nonvirtual or virtual bus, can influence the speed, size, and clarity of the generated code.
- Some useful bus implementation techniques are not immediately obvious.

When you work with buses, these guidelines help you to improve the results. The guidelines describe techniques to:

- Simplify the layout of the model.
- Increase the efficiency of generated code.
- Define data structures for function (subsystem) interfaces.
- Define data structures that match existing data structures in external C code.

There are some trade-offs among speed, size, and clarity. For example, the code for nonvirtual buses is easier to read because the buses appear in the code as structures, but the code for virtual buses is faster because virtual buses do not require copying signal data. Apply some of the guidelines based on where you are in the application development process.

Set Bus Diagnostics

Simulink provides diagnostics that you can use to optimize bus usage. Set the following values on the **Configuration Parameters > Diagnostics > Connectivity** pane.

Buses	
Unspecified bus object at root Outport block:	warning
Element name mismatch:	warning
Mux blocks used to create bus signals:	error
Bus signal treated as vector:	warning
Non-bus signals treated as bus signals:	none
Repair bus selections:	Warn and repair

Optimize Virtual and Nonvirtual Buses

Virtual buses are graphical conveniences that do not affect generated code. As a result, the code generation engine is able to fully optimize the signals in the bus. Use virtual buses rather than nonvirtual buses wherever possible. You can convert between virtual and nonvirtual buses by using Signal Conversion blocks. In some cases, Simulink automatically converts a virtual bus to a nonvirtual bus when required. For example, a Stateflow chart converts an input virtual bus to a nonvirtual bus.

To bundle function-call signals, you must use a virtual bus.

You must use nonvirtual buses for:

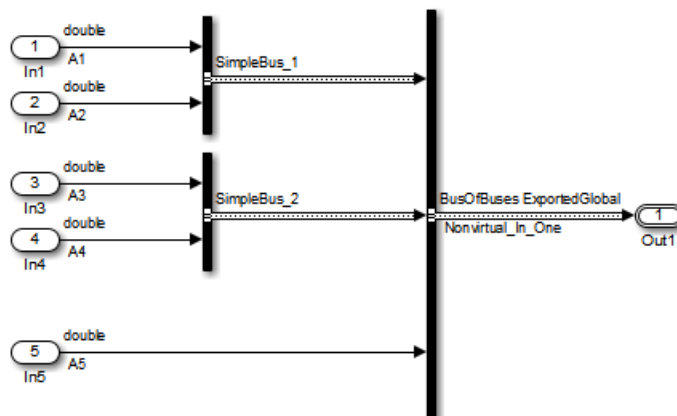
- Nonauto storage classes
- Generating a specific structure from the bus
- Root-level Inport or Outport blocks when the bus has mixed data types

Avoid Nonlocal Nested Buses in Nonvirtual Buses

Buses can contain subordinate buses. To generate efficient code, set the storage classes of subordinate buses to Auto. Setting the storage class to Auto eliminates:

- Allocation of redundant memory for the subordinate bus signal and for the parent bus signal
- Additional copy operations (copying data to the subordinate bus, and then copying from the subordinate bus to the final bus)

This model contains nonvirtual bus signals. The subordinate bus signals Sub_Bus_1 and Sub_Bus_2 use the storage class Auto.



The generated code algorithm efficiently assigns the input signal data to the bus signals.

```
void ex_nonvirtual_buses_step(void)
{
    Nonvirtual_In_One.SimpleBus_1.A1 = A1;
    Nonvirtual_In_One.SimpleBus_1.A2 = A2;
    Nonvirtual_In_One.SimpleBus_2.A3 = A3;
    Nonvirtual_In_One.SimpleBus_2.A4 = A4;
    Nonvirtual_In_One.A5 = A5;
}
```

See Also

Simulink.Bus

Related Examples

- “Organize Data into Structures in Generated Code” on page 15-147
- “Modify Sample Times for Nonvirtual Buses”

Control Signal and State Initialization in the Generated Code

To initialize signals and discrete states with custom values for simulation and code generation, you can use signal objects and block parameters. Data initialization increases application reliability and is a requirement of safety critical applications. Initializing signals for both simulation and code generation can expedite transitions between phases of Model-Based Design.

For basic information about specifying initial values for signals and discrete states in a model, see “Initialize Signals and Discrete States”.

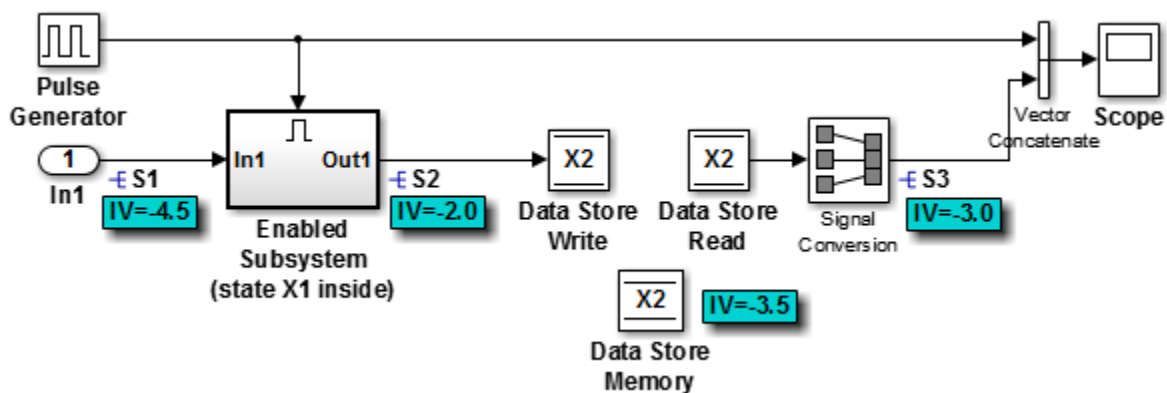
Signal and State Initialization in the Generated Code

The initialization behavior for code generation is the same as that for model simulation with the following exceptions:

- RSim executables can use the **Data Import/Export** pane of the Configuration Parameters dialog box to load input values from MAT-files. GRT and ERT executables cannot load input values from MAT-files.
- The initial value for a block output signal or root level input or output signal can be overwritten by an external (calling) program.
- Setting the initial value for persistent signals is relevant if the value is used or viewed by an external application.

When you generate code, initialization statements are placed in *model.c* or *model.cpp* in the model's initialize code.

For example, consider the model *rtwdemo_sigobj_iv*.



If you create and initialize signal objects in the base workspace, the code generator places initialization code for the signals in the file *rtwdemo_sigobj_iv.c* under the *rtwdemo_sigobj_iv_initialize* function, as shown below.

```
/* Model initialize function */
void rtwdemo_sigobj_iv_initialize(void)
{
    .
    .
    .
}
```

```

/* exported global signals */
S3 = -3.0;

S2 = -2.0;
.
.
.
/* exported global states */
X1 = 0.0;
X2 = 0.0;

/* external inputs */
S1 = -4.5;
.
.
.

```

The following code shows the initialization code for the enabled subsystem's Unit Delay block state X1 and output signal S2.

```

void MdlStart(void) {
    .
    .
    .
    /* InitializeConditions for UnitDelay: '<S2>/Unit Delay' */
    X1 = aa1;

    /* Start for enable system: '<Root>/Enabled Subsystem (state X1 inside)' */

    /* virtual outports code */

    /* (Virtual) Outport Block: '<S2>/Out1' */

    S2 = aa2;
}

```

For an enabled subsystem, the initial value is also used as a reset value if the subsystem's Outport block parameter **Output when disabled** is set to **reset**. The following code from `rtwdemo_sigobj_iv.c` shows the assignment statement for S3 as it appears in the model output function `rtwdemo_sigobj_iv_output`.

```

/* Model output function */

static void rtwdemo_sigobj_iv_output(void)
{
    .
    .
    .
    /* Disable for enable system: '<Root>/Enabled Subsystem (state X1 inside)' */

    /* (Virtual) Outport Block: '<S2>/Out1' */

    S2 = aa2;
}

```

Generate Tunable Initial Conditions

You can represent initial conditions for signals and states by creating tunable global variables in the generated code. These variables allow you to restart an application by using initial conditions that are stored in memory.

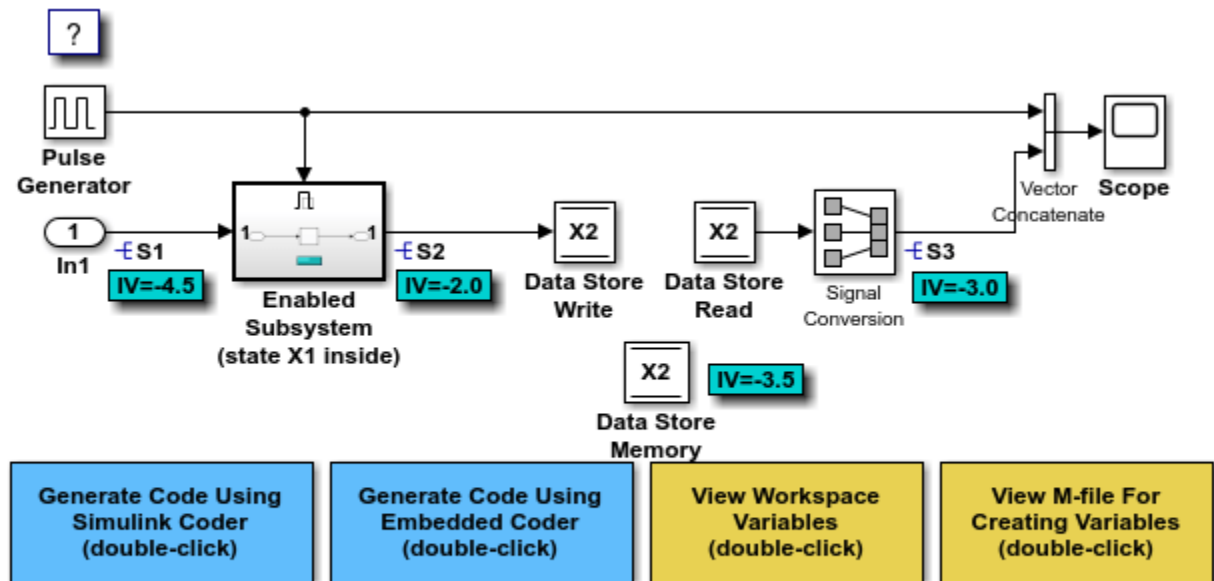
If you set **Configuration Parameters > Optimization > Default parameter behavior** to **Tunable**, initial conditions appear as tunable fields of the global parameters structure.

Whether you set **Default parameter behavior** to **Tunable** or **Inlined**, you can use a tunable parameter to specify the `InitialValue` property of a signal object or the **Initial condition** parameter of a block. For basic information about tunable parameters, see “Create Tunable Calibration Parameter in the Generated Code” on page 15-112.

This example shows how to use tunable parameters to specify initial conditions for signals and states.

Explore Example Model

Open the example model `rtwdemo_sigobj_iv` and configure it to show the generated names of blocks. The signal `S2` uses a `Simulink.Signal` object in the base workspace.



Copyright 1994-2020 The MathWorks, Inc.

Double-click the `Simulink.Signal` object `S2` to view its properties. The **Initial value** property is set to `aa2`. The object uses the variable `aa2` to specify an initial condition for the signal `S2`. The **Storage class** property is set to `ExportedGlobal`. To use a `Simulink.Signal` object to initialize a signal, the signal object must use a storage class other than `Auto` or, if the corresponding data category in the Code Mapping Editor uses a storage class setting other than `Default`, `Model default`.

On the **Optimization** pane, in the Configuration Parameters dialog box, click **Configure**. The variable `aa2` is a tunable parameter that uses the storage class `ExportedGlobal`.

In the model, open the Enabled Subsystem. In the Output block dialog box, the parameter **Output when disabled** is set to `reset`. When the subsystem becomes disabled, the output signal `S2` resets to the initial value `aa2`.

Open the Unit Delay block dialog box. On the **State Attributes** tab, the **State name** box is set to `X1`.

Open the Enable block dialog box. The parameter **States when enabling** is set to `reset`. When the subsystem transitions from a disabled state to an enabled state, it resets internal block states, such as `X1`, to their initial values.

In the base workspace, double-click the `Simulink.Signal` object `X1` to view its properties. The **Initial value** property is set to `aa1`.

Double-click the Simulink.Parameter object `aa1` to view its properties. The **Storage class** property is set to `ExportedGlobal`. You can generate tunable initial conditions for block states by using tunable parameters such as `aa1` and Simulink.Signal objects such as `X1`.

Generate and Inspect Code

Generate code with the example model.

```
### Starting build procedure for: rtwdemo_sigobj_iv
### Successful completion of build procedure for: rtwdemo_sigobj_iv
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_sigobj_iv	Code generated and compiled.	Code generation information file does not exist

1 of 1 models built (0 models already up to date)

Build duration: 0h 0m 11.57s

In the code generation report, view the file `rtwdemo_sigobj_iv.c`. The code uses global variables to represent the block state `X1` and the signal `S2`.

```
/* Exported block states */
real_T X1;                                /* '<S1>/Unit Delay' */

/* Exported block signals */
real_T S1;                                /* '<Root>/In1' */
real_T S3;                                /* '<Root>/Signal Conversion' */
real_T S2;                                /* '<S1>/Unit Delay' */
```

The code uses global variable to represent the tunable parameter `aa1`.

```
/* Exported block parameters */
real_T aa1 = -2.5;                         /* Variable: aa1
```

The model initialization function uses the tunable parameter `aa1` to initialize the state `X1`. The function also uses the tunable parameter `aa2` to initialize the signal `S2`.

```
/* SystemInitialize for Enabled SubSystem: '<Root>/Enabled Subsystem (state X1 inside)' */
/* InitializeConditions for UnitDelay: '<S1>/Unit Delay' */
X1 = aa1;

/* SystemInitialize for UnitDelay: '<S1>/Unit Delay' incorporates:
 * Output: '<S1>/Out1'
 */
S2 = 0.0;
```

In the model step function, when the Enabled Subsystem transitions from a disabled state to an enabled state, the Unit Delay block state `X1` resets to its initial value.

```
if (rtb_PulseGenerator > 0) {
```

```

if (!rtwdemo_sigobj_iv_DW.EnabledSubsystemstateX1inside_M) {
    /* InitializeConditions for UnitDelay: '<S1>/Unit Delay' */
    X1 = aal;
    rtwdemo_sigobj_iv_DW.EnabledSubsystemstateX1inside_M = true;
}

/* UnitDelay: '<S1>/Unit Delay' */
S2 = X1;

/* Update for UnitDelay: '<S1>/Unit Delay' incorporates:
 * Inport: '<Root>/In1'
 */
X1 = S1;
} else if (rtwdemo_sigobj_iv_DW.EnabledSubsystemstateX1inside_M) {
    /* Disable for UnitDelay: '<S1>/Unit Delay' incorporates:
     * Outport: '<S1>/Out1'
     */
    S2 = 0.0;
    rtwdemo_sigobj_iv_DW.EnabledSubsystemstateX1inside_M = false;
}

/* End of Outputs for SubSystem: '<Root>/Enabled Subsystem (state X1 inside)' */

/* DataStoreWrite: '<Root>/Data Store Write' */
X2 = S2;

/* SignalConversion: '<Root>/Signal Conversion' incorporates:
 * DataStoreRead: '<Root>/Data Store Read'
 */
S3 = X2;
}

/* Model initialize function */
void rtwdemo_sigobj_iv_initialize(void)
{
    /* Registration code */

    /* initialize error status */
    rtmSetErrorStatus(rtwdemo_sigobj_iv_M, (NULL));

    /* block I/O */

    /* exported global signals */
    S3 = -3.0;

    /* states (dwork) */
    (void) memset((void *)&rtwdemo_sigobj_iv_DW, 0,
        sizeof(DW_rtwdemo_sigobj_iv_T));

    /* exported global states */
    X1 = 0.0;
    X2 = 0.0;

    /* external inputs */
    S1 = -4.5;

    /* Start for DiscretePulseGenerator: '<Root>/Pulse Generator' */
    rtwdemo_sigobj_iv_DW.clockTickCounter = -2;
}

```

```

/* Start for DataStoreMemory: '<Root>/Data Store Memory' */
X2 = -3.5;

/* SystemInitialize for Enabled SubSystem: '<Root>/Enabled Subsystem (state X1 inside)' */
/* InitializeConditions for UnitDelay: '<S1>/Unit Delay' */
X1 = aa1;

/* SystemInitialize for UnitDelay: '<S1>/Unit Delay' incorporates:
 *   Output: '<S1>/Out1'
 */
S2 = 0.0;

/* End of SystemInitialize for SubSystem: '<Root>/Enabled Subsystem (state X1 inside)' */
}

/* Model terminate function */
void rtwdemo_sigobj_iv_terminate(void)
{
/* (no terminate code required) */
}

```

If the Enabled Subsystem becomes disabled during code execution, the algorithm uses the tunable initial condition aa2 to set the value of the signal S2.

```

} else {

} else if (rtwdemo_sigobj_iv_DW.EnabledSubsystemstateX1inside_M) {
/* Disable for UnitDelay: '<S1>/Unit Delay' incorporates:
 *   Output: '<S1>/Out1'
 */
S2 = 0.0;
rtwdemo_sigobj_iv_DW.EnabledSubsystemstateX1inside_M = false;
}

```

Generate Tunable Initial Condition Structure for Bus Signal

When you use a MATLAB® structure to specify initialization values for the signal elements in a bus, you can create a tunable global structure in the generated code.

If you set **Configuration Parameters > Optimization > Default parameter behavior** to **Tunable**, the initial condition appears as a tunable substructure of the global parameters structure.

Whether you set **Default parameter behavior** to **Tunable** or **Inlined**, you can specify the initial condition by using a tunable `Simulink.Parameter` object whose value is a structure. If you apply a storage class other than `Auto` to the parameter object, the structure is tunable in the generated code.

To generate efficient code by avoiding data type mismatches between the structure and the bus signal, use either:

- Typed expressions to specify the values of the structure fields. Match the data type of each field with the data type of the corresponding signal element.

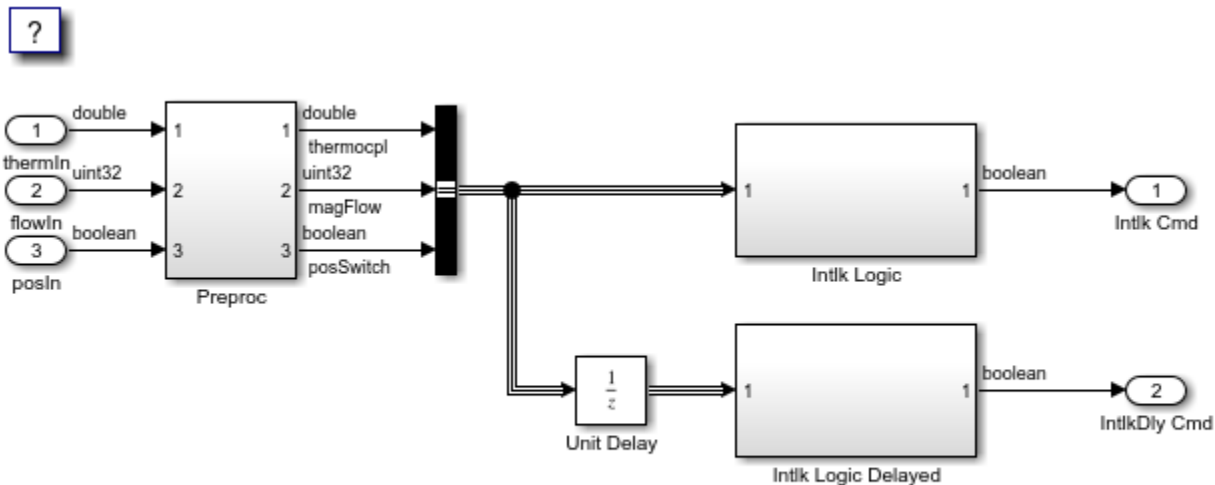
- A `Simulink.Bus` object to control the data types of the structure fields and the signal elements.

For basic information about using structures to initialize bus signals, and to decide how to control field data types, see “Specify Initial Conditions for Bus Elements”.

Generate Tunable Initial Condition Structure

This example shows how to use a tunable structure parameter to initialize a virtual bus signal.

Open the example model `rtwdemo_tunable_init_struct` and configure it to show the generated names of blocks.



Copyright 2015-2020 The MathWorks, Inc.

On the **Modeling** tab, click **Model Data Editor**.

On the **Inports/Outputs** tab, the **Data Types** column shows that each Inport block in the model uses a different output data type.

Open the Bus Creator block dialog box. The block output is a virtual bus.

In the Configuration Parameters dialog box, open the **Optimization** pane. The configuration parameter **Default parameter behavior** is set to **Tunable**. By default, block parameters, including initial conditions, appear in the generated code as tunable fields of the global parameters structure.

In the Model Data Editor, inspect the **States** tab.

For the Unit Delay block, set **Initial Value** to a structure that specifies an initial condition for each of the three signal elements. To generate efficient code, match the data types of the structure fields with the data types of the corresponding signal elements. For example, set **Initial Value** to the expression `struct('thermocpl',15.23,'magFlow',uint32(79),'posSwitch',false)`.

```
set_param('rtwdemo_tunable_init_struct/Unit Delay','InitialCondition',...
'struct('thermocpl',15.23,'magFlow',uint32(79),'posSwitch',false)')
```

Generate code from the example model.

```
### Starting build procedure for: rtwdemo_tunable_init_struct
### Successful completion of build procedure for: rtwdemo_tunable_init_struct
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_tunable_init_struct	Code generated and compiled.	Code generation information file does

1 of 1 models built (0 models already up to date)

Build duration: 0h 0m 15.124s

In the code generation report, view the file `rtwdemo_tunable_init_struct_types.h`. The code defines a structure type whose fields use the data types that you specified in the struct expression.

```
#ifndef DEFINED_TYPEDEF_FOR_struct_mqGiljsItE0G7cf1bNqMu_
#define DEFINED_TYPEDEF_FOR_struct_mqGiljsItE0G7cf1bNqMu_

typedef struct {
    real_T thermocpl;
    uint32_T magFlow;
    boolean_T posSwitch;
} struct_mqGiljsItE0G7cf1bNqMu;
```

View the file `rtwdemo_tunable_init_struct.h`. The struct type definition of the global parameters structure contains a substructure, `UnitDelay_InitialCondition`, which represents the **Initial condition** parameter of the Unit Delay block.

```
struct P_rtwdemo_tunable_init_struct_T {
    struct_mqGiljsItE0G7cf1bNqMu UnitDelay_InitialCondition;
```

View the file `rtwdemo_tunable_init_struct_data.c`. This source file allocates memory for the global parameters structure. The substructure `UnitDelay_InitialCondition` appears.

```
/* Block parameters (default storage) */
P_rtwdemo_tunable_init_struct_T rtwdemo_tunable_init_struct_P = {
    /* Mask Parameter: UnitDelay_InitialCondition
    * Referenced by:
    *   '<Root>/Unit Delay'
    *   '<Root>/Unit Delay'
    *   '<Root>/Unit Delay'
    */
    {
        15.23,
        79U,
        0
    },
};
```

View the file `rtwdemo_tunable_init_struct.c`. The model initialization function uses the fields of the substructure to initialize the block states.

```
/* InitializeConditions for UnitDelay generated from: '<Root>/Unit Delay' */
```

```

rtwdemo_tunable_init_struct_DW.UnitDelay_1_DSTATE =
    rtwdemo_tunable_init_struct_P.UnitDelay_InitialCondition.thermocpl;

/* InitializeConditions for UnitDelay generated from: '<Root>/Unit Delay' */
rtwdemo_tunable_init_struct_DW.UnitDelay_2_DSTATE =
    rtwdemo_tunable_init_struct_P.UnitDelay_InitialCondition.magFlow;

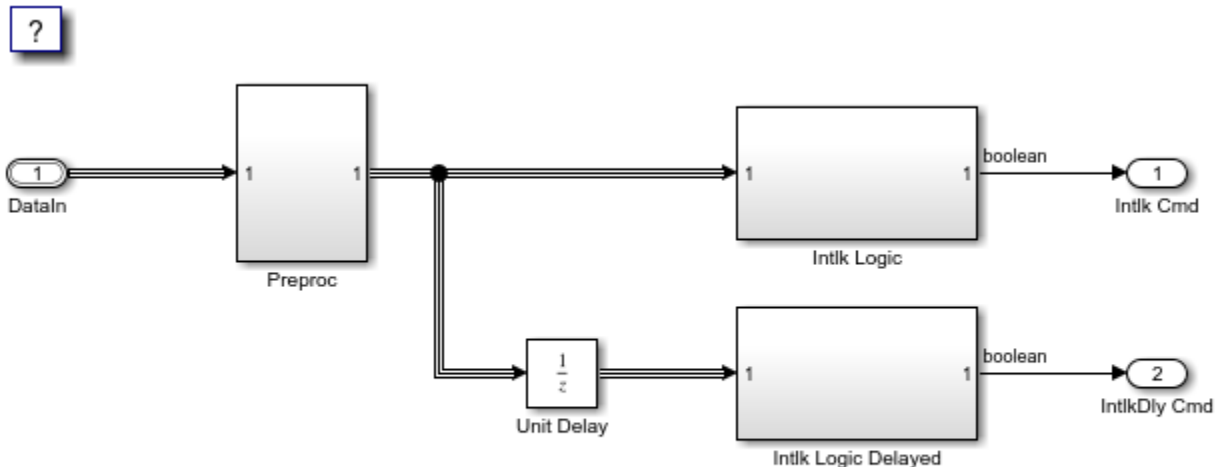
/* InitializeConditions for UnitDelay generated from: '<Root>/Unit Delay' */
rtwdemo_tunable_init_struct_DW.UnitDelay_3_DSTATE =
    rtwdemo_tunable_init_struct_P.UnitDelay_InitialCondition.posSwitch;

```

Use Bus Object to Specify Data Types

If you create a bus object, you can use it to specify the data type of the bus signal and the tunable initial condition structure. Before code generation, the `Simulink.Parameter` object casts the values of the structure fields to the data types of the signal elements. For basic information about creating bus objects and using them in models, see “Specify Bus Properties with Simulink.Bus Object Data Types”.

Open the example model `rtwdemo_init_struct_busobj` and configure it to show the generated names of blocks.



Copyright 2015-2020 The MathWorks, Inc.

In the base workspace, double-click the `Simulink.Bus` object `ComponentData`. The object defines three signal elements: `thermocpl`, `magFlow`, and `posSwitch`. The elements each use a different data type.

In the model, open the Model Data Editor (on the **Modeling** tab, click **Model Data Editor**). The **Imports/Outputs** tab shows that for the Inport block `DataIn`, the output data type (**Data Type** column) is set to `Bus: ComponentData`.

At the command prompt, create the structure parameter `initStruct`. You can specify the field values by using untyped expressions. To improve readability, specify the field `posSwitch` with a Boolean value.

```
initStruct = struct(...
    'thermocpl',15.23,...
    'magFlow',79,...
    'posSwitch',false ...
);
```

```
initStruct = Simulink.Parameter(initStruct);
```

In the Model Data Editor, inspect the **Parameters** tab.

In the model, click the Unit Delay block. The Model Data Editor highlights the row that corresponds to the **Initial condition** parameter of the block.

In the Model Data Editor, set the parameter value (**Value** column) to `initStruct`.

Click the **Show/refresh additional information** button. The parameter object, `initStruct`, appears in the data table as a row.

Use the **Data Type** column to set the data type of `initStruct` to `Bus: ComponentData`.

```
initStruct.DataType = 'Bus: ComponentData';
```

Set the **Change View** drop-down list to `Code`.

Use the **Storage Class** column to apply the storage class `ExportedGlobal` to `initStruct`.

```
initStruct.StorageClass = 'ExportedGlobal';
```

Generate code from the example model.

```
### Starting build procedure for: rtwdemo_init_struct_busobj
### Successful completion of build procedure for: rtwdemo_init_struct_busobj
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_init_struct_busobj	Code generated and compiled.	Code generation information file does not exist

1 of 1 models built (0 models already up to date)

Build duration: 0h 0m 10.224s

In the code generation report, view the file `rtwdemo_init_struct_busobj_types.h`. The code creates a structure type `ComponentData` whose fields use the data types in the bus object.

```
#ifndef DEFINED_TYPEDEF_FOR_ComponentData_
#define DEFINED_TYPEDEF_FOR_ComponentData_

typedef struct {
    real_T thermocpl;
    uint32_T magFlow;
    boolean_T posSwitch;
} ComponentData;
```

View the file `rtwdemo_init_struct_busobj.c`. The code creates a global variable to represent the tunable parameter object `initStruct`.


```

/* Exported block parameters */
ComponentData initStruct = {
    15.23,
    79U,
    0
};
/* Variable: initStruct

```

The model initialization function uses the structure fields to initialize the block states.

```

/* InitializeConditions for UnitDelay generated from: '<Root>/Unit Delay' */
rtwdemo_init_struct_busobj_DW.UnitDelay_1_DSTATE = initStruct.thermocpl;

/* InitializeConditions for UnitDelay generated from: '<Root>/Unit Delay' */
rtwdemo_init_struct_busobj_DW.UnitDelay_2_DSTATE = initStruct.magFlow;

/* InitializeConditions for UnitDelay generated from: '<Root>/Unit Delay' */
rtwdemo_init_struct_busobj_DW.UnitDelay_3_DSTATE = initStruct.posSwitch;

```

To change the data type of a signal element, specify the new type in the bus object. The signal element in the model uses the new type. Before simulation and code generation, the parameter object `initStruct` casts the corresponding structure field to the new type.

See Also

State Reader | State Writer

Related Examples

- “Initialization Behavior Summary for Signal Objects”
- “Specify Initial Conditions for Bus Elements”
- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Startup, Reset, and Shutdown Function Interfaces” on page 2-8
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49
- “Initialization of Signal, State, and Parameter Data in the Generated Code” on page 15-188
- “Remove Zero-Initialization Code” (Embedded Coder)

Initialization of Signal, State, and Parameter Data in the Generated Code

Signal lines, block parameters, and block states in a model can appear in the generated code as data, for example, as global variables. By default, the code initializes this data before execution of the primary algorithm. To match the numerics of simulation in Simulink, the code generator chooses the initial values based on specifications that you make in the model.

By understanding how the generated code initializes data, you can:

- Model a system that reinitializes states during execution, which means that the application can restart the entire system.
- Store initial values in memory as variables, which can persist between execution runs. You can then overwrite these values before starting or restarting a system.
- Generate efficient code by eliminating storage for invariant initial values and by preventing the generation of code that unnecessarily or redundantly initializes data.

For basic information about initializing signals and states in a model, see “Initialize Signals and Discrete States” and “Load State Information”.

Static Initialization and Dynamic Initialization

To initialize a data item such as a global variable, an application can use static or dynamic initialization.

- Static initialization occurs in the same statement that defines (allocates memory for) the variable. The initialization does not occur inside a function definition.

The code can be more efficient because none of the generated model functions execute initialization statements.

- Dynamic initialization occurs inside a function. For each model or nonvirtual subsystem, the code generator typically creates one or more functions that are dedicated to initialization.

The generated code or your code can restart a system by calling an initialization function during execution.

Real-World Ground Initialization Requiring Nonzero Bit Pattern

Each data item has a real-world ground value. This value can depend on the data type of the item. For example, for a signal whose data type is `double` or `int8`, the real-world ground value is zero. For an enumeration, the ground value is the default enumeration member.

For some kinds of data, to represent a real-world ground value, a computer stores zero in memory (all bits zero). However, for some other data, a computer stores a nonzero value in memory. This data includes, for example:

- Fixed-point data with bias. The code initializes such a data item to the stored integer value that, given the scaling and bias, represents real-world zero.
- An enumeration whose default member maps to an integer value other than zero. For example, if the default member is `High` with an underlying integer value of 3, the code initializes such a data item to `High`.

Initialization of Signal and State Data

In a model, you can control signal and state initial values through block parameters. For example, to set the initial value of a Unit Delay block, you use the **Initial condition** parameter. In some cases, you can also use the `InitialValue` property of a `Simulink.Signal` object. For most of these block parameters, the default value is 0.

You can also initialize states by using State Writer blocks in Initialize Function subsystems and the **Initial states** model configuration parameter.

By default, the generated code dynamically initializes signal and state data (and other data, such as the model error status) in the generated initialization function. The function, named `model_initialize` by default, performs signal and state initialization operations in this order:

- 1 Initializes the signal and state data in the default generated structures, such as the `DWork` structure, to a stored value of zero.
- 2 Initializes additional signal and state data that are not in the default generated structures to the relevant real-world ground value.

This initialization applies only to data that meet both of these criteria:

- The generated code defines (allocates memory for) the data.
- The data use a storage class other than `Auto` (the default storage class) or `Model default` (when the Code Mapping Editor specifies the `Default` storage class, the default setting).

For example, the code applies this operation to data items that use the storage class `ExportedGlobal`.

- 3 Initializes each signal and state to the real-world value that the model specifies, for example, through the **Initial condition** parameter of a Unit Delay block.
- 4 Initializes each state to the real-world value that you assign by using a State Writer block. The function performs this initialization only if you use an Initialize Function subsystem in the model.
- 5 Initializes each state to the real-world value that you specify with the configuration parameter **Initial states**.

After this initialization function executes, each data item has the last real-world value that the function assigned. For example, if you use a State Writer block to initialize a block state to 5 while also using the **Initial states** configuration parameter to initialize the same state to 10, the state ultimately uses 10 as an initial value.

memset for Bulk Initialization

To initialize signal or state data items that have contiguous storage to a stored value of zero, the generated code can call `memset` in an initialization function. Data items that have contiguous storage include the `DWork` structure, arrays, or data items that use a multiword data type.

If your application requires it, you can prevent the generated code from using `memset` for initializing floating-point data to stored zero. See [Use memset to initialize floats and doubles to 0.0](#).

Tunable Initial Values

You can configure the way that tunable block parameters, such as the **Gain** parameter of a Gain block, appear in the generated code. Most block parameters that set initial values (for example,

Initial condition) are tunable. For example, the configuration parameter **Default parameter behavior** can determine whether the initial values appear in the generated code as inlined constants or as tunable global data. You can also use parameter objects and storage classes to control the representation of these initial values.

For tunable initial values, in the model initialization function, the right-hand side of the assignment statement is a global variable, structure field, or other data whose value you can change in memory.

To make initial values tunable in the generated code, see “Control Signal and State Initialization in the Generated Code” on page 15-177.

Initialization of Parameter Data

The generated code statically initializes parameter data to the values that you specify in Simulink.

To express the initial value of a tunable parameter (a global variable) as a mathematical expression involving system constants or other macros, which requires Embedded Coder, see “Initialization by Mathematical Expression (Embedded Coder)” on page 15-190.

Kinds of Parameter Data

These model elements appear in the generated code as parameter data:

- Tunable block parameters, such as the **Gain** parameter of a Gain block, when you set **Default parameter behavior** to Tunable. Each parameter appears as a field of a dedicated global structure.
- Some tunable block parameters when you set **Default parameter behavior** to Inlined. When the code generator cannot inline the value of a parameter as a literal number, the parameter appears as a field of a dedicated global const structure.
- `Simulink.Parameter` objects to which you apply a storage class that has an exported data scope. For example, the built-in storage class `ExportedGlobal` has an exported data scope, but the storage class `ImportedExtern` does not.

Initialization by Mathematical Expression (Embedded Coder)

You can set the value of a parameter object (such as `Simulink.Parameter`) to a mathematical expression involving numbers, MATLAB variables, and other parameter objects. See “Set Variable Value by Using a Mathematical Expression”.

If you use this technique with Embedded Coder, you can generate code that initializes the corresponding parameter data (global variable) by using the specified expression. For an example, see “Initialize Parameter Value From System Constant or Other Macro (Embedded Coder)” (Embedded Coder). For general information, including limitations, see “Code Generation of Parameter Objects With Expression Values” on page 15-125.

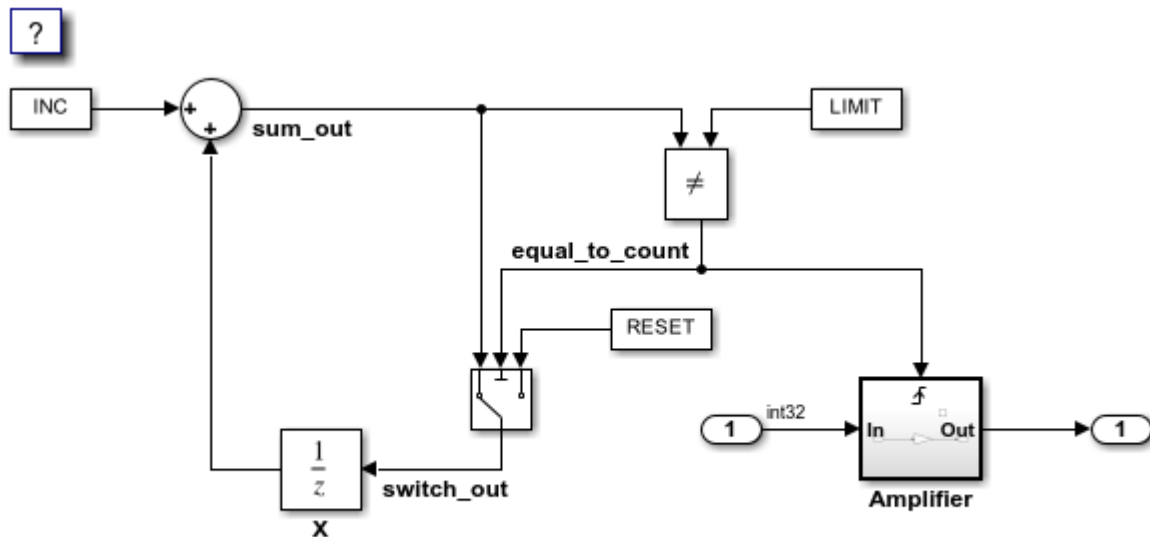
Data Initialization in the Generated Code

This example shows how the generated code initializes signal, state, and parameter data.

Explore Example Model

Open the example model, `rtwdemo_rtintro`.

```
open_system('rtwdemo_rtwinintro')
```



Copyright 1994-2020 The MathWorks, Inc.

In the Apps gallery, click **Simulink Coder**.

Open the Model Data Editor. On the **Modeling** tab, click **Model Data Editor**.

Inspect the **States** tab. For the Unit Delay block state, the **Initial Value** is set to 0, the default, which means the initial value of the state is zero. The state is named X.

Set **Initial Value** to a nonzero number, for example, 5.

```
set_param('rtwdemo_rtwinintro/X', 'InitialCondition', '5')
```

Open the Code Mappings editor. In the **C Code** tab, select **Code Interface > Individual Element Code Mappings**.

On the **Signals/States** tab, under **States** set **Storage Class** for X to `ExportedGlobal`. With this setting, the block state appears in the generated code as a separate global variable.

```
coder.mapping.utils.create('rtwdemo_rtwinintro');
cm = coder.mapping.api.get('rtwdemo_rtwinintro');
setState(cm, 'rtwdemo_rtwinintro/X', 'StorageClass', 'ExportedGlobal');
```

In the model, open the Amplifier subsystem, which is a triggered subsystem.

In the Model Data Editor, inspect the **Parameters** tab. To set and inspect initial values of signals and states, you can use the **Parameters** tab instead of the **States** tab.

The Model Data Editor shows that for the Output block, the **Initial output** (`InitialOutput`) parameter is set to 0, the default. This subsystem output requires an initial value because the subsystem executes conditionally. Leave the initial value at the default.

In the model, clear **Configuration Parameters > Signal storage reuse**. When you clear this setting, signal lines appear in the generated code as fields of a generated structure whose purpose is

to store signal data. This representation of the signals makes it easier to see how the code generator initializes data.

```
set_param('rtwdemo_rtwintr', 'OptimizeBlockIOStorage', 'off')
```

In the model, inspect the configuration parameter **Configuration Parameters > Optimization > Default parameter behavior**. The configuration parameter is set to `Inlined`, which means block parameters, including initial values, appear in the generated code as inlined literals or as `const` data.

Navigate to the root level of the model.

In the Model Data Editor, click the **Show/refresh additional information** button.

Near the **Filter contents** box, click the **Filter using selection** button.

In the model, click the Constant block labeled `INC`. The Model Data Editor shows that the **Constant value** (Value) parameter of the block is set to `INC`. `INC` is a MATLAB variable in the base workspace.

For `INC`, set the value in the **Value** column to `Simulink.Parameter(uint8(1))`. MATLAB converts `INC` to a `Simulink.Parameter` object.

In the Code Mappings editor, on the **Parameters** tab, apply the storage class `ExportedGlobal` to `INC`. With this setting, the generated code defines `INC` as a global variable. Concerning initialization, `INC` is an item of parameter data.

```
INC = Simulink.Parameter(INC);
INC.StorageClass = 'ExportedGlobal';
```

Generate and Inspect Code

To build the model and generate code, press **Ctrl+B**.

```
slbuild('rtwdemo_rtwintr');

### Starting build procedure for: rtwdemo_rtwintr
### Successful completion of build procedure for: rtwdemo_rtwintr
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_rtwintr	Code generated and compiled.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.449s

In the generated file `rtwdemo_rtwintr.c`, outside the definition of a function, the code statically initializes the parameter data `INC`. The value of `INC` is 1.

```
file = fullfile('rtwdemo_rtwintr_grt_rtw', 'rtwdemo_rtwintr.c');
rtwdemodbtype(file, '/* Exported block parameters */', 'uint8_T INC = 1U;', 1, 1)
```

```
/* Exported block parameters */
uint8_T INC = 1U;                                /* Variable: INC
```

In the same file, inspect the definition of the `rtwdemo_rtwinintro_initialize` function. First, the function uses `memset` to initialize the internal signals in the model to a stored value of 0.

```
rtwdemodbtype(file, '/* block I/O */', 'sizeof(B_rtwdemo_rtwinintro_T);', 1, 1)
```

```
/* block I/O */
(void) memset(((void *) &rtwdemo_rtwinintro_B), 0,
             sizeof(B_rtwdemo_rtwinintro_T));
```

The function then initializes the Unit Delay state, `X`, to a ground value. In this case, the ground value is zero.

```
rtwdemodbtype(file, '/* exported global states */', 'X = 0U;', 1, 1)
```

```
/* exported global states */
X = 0U;
```

The function also initializes other data, including the root-level inputs and outputs (Inport and Output blocks), to ground values.

```
rtwdemodbtype(file, '/* external inputs */', 'Amplifier_Trig_ZCE = POS_ZCSIG;', 1, 1)
```

```
/* external inputs */
rtwdemo_rtwinintro_U.Input = 0;

/* external outputs */
rtwdemo_rtwinintro_Y.Output = 0;
rtwdemo_rtwinintro_PrevZCX.Amplifier_Trig_ZCE = POS_ZCSIG;
```

The function then initializes `X` to the value that you specified in the **Initial condition** block parameter.

```
rtwdemodbtype(file, '/* InitializeConditions for UnitDelay: '<Root>/X' */', ...
             'X = 5U;', 1, 1)
```

```
/* InitializeConditions for UnitDelay: '<Root>/X' */
X = 5U;
```

Finally, the function initializes the output of the Amplifier subsystem.

```
rtwdemodbtype(file, 'SystemInitialize for Triggered SubSystem', ...
             'End of SystemInitialize for SubSystem', 1, 1)
```

```
/* SystemInitialize for Triggered SubSystem: '<Root>/Amplifier' */
/* SystemInitialize for Output: '<Root>/Output' incorporates:
 * Output: '<S1>/Out'
 */
rtwdemo_rtwinintro_Y.Output = 0;
```

Inspect Difference Between Stored Value and Real-World Value

For some data, even if the real-world initial value is zero, a computer stores a nonzero value in memory. To observe this difference, apply a slope-bias fixed-point data type to the block state, X. To run this example, you must have Fixed-Point Designer™.

In the Model Data Editor, inspect the **Signals** tab.

In the model, click the output signal of the Switch block, `switch_out`.

Use the **Data Type** column of the Model Data Editor to set the signal data type to `fixdt(1,16,1,3)`. This expression represents a fixed-point data type with a slope of 1 and a bias of 3.

```
set_param('rtwdemo_rtwinintro/Switch','OutDataTypeStr',...
    'fixdt(1,16,1,3)')
```

In the model, select the Sum block. In the Property Inspector, under **Signal Attributes**, clear **Require all inputs to have the same data type**.

```
set_param('rtwdemo_rtwinintro/Sum','InputSameDT','off')
```

To prevent compilation errors on different platforms, select the model configuration parameter **Generate code only**. This setting causes the model to generate only code.

```
set_param('rtwdemo_rtwinintro','GenCodeOnly','on')
```

To build the model and generate code, press **Ctrl+B**.

```
slbuild('rtwdemo_rtwinintro')

### Starting build procedure for: rtwdemo_rtwinintro
### Successful completion of code generation for: rtwdemo_rtwinintro
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_rtwinintro	Code generated.	Generated code was out of date.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 10.933s
```

The model initialization function first initializes X to a real-world ground value, 0. Due to the slope-bias data type, which has bias 3, this real-world value corresponds to a stored value of -3.

```
rtwdemodbtype(file,'/* exported global states */','X = -3;',1,1)
```

```
/* exported global states */
X = -3;
```

The function then initializes X to the real-world initial value that you specified in the **Initial condition** block parameter, 5. This real-world value corresponds to a stored value of 2.

```
rtwdemodbtype(file,'/* InitializeConditions for UnitDelay: '<Root>/X' */',...
    'X = 2;',1,1)
```



```
/* InitializeConditions for UnitDelay: '<Root>/X' */
X = 2;
```

Modeling Goals

Goal	More Information
Explicitly model initialization behavior by using blocks	<p>You can explicitly model initialization and reset behavior by using Initialize Function and Reset Function subsystems. In the subsystems, use State Writer blocks to calculate and assign an initial value for a state dynamically. The corresponding code appears in the model initialization function.</p> <p>For more information, see “Startup, Reset, and Shutdown Function Interfaces” on page 2-8.</p>
Prevent generation of code that explicitly initializes data to zero	<p>If your application environment already initializes global variables to zero, for more efficient code, you can prevent the generation of statements that explicitly initialize global variables to zero. This optimization applies only to signals and states whose initial values are stored in memory as zero. For example, the code generator does not apply the optimization to:</p> <ul style="list-style-type: none"> • Data for which you specify a nonzero initial value by using a block parameter. • Data whose real-world initial value is zero but whose corresponding stored value is not zero. • Enumerated data whose default member maps to a nonzero integer. <p>The optimization requires Embedded Coder. For more information, see “Remove Zero-Initialization Code” (Embedded Coder).</p>
Generate code that imports data from your external code	<p>You can generate code that reuses (imports) data that your external code defines. For example, you can apply the storage class <code>ImportedExtern</code> to a signal line, block state, or parameter object. For imported data:</p> <ul style="list-style-type: none"> • The generated code does not initialize parameter data. Your code must initialize imported parameter data. • The generated initialization functions dynamically initialize some signal and state data. Unlike data that the generated code allocates, the code does not initialize imported signal or state data to a stored value of zero. Instead, the code immediately initializes the data to the real-world value that you specify in Simulink. <p>For more information, including how to prevent the generated code from initializing imported data, see “Exchange Data Between External Calling Code and Generated Code” on page 30-67.</p>

See Also

`model_initialize`

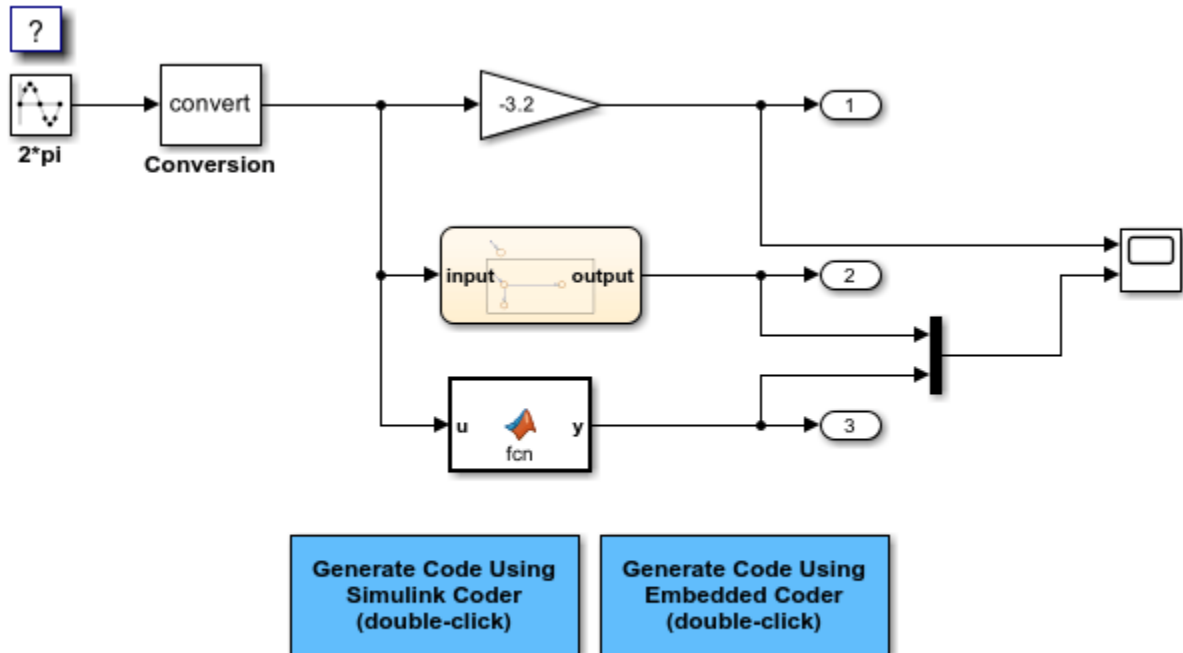
Related Examples

- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Optimize Generated Code Using Fixed-Point Data with Simulink, Stateflow, and MATLAB

This model shows fixed-point code generation in Simulink®, Stateflow®, and MATLAB®.

```
open_system('rtwdemo_fixpt1');
```



Copyright 1994-2020 The MathWorks, Inc.

See Also

More About

- "Fixed Point"

Declare Workspace Variables as Tunable Parameters Using the Model Parameter Configuration Dialog Box

You can use the Model Parameter Configuration dialog box to declare numeric MATLAB variables in the base workspace as tunable parameters. You can select code generation options, such as storage class, for each tunable parameter.

However, it is a best practice to instead use parameter objects to declare tunable parameters. Do not use the Model Parameter Configuration dialog box to select parameter objects in the base workspace. To use parameter objects, instead of the Model Parameter Configuration dialog box, to declare tunable parameters, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.

Note You cannot use the Model Parameter Configuration dialog box to declare tunable parameters for a referenced model. Use `Simulink.Parameter` objects instead.

Declare Existing Workspace Variables as Tunable Parameters

Use the Model Parameter Configuration dialog box to declare existing workspace variables as tunable parameters for a model.

- 1 In the Configuration Parameters dialog box, on the **Code Generation > Optimization** pane, set the **Default parameter behavior** parameter to **Tunable** and click **Configure**.
- 2 In the Model Parameter Configuration dialog box, under **Source list**, select a method to populate the list of available workspace variables.
 - Select **MATLAB workspace** to view numeric variables that are defined in the base workspace.
 - Select **Referenced workspace variables** to view only the numeric variables in the base workspace that the model uses. Selecting this option begins a diagram update and a search for used variables, which can take time for a large model.
- 3 In the Model Parameter Configuration dialog box, under **Source list**, select one or more workspace variables.
- 4 Click **Add to table**. The variables appear as tunable parameters under **Global (tunable) parameters**, and appear in italic font under **Source list**.
- 5 Optionally, select a parameter under **Global (tunable) parameters**, and adjust the code generation settings for the parameter. For more information about adjusting the code generation options for tunable parameters, see “Set Tunable Parameter Code Generation Options” on page 15-199
- 6 Click **OK** to apply your selection of tunable parameters and close the dialog box.

Declare New Tunable Parameters

Use the Model Parameter Configuration dialog box to declare new tunable parameters. You can use this technique to declare the names of tunable parameters, and to adjust their code generation settings, before you create the corresponding workspace variables.

- 1 In the Configuration Parameters dialog box, on the **Optimization** pane, set the **Default parameter behavior** parameter to **Tunable** and click **Configure**.

- 2 In the Model Parameter Configuration dialog box, under **Global (tunable) parameters**, click **New**.
- 3 Under the **Name** column, specify a name for the new tunable parameter.
- 4 Optionally, adjust the code generation settings for the new parameter. For more information about adjusting the code generation options for tunable parameters, see “Set Tunable Parameter Code Generation Options” on page 15-199
- 5 Click **OK** to apply your changes and close the dialog box.

Set Tunable Parameter Code Generation Options

To set the properties of tunable parameters listed under **Global (tunable) parameters** in the Model Parameter Configuration dialog box, select a parameter and specify a storage class and, optionally, a storage type qualifier.

Property	Description
Storage class	<p>Select one of the following to use for code generation:</p> <ul style="list-style-type: none"> • Model default • ExportedGlobal • ImportedExtern • ImportedExternPointer <p>For more information about tunable parameter storage classes, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.</p>
Storage type qualifier	<p>For variables with a storage class other than Auto, you can add a qualifier (such as <code>const</code> or <code>volatile</code>) to the generated storage declaration. To do so, you can select a predefined qualifier from the list, or add qualifiers not in the list by typing them in. The code generator does not check the storage type qualifier for validity, and includes the qualifier text in the generated code without checking syntax .</p>

Programmatically Declare Workspace Variables as Tunable Parameters

Tune Parameters from the Command Line

When parameters are MATLAB workspace variables, the Model Parameter Configuration dialog box is the recommended way to see or set the properties of tunable parameters. In addition to that dialog box, you can also use MATLAB `get_param` and `set_param` commands.

Note You can also use `Simulink.Parameter` objects for tunable parameters. See “Create Tunable Calibration Parameter in the Generated Code” on page 15-112 for details.

The following commands return the tunable parameters and corresponding properties:

- `get_param(gcs, 'TunableVars')`

- `get_param(gcs, 'TunableVarsStorageClass')`
- `get_param(gcs, 'TunableVarsTypeQualifier')`

The following commands declare tunable parameters or set corresponding properties:

- `set_param(gcs, 'TunableVars', str)`

The argument `str` (character vector) is a comma-separated list of variable names.

- `set_param(gcs, 'TunableVarsStorageClass', str)`

The argument `str` (character vector) is a comma-separated list of storage class settings.

The valid storage class settings are

- `Auto`
- `ExportedGlobal`
- `ImportedExtern`
- `ImportedExternPointer`
- `set_param(gcs, 'TunableVarsTypeQualifier', str)`

The argument `str` (character vector) is a comma-separated list of storage type qualifiers.

The following example declares the variable `k1` to be tunable, with storage class `ExportedGlobal` and type qualifier `const`. The number of variables and number of specified storage class settings must match. If you specify multiple variables and storage class settings, separate them with a comma.

```
set_param(gcs, 'TunableVars', 'k1')
set_param(gcs, 'TunableVarsStorageClass', 'ExportedGlobal')
set_param(gcs, 'TunableVarsTypeQualifier', 'const')
```

Share Data Between Code Generated from Simulink, Stateflow, and MATLAB

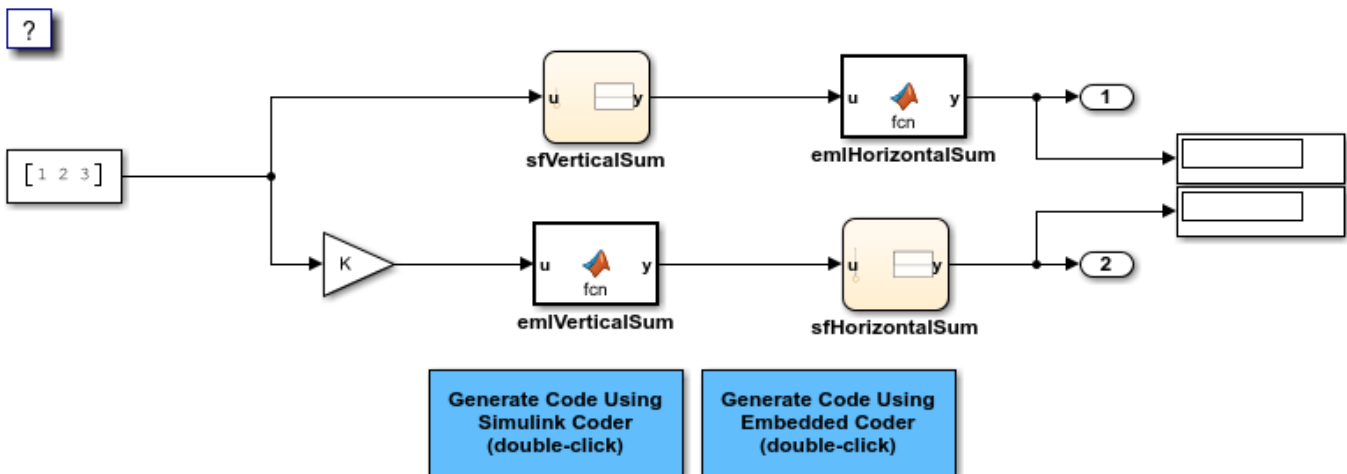
Stateflow® and MATLAB® Coder™ can fully define their data definitions, or they can inherit them from Simulink®. Data definition capabilities include:

- Inheriting input/output data types and sizes from Simulink.
- Parameterized data types and sizes. That is, data type and size may be specified as a function of another data's type and size, e.g., $\text{type}(y)=\text{type}(u)$ and $\text{size}(y)=\text{size}(u)$.
- Inferred output size and type from Simulink via signal attribute back propagation.
- Parameter scoped data, which allows referencing Simulink parameters in Stateflow and MATLAB.

Open Example Model

Open the example model `rtwdemo_dynamicio`.

```
open_system('rtwdemo_dynamicio')
```



Copyright 1994-2020 The MathWorks, Inc.

Instructions

- 1 Compile the model (on the **Simulation** tab, under **Prepare**, click **Update Model**) and note the displayed signal types and sizes.
- 2 Change the data type and/or size of the Constant block and recompile the model. Note that the attributes of the signals automatically adapt to the Constant block specification.
- 3 Generate and inspect code using the blue buttons in the model. Note that K is shared by the Gain and `sfVerticalSum` block.

Notes

- The data type and size of Stateflow and MATLAB data is inherited from Simulink.

- The Gain block and the Stateflow chart sfVerticalSum share the Simulink parameter K, which is defined in the MATLAB workspace as a Simulink.Parameter with Model default storage class (i.e., rtP.K in the generated code).

Configuring Data and Functions in the Generated Code

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Code Definition and Mapping Limitations and Considerations” on page 16-8
- “Migration of Model Data Configurations to Code Mappings” on page 16-11
- “Configure Root-Level Inport Blocks for C Code Generation” on page 16-18
- “Configure Root-Level Outport Blocks for C Code Generation” on page 16-27
- “Configure Signal Data for C Code Generation” on page 16-34
- “Configure Parameters for C Code Generation” on page 16-44
- “Configure Block States for C Code Generation” on page 16-53
- “Configure Data Stores for C Code Generation” on page 16-60
- “Manage Multiple Code Mappings for a Model” on page 16-68
- “Configure Model Data Interface for C Code Generation” on page 16-72
- “Configure Generated C Function Interface for Model Entry-Point Functions” on page 16-79

C Code Generation Configuration for Model Interface Elements

The code generator provides tooling for you to control the representation of the data and, if you have Embedded Coder, entry-point functions in the generated code. For example, you can configure data and functions for code generation to comply with code standards and guidelines, integrate generated code with external code, or address memory requirements.

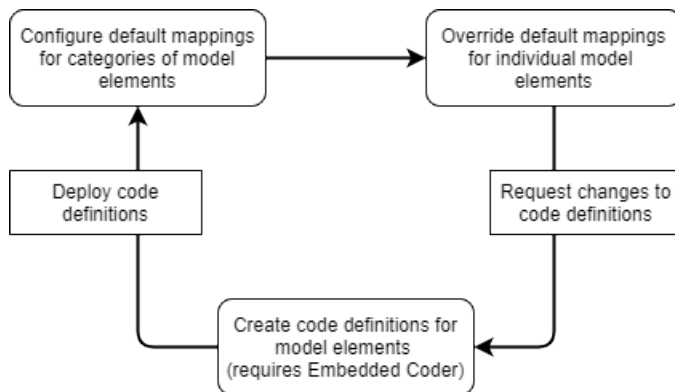
The code generator produces code for model data elements and functions:

- Data such as inports, signals, states, and parameters, which your application code can read from and write to.
- Entry-point functions such as `model_step`, which your application code calls to execute the model algorithm.

You can configure model data elements and functions for code generation by using the **Code Mappings Editor - C** or code mappings programming interface (`coder.mapping.api.CodeMapping`). Embedded Coder is required to configure functions. For data elements, use the Code Mappings editor or code mappings API instead of, or in addition to, creating and configuring Simulink data objects. The editor and API enable you to configure:

- Default code generation settings that apply throughout a model for a category of data elements
- Configuration settings that apply to individual data elements

This diagram shows the overall workflow:



Migration

When you open a model created in a release before R2020b, Simulink® migrates code definitions specified in the model to the model code mappings.

If necessary, as part of the migration, Simulink configures the Embedded Coder Dictionary that the model uses. When you save the model, the code mappings are saved as part of the model.

For more information, such as effects of migration and what to consider before migrating, see “Migration of Model Data Configurations to Code Mappings” on page 16-11 .

Code Definitions

Code definitions include storage classes. A storage class defines properties, such as appearance and location, that the code generator uses when producing code for the associated data.

Memory sections control the placement of data and function definitions in memory. In some cases, you can select a specific memory section that you want when configuring data elements and functions.

The code generator provides predefined code definitions for data elements. If you have Embedded Coder, you can use the Embedded Coder Dictionary to create and share code definitions, for example, to standardize code that you generate from multiple models or to create definitions for different application run-time environments. Creating custom definitions enables you to achieve code generation objectives that predefined definitions cannot satisfy. For data elements, you also have the option of creating code definitions created in packages by using Embedded Coder Custom Storage Class Designer.

Code definitions that you create by using the Embedded Coder dictionary appear in the **Code Mappings Editor - C** and are accessible by using the code mappings API. For some definitions that you create by using the Custom Storage Class Designer, you can make the definitions appear in the Code Mappings editor by configuring an Embedded Coder Dictionary to refer to them.

By default, code definitions that you create by using the Embedded Coder Dictionary for a model are stored in the model file. You can share code definitions between models and projects by storing the definitions in a Simulink data dictionary. With a data dictionary, to modify a shared definition, you change a definition in one place—the Embedded Coder Dictionary.

For more information, see “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder) and “Create Storage Classes by Using the Custom Storage Class Designer” (Embedded Coder).

Data Default Configurations

Reduce the effort of preparing a model for C code generation by specifying default configurations for categories of data elements, for example inports or model workspace parameters, across a model. Applying default configurations can save time and reduce the risk of introducing errors in code, especially for larger models and models from which you generate multi-instance code.

Default configurations apply throughout a model, excluding referenced models. Simulink saves a separate set of code mappings as part of each model in a model reference hierarchy.

If a model includes a significant number of elements of a given category (for example, more than 10), it is more efficient to configure the category by using a default setting, and then override that setting for special cases.

The default settings can help you to reduce manual data entry.

- As you add blocks and signals to a model, new data elements inherit the default settings.
- To change code generation settings for many data elements at once, make the changes in one place—the default settings.
- For efficiency, use naming rules as you make changes elsewhere in the model.

You configure data defaults by using the **Code Mappings Editor - C Data Defaults** tab or the `setDataDefault` function.

Model Data Categories

You configure data defaults by using the **Code Mappings Editor - C Data Defaults** tab or the `setDataDefault` function. In the **Data Defaults** tab or function call, you can select or specify these categories of data elements.

Model Element Category	Description
Inports	Root-level input ports of a model, such as Inport and In Bus Element blocks.
Outports	Root-level output ports of a model, such as Outport and Out Bus Element blocks.
Signals, states, and internal data	Data elements that are internal to the model, such as block output signals, discrete block states, data stores, and zero-crossing signals.
Shared local data stores	Data Store Memory blocks that have the block parameter Share across model instances set. These data stores are accessible only in the model where they are defined. The data store value is shared across instances of the model.
Global data stores	Data stores that are defined by a signal object in the base workspace or in a data dictionary. Multiple models in an application can use these data stores. To view and configure these data stores in the Code Mappings editor, click the Refresh link to the right of the category name. Clicking this link updates the model diagram.
Model parameters	Parameters that are defined within a model, such as parameters in the model workspace. Excludes model arguments.
External parameters	Parameters that you define as objects in the base workspace or in a data dictionary. Multiple models in an application can use these parameters. To view and configure these parameters in the Code Mappings editor, click the Refresh link to the right of the category name. Clicking this link updates the model diagram.

Considerations for Configuring Default Code Generation Settings for Data

Before configuring default code generation settings for data elements, take into account the considerations listed in this table.

Consideration	More Information
What categories are relevant to your model?	The data element categories in the preceding table

Consideration	More Information
Does the model use several instances of data that is in a category? If the answer is yes, applying default mappings is beneficial. Otherwise, consider configuring code generation for each data element individually.	<ul style="list-style-type: none"> • “Configure Root-Level Inport Blocks for C Code Generation” on page 16-18 • “Configure Root-Level Outport Blocks for C Code Generation” on page 16-27 • “Configure Signal Data for C Code Generation” on page 16-34 • “Configure Parameters for C Code Generation” on page 16-44 • “Configure Block States for C Code Generation” on page 16-53 • “Configure Data Stores for C Code Generation” on page 16-60
Which storage class aligns with your code generation requirements for each category?	“Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75
Do you want to prevent optimizations from eliminating specific data from the code?	“Elimination of Categories of Data by Optimizations” on page 16-5

After you configure one or more categories of model elements, setting memory sections and **Shared utilities identifier format** model configuration parameters has no effect on the model configuration. When you open a coder app, Simulink migrates the model configuration parameter settings to the Code Mappings editor. As part of the migration, Simulink configures the Embedded Coder Dictionary that the model uses, as described in “Migration of Memory Section and Shared Utility Settings from Configuration Parameters to Code Mappings” (Embedded Coder).

Elimination of Categories of Data by Optimizations

Code generation optimizations can eliminate data from the code, which means your application code cannot interact with the data. For general information about data elimination by optimizations, see “How Generated Code Stores Internal Signal, State, and Parameter Data” (Embedded Coder).

The optimizations can eliminate data in these categories only:

- **Model parameters**
- **Model parameter arguments**
- **External parameters**
- **Internal data**

After eliminating data with optimizations, the code generator applies the default code generation settings in the code mappings to the remaining data. For parameters and signal lines, optimizations can eliminate data for an entire category. If this elimination occurs, default settings that you specify for that category do not apply to data.

To prevent optimizations from eliminating individual data elements, explicitly apply a storage class to the individual element. The storage class controls the appearance of the data in the generated code. To force a data element to use the default storage class that you specify, explicitly apply the storage class `Model default`. See “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75.

Shared Dictionary Data Defaults

If you link a model to a Simulink data dictionary, which includes a coder dictionary that configures default code definitions for categories of data, you can use the Code Mappings editor to apply the dictionary defaults. In the Code Mappings editor, on the **Data Defaults** tab, select a category and set the storage class to `Dictionary Default`. If someone makes a change to the default settings in the shared coder dictionary, the code generator applies the updated default settings when producing code for your model. See “Configure Default Code Mapping in a Shared Dictionary” (Embedded Coder).

Configurations for Individual Data Elements

After you configure model-wide default settings, you can override the defaults for individual data elements. An individual data element configuration applies to a specific data element in a model. Configure data elements individually when:

- The model includes a few elements of a given category that have unique source, naming, or placement requirements.
- You configured default settings and you want to override the default setting for an individual element.

For individual data elements, initially the storage class is set to `Auto`, which means that the code generator might eliminate or change the representation of relevant code for optimization purposes. If optimizations are not possible, the code generator applies the model default configuration.

- To avoid optimizations and force the code generator to use the default configuration, set the storage class to `Model default`.
- To override the default configuration, specify the storage class that meets the code generation requirements for that data element.

If a data element in a model is unnamed (for example, a signal has no label or a state has no name), you must specify a code identifier for the data element when configuring it for C code generation. Set the code identifier by specifying a value for the storage class property **Identifier**.

Configure Data

- 1 Open the Simulink Coder app.
- 2 Configure default mappings for categories of data elements. Select **Code Interface > Default Code Mappings**. In the Code Mappings editor, on the **Data Defaults** tab, select a model element category and set the storage class. In the Property Inspector, set storage class properties to align with model requirements.
- 3 Decide whether to override default configuration settings for individual data elements. If you choose not to override settings, go to step 8.
- 4 Identify MATLAB variables that you want to configure individually and store the configurations in the model file. In the Model Explorer, convert the variables to `Simulink.Parameter` objects. The resulting data objects appear in the Code Mappings editor as model parameters.
- 5 Identify signals representing algorithm data that you want to monitor while the generated code executes and configure those signals individually. For each signal that you identify, do one of the following:

- For signal data that is specific to a model (not shared), add the signals to the Code Mappings editor.
 - For shared signal data, create and configure `Simulink.Signal` objects.
- 6** Configure individual data elements. In the Code Mappings editor, click a category tab, select a data element, and set the storage class. In the Property Inspector, set storage class property values. To change the setting of model configuration parameter **Default parameter behavior**, click the link indicating the behavior when the storage class is set to `Auto`. In the Model Configuration Parameters dialog box, change the parameter setting to `Tunable` or `Inline`.

For information on configuring specific types of data elements, see:

- 7** To view and configure external data objects that are stored in the base workspace or a data dictionary, in the Code Mappings editor, on the **Data Defaults** tab, click the `Refresh` link to the right of the category name.
- 8** Generate and review code.

For more details and examples, see:

- **Code Mappings Editor - C**
- “Configure Root-Level Inport Blocks for C Code Generation” on page 16-18
- “Configure Root-Level Outport Blocks for C Code Generation” on page 16-27
- “Configure Signal Data for C Code Generation” on page 16-34
- “Configure Parameters for C Code Generation” on page 16-44
- “Configure Block States for C Code Generation” on page 16-53
- “Configure Data Stores for C Code Generation” on page 16-60

See Also

Code Mappings Editor - C | `coder.mapping.api.CodeMapping`

More About

- “Configure Root-Level Inport Blocks for C Code Generation” on page 16-18
- “Configure Root-Level Outport Blocks for C Code Generation” on page 16-27
- “Configure Signal Data for C Code Generation” on page 16-34
- “Configure Parameters for C Code Generation” on page 16-44
- “Configure Block States for C Code Generation” on page 16-53
- “Configure Data Stores for C Code Generation” on page 16-60
- “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75

Code Definition and Mapping Limitations and Considerations

Limitations

- If a data element in a model is unnamed (for example, a signal has no label or a state has no name), you must specify a code identifier for the data element when configuring it for C code generation. Set the code identifier by specifying a value for the storage class property **Identifier**.
- You cannot use code mappings to configure MATLAB variables that are used as block parameters. Convert the variables to `Simulink.Parameter` objects in the model workspace. The resulting data objects appear in the Code Mappings editor as model parameters.
- Use code mappings to configure only one discrete state per block in a model.
- If a model contains a tunable, nonfinite parameter, for example, with a value of `inf`, the code generator does not apply a specified default parameter mapping. The parameter appears as a field in the default parameter data structure `rtP` (see “Real-Time Model Data Structure” on page 15-32).

Default Code Mapping Constraints

Default code mappings for categories **External parameters** and **Global data stores** must be the same for models throughout the model reference hierarchy.

Model Editing Operations and Individual Data Element Code Mappings

While editing a model in the context of a coder app, you should be aware of the relationship between Simulink model editing tools, such as copy and undo, and code mappings. In the context of a coder app, when you perform copy-and-paste or cut-and-paste editing operations within a model, the coder app preserves the code mappings for individual data elements. For example, within a model, if you copy and paste the source block of a signal that is configured with a specific storage class, the pasted version of the signal port retains the same storage class.

While you can use undo and redo operations for editing actions on model data elements, they are unavailable while configuring code mappings. For example, if you delete a state block, and then click **Undo**, the block reappears in the model canvas and the code configuration for the state reappears in the code mappings. In the Code Mappings editor, if you change the storage class setting for a state from `Auto` to `Model default`, and then click **Undo**, the storage class does not revert to `Auto`. In this case, the change pertains to the code mapping, not to the model data element.

Storage Classes and Reentrant, Multi-Instance Models and Components

The code that you generate from a model or component (referenced model or subsystem) is multi-instance code if it allows your application to maintain multiple independent instances of the component during execution. For example, you can generate reentrant, multi-instance code from an entire model by setting the model configuration parameter **Code interface packaging** to `Reusable function`. For general information about multi-instance models and components, see “Model Composition and Reuse”.

Under some circumstances, applying storage classes can:

- Prevent you from generating multi-instance code.
- Cause some data elements to appear in the generated code as singletons, which means that each instance of the model or component directly accesses the same shared, global data, creating dependencies between the instances.

To avoid errors and unexpected generation of single-instance code and singleton data, observe the guidelines and limitations below.

Directly Applied Storage Classes

When you apply a storage class directly to a data element (see “C Code Generation Configuration for Model Interface Elements” on page 16-2):

- To generate multi-instance code, you must apply the storage class only to parameter objects, global data stores, shared local data stores, and root-level I/O.
- The storage class yields only singleton data.

Storage Classes Applied by Default

When you apply a default storage class (see “C Code Generation Configuration for Model Interface Elements” on page 16-2):

- For these data categories, you can generate only singleton data:
 - **Model parameters**
 - **External parameters**
 - **Global data stores**
 - **Shared local data stores**
- For **Internal data**, you can generate only instance-specific data, which means each instance of the model or component operates on a separate copy of the data. For this category, you must use the example storage class `SignalStruct`, which appears after preparing a model for code generation with the Quick Start tool. Alternatively, you can create and use your own structured storage class by using an Embedded Coder Dictionary.

For more information, see “What Is Reentrant Code?” on page 10-3 and “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75.

Copy Code Mappings When Converting Subsystems to Referenced Models

You can copy code mappings when you convert a subsystem to a referenced model.

- In the **Model Reference Conversion Advisor**, select **Copy code mappings**.
- In the MATLAB command window, use the `Simulink.SubSystem.convertToModelReference` function with the comma-separated pair consisting of `'CopyCodeMappings'` and `true`.

See Also

Code Mappings Editor - C | `coder.mapping.api.CodeMapping`

More About

- “C Code Generation Configuration for Model Interface Elements” on page 16-2

Migration of Model Data Configurations to Code Mappings

Starting in R2020b, the recommended approach to configure model data for code generation is to use the Code Mappings editor or code mappings API. From one interface, you can configure default settings for categories of data, and then override those settings, as needed, for specific data elements. For parameters, you can specify whether they are inlined or tunable by default. When producing code for data, the code generator uses the code definitions that you specify to determine how to represent the data in the generated code.

When you open a model created in a release before R2020b, Simulink migrates code definitions specified in the model to the model code mappings.

Effects of Migration

For data configured within a model, the migration process replaces existing storage class specifications with equivalent code mappings. Data configured for code generation within a model includes data represented by:

- Outport blocks
- Signal lines
- Block states
- Data stores
- Parameter objects in the model workspace

This table lists specific model conditions and their corresponding migration results.

Condition	Migration Results
Model is a library model.	Simulink discards the code configuration associated with data configured within the model.
Data element is configured for code generation within the model and the data element is associated with a <code>Simulink.CoderInfo</code> object, the <code>StorageClass</code> property for that object is set to <code>Custom</code> , and the <code>CustomStorageClass</code> property is set to <code>Default</code> .	Simulink changes the storage class setting from <code>Custom (Default)</code> to <code>ExportedGlobal</code> and displays a warning.

Condition	Migration Results
<p>Model uses data objects configured for use with the <code>TypeQualifier</code> storage class property for a built-in storage class, such as <code>ExportedGlobal</code> or <code>ImportedExtern</code>. You might have set the property as:</p> <ul style="list-style-type: none"> • <code>CoderInfo.TypeQualifier</code> property for data objects in a workspace or data dictionary • Port parameter <code>RTWStorageTypeQualifier</code> • Block parameter <code>RTWStateStorageTypeQualifier</code> for Data Store Memory, Discrete Filter, Discrete State-Space, Discrete-Time Integrator, Discrete Transfer Fcn, Discrete Zero-Pole, and Memory blocks 	<p>Simulink issues a warning indicating that the property is no longer supported. Other interfaces are available for achieving the same results.</p>
<p>Model uses the <code>Simulink.CoderInfo</code> object <code>Alignment</code> property for data configured within the model.</p>	<p>Simulink issues a warning indicating that the property is no longer supported for model-specific data. You can use the property for data stored in the base workspace or a data dictionary.</p>
<p>You initialize the model workspace for the model from an external data source, such as a MAT-file, MATLAB file, or data dictionary.</p>	<p>Simulink moves code configuration for the parameter object to the code mappings for that model.</p>
<p>For an individual subsystem from which you generate code and executable programs by right-clicking the Subsystem block, you configure the storage class and storage class properties for the subsystem input and output signals outside of the subsystem.</p>	<p>Generated code does not include variables for the input and output interface data.</p>

If you perform these operations, you might need to take action:

- Use the Simulink Editor to copy a block line to another model. Simulink does not copy the code configuration information associated with the copied modeling element. For more information, see “Copy Blocks and Signal Lines Between Models” (Embedded Coder).
- Switch the system target file setting for a model between a GRT-based and ERT-based system target file. Simulink ignores the code configuration information for the system target file of the source version of the copied model. For more information, see “Switch Between GRT- and ERT-Based System Target Files” (Embedded Coder).

Considerations Before Migrating

Before migration, consider the preparation suggested in the following sections if these conditions exist:

- Data configured within a library model
- Storage class `TypeQualifier` property
- Data alignment for code generation

- Code configuration for parameter objects initialized in the model workspace
- Code configuration for subsystem I/O interface

Data Configured Within Library Model

If your model is a library model, reconfigure code generation for data. Reconfigure code generation for data in the context of models that use the library (see “Code Interface Configuration”).

To avoid losing the code configuration for data configured within the model, in an earlier release, load the model and create and run a MATLAB script that:

- 1 Loads the model data.
- 2 Finds data configured for code generation within the model.
- 3 Converts the representation of the data to data objects that you store in an external source, such as a MAT-file, MATLAB file, or data dictionary.
- 4 Updates the model to use the external data objects.

Storage Class TypeQualifier Property

If your model configures data objects with built-in storage classes, such as `ExportedGlobal` and `ImportedExtern`, and uses the storage class `TypeQualifier` property, when you load the model in release R2020b or later, Simulink issues a warning. To eliminate the warning and update a model to use a supported alternative, in an earlier release, load the model and create and run a MATLAB script that:

- 1 Loads the model data objects from the model workspace, base workspace, or data dictionary.
- 2 Finds data objects with the `TypeQualifier` property set to a non-empty string value.
- 3 Changes the storage class setting to a storage class predefined with the required type qualifier (for example, storage class `Const` includes qualifier `const` in data declarations and definitions).

This example script loads model data objects from the base workspace, finds instances of data configured with the storage class `TypeQualifier` property, and changes the storage class setting to `Const`.

```
vars = who;

for idx = 1:length(vars)
    thisVar = evalin('base',vars{idx});
    if (isa(thisVar,'Simulink.Parameter') && (isequal(thisVar.CoderInfo.TypeQualifier, 'const')))
        thisVar.CoderInfo.TypeQualifier = '';
        thisVar.CoderInfo.StorageClass = 'Custom';
        thisVar.CoderInfo.CustomStorageClass = 'Const';
    end
end
```

Data Alignment for Code Generation

To preserve data alignment information configured for a model, represent data with data objects outside of the model. For example, open the model in a previous release of Simulink and perform the actions listed in this table.

Type of Data Element	Action
Parameter object in the model workspace	Use the Model Explorer to move the object from the model workspace to the base workspace or a data dictionary. Then, set the storage class and alignment properties of the object. Save the model.
Signal line	Open the Signal Properties dialog box, specify a name for the signal and select Signal name must resolve to Simulink signal object (storage class must be Auto). Create a <code>Simulink.Signal</code> object in the base workspace or a data dictionary and set the storage class and alignment properties of the object. Save the model.
Block state	Open the Block Parameters dialog box, specify a name for the state and select State name must resolve to Simulink signal object (storage class must be Auto). Create a <code>Simulink.Signal</code> object in the base workspace or a data dictionary and set the storage class and alignment properties. Save the model.

Alternatively, create and run a MATLAB script that:

- 1 Loads the model data.
- 2 Finds data objects that set `Simulink.CoderInfo.Alignment`.
- 3 Converts the representation of the data to data objects that you store in an external source, such as a MAT-file, MATLAB file, or data dictionary.
- 4 Sets `Simulink.CoderInfo` object property `Alignment` for the new data objects.

For example, this script checks whether `Alignment` is set for parameter, lookup table, and breakpoint data objects and captures the data objects found in MAT-file `savedData.mat`.

```
filename = 'savedData.mat';
varList = who;
tosaveVars = {};
for idx = 1 : length(varList)
    variable = eval(varList{idx});
    if ((isa(variable, 'Simulink.Parameter') || ...
        isa(variable, 'Simulink.LookupTable') || ...
        isa(variable, 'Simulink.Breakpoint')) && ...
        variable.CoderInfo.Alignment ~= -1)
        tosaveVars{end+1} = varList{idx};
    end
end
if ~isempty(tosaveVars)
    save('savedData.mat', tosaveVars{:});
end
```

Evaluate the script in the model workspace with these commands:

```
mws = get_param(bdroot, 'ModelWorkspace');
evalin(mws, 'SaveMWSParamsWithAlignment');
```

For parameter objects in the model workspace, alternatively, you can use the Model Explorer interface to move the objects from the model workspace to the base workspace or a data dictionary. Then, select the **Alignment** property for the storage class.

Code Configuration for Parameter Objects Initialized in Model Workspace

After loading a model in release R2020b or later and initializing the model workspace for the model from an external data source, such as a MAT-file, MATLAB file, or data dictionary, Simulink:

- Saves the contents of the `Simulink.CoderInfo` object to the model code mappings.
- Removes the `Simulink.CoderInfo` object from the `Simulink.Parameter` object defined in the external data source.

Use the Code Mappings editor or the code mappings API to get and set parameter code configuration settings. See “Code Interface Configuration”.

Code Configuration for Subsystem I/O Interface

In release R2020b or later, for individual subsystems from which you generate code and executable programs by right-clicking the Subsystem block, to include signal data in the generated subsystem code interface, configure the storage class and storage class properties for the subsystem input and output signals within the subsystem. If you configure the signals outside of the subsystem, the generated code does not include variables for the input and output interface data.

For more information, see “Generate Code and Executables for Individual Subsystems” on page 7-2.

Copy Blocks and Signal Lines Between Models

Starting in R2020b, when you use the Simulink Editor to copy a block or signal line to another model, Simulink does not copy the code configuration information associated with the copied modeling element.

To work around this change, copy and paste the code configuration information for data elements by using the code mappings API.

- 1 Open the source and destination models for the copy operation.
- 2 Use the code mappings API to get the code mappings object for the source model.
- 3 Use the code mappings API to get code mapping information for the model element that you are copying.
- 4 If a code mappings object does not exist for the destination model, use the code mappings API to create one.
- 5 For the model element that you are copying to the destination model, use the code mappings API to set code mapping information to the same values as for the source model.
- 6 Close the source model.
- 7 Save and close the destination model.

For example:

```
.
.
.
srcMdl = 'my_src_mdl';
destMdl = 'my_dest_mdl';

open_system(srcMdl);
open_system(destMdl);

% Get data configuration from source model
```

```
srcCM = coder.mapping.api.get(srcMdl);
srcSC = getSignal(srcCM, srcPort, 'StorageClass')

% Set data configuration for destination model to setting of source model
destCM = coder.mapping.api.get(destMdl);
setSignal(destCM, destPort, 'StorageClass', srcSC);

close_system(srcMdl);
save_system(destMdl);
close_system(destMdl);
.
.
.
```

Switch Between GRT- and ERT-Based System Target Files

Starting in R2020b, when you switch the system target file setting for a model between a GRT-based and ERT-based system target file, Simulink applies alternative code configurations for each system target file. This switch can occur if you:

- Alternate between the Simulink Coder and Embedded Coder apps.
- Change the active configuration set for a model.
- Change the setting of model configuration parameter **System target file**.

To preserve the code configuration information for data configured within the model, create and run a MATLAB script that uses the code mappings API to copy relevant code mappings.

- 1 Open the model with the source system target file and code mappings that you want to preserve.
- 2 Use the code mappings API to get the code mappings object for the model and get data mappings objects for the data elements.
- 3 Use the code mappings API to get the code mappings object for the model and set data mappings objects for the data elements.
- 4 Save and close the model.

This script fragment shows how to use the code mappings API to preserve the storage class of a signal object when switching from a GRT system target file to a an ERT system target file.

```
.
.
.
mymdl = 'my_shared_mdl';

open_system(mymdl);

% Get data configuration for Simulink Coder (GRT-based) code generation
scCodeMapping = coder.mapping.api.get(model, 'SimulinkCoderC');
scStorageClass = getSignal(scCodeMapping, srcPort, 'StorageClass')

% Set data configuration for Embedded Coder (ERT-based) code generation
ecCodeMapping = coder.mapping.api.get(model, 'EmbeddedCoderC');
setSignal(ecCodeMapping, srcPort, 'StorageClass', scStorageClass)

save_system(mymdl);
close_system(mymdl);
.
```


Workflow Changes

Starting in R2020b, configure model data elements for code generation by using the Code Mappings editor or code mappings API instead of or in addition to creating and configuring Simulink data objects. Use this workflow:

- 1 Open the **Simulink Coder** app.
- 2 In the Code Mappings editor, configure default behavior for storage representation. .
- 3 Decide whether to override default configuration settings for individual data elements. If you choose not to override settings, skip to step 8.
- 4 Identify MATLAB variables that you want to configure individually and store the configurations in the model workspace. In the Model Explorer, convert the variables to `Simulink.Parameter` objects. The resulting data objects appear in the Code Mappings editor as model parameters.
- 5 Identify signals representing algorithm data that you want to monitor while the generated code executes and configure individually. For each signal that you identify, do one of the following:
 - For signal data that is specific to a model (not shared), add the signals to the Code Mappings editor. In the model, pause on the ellipsis above a signal line to open the action bar. Click the **Add Signal** button to add the signal to the Code Mappings editor.
 - For shared signal data, create and configure `Simulink.Signal` objects. See “C Code Generation Configuration for Model Interface Elements” on page 16-2 and “Control Signal and State Initialization in the Generated Code” on page 15-177.
- 6 Configure individual data elements. In the Code Mappings editor, click a category tab, select a data element, and set the storage class. In the Property Inspector, set storage class property values. To change the setting of model configuration parameter **Default parameter behavior** click the link indicating the behavior when the storage class is set to `Auto`. In the Model Configuration Parameters dialog box, change the parameter setting to `Tunable` or `Inline`.
- 7 To view external data objects that are stored in the base workspace or a data dictionary, in the Code Mappings editor, click the Code Mappings editor **Refresh** button.
- 8 Generate and review code.

See Also

More About

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Control Data and Function Interface in Generated Code” on page 15-44
- “Control Signal and State Initialization in the Generated Code” on page 15-177

Configure Root-Level Inport Blocks for C Code Generation

To use code that you generate from a model, you call generated entry-point functions. The calling environment and generated functions exchange input and output data, for example, as global variables or function arguments. Root-level Inport blocks (inports) constitute the input data of the interface. To integrate and deploy the generated code into applications, you can customize how the code generator produces the interface code, including how input data is declared and handled. Customizations can:

- Minimize the modifications that you must make to existing code.
- Generate stable interfaces that do not change or minimally change when you make changes to your model.
- Generate code that exchanges data more efficiently (for example, by using pointers and pass-by-reference arguments for nonscalar data).

For code generation, examples show how to customize the inport interface for model `rtwdemo_configrpininterface`. You can configure code mappings by using the **Code Mappings Editor - C** or code mappings API (`coder.mapping.api.CodeMapping`).

Choose Customization Options for Inport Interface

By default, root-level inports in a model appear in generated code as fields of a global data structure named `model_ExtU`. Based on your code interface requirements, decide whether to customize generation of root-level inport data. If you do not configure customizations, the code generator determines whether to eliminate or change the representation of inports in generated code for optimization purposes. If you configure customizations, decide:

- Whether to set up a default configuration

If a model includes a significant number (for example, more than 10) of root-level inports, it is more efficient to configure the inports with a default setting, and then override that setting for special cases. If the model includes a few root-level inports that have unique source, naming, or placement requirements, consider configuring the inports individually.

- How to declare and handle root-level inport data in the generated interface
 - As separate global variables
 - To read input data from global variables defined in external code
 - For referenced model inports, as global variables (`void-void`)
 - As calls to access functions. Requires Embedded Coder
 - As entry-point function arguments. Requires Embedded Coder

For more information about these options, see “Control Data and Function Interface in Generated Code” on page 15-44.

Other considerations include whether to:

- Name inports in the generated code by using Inport block labels that appear in the model or by using unique code identifiers.
- Include the `volatile` type qualifier in global variable definitions and declarations. Requires Embedded Coder. See “Protect Global Data with `const` and `volatile` Type Qualifiers” (Embedded Coder).

- Place data read into a specific area of memory. Requires Embedded Coder. See “Control Data and Function Placement in Memory by Inserting Pragmas” (Embedded Coder).

For a list of interface requirements that are relevant to inports with corresponding storage classes and storage class properties, see “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75.

For example model `rtwdemo_configrpinterface`, inport interface requirements are:

- Get data for root-level inports `In1`, `In2`, `In3`, and `In4` from an external header file. The data read with `In1` is used for computing a value that is stored in memory, and then used in an if-else condition of code generated for the Switch block. Data read from the other three inports is used in lookup tables `Table1` and `Table2`.
- Data for inport `In1` must be passed by value.
- Data for inports `In2`, `In3`, and `In4` must be passed by reference.
- Variables representing the inports in the generated code must be named `input1`, `input2`, `input3`, and `input4`.

For this example, configure inports in `rtwdemo_configrpinterface` to meet these code generation requirements.

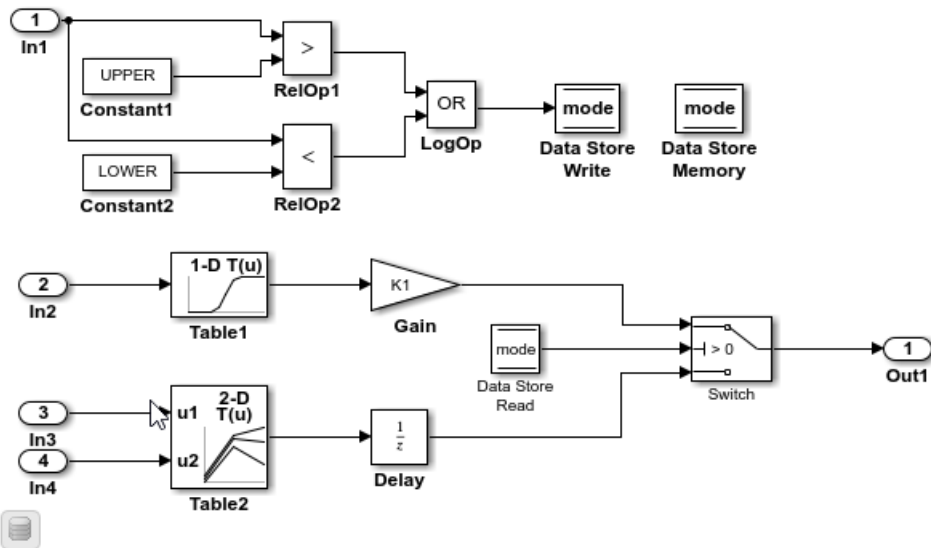
Configure Default Code Generation Settings for Root-Level Inports

A default code generation setting for root-level inports can reduce the effort of preparing a model for code generation, especially if a model has a significant number of inports. Choose configuration settings once, and the code generator applies those settings to the inports across the model. Simulink stores the default configuration as part of the model.

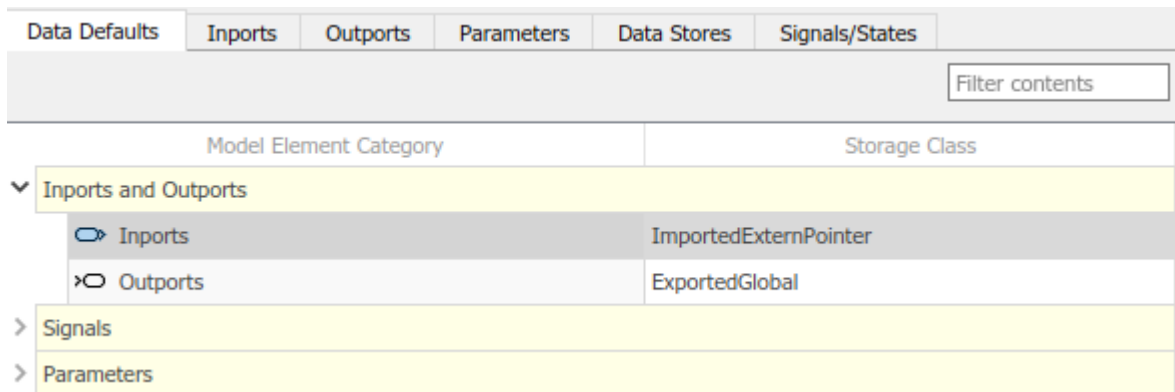
Consider configuring default code generation settings for model root-level inports if your model uses multiple root-level inports that do not have unique requirements.

This example shows how to use the **Code Mappings Editor - C** to configure default settings for root-level inports for model `rtwdemo_configrpinterface`. Specify that the inports read input from variables declared in an external header file.

- 1 Open example model `rtwdemo_configrpinterface`. Save a copy of the model to a writable location.



- 2 Open the **Simulink Coder** app.
- 3 In the **C Code** tab, select **Code Interface > Default Code Mappings**.
- 4 In the Code Mappings editor, under **Inports and Outports**, select category **Inports**. Set the default storage class to ImportedExternPointer.



- 5 Save the model.

Configure Code Generation Settings for Individual Root-Level Inports

You can configure individual root-level inports for code generation. For example, if a model has two root-level inports that have unique code generation requirements, configure the inports individually. Or, if you configure default settings for inports, configure individual inports to use the default settings or unique settings.

If your model meets at least one of these criteria, consider configuring code generation settings for root-level inports individually:

- Uses multiple inports that have unique requirements.
- Uses few inports.
- Has a default configuration for inports and you need to override the configuration for some specific inports.

This example shows how to use the Code Mappings editor to override the default code generation configuration for one of the inports in model `rtwdemo_configrpininterface`. In the preceding example, you set the default storage class for inports to `ImportedExternPointer`.





The example also shows how to configure identifiers that the code generator uses for naming the four inports in the generated code. You can specify code generation identifiers, for example for integration, without having to modify the model design.


- 1 If you have not already done so, complete the steps in “Configure Default Code Generation Settings for Root-Level Inports” on page 16-19.
- 2 In the Code Mappings editor, click the **Inports** tab. The editor lists the names of Inport blocks and bus elements that are in the model. If a port resolves to a signal object, a resolve-to-signal-object icon appears to the right of the element name. The storage class for each inport is set to `Auto`, which means that the code generator might eliminate or change the representation of relevant code for optimization purposes. If optimizations are not possible, the code generator applies the model default configuration. For this example, the model default configuration specifies storage class `ImportedExternPointer`.
 - To avoid optimizations and force the code generator to use the default configuration, set the storage class to `Model default`.
 - To override the default configuration, specify the storage class that meets the code generation requirements for that inport.
- 3 Configure the code generator to apply the default storage class setting to inports `In2`, `In3`, and `In4`. Select the rows for the three inports. Then, for one of the selected inports, set the storage class to `Model default: ImportedExternPointer`.

Data Defaults		Inports	Outputs	Parameters	Data Stores	Signals/States
Filter contents						
		Source	Storage Class			
⊞	In1		Auto			
⊞	In2		Auto			
⊞	In3		Auto			
⊞	In4		Model default: ImportedExternPointer			

The storage class for the three selected inports changes to `Model default: ImportedExternPointer`.

- 4 For inport `In1`, override the default storage class setting. Set the storage class to `ImportedExtern`.

Data Defaults	Inports	Outports	Parameters	Data Stores	Signals/States
Filter contents					
Source		Storage Class			
	In1	ImportedExtern			
	In2	Model default: ImportedExternPointer			
	In3	Model default: ImportedExternPointer			
	In4	Model default: ImportedExternPointer			

- 5 Configure code identifiers for the inports so that the interface argument names in the generated code match the interface names in the external header and definition files. In the Code Mappings editor, select the row for each inport. Click the  icon and set the **Identifier** property to these values.

Inport	Code Identifier
In1	input1
In2	input2
In3	input3
In4	input4

- 6 Save the model.
- 7 Generate and view the code. For example, in `rtwdemo_configrpininterface.c` find where variables `input1`, `input2`, `input3`, and `input4` are used in the step entry-point function.

```
void rtwdemo_configrpininterface_step(void)
{
    mode = ((input1 > rtwdemo_configrpininterface_UPPER) || (input1 <
        rtwdemo_configrpininterface_LOWER));

    dout_Table1 = look1_binlc((*input2), (&(mp_Table1.BP[0])),
        (&(mp_Table1.Table[0])), 10U);
    .
    .
    .
    dout_Table2 = look2_binlc((*input3), (*input4), (&(mp_Table2.BP1[0])),
        (&(mp_Table2.BP2[0])), (&(mp_Table2.Table[0])),
        (&(rtwdemo_configrpininterface_Table2_maxIndex[0])), 3U);

    dstate_X = dout_Table2;
}
}
```

Configure Code Generation Settings for Root-Level Inports Programmatically

To automate configuration of root-level inports for code generation, use the programming interface for code mappings. For example, when creating custom block libraries or part of an application test environment, use the programming interface to automate data configuration.

This example shows how to use the programming interface to configure default settings for root-level inports for model `rtwdemo_configrpininterface`. Three of the four inports read input from global pointer variables defined in an external file. You configure the generated code for inports to read

input from global pointer variables by default. Then, override the default code generation configuration for In1, which requires that input data be passed by value.

In addition, you configure code identifiers for the inports so that global variable names in the generated code match the variable names defined in an external file.

- 1 Open the example model.

```
open_system('rtwdemo_configrpinterface')
```

- 2 Create object `cm` by calling function `coder.mapping.api.get`. The object stores the data code generation configuration for model `rtwdemo_configrpinterface`.

```
cm = coder.mapping.api.get('rtwdemo_configrpinterface');
```

- 3 Configure default settings for inports by calling function `setDataDefault`. For the arguments, specify these values:

- The object returned by `coder.mapping.api.get`
- Inports for the default category
- Property name `StorageClass` with property value `ImportedExternPointer`

```
setDataDefault(cm,'Inports','StorageClass','ImportedExternPointer')
```

- 4 Verify your default configuration for inports. Issue a call to `getDataDefault` that specifies the object returned by `coder.mapping.api.get`, category `Inports`, and `StorageClass`.

```
getDataDefault(cm,'Inports','StorageClass')
```

```
ans =
```

```
    'ImportedExternPointer'
```

- 5 Override the default inport configuration for inport In1. The requirements for that inport differ from requirements of the other three inports.

By default, Simulink sets the storage class for individual inports to `Auto`. When the storage class is `Auto`, the code generator:

- Determines whether to eliminate the data from the generated code for optimization purposes.
- If retaining the data, determines how to efficiently represent the data in the generated code, taking into account default configuration settings.

To control the storage class configuration for an inport, configure settings for the inport with function `setInport`. Configure code generated for inport In1 to read input data from global variables defined in an external file, as specified in the requirements. Issue a call to function `setInport` that specifies the object returned by `coder.mapping.api.get`, Inport block named In1, property name `StorageClass`, and property value `ImportedExtern`.

```
setInport(cm,'In1','StorageClass','ImportedExtern')
```

- 6 Verify your configuration for inport In1 by calling function `getInport`.

```
getInport(cm,'In1','StorageClass')
```

```
ans =
```

```
    'ImportedExtern'
```

- 7 Configure the code generator to apply your default inport settings to inports In2, In3, and In4. For each inport, call function `setInport`. Specify the object returned by `coder.mapping.api.get`, the Inport block name, property name `StorageClass`, and property value `Model default`.

```
setInport(cm,'In2','StorageClass','Model default')
setInport(cm,'In3','StorageClass','Model default')
setInport(cm,'In4','StorageClass','Model default')
```

- 8 Verify your configuration changes for inports In2, In3, and In4 with calls to function `getInport`.

```
getInport(cm,'In2','StorageClass')
```

```
ans =
```

```
    'Model default'
```

```
getInport(cm,'In3','StorageClass')
```

```
ans =
```

```
    'Model default'
```

```
getInport(cm,'In4','StorageClass')
```

```
ans =
```

```
    'Model default'
```

- 9 Configure code identifiers for the inports so that the global variables that represent inports in the generated code match the variable names in the external definition file. For each inport, call function `setInport`. Specify the object returned by `coder.mapping.api.get`, the Inport block name, property name `Identifier`, and one of these property values.

Inport	Code Identifier
In1	input1
In2	input2
In3	input3
In4	input4

```
setInport(cm,'In1','Identifier','input1');
setInport(cm,'In2','Identifier','input2');
setInport(cm,'In3','Identifier','input3');
setInport(cm,'In4','Identifier','input4');
```

- 10 Verify your configuration changes for the inports by using calls to function `getInport`.

```
getInport(cm,'In1','Identifier')
```

```
ans =
```

```
    'input1'
```

```
getInport(cm,'In2','Identifier')
```



```

ans =
    'input2'

getInport(cm,'In3','Identifier')

ans =
    'input3'

getInport(cm,'In4','Identifier')

ans =
    'input4'

```

11 Save the model.

12 Generate and view the code. For example, in `rtwdemo_configrpinterface.c` find where variables `input1`, `input2`, `input3`, and `input4` are used in the step entry-point function.

```

void rtwdemo_configrpinterface_step(void)
{
    mode = ((input1 > rtwdemo_configrpinterface_UPPER) || (input1 <
        rtwdemo_configrpinterface_LOWER));

    dout_Table1 = look1_binlc((*input2), (&(mp_Table1.BP[0])),
        (&(mp_Table1.Table[0])), 10U);
    .
    .
    .
    dout_Table2 = look2_binlc((*input3), (*input4), (&(mp_Table2.BP1[0])),
        (&(mp_Table2.BP2[0])), (&(mp_Table2.Table[0])),
        (&(rtwdemo_configr_Table2_maxIndex[0])), 3U);

    dstate_X = dout_Table2;
}
}

```

Choose Storage Class and Storage Class Properties for Root-Level Inports

Depending on your code generation requirements, choose from these storage classes to configure code generation for root-level inports.

Requirements	Storage Class
Enable optimizations, potentially generating more efficient code.	“Auto” on page 15-77 (Individual mappings only)
For data elements that cannot be optimized, represent data as a field of a standard data structure.	“Default” on page 15-77 (Default mapping only)
Prevent optimizations from eliminating storage for a data element and use the default mapping for the data element category.	“Model Default” on page 15-77 (Individual mappings only) , Dictionary Default (Individual mappings only)
Generate a global variable definition and declaration.	“ExportedGlobal” on page 15-78

Requirements	Storage Class
Generate code that reads from and writes to a global variable or global variable pointer defined by your external code.	"ImportedExtern, ImportedExternPointer" on page 15-78

The list of available storage classes might include other project-specific storage classes defined in an Embedded Coder Dictionary. If you have special requirements that are not met by the listed storage classes and you have Embedded Coder software, you can define a storage class. See "Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture" (Embedded Coder).

For an individual inport, use the **Identifier** storage class property to configure a name for the variable representing the inport in the generated code. If you leave the **Identifier** property blank, the code generator uses the name of the signal attached to the Inport block. If the signal label is empty, the code generator uses the name of the Inport block.

See Also

Code Mappings Editor - C | `coder.mapping.api.CodeMapping`

More About

- "C Code Generation Configuration for Model Interface Elements" on page 16-2
- "Choose Storage Class for Controlling Data Representation in Generated Code" on page 15-75
- "Control Data and Function Interface in Generated Code" on page 15-44
- "Configure Model Data Elements for ASAP2 File Generation" on page 39-2

Configure Root-Level Output Blocks for C Code Generation

To use code that you generate from a model, you call generated entry-point functions. The calling environment and generated functions exchange input and output data, for example, as global variables or function arguments. Root-level Outputport blocks (outputports) constitute the output data of the interface. To integrate and deploy the generated code into larger applications, you can customize how the code generator produces the interface code, including how output data is declared and handled. Customizations can:

- Minimize the modifications that you make to existing code.
- Generate stable interfaces that do not change or minimally change when you make changes to your model.
- Generate code that exchanges data more efficiently (for example, by using pointers and pass-by-reference arguments for nonscalar data).

For code generation, examples show how to customize the outputport interface for the model `rtwdemo_configrpinterface`. You can configure code mappings by using the **Code Mappings Editor - C** or code mappings API (`coder.mapping.api.CodeMapping`).

Choose Customization Options for Outputport Interface

By default, root-level outputports in a model appear in generated code as fields of a global data structure named `model_ExtY`. Based on your code interface requirements, decide whether to customize generation of outputport data. If you do not configure customizations, the code generator determines whether to eliminate or change the representation of outputports in generated code for optimization purposes. If you configure customizations, decide:

- Whether to set up a default configuration

If a model includes a significant number (for example, more than 10) of root-level outputports, it is more efficient to configure the outputports with a default setting, and override that setting for special cases. If the model includes a few outputports that have unique source, naming, or placement requirements, consider configuring the outputports individually.

- How to declare and handle outputport data in the generated interface
 - As separate global variables
 - To write output data to global variables defined in external code
 - For referenced model outputports, as global variables (`void-void`)
 - As calls to access functions. Requires Embedded Coder
 - As entry-point function arguments. Requires Embedded Coder

For more information about these options, see “Control Data and Function Interface in Generated Code” on page 15-44.

Other considerations include whether to:

- Name outputports in the generated code by using Outputport block labels that appear in the model or by using unique code identifiers.
- Include the `volatile` type qualifier in global variable definitions and declarations. Requires Embedded Coder. See “Protect Global Data with `const` and `volatile` Type Qualifiers” (Embedded Coder).

- Write data from a specific area of memory. Requires Embedded Coder. See “Control Data and Function Placement in Memory by Inserting Pragmas” (Embedded Coder).

For a list of lists interface requirements that are relevant to outports with corresponding storage classes and storage class properties, see “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75.

Outport interface requirements for example model `rtwdemo_configrpinterface` are:

- Write data for root-level outport `Out1` to an external header file. The data written with `Out1` is the output of code generated for the Switch block.
- The variable representing the outport in the generated code must be named `output`.

For this example, configure the outport in `rtwdemo_configrpinterface` to meet these code generation requirements.

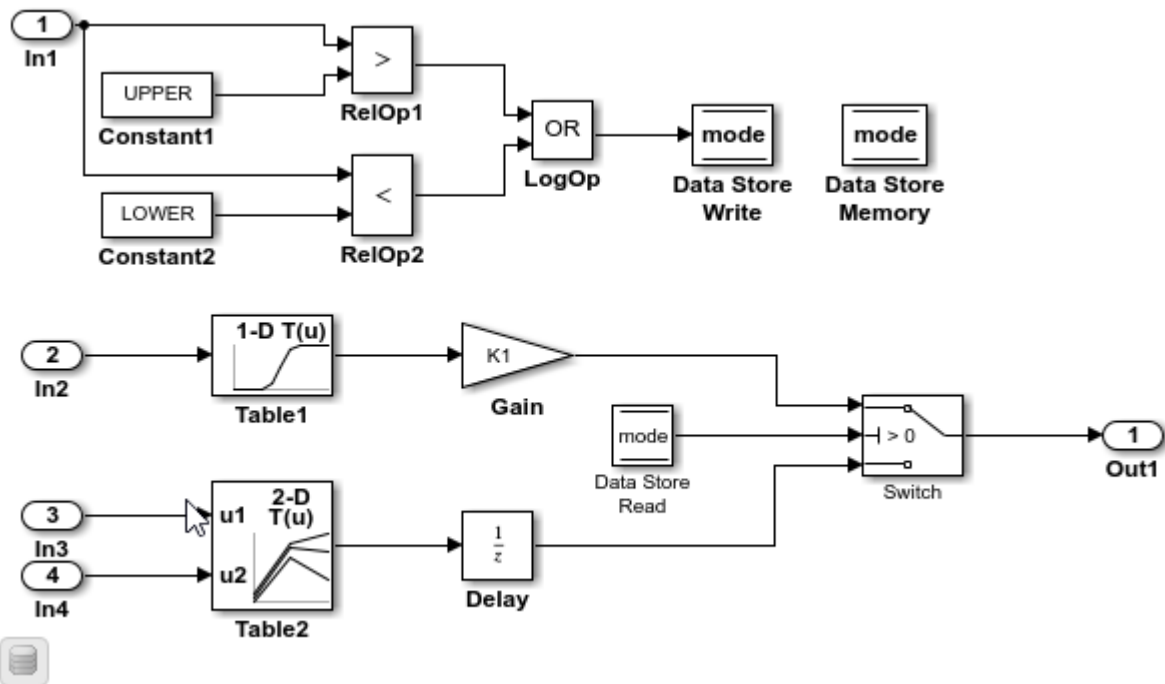
Configure Default Code Generation Settings for Root-Level Outports

A default code generation setting for root-level outports can reduce the effort of preparing a model for code generation, especially if a model has a significant number of outports. Choose configuration settings once, and the code generator applies those settings to outports across the model. Simulink stores the default configuration as part of the model.

Consider configuring default code generation settings for model outports if your model uses multiple root-level outports that do not have unique requirements.

The example model `rtwdemo_configrpinterface` includes one root-level outport. This example shows how to use the **Code Mappings Editor - C** to configure default settings for root-level outports. Specify that the code generator declare and define variables to represent root outports in generated external header and definition files.

- 1 Open example model `rtwdemo_configrpinterface`. Save a copy of the model to a writable location.



- 2 Open the **Simulink Coder** app.
- 3 In the **C Code** tab, select **Code Interface > Default Code Mappings**.
- 4 In the Code Mappings editor, under **Inports and Outports**, select category **Outports**. Set the default storage class to **ExportedGlobal**.

- 5 Save the model.

Configure Code Generation Settings for Individual Root-Level Outports

You can configure individual root-level outports for code generation. For example, if a model has two root-level outports that have unique code generation requirements, configure the outports individually. Or, if you configure default settings for outports, configure individual outports to use the default settings or unique settings.

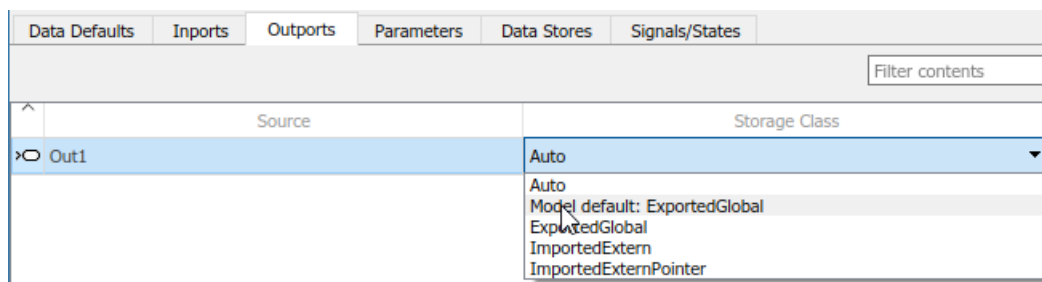
If your model meets at least one of these criteria, consider configuring code generation settings for outports individually:

- Uses multiple outports that have unique requirements.
- Uses few outports.
- Has a default configuration for outports and you need to override the configuration for some specific outports.


This example shows how to use the Code Mappings editor to apply the default code generation configuration for outports in model `rtwdemo_configrpinterface`. In the preceding example, you set the default storage class for outports to `ExportedGlobal`.

The example also shows how to configure an identifier that the code generator uses for naming the outport in the generated code. You can specify code generation identifiers, for example for integration, without modifying the model design.

- 1 If you have not already done so, complete the steps in “Configure Default Code Generation Settings for Root-Level Outports” on page 16-28.
- 2 In the Code Mappings editor, click the **Outports** tab. The editor lists the names of Outport blocks and bus elements that are in the model. If a port resolves to a signal object, a resolve-to-signal-object icon appears to the right of the element name. The storage class for the outport in the example model is set to `Auto`, which means that the code generator might eliminate or change the representation of relevant code for optimization purposes. If optimizations are not possible, the code generator applies the model default configuration. For this example, the model default configuration specifies storage class `ExportedGlobal`.
 - To avoid optimizations and force the code generator to use the default configuration, set the storage class to `Model default`.
 - To override the default configuration, specify the storage class that meets the code generation requirements for that outport.
- 3 Configure the code generator to apply the default storage class setting to outport `Out1`. Select the row for the outport. Set the storage class to `Model default: ExportedGlobal`.



The storage class for the selected outport changes to `Model default: ExportedGlobal`.

- 4 Configure the code identifier for the output so that the interface argument name in the generated code matches the interface name used by the external code. In the Code Mappings editor, select the row for the output. Click the  icon and set the **Identifier** property to output.
- 5 Save the model.
- 6 Generate and view the code. For example, in the file `rtwdemo_configrpinterface.c`, find where variable `output` is used in the step entry-point function.

```
if (mode) {
    output = (real_T)mp_K1 * dout_Table1;
} else {
    output = dstate_X;
}
```

Configure Code Generation Settings for Root-Level Outputs Programmatically

To automate configuration of root-level outputs for code generation, use the programming interface for code mappings. For example, when creating custom block libraries or part of an application test environment, use the programming interface to automate data configuration.

This example shows how to use the programming interface to configure default settings for root-level outputs for model `rtwdemo_configrpinterface`. The root-level output writes output to a separate global variable.

Configure code an identifier for the output so that global variable name in the generated code matches the variable name in the external code interface.

- 1 Open the example model.

```
open_system('rtwdemo_configrpinterface')
```

- 2 Create object `cm` by calling function `coder.mapping.api.get`. The object stores the code generation configuration for data elements in model `rtwdemo_configrpinterface`.

```
cm = coder.mapping.api.get('rtwdemo_configrpinterface');
```

- 3 Configure default settings for outputs by calling function `setDataDefault`. For the arguments, specify these values.

- The object returned by `coder.mapping.api.get`
- Outputs for the default category
- Property name `StorageClass` that has property value `ExportedGlobal`

```
setDataDefault(cm, 'Outputs', 'StorageClass', 'ExportedGlobal')
```

- 4 Verify your default configuration for outputs. Issue a call to `getDataDefault` that specifies the object returned by `coder.mapping.api.get`, category `Outputs`, and `StorageClass`.

```
getDataDefault(cm, 'Outputs', 'StorageClass')
```

```
ans =
```

```
    'ExportedGlobal'
```

- 5 Apply the default output configuration for output `Out1`.

By default, Simulink sets the storage class for individual outputs to `Auto`. When the storage class is `Auto`, the code generator:

- Determines whether to eliminate the data from the generated code for optimization purposes.
- If retaining the data, determines how to efficiently represent the data in the generated code, taking into account default configuration settings.

To control the configuration for an output, call function `setOutputport`.

Issue a call to `setOutputport` that specifies:

- Object returned by `coder.mapping.api.get`
- Output block name `Out1`
- Default storage class previously set for the output by using property `StorageClass` and property value `Model default`
- Property `Identifier` and property value `output`

```
setOutputport(cm,'Out1','StorageClass','Model default','Identifier','output');
```

- 6 Verify your configuration changes by calling function `getOutputport`. Specify the object returned by `coder.mapping.api.get`, name of the output block, and property `StorageClass` or `Identifier`.

```
getOutputport(cm,'Out1','StorageClass')
```

```
ans =
```

```
    'Model default'
```

```
getOutputport(cm,'Out1','Identifier')
```

```
ans =
```

```
    'output'
```

- 7 Save the model.

- 8 Generate and view the code. For example, in the file `rtwdemo_configrpininterface.c`, find where variable `output` is used in the step entry-point function.

```
if (mode) {
    output = (real_T)mp_K1 * dout_Table1;
} else {
    output = dstate_X;
}
```

Choose Storage Class and Storage Class Properties for Root-Level Outputs

Depending on your code generation requirements, choose from these storage classes to configure code generation for root-level outputs.

Requirements	Storage Class
Enable optimizations, potentially generating more efficient code.	"Auto" on page 15-77 (Individual mappings only)

Requirements	Storage Class
For data elements that cannot be optimized, represent data as a field of a standard data structure.	“Default” on page 15-77 (Default mapping only)
Prevent optimizations from eliminating storage for a data element and use the default mapping for the data element category.	“Model Default” on page 15-77 (Individual mappings only) , Dictionary Default (Individual mappings only)
Generate a global variable definition and declaration.	“ExportedGlobal” on page 15-78
Generate code that reads from and writes to a global variable or global variable pointer defined by your external code.	“ImportedExtern, ImportedExternPointer” on page 15-78

The list of available storage classes might include other project-specific storage classes defined in an Embedded Coder Dictionary. If you have special requirements that are not met by the listed storage classes and you have Embedded Coder software, you can define a storage class. See “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder).

For an individual outport, use the **Identifier** storage class property to configure a name for the variable representing the outport in the generated code. If you leave the **Identifier** property blank, the code generator uses the **Signal name** parameter of the Outport block. If this parameter is empty, the code generator uses the name of the Outport block.

See Also

Code Mappings Editor - C | `coder.mapping.api.CodeMapping`

More About

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75
- “Control Data and Function Interface in Generated Code” on page 15-44
- “Configure Model Data Elements for ASAP2 File Generation” on page 39-2

Configure Signal Data for C Code Generation

Signal data consists of variables that store intermediate results in global memory as generated algorithmic code computes values of block output signals from block input signals, excluding signals that connect to root Inport and Outport blocks. Signal data is not signal lines in a model diagram. For example, in these cases, a signal line has no data representation in generated code:

- Feeds a function-call port
- Exits a message block, such as the Send block
- Crosses subsystem boundaries with no computation blocks in between

When you configure a model for code generation, you choose the block input and output signals to make accessible in the generated code. For example, configure signal data to:

- Make the data accessible for interaction and monitoring while the generated code executes.
- Minimize the amount of data that is stored in memory.
- Control where the code generator places signal data in memory.
- Promote signal data to the model interface so that other components and systems can access that data.
- Improve readability and traceability of the generated code.

The code generator checks for signal line configuration consistency. For example, the code generator verifies that the configuration of a signal line feeding into a subsystem matches the configuration of the signal line that exits the corresponding subsystem inport.

For code generation, examples show how to configure block signals for the model `rtwdemo_configrpinterface`. You can configure code mappings by using the **Code Mappings Editor - C** or code mappings programming interface (`coder.mapping.api.CodeMapping`).

Choose Code Configuration Options for Signals

Based on your code generation requirements, decide which block signals to represent in the generated code and how to represent the signal data. By default, signals in a model appear in generated code as fields of a global data structure named `model_B`. If you do not configure customizations, the code generator determines whether to eliminate or change the representation of signals in generated code for optimization purposes. If you configure customizations, decide:

- Which signals to make accessible in the generated code

You must add signals that you want to make accessible in the generated code to the model code mappings.

- Whether to set up a default configuration

If you need to gain access to a significant number (for example, more than 10) of signals, it is more efficient to configure the signals with default settings and then override those settings for special cases. If you need to gain access to a few signals that have unique source, naming, or placement requirements, consider configuring the signals individually.

- How to declare and handle signal data in the generated code
 - As separate global variables

- To read input data from global variables defined in external code
- As calls to access functions. Requires Embedded Coder

For more information about these options, see “Control Data and Function Interface in Generated Code” on page 15-44.

Other considerations include whether to:

- Name signals in the generated code by using signal labels that appear in the model or by using unique code identifiers.
- Include the `volatile` type qualifier in global variable definitions and declarations. Requires Embedded Coder. See “Protect Global Data with `const` and `volatile` Type Qualifiers” (Embedded Coder).
- Place signal data into a specific area of memory. Requires Embedded Coder. See “Control Data and Function Placement in Memory by Inserting Pragmas” (Embedded Coder).

For a list of interface requirements that are relevant to signals with corresponding storage classes and storage class properties, see “Choose Storage Class and Storage Class Properties for Data Stores” on page 16-66.

Signal requirements for example model `rtwdemo_configrpinterface` are:

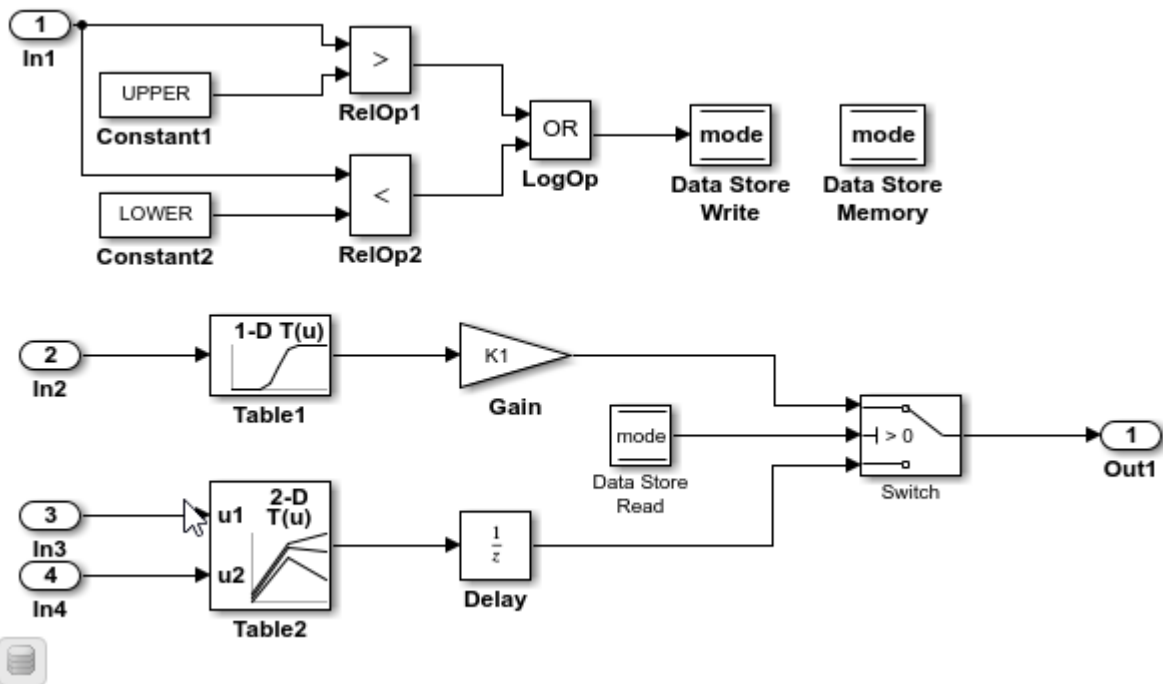
- Configure signals that feed into and exit from the Switch block for code generation. Retain that signal data for monitoring while the generated code executes.
- Represent signals as separate global variables.
- Apply prefix `dout_` to names of variables that represent signals.

Set the default representation of signals in the generated code as global variables that have the `static` type qualifier. Then, you configure the output signals of the two lookup table blocks to use the default storage class and unique code identifiers that include the required prefix `dout_`.

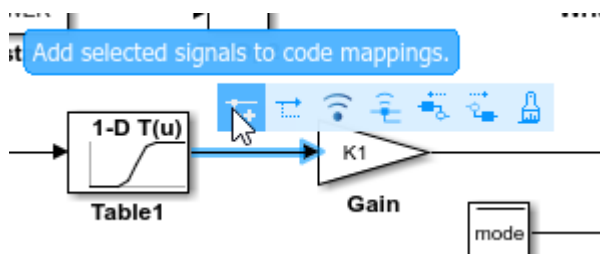
Add Signals to Model Code Mappings

Before you can configure a signal for code generation, add the signal to the model code mappings. Add the output signals for the two lookup table blocks to the model code mappings.

- 1 Open example model `rtwdemo_configrpinterface`. Save a copy of the model to a writable location.



- 2 Open the **Simulink Coder** app.
- 3 In the **C Code** tab, select **Code Interface > Individual Element Code Mappings**.
- 4 In the Code Mappings editor, click the **Signals/States** tab. No signals are listed.
- 5 Add signals to the code mappings. For the output signal of lookup table blocks Table1 and Table2:
 - a In the model, select the signal.
 - b Pause on the ellipsis that appears above or below the signal line to open the action bar. Click the **Add Signal** button.



In the Code Mappings editor, the **Signals** node expands and lists the two signals that you added.

Data Defaults			Inports	Outports	Parameters	Data Stores	Signals/States
Source		Storage Class		Path			
Filter contents							
Signals (2) <ul style="list-style-type: none"> Table1:1 Auto rtwdemo_configrpinterface Table2:1 Auto rtwdemo_configrpinterface 							
States (1)							

Configure Default Code Generation Settings for Signals

A default code generation setting for signals can reduce the effort of preparing a model for code generation, especially if a model has a significant number of signals that you want to monitor while the generated code executes. Choose configuration settings once, and the code generator applies those settings to signals across the model. Simulink stores the default configuration as part of the model.

Consider configuring default code generation settings for model signals if your model uses multiple signals that do not have unique requirements.

This example shows how to use the **Code Mappings Editor - C** to set the default storage class for signals in the model `rtwdemo_configrpinterface` to `ExportedGlobal`. With that storage class setting, the code generator represents signal data in the generated code as global variables.

- 1 If you have not already done so, add signals to the model code mappings as described in “Add Signals to Model Code Mappings” on page 16-35.
- 2 In the **C Code** tab, select **Code Interface > Default Code Mappings**.
- 3 In the Code Mappings editor, on the **Data Defaults** tab, under **Signals**, select category **Signals, states, and internal data**. Set the default storage class to `ExportedGlobal`.

Data Defaults		Inports	Outports	Parameters	Data Stores	Signals/States
Model Element Category		Storage Class				
Filter contents						
Inports and Outports						
Signals						
<input checked="" type="checkbox"/> Signals, states, and internal data		Default				
<input type="checkbox"/> Shared local data stores		Default				
<input type="checkbox"/> Global data stores		ExportedGlobal				
		ImportedExtern				
		ImportedExternPointer				
Parameters						

- 4 Save the model.


Configure Code Generation Settings for Individual Signals

You can configure individual signals for code generation. For example, if a model has two signals that have unique code generation requirements, configure the signals individually. Or, if you configure default settings for signals, you can override those settings for specific signals.

If your model meets at least one of these criteria, consider configuring code generation settings for signals individually:

- Uses multiple signals that have unique source, naming, or placement requirements.
- Uses a few signals.
- Has a default configuration for signals and you need to override the configuration for some specific signals.

This example shows how to use the Code Mappings editor to apply your default storage class setting to the output signals of lookup table blocks `Table1` and `Table2` in model `rtwdemo_configinterface`. The example also shows how to configure code identifiers for those output signals. You can specify code generation identifiers, for example for integration, without modifying the model design.

- 1 If you have not already done so, complete the steps in “Configure Default Code Generation Settings for Signals” on page 16-37.
- 2 In the Code Mappings editor, click the **Signals/States** tab. Expand **Signals**. The editor lists the names or block port identifiers of signals that you added to the code mappings. If a signal resolves to a signal object, a resolve-to-signal-object icon appears to the right of the element name or port identifier. The storage class for each signal is set to `Auto`, which means that the code generator might eliminate or change the representation of relevant code for optimization purposes. If optimizations are not possible, the code generator applies the model default configuration. For this example, the model default configuration specifies storage class `ExportedGlobal`.
 - To avoid optimizations and force the code generator to use the default configuration, set the storage class to `Model default`.
 - To override the default configuration, specify the storage class that meets the code generation requirements for that signal.
- 3 In the Code Mappings editor, select the output signals for blocks `Table1` and `Table2`. Set the storage class to `Model default: ExportedGlobal`.
- 4 Configure the code identifier for the output signals for the two lookup table blocks with names that include the prefix `dout_`. In the Code Mappings editor, select signal `Table1:1`. Click the  icon and set the **Identifier** property to `dout_Table1`. For signal `Table2:1`, set **Identifier** to `dout_Table2`.

The screenshot displays the Simulink environment for the model 'rtwdemo_configrinterface'. The main workspace shows a block diagram with inputs In1, In2, In3, and In4. In1 is compared against 'UPPER Constant1' and 'LOWER Constant2' using 'RelOp1' and 'RelOp2' blocks. The results are combined in an 'OR' block, followed by a 'LogOp' block. In2 is processed by a '1-D T(u)' block (Table1), which is highlighted with a red box. The output of Table1 is multiplied by a 'Gain' block (K1). In3 and In4 are processed by a '2-D T(u)' block (Table2), followed by a 'Delay' block. The outputs of the Gain and Delay blocks are read from 'Data Store Read' blocks. A 'Switch' block selects between the Gain output and the Delay output based on a 'mode' signal. The final output is 'Out1'. Below the workspace, the 'Code Mappings - Component Interface' window is open, showing a table with columns for Source, Storage Class, and Path. The table lists 'Table1:1' and 'Table2:1' under the 'Signals (2)' category, both with a storage class of 'Model default: ExportedGlobal' and a path of 'rtwdemo_configrinterface'. A 'States (1)' category lists 'X' with the same storage class and path. To the right, the 'Property Inspector' window is open, showing the 'Measurement' settings for 'dout_Table1', including 'Calibration Access' set to 'NoCalibration', 'Display Identifier', 'Format', and 'Compu Method' fields, and an 'Open in Property Inspector' button.

Source	Storage Class	Path
Table1:1	Model default: ExportedGlobal	rtwdemo_configrinterface
Table2:1	Model default: ExportedGlobal	rtwdemo_configrinterface
States (1)		
X	Model default: ExportedGlobal	rtwdemo_configrinterface

- 5 Save the model.
- 6 Generate and view the code. For example, in `rtwdemo_configrinterface.c`, find the data definitions for the lookup table data.

```
real_T dout_Table1;
real_T dout_Table2;
```

Find where lookup table data is used in the step entry-point function.

```
.
.
.
dout_Table1 = look1_binlc((*input2), (&(mp_Table1.BP[0])),
    (&(mp_Table1.Table[0])), 10U);

if (mode) {
    output = (real_T)mp_K1 * dout_Table1;
} else {
    output = dstate_X;
}

dout_Table2 = look2_binlc((*input3), (*input4), (&(mp_Table2.BP1[0])),
    (&(mp_Table2.BP2[0])), (&(mp_Table2.Table[0])),
    (&(rtwdemo_configr_Table2_maxIndex[0])), 3U);}

dstate_X = dout_Table2;
}
```

Remove Signals from Model Code Mappings

- 1 In the **C Code** tab, select **Code Interface > Individual Element Code Mappings**.
- 2 In the Code Mappings editor, click the **Signals/States** tab.
- 3 For each signal that you want to remove:
 - a In the model, select the signal.
 - b Pause on the ellipsis that appears above or below the signal line to open the action bar. Click the **Remove Signal** button.

Configure Code Generation Settings for Signals Programmatically

To automate configuration of signals for code generation, use the programming interface for code mappings. For example, when creating custom block libraries or part of an application test environment, use the programming interface to automate data configuration.

This example shows how to use the programming interface to configure signals for model `rtwdemo_configrpinterface`. Set the default representation of signals in the generated code as global variables. Then, configure the output signals of the two lookup table blocks to use the default storage class and unique code identifiers that include the required prefix `dout_`.

- 1 Open the example model.

```
open_system('rtwdemo_configrpinterface')
```

- 2 Create object `cm` by calling function `coder.mapping.api.get`. The object stores the code generation configuration for data for model `rtwdemo_configrpinterface`.

```
cm = coder.mapping.api.get('rtwdemo_configrpinterface');
```

- 3 Configure default settings for signals by calling function `setDataDefault`. For the arguments, specify these values:

- The object returned by `coder.mapping.api.get`
- `InternalData` for the default category
- Property name `StorageClass` with property value `ExportedGlobal`

```
setDataDefault(cm, 'InternalData', 'StorageClass', 'ExportedGlobal');
```

- 4 Verify your default configuration for signals. Issue a call to `getDataDefault` that specifies the object returned by `coder.mapping.api.get` and category `InternalData`. Specify the third argument as property `StorageClass`.

```
getDataDefault(cm, 'InternalData', 'StorageClass')
```

```
ans =
```

```
    'ExportedGlobal'
```

- 5 Get handles to the output ports for lookup table blocks `Table1` and `Table2`. You use the port handles to add the signal data to the model code mappings.

```
lut1D_ports = get_param('rtwdemo_configrpinterface/Table1', 'PortHandles');
lut2D_ports = get_param('rtwdemo_configrpinterface/Table2', 'PortHandles');
```

```
lut1D_outPort = lut1D_ports.Outport;
lut2D_outPort = lut2D_ports.Outport;
```


- 6** Add the output signals of the lookup table blocks to the model code mappings.

```
addSignal(cm,[lut1D_outPort,lut2D_outPort]);
```

- 7** Apply the default configuration for signals to the output signal of the lookup table blocks.

By default, Simulink sets the storage class for individual signals to `Auto`. The code generator:

- Determines whether to eliminate the data from the generated code for optimization purposes.
- If retaining the data, determines how to efficiently represent the data in the generated code, taking into account default configuration settings.

To control the configuration for a signal, call function `setSignal`.

For each signal, issue a call to `setSignal` that specifies:

- Object returned by `coder.mapping.api.get`
- Port handle for the signal, `lut1D_outport` or `lut2D_outport`.
- Default storage class previously set for signals by using property `StorageClass` and property value `Model default`.
- Property `Identifier` and property value `dout_Table1` or `dout_Table2`

```
setSignal(cm,lut1D_outPort,'StorageClass','Model default','Identifier','dout_Table1');
setSignal(cm,lut2D_outPort,'StorageClass','Model default','Identifier','dout_Table2');
```

- 8** Verify your configuration settings by calling function `getSignal`. Specify the object returned by `coder.mapping.api.get`, the port handle for the signal (`lut1D_outport` or `lut2D_outport`), and property `StorageClass` or `Identifier`.

```
getSignal(cm,lut1D_outPort,'StorageClass')
```

```
ans =
```

```
    'Model default'
```

```
getSignal(cm,lut1D_outPort,'Identifier')
```

```
ans =
```

```
    'dout_Table1'
```

```
getSignal(cm,lut2D_outPort,'StorageClass')
```

```
ans =
```

```
    'Model default'
```

```
getSignal(cm,lut2D_outPort,'Identifier')
```

```
ans =
```

```
    'dout_Table2'
```

- 9** Save the model.

- 10** Generate and view the code. For example, in `rtwdemo_configrpininterface.c`, find the data definitions for the lookup table data.

```
real_T dout_Table1;
real_T dout_Table2;
```

Find where lookup table data is used in the step entry-point function.

```
.
.
.
    dout_Table1 = look1_binlc((*input2), (&(mp_Table1.BP[0])),
        (&(mp_Table1.Table[0])), 10U);

    if (mode) {
        output = (real_T)mp_K1 * dout_Table1;
    } else {
        output = dstate_X;
    }

    dout_Table2 = look2_binlc((*input3), (*input4), (&(mp_Table2.BP1[0])),
        (&(mp_Table2.BP2[0])), (&(mp_Table2.Table[0])),
        (&(rtwdemo_configr_Table2_maxIndex[0])), 3U);}

    dstate_X = dout_Table2;
}
```

Choose Storage Class and Storage Class Properties for Signals

Depending on your code generation requirements, choose from these storage classes to configure code generation for block signals.

Requirements	Storage Class to Choose
Enable optimizations, potentially generating more efficient code.	“Auto” on page 15-77 (Individual mappings only)
For data elements that cannot be optimized, represent data as a field of a standard data structure.	“Default” on page 15-77 (Default mapping only)
Prevent optimizations from eliminating storage for a data element and use the default storage class for the data element category.	“Model Default” on page 15-77 (Individual mappings only) , Dictionary Default (Individual mappings only)
Generate a global variable definition and declaration.	“ExportedGlobal” on page 15-78
Generate code that reads from and writes to a global variable or global variable pointer defined by your external code.	“ImportedExtern, ImportedExternPointer” on page 15-78

The list of available storage classes might include other project-specific storage classes defined in an Embedded Coder Dictionary. If you have special requirements that are not met by the listed storage classes and you have Embedded Coder software, you can define a storage class. See “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder).

For an individual signal, use the **Identifier** storage class property to configure a name for the variable representing the signal in the generated code. If you leave the **Identifier** property blank, the code generator uses the signal label. If the signal label is empty, the code generator uses the name of the block that outputs the signal.

See Also

Code Mappings Editor - C | `coder.mapping.api.CodeMapping`

More About

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75
- “Configure Model Data Elements for ASAP2 File Generation” on page 39-2

Configure Parameters for C Code Generation

When configuring a model for code generation, you can identify and configure parameters for tunability, for example for calibration. Types of parameters that you can configure are listed in this table.

Type of Parameter	Description
Model parameter	Parameter defined within a model, such as a parameter in the model workspace.
External parameter	Parameter defined as an object in the base workspace or in a data dictionary.

To configure a parameter for code generation, you must associate the parameter with a data object. For example, before configuring a MATLAB variable for code generation, in the Model Explorer, convert the variable to a parameter object.

When you open a model in the **Simulink Coder** app, model parameters that are associated with data objects appear in the Code Mappings editor. In the editor, you can configure the parameters for code generation. If a model uses external parameters, you can add them to the Code Mappings editor by clicking the **Refresh** link to the right of the parameter name. That link initiates an update diagram and, in the editor view, adds external parameters that are used by the model.

Configure parameter data to:

- Make the data accessible for interaction while the generated code executes.
- Minimize the amount of data that is stored in memory.
- Promote parameter data to the model interface so that other components and systems can access that data.
- Improve readability and traceability of the generated code.

To control whether the parameters throughout a model are tunable, you can gain access to model configuration parameter **Default parameter behavior** by using the 'Auto' will be tunable/inline link on the Code Mappings editor **Data Defaults** tab. For example, you can use that link to make parameters tunable in these cases:

- Refine parameter settings during rapid prototyping
- Calibrate parameters
- Optimize parameters for production code

For code generation, examples show how to configure model parameters and model parameter arguments for the model `rtwdemo_configrpinterface`. You can configure code mappings by using the **Code Mappings Editor - C** or code mappings programming interface (`coder.mapping.api.CodeMapping`).

Choose Customization Options for Parameters

By default, parameters in a model appear in generated code as fields of a global data structure named `model_P`. Based on your code interface requirements, decide whether to customize generation of parameter data. If you do not configure customizations, the code generator determines whether to eliminate or change the representation of parameters in generated code for optimization purposes. If you configure customizations, decide:

- Whether to set up a default configuration

If a model includes a significant number of parameters of a category that must be tunable (for example, more than 10), it is more efficient to configure the parameters of that category by using a default setting, and then override that setting for special cases. If the model includes a few parameters of a given category that have unique source, naming, or placement requirements, consider configuring the parameters individually.

- How to declare and handle model parameter data in the generated code
 - As separate global variables
 - To read model parameter data from global variables defined in external code
 - As calls to access functions. Requires Embedded Coder

For more information about these options, see “Control Data and Function Interface in Generated Code” on page 15-44.

Other considerations for model parameters include whether to:

- Name parameters in the generated code by using parameter names in the model or by using unique code identifiers.
- Support preprocessor conditionals defined by compiler flags or options. Requires Embedded Coder. See “Use Variant Models to Generate Code That Uses C Preprocessor Conditionals” (Embedded Coder).
- Include the `static` type qualifier in global variable definitions and declarations, for example, to prevent name clashes. Requires Embedded Coder. See “Prevent Name Clashes by Configuring Data Item as static” (Embedded Coder).
- Include the `const`, `volatile`, or `const` and `volatile` type qualifier in global variable definitions and declarations. Requires Embedded Coder. See “Protect Global Data with `const` and `volatile` Type Qualifiers” (Embedded Coder).
- Generate a macro (`#define`) or code that uses a macro defined in an external header file. Requires Embedded Coder. See “Macro Definitions (`#define`)” (Embedded Coder).
- Generate a global data structure with a name that you specify. Requires Embedded Coder. See “Organize Data into Structures in Generated Code” (Embedded Coder).
- Place parameter data into a specific area of memory. Requires Embedded Coder. See “Control Data and Function Placement in Memory by Inserting Pragmas” (Embedded Coder).

For a list of interface requirements that are relevant to parameters that have corresponding storage classes and storage class properties, see “Choose Storage Class and Storage Class Properties for Data Stores” on page 16-66.

Parameter requirements for example model `rtwdemo_configrpinterface` are:

- By default, retain model parameters in the generated code for tuning. Do not optimize the code by inlining the parameters.
- Apply prefix `mp_` to names of variables that represent model parameters.

For this example, configure model parameters in `rtwdemo_configrpinterface` to meet these code generation requirements.

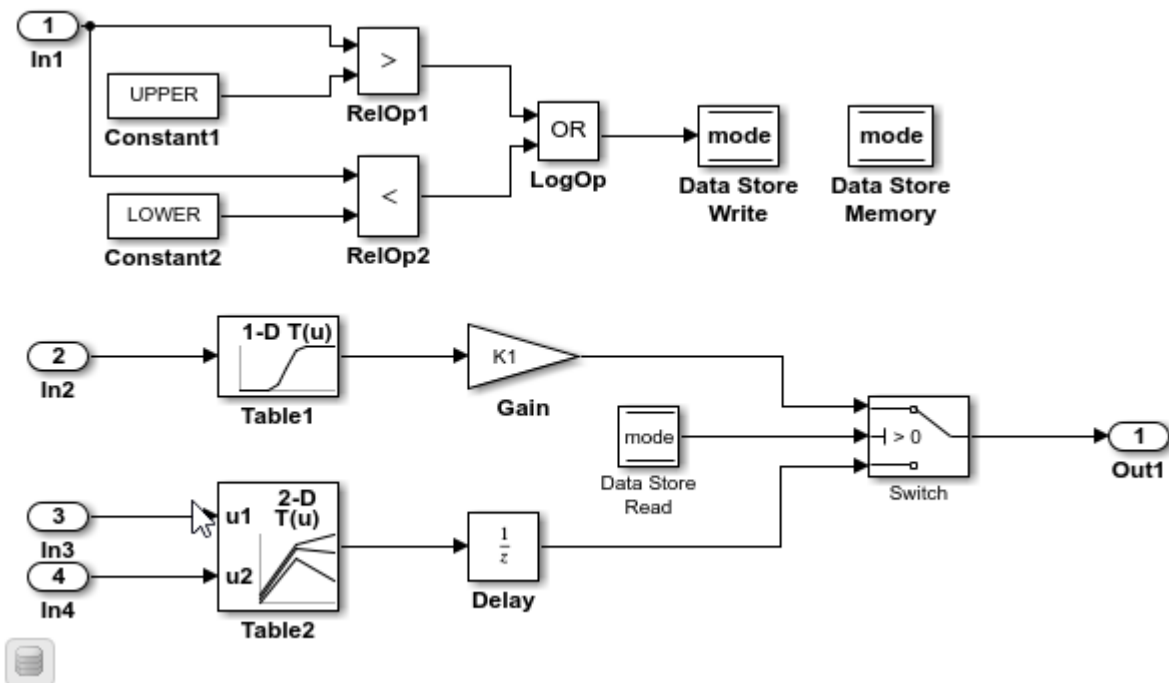
Configure Default Code Generation Settings for Parameters

Default code generation settings for parameters can reduce the effort of preparing a model for code generation, especially if a model has a significant number of parameters with which you need to interact while the generated code executes. Choose configuration settings once, and the code generator applies those settings to parameters across the model. Simulink stores the default configuration as part of the model.

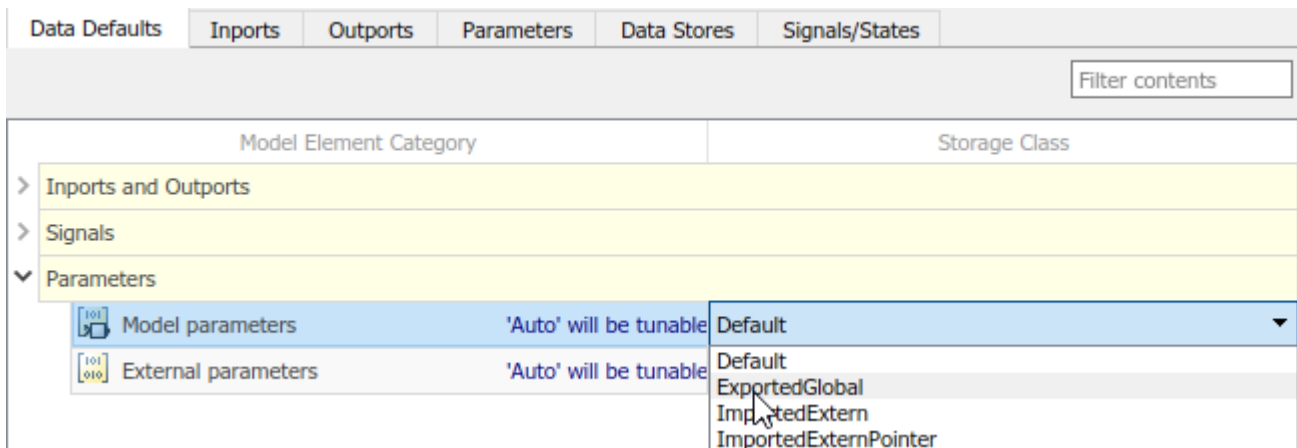
Consider configuring default code generation settings for model parameters if your model uses multiple parameters for the same category that do not have unique requirements.

This example shows how to use the **Code Mappings Editor - C** to configure default settings for model parameters for the model `rtwdemo_configrpinterface`. Configure model parameters so that they are tunable and are defined and declared in the generated code as separate global variables.

- 1 Open model `rtwdemo_configrpinterface`. Save a copy of the model to a writable location.



- 2 Open the **Simulink Coder** app.
- 3 In the **C Code** tab, select **Code Interface > Default Code Mappings**.
- 4 In the Code Mappings editor, under **Parameters**, select category **Model parameters**. Link text 'Auto' will be inlined indicates that the code generator is configured to inline model parameters by default. A requirement for this example is model parameters be tunable. Click 'Auto' will be inlined.
- 5 In the Model Configuration Parameters dialog box, set model configuration parameter **Default parameter behavior** to Tunable. Save the change and close the dialog box. In the Code Mappings editor, the link text changes to 'Auto' will be tunable.
- 6 In the Code Mappings, with the **Model parameters** category still selected, set the storage class to ExportedGlobal.



7 Save the model.

Configure Code Generation for Individual Parameters

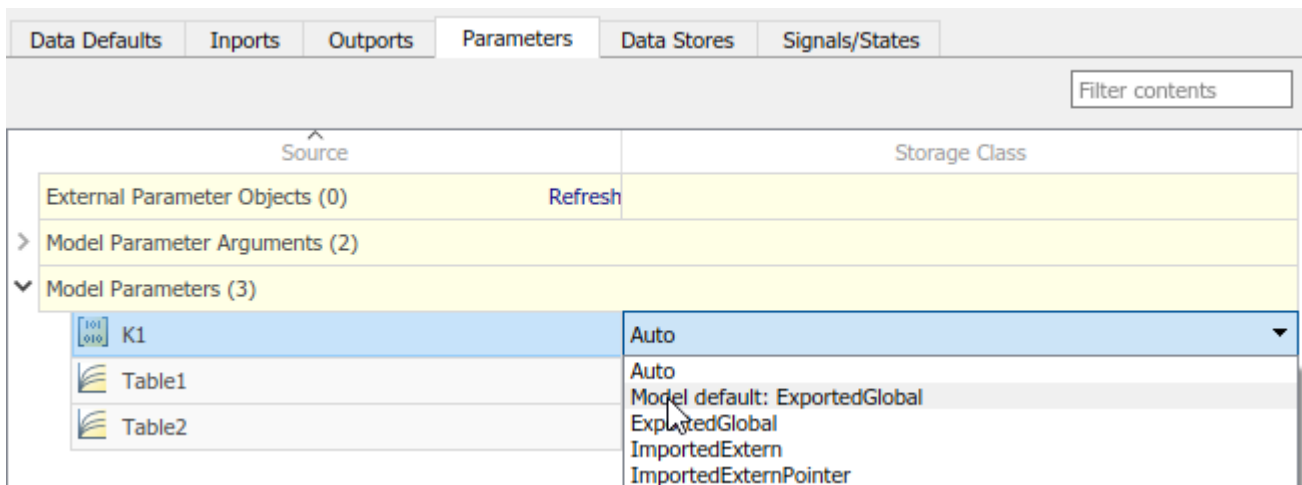
You can configure individual parameters for code generation. For example, if a model has two parameters of the same category that have unique code generation requirements, configure the parameters individually. Or, if you configure default settings for a category of parameters, you can override those settings for specific parameters.

If your model meets at least one of these criteria, consider configuring code generation settings for parameters individually:

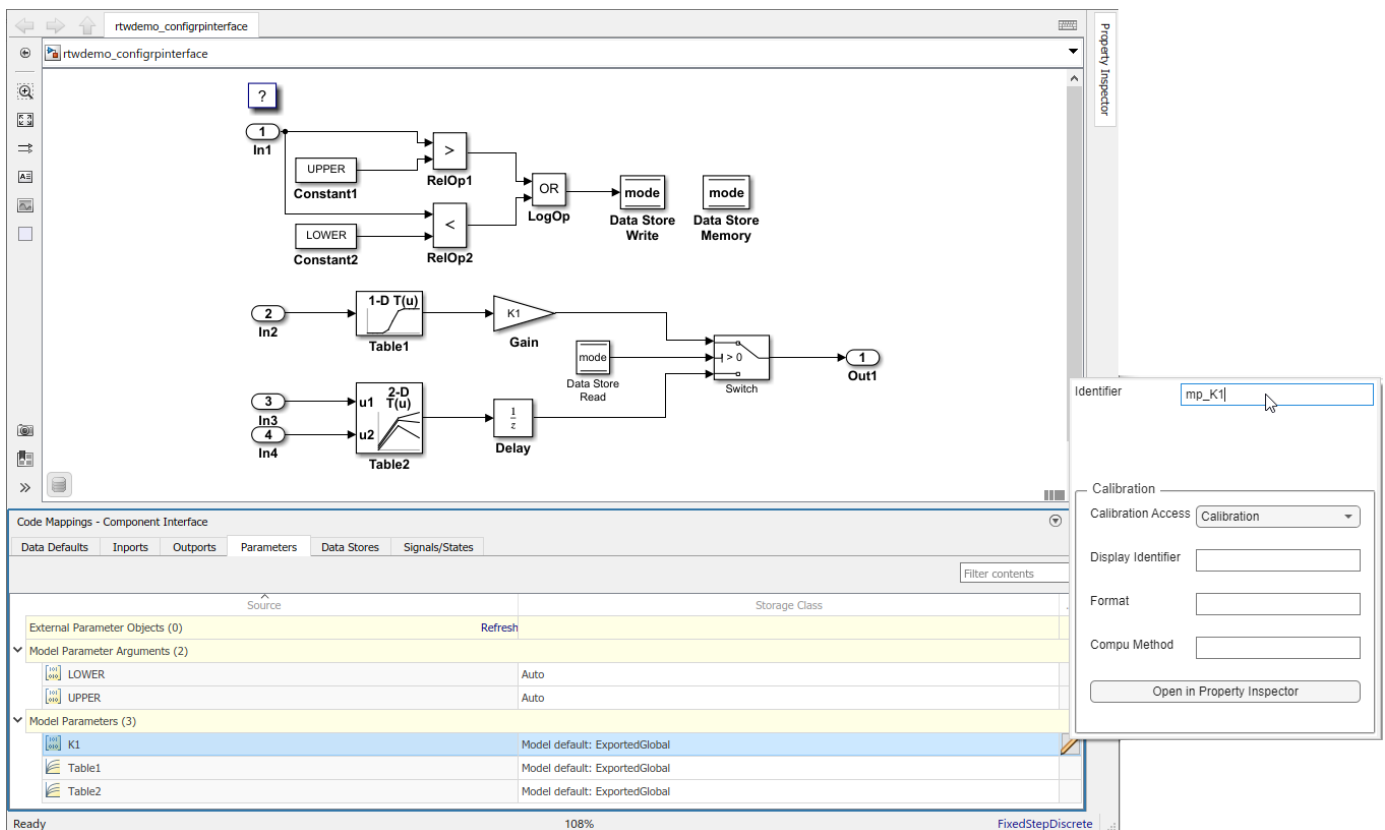
- Uses multiple parameters of the same category that have unique source, naming, or placement requirements.
- Uses a few parameters of the same category.
- Has a default configuration for a category of parameters and you need to override the configuration for some specific parameters.

This example shows how to use the Code Mappings editor to apply your default storage class setting for model parameters to parameters `K1`, `Table1`, and `Table2` in model `rtwdemo_configinterface`. You configure code identifiers for those parameters. You can specify code generation identifiers, for example for integration, without modifying the model design.

- 1 If you have not already done so, complete the steps in “Configure Default Code Generation Settings for Parameters” on page 16-46.
- 2 Expand **Model Parameters**. By default, the storage class for each parameter is set to `Auto`. Use the model default configuration, which specifies storage class `ExportedGlobal`.
 - To avoid optimizations and force the code generator to use the default configuration, set the storage class to `Model default`.
 - To override the default configuration, specify the storage class that meets the code generation requirements for that parameter.
- 3 In the Code Mappings editor, under **Model Parameters**, select parameters `K1`, `Table1`, and `Table2`. Set the storage class to `Model default: ExportedGlobal`.



- Configure the code identifiers for model parameters with names that include the prefix `mp_`. In the Code Mappings editor, select model parameter `K1`. Click the icon and set the storage class property **Identifier** to `mp_K1`. For parameters `Table1` and `Table2`, set **Identifier** to `mp_Table1` and `mp_Table2`.



- Save the model.
- Generate and view the code. For example, find the data definitions for model parameter `mp_K1` in `rtwdemo_configrinterface.c`.


```
int8_T mp_K1 = 2;
```


Find where the parameter is used in the step entry-point function.

```
if (mode) {
    output = (real_T)mp_K1 * dout_Table1;
} else {
    output = dstate_X;
}
```

Configure Code Generation Settings for Parameters Programmatically

To automate configuration of parameters for code generation, use the programming interface for code mappings. For example, when creating custom block libraries or part of an application test environment, use the programming interface to automate data configuration.

This example shows how to configure default settings for model parameters for model `rtwdemo_configrpinterface`. Configure model parameters so that they are tunable and are defined and declared in the generated code as separate global variables. Apply your default storage class setting for model parameters to parameters `K1`, `Table1`, and `Table2`. Configure code identifiers for those parameters.

- 1 Open the example model.

```
open_system('rtwdemo_configrpinterface')
```

- 2 Set model configuration parameter **Default parameter behavior** to Tunable.

```
model='rtwdemo_configrpinterface';
set_param(model,'DefaultParameterBehavior','Tunable');
```

- 3 Create object `cm` by calling function `coder.mapping.api.get`. The object stores the code generation configuration for data elements in model `rtwdemo_configrpdefaults`.

```
cm = coder.mapping.api.get('rtwdemo_configrpinterface');
```

- 4 Configure default settings for model parameters by calling function `setDataDefault`. For the arguments, specify these values:

- The object returned by `coder.mapping.api.get`
- `ModelParameters` for the default category
- Property name `StorageClass` with property value `ExportedGlobal`

```
setDataDefault(cm,'ModelParameters','StorageClass','ExportedGlobal');
```

- 5 Verify your default configuration for model parameters. Issue calls to `getDataDefault` that specify the object returned by `coder.mapping.api.get`, category `ModelParameters`, and `StorageClass`.

```
getDataDefault(cm,'ModelParameters','StorageClass')
```

```
ans =
```

```
    'ExportedGlobal'
```

- 6 Apply the default configuration for model parameters to parameters `K1`, `Table1`, and `Table2`.

By default, Simulink sets the storage class for individual parameters to `Auto`. When the storage class is `Auto`, the code generator:

- Determines whether to eliminate the data from the generated code for optimization purposes.

- If retaining the data, determines how to efficiently represent the data in the generated code, taking into account default configuration settings.

Configure the code generator to apply your default model parameter settings to parameters K1, Table1, and Table2. For each parameter, call function `setModelParameter`. Specify the object returned by `coder.mapping.api.get`, the parameter name, property name `StorageClass`, and property value `Model default`.

```
setModelParameter(cm,'K1','StorageClass','Model default');
setModelParameter(cm,'Table1','StorageClass','Model default');
setModelParameter(cm,'Table2','StorageClass','Model default');
```

- 7 Verify your configuration changes for parameters K1, Table1, and Table2 by using calls to function `getModelParameter`.

```
getModelParameter(cm,'K1','StorageClass')

ans =

    'Model default'

getModelParameter(cm,'Table1','StorageClass')

ans =

    'Model default'

getModelParameter(cm,'Table2','StorageClass')

ans =

    'Model default'
```

- 8 Configure code identifiers for the model parameters. For each parameter, call function `setModelParameter`. Specify the object returned by `coder.mapping.api.get`, the parameter name, property name `Identifier`, and one of these property values.

Model Parameter	Code Identifier
K1	mp_K1
Table1	mp_Table1
Table2	mp_Table2

```
setModelParameter(cm,'K1','Identifier','mp_K1');
setModelParameter(cm,'Table1','Identifier','mp_Table1');
setModelParameter(cm,'Table2','Identifier','mp_Table2');
```

- 9 Verify your configuration changes for the model parameters by using calls to function `getModelParameter`.

```
getModelParameter(cm,'K1','Identifier')

ans =

    'mp_K1'

getModelParameter(cm,'Table1','Identifier')
```

```

ans =

    'mp_Table1'

getModelParameter(cm, 'Table2', 'Identifier')

ans =

    'mp_Table2'

```

10 Save the model.

11 Generate and view the code. For example, find the data definitions for model parameter mp_K1 in `rtwdemo_configrinterface.c`.

```
int8_T mp_K1 = 2;
```

Find where the parameter is used in the step entry-point function.

```

if (mode) {
    output = (real_T)mp_K1 * dout_Table1;
} else {
    output = dstate_X;
}

```

Choose Storage Class and Storage Class Properties for Model Parameters

Depending on your code generation requirements, choose from these storage classes to configure code generation for model parameters. For constants, only `Default` applies. For model parameter arguments, only `Auto`, `Default`, and `Model default` apply.

Requirements	Storage Class
Enable optimizations, potentially generating more efficient code.	"Auto" on page 15-77 (Individual mappings only)
For data elements that cannot be optimized, represent data as a field of a standard data structure.	"Default" on page 15-77 (Default mapping only)
Prevent optimizations from eliminating storage for a data element and use the default mapping for the data element category.	"Model Default" on page 15-77 (Individual mappings only), Dictionary Default (Individual mappings only)
Generate a global variable definition and declaration.	"ExportedGlobal" on page 15-78
Generate code that reads from and writes to a global variable or global variable pointer defined by your external code.	"ImportedExtern, ImportedExternPointer" on page 15-78

The list of available storage classes might include other project-specific storage classes defined in an Embedded Coder Dictionary. If you have special requirements that are not met by the listed storage classes and you have Embedded Coder software, you can define a storage class. See "Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture" (Embedded Coder).

For an individual model parameter, use the **Identifier** storage class property to configure a name for the variable representing the parameter in the generated code.

See Also

Code Mappings Editor - C | `coder.mapping.api.CodeMapping`

More About

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75
- “Configure Model Data Elements for ASAP2 File Generation” on page 39-2

Configure Block States for C Code Generation

Block states, such as the state of a Unit Delay block, retain a state value between execution cycles of an algorithm. When configuring a model for code generation, you can configure state data to:

- Minimize the amount of data that is stored in memory.
- Control where the code generator places state data in memory.
- Promote state data to the model interface so that other components and systems can access that data.
- Improve readability and traceability of the generated code.

For code generation, examples show how to configure a block state for model `rtwdemo_configrpinterface`. You can configure code mappings by using the **Code Mappings Editor - C** or code mappings programming interface (`coder.mapping.api.CodeMapping`).

Choose Code Configuration Options for States

Based on your code generation requirements, decide how to represent state data. By default, states in a model appear in generated code as fields of a global data structure named `model_DW`. If you do not configure customizations, the code generator determines whether to eliminate or change the representation of states in generated code for optimization purposes. If you configure customizations, decide:

- Whether to set up a default configuration

If you need to gain access to a significant number of states (for example, more than 10), it is more efficient to configure the states with default settings and then override those settings for special cases. If you need to gain access to a few states that have unique source, naming, or placement requirements, consider configuring the states individually.

- How to declare and handle state data in the generated code
 - As separate global variables
 - To read input data from global variables defined in external code
 - As calls to access functions. Requires Embedded Coder

For more information about these options, see “Control Data and Function Interface in Generated Code” on page 15-44.

Other considerations include whether to:

- Name states in the generated code by using state labels that appear in the model or by using unique code identifiers.
- Include the `volatile` type qualifier in global variable definitions and declarations. Requires Embedded Coder. See “Protect Global Data with `const` and `volatile` Type Qualifiers” (Embedded Coder).
- Place state data into a specific area of memory. Requires Embedded Coder. See “Control Data and Function Placement in Memory by Inserting Pragmas” (Embedded Coder).

For a list of interface requirements that are relevant to states with corresponding storage classes and storage class properties, see “Choose Storage Class and Storage Class Properties for Data Stores” on page 16-66.

State requirements for example model `rtwdemo_configrpinterface` are:

- Retain the state data of the Unit Delay block for accessibility while the generated code executes.
- Represent the state as a separate global variable.
- Apply prefix `dtemp_` to the name of the variable that represents the state.

For this example, you set the default representation of the state in the generated code as a global variable with the `static` type qualifier. Then, you configure the state of the Unit Delay block to use the default storage class and a unique code identifier that includes the required prefix `dtemp_`. The code identifier capability enables you to specify code generation identifiers without having to modify the model design.

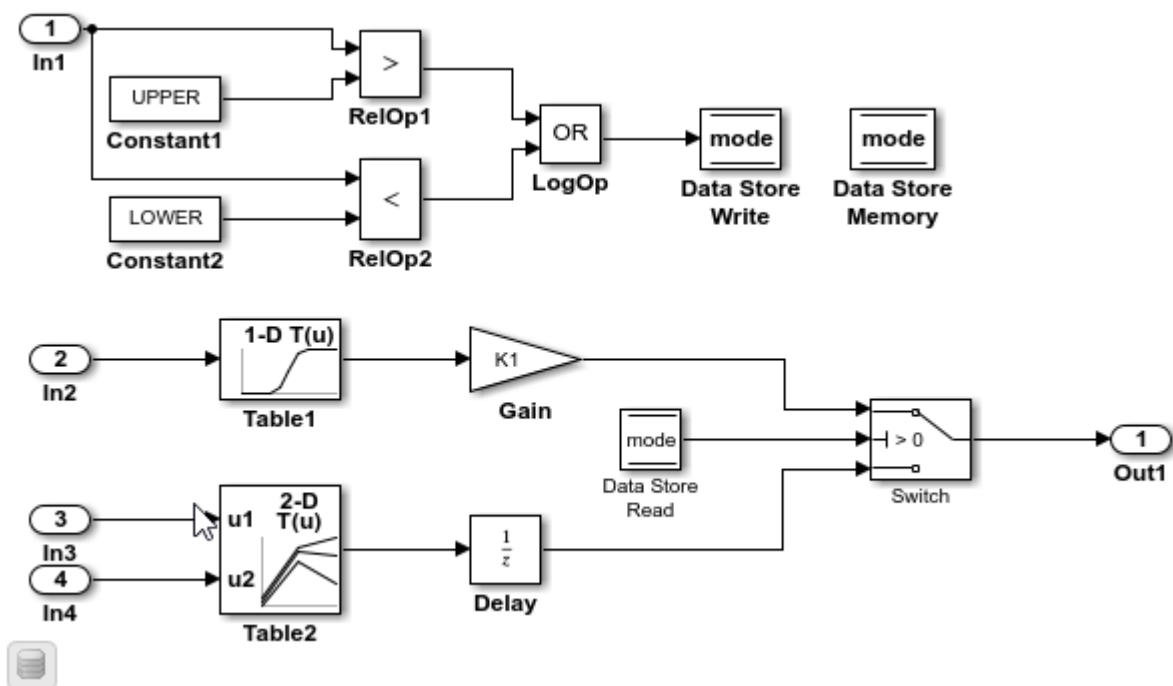
Configure Default Code Generation Settings for States

A default code generation setting for states can reduce the effort of preparing a model for code generation, especially if a model has a significant number of states that you want to gain access to while the generated code executes. Choose configuration settings once, and the code generator applies those settings to states across the model. Simulink stores the default configuration as part of the model.

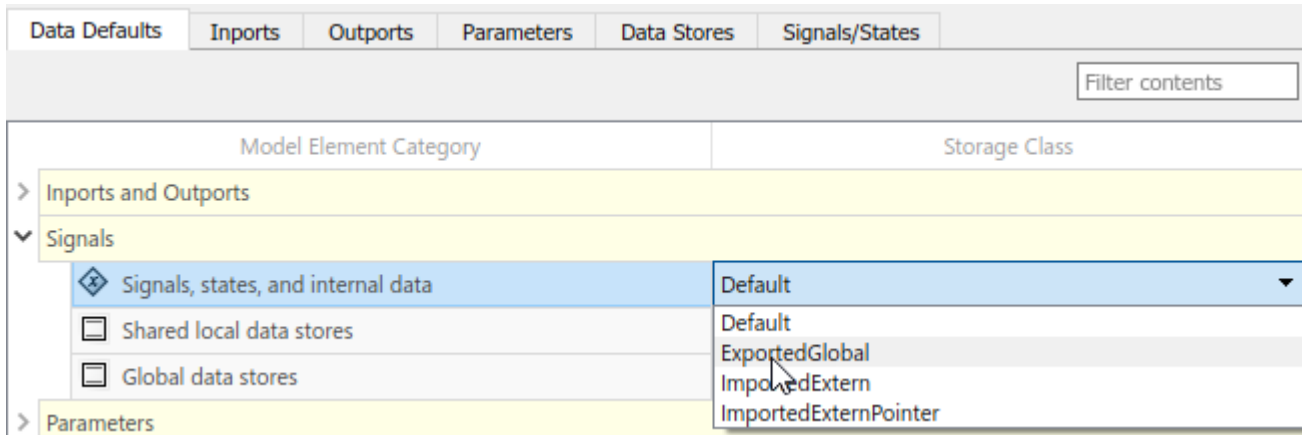
Consider configuring default code generation settings for model states if your model uses multiple states that do not have unique requirements.

This example shows how to use the **Code Mappings Editor - C** to configure default settings for states. Use the Code Mappings editor to set the default storage class for states in model `rtwdemo_configrpinterface` to `ExportedGlobal`. With that storage class setting, the code generator represents state data in the generated code as global variables.

- 1 Open model `rtwdemo_configrpinterface`. Save a copy of the model to a writable location.



- 2 Open the **Simulink Coder** app.
- 3 In the **C Code** tab, select **Code Interface > Default Code Mappings**.
- 4 In the Code Mappings editor, under **Signals**, select category **Signals, states, and internal data**. Set the default storage class to `ExportedGlobal`.



- 5 Save the model.

Configure Code Generation Settings for Individual States

You can configure individual states for code generation. For example, if a model has two states that have unique code generation requirements, configure the states individually. Or, if you configure default settings for states, you can override those settings for specific states.


If your model meets at least one of these criteria, consider configuring code generation settings for states individually:

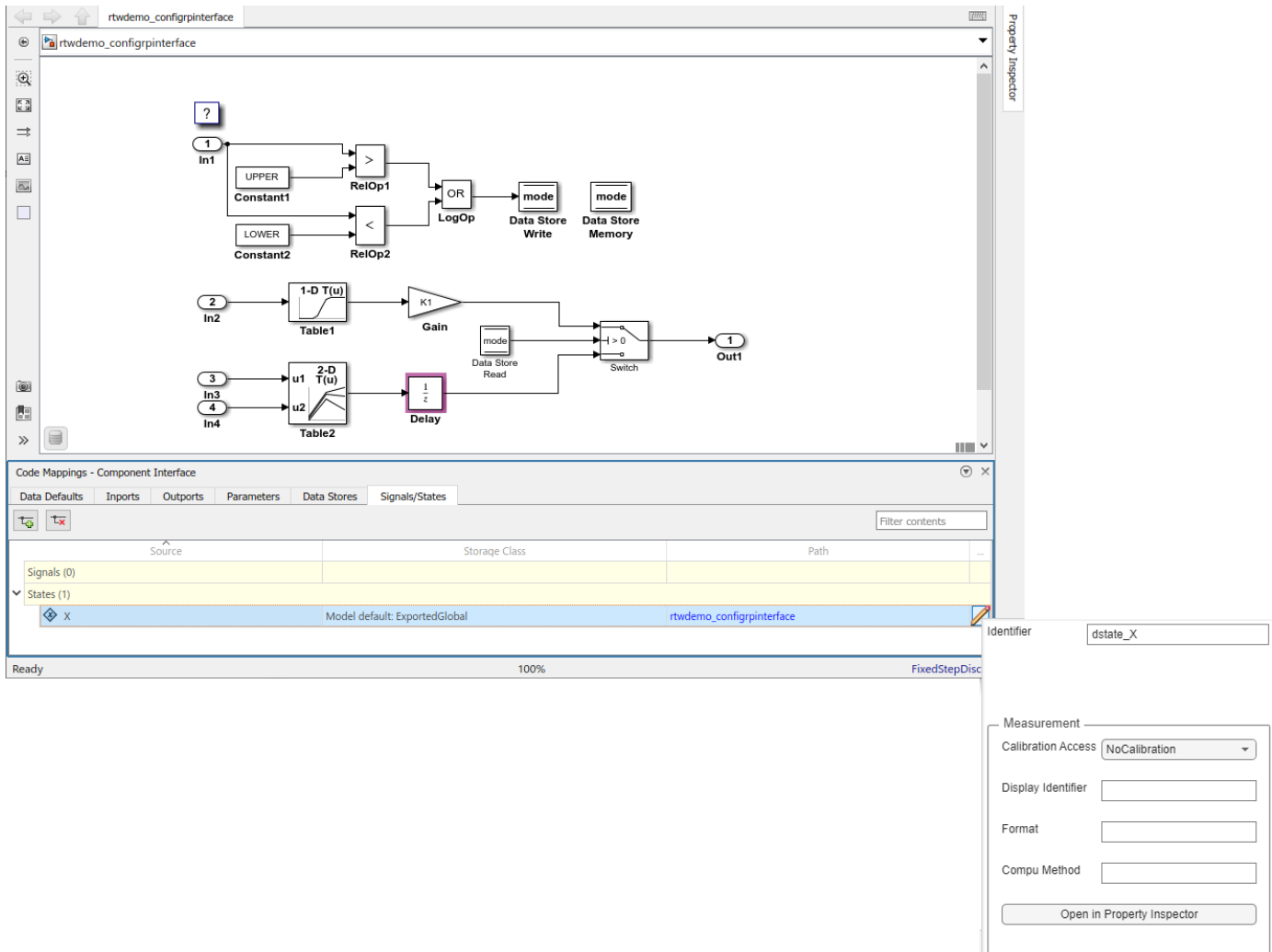
- Uses multiple states that have unique source, naming, or placement requirements.
- Uses a few states.
- Has a default configuration for states and you need to override the configuration for some specific states.

This example shows how to use the Code Mappings editor to apply your default storage class setting to the Unit Delay block state X in model `rtwdemo_configinterface`.

The example also shows how to configure a code identifier for that state. You can specify code generation identifiers, for example for integration, without modifying the model design.

- 1 If you have not already done so, complete the steps in “Configure Default Code Generation Settings for States” on page 16-54.
- 2 In the Code Mappings editor, click the **Signals/States** tab. Expand **States**. The storage class for the state is set to `Auto`, which means that the code generator might eliminate or change the representation of relevant code for optimization purposes. If optimizations are not possible, the code generator applies the model default configuration. For this example, the model default configuration specifies storage class `ExportedGlobal`.
 - To avoid optimizations and force the code generator to use the default configuration, set the storage class to `Model default`.

- To override the default configuration, specify the storage class that meets the code generation requirements for that state.
- 3 In the Code Mappings editor, select state X. Set the storage class to Model default: ExportedGlobal.
- 4 Configure the code identifier for the state with a name that includes the prefix dstate_. In the Code Mappings editor, select state X. Click the  icon and set the storage class property **Identifier** to dstate_X.



The screenshot shows a Simulink model window titled 'rtwdemo_configrpininterface'. The model contains several blocks: 'In1' (input), 'UPPER Constant1', 'LOWER Constant2', 'RelOp1' (greater than), 'RelOp2' (less than), 'OR LogOp', 'Data Store Write', 'Data Store Memory', '1-D T(u) Table1', 'Gain K1', '2-D T(u) Table2', 'Delay', 'Data Store Read', 'Switch', and 'Out1' (output). Below the model is the 'Code Mappings - Component Interface' window. It has tabs for 'Data Defaults', 'Inports', 'Outports', 'Parameters', 'Data Stores', and 'Signals/States'. The 'Signals/States' tab is active, showing a table with columns for Source, Storage Class, and Path. Under 'States (1)', state 'X' is listed with 'Storage Class' set to 'Model default: ExportedGlobal' and 'Path' set to 'rtwdemo_configrpininterface'. A pencil icon is visible next to the 'Storage Class' cell. To the right of the table, the 'Identifier' property is set to 'dstate_X'. Below the table, there are fields for 'Measurement', 'Calibration Access' (set to 'NoCalibration'), 'Display Identifier', 'Format', and 'Compu Method', along with an 'Open in Property Inspector' button.

- 5 Save the model.
- 6 Generate and view the code. For example, in `rtwdemo_configrpininterface.c`, find the data definitions for the state data.

```
real_T dstate_X;
```

Find where the state data is used in the step entry-point function.

```

.
.
.
if (mode) {

```



```

    output = (real_T)mp_K1 * dout_Table1;
  } else {
    output = dstate_X;
  }
  .
  .
  .
  dstate_X = dout_Table2;
}

```

Configure Code Generation Settings for States Programmatically

To automate configuration of states for code generation, use the programming interface for code mappings. For example, when creating custom block libraries or as part of creating an application test environment, use the programming interface to automate data configuration.

This example shows how to use the programming interface to configure states for model `rtwdemo_configrpinterface`. Set the default representation of states in the generated code as global variables. Then, configure state X of the Unit Delay block to use the default storage class and unique code identifier that includes the required prefix `dstate_`.

- 1 Open the example model.

```
open_system('rtwdemo_configrpinterface')
```

- 2 Create object `cm` by calling function `coder.mapping.api.get`. The object stores the code generation configuration for data for model `rtwdemo_configrpinterface`.

```
cm = coder.mapping.api.get('rtwdemo_configrpinterface');
```

- 3 Configure default settings for states by calling function `setDataDefault`. For the arguments, specify these values:

- The object returned by `coder.mapping.api.get`
- `InternalData` for the default category
- Property name `StorageClass` with property value `ExportedGlobal`

```
setDataDefault(cm, 'InternalData', 'StorageClass', 'ExportedGlobal');
```

- 4 Verify your default configuration for states. Issue a call to `getDataDefault` that specifies the object returned by `coder.mapping.api.get` and category `InternalData`. Specify the third argument as property `StorageClass`.

```
getDataDefault(cm, 'InternalData', 'StorageClass')
```

```
ans =
```

```
    'ExportedGlobal'
```

- 5 Apply the default configuration for states to state X.

By default, Simulink sets the storage class for individual states to `Auto`. The code generator:

- Determines whether to eliminate the data from the generated code for optimization purposes.
- If retaining the data, determines how to efficiently represent the data in the generated code, taking into account default configuration settings.

To control the configuration for a state, call function `setState`.

Issue a call to `setState` that specifies:

- Object returned by `coder.mapping.api.get`
- State name `X`
- Default storage class previously set for states by using property `StorageClass` and property value `Model default`.
- Property `Identifier` and property value `dstate_X`

```
setState(cm,'X','StorageClass','Model default','Identifier','dstate_X');
```

- 6 Verify your configuration changes by calling function `getState`. Specify the object returned by `coder.mapping.api.get`, name of the state, and property `StorageClass` or `Identifier`.

```
getState(cm,'X','StorageClass')
```

```
ans =
```

```
    'Model default'
```

```
getState(cm,'X','Identifier')
```

```
ans =
```

```
    'dstate_X'
```

- 7 Save the model.

- 8 Generate and view the code. For example, in `rtwdemo_configrpininterface.c`, find the data definitions for state data.

```
real_T dstate_X;
```

Find where the state data is used in the step entry-point function.

```
.
.
.
if (mode) {
    output = (real_T)mp_K1 * dout_Table1;
} else {
    output = dstate_X;
}
.
.
.
dstate_X = dout_Table2;
}
```

Choose Storage Class and Storage Class Properties for States

Depending on your code generation requirements, choose from these storage classes to configure code generation for block states.

Requirements	Storage Class
Let the code generator handle how to represent the state in the generated code. For example, for optimization purposes, the code generator might eliminate or change the representation of data.	“Auto” on page 15-77 (Individual mappings only)
For data elements that cannot be optimized, represent data as a field of a standard data structure.	“Default” on page 15-77 (Default mapping only)

Requirements	Storage Class
Prevent optimizations from eliminating storage for a data element and use the default mapping for the data element category.	“Model Default” on page 15-77 (Individual mappings only) , Dictionary Default (Individual mappings only)
Generate a global variable definition and declaration.	“ExportedGlobal” on page 15-78
Generate code that reads from and writes to a global variable or global variable pointer defined by your external code.	“ImportedExtern, ImportedExternPointer” on page 15-78

The list of available storage classes might include other project-specific storage classes defined in an Embedded Coder Dictionary. If you have special requirements that are not met by the listed storage classes and you have Embedded Coder software, you can define a storage class. See “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder).

For an individual state, use the **Identifier** storage class property to configure a name for the variable representing the state in the generated code. If you leave the **Identifier** property blank, the code generator uses the **State name** parameter of the block that has the state. If this parameter is empty, the code generator uses the name of the block.

See Also

Code Mappings Editor - C | `coder.mapping.api.CodeMapping`

More About

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75
- “Configure Model Data Elements for ASAP2 File Generation” on page 39-2

Configure Data Stores for C Code Generation

Data stores enable subsystems and referenced models to share data without using I/O ports to pass the data from level to level (see “Data Store Basics” and “Data Stores in Generated Code” on page 15-94). Types of data stores include the types listed in this table.

Type of Data Store	Description
Local data store	Data store that is accessible from anywhere in a model hierarchy that is at or below the level at which you define the data store. You can define a local data store graphically in a model by including a Data Store Memory block.
Shared local data store	Data Store Memory block with the block parameter Share across model instances set. These data stores are accessible only in the model where they are defined. The data store value is shared across instances of the model.
Global data store	Data store that is defined by a signal object in the base workspace or in a data dictionary. Multiple models in an application can use these data stores.

When you open a model in the **Simulink Coder** app, local and shared local data stores appear in the Code Mappings editor, where you can configure the data stores for code generation. If a model uses global data stores, you can view them in the Code Mappings editor by clicking the **Refresh** link that appears to the right of the data store name. That link initiates an update diagram and adds global data stores that the model uses in the editor view.

Configure data stores to:

- Make data accessible for interaction while the generated code executes.
- Control where the code generator places data in memory.
- Improve readability and traceability of generated code.

For code generation, examples show how to configure a data store for model `rtwdemo_configrpininterface`. You can configure code mappings by using the **Code Mappings Editor - C** or code mappings API (`coder.mapping.api.CodeMapping`).

Choose Customization Options for Data Stores

By default, local data stores in a model appear in generated code as fields of a global data structure named `model_DW`. Shared local data stores appear as a field of global data structure `model_SharedDSM`. Based on your code interface requirements, decide whether to customize code generation of data stores. If you do not configure customizations, the code generator determines whether to eliminate or change the representation of data stores in generated code for optimization purposes. If you configure customizations, decide:

- Whether to set up a default configuration

If a model includes a significant number (for example, more than 10) of data stores for a given category that must be accessible during program execution, it is more efficient to configure the data stores with a default setting, and then override that setting for special cases. If the model includes a few data stores for a given category that have unique source, naming, or placement requirements, consider configuring the data stores individually.

- How to declare and handle model data stores in the generated code
 - As separate global variables
 - To read from data stores as global variables defined in external code
 - As calls to access functions. Requires Embedded Coder

For more information about these options, see “Control Data and Function Interface in Generated Code” on page 15-44.

Other considerations for model parameters include whether to:

- Name data stores in the generated code by using data store names in the model or by using unique code identifiers.
- Include the `static` type qualifier in global variable definitions and declarations, for example, to prevent name clashes. Requires Embedded Coder. See “Prevent Name Clashes by Configuring Data Item as static” (Embedded Coder).
- Include the `volatile` type qualifier in global variable definitions and declarations. Requires Embedded Coder. See “Protect Global Data with `const` and `volatile` Type Qualifiers” (Embedded Coder).
- Generate a global data structure with a name that you specify. Requires Embedded Coder. See “Organize Data into Structures in Generated Code” (Embedded Coder).
- Place data stores in a specific area of memory. Requires Embedded Coder. See “Control Data and Function Placement in Memory by Inserting Pragmas” (Embedded Coder).

For a list of interface requirements that are relevant to data stores with corresponding storage classes and storage class properties, see “Choose Storage Class and Storage Class Properties for Data Stores” on page 16-66.

For example model `rtwdemo_configrpinterface`, data store requirements are:

- Represent local and shared local data stores as separate global variables.
- Apply prefix `ds_` to names of variables that represent data stores.

For this example, you set the default representation for local and shared local data stores in the generated code as global variables. Then, you configure the local data store in the model to use the default storage class and a unique code identifier that includes the required prefix `ds_`. The code identifier capability enables you to specify code generation identifiers without having to modify the model design.

Configure Default Code Generation Settings for Data Stores

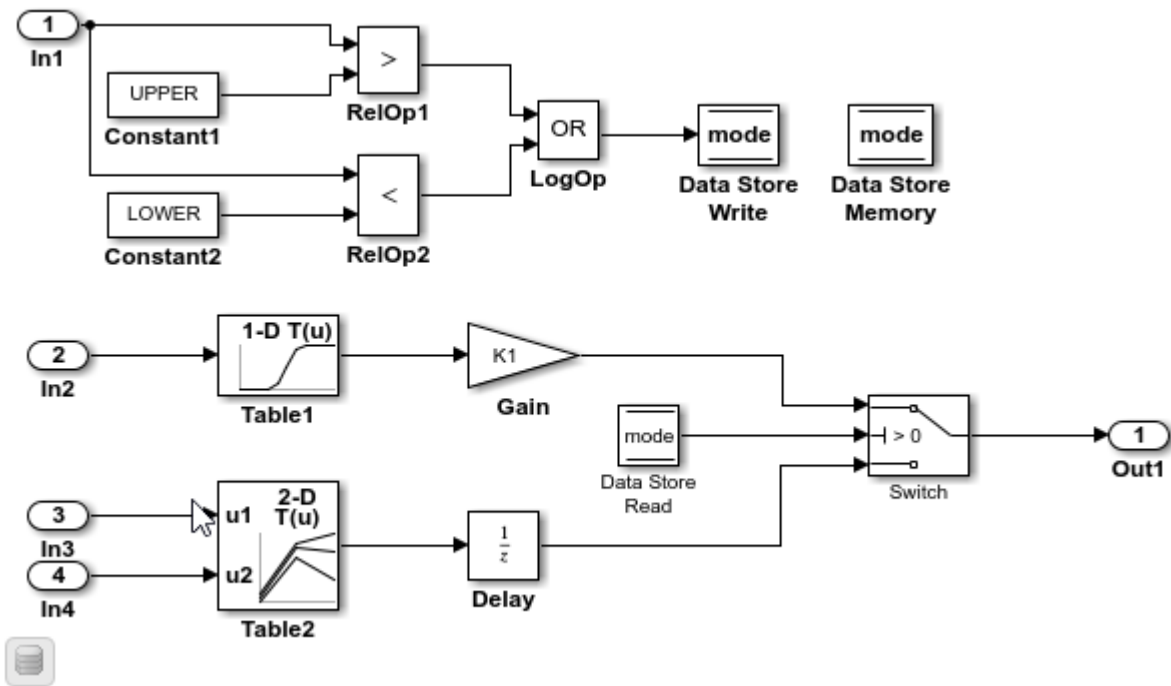
Default code generation settings for data stores can reduce the effort of preparing a model for code generation, especially if a model has a significant number of data stores that you want to gain access to while the generated code executes. Choose configuration settings once, and the code generator applies those settings to data stores across the model. Simulink stores the default configuration as part of the model.

Consider configuring default code generation settings for model data stores if your model uses multiple data stores for a given category that do not have unique requirements.

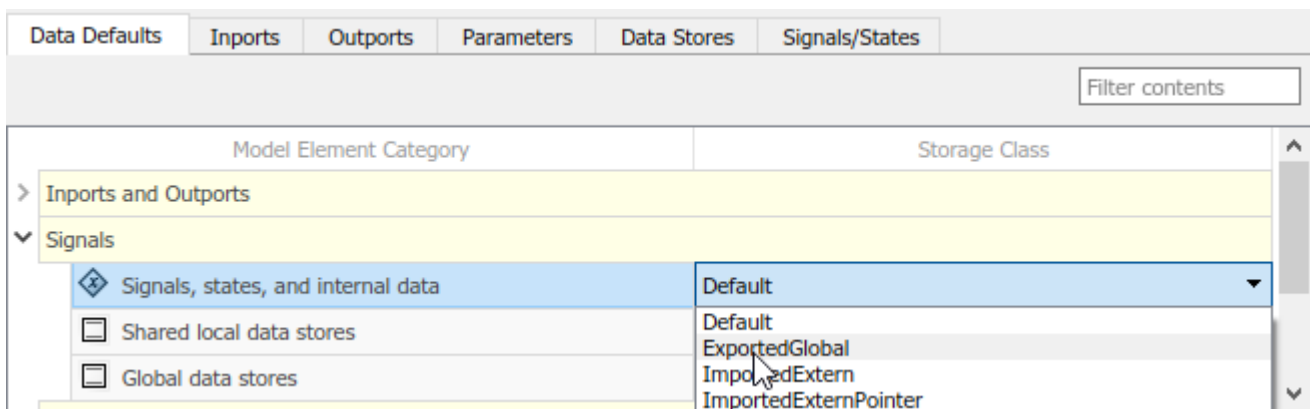
This example shows how to use the **Code Mappings Editor - C** to configure default settings for data stores. Use the Code Mappings editor to set the default storage class for local and shared local data

stores in model `rtwdemo_configrpinterface` to `ExportedGlobal`. With those storage class settings, the code generator represents the data stores in the generated code as global variables.

- 1 Open model `rtwdemo_configrpinterface`. Save a copy of the model to a writable location.



- 2 Open the **Simulink Coder** app.
- 3 In the **C Code** tab, select **Code Interface > Default Code Mappings**.
- 4 Configure default code mappings for local data stores. In the Code Mappings editor, on the **Data Defaults** tab, expand the **Signals** node. Select category **Signals, states, and internal data**. Set the storage class to `ExportedGlobal`.



- 5 Configure default code mappings for shared local data stores. Select category **Shared local data stores**. Set the storage class to `ExportedGlobal`.
- 6 Save the model.


Configure Code Generation for Individual Data Stores

You can configure individual data stores for code generation. For example, if a model has two data stores of the same category that have unique code generation requirements, configure the data stores individually. Or, if you configure default settings for a category of data stores, you can override those settings for specific data stores.

If your model meets at least one of these criteria, consider configuring code generation settings for data stores individually:

- Uses multiple data stores of the same category that have unique source, naming, or placement requirements.
- Uses a few data stores of the same category.
- Has a default configuration for a category of data stores and you need to override the configuration for some specific states.

This example shows how to use the Code Mappings editor to apply your default storage class setting to the Data Store Memory block `mode` in model `rtwdemo_configinterface`. The example also shows how to configure a code identifier for that data store.

- 1 If you have not already done so, complete the steps in “Configure Default Code Generation Settings for Data Stores” on page 16-61.
- 2 In the Code Mappings editor, click the **Data Stores** tab. Expand **Local Data Stores**. The storage class for data store `mode` is set to `Auto`, which means that the code generator might eliminate or change the representation of relevant code for optimization purposes. If optimizations are not possible, the code generator applies the model default configuration. For this example, the model default configuration specifies the storage class `ExportedGlobal`.
 - To avoid optimizations and force the code generator to use the default configuration, set the storage class to `Model default`.
 - To override the default configuration, specify the storage class that meets the code generation requirements for that data store.
- 3 In the Code Mappings editor, select local data store `mode`. Set the storage class to `Model default: ExportedGlobal`.
- 4 Configure the code identifier for the data store with a name that includes the prefix `ds_`. In the Code Mappings editor, select shared local data store `mode`. Click the  icon and set the storage class property **Identifier** to `ds_mode`.

The screenshot displays a Simulink model window titled 'rtwdemo_configrinterface'. The model contains several blocks: 'In1' (input), 'UPPER Constant1', 'LOWER Constant2', 'RelOp1' (greater-than), 'RelOp2' (less-than), 'OR', 'LogOp', 'Data Store Write', 'Data Store Memory', '1-D T(u) Table1', 'Gain K1', 'Data Store Read', '2-D T(u) Table2', 'Delay 1/z', 'Switch', and 'Out1'. The 'Code Mappings - C' window is open, showing the 'Data Stores' tab. It lists 'mode' as a local data store with the storage class 'Model default: ExportedGlobal' and path 'rtwdemo_configrinterface'. A property inspector for 'ds_mode' is also visible, showing settings for Identifier, Measurement, Calibration Access, Display Identifier, Format, and Compu Method.

- 5 Save the model.
- 6 Generate and view the code. For example, in `rtwdemo_configrinterface.c`, find the data definitions for the data store.

```
boolean_T ds_mode;
```

Find where the state data is used in the step entry-point function.

```
.
.
.
ds_mode = ((input1 > rtwdemo_configrinterface_UPPER) || (input1 <
    rtwdemo_configrinterface_LOWER));
.
.
.
if (ds_mode) {
    output = (real_T)mp_K1 * dout_Table1;
} else {
    output = dstate_X;
}
.
.
.
```

Configure Code Generation Settings for Data Stores Programmatically

To automate configuration of data stores for code generation, use the programming interface for code mappings. For example, when creating custom block libraries or part of an application test environment, use the programming interface to automate data configuration.

This example shows how to use the programming interface to configure data stores for model `rtwdemo_configrpinterface`. Set the default representation of local and shared local data stores in the generated code as global variables. Configure the Data Store Memory block mode to use the default storage class and a unique code identifier that includes the required prefix `ds_`.

- 1 Open the example model.

```
open_system('rtwdemo_configrpinterface')
```

- 2 Create object `cm` by calling function `coder.mapping.api.get`. The object stores the data code generation configuration for model `rtwdemo_configrpinterface`.

```
cm = coder.mapping.api.get('rtwdemo_configrpinterface');
```

- 3 Configure default settings for local data stores by calling function `setDataDefault`. For the arguments, specify these values:

- The object returned by `coder.mapping.api.get`
- `InternalData` for the default category
- Property name `StorageClass` with property value `ExportedGlobal`

```
setDataDefault(cm,'InternalData','StorageClass','ExportedGlobal');
```

- 4 Configure default settings for shared local data stores by calling function `setDataDefault`. Specify these values for arguments:

- The object returned by `coder.mapping.api.get`.
- `SharedLocalDataStore` for the default category.
- Property name `StorageClass` with property value `ExportedGlobal`.

```
setDataDefault(cm,'SharedLocalDataStore','StorageClass','ExportedGlobal');
```

- 5 Verify your default configuration settings for local and shared local data stores. Issue a call to `getDataDefault` that specifies the object returned by `coder.mapping.api.get`, category `InternalData`, and the property `StorageClass`. In a second call to `getDataDefault`, replace the category with `SharedLocalDataStore`.

```
getDataDefault(cm,'InternalData','StorageClass')
```

```
ans =
```

```
    'ExportedGlobal'
```

```
getDataDefault(cm,'SharedLocalDataStore','StorageClass')
```

```
ans =
```

```
    'ExportedGlobal'
```

- 6 Apply the default configuration for local data store mode.

By default, Simulink sets the storage class for data stores to `Auto`. When the storage class is `Auto`, the code generator:

- Determines whether to eliminate the data from the generated code for optimization purposes.
- If retaining the data, determines how to efficiently represent the data in the generated code, taking into account default configuration settings.

To control the configuration for an individual data store, call function `setDataStore`. Issue a call to `setDataStore` that specifies:

- Object returned by `coder.mapping.api.get`
- Data store name `mode`
- Default storage class previously set for the data store by using property `StorageClass` and property value `Model default`.
- Property `Identifier` and property value `ds_mode`

```
setDataStore(cm, 'mode', 'StorageClass', 'Model default', 'Identifier', 'ds_mode');
```

- 7 Verify your configuration changes by calling function `getDataStore`. Specify the object returned by `coder.mapping.api.get`, the name of the data store, and property `StorageClass` or `Identifier`.

```
getDataStore(cm, 'mode', 'StorageClass')
```

```
ans =
```

```
    'Model default'
```

```
getDataStore(cm, 'mode', 'Identifier')
```

```
ans =
```

```
    'ds_mode'
```

- 8 Save the model.

- 9 Generate and view the code. For example, in `rtwdemo_configrpininterface.c`, find the data definition for the data store.

```
boolean_T ds_mode;
```

Find where the data store is used in the step entry-point function.

```
.
.
.
    ds_mode = ((input1 > rtwdemo_configrpininterface_UPPER) || (input1 <
        rtwdemo_configrpininterface_LOWER));
.
.
.
    if (ds_mode) {
        output = (real_T)mp_K1 * dout_Table1;
    } else {
        output = dstate_X;
    }
.
.
.
```

Choose Storage Class and Storage Class Properties for Data Stores

Depending on your code generation requirements, choose from these storage classes to configure code generation for data stores.

Requirements	Storage Class
Enable optimizations, potentially generating more efficient code.	“Auto” on page 15-77 (Individual mappings only)
For data elements that cannot be optimized, represent data as a field of a standard data structure.	“Default” on page 15-77 (Default mapping only)
Prevent optimizations from eliminating storage for a data element and use the default storage class for the data element category.	“Model Default” on page 15-77 (Individual mappings only) , Dictionary Default (Individual mappings only)
Generate a global variable definition and declaration.	“ExportedGlobal” on page 15-78
Generate code that reads from and writes to a global variable or global variable pointer defined by your external code.	“ImportedExtern, ImportedExternPointer” on page 15-78

The list of available storage classes might include other project-specific storage classes defined in an Embedded Coder Dictionary. If you have special requirements that are not met by the listed storage classes and you have Embedded Coder software, you can define a storage class. See “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder).

For an individual data store, use the **Identifier** storage class property to configure a name for the variable representing the data store in the generated code.

See Also

Code Mappings Editor - C | `coder.mapping.api.CodeMapping`

More About

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75
- “Configure Model Data Elements for ASAP2 File Generation” on page 39-2

Manage Multiple Code Mappings for a Model

Code mappings separate code generation configurations from model designs. This separation enables you to design models that are platform-neutral and ready to deploy to different target platform environments. If you have access to the necessary products, you can associate a model with code mappings for several different platforms.

Platform	System Target File	Programming Language	Required Products
C rapid prototyping	GRT-based	C	Simulink Coder
C production	ERT-based	C	Embedded Coder
C++ production	ERT-based	C++	Embedded Coder
AUTOSAR classic	AUTOSAR	C	AUTOSAR Blockset, Embedded Coder (for generating code)
AUTOSAR adaptive	AUTOSAR Adaptive	C++	AUTOSAR Blockset, Embedded Coder (for generating code)

You can generate code for the different platforms from a model or model hierarchy. Associate the models in a model hierarchy with the same platform.

Simulink activates an existing code mapping when you do one of the following:

- In the Simulink Editor, open a coder app for a model and specify an output type that supports code mappings.
- Use the code mappings API to create a code mapping.

The Code Mappings editor and code mappings programming interface display and operate on the active code mapping.

Configure Alternative Code Mappings by Using Configuration Sets

You can configure alternative code mappings for a model by creating multiple model configuration sets and changing which configuration set is active. Simulink creates or opens an existing code mapping for a model based on settings of these model configuration parameters:

- **System target file**
- **Language**
- **Code interface packaging**

Create a configuration set for each platform for which you want to generate code for a model. Code mappings are supported for these combinations of parameter settings.

Platform	System target file	Language	Code interface packaging
C/C++ rapid prototyping	grt.tlc or derivative of grt.tlc	C/C++	Nonreusable function or Reusable function
C/C++ production	ert.tlc or derivative of ert.tlc	C/C++	Nonreusable function or Reusable function
C++ production	ert.tlc or derivative of ert.tlc	C++	C++ class
AUTOSAR classic	autosar.tlc	C	Nonreusable function or Reusable function
AUTOSAR adaptive	autosar_adaptive.tlc	C++	C++ class

For C++ platforms, reusable and nonreusable functions use C code mappings.

Activate the configuration set of interest. Then, open the corresponding code mappings in the Simulink Editor, as described in “Activate Code Mappings in the Simulink Editor” on page 16-69 or the code mappings API in “Create and Activate Code Mappings Programmatically” on page 16-70.

Activate Code Mappings in the Simulink Editor

When you open a model in the Simulink Editor, Simulink activates an existing code mapping for the model based on settings of these model configuration parameters:

- **System target file**
- **Language**
- **Code interface packaging**

When you open one of the following coder apps, Simulink checks whether the system target file setting of the active configuration set aligns with your coder app selection, which can be one of these:

- **Simulink Coder** (GRT-based system target file)
- **Embedded Coder** (ERT-based system target file)
- **AUTOSAR Component Designer** (AUTOSAR system target file)

When Simulink detects a misalignment, the editor displays a message that identifies the current system target file setting and questions whether you want to change it and open the selected coder app. You can cancel the operation and open the app that aligns with the current system target file setting or continue with changing the system target file and open the app that you selected.

If the current system target file aligns with your coder app selection, Simulink opens the coder app and adds a code tab (for example, **C Code** or **AUTOSAR**) to the editor toolstrip.

After a coder app opens, on the toolstrip, for **Output**, select the C or C++ code setting or, for AUTOSAR, select one of the AUTOSAR platforms. Then, use the **Code Interface** menu to open the Code Mappings editor.

Create and Activate Code Mappings Programmatically

You can use the code mappings functions `coder.mapping.utils.create` and `coder.mapping.api.get` to create and activate code mappings for GRT- and ERT-based model platforms.

- 1 Set up multiple model configuration set objects (`Simulink.ConfigSet`) for the model. For example, you might set up GRT and ERT C code configurations or two ERT code configurations, one for generating reusable functions and one for generating nonreusable function code. Name the configuration sets and attach them to your model.

```
grtConfigObj = Simulink.ConfigSet;
set_param(grtConfigObj, 'Name', 'grtConfig');
attachConfigSet('myModel', grtConfigObj);
```

```
ertConfigObj = Simulink.ConfigSet;
set_param(ertConfigObj, 'Name', 'ertConfig');
attachConfigSet('myModel', ertConfigObj);
```

- 2 Activate and create a code mapping for one of the configuration sets by calling `setActiveConfigSet` and `coder.mapping.utils.create`.

```
setActiveConfigSet('myModel', 'grtConfig');
grt_cm = coder.mapping.utils.create('myModel');
```

- 3 Activate and create a code mapping for the second configuration set.

```
setActiveConfigSet('myModel', 'ertConfig')
ert_cm = coder.mapping.utils.create('myModel');
```

- 4 Activate the model code mappings by using calls to `coder.mapping.api.get` that specify the code mapping type, `SimulinkCoderC` or `EmbeddedCoderC`.

```
grt_cm = coder.mapping.api.get('myModel', 'SimulinkCoderC');
.
.
.
# Configure model data elements
.
.
..
ert_cm = coder.mapping.api.get('myModel', 'EmbeddedCoderC');
```

Code Mappings and Embedded Coder Dictionary

If you have Embedded Coder, you can use the Embedded Coder Dictionary to generate standardized code from multiple models. Because the definitions in the Embedded Coder Dictionary for a model are separate from your model elements, you can share the definitions by storing the Embedded Coder Dictionary in a Simulink data dictionary outside of your model. See “Separate C Code Interface Configuration Definition from Model-Specific Configurations” (Embedded Coder).

See Also

Code Mappings Editor - C | **Code Mappings - C++ Editor** | `Simulink.ConfigSet` | `setActiveConfigSet` | `coder.mapping.api.CodeMapping`

More About

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- Code interface packaging
- “Manage Configuration Sets for a Model”

Configure Model Data Interface for C Code Generation

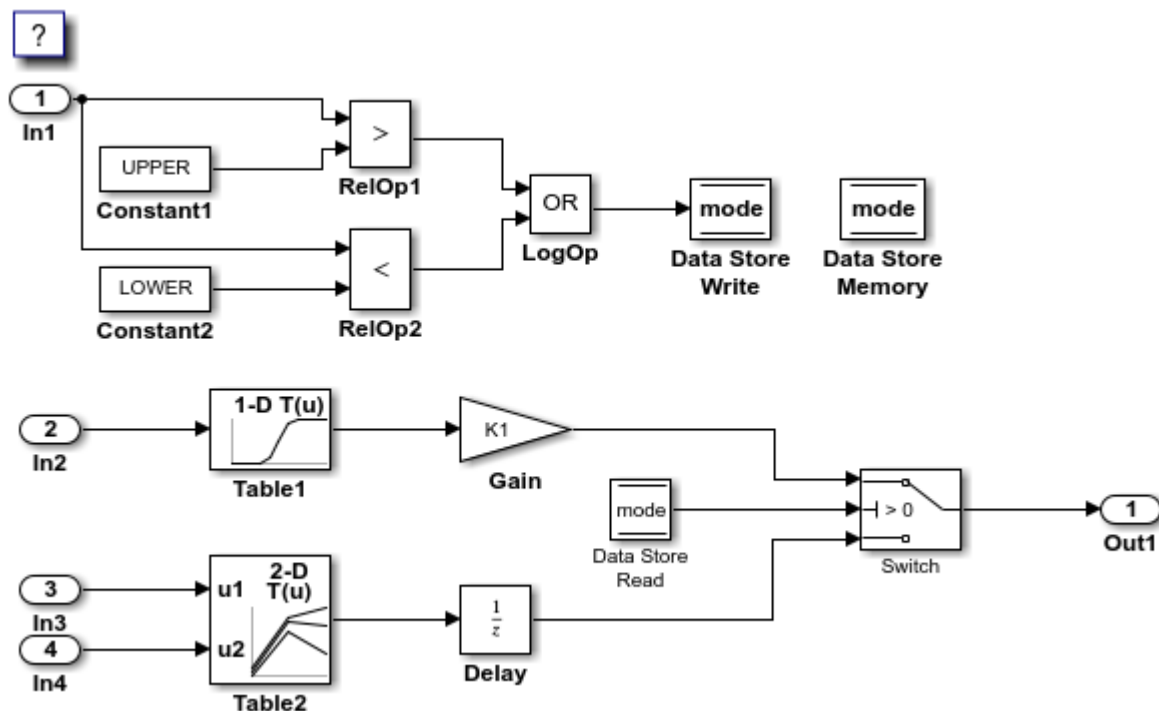
To integrate the generated code with external code or to generate code that complies with coding standards and guidelines, you can configure how data elements in a model, such as block parameters and signals lines, appear in the generated code. You can configure default code generation settings for categories of data elements and configure individual elements by using the Code Mappings editor or code mappings API. For example, you can configure settings that control placement of declarations and definitions in generated and external (exported and imported) files and naming of variables that represent data elements in the generated code.

This example shows how to configure data elements for code generation for example model `rtwdemo_configrpinterface`.

Explore Example Model

Open example model `rtwdemo_configrpinterface`.

```
model = 'rtwdemo_configrpinterface';
open_system(model)
```



The example shows how to configure these model data elements for code generation:

- Inport blocks In1, In2, In3, and In4
- Outport block Out1
- Model parameters K1, Table1, and Table2
- Local data store mode

- Signal lines sourced from lookup table blocks Table1 and Table2
- State X (delay)

Configure Default Settings for Inport Blocks

- 1 Open the **Simulink Coder** app.
- 2 In the **C Code** tab, select **Code Interface > Default Code Mappings**.
- 3 In the Code Mappings editor, under **Inports and Outports**, select category **Inports**. Set the default storage class to `ImportedExternPointer`.

Configure Default Settings for Output Block

On the **Data Defaults** tab, under **Inports and Outports**, select category **Outports**. Set the default storage class to `ExportedGlobal`.

Configure Default Settings for Parameters

- 1 On the **Data Defaults** tab, under **Parameters**, select category **Model parameters**. Link text 'Auto' will be inlined indicates that the code generator is configured to inline model parameters by default. For this example, configure the model parameters to be tunable. Click 'Auto' will be inlined. The Model Configuration Parameters dialog box opens.
- 2 Set model configuration parameter **Default parameter behavior** to **Tunable**. Save the change and close the dialog box. In the Code Mappings editor, the link text changes to 'Auto' will be tunable.
- 3 In the Code Mappings editor, with the **Model parameters** category selected, set the storage class to `ExportedGlobal`.

Configure Default Settings for Signals, States, and Data Stores

- 1 On the **Data Defaults** tab, under **Signals**, select category **Signals, states, and internal data**. Set the default storage class to `ExportedGlobal`.
- 2 Configure default code mappings for shared local data stores. Select category **Shared local data stores**. Set the storage class to `ExportedGlobal`.
- 3 Save the model.

Configure Code Generation Settings for Individual Inport Blocks

- 1 In the Code Mappings editor, click the **Inports** tab. The editor lists the names of root-level **Inport** and **In Bus Element** blocks that are in the model. If a port resolves to a signal object, a resolve-to-signal-object icon appears to the right of the element name. The storage class for each inport is set to `Auto`, which means that the code generator might eliminate relevant code for optimization purposes. If optimizations are not possible, the code generator applies the model default configuration. For this example, the model default configuration specifies storage class `ImportedExternPointer`. To avoid optimizations and force the code generator to use the default configuration, set the storage class to `Model default`. To override the default configuration, specify the storage class that meets the code generation requirements for that inport.
- 2 Configure the code generator to apply the default storage class setting to inports **In2**, **In3**, and **In4**. Select the rows for the three inports. Then, for one of the selected inports, set the storage class to `Model default: ImportedExternPointer`. The storage class for the three selected inports changes to `Model default: ImportedExternPointer`.

- 3 For inport In1, override the default storage class setting. Set the storage class to ImportedExtern.
- 4 Configure code identifiers for the inports so that the variables in the generated code match the interface names in the external header and definition files. In the Code Mappings editor, select the row for each inport. In the Property Inspector, set the **Identifier** property to these values: input1, input2, input3, and input4.

Configure Code Generation Settings for Individual Outputs

- 1 In the Code Mappings editor, click the **Outputs** tab. The editor lists the names of root-level **Outport** and **Out Bus Element** blocks and bus elements that are in the model. If a port resolves to a signal object, a resolve-to-signal-object icon appears to the right of the element name. The storage class for the outport in the example model is set to **Auto**, which means that the code generator might eliminate relevant code for optimization purposes.
- 2 To avoid optimizations and force the code generator to use the default configuration, set the storage class to **ExportedGlobal**. The storage class for the selected outport changes to **Model default: ExportedGlobal**.
- 3 Configure the code identifier for the outport so that the variable in the generated code matches the interface name used by the external code. In the Code Mappings editor, select the row for the outport. In the Property Inspector, set the **Identifier** property to **output**.

Configure Code Generation Settings for Individual Parameters

- 1 In the Code Mappings editor, click the **Parameters** tab. Expand **Model Parameter Arguments**. By default, the storage class for each model parameter argument is set to **Auto**, which means that the code generator might eliminate relevant code for optimization purposes. If optimizations are not possible, the code generator applies the model default configuration. For this example, leave the storage class set to **Auto**.
- 2 Expand **Model Parameters**. By default, the storage class for each model parameter is set to **Auto**. For this example, you want to avoid optimizations and for the code generator to use the default storage class setting, **ExportedGlobal**.
- 3 In the Code Mappings editor, under **Model Parameters**, select parameters **K1**, **Table1**, and **Table2**. Set the storage class to **Model default: ExportedGlobal**.
- 4 Configure the code identifiers for model parameters with names that include the prefix **mp_**, as the requirements specify. In the Code Mappings editor, select model parameter **K1**. In the Property Inspector, expand the **Code** node. Set the storage class property **Identifier** to **mp_K1**. For parameters **Table1** and **Table2**, set **Identifier** to **mp_Table1** and **mp_Table2**.

Configure Code Generation Settings for Individual Data Stores

- 1 In the Code Mappings editor, click the **Data Stores** tab. Expand **Local Data Stores**. The storage class for data store mode is set to **Auto**, which means that the code generator might eliminate relevant code for optimization purposes. For this example, you want to avoid optimizations and for the code generator to use the default storage class setting, **ExportedGlobal**.
- 2 In the Code Mappings editor, select local data store mode. Set the storage class to **Model default: ExportedGlobal**.
- 3 Configure the code identifier for the data store with a name that includes the prefix **ds_**. In the Code Mappings editor, select shared local data store mode. In the Property Inspector, expand the **Code** node. Set the storage class property **Identifier** to **ds_mode**.

Configure Code Generation Settings for Individual States

- 1 In the Code Mappings editor, click the **Signals/States** tab. Expand **States**. The storage class for the state is set to **Auto**, which means that the code generator might eliminate relevant code for optimization purposes. For this example, you want to avoid optimizations and for the code generator to use the default storage class setting, **ExportedGlobal**.
- 2 In the Code Mappings editor, select state X. Set the storage class to **Model default: ExportedGlobal**.
- 3 Configure the code identifier for the state with a name that includes the prefix **dstate_**, as the requirements specify. In the Code Mappings editor, select state X. In the Property Inspector, expand the **Code** node. Then, set the storage class property **Identifier** to **dstate_X**.

Configure Code Generation Settings for Individual Signals

Add signals that you want to configure to the model code mappings. For this example, add to the model code mappings the output signals for the two lookup table blocks. Then, configure those signals.

- 1 In the Code Mappings editor, click the **Signals/States** tab. No signals are listed.
- 2 Add signals to the code mappings. For the output signals of lookup table blocks **Table1** and **Table2**, in the model, select a signal, pause on the ellipsis that appears above or below the signal line to open the action bar, and click the **Add Signal** button. In the Code Mappings editor, the **Signals** node expands and lists the names or block identifiers of the two signals that you added. If a signal resolves to a signal object, a resolve-to-signal-object icon appears to the right of the element name or port identifier. The storage class for each signal is set to **Auto**, which means that the code generator might eliminate relevant code for optimization purposes. For this example, you want to avoid optimizations and for the code generator to use the default storage class setting, **ExportedGlobal**.
- 3 In the Code Mappings editor, select the output signals for blocks **Table1** and **Table2**. Set the storage class to **Model default: ExportedGlobal**.
- 4 Configure the code identifier for the output signals for the two lookup table blocks with names that include the prefix **dout**. In the Code Mappings editor, select signal **Table1:1**. In the Property Inspector, expand the **Code** node. Set the storage class property **Identifier** to **dout_Table1**. For signal **Table2:1**, set **Identifier** to **dout_Table2**.
- 5 Save the model.

Configure Data for Code Generation Programmatically

Alternatively, use the code mappings API to configure model data elements programmatically.

1. Get the object that represents the code mappings for the model by calling `coder.mapping.api.get`. You specify the returned object as the first argument in subsequent calls to code mapping functions.

```
cm = coder.mapping.api.get(model);
```

2. Configure default code generation settings for categories of data elements with calls to `setDataDefaults`. In each call, include the object returned by the call to `coder.mapping.api.get`, the name of a data element category, and name-value pair arguments that specify the storage class and storage class properties. Use `setDataDefaults` to configure these default settings:

- Inport blocks with storage class ImportedExternPointer
- Outport blocks with storage class ExportedGlobal
- Model parameters with storage class ExportedGlobal
- Internal data (signals, states, and local data stores) with storage class ExportedGlobal
- Shared local data stores with storage class ExportedGlobal

```
setDataDefault(cm, 'Inports', 'StorageClass', 'ImportedExternPointer');
setDataDefault(cm, 'Outports', 'StorageClass', 'ExportedGlobal');
setDataDefault(cm, 'ModelParameters', 'StorageClass', 'ExportedGlobal');
setDataDefault(cm, 'InternalData', 'StorageClass', 'ExportedGlobal');
setDataDefault(cm, 'SharedLocalDataStore', 'StorageClass', 'ExportedGlobal');
```

Set model configuration parameter `DefaultParameterBehavior` to `Tunable`, so that model parameters are accessible during program execution.

```
set_param(model, 'DefaultParameterBehavior', 'Tunable');
```

3. Configure individual inports with calls to `setInport`. To avoid optimizations, apply the default configuration for inports to `In2`, `In3`, and `In4`. For `In1`, override the default configuration, by setting the storage class to `ImportedExtern`. Also, configure code identifiers for the inports so that the global variables that represent inports in the generated code match the variable names in the external definition file. In each call to `setInport`, specify the object returned by `coder.mapping.api.get`, the Inport block name, and name-value pair arguments for storage class and property settings.

```
setInport(cm, 'In1', 'StorageClass', 'ImportedExtern', 'Identifier', 'Input1');
setInport(cm, 'In2', 'StorageClass', 'Model default', 'Identifier', 'Input2');
setInport(cm, 'In3', 'StorageClass', 'Model default', 'Identifier', 'Input3');
setInport(cm, 'In4', 'StorageClass', 'Model default', 'Identifier', 'Input4');
```

4. Configure individual data element settings for Outport block `Out1` with a call to `setOutport`. To avoid optimizations, apply the default configuration. Also, configure a code identifier for the outport. In the call to `setOutport`, specify the object returned by `coder.mapping.api.get`, the Outport block name, and name-value pair arguments for the storage class and code identifier property settings.

```
setOutport(cm, 'Out1', 'StorageClass', 'Model default', 'Identifier', 'output');
```

5. Configure individual model parameters with calls to `setModelParameter`. To avoid optimizations, apply the default configuration. Also, configure code identifiers for the parameters. In each call to `setModelParameter`, specify the object returned by `coder.mapping.api.get`, the parameter name, and name-value pair arguments for the storage class and code identifier property settings.

```
setModelParameter(cm, 'K1', 'StorageClass', 'Model default', 'Identifier', 'mp_K1');
setModelParameter(cm, 'Table1', 'StorageClass', 'Model default', 'Identifier', 'mp_Table1');
setModelParameter(cm, 'Table2', 'StorageClass', 'Model default', 'Identifier', 'mp_Table2');
```

6. Configure individual data element settings for local data store `mode` with a call to `setDataStore`. To avoid optimizations, apply the default configuration. Also, configure a code identifier for the data store. In the call to `setDataStore`, specify the object returned by `coder.mapping.api.get`, the data store name, and name-value pair arguments for the storage class and code identifier property settings.

```
setDataStore(cm, 'mode', 'StorageClass', 'Model default', 'Identifier', 'ds_mode');
```

7. Configure individual data element settings for state X with a call to `setState`. To avoid optimizations, apply the default configuration. Also, configure a code identifier for the state. In the call to `setState`, specify the object returned by `coder.mapping.api.get`, the path for the block that uses the state, and name-value pair arguments for the storage class and code identifier property settings.

```
setState(cm,'rtwdemo_configrpinterface/Delay','StorageClass','Model default',...
        'Identifier','dstate_X');
```

8. Add signals to the code mappings and configure them with calls to `setSignal`. To add the port handles, add the signal data to the model code mappings with calls to `get_param`. Then, specify the block handles in a call to `addSignals`. To avoid optimizations, apply your default configuration for signals. Also, configure code identifiers for the signals. In each call to `setSignal`, specify the object returned by `coder.mapping.api.get`, the path or block handle of the signal source, and name-value pair arguments for the storage class and code identifier property settings.

```
lut1D_ports = get_param('rtwdemo_configrpinterface/Table1','PortHandles');
lut2D_ports = get_param('rtwdemo_configrpinterface/Table2','PortHandles');

lut1D_outPort = lut1D_ports.Outport;
lut2D_outPort = lut2D_ports.Outport;

addSignal(cm,[lut1D_outPort,lut2D_outPort]);

setSignal(cm,lut1D_outPort,'StorageClass','Model default','Identifier','dout_Table1');
setSignal(cm,lut2D_outPort,'StorageClass','Model default','Identifier','dout_Table2');
```

Generate and View Code

1. Generate code.

```
slbuild(model)

### Starting build procedure for: rtwdemo_configrpinterface
### Successful completion of code generation for: rtwdemo_configrpinterface

Build Summary

Top model targets built:

Model                Action                Rebuild Reason
=====
rtwdemo_configrpinterface  Code generated.  Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 37.308s
```

2. View the code. For example, in `rtwdemo_configrpinterface.c`, find where these variables are used in the step entry-point function: `input1`, `input2`, `input3`, `input4`, `output`, `mp_K1`, `ds_mode`, `dstate_X`, `dout_Table1`, and `dout_Table2`.

Related Topics

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Configure Root-Level Inport Blocks for C Code Generation” on page 16-18
- “Configure Root-Level Outport Blocks for C Code Generation” on page 16-27

- “Configure Signal Data for C Code Generation” on page 16-34
- “Configure Parameters for C Code Generation” on page 16-44
- “Configure Block States for C Code Generation” on page 16-53
- “Configure Data Stores for C Code Generation” on page 16-60
- Code Mappings Editor - C

Configure Generated C Function Interface for Model Entry-Point Functions

What Is an Entry-Point Function?

An entry point is a location in code where a transfer of program control (execution) occurs. The main function (`main()`) is the entry point to a C/C++ program and is called when the application starts executing. Calls to other functions, for example from the `main` function, provide entry points to function code. Program control is transferred to the called function. The function code executes, and when finished, returns control to the `main` or other calling function.

When producing code for a model, the code generator defines a set of entry-point functions that you can call to execute the generated code. You can call the generated functions from external code or from a version of the generated main function that you modify.

The Code Interface Report section of the code generation report lists the entry-point functions that the code generator produces for a model. For more information, see “Analyze Generated Data Code Interface” (Embedded Coder).

Types of Generated Entry-Point Functions

The code generator produces entry-point functions for these types of model functions.

Type of Model Function	Model Source Name	Default Generated Function Name	Description
Exported function (requires Embedded Coder)	ExportedFunction: <i>slIdentifier</i> , where <i>slIdentifier</i> is the name of the function-call Inport block in the model	<i>function-call-inport-block-name</i> or <i>signal-label</i> (if specified)	For an exported-function model, the exported function for a subsystem.
Initialize function	Initialize	<i>model_initialize</i>	Initialization code for a model. At the start of the application code, call the function <i>once</i> . Do not use this function to reset the real-time model data structure (rtM).

Type of Model Function	Model Source Name	Default Generated Function Name	Description
Partition function	Partition: <i>sIdentifier</i> , where <i>sIdentifier</i> is a partition that was created explicitly from a block in the model and shown in the Simulink® Schedule Editor (for example, P1), or a task name in the Concurrent Execution dialog	<i>model_stepn</i> , where <i>n</i> uniquely identifies the function generated for one of the model sample periods	For a model partition, output and update code. Model configuration parameter Single output/update function is selected (the default).
Partition update function	PartitionUpdate: <i>sIdentifier</i> , where <i>sIdentifier</i> is a partition that was created explicitly from a block in the model and shown in the Simulink® Schedule Editor (for example, P1), or a task name in the Concurrent Execution dialog	<i>model_outputn</i> and <i>model_updaten</i> , where <i>n</i> uniquely identifies the function generated for one of the model sample periods	For a model partition, output and update code in separate functions. Model configuration parameter Single output/update function is cleared.
Periodic multitasking function	Periodic: <i>sIdentifier</i> , where <i>sIdentifier</i> is an annotation that corresponds to the sample time period for a periodic or continuous rate of a multitasking model (for example, D1)	<i>model_stepn</i> , where <i>n</i> uniquely identifies the function generated for one of the model sample periods	For blocks in a rate-based model configured for multitasking, output and update code. The code generator produces a function for each sample period. Model configuration parameter Single output/update function is selected (the default).
Periodic multitasking update function	PeriodicUpdate: <i>sIdentifier</i> , where <i>sIdentifier</i> is an annotation that corresponds to the sample time period for a periodic or continuous rate of a multitasking model (for example, D1)	<i>model_outputn</i> and <i>model_updaten</i> , where <i>n</i> uniquely identifies the function generated for one of the model sample periods	For blocks in a rate-based model configured for multitasking, output and update code in separate functions. The code generator produces output and update functions for each sample period. Model configuration parameter Single output/update function is cleared.

Type of Model Function	Model Source Name	Default Generated Function Name	Description
Periodic single-tasking function	Periodic	<i>model_step</i>	For blocks in a rate-based model configured for single-tasking, output and update code. Model configuration parameter Single output/update function is selected (the default).
Periodic single-tasking update function	PeriodicUpdate	<i>model_output</i> and <i>model_update</i>	For blocks in a rate-based model configured for single-tasking, output and update code in separate functions. Model configuration parameter Single output/update function is cleared.
Reset function	Reset: <i>slIdentifier</i> , where <i>slIdentifier</i> is the name of the reset function in the model	<i>model_reset-function-name</i>	If the model includes a Reset Function block, reset code generated. To reset conditions or state, call the function from the application code.
Simulink Function	Simulink Function: <i>slIdentifier</i> , where <i>slIdentifier</i> is the name of the Simulink Function block in the model	<i>function-name</i> for a global Simulink Function block or <i>model_function-name</i> for a scoped Simulink Function block	For an exported-function model, the exported function for a Simulink Function block.
Terminate function	Terminate	<i>model_terminate</i>	Code for turning off a system. For ERT-based models, you can suppress generation of this function by clearing the model configuration parameter Terminate function required (Embedded Coder) (set by default).

If you have Embedded Coder software, to facilitate integration of external and generated code and achieve compliance with code standards and guidelines, you can configure how the code generator produces function interfaces from a model or subsystem.

Configure Whether Entry-Point Functions Are Reusable

By default, for top models, the code generator produces code that is not reusable or reentrant. Entry-point functions have a `void-void` interface. Code communicates with other code by directly accessing global data structures that reside in shared memory.

If your application requires reusable, multi-instance entry-point function code, you can configure the code generator to call each function (instance) with unique data. In this case, the code is reentrant.

You configure whether entry-point functions are reusable with the model configuration parameter **Code interface packaging** and related parameters. The parameter settings that you choose depend on factors such as configuration selections for the system target file, programming language, and argument interface.

Default Configurations for Single-Instance C Entry-Point Functions

By default, for GRT- and ERT-based system target files, the code generator produces single-instance C entry-point functions. The generated code:

- Creates an execution function without arguments (`void-void`).
- Allocates memory statically (at compile time) for model data structures.

The default model configuration parameter settings for configuring single-instance entry-point function code are:

- **Language** set to C.
- **Code interface packaging** set to `Nonreusable function`.

Generate Reusable, Multi-Instance C Entry-Point Functions

You can configure the code generator to produce reusable entry-point functions in C for either a GRT- or ERT-based system target file. However, the function interfaces that the code generator produces by default varies. Assuming that model configuration parameter **Language** is set to C and **Code interface packaging** is set to `Reusable function`, the code generator produces this entry-point function code for each system target file scenario.

System Target File	Interface
GRT-based	<ul style="list-style-type: none"> • Reusable, multi-instance C entry-point functions that are reentrant. • Packs values of model root-level Inport blocks and Outport blocks into the real-time model data structure. Passes that structure to the execution function as an argument by reference. • Allocates memory dynamically at runtime for the data of a model instance. Allocates the memory by calling a function, such as <code>malloc</code>.
ERT-based	<ul style="list-style-type: none"> • Reusable, multi-instance C entry-point functions that are reentrant. • Passes the value of each model root-level Inport block and Outport block to the execution function as a separate argument. • Allocates memory statically for model data structures.

If you are using an ERT-based system target file and want to generate reusable, multi-instance C entry-point functions that are reentrant, consider:

- Using dynamic memory allocation to initialize model data structures. Select **Use dynamic memory allocation for model initialization** (Embedded Coder).
- Packing values of model root-level Inport blocks into a structure, packing values of root-level Outport blocks into a second structure, and passing the structures to the execution function as arguments by reference. Set **Pass root-level I/O as** (Embedded Coder) to **Structure reference**.
- Packing values of model root-level Inport blocks and Outport blocks into the real-time model data structure and passing that structure to the execution function as an argument by reference. Set **Pass root-level I/O as** (Embedded Coder) to **Part of model data structure**.

How to Interface with Generated Entry-Point Functions

- 1 After generating code for a model, use the **Code** view to review the generated entry-point functions and, if applicable, variables representing external input and output ports.
- 2 Add `#include` statements to your external code that include the generated header files that declare the model entry-point functions.
- 3 Add an `#include` statement that includes the generated file `rtwtypes.h`. This file provides type definitions, `#define` statements, and enumerations.
- 4 Initialize target-specific data structures and hardware, such as ADCs or DACs.
- 5 If applicable, initialize data for each instance of a reusable model.
- 6 If applicable, write input data to generated variables that represent model Inport blocks.
- 7 Call the generated entry-point functions or set up use of the `rt_OneStep` function.
- 8 If applicable, read data from generated variables that represent model Outport blocks.

For more information, see “Deploy Applications to Target Hardware” (Embedded Coder).

C Function Interface Customization Limitations

These limitations apply to customizations for generated C function interfaces:

- You must select the model configuration parameter **Single output/update function**.
- Multirate models are supported, but you must configure the models for single tasking.
- You must configure root-level inports and outports to use the `Default` storage class.
- If you choose to customize a function interface, you must provide your own custom `main` program. You cannot configure the function interface with the static `rt_main.c` that MathWorks provides. Specifying a function interface configuration other than the default creates a mismatch between the generated code and the default static `rt_main.c`.
- The code generator removes the data structure for the root inports of the model unless a subsystem implemented by a nonreusable function uses the value of one or more of the inports.
- The code generator removes the data structure for the root outports of the model except when you enable MAT-file logging or if the sample time of one or more of the outports is not the fundamental base rate (including a constant rate).
- If you copy a subsystem block to create a block in a new model or the same model, the function interface information from the original subsystem block is not copied to the new subsystem block.
- If you have Stateflow, for a Stateflow chart that uses a model root inport value or that calls a subsystem that uses a model root inport value, do one of the following to generate code:

- Clear the **Execute (enter) Chart At Initialization** check box in the Stateflow chart.
- Make the Stateflow function a nonreusable function.
- Insert a Simulink Signal Conversion block immediately after the root inport. In the Signal Conversion block parameters dialog box, select **Exclude this block from 'Block reduction' optimization**.
- If a model root inport value connects to a Simscape conversion block, insert a Simulink Signal Conversion block between the root inport and the Simscape conversion block. In the Signal Conversion block parameters dialog box, select **Exclude this block from 'Block reduction' optimization**.
- When building a referenced model that is configured with a function interface, do not use virtual buses as inputs or outputs to the referenced model. Use nonvirtual buses instead.
- If the C function interface is not the default, the value is ignored for the model configuration parameter **Pass fixed-size scalar root inputs by value for code generation**. For more information, see Pass fixed-size scalar root inputs by value for code generation.

See Also

Simulink Function | Function Caller

More About

- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Simulink Function Blocks and Code Generation” on page 10-6

Entry-Point Functions and Scheduling in Simulink Coder

- “Configure Entry-Point Function Interfaces for Simulink Function and Function Caller Blocks” on page 17-2
- “Generate C++ Class Interface to Model or Subsystem Code” on page 17-15
- “Execution of Code Generated from a Model” on page 17-18
- “Rapid Prototyping Model Functions” on page 17-27

Configure Entry-Point Function Interfaces for Simulink Function and Function Caller Blocks

With Embedded Coder, you can customize the generated C and C++ function interfaces for Simulink Function and Function Caller blocks. Function code interface configuration supports easier integration of generated code with functions or function calls in external code and customizations for coding standards or design requirements.

You can customize generated C and C++ function interfaces for:

- Global Simulink functions
- Exported scoped Simulink functions

You cannot customize the generated function interface for a scoped Simulink Function block that is not located at the root level of a model. For more information, see “Scoped, Global, and Port-Scoped Simulink Function Blocks Overview”.

By opening a dialog box from a selected Simulink Function or Function Caller block, you can customize the function prototype generated for that block. Your changes for the selected block also update other corresponding Simulink Function and Function Caller blocks in the model. The function visibility, global or scoped, set in the function Trigger Port block determines which function attributes you can modify:

- For a global function, you can change the function name, and the names, type qualifiers, and order of function arguments.
- For an exported scoped function, you can change the argument type qualifiers and the order of arguments. You cannot change the function and argument names.

Your changes do not graphically alter the model and do not affect the Simulink function prototype defined in the block.

Embedded Coder supports Simulink function code interface configuration for models configured with ERT-based system target files, except for AUTOSAR system target files.

Configure Generated C Function Interface for Global Simulink Function Block

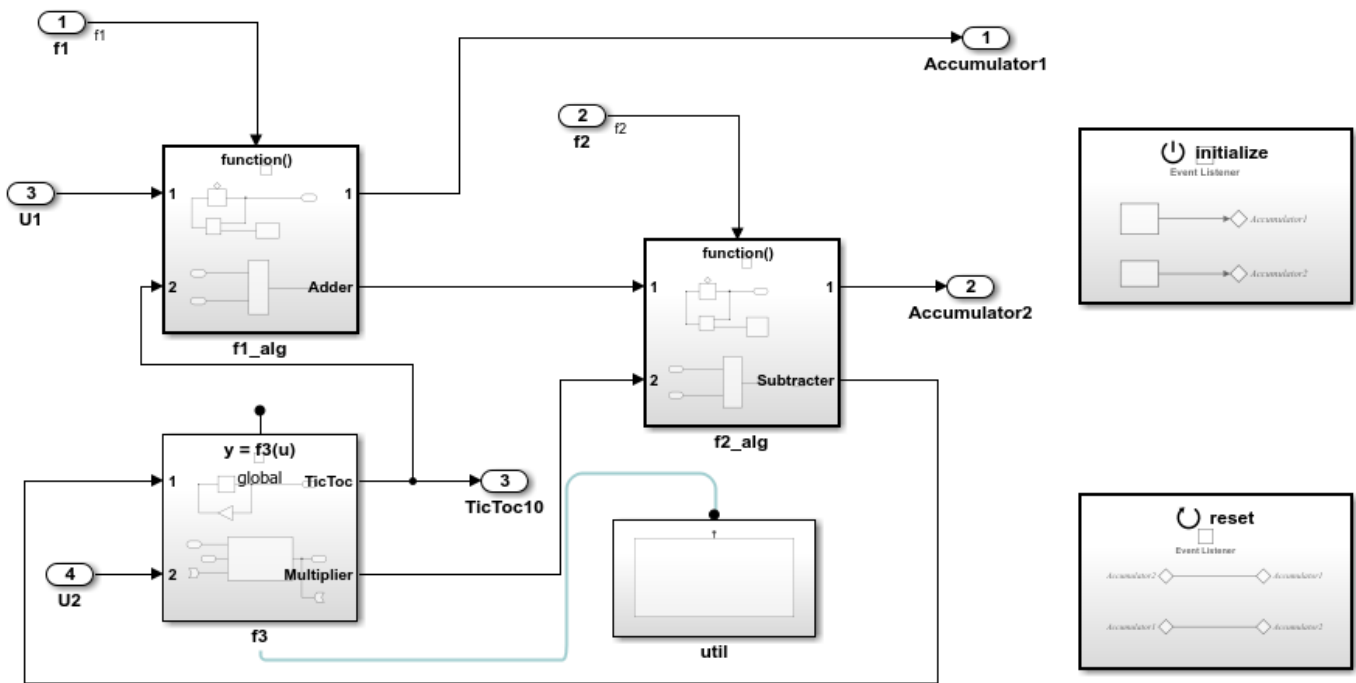
This example shows how to customize the C function interface that the code generator produces for a global Simulink Function block.

- 1 Open the example model.
- 2 Open the Configure C/C++ Function Interface Dialog Box.
- 3 Configure the Simulink function prototype.
- 4 Generate and review the code.

Open Example Model

Open example model `SimulinkFunctions`. Save a copy of the model to a writable location.

```
open_system('SimulinkFunctions');
```



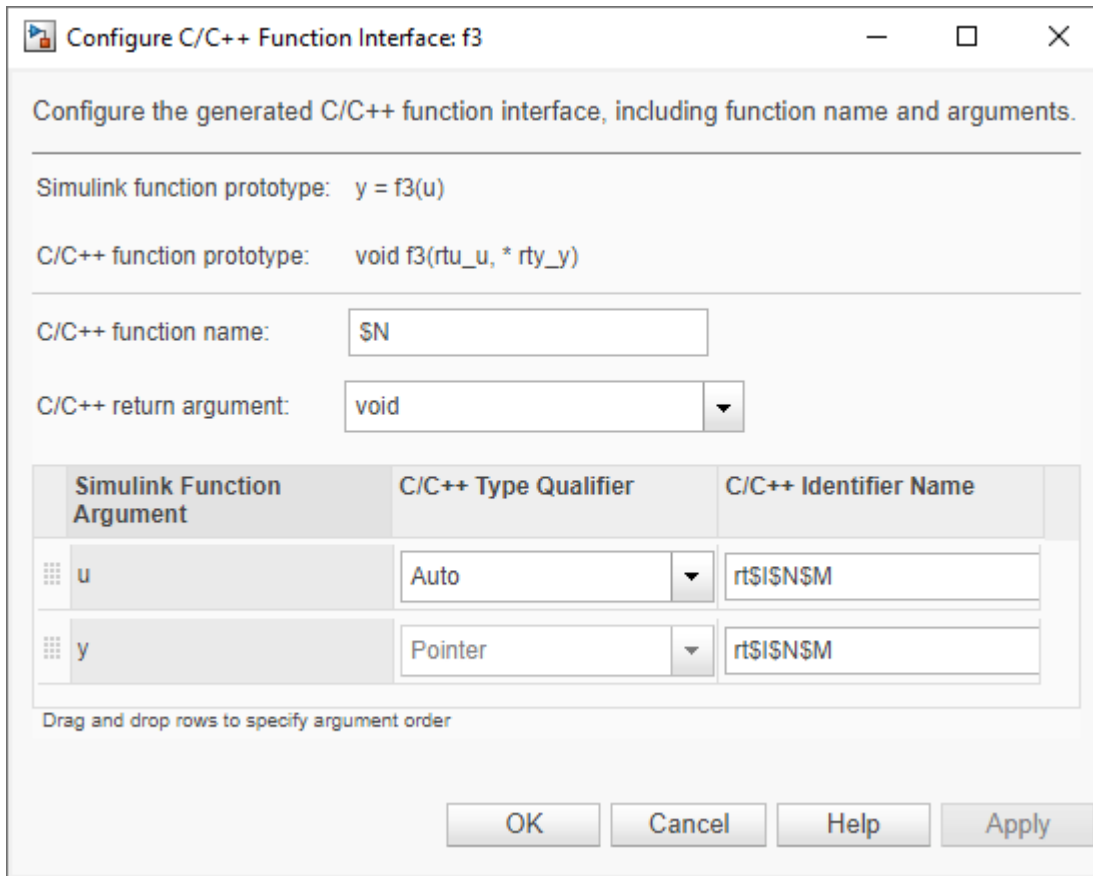
Open Configure C/C++ Function Interface Dialog Box

In the apps gallery, open the **Embedded Coder** app.

On the **C Code** tab, select **Code Interface > Individual Element Code Mappings**.

In the Code Mappings editor, click the **Functions** tab.

In the row for Simulink Function: f3, click the function preview hyperlink. The Configure C/C++ Function Interface dialog box shows **Simulink function prototype** as $y = f3(u)$ and **C/C++ function prototype** as $y = \text{SimulinkFunctions_f3}(\text{rtu_u})$. The preview for **C/C++ function prototype** is updated as you make changes.



Configure Simulink Function Prototype

Configure the function name and arguments.

- Modify the function and argument identifier names. Model configuration parameter **Subsystem method arguments** defines the default naming rule for Simulink Function arguments. Changes that you make in the C/C++ Function Interface dialog box override the default naming rule. In the **C/C++ function name field**, and in the **C/C++ Identifier Name** column for each Simulink function argument, enter a custom name or naming rule. Specify valid C-identifier characters, identifier format macros, or a combination of characters and macros. For this example, name the function `function3` and, for the arguments, enter naming rule `NM`. To see tips about available macros, place your cursor over **C/C++ function name** and **C/C++ Identifier Name**. For more information about identifier naming rules, see “Identifier Format Control” (Embedded Coder).
- For argument `u`, set **C/C++ Type Qualifier** to `Pointer` to `const`.
- Reorder the arguments. Drag the `y` argument row above the `u` argument row.

Configure the generated C/C++ function interface, including function name and arguments.

Simulink function prototype: `y = f3(u)`

C/C++ function prototype: `void f3(* rty_y, rtu_u)`

C/C++ function name:

C/C++ return argument:

Simulink Function Argument	C/C++ Type Qualifier	C/C++ Identifier Name
y	Pointer	rt\$N\$M
u	Auto	rt\$N\$M

Drag and drop rows to specify argument order

OK Cancel Help Apply

Verify your changes by checking the function prototype preview.

Click **Apply**. Your modifications, whether made to a Simulink Function block or a Function Caller block, affect code generation for the Simulink Function block and corresponding Function Caller blocks in the model.

Optionally, you can change **C/C++ return argument** from `void` to `y`. In that case, the C function prototype is `y = function3(const * u)`.

Save your changes by clicking **OK**. Then, save the model.

Generate and Review Code

Make sure that the Simulink Function block is not selected. Then, generate code for the model.

In the **Code** view, open the generated file `SimulinkFunctions.c` and search for `function3`. The generated function code reflects the changes to the generated C function prototype.

```
void function3(real_T *y, const real_T *u)
{
    rtY.TicToc10 = rtDWork.Delay_DSTATE;

    rtDWork.Delay_DSTATE = (int8_T)(int32_T)-(int32_T)rtY.TicToc10;

    adder(rtB.Subtract, rtU.U2, *u, &rtB.FunctionCaller);
}
```

```
*y = rtB.FunctionCaller;  
}
```

More About

- “Generate Reentrant Code from Simulink Function Blocks” (Embedded Coder)
- “Simulink Functions Overview”
- “Scoped Simulink Function Blocks in Models”
- “Simulink Function Blocks in Referenced Models”

Configure Generated C Function Interface for Exported Scoped Simulink Function Block

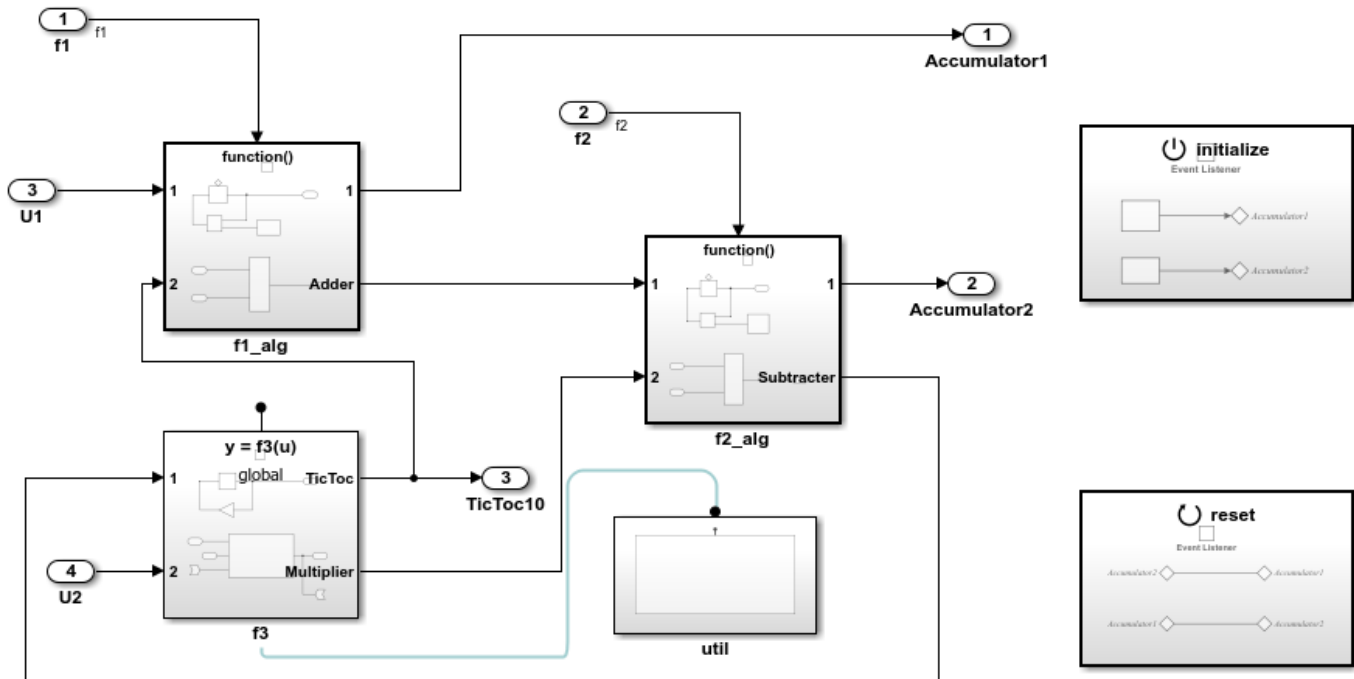
This example shows how to customize the C function interface that the code generator produces for an exported, scoped Simulink Function block.

For an exported, scoped function, you can modify the generated return argument, the argument type qualifiers, and the order of arguments. You cannot change the generated function name and argument names.

- 1** Open the example model.
- 2** Change the visibility of the Simulink function.
- 3** Open the Configure C/C++ Function Interface dialog box.
- 4** Configure the Simulink Function prototype.
- 5** Generate and review the code.

Open Example Model

Open example model `SimulinkFunctions`. Save a copy of the model to a writable location.



Change Visibility of Simulink Function

Change the visibility of the global function `f3` from global to scoped. Open the Simulink Function block `f3`. Inside the function block, double-click the Trigger Port block `f3`. In the block parameters dialog box, set **Function visibility** to **scoped**. Click **Apply** and **OK**.

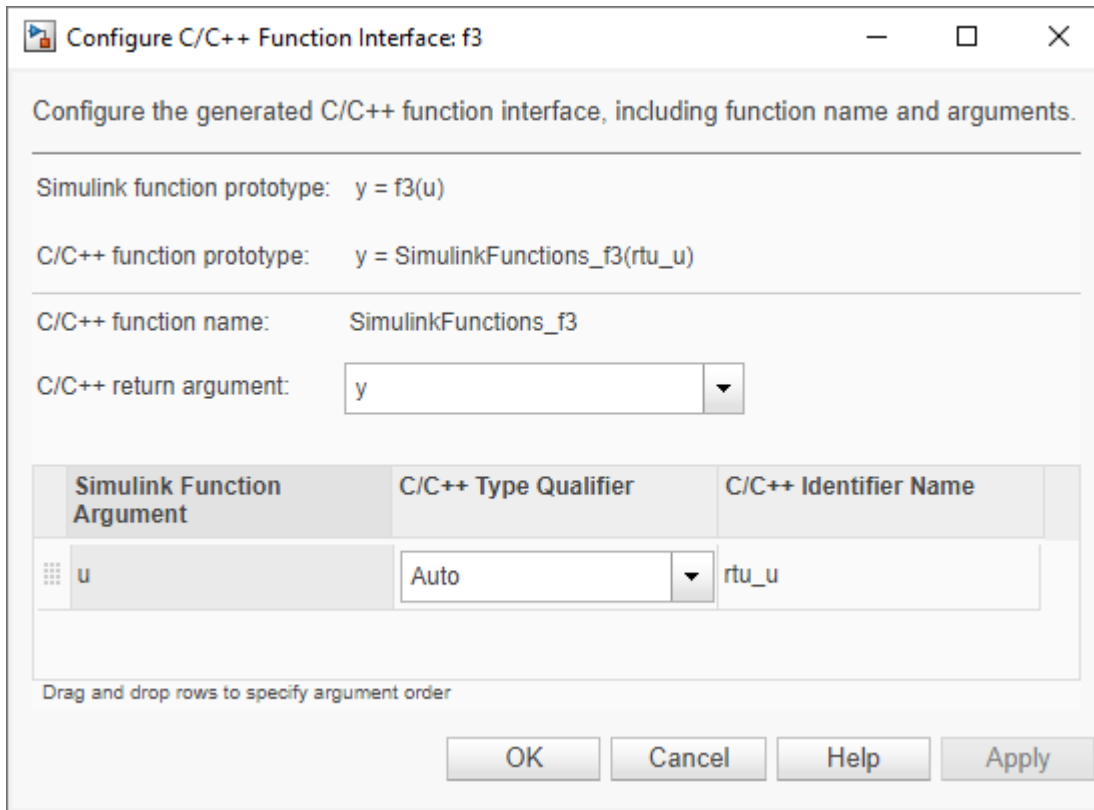
Open Configure C/C++ Function Interface Dialog Box

In the apps gallery, open the **Embedded Coder** app.

On the **C Code** tab, select **Code Interface > Individual Element Code Mappings**.

In the Code Mappings editor, click the **Functions** tab.

In the row for Simulink Function: `f3`, click the function preview hyperlink. The Configure C/C++ Function Interface dialog box shows **Simulink function prototype** as `y = f3(u)` and **C/C++ function prototype** as `void f3(rtu_u, * rty_y)`. The preview for **C/C++ function prototype** is updated as you make changes.



For models that you configure for C multi-instantiable code, the interface preview for scoped Simulink functions includes a `self` argument. The `self` argument is a pointer to a version of the real-time model (`RT_MODEL`) data structure that stores multi-instance data associated with reusable functions.

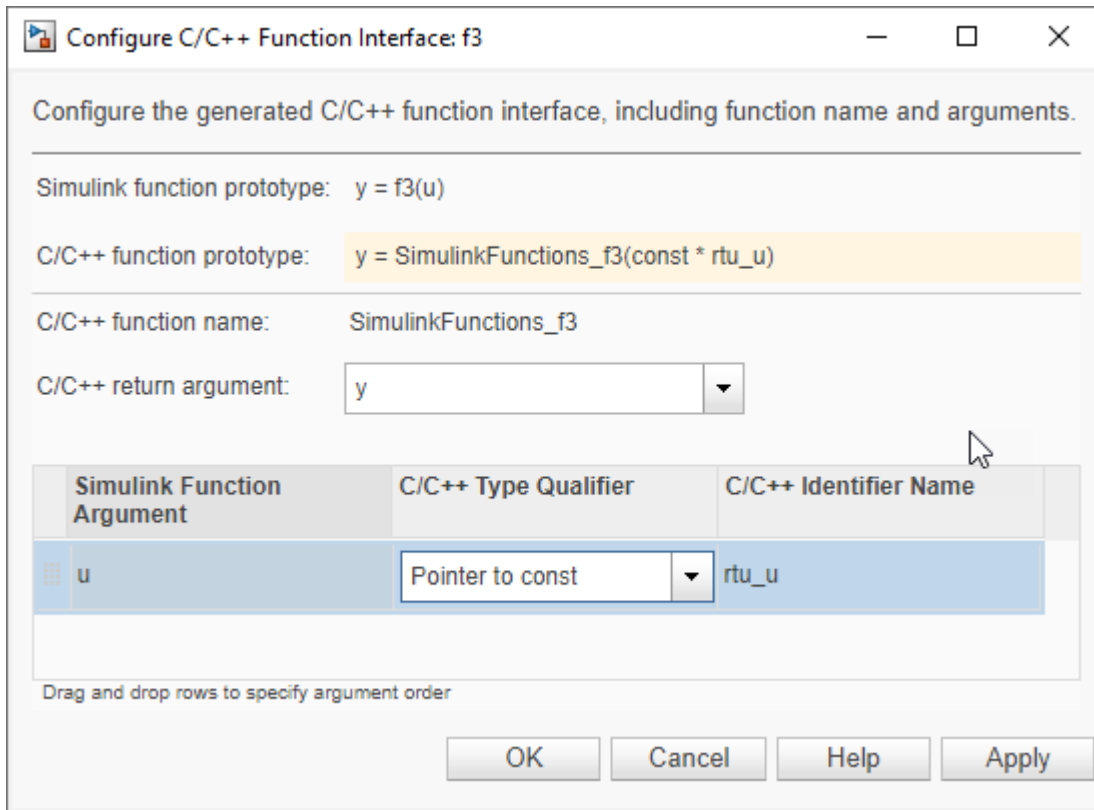
If model configuration parameter **Code interface packaging** is set to `Reusable function` and **Total number of instances allowed per top model** is set to `Multiple`, the argument appears as `self`, indicating that the code generator produces a `self` structure.

If model configuration parameter **Code interface packaging** is set to `Reusable function` or **Total number of instances allowed per top model** is set to `Multiple`, the argument appears as `[self]`, indicating that the argument is optional. The code generator produces a `self` structure only if you use the model as a top model and **Code interface packaging** is set to `Reusable function` or if you use the model as a referenced model and **Total number of instances allowed per top model** is set to `Multiple`.

For more information about configuring a model for multi-instantiable code generation, see “Generate Reentrant Code from Simulink Function Blocks” (Embedded Coder).

Configure Simulink Function Prototype

Configure the function arguments. For this example, configure the `u` argument. Set **C/C++ type Qualifier** to `Pointer to const`.



Verify your change by checking the function prototype preview.

Click **Apply**. Your modifications, whether made to a Simulink Function block or a Function Caller block, affect code generation for the Simulink Function block and corresponding Function Caller blocks in the model.

Optionally, you can change **C/C++ return argument** from `y` to `void`. In that case, the C function prototype is `void = SimulinkFunctions_f3(* rty_y, const * rtu_u)`.

Save your changes by clicking **OK**. Then, save the model.

Generate and Review Code

Make sure that the Simulink Function block is not selected. Then, generate code for the model.

In the **Code** view, open the generated file `SimulinkFunctions.c` and search for `SimulinkFunctions_f3`. The generated function code reflects the changes to the generated C function prototype.

```
real_T SimulinkFunctions_f3(const real_T *rtu_u)
{
    real_T rty_y_0;

    rtY.TicToc10 = rtDWork.Delay_DSTATE;

    rtDWork.Delay_DSTATE = (int8_T)(int32_T)-(int32_T)rtY.TicToc10;

    adder(rtB.Subtract, rtU.U2, *rtu_u, &rtB.FunctionCaller);
}
```

```
    rty_y_0 = rtB.FunctionCaller;  
    return rty_y_0;  
}
```

More About

- “Generate Reentrant Code from Simulink Function Blocks” (Embedded Coder)
- “Simulink Functions Overview”
- “Scoped Simulink Function Blocks in Models”
- “Simulink Function Blocks in Referenced Models”

Configure Generated C++ Function Interface for Exported Scoped Simulink Function Block

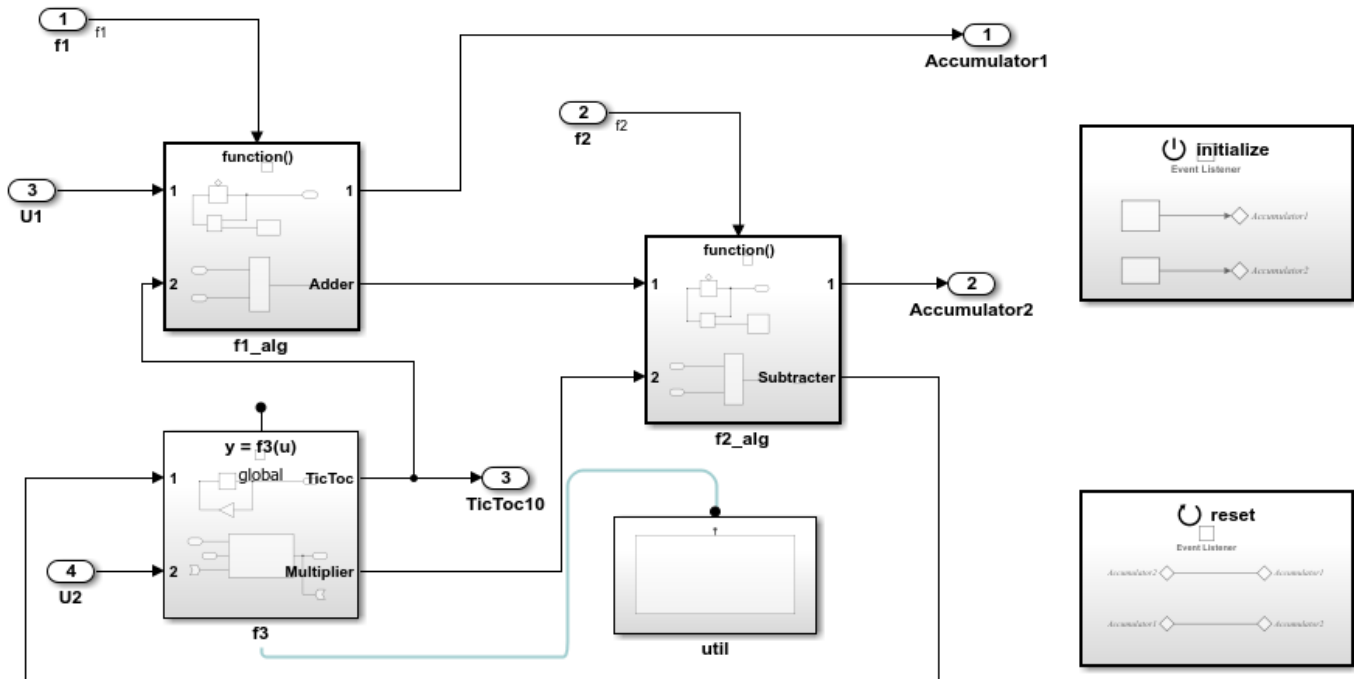
This example shows how to customize the C++ function interface that the code generator produces for an exported, scoped Simulink Function block.

For an exported, scoped function, you can modify the generated return argument, the argument type qualifiers, and the order of arguments. You cannot change the generated function name and argument names.

- 1 Open the example model.
- 2 Change the visibility of the Simulink function.
- 3 Change the programming language setting for code generation to C++.
- 4 Open the Configure C/C++ Function Interface dialog box.
- 5 Configure the Simulink Function prototype.
- 6 Generate and review the code.

Open Example Model

Open example model `SimulinkFunctions`. Save a copy of the model to a writable location.



Change Visibility of Simulink Function

Change the visibility of the global function f3 from global to scoped. Open the Simulink Function block f3. Inside the function block, double-click the Trigger Port block f3. In the block parameters dialog box, set **Function visibility** to scoped. Click **Apply** and **OK**.

Change Language Setting for Code Generation

Set the model configuration parameter **Language** to C++.

Open Configure C/C++ Function Interface Dialog Box

In the Simulink Editor, right-click the Simulink Function block f3. In the context menu, select **C/C++ Code > Configure C/C++ Function Interface**. The dialog box shows **Simulink function prototype** as $y = f3(u)$ and **C/C++ function prototype** as $y = \text{rtwdemo_functions_f3}(\text{rtu_u})$. The preview for **C/C++ function prototype** is updated as you make changes.

In the Code Mappings editor, click the **Functions** tab.

In the row for Simulink Function:f3, click the function preview hyperlink. The Configure C/C++ Function Interface dialog box shows **Simulink function prototype** as $y = f3(u)$ and **C/C++ function prototype** as $\text{void f3}(\text{rtu_u}, * \text{rty_y})$. The preview for **C/C++ function prototype** is updated as you make changes.

Configure the generated C/C++ function interface, including function name and arguments.

Simulink function prototype: `y = f3(u)`

C/C++ function prototype: `y = SimulinkFunctions_f3(rtu_u)`

C/C++ function name: `SimulinkFunctions_f3`

C/C++ return argument: `y`

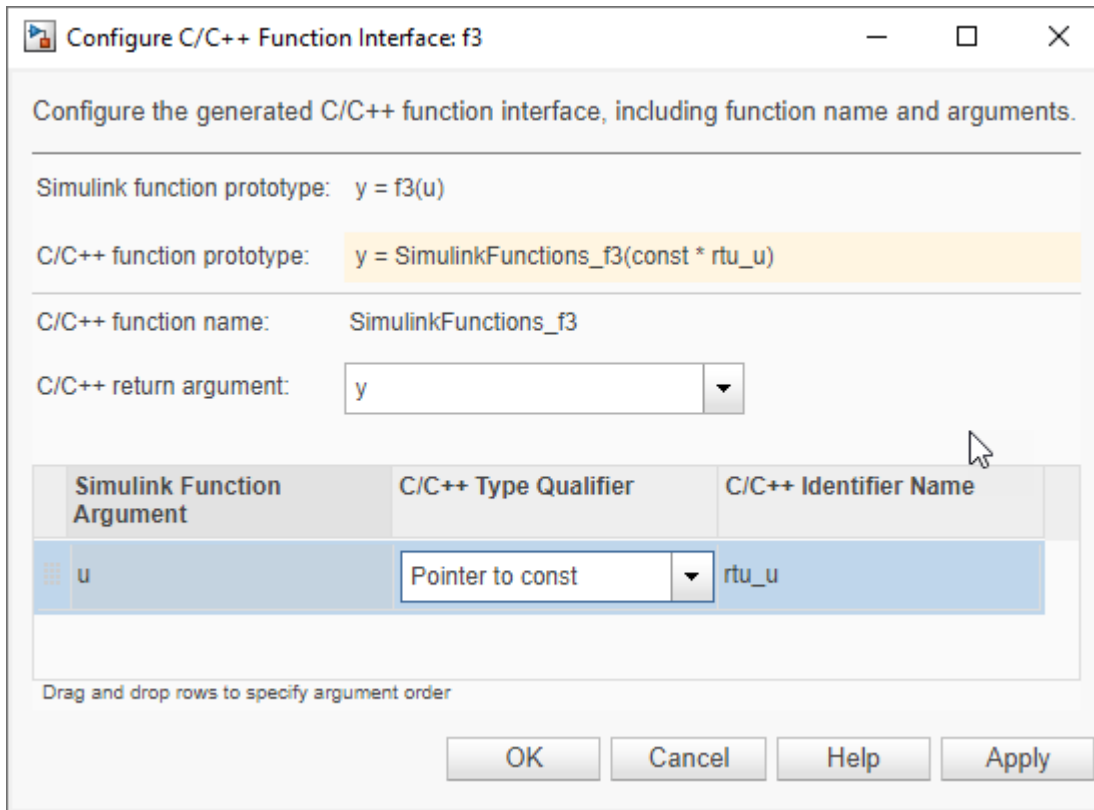
Simulink Function Argument	C/C++ Type Qualifier	C/C++ Identifier Name
u	Auto	rtu_u

Drag and drop rows to specify argument order

OK Cancel Help Apply

Configure Simulink Function Prototype

Configure the function arguments. For this example, configure the `u` argument. Set **C/C++ type Qualifier** to `Pointer to const`.



Verify your change by checking the function prototype preview.

Click **Apply**. Your modifications, whether made to a Simulink Function block or a Function Caller block, affect code generation for the Simulink Function block and corresponding Function Caller blocks in the model.

Optionally, you can change **C/C++ return argument** from `y` to `void`. In that case, the C++ function prototype is `void = SimulinkFunctions_f3(* rty_y, const * rtu_u)`.

Save your changes by clicking **OK**. Then, save the model.

Generate and Review Code

Make sure that the Simulink Function block is not selected. Then, generate code for the model.

In the **Code** view, open the generated file `SimulinkFunctions.cpp` and search for function `f3`. The generated function code reflects the changes to the generated C++ function prototype.

```
real_T SimulinkFunctionsModelClass::f3(const real_T *rtu_u)
{
    real_T rty_y_0;

    rtY.TicToc10 = rtDWork.Delay_DSTATE;

    rtDWork.Delay_DSTATE = static_cast<int8_T>(static_cast<int32_T>
        (-static_cast<int32_T>(rtY.TicToc10)));
```

```
    adder(rtB.Subtract, rtU.U2, *rtu_u, &rtB.FunctionCaller);  
    rty_y_0 = rtB.FunctionCaller;  
    return rty_y_0;  
}
```

More About

- “Generate Reentrant Code from Simulink Function Blocks” (Embedded Coder)
- “Simulink Functions Overview”
- “Scoped Simulink Function Blocks in Models”
- “Simulink Function Blocks in Referenced Models”

Simulink Function Code Interface Limitations

- For C++ code generation, global Simulink functions are incompatible with C++ class interfaces for model entry-point functions.
- Simulink function code interface configuration does not support Simulink functions and function callers in Stateflow.
- The Argument Outport block cannot be a return argument for a Simulink function when its data type is set to one of the following types:
 - Complex types
 - Bus types
 - Image types
 - Non-scalar types
 - Multiword types

See Also

Related Examples

- “Simulink Function Blocks and Code Generation” on page 10-6
- “Generate Reentrant Code from Simulink Function Blocks” on page 10-18

Generate C++ Class Interface to Model or Subsystem Code

To generate a C++ class interface to model code, set model configuration parameter Code interface packaging to C++ class. The generated interface encapsulates required model data into C++ class attributes and model entry point functions into C++ class methods. The benefits of C++ class encapsulation include:

- Greater control over access to model data
- Ability to create multiple instances of model classes
- Easier integration of model code into C++ programming environments

C++ class encapsulation also works for right-click builds of nonvirtual subsystems. (For information on requirements that apply, see “Generate C++ Class Interface to Nonvirtual Subsystem Code” on page 17-16.)

Generate C++ Class Interface to Model Code

To generate encapsulated C++ class code from a model:

- 1 Set model configuration parameter **Language** to C++. This selection also enables C++ class code interface packaging for the model.
- 2 Verify that model configuration parameter **Code interface packaging** is set to C++ class.
- 3 Examine the setting of **Multi-instance code error diagnostic**. Leave the parameter at its default value Error unless you need to alter the severity level for diagnostics displayed when a model violates requirements for generating multi-instance code.



- 4 Generate code for the model.
- 5 Examine the C++ model class code in the generated files *model.h* and *model.cpp*. For example, the following code excerpt from the H file generated for the example model *SecondOrderSystem* shows the C++ class declaration for the model.

```
/* Class declaration for model SecondOrderSystem */
class SecondOrderSystemModelClass {
  /* public data and function members */
public:
  /* External outputs */
  ExtY_SecondOrderSystem_T SecondOrderSystem_Y;

  /* Model entry point functions */

  /* model initialize function */
  void initialize();

  /* model step function */
  void step();

  /* model terminate function */
  void terminate();

  /* Constructor */
  SecondOrderSystemModelClass();

  /* Destructor */
  ~SecondOrderSystemModelClass();
};
```

```

/* Real-Time Model get method */
RT_MODEL_SecondOrderSystem_T * getRTM();
...
};

```

For more information about generating and calling model entry-point functions, see “Configure Generated C Function Interface for Model Entry-Point Functions” on page 16-79.

Note If you have an Embedded Coder license and you have selected an ERT-based system target file for your model, use additional **Code Generation > Interface** pane parameters to customize the generated C++ class interface.

Generate C++ Class Interface to Nonvirtual Subsystem Code

You can generate C++ class interfaces for right-click builds of nonvirtual subsystems, if the following requirements are met:

- The model is configured for the C++ language and C++ class code interface packaging.
- The subsystem is convertible to a Model block using the function `Simulink.SubSystem.convertToModelReference`. For referenced model conversion requirements, see the Simulink reference page `Simulink.SubSystem.convertToModelReference`.

To configure C++ class interfaces for a subsystem that meets the requirements:

- 1 Open the containing model and select the subsystem block.
- 2 Right-click the subsystem and select **C/C++ Code > Build This Subsystem**.
- 3 When the subsystem build completes, examine the C++ class interfaces in the generated files and the HTML code generation report. For more information about generating and calling model entry-point methods, see “Configure Generated C Function Interface for Model Entry-Point Functions” on page 16-79.

If you have an Embedded Coder license and you have selected an ERT-based system target file for your model, you can use the MATLAB command `RTW.configSubsystemBuild` to customize the generated C++ class interface to subsystem code.

C++ Class Interface Limitations

- Among the data exchange interfaces available on the **Interface** pane of the Configuration Parameters dialog box, only the C API interface is supported for C++ class code generation. If you select **External mode** or **ASAP2 interface**, code generation fails with a validation error.
- If a model root inport value connects to a Simscape conversion block, you must insert a Simulink Signal Conversion block between the root inport and the Simscape conversion block. On the Simulink Signal Conversion block parameter dialog box, select **Exclude this block from 'Block reduction' optimization**.
- You cannot use a C++ class interface in cases when a referenced model cannot have a combined output/update function. Cases include a model that
 - Has a continuous sample time
 - Saves states

See Also

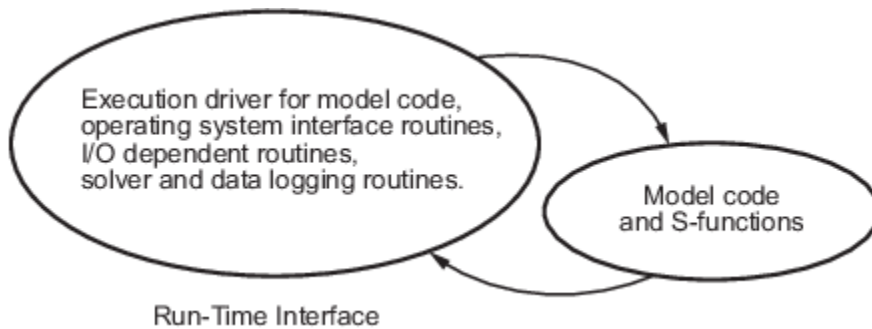
More About

- “Generate Reentrant Code from Top Models” on page 10-13
- “Configure Generated C Function Interface for Model Entry-Point Functions” on page 16-79
- “Execution of Code Generated from a Model” on page 17-18
- “Rapid Prototyping Model Functions” on page 17-27

Execution of Code Generated from a Model

The code generator produces algorithmic code as defined by your model. You can include external (for example, custom or legacy) code in a model by using techniques explained in “Choose an External Code Integration Workflow” on page 30-2.

The code generator also provides an interface that executes the generated model code. The interface and model code are compiled together to create an executable program. The next figure shows a high-level object-oriented view of the executable.



The Object-Oriented View of a Real-Time Program

In general, the conceptual design of the model execution driver does not change between the rapid prototyping and embedded style of generated code. The following sections describe model execution for single-tasking and multitasking environments both for simulation (non-real-time) and for real time. For most model code, the multitasking environment provides the most efficient model execution (that is, fastest sample rate).

The following concepts are useful in describing how model code executes.

- **Initialization:** *model_initialize* initializes the interface code and the model code.
- **ModelOutputs:** Calls blocks in your model that have a sample hit at the current time and has them produce their output. *model_output* can be done in major or minor time steps. In major time steps, the output is a given simulation time step. In minor time steps, the interface integrates the derivatives to update the continuous states.
- **ModelUpdate:** *model_update* calls blocks that have a sample hit at the current point in time and has them update their discrete states or similar type objects.
- **ModelDerivatives:** Calls blocks in your model that have continuous states and has them update their derivatives. *model_derivatives* is only called in minor time steps.
- **ModelTerminate:** *model_terminate* terminates the program if it is designed to run for a finite time. It destroys the real-time model data structure, deallocates memory, and can write data to a file.

Program Execution

A real-time program cannot require 100% of the CPU time. This requirement provides an opportunity to run background tasks during the free time.

Background tasks include operations such as writing data to a buffer or file, allowing access to program data by third-party data monitoring tools, or updating program parameters.

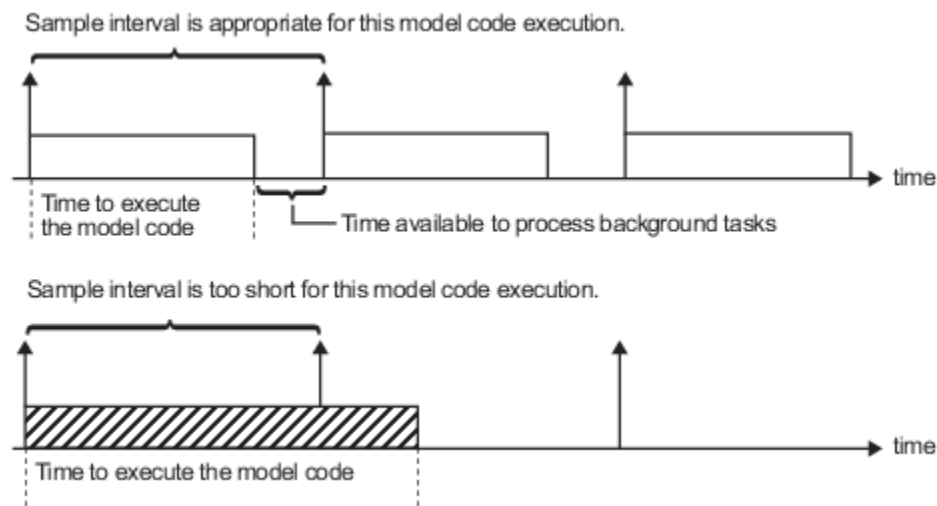
It is important, however, that the program be able to preempt the background task so the model code can execute in real time.

The way the program manages tasks depends on capabilities of the environment in which it operates.

Program Timing

Real-time programs require careful timing of the task invocations (either by using an interrupt or a real-time operating system tasking primitive) so that the model code executes to completion before another task invocation occurs. The timing includes time to read and write data to and from external hardware.

The next figure illustrates interrupt timing.



Task Timing

The sample interval must be long enough to allow model code execution between task invocations.

In the figure above, the time between two adjacent vertical arrows is the sample interval. The empty boxes in the upper diagram show an example of a program that can complete one step within the interval and still allow time for the background task. The gray box in the lower diagram indicates what happens if the sample interval is too short. Another task invocation occurs before the task is complete. Such timing results in an execution error.

If the real-time program is designed to run forever (that is, the final time is 0 or infinite so that the `while` loop never exits), then the shutdown code does not execute.

For more information on how the timing engine works, see "Absolute and Elapsed Time Computation" on page 4-2.

External Mode Communication

External mode allows communication between the Simulink block diagram and the standalone program that is built from the generated code. In this mode, the real-time program functions as an interprocess communication server, responding to requests from the Simulink engine.

Data Logging in Single-Tasking and Multitasking Model Execution

“Configure Model for Debugging” on page 20-34 explains how you can save system states, outputs, and time to a MAT-file at the completion of the model execution. The LogTXY function, which performs data logging, operates differently in single-tasking and multitasking environments.

If you examine how LogTXY is called in the single-tasking and multitasking environments, notice that for single-tasking LogTXY is called after ModelOutputs. During this ModelOutputs call, blocks that have a hit at time t execute, whereas in multitasking, LogTXY is called after ModelOutputs(tid=0), which executes only the blocks that have a hit at time t and that have a task identifier of 0. This results in differences in the logged values between single-tasking and multitasking logging. Specifically, consider a model with two sample times, the faster sample time having a period of 1.0 second and the slower sample time having a period of 10.0 seconds. At time $t = k*10$, $k=0,1,2\dots$ both the fast (tid=0) and slow (tid=1) blocks execute. When executing in multitasking mode, when LogTXY is called, the slow blocks execute, but the previous value is logged, whereas in single-tasking the current value is logged.

Another difference occurs when logging data in an enabled subsystem. Consider an enabled subsystem that has a slow signal driving the enable port and fast blocks within the enabled subsystem. In this case, the evaluation of the enable signal occurs in a slow task, and the fast blocks see a delay of one sample period; thus the logged values will show these differences.

To summarize differences in logged data between single-tasking and multitasking, differences will be seen when

- A root output block has a sample time that is slower than the fastest sample time
- A block with states has a sample time that is slower than the fastest sample time
- A block in an enabled subsystem where the signal driving the enable port is slower than the rate of the blocks in the enabled subsystem

For the first two cases, even though the logged values are different between single-tasking and multitasking, the model results are not different. The only real difference is where (at what point in time) the logging is done. The third (enabled subsystem) case results in a delay that can be seen in a real-time environment.

Non-Real-Time Single-Tasking Systems

This pseudocode shows the execution of a model for a non-real-time single-tasking system.

```
main()
{
  Initialization
  While (time < final time)
    ModelOutputs      -- Major time step.
    LogTXY            -- Log time, states and root outputs.
    ModelUpdate       -- Major time step.
    Integrate         -- Integration in minor time step for
                      -- models with continuous states.

    ModelDerivatives
    Do 0 or more
      ModelOutputs
      ModelDerivatives
    EndDo -- Number of iterations depends upon the solver
    Integrate derivatives to update continuous states.
```



```

    EndIntegrate
  EndWhile
  Termination
}

```

The initialization phase begins first. This consists of initializing model states and setting up the execution engine. The model then executes, one step at a time. First `ModelOutputs` executes at time t , then the workspace I/O data is logged, and then `ModelUpdate` updates the discrete states. Next, if your model has continuous states, `ModelDerivatives` integrates the continuous states' derivatives to generate the states for time $t_{new} = t + h$, where h is the step size. Time then moves forward to t_{new} and the process repeats.

During the `ModelOutputs` and `ModelUpdate` phases of model execution, only blocks that reach the current point in time execute.

Non-Real-Time Multitasking Systems

This pseudocode shows the execution of a model for a non-real-time multitasking system.

```

main()
{
  Initialization
  While (time < final time)
    ModelOutputs(tid=0)  -- Major time step.
    LogTXY               -- Log time, states, and root
                        -- outputs.
    ModelUpdate(tid=0)   -- Major time step.
    Integrate            -- Integration in minor time step for
                        -- models with continuous states.
    ModelDerivatives
    Do 0 or more
      ModelOutputs(tid=0)
      ModelDerivatives
    EndDo (Number of iterations depends upon the solver.)
    Integrate derivatives to update continuous states.
  EndIntegrate
  For i=1:NumTids
    ModelOutputs(tid=i) -- Major time step.
    ModelUpdate(tid=i)  -- Major time step.
  EndFor
  EndWhile
  Termination
}

```

Multitasking operation is more complex than single-tasking execution because the output and update functions are subdivided by the *task identifier* (`tid`) that is passed into these functions. This allows for multiple invocations of these functions with different task identifiers using overlapped interrupts, or for multiple tasks when using a real-time operating system. In simulation, multiple tasks are emulated by executing the code in the order that would occur if preemption did not exist in a real-time system.

Multitasking execution assumes that task rates are multiples of the base rate. The Simulink product enforces this when you create a fixed-step multitasking model. The multitasking execution loop is very similar to that of single-tasking, except for the use of the task identifier (`tid`) argument to `ModelOutputs` and `ModelUpdate`.

You cannot use `tid` values from code generated by a target file and not by Simulink Coder. Simulink Coder tracks the use of `tid` when generating code for a specific subsystem or function type. When you generate code in a target file, this argument cannot be tracked because the scope does not have subsystem or function type. Therefore, `tid` becomes an undefined variable and your target file fails to compile.

Real-Time Single-Tasking Systems

This pseudocode shows the execution of a model in a real-time single-tasking system where the model is run at interrupt level.

```
rtOneStep()
{
    Check for interrupt overflow
    Enable "rtOneStep" interrupt
    ModelOutputs    -- Major time step.
    LogTXY         -- Log time, states and root outputs.
    ModelUpdate    -- Major time step.
    Integrate      -- Integration in minor time step for models
                  -- with continuous states.
        ModelDerivatives
        Do 0 or more
            ModelOutputs
            ModelDerivatives
        EndDo (Number of iterations depends upon the solver.)
        Integrate derivatives to update continuous states.
    EndIntegrate
}

main()
{
    Initialization (including installation of rtOneStep as an
    interrupt service routine, ISR, for a real-time clock).
    While(time < final time)
        Background task.
    EndWhile
    Mask interrupts (Disable rtOneStep from executing.)
    Complete any background tasks.
    Shutdown
}
```

Real-time single-tasking execution is very similar to non-real-time single-tasking execution, except that instead of free-running the code, the `rt_OneStep` function is driven by a periodic timer interrupt.

At the interval specified by the program's base sample rate, the interrupt service routine (ISR) preempts the background task to execute the model code. The base sample rate is the fastest in the model. If the model has continuous blocks, then the integration step size determines the base sample rate.

For example, if the model code is a controller operating at 100 Hz, then every 0.01 seconds the background task is interrupted. During this interrupt, the controller reads its inputs from the analog-to-digital converter (ADC), calculates its outputs, writes these outputs to the digital-to-analog converter (DAC), and updates its states. Program control then returns to the background task. These steps must occur before the next interrupt.

Real-Time Multitasking Systems

This pseudocode shows how a model executes in a real-time multitasking system where the model is run at interrupt level.

```

rtOneStep()
{
  Check for interrupt overflow
  Enable "rtOneStep" interrupt
  ModelOutputs(tid=0)    -- Major time step.
  LogTXY                 -- Log time, states and root outputs.
  ModelUpdate(tid=0)     -- Major time step.
  Integrate              -- Integration in minor time step for
                        -- models with continuous states.

  ModelDerivatives
  Do 0 or more
    ModelOutputs(tid=0)
    ModelDerivatives
  EndDo (Number of iterations depends upon the solver.)
  Integrate derivatives and update continuous states.
EndIntegrate
For i=1:NumTasks
  If (hit in task i)
    ModelOutputs(tid=i)
    ModelUpdate(tid=i)
  EndIf
EndFor
}

main()
{
  Initialization (including installation of rtOneStep as an
  interrupt service routine, ISR, for a real-time clock).
  While(time < final time)
    Background task.
  EndWhile
  Mask interrupts (Disable rtOneStep from executing.)
  Complete any background tasks.
  Shutdown
}

```

Running models at interrupt level in a real-time multitasking environment is very similar to the previous single-tasking environment, except that overlapped interrupts are employed for concurrent execution of the tasks.

The execution of a model in a single-tasking or multitasking environment when using real-time operating system tasking primitives is very similar to the interrupt-level examples discussed above. The pseudocode below is for a single-tasking model using real-time tasking primitives.

```

tSingleRate()
{
  MainLoop:
  If clockSem already "given", then error out due to overflow.
  Wait on clockSem
  ModelOutputs          -- Major time step.
  LogTXY                -- Log time, states and root
                        -- outputs
}

```

```

    ModelUpdate           -- Major time step
    Integrate             -- Integration in minor time step
                        -- for models with continuous
                        -- states.

    ModelDerivatives
    Do 0 or more
        ModelOutputs
        ModelDerivatives
    EndDo (Number of iterations depends upon the solver.)
    Integrate derivatives to update continuous states.
EndIntegrate
EndMainLoop
}

main()
{
    Initialization
    Start/spawn task "tSingleRate".
    Start clock that does a "semGive" on a clockSem semaphore.
    Wait on "model-running" semaphore.
    Shutdown
}

```

In this single-tasking environment, the model executes as real-time operating system tasking primitives. In this environment, create a single task (tSingleRate) to run the model code. This task is invoked when a clock tick occurs. The clock tick gives a clockSem (clock semaphore) to the model task (tSingleRate). The model task waits for the semaphore before executing. The clock ticks occur at the fundamental step size (base rate) for your model.

Multitasking Systems Using Real-Time Tasking Primitives

This pseudocode is for a multitasking model using real-time tasking primitives.

```

tSubRate(subTaskSem,i)
{
    Loop:
        Wait on semaphore subTaskSem.
        ModelOutputs(tid=i)
        ModelUpdate(tid=i)
    EndLoop
}

tBaseRate()
{
    MainLoop:
        If clockSem already "given", then error out due to overflow.
        Wait on clockSem
        For i=1:NumTasks
            If (hit in task i)
                If task i is currently executing, then error out due to
                overflow.
                Do a "semGive" on subTaskSem for task i.
            EndIf
        EndFor
        ModelOutputs(tid=0)    -- major time step.
        LogTXY                 -- Log time, states and root outputs.
        ModelUpdate(tid=0)     -- major time step.
        Loop:                  -- Integration in minor time step for
}

```

```

-- models with continuous states.
ModelDerivatives
Do 0 or more
  ModelOutputs(tid=0)
  ModelDerivatives
EndDo (number of iterations depends upon the solver).
Integrate derivatives to update continuous states.
EndLoop
EndMainLoop
}
main()
{
  Initialization
  Start/spawn task "tSubRate".
  Start/spawn task "tBaseRate".

  Start clock that does a "semGive" on a clockSem semaphore.
  Wait on "model-running" semaphore.
  Shutdown
}

```

In this multitasking environment, the model is executed using real-time operating system tasking primitives. Such environments require several model tasks (tBaseRate and several tSubRate tasks) to run the model code. The base rate task (tBaseRate) has a higher priority than the subrate tasks. The subrate task for tid=1 has a higher priority than the subrate task for tid=2, and so on. The base rate task is invoked when a clock tick occurs. The clock tick gives a clockSem to tBaseRate. The first thing tBaseRate does is give semaphores to the subtasks that have a hit at the current point in time. Because the base rate task has a higher priority, it continues to execute. Next it executes the fastest task (tid=0), consisting of blocks in your model that have the fastest sample time. After this execution, it resumes waiting for the clock semaphore. The clock ticks are configured to occur at the fundamental step size for your model.

Rapid Prototyping and Embedded Model Execution Differences

The rapid prototyping program framework provides a common application programming interface (API) that does not change between model definitions.

The Embedded Coder product provides a different framework called the embedded program framework. The embedded program framework provides an optimized API that is tailored to your model. When you use the embedded style of generated code, you are modeling how you would like your code to execute in your embedded system. Therefore, the definitions defined in your model should be specific to your embedded targets. Items such as the model name, parameter, and signal storage class are included as part of the API for the embedded style of code.

One major difference between the rapid prototyping and embedded style of generated code is that the latter contains fewer entry-point functions. The embedded style of code can be configured to have only one function, *model_step*.

Thus, model execution code eliminates Loop...EndLoop statements and groups ModelOutputs, LogTXY, and ModelUpdate into a single statement, *model_step*.

For more information about how generated embedded code executes, see "Configure Generated C Function Interface for Model Entry-Point Functions" on page 16-79.

See Also

More About

- “Time-Based Scheduling and Code Generation” (Embedded Coder)
- “Sample Times in Subsystems”
- “Sample Times in Systems”
- “Time-Based Scheduling Example Models” (Embedded Coder)

Rapid Prototyping Model Functions

Rapid prototyping code defines the following functions that interface with the main program (`main.c` or `main.cpp`):

- `Model()`: The model registration function. This function initializes the work areas (for example, allocating and setting pointers to various data structures) used by the model. The model registration function calls the `MdlInitializeSizes` and `MdlInitializeSampleTimes` functions. These two functions are very similar to the S-function `mdlInitializeSizes` and `mdlInitializeSampleTimes` methods.
- `MdlStart(void)`: After the model registration functions `MdlInitializeSizes` and `MdlInitializeSampleTimes` execute, the main program starts execution by calling `MdlStart`. This routine is called once at startup.

The function `MdlStart` has four basic sections:

- Code to initialize the states for each block in the root model that has states. A subroutine call is made to the “initialize states” routines of conditionally executed subsystems.
- Code generated by the one-time initialization (start) function for each block in the model.
- Code to enable the blocks in the root model that have enable methods, and the blocks inside triggered or function-call subsystems residing in the root model. Simulink blocks can have enable and disable methods. An enable method is called just before a block starts executing, and the disable method is called just after the block stops executing.
- Code for each block in the model whose output value is constant. The block code appears in the `MdlStart` function only if the block parameters are not tunable in the generated code and if the code generator cannot eliminate the block code through constant folding.
- `MdlOutputs(int_T tid)`: `MdlOutputs` updates the output of blocks. The `tid` (task identifier) parameter identifies the task that in turn maps when to execute blocks based upon their sample time. This routine is invoked by the main program during major and minor time steps. The major time steps are when the main program is taking an actual time step (that is, it is time to execute a specific task). If your model contains continuous states, the minor time steps will be taken. The minor time steps are when the solver is generating integration stages, which are points between major outputs. These integration stages are used to compute the derivatives used in advancing the continuous states.
- `MdlUpdate(int_T tid)`: `MdlUpdate` updates the states and work vector state information (that is, states that are neither continuous nor discrete) saved in work vectors. The `tid` (task identifier) parameter identifies the task that in turn indicates which sample times are active, allowing you to conditionally update only states of active blocks. This routine is invoked by the interface after the major `MdlOutputs` has been executed. The solver is also called, and `model_Derivatives` is called in minor steps by the solver during its integration stages. All blocks that have continuous states have an identical number of derivatives. These blocks are required to compute the derivatives so that the solvers can integrate the states.
- `MdlTerminate(void)`: `MdlTerminate` contains any block shutdown code. `MdlTerminate` is called by the interface, as part of the termination of the real-time program.

The contents of the above functions are directly related to the blocks in your model. A Simulink block can be generalized to the following set of equations.

$$y = f_0(t, x_c, x_d, u)$$

Output y is a function of continuous state x_c , discrete state x_d , and input u . Each block writes its specific equation in a section of `MdlOutputs`.

$$x_{d+1} = f_u(t, x_d, u)$$

The discrete states x_d are a function of the current state and input. Each block that has a discrete state updates its state in `MdlUpdate`.

$$\dot{x} = f_d(t, x_c, u)$$

The derivatives \dot{x} are a function of the current input. Each block that has continuous states provides its derivatives to the solver (for example, `ode5`) in `model_Derivatives`. The derivatives are used by the solver to integrate the continuous state to produce the next value.

The output, y , is generally written to the block I/O structure. Root-level Output blocks write to the external outputs structure. The continuous and discrete states are stored in the states structure. The input, u , can originate from another block's output, which is located in the block I/O structure, an external input (located in the external inputs structure), or a state. These structures are defined in the `model.h` file that the Simulink Coder software generates.

The next example shows the general contents of the rapid prototyping style of C code written to the `model.c` file.


```
/*
 * Version, Model options, TLC options,
 * and code generation information are placed here.
 */
<includes>
void MdlStart(void)
{
    /*
     * State initialization code.
     * Model start-up code - one time initialization code.
     * Execute any block enable methods.
     * Initialize output of any blocks with constant sample times.
     */
}

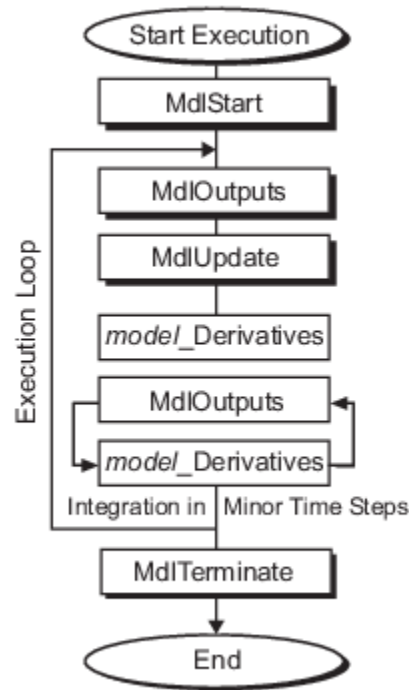
void MdlOutputs(int_T tid)
{
    /* Compute:  $y = f_0(t, x_c, x_d, u)$  for each block as needed. */
}

void MdlUpdate(int_T tid)
{
    /* Compute:  $x_{d+1} = f_u(t, x_d, u)$  for each block as needed. */

    /* Compute:  $dx_c = f_d(t, x_c, u)$  for each block in model_derivatives
        as needed. */
}

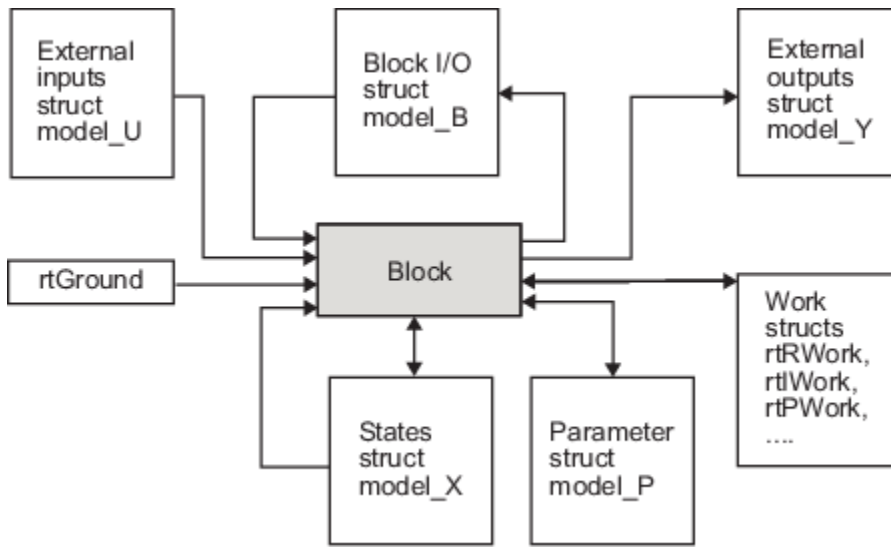
void MdlTerminate(void)
{
    /* Perform shutdown code for any blocks that
        have a termination action */
}
}
```

This figure shows a flow chart describing the execution of the rapid prototyping generated code.



Rapid Prototyping Execution Flow Chart

Each block places code in specific Mdl routines according to the algorithm that it is implementing. Blocks have input, output, parameters, and states, as well as other general items. For example, in general, block inputs and outputs are written to a block I/O structure (*model_B*). Block inputs can also come from the external input structure (*model_U*) or the state structure when connected to a state port of an integrator (*model_X*), or ground (*rtGround*) if unconnected or grounded. Block outputs can also go to the external output structure (*model_Y*). This figure shows the general mapping between these items.



Data View of the Generated Code

The following list defines the structures shown in the preceding figure:

- Block I/O structure (*model_B*): This structure consists of persistent block output signals. The number of block output signals is the sum of the widths of the data output ports of the nonvirtual blocks in your model. If you activate block I/O optimizations, the Simulink and Simulink Coder products reduce the size of the *model_B* structure by
 - Reusing the entries in the *model_B* structure
 - Making other entries local variables

See “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49 for more information on these optimizations.

Structure field names are determined either by the block's output signal name (when present) or by the block name and port number when the output signal is left unlabeled.

- Block states structures: The continuous states structure (*model_X*) contains the continuous state information for blocks in your model that have continuous states. Discrete states are stored in a data structure called the *DWork vector* (*model_DWork*).
- Block parameters structure (*model_P*): The parameters structure contains block parameters that can be changed during execution (for example, the parameter of a Gain block).
- External inputs structure (*model_U*): The external inputs structure consists of the root-level Inport block signals. Field names are determined by either the block's output signal name, when present, or by the Inport block's name when the output signal is left unlabeled.
- External outputs structure (*model_Y*): The external outputs structure consists of the root-level Outport blocks. Field names are determined by the root-level Outport block names in your model.
- Real work, integer work, and pointer work structures (*model_RWork*, *model_IWork*, *model_PWork*): Blocks might have a need for real, integer, or pointer work areas. For example, the Memory block uses a real work element for each signal. These areas are used to save internal states or similar information.

See Also

More About

- “Time-Based Scheduling and Code Generation” (Embedded Coder)
- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Array Layout

Column-Major Layout to Row-Major Layout Conversion of Models with Lookup Table Blocks

Simulink® Coder™ supports row-major array layout for code generation. You can integrate existing applications that use row-major array layout with the generated code in row-major array layout. When you switch an existing model with lookup table (LUT) blocks from the column-major array layout to the row-major array layout, it is recommended to convert the LUT blocks from the column-major algorithm to the row-major algorithm. The code generated by using row-major algorithm performs with the best speed and memory usage when operating on table data with row-major array layout. The code generated by using column-major algorithm performs best with column-major array layout.

This example shows the workflow of converting a model with LUT blocks from column-major layout to row-major layout to achieve the best performance on a row-major array layout.

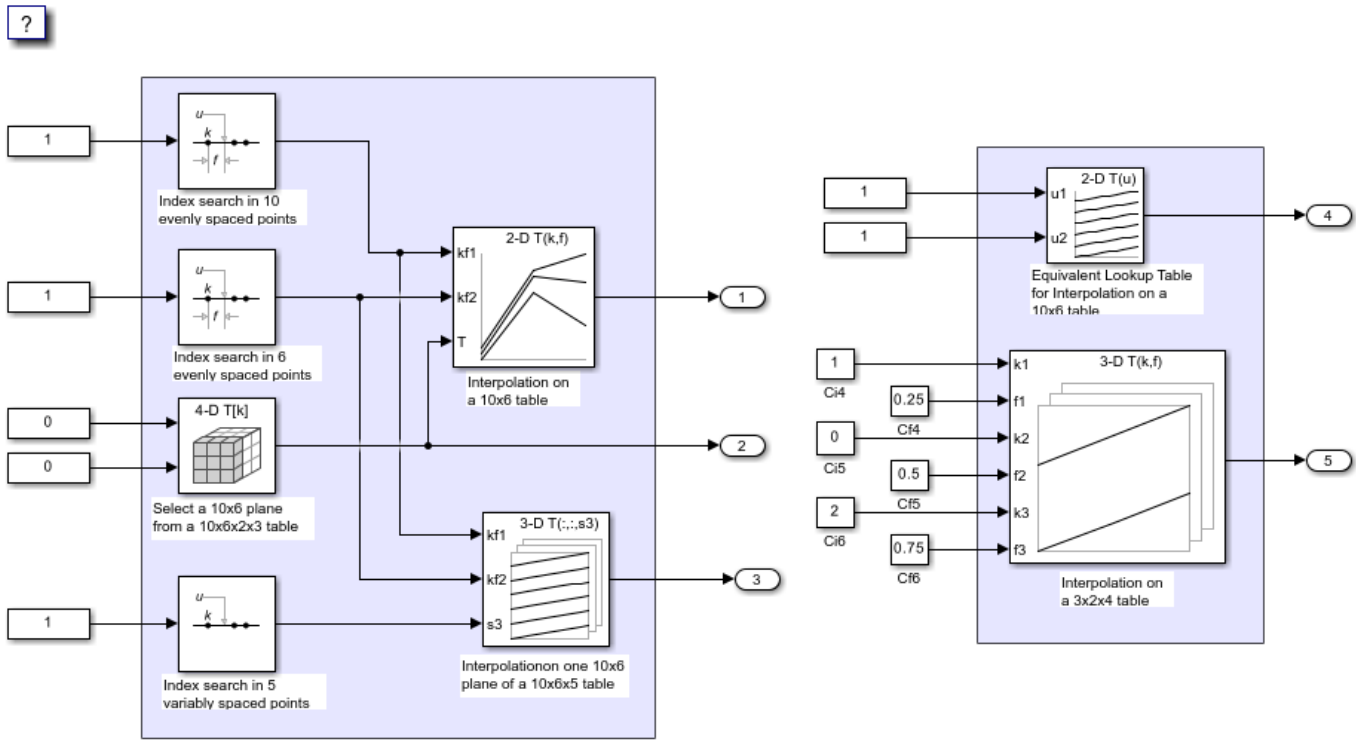
In this example, you:

- Identify the array layout and select the optimized algorithm.
- Preserve semantics through table permutation.
- Generate code by using a row-major algorithm and an array layout.

Simulate and Generate Code by Using Column-Major Algorithms

1. Open the example model RowLUTCo1ToRow.

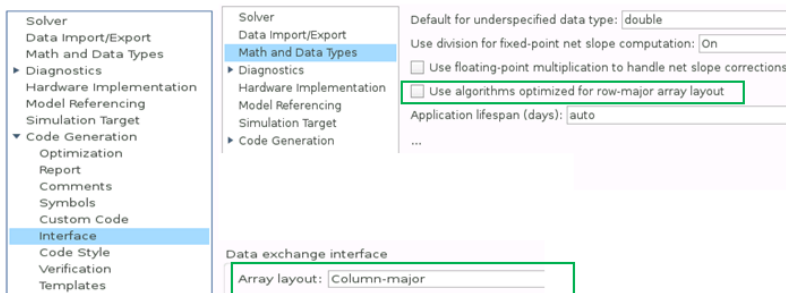
```
open_system('RowLUTCo1ToRow');
```



Copyright 2018-2022 The MathWorks, Inc.

By default, Simulink configures a model to use column-major algorithms and a column-major array layout. These parameters are the configuration parameters in the Configuration Parameters dialog box.

- Math and Data Types > “Use algorithms optimized for row-major array layout” — This parameter affects simulation and code generation.
- Code Generation > Interface > Array layout — This parameter affects only code generation.



2. On the **Modeling** tab, click **Run** to simulate the model and observe the output logged in workspace variable `yout`.

3. Change your current folder in MATLAB® to a writable folder. On the **C Code** tab, click **Build** to generate C code.

Select Optimized Algorithms for Row-Major Array Layout

Table data with row-major array layout is frequently used in the field of calibration. To interface the row-major table data with the existing model, update the column-major model to operate efficiently on row-major table data.

Use the algorithm that is optimized for the specified array layout to achieve the best performance. For example, use row-major algorithms when **Array layout** is set as Row-major during code generation.

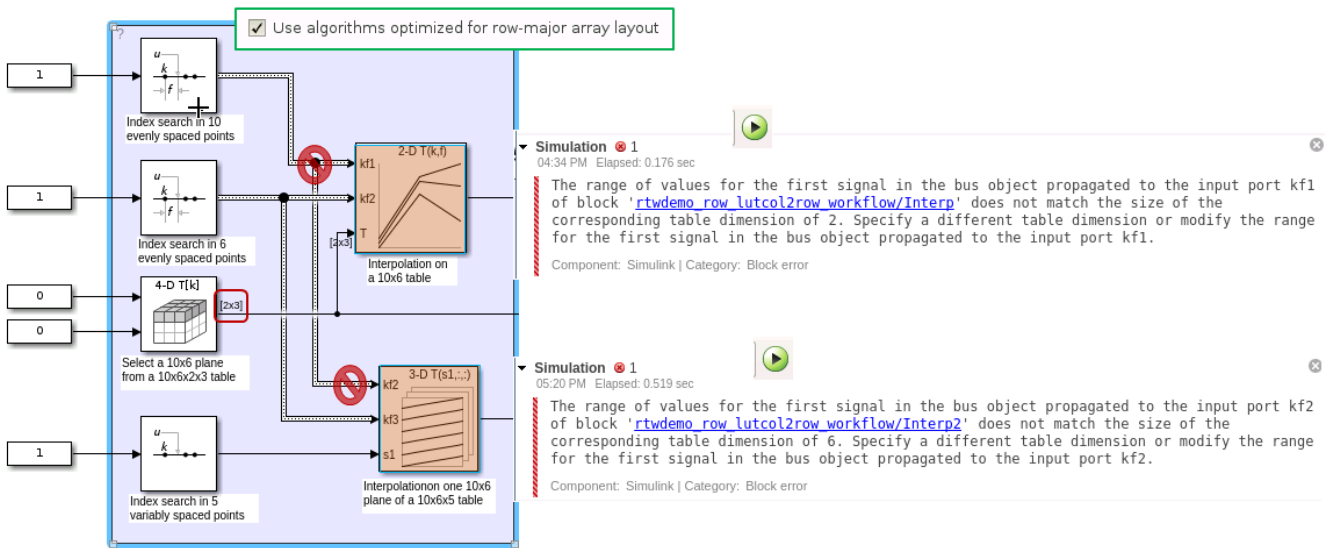
Array Layout	Algorithm	Cache-Friendly Algorithm
Column-major	Column-major	✓ (Recommended)
Row-major	Row-major	✓ (Recommended)
Row-major	Column-major	⚠ (Not recommended)
Column-major	Row-major	⚠ (Not recommended)

1. To enable row-major algorithms, open the Configuration Parameters dialog box. On the **Math and Data Types** pane, select the configuration parameter **Use algorithms optimized for row-major array layout**. Alternatively, in the MATLAB Command Window, enter:

```
set_param('RowLUTColToRow', 'UseRowMajorAlgorithm', 'on');
```

2. Click **Run** to simulate the model. Simulink reports errors because it encounters inconsistent breakpoint and table data between prelookup and interpolation blocks. The causes of this error are the two semantic changes that occur when you switch from column-major algorithms to row-major algorithms, that is, when you:

- Select a plane from a 3-D table in interpolation block.
- Select a plane from a 4-D table through a direct lookup table block.



Dimension of the Selected Plane (Semantic Changes)

	Column-major algorithm	Row-major algorithm
Select a plane from a 4-D table (10x6x2x3)	10x6	2x3
Select a plane from a 3-D table (10x6x5)	10x6	6x5

Preserve Semantics by Using Table Permutation

1. For subtable selection before interpolation, or direct lookup that outputs a vector or 2-D matrix, the model semantics change when you switch from a column-major algorithm to a row-major algorithm by selecting the configuration parameter **Use algorithms optimized for row-major array layout**. To preserve the semantics and fix the previous errors, permute the table data by using these commands:

```
T4d_str = get_param('RowLUTColToRow/Direct LUT', 'Table');
set_param('RowLUTColToRow/Direct LUT', 'Table', ...
['permute(', T4d_str, ', [3,4,1,2]')]);
```

```
T3d_str = get_param('RowLUTColToRow/Interp2', 'Table');
set_param('RowLUTColToRow/Interp2', 'Table', ...
['permute(', T3d_str, ', [3,1,2]')]);
```

2. Before you import table data from a file, you must permute the table data in the file. This permutation keeps the table tunable throughout the simulation and code generation workflow.

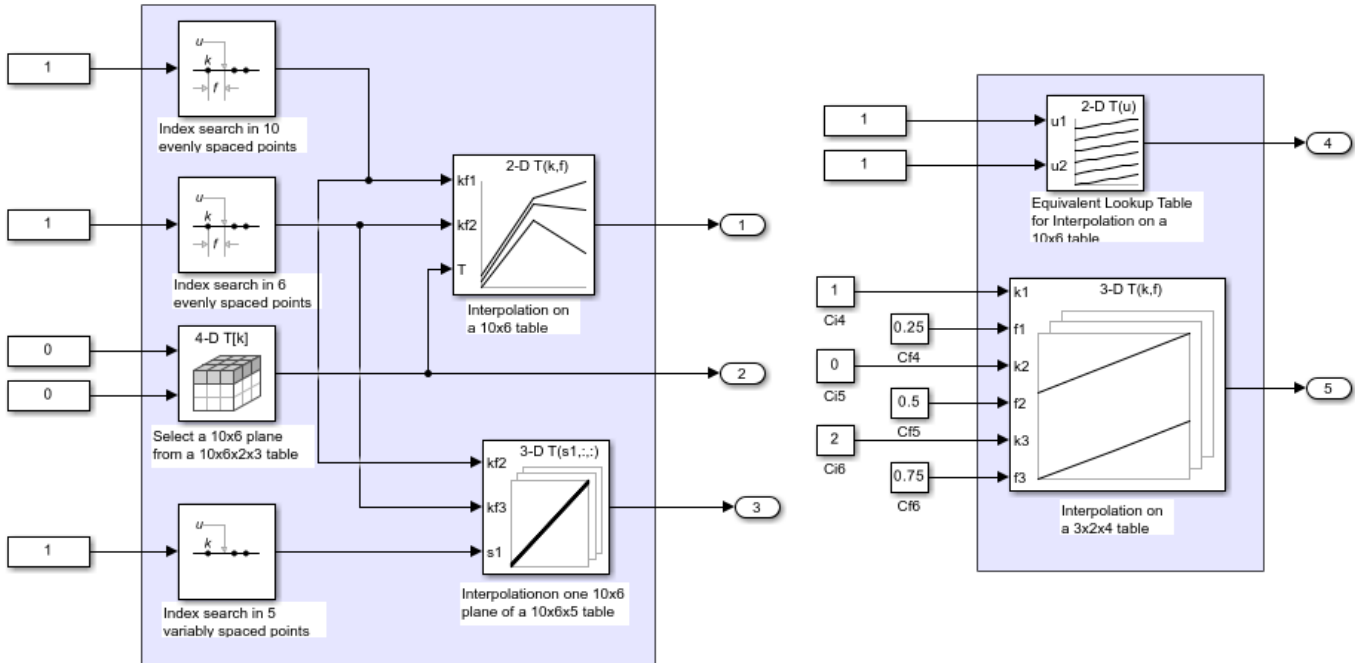
Code Generation by Using Row-Major Algorithm and Array Layout

After permuting the table data, Simulink configures the model RowLUTColToRow for row-major simulation. The model is equivalent to the preconfigured model RowLUTColToRowPreconfigured that has permuted table data and uses a row-major algorithm.

1. Open the example model RowLUTColToRowPreconfigured.

```
open_system('RowLUTColToRowPreconfigured');
```

?

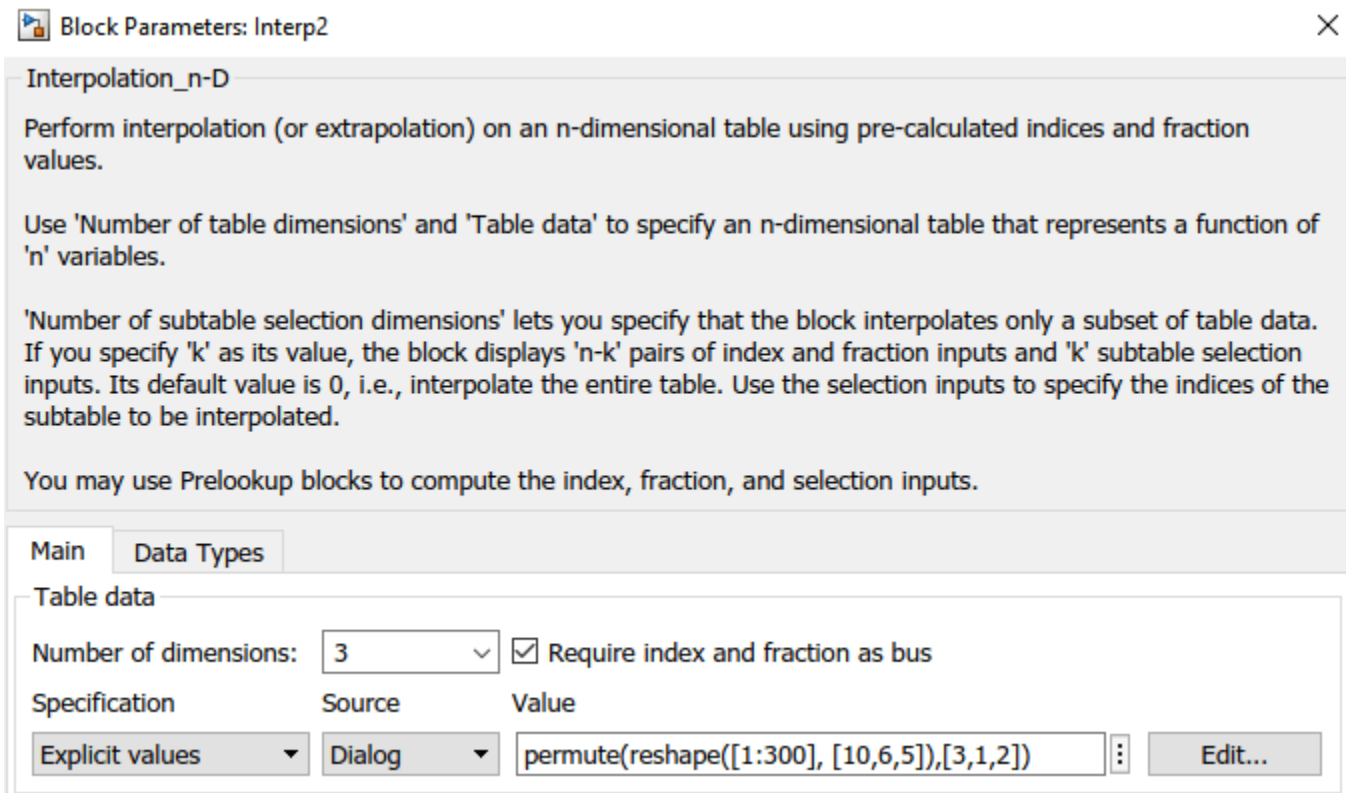
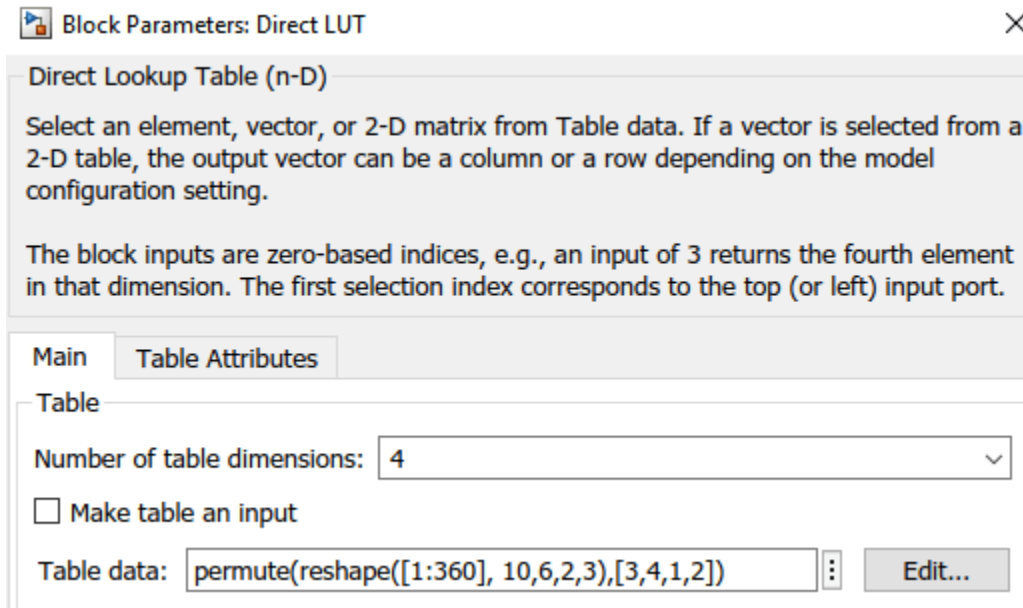


Copyright 2018-2022 The MathWorks, Inc.

2. To set up these models for row-major code generation, open the Configuration Parameters dialog box. In addition to enabling the **Use algorithms optimized for row-major array layout** configuration parameter, on the **Code Generation > Interface** pane, set the configuration parameter **Array Layout** to Row-Major option. The **Array Layout** parameter enables the model for row-major code generation. Alternatively, in the MATLAB Command Window, enter:

```
% For model 'RowLUTColToRowPreconfigured'
set_param('RowLUTColToRowPreconfigured', 'ArrayLayout', 'Row-major');
% For model 'RowLUTColToRow'
set_param('RowLUTColToRow', 'ArrayLayout', 'Row-major');
```

3. In the block dialog boxes, examine the permuted 3-D table.



4. Change your current folder in MATLAB to a writable folder. On the **C Code** tab, click **Build** to generate C code. In the generated code, observe the table data with row-major array layout

```

21 /* Block parameters (default storage) */
22 P rtP = {
23   /* Variable: Tbl_1
24     * Referenced by: '<Root>/2-D Lookup Table'
25     */
26   { 1.0, 11.0, 21.0, 31.0, 41.0, 51.0, 2.0, 12.0, 22.0, 32.0, 42.0, 52.0, 3.0,
27     13.0, 23.0, 33.0, 43.0, 53.0, 4.0, 14.0, 24.0, 34.0, 44.0, 54.0, 5.0, 15.0,
28     25.0, 35.0, 45.0, 55.0, 6.0, 16.0, 26.0, 36.0, 46.0, 56.0, 7.0, 17.0, 27.0,
29     37.0, 47.0, 57.0, 8.0, 18.0, 28.0, 38.0, 48.0, 58.0, 9.0, 19.0, 29.0, 39.0,
30     49.0, 59.0, 10.0, 20.0, 30.0, 40.0, 50.0, 60.0 }
31 };

```

In the generated code, the memcpy function replaces the for loops. Using memcpy reduces the amount of memory for storing data. This optimization improves execution speed.

```

83 /* Model step function */
84 void rtwdemo_row_lutcol2row_workflow_rowrow_step(void)
85 {
86   uint32_T rtb_Prelookup_Index;
87   real_T rtb_Prelookup_Fraction;
88   uint32_T rtb_Prelookup1_Index;
89   real_T rtb_Prelookup1_Fraction;
90   uint32_T rtb_Prelookup2;
91   real_T frac[2];
92   uint32_T bpIndex[2];
93   real_T frac_0[2];
94   uint32_T bpIndex_0[3];
95   real_T frac_1[3];
96   uint32_T bpIndex_1[3];
97
98   /* LookupNDDirect: '<Root>/Direct LUT'
99     *
100    * About '<Root>/Direct LUT':
101    * 4-dimensional Direct Look-Up returning a 2-D Matrix,
102    * which is contiguous for row-major array
103    */
104   memcpy(&rtY.Out2[0], &rtCP_DirectLUT_table[0], 60U * sizeof(real_T));
105
106   /* PreLookup: '<Root>/PreLookup' incorporates:
107     * Constant: '<Root>/Constant1'
108     */
109   rtb_Prelookup_Index = plook_evenca(1.0, rtCP_Prelookup_BreakpointsData[0],
110   rtCP_Prelookup_BreakpointsData[1] - rtCP_Prelookup_BreakpointsData[0], 9U,
111   &rtb_Prelookup_Fraction);

```

Observe the algorithms optimized for row-major data.

```

17 real_T intrp2d_la(const uint32_T bpIndex[], const real_T frac[], const real_T
18                 table[], const uint32_T stride, const uint32_T maxIndex[])
19 {
20     real_T y;
21     real_T yR_1d;
22     uint32_T offset_1d;
23
24     /* Row-major Interpolation 2-D
25     Interpolation method: 'Linear point-slope'
26     Use last breakpoint for index at or above upper limit: 'on'
27     Overflow mode: 'wrapping'
28     */
29     offset_1d = bpIndex[1U] * stride + bpIndex[0U];
30     if (bpIndex[0U] == maxIndex[0U]) {
31         y = table[offset_1d];
32     } else {
33         y = (table[offset_1d + 1U] - table[offset_1d]) * frac[0U] + table[offset_1d];
34     }
35
36     if (bpIndex[1U] == maxIndex[1U]) {
37     } else {
38         offset_1d += stride;
39         if (bpIndex[0U] == maxIndex[0U]) {
40             yR_1d = table[offset_1d];
41
42         close_system('RowLUTColToRow',0);
43         close_system('RowLUTColToRowPreconfigured',0);

```

See Also

Related Examples

- “Code Generation of Matrices and Arrays” on page 24-63
- “Interpolation Algorithm for Row-Major Array Layout” on page 18-10

Interpolation Algorithm for Row-Major Array Layout

This example illustrates the interpolation algorithm in 2-D and 3-D Lookup Table that is optimized for row-major array layout. The interpolation algorithm that is optimized for column-major array layout is also presented as a reference. The code generated by using row-major interpolation algorithm performs with the best speed and memory usage when operating on table data with row-major array layout. The code generated by using column-major interpolation algorithm performs best with column-major array layout.

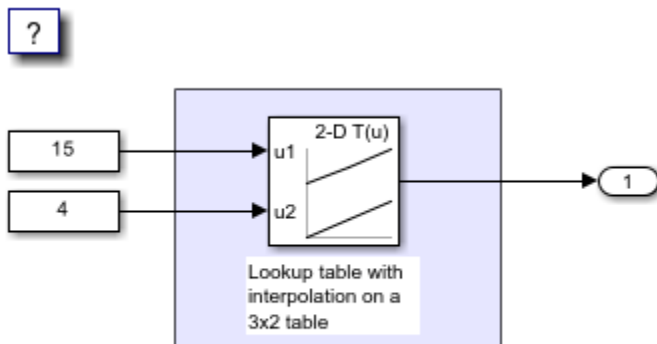
In this example, you:

- Interpolate on a 2-D Lookup Table with column-major and row-major algorithm.
- Generate code with a row-major algorithm and an array layout.
- Identify the array layout and select the optimized algorithm.
- Interpolate on a 3-D Lookup Table with column-major and row-major algorithm.

Simulate with 2-D Row-Major Algorithm

1. Open the example model RowLUT2D.

```
model = 'RowLUT2D';
open_system(model);
```



Copyright 2018-2022 The MathWorks, Inc.

2. By default, Simulink® configures a model with column-major algorithm and column-major array layout. The model RowLUT2D is configured to use column-major algorithm. Simulate the model. To observe the output, open the Data Inspector from the **Simulation** tab. The output value is 4.

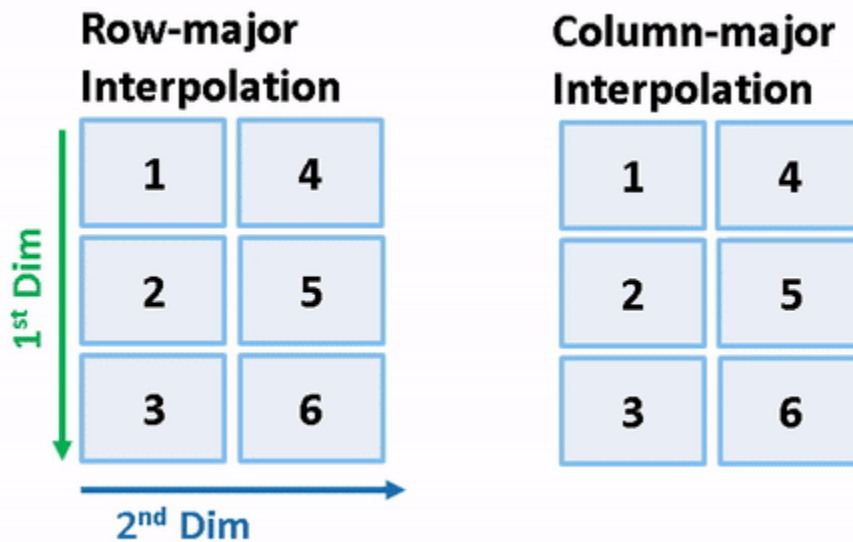
3. To enable row-major algorithms, open the Configuration Parameters dialog box. On the **Math and Data Types** pane, enable the configuration parameter “Use algorithms optimized for row-major array layout”. Alternatively, in the MATLAB® Command Window, enter:

```
set_param(model, 'UseRowMajorAlgorithm', 'on');
```

4. Simulate the model and observe those results in the Data Inspector. The output value is 4.

The column-major and row-major algorithms differ only in the interpolation order. In some cases, due to different operation order on the same data set, you might experience minor numeric differences in

the outputs of column-major and row-major algorithms. For the 2-D Lookup Table used in the example model, the interpolation algorithm is illustrated here.



Generate Code by Using Row-Major Algorithm and Array Layout

The 2-D table data used in model RowLUT2D is:

Table_3by2.Value

ans =

1	4
2	5
3	6

1. Open the Configuration Parameters dialog box. In addition to enabling the **Use algorithms optimized for row-major array layout** configuration parameter, on the **Code Generation > Interface** pane, set the configuration parameter Array layout to Row-Major option. This configuration parameter enables the model for row-major code generation. Alternatively, in the MATLAB Command Window, enter:

```
set_param(model, 'ArrayLayout', 'Row-major');
```

2. Change your current folder in MATLAB to a writable folder. On the **C Code** tab, click **Build** to generate C code.

3. In the generated code, observe the table data with row-major array layout. For comparison, here is the table data in the generated code for column-major array layout.

2-D Table with row-major array layout

```

/* Block parameters (default storage) */
P_rtwdemo_row_lut2d_T rtwdemo_row_lut2d_P = {
  /* Variable: Table_3by2
   * Referenced by: '<Root>/2-D Lookup Table'
   */
  { 1.0, 4.0, 2.0, 5.0, 3.0, 6.0 }
};

```

2-D Table with column-major array layout

```

/* Block parameters (default storage) */
P_rtwdemo_row_lut2d_T rtwdemo_row_lut2d_P = {
  /* Variable: Table_3by2
   * Referenced by: '<Root>/2-D Lookup Table'
   */
  { 1.0, 2.0, 3.0, 4.0, 5.0, 6.0 }
};

```

In the generated code, the table data is in row-major order and the interpolation algorithm is optimized for row-major array layout. The row-major algorithm operates on table data that is contiguous in memory. This leads to faster cache access, making these algorithms cache-friendly.

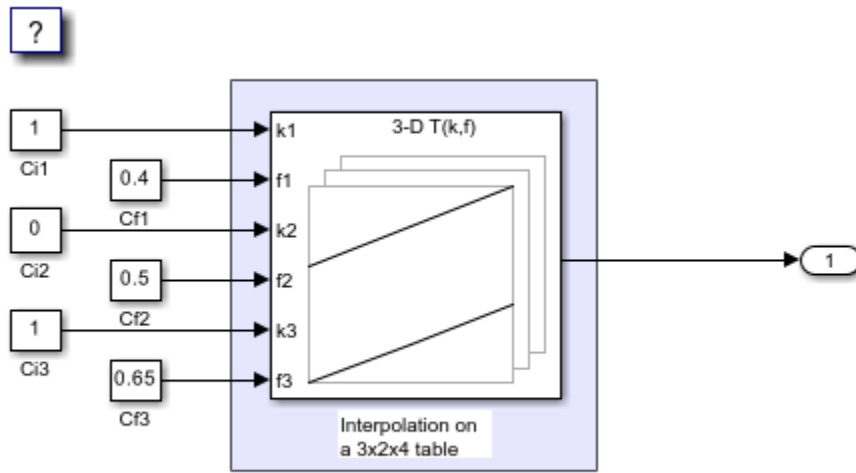
This table summarizes the relationship between array layout and cache-friendly algorithms. It is recommended to use the algorithm that is optimized for the specified array layout to achieve good performance. For example, use row-major interpolation algorithm when the array layout is set to Row-Major for code generation.

Array Layout	Algorithm	Cache-Friendly Algorithm
Column-major	Column-major	✓ (Recommended)
Row-major	Row-major	✓ (Recommended)
Row-major	Column-major	⚠ (Not recommended)
Column-major	Row-major	⚠ (Not recommended)

Interpolation on a 3-D Table

1. Open the example model RowInterpolationAlgorithm.

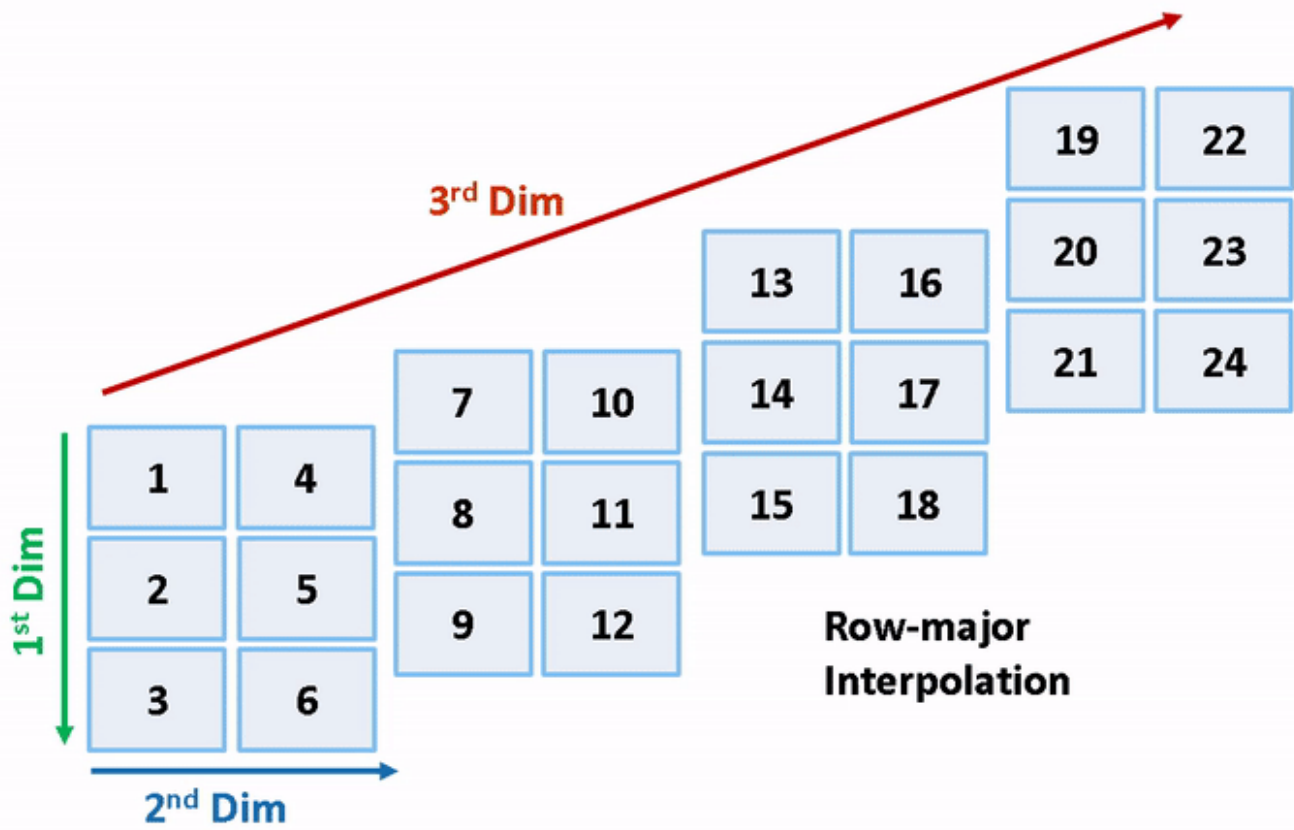
```
open_system('RowInterpolationAlgorithm');
```

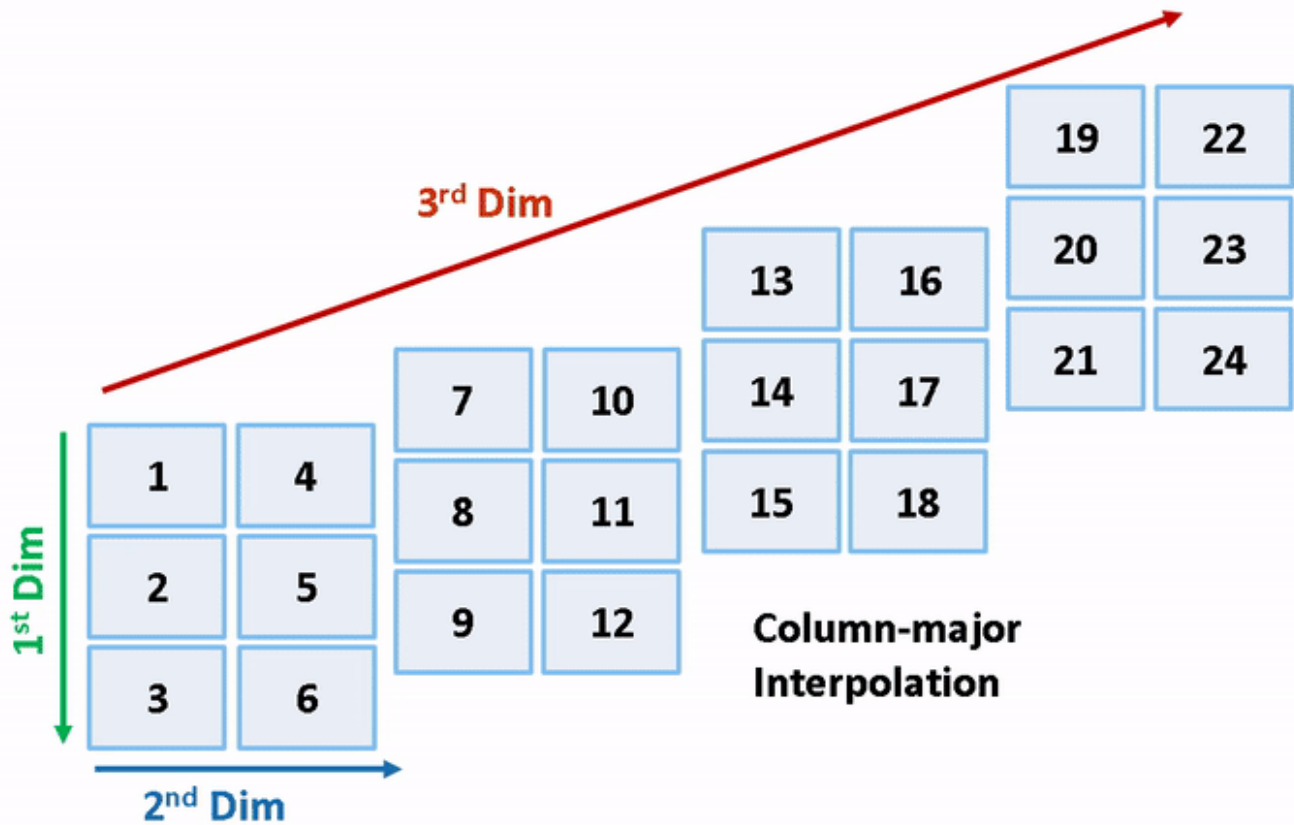



Copyright 2018-2022 The MathWorks, Inc.

2. Generate code for the model by using the preceding procedure. Simulate and generate code from the model by repeating the steps performed on RowLUT2D model.

The row-major and column-major interpolations on the 3-D table used in the example model are illustrated here.





```
close_system('RowLUT2D',0);
close_system('RowInterpolationAlgorithm', 0);
```

See Also

Related Examples

- "Code Generation of Matrices and Arrays" on page 24-63
- "Interpolation with Subtable Selection Algorithm for Row-Major Array Layout" on page 18-16

Interpolation with Subtable Selection Algorithm for Row-Major Array Layout

This example illustrates the algorithm for interpolating a subtable that the interpolation block selects from a higher dimension table. The interpolation algorithm with subtable selection is optimized for row-major array layout. As a reference, see the interpolation algorithm with subtable selection that is optimized for column-major array layout. The code generated by using row-major interpolation algorithm performs with the best speed and memory usage when operating on table data with row-major array layout. The code generated by using column-major algorithm performs best with column-major array layout.

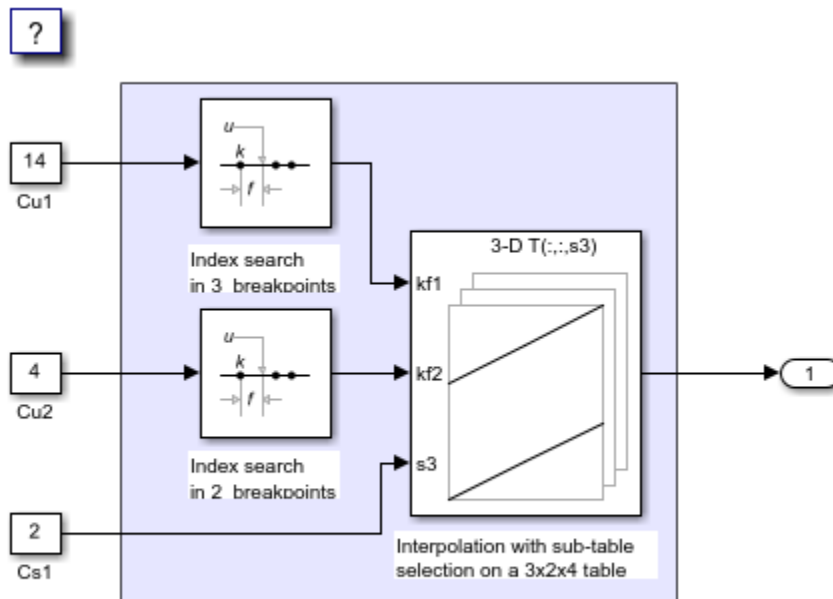
In this example, you:

- Interpolate on a selected subtable with column-major and row-major algorithm.
- Preserve block semantics through table permutation.
- Generate code with row-major algorithm and array layout.

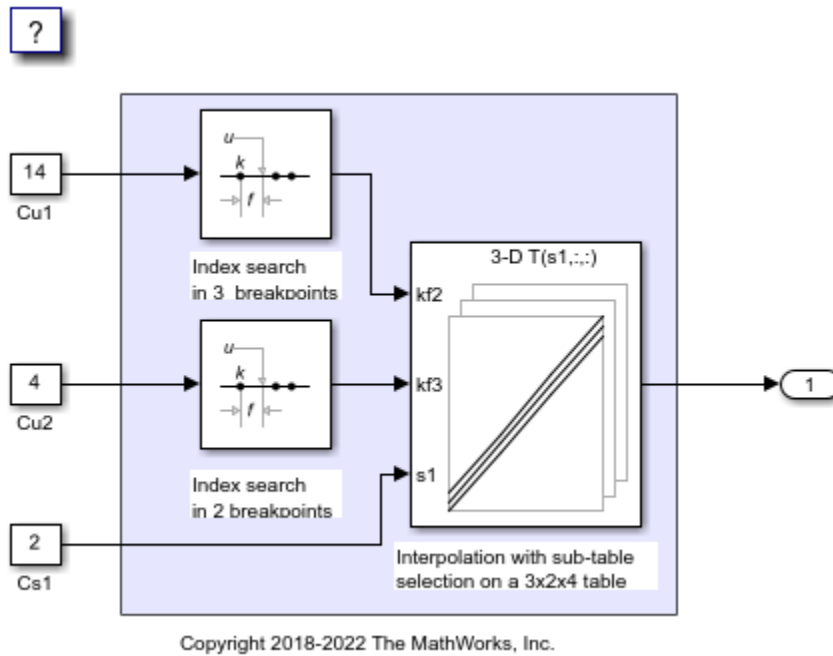
Simulate by Using Row-Major Algorithm

1. Open example model SubtableInterpolationCol and SubtableInterpolationRow.

```
open_system('SubtableInterpolationCol');
open_system('SubtableInterpolationRow');
```



Copyright 2018-2022 The MathWorks, Inc.



2. By default, Simulink® configures a model with column-major algorithm and column-major array layout. The model `SubtableInterpolationCol` is configured to use a column-major algorithm. Run the model and observe the output stored in workspace variable `yout`.

3. To enable row-major algorithms, open the Configuration Parameters dialog box. On the **Math and Data Types** pane, select the configuration parameter “Use algorithms optimized for row-major array layout” Alternatively, in the MATLAB® Command Window, enter:

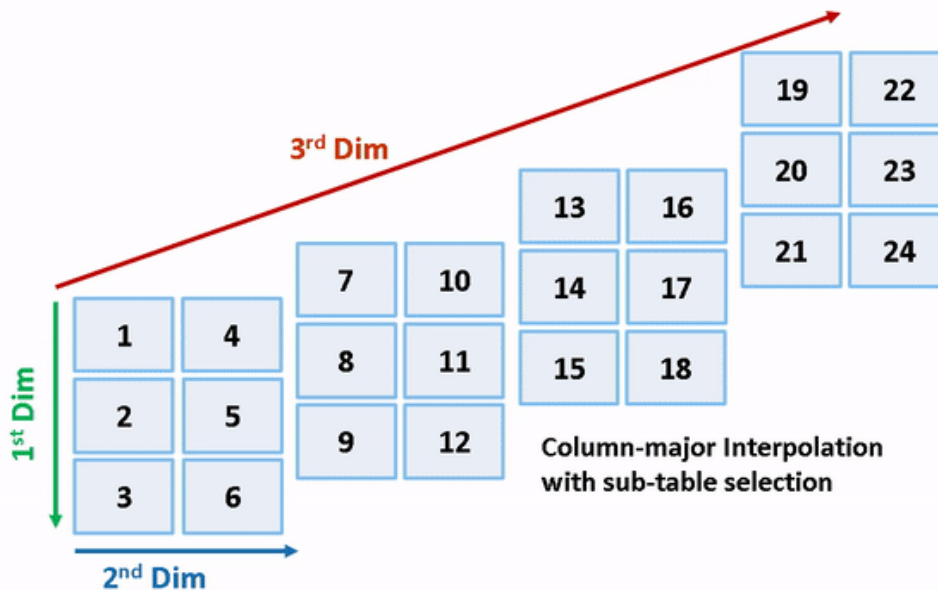
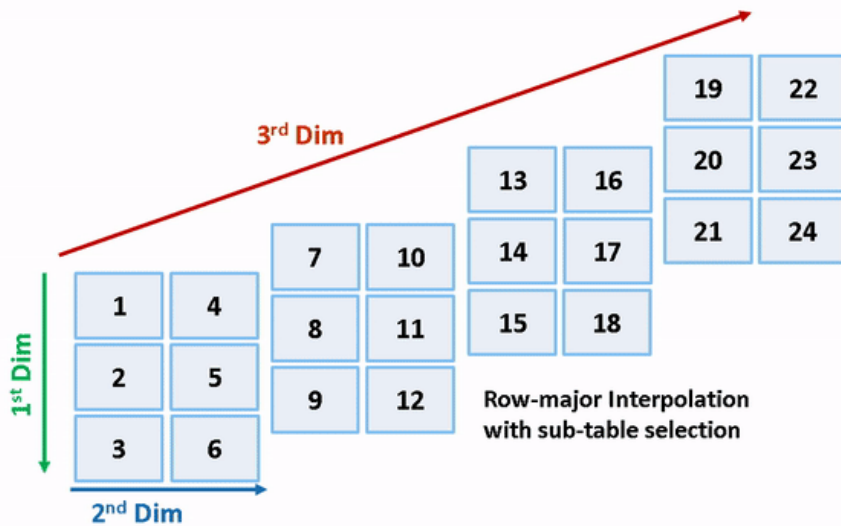
```
set_param('SubtableInterpolationCol', 'UseRowMajorAlgorithm', 'on');
```

4. Simulate the model and observe the error.

The range of values for the first signal in the bus object propagated to the input port `kf2` of block '`rtwdemo_col_interpelsubtable/Interp`' does not match the size of the corresponding table dimension of 2. Specify a different table dimension or modify the range for the first signal in the bus object propagated to the input port `kf2`.

Component: Simulink | Category: Block error

The column-major and row-major algorithms differ in terms of subtable selection and interpolation order. The subtable selection is performed inside the original table. No extra memory is allocated for the subtable. The selected subtable is contiguous in memory. The interpolation order is cache-friendly for column-major algorithms with column-major array layouts and row-major algorithms with row-major array layouts. This illustration compares the row-major and column-major interpolations with subtable selection.



Due to a change in semantics, column-major and row-major interpolations are performed on different subtables or data sets. This interpolation leads to different numeric outputs or an error.

Preserve Semantics Through Table Permutation

With subtable selection, the model semantics change when switching from a column-major algorithm to a row-major algorithm. To preserve the semantics or ensure that the same subtable is selected for interpolation, you need to permute the table data. Otherwise, Simulink might report error if it encounters inconsistent breakpoint and table data between prelookup and interpolation blocks.

1. The block SubtableInterpolationCol/Interp has 3-D table data given as **T3d = reshape([1:24], 3,2,4)** and one selection port with input **2** (0-based index). The selected subtable is

T3d(:, :, 3) (1-based index in MATLAB) for the column-major algorithm. To preserve the semantics for the row-major algorithm on the same model, that is, select the same subtable with same index and selection port inputs, permute the table as **T3d_p = permute(T3d, [3,1,2])**. The selected subtable is **T3d_p(3, :, :)** (1-based index) for the row-major algorithm.

```
T3d_str = get_param('SubtableInterpolationCol/Interp', 'Table');
set_param('SubtableInterpolationCol/Interp', 'Table', ...
    ['permute(', T3d_str, ', [3,1,2]')]);
```

2. When you import table data from a file, you must permute the table data in the file before importing it. This permutation keeps the table tunable throughout the simulation and code generation workflow.

Generate Code by Using Row-Major Algorithm and Array Layout

After permuting the table data, model `SubtableInterpolationCol` is configured for row-major simulation. The model is equivalent to the preconfigured model `SubtableInterpolationRow` that uses a row-major algorithm.

1. To set up the model for row-major code generation, open the Configuration Parameters dialog box. In addition to selecting the **Use algorithms optimized for row-major array layout** configuration parameter, on the **Code Generation > Interface** pane, set the configuration parameter Array layout to Row-Major option. This configuration parameter enables the model for row-major code generation. Alternatively, in the MATLAB Command Window, enter:

```
set_param('SubtableInterpolationCol', 'ArrayLayout', 'Row-major');
```

2. In the dialog box, examine the permuted 3-D table data and the selected 2-D subtable.

Block Parameters: Interp ×

Interpolation_n-D

Perform interpolation (or extrapolation) on an n-dimensional table using pre-calculated indices and fraction values.

Use 'Number of table dimensions' and 'Table data' to specify an n-dimensional table that represents a function of 'n' variables.

'Number of subtable selection dimensions' lets you specify that the block interpolates only a subset of table data. If you specify 'k' as its value, the block displays 'n-k' pairs of index and fraction inputs and 'k' subtable selection inputs. Its default value is 0, i.e., interpolate the entire table. Use the selection inputs to specify the indices of the subtable to be interpolated.

You may use Prelookup blocks to compute the index, fraction, and selection inputs.

Main Data Types

Table data

Number of dimensions: Require index and fraction as bus

Specification	Source	Value	
<input type="text" value="Explicit values"/>	<input type="text" value="Dialog"/>	<input type="text" value="permute(reshape([1:24], 3,2,4), [3,1,2])"/>	<input type="button" value="Edit..."/>

Algorithm

Interpolation method:

Extrapolation method: Valid index input may reach last index

Diagnostic for out-of-range input:

Number of sub-table selection dimensions:

3. Change your current folder in MATLAB to a writable folder. On the **C Code** tab, click **Build** to generate C code. In the generated code, observe the 2-D interpolation algorithm optimized for row-major data.


```

93 static real_T intrp2d_la(const uint32_T bpIndex[], const real_T frac[], const
94   real_T table[], const uint32_T stride, const uint32_T maxIndex[])
95 {
96   real_T y;
97   real_T yR_ld;
98   uint32_T offset_ld;
99
100  /* Row-major Interpolation 2-D
101     Interpolation method: 'Linear point-slope'
102     Use last breakpoint for index at or above upper limit: 'on'
103     Overflow mode: 'wrapping'
104  */
105  offset_ld = bpIndex[1U] * stride + bpIndex[0U];
106  if (bpIndex[0U] == maxIndex[0U]) {
107    y = table[offset_ld];
108  } else {
109    y = (table[offset_ld + 1U] - table[offset_ld]) * frac[0U] + table[offset_ld];
110  }
111
112  if (bpIndex[1U] == maxIndex[1U]) {
113  } else {
114    offset_ld += stride;
115    if (bpIndex[0U] == maxIndex[0U]) {
116      yR_ld = table[offset_ld];
117    } else {
118      yR_ld = (table[offset_ld + 1U] - table[offset_ld]) * frac[0U] +
119        table[offset_ld];
120    }
121
122    y += (yR_ld - y) * frac[1U];
123  }
124
125  return y;
126 }

```

```
close_system('SubtableInterpolationCol',0);
```

See Also

Related Examples

- “Code Generation of Matrices and Arrays” on page 24-63
- “Interpolation Algorithm for Row-Major Array Layout” on page 18-10

Direct Lookup Table Algorithm for Row-Major Array Layout

This example shows how Simulink® selects a vector or a 2-D matrix from table data. In a 2-D table, the output vector can be a column or a row depending on the model configuration setting **Use algorithms optimized for row-major array layout**. In this example, the Direct Lookup Table algorithm is optimized for row-major array layout. The Direct Lookup Table algorithm that is optimized for column-major array layout is also presented as a reference. The code generated by using row-major interpolation algorithm performs with the best speed and memory usage when operating on table data with row-major array layout. The code generated by using column-major algorithm performs best with column-major array layout.

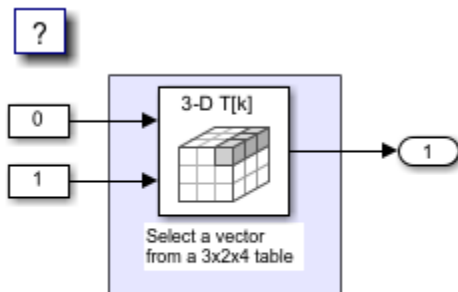
In this example, you:

- Output a vector or a plane by using direct lookup with a column-major or a row-major algorithm.
- Preserve semantics when switching from a column-major algorithm to a row-major algorithm.
- Generate code by using a row-major algorithm and an array layout.

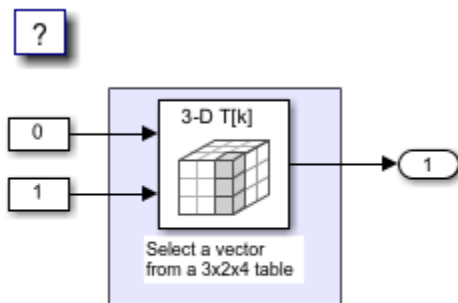
Simulate by Using Row-Major Algorithm — Output a Vector from 3-D Table

Open example models RowDLUT3DSelectVector and ColumnDLUT3DSelectVector.

```
open_system('RowDLUT3DSelectVector');
open_system('ColumnDLUT3DSelectVector');
```



Copyright 2018-2022 The MathWorks, Inc.



Copyright 2018-2022 The MathWorks, Inc.

1. By default, Simulink configures a model with column-major algorithm and column-major array layout. The model `ColumnDLUT3DSelectVector` is preconfigured to use column-major algorithms. Simulate the model and observe the output stored in workspace variable `yout`.

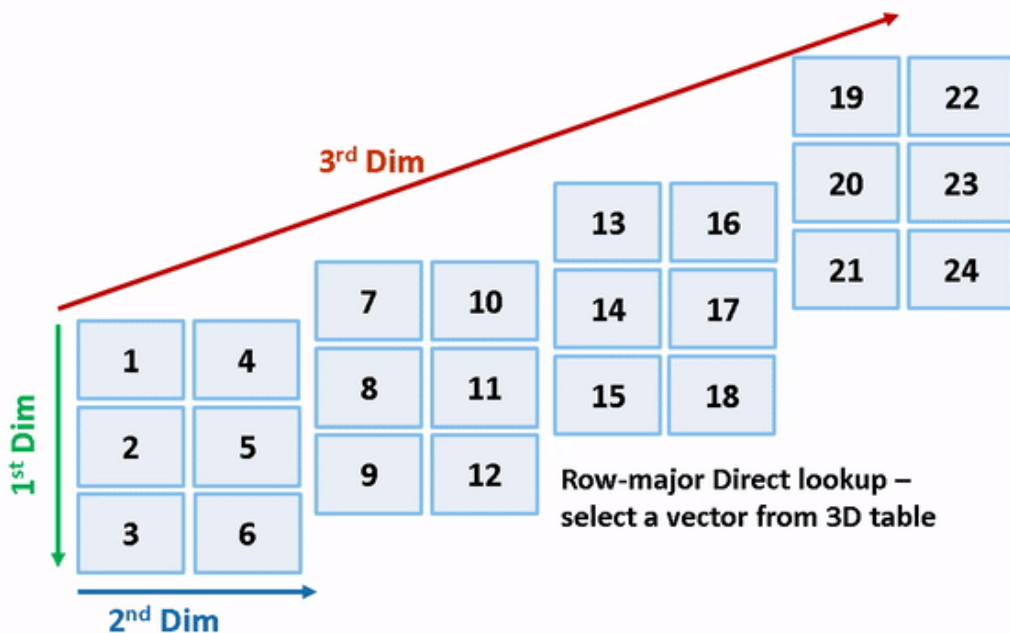
2. To enable row-major algorithms, open the Configuration Parameters dialog box. On the **Math and Data Types** pane, select the configuration parameter “Use algorithms optimized for row-major array layout” Alternatively, in the MATLAB® Command Window, enter:

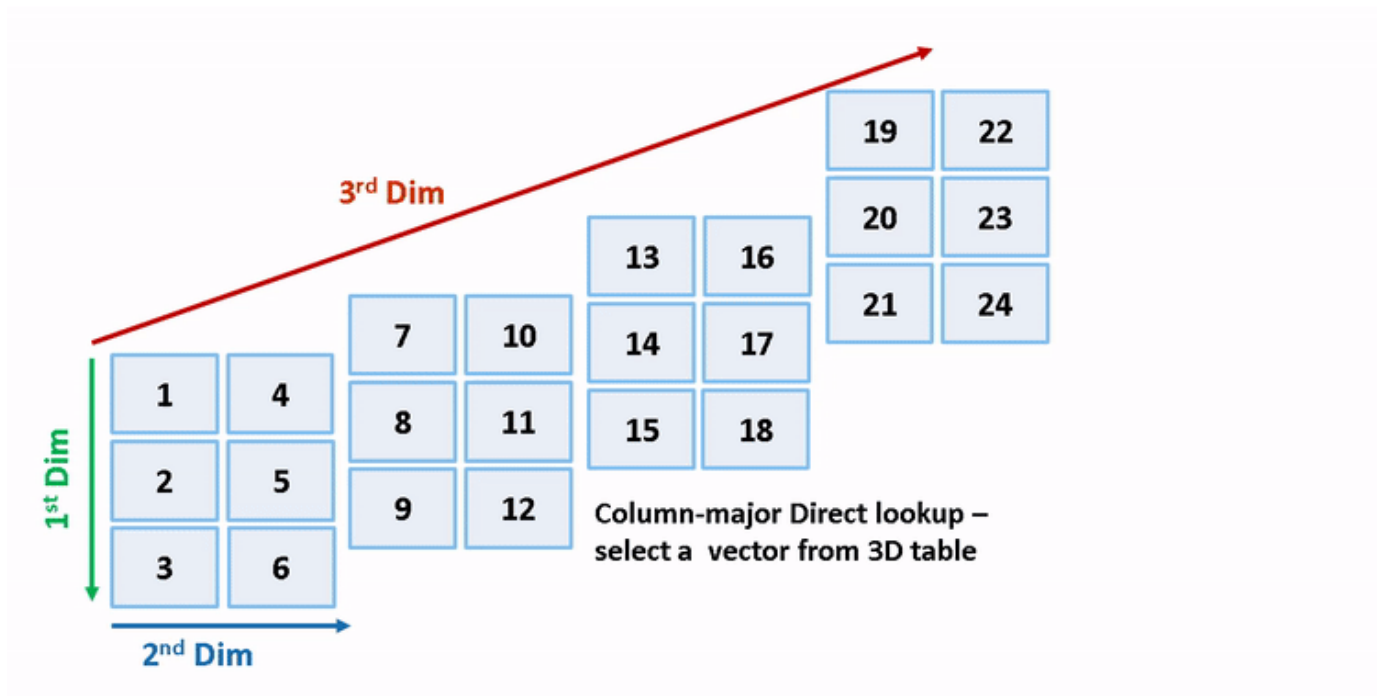
```
set_param('ColumnDLUT3DSelectVector','UseRowMajorAlgorithm','on');
```

3. On the **Simulation** tab, click **Run** to simulate the model. Observe the change in output dimension and numeric values logged in workspace variable `yout`.

The column-major and row-major algorithms differ semantically in the selection of output vector. For example, in a 2-D table, Simulink selects a column vector as output for column-major algorithm and a row vector for row-major algorithm. In a table with a 3-D or higher dimension, Simulink selects the output vector from the first dimension of the table for a column-major algorithm and from the last dimension of the table for a row-major algorithm. The elements of the selected vector are contiguous in the table storage memory. In this example, the last dimension is the third dimension of the 3-D table. Due to semantic change, column-major and row-major direct lookup table algorithms output different vector size and numeric values.

These illustrations compare the vector output of row-major and column-major direct lookup table algorithms in a 3-D table.





Preserve Semantics by Using Table Permutation

For a direct lookup table that outputs a vector or 2-D matrix, the model semantics change when you switch from a column-major algorithm to a row-major algorithm. To preserve the semantics or ensure the same output given the same block I/O connections, you must permute the table data. Otherwise, Simulink propagates incorrect dimensions to downstream blocks.

1. The block `ColumnDLUT3DSelectVector/Direct Lookup Table (n-D)` has 3-D table data **T3d** = `reshape([1:24], 3,2,4)` and two input ports with value **0** and **1** (both are 0-based indices). The selected output vector is **T3d(:,1,2)** (1-based index) for a column-major algorithm. To preserve the semantics for a row-major algorithm on the same model, that is, select the same vector with same index port inputs, permute the table as **T3d_p** = `permute(T3d, [2,3,1])`. For a row-major algorithm, the selected vector is **T3d_p(1,2,:)**.

```
T3d_str = get_param('ColumnDLUT3DSelectVector/Direct Lookup Table (n-D)', 'Table');
set_param('ColumnDLUT3DSelectVector/Direct Lookup Table (n-D)', 'Table', ...
['permute(', T3d_str, ', [2,3,1]')]);
```

2. When you import table data from a file, you must permute the table data in the file before importing it. This permutation keeps the table tunable throughout the simulation and code generation workflow.

Code Generation by Using Row-Major Algorithm and Array Layout

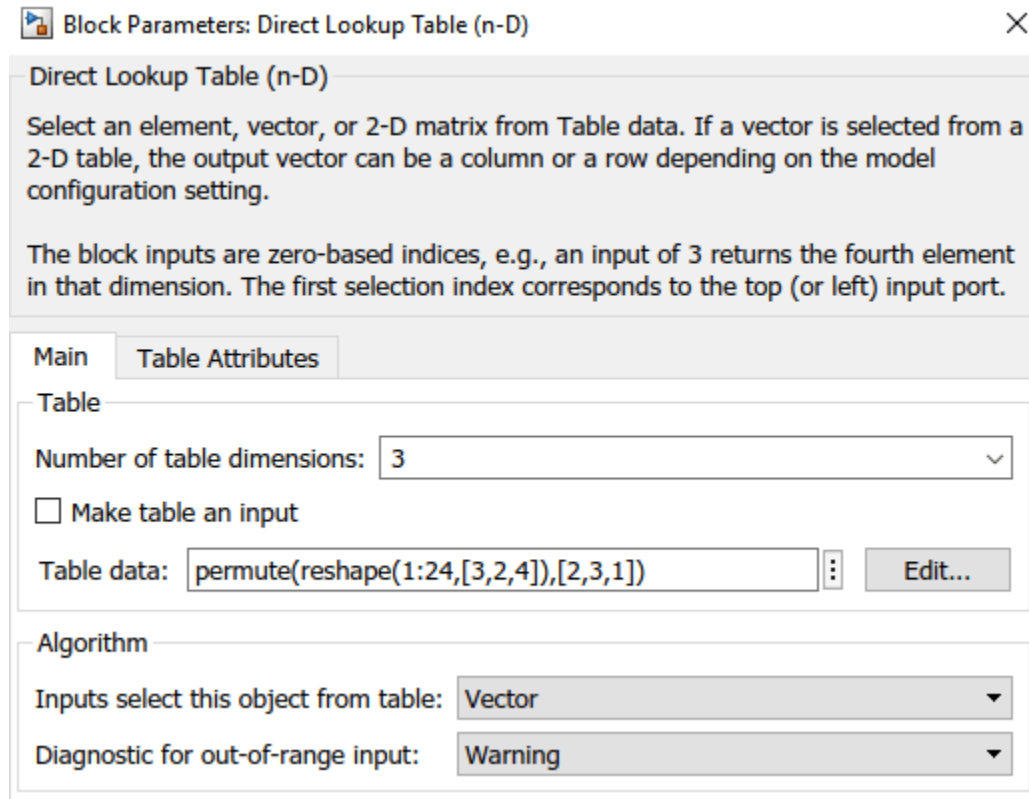
After permuting the table data, Simulink configures the model `ColumnDLUT3DSelectVector` for row-major simulation. The model is equivalent to the preconfigured model `RowDLUT3DSelectVector` that has permuted table data and uses a row-major algorithm.

1. To set up these models for row-major code generation, open the Configuration Parameters dialog box. In addition to enabling the **Use algorithms optimized for row-major array layout** configuration parameter, on the **Code Generation > Interface** pane, set the configuration

parameter Array layout to the Row-Major option. This configuration parameter enables the model for row-major code generation. Alternatively, in the MATLAB Command Window, enter:

```
% For model 'ColumnDLUT3DSelectVector'
set_param('ColumnDLUT3DSelectVector', 'ArrayLayout', 'Row-major');
% For model 'RowDLUT3DSelectVector'
set_param('RowDLUT3DSelectVector', 'ArrayLayout', 'Row-major');
```

2. In the Direct Lookup Table (n-D) block dialog box, examine the permuted 3-D table data.

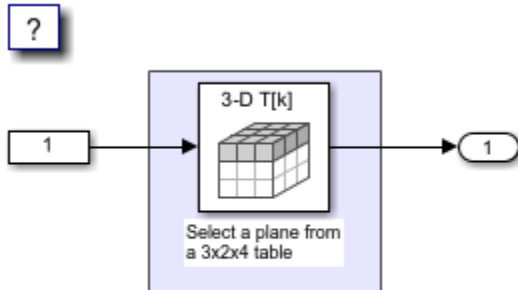


3. Change your current folder in MATLAB to a writable folder. On the **C Code** tab, click **Build** to generate C code. In the generated code, the `memcpy` function replaces the `for` loops. Using `memcpy` reduces the amount of memory for storing data. This optimization improves execution speed.

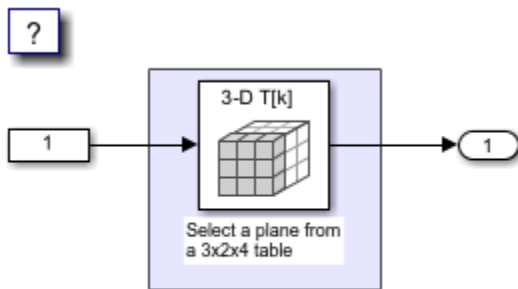
```
34 void rtwdemo_row_dlut3d_selvector_step(void)
35 {
36     /* Outport: '<Root>/Out2' incorporates:
37      * LookupNDDirect: '<Root>/Direct Lookup Table (n-D)'
38      *
39      * About '<Root>/Direct Lookup Table (n-D)':
40      * 3-dimensional Direct Look-Up returning a Vector,
41      * which is contiguous for row-major array
42      */
43     memcpy(&rtY.Out2[0], &rtCP_DirectLookupTablenD_table[3], 3U * sizeof(real_T));
44 }
```

Simulate by Using Row-Major Algorithm – Output a Plane from 3-D Table

```
open_system('RowDLUT3DSelectPlane');
open_system('ColumnDLUT3DSelectPlane');
```

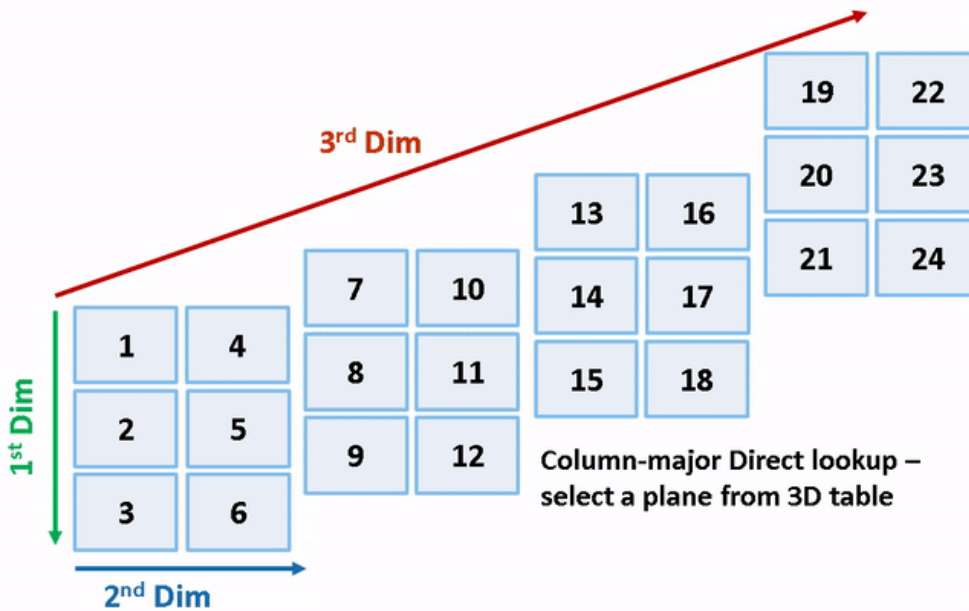
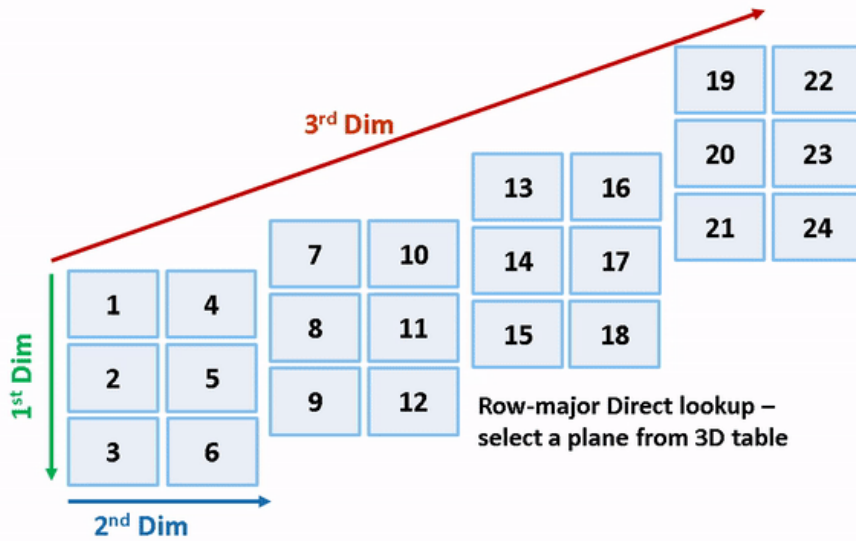


Copyright 2018-2022 The MathWorks, Inc.



Copyright 2018-2020 The MathWorks, Inc.

1. Open the example model RowDLUT3DSelectPlane that outputs a plane or 2-D matrix from a 3-D table.
2. Simulate and generate code from the model by repeating the steps performed on ColumnDLUT3DSelectVector. The row-major and column-major direct lookup algorithms that output a 2-D matrix from a 3-D table are illustrated here.



```
close_system('RowDLUT3DSelectVector',0);
close_system('ColumnDLUT3DSelectVector',0);
close_system('RowDLUT3DSelectPlane',0);
close_system('ColumnDLUT3DSelectPlane',0);
```

See Also

Related Examples

- “Code Generation of Matrices and Arrays” on page 24-63

- “Interpolation Algorithm for Row-Major Array Layout” on page 18-10

Generate Row-Major Code for S-Functions

You can generate row-major code for models that contain S-functions. By default, the code generator generates column-major code. To learn more about row-major code generation, see “Code Generation of Matrices and Arrays” on page 24-63.

For an existing model that contains S-functions, when you set the configuration parameter **Array layout** as **Row-major**, the configuration parameter **External functions compatibility for row-major code generation** is enabled and set to **error** by default. When you try to build the existing model, you get an error because the S-functions are not enabled for row-major code generation by default. You can test the compatibility of your S-function for row-major code generation by using the **External functions compatibility for row-major code generation** configuration parameter.

This workflow is also applicable to C Caller blocks in a model.

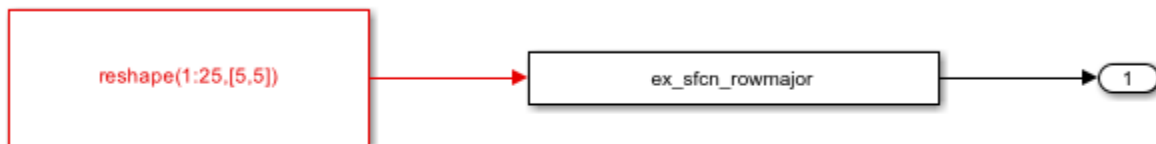
Example

1. Open example model `ex_sfcn_rowmajor_unset`. The model needs these additional files:

- S-function: `ex_sfcn_rowmajor.c`
- TLC: `ex_sfcn_rowmajor.tlc`

Save these files to your local folder.

```
open_system('ex_sfcn_rowmajor_unset');
```



Copyright 2021 The MathWorks, Inc.

2. The model is configured with **Array layout** set to **Column-major**. To enable row-major code generation, set **Array layout** to **Row-major**.

3. When you build the model, the code generator terminates the build and you see this error message:

```
### Build procedure for model: 'ex_sfcn_rowmajor_unset' aborted due to an error.
S-function "ex_sfcn_rowmajor" with SSArrayLayout set to 'SS_UNSET' does not support row major code
generation. To test this S-function, set the configuration parameter 'External functions
compatibility for row-major code generation' to 'warning' or 'none'.
Component: Simulink | Category: Block diagram error
```

4. To proceed, do one of the following:

- If you want to test your existing S-functions with the row-major code for the model, change the setting of the configuration parameter **External functions compatibility for row-major code**

generation to warning or none. The code generator completes the build without generating row-major code for the S-functions.

- If you want to update your S-functions so that they are compatible with row-major array layout, use the S-function API to enable the S-function for row-major code generation.

To test your model for S-functions with unspecified array layout, you can also run the Model Advisor checks on the model. Select the **Identify TLC S-functions with unset array layout** check and click the **Run This Check** button. If the model includes S-functions with unspecified array layout, you see a warning such as:

The screenshot shows a Model Advisor panel titled "Identify TLC S-Functions with unset array layout". It includes a "Run This Check" button and a "View by" dropdown menu set to "Recommended Action". The result is a warning: "List all S-Functions which have SSArrayLayout set to SS_UNSET. These S-Functions can have numerical implications in the generated code." Below this, a "Warning" section states: "The following TLC S-Functions have SSArrayLayout set to SS_UNSET whose algorithms need evaluation." A bullet point lists the function: [ex_sfcn_rowmajor_unset/S-Function](#).

To specify the array layout of the user-defined S-function, use the `ssSetArrayLayoutForCodeGen` function of the SimStruct API. You can set the enumerated type `SSArrayLayout` to:

- `SS_UNSET` - This setting is the default setting that disables the block for row-major code generation.
- `SS_COLUMN_MAJOR` - Specify the block for column-major code generation only.
- `SS_ROW_MAJOR` - Specify the block for row-major code generation only.
- `SS_ALL` - Specify the block as allowed for code generation regardless of the array layout.

5. Update the `ex_sfcn_rowmajor.c` file by adding the `ssSetArrayLayoutForCodeGen` in the `mdlInitializeSizes` method:

```
static void mdlInitializeSizes(SimStruct *S)
{
    /* Specify array layout of the S-function */
    ssSetArrayLayoutForCodeGen(S, SS_ROW_MAJOR);
    .
    .
    .
}
```

If your S-function is not affected by an array layout, set `SSArrayLayout` to `SS_ALL`.

6. Compile the S-function by using this command in the MATLAB® Command Window:

```
mex ex_sfcn_rowmajor.c
```

```
Building with 'Microsoft Visual C++ 2019 (C)'.  
MEX completed successfully.
```

7. To build the model and generate code, press **Ctrl+B**.

If you generate S-functions by using S-Function Builder, use the parameter **Array layout** to specify the majority of the S-function. For more information, see “Build S-Functions Automatically Using S-Function Builder”. If you use the Legacy Code Tool to integrate C functions, use the `convertNDArrayToRowMajor` option in `legacy_code`. The S-Function Builder and Legacy Code Tool apply preceding transposes when the S-function is set to row-major array layout during simulation in normal mode.

See Also

Related Examples

- “Integrate C Code Using C Caller Blocks”

Row-Major Algorithms for Row-Major Array Layout

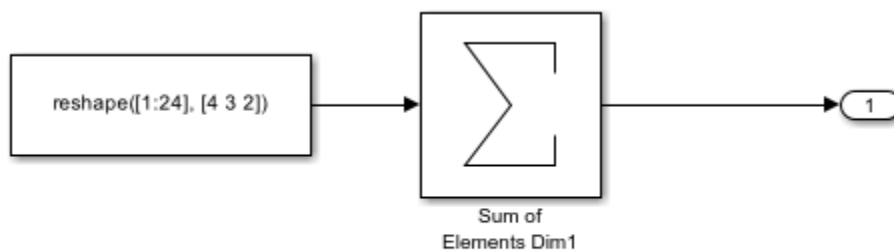
This example shows how to use row-major algorithms to generate efficient code. You can enable the “Use algorithms optimized for row-major array layout” configuration parameter to enable efficient row-major algorithms that are optimized for row-major array layout. The code that you generate by using row-major algorithms performs with better speed and efficient memory usage when operating on data with row-major array layout.

In this example, you operate on row-major data by first using the default column-major algorithms, and then using the row-major algorithms. This comparison helps in identifying the appropriate algorithm settings to achieve different requirements.

Generate Code by Using the Default Column-Major Algorithms for Row-Major Array Layout

Open the example model `ex_rowmajor_algorithm`.

```
model = 'ex_rowmajor_algorithm';
open_system(model);
```



Copyright 2021 The MathWorks, Inc.

The model contains a Sum of Elements block and the input of the block is an array. By default, Simulink® configures a model with column-major algorithms and column-major array layout. In this example, you configure the array layout of this model as row-major. To specify the array layout, open the **Embedded Coder** app, and then open the Configuration Parameters dialog box. On the **Code Generation > Interface** pane, set the configuration parameter **Array layout** to Row-Major option. Alternatively, in the MATLAB® Command Window, enter:

```
set_param(model, 'ArrayLayout', 'Row-major');
```

Change your current folder in MATLAB to a writable folder. Then generate code from the model by using the `slbuild` function or by pressing **Ctrl+B**.

```
currentDir = pwd;
[~,cgDir] = rtwdemodir();
% Generate code and capture the code generation information to Value
Value=evalc('slbuild(''ex_rowmajor_algorithm'')');
```

Inspect the generated `ex_rowmajor_algorithm_step` step function in the `ex_rowmajor_algorithm.c`.

```
file = fullfile('ex_rowmajor_algorithm_ert_rtw', 'ex_rowmajor_algorithm.c');
rtwdemodbtype(file, '/* Model step function */', '/* Model initialize function', 1, 1);
```

```

/* Model step function */
void ex_rowmajor_algorithm_step(void)
{
    int32_T i;
    int32_T i_0;
    int32_T tmp;

    /* Sum: '<Root>/Sum of Elements Dim1' incorporates:
     * Constant: '<Root>/Constant1'
     */
    ex_rowmajor_algorithm_Y.Out2 = -0.0F;
    for (i = 0; i < 2; i++) {
        for (i_0 = 0; i_0 < 3; i_0++) {
            tmp = (i_0 << 1) + i;
            ex_rowmajor_algorithm_Y.Out2 =
                ((ex_rowmajor_algorithm_ConstP.Constant1_Value[tmp] +
                 ex_rowmajor_algorithm_Y.Out2) +
                 ex_rowmajor_algorithm_ConstP.Constant1_Value[tmp + 6]) +
                ex_rowmajor_algorithm_ConstP.Constant1_Value[tmp + 12]) +
                ex_rowmajor_algorithm_ConstP.Constant1_Value[tmp + 18];
        }
    }

    /* End of Sum: '<Root>/Sum of Elements Dim1' */
}

```

When **Array layout** is set to Row-major and the **Use algorithms optimized for row-major array layout** configuration parameter is set to off, the code generator uses column-major algorithms. The algorithms traverse the data in column-major order though the data is in row-major order. This process requires some extra operations in the generated code, which makes the code less efficient. When the **Use algorithms optimized for row-major array layout** parameter is set to off, the code generator does not include other vectorization optimizations such as single-instance, multiple-data (SIMD) optimizations.

Generate Code by Using the Row-Major Algorithms for Row-Major Array Layout

To enable the row-major algorithms, on the **Math & Data Types** pane, select the configuration parameter “Use algorithms optimized for row-major array layout” check box. This parameter enables the algorithms that are optimized for row-major array layout. Alternatively, in the MATLAB Command Window, enter:

```
set_param(model, 'UseRowMajorAlgorithm', 'on');
```

After enabling the row-major algorithms, generate code.

```
Value=evalc('slbuild(''ex_rowmajor_algorithm'')');
```

Inspect the generated `ex_rowmajor_algorithm_step` step function in the `ex_rowmajor_algorithm.c`.

```
file = fullfile('ex_rowmajor_algorithm_ert_rtw', 'ex_rowmajor_algorithm.c');
rtwdemodbtype(file, '/* Model step function */', '/* Model initialize function', 1, 1);
```

```
/* Model step function */
```

```
void ex_rowmajor_algorithm_step(void)
{
    int32_T i;

    /* Sum: '<Root>/Sum of Elements Dim1' incorporates:
     * Constant: '<Root>/Constant1'
     */
    ex_rowmajor_algorithm_Y.Out2 = -0.0F;
    for (i = 0; i < 24; i++) {
        ex_rowmajor_algorithm_Y.Out2 +=
            ex_rowmajor_algorithm_ConstP.Constant1_Value[i];
    }

    /* End of Sum: '<Root>/Sum of Elements Dim1' */
}
```

When the **Array layout** is set to Row-major and the **Use algorithms optimized for row-major array layout** configuration parameter is set to on, the code generator generates efficient code. The code generator uses row-major algorithms that traverse the data in row-major order, which reduces the number of operations in the generated code. However, you might notice a minor numeric difference between the outputs of column-major and row-major algorithms due to the operation order.

See Also

Related Examples

- “Code Generation of Matrices and Arrays” on page 24-63
- “Interpolation Algorithm for Row-Major Array Layout” on page 18-10

Generate Row-Major Code for Model That Contains a MATLAB Function Block

Programming languages and environments assume a single array layout for all data. MATLAB and Fortran use column-major layout by default, whereas C and C++ use row-major layout. With Simulink Coder, you can generate C/C++ code that uses row-major layout or column-major layout.

To learn more about row-major code generation, see “Code Generation of Matrices and Arrays” on page 24-63.

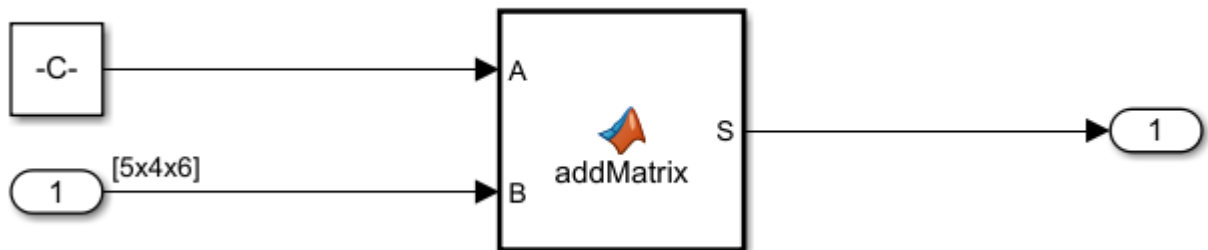
MATLAB Function blocks enable you to define custom functionality in Simulink models by using the MATLAB language. You can generate row-major code for models that contain a MATLAB Function block by using row-major or column-major data. For more information on MATLAB Function blocks, see “Implement MATLAB Functions in Simulink with MATLAB Function Blocks”.

By default, the code generator generates column-major code. For C/C++ code generation, you can specify the array layout at the model level by using the **Array layout** model configuration parameter. Setting this parameter to Row-major enables the model for row-major code generation. To enable the MATLAB Function block in your model for row-major code generation, use the `coder.rowMajor` function at the function level inside the block.

Row-Major Code Generation

For certain algorithms, row-major layout provides more efficient memory access. You get efficient code when you generate code for a model that uses row-major array layout and the model contains a MATLAB Function block that uses an algorithm for row-major data.

- 1 Consider an example model `ex_row_major_MLFB`.



This model contains a Constant block that has a [5 4] matrix. To specify the matrix, set **Constant value** to:

```
reshape(1:20,5,4)
```

The Inport block also specifies a [5 4] matrix. To specify the matrix, set the **Port dimensions** to [5 4].

- 2 In the Configuration Parameters dialog box, set **Array layout** to Row-major.
- 3 Write a function for matrix addition called `addMatrix`. The MATLAB Function block inherits the array layout setting from the model configuration parameter **Array layout** unless specified otherwise.

Optionally, you can use `coder.rowMajor` to explicitly set the array layout of the MATLAB Function block to row-major layout.

```
function S = addMatrix(A,B)
S = zeros(size(A));
for row = 1:size(A,1)
    for col = 1:size(A,2)
        S(row,col) = A(row,col) + B(row,col);
    end
end
```

- 4 Generate code for the model. From the **C Code** tab, click **Build**.

The code generator produces this C code:

```
for (b_row = 0; b_row < 5; b_row++) {
    for (b_col = 0; b_col < 4; b_col++) {
        rtb_S_tmp = (b_row << 2) + b_col;
        rtb_S[rtb_S_tmp] = ex_row_major_MLFB_P.Constant_Value[rtb_S_tmp] +
            ex_row_major_MLFB_U.Inport1[rtb_S_tmp];
    }
}
```

The generated code has two `for` loops. The first `for` loop accesses the rows and the second `for` loop accesses the columns. When the array layout of the MATLAB Function block and the model is the same, the generated code is efficient because no transposes or conversion are required.

Mixed-Majority Code Generation

You can generate mixed-majority code when you have a model that operates on row-major data and a MATLAB Function block that operates on column-major data. When you generate code for a model that uses column-major layout, and the model contains a MATLAB Function block that uses row-major layout, then the code generator converts the block input data to row-major and the block output data back to column-major data, as needed. You can also generate mixed majority code when you have a model that operates on column-major data and a MATLAB Function block that operates on row-major data.

Array layout conversions can affect performance.

- 1 Consider the example model `ex_row_major_MLFB`. For more information on the example model, see “Row-Major Code Generation” on page 18-35.

In the Configuration Parameters dialog box, set **Array layout** to Row-major.

- 2 Update the `addMatrix` function in the MATLAB Function block for column-major data by using the `coder.columnMajor` function.

```
function S = addMatrix(A,B)
coder.columnMajor;
S = zeros(size(A));
for row = 1:size(A,1)
    for col = 1:size(A,2)
        S(row,col) = A(row,col) + B(row,col);
    end
end
```


You can generate mixed-majority code by using the MATLAB Function block. In this case, you configure the model for row-major array layout and the MATLAB Function block for column-major array layout.

- 3 Generate code for the model. From the **C Code** tab, click **Build**.

The code generator produces this C code:

```
for (b_row = 0; b_row < 4; b_row++) {
    for (b_col = 0; b_col < 5; b_col++) {
        B_tmp = (b_col << 2) + b_row;
        B_tmp_0 = b_col + 5 * b_row;
        B[B_tmp_0] = ex_row_major_MLFB_19b_U.Inport1[B_tmp];
        A[B_tmp_0] = ex_row_major_MLFB_19b_P.Constant_Value[B_tmp];
    }
}

for (b_row = 0; b_row < 5; b_row++) {
    /* Output: '<Root>/Output' */
    for (b_col = 0; b_col < 4; b_col++) {
        B_tmp = 5 * b_col + b_row;
        ex_row_major_MLFB_19b_Y.Outport[b_col + (b_row << 2)] = A[B_tmp] + B[B_tmp];
    }
}
```

The inputs to the MATLAB Function block exist in a row-major environment. The code generator performs a conversion operation on inputs before they are fed to the MATLAB Function block because the block is column-major layout. After processing the algorithm in the MATLAB Function block, the code generator converts the data back to row-major data before passing the data to an Output.

See Also

Related Examples

- “Interface with Row-Major Data in MATLAB Function Block”
- “Specify Array Layout in Functions and Classes”

Configuring Data in Generated Code

- “Configure Multi-Instance Code Generation” on page 19-2
- “Configure Data Interfaces” on page 19-3
- “Configure Internal Data” on page 19-4

Configure Multi-Instance Code Generation

By default, for top models, the code generator produces code that is not reentrant. Entry-point functions have a `void-void` interface. Code communicates with other code by sharing access to global data structures that reside in shared memory.

For applications that can benefit from code reuse and require that each use or instance of the code maintains its own unique data, configure a model such that the code generator produces reentrant, multi-instantiable code. Multiple programs can use reentrant code simultaneously. When you configure a model for reentrancy, the execution (step) entry-point function uses root-level input and output arguments instead of global data structures.

Configure a Top Model for Multi-Instance Code Generation

Set model configuration parameter **Code interface packaging** to `Reusable` function.

Apply additional diagnostic and code generation control by setting these model configuration parameters:

- To select the severity level for diagnostic messages that the code generator displays when a model does not meet requirements for multi-instance code, set parameter **Multi-instance code error diagnostic** to `None`, `Warning`, or `Error`.
- To control how the generated code passes root-level model input and output to the reusable execution (step) function (requires Embedded Coder), set parameter **Pass root-level I/O as** to `Individual arguments`, `Structure reference`, or `Part of model data structure`.
- To include a function in the generated file `model.c` that uses `malloc` to dynamically allocate memory for model data structures (requires Embedded Coder), set parameter **Use dynamic memory allocation for model initialization** to `On`.

For more information, see “Generate Reentrant Code from Top Models” (Embedded Coder).

Configure a Referenced Model for Multi-Instance Code Generation

Set model configuration parameter **Total number of instances allowed per top model** to `Multiple`. With this parameter setting, a model can be referenced more than once in a model hierarchy, if it contains not constructs that preclude multiple references. An error occurs if the model cannot be referenced multiple times, even if only one reference exists.

For more information, see “Generate Reentrant Code from Simulink Function Blocks” (Embedded Coder).

Configure Data Interfaces

- 1 Click the **Model Data Editor** tab.
- 2 Update the model to include the latest model data by clicking the update model data button.
- 3 In the Model Data Editor, with the view set to **Code**, click the tab for the type of data element that you want to configure.
- 4 In the filter text box, enter a string that will help you find the row for that data element.
- 5 In the **Storage Class** column, select a storage class.
- 6 In the table or Property Inspector, set values for available properties.
- 7 Save the model.

Configure Internal Data

- 1 Click the **Model Data Editor** tab.
- 2 Update the model to include the latest model data by clicking the update model data button.
- 3 In the Model Data Editor, with the view set to **Code**, click the tab for the type of data element that you want to configure.
- 4 In the filter text box, enter a string that will help you find the row for the data element.
- 5 In the **Storage Class** column, select a storage class. For example, you might want to change the storage class setting for Model Workspace parameters to `Model default`.
- 6 In the table or Property Inspector, set values for available properties.
- 7 Save the model.

Code Generation

Configuration for Simulink Coder

- “Code Generation Configuration” on page 20-2
- “Configure Code Generation Parameters for Model Programmatically” on page 20-4
- “Configure Model from Command Line” on page 20-6
- “Use Configuration Reference to Select Code Generation Target” on page 20-11
- “Check Model and Configuration for Code Generation” on page 20-16
- “Application Objectives Using Code Generation Advisor” on page 20-18
- “Simulink Coder Model Advisor Checks for Standards and Code Efficiency” on page 20-21
- “Configure Code Comments” on page 20-22
- “Include MATLAB Code as Comments in Generated Code” on page 20-23
- “Construction of Generated Identifiers” on page 20-27
- “Identifier Name Collisions and Mangling” on page 20-28
- “Specify Identifier Length to Avoid Naming Collisions” on page 20-29
- “Specify Reserved Names for Generated Identifiers” on page 20-30
- “Reserved Keywords” on page 20-31
- “Configure Model for Debugging” on page 20-34

Code Generation Configuration

When you are ready to generate code for a model, you can modify the model configuration parameters specific to code generation. The code generation parameters determine how the code generator produces code and builds an executable program from your model.

The model configuration parameters for code generation are in the **Code Generation** pane in the Configuration Parameters dialog box. The content of the **Code Generation** pane and its subpanes can change depending on the target that you specify. Some configuration options are available only with the Embedded Coder product.

Your application objectives can include a combination of these code generation objectives: debugging, traceability, execution efficiency, and safety precaution. There are tradeoffs associated with these configuration choices, such as execution speed and memory usage. To help configure a model to achieve your application objectives, use the Model Advisor and the Code Generation Advisor.

Open the Model Configuration for Code Generation

To modify the model configuration parameters for code generation, open the **Code Generation** pane. There are several different ways to open the **Code Generation** pane from the Simulink editor:

- To open the Configuration Parameters dialog box, click the model configuration parameters icon.



Then, click **Code Generation** in the left pane.

- On the **C Code** tab, select **Settings > Code Generation Settings**.
- On the **Modeling** tab, click **Model Explorer**, or from the MATLAB command line, type `daexplr` and press **Enter**. In the left pane of the Model Explorer, expand the node for the current model and click the Configurations node. In the **Contents** pane, right-click the configuration and select **Open** from the context menu. Then click **Code Generation** in the left pane.

Note In the Configuration Parameters dialog box, when you change the value of a check box, menu selection, or edit field, the white background of the element changes color to indicate that you made an unsaved change. When you click **OK**, **Cancel**, or **Apply**, the background resets to white.

Configuration Tools

To help you configure your model for code generation and to check your configuration against your code generation objectives, Simulink Coder and Embedded Coder provide several tools.

Goal	Approach	More Information
Automate configuration.	At the MATLAB command line, use the <code>set_param</code> function	“Configure Code Generation Parameters for Model Programmatically” on page 20-4

Goal	Approach	More Information
Configure your model for code generation quickly and easily.	Quick Start tool	<ul style="list-style-type: none"> • “Generate Code by Using the Quick Start Tool” on page 23-2 • “Generate Code by Using the Quick Start Tool” (Embedded Coder)
Use a template to create a model configured for code generation, ready for you to add your own blocks.	Code generation templates	“Generate Code and Simulate Models in a Project” on page 24-97
To configure your model for code generation, use Simulink blocks and predefined or custom MATLAB scripts.	Code Generation Wizards blocks	“Configure and Optimize Model with Configuration Wizard Blocks” (Embedded Coder)
Verify that your model meets standards and guidelines.	Model Advisor	“Check Your Model Using the Model Advisor”
Verify that your model meets your application objectives.	Code Generation Advisor	“Application Objectives Using Code Generation Advisor” on page 20-18

See Also

Related Examples

- “Model Configuration Sets”
- “Configure Code Generation Parameters for Model Programmatically” on page 20-4
- “Application Objectives Using Code Generation Advisor” on page 20-18

Configure Code Generation Parameters for Model Programmatically

You can modify code generation parameters for the active configuration set in the Configuration Parameters dialog box or from the MATLAB command line. Use the command-line approach for creating a script that automates setting parameters for an established model configuration.

Modify Parameters to Support Execution efficiency

In this example, you modify the configuration parameters to support the Code Generation Advisor application objective, Execution efficiency.

Step 1. Open a model.

```
openExample('slexAircraftExample')
```

Step 2. Get the active configuration set.

```
cs = getActiveConfigSet(model);
```

Step 3. Select the Generic Real-Time (GRT) target.

```
switchTarget(cs, 'grt.tlc', []);
```

Step 4. To optimize execution speed, modify parameters.

If your application objective is Execution efficiency, use `set_param` to modify these parameters:

```
set_param(cs, 'MatFileLogging', 'off');
set_param(cs, 'SupportNonFinite', 'off');
set_param(cs, 'RTWCompilerOptimization', 'on');
set_param(cs, 'OptimizeBlockIOStorage', 'on');
set_param(cs, 'EnhancedBackFolding', 'on');
set_param(cs, 'ConditionallyExecuteInputs', 'on');
set_param(cs, 'DefaultParameterBehavior', 'Inlined');
set_param(cs, 'BooleanDataType', 'on');
set_param(cs, 'BlockReduction', 'on');
set_param(cs, 'ExpressionFolding', 'on');
set_param(cs, 'LocalBlockOutputs', 'on');
set_param(cs, 'EfficientFloat2IntCast', 'on');
set_param(cs, 'BufferReuse', 'on');
```

Step 5. Save the model configuration to a file.

Save the model configuration to a file, 'Exec_efficiency_cs.m', and view the parameter settings.

```
saveAs(cs, 'Exec_Efficiency_cs');  
dbtype Exec_Efficiency_cs 1:50
```

See Also

More About

- “Code Generation Configuration” on page 20-2
- “Application Objectives Using Code Generation Advisor” on page 20-18

Configure Model from Command Line

The code generator provides model configuration parameters for customizing generated code. Depending on how you use and interact with the generated code, you make configuration decisions. You choose a configuration that best matches your needs for debugging, traceability, code efficiency, and safety precaution.

It is common to automate the model configuration process by using a MATLAB® script once you have decided upon a desired configuration.

The example describes:

- Concepts of working with configuration parameters
- Documentation to understand the code generation options
- Tools and scripts to automate the configuration of a model

Configuration Parameter Workflows

There are many workflows for Configuration Parameters that include persistence within a single model or persistence across multiple models. Depending on your needs, you can work with configuration sets as copies or references. This example shows the basics steps for working directly with the active configuration set of a model. For a comprehensive description of configuration set features and workflows, see “Manage Configuration Sets for a Model” in the Simulink® documentation.

Configuration Set Basics

Load a model into memory.

```
model='rtwdemo_configwizard';
load_system(model)
```

Obtain the model's active configuration set.

```
cs = getActiveConfigSet(model);
```

Simulink® Coder™ exposes a subset of the code generation options. If you are using Simulink® Coder™, select the Generic Real-Time (GRT) target.

```
switchTarget(cs, 'grt.tlc', []);
```

Embedded Coder® exposes the complete set of code generation options. If you are using Embedded Coder®, select the Embedded Real-Time (ERT) target.

```
switchTarget(cs, 'ert.tlc', []);
```

To automate configuration of models built for GRT- and ERT-based targets, the configuration set **IsERTTarget** attribute is useful.

```
isERT = strcmp(get_param(cs, 'IsERTTarget'), 'on');
```

You can interact with code generation options via the model or the configuration set. This example gets and sets options indirectly via the model.

```
deftParamBehvr = get_param(model, 'DefaultParameterBehavior'); % Get
set_param(model, 'DefaultParameterBehavior', deftParamBehvr) % Set
```

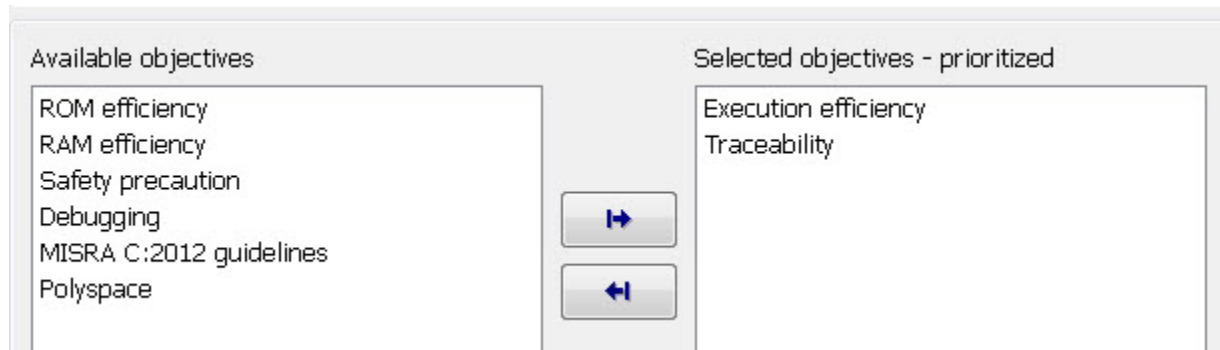
This example gets and sets options directly via the configuration set.

```
if isERT
    lifespan = get_param(cs, 'LifeSpan'); % Get LifeSpan
    set_param(cs, 'LifeSpan', lifespan) % Set LifeSpan
end
```

Configuration Option Summary

The full list of code generation options are documented with tradeoffs for debugging, traceability, code efficiency, and safety precaution. For Simulink® Coder™, see the Simulink® Coder™ version of “Recommended Settings Summary for Model Configuration Parameters”. For Embedded Coder®, see the Embedded Coder® version of “Recommended Settings Summary for Model Configuration Parameters” (Embedded Coder).

Use Code Generation Advisor to obtain a model configuration optimized for your goals. In the Set Objectives dialog box, you can set and prioritize objectives.



You can find documentation about the Code Generation Advisor in “Application Objectives Using Code Generation Advisor” on page 20-18. You can find additional documentation specific to Embedded Coder® in “Configure Model for Code Generation Objectives by Using Code Generation Advisor” (Embedded Coder).

Parameter Configuration Scripts

Simulink® Coder™ provides an example configuration script that you can use as a starting point for your application. A list of the most relevant GRT and ERT code generation options is contained in `rtwconfiguremodel.m`.

Alternatively, you can generate a MATLAB function that contains the complete list of model configuration parameters by using the configuration set `saveAs` function.

```
% Go to a temporary writable directory.
currentDir = pwd;
rtwdemodir();

% Save the model's configuration parameters to file 'MyConfig.m'.
saveAs(cs, 'MyConfig')

% Display the first 50 lines of MyConfig.m.
dbtype MyConfig 1:50
```

```

1  function cs = MyConfig()
2  % MATLAB function for configuration set generated on 04-Mar-2023 00:33:28
3  % MATLAB version: 9.14.0.2206163 (R2023a)
4
5  cs = Simulink.ConfigSet;
6
7  % Original configuration set version: 23.0.0
8  if cs.versionCompare('23.0.0') < 0
9      error('Simulink:MFileVersionViolation', 'The version of the target configuration set is
10 end
11
12 % Character encoding: UTF-8
13
14 % Do not change the order of the following commands. There are dependencies between the pa
15 cs.set_param('Name', 'Configuration'); % Name
16 cs.set_param('Description', ''); % Description
17
18 % Original configuration set target is ert.tlc
19 cs.switchTarget('ert.tlc','');
20
21 cs.set_param('HardwareBoard', 'None'); % Hardware board
22
23 cs.set_param('TargetLang', 'C'); % Language
24
25 cs.set_param('CodeInterfacePackaging', 'Nonreusable function'); % Code interface packagin
26
27 cs.set_param('GenerateAllocFcn', 'off'); % Use dynamic memory allocation for model initi
28
29 cs.set_param('Solver', 'FixedStepDiscrete'); % Solver
30
31 % Solver
32 cs.set_param('StartTime', '0.0'); % Start time
33 cs.set_param('StopTime', '48'); % Stop time
34 cs.set_param('SolverType', 'Fixed-step'); % Type
35 cs.set_param('SampleTimeConstraint', 'STIndependent'); % Periodic sample time constraint
36 cs.set_param('ConcurrentTasks', 'off'); % Allow tasks to execute concurrently on target
37 cs.set_param('AllowMultiTaskInputOutput', 'off'); % Allow multiple tasks to access inputs
38
39 % Data Import/Export
40 cs.set_param('Decimation', '1'); % Decimation
41 cs.set_param('LoadExternalInput', 'off'); % Load external input
42 cs.set_param('SaveFinalState', 'off'); % Save final state
43 cs.set_param('LoadInitialState', 'off'); % Load initial state
44 cs.set_param('LimitDataPoints', 'off'); % Limit data points
45 cs.set_param('SaveFormat', 'StructureWithTime'); % Format
46 cs.set_param('SaveOutput', 'off'); % Save output
47 cs.set_param('SaveState', 'off'); % Save states
48 cs.set_param('SignalLogging', 'on'); % Signal logging
49 cs.set_param('DSMLogging', 'on'); % Data stores
50 cs.set_param('StreamToWks', 'on'); % Stream To Workspace blocks

```

Each parameter setting in the generated file includes a comment for the corresponding parameter string in the Configuration Parameters dialog box.

```

% Return to previous working directory.
cd(currentDir)

```


Configuration Wizard Blocks

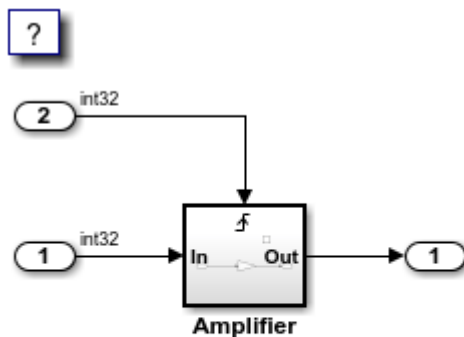
Embedded Coder® provides a set of Configuration Wizard blocks to obtain an initial configuration of a model for a specific goal. For more information, see “Configure and Optimize Model with Configuration Wizard Blocks” (Embedded Coder).

The predefined blocks provide configuration for:

- ERT optimized for fixed point
- ERT optimized for floating point
- GRT optimized for fixed and floating point
- GRT debug settings for fixed and floating point
- Custom (you provide the script)

Put the block into a model and double-click it to configure the model. Open model `rtwdemo_configwizard` and click **Open Configuration Wizard Library** to interact with the blocks.

```
open_system(model)
```



[Open Configuration Parameters](#)

[Open Configuration Wizard Library](#)

Copyright 1994-2020 The MathWorks, Inc.

To use configuration wizard blocks in the `rtwdemo_configwizard` model follow these steps:

- Open the Configuration Wizard Library by clicking the link provided in the model.
- Open the Model's Configuration Parameters by clicking the link provided in the model.
- Drag and drop a Configuration Wizard Block, for example ERT (optimized for fixed point), from the wizard library into the model.
- Double-click the wizard block.

The Configuration Parameter options are modified automatically.

```
% cleanup
rtwdemoclean;
close_system(model,0)
```

Summary

Simulink provides a rich set of MATLAB functions to automate the configuring a model for simulation and code generation. Simulink Coder and Embedded Coder® provide additional functionality specific for code generation. The Code Generation Advisor optimizes the model configuration based on a set of prioritized goals. You can save the optimal configuration to a MATLAB file by using the configuration set saveAs function, and reuse it across models and projects.

See Also**More About**

- “Code Generation Configuration” on page 20-2
- “Application Objectives Using Code Generation Advisor” on page 20-18

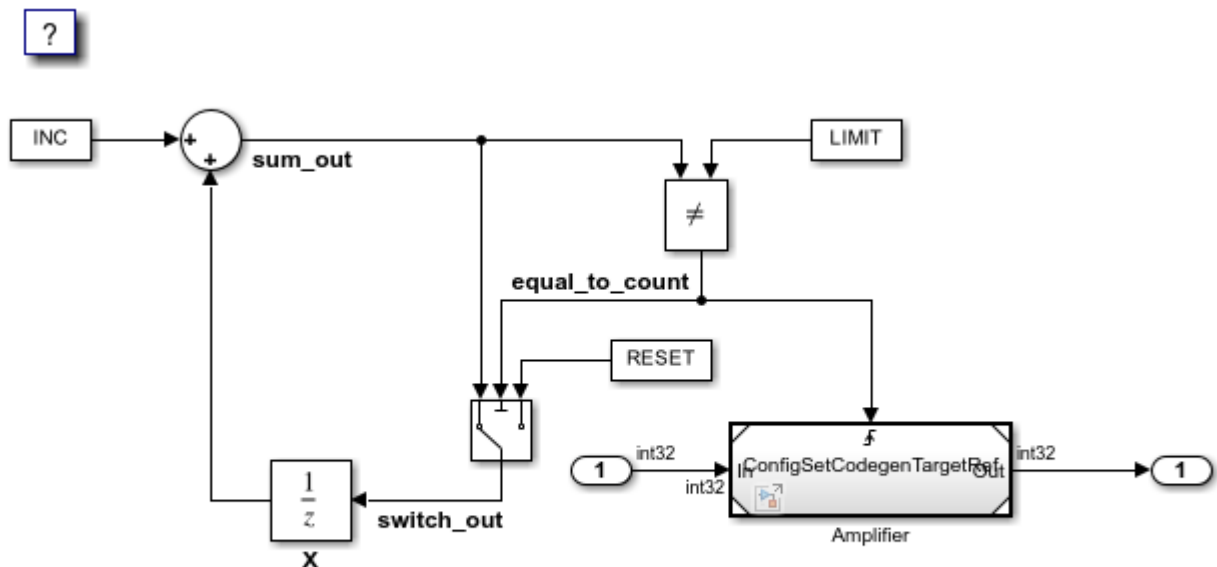
Use Configuration Reference to Select Code Generation Target

This example shows how to use a configuration reference to select a code generation target for a model reference hierarchy without modifying individual models.

Open Example Model

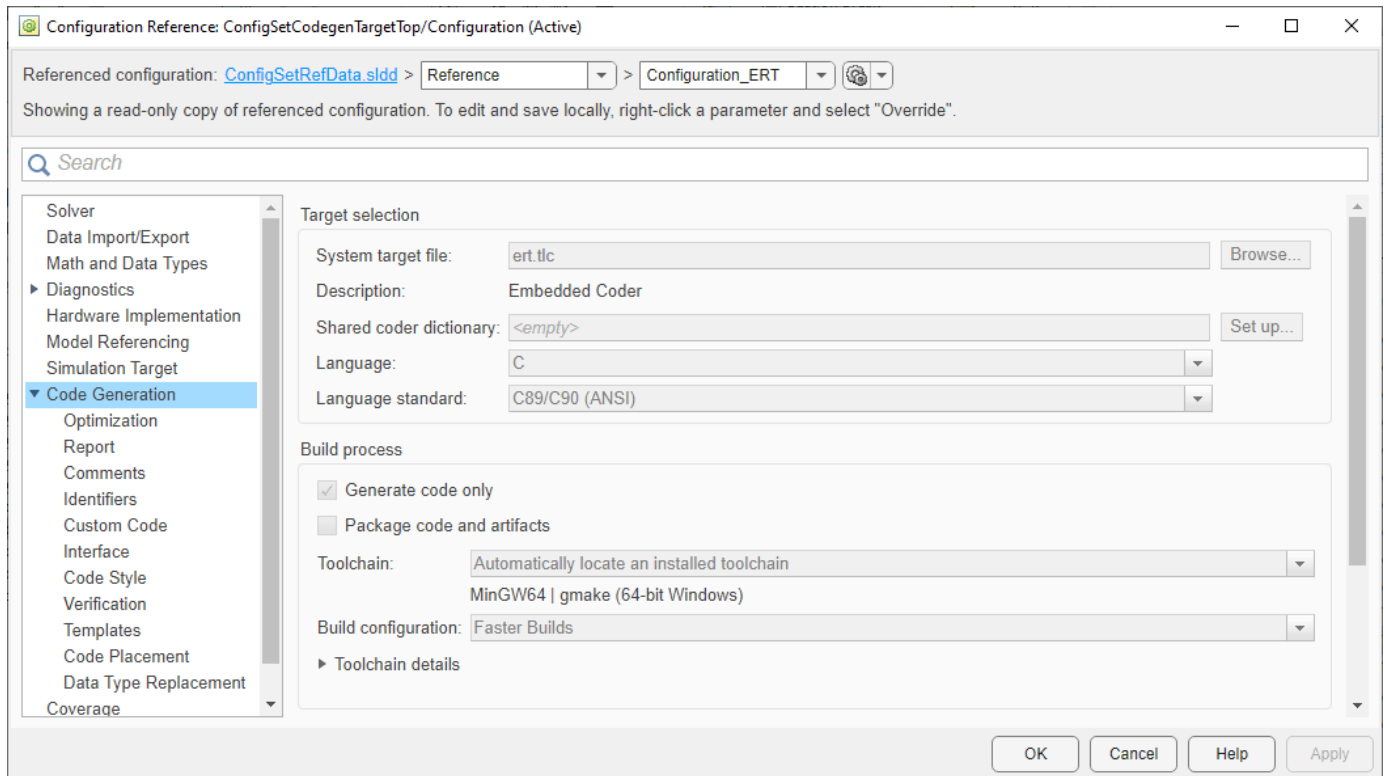
Open the example model `ConfigSetCodegenTargetTop`.

```
open_system('ConfigSetCodegenTargetTop');
```



Copyright 2017 The MathWorks, Inc.

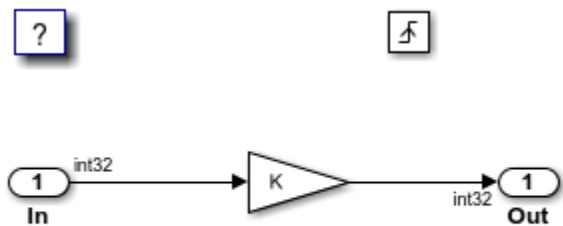
To open the active configuration set for the model `ConfigSetCodegenTargetTop`, on the **Modeling** tab, click the **Model Configuration Parameters** button.



Because the model is using a referenced configuration, the Configuration Reference dialog box opens and displays a read-only view of the referenced configuration set. Information at the top of the dialog box indicates that the model is using the configuration set, Reference, located in the data dictionary, ConfigSetRefData.sldd. In this example, Reference is another configuration reference, Configuration_ERT.

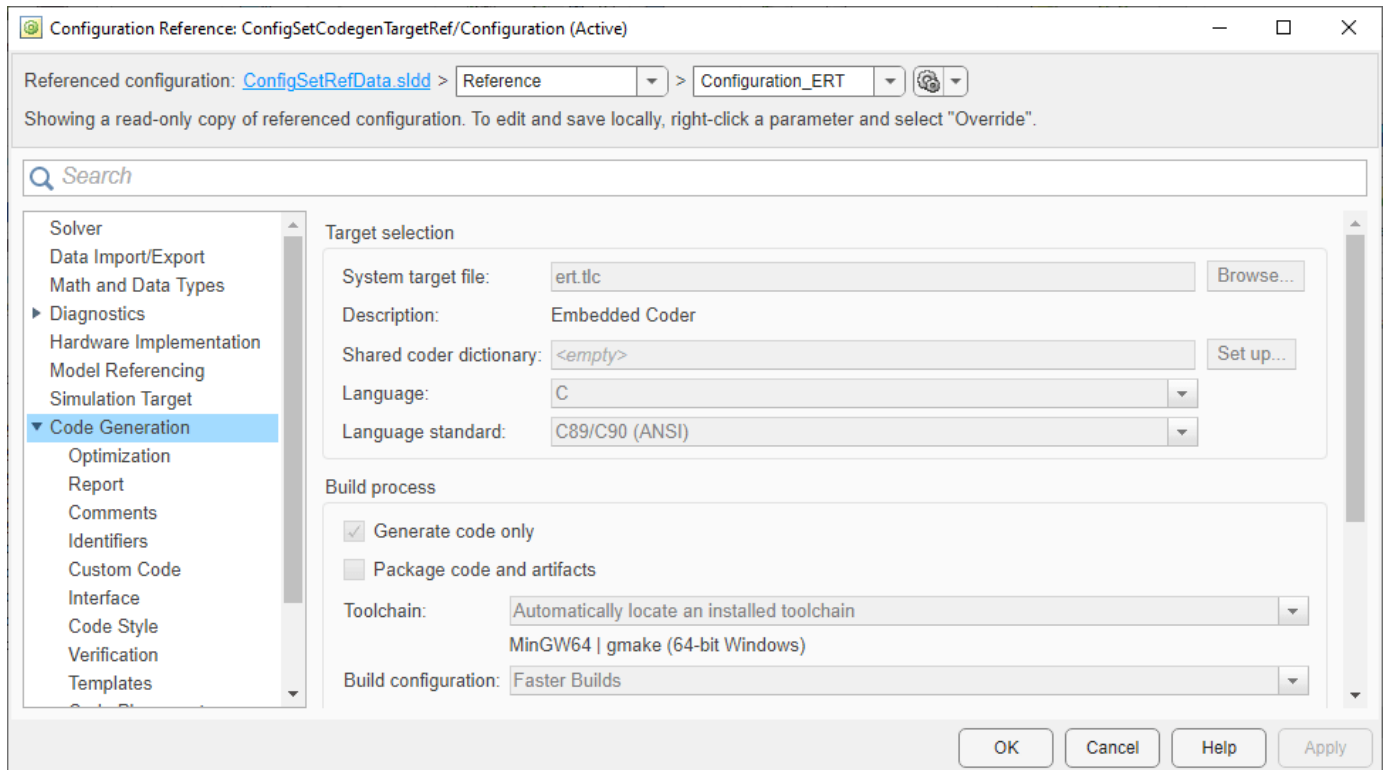
In the model editor, right-click the Model block, Amplifier, and select **Open As Top Model**. This opens the referenced model ConfigSetCodegenTargetRef.

```
open_system('ConfigSetCodegenTargetTop/Amplifier');
```



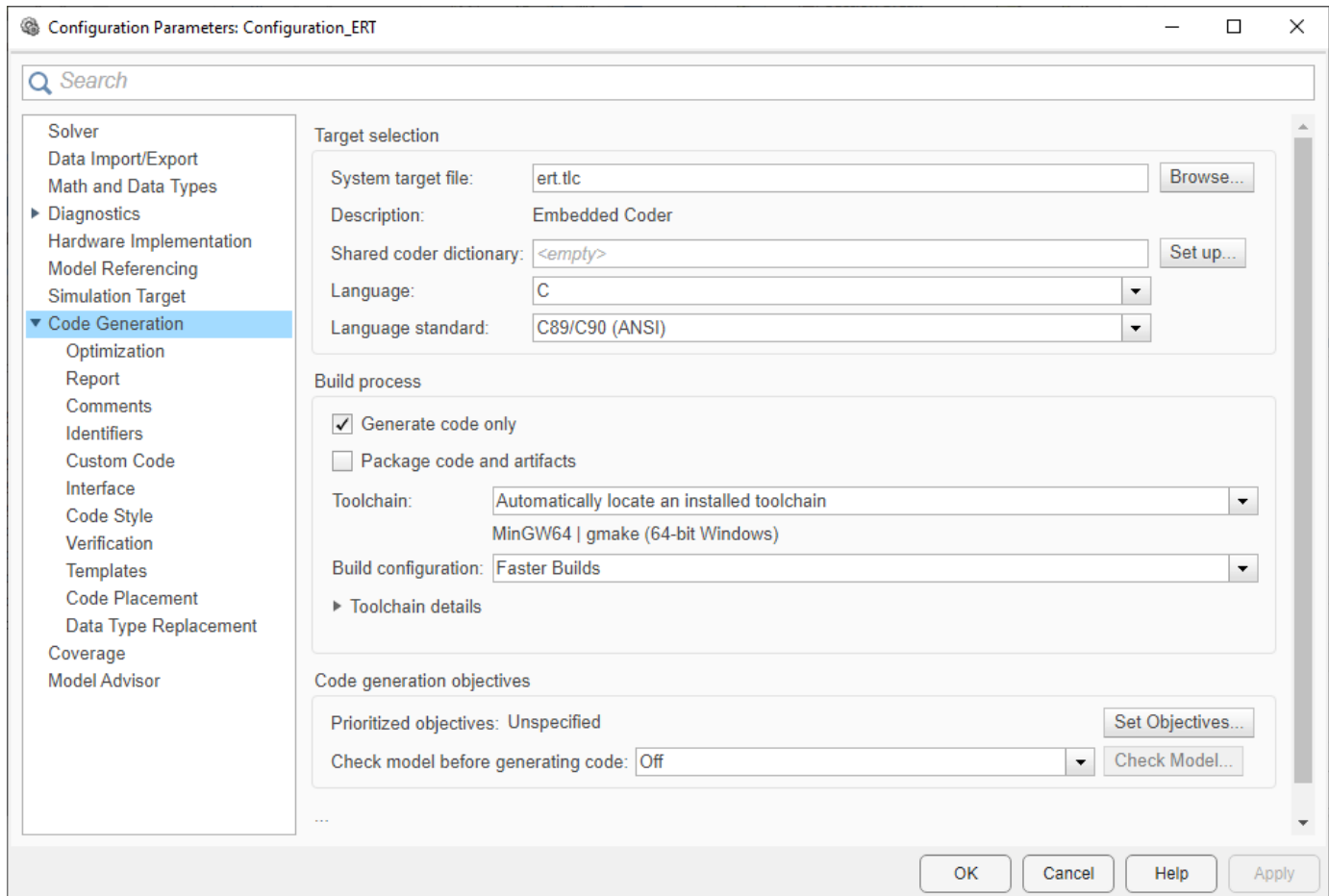
Copyright 2017 The MathWorks, Inc.

Open the active configuration set for this model. The name and location for the referenced configuration is the same as in ConfigSetCodegenTargetTop.



Open the Referenced Configuration Set

If you need to change and apply parameter values in the configuration set, open `Configuration_ERT` for edit in the Configuration Parameters dialog box by clicking the Model Configuration Parameters icon at the top of either of the Configuration Reference dialog boxes.



Generate Code for ERT Target

The referenced configuration set is customized for ERT code generation. To generate code, press **Ctrl +B**.

```
slbuild('ConfigSetCodegenTargetTop');
```

```
### Starting serial model reference code generation build.
### Starting build procedure for: ConfigSetCodegenTargetRef
### Successful completion of code generation for: ConfigSetCodegenTargetRef
### Starting build procedure for: ConfigSetCodegenTargetTop
### Successful completion of code generation for: ConfigSetCodegenTargetTop
```

Build Summary

Code generation targets built:

Model	Action	Rebuild Reason
ConfigSetCodegenTargetRef	Code generated.	ConfigSetCodegenTargetRef.c does not exist.

Top model targets built:

Model	Action	Rebuild Reason
-------	--------	----------------

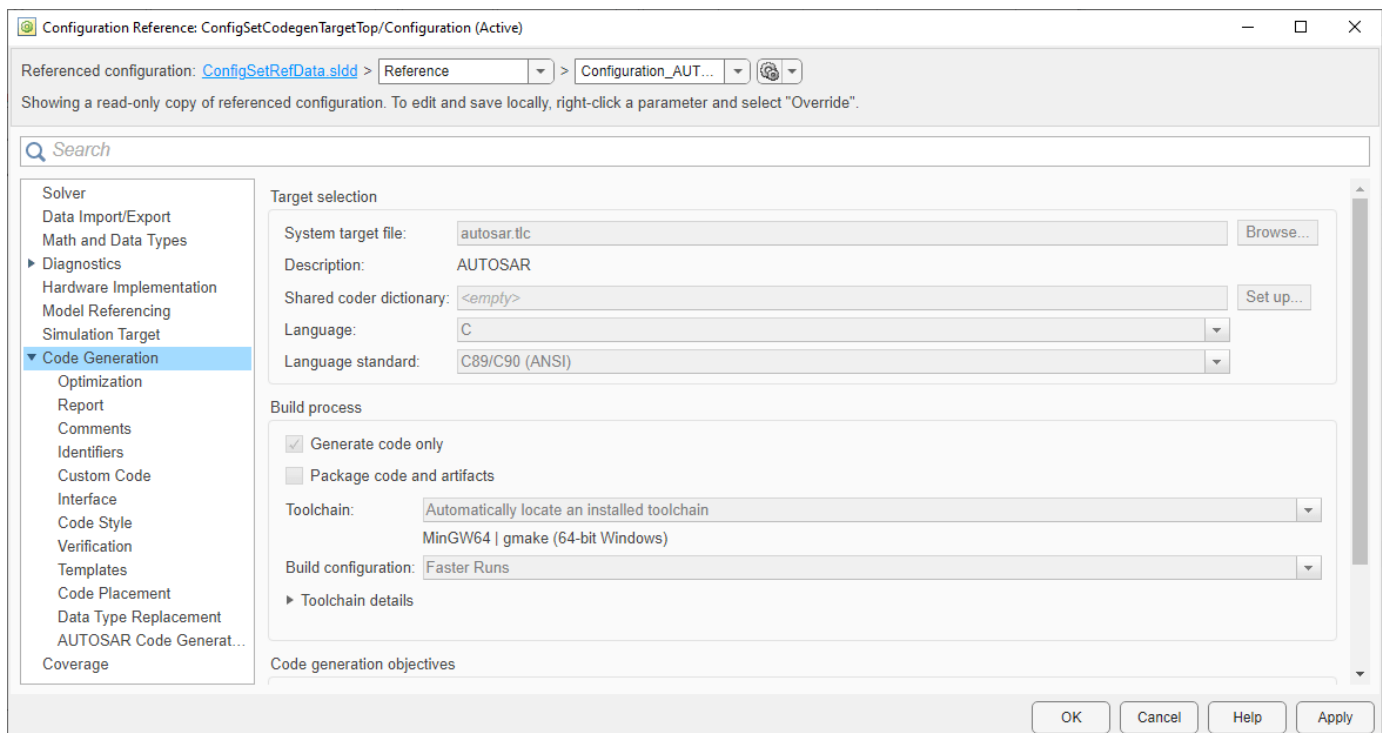
```
ConfigSetCodegenTargetTop Code generated. Code generation information file does not exist.
```

```
2 of 2 models built (0 models already up to date)
Build duration: 0h 0m 20.493s
```

The code generation report displays once code generation is complete.

Switch Targets and Generate Code for AUTOSAR Target

For this model there is another configuration set, Configuration_AUTOSAR, that is customized for AUTOSAR code generation. At the top of the Configuration Reference dialog box, in the rightmost drop-down list, select Configuration_AUTOSAR. `slexConfigSetREfExample` and its referenced model, `slexConfigSetMdlRef`, now both use the configuration set, Configuration_AUTOSAR.



Because you edited the configuration reference, the data dictionary has unsaved changes. The configurations are stored outside of the models so the models do not have unsaved changes.

If you have downloaded the AUTOSAR package, you can now generate code for an AUTOSAR target. In the model editor window, press **Ctrl+B**.

Check Model and Configuration for Code Generation

You can use the Model Advisor checks to assess model readiness to generate code. To check and configure your model for code generation application objectives such as traceability or debugging, use the Code Generation Advisor.

For information about	See
Model Advisor	"Run Model Advisor Checks"
Code Generation Advisor	"Application Objectives Using Code Generation Advisor" on page 20-18
Checks available with Simulink Coder	"Model Advisor Checks"
Checks available with Embedded Coder	"Embedded Coder Checks" (Embedded Coder)

Check Model for Code Efficiency with Model Advisor

To check model `rtwdemo_throttlecntrl` for code efficiency, use the Model Advisor.

- 1 Open `rtwdemo_throttlecntrl`. Save a copy as `throttlecntrl` in a writable location on your MATLAB path.
- 2 To start the Model Advisor, on the **Modeling** tab, click **Model Advisor**. A dialog box opens showing the model system hierarchy.
- 3 Click `throttlecntrl` and then click **OK**. The Model Advisor window opens.
- 4 Expand **By Task > Code Generation Efficiency**. To check your model for code generation efficiency, use the checks in the folder. By default, checks that do not trigger an Update Diagram are selected. The checks available for code generation efficiency depend on whether you have a Simulink Coder or Embedded Coder license.
- 5 In the left pane, select the remaining checks. Right-click on the **Code Generation Efficiency** folder and select **Run Selected Checks**.

The results shows a **Run Summary** that flags check warnings. The warnings identify issues that impact code efficiency. For more information about the report, see "View Model Advisor Check Reports".

Check Model During Code Generation by using the Code Generation Advisor

To review a model as part of the code generation process, use the Code Generation Advisor.

- 1 To specify your code generation objectives, on the **Configuration Parameters > Code Generation** pane, choose a value for the **Select objective** parameter.
- 2 Select one of the following values for the parameter **Check model before generating code**:
 - On (proceed with warnings)
 - On (stop for warnings)
- 3 If you want to only generate code, select **Generate code only**. Otherwise clear the check box to build an executable.
- 4 Apply your changes, and press **Ctrl+B** to generate code. The Code Generation Advisor starts and reviews the top model and subsystems.

If the Code Generation Advisor issues failures or warnings, and you specified:

- On (proceed with warnings) — The Code Generation Advisor window opens while the build process proceeds. After the build process is complete, you can review the results.
 - On (stop for warnings) — The build process halts and displays the Diagnostic Viewer. To continue, you must review and resolve the Code Generation Advisor results or clear the **Check model before generating code** parameter.
- 5 In the Code Generation Advisor dialog box, review the results by selecting a check from the left pane. The results for that check display in the right pane.
 - 6 After reviewing the check results, you can choose to fix warnings and failures as described in “Fix a Model Advisor Check Warning or Failure”.

Note When you specify an efficiency or Safety precaution objective, the Code Generation Advisor includes additional checks. When you make changes to one of these additional checks, previous check results can potentially become invalid and need to be rerun.

For more information, see “Set Objectives — Code Generation Advisor Dialog Box” (Embedded Coder)

See Also

Related Examples

- “Check Your Model Using the Model Advisor”
- “Application Objectives Using Code Generation Advisor” on page 20-18

Application Objectives Using Code Generation Advisor

In this section...

“High-Level Code Generation Objectives” on page 20-18

“Configure Model for Code Generation Objectives Using Code Generation Advisor” on page 20-19

“Configure Model for Code Generation Objectives by Using Configuration Parameters Dialog Box” on page 20-19

Consider how your application objectives, such as efficiency or debugging, map to code generation options in a model configuration set. Parameters that you set in the Configuration Parameters dialog box specify the behavior of a model in simulation and the code generated for the model.

Before generating code, or as part of the code generation process, you can use the Code Generation Advisor to review a model. When you choose to review a model before generating code, you specify which model, subsystem, or referenced model the Code Generation Advisor reviews. When you choose to review a model as part of the code generation process, the Code Generation Advisor reviews the entire system. The Code Generation Advisor uses the information presented in “Recommended Settings Summary for Model Configuration Parameters” to determine the parameter values that meet your objectives. To configure objectives for a model with an ERT-based target, see “Configure Model for Code Generation Objectives Using Code Generation Advisor” (Embedded Coder).

Setting a code generation objective, and then running the Code Generation Advisor provides information on how to meet the code generation objective for your model. The Code Generation Advisor does not alter the generated code. You can use the Code Generation Advisor to make the suggested changes to your model. The generated code is changed only after you modify your model and regenerate code. When you use the Code Generation Advisor to set the code generation objective and check your model, the generated code includes comments identifying which objective you specified, the checks the Code Generation Advisor ran on the model, and the results of running the checks.

If a model uses a configuration reference, you can run the Code Generation Advisor to review your configuration parameter settings. However, the Code Generation Advisor cannot modify the configuration parameter settings.

High-Level Code Generation Objectives

Depending on the type of application that your model represents, you are likely to have specific high-level code generation objectives. For example, debugging might be more critical than efficient execution. If you have a specific objective, you can quickly configure your model to meet that objective by selecting from these code generation objectives:

- Execution efficiency — Configure code generation settings to achieve fast execution time.
- Debugging — Configure code generation settings to debug the code generation build process.

For a list of objectives that are available for models with ERT-based targets, see “High-Level Code Generation Objectives” (Embedded Coder).

Configure Model for Code Generation Objectives Using Code Generation Advisor

This example shows how to use the Code Generation Advisor to check and configure your model to meet code generation objectives:

- 1 On the **C Code** tab, click **C/C++ Code Advisor**.
- 2 In the System Selector window, select the model or subsystem that you want to review, and then click **OK**.
- 3 In the Code Generation Advisor, on the **Code Generation Objectives** pane, select a code generation objective from the drop-down list (GRT-based targets). As you select objectives, on the left pane, the Code Generation Advisor updates the list of checks it will run on your model. If your model is configured with an ERT-based target, more objectives are available.
- 4 Click **Run Selected Checks** to run the checks listed in the left pane of the Code Generation Advisor.
- 5 In the Code Generation Advisor window, review the results for **Check model configuration settings against code generation objectives** by selecting it from the left pane. The results for that check are displayed in the right pane.

Check model configuration settings against code generation objectives triggers a warning for these issues:

- Parameters are set to values other than the value recommended for the specified code generation objective.
- The selected code generation objective differs from the objective set in the model.

Click **Modify Parameters** to set:

- Parameters to the value recommended for the specified code generation objective.
 - Code generation objective in the model to the objective specified in the Code Generation Advisor.
- 6 In the Code Generation Advisor window, review the results for the remaining checks by selecting them from the left pane. The results for the checks display in the right pane.
 - 7 After reviewing the check results, you can choose to fix warnings and failures, as described in “Fix a Model Advisor Check Warning or Failure”.

When you specify a the execution efficiency objective, the Code Generation Advisor includes additional checks. When you make changes to one of these additional checks, previous check results can potentially become invalid and need to be rerun.

Configure Model for Code Generation Objectives by Using Configuration Parameters Dialog Box

This example shows how to check and configure the code generation objectives in the Configuration Parameters dialog box:

- 1 Open the Configuration Parameters dialog box and select **Code Generation**.
- 2 Select or confirm selection of a System target file.
- 3 Specify the objective using the **Select objectives** drop-down list.

- 4** Click **Check Model** to run the model checks.
- 5** In the System Selector window, select the model or subsystem that you want to review, and then click **OK**. The Code Generation Advisor dialog box opens. The Code Generation Advisor uses the code generation objective to determine which model checks to run.
- 6** On the left pane, the Code Generation Advisor lists the checks run on the model and the results. Click each warning to see the suggestions for changes that you can make to your model to pass the check.
- 7** Determine which changes to make to your model. On the right pane of the Code Generation Advisor, follow the instructions listed for each check to modify the model.

Simulink Coder Model Advisor Checks for Standards and Code Efficiency

To check that your model meets standards and is ready to generate code, you can use the Model Advisor checks available with Simulink Coder.

- On the **Modeling** tab, click **Model Advisor**.
- In the Model Advisor window, expand the **By Product** folder. The **Simulink Coder** folder contains Model Advisor checks that you can run. For a complete list of checks, see “Model Advisor Checks”

For more information, see “Check Your Model Using the Model Advisor”.

See Also

Related Examples

- Modeling Guidelines for Code Generation

Configure Code Comments

Configure how the code generator inserts comments into generated code by modifying parameters on the **Code Generation > Comments** pane.

Goal	Action
Include comments in generated code	Include comments. Selecting this parameter allows you to select one or more autogenerated comment types to be placed in the code.
Include comments that describe a Simulink block's code	Simulink block comments. Selecting this parameter includes the comments before the Simulink block's code in the generated code. In Embedded Coder, use Trace to model using (Embedded Coder) to select Block path or Simulink identifier as the comment format.
Include comments that describe Stateflow objects	Stateflow object comments. Selecting this parameter includes the comments describing Stateflow objects in the generated code. In Embedded Coder, use Trace to model using (Embedded Coder) to select Block path or Simulink identifier as the comment format.
Include MATLAB source code as comments	MATLAB source code as comments. Selecting this parameter inserts these comments preceding the associated generated code. The function signature is included in the function banner.
Include comments for eliminated blocks	Show eliminated blocks. Selecting this parameter includes comments for blocks that were eliminated as the result of optimizations, such as inlining parameters.
Include parameter comments regardless of the number of parameters	Verbose comments for 'Model default' storage class. Selecting this parameter includes comments for parameter variable names and names of source blocks in the model parameter structure declaration in <i>model_prm.h</i> . If you do not select this parameter, parameter comments are generated if less than 1000 parameters are declared. This reduces the size of the generated file for models with a large number of parameters.
Include MATLAB user comments	MATLAB user comments (Embedded Coder). Selecting this parameter includes function description comments and other user comments from MATLAB code as comments in the generated code.
Specify comment style	Comment style (Embedded Coder). Select Auto , Multi-line or Single-line as the style of comments in the generated code.

Note When you configure the code generator to produce code that includes comments, the code generator includes text for model parameters, block names, signal names, and Stateflow object names in the generated code comments. If the text includes characters that are unrepresented in the character set encoding for the model, the code generator replaces the characters with XML escape sequences. For example, the code generator replaces the Japanese full-width Katakana letter ア with the escape sequence `ア`. For more information, see “Internationalization and Code Generation” on page 22-2.

See Also

More About

- “Model Configuration Parameters: Comments” (Embedded Coder)

Include MATLAB Code as Comments in Generated Code

If you have Simulink Coder, you can include MATLAB source code as comments in the code generated for a MATLAB Function block. Including this information in the generated code enables you to:

- Correlate the generated code with your source code.
- Understand how the generated code implements your algorithm.
- Evaluate the quality of the generated code.

When you select the **MATLAB source code as comments** parameter, the generated code includes:

- The source code as a comment immediately after the traceability tag. When you enable traceability and generate code for ERT targets (requires Embedded Coder), the traceability tags are hyperlinks to the source code. For more information, see “Use Traceability in MATLAB Function Blocks” (Embedded Coder).

For examples and information on the location of the comments in the generated code, see “Location of Comments in Generated Code” (Embedded Coder).

- The function help text in the function body in the generated code. The function help text is the first comment after the MATLAB function signature. It provides information about the capabilities of the function and how to use it.

Note With Embedded Coder, you can also include the function help text in the generated code function banner. For more information, see “Include MATLAB User Comments in Generated Code” (Embedded Coder).

Include MATLAB Code as Comments in the Generated Code

To include MATLAB source code as comments in the code generated for a MATLAB Function block:

- 1 On the **C Code** tab, click **Configuration Parameters**.
- 2 In the **Code Generation > Comments** pane, select **MATLAB source code as comments** and click **Apply**.

Location of Comments in Generated Code

The comments containing the source code appear after the traceability tag in the generated code as follows.

```
/* '<S2>:1:18' for y = 1 : 2 : (HEIGHT-4) */
```

Selecting the **Stateflow object comments** parameter generates the traceability comment '`<S2>:1:18'`. Selecting the **MATLAB source code as comments** parameter generates the `for y = 1 : 2 : (HEIGHT-4)` comment.

Straight-Line Source Code

The comment containing the source code precedes the generated code that implements the source code statement. This comment appears after comments that you add, which precede the generated

code. The comments are separated from the generated code because the statements are assigned to function outputs.

MATLAB Code

```
function [x y] = straightline(r,theta)
%#codegen
% Convert polar to Cartesian
x = r * cos(theta);
y = r * sin(theta);
```

Commented C Code

```
/* MATLAB Function 'straightline': '<S1>:1' */
/* Convert polar to Cartesian */
/* '<S1>:1:4' x = r * cos(theta); */
/* '<S1>:1:5' y = r * sin(theta); */
straightline0_Y.x = straightline0_U.r * cos(straightline0_U.theta);

/* Output: '<Root>/y' incorporates:
 * Inport: '<Root>/r'
 * Inport: '<Root>/theta'
 * MATLAB Function Block: '<Root>/straightline'
 */
straightline0_Y.y = straightline0_U.r * sin(straightline0_U.theta);
```

If Statements

The comment for the `if` statement immediately precedes the code that implements the statement. This comment appears after comments that you add, which precede the generated code. The comments for the `elseif` and `else` clauses appear immediately after the code that implements the clause and before the code generated for statements in the clause.

MATLAB Code

```
function y = ifstmt(u,v)
%#codegen
if u > v
    y = v + 10;
elseif u == v
    y = u * 2;
else
    y = v - 10;
end
```

Commented C Code

```
/* MATLAB Function 'MLFcn': '<S1>:1' */
/* '<S1>:1:3' if u > v */
if (MLFcn_U.u > MLCfn_U.v) {
    /* Output: '<Root>/y' */
    /* '<S1>:1:4' y = v + 10; */
    MLCfn_Y.y = MLCfn_U.v + 10.0;
} else if (MLFcn_U.u == MLCfn_U.v) {
    /* Output: '<Root>/y' */
    /* '<S1>:1:5' elseif u == v */
    /* '<S1>:1:6' y = u * 2; */
    MLCfn_Y.y = MLCfn_U.u * 2.0;
} else {
```



```

/* Output: '<Root>/y' */
/* '<S1>:1:7' else */
/* '<S1>:1:8' y = v - 10; */
MLFcn_Y.y = MLFcn_U.v - 10.0;

```

For Statements

The comment for the `for` statement header immediately precedes the generated code, which implements the header. This comment appears after comments that you add, which precede the generated code.

MATLAB Code

```

function y = forstmt(u)
%#codegen
y = 0;
for i=1:u
    y = y + 1;
end

```

Commented C Code

```

/* MATLAB Function 'MLFcn': '<S1>:1' */
/* '<S1>:1:3' y = 0; */
rtb_y = 0.0;

/* '<S1>:1:5' for i=1:u */
for (i = 1.0; i <= MLFcn_U.u; i++) {
    /* '<S1>:1:6' y = y + 1; */
    rtb_y++;
}

```

While Statements

The comment for the `while` statement header immediately precedes the generated code, which implements the statement header. This comment appears after comments that you add, which precede the generated code.

Switch Statements

The comment for the `switch` statement header immediately precedes the generated code that implements the statement header. This comment appears after comments that you add that precede the generated code. The comments for the `case` and `otherwise` clauses appear immediately after the generated code that implements the clause, and before the code generated for statements in the clause.

Include MATLAB User Comments in Generated Code

MATLAB user comments include the function help text and other comments. The function help text is the first comment after the MATLAB function signature. It provides information about the capabilities of the function and how to use it. You can include the MATLAB user comments in the code generated for a MATLAB Function block.

- 1 On the **C Code** tab, click **Configuration Parameters**.
- 2 In the **Code Generation > Comments** pane, select MATLAB user comments (Embedded Coder) and click **Apply**.

Limitations of MATLAB Source Code as Comments

For including MATLAB source code as comments, the MATLAB Function block has these limitations :

- You cannot include MATLAB source code as comments for:
 - MathWorks toolbox functions
 - P-code
 - Simulation targets
 - Stateflow Truth Table blocks
- The appearance or location of comments can vary depending on these conditions:
 - Comments might still appear in the generated code even if the implementation code is eliminated, for example, due to constant folding.
 - Comments might be eliminated from the generated code if a complete function or code block is eliminated.
 - For certain optimizations, the comments might be separated from the generated code.
 - The generated code includes legally required comments from the MATLAB source code, even if you do not choose to include source code comments in the generated code.

See Also

More About

- “Model Configuration Parameters: Comments” (Embedded Coder)
- “Verify Generated Code by Using Code Tracing” (Embedded Coder)
- “Use Traceability in MATLAB Function Blocks” (Embedded Coder)

Construction of Generated Identifiers

For GRT and RSim targets, the code generator automatically constructs identifiers for variables and functions in the generated code. These identifiers represent:

- Signals and parameters that have `Auto` storage class
- Subsystem function names that are not user-defined
- Stateflow names

The components of a generated identifier include

- The root model name, followed by
- The name of the generating object (signal, parameter, state, and so on), followed by
- Unique *name-mangling* text

The code generator conditionally generates the name-mangling text to resolve potential conflicts with other generated identifiers.

To configure how the code generator names identifiers and objects, see:

- “Specify Identifier Length to Avoid Naming Collisions” on page 20-29
- “Specify Reserved Names for Generated Identifiers” on page 20-30

The code generator reserves certain words for its own use as keywords of the generated code language. For more information, see “Reserved Keywords” on page 20-31.

With an Embedded Coder license, you can specify parameters to control identifier formats, mangle length, scalar inlined parameters, and Simulink data object naming rules. For more information, see “Customize Generated Identifier Naming Rules” (Embedded Coder).

Identifier Name Collisions and Mangling

In identifier generation, a circumstance that would cause generation of two or more identical identifiers is called a *name collision*. When a potential name collision exists, unique *name-mangling* text is generated and inserted into each of the potentially conflicting identifiers. Each set of name-mangling characters is unique for each generated identifier.

Identifier Name Collisions with Referenced Models

Referenced models can introduce additional naming constraints. Within a model that uses referenced models, collisions between the names of the models cannot exist. When you generate code from a model that includes referenced models, the **Maximum identifier length** parameter must be large enough to accommodate the root model name and name-mangling text. A code generation error occurs if **Maximum identifier length** is too small.

When a name conflict occurs between an identifier within the scope of a higher-level model and an identifier within the scope of a referenced model, the identifier from the referenced model is preserved. Name mangling is performed on the identifier from the higher-level model.

For more information on referenced models, see “Parameterize Instances of a Reusable Referenced Model”.

Specify Identifier Length to Avoid Naming Collisions

The length of a generated identifier is limited by the **Maximum identifier length** parameter specified on the **Identifiers** pane of the Configuration Parameters dialog box. The **Maximum identifier length** field allows you to limit the number of characters in function, type definition, and variable names. The default is 31 characters. This is also the minimum length you can specify. The maximum is 256 characters.

When there is a potential name collision between two identifiers, name-mangling text is generated. The text has the minimum number of characters required to avoid the collision. The other symbol components are then inserted. If **Maximum identifier length** is not large enough to accommodate full expansions of the other components, they are truncated. To avoid this outcome, it is good practice to:

- Avoid name collisions by not using default block names (for example, `Gain1`, `Gain2...`) when the model includes multiple blocks of the same type.
- For subsystems, make them atomic and reusable.
- Increase the **Maximum identifier length** parameter to accommodate the length of the identifier you expect to generate.

Specify Reserved Names for Generated Identifiers

You can specify a set of reserved keywords that the code generation process should not use, facilitating code integration where functions and variables from external environments are unknown in the Simulink model. To create a list of reserved names, open the Configuration Parameters dialog box. On the **Code Generation > Identifiers** pane, enter the keywords in the Reserved names field.

If your model contains MATLAB Function or Stateflow blocks, the code generation process can use the reserved names specified for those blocks if you select **Use the same reserved names as Simulation Target** on the **Code Generation > Identifiers** pane.

Reserved Keywords

In this section...

“C Reserved Keywords” on page 20-31

“C++ Reserved Keywords” on page 20-31

“Other Common Reserved Keywords” on page 20-32

“Code Generation Code Replacement Library Keywords” on page 20-32

Generator keywords are reserved for internal use. Do not use them in models as identifiers or function names. Also avoid using C reserved keywords in models as identifiers or function names. If your model contains reserved keywords, the code generator attempts to replace those keywords when it generates identifiers in the code. If the code generator cannot replace the reserved keyword, then the generated code uses reserved keywords as identifiers and might not compile. To avoid generating identifiers that do not match identifiers in your model, do not use reserved keywords as identifiers.

Note You can register additional reserved identifiers in the Simulink environment. For more information, see “Specify Reserved Names for Generated Identifiers” on page 20-30.

C Reserved Keywords

These are the reserved keywords that the C language specifies. The code generator attempts to rename identifiers that match these keywords.

auto	double	int	struct
break	else	long	switch
case	enum	register	typedef
char	extern	return	union
const	float	short	unsigned
continue	for	signed	void
default	goto	sizeof	volatile
do	if	static	while

C++ Reserved Keywords

These are the reserved keywords that the C++ language specifies in addition to the C reserved keywords. If you generate C++ code, the code generator attempts to rename identifiers that match these keywords.

bool	false	protected	try
catch	friend	public	typeid
class	inline	reinterpret_cast	typename
const_cast	mutable	static_cast	using

delete	namespace	template	virtual
dynamic_cast	new	this	wchar_t
explicit	operator	throw	
export	private	true	

Other Common Reserved Keywords

The code generator attempts to replace some keywords that are not part of the C or C++ language specification. See examples of some of these keywords in this table.

abs	int16_T	MAX_uint8_T*	RT_MALLOC
asm	int32_T	MAX_uint16_T*	rtInf
boolean_T	int64_T	MAX_uint32_T*	rtMinusInf
byte_T	INTEGER_CODE	MAX_uint64_T	rtNaN
char_T	LINK_DATA_BUFFER_SIZE	MIN_int8_T*	SeedFileBuffer
cint8_T	LINK_DATA_STREAM	MIN_int16_T*	SeedFileBufferLen
cint16_T	localB	MIN_int32_T*	single
cint32_T	localC	MIN_int64_T	TID01EQ
creal_T	localDWork	MODEL	time_T
creal32_T	localP	MT	tm
creal64_T	localX	NCSTATES	uint_T
cuint8_T	localXdis	NULL	uint8_T
cuint16_T	localXdot	NUMST	uint16_T
cuint32_T	localZCE	pointer_T	uint32_T
ERT	localZCSV	PROFILING_ENABLED	uint64_T
fortran	matrix	PROFILING_NUM_SAMPLES	UNUSED_PARAMETER
HAVESTDIO	MAX_int8_T*	real_T	USE_RTMODEL
id_t	MAX_int16_T*	real32_T	VCAST_FLUSH_DATA
int_T	MAX_int32_T*	real64_T	vector
int8_T	MAX_int64_T	RT	

*Not reserved if you specify a replacement identifier.

Code Generation Code Replacement Library Keywords

The list of code replacement library reserved keywords for your development environment varies depending on which libraries are registered. The list of available code replacement libraries varies depending on other installed products (for example, a target product), or if you used Embedded Coder to create and register custom code replacement libraries.

To generate a list of reserved keywords for libraries currently registered in your environment, use this MATLAB function:

```
lib_ids = RTW.TargetRegistry.getInstance.getTflReservedIdentifiers()
```


This function returns an array of library keywords. Specifying the input argument is optional.

Note To list the libraries currently registered in your environment, use the MATLAB command `crviewer`.

To generate a list of reserved keywords for a specific library that you are using to generate code, call the function passing the name of the library as displayed in the **Code replacement library** menu on the **Code Generation > Interface** pane of the Configuration Parameters dialog box. For example,

```
lib_ids = RTW.TargetRegistry.getInstance.getTflReservedIdentifiers('GNU C99 extensions')
```

Here is a partial example of the function output:

```
>> lib_ids = ...
    RTW.TargetRegistry.getInstance.getTflReservedIdentifiers('GNU C99 extensions')

lib_ids =

    'exp10'
    'exp10f'
    'acosf'
    'acoshf'
    'asinf'
    'asinhf'
    'atanf'
    'atanhf'
    ...
    'rt_lu_cplx'
    'rt_lu_cplx_sgl'
    'rt_lu_real'
    'rt_lu_real_sgl'
    'rt_mod_boolean'
    'rt_rem_boolean'
    'strcpy'
    'utAssert'
```

Note Some of the returned keywords appear with the suffix `$N`, for example, `'rt_atan2$N'`. `$N` expands into the suffix `_snf` only if nonfinite numbers are supported. For example, `'rt_atan2$N'` represents `'rt_atan2_snf'` if nonfinite numbers are supported and `'rt_atan2'` if nonfinite numbers are not supported. As a precaution, you should treat both forms of the keyword as reserved.

Configure Model for Debugging

In the Configuration Parameters dialog box, use parameters on the **Diagnostics** pane and debugging parameters to configure a model such that the generated code and the build process are set for debugging. You can set parameters that apply to the model compilation phase, the target language code generation phase, or both.

Parameters in the following table will be helpful if you are writing TLC code for customizing targets, integrating legacy code, or developing new blocks.

To...	Select...
Display progress information during code generation in the MATLAB Command Window	Verbose build. Compiler output also displays.
Prevent the build process from deleting the <i>model.rtw</i> file from the build folder at the end of the build	Retain .rtw file. This parameter is useful if you are modifying the target files, in which case you need to look at the <i>model.rtw</i> file.
Instruct the TLC profiler to analyze the performance of TLC code executed during code generation and generate a report	Profile TLC. The report is in HTML format and can be read in your web browser.
Start the TLC debugger during code generation	Start TLC debugger when generating code. Alternatively, enter the argument <code>-dc</code> for the System target file parameter on the Code Generation pane. To start the debugger and run a debugger script, enter <code>-df filename</code> for System target file .
Generate a report containing statistics indicating how many times the code generator reads each line of TLC code during code generation	Start TLC coverage when generating code. Alternatively, enter the argument <code>-dg</code> for the System Target File parameter on the Code Generation pane.
Halt a build if a user-supplied TLC file contains an <code>%assert</code> directive that evaluates to <code>FALSE</code>	<p>Enable TLC assertion. Alternatively, you can use MATLAB commands to control TLC assertion handling.</p> <p>To set the flag on or off, use the <code>set_param</code> command. The default is off.</p> <pre>set_param(model, 'TLCAssertion', 'on off')</pre> <p>To check the current setting, use <code>get_param</code>.</p> <pre>get_param(model, 'TLCAssertion')</pre>

To...	Select...
Detect loss of tunability	<p data-bbox="589 300 1479 485">“Detect loss of tunability” on the Diagnostics > Data Validity pane. You can use this parameter to report loss of tunability when an expression is reduced to a numeric expression. This can occur if a tunable workspace variable is modified by Mask Initialization code, or is used in an arithmetic expression with unsupported operators or functions. Possible values are:</p> <ul data-bbox="589 510 1479 632" style="list-style-type: none"><li data-bbox="589 510 1479 548">• none — Loss of tunability can occur without notification.<li data-bbox="589 552 1479 590">• warning — Loss of tunability generates a warning (default).<li data-bbox="589 594 1479 632">• error — Loss of tunability generates an error. <p data-bbox="589 657 1479 716">For a list of supported operators and functions, see “Tunable Expression Limitations” on page 15-122</p>

To...	Select...
Enable model verification (assertion) blocks	<p>“Model Verification block enabling”. Use this parameter to enable or disable model verification blocks such as Assert, Check Static Gap, and related range check blocks. The diagnostic applies to generated code and simulation behavior. For example, simulation and code generation ignore this parameter when model verification blocks are inside an S-function. Possible values are:</p> <ul style="list-style-type: none"> • User local settings • Enable All • Disable All <p>For Assertion blocks not disabled, generated code for a model includes one of the following statements, depending on the blocks input signal type (Boolean, real, or integer, respectively).</p> <pre>utAssert(input_signal); utAssert(input_signal != 0.0); utAssert(input_signal != 0);</pre> <p>If the model contains variant blocks with the <code>Variant activation time</code> parameter set to <code>startup</code> and the <code>Allow zero active variant controls</code> parameter set to <code>off</code>, the <code>model_initialize</code> function in the generated code contains this assertion.</p> <pre>utAssert(<variant condition expression on the block>);</pre> <p>This statement checks that at least one variant choice is active for such variant blocks.</p> <p>By default, <code>utAssert</code> does not change generated code. For assertions to abort execution, you must enable them by specifying the following <code>make_rtw</code> command for Code Generation > Make command parameter:</p> <pre>make_rtw OPTS="-DDOASSERTS"</pre> <p>Use the following variant if you want triggered assertions to print the assertion statement instead of aborting execution:</p> <pre>make_rtw OPTS="-DDOASSERTS -DPRINT_ASSERTS"</pre> <p><code>utAssert</code> is defined as <code>#define utAssert(exp) assert(exp)</code>.</p> <p>To customize assertion behavior, provide your own definition of <code>utAssert</code> in a handwritten header file that overrides the default <code>utAssert.h</code>. For details on how to include a customized header file in generated code, see “Integrate External Code by Using Model Configuration Parameters” on page 30-27.</p> <p>When running a model in accelerator mode, the Simulink engine calls back to itself to execute assertion blocks instead of using generated code. Thus, user-defined callbacks are still called when assertions fail.</p>

For more information about the TLC debugging options, see Debugging on “Target Language Compiler”. Also, consider using the Model Advisor as a tool for troubleshooting model builds. For more information about the Model Advisor, see “Run Model Advisor Checks”.

See Also

More About

- “Tunable Expression Limitations” on page 15-122
- “Integrate External Code by Using Model Configuration Parameters” on page 30-27
- “Target Language Compiler”

System Target File Configuration

- “Configure a System Target File” on page 21-2
- “Configure STF-Related Code Generation Parameters” on page 21-5
- “Configure Standard Math Library for Target System” on page 21-12
- “Compare System Target File Support Across Products” on page 21-17

Configure a System Target File

To configure a model for code generation, follow the steps in “Select a Solver That Supports Code Generation” on page 21-2 and “Select a System Target File from STF Browser” on page 21-2. When you select a system target file, other model configuration parameters change to serve requirements of the execution environment. For example:

- Code interface parameters
- Build process parameters, such as the toolchain or template makefile
- Target hardware parameters, such as word size and byte ordering

After selecting a system target file, you can modify model configuration parameter settings.

You can switch between different system target files in a single workflow for different code generation purposes (for example, rapid prototyping versus production code deployment). To switch, set up different configuration sets for the same model and switch the active configuration set for the current operation. For more information on how to set up configuration sets and change the active configuration set, see “Manage Configuration Sets for a Model”.

Select a Solver That Supports Code Generation

To build a model, the model configuration must specify a solver that is compatible with code generation for the system target file. Few system target files support code generation with variable-step solvers or for models with a nonzero start time.

- For GRT, ERT, and ERT-based system target files, set the **Type** configuration parameter to **Fixed-step**.
- For Rapid Simulation (Rsim) or S-Function (rtwscfn) system target files, set the **Type** configuration parameter to **Fixed-step** or **Variable-step**.

For more information about the requirement that you use a fixed-step solver to generate code for a realtime system target file, see “Time-Based Scheduling and Code Generation” (Embedded Coder).

The Rapid Simulation (RSim) system target file supports a nonzero start time when the **Solver selection** configuration parameter is set to **Use Simulink solver module**. Other system target files do not support a nonzero start time -- the code generator does not produce code and the build process produces an error.

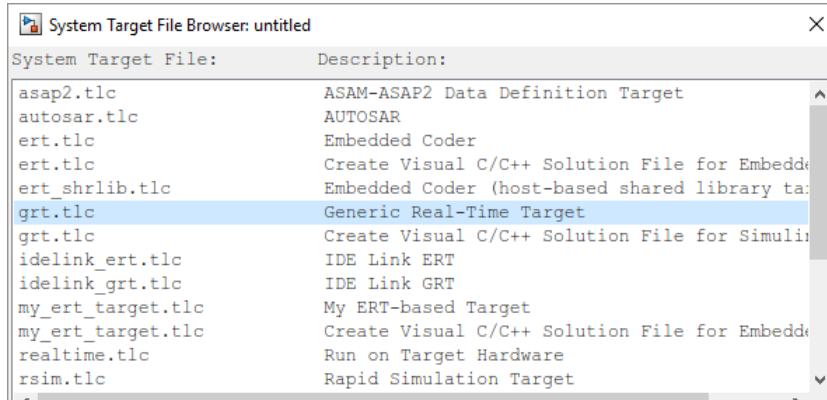
Select a System Target File from STF Browser

After you select a solver (see “Select a Solver That Supports Code Generation” on page 21-2), specify a value for the **System target file** configuration parameter. Enter the name of your system target file directly in the field or use the System Target File Browser to select a file.

To use the System Target File Browser:

- 1 In the configuration parameters dialog box, next to the **System target file** field, click the **Browse** button. The browser displays a list of available system target files, including customizations.

- From the list, select your file, for example, the GRT system target file (`grt.tlc`). The background of the list box turns yellow to indicate that the choice is not applied. Click **Apply** or **OK**.



System Target File Browser

You also can select a system target file programmatically from MATLAB code, as described in “Select a System Target File Programmatically” on page 21-3.

After selecting a system target file, you can modify model configuration parameter settings. Selecting a system target file for your model selects either the toolchain approach or template makefile approach for build process control. For more information about these approaches, see “Configure Toolchain (ToolchainInfo) or Template Makefile Build Process” on page 31-20.

If you want to switch between different system target files in a single workflow for different code generation purposes, set up different configuration sets for the same model. Switch the active configuration set for the current operation. This approach is useful for switching between rapid prototyping and production code deployment. For more information on how to set up configuration sets and change the active configuration set, see “Manage Configuration Sets for a Model”.

Select a System Target File Programmatically

Simulink models store model-wide parameters and system target file-specific data in *configuration sets*. Every configuration set contains a component that defines the structure of a particular system target file and the current values of relevant options. Simulink loads some of this information from the system target file that you specify. You can configure models to generate alternative code by copying and modifying old or adding new configuration sets and browsing to select a new system target file. Then, you can interactively select an active configuration from among these sets (only one configuration set can be active at a given time).

Scripts that automate system target file selection must emulate this process.

To program system target file selection:

- Obtain a handle to the active configuration set with a call to the `getActiveConfigSet` function.
- Define character vector variables that correspond to the required system target file, toolchain or template makefile, and/or make command settings. For example, for the ERT system target file, you would define variables for the character vectors `'ert.tlc'`, `'ert_default_tmf'`, and `'make_rtw'`.

- 3 Select the system target file with a call to the `switchTarget` function. In the function call, specify the handle for the active configuration set and the system target file.
- 4 Set the `TemplateMakefile` and `MakeCommand` configuration parameters to the corresponding variables created in step 2.

For example:

```
cs = getActiveConfigSet(model);
stf = 'ert.tlc';
tmf = 'ert_default_tmf';
mc = 'make_rtw';
switchTarget(cs,stf,[]);
set_param(cs,'TemplateMakefile',tmf);
set_param(cs,'MakeCommand',mc);
```

For more information about selecting system target files programmatically, see `switchTarget`.

Develop Custom System Target Files

You can create your own system target files that interface with external code or operating environments.

For more information on how to make custom system target files appear in the System Target File Browser and display relevant controls, see “Custom Targets” on page 46-2 and the topics it references.

See Also

`getActiveConfigSet` | `switchTarget`

More About

- “Configure STF-Related Code Generation Parameters” on page 21-5
- “Choose a Code Replacement Library” on page 26-6
- “Configure Standard Math Library for Target System” on page 21-12
- “Compare System Target File Support Across Products” on page 21-17
- “Custom Targets” on page 46-2

Configure STF-Related Code Generation Parameters

Many model configuration parameters for code generation are specific to GRT, ERT, or ERT-based system target files.

Specify Generated Code Interfaces

Use interface model configuration parameters to control which libraries to use when generating code, whether to include support for an API in generated code, and other interface options.

To...	Select or Enter...
Specify the language standard that the code generator uses when generating code.	<p>Select C89/C90 (ANSI), C99 (ISO), or C++03 (ISO) for the Language standard parameter.</p> <p>Selecting C89/C90 (ANSI) provides the ANSI^a C set of library functions. For example, selecting C89/C90 (ANSI) results in generated code that calls <code>sin()</code> whether the input argument is double precision or single precision. However, if you select C99 (ISO), the generated code calls the function <code>sinf()</code> when the input argument is single precision. If your compiler supports the ISO^b C math extensions, selecting the ISO C library can result in more efficient code.</p> <p>For more information, see Language standard.</p> <p>The options for this parameter have dependencies. See Interface Dependencies.</p>
Specify an application-specific library that the code generator uses when generating code.	<p>If you generate application-specific C or C++ code for math functions or operations, select a value for Code replacement library. Otherwise, specify None.</p> <p>For more information about code replacement libraries, see “Choose a Code Replacement Library” on page 26-6 and Code replacement library.</p> <p>The options for this parameter have dependencies. See Interface Dependencies.</p>
Direct where the code generator places fixed-point and other utility code.	<p>Select Auto or Shared location for Shared code placement. The shared location directs code for utilities to be placed within the <code>slprj</code> folder in your working folder, which is used for building referenced models. If you select Auto,</p> <ul style="list-style-type: none"> • When the model contains Model blocks, places utility code within the <code>slprj/target/_sharedutils</code> folder. • When the model does not contain Model blocks, places utility code in the build folder (generally, in <code>model.c</code> or <code>model.cpp</code>).

To...	Select or Enter...
Specify text to be added to the variable names used when logging data to MAT-files and to distinguish logging data from code generation and simulation applications.	Enter a prefix or suffix, such as <code>rt_</code> or <code>_rt</code> , for the MAT-file variable name modifier parameter. The code generator prefixes or appends the text to the variable names for system outputs, states, and simulation time specified in the Data Import/Export pane. See “Log Program Execution Results” on page 33-2 for information on MAT-file data logging.
Specify data exchange APIs to be included in generated code.	<p>Select one or more C API options, the ASAP2 interface option, or the External mode option. When you select External mode, other options appear. The data exchange APIs are independent, and you can select combinations of these APIs. For example, you could choose C API and external mode.</p> <p>For more information on working with these interfaces, see “Exchange Data Between Generated and External Code Using C API” on page 34-2, “Export ASAP2 File for Data Measurement and Calibration” on page 35-2, and “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” on page 32-2.</p> <p>The options for this parameter have dependencies. See Interface Dependencies.</p>

a ANSI is a registered trademark of the American National Standards Institute, Inc.
b ISO is a registered trademark of the International Organization for Standardization.

Note Before setting **Language standard** or **Code replacement library**, verify that your compiler supports the library you want to use. If you select a parameter value that your toolchain does not support, compiler errors can occur. For example, if you select the language standard C99 (ISO) and your compiler does not support the ISO C math extensions, compile-time errors could occur.

When the Embedded Coder product is installed on your system, the **Code Generation > Interface** pane expands to include several additional options. For descriptions of **Code Generation > Interface** pane parameters, see “Model Configuration Parameters: Code Generation Interface”.

Several interface parameters have dependencies on settings of other parameters. The following table summarizes the dependencies.

Interface Dependencies

Parameter	Dependencies?	Dependency Details
Language standard	Yes	Available values depend on Language selection.
Code replacement library	Yes	Available values depend on product licensing and other parameters. For more information, see Code replacement library.
Shared code placement	No	
Support: floating-point numbers (ERT system target files only)	No	
Support: non-finite numbers	Yes (ERT) No (GRT)	For ERT system target files, enabled by Support floating-point numbers
Support: complex numbers (ERT system target files only)	No	
Support: absolute time (ERT system target files only)	No	
Support: continuous time (ERT system target files only)	Yes	Requires that you disable Remove error status field in real-time model data structure .
Support: non-inlined S-functions (ERT system target files only)	Yes	Requires that you enable Support floating-point numbers and Support non-finite numbers
Classic call interface	Yes	Requires that you disable Single output/update function . For ERT system target files, requires that you enable Support floating-point numbers .
Single output/update function	Yes	Disable for Classic call interface
Terminate function required (ERT system target files only)	Yes	
Code interface packaging	Yes	Available values depend on Language selection.
Multi-instance code error diagnostic	Yes	Set Code interface packaging to Reusable function or C++ class
Pass root-level I/O as (ERT system target files only)	Yes	Set Code interface packaging to Reusable function
Use dynamic memory allocation for model initialization (ERT system target files only)	Yes	Set Code interface packaging to Reusable function
MAT-file logging	Yes	For GRT system target files, requires that you enable Support non-finite numbers ; for ERT system target files, requires that you enable Support floating-point numbers , Support non-finite numbers , and Terminate function required

Parameter	Dependencies?	Dependency Details
MAT-file file variable name modifier	Yes	Enabled by MAT-file logging
Remove error status field in real-time model data structure (ERT system target files only)	Yes	Requires that you disable Support: continuous time .
Generate C API for: signals	No	
Generate C API for: parameters	No	
Generate C API for: states	No	
Generate C API for: root-level I/O	No	
ASAP2 interface	No	
External mode	No	
Transport layer	Yes	Enable External mode
MEX-file arguments	Yes	Enable External mode
Static memory allocation	Yes	Enable External mode
Static memory buffer size	Yes	Enable Static memory allocation

Configure Numeric Data Support

By default, ERT system target files support code generation for integer, floating-point, nonfinite, and complex numbers.

To Generate Code That Supports...	Do...
Integer data only	Clear Support floating-point numbers . If noninteger data or expressions are encountered during code generation, an error message reports the offending blocks and parameters.
Floating-point data	Select Support floating-point numbers .
Nonfinite values (for example, NaN, Inf)	Select Support floating-point numbers and Support non-finite numbers .
Complex data	Select Support complex numbers .

For more information, see “Model Configuration Parameters: Code Generation Interface”.

Configure Time Value Support

Certain blocks require the value of absolute time, elapsed time, or continuous time. Absolute time is the time from the start of program execution to the present time. Elapsed time is the time elapsed between two trigger events. Depending on the blocks used, your model could require adjustment of the configuration settings for supported time values.

To...	Select...
Generate code that creates and maintains integer counters for blocks that use absolute or elapsed time values (default).	Support absolute time. For further information on the allocation and operation of absolute and elapsed timers, see “Absolute and Elapsed Time Computation” on page 4-2. If you do not select this parameter and the model includes a block that uses absolute or elapsed time values, the build process generates an error.
Generate code for blocks that rely on continuous time.	Support continuous time. If you do not select this parameter and the model includes continuous-time blocks, the build process generates an error.

For more information, see “Model Configuration Parameters: Code Generation Interface”.

Configure Noninlined S-Function Support

To generate code for noninlined S-Functions in a model, select **Support noninlined S-functions**. The generation of noninlined S-functions requires floating-point and nonfinite numbers. Thus, when you select **Support non-inlined S-functions**, the ERT system target file selects **Support floating-point numbers** and **Support non-finite numbers**.

When you select **Support non-finite numbers** and the model includes a C MEX S-function that does not have a corresponding TLC implementation (for inlining code generation), the build process generates an error.

Inlining S-functions is highly advantageous in production code generation, for example in implementing device drivers. To enforce the use of inlined S-functions for code generation, clear **Support non-inlined S-functions**.

When generating code for a model that contains noninlined S-functions with an ERT system target file, there can be a mismatch between the simulation and code generation results when either of the following is true:

- Model configuration parameter `GenCodeOnly` is set to off.
- Model configuration parameter `ProdEqTarget` is set to off.

To avoid such a mismatch, set `ProdEqTarget` to on or set `GenCodeOnly` to on.

Configure Model Function Generation and Argument Passing

For ERT system target files, you can configure model for how functions are generated and how arguments are passed to the functions.

To...	Do...
Generate model function calls that are compatible with the main program module of the pre-R2012a GRT system target file (<code>grt_main.c</code> or <code>.cpp</code>).	Select Classic call interface and MAT-file logging . In addition, clearing Remove error status field in real-time model data structure . Classic call interface provides a quick way to use code generated in R2012a or higher with a pre-R2012a GRT-based custom system target file by generating wrapper function calls that interface to the generated code.

To...	Do...
Reduce overhead and use more local variables by combining the output and update functions in a single <i>model_step</i> function.	Select Single output/update function Errors or unexpected behavior can occur if a Model block is part of a cycle and the model configuration enables Single output/update function (the default). For more information about direct feed through, see “Algebraic Loop Concepts”.
Generate a <i>model_terminate</i> function for a model not designed to run indefinitely.	Select Terminate function required (Embedded Coder). For more information, see the description of <i>model_terminate</i> .
Generate reusable, reentrant code from a model or subsystem.	Select Generate reusable code . See “Configure Code Reuse Support” on page 21-10 for details.
Statically allocate model data structures and access them directly in the model code.	Clear Generate reusable code . The generated code is not reusable or reentrant. See “Configure Generated C Function Interface for Model Entry-Point Functions” on page 16-79 for information on the calling interface generated for model functions in this case.
Suppress the generation of an error status field in the real-time model data structure, <i>rtModel</i> , for example, if you do not require to log or monitor error messages.	Select Remove error status field in real-time model data structure . Selecting this parameter can also cause the code generator to omit the <i>rtModel</i> structure from the generated code. When generating code for multiple integrated models, set this parameter the same for all of the models. Otherwise, the integrated application could exhibit unexpected behavior. For example, if you select the option in one model but not in another, it is possible that the integrated application could not register the error status. Do not select this parameter if you select the MAT-file logging option. The two options are incompatible.
Open the Configure C Step Function Interface dialog box and modify the <i>model_step</i> function prototype.	In the Code Mapping Editor, on the Entry-Point Functions tab, click in the Function Name column for the base-rate step function. Click the three vertical dots that appear. Then click Configure Prototype . Unless a function prototype was previously configured, the dialog box opens showing a preview of the step function interface without arguments (<i>void-void</i>). The dialog box also shows the current function name. If arguments have previously been configured for the model, the dialog box displays the current settings. You can change the name and argument settings for the step function. For more information, see “Configure Name and Arguments for Individual Step Functions” (Embedded Coder).

For more information, see “Model Configuration Parameters: Code Generation Interface”.

Configure Code Reuse Support

For GRT, ERT, GRT-based, and ERT-based system target files, you can configure how a model reuses code by setting the **Code interface packaging** configuration parameter value to **Reusable function**.

The **Pass root-level I/O as** configuration parameter provides options that control how model inputs and outputs at the root level of the model are passed to the *model_step* function.

To...	Select...
Pass each root-level model input and output argument to the <i>model_step</i> function individually (the default)	Code interface packaging > Reusable function and Pass root-level I/O as > Individual arguments.
Pack root-level input arguments and root-level output arguments into separate structures that are then passed to the <i>model_step</i> function	Code interface packaging > Reusable function and Pass root-level I/O as > Structure reference.
Pack root-level input arguments and root-level output arguments into the model data structure to support reentrant multi-instance code from a model for ERT system target file	Code interface packaging > Reusable function and Pass root-level I/O as > Part of model data structure.

If using the **Code interface packaging** > Reusable function selection, consider using the **Use dynamic memory allocation for model initialization** option to control whether an allocation function is generated. This option applies for ERT system target files.

Sometimes, selecting **Code interface packaging** as Reusable function can generate code that compiles but is not reentrant. For example, if a signal, DWork structure, or parameter data has a storage class other than Auto, global data structures are generated. To handle such cases, use the **Multi-instance code error diagnostic** parameter to choose the severity levels for diagnostics.

Sometimes, the code generator is unable to generate valid and compilable code. For example, if the model contains one of the following, the generated code is invalid.

- An S-function that is not code-reuse compliant
- A subsystem triggered by a wide function-call trigger

In these cases, the build terminates after reporting the problem.

For more information, see “Generate Reentrant Code from Top Models” (Embedded Coder) and “Model Configuration Parameters: Code Generation Interface”.

See Also

More About

- “Configure a System Target File” on page 21-2
- “Choose a Code Replacement Library” on page 26-6
- “Configure Standard Math Library for Target System” on page 21-12
- “Compare System Target File Support Across Products” on page 21-17

Configure Standard Math Library for Target System

Specify standard library extensions that the code generator uses for math operations. When you generate code for a new model or with a new configuration set object, the code generator uses the ISO®/IEC 9899:1999 C (C99 (ISO)) library by default. For preexisting models and configuration set objects, the code generator uses the library specified by the **Standard math library** parameter.

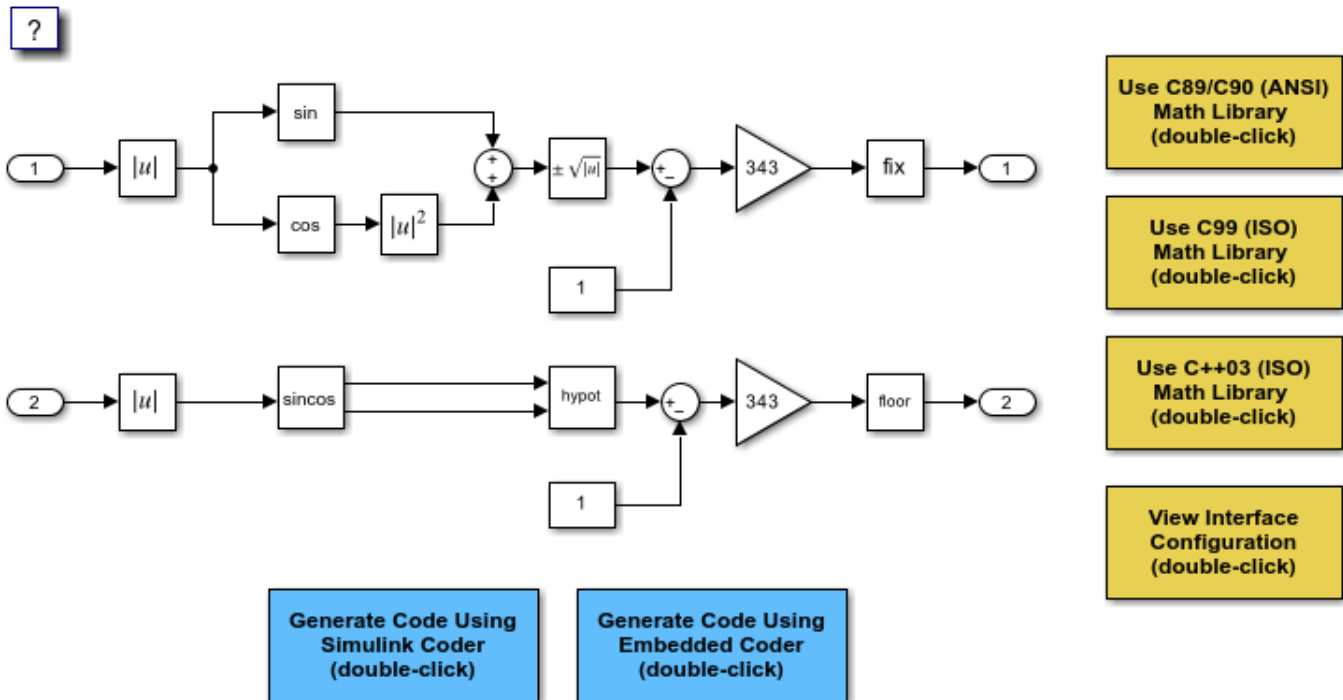
If your compiler supports the ISO®/IEC 9899:1990 (C89/C90 (ANSI)), ISO/IEC 14882:2003(C++03 (ISO)) or ISO/IEC 14882:2011(C++11 (ISO)) math library extensions, you can change the standard math library setting. The C++03 (ISO) or C++11 (ISO) library is an option when you select C++ for the programming language.

The C99 library leverages the performance that a compiler offers over standard ANSI C. When using the C99 library, the code generator produces calls to ISO C functions when possible. For example, the generated code calls the function `sqrtf()`, which operates on single-precision data, instead of `sqrt()`.

To change the library setting, use the **Configuration Parameters>Standard math library** parameter. The command-line equivalent is `TargetLangStandard`.

Generate and Inspect ANSI C Code

1. Open the example model `rtwdemo_clibsup`.



Copyright 1994-2020 The MathWorks, Inc.

2. Generate code.

```
### Starting build procedure for: rtwdemo_clibsup
### Successful completion of code generation for: rtwdemo_clibsup
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_clibsup	Code generated.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 12.185s

3. Examine the code in the generated file `rtwdemo_clibsup.c`. Note that the code calls the `sqrt` function.

```
if (rtb_Abs2 < 0.0F) {
    rtb_Abs2 = -(real32_T)sqrt((real32_T)fabs(rtb_Abs2));
} else {
    rtb_Abs2 = (real32_T)sqrt(rtb_Abs2);
}

/* Gain: '<Root>/Gain1' incorporates:
 * Constant: '<Root>/Constant'
 * Sqrt: '<Root>/get_hypot1'
 * Sum: '<Root>/Sum'
 *
 * About '<Root>/get_hypot1':
 * Operator: signedSqrt
 */
rtb_Abs2 = (rtb_Abs2 - 1.0F) * 343.0F;

/* Rounding: '<Root>/fix1' */
if (rtb_Abs2 < 0.0F) {
    /* Outport: '<Root>/Out1' */
    rtwdemo_clibsup_Y.Out1 = (real32_T)ceil(rtb_Abs2);
} else {
    /* Outport: '<Root>/Out1' */
    rtwdemo_clibsup_Y.Out1 = (real32_T)floor(rtb_Abs2);
}

/* End of Rounding: '<Root>/fix1' */

/* Abs: '<Root>/Abs2' incorporates:
 * Inport: '<Root>/In2'
 */
rtb_Abs2 = (real32_T)fabs(rtwdemo_clibsup_U.In2);

/* Outport: '<Root>/Out2' incorporates:
 * Constant: '<Root>/Constant1'
 * Gain: '<Root>/Gain2'
 * Math: '<Root>/get_hypot2'
 * Rounding: '<Root>/fix2'
 * Sum: '<Root>/Sum2'
 * Trigonometry: '<S1>/cos'
 * Trigonometry: '<S1>/sin'
```

```

    */
    rtwdemo_clibsup_Y.Out2 = (real32_T)floor((rt_hypotf_snf((real32_T)sin(rtb_Abs2),
        (real32_T)cos(rtb_Abs2)) - 1.0F) * 343.0F);
}

/* Model initialize function */
void rtwdemo_clibsup_initialize(void)
{
    /* Registration code */

    /* initialize non-finites */
    rt_InitInfAndNaN(sizeof(real_T));

    /* initialize error status */
    rtmSetErrorStatus(rtwdemo_clibsup_M, (NULL));

    /* external inputs */
    (void)memset(&rtwdemo_clibsup_U, 0, sizeof(ExtU_rtwdemo_clibsup_T));

    /* external outputs */
    (void)memset(&rtwdemo_clibsup_Y, 0, sizeof(ExtY_rtwdemo_clibsup_T));
}

/* Model terminate function */
void rtwdemo_clibsup_terminate(void)
{
    /* (no terminate code required) */
}

```

Generate and Inspect ISO C Code

1. Change the setting of **Standard math library** to C99 (ISO). Alternatively, at the command line, set TargetLangStandard to C99 (ISO).

2. Regenerate the code.

```

### Starting build procedure for: rtwdemo_clibsup
### Successful completion of code generation for: rtwdemo_clibsup

```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_clibsup	Code generated.	Code generation information file does not exist.

```

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.423s

```

3. Reexamine the code in the generated file `rtwdemo_clibsup.c`. Now the generated code calls the function `sqrtf` instead of `sqrt`.

```

if (rtb_Abs2 < 0.0F) {
    rtb_Abs2 = -sqrtf(fabsf(rtb_Abs2));
} else {
    rtb_Abs2 = sqrtf(rtb_Abs2);
}

```

```

}

/* Output: '<Root>/Out1' incorporates:
 * Constant: '<Root>/Constant'
 * Gain: '<Root>/Gain1'
 * Rounding: '<Root>/fix1'
 * Sqrt: '<Root>/get_hypot1'
 * Sum: '<Root>/Sum'
 *
 * About '<Root>/get_hypot1':
 * Operator: signedSqrt
 */
rtwdemo_clibsup_Y.Out1 = truncf((rtb_Abs2 - 1.0F) * 343.0F);

/* Abs: '<Root>/Abs2' incorporates:
 * Inport: '<Root>/In2'
 */
rtb_Abs2 = fabsf(rtwdemo_clibsup_U.In2);

/* Output: '<Root>/Out2' incorporates:
 * Constant: '<Root>/Constant1'
 * Gain: '<Root>/Gain2'
 * Math: '<Root>/get_hypot2'
 * Rounding: '<Root>/fix2'
 * Sum: '<Root>/Sum2'
 * Trigonometry: '<S1>/cos'
 * Trigonometry: '<S1>/sin'
 */
rtwdemo_clibsup_Y.Out2 = floorf((rt_hypotf_snf(sinf(rtb_Abs2), cosf(rtb_Abs2))
- 1.0F) * 343.0F);
}

/* Model initialize function */
void rtwdemo_clibsup_initialize(void)
{
    /* Registration code */

    /* initialize non-finites */
    rt_InitInfAndNaN(sizeof(real_T));

    /* initialize error status */
    rtmSetErrorStatus(rtwdemo_clibsup_M, (NULL));

    /* external inputs */
    (void)memset(&rtwdemo_clibsup_U, 0, sizeof(ExtU_rtwdemo_clibsup_T));

    /* external outputs */
    (void)memset(&rtwdemo_clibsup_Y, 0, sizeof(ExtY_rtwdemo_clibsup_T));
}

/* Model terminate function */
void rtwdemo_clibsup_terminate(void)
{
    /* (no terminate code required) */
}

```

Related Information

- Language standard
- “Configure a System Target File” on page 21-2
- “Configure STF-Related Code Generation Parameters” on page 21-5
- “Compare System Target File Support Across Products” on page 21-17
- “Replace Code Generated from Simulink Models” on page 26-8

Compare System Target File Support Across Products

When you select a system target file (such as `grt.tlc`), the selection defines the run-time environment and the code generation features. Identify the system target file features that match your code generation workflow goals.

The code generator uses the system target file to produce code intended for execution on certain target hardware or on a certain operating system. The system target file invokes other run-time environment-specific files. For more information on configuring model code generation parameters for target hardware, see “Configure Run-Time Environment Options” on page 11-2.

Different types of system target files support a selection of generated code features. In the system target file, the value of the `CodeFormat` TLC variable and corresponding `rtwgensettings.DerivedFrom` field value identify the system target file type and generated code features. These selections apply your code generation control decisions at several points in the code generation process. Your selections include whether and how the model build generates:

- Certain data structures (for example, `SimStruct` or `rtModel`)
- Static or dynamic memory allocation code
- Calling interface for generated model functions

For custom system target file development, the `CodeFormat` value differs among code generation targets:

- If the system target file does not include a value for the `CodeFormat` TLC variable, the default value is `RealTime` for generic real-time target (GRT). The corresponding `rtwgensettings.DerivedFrom` field value is `grt.tlc` (default value).
- If you are developing a custom system target file and you have Embedded Coder software, consider setting the `CodeFormat` TLC variable value to `Embedded-C` for embedded real-time target (ERT). The corresponding `rtwgensettings.DerivedFrom` field value is `ert.tlc`. The ERT system target file supports more generated code features than the GRT system target file.

This example shows how the value for the `CodeFormat` TLC variable and corresponding `rtwgensettings.DerivedFrom` field value are set in `ert.tlc`.

```
%assign CodeFormat = "Embedded-C"

/%
  BEGIN_RTW_OPTIONS
  rtwgensettings.DerivedFrom = 'ert.tlc';
  END_RTW_OPTIONS
%/
```

Note Use the value for the `CodeFormat` TLC variable with its corresponding `rtwgensettings.DerivedFrom` field value to generate code for the model. If you do not select a value explicitly, the default values apply. For the `rtwgensettings.DerivedFrom` field, use `'ert.tlc'` or `'grt.tlc'`. For more information, see “System Target File Structure” on page 46-19.

For a description of the optimized call interface generated by default for the GRT and ERT system target files, see “Configure Generated C Function Interface for Model Entry-Point Functions” on page 16-79.

Code generation with the GRT and ERT system target files uses the real-time model data structure (`rtModel`). This structure encapsulates model-specific information in a much more compact form than the simulation structure, `SimStruct`. Many efficient features of generated code depend on generation of `rtModel` rather than `SimStruct`, including:

- Integer absolute and elapsed timing services
- Independent timers for asynchronous tasks
- Generation of improved C API code for signal, state, and parameter monitoring
- Pruning the data structure to minimize its size (ERT-derived system target files only)

For a description of the `rtModel` data structure, see “Real-Time Model Data Structure” on page 15-32.

Compare Product System Target Files

You can select from a range of system target files by using the System Target File Browser. This selection lets you experiment with configuration options and save your model with different configurations.

You cannot build or generate code for non-GRT system target files, unless you have the required software on your system. For example, you require Embedded Coder for ERT system target files, Simulink Desktop Real-Time for SLDRT system target files, and so on.

Selecting a system target file for your model selects either the toolchain approach or the template makefile approach for build process control. For more information about these approaches, see “Approaches for Building Code Generated from Simulink Models” on page 31-11.

System Target Files Available from System Target File Browser

Supported System Target File	File Names	Reference
Embedded Coder (for PC or UNIX platforms)	ert.tlc ert_shrllib.tlc	"Configure a System Target File" (Embedded Coder)
Create Visual C++ [®] Solution File for Embedded Coder	ert.tlc (Requires CMake toolchain. See note.)	"Configure a System Target File" (Embedded Coder)
Embedded Coder for AUTOSAR	autosar.tlc	"Develop a Model That Complies with the AUTOSAR Standard" (Embedded Coder)
Generic Real-Time (for PC or UNIX platforms)	grt.tlc	"Compare Generated Code Features by STF" on page 21-23
Create Visual C++ Solution File	grt.tlc (Requires CMake toolchain. See note.)	"Compare Generated Code Features by STF" on page 21-23
Rapid Simulation (default for PC or UNIX platforms)	rsim.tlc	"Accelerate, Refine, and Test Hybrid Dynamic System on Host Computer by Using RSim System Target File" on page 29-2
Rapid Simulation for LCC compiler	rsim.tlc	"Accelerate, Refine, and Test Hybrid Dynamic System on Host Computer by Using RSim System Target File" on page 29-2
Rapid Simulation for UNIX platforms	rsim.tlc	"Accelerate, Refine, and Test Hybrid Dynamic System on Host Computer by Using RSim System Target File" on page 29-2
Rapid Simulation for Visual C++ compiler	rsim.tlc	"Accelerate, Refine, and Test Hybrid Dynamic System on Host Computer by Using RSim System Target File" on page 29-2
S-Function for PC or UNIX platforms	rtwsfcn.tlc (To be removed)	"Use S-Function Target for Model or Subsystem" on page 29-40
S-Function for LCC	rtwsfcn.tlc (To be removed)	"Use S-Function Target for Model or Subsystem" on page 29-40
S-Function for UNIX platforms	rtwsfcn.tlc (To be removed)	"Use S-Function Target for Model or Subsystem" on page 29-40
S-Function for Visual C++ compiler	rtwsfcn.tlc (To be removed)	"Use S-Function Target for Model or Subsystem" on page 29-40

Supported System Target File	File Names	Reference
ASAM-ASAP2 Data Definition	asap2.tlc (Will be removed in a future release. See “Generate ASAP2 and CDF Calibration Files” on page 36-2 for information about how to generate A2L Files)	“Generate ASAP2 and CDF Calibration Files” on page 36-2
Simulink Desktop Real-Time	sldrt.tlc sldrtert.tlc	“Set Run in Kernel Mode Code Generation Parameters” (Simulink Desktop Real-Time)
Simulink Real-Time	slrealtime.tlc	“Simulink Real-Time Options Pane” (Simulink Real-Time)

Note To create and build a Visual C++ Solution (.sln) file with Debug configuration, in the Toolchain field, specify a CMake toolchain, for example, Microsoft Visual Studio Project 2019 | CMake (64-bit Windows) or Microsoft Visual Studio Project 2017 | CMake (64-bit Windows). For more information about CMake toolchains, see “Configure CMake Build Process” on page 31-13.

Compare Code Styles and STF Support

The code generator produces two styles of code. One code style is suitable for rapid prototyping (and simulation by using code generation). The other style is suitable for embedded applications. This table maps system target files to corresponding code styles.

Code Styles Listed by System Target File

System Target File	Code Style	Purpose
Embedded Coder embedded real-time (ERT)	Embedded	A starting point for embedded application development of C/C++ generated code.
Simulink Coder generic real-time (GRT)	Rapid prototyping	A starting point for creating a rapid prototyping target hardware that does not use real-time operating system tasking primitives and for verifying the generated C/C++ code on your desktop computer.
Rapid simulation (RSim)	Rapid prototyping	Provides non-real-time simulation on your desktop computer and a high-speed or batch simulation tool.
S-function	Rapid prototyping	Creates a C MEX S-function for simulation within another Simulink model.
Simulink Desktop Real-Time	Rapid prototyping	Runs a model in real time at interrupt level while your desktop computer runs Microsoft Windows in the background.
Simulink Real-Time	Rapid prototyping	Runs a model in real time on a desktop computer running the Simulink Real-Time kernel.

Third-party vendors supply additional system target files to support code generation for their products. For more information about third-party products, see the vendor website or the MathWorks Connections program web page: <https://www.mathworks.com/products/connections>.

Compare Generated Code Features by Product

The code generation process for real-time system target files (such as GRT) provides many embedded code optimizations. Selecting an ERT-based system target file offers more extensive features than GRT. The system target file selection determines the available features for the code generation product. As you select the code generation target that matches your development process, use this table to compare code generation features available with Simulink Coder and features available with Embedded Coder.

Compare Code Generation Features for Simulink Coder Versus Embedded Coder

Feature	Simulink Coder	Embedded Coder
rtModel data structure	<ul style="list-style-type: none"> Full rtModel structure generated GRT variable declaration: <code>rtModel_model model_M_;</code> 	<ul style="list-style-type: none"> rtModel is optimized for the model Optional suppression of error status field and data logging fields ERT variable declaration: <code>RT_MODEL_model model_M_;</code>
Custom storage classes (CSCs)	Code generation ignores CSCs. Objects are assigned a CSC default to Auto storage class.	Code generation with CSCs is supported
HTML code generation report	Basic HTML code generation report	Enhanced report with additional detail and hyperlinks to the model
Symbol formatting	Symbols (for signals, parameters, and so on) are generated in accordance with hard-coded default	Detailed control over generated symbols
User-defined maximum identifier length for generated symbols	Supported	Supported
Generation of terminate function	Generated	Option to suppress the terminate function
Combined output/update function	Separate output/update functions are generated	Option to generate combined output/update function
Optimized data initialization	Not available	Options to suppress generation of unnecessary initialization code for zero-valued memory, I/O ports, and so on
Comments generation	Basic options to include or suppress comment generation	Options to include Simulink block descriptions, Stateflow object descriptions, and Simulink data object descriptions in comments
Module Packaging Features (MPF)	Not supported	Extensive code customization features See “Manage Replacement of Simulink Data Types in Generated Code” (Embedded Coder) and “MPT Data Object Properties” (Embedded Coder).
System target file-optimized data types header file	Requires full <code>tmwtypes.h</code> header file	Generates optimized <code>rtwtypes.h</code> header file, including definitions required by the system target file
User-defined types	User-defined types default to base types in code generation	User-defined data type aliases are supported in code generation
Rate grouping	Not supported	Supported
Auto-generation of main program module	Not supported. Static main program module is provided.	Automated and customizable generation of main program module is supported (static main program also available)
Reusable (multi-instance) code generation	Option to generate reusable code with dynamic memory allocation	Option to generate reusable code with static or dynamic memory allocation

Feature	Simulink Coder	Embedded Coder
Software constraint options	Support for floating-point, complex, and nonfinite numbers is enabled	Options to enable or disable support for floating-point, complex, and nonfinite numbers
Application life span	Defaults to <code>inf</code>	User-specified. Determines most efficient word size for integer timers
ANSI-C/C++ code generation	Supported	Supported
ISO-C/C++ code generation	Supported	Supported
GNU [®] -C/C++ code generation	Supported	Supported
Generate scalar inlined parameters as <code>#DEFINE</code> statements	Not supported	Supported
MAT-file variable name modifier	Supported	Supported
Data exchange: C API, ASAP2, external mode	Supported	Supported

Compare Generated Code Features by STF

The code generator supports a selection of generated code features for different types of system target files. In each system target file, the value of the `CodeFormat` TLC variable identifies the set of features.

This table summarizes how different system target files support applications.

Application	System Target File (STF)
Fixed- or variable-step acceleration	RSIM, S-Function, Model Reference
Fixed-step real-time deployment	GRT, ERT, Simulink Real-Time, Simulink Desktop Real-Time, ...

This table summarizes the various options available for each **System target file** selection, with the exceptions noted.

Features Supported in Code Generated for System Target Files (STF)

Feature	System Target Files (STF)							
	grt.tlc (See note 1.)	ert.tlc (See note 1.)	ert_shrlib.tlc (See note 1.)	rtwsfcn.tlc (See note 1.)	rsim.tlc (See note 1.)	sldrt.tlc (See note 1.)	slrealtime.tlc (See note 1.)	Other (See note 1.)
Static memory allocation	X	X				X	X	X
Dynamic memory allocation	X (See notes 4, 5.)	X (See notes 4, 5.)		X	X		X	
Continuous time	X	X		X	X	X	X	
C/C++ MEX S-functions (noninlined)	X	X		X	X	X	X	
S-function (inlined)	X	X		X	X	X	X	X
Minimize RAM/ROM usage		X					X ²	X
Supports external mode	X	X			X	X	X	
Rapid prototyping	X					X	X	X
Production code		X					X ²	X (See note 3.)
Batch parameter tuning and Monte Carlo methods			X		X			
System-level Simulator			X					
Executes in hard real time	X (See note 3.)	X (See note 3.)				X	X	X ⁵

Feature	System Target Files (STF)							
	grt.tlc (See note 1.)	ert.tlc (See note 1.)	ert_shrlib.tlc (See note 1.)	rtwsfcn.tlc (See note 1.)	rsim.tlc (See note 1.)	sldrt.tlc (See note 1.)	slrealtime.tlc (See note 1.)	Other (See note 1.)
Non-real-time executable included	X	X			X			
Multiple instances of model	X (See notes 4, 5.)	X (See notes 4, 5.)		X ⁴			X (See notes 4, 5.)	X (See notes 4, 5.)
Supports variable-step solvers				X	X			
Supports SIL/PIL	X	X						X

Notes

- 1 System Target Files:
 - grt.tlc — Generic real-time target
 - ert.tlc — Embedded real-time target
 - ert_shrlib.tlc — Embedded real-time target shared library
 - rtwsfcn.tlc (To be removed) — S-function target
 - rsim.tlc — Rapid simulation target
 - sldrt.tlc — Simulink Desktop Real-Time target
 - slrealtime.tlc — Simulink Real-Time target
 - Other — The Simulink Coder embedded real-time capabilities support other system target files
- 2 Does not apply to GRT-based system target files. Applies to only ERT-based system target files.
- 3 The default GRT and ERT `rt_main` files emulate execution of hard real time. When explicitly connected to a real-time clock, they execute in hard real time.
- 4 You can generate code for multiple instances of a Stateflow chart or subsystem containing a chart, except when the chart contains exported graphical functions.
- 5 In the Configuration Parameters dialog box, for the **Code interface packaging** field, select Reusable function.

See Also

More About

- “Configure a System Target File” on page 21-2
- “Configure STF-Related Code Generation Parameters” on page 21-5
- “Choose a Code Replacement Library” on page 26-6
- “Configure Standard Math Library for Target System” on page 21-12

Internationalization Support in Simulink Coder

Internationalization and Code Generation

Internationalization support in software development tooling is vital for enabling efficient globalization. If there is any possibility of future collaboration with others across locales, consider internationalization from project inception. Internationalization can prevent rework or having to develop a new model design. The relevant requirement concerns locale settings.

Locale Settings

On a computer, a locale setting defines the language (character set encoding) for the user interface and the display formats for information such as time, date, and currency. The encoding dictates the number of characters that a locale can render. For example, the US-ASCII coded character set (codeset) defines 128 characters. A Unicode® codeset, such as UTF-8, defines more than 1,100,000 characters.

For code generation, the locale setting determines the character set encoding of generated file content. To avoid garbled text or incorrectly displayed characters, the locale setting for your MATLAB session must be compatible with the setting for your compiler and operating system. For information on finding and changing the operating system setting, see “Internationalization” or see the operating system documentation.

Prepare to Generate Code for Mixed Languages and Locales

To prepare to generate code for a model, identify:

- The operating system locale.
- The locale of the MATLAB session.
- Code generation requirements for:
 - Target Language Compiler files
 - Code generation template files that include comments (requires Embedded Coder)

Character Set Limitations

Target language compiler files support user default encoding only. To produce international, custom generated code that is portable, use the 7-bit ASCII character set.

XML Escape Sequence Replacements

The code generator replaces characters that are not represented in the character set encoding of a model with XML escape sequences. Escape sequence replacements occur for block, signal, and Stateflow object names that appear in:

- Generated code comments
- Code generation reports
- Block paths logged to MAT-files
- Block paths logged to C API files *model_capi.c* (or *.cpp*) and *model_capi.h*

Generate and Review Code with Mixed Languages and Mixed Locales

This example shows how to use the code generator to produce and review code for use in mixed languages and mixed locales.

Before using this example, see “Internationalization and Code Generation” on page 22-2 or “Internationalization and Code Generation” (Embedded Coder).

The configuration for model `MixedLanguagesAndLocales` uses system target file `ert.tlc`. To see internationalization and localization support with Simulink® Coder™, configure the model to use the `grt.tlc` system target file. The example indicates the support that is specific to Embedded Coder (for example, code generation templates).

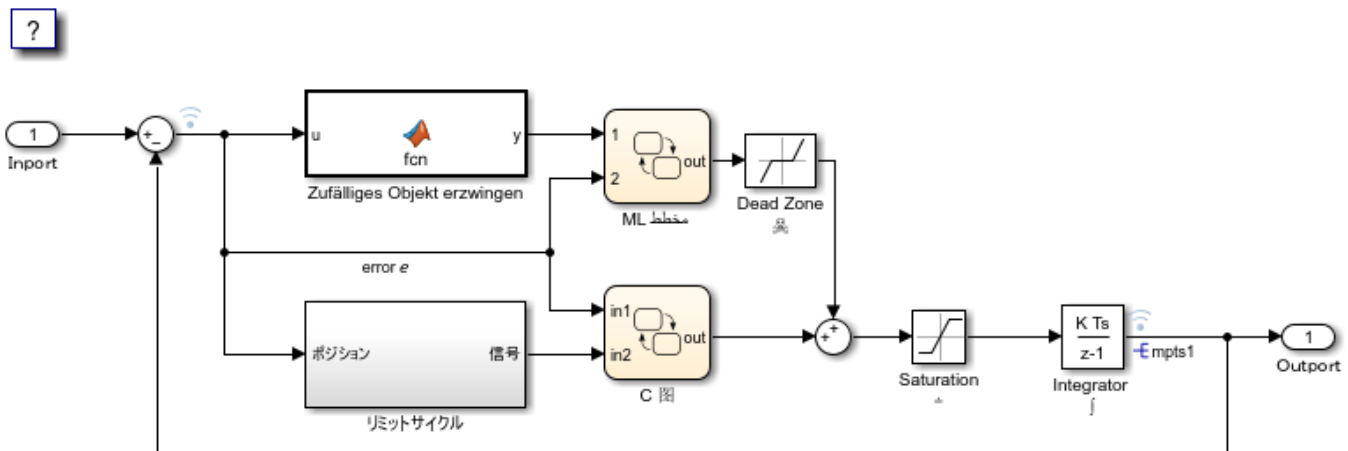
The model configuration specifies files and settings that control how the code generator handles localization for:

- C and C++ API interfaces
- Code generation template (CGT) files (requires Embedded Coder)
- Target Language Compiler (TLC) files that apply code customizations (requires Embedded Coder)

Open Example Model

Open example model `MixedLanguagesAndLocales`. Labels in the model appear in multiple languages (Arabic, Chinese, English, German, and Japanese) and various Unicode symbols.

```
model = 'MixedLanguagesAndLocales';
open_system(model);
```



Copyright 2015-2022 The MathWorks, Inc.

Open Embedded Coder App

In the Apps tab, select **Embedded Coder**.

Verify Locale Settings

Verify that the locale setting for your MATLAB® software is compatible with your compiler. See the documentation for your operating system or the following MATLAB documentation:

- “Set Locale on Microsoft Windows Platforms”
- “Set Locale on Linux Platforms”
- “Set Locale on macOS Platforms”

Configure Code Generation Template File

To use a code generation template file with unicode characters when generating code, complete these steps (requires Embedded Coder). Otherwise, go to the next section.

1. Open the Configuration Parameters dialog box.
2. Navigate to the **Code Generation > Template** pane. The model is configured to use the code generation template file `MixedLanguagesAndLocales.cgt`. That file adds comments to the top of generated code files. For the code generator to apply escape sequence replacements for the `.cgt` file, enable replacements by specifying:

```
<encodingIn = "encoding-name">
```

3. Open the file `MixedLanguagesAndLocales.cgt`.

```
edit MixedLanguagesAndLocales.cgt
```

4. Find the line of code that enables escape sequence replacements for the character set encoding UTF-8.

```
<encodingIn = "UTF-8">
```

5. Close the file `MixedLanguagesAndLocales.cgt`.

Configure Code Customization File

To use file customization templates with unicode characters when generating code, complete these steps (requires Embedded Coder). Otherwise, go to the next section.

You can specify customizations to generated code files by using TLC code. TLC files support user default encoding only. To produce international custom generated code that is portable, use the 7-bit ASCII character set.

1. Open the Configuration Parameters dialog box.
2. Navigate to the **Code Generation > Template** pane. The model is configured to use the code customization file `example_file_process.tlc`. That file customizes the generated code just before the code generator writes the code files. For example, the file adds a C source file, corresponding include file, and `#define` and `#include` statements.

3. Open the file `example_file_process.tlc`.

```
edit example_file_process.tlc
```

4. Before generating code, check that the following line of code is uncommented:

```
assign ERTCustomFileTest = TLC_TRUE
```

5. Close the file `example_file_process.tlc`.

Generate C Code and Report

Generate C code and a code generation report.

```
evalc('slbuild(model)');
```

Review Generated Code

Use the code generation report to review the generated code. For characters that are not in the current MATLAB character set encoding, the code generator uses escape sequence replacements to render characters correctly in the code generation report.

1. If the code generation report for model `MixedLanguagesAndLocales` is not open, in the Command Window, type:

```
coder.report.open(model);
```

2. Review the generated code in `MixedLanguagesAndLocales.c` and `MixedLanguagesAndLocales.h`. Names of model elements appear in code comments as replacement names in the local language.

3. Open the Traceability Report. The report maintains traceability information, even when the name contains characters that are not represented in the current encoding. Names of model elements appear in the report as replacement names in the local language.

4. Scroll down to and click the code location link for the first Chart (State 'Selection' <S2>:23). The report view changes to show the corresponding code in `MixedLanguagesAndLocales.c`.

5. In the code comment, click the <S2>:23 link. The model window shows the chart in a new tab.

6. In the model window, right-click that chart. From the context menu, select **C/C++ Code > Navigate to C/C++ Code**. The report view changes to show the named constant section of code for that chart.

7. Close the code generation report, Model Advisor, and model. In the Command Window, type:

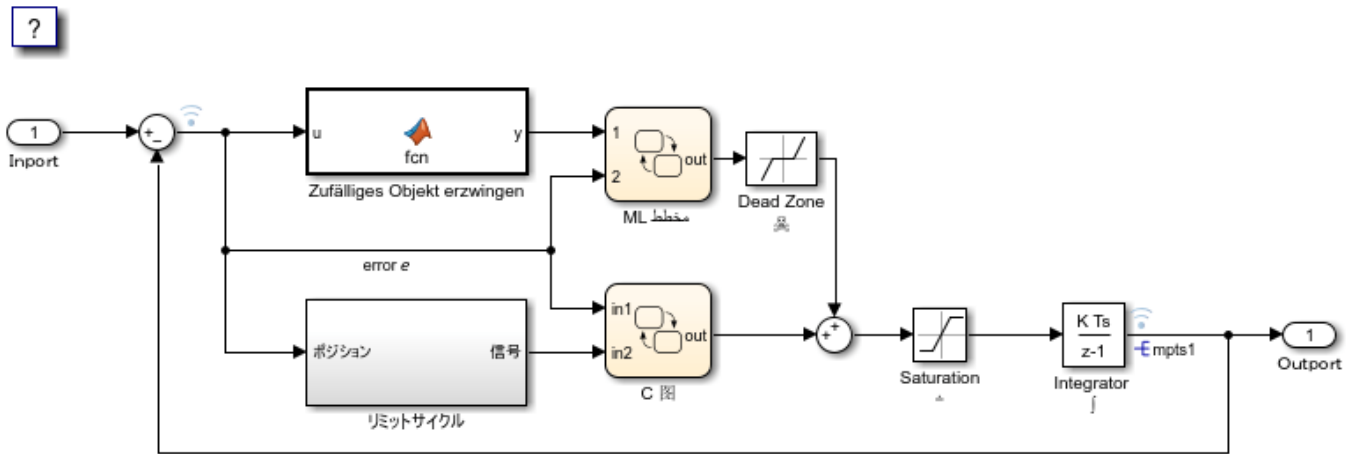
```
coder.report.close();
bdclose(model);
```

Generate C++ Code

Generate C++ code and a code generation report.

1. Open the model.

```
open_system(model);
```



Copyright 2015-2022 The MathWorks, Inc.

2. Delete the `slprj\ert_sharedutils` folder that was produced when you generated C code.

```
rmmdir('slprj/ert/_sharedutils','s')
```

3. Set model configuration parameter **Language** to C++. By default, this parameter setting changes the setting of model configuration parameter **Code interface packaging** to C++ class. Alternatively, in the Command Window, type:

```
set_param(model, 'TargetLang', 'C++');
```

4. Generate C++ code and a code generation report.

```
evalc('slbuild(model)');
```

5. To see internationalization and localization support, review the generated code. See **Review the Generated Code**.

6. Close the code generation report and model. In the Command Window, type:

See Also

More About

- “Locale Setting Concepts for Internationalization”

Basic Code Generation for Simulink Coder

- “Generate Code by Using the Quick Start Tool” on page 23-2
- “Generate Code Using Simulink Coder” on page 23-4
- “Aircraft Position Radar Model” on page 23-10

Generate Code by Using the Quick Start Tool

The Quick Start tool helps you prepare a model for generating code. To start the tool, open the Simulink Coder app. On the **C Code** tab, click **Quick Start**.

After you start the tool, you must answer these questions about the code that you want to generate:

- What is the type of code output for your generated code?
- Does your application require reentrant, multi-instance code?
- What is your primary code generation objective?

The tool presents parameter changes required to generated code. If you choose to generate code, the tool applies the parameter changes to your configuration set and generates the code. If your system contains referenced models, the Quick Start tool also applies parameter changes to the referenced models. After code generation, you can view the code generation report and find information on building, customizing, optimizing, and packaging the code. To further customize your generated code, click **Finish**, and then use the Simulink Coder app. The app provides the tools to control the names and representation of the model data in the generated code. For more information, see **Simulink Coder**.

Configuration Parameter Changes for Models That Have a Configuration Reference

To apply configuration parameter changes to a model that has an active configuration reference, the Quick Start tool:

- Creates a `Simulink.ConfigSet` object `QuickStart_timestamp` in the workspace or data dictionary that contains the original configuration set. The new object is a copy of the original configuration set that has the parameter changes applied.
- Creates a `Simulink.ConfigSetRef` object that points to the new configuration set object.
- Attaches the new configuration reference to the model and makes it the active configuration.
- If your system contains referenced models that share your original configuration set, the tool attaches the new configuration reference to the referenced models and makes the new configuration reference the active configuration.

To restore the original configuration set, activate the original `Simulink.ConfigSetRef` object.

Note If the Quick Start tool creates the configuration set object in the MATLAB workspace, you must save the object to preserve the configuration set after the MATLAB session ends. For more information, see “Save a Configuration Set”.

Next Steps

After you have generated code by using Quick Start tool, possible next steps are:

- “Preserve Variables in Generated Code” on page 15-2
- “Build Process Workflow for Real-Time Systems” on page 31-39
- “Manage Configuration Sets for a Model”

- “Relocate or Share Generated Code” on page 31-56

To control the names and representation of the model data and functions in the generated code, use the Model Data Editor in the Simulink Coder app. For more information, see **Simulink Coder**.

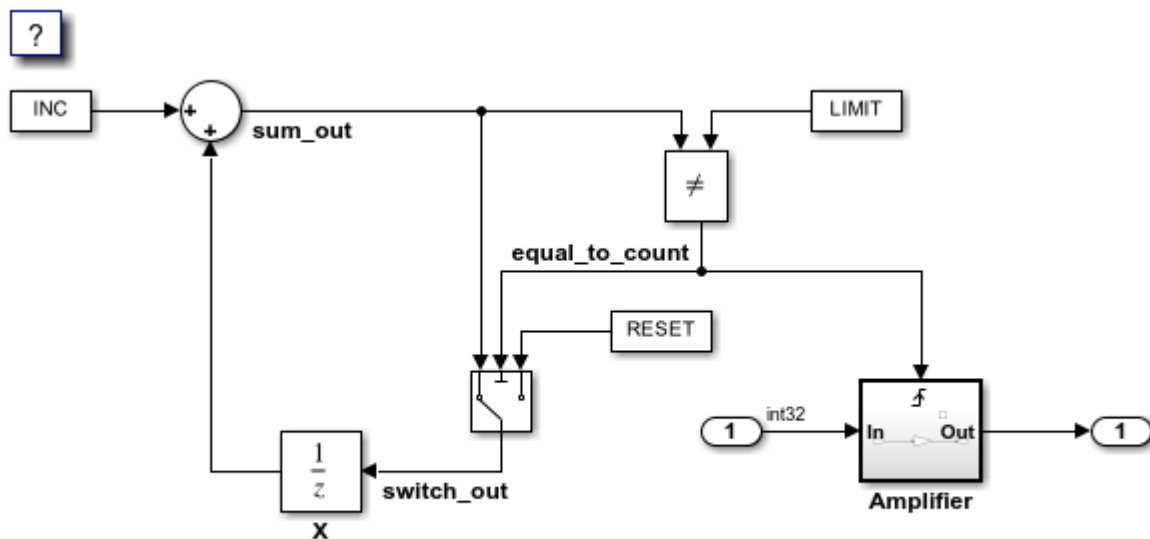
Generate Code Using Simulink Coder

This example shows how to select a system target file for a Simulink® model, generate C code for real-time simulation, and view generated files.

The model represents an 8-bit counter that feeds a triggered subsystem that is parameterized by constant blocks INC, LIMIT, and RESET. Input and Output represent I/O for the model. The Amplifier subsystem amplifies the input signal by gain factor K, which updates when signal equal_to_count is true.

1. Open model `rtwdemo_rtwintr0` and save a copy to a writable location.

```
model='rtwdemo_rtwintr0';
open_system(model)
```

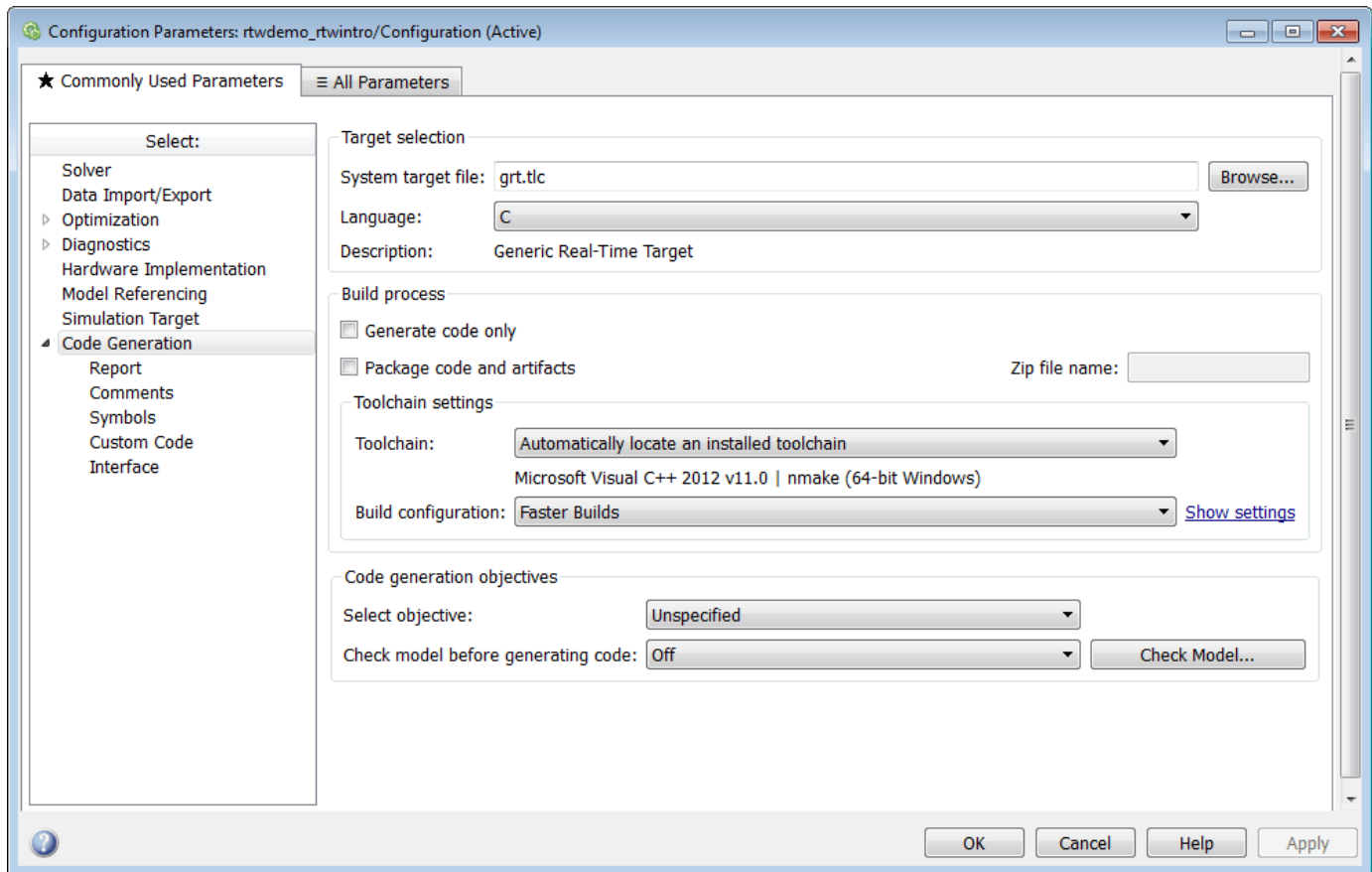


Copyright 1994-2020 The MathWorks, Inc.

2. Open the **Simulink Coder** app.
3. Open the Model Configuration Parameters dialog box. In the **C Code** tab, click **Settings**. Alternatively, from the command line, enter:

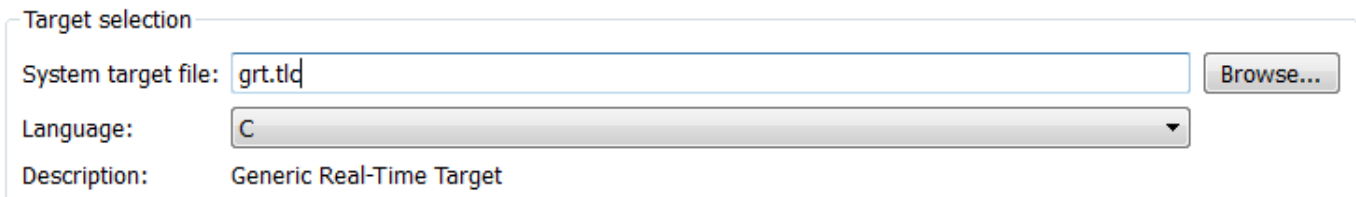
```
cs = getActiveConfigSet(model);
openDialog(cs);
```

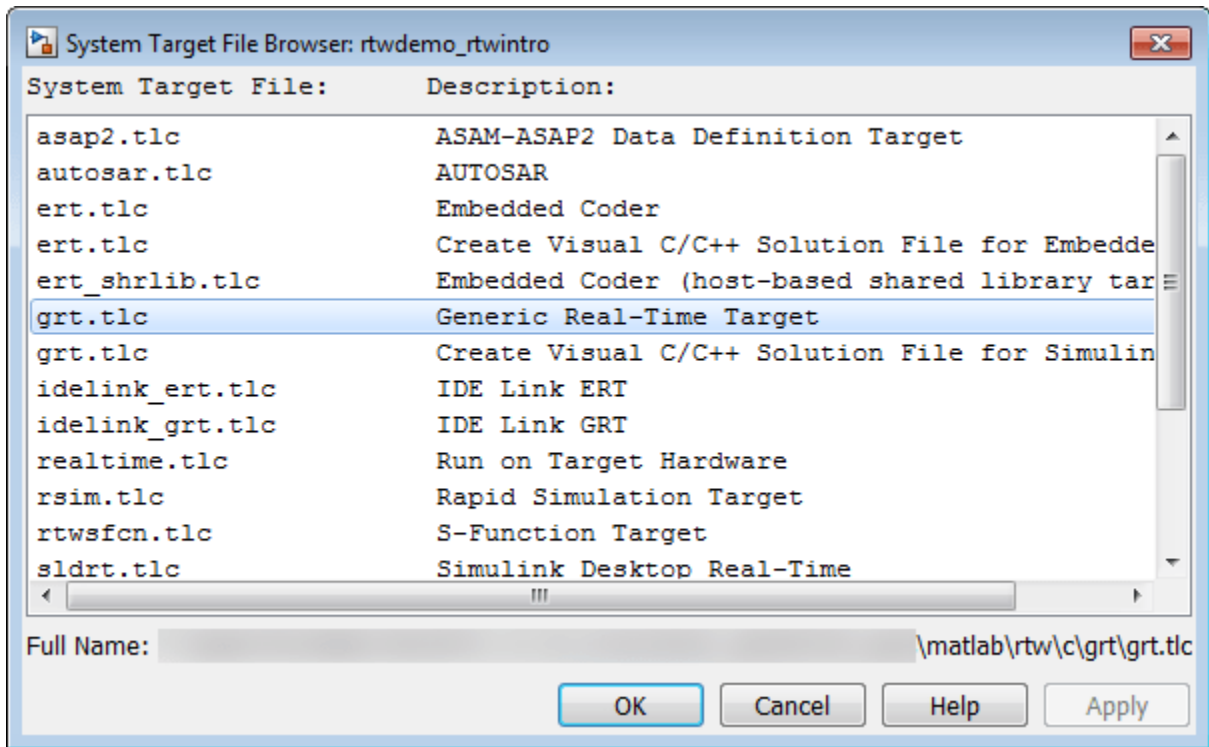
4. Select the **Code Generation** tab.



5. In the **Target Selection** pane, click **Browse** to select a system target file.

You can generate code for a specific target computer environment or purpose. Some options are provided by built-in system target files, which control the code generation process for a target computer environment.





6. Select the **Generic Real-Time (GRT)** system target file and click **Apply**.

Optionally, in the **Code Generation Advisor** pane set the parameter **Select objective** to **Execution efficiency** or **Debugging**. Then, to identify and systematically change parameters to meet your objectives, click **Check model...**

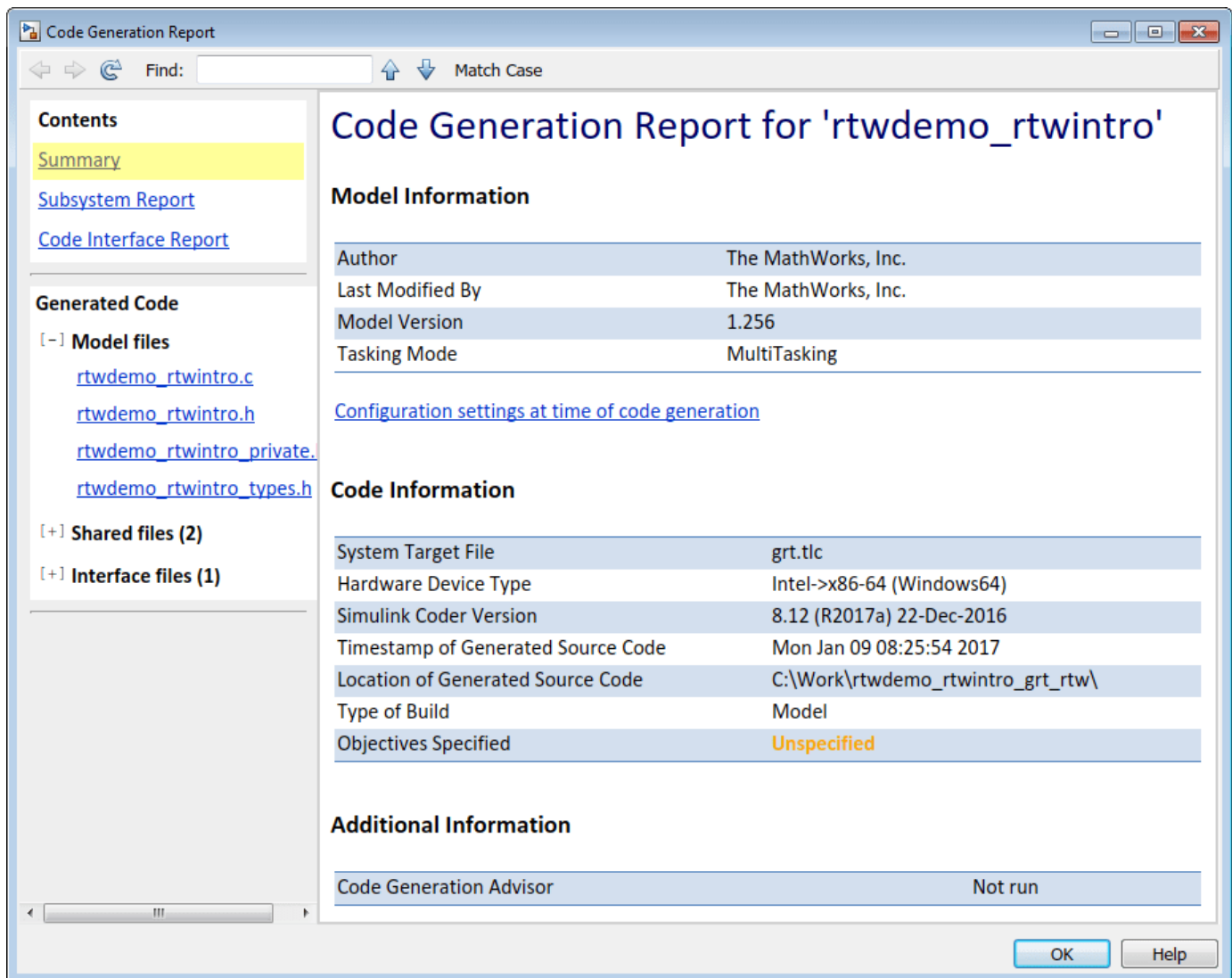
7. Generate code for the model by using one of these options:

- On the **C Code** tab, click **Build**.
- Press **Ctrl+B**.



8. View the code generation report.

The report includes links to model files such as `rtwdemo_rtwintr0.c` and associated utility and header files.



This figure contains a portion of `rtwdemo_rtwintr0.c`.

```

Step function for model: rtwdemo_rtwintr
File: rtwdemo_rtwintr.c

1  /* Model step function */
2  void rtwdemo_rtwintr_step(void)
3  {
4      uint8_T rtb_sum_out;
5      boolean_T rtb_equal_to_count;
6
7      /* Sum: '<Root>/Sum' incorporates:
8       * Constant: '<Root>/INC'
9       * UnitDelay: '<Root>/X'
10     */
11     rtb_sum_out = (uint8_T)(1U + (uint32_T)rtwdemo_rtwintr_DWork.X);
12
13     /* RelationalOperator: '<Root>/RelOpt' incorporates:
14      * Constant: '<Root>/LIMIT'
15     */
16     rtb_equal_to_count = (rtb_sum_out != 16);
17
18     /* Outputs for Triggered SubSystem: '<Root>/Amplifier' incorporates:
19      * TriggerPort: '<S1>/Trigger'
20     */
21     if (rtb_equal_to_count && (rtwdemo_rtwintr_PrevZCSigState.Amplifier_Trig_ZCE
22         != POS_ZCSIG)) {
23         /* Outport: '<Root>/Output' incorporates:
24          * Gain: '<S1>/Gain'
25          * Inport: '<Root>/Input'
26         */
27         rtwdemo_rtwintr_Y.Output = rtwdemo_rtwintr_U.Input << 1;
28     }
29
30     rtwdemo_rtwintr_PrevZCSigState.Amplifier_Trig_ZCE = (uint8_T)
31         (rtb_equal_to_count ? (int32_T)POS_ZCSIG : (int32_T)ZERO_ZCSIG);
32
33     /* End of Outputs for SubSystem: '<Root>/Amplifier' */
34
35     /* Switch: '<Root>/Switch' */
36     if (rtb_equal_to_count) {
37         /* Update for UnitDelay: '<Root>/X' */
38         rtwdemo_rtwintr_DWork.X = rtb_sum_out;
39     } else {
40         /* Update for UnitDelay: '<Root>/X' incorporates:
41          * Constant: '<Root>/RESET'
42         */
43         rtwdemo_rtwintr_DWork.X = 0U;
44     }
45
46     /* End of Switch: '<Root>/Switch' */
47 }

```

9. Close the model.

```
bdclose(model)
rtwdemoclean;
```

Related Topics

- “Configure Model, Generate Code, and Simulate” on page 24-2
- “Configure a System Target File” on page 21-2
- “Generate Code Using Embedded Coder” (Embedded Coder)
- “Generate Code and Simulate Models in a Project” on page 24-97
- “Generate Code and Simulate Models Programmatically with Project” on page 24-100

Aircraft Position Radar Model

This model shows the code generated for a Simulink® model containing a MATLAB® script.

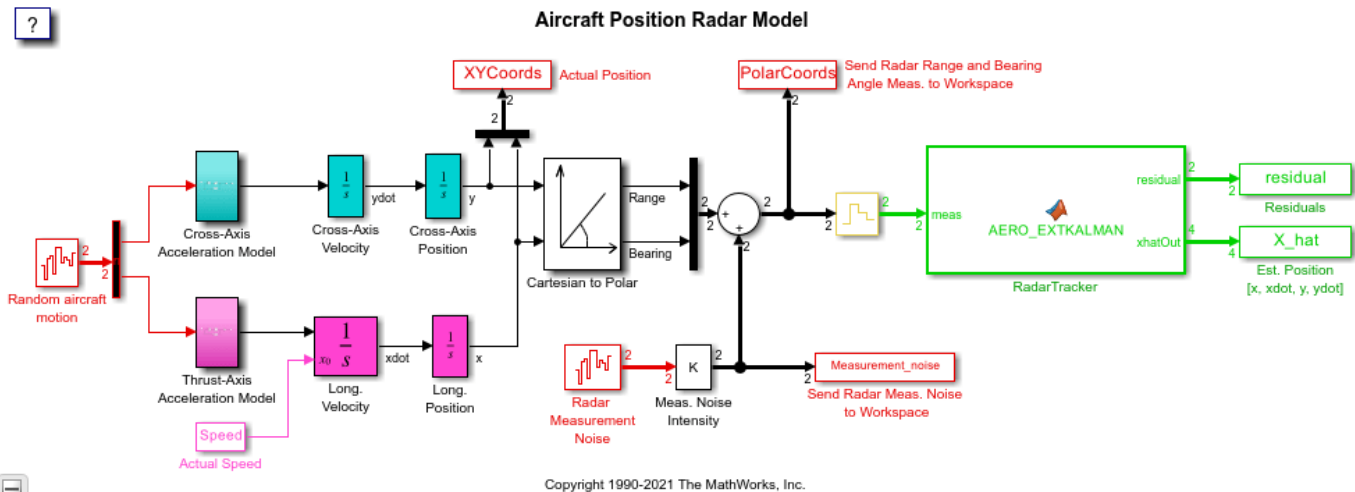
The model contains an Extended Kalman Filter that estimates aircraft position from radar measurements. The MATLAB script `rtwdemo_eml_aero_radar.m` contains data for running the model. The estimated and actual positions are saved to the workspace and are plotted at the end of the simulation by the program `rtwdemo_aero_radplot` (called from the simulation automatically).

Review and Simulate the Model

In this section you should review the model and perform a simulation.

Open the Simulink model.

```
model='rtwdemo_eml_aero_radar';
open_system(model)
rtwdemo_eml_aero_radar([],[],[],'compile');
rtwdemo_eml_aero_radar([],[],[],'term');
```



Open the MATLAB Function block `RadarTracker` in the MATLAB Editor.

```
open_system([model, '/RadarTracker'])
```



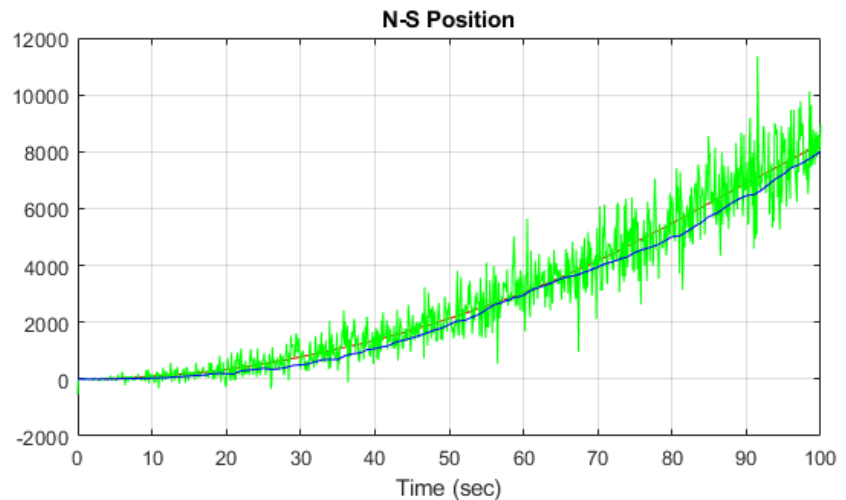
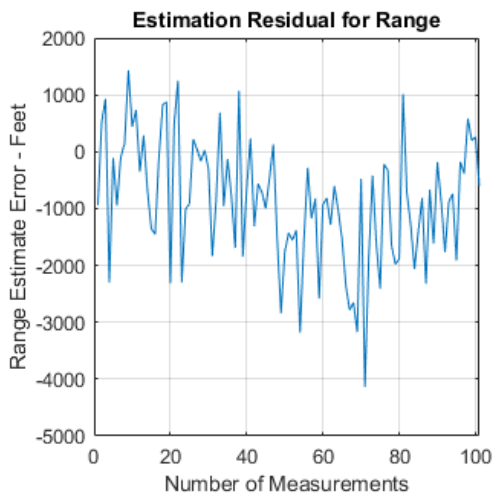
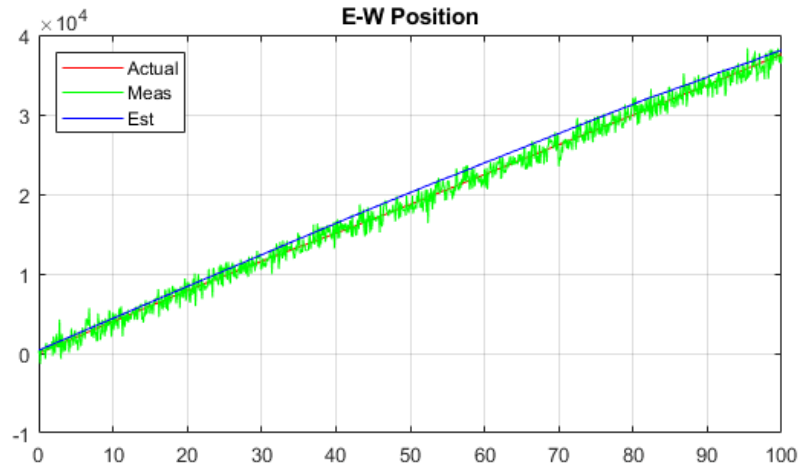
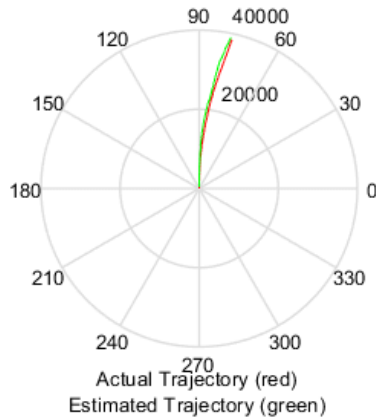
```

Block: rtwdemo_eml_aero_radar/RadarTracker
File Edit Text Go Cell Tools Debug Desktop Window Help
1 function [residual, xhatOut] = AERO_EXTKALMAN(meas, deltat)
2 % AERO_EXTKALMAN Radar Data Processing Tracker Using an Extended Kalman
3 % Filter. Radar update time deltat is inherited from workspace.
4
5 % Initialization
6 persistent P
7 persistent xhat
8 if isempty(P)
9     xhat = [0.001; 0.01; 0.001; 400];
10    P = zeros(4);
11 end
12
13 % 1. Compute Phi, Q, and R
14 Phi = [1 deltat 0 0; 0 1 0 0; 0 0 1 deltat; 0 0 0 1];
15 Q = diag([0 .005 0 .005]);
16 R = diag([300^2 0.001^2]);
17
18 % 2. Propagate the covariance matrix:
19 P = Phi*P*Phi' + Q;
20
21 % 3. Propagate the track estimate::
22 xhat = Phi*xhat;
23
24 % 4 a). Compute observation estimates:
25 Rangehat = sqrt(xhat(1)^2+xhat(3)^2);
26 Bearinghat = atan2(xhat(3),xhat(1));
27
28 % 4 b). Compute observation vector y and linearized measurement matrix M
29 yhat = [Rangehat;
30         Bearinghat];
31 M = [ cos(Bearinghat)          0 sin(Bearinghat)          0
32       -sin(Bearinghat)/Rangehat 0 cos(Bearinghat)/Rangehat 0 ];
33
34 % 4 c). Compute residual (Estimation Error)
35 residual = meas - yhat;
36
37 % 5. Compute Kalman Gain:
38 W = P*M'*inv(M*P*M'+ R); %#ok
39
40 % 6. Update estimate
41 xhat = xhat + W*residual;
42
43 % 7. Update Covariance Matrix
44 P = (eye(4) -W*M)*P*(eye(4) -W*M)' + W*R*W';
45
46 xhatOut = xhat;
47 end

```

Simulate the model and review the results (displayed automatically).

```
sim(model)
```



Generate Code for the Model

In this section you will generate code for the Kalman Filter portion of the model using the subsystem build functionality provided by Simulink Coder. In the first build, the model is configured to generate code using Simulink Coder™. In the second build, the model is configured to generate code using Embedded Coder®.

```
% Create a temporary folder (in your system's temporary folder) for the
% build and inspection process.
currentDir = pwd;
[~,cgDir] = rtwdemodir();
```

Configure and build the model using Simulink Coder.

```
rtwconfiguredemo(model, 'GRT')
slbuild([model, '/RadarTracker'])
```

```
### Starting build procedure for: RadarTracker
### Successful completion of build procedure for: RadarTracker
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
RadarTracker	Code generated and compiled	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 19.709s

Configure and build the model using Embedded Coder.

```
rtwconfiguredemo(model, 'ERT')
slbuild([model, '/RadarTracker'])
```

```
### Starting build procedure for: RadarTracker
### Successful completion of build procedure for: RadarTracker
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
RadarTracker	Code generated and compiled	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 16.443s

A portion of RadarTracker.c is listed below.

```
cfile = fullfile(cgDir, 'RadarTracker_ert_rtw', 'RadarTracker.c');
rtwdemodbtype(cfile, /* Model step', /* Model initialize', 1, 0);
```

```
/* Model step function */
void RadarTracker_step(void)
{
    real_T P_d_tmp[16];
    real_T Phi_0[16];
    real_T Q[16];
    real_T Q_0[16];
    real_T M[8];
    real_T W[8];
    real_T tmp[8];
    real_T x_tmp[8];
    real_T Phi_1[4];
    real_T Bearinghat;
    real_T M_tmp;
    real_T M_tmp_0;
    real_T Rangehat;
    real_T r;
    real_T tmp_0;
    int32_T Phi_tmp;
    int32_T Phi_tmp_tmp;
    int32_T i;
    int32_T j;
```

```

int8_T Phi[16];
static const real_T e[4] = { 0.0, 0.005, 0.0, 0.005 };

static const real_T c_b[4] = { 90000.0, 0.0, 0.0, 1.0E-6 };

/* MATLAB Function: '<Root>/RadarTracker' incorporates:
 * Inport: '<Root>/meas'
 */
Phi[0] = 1;
Phi[4] = 1;
Phi[8] = 0;
Phi[12] = 0;
Phi[2] = 0;
Phi[6] = 0;
Phi[10] = 1;
Phi[14] = 1;
Phi[1] = 0;
Phi[3] = 0;
Phi[5] = 1;
Phi[7] = 0;
Phi[9] = 0;
Phi[11] = 0;
Phi[13] = 0;
Phi[15] = 1;
memset(&Q[0], 0, sizeof(real_T) << 4U);
for (j = 0; j < 4; j++) {
    Q[j + (j << 2)] = e[j];
    for (i = 0; i < 4; i++) {
        Phi_tmp_tmp = i << 2;
        Phi_tmp = j + Phi_tmp_tmp;
        Phi_0[Phi_tmp] = 0.0;
        Phi_0[Phi_tmp] += rtDW.P_d[Phi_tmp_tmp] * (real_T)Phi[j];
        Phi_0[Phi_tmp] += rtDW.P_d[Phi_tmp_tmp + 1] * (real_T)Phi[j + 4];
        Phi_0[Phi_tmp] += rtDW.P_d[Phi_tmp_tmp + 2] * (real_T)Phi[j + 8];
        Phi_0[Phi_tmp] += rtDW.P_d[Phi_tmp_tmp + 3] * (real_T)Phi[j + 12];
    }
}

for (i = 0; i < 4; i++) {
    Phi_1[i] = 0.0;
    for (j = 0; j < 4; j++) {
        Phi_tmp_tmp = (j << 2) + i;
        rtDW.P_d[Phi_tmp_tmp] = (((Phi_0[i + 4] * (real_T)Phi[j + 4] + Phi_0[i] *
            (real_T)Phi[j]) + Phi_0[i + 8] * (real_T)Phi[j + 8]) + Phi_0[i + 12] *
            (real_T)Phi[j + 12]) + Q[Phi_tmp_tmp];
        Phi_1[i] += (real_T)Phi[Phi_tmp_tmp] * rtDW.xhat[j];
    }
}

rtDW.xhat[0] = Phi_1[0];
rtDW.xhat[1] = Phi_1[1];
rtDW.xhat[2] = Phi_1[2];
rtDW.xhat[3] = Phi_1[3];
Rangehat = sqrt(rtDW.xhat[0] * rtDW.xhat[0] + rtDW.xhat[2] * rtDW.xhat[2]);
Bearinghat = atan2(rtDW.xhat[2], rtDW.xhat[0]);
M_tmp = sin(Bearinghat);
M_tmp_0 = cos(Bearinghat);
M[0] = M_tmp_0;

```

```

M[2] = 0.0;
M[4] = M_tmp;
M[6] = 0.0;
M[1] = -M_tmp / Rangehat;
M[3] = 0.0;
M[5] = M_tmp_0 / Rangehat;
M[7] = 0.0;
rtY.residual[0] = rtU.meas[0] - Rangehat;
rtY.residual[1] = rtU.meas[1] - Bearinghat;
for (i = 0; i < 2; i++) {
    for (j = 0; j < 4; j++) {
        Phi_tmp_tmp = (j << 1) + i;
        x_tmp[j + (i << 2)] = M[Phi_tmp_tmp];
        W[Phi_tmp_tmp] = 0.0;
        Phi_tmp = j << 2;
        W[Phi_tmp_tmp] += rtDW.P_d[Phi_tmp] * M[i];
        W[Phi_tmp_tmp] += rtDW.P_d[Phi_tmp + 2] * M[i + 4];
    }
}

for (i = 0; i < 2; i++) {
    for (j = 0; j < 2; j++) {
        Phi_tmp_tmp = j << 2;
        Phi_tmp = (j << 1) + i;
        Phi_1[Phi_tmp] = ((x_tmp[Phi_tmp_tmp + 1] * W[i + 2] + x_tmp[Phi_tmp_tmp]
            * W[i]) + x_tmp[Phi_tmp_tmp + 2] * W[i + 4]) +
            x_tmp[Phi_tmp_tmp + 3] * W[i + 6]) + c_b[Phi_tmp];
    }
}

if (fabs(Phi_1[1]) > fabs(Phi_1[0])) {
    r = Phi_1[0] / Phi_1[1];
    Rangehat = 1.0 / (r * Phi_1[3] - Phi_1[2]);
    Bearinghat = Phi_1[3] / Phi_1[1] * Rangehat;
    M_tmp = -Rangehat;
    M_tmp_0 = -Phi_1[2] / Phi_1[1] * Rangehat;
    Rangehat *= r;
} else {
    r = Phi_1[1] / Phi_1[0];
    Rangehat = 1.0 / (Phi_1[3] - r * Phi_1[2]);
    Bearinghat = Phi_1[3] / Phi_1[0] * Rangehat;
    M_tmp = -r * Rangehat;
    M_tmp_0 = -Phi_1[2] / Phi_1[0] * Rangehat;
}

for (i = 0; i < 4; i++) {
    for (j = 0; j < 2; j++) {
        Phi_tmp_tmp = j << 2;
        Phi_tmp = i + Phi_tmp_tmp;
        tmp[Phi_tmp] = 0.0;
        tmp[Phi_tmp] += x_tmp[Phi_tmp_tmp] * rtDW.P_d[i];
        tmp[Phi_tmp] += x_tmp[Phi_tmp_tmp + 1] * rtDW.P_d[i + 4];
        tmp[Phi_tmp] += x_tmp[Phi_tmp_tmp + 2] * rtDW.P_d[i + 8];
        tmp[Phi_tmp] += x_tmp[Phi_tmp_tmp + 3] * rtDW.P_d[i + 12];
    }
}

W[i] = 0.0;
W[i] += tmp[i] * Bearinghat;

```

```

    r = tmp[i + 4];
    W[i] += r * M_tmp;
    tmp_0 = W[i] * rtY.residual[0];
    W[i + 4] = 0.0;
    W[i + 4] += tmp[i] * M_tmp_0;
    W[i + 4] += r * Rangehat;
    rtDW.xhat[i] += W[i + 4] * rtY.residual[1] + tmp_0;
}

for (i = 0; i < 16; i++) {
    Phi[i] = 0;
}

Phi[0] = 1;
Phi[5] = 1;
Phi[10] = 1;
Phi[15] = 1;
memset(&Q[0], 0, sizeof(real_T) << 4U);
for (j = 0; j < 4; j++) {
    Q[j + (j << 2)] = 1.0;
    for (i = 0; i < 4; i++) {
        Phi_tmp_tmp = (i << 2) + j;
        P_d_tmp[Phi_tmp_tmp] = 0.0;
        Phi_tmp = i << 1;
        P_d_tmp[Phi_tmp_tmp] += M[Phi_tmp] * W[j];
        P_d_tmp[Phi_tmp_tmp] += M[Phi_tmp + 1] * W[j + 4];
    }
}

for (i = 0; i < 16; i++) {
    Q_0[i] = Q[i] - P_d_tmp[i];
}

for (i = 0; i < 4; i++) {
    for (j = 0; j < 4; j++) {
        Phi_tmp_tmp = j << 2;
        Phi_tmp = i + Phi_tmp_tmp;
        Q[Phi_tmp] = 0.0;
        Q[Phi_tmp] += rtDW.P_d[Phi_tmp_tmp] * Q_0[i];
        Q[Phi_tmp] += rtDW.P_d[Phi_tmp_tmp + 1] * Q_0[i + 4];
        Q[Phi_tmp] += rtDW.P_d[Phi_tmp_tmp + 2] * Q_0[i + 8];
        Q[Phi_tmp] += rtDW.P_d[Phi_tmp_tmp + 3] * Q_0[i + 12];
        Phi_0[j + (i << 2)] = (real_T)Phi[Phi_tmp] - P_d_tmp[Phi_tmp];
    }

    M[i] = 0.0;
    M[i] += W[i] * 90000.0;
    M[i + 4] = 0.0;
    M[i + 4] += W[i + 4] * 1.0E-6;
}

for (i = 0; i < 4; i++) {
    for (j = 0; j < 4; j++) {
        Phi_tmp_tmp = j << 2;
        Phi_tmp = i + Phi_tmp_tmp;
        Q_0[Phi_tmp] = 0.0;
        Q_0[Phi_tmp] += Phi_0[Phi_tmp_tmp] * Q[i];
        Q_0[Phi_tmp] += Phi_0[Phi_tmp_tmp + 1] * Q[i + 4];
    }
}

```

```

        Q_0[Phi_tmp] += Phi_0[Phi_tmp_tmp + 2] * Q[i + 8];
        Q_0[Phi_tmp] += Phi_0[Phi_tmp_tmp + 3] * Q[i + 12];
        P_d_tmp[Phi_tmp] = 0.0;
        P_d_tmp[Phi_tmp] += M[i] * W[j];
        P_d_tmp[Phi_tmp] += M[i + 4] * W[j + 4];
    }
}

for (i = 0; i < 16; i++) {
    rtDW.P_d[i] = Q_0[i] + P_d_tmp[i];
}

/* Outport: '<Root>/xhatOut' incorporates:
 * MATLAB Function: '<Root>/RadarTracker'
 */
rtY.xhatOut[0] = rtDW.xhat[0];
rtY.xhatOut[1] = rtDW.xhat[1];
rtY.xhatOut[2] = rtDW.xhat[2];
rtY.xhatOut[3] = rtDW.xhat[3];
}

```

You can view the entire generated code in a detailed HTML report, with bi-directional traceability between model and code.

```
web(fullfile(cgDir, 'RadarTracker_ert_rtw', 'html', 'index.html'))
```

Close the model and cleanup.

```

bdclose(model)
rtwdemoclean;
cd(currentDir)

```


Source Code Generation in Simulink Coder

- “Configure Model, Generate Code, and Simulate” on page 24-2
- “Configure Model and Generate Code” on page 24-11
- “Configure Data Interface” on page 24-16
- “Call External C Functions” on page 24-23
- “Reload Generated Code” on page 24-29
- “Manage Build Process Folders” on page 24-30
- “Manage Build Process Files” on page 24-35
- “Manage Build Process File Dependencies” on page 24-43
- “Add Build Process Dependencies” on page 24-52
- “Build Process Support for File and Folder Names” on page 24-57
- “Code Generation of Matrices and Arrays” on page 24-63
- “Cross-Release Shared Utility Code Reuse” on page 24-69
- “Cross-Release Code Integration” on page 24-71
- “Integration of Code from Multiple Folders” on page 24-82
- “Integrate Generated Code by Using Cross-Release Workflow” on page 24-85
- “Compile Code in Another Development Environment” on page 24-93
- “Generate Code and Simulate Models in a Project” on page 24-97
- “Generate Code and Simulate Models Programmatically with Project” on page 24-100

Configure Model, Generate Code, and Simulate

In this section...

“About This Example” on page 24-2
“Functional Design of the Model” on page 24-2
“View the Top Model” on page 24-2
“View the Subsystems” on page 24-3
“Simulation Test Environment” on page 24-4
“Run Simulation Tests” on page 24-8
“Key Points” on page 24-9
“Learn More” on page 24-10

About This Example

Learning Objectives

- Learn about the functional behavior of the example model.
- Learn about the role of the example test harness and its components.
- Run simulation tests on a model.

Prerequisites

- Ability to open and modify Simulink models and subsystems.
- Understand subsystems and how to view subsystem details.
- Understand referenced models and how to view referenced model details.
- Ability to set model configuration parameters.

Required Files

Before you use each example model file, place a copy in a writable location and add it to your MATLAB path.

- `rtwdemo_throttlecntrl` model file
- `rtwdemo_throttlecntrl_testharness` model file

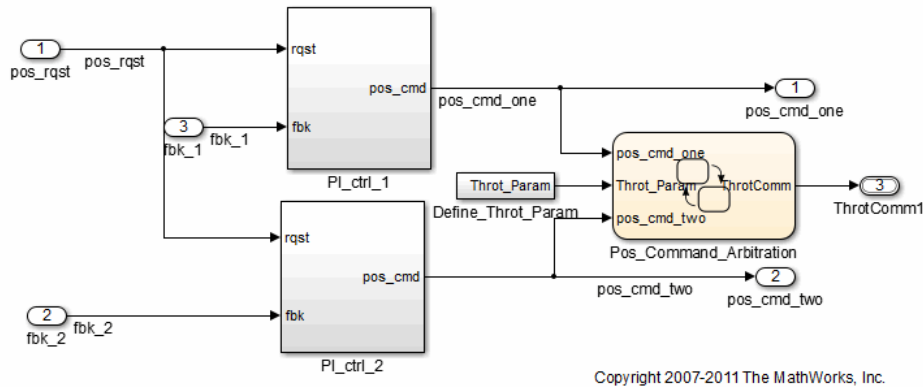
Functional Design of the Model

This example uses a simple, but functionally complete, example model of a throttle controller. The model features redundant control algorithms. The model highlights a standard model structure and a set of basic blocks in algorithm design.

View the Top Model

Open `rtwdemo_throttlecntrl` and save a copy as `throttlecntrl` in a writable location on your MATLAB path.

Note This model uses Stateflow software.



The top level of the model consists of the following elements:

Subsystems	PI_ctrl_1 PI_ctrl_2 Define_Throt_Param Pos_Command_Arbitration
Top-level input	pos_rqst fbk_1 fbk_2
Top-level output	pos_cmd_one pos_cmd_two ThrotComm1
Signal routing	
Omit blocks that change the value of a signal, such as Sum and Integrator	

The layout uses a basic architectural style for models:

- Separation of calculations from signal routing (lines and buses)
- Partitioning into subsystems

You can apply this style to a wide range of models.

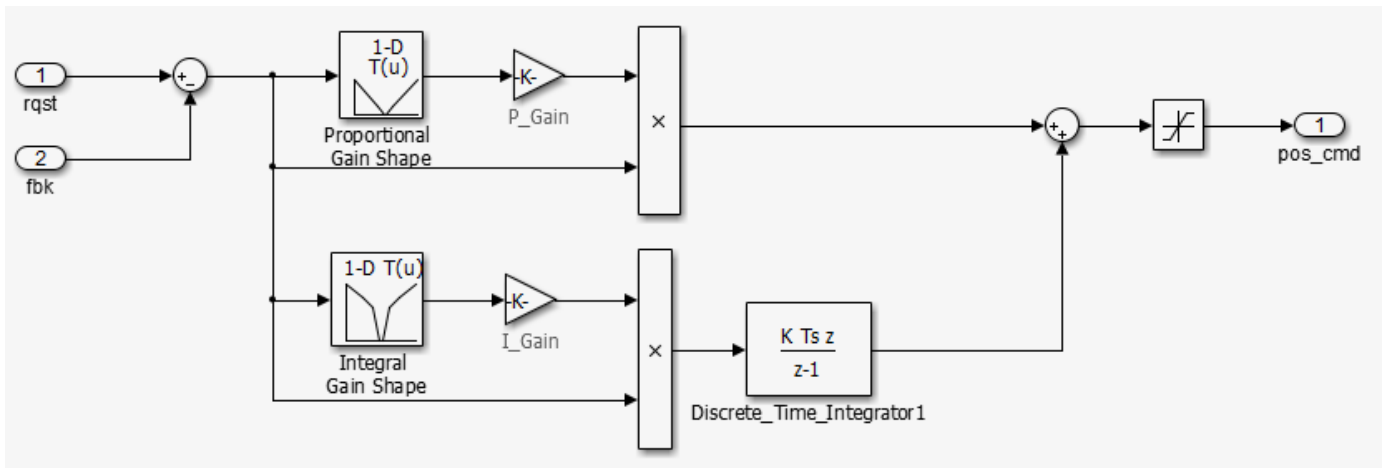
View the Subsystems

Explore two of the subsystems in the top model.

- 1 If not already open, open `throttlectrl`.

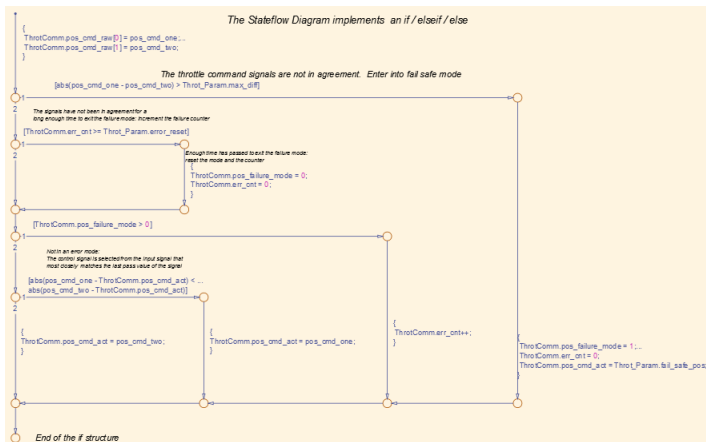
Two subsystems in the top model represent proportional-integral (PI) controllers, `PI_ctrl_1` and `PI_ctrl_2`. At this stage, these identical subsystems, use identical data.

- 2 Open the `PI_ctrl_1` subsystem.



The PI controllers in the model are from a *library*, a group of related blocks or models for reuse. Libraries provide one of two methods for including and reusing models. The second method, model referencing, is described in “Simulation Test Environment” on page 24-4. You cannot edit a block that you add to a model from a library. Edit the block in the library so that instances of the block in different models remain consistent.

- 3 Open the Pos_Command_Arbitration subsystem. This Stateflow chart performs basic error checking on the two command signals. If the command signals are too far apart, the Stateflow diagram sets the output to a fail_safe position.



- 4 Close throttlecntrl.

Simulation Test Environment

To test the throttle controller algorithm, incorporate it into a test harness. A test harness is a model that evaluates the control algorithm and offers the following benefits:

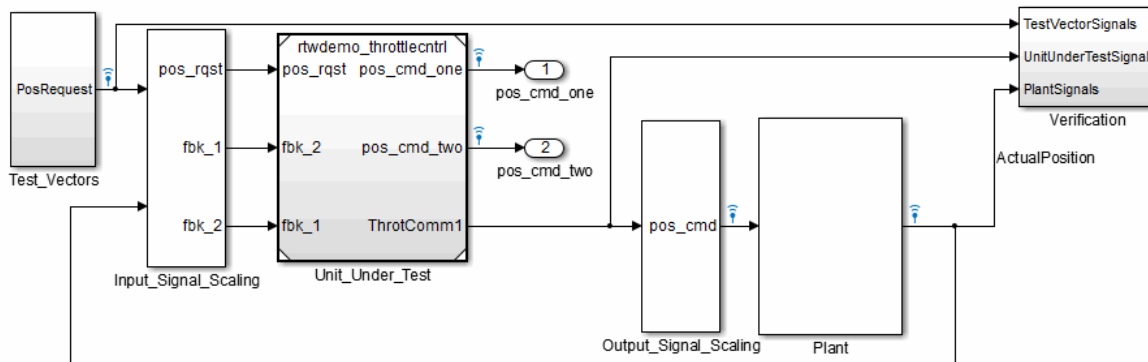
- Separates test data from the control algorithm.
- Separates the plant or feedback model from the control algorithm.
- Provides a reusable environment for multiple versions of the control algorithm.

The test harness model for this example implements a common simulation testing environment consisting of the following parts:

- Unit under test
- Test vector source
- Evaluation and logging
- Plant or feedback system
- Input and output scaling

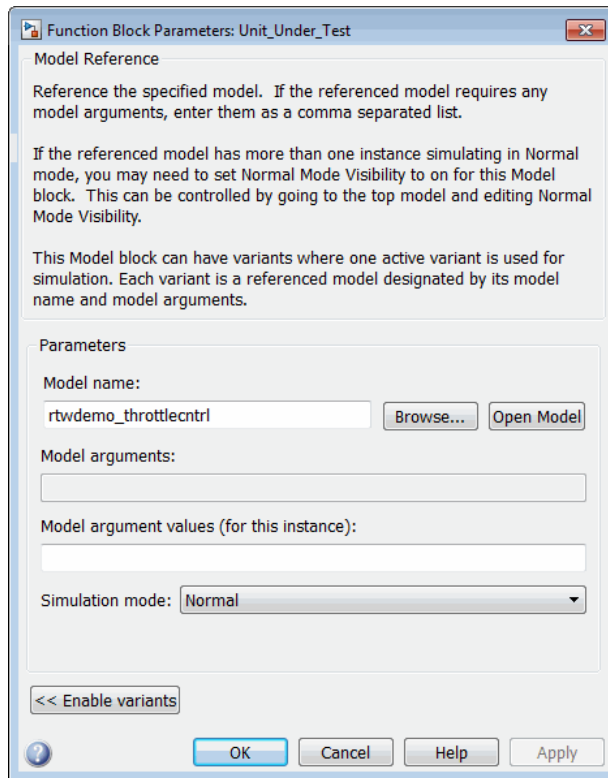
Explore the simulation testing environment.

- 1 Open the test harness model `rtwdemo_throttlecntrl_testharness` and save a copy as `throttlecntrl_testharness` in a writable location on your MATLAB path.



Copyright 2007-2011 The MathWorks, Inc.

- 2 Set up your `throttlecntrl` model as the control algorithm of the test harness.
 - a Open the `Unit_Under_Test` block and view the control algorithm.
 - b View the model reference parameters by right-clicking the `Unit_Under_Test` block and selecting **Block Parameters (ModelReference)**.



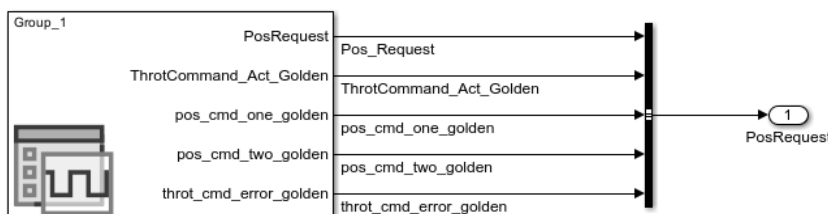
rtwdemo_throttlectrl appears as the name of the referenced model.

- c Change the value of **Model name** to throttlectrl.
- d Update the test harness model diagram by clicking **Simulation > Update Diagram**.

The control algorithm is the unit under test, as indicated by the name of the Model block, Unit_Under_Test.

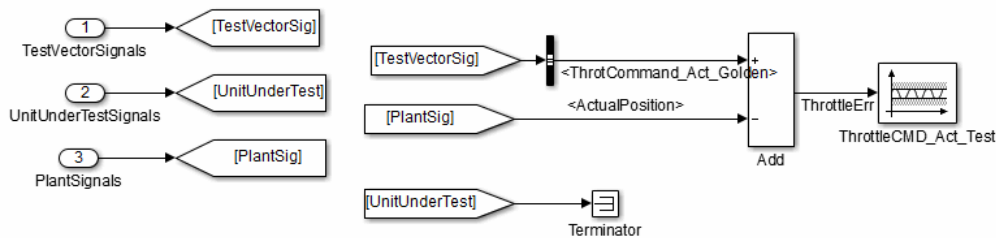
The Model block provides a method for reusing components. From the top model, it allows you to reference other models (directly or indirectly) as compiled functions. By default, Simulink software recompiles the model when the referenced models change. Compiled functions have the following advantages over libraries:

- Simulation time is faster for large models.
 - You can directly simulate compiled functions.
 - Simulation requires less memory. Only one copy of the compiled model is in memory, even when the model is referenced multiple times.
- 3 Open the test vector source, implemented in this test harness as the Test_Vectors subsystem.



The subsystem uses a Signal Editor block for the test vector source. The block has data that drives the simulation (PosRequest) and provides the expected results used by the Verification subsystem. This example test harness uses only one set of test data. Typically, create a test suite that fully exercises the system.

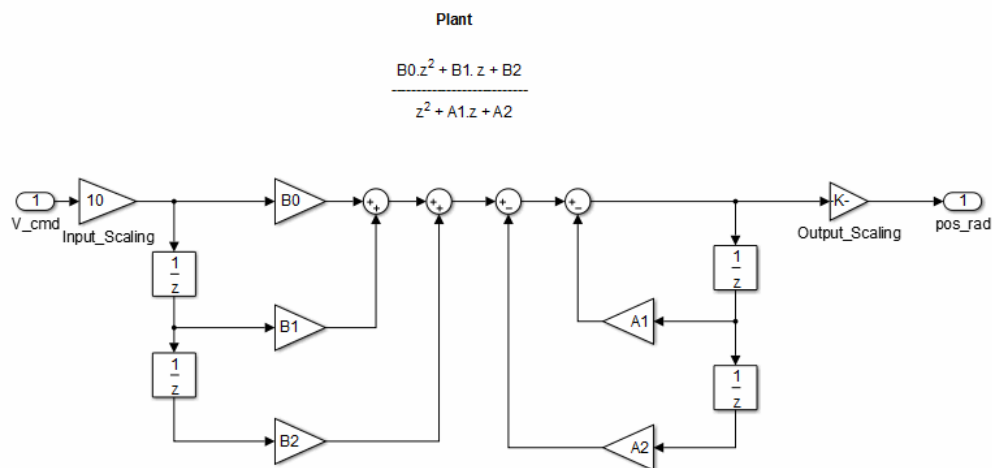
- 4 Open the evaluation and logging subsystem, implemented in this test harness as subsystem Verification.



A test harness compares control algorithm simulation results against golden data — test results that exhibit the desired behavior for the control algorithm as certified by an expert. In the Verification subsystem, an Assertion block compares the simulated throttle value position from the plant against the golden value from the test harness. If the difference between the two signals is greater than 5%, the test fails and the Assertion block stops the simulation.

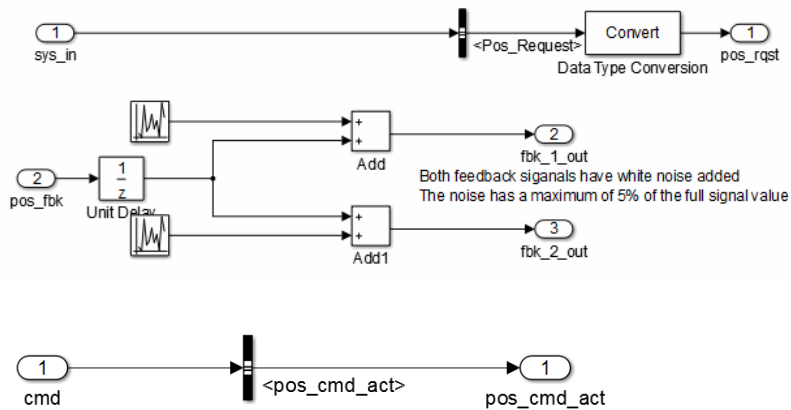
Alternatively, you can evaluate the simulation data after the simulation completes execution. Perform the evaluation with either MATLAB scripts or third-party tools. Post-execution evaluation provides greater flexibility in the analysis of data. However, it requires waiting until execution is complete. Combining the two methods can provide a highly flexible and efficient test environment.

- 5 Open the plant or feedback system, implemented in this test harness as the Plant subsystem.



The Plant subsystem models the throttle dynamics with a transfer function in canonical form. You can create plant models to varying levels of fidelity. It is common to use different plant models at different stages of testing.

- 6 Open the input and output scaling subsystems, implemented in this test harness as Input_Signal_Scaling and Output_Signal_Scaling.



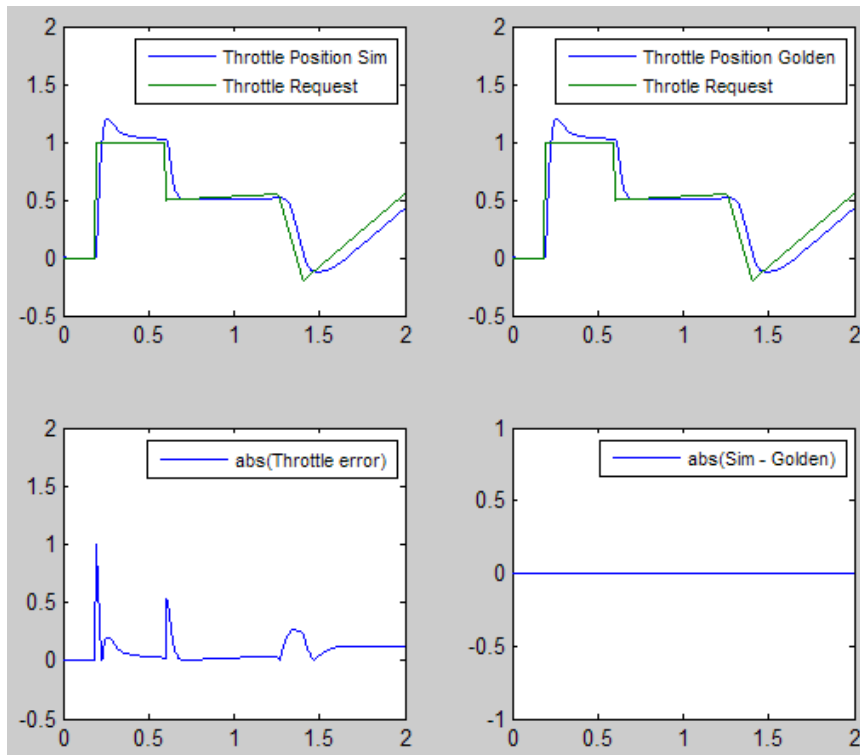
The subsystems that scale input and output perform the following primary functions:

- Select input signals to route to the unit under test.
- Select output signals to route to the plant.
- Rescale signals between engineering units and units that are writable for the unit under test.
- Handle rate transitions between the plant and the unit under test.

7 Save and close `throttlecntrl_testharness`.

Run Simulation Tests

- 1 Check that your working folder is set to a writable folder, such as the folder into which you placed copies of the example model files.
- 2 Open your copy of the test harness model, `throttlecntrl_testharness`.
- 3 Start a test harness model simulation. When the simulation is complete, the following results appear.



The lower-right hand plot shows the difference between the expected (golden) throttle position and the throttle position that the plant calculates. If the difference between the two values is greater than ± 0.05 , the simulation stops.

- 4 Save and close throttle controller and test harness models.

Key Points

- A basic model architecture separates calculations from signal routing and partitions the model into subsystems
- Two options for model reuse include block libraries and model referencing.
- If you represent your control algorithm in a test harness as a Model block, specify the name of the control algorithm model in the Model Reference Parameters dialog box.
- A test harness is a model that evaluates a control algorithm. Typically, a harness consists of a unit under test, a test vector source, evaluation and logging, a plant or feedback system, and input and output scaling components.
- The unit under test is the control algorithm being tested.
- The test vector source provides the data that drives the simulation which generates results used for verification.
- During verification, the test harness compares control algorithm simulation results against golden data and logs the results.
- The plant or feedback component of a test harness models the environment that is being controlled.
- When developing a test harness,

- Scale input and output components.
- Select input signals to route to the unit under test.
- Select output signals to route to the plant.
- Rescale signals between engineering units and units that are writable for the unit under test.
- Handle rate transitions between the plant and the unit under test.

Learn More

- “Support Model Referencing” on page 46-61
- “Code Generation”
- “Signal Groups”

Configure Model and Generate Code

In this section...

“About This Example” on page 24-11
 “Configure the Model for Code Generation” on page 24-11
 “Save Your Model Configuration as a MATLAB Function” on page 24-12
 “Check Model Conditions and Configuration Settings” on page 24-13
 “Generate Code for the Model” on page 24-13
 “Review the Generated Code” on page 24-13
 “Generate an Executable” on page 24-14
 “Key Points” on page 24-14

About This Example

Learning Objectives

- Configure a model for code generation.
- Use model checking tools to discover conditions and configuration settings resulting in generation of inaccurate or inefficient code.
- Generate code from a model.
- Locate and identify generated code files.
- Review generated code.

Prerequisites

- Ability to open and modify Simulink models and subsystems.
- Ability to set model configuration parameters.
- Ability to use the Code Generation Advisor.
- Ability to read C code.
- An installed, supported C compiler.

Required Files

rtwdemo_throttlecntrl model file

Configure the Model for Code Generation

Model configuration parameters determine the method for generating the code and the resulting format.

- 1 Open `rtwdemo_throttlecntrl` and save a copy as `throttlecntrl` in a writable location on your MATLAB path. Then, open the **Simulink Coder** app.
- 2 Open the Configuration Parameters dialog box, **Solver** pane. To generate code for a model, you must configure the model to use a fixed-step solver. The following table shows the solver configuration for this example.

Parameter	Setting	Effect on Generated Code
Type	Fixed-step	Maintains a constant (fixed) step size, which is required for code generation
Solver	discrete (no continuous states)	Applies a fixed-step integration technique for computing the state derivative of the model
Fixed-step size	.001	Sets the base rate; must be the lowest common multiple of the rates in the system

Solver options

Type: Solver:

► Additional options

- 3 Set model configuration parameter **System target file** to `grt.tlc`.

Note The GRT (Generic Real-Time Target) configuration requires a fixed-step solver. However, the `rsim.tlc` system target file supports variable step code generation.

The system target file defines an environment for generating and building code for execution on a certain hardware or operating system platform. For example, one property of a system target file is the value for the `CodeFormat` TLC variable. The GRT configuration requires a fixed step solver and the `rsim.tlc` supports variable step code generation.

- 4 Set model configuration parameter **Include directories** as follows:

```
"$matlabroot$\toolbox\rtw\rtwdemos\EmbeddedCoderOverview\"
```

This folder includes files that are required to build an executable for the model.

- 5 Close the Model Configuration Parameters dialog box.

Save Your Model Configuration as a MATLAB Function

You can save the settings of model configuration parameters as a MATLAB function by using the `getActiveConfigSet` function. In the MATLAB Command Window, enter:

```
thcntrlAcs = getActiveConfigSet('throttlecntrl');
thcntrlAcs.saveAs('throttlecntrlModelConfig');
```

You can then use the resulting function (for example, `throttlecntrlModelConfig`) to:

- Archive the model configuration.
- Compare different model configurations by using differencing tools.
- Set the configuration of other models.

For example, you can set the configuration of model `myModel` to match the configuration of the throttle controller model by opening `myModel` and entering:

```
myModelAcs = throttlecntrlModelConfig;
attachConfigSet('myModel', myModelAcs, true);
setActiveConfigSet('myModel', myModelAcs.Name);
```

For more information, see “Save a Configuration Set” and “Load a Saved Configuration Set”.

Check Model Conditions and Configuration Settings

Before generating code for a model, use the Code Generation Advisor to check the model conditions and configuration settings. The check identifies issues that can result in inaccurate or inefficient code.

- 1 Open model `throttlecntrl`. Then, open the **Simulink Coder** app.
- 2 In the **C Code** tab, click **C/C++ Code Advisor**. A dialog box opens showing the model system hierarchy.
- 3 Click `rtwdemo_throttlecntrl` and then click **OK**. The Code Generation Advisor window opens.
- 4 Right-click on the **Code Generation Advisor** folder and select **Run Selected Checks**.
- 5 To review the results for check **Check model configuration settings against code generation objectives**, select the check in the left pane. The results are displayed in the right pane.
- 6 After reviewing the check results, you can choose to fix warnings and failures as described in “Fix a Model Advisor Check Warning or Failure”.

Generate Code for the Model

- 1 Open `throttlecntrl`. Then, open the **Simulink Coder** app.
- 2 Select model configuration parameter **Generate code only**.
- 3 Select model configuration parameter **Create code generation report** and click **Apply**.
- 4 Generate code.

Messages appear in the Diagnostics Viewer. The code generator produces standard C and header files, and an HTML code generation report. The code generator places the files in a *build folder*, a subfolder named `throttlecntrl_grt_rtw` under your current working folder.

Review the Generated Code

- 1 In the code generation report, click the link for the generated C model file and review the generated code. Look for these items in the report:
 - Identification, version, timestamp, and configuration comments.
 - Links to help you navigate within and between files
 - Data definitions
 - Scheduler code
 - Controller code
 - Model initialization and termination functions

- Call interface for the GRT system target file — output, update, initialization, start, and terminate
- 2 Save and close model `throttlecntrl`.

Consider examining the following files. In the code generation report **Contents** pane, click the links.

File	Description
<code>throttlecntrl.c</code>	C file that contains the scheduler, controller, initialization, and interface code
<code>throttlecntrl_data.c</code>	C file that assigns values to generated data structures
<code>throttlecntrl.h</code>	Header file that defines data structures
<code>throttlecntrl_private.h</code>	Header file that defines data used only by the generated code
<code>throttlecntrl_types.h</code>	Header file that defines the model data structure

For more information, see “Manage Build Process File Dependencies” on page 24-43.

At this point, consider logging data to a MAT-file. For an example, see “Log Data for Analysis” on page 33-2.

Generate an Executable

- 1 Open `throttlecntrl`. Then, open the **Simulink Coder** app.
- 2 Clear model configuration parameter **Generate code only** and click **Apply**.
- 3 Build the model. Watch the messages in the Diagnostics Viewer. The code generator uses a template make file associated with your system target file selection to create an executable file. You can run this program on your workstation, independent of external timing and events.
- 4 Check your working folder for the file `throttlecntrl.exe`.
- 5 Run the executable. In the Command Window, enter `!throttlecntrl`. The `!` character passes the command that follows it to the operating system, which runs the standalone program.

The program produces one line of output in the Command Window:

```
** starting the model **
```

At this point, consider logging data to a MAT-file. For an example, see “Log Data for Analysis” on page 33-2.

Tip For UNIX platforms, run the executable program in the Command Window with the syntax `!./executable_name`. If preferred, run the executable program from an OS shell with the syntax `./executable_name`. For more information, see “Run External Commands, Scripts, and Programs”.

Key Points

- To generate code, change the model configuration to specify a fixed-step solver then select a system target file. Using the `grt.tlc` file requires a fixed-step solver. If the model contains continuous time blocks, you can use a variable-step solver with the `rsim.tlc` system target file.

- After debugging a model, consider configuring a model with parameter inlining enabled.
- Use the `getActiveConfigSet` function to save a model configuration for future use or to apply it to another model.
- Before generating code, consider checking a model with the Code Generation Advisor.
- The code generator places generated files in a subfolder (*model_grt_rtw*) of your working folder.

See Also

More About

- “Code Generation”
- “Model Configuration Sets”
- “Check Your Model Using the Model Advisor”

Configure Data Interface

About This Example

Learning Objectives

- Configure the data interface for code generated for a model.
- Control the name, data type, and data storage class of signals and parameters in generated code.

Prerequisites

- Understanding ways to represent and use data and signals in models.
- Familiarity with representing data constructs as data objects.
- Ability to read C code.

Required File

rtwdemo_throttlenctrl_datainterface model file

Declare Data

Most programming languages require that you declare data before using it. The declaration specifies the following information:

Data Attribute	Description
Scope	The region of the program that has access to the data
Duration	The period during which the data is resident in memory
Data type	The amount of memory allocated for the data
Initialization	An initial value, a pointer to memory, or NULL. If you do not provide an initial value, most compilers assign a zero value or a null pointer.

The following data types are supported for code generation.

Supported Data Types

Name	Description
double	Double-precision floating point
single	Single-precision floating point
int8	Signed 8-bit integer
uint8	Unsigned 8-bit integer
int16	Signed 16-bit integer
uint16	Unsigned 16-bit integer
int32	Signed 32-bit integer
uint32	Unsigned 32-bit integer
Fixed-point data types	8-, 16-, 32-bit word lengths

A storage class is the scope and duration of a data item. For more information about storage classes, see “C Code Generation Configuration for Model Interface Elements” on page 16-2.

Use Data Objects

In Simulink models and Stateflow charts, the following methods are available for declaring data: data objects and direct specification. This example uses the data object method. Both methods allow full control over the data type and storage class. You can mix the two methods in a single model.

In the MATLAB and Simulink environment, you can use data objects in various ways. This example focuses on the following types of data objects:

- Signal
- Parameter
- Bus

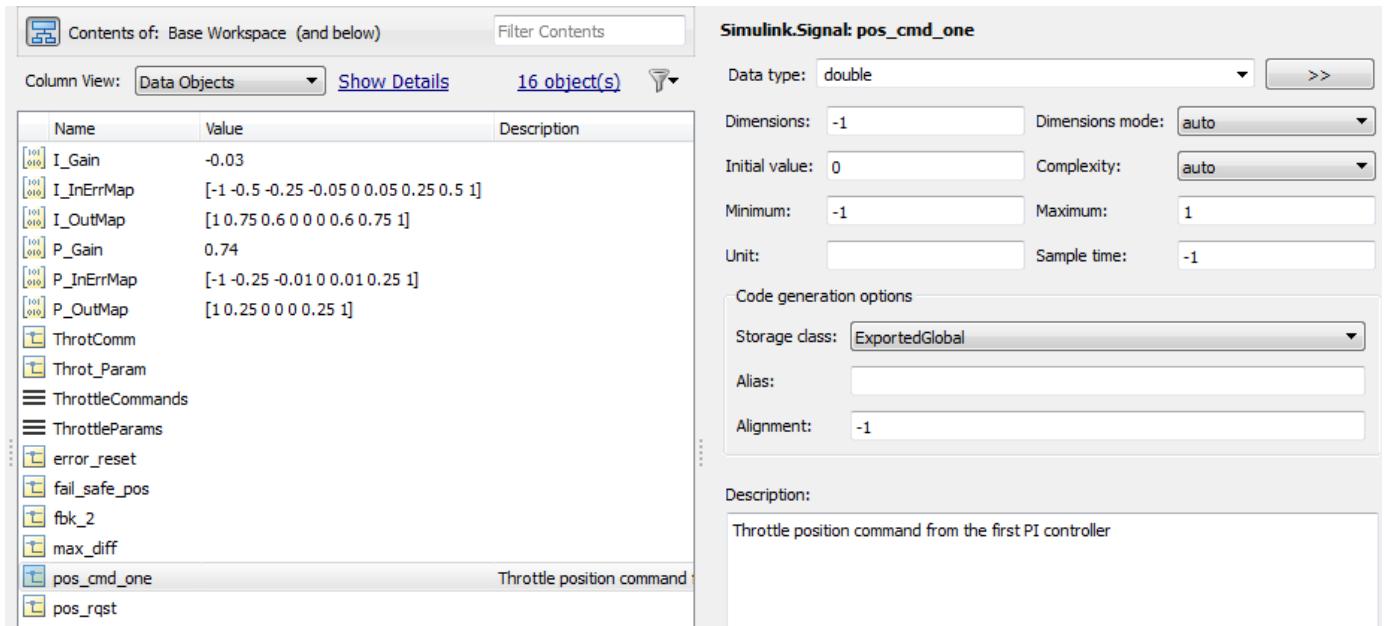
To configure the data interface for your model using the data object method, in the MATLAB base workspace, you define data objects. Then, associate them with your Simulink model or embedded Stateflow chart. When you build your model, the build process uses the associated base workspace data objects in the generated code.

You can set the values of the data object properties, which include:

- Data type
- Storage class
- Value (parameters)
- Initial value (signals)
- Identifier (define a different name in the generated code)
- Dimension (typically inherited for parameters)
- Complexity (inherited for parameters)
- Unit (physical measurement unit)
- Minimum value
- Maximum value
- Description (used to document your data objects — does not affect simulation or code generation)

You can create and inspect base workspace data objects by entering commands in the MATLAB Command Window or by using Model Explorer. To explore base workspace signal data objects, use these steps:

- 1 Open `rtwdemo_throttlectrl_datainterface` and save a copy as `throttlectrl_datainterface` in a writable location on your MATLAB path.
- 2 Open Model Explorer.
- 3 Select **Base Workspace**.
- 4 Select the `pos_cmd_one` signal object for viewing.



You can also view the definition of a signal object. In the MATLAB Command Window, enter `pos_cmd_one`:

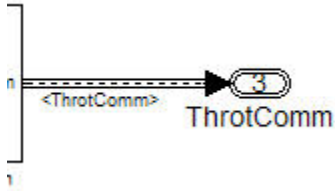
```
pos_cmd_one =
Signal with properties:
    CoderInfo: [1x1 Simulink.CoderInfo]
    Description: 'Throttle position command from the first PI controller'
    DataType: 'double'
    Min: -1
    Max: 1
    Unit: ''
    Dimensions: -1
    DimensionsMode: 'auto'
    Complexity: 'auto'
    SampleTime: -1
    InitialValue: '0'
```

- To view other signal objects, in Model Explorer, click the object name or in the MATLAB Command Window, enter the object name. This table summarizes object characteristics for some of the data objects in this model.

Object Characteristics	pos_cmd_one	pos_rqst	P_InErrMap	ThrotComm*	ThrottleCommands*
Description	Top-level output	Top-level input	Calibration parameter	Top-level output structure	Bus definition
Data type	Double	Double	Auto	Auto	Structure
Storage class	Exported global	Imported extern pointer	Constant	Exported global	None

* `ThrottleCommands` defines a Bus object; `ThrotComm` is an instantiation of the bus. If the bus is a nonvirtual bus, the signal generates a structure in the C code.

You can use a bus definition (`ThrottleCommands`) to instantiate multiple instances of the structure. In a model diagram, a bus object appears as a wide line with central dashes, as shown.



Add New Data Objects

You can create data objects for named signals, states, and parameters. To associate a data object with a construct, the construct must have a name.

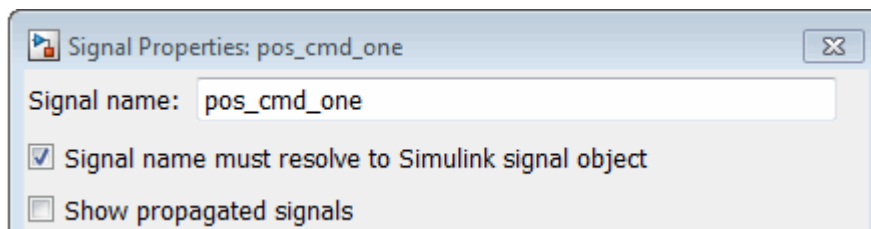
To find constructs for which you can create data objects, use the Data Object Wizard. This tool finds the constructs and then creates the objects for you. The model includes two signals that are not associated with data objects: `fbk_1` and `pos_cmd_two`.

To find the signals and create data objects for them:

- 1 In the Simulink Editor, in the **Modeling** tab, under **Design**, click **Data Object Wizard**. The Data Object Wizard dialog box opens.
- 2 To find candidate constructs, click **Find**. Constructs `fbk_1` and `pos_cmd_two` appear in the dialog box.
- 3 To select both constructs, click **Select All**.
- 4 In the table, under **Class**, make sure that each proposed data object uses the class `Simulink.Signal`. To change the class of the objects, click **Change Class**.
- 5 To create the data objects, click **Create**. Constructs `fbk_1` and `pos_cmd_two` are removed from the dialog box.
- 6 Close the Data Object Wizard.
- 7 In the **Contents** pane of the Model Explorer, find the newly created objects `fbk_1` and `pos_cmd_two`.

Enable Data Objects for Generated Code

- 1 Enable a signal to appear in generated code.
 - a In the Simulink Editor, right-click the `pos_cmd_one` signal line and select **Properties**. A Signal Properties dialog box opens.
 - b Make sure that you select the **Signal name must resolve to Simulink signal object** parameter.



- 2 Enable signal object resolution for the signals in the model. In the MATLAB Command Window, enter:

```
disableimplicitsignalresolution('throttlecntrl_datainterface')
```

- 3 Save and close throttlecntrl_datainterface.

Effects of Simulation on Data Typing

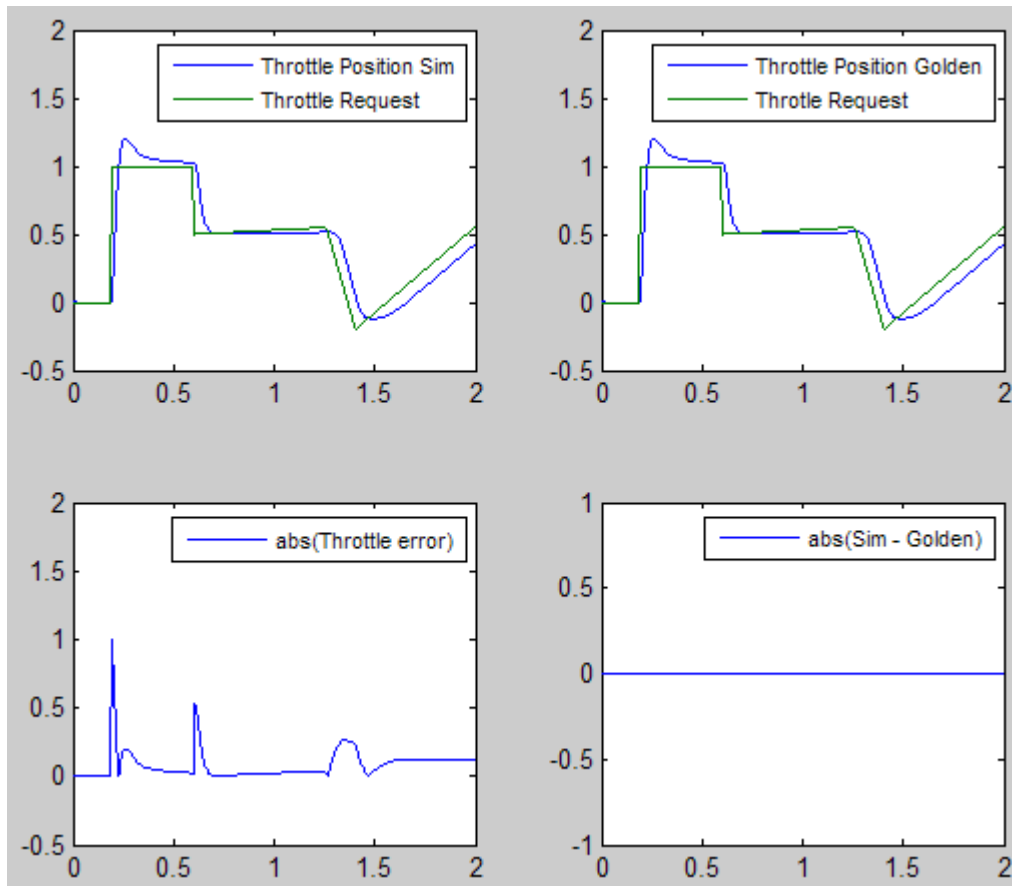
In the throttle controller model, the data types are set to `double`. Because Simulink software uses the `double` data type for simulation, do not expect changes in the model behavior when you run the generated code. You verify this effect by running the test harness.

Before you run your test harness, update it to include the `throttlecntrl_datainterface` model.

Note The following procedure requires a Stateflow license.

- 1 Open `throttlecntrl_datainterface`.
- 2 Open your copy of test harness, `throttlecntrl_testharness`.
- 3 Right-click the `Unit_Under_Test Model` block and select **Block Parameters (ModelReference)**.
- 4 Set **Model name** to `throttlecntrl_datainterface`. Click **OK**.
- 5 Update the test harness model diagram.
- 6 Simulate the test harness.

The resulting plot shows that the difference between the golden and simulated versions of the model remains zero.



7 Save and close `throttlecntrl_testharness`.

Manage Data

Data objects exist in a separate file from the model in the base workspace. To save the data manually, in the MATLAB Command Window, enter `save`.

The separation of data from the model provides the following benefits:

- One model, multiple data sets:
 - Use of different parameter values to change the behavior of the control algorithm (for example, for reusable components with different calibration values)
 - Use of different data types to change targeted hardware (for example, for floating-point and fixed-point targeted hardware)
- Multiple models, one data set:
 - Sharing data between models in a system
 - Sharing data between projects (for example, transmission, engine, and wheel controllers can use the same CAN message data set)

Key Points

- You can declare data in Simulink models and Stateflow charts by using data objects or direct specification.
- From the Model Explorer or from the command line in the MATLAB Command Window, manage (create, view, configure, and so on) base workspace data.
- The Data Object Wizard provides a quick way to create data objects for constructs such as signals, buses, and parameters.
- Configure data objects explicitly to appear by name in generated code.
- Separation of data from model provides several benefits.

See Also

More About

- “Load Signal Data for Simulation”
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49
- “Organize Parameter Data into a Structure by Using Struct Storage Class” (Embedded Coder)
- “Control Placement of Global Data Definitions and Declarations in Generated Files” (Embedded Coder)

Call External C Functions

In this section...

“About This Example” on page 24-23
 “Include External C Functions in a Model” on page 24-23
 “Create a Block That Calls a C Function” on page 24-23
 “Validate External Code in the Simulink Environment” on page 24-25
 “Validate C Code as Part of a Model” on page 24-26
 “Call a C Function from Generated Code” on page 24-27
 “Key Points” on page 24-27

About This Example

Learning Objectives

- Evaluate a C function as part of a model simulation.
- Call an external C function from generated code.

Prerequisites

- Ability to open and modify Simulink models and subsystems.
- Ability to set model configuration parameters.
- Ability to read C code.
- An installed, supported C compiler.

Required Files

- `rtwdemo_throttlectrl_extfuncall` model file
- `rtwdemo_ValidateLegacyCodeVrsSim` model file
- `/toolbox/rtw/rtwdemos/EmbeddedCoderOverview/stage_4_files/SimpleTable.c`
- `/toolbox/rtw/rtwdemos/EmbeddedCoderOverview/stage_4_files/SimpleTable.h`

Include External C Functions in a Model

Simulink models are one part of Model-Based Design. For many applications, a design also includes a set of pre-existing C functions created, tested (verified), and validated outside of a MATLAB and Simulink environment. You can integrate these functions easily into a model and the generated code. You can use external C code in the generated code to access hardware devices and external data files during rapid simulation runs.

This example shows you how to create a custom block that calls an external C function. When the block is part of the model, you can take advantage of the simulation environment to test the system further.

Create a Block That Calls a C Function

To specify a call to an external C function, use an S-Function block. You can automate the process of creating the S-Function block by using the Simulink Legacy Code Tool. Using this tool, specify an

interface for your external C function. The tool then uses that interface to automate creation of an S-Function block.

- 1 Make copies of the files `SimpleTable.c` and `SimpleTable.h`, located in the folder `matlabroot/toolbox/rtw/rtwdemos/EmbeddedCoderOverview/stage_4_files` (open). Put the copies in your working folder.
- 2 Create an S-Function block that calls the specified function at each time step during simulation:

- a In the MATLAB Command Window, create a function interface definition structure:

```
def=legacy_code('initialize')
```

The data structure `def` defines the function interface to the external C code.

```
def =
```

```

      SFunctionName: ''
InitializeConditionsFcnSpec: ''
      OutputFcnSpec: ''
      StartFcnSpec: ''
      TerminateFcnSpec: ''
      HeaderFiles: {}
      SourceFiles: {}
      HostLibFiles: {}
      TargetLibFiles: {}
      IncPaths: {}
      SrcPaths: {}
      LibPaths: {}
      SampleTime: 'inherited'
      Options: [1x1 struct]
```

- b Populate the function interface definition structure by entering the following commands:

```
def.OutputFcnSpec=['double y1 = SimpleTable(double u1,',...
  'double p1[], double p2[], int16 p3)'];
def.HeaderFiles = {'SimpleTable.h'};
def.SourceFiles = {'SimpleTable.c'};
def.SFunctionName = 'SimpTableWrap';
```

- c Create the S-function:

```
legacy_code('sfcn_cmex_generate', def)
```

- d Compile the S-function:

```
legacy_code('compile', def)
```

- e Create the S-Function block:

```
legacy_code('slblock_generate', def)
```

A new model window opens that contains the `SimpTableWrap` block.

Tip Creating the S-Function block is a one-time task. Once the block exists, you can reuse it in multiple models.

- 3 Save the model to your working folder as: `s_func_simpltablewrap`.
- 4 Create a Target Language Compiler (TLC) file for the S-Function block:

```
legacy_code('sfcn_tlc_generate', def)
```


The TLC file is the component of an S-function that specifies how the code generator produces the code for a block.

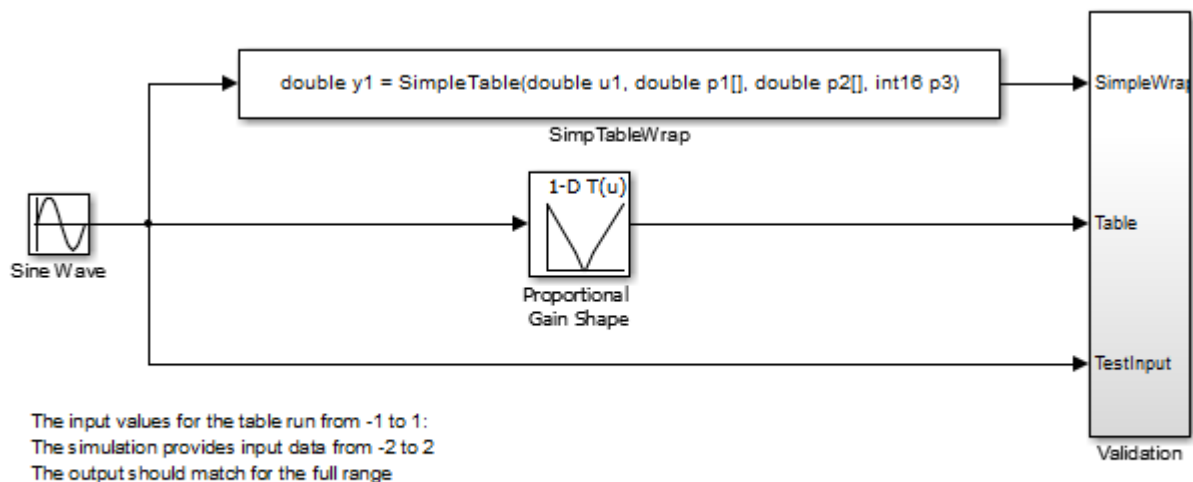
For more information on using the Legacy Code Tool, see:

- “Integrate C Functions Using Legacy Code Tool”
- “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6

Validate External Code in the Simulink Environment

When you integrate external C code with a Simulink model, before using the code, validate the functionality of the external C function code as a standalone component.

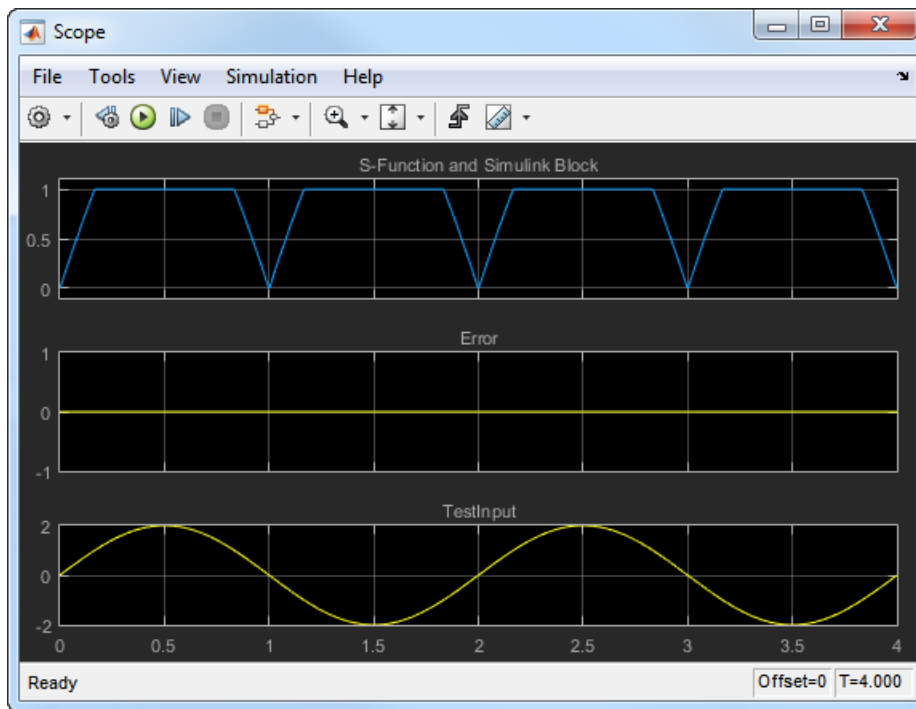
- 1 Open the model `rtwdemo_ValidateLegacyCodeVrsSim`. This model validates the S-function block that you created.



Copyright 2007-2011 The MathWorks, Inc.

- The Sine Wave block produces output values from [-2 : 2].
 - The input range of the lookup table is from [-1 : 1].
 - The output from the lookup table is the absolute value of the input.
 - The lookup table output clips the output at the input limits.
- 2 Simulate the model.
 - 3 View the validation results by opening the `Validation` subsystem and, in that subsystem, clicking the Scope block.

The following figure shows the validation results. The external C code and the Simulink Lookup table block provide the same output values.



- 4 Close the validation model.

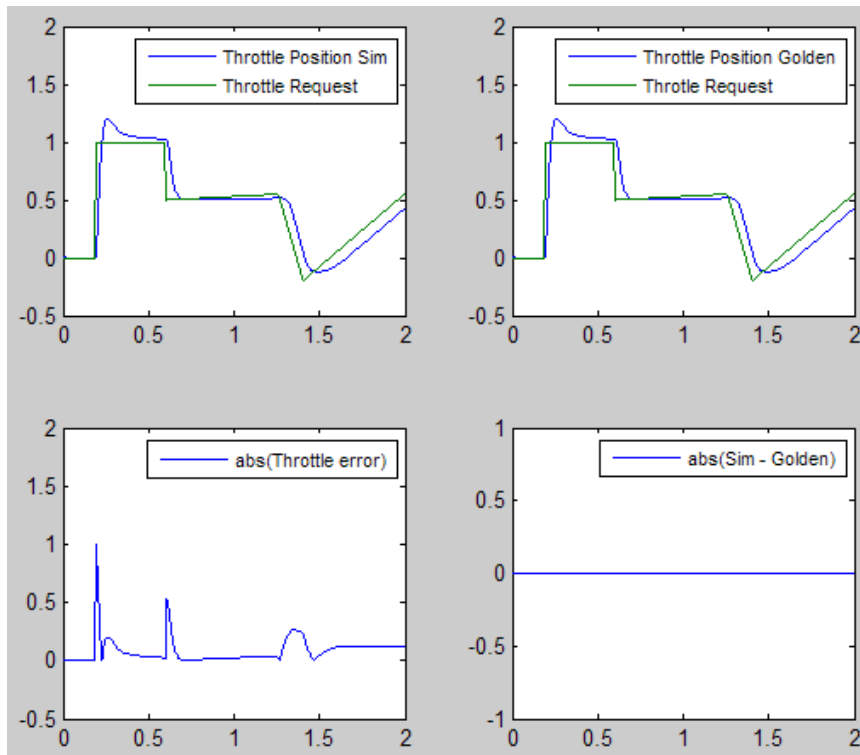
Validate C Code as Part of a Model

After you validate the functionality of the external C function code as a standalone component, validate the S-function in the model. Use the test harness model to complete the validation.

Note The following procedure requires a Stateflow license.

- 1 Open `rtwdemo_throttlectrl_extfunccall` and save a copy to `throttlectrl_extfunccall` in a writable folder on your MATLAB path.
- 2 Examine the `PI_ctrl_1` and `PI_ctrl_2` subsystems.
 - a Lookup blocks have been replaced with the block you created using the Legacy Code Tool.
 - b Examine the block parameter settings for `SimpTableWrap` and `SimpTableWrap1`.
 - c Close the Block Parameter dialog boxes and the PI subsystem windows.
- 3 Open the test harness model, right-click the `Unit_Under_Test Model` block, and select **Block Parameters (ModelReference)**.
- 4 Set **Model name** to `throttlectrl_extfunccall`. Click **OK**.
- 5 Update the test harness model diagram.
- 6 Simulate the test harness.

The simulation results match the expected golden values.



- 7 Save and close `throttlecntrl_extfunccall` and `throttlecntrl_testharness`.

Call a C Function from Generated Code

The code generator uses a TLC file to process the S-Function block. Calls to C code embedded in an S-Function block:

- Can use data objects.
- Are subject to expression folding, an operation that combines multiple computations into a single output calculation.

- 1 Open `throttlecntrl_extfunccall`.
- 2 Generate code for the model.
- 3 Examine the generated code in the file `throttlecntrl_extfunccall.c`.
- 4 Close `throttlecntrl_extfunccall` and `throttlecntrl_testharness`.

Key Points

- You can easily integrate external functions into a model and generated code by using the Legacy Code Tool.
- Validate the functionality of external C function code which you integrate into a model as a standalone component.
- After you validate the functionality of external C function code as a standalone component, validate the S-function in the model.

See Also

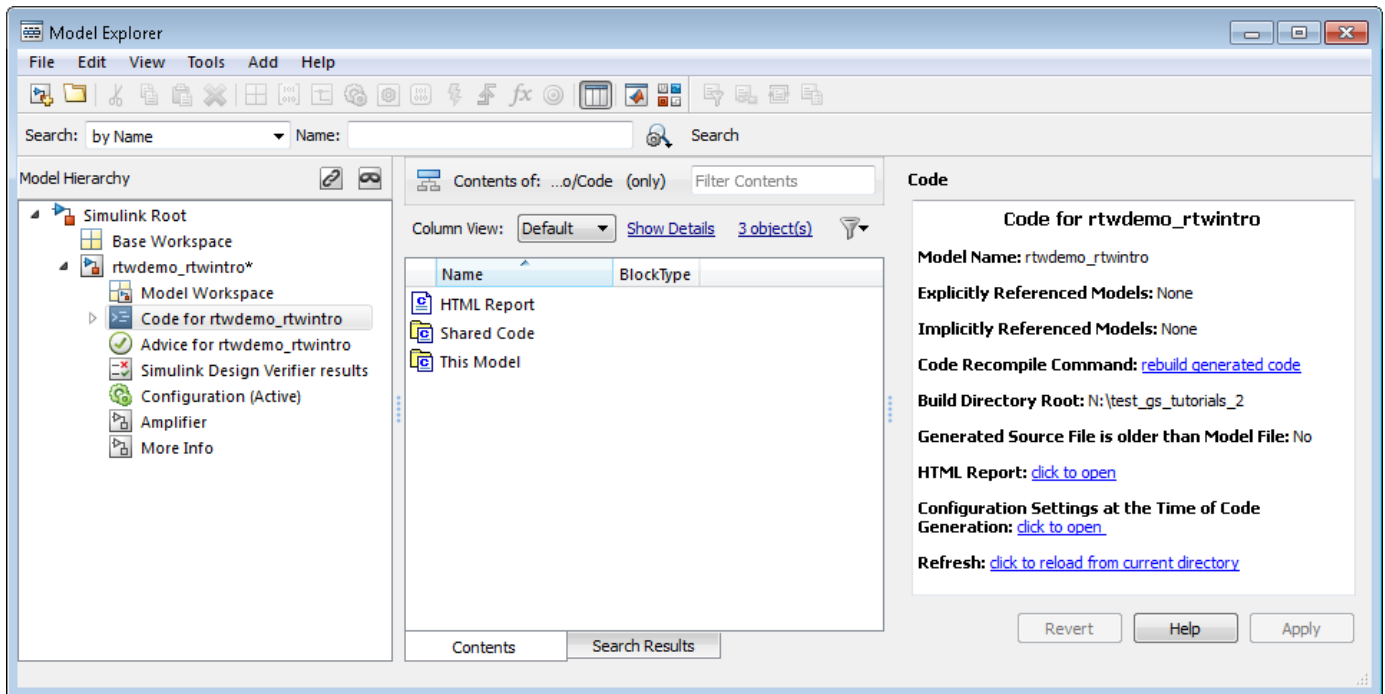
More About

- “Integrate C Functions Using Legacy Code Tool”
- “S-Functions and Code Generation” on page 14-2

Reload Generated Code

You can reload the code generated for a model from the Model Explorer.

- 1 Click the **Code for *model*** node in the **Model Hierarchy** pane.
- 2 In the **Code** pane, click the **Refresh** link.



The code generator reloads the code for the model from the build folder.

See Also

More About

- “Rebuild a Model” on page 31-48
- “Control Regeneration of Top Model Code” on page 31-49

Manage Build Process Folders

The build process places generated files from Simulink diagram updates and model builds into a hierarchy of folders that is specified by default. You can change the default specification for build process folders, if, for example:

- Your company does not use the current working folder as the location for the code generation folder or the simulation cache folder.
- You place the code generation folder under version control, but do not place the simulation cache folder under version control.

The MATLAB session parameters `CacheFolder`, `CodeGenFolder`, and `CodeGenFolderStructure` are file generation control parameters that specify the folder locations for the build process. At the start of a MATLAB session, these Simulink preferences determine the values of the parameters:

“**Simulation cache folder**” - `CacheFolder`

“**Code generation folder**” - `CodeGenFolder`

“**Code generation folder structure**” - `CodeGenFolderStructure`

To modify the parameters during a MATLAB session, use `Simulink.fileGenControl`. The parameter values that you set expire at the end of the MATLAB session.

File Generation Control Parameters

This table provides information about how you can use the parameters to manage build process folders.

MATLAB Session Parameter	Simulink Preference	Description
CacheFolder	“ Simulation cache folder ”	<p>The build process places generated files from Simulink diagram updates and model build artifacts for simulation and code generation in the simulation cache folder <i>simulationCacheFolder</i>. The folder is a root folder.</p> <p>By default (<code>CacheFolder = ''</code>), <i>simulationCacheFolder</i> is the current working folder, <code>pwd</code>.</p> <p>You can use the parameter to specify another folder. For example, if you want to:</p> <ul style="list-style-type: none"> • Separate generated files from the models and other source material. • Reuse or share previously built simulation targets without having to set the current working folder back to a previous working folder.

MATLAB Session Parameter	Simulink Preference	Description
CodeGenFolder	“Code generation folder”	<p>The build process, which uses system target files to generate production code from a Simulink model, places the production code in the code generation folder <i>codeGenerationFolder</i>. The folder is a root folder.</p> <p>If you choose to generate an executable program file, the build process writes the file <i>model.exe</i> (Windows) or <i>model</i> (UNIX) to the folder.</p> <p>By default (<code>CodeGenFolder = ''</code>), <i>codeGenerationFolder</i> is the current working folder, <code>pwd</code>.</p> <p>You can use the parameter to specify another folder. For example, if you want to separate generated production code from:</p> <ul style="list-style-type: none"> • Models and other source material. • Generated simulation and code generation artifacts. <p>If you specify the root folder of a drive as the code generation folder, the build process cannot generate code for your model. For example, <code>C:\</code>.</p>
CodeGenFolderStructure	“Code generation folder structure”	<p>To specify the folder structure within the code generation folder, use the parameter. For example, if you configure models for different target environments, you can specify a separate subfolder for the generated code from each model.</p>

Build Process Folders

This table provides information about how `CodeGenFolderStructure` controls the folder structure within the simulation cache folder and the code generation folder.

Folder Name when CodeGenFolderStructure = 'ModelSpecific'	Folder Name when CodeGenFolderStructure = 'TargetEnvironmentSubfolder'	Description
<p><i>codeGenerationFolder/model_target_rtw</i></p> <p>The default for <i>target</i> is the name of the selected system target file, for example, <i>grt</i>, <i>ert</i>, and <i>rsim</i>. You can change <i>target</i> with the <i>rtwgensettings.BuildDirSuffix</i> field in the system target file.</p>	<p><i>codeGenerationFolder/targetSpecific/model</i></p> <p>The build process uses configuration information for the system target file and the hardware device to produce a unique label for the subfolder, <i>targetSpecific</i>.</p>	<p>Build folder, which stores generated source code and other files created by the build process.</p> <p>Contains the generated code modules, <i>model.c</i> and <i>model.h</i>, and generated makefile, <i>model.mk</i>.</p> <p><i>model.mk</i> is for compiling and linking code generated from model components.</p> <p><i>model</i> is the name of the source model.</p>
<p><i>codeGenerationFolder/model_target_rtw/services</i></p>	<p><i>codeGenerationFolder/targetSpecific/model/services</i></p>	<p>Folder created when you generate code for a component model that uses a service code interface configuration.</p> <p>Contains <i>services.h</i>, the header file that specifies function prototypes for target platform services.</p> <p>For more information, see “Generate and Verify Code” (Embedded Coder) and “Identify and Package Files for Deployment” (Embedded Coder).</p>
<p><i>codeGenerationFolder/model_target_rtw/services/lib</i></p>	<p><i>codeGenerationFolder/targetSpecific/model/services/lib</i></p>	<p>Contains <i>buildInfo.mat</i>, which you use for building the component model library that represents the generated code compiled against <i>services.h</i>.</p> <p>For more information, see “Generate and Verify Code” (Embedded Coder) and “Identify and Package Files for Deployment” (Embedded Coder).</p>
<p><i>codeGenerationFolder/model_target_rtw/html</i></p>	<p><i>codeGenerationFolder/targetSpecific/model/html</i></p>	<p>Code generation report folder that contains report files generated by the build process.</p>

Folder Name when CodeGenFolderStructure = 'ModelSpecific'	Folder Name when CodeGenFolderStructure = 'TargetEnvironmentSubfolder'	Description
<code>codeGenerationFolder/slprj/target/model</code>	<code>codeGenerationFolder/targetSpecific/_ref/model</code>	Model reference target files.
<code>codeGenerationFolder/slprj/target/model/referenced_model_includes</code>	<code>codeGenerationFolder/targetSpecific/_ref/model/referenced_model_includes</code>	Header files from models referenced by <code>model</code> .
<code>codeGenerationFolder/slprj/target/model/tmwinternal</code>	<code>codeGenerationFolder/targetSpecific/_ref/model/tmwinternal</code>	MAT-files used during code generation.
<code>codeGenerationFolder/slprj/target/_sharedutils</code>	<code>codeGenerationFolder/targetSpecific/_shared</code>	Utility functions for model reference system target files, which are shared across models. Folder also contains <code>rtwshared.mk</code> for compiling generated shared utility code
<code>simulationCacheFolder/slprj/sim/model</code>	<code>simulationCacheFolder/slprj/sim/model</code>	Simulation target files for referenced models.
<code>simulationCacheFolder/slprj/sim/model/tmwinternal</code>	<code>simulationCacheFolder/slprj/sim/model/tmwinternal</code>	MAT-files used during code generation.
<code>simulationCacheFolder/slprj/sim/_sharedutils</code>	<code>simulationCacheFolder/slprj/sim/_sharedutils</code>	Utility functions for simulation system target files, which are shared across models.

If the system target file is ERT-based, then these model configuration parameters also control the location of shared utility code:

- Shared code placement (UtilityFuncGeneration)
- Existing shared code (Embedded Coder) (ExistingSharedCode)

You can use `RTW.getBuildDir` to display build folder information for the model.

See Also

`RTW.getBuildDir` | `Simulink.fileGenControl`

More About

- “Manage Build Process Files” on page 24-35
- “Manage Build Process File Dependencies” on page 24-43
- “Add Build Process Dependencies” on page 24-52
- “Build Process Support for File and Folder Names” on page 24-57

- “Build Process Workflow for Real-Time Systems” on page 31-39
- “Generate Code for Model Reference Hierarchy” on page 8-2
- “Cross-Release Shared Utility Code Reuse” on page 24-69
- “Cross-Release Code Integration” on page 24-71
- “Generate Code and Simulate Models in a Project” on page 24-97
- “Generate Code and Simulate Models Programmatically with Project” on page 24-100

Manage Build Process Files

To apply generated code source and header files from the build process, it is helpful to understand the files that the build process generates and the conditions that control file generation. This information provides access to generated code resources, such as:

- Public interface to the model entry points
- Enumerated types corresponding to built-in data types
- Data structures that describe the model signals, states, and parameters

The code generator creates *model*.*** files during the code generation and build process. You can customize the file names for generated header, source, and data files. For more information, see “Customize Generated File Names” (Embedded Coder). The code generator creates additional folders and dependency files to support shared utilities and model references. For more information about the folders that the build process creates, see “Manage Build Process Folders” on page 24-30. For an example that shows how to use a project to manage build process folders, see “Generate Code and Simulate Models in a Project” on page 24-97.

Depending on model architectures and code generation options, the build process for a GRT-based system target file can produce files that the build process does not generate for an ERT-based system target file. Also, for ERT-based system target files, the build process packages generated files differently than for GRT-based system target files. See “Manage File Packaging of Generated Code Modules” (Embedded Coder).

Note By default, the build process deletes foreign (not generated) source files in the build folder. It is possible to preserve foreign source files in the build folder by following the guidelines in “Preserve External Code Files in Build Folder” on page 30-34.

The table describes the principal generated files. Within the generated file names shown in the table, the *model* represents the name of the model for which you are generating code. The *subsystem* represents the name of a subsystem within the model. When you select the **Create code generation report** parameter, the code generator produces a set of HTML files. There is one HTML file for each source file plus a *model_contents.html* index file in the *html* subfolder within your build folder. The source and header files in the table have dependency relationships. For descriptions of other file dependencies, see “Manage Build Process File Dependencies” on page 24-43 and “Add Build Process Dependencies” on page 24-52.

File	Description
<code>builtin_typeid_types.h</code>	<p>Defines an enumerated type corresponding to built-in data types.</p> <p>A model build generates this file when one or more of these conditions apply:</p> <ul style="list-style-type: none"> • Your model contains a Stateflow chart that uses messages. • Your model configuration enables MAT-file logging. • Your model configuration enables C API options in Code Generation > Interface.

File	Description
<i>model.bat</i>	<p>Contains Windows batch file commands that set the compiler environment and invoke the make utility.</p> <p>For more information about using this file, see “model.bat” on page 24-39.</p>
<i>model.c</i> <i>model.cpp</i>	<p>Corresponds to the model file.</p> <p>The Target Language Compiler generates this C or C++ source code file. The file contains:</p> <ul style="list-style-type: none"> • Include files <i>model.h</i> and <i>model_private.h</i> • Data, except data placed in <i>model_data.c</i> • Model-specific scheduler code • Model-specific solver code • Model registration code • Algorithm code • Optional GRT wrapper functions
<i>model.exe</i> (Windows platform)	Executable program file.
<i>model</i> (UNIX and Macintosh platforms)	A model build generates this file unless you explicitly specify that the code generator produce code only. The build generates the executable in the current folder (not the build folder) under control of the make utility of your development system.
<i>model.h</i>	<p>Defines model data structures and a public interface to the model entry points and data structures. Provides an interface to the real-time model data structure (<i>model_rtM</i>) via access macros.</p> <p>Subsystem .c or .cpp files in the model include <i>model.h</i>. This file includes:</p> <ul style="list-style-type: none"> • Exported Simulink data symbols • Model data structures, including <i>rtM</i> • Model entry-point functions <p>For more information, see “model.h” on page 24-39.</p>
<i>model.mk</i>	<p>Generated makefile that controls compiling and linking the generated code into the final binary file by the make utility of your development system.</p> <p>If you set the MAKEFLAGS environment variable, do not select options with this variable that conflict with the current make utility used by the build process.</p>
<i>model.rtw</i>	<p>Represents the compiled model.</p> <p>By default, the build process deletes this ASCII file when the build process is complete. You can choose to retain the file for inspection.</p>

File	Description
<i>model_capi.h</i> <i>model_capi.c</i>	<p>Contain data structures that describe the model signals, states, and parameters without using external mode.</p> <p>For more information, see “Exchange Data Between Generated and External Code Using C API” on page 34-2.</p>
<i>model_data.c</i>	<p>Contains (if conditionally generated) declarations for the parameters data structure and the constant block I/O data structure, and zero representations for structure data types that the model uses.</p> <p>A model build generates this file when the model uses these data structures. The extern declarations for structures appear in <i>model.h</i>. When present, this file contains:</p> <ul style="list-style-type: none"> • Constant block I/O parameters • Include files <i>model.h</i> and <i>model_private.h</i> • Definitions for the zero representations for user-defined structure data types that the model uses • Constant parameters
<i>model_dt.h</i>	<p>Declares structures that contain data type and data type transition information for generated model data structures for supporting external mode.</p>
<i>model_private.h</i>	<p>Contains local define constants and local data for the model and subsystems.</p> <p>The generated source files from the model build include this file. When you interface external code with generated code from a model, include <i>model_private.h</i>. The file contains:</p> <ul style="list-style-type: none"> • Imported Simulink data symbols • Stateflow entry points • Simulink Coder details (various macros, enums, and so forth, that are private to the code) <p>For more information, see “Manage Build Process File Dependencies” on page 24-43.</p>
<i>model_reference_types.h</i>	<p>Contains type definitions for timing bridges.</p> <p>A model build generates this file for a referenced model or a model containing model reference blocks.</p>
<i>model_targ_data_map.m</i>	<p>Contains MATLAB language commands that external mode uses to initialize the external mode connection.</p>
<i>model_types.h</i>	<p>Provides forward declarations for the real-time model data structure and the parameters data structure.</p> <p>The generated header files from the model build include this file. Function declarations of reusable functions can use these structures.</p>

File	Description
multiword_types.h	<p>Contains type definitions for multiple-word wide data types and their word-size chunks. If your code uses multiword data types, include this header file.</p> <p>A model build generates this file when one or more of these conditions apply:</p> <ul style="list-style-type: none"> Your model uses multiword data types. Your model configuration enables MAT-file logging. Your model configuration enables Code Generation > Interface > External mode.
rtGetInf.c rtGetInf.h rtGetNaN.c rtGetNaN.h rt_nonfinite.c rt_nonfinite.h	<p>Declares and initializes global nonfinite values for inf, minus inf, and nan. Provides nonfinite comparison functions.</p> <p>A model build generates these files when one or more of these conditions apply:</p> <ul style="list-style-type: none"> The model contains S-functions. The generated code from the model requires nonfinite numbers. Your model configuration enables MAT-file logging. Your model configuration selects <code>grt.tlc</code> as the System target file and enables the Classic call interface.
rtmodel.h	<p>Contains <code>#include</code> directives required by static main program modules such as <code>rt_main.c</code>.</p> <p>The build process does not create these modules at code generation time. The modules include <code>rtmodel.h</code> to access model-specific data structures and entry points. If you create your own main program module, make sure to include <code>rtmodel.h</code>.</p>
rtwtypes.h	<p>Provides type definitions, <code>#define</code> statements, and enumerations.</p> <p>For ERT system target files, the code generator creates <code>rtwtypes.h</code> only if one of these conditions applies:</p> <ul style="list-style-type: none"> Data type replacement (Embedded Coder) is set to Use <code>coder typedefs</code>. Data type replacement (Embedded Coder) is set to Use <code>C data types with fixed-width integers</code> and Coder typedefs compatibility (Embedded Coder) is selected. If you use legacy custom code or static source files (under <code>matlabroot</code>) that require Simulink Coder data type definitions, the settings force the generation of <code>rtwtypes.h</code>. <p>For ERT-based system target files that do not generate a GRT interface and do not have noninlined S-functions, <code>rtwtypes.h</code> does not include <code>simstruc_types.h</code>.</p> <p>For more information, see “<code>rtwtypes.h</code>” on page 24-40 and “Manage Build Process File Dependencies” on page 24-43.</p>

File	Description
<code>rtw_proj.tmw</code>	Marker files.
<code>sl_proj.tmw</code>	The build process generates these files to help the make utility determine when to recompile and link the generated code.
<code>rt_defines.h</code>	Contains type definitions for special mathematical constants (such as π and e) and defines the <code>UNUSED_PARAMETER</code> macro. A model build generates this file when the generated code requires a mathematical constant definition or when the function body does not access a required model function argument.
<code>rt_sfcn_helper.h</code>	Provide functions that the noninlined S-functions use in a model.
<code>rt_sfcn_helper.c</code>	The noninlined S-functions use functions <code>rt_CallSys</code> , <code>rt_enableSys</code> , and <code>rt_DisableSys</code> to call downstream function-call subsystems.
<code>subsystem.c</code>	Contains C source code for each noninlined nonvirtual subsystem or copy the code when the subsystem is configured to place code in a separate file.
<code>subsystem.h</code>	Contains exported symbols for noninlined nonvirtual subsystems.

model.bat

This file contains Windows batch file commands that set the compiler environment and invoke the make utility.

If you are using the toolchain approach for the build process, you also can use this batch file to extract information from the generated makefile, `model.mk`. The information includes macro definitions and values that appear in the makefile, such as `CFLAGS` (C compiler flags) and `CPP_FLAGS` (C++ compiler flags). With the folder containing `model.bat` selected as the current working folder, in the Command Window, type:

```
>> system('model.bat info')
```

On UNIX and Macintosh platforms, the code generator does not create the `model.bat` file. To extract information for toolchain approach builds from the generated makefile on these systems, in the Command Window, type:

```
>> system('gmake -f model.mk info')
```

model.h

The header file `model.h` declares model data structures and a public interface to the model entry points and data structures. This header file also provides an interface to the real-time model data structure (`model_M`) by using access macros. If your code interfaces to model functions or model data structures, include `model.h`:

- Exported global signals

```
extern int32_T INPUT;    /* '<Root>/In' */
```
- Global structure definitions

```
/* Block parameters (auto storage) */
extern Parameters_mymodel mymodel_P;
```

- Real-time model (RTM) macro definitions

```
#ifndef rtmGetSampleTime
# define rtmGetSampleTime(rtm, idx)
((rtm)->Timing.sampleTimes[idx])
#endif
```

- Model entry point functions (ERT example)

```
extern void mymodel_initialize(void);
extern void mymodel_step(void);
extern void mymodel_terminate(void);
```

The `main.c` (or `.cpp`) file includes `model.h`. If the model build generates the `main.c` (or `.cpp`) file from a TLC script, the TLC source can include `model.h`.

```
#include "%<CompiledModel.Name>.h"
```

If `main.c` is a static source file, you can use the fixed header file name `rtmodel.h`. This file includes the `model.h` header file:

```
#include "model.h" /* If main.c is generated */
```

or

```
#include "rtmodel.h" /* If static main.c is used */
```

Other external source files can require to include `model.h` to interface to model data, for example exported global parameters or signals. The `model.h` file itself can have additional header dependencies due to requirements of generated code. See “System Header Files” on page 24-44 and “Code Generator Header Files” on page 24-46.

To reduce dependencies and reduce the number of included header files, see “Manage Build Process File Dependencies” on page 24-43.

rtwtypes.h

When the configuration parameter **Data type replacement** is set to `Use coder typedefs`, the generated code includes `rtwtypes.h`. The header file `rtwtypes.h` defines data types, structures, and macros required by the generated code.

Often, the generated code requires that integer operations overflow or underflow at specific values. For example, when the code expects a 16-bit integer, the code does not accept an 8-bit or a 32-bit integer type. The C language does not set a standard for the number of bits in types such as `char` and `int`.

Note Before the creation of the C99 language standard, there was no universally accepted data type for sized integers. The C99 standard provides the header files `stdint.h` and `stdbool.h`. The `stdint.h` file contains sized-integer implementations that were previously unavailable.

To accommodate this feature of the C language, the generated code uses sized integer types, such as `int8_T` and `uint32_T`, which are not standard C types. In `rtwtypes.h`, the generated code maps these sized-integer types to the corresponding C keyword base type using information in the **Hardware Implementation** pane of the Configuration Parameters dialog box.

The code generator produces an optimized version of `rtwtypes.h` for ERT-based system target files when these conditions apply:

- **Configuration Parameters > Code Generation > Interface > Advanced parameters > Classic call interface** is not selected.
- The model does not contain noninlined S-functions.

The optimized `rtwtypes.h` file contains definitions that are specific to the hardware configuration of the model.

The location of `rtwtypes.h` depends on whether the build process uses the *shared utilities* location. If it uses a shared location, the code generator places `rtwtypes.h` in `slprj/target/_sharedutils`. Otherwise, it places `rtwtypes.h` in the build folder (`model_target_rtw`). See “Specify Generated Code Interfaces” on page 21-5 and “Control Placement of `rtwtypes.h` for Shared Utility Code” on page 10-54.

Source files include the `rtwtypes.h` header file when the source files use code generator type names or other code generator definitions. For example, files that declare variables by using a code generator data type:

```
uint32_T myvar
```

In some cases, an S-function, custom source file is compiled, first, by the `mex` command, and second, by the code generation build. During `mex` compilation, `rtwtypes.h` is typically unavailable. To obtain the definitions normally supplied by `rtwtypes.h`, the custom source file can conditionally (through the `MATLAB_MEX_FILE` macro definition) include `tmwtypes.h`:

```
#ifdef MATLAB_MEX_FILE
#include "tmwtypes.h"
#else
#include "rtwtypes.h"
#endif
```

A source file for the code generator `main.c` (or `.cpp`) file includes `rtwtypes.h` without preprocessor checks.

```
#include "rtwtypes.h"
```

Custom source files that the Target Language Compiler generates can also emit the `include` statements into their generated file.

If the **Data type replacement** configuration parameter is set to Use C data types with fixed-width integers, the code generator does not create `rtwtypes.h` because the generated code does not require definitions from `rtwtypes.h`. If the generated code uses static files under `matlabroot` that require Simulink Coder data type definitions, the software generates a simplified version of `rtwtypes.h`. If you have custom code that includes `rtwtypes.h`, you can force the generation of the simplified version of `rtwtypes.h` by selecting the **Coder typedefs compatibility** check box.

Simplified `rtwtypes.h`

```
/*
 * File: rtwtypes.h
 *
 * This version of rtwtypes.h is generated for
 * compatibility with custom source code or static
 * source files that are located under matlabroot.
 * Automatically generated code does not have to
 * include this file.
```

```
*/  
  
#ifndef RTWTYPES_H  
#define RTWTYPES_H  
  
#include <stdint.h>  
#include <stdbool.h>  
  
typedef int8_t int8_T;  
typedef uint8_t uint8_T;  
typedef int16_t int16_T;  
typedef uint16_t uint16_T;  
typedef int32_t int32_T;  
typedef uint32_t uint32_T;  
typedef bool boolean_T;  
typedef char char_T;  
typedef char byte_T;  
typedef int int_T;  
typedef unsigned char uchar_T;  
typedef unsigned int uint_T;  
typedef unsigned long ulong_T;  
  
#define MIN_int8_T INT8_MIN  
#define MAX_int8_T INT8_MAX  
#define MAX_uint8_T UINT8_MAX  
#define MIN_int16_T INT16_MIN  
#define MAX_int16_T INT16_MAX  
#define MAX_uint16_T UINT16_MAX  
#define MIN_int32_T INT32_MIN  
#define MAX_int32_T INT32_MAX  
#define MAX_uint32_T UINT32_MAX  
  
#endif /* RTWTYPES_H */
```

See Also

More About

- “Manage Build Process Folders” on page 24-30
- “Manage Build Process File Dependencies” on page 24-43
- “Add Build Process Dependencies” on page 24-52
- “Build Process Support for File and Folder Names” on page 24-57
- “Build Process Workflow for Real-Time Systems” on page 31-39
- “Generate Code and Simulate Models in a Project” on page 24-97
- “Generate Code and Simulate Models Programmatically with Project” on page 24-100

Manage Build Process File Dependencies

An important control of the size of generated code is managing the number and size of included files (dependencies). To reduce the number of system header files and generated header files that generated code requires, it is helpful to understand the dependencies that the build process generates and the conditions that lead to dependencies.

The dependency relationships among generated source and header files appear in the figure. Arrows coming from a file point to files it includes. Other dependencies exist, for example, on Simulink header files `tmwtypes.h` and `simstruc_types.h`, plus C or C++ library files. The figure maps inclusion relations between only those files that are generated in the build folder. These files can reference utility and model reference code located in a code generation folder. For more information about the folders and files that the build process creates, see “Manage Build Process Folders” on page 24-30 and “Manage Build Process Files” on page 24-35.

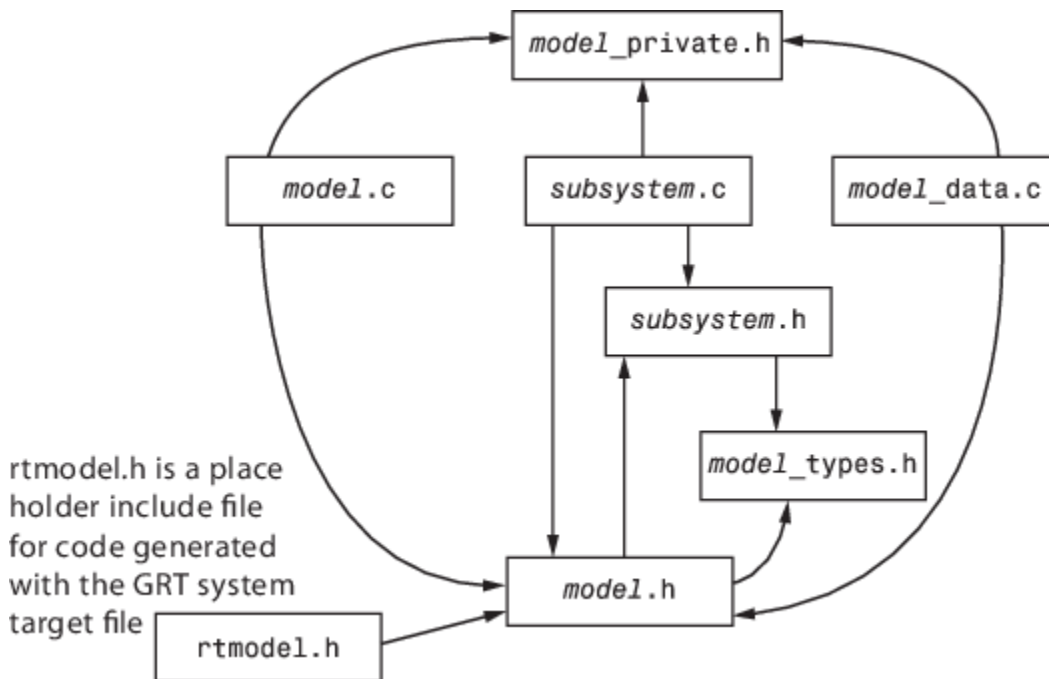
The two tables identify the conditions that control creation of dependency files for GRT and ERT targets. To manage build-related dependencies, consider how these conditions apply to your model and code generation process. Then, configure model parameters and code generation options to manage build process file dependencies.

Due to differences in file packaging options for code generated with ERT-based system target files, the file dependencies differ slightly from file packaging for code generated with GRT-based system target files. See “Manage File Packaging of Generated Code Modules” (Embedded Coder).

The parent system header files (`model.h`) include child subsystem header files (`subsystem.h`). In more layered models, subsystems similarly include their children's header files in the model hierarchy. As a consequence, subsystems are able to view recursively into their descendant subsystems and view into the root system because every `subsystem.c` or `subsystem.cpp` includes `model.h` and `model_private.h`.

In the figure, files `model.h`, `model_private.h`, and `subsystem.h` depend on the header file `rtwtypes.h`. For ERT system target files, the dependency on `rtwtypes.h` exists only if **Data type replacement** is set to Use `coder typedefs`.

If you use system target files that are not based on the ERT system target file, the source files that you generate can have additional dependencies on `tmwtypes.h` and `simstruc_types.h`.



System Header Files

The system header files make function declarations, type definitions, and macro definitions available to the legacy or external code. Some code generation scenarios require including header files that are specific to the code generator product.

The code generator includes some system header files for broadly defined cases. For example, generated code includes `<stddef.h>` when the model contains a utility function that requires this header file. This approach helps identify header file dependencies:

- 1 Set model configuration parameter **Shared code placement** to 'Shared location' and build the model. The code generator places the utility functions in `__sharedutils` folder.
- 2 Use a find-in-file utility (for example, `grep` utility) to search the `.c` and `.h` files in the `__sharedutils` folder for `#include`. The search results list the utilities with header file dependencies.
- 3 Use this information to identify utilities to remove from the model and reduce header file dependencies in the generated code.

For more information, see “Generate Shared Utility Code for Fixed-Point Functions” on page 10-60.

System Header File	Description and Inclusion Conditions for GRT or ERT System Target Files
<math.h>	<p>Defines math constants</p> <p>GRT—Generated code does not include this file.</p> <p>ERT—Generated code includes this file when the code honors your model configuration for solver Stop time and either:</p> <ul style="list-style-type: none"> • Your model configuration enables parameter MAT-file logging. See MAT-file logging. • Your model configuration enables parameter External mode.
<float.h>	<p>Provides floating-point math functions</p> <p>GRT—Generated code includes this file when your model contains a floating-point math function.</p> <p>ERT—Generated code includes this file when your model contains a floating-point math function, unless a code replacement library entry overrides the function. For more information, see “Choose a Code Replacement Library” on page 26-6.</p>
<stddef.h>	<p>Defines NULL</p> <p>GRT and ERT—Generated code includes this file when your model contains a utility function that requires this file.</p>
<stdio.h>	<p>Provides file I/O functions</p> <p>GRT—Generated code includes this file when your model includes a To File block.</p> <p>ERT—Generated code includes this file when either:</p> <ul style="list-style-type: none"> • Your model includes a To File block. • Your model configuration enables parameter MAT-file logging. See MAT-file logging.
<stdlib.h>	<p>Provides utility functions such as the integer versions of <code>div()</code> and <code>abs()</code></p> <p>GRT—Generated code includes this file when either:</p> <ul style="list-style-type: none"> • Your model includes a Stateflow chart. • Your model includes a math function block configured for <code>mod()</code> or <code>rem()</code>, which generate calls to <code>div()</code>. <p>ERT—Generated code includes this file when either:</p> <ul style="list-style-type: none"> • Your model includes a Stateflow chart, and you select model configuration parameter Support: floating-point numbers. • Your model includes a math function block configured for <code>mod()</code> or <code>rem()</code>, which generate calls to <code>div()</code>.

System Header File	Description and Inclusion Conditions for GRT or ERT System Target Files
<string.h>	<p>Provides memory functions such as <code>memset()</code> and <code>memcpy()</code></p> <p>GRT—Generated code includes this file when your model initialization code calls <code>memset()</code>.</p> <p>ERT—Generated code includes this file when a block or model initialization code calls <code>memcpy()</code> or <code>memset()</code>.</p> <p>For a list of relevant blocks, in the Command Window, type:</p> <pre>showblockdatatypetable</pre> <p>Look for blocks with the N2 note. To omit calls to <code>memset()</code> from model initialization code, select model configuration parameters Remove root level I/O zero initialization and Remove internal data zero initialization.</p>

Code Generator Header Files

Dependencies in the table for generated header files apply to the system target files `grt.tlc` and `ert.tlc`. System target files derived from these base system target files can have additional header dependencies. Code generation for blocks from blocksets, embedded targets, and custom S-functions can introduce additional header dependencies.

Header File	Description and Inclusion Conditions for GRT or ERT System Target Files
<code>builtin_typeid_types.h</code>	<p>Defines an enumerated type corresponding to built-in data types</p> <p>GRT and ERT—Generated code includes this file when one or more of these conditions apply:</p> <ul style="list-style-type: none"> Your model contains a Stateflow chart that uses messages. Your model configuration enables: MAT-file logging. See MAT-file logging. Your model configuration selects C API options at Code Generation > Interface.
<code>dt_info.h</code>	<p>Defines data structures for external mode</p> <p>GRT and ERT—Generated code includes this file when your model configuration enables external mode.</p>
<code>ext_work.h</code>	<p>Defines external mode functions</p> <p>GRT and ERT—Generated code includes this file when your model configuration enables external mode.</p>

Header File	Description and Inclusion Conditions for GRT or ERT System Target Files
fixedpoint.h	<p>Provides fixed-point support for noninlined S-functions</p> <p>GRT—Generated code includes this file.</p> <p>ERT—Generated code includes this file when either:</p> <ul style="list-style-type: none"> • Your model uses noninlined S-functions. • Your model configuration selects parameter Classic call interface.
model_reference_types.h	<p>Contains type definitions for timing bridges</p> <p>GRT and ERT—Generated code includes this file when building a reference model or building a model that contains model blocks.</p>
model_types.h	<p>Defines model-specific data types</p> <p>GRT and ERT—Generated code includes this file.</p>
multiword_types.h	<p>Contains type definitions for multiword-wide data types and their word-size chunks</p> <p>GRT and ERT—Generated code includes this file when one or more of these conditions apply:</p> <ul style="list-style-type: none"> • Your model uses multiword data types. • Your model configuration selects parameter MAT-file logging. See MAT-file logging. • Your model configuration selects External mode. <p>For a model that uses multiword data types, the code generator overwrites the file if the data types are greater in length than those of the model for which code was last generated. To avoid overwriting this file, set:</p> <ul style="list-style-type: none"> • <code>MultiwordTypeDef</code> to 'User defined' • <code>MultiwordLength</code> to the biggest length that is required by your models.

Header File	Description and Inclusion Conditions for GRT or ERT System Target Files
rtGetInf.h rtGetNaN.h rt_nonfinite.h	<p>Support nonfinite numbers</p> <p>GRT—Generated code includes this file when one or more of these conditions apply:</p> <ul style="list-style-type: none"> • Your model contains S-functions. • The generated code requires nonfinite numbers. • Your model configuration selects parameter MAT-file logging. See MAT-file logging. • Your model configuration selects parameter Classic call interface. <p>ERT—Generated code includes this file when one or more of these conditions apply:</p> <ul style="list-style-type: none"> • Your model contains S-functions. • The generated code requires nonfinite numbers. • Your model configuration selects parameter MAT-file logging. See MAT-file logging.
rt_defines.h	<p>Contains type definitions for special mathematical constants (such as π and e) and defines the <code>UNUSED_PARAMETER</code> macro</p> <p>GRT and ERT—Generated code includes this file when either:</p> <ul style="list-style-type: none"> • The generated code requires a mathematical constant definition. • The function body does not access a required model function argument.
rt_logging.h	<p>Supports MAT-file logging and includes:</p> <p>rtwtypes.h builtin_typeid_types.h multiword_types.h rt_mxclassid.h rtw_matlogging.h</p> <p>GRT—Generated code includes this file.</p> <p>ERT—Generated code includes this file when you model configuration selects parameter MAT-file logging. See MAT-file logging.</p>
rt_mxclassid.h	<p>Defines mxArray class ID enumerations</p> <p>GRT and ERT—Generated code includes this file when the code includes <code>rt_logging.c</code>.</p>

Header File	Description and Inclusion Conditions for GRT or ERT System Target Files
rtw_continuous.h	<p>Supports continuous time</p> <p>GRT—Generated code includes this file when the code includes <code>simstruc_types.h</code>.</p> <p>ERT—Generated code includes this file when your model configuration selects parameter Support: continuous time and when the code does not already include <code>simstruc.h</code>.</p>
rtw_extmode.h	<p>Supports external mode</p> <p>GRT—Generated code includes this file when the code includes <code>simstruc_types.h</code>.</p> <p>ERT—Generated code includes this file when your model configuration selects external mode and when the code does not already include <code>simstruc.h</code>.</p>
rtw_matlogging.h	<p>Supports MAT-file logging</p> <p>GRT—Generated code includes this file when the code includes <code>simstruc_types.h</code> and <code>rt_logging.h</code>.</p> <p>ERT—Generated code includes this file when the code includes <code>rt_logging.h</code>.</p>
rtw_solver.h	<p>Supports continuous states</p> <p>GRT—Generated code includes this file when the code includes <code>simstruc_types.h</code>.</p> <p>ERT—Generated code includes this file when your model configuration selects parameter Support: continuous time and when the code does not already include <code>simstruc.h</code>.</p>

Header File	Description and Inclusion Conditions for GRT or ERT System Target Files
rtwtypes.h	<p>Defines code generator data types</p> <p>GRT—Generated code includes this file. Uses a verbose version of the file, which includes <code>tmwtypes.h</code>.</p> <p>ERT—Generated code includes this file if one of these conditions applies:</p> <ul style="list-style-type: none"> • Data type replacement is set to <code>Use coder typedefs</code>. • Data type replacement is set to <code>Use C data types with fixed-width integers</code> and Coder typedefs compatibility is selected. <p>See “<code>rtwtypes.h</code>” on page 24-40.</p> <p>The code generator overwrites the previously generated <code>rtwtypes.h</code> when you enable (previously disabled) support for:</p> <ul style="list-style-type: none"> • Complex numbers (<code>SupportComplex</code> set to 'on'). • Noninlined S-functions (<code>SupportNonInlinedSFcns</code> set to 'on') <p>To avoid rewriting <code>rtwtypes.h</code>, you can:</p> <ul style="list-style-type: none"> • Specify support for complex datatypes for your models even if the models do not currently use complex data types. • Disable support for noninlined S-functions. In this case, the use of a noninlined S-function produces an error. To avoid the error, convert the S-function to an inlined S-function. For more information, see “Inlining S-Functions” (Embedded Coder).
simstruc.h	<p>Supports calling noninlined S-functions that use the <code>Simstruct</code> definition; also includes:</p> <pre>limits.h string.h tmwtypes.h simstruc_types.h</pre> <p>GRT—Generated code includes this file.</p> <p>ERT—Generated code includes this file when either:</p> <ul style="list-style-type: none"> • Your model uses noninlined S-functions. • Your model configuration selects parameter Classic call interface.

Header File	Description and Inclusion Conditions for GRT or ERT System Target Files
simstruc_types.h	<p>Provides definitions that the generated code uses and includes the header files:</p> <pre>rtw_matlogging.h rtw_extmode.h rtw_continuous.h rtw_solver.h sysran_types.h</pre> <p>GRT—Generated code includes this file when the code includes <code>rtwtypes.h</code>.</p> <p>ERT—Generated code does not include this file. For ERT, <code>rtwtypes.h</code> contains definitions, and <code>model.h</code> contains header files.</p>
sysran_types.h	<p>Supports external mode</p> <p>GRT—Generated code includes this file when the code includes <code>simstruc_types.h</code>.</p> <p>ERT—Generated code includes this file when your model configuration selects external mode and when the code does not already include <code>simstruc.h</code>.</p>
zero_crossing_types.h	<p>Contains zero-crossing definitions for models with triggered subsystems where the trigger is rising, falling, or either. File is generated only if required by the model as determined by the data type of the trigger signal.</p> <p>GRT—Generated code does not include this file for GRT code generation targets.</p> <p>ERT—Generated code includes this file when a model has a conditionally executed subsystem where a trigger uses zero crossing detection.</p> <p>If generated, the content of <code>zero_crossing_types.h</code> is always the same.</p>

See Also

More About

- “Manage Build Process Folders” on page 24-30
- “Manage Build Process Files” on page 24-35
- “Add Build Process Dependencies” on page 24-52
- “Build Process Support for File and Folder Names” on page 24-57
- “Build Process Workflow for Real-Time Systems” on page 31-39

Add Build Process Dependencies

When you specify a system target file for code generation, the code generator can build a standalone executable program that can run on the development computer. To build the executable program, the code generator uses the selected compiler and the makefile generated by the toolchain or template makefile (TMF) build process approach. Part of the makefile generation process is to add source file, header file, and library file information (the dependencies) in the generated makefile for a compilation. Or, for a specific application, you can add the generated files and file dependencies through a configuration management system.

The generated code for a model consists of a small set of files. (See “Manage Build Process Files” on page 24-35.) These files have dependencies on other files, which occur due to:

- Header file inclusions
- Macro declarations
- Function calls
- Variable declarations

The model or external code introduces dependencies for various reasons:

- Blocks in a model generate code that makes function calls. These calls can occur in several forms:
 - Included source files (not generated) declare the called functions. In cases such as a blockset, manage these source file dependencies by compiling them into a library file.
 - The generated code makes calls to functions in the run-time library provided by the compiler.
 - Some function dependencies also are generated files, which are referred to as shared utilities. Some examples are fixed-point utilities and non-finite support functions. These dependencies are referred to as shared utilities. The generated functions can appear in files in the build folder for standalone models or in the `_sharedutils` folder under the `slprj` folder for builds that involve model reference.
- Models with continuous time require solver source code files.
- Code generator options such as external mode, C API, and MAT-file logging.
- External code specifies dependencies.

File Dependency Information for the Build Process

The code generator provides several mechanisms to input file dependency information into the build process. The mechanisms depend on whether your dependencies are block-based or are model- or system target file-based.

For block dependencies, consider using:

- S-functions and blocksets
 - Add folders that contain S-function MEX-files that the model uses to the header include path.
 - Create makefile rules for these folders to allow for finding source code.
 - Specify additional source file names with the S-Function block parameter `SFunctionModules`.
 - Specify additional dependencies with the `rtwmakecfg.m` mechanism. See “Use `rtwmakecfg.m` API to Customize Generated Makefiles” on page 44-17.

For more information on applying these approaches to legacy or external code integration, see “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6.

- S-Function Builder block, which provides its own UI for specifying dependency information

For model- or system target file-based dependencies, such as external header files, consider using:

- The **Code Generation > Custom Code** pane in the Configuration Parameters dialog box. You can specify additional libraries, source files, and include folders.
- TLC functions `LibAddToCommonIncludes()` and `LibAddToModelSources()`. You can specify dependencies during the TLC phase. See “`LibAddToCommonIncludes(incFileName)`” and “`LibAddSourceFileCustomSection(file, builtInSection, newSection)`”. The Embedded Coder product also provides a TLC-based customization template for generating additional source files.

Generated Makefile Dependencies

For toolchain approach or template makefile (TMF) approach build processes, the code generator generates a makefile. For TMFs, the generated makefile provides token expansion in which the build process expands different tokens in the makefile to include the additional dependency information. The resulting makefile contains the complete dependency information. See “Customize Template Makefiles” on page 46-44.

The generated makefile contains:

- Names of the source file dependencies
- Folders where source files are located
- Location of the header files
- Precompiled library dependencies
- Libraries that the make utility compiles and creates

A property of make utilities is that you do not have to specify the specific location for a given source C or C++ file. If a rule exists for that folder and the source file name is a prerequisite in the makefile, the make utility can find the source file and compile it. The C or C++ compiler (preprocessor) does not require absolute paths to the headers. The compiler finds header file with the name of the header file by using an `#include` directive and an include path. The generated C or C++ source code depends on this standard compiler capability.

Libraries are created and linked against, but occlude the specific functions that the program calls.

These properties can make it difficult to determine the minimum list of file dependencies manually. You can use the makefile as a starting point to determine the dependencies in the generated code.

Another approach to determining the dependencies is using linker information, such as a linker map file, to determine the symbol dependencies. The map file provides the location of code generator and blockset source and header files to help in locating the dependencies.

Code Generator Static File Dependencies

Several locations in the MATLAB folder tree contain static file dependencies specific to the code generator:

- `matlabroot/rtw/c/src` (open)

This folder has subfolders and contains additional files must be compiled. Examples include solver functions (for continuous time support), external mode support files, C API support files, and S-

function support files. Include source files in this folder into the build process with the SRC variables of the makefile.

- Header files in the folder `matlabroot/extern/include`
- Header files in the folder `matlabroot/simulink/include`

These folders contain additional header file dependencies such as `tmwtypes.h`, `simstruc_types.h`, and `simstruc.h`.

Note For ERT-based system target files, you can avoid several header dependencies. ERT-based system target files generate the minimum set of type definitions, macros, and so on, in the file `rtwtypes.h`.

Blockset Static File Dependencies

Blockset products with S-function code apply the `rtwmakecfg.m` mechanism to provide the code generator with dependency information. The `rtwmakecfg.m` file from the blockset contains the list of include path and source path dependencies for the blockset. Typically, blocksets create a library from the source files to which the generated model code can link. The libraries are created and identified when you use the `rtwmakecfg.m` mechanism.

To locate the `rtwmakecfg.m` files for blocksets in your MATLAB installed tree, use the following command:

```
>> which -all rtwmakecfg.m
```

If the model that you are compiling uses one or more of the blocksets listed by the `which` command, you can determine folder and file dependency information from the respective `rtwmakecfg.m` file.

Folder Dependency Information for the Build Process

You can add `#include` statements to generated code. Such references can come from several sources, including TLC scripts for inlining S-functions, custom storage classes, bus objects, and data type objects. The included files consist of header files for external code or other customizations. You can specify compiler include paths with the `-I` compiler option. The build process uses the specified paths to search for included header files.

Usage scenarios for the generated code include, but are not limited to, the following:

- A custom build process compiles generated code that requires an environment-specific set of `#include` statements.

In this scenario, the build process invokes the code generator when you select model configuration parameter **Generate code only**. Consider using fully qualified paths, relative paths, or just the header file names in the `#include` statements. Use include paths.

- The build process compiles the generated code.

In this case, you can specify compiler include paths (`-I`) for the build process in several ways:

- Specify additional include paths on the **Code Generation > Custom Code** pane in the Configuration Parameters dialog box. The code generator propagates the include paths into the generated makefile.

- The `rtwmakecfg.m` mechanism allows S-functions to introduce additional include paths into the build process. The code generator propagates the include paths into the generated makefile.
- When building a model that uses a custom system target file and is makefile-based, you can directly add the include paths into the template makefile that the system target file uses.
- Use the `make` command to specify a `USER_INCLUDES` make variable that defines a folder in which the build process searches for included files. For example:

```
make_rtw USER_INCLUDES=-Id:\work\feature1
```

The build process passes the custom includes to the command-line invocation of the `make` utility, which adds them to the overall flags passed to the compiler.

Use #include Statements and Include Paths

Consider the following approaches for using `#include` statements and include paths with the build process to generate code that remains portable and minimizes compatibility problems with future versions.

Assume that additional header files are:

```
c:\work\feature1\foo.h
c:\work\feature2\bar.h
```

- An approach is to include in the `#include` statements only the file name, such as:

```
#include "foo.h"
#include "bar.h"
```

Then, the include path passed to the compiler contains folders in which the headers files exist:

```
cc -Ic:\work\feature1 -Ic:\work\feature2 ...
```

- Another approach is to use relative paths in `#include` statements and provide an anchor folder for these relative paths using an include path, for example:

```
#include "feature1\foo.h"
#include "feature2\bar.h"
```

Then, specify the anchor folder (for example `\work`) to the compiler:

```
cc -Ic:\work ...
```

Avoid These Folder Dependencies

When using the build process, avoid dependencies on folders in the build process “Code generation folder”, such as the `model_ert_rtw` folder or the `slprj` folder. Do not use paths in `#include` statements that are relative to the location of the generated source file. For example, if your MATLAB code generation folder is `c:\work`, the build process generates the `model.c` source file into a subfolder such as:

```
c:\work\model_ert_rtw\model.c
```

The `model.c` file has `#include` statements of the form:

```
#include "..\feature1\foo.h"
#include "..\feature2\bar.h"
```

It is preferable to use one of the other suggested approaches because the relative path creates a dependency on the code generator folder structure.

See Also

More About

- “Manage Build Process Folders” on page 24-30
- “Manage Build Process Files” on page 24-35
- “Manage Build Process File Dependencies” on page 24-43
- “Build Process Support for File and Folder Names” on page 24-57
- “Build Process Workflow for Real-Time Systems” on page 31-39

Build Process Support for File and Folder Names

Filename with Spaces

For the build process that uses `ToolchainInfo` objects, only these toolchains support the use of filenames containing spaces:

- GNU `gcc/g++` | `gmake` (64-bit Linux) on Linux®
- MinGW64 | `gmake` (64-bit Windows) on Windows
- Xcode with Clang | `gmake` (64-bit Mac) on Mac

The build process that uses template makefiles does not support the use of filenames containing spaces.

Folder Names with Spaces

On a Windows system, the code generator maps a drive corresponding to the MATLAB installation folder for either of these conditions:

- The `matlabroot` folder is a UNC location.
- The path the `matlabroot` folder contains spaces, and the system has no alternative name support.

These folder paths can contain spaces:

- The path to your MATLAB installation folder (`matlabroot`). For example, `C:\Program Files\MATLAB\R2015b`
- The path to the current working folder where you start the build (`pwd`). For example, `C:\Users\username\Documents\My Work`.
- The path to the installation folder for a compiler that the build process uses.

If your work environment includes one or more of the preceding scenarios, use the following support mechanisms for the build process:

- If you are using the toolchain approach to build generated code, the system support for spaces in folder names influences toolchain operation:
 - For Linux systems and Windows systems with 8.3 name creation enabled, the toolchain manages spaces in folder names by using alternative names from the operating system. The toolchain uses the `TransformPathsWithSpaces` attribute to manage these names.

```
addAttribute(toolchainObject, 'TransformPathsWithSpaces', true);
```

The security permissions of drives and folders can determine whether the toolchain transforms the path. For example, if the path contains a folder with a security configuration that forbids 8.3 path transformations, the toolchain does not transform the path and the build process produces a warning.

- For Windows systems with 8.3 name creation disabled, the toolchain manages spaces in folder names by mapping a network drive using a batch file (.bat). This operation requires adding the `RequiresBatchFile` attribute to the toolchain definition.

```
addAttribute(toolchainObject, 'RequiresBatchFile', true);
```

When developing a toolchain for a Windows system, set both attributes. For more information about the toolchain attributes, see `addAttribute`.

- If you are using the template makefile approach to build generated code, the template makefile (.tmf) requires code to manage spaces in folder names. When the alternative folder names (Windows short names) differ from the file system folder names (Windows long names), add this code to the makefile.

```
ALT_MATLAB_ROOT      = |>ALT_MATLAB_ROOT<|
ALT_MATLAB_BIN       = |>ALT_MATLAB_BIN<|
!if "$(MATLAB_ROOT)" != "$(ALT_MATLAB_ROOT)"
MATLAB_ROOT = $(ALT_MATLAB_ROOT)
!endif
!if "$(MATLAB_BIN)" != "$(ALT_MATLAB_BIN)"
MATLAB_BIN = $(ALT_MATLAB_BIN)
!endif
```

When the values of the location tokens are not equal, this code replaces `MATLAB_ROOT` with `ALT_MATLAB_ROOT`. The replacement indicates that the path to your MATLAB installation folder includes spaces. This code applies the same type of replacement for `MATLAB_BIN` with `ALT_MATLAB_BIN`. The preceding code is specific to `nmake`. For platform-specific examples, see the supplied template makefiles.

With either build approach, when there is an issue with support for creation of alternate names (short names), build errors can occur on Windows. If a build generates an error message similar to the following message, see “Troubleshooting Errors When Folder Names Have Spaces” on page 24-60.

```
NMAKE : fatal error U1073: don't know how to make ' ...
```

When using operating system commands, such as `system` or `dos`, enclose paths that specify executable files or command parameters in double quotes (" "). For example:

```
system('dir "D:\Applications\Common Files"')
```

This table provides a summary of build folder support and limitations for Windows.

Build Process Folders	Approach for Paths with UNC or Spaces	Support for Windows
<p>matlabroot folder</p> <p>The matlabroot value is derived from the MATLAB installation location.</p>	<p>During a build, a UNC location such as: <code>\\networkdrive\matlab\R20xxb</code> could be remapped as: <code>T:\</code></p> <p>During a build on a Windows system with short filename (8.3) support (default for Windows using NTFS), the build process uses the Windows API <code>getShortPathName()</code> for the folder location.</p> <p>During a build on a Windows system without short filename (8.3) support (systems using ReFS or using NTFS with 8.3 support disabled), a location with spaces in the path such as: <code>C:\Program Files\MATLAB\R20xxb</code> could be remapped as: <code>T:\R20xxb</code></p>	<p>Build process folder support available independent of file system (NTFS or ReFS) or file system configuration for short filename support.</p> <p>Limitations:</p> <p>On systems that require drive mapping for the installation location, the build process requires that a drive letter is available for mapping.</p> <p>On systems without short filename (8.3) support (using ReFS or using NTFS with 8.3 support disabled), the final folder in the installation location cannot contain spaces. For example, a final folder name: <code>C:\Program Files\MATLAB\R20xxb sp1</code> is not supported.</p>
<p>Code generation folder</p> <p>Simulation cache folder</p>	<p>For UNC locations, build process temporarily maps a drive by using the shell commands <code>pushd</code> and <code>popd</code>.</p>	<p>Build process folder support is available independent of file system (NTFS or ReFS) or file system configuration for short path name support.</p>
<p>Custom code source file locations—among others, these locations include folders specified by:</p> <ul style="list-style-type: none"> • <code>rtwmakecfg.m</code> • Model configuration parameter <p>Additional build information</p> <ul style="list-style-type: none"> • Code replacement library 	<p>For paths with spaces, build process uses the Windows short path name (8.3) by using the Windows API: <code>getShortPathName()</code></p>	<p>Build process folder support depends on NTFS file system and requires Windows default support. Registry sets value of 2 or 0 for: <code>NtfsDisable8dot3NameCreation</code></p> <p>Limitations: Build process does not support spaces in the path to these folders for:</p> <ul style="list-style-type: none"> • NTFS file system with short path name support disabled • ReFS file system (this file system does not support short path names)

Troubleshooting Errors When Folder Names Have Spaces

On Windows, when there is an issue with support for creation of short filenames, build process errors can occur. When this issue affects a build, you see an error message similar to:

```
NMAKE : fatal error U1073: don't know how to make 'C:\Work\My'
```

This message can occur if a space in the folder name (C:\Work\My Models) prevents the build process from finding the model or a file to build. For descriptions of the build-related folders that are sensitive to a space in the folder name or path, see “Folder Names with Spaces” on page 24-57.

To avoid issues from folder names with spaces when Windows short filename support for filenames is disabled, do not use paths with spaces. For example, install third-party software to paths without spaces. Do not use paths with spaces for folders containing your models, source files, or libraries.

An issue can occur with builds that use folder names with spaces, because it is possible to disable Windows alternate name support. The build process uses this alternate name support on Windows systems. There are many terms for this file, folder, and path alternate name support:

- 8.3 name
- DOS path
- short filename (SFN, ShortFileName)
- long name alias
- Windows path alias

Verify the type of file system that the drive uses. In Windows Explorer, right-click the drive icon and select properties.

- If the file system is ReFS (Resilient File System), it is an issue. The ReFS does not provide short filename support. Except for the MATLAB installation folder, the build process does not support folder names with spaces for the ReFS file system. If your work environment requires short filename support for the build folder or for additional external code folders, do not use ReFS.
- If the file system is NTFS (New Technology File System), it is possible that the build error is related to a registry setting incompatibility. Continue with troubleshooting steps.

The error could stem from an issue with short filename support on a system using NTFS. Check the Windows registry setting that enables the creation of short names for files, folders, and paths.

- 1 Open the Windows command prompt, running as administrator. For example, from the Windows Start menu, type `cmd`, right-click the `cmd.exe` icon, and select `Run as administrator`.
- 2 Change to the `windows\system32` folder and query the `NtfsDisable8dot3NameCreation` status by typing:


```
> fsutil 8dot3name query
```
- 3 If the registry state of `NtfsDisable8dot3NameCreation` is not 0 (enable 8dot3 name creation for all volumes on the system), change the value to 0 by typing:


```
> fsutil 8dot3name set 0
```

For more information about enabling creation of short names. See [https://learn.microsoft.com/en-us/previous-versions/windows/it-pro/windows-server-2012-R2-and-2012/ff621566\(v=ws.11\)](https://learn.microsoft.com/en-us/previous-versions/windows/it-pro/windows-server-2012-R2-and-2012/ff621566(v=ws.11)).

Changing the registry setting enables creation of short names only for files and folders that are created after the change.

- 4 To create short names for files created while short name creation was disabled, at the Windows command line, use the `fsutil` utility.

To set the short name, the syntax is:

```
> fsutil file setshortname <FileName> <ShortName>
```

For example, to create the short name `PROGRA~1` for the long name `C:\Program Files`, type:

```
> fsutil file setshortname "C:\Program Files" PROGRA~1
```

The `C:\Program Files` folder name is in quotations because it has spaces.

- 5 To verify that the short name was created, use the `dir` command with `/x` option to show short names.

```
> dir C:\ /x
```

Folder Names with Special Characters

The build process might produce an error if a build-related folder path contains:

- Unicode characters that do not belong to the system locale. This limitation does not apply if the build process uses a Microsoft Visual C++ compiler.
- A Japanese (multibyte) character where the final byte is equal to the `5C` hexadecimal character. The make and compiler tools might incorrectly interpret the final byte as the `'\'` (backslash) character.

Very Long Folder Paths

For the MinGW® compiler, the build process produces an error when the command line length exceeds the Windows limit of 32,767 characters. If this error occurs, check the length of include paths. You can reduce the command line length by building the generated code in a code generation folder that has a shorter name

See Also

`addAttribute`

More About

- “Manage Build Process Folders” on page 24-30
- “Manage Build Process Files” on page 24-35
- “Manage Build Process File Dependencies” on page 24-43
- “Add Build Process Dependencies” on page 24-52
- “Build Process Workflow for Real-Time Systems” on page 31-39

External Websites

- <https://www.mathworks.com/matlabcentral/answers/95399-why-is-the-build-process-failing-with-error-code-nmake-fatal-error-u1073-don-t-know-how-to-make>
- [https://learn.microsoft.com/en-us/previous-versions/windows/it-pro/windows-2000-server/cc959352\(v=technet.10\)](https://learn.microsoft.com/en-us/previous-versions/windows/it-pro/windows-2000-server/cc959352(v=technet.10))

- [`https://learn.microsoft.com/en-us/previous-versions/windows/it-pro/windows-server-2012-R2-and-2012/ff621566\(v=ws.11\)`](https://learn.microsoft.com/en-us/previous-versions/windows/it-pro/windows-server-2012-R2-and-2012/ff621566(v=ws.11))

Code Generation of Matrices and Arrays

MATLAB stores matrix data and arrays (1-D, 2-D, ...) in column-major format as a vector. Simulink and the code generator can store array data in column-major or row-major format. For an array stored in column-major layout, the elements of the columns are contiguous in memory. In row-major layout, the elements of the rows are contiguous. Array layout is also referred to as order, format, and representation. The order in which elements are stored can be important for integration, usability, and performance. Certain algorithms perform better on data stored in a particular order.

Programming languages and environments assume a single array layout for all data. MATLAB and Fortran use column-major layout by default, whereas C and C++ use row-major layout. With Simulink Coder, you can generate C/C++ code that uses row-major layout or column-major layout.

Array Storage in Computer Memory

Computer memory stores data in terms of one-dimensional arrays. For example, when you declare a 3-by-3 matrix, the software stores this matrix as a one-dimensional array with nine elements. By default, MATLAB stores these elements with a column-major array layout. The elements of each column are contiguous in memory.

Consider the matrix A:

```
A =
    1    2    3
    4    5    6
    7    8    9
```

A translates to an array of length 9 in this order:

```
A(1) = A(1,1) = 1;
A(2) = A(2,1) = 4;
A(3) = A(3,1) = 7;
A(4) = A(1,2) = 2;
A(5) = A(2,2) = 5;
```

and so on.

In column-major format, the next element of an array in memory is accessed by incrementing the first index of the array. For example, these element pairs are stored sequentially in memory:

- $A(i)$ and $A(i+1)$
- $B(i, j)$ and $B(i+1, j)$
- $C(i, j, k)$ and $C(i+1, j, k)$

The matrix A is represented in memory by default with this arrangement:

```
1    4    7    2    5    8    3    6    9
```

In row-major array layout, the programming language stores row elements contiguously in memory. In row-major layout, the elements of the array are stored as:

```
1    2    3    4    5    6    7    8    9
```

You can store the N-dimensional arrays in column-major or row-major layout. In column-major layout, the elements from the first (leftmost) dimension or index are contiguous in memory. In row-major layout, the elements from the last (rightmost) dimension or index are contiguous.

For more information on the internal representation of MATLAB data, see “MATLAB Data”.

Code generation software uses column-major format by default for several reasons:

- The world of signal and array processing is largely in column-major array layout: MATLAB, LAPack, Fortran90, and DSP libraries.
- A column is equivalent to a channel in frame-based processing. In this case, column-major storage is more efficient.
- A column-major array is self-consistent with its component submatrices:
 - A column-major 2-D array is a simple concatenation of 1-D arrays.
 - A column-major 3-D array is a simple concatenation of 2-D arrays.
 - The stride is the number of memory locations to index to the next element in the same dimension. The stride of the first dimension is one element. The stride of the nth dimension element is the product of the sizes of the lower dimensions.
 - Row-major n-D arrays have their stride of 1 for the highest dimension. Submatrix manipulations are typically accessing a scattered data set in memory, which does not allow for efficient indexing.

C uses row-major format. MATLAB and Simulink use column-major format by default. You can configure the code generation software to generate code with a row-major array layout. If you are integrating external C code with the generated code, see the considerations listed in this table.

Action	Consider
Configure array layout of the model for code generation.	In the Configuration Parameters dialog box, set model configuration parameter Array layout to Column-major or Row-major.
Enable efficient row-major algorithms for simulation and code generation.	Select model configuration parameter Use algorithms optimized for row-major array layout .

Action	Consider
Integrate external C code functions in row-major array layout with the generated code.	<p>Create S-functions that integrate external code functions with the generated code by using:</p> <ul style="list-style-type: none"> • The SimStruct function <code>ssSetArrayLayoutForCodeGen</code>. • The “Set Array Layout” setting in the S-Function Builder block. • The Legacy Code Tool option <code>convert2DMatrixToRowMajor</code> in <code>legacy_code</code>. Using <code>convert2DMatrixToRowMajor</code> impacts code efficiency. <p>Use the C Caller block to call external C functions into Simulink. Specify array layout of custom C functions by using model configuration parameter Default function array layout.</p> <p>You can also use <code>coder.ceval</code> in a MATLAB Function block. See “Interface with Row-Major Data in MATLAB Function Block”.</p>

Code Generator Matrix Parameters

The compiled model file, `model.rtw`, represents matrices as character vectors in MATLAB syntax, without an implied storage format. This format enables you to copy the character vector out of an `.rtw` file, paste it into a MATLAB file, and have MATLAB recognize it.

Column-Major Layout

For example, the 3-by-3 matrix in the Constant block

```

1  2  3
4  5  6
7  8  9

```

is stored in `model.rtw` as

```

Parameter {
    Identifier          "Constant_Value"
    LogicalSrc         P0
    Protected          no
    Value              [1.0, 4.0, 7.0, 2.0, 5.0, 8.0, 3.0, 6.0, 9.0]
    CGTypeIdx          18
    ContainerCGTypeIdx 19
    ReferencedBy       Matrix(1,4)
    GraphicalRef       Matrix(1,2)
    BHMPrmIdx          0
    GraphicalSource    [0, 1]
    OwnerSysIdx        [0, -1]
    VarGroupIdx        [1, 0]
}
[[0, -1, 0, 0];]
[[0, 1];]

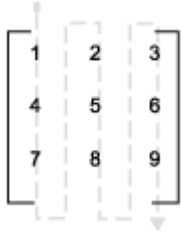
```

```

    WasAccessedAsVariable 1
}

```

The `model_data.c` file declares the actual storage for the matrix parameter. You can see that the format is in column-major layout.



```

Parameters model_P = {
  /* Expression: [ [1,2,3] ; [4,5,6] ; [7,8,9]]
   * Referenced by: '<Root>/Constant '
   */
  { 1.0, 4.0, 7.0, 2.0, 5.0, 8.0, 3.0, 6.0, 9.0 }
};

```

Row-Major Layout

For example, the 3-by-3 matrix in the Constant block

```

1  2  3
4  5  6
7  8  9

```

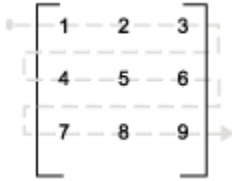
is stored in `model.rtw` as

```

Parameter {
  Identifier          "Constant_Value"
  LogicalSrc          P0
  Protected           no
  Value               [1.0, 2.0, 3.0, 4.0, 5.0, 6.0, 7.0, 8.0, 9.0]
  CGTypeIdx           18
  ContainerCGTypeIdx  19
  ReferencedBy        Matrix(1,4)
  GraphicalRef        Matrix(1,2)
  BHMPrmIdx           0
  GraphicalSource     [0, 1]
  OwnerSysIdx         [0, -1]
  VarGroupIdx         [1, 0]
  WasAccessedAsVariable 1
}

```

The `model_data.h` file declares the actual storage for the matrix parameter. You can see that the format is in row-major layout.



```
Parameters model_P = {
  /* Expression: [ [1,2,3] ; [4,5,6] ;[7,8,9]]
   * Referenced by: '<Root>/Constant '
   */
  { 1.0, 2.0, 3.0, 4.0, 5.0, 6.0, 7.0, 8.0, 9.0 }
};
```

Internal Data Storage for Complex Number Arrays

Simulink and code generator internal data storage formatting differs from MATLAB internal data storage formatting only in the storage of complex number arrays. In MATLAB, the real and imaginary parts are stored in separate arrays. In Simulink and the code generator, the parts are stored in an interleaved format. The numbers in memory alternate real, imaginary, real, imaginary, and so forth. This convention allows efficient implementations of small signals on Simulink lines, for Mux blocks, and other virtual signal manipulation blocks. For example, the signals do not actively copy their inputs, just the references.

Unsupported Blocks for Row-Major Code Generation

The code generator does not support these blocks for code generation in row-major array layout.

Continuous

- Derivative
- Integrator
- Integrator Limited
- Integrator, Second-Order
- Integrator, Second-Order Limited
- PID Controller
- PID Controller (2DOF)
- State-Space
- Transfer Fcn
- Transport Delay
- Variable Time Delay
- Variable Transport Delay
- Zero-Pole

User-Defined Functions

- Interpreted MATLAB Function
- Level-2 MATLAB S-Function

Sources

- From Spreadsheet

Signal Attributes

- IC

Limitations for Row-Major Code Generation

- Variable-size signals are not supported for row-major code generation.

See Also**More About**

- “MATLAB Data”
- “Mapping MATLAB Types to Types in Generated Code”
- “Row-Major and Column-Major Array Layouts”
- “Dimension Preservation of Multidimensional Arrays” (Embedded Coder)
- “Preserve Dimensions of Bus Elements in Generated Code” (Embedded Coder)
- “Interface with Row-Major Data in MATLAB Function Block”

Cross-Release Shared Utility Code Reuse

In this section...

“Workflow to Reuse Shared Utility Code” on page 24-69

“Required Editing for Shared Utility Code Reuse” on page 24-70

When you generate code for a model, the code generator by default creates shared utility files that the model requires. When you generate code with different releases, the code generators can produce functionally identical shared files that contain some nonfunctional differences. For example, different comments and different coding style. When you use the same release to generate code for different models in different folders, you can also produce shared files with nonfunctional differences. For example, if you specify different `ParenthesesLevel` or `ExpressionFolding` values for the models, the code generator can produce shared files that contain different comments or different coding styles.

Integrated code that includes functionally identical shared files:

- Is more expensive to verify because each shared file requires verification.
- Produces compilation errors if the shared files define duplicate symbols.

If you have an Embedded Coder license, you can avoid these issues by specifying the reuse of shared code from an existing folder, for example, a read-only library of verified code. In this case, the code generator does not create new shared utility files. The build process uses external code or previously generated shared utility code from the folder. An administrator maintains and updates the read-only library.

Workflow to Reuse Shared Utility Code

- 1 In the Configuration Parameters dialog box:
 - a In the **Existing shared code** field, enter the full path to your shared code folder.
 - b Verify that the **Use only existing shared code** diagnostic is set to error (default).
- 2 Remove the `slprj` folder or move to a new working folder.
- 3 Build your model. If you do not see an error, your shared code folder contains the required shared utility files.
- 4 If files are missing from the existing shared code folder, you see an error. To continue code generation with a locally generated version of the missing shared utility files:
 - a Set the **Use only existing shared code** diagnostic to warning.
 - b Rebuild your model. The code generation process uses a locally generated version of the missing shared utility files.
 - c Provide the administrator of the verified code library with your model and information about missing shared utility files. With the model, the administrator generates the required shared utility files. Using `sharedCodeUpdate`, the administrator adds the files to the existing shared code folder.

If you require reuse of shared code for a component exported from a previous release, provide the administrator with information about the build folder location for the component. The administrator can use `sharedCodeUpdate` to copy the shared code for the component to the existing shared code folder.

- d When the files are available in the existing shared code folder, repeat steps 1-3.

If the shared utility code is generated from library subsystems that are shared across models on page 10-36, you cannot reuse the code *across* releases because the code is release-specific—the symbol name and file name mangling includes the release number. The administrator must add the shared utility code generated for each release to the shared code folder.

The `sharedCodeUpdate` function can add files to the shared code folder that have identical content but different file and function names. This behavior is useful when you have different model components that require their own shared utility functions. Although some code is duplicated, the different model components can access the shared utility functions with which they were verified. To force model components to have their own versions of shared utility functions, configure naming rules to insert the model name into shared utility identifiers (Embedded Coder).

Required Editing for Shared Utility Code Reuse

For most shared utility code files, you can specify central copies that you can reuse across releases without modifying the files. With some files, for example, `rtwtypes.h`, and `zero_crossing_types.h`, there are situations where manual editing is required to produce central copies that you can use with generated code from different releases. For example:

- The `rtwtypes.h` file generated by releases up to and including R2013a contains a checksum.

```
/* This ID is used to detect inclusion
   of an incompatible rtwtypes.h */
#define RTWTYPES_ID_C08S16I32L64N64F0
```

For each version of this `rtwtypes.h` file that you want to include in your integration, copy the corresponding `#define` statement into your central copy of `rtwtypes.h`.

- In R2015a, the zero-crossing definitions moved from `rtwtypes.h` into `zero_crossing_types.h`. To create an `rtwtypes.h` file that is compatible with generated model code from different releases, in your central copy of `rtwtypes.h`, insert this statement.

```
#include "zero_crossing_types.h"
```

Remove definitions from `rtwtypes.h` that `zero_crossing_types.h` provides.

See Also

`sharedCodeUpdate` | `crossReleaseImport`

Related Examples

- “Cross-Release Code Integration” (Embedded Coder)

More About

- “Generate Shared Utility Code” on page 10-52
- “Generate Shared Utility Code for Fixed-Point Functions” on page 10-60
- “Generate Shared Utility Code for Custom Data Types” on page 10-62

Cross-Release Code Integration

In this section...

“Workflow” on page 24-71

“Limitations” on page 24-73

“Simulink.Bus Support” on page 24-74

“Root-Level I/O Through Global Variables in Generated Code” on page 24-75

“Communicate Between Current and Previous Release Components Through Global Data Stores” on page 24-77

“Parameter Tuning” on page 24-78

“Use Multiple Instances of Code Generated from Reusable Referenced Model” on page 24-79

“Compare Simulation Behavior of Model Component in Current Release and Generated Code from Previous Release” on page 24-80

“Import AUTOSAR Code from Previous Releases” on page 24-81

If you have an Embedded Coder license, you can integrate generated C code from previous releases (R2010a and later) with generated code from the current release when the source models are single-rate and the generated C code is from:

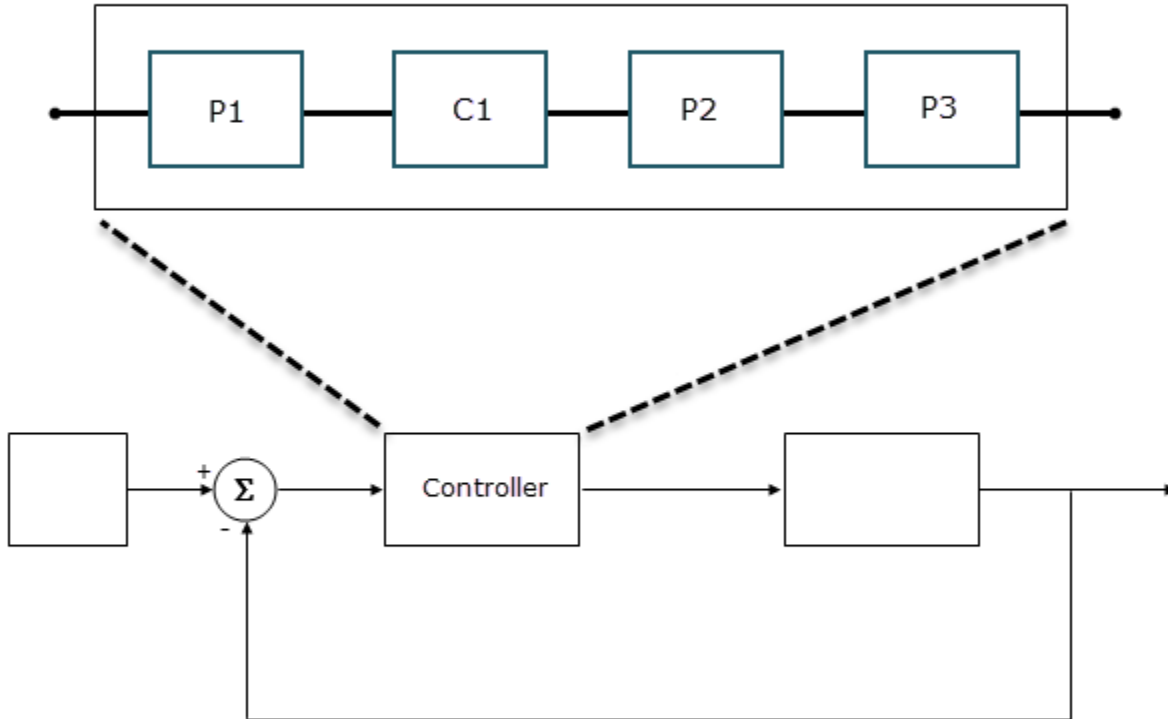
- Top-model or subsystem build processes that use the nonreusable function code interface.
- Single or multiple instance model reference build processes that suppress error status monitoring (`SuppressErrorStatus` is on).

If you can reuse existing code without modification, you can reduce the cost of reverification.

Note In future releases, `crossReleaseImport` will support the import of generated code from only the previous eight releases.

Workflow

Consider this control system model.



The Controller Model block references a model that consists of three components:

- P1 is a Model block, which references a model developed with a previous release, for example, R2015b. The generated model code, with the standalone code interface, is in the folder P1_ert_rtw.
- C1 is a Model block, which references a model that you are developing in the current release.
- P2 is a subsystem block developed with a previous release, for example, R2016a. The generated subsystem code is in the folder P2_ert_rtw.
- P3 is a Model block, which references a model developed with a previous release, for example, R2016b. The generated model code, with the model reference code interface, is in the folder slprj/ert/P3.

To integrate code from previous releases with generated code from the current release, use this workflow:

1 Specify an existing shared code folder

You can specify the reuse of shared code on page 24-69 from an existing folder, for example, a library of verified code that an administrator maintains and updates. Specify the shared code folder for model components that require code generation in the current release, for example, Controller and C1.

In the Configuration Parameters dialog box, in the **Existing shared code** field, enter the full path to the shared code folder.

2 Import components into current release

From the current release, import generated component code from previous releases as software-in-the-loop (SIL) blocks or processor-in-the-loop (PIL) blocks. For example:

```
crossReleaseImport(folderPathForP1_ert_rtw, 'Controller', ...
'SimulationMode', 'SIL');
crossReleaseImport(folderPathForP2_ert_rtw, 'Controller', ...
'SimulationMode', 'SIL');
crossReleaseImport(folderPathForP3, 'Controller', ...
'SimulationMode', 'SIL');
```

In the current working folder, the `crossReleaseImport` function creates the `xrel` subfolder that contains software-in-the-loop (SIL) blocks and subfolders:

- P1_R2015b_sil and P1_R2015b_sil_resources
- P2_R2016a_sil and P2_R2016a_sil_resources
- P3_R2016b_sil and P3_R2016b_sil_resources

3 Incorporate components into current release model

To replace components with SIL or PIL blocks, use the Simulink Editor or the `pil_block_replace` command. For example, replace:

- P1 with P1_R2015b_sil.
- P2 with P2_R2016a_sil.
- P3 with P3_R2016b_sil.

When you run a model simulation, the simulation runs the previous release code through the SIL or PIL blocks.

When you build the `Controller` model (`slbuild('Controller')`), the code generator does not generate new code for the components represented by the SIL or PIL blocks. The model code calls code generated by previous releases.

The Simulink Code Inspector supports call-site validation for a cross-release SIL or PIL block. The block provides a specification of the expected function call, which is derived from the generated code. During analysis, the Simulink Code Inspector compares the expected function call with the block behavior. For information about analysis limitations, see “S-Function” (Simulink Code Inspector).

Limitations

The cross-release code integration workflow does not support:

- Export-function models for ERT code.
- Simulink Function and Function Caller blocks across the boundaries of ERT code generated by different releases.
- The integration of generated code from releases before R2010a.
- The import of generated code from the current release into a previous release (forward compatibility).

- The export of files located in the MATLAB root folder of the previous release, for example, blockset library files.
- The export and import of generated code from models with noninlined S-functions.
- C-API (Embedded Coder).

At the end of the model build process, the code generation report displays shared files that are directly used by the integration model, for example, Controller. The report does not display shared files used by the components of the model, for example, P1 and P2.

If your model has:

- A cross-release SIL or PIL block, you cannot run rapid accelerator mode simulations.
- A Model block that references a model containing a cross-release SIL or PIL block, you cannot run accelerator mode or rapid accelerator mode simulations.

You can run:

- Top-model SIL simulations of models that contain cross-release SIL or PIL blocks. The models must not contain cross-release AUTOSAR Model blocks.
- Model block SIL simulations of referenced models that contain cross-release SIL or PIL blocks. The referenced models must not contain cross-release AUTOSAR Model blocks.

For information about other cross-release AUTOSAR code integration limitations, see “Import AUTOSAR Code from Previous Releases” on page 24-81.

Simulink.Bus Support

To use a bus object as a data type in cross-release ERT code integration, use one of these approaches.

Approach	Code Export	Code Import
Automatic	<p>In the previous release, before generating code, set the <code>DataScope</code> property of the <code>Simulink.Bus</code> object to <code>Auto</code>. Do not assign a value to the <code>HeaderFile</code> property.</p> <p>The code generator creates the <code>Simulink.Bus</code> data type definition in the default header file (Embedded Coder), which is located in the code generation folder for the model.</p>	<p>In the current release, before running <code>crossReleaseImport</code>, set the <code>DataScope</code> property of the <code>Simulink.Bus</code> object to <code>Auto</code>.</p> <p>When you build the integration model, the build process uses the <code>Simulink.Bus</code> data type from the header file in the imported code.</p>

Approach	Code Export	Code Import
Exported bus	<p>In the previous release, before generating code, specify these properties of the <code>Simulink.Bus</code> object:</p> <ul style="list-style-type: none"> • <code>DataScope</code> — Set to <code>Exported</code>. • <code>HeaderFile</code> — Specify a file name, for example, <code>prevRelBusType</code>. <p>The code generator creates <code>prevRelBusType.h</code> in the shared utility code folder. This header file contains the definition for the <code>Simulink.Bus</code> data type. Use <code>sharedCodeUpdate</code> to add <code>prevRelBusType.h</code> to the shared code folder that <code>ExistingSharedCode</code> specifies.</p> <p>For R2010a and R2010b, the <code>DataScope</code> property is not available. Do not assign a value to the <code>HeaderFile</code> property. The code generator creates the <code>Simulink.Bus</code> data type definition in <code>modelName_types.h</code>, which is located in the code generation folder for the model.</p>	<p>In the current release, before running <code>crossReleaseImport</code>, set the <code>DataScope</code> property of the <code>Simulink.Bus</code> object to <code>Imported</code>.</p> <p>When you build the integration model that incorporates the imported SIL or PIL block, the build process uses the <code>Simulink.Bus</code> data type definition in <code>prevRelBusType.h</code>.</p> <p>If the imported code is from R2010a or R2010b, specify these properties of the <code>Simulink.Bus</code> object:</p> <ul style="list-style-type: none"> • <code>DataScope</code> — Set to <code>Imported</code>. • <code>HeaderFile</code> — Set to file path for <code>modelName_types.h</code>, which is in the imported code folder. <p>When you build the integration model, the build process uses the <code>Simulink.Bus</code> data type definition in <code>modelName_types.h</code>.</p>
Imported bus	<p>In the previous release, before generating code, specify these properties of the <code>Simulink.Bus</code> object:</p> <ul style="list-style-type: none"> • <code>DataScope</code> — Set to <code>Imported</code>. • <code>HeaderFile</code> — Specify a path to a file that contains the <code>Simulink.Bus</code> data type definition, for example, <code>aBusType.h</code>. <p>For R2010a and R2010b, the <code>DataScope</code> property is not available. For the <code>HeaderFile</code> property, specify a path to a file that contains the <code>Simulink.Bus</code> data type definition, for example, <code>aBusType.h</code>.</p>	<p>In the current release, after importing generated code, you do not have to change the <code>Simulink.Bus</code>.</p> <p>When you build the integration model that incorporates the imported SIL or PIL block, the build process uses the <code>Simulink.Bus</code> data type definition from <code>aBusType.h</code>.</p> <p>If the imported code is from R2010a or R2010b, specify these properties of the <code>Simulink.Bus</code> object:</p> <ul style="list-style-type: none"> • <code>DataScope</code> — Set to <code>Imported</code>. • <code>HeaderFile</code> — Set to <code>aBusType.h</code>. <p>When you build the integration model, the build process uses the <code>Simulink.Bus</code> data type definition in <code>aBusType.h</code>.</p>

Root-Level I/O Through Global Variables in Generated Code

When imported ERT code from a previous release implements an input or output port through a global variable, the code that is generated from the integration model depends on the properties of the signal connected to the port in the integration model.

If the integration model signal does not map to a generated code variable that has the same name as the signal in the imported code, the generated code:

- Defines the variable if it is declared but undefined in the imported code.
- Creates additional code to copy data between the variable generated from the integration model and the variable in the imported code.

If the integration model signal maps to a variable with the same name as the signal in the imported code, the variable used by the imported code is also used by the code generated from the integration model. You must use a compatible storage class for the signal.

Storage Class Property	Property Value Support
Type	Unstructured only
Data access	<p>Cannot connect a port implemented as a <code>Pointer</code> variable in the imported code to a signal of the same name that is implemented through a <code>Direct</code> storage class in the integration model</p> <p>Cannot connect a port implemented as a <code>Direct</code> variable in the imported code to a signal of the same name that is implemented as a <code>Pointer</code> in the integration model</p>

Storage Class Property	Property Value Support
Data scope	<p>If the imported code declares but does not define the variable (i.e. code is generated from a signal that uses the Imported value), then one of the following is required:</p> <ul style="list-style-type: none"> • The integration model uses the Exported value. • External code defines the variable. <p>If the imported code defines the variable (i.e. code is generated from a signal that uses the Exported value):</p> <ul style="list-style-type: none"> • For signals in the integration model that are connected to root-level I/O ports: <ul style="list-style-type: none"> • Set the <code>EnableDataOwnership</code> configuration parameter to on. • Use a custom storage class with these properties: <ul style="list-style-type: none"> • Type - Unstructured • Data access - Direct • Data scope - Exported • Owner - Non-empty value that is not the name of the integration model. For example, the value can be the name of the imported component. • For signals in the integration model that are not connected to root-level I/O ports, use a custom storage class with these property values: <ul style="list-style-type: none"> • Data scope - Imported • Data access - Direct

Communicate Between Current and Previous Release Components Through Global Data Stores

Current and previous release components can communicate through global data stores associated with Simulink. Signal objects in the MATLAB base workspace or a Simulink data dictionary. The workflow described here applies to cross-release ERT code integration.

Code Generation Configuration

Before generating model component code in the old release, configure data store memory to use a storage class that imports external code.

If a top-model or subsystem build process generates the code, set the **Storage class** property of the `Simulink.Signal` objects to one of these classes:

- `ImportedExtern`
- `ImportedExternPointer`
- `ImportFromFile` custom storage class

If a model reference build process generates the code, you can also use these classes:

- `ExportedGlobal`
- `ExportToFile` custom storage class

Import Configuration

In the current release, before you run `crossReleaseImport`, define a `Simulink.Signal` object in the MATLAB base workspace or a Simulink data dictionary for each global data store of the component that you want to import:

- 1 For the object name and the **Data type**, **Complexity**, and **Dimensions** object properties, specify values that match values of the corresponding object in the component that you want to import.
- 2 For the **Storage class** property, specify a value that is compatible. If the previous release value is `ImportedExtern`, specify one of these values for the current release:
 - `ImportedExtern`
 - `ExportedGlobal`
 - `ImportFromFile` or `ExportToFile` custom storage class.

If the previous release value is `ImportedExternPointer`, then specify `ImportedExternPointer` for the current release.

- 3 For the **Identifier** property, specify a matching value only if the property is specified for the object in the component that you want to import.
- 4 Save the `Simulink.Signal` objects. The objects are required each time you simulate or build the imported SIL or PIL block.

For more information, see “Data Stores in Generated Code” on page 15-94.

Parameter Tuning

The cross-release ERT code integration workflow supports parameter tuning in an integration model that contains component code with tunable parameters from previous releases.

In the current release, before you run `crossReleaseImport`, for each tunable parameter of the component that you want to import:

- 1 Define a `Simulink.Parameter` object in the MATLAB base workspace or a Simulink data dictionary.
- 2 For the object name and the **Data type**, **Complexity**, and **Dimensions** object properties, specify values that match the previous release object values. If the variable in the imported source code does not have the same name as the object, for the **Identifier** property, specify a value that matches the variable name.
- 3 If the previous release object uses the `GetSet` custom storage class, specify:

- A matching value for the **Storage class** object property.
- The header file that defines the `get` and `set` access functions. For details, see “Control Data Representation by Configuring Storage Class Properties” (Embedded Coder) and “Access Data Through Functions with Storage Class GetSet” (Embedded Coder).

Support of the `GetSet` custom storage class from a previous release applies only for R2011a and later releases.

For code generated by a top-model or subsystem build process in a previous release, these limitations apply:

- If a tunable parameter specified by a `Simulink.Parameter` object in the previous release has storage class set to `ExportedGlobal` and the storage class of the `Simulink.Parameter` object in the current release is also `ExportedGlobal`, an error occurs when you build the integration model.
- On a Mac operating system, if a tunable parameter specified by a `Simulink.Parameter` object in the previous release has storage class set to `ExportedGlobal`, you cannot build the integration model if the storage class of the `Simulink.Parameter` object in the current release (with the same name or alias) is `ImportedExtern`. To work around this limitation, modify the default settings:

- 1 Get the build tool from the default toolchain.

```
tc = coder.make.getDefaultToolchain;
cComp = tc.getBuildTool('C Compiler');
```

- 2 Extract the C compiler standard options.

```
stdMaps = cComp.SupportedStandard.getLangStandardMaps;
optionValues = stdMaps.getCompilerOptions('*');
```

- 3 Remove `-fno-common` from the standard options for the C and C++ compilers.

```
optionToRemove = '-fno-common';
optionsToKeep = strrep(optionValues, optionToRemove, '');
c_standard_opts_id = '$(C_STANDARD_OPTS)';

custToolChainOpts = get_param(model, 'CustomToolchainOptions');
custToolChainOpts{2} = ...
    strrep(custToolChainOpts{2}, c_standard_opts_id, optionsToKeep);

set_param(model, 'CustomToolchainOptions', custToolChainOpts);
```

Use Multiple Instances of Code Generated from Reusable Referenced Model

Using a reusable referenced model, you can specify unique model argument values for each instance of the referenced model in a parent model. Import previously generated code for the reusable referenced model into the current release as a parameterized cross-release SIL or PIL block, and then insert multiple instances of the block into an integration model. For each block instance, you can specify unique model argument values.

- 1 Update existing shared code folder.

```
sharedCodeUpdate(sourceFolder, destinationFolder)
```

- 2 Import generated code for the reusable referenced model into the current release as a parameterized cross-release block, for example, a SIL block.

```
handleSILBlock = crossReleaseImport(reusableReferencedModelcodeLocation, ...
    configSetIntegrationModel, 'SimulationMode', 'SIL');
```

- 3 In the integration model, replace, for example, two instances of the referenced model with instances of the cross-release SIL block.

```
open_system(integrationModel);

blockInstanceName1 = 'refModelInstanceName1';
blockInstanceName2 = 'refModelInstanceName2';

SILBlockFullName = getfullname(handleSILBlock);
replace_block(integrationModel, 'Name', blockInstanceName1, ...
    SILBlockFullName, 'noprompt');
replace_block(integrationModel, 'Name', blockInstanceName2, ...
    SILBlockFullName, 'noprompt');
```

- 4 Suppose the reusable referenced model has model arguments *paramA* and *paramB*, and the values for the referenced model instances, *refModelInstanceName1* and *refModelInstanceName2*, are workspace variables. In this case, you can specify argument values for the cross-release block instances that are workspace variables.

```
pathToBlockInstanceName1 = [integrationModel, '/', blockInstanceName1];
pathToBlockInstanceName2 = [integrationModel, '/', blockInstanceName2];

set_param(pathToBlockInstanceName1, 'paramA', 'paramA_instName1');
set_param(pathToBlockInstanceName1, 'paramB', 'paramB_instName1');

set_param(pathToBlockInstanceName2, 'paramA', 'paramA_instName2');
set_param(pathToBlockInstanceName2, 'paramB', 'paramB_instName2');
```

- 5 To specify unique model argument values for the block instances in the integration model, assign values to the workspace variables, *paramA_instName1*, *paramB_instName1*, *paramA_instName2*, and *paramB_instName2*.

Compare Simulation Behavior of Model Component in Current Release and Generated Code from Previous Release

In a previous release, suppose that you developed a model component, generated code for the component, and tested and deployed the generated code. Now, in the current release, you want to add features to the model component and use the model component in system development and code generation. Before you proceed, you can compare the functional behavior of the model component and the generated code from the previous release.

To test the numerical equivalence between the model component and the generated code from the previous release, use Simulink Test. With the Test Manager (Simulink Test), you can perform back-to-back tests and output comparisons:

- 1 Bring the model component into the current release as a Model block with the **Simulation mode** block parameter set to **Normal**.
- 2 With the Model block, create a top model that specifies test input data.
- 3 Import the code generated in the previous release into the current release as a SIL block.
- 4 With the SIL block, create another top model that specifies the same the test input data.
- 5 In the Test Manager, create an equivalence test case that runs simulations of the top models and compares outputs.
- 6 Run the test case and review results.

For more information, see “Test Two Simulations for Equivalence” (Simulink Test).

Note If you want to compare the behavior of generated code from the current and previous release, in step 1, specify these Model block parameters:

- Set **Simulation mode** to Software-in-the-loop (SIL) or Processor-in-the-loop (PIL).
 - Set **Code interface** to Top model.
-

Import AUTOSAR Code from Previous Releases

If you install the Embedded Coder Support Package for AUTOSAR Standard, you can import into the current release AUTOSAR component code that you generated in a previous release.

When you run `crossReleaseImport`, the function imports the AUTOSAR code as a cross-release Model block instead of a SIL or PIL block. The **Simulation mode** parameter of the Model block is set to Software-in-the-loop (SIL) or Processor-in-the-loop (PIL). Incorporate the Model block into the current release model.

These limitations apply:

- The cross-release workflow does not support per-instance memory that accesses NVRAM.
- Tunable parameters must be mapped to AUTOSAR calibration parameters or AUTOSAR internal calibration parameters.
- If the original model uses variants or symbolic dimensions (dimension variants), the imported model can use only the same variant and symbolic dimension configuration that was used when generating the code in the previous release.

See Also

`sharedCodeMATLABVersions` | `sharedCodeUpdate` | `crossReleaseImport`

More About

- “Cross-Release Shared Utility Code Reuse” (Embedded Coder)
- “Generate Shared Utility Code” on page 10-52
- “Generate Shared Utility Code for Fixed-Point Functions” on page 10-60
- “Generate Shared Utility Code for Custom Data Types” on page 10-62
- “Integrate Generated Code by Using Cross-Release Workflow” (Embedded Coder)

Integration of Code from Multiple Folders

To help you to integrate generated code from multiple code generation folders, this table provides information about actions you can take.

Action	Information	Type of Integration Supported
Use consistent configuration parameters across models	<p>If you want to integrate a group of models, aim to use a configuration set that is consistent across the models.</p> <p>For each model, you can create a hash table from the shared utilities checksum file, <code>checksummap.mat</code>. The key-value pairs in the hash table give you information about model parameters. This information can help you to determine the parameter values that you must use across the models. For more information, see “Manage the Shared Utility Code Checksum” (Embedded Coder)</p> <p>To determine whether parameter values are consistent across the models, for each model, you can run checks for compliance with modeling guidelines that specify parameter values. For more information, see “Model Advisor Checks for High-Integrity Systems Modeling Guidelines”.</p> <p>You can specify the same configuration parameters for a group of models. For example, for each model, use a configuration reference to access the same configuration set from a data dictionary. When you use the same configuration set, the generated data types for each model are the same and the corresponding <code>rtwtypes.h</code> files are equivalent except for comments. To integrate code from the different models, you can choose a single <code>rtwtypes.h</code> file. For more information, see:</p> <ul style="list-style-type: none"> • “Share a Configuration with Multiple Models” • “What Is a Data Dictionary?” 	Same-release, cross-release

Action	Information	Type of Integration Supported
Reuse shared utility code	<p>You can reuse data and functions across software components by specifying a shared utilities folder for your models. In the Configuration Parameters dialog box, set Shared code placement to Shared location.</p> <p>If you want software components to share a utilities folder, generate code from a common working folder or specify the same code generation folder through the file generation control parameter, <code>CodeGenFolder</code>. For more information, see “Manage Build Process Folders” (Embedded Coder).</p> <p>Identically-named utility files generated in different releases are functionally equivalent even if file style or comments differ. For cross-release code integration, you can use the <code>ExistingSharedCode</code> parameter to specify the reuse of shared utility code from an existing folder. For more information, see:</p> <ul style="list-style-type: none"> • “Workflow to Reuse Shared Utility Code” (Embedded Coder) • Existing shared code (Embedded Coder) 	Same release, cross-release
Avoid generated header file updates	<p>In general, when you build a model, existing header files (and source files) in the <code>_sharedutils</code> or <code>_shared</code> folder are not regenerated. However, there are instances when the code generator overwrites existing header files in the folder. The table in “Code Generator Header Files” (Embedded Coder), which describes header file generation dependencies, provides information about how you can avoid overwriting some header files, for example, <code>rtwtypes.h</code> and <code>multiword_types.h</code>.</p> <p>For more information, see:</p> <ul style="list-style-type: none"> • “Build Process Folders” (Embedded Coder) • “Control Placement of <code>rtwtypes.h</code> for Shared Utility Code” (Embedded Coder) 	N/A

Action	Information	Type of Integration Supported
Use cross-release SIL or PIL block	You can use the cross-release SIL or PIL block to integrate generated code from previous releases (R2010a and later) with generated code from the current release. For more information, see “Cross-Release Code Integration” (Embedded Coder).	Same release, cross-release
Use CRL and ExistingSharedCode	<p>You can use the code replacement library (CRL) and ExistingSharedCode parameter approaches either separately or jointly:</p> <ul style="list-style-type: none"> • The CRL approach supports customization of code generation and eliminates the generation of some shared utility files. You must manage the use of utility files that are not replaced by CRL. For more information, see “Develop a Code Replacement Library” (Embedded Coder). • The ExistingSharedCode approach, which is based on previously generated utility files, also supports customization and suppresses regeneration of utility files. If customization is not required, the ExistingSharedCode approach is simpler to set up and manage. For more information, see “Cross-Release Shared Utility Code Reuse” (Embedded Coder). 	Same release, cross-release

See Also

More About

- “Cross-Release Code Integration” (Embedded Coder)

Integrate Generated Code by Using Cross-Release Workflow

This example shows how to reuse previously generated code by creating a cross-release software-in-the-loop (SIL) block and incorporating the block within an integration model. For more information about the workflow, see “Cross-Release Code Integration” (Embedded Coder).

Generate Code from a Model

In the cross-release code integration workflow, you create a software-in-the-loop (SIL) or processor-in-the-loop (PIL) block from code that you previously generated.

This example uses generated code from the current release.

```
model = 'rtwdemo_crossrelease_counter';
close_system(model,0)
load_system(model)
set_param(model, 'SimulationCommand', 'update');
open_system(model)
slbuild(model);

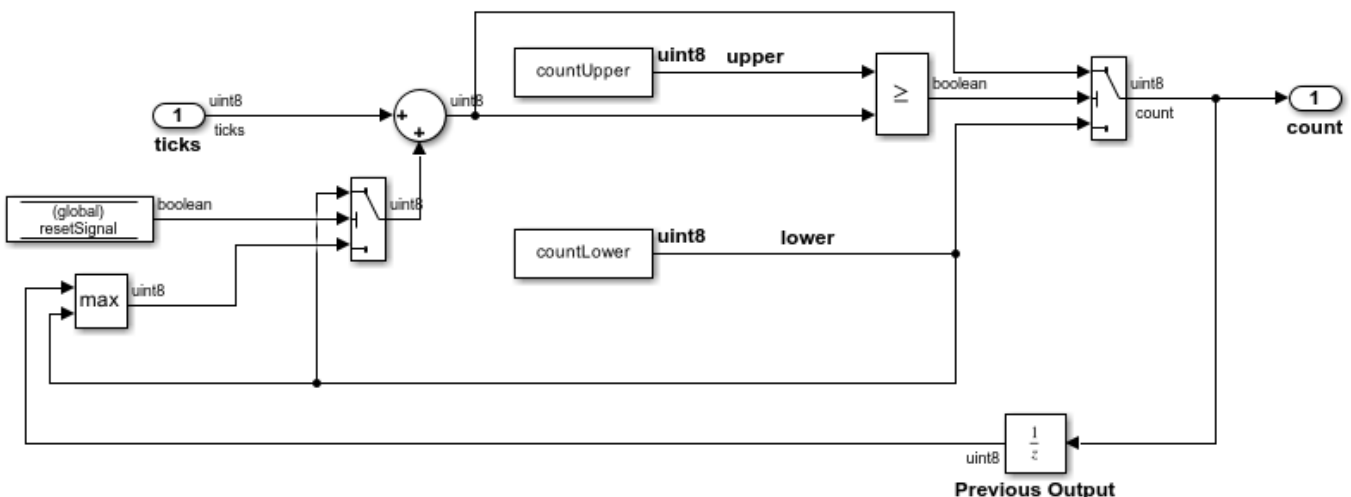
### Starting build procedure for: rtwdemo_crossrelease_counter
### Successful completion of code generation for: rtwdemo_crossrelease_counter
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_crossrelease_counter	Code generated.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 12.498s



Copyright 2017-2020 The MathWorks, Inc.

Identify the build folder that contains the generated code.

```
buildFolder = RTW.getBuildDir(model).BuildDirectory;
```

Identify the folder that contains code for shared utilities associated with the generated code.

```
previousSharedCodeFolder = RTW.getBuildDir(model).SharedUtilsTgtDir;
```

To reuse generated code from a previous release:

- 1 On the **Simulation** tab, in the **File** section, select **Save > Previous Version**.
- 2 In the **Save as type field**, specify the release and model type.
- 3 Click **Save**.
- 4 Use the previous release to open the saved model, and then generate code.
- 5 Set the value of `buildFolder` to the location of the code generated in the previous release.
- 6 Set the value of `previousSharedCodeFolder` to the location of the shared utilities generated in the previous release.

Manage Shared Code

Add the generated shared source code files to a repository folder that is used by the integration model.

```
sharedCodeRepo = 'SharedCodeRepo';
mkdir(sharedCodeRepo);
sharedCodeUpdate(previousSharedCodeFolder, sharedCodeRepo, 'Interactive', false);
```

The following files will be copied from `slprj\ert_sharedutils` to `SharedCodeRepo\R2023a`:

```
rtwtypes.h
```

Files copied from `slprj\ert_sharedutils` to `SharedCodeRepo\R2023a`.

Import Code

Load the integration model.

```
integrationModel = 'rtwdemo_crossrelease_integration';
close_system(integrationModel, 0);
load_system(integrationModel);
```

Modify the Simulink configuration set so that it references the existing shared code library.

```
cs = getActiveConfigSet(integrationModel);
set_param(cs, 'ExistingSharedCode', fullfile(pwd, sharedCodeRepo));
```

Create the cross-release SIL block.

```
blockHandle = crossReleaseImport(buildFolder, cs, 'SimulationMode', 'SIL');

### Starting import process for component: rtwdemo_crossrelease_counter_R2023a
### Starting build process for SIL block: rtwdemo_crossrelease_counter_R2023a
```



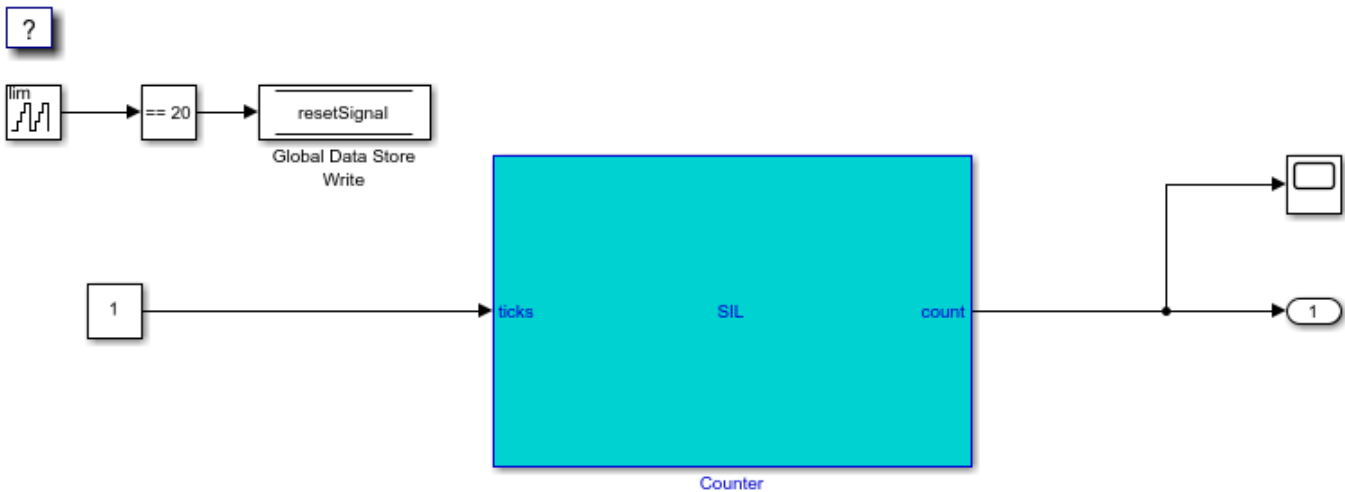
Incorporate Cross-Release SIL Block into Integration Model

To replace a block in the integration model with the cross-release block, use `pil_block_replace`. This function preserves the block size, line connections, and priority.

```
srcBlock = getfullname(blockHandle);
dstBlock = [integrationModel, '/', 'Counter'];
pil_block_replace(srcBlock, dstBlock)
open_system(integrationModel)
```

Successfully swapped the following blocks:

```
untitled/rtwdemo_crossrelease_counter_R2023a_sil
rtwdemo_crossrelease_integration/Counter
```

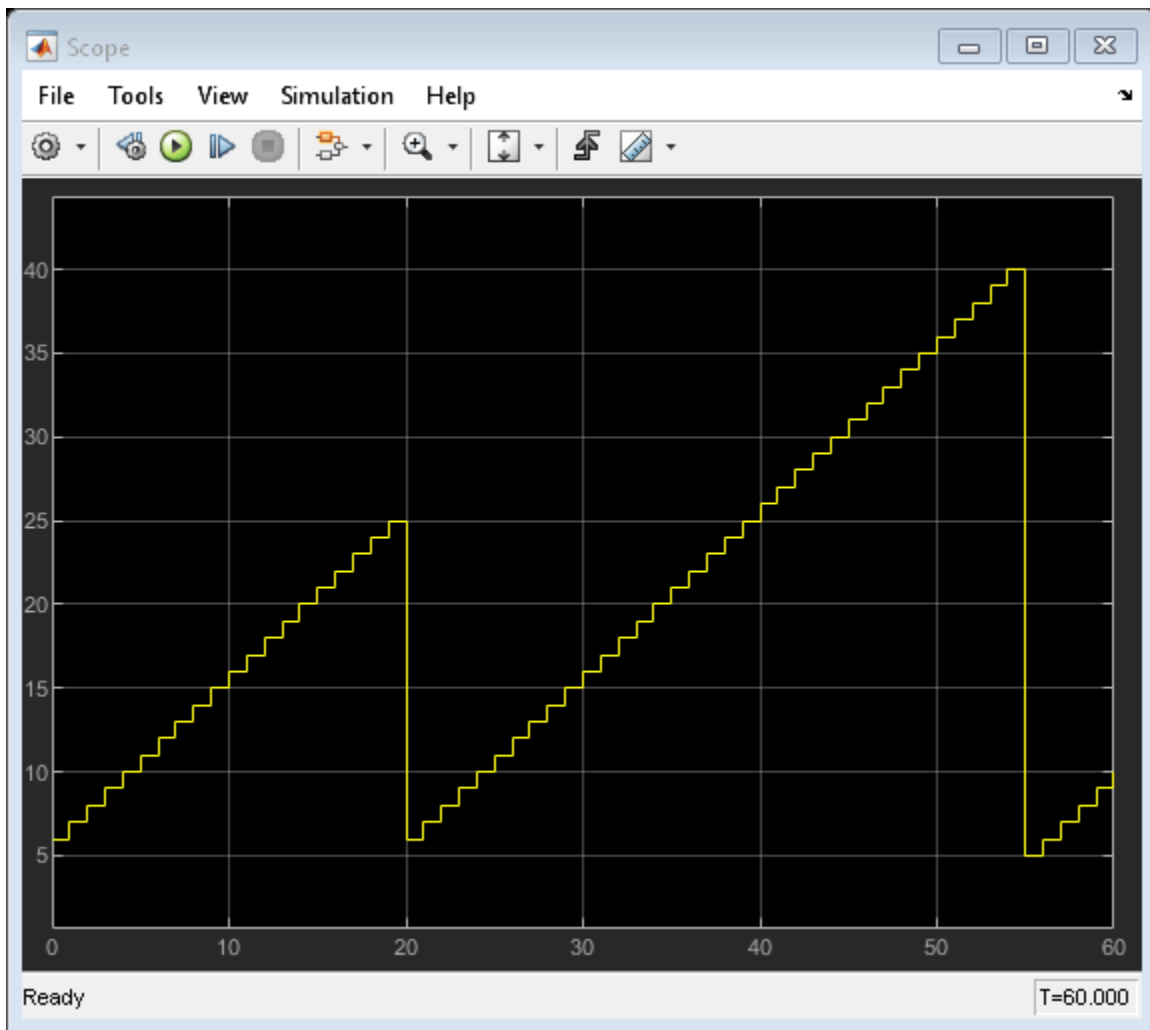


Simulate Integration Model

Run a simulation of the integration model.

```
sim(integrationModel)
```

```
### Preparing to start SIL block simulation: <a href="matlab: targets_hyperlink_manager('run',1)
### Starting SIL simulation for component: rtwdemo_crossrelease_counter_R2023a_sil
### Application stopped
### Stopping SIL simulation for component: rtwdemo_crossrelease_counter_R2023a_sil
```

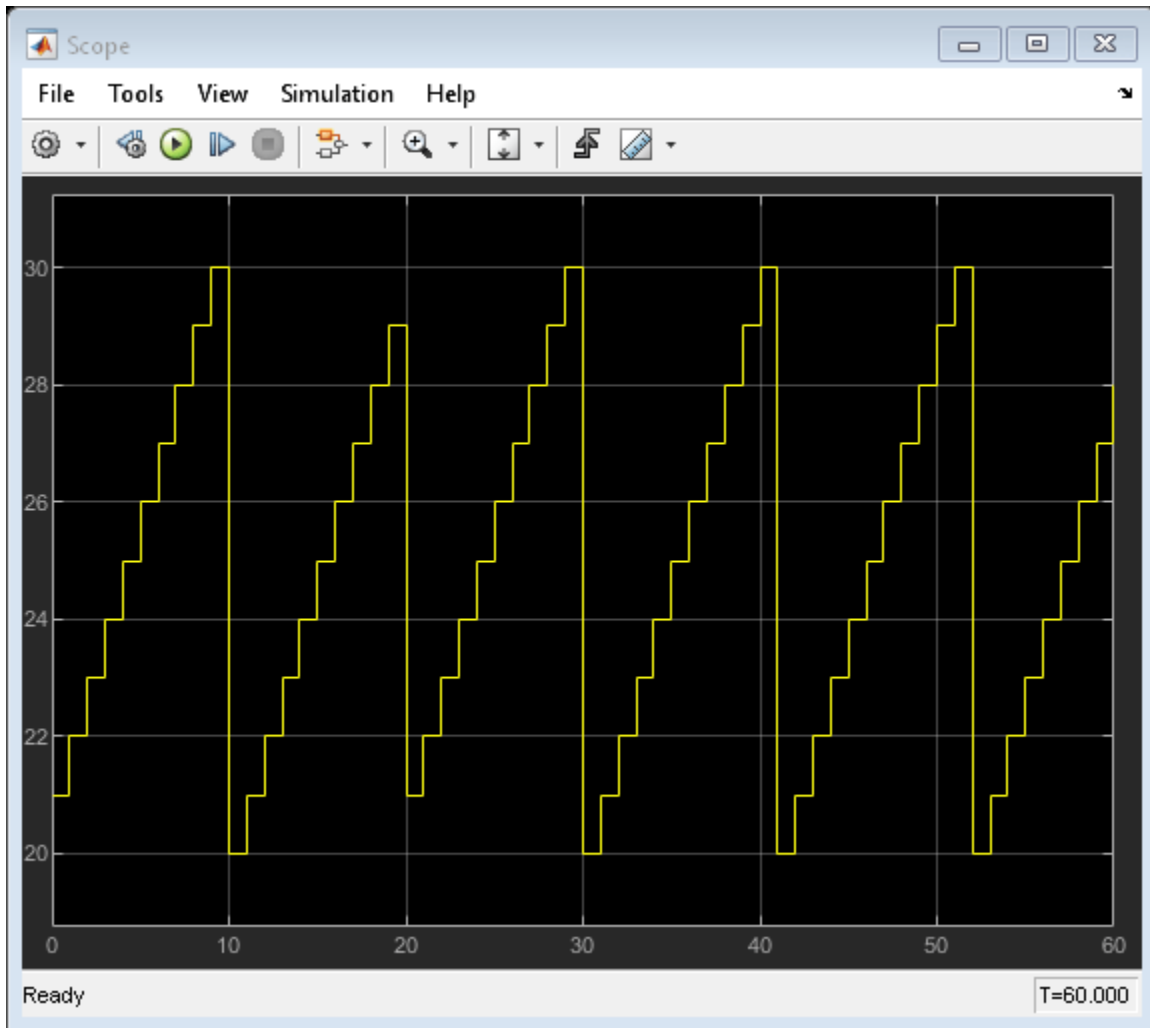


Tune Parameters

The source model for the cross-release SIL block references two tunable parameters, which are controlled by Simulink.Parameter objects in the base workspace. Use these parameters to modify the behavior of the SIL simulation.

```
countUpper.Value = 30;
countLower.Value = 20;
yout_returned = sim(integrationModel, 'ReturnWorkspaceOutputs', 'on');
```

```
### Preparing to start SIL block simulation: <a href="matlab: targets_hyperlink_manager('run',1)
### Starting SIL simulation for component: rtwdemo_crossrelease_counter_R2023a_sil
### Application stopped
### Stopping SIL simulation for component: rtwdemo_crossrelease_counter_R2023a_sil
```

Configure Storage Class for Signals in Integration Model

Configure:

- Signal names in the integration model to match the names used in the imported code.
- Complementary storage classes.

In this case, the `ticks` input and `count` output are implemented through the `ImportedExtern` storage class in the imported code. If the storage class for the signals connected to the input and output ports of the cross-release block in the integration model is `ExportedGlobal`, the integration model must provide the definitions for the variables.

If the signal names do not match, the integration model generates additional code to copy data between the signal implemented by the integration model and the signal implemented by the imported code.

```
hLines = get_param(dstBlock, 'LineHandles');
```

```
% Use the code mappings API to configure the storage class on the signals.
cm = coder.mapping.api.get(integrationModel);
```

```

ticksPortHandle = get(hLines.Inport(1), 'SrcPortHandle');
set(ticksPortHandle, 'Name', 'ticks');
addSignal(cm, ticksPortHandle);
setSignal(cm, ticksPortHandle, 'StorageClass', 'ExportedGlobal');

countPortHandle = get(hLines.Outport(1), 'SrcPortHandle');
set(countPortHandle, 'Name', 'count');
addSignal(cm, countPortHandle);
setSignal(cm, countPortHandle, 'StorageClass', 'ExportedGlobal');

```

Configure Storage Class for Parameters and Data Store Memory in Integration Model

The parameters and data stores are implemented through the ImportedExtern storage class in the imported code. If you configure the parameters and data stores to use the ExportedGlobal storage class, the integration model must provide the definitions for the variables.

```

resetSignal.CoderInfo.StorageClass = 'ExportedGlobal';
countLower.CoderInfo.StorageClass = 'ExportedGlobal';
countUpper.CoderInfo.StorageClass = 'ExportedGlobal';

```

Generate Code from Integration Model

Once the required shared code is in the shared code repository, remove the previously generated shared utilities folder.

```

if isfolder(RTW.getBuildDir(integrationModel).SharedUtilsTgtDir)
    rmdir(RTW.getBuildDir(integrationModel).SharedUtilsTgtDir, 's');
end

```

Delete the scope and connecting line that do not affect code generation.

```

scopeBlock = [integrationModel, '/', 'Scope'];
hScopeLines = get_param(scopeBlock, 'LineHandles');
hScopeLine = hScopeLines.Inport(1);
assert(strcmp(get(hScopeLine, 'SegmentType'), 'branch'));
delete_line(hScopeLine);
delete_block(scopeBlock);

```

Generate code.

```

slbuild(integrationModel);

### Starting build procedure for: rtwdemo_crossrelease_integration
### Successful completion of code generation for: rtwdemo_crossrelease_integration

```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_crossrelease_integration	Code generated.	Code generation information file does not exist

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 15.64s

Examine Call to Imported Code

To examine the cross-release block code within the integration model code, use the `rtwtrace` utility.

```
rtwtrace(dstBlock);
```

Test Code Generated from Integration Model

Run a top-model software-in-the-loop (SIL) simulation of the integration model. The simulation runs the code generated from the integration model, which calls the imported code.

Log the simulation output in the workspace.

```
set_param(integrationModel, 'SimulationMode', 'software-in-the-loop (sil)');
yout_SIL = sim(integrationModel, 'ReturnWorkspaceOutputs', 'on');
plot(yout_SIL.yout{1}.Values);
```

```
### Starting build procedure for: rtwdemo_crossrelease_integration
### Successful completion of build procedure for: rtwdemo_crossrelease_integration
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_crossrelease_integration	Code generated and compiled.	S-function rtwdemo_crossrelease_

```
1 of 1 models built (0 models already up to date)
```

```
Build duration: 0h 0m 12.667s
```

```
### Preparing to start SIL simulation ...
```

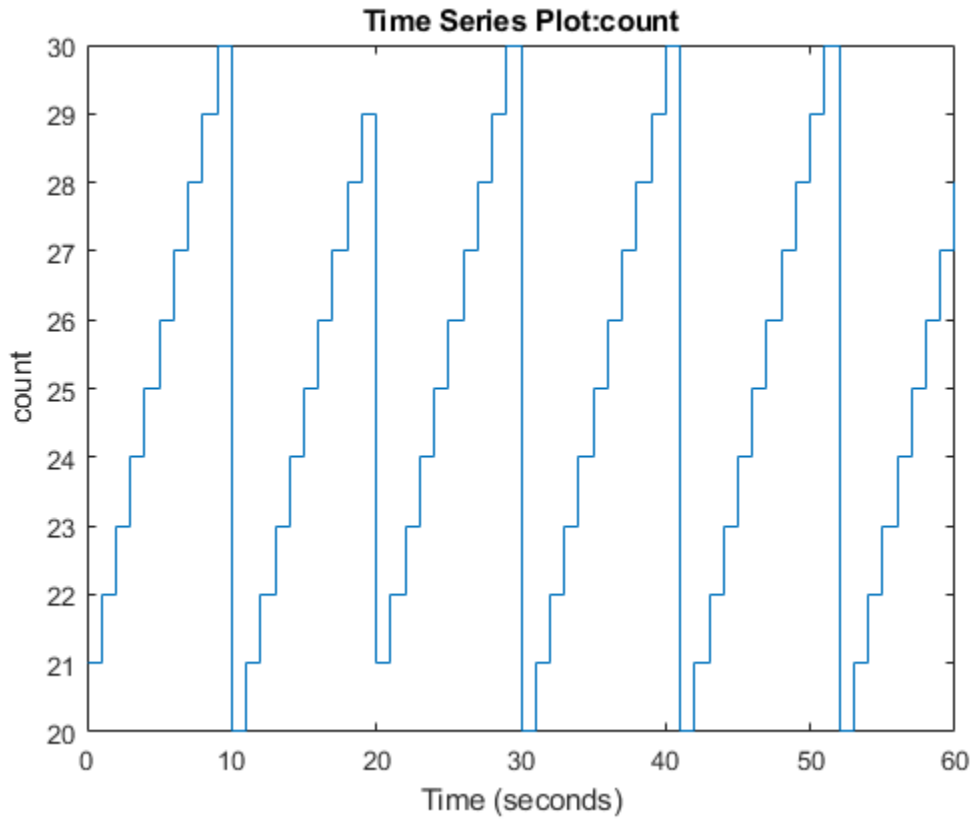
```
Building with 'Microsoft Visual C++ 2019 (C)'.
MEX completed successfully.
```

```
### Updating code generation report with SIL files ...
```

```
### Starting SIL simulation for component: rtwdemo_crossrelease_integration
```

```
### Application stopped
```

```
### Stopping SIL simulation for component: rtwdemo_crossrelease_integration
```



Compare Simulation Outputs

Compare the outputs from the simulation where only the imported code ran in SIL mode with the outputs from the simulation where the integration model ran as a top-model in SIL mode.

```
max(abs(yout_SIL.yout{1}.Values.Data - yout_returned.yout{1}.Values.Data))
```

```
ans =
```

```
uint8
```

```
0
```

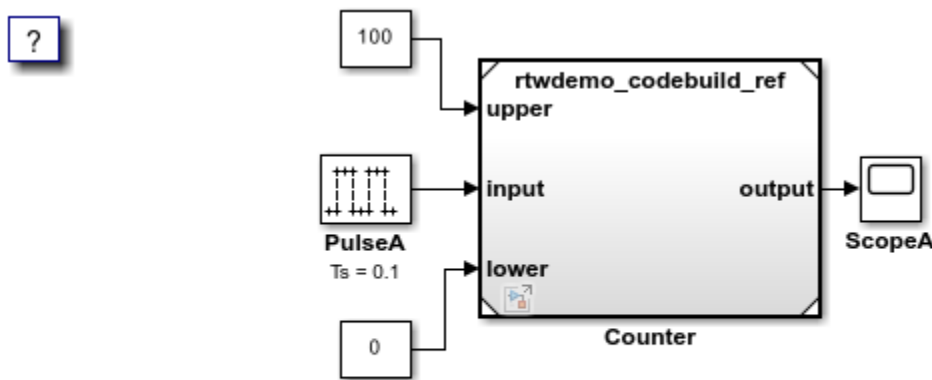
Compile Code in Another Development Environment

This example shows how you can relocate generated code and compile it in another development environment. Use packNGo to create a ZIP file that contains generated source code, build information, and CMake configuration files. In your target development environment, use the codebuild function to build the final library or executable file.

Configure the Model

Open the model.

```
rtwdemo_codebuild
```



Copyright 2016-2021 The MathWorks, Inc.

Do not automatically compile the code, generate a makefile, or package generated code and artifacts.

```
set_param('rtwdemo_codebuild', 'GenerateMakefile', 'off');
set_param('rtwdemo_codebuild', 'PackageGeneratedCodeAndArtifacts', 'off');
```

Generate Code and CMake Configuration Files

Run code generation command.

```
slbuild('rtwdemo_codebuild');

### Starting serial model reference code generation build
### Successfully updated the model reference code generation target for: rtwdemo_codebuild_ref
### Starting build procedure for: rtwdemo_codebuild
### Successful completion of code generation for: rtwdemo_codebuild
```

Build Summary

Code generation targets built:

Model	Action	Rebuild Reason
rtwdemo_codebuild_ref	Code generated	rtwdemo_codebuild_ref.c does not exist.

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_codebuild	Code generated	Code generation information file does not exist.

2 of 2 models built (0 models already up to date)
Build duration: 0h 0m 56.832s

Generate the CMakeLists.txt configuration files.

```
buildFolder = RTW.getBuildDir('rtwdemo_codebuild').BuildDirectory;
codebuild(buildFolder, 'BuildMethod', 'cmake');
```

Package Generated Code and CMake Configuration Files

Run packNGo, packaging the files hierarchically.

```
packNGo(buildFolder, 'packType', 'hierarchical', 'nestedZipFiles', false);
```

Switch to Another Development Environment

This is an optional step. For example, if your current development environment is a Linux computer, you can copy the zip file to a Windows computer and then perform the subsequent steps on that computer.

Unzip Source Code and Build Information Files

Unzip the files.

```
rtwdemo_codebuild_files = unzip('rtwdemo_codebuild.zip');
```

Identify folder containing unzipped code for the top component.

```
rtwdemo_codebuild_top_component_folder = fileparts(rtwdemo_codebuild_files{1});
```

Build Executable File Using Default Toolchain

Identify the default toolchain (based on compiler selected by mex -setup).

```
defaultToolchain = coder.make.getDefaultToolchain;
```

Build the executable file.

```
codebuild(rtwdemo_codebuild_top_component_folder, 'BuildMethod', defaultToolchain);
```

Check that executable file is produced.

```
dir(fullfile(rtwdemo_codebuild_top_component_folder, '..', 'rtwdemo_codebuild*'))
```

```
rtwdemo_codebuild.exe
rtwdemo_codebuild_grt_rtw
```

Build Executable File Using Default Template Makefile

Select a template makefile for the current platform.

```

if ispc
    % With ert_vcx64.tmf, you must have Microsoft Visual C++ installed. If
    % it is not installed, specify ert_lcc64.tmf to use lcc-win64 instead.
    templateMakefile = 'ert_vcx64.tmf';
else
    templateMakefile = 'ert_unix.tmf';
end

```

Build the executable file.

```
codebuild(rtwdemo_codebuild_top_component_folder, 'BuildMethod', templateMakefile);
```

Build Shared Library

On Windows, create a definition file that specifies exported symbols.

```

if ispc
    exportsFile = fullfile...
        (rtwdemo_codebuild_top_component_folder, 'rtwdemo_codebuild.def');
    fid = fopen(exportsFile, 'w');
    fwrite(fid, ['EXPORTS' newline]);
    fwrite(fid, ['rtwdemo_codebuild_initialize' newline]);
    fwrite(fid, ['rtwdemo_codebuild_step' newline]);
    fwrite(fid, ['rtwdemo_codebuild_terminate' newline]);
    fclose(fid);
end

```

Build the shared library.

```
codebuild(rtwdemo_codebuild_top_component_folder, 'BuildVariant', 'SHARED_LIBRARY');
```

Build Static Library

```
codebuild(rtwdemo_codebuild_top_component_folder, 'BuildVariant', 'STATIC_LIBRARY');
```

Use CMake Configuration Files

With the CMakeLists.txt files, you can use the third-party tool CMake to generate makefiles or workspaces for a compiler environment of your choice.

```

% Compute the CMake command for the current platform.
if ispc
    cmakeCommand = fullfile(matlabroot, 'bin', computer('arch'), 'cmake', 'bin', 'cmake.exe');
else
    cmakeCommand = fullfile(matlabroot, 'bin', computer('arch'), 'cmake', 'bin', 'cmake');
end

```

Using CMake, build the executable file.

```

cd(rtwdemo_codebuild_top_component_folder)
[status1, cmdout1] = system([cmakeCommand ' .']);
[status2, cmdout2] = system([cmakeCommand ' --build .']);

```

See Also

packNGo | codebuild

More About

- “Relocate or Share Generated Code” on page 31-56
- “Configure CMake Build Process” on page 31-13
- “Deploy Component Algorithm as Component Model Library by Using CMake” (Embedded Coder)
- “Code Compilation Customization”

Generate Code and Simulate Models in a Project

This example shows how to use the code generation template for a new Project. The code generation project template for Project includes multiple models. The project template also provides utilities (.m scripts) that help you generate controller code and run simulations of the harness model.

Create a Project

Create a new Project from the code generation project template.

To create this project from the Simulink® start page, in the Command Window, type:

```
simulink
```

Select the Code Generation template from the start page, and create the exampleCodeGen project.

Generate Code

Generate controller code for the `feedback_control.slx` model.

To generate controller code, select the **Project Shortcuts** tab and select the **Generate Controller Code** shortcut.

This shortcut runs the `generate_controller_code.m` script in the utilities folder of the project. The script builds the `feedback_control.slx` model in the controller folder of the project.



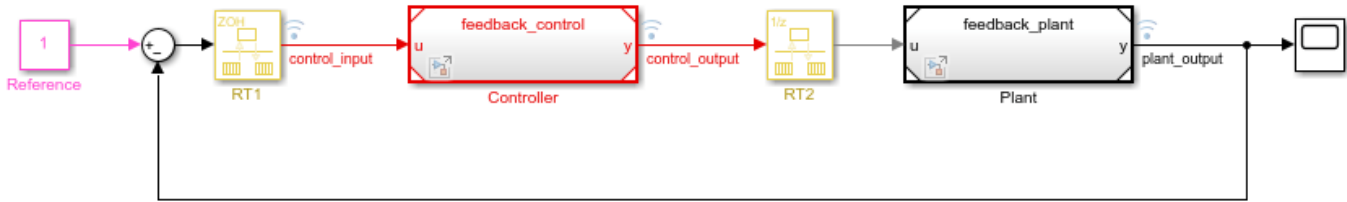
Copyright 2012-2017 The MathWorks, Inc.

Simulate Models

Simulate the top-level harness `feedback_harness.slx` model.

To open the harness model for simulation, select the **Project Shortcuts** tab and select the **Feedback Harness** shortcut. This shortcut opens the `feedback_harness.slx` model in the harnesses folder of the project.

To simulate the model, click **Run**.

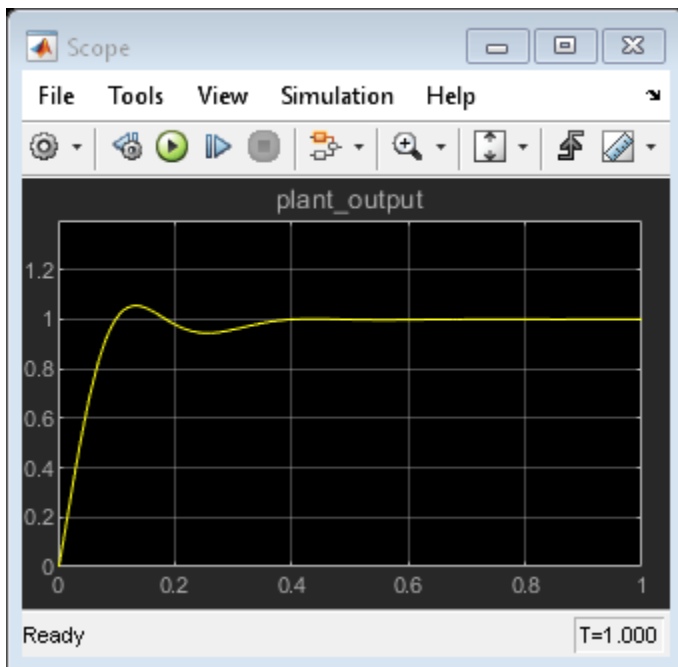


Copyright 2012 The MathWorks, Inc.

Observe Simulation Output

Open the Scope block in the model and observe the simulation output.

To open the scope and observe simulation, in the Simulink window, double-click the Scope block.

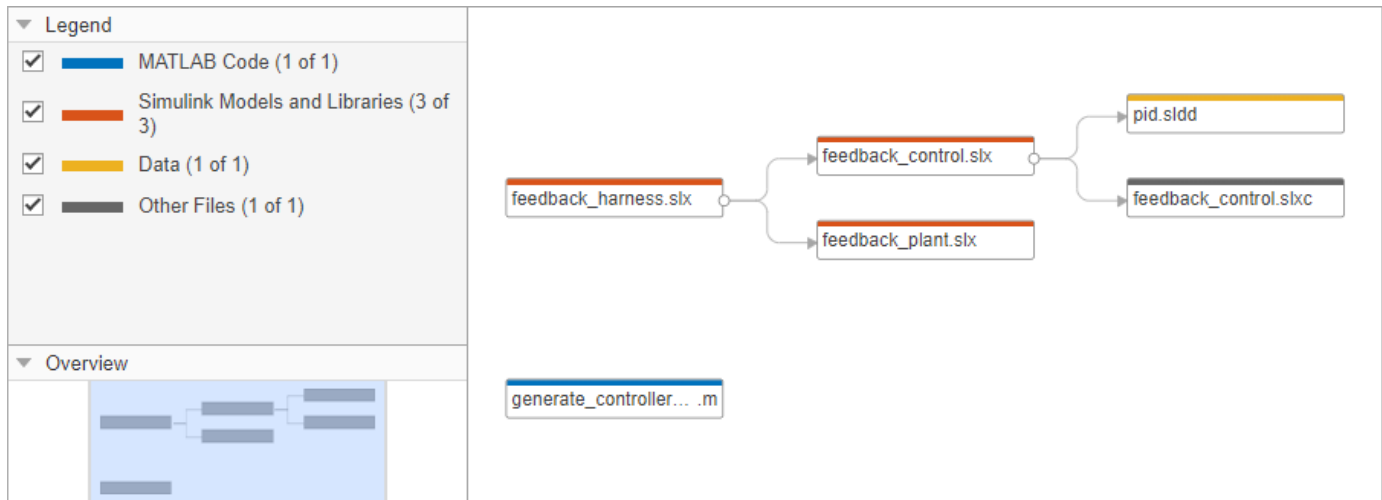


Use the Dependency Analyzer View

To view file dependencies, use the **Dependency Analyzer** view of the project.

From the **Dependency Analyzer** view, you can:

- Observe file dependencies.
- Hover over the dependency arrows to examine the dependency type.
- Double-click on a file or a model to open it.



More Information

- “Work with Project Files”
- `Simulink.createFromTemplate`
- “Automate Startup Tasks”
- “Automate Shutdown Tasks”

Related Examples

- “Create a New Project from a Folder”
- “Create a Project from a Model”

Generate Code and Simulate Models Programmatically with Project

This example shows how to use the Project API to create a new project and automate project tasks for manipulating files. It covers how to programmatically create a blank project, add files and folders, set up the project path, define project shortcuts, generate code, and simulate the harness model.

Project helps you organize large designs by managing and sharing files and settings, finding required files, and interacting with source control.

Manage a Model-Based Design Project

The example models provide a controller that positions a camera fixed to a DC motor that tracks a green ball.

The MBD approach leads to modeling the controller operations:

- Model plant: The plant is a DC motor that rotates to reposition the camera.
- Model controller: The design uses a PID controller with feedback on angular position for positioning the DC motor.
- Model high-level ball tracking scheduler: The scheduler checks for validity of the position of the green ball and the saturation of the reference angle. The scheduler handles the instances of no ball on the screen to track.

The design and Project have the following folders.

Plant -- The plant provides:

- Models with continuous-time dynamic systems that use basic Simulink® blocks.
- Models with linear time invariant (LTI) systems that use the LTI system block.
- Import and export of simulation data to and from the workspace
- Loading of parameters into the base workspace that use data dictionary and model callbacks.

Controller -- The controller provides:

- Models with discrete-time dynamic systems that use basic Simulink blocks.
- Models with discrete-time dynamic systems that use the PID Controller Block.
- Enforcement of signal saturation that use a PWM Conversion subsystem.

Ball Tracking -- The ball tracking provides:

- The basic ball tracking algorithm.
- The terminator and ground blocks as placeholders.
- A hybrid controller that generates motor reference angles using Stateflow®.

Harnesses -- The harnesses provide top-level simulation of the full system.

Create a Blank Project and Add Files

1. Create a blank project. On the **Home** tab, click **New > Project > Blank Project**.

Alternatively, create this project and use `currentProject` to get a project object to manipulate the project from the Command Window:

```
matlab.project.createProject('modelBasedDesignCodeGenProject');
proj_modelBasedDesignCodeGen = currentProject

proj_modelBasedDesignCodeGen =
    Project with properties:

        Name: "modelBasedDesignCodeGenProject"
    SourceControlIntegration: ""
        RepositoryLocation: ""
    SourceControlMessages: [1x0 string]
        ReadOnly: 0
        TopLevel: 1
    Dependencies: [1x1 digraph]
        Categories: [1x1 matlab.project.Category]
        Files: [1x0 matlab.project.ProjectFile]
        Shortcuts: [1x0 matlab.project.Shortcut]
    ProjectPath: [1x0 matlab.project.PathFolder]
    ProjectReferences: [1x0 matlab.project.ProjectReference]
    StartupFiles: [1x0 string]
    ShutdownFiles: [1x0 string]
    DefinitionFileType: FixedPathMultiFile
    Description: ""
    RootFolder: "C:\Users\ktripp\MATLAB\Projects\modelBasedDesignCodeGenProject1"
    DependencyCacheFile: ""
    SimulinkCodeGenFolder: ""
    ProjectStartupFolder: "C:\Users\ktripp\MATLAB\Projects\modelBasedDesignCodeGenProject1"
    SimulinkCacheFolder: ""
```

2. Copy the example files to folders within the Project.

Alternatively, in the Command Window, type:

```
copyfile(fullfile(matlabroot, 'toolbox', 'rtw', 'rtwdemos', 'slproject_codegen_demo'), ...
fullfile(pwd), 'f');
```

3. Add existing files to the project:

- Click the **Project Files View** button and select **All files View**.
- Select the folders `ball_tracking`, `controller`, `harnesses`, `plant`, and `utilities`, right-click the icon for one of the folders, and select **Add Folder to Project (Including Child Files)**.

Alternatively, add these files to the project programmatically from the Command Window:

```
addFolderIncludingChildFiles(proj_modelBasedDesignCodeGen, 'ball_tracking');
addPath(proj_modelBasedDesignCodeGen, 'ball_tracking');
addFolderIncludingChildFiles(proj_modelBasedDesignCodeGen, 'controller');
addPath(proj_modelBasedDesignCodeGen, 'controller');
addFolderIncludingChildFiles(proj_modelBasedDesignCodeGen, 'harnesses');
addPath(proj_modelBasedDesignCodeGen, 'harnesses');
addFolderIncludingChildFiles(proj_modelBasedDesignCodeGen, 'plant');
addPath(proj_modelBasedDesignCodeGen, 'plant');
addFolderIncludingChildFiles(proj_modelBasedDesignCodeGen, 'utilities');
addPath(proj_modelBasedDesignCodeGen, 'utilities');
```

4. Select the `work` folder, right-click the icon for one of the folders, and select **Project Path > Add to the Project Path (Including Subfolders)**. If you choose to add this folder to the path, generated files (artifacts) from simulation and code generation are available on the project path. If you choose not to include the generated files in the project, the dependency graph does not display these files.

Alternatively, add this folder to the project path from the Command Window:

```
addPath(proj_modelBasedDesignCodeGen, 'work');
```

5. To make the project more accessible, define and set properties for project shortcuts. In the **Files** View, select the **Project** tab. Open the `utilities` folder node.

6. Select the files `clean_up_project.m`, `generate_controller_code.m`, and `set_up_project.m`, right-click the icon for one of the files, and select **Create Shortcut**.

7. To set the current working folder, set the simulation cache folder, and run other setup scripts when you open the project:

- Select the `set_up_project.m` script.
- Right-click the file.
- Select **Run at Startup**.

Alternatively, add this setup script to project startup from the Command Window:

```
addStartupFile(proj_modelBasedDesignCodeGen, fullfile(pwd, '/utilities/set_up_project.m'));
```

8. To restore previous settings and run other clean-up scripts when you close the project:

- Select the `clean_up_project.m` script.
- Right-click the file.
- Select **Run at Shutdown**.

Alternatively, add the clean-up script to project shutdown programmatically from the Command Window:

```
addShutdownFile(proj_modelBasedDesignCodeGen, fullfile(pwd, '/utilities/clean_up_project.m'));
```

9. Reopen the project to run the **Startup** shortcut.

Alternatively, reload the project and run the `set_up_project` from the Command Window:

```
reload(proj_modelBasedDesignCodeGen);
run('set_up_project');
```

Generate Code for the Controller from the Dependency Graph

1. On the **Project** tab, click **Dependency Analyzer** to run a dependency analysis on all the files in your project.

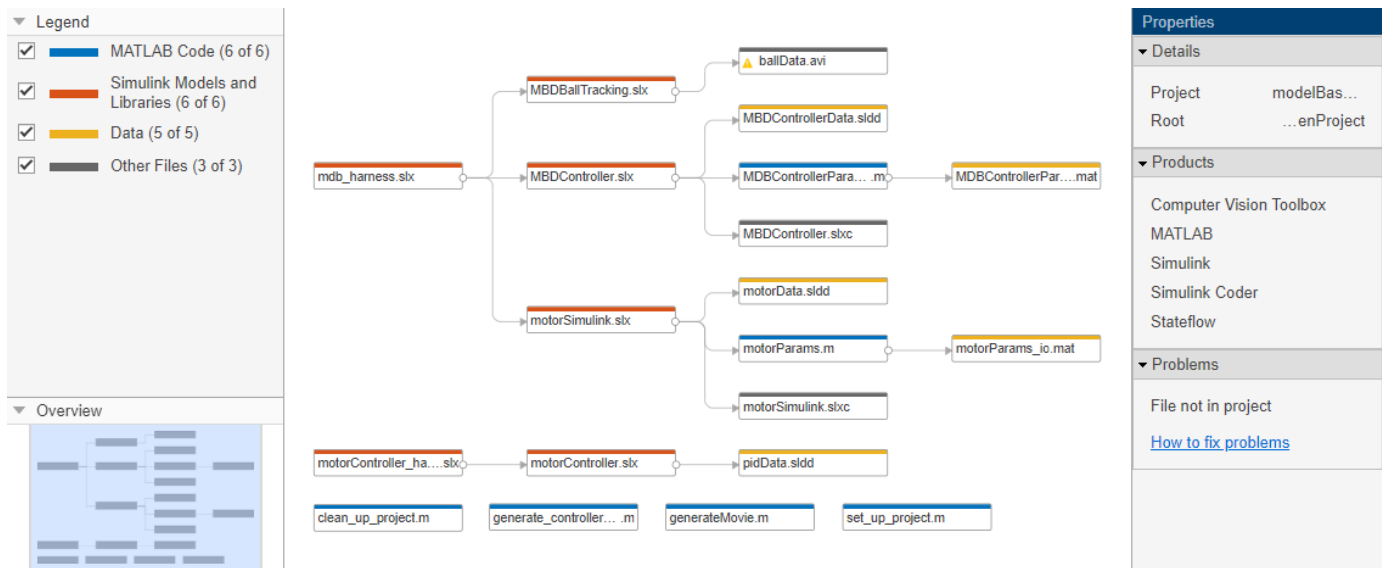
The dependency graph displays your project structure and the files dependencies. It shows how files relate to each other. Each item in the graph represents a file, and each arrow is a dependency.

The contents of the `work` folder do not appear in the dependency graph. This setup simplifies the view to indicate the dependencies in model files and `.m` script file.

a. Examine the **Properties** pane on the right for useful information about your project, such as required products and problem files.

b. Use the **Zoom In** button to zoom into the dependencies map. Use arrow keys or click-drag with the mouse roller wheel to move around in the map.

c. Hover over the dependency arrows to examine the dependency type. For example, the MATLAB® scripts and functions are being run by model callbacks.



2. Select the `MBDController.slx` model on the dependency map. Right-click the model and select **Open**.

3. When the model opens in Simulink, click the **Build** button to generate code for the model. Or, use the `generate_controller_code` shortcut in the project to generate code.

Alternatively, generate code for the model from the Command Window. For the sake of this example, set write permissions to `MBDController.slxc` then generate code for the model:

```
fileattrib('MBDController.slxc', '+w');
evalc('slbuild(''MBDController'');
```

4. Close the project from the Project window.

Alternatively, close this project and run `clean_up_project` from the Command Window:

```
run('clean_up_project');
close(proj_modelBasedDesignCodeGen);
```

See Also

`currentProject`, `openProject`

More Information

- “Work with Project Files”
- `Simulink.createFromTemplate`

- “Automate Startup Tasks”
- “Automate Shutdown Tasks”

Related Examples

- “Create a New Project from a Folder”
- “Create a Project from a Model”

Report Generation in Simulink Coder

- “Reports for Code Generation” on page 25-2
- “Generate Code Generation Report” on page 25-5
- “Document Generated Code with Simulink Report Generator” on page 25-7
- “Get Code Description of Generated Code” on page 25-13
- “Get Generated Type and Name of Inport Structures” on page 25-20
- “Analyze Generated Data Code Interface” on page 25-23

Reports for Code Generation

In this section...

“HTML Code Generation Report Content” on page 25-2

“HTML Code Generation Report Location” on page 25-2

“HTML Code Generation Report for Referenced Models” on page 25-2

“HTML Code Generation Report Extensions” on page 25-3

The code generator software produces an HTML code generation report so that you can view and analyze the generated code. When you build your model, the code generation process produces an HTML file that is displayed in an HTML browser or in the Model Explorer. For an example, see “Generate Code Generation Report” on page 25-5.

If you have Simulink Report Generator, you can document your code generation project in multiple formats, including HTML, PDF, RTF, Microsoft Word, and XML. For an example of how to create a Microsoft Word report, see “Document Generated Code with Simulink Report Generator” on page 25-7.

HTML Code Generation Report Content

The code generation report includes:

- The **Summary** section that contains model and code information, including **Author**, **Tasking Mode**, **System Target File**, **Hardware Device Type**, and code generation objectives information. The **Configuration settings at the time of code generation** link opens a noneditable view of the Configuration Parameters dialog box. The dialog box shows the Simulink model settings at the time of code generation, including TLC options.
- The **Subsystem Report** section that contains information on nonvirtual subsystems in the model.
- The **Code Interface Report** section that provides information about the generated code interface, including model entry-point functions and input/output data. For more information, see “Analyze Generated Data Code Interface” on page 25-23.
- In the **Code** section on the **Contents** pane, you can click the names of source code files generated from your model to view their contents.

HTML Code Generation Report Location

The default location for the code generation report files is in the `html` subfolder of the build folder, `model_target_rtw/html/`. `target` is the name of the **System target file** specified on the **Code Generation** pane. The default name for the top-level HTML report file is `index.html`. For more information on the location of the build folder, see “Manage Build Process Folders” on page 24-30.

HTML Code Generation Report for Referenced Models

To generate a code generation report for a top model and code generation reports for each referenced model, you need to specify the **Create code generation report** on the **Code Generation > Report** pane for the top model and each referenced model. You can open the code generation report of a referenced model in one of two ways:

- From the top-model code generation report, you can access the referenced model code generation report by clicking the name of the top model above the generated code. In the drop-down list of the model reference hierarchy, click the referenced model. To navigate back to the top model code generation report, use the **Back** button at the top of the left navigation pane or use the model reference hierarchy context menu.
- In the referenced model diagram window, on the **C Code** tab, click **Open Report**.

For more information, see “Generate Code for Model Reference Hierarchy” on page 8-2.

HTML Code Generation Report Extensions

If you have Embedded Coder, the code generator enhances the HTML code generation report. Configure your model to include these sections in the report:

- The **Traceability Report** section enables you to account for **Eliminated / Virtual Blocks** that are untraceable versus the listed **Traceable Simulink Blocks / Stateflow Objects / MATLAB Scripts**. This section provides a complete mapping between model elements and code. For more information, see “Customize Traceability Reports” (Embedded Coder).
- The **Static Code Metrics Report** section provides generated code statistics. Metrics are estimated from static analysis of the generated code. To generate a static code metrics report without generating a code generation report, use the function `coder.report.generateCodeMetrics`. For more information, see “Static Code Metrics” (Embedded Coder).
- The **Code Replacements Report** section enables you to account for code replacement library (CRL) functions that were used during code generation, providing a mapping between each replacement instance and the Simulink block that triggered the replacement. For more information, see “Analyze Code Replacements in Generated Code” (Embedded Coder).
- The **Coder Assumptions** section provides a list of:
 - Code generation assumptions for your target hardware that you can check.
 - Expected results for the assumption checks.

For more information, see “Check Code Generation Assumptions” (Embedded Coder).

- The model web view displays an interactive model diagram within the code generation report and supports traceability between the source code and the model. You can then share your model and generated code outside of the MATLAB environment. For more information, see “Web View of Model in Code Generation Report” (Embedded Coder).

On the **Contents** pane, in the **Generated Files** section, you can click the names of source code files generated from your model to view their contents in a MATLAB web browser window. In the displayed source code:

- If you enable code-to-model traceability, hyperlinks within the displayed source code navigate to the blocks or subsystems from which the code is generated. For more information, see “Code-to-Model Traceability” (Embedded Coder).
- If you enable model-to-code traceability, you can navigate to the generated code for a block in the model. For more information, see “Model-to-Code Traceability” (Embedded Coder).
- If you set the **Code coverage tool** parameter on the **Code Generation > Verification** pane, you can view the code coverage data and annotations. For more information, see “Configure Code Coverage with Third-Party Tools” (Embedded Coder).

- If you select the **Generate static code metrics** check box on the **Code Generation > Report** pane, you can view code metrics information and navigate to code definitions and declarations in the generated code. For more information, see “View Static Code Metrics and Definitions Within the Generated Code” (Embedded Coder).

Note To view the contents of your generated code and navigate between model and code, you can also use the Code view in the Code perspective. In the perspective, click the **Code** tab in the bottom-right corner.

See Also

Related Examples

- “Verify Generated Code by Using Code Tracing” (Embedded Coder)

Generate Code Generation Report

You can configure a model to generate a code generation report when you build the model. If you want to separate report generation from code generation, you can manually generate a report after generating code. If you do not change the model after generating code, you can generate a report without regenerating the code.

If you have Embedded Coder, you can include optional sections in the report, such as reports on traceability and code replacement. For more information, see “Reports for Code Generation” on page 25-2.

Configure Model

Configure a model to generate code without generating a report.

- 1 Open a model, for example, `rtwdemo_sum`.

```
model = 'rtwdemo_sum';  
open_system(model);
```
- 2 On the **C Code** tab, from the **Open Report** list, select **Report Options**. The Configuration Parameters dialog box opens with the **Code Generation > Report** pane visible.
- 3 Clear the **Create code generation report** parameter. When you generate code, the report is not generated. Instead, you generate the report manually.
- 4 If you have Embedded Coder, verify that **Eliminated / virtual blocks (Embedded Coder)** is selected. This parameter enables an optional **Traceability Report** section in the code generation report.
- 5 Generate code.

Generate Report

On the **C Code** tab, select **Open Report**. The report generation process writes the code generation report files to the `html` subfolder of the build folder (see “HTML Code Generation Report Location” on page 25-2) without regenerating the code. Next, a MATLAB web browser window opens and displays the code generation report.

If you selected **Eliminated / virtual blocks**, the code generation report includes a **Traceability Report** section.

To open the code generation report at any time after generating code, on the **C Code** tab, select **Open Report**. If a report already exists, it opens without regenerating the code or report.

Share Report

To share the code generation report, you can package the code generation report files and supporting files into a ZIP file for transfer.

- 1 In the **Current Folder** browser, select these folders:

- `/slprj`
 - `model_target_rtw`
- 2** To open the context menu, right click one of the selected folders.
 - 3** In the context menu, select **Create Zip File**. A file appears in the **Current Folder** browser.
 - 4** Name the ZIP file.

If your model is part of a project, you can include the code generation report when archiving the project. For more information, see “Archive Projects”.

Limitation

After building your model or generating the code generation report, if you modify legacy or custom code, you must rebuild your model or regenerate the report for the code generation report to include the updated legacy source files. For example, if you modify your legacy code, and then use the **Open Report** button to open an existing report, the software does not check if the legacy source file is out of date compared to the generated code. The code generation report is not regenerated and the report includes the out-of-date legacy code.

To reflect changes to legacy or custom code in the code generation report, rebuild the model before using the **Open Report** button.

See Also

More About

- “Reports for Code Generation” (Embedded Coder)
- “Configure and Generate Code Generation Report Programmatically” (Embedded Coder)

Document Generated Code with Simulink Report Generator

In this section...

- “Generate Code for the Model” on page 25-7
- “Open the Report Generator” on page 25-8
- “Set Report Name, Location, and Format” on page 25-10
- “Include Models and Subsystems in a Report” on page 25-11
- “Customize the Report” on page 25-11
- “Generate the Report” on page 25-12

The Simulink Report Generator software creates documentation from your model in multiple formats, including HTML, PDF, RTF, Microsoft Word, and XML. This example shows one way to document a code generation project in Microsoft Word. The generated report includes:

- System snapshots (model and subsystem diagrams)
- Block execution order list
- Simulink Coder and model version information for generated code
- List of generated files
- Optimization configuration parameter settings
- System target file selection and build process configuration parameter settings
- Subsystem map
- File name, path, and generated code listings for the source code

To adjust Simulink Report Generator settings to include custom code and then generate a report for a model, complete the following tasks:

- 1 “Generate Code for the Model” on page 25-7
- 2 “Open the Report Generator” on page 25-8
- 3 “Set Report Name, Location, and Format” on page 25-10
- 4 “Include Models and Subsystems in a Report” on page 25-11
- 5 “Customize the Report” on page 25-11
- 6 “Generate the Report” on page 25-12

A Simulink Report Generator license is required for the following report formats: PDF, RTF, Microsoft Word, and XML. For more information on generating reports in these formats, see the Simulink Report Generator documentation.

Generate Code for the Model

Before you use the Report Generator to document your project, generate code for the model.

- 1 In the MATLAB Current Folder browser, navigate to a folder where you have write access.
- 2 Create a working folder from the MATLAB command line by typing:

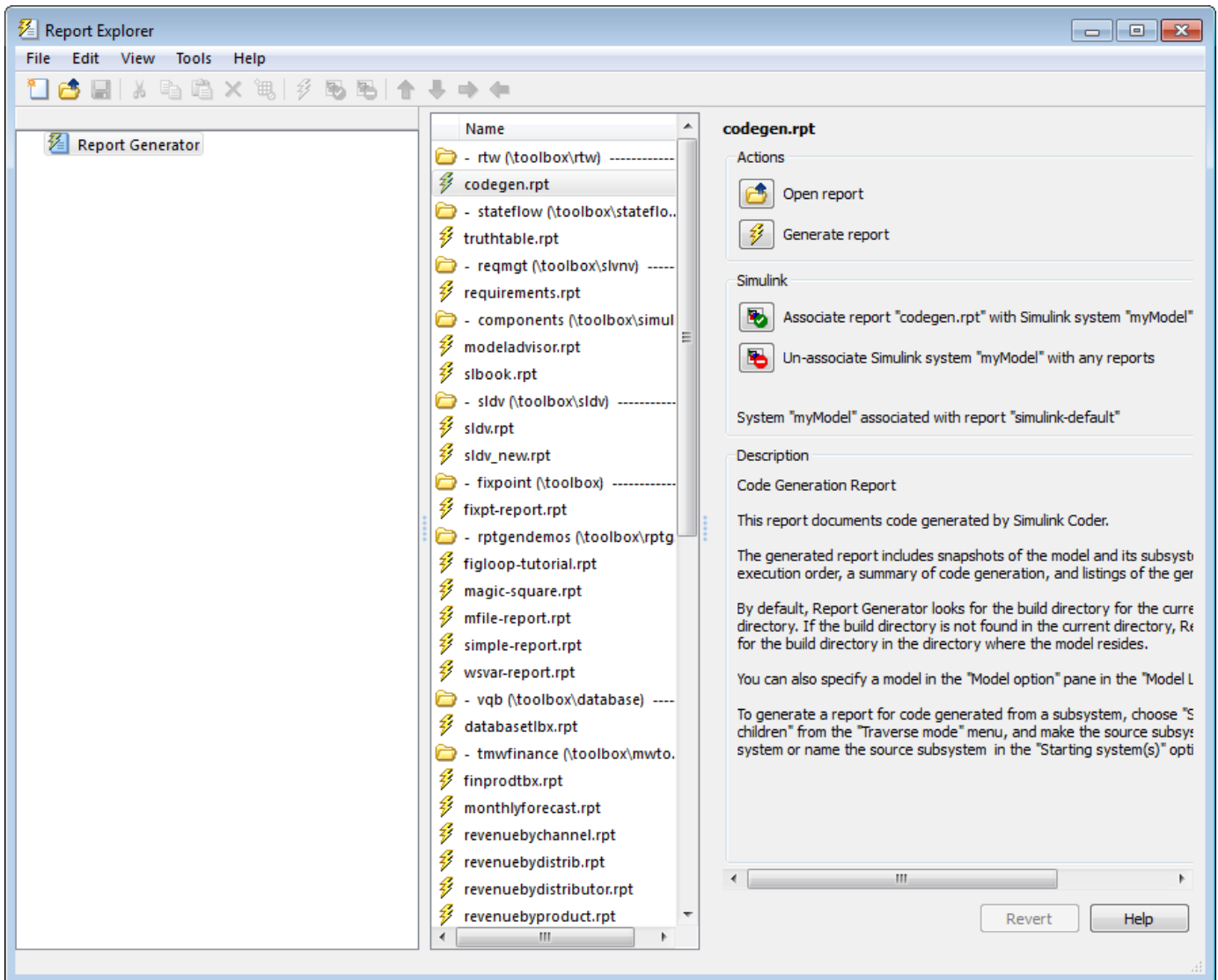
```
mkdir report_ex
```

- 3 Make `report_ex` your working folder:
`cd report_ex`
- 4 Open the `slexAircraftExample` model. In the Command Window, enter:
`openExample('slexAircraftExample')`
- 5 On the **Simulation** tab, in the **Save** list, select **Save as**. Navigate to the working folder, `report_ex`, and save a copy of the `slexAircraftExample` model as `myModel`.
- 6 Open the Configuration Parameters dialog box. Set the **Solver > Solver selection > Type** parameter to `Fixed-step`.
- 7 Select the **Code Generation** pane. Select **Generate code only**.
- 8 Click **Apply**.
- 9 In the model window, press **Ctrl+B**. The build process generates code for the model.

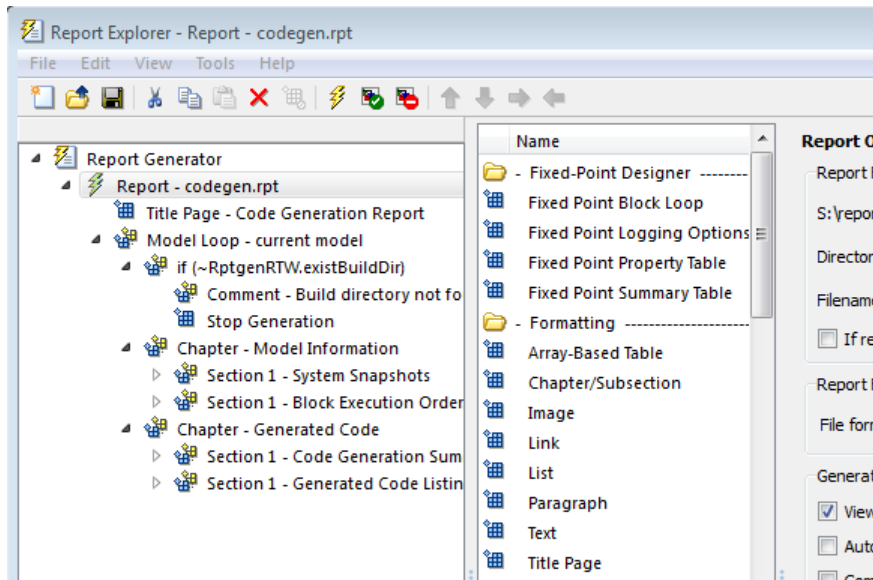
Open the Report Generator

After you generate the code, open the Report Generator.

- 1 Open the Report Generator app. In the model diagram window, in the **Apps** gallery, click **Report Generator**.
- 2 In the Report Explorer window, in the options pane (center), click the folder **rtw (\toolbox\rtw)**. Click the setup file that it contains, **codegen.rpt**.



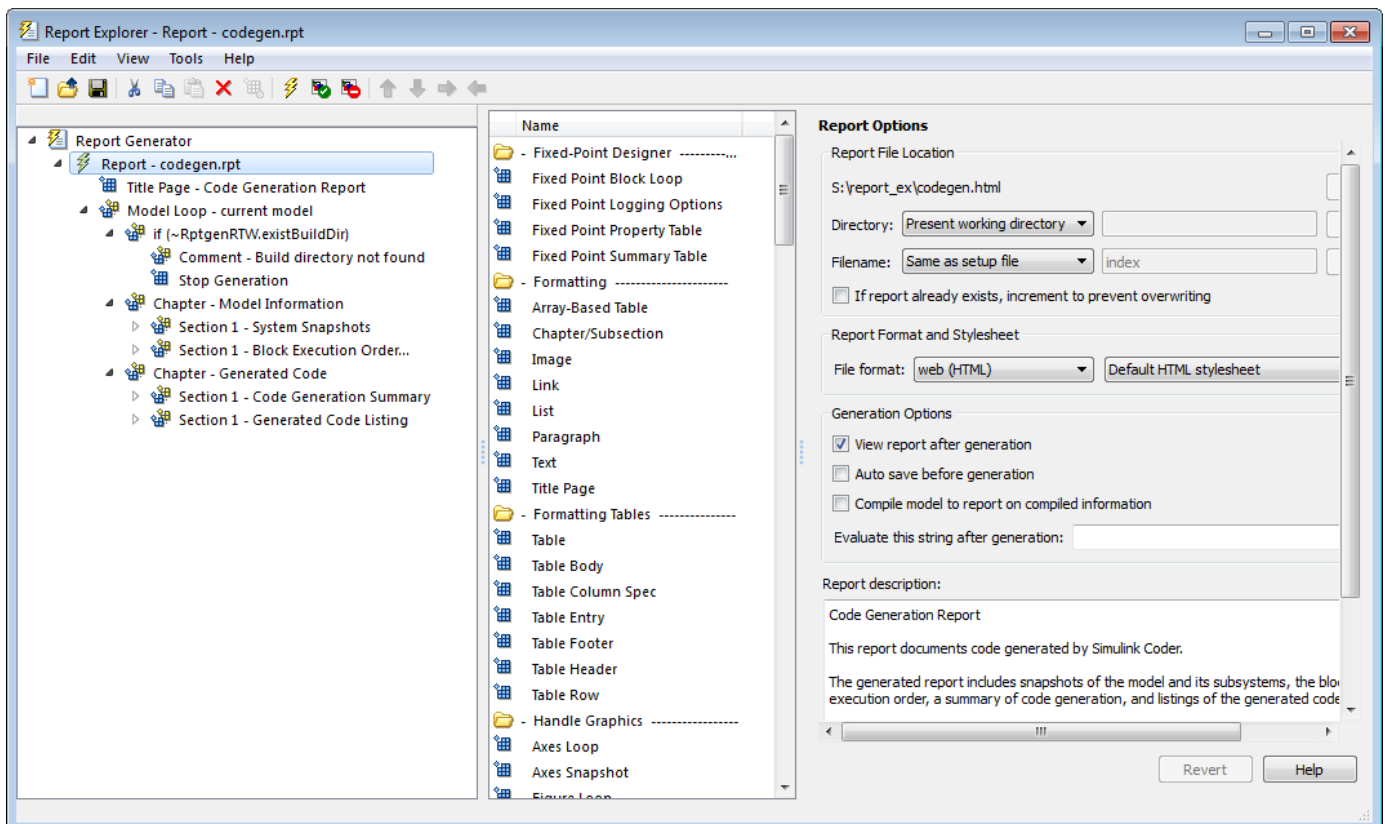
- 3 Double-click **codegen.rpt** or select it and click the **Open report** button. The Report Explorer displays the structure of the setup file in the outline pane (left).



Set Report Name, Location, and Format

Before generating a report, you can specify report output options, such as the folder, file name, and format. For example, to generate a Microsoft Word report named `MyCGModelReport.rtf`:

- 1 In the properties pane, under **Report Options**, review the options listed.

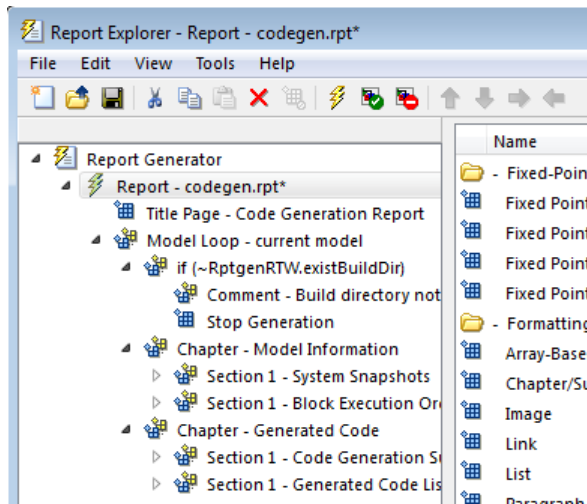


- 2 Leave the **Directory** field set to Present working directory.
- 3 For **Filename**, select Custom: and replace index with the name MyModelCGReport.
- 4 For **File format**, specify Rich Text Format and replace Standard Print with Numbered Chapters & Sections.

Include Models and Subsystems in a Report

Specify the models and subsystems that you want to include in the generated report by setting options in the Model Loop component.

- 1 In the outline pane (left), select **Model Loop**. Report Generator displays Model Loop component options in the properties pane.
- 2 If not already selected, select Current block diagram for the **Model name** option.
- 3 In the outline pane, click **Report - codegen.rpt***.



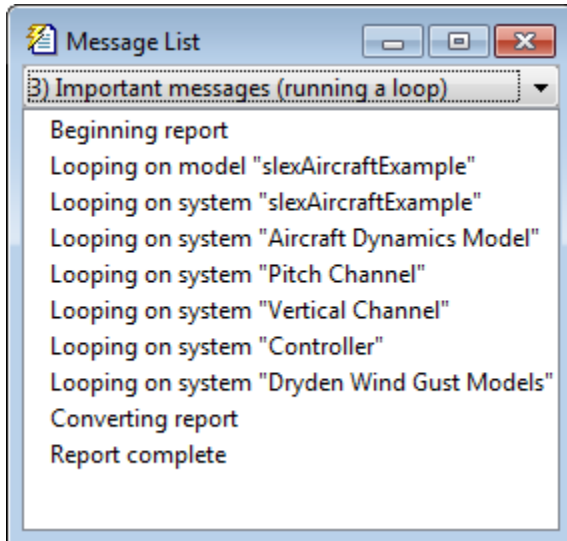
Customize the Report

After specifying the models and subsystems to include in the report, you can customize the sections included in the report.

- 1 In the outline pane (left), expand the node **Chapter - Generated Code**. By default, the report includes two sections, each containing one of two report components.
- 2 Expand the node **Section 1 — Code Generation Summary**.
- 3 Select **Code Generation Summary**. Options for the component are displayed in the properties pane.
- 4 Click **Help** to review the report customizations that you can make with the Code Generation Summary component. For this example, do not customize the component.
- 5 In the Report Explorer window, expand the node **Section 1 — Generated Code Listing**.
- 6 Select **Import Generated Code**. Options for the component are displayed in the properties pane.
- 7 Click **Help** to review the report customizations that you can make with the Import Generated Code component.

Generate the Report

After you adjust the report options, from the **Report Explorer** window, generate the report by clicking **File > Report**. A **Message List** dialog box opens, which displays messages that you can monitor as the report is generated. Model snapshots also appear during report generation. The **Message List** dialog box might be hidden behind other dialog boxes.



When the report is complete, open the report, `MyModelCGReport.rtf` in the folder `report_ex` (in this example).

See Also

More About

- "Reports for Code Generation" (Embedded Coder)
- "Generate Code Generation Report" (Embedded Coder)
- "Configure and Generate Code Generation Report Programmatically" (Embedded Coder)

Get Code Description of Generated Code

You can use the code descriptor API to obtain metadata about the generated code. For each model build, the code generator, by default, creates a `codedescriptor.dmr` file in the build folder. When simulating the model in Accelerator and Rapid Accelerator modes, the `codedescriptor.dmr` is not generated.

You can use the code descriptor API once the code is generated. Use the code descriptor API to describe these items in the generated code:

- Data Interfaces: inports, outports, parameters, data stores, and internal data.
- Function Interfaces: initialize, output, update, and terminate.
- Run-time information of the data and function interfaces, such as timing requirements of each interface entity.
- Model hierarchy information and code description of referenced models.

Get Data Interface Information

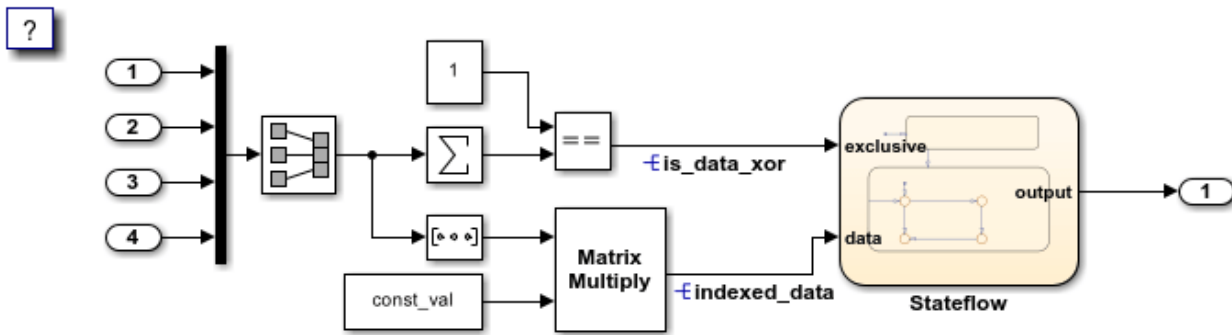
The `coder.descriptor.DataInterface` object describes various properties for a specified data interface in the generated code. In the model `rtwdemo_comments`, there are four inports, one outport, and a tunable external parameter. For more information about the data interfaces in your model, use the `coder.codedescriptor.CodeDescriptor` object and its methods.

1. Create a temporary folder for the build and inspection process.

```
currentDir = pwd;  
[~,cgDir] = rtwdemodir();
```

2. Open and build the model.

```
open_system('rtwdemo_comments');  
evalc('slbuild(''rtwdemo_comments'')');
```



- ▶ Simulink Block
- ▶ Stateflow State
- ▶ Stateflow Transition
- ▶ Simulink Signal Object
- ▶ Simulink Parameter Object
- ▶ MPT Signal Object
- ▶ MPT Callback Function

- ▶ Open International Example
- ▶ Open Comments Settings
- ▶ Comments Documentation
- ▶ Customize banners via code templates

Generate Code Using
Embedded Coder
(double-click)

DOC
Text
History

Copyright 1994–2020 The MathWorks, Inc.

3. Create a `coder.codedescriptor.CodeDescriptor` object for the required model by using the `getCodeDescriptor` function.

```
codeDescriptor = coder.getCodeDescriptor('rtwdemo_comments');
```

4. To obtain a list of all the data interface types in the generated code, use the `getDataInterfaceTypes` method.

```
dataInterfaceTypes = codeDescriptor.getDataInterfaceTypes()
```

```
dataInterfaceTypes =
```

```
5x1 cell array
```

```
{'Inports'           }
{'Outports'          }
{'Parameters'        }
{'ExternalParameterObjects'}
{'InternalData'      }
```

To obtain a list of all the supported data interfaces, use the `getAllDataInterfaceTypes` method.

5. To obtain more information about a particular data interface type, use the `getDataInterfaces` method.

```
dataInterface = codeDescriptor.getDataInterfaces('Inports');
```

This method returns properties of the Inport blocks in the generated code.

6. Because this model has four inports, `dataInterface` is an array of `coder.descriptor.DataInterface` objects. Obtain the details of the first Inport of the model by accessing the first location in the array.

```
dataInterface(1)
```

```
ans =
```

```
DataInterface with properties:
    Type: [1x1 coder.descriptor.types.Type]
        SID: 'rtwdemo_comments:99'
    GraphicalName: 'In1'
    VariantInfo: [1x0 coder.descriptor.VariantInfo]
    Implementation: [1x1 coder.descriptor.DataImplementation]
    Timing: [1x1 coder.descriptor.TimingInterface]
    Unit: ''
    Range: [1x0 coder.descriptor.Range]
```

Get Function Interface Information

The function interfaces are the entry-point functions in the generated code. In the model `RollAxisAutopilot`, the entry-point functions are `model_initialize`, `model_step`, and `model_terminate`. For more information about the function interfaces in your model, use the `coder.descriptor.FunctionInterface` object.

2. Open and build the model.

```
cd(currentDir)
open_system('RollAxisAutopilot')
slbuild('RollAxisAutopilot')

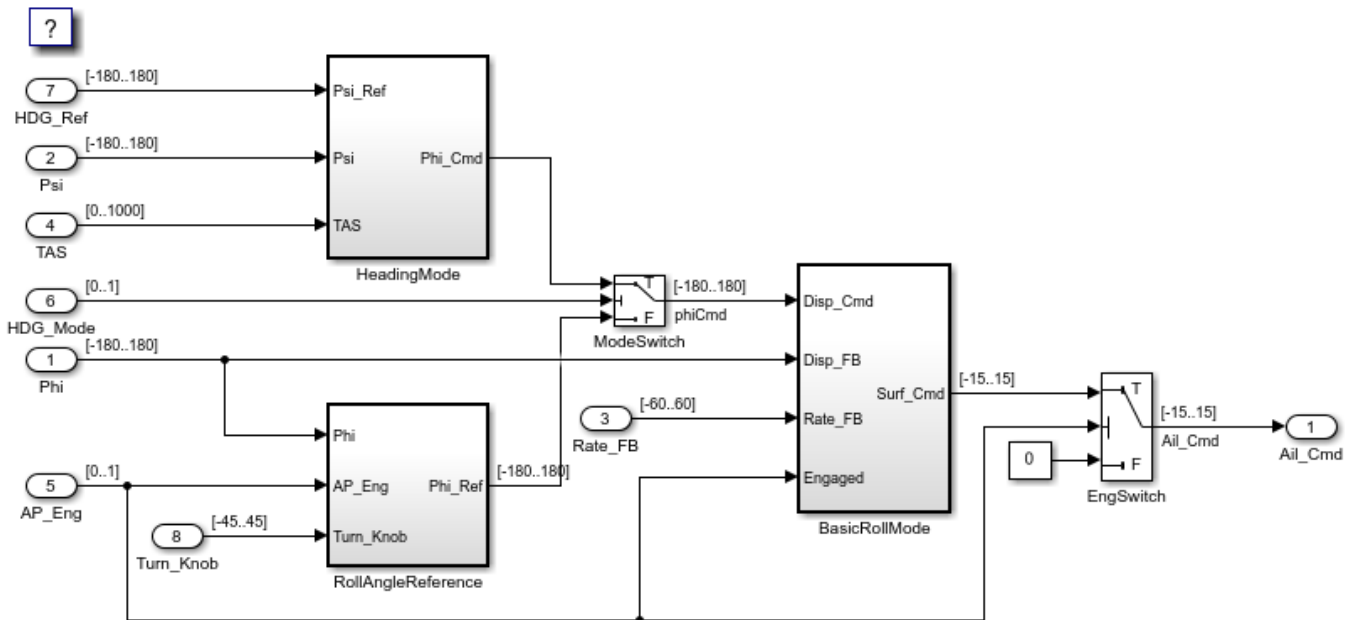
### Starting build procedure for: RollAxisAutopilot
### Successful completion of build procedure for: RollAxisAutopilot

Build Summary

Top model targets built:

Model          Action          Rebuild Reason
=====
RollAxisAutopilot  Code generated and compiled.  Code generation information file does not exist

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 13.721s
```



Copyright 1990-2020 The MathWorks, Inc.

3. Create a `coder.codedescriptor.CodeDescriptor` object for the required model by using the `getCodeDescriptor` function.

```
codeDescriptor = coder.getCodeDescriptor('RollAxisAutopilot');
```

4. To obtain a list of all the function interface types in the generated code, use the `getFunctionInterfaceTypes` method.

```
functionInterfaceTypes = codeDescriptor.getFunctionInterfaceTypes()
```

```
functionInterfaceTypes =
```

```
2x1 cell array
```

```
 {'Initialize'}  
 {'Output' }
```

To obtain a list of all the supported function interfaces, use the `getAllFunctionInterfaceTypes` method.

5. To obtain more information about a particular function interface type, use the `getFunctionInterfaces` method.

```
functionInterface = codeDescriptor.getFunctionInterfaces('Initialize')
```

```
functionInterface =
```

```
FunctionInterface with properties:  
  Prototype: [1x1 coder.descriptor.types.Prototype]
```



```

ActualReturn: [1x0 coder.descriptor.DataInterface]
VariantInfo: [1x0 coder.descriptor.VariantInfo]
FunctionOwner: [1x0 coder.descriptor.TypedRegion]
    Timing: [1x1 coder.descriptor.TimingInterface]
ActualArgs: [1x0 coder.descriptor.DataInterface Sequence]

```

6. You can further expand on the properties to obtain detailed information. To get the function return value, name, and arguments:

```
functionInterface.Prototype
```

```
ans =
```

```

Prototype with properties:
    Name: 'RollAxisAutopilot_initialize'
    Return: [1x0 coder.descriptor.types.Argument]
    HeaderFile: 'RollAxisAutopilot.h'
    SourceFile: 'RollAxisAutopilot.c'
    Arguments: [1x0 coder.descriptor.types.Argument Sequence]

```

Get Model Hierarchy Information

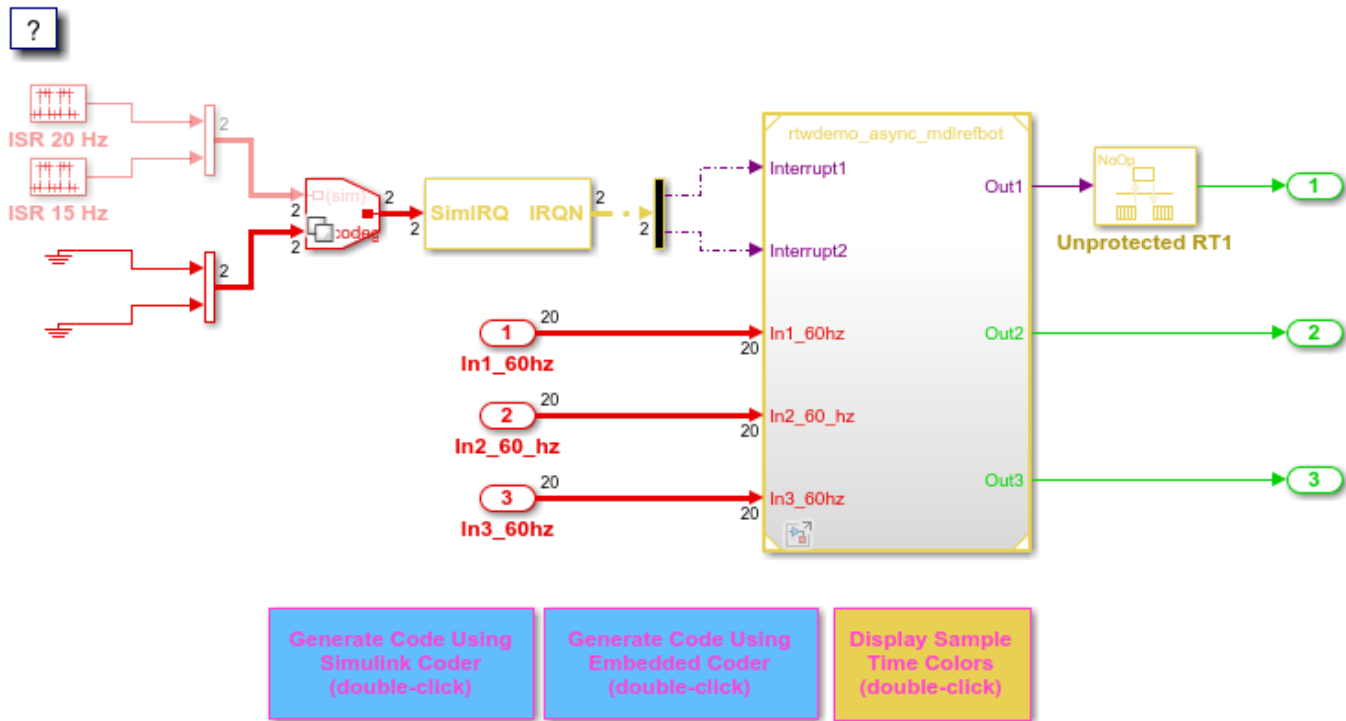
Use the `coder.codedescriptor.CodeDescriptor` object to get the entire model hierarchy information. The model `rtwdemo_async_mdltreftop` has model `rtwdemo_async_mdltreftbot` as the referenced model.

1. Create a temporary folder for the build and inspection process.

```
currentDir = pwd;
[~,cgDir] = rtwdemodir();
```

2. Open and build the model.

```
open_system('rtwdemo_async_mdltreftop');
evalc('slbuild(''rtwdemo_async_mdltreftop'')');
```



Copyright 2010-2020 The MathWorks, Inc.

3. Create a `coder.CodeDescriptor` object for the required model by using the `getCodeDescriptor` function.

```
codeDescriptor = coder.getCodeDescriptor('rtwdemo_async_mdrefbot');
```

4. Get a list of all the referenced models by using the `getReferencedModelNames` method.

```
refModels = codeDescriptor.getReferencedModelNames()
```

```
refModels =
```

```
1x1 cell array
```

```
 {'rtwdemo_async_mdrefbot'}
```

5. To obtain the `coder.CodeDescriptor` object for the referenced model, use the `getReferencedModelCodeDescriptor` method.

```
refCodeDescriptor = codeDescriptor.getReferencedModelCodeDescriptor('rtwdemo_async_mdrefbot');
```

You can now use the `refCodeDescriptor` object to obtain more information about the referenced model by using all the available methods in the Code Descriptor API.

See Also

Related Examples

- “Analyze Generated Data Code Interface” (Embedded Coder)
- `coder.codedescriptor.CodeDescriptor`
- “Get Generated Type and Name of Inport Structures” on page 25-20

Get Generated Type and Name of Inport Structures

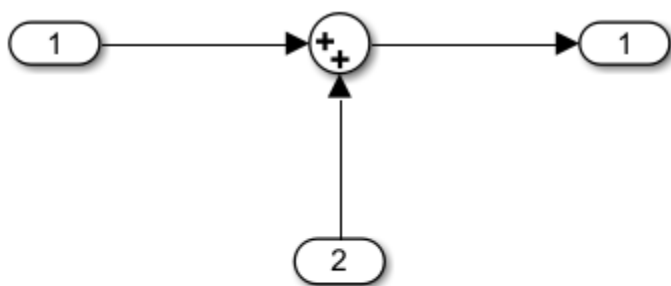
To obtain the generated type and name of inport and outport structures, use the code descriptor API. The code generator stores this information within a `DataImplementation` object field on a `DataInterface` object when it generates the code.

To interface your generated code with other components of your application, use the code descriptor API.

Open and Build Model

Open the model.

```
open_system('rtwdemo_sum');
```



Copyright 2015-2018 The MathWorks, Inc.

Build the model.

```
evalc('slbuild(''rtwdemo_sum'')');
```

The generated code includes a structure representing the inports.

```
/* External inputs (root inport signals with default storage) */
ExtU rtU;
```

Get Code Descriptor and Inport Structure Names and Types

Get the code descriptor object.

```
codeDescObj = coder.getCodeDescriptor('rtwdemo_sum')
```

```
codeDescObj =
```

```
CodeDescriptor with properties:
```

```
ModelName: 'rtwdemo_sum'
```

```
BuildDir: 'C:\TEMP\Bdoc23a_2213998_3568\ib570499\4\tp50229dff\simulinkcoder-ex78186698\rtwdemo_sum'
```

Get the list of data interface types in the generated code.

```
dataInterfaceTypes = codeDescObj.getDataInterfaceTypes()
```

```
dataInterfaceTypes = 2x1 cell
    {'Inports' }
    {'Outports'}
```

Get the `DataInterface` object that represents the inports.

```
inports = codeDescObj.getDataInterfaces('Inports')
```

```
inports =
    1x2 DataInterface array with properties:
```

```
    Type
    SID
    GraphicalName
    VariantInfo
    Implementation
    Timing
    Unit
    Range
```

Get the `Implementation` object that represents the first inport. In this example, the inports are each represented as a `StructExpression`, which is a subclass of `DataImplementation`.

```
impl = inports(1).Implementation
```

```
impl =
    StructExpression with properties:
        Type: [1x1 coder.descriptor.types.Type]
        BaseRegion: [1x1 coder.descriptor.TypedRegion]
        ElementIdentifier: 'In1'
        Variant: ''
```

Get the `Type` of the first inport. The `Type` of the inport is `real_T`.

```
implType = impl.Type
```

```
implType =
    Double with properties:
        Identifier: 'real_T'
        Name: 'double'
        ReadOnly: 0
        Volatile: 0
        WordLength: 64
        Signedness: 1
```

Get the `BaseRegion` of the first inport. `BaseRegion` represents the structure that contains the inport. The `BaseRegion` is `rtU`.

```
baseRegion = impl.BaseRegion
```

```
baseRegion =
    Variable with properties:
        Type: [1x1 coder.descriptor.types.Type]
        Identifier: 'rtU'
        VarOwner: 'rtwdemo_sum'
```

```
StorageSpecifier: ''
```

Get the Type of the BaseRegion. The Type of the BaseRegion is ExtU.

```
baseRegionType = baseRegion.Type
```

```
baseRegionType =  
  Struct with properties:  
    Identifier: 'ExtU'  
    Name: 'ExtU'  
    ReadOnly: 0  
    Volatile: 0  
    Elements: [1x2 coder.descriptor.types.AggregateElement Sequence]  
    Checksum: [1x0 Integer Sequence]
```

Close Model

```
bdclose('rtwdemo_sum');
```

See Also

Related Examples

- “Get Code Description of Generated Code” on page 25-13

Analyze Generated Data Code Interface

In this section...

- “Navigating Code Interface Report Subsections” on page 25-23
- “Interpreting the Entry-Point Functions Subsection” on page 25-24
- “Interpreting the Inports and Outports Subsections” on page 25-27
- “Interpreting the Interface Parameters Subsection” on page 25-28
- “Interpreting the Data Stores Subsection” on page 25-29
- “Code Interface Report Limitations” on page 25-29

When you generate a code generation report, you can include a code interface report, which documents the generated code interface including model entry-point functions and interface data. The code interface information helps you to review the generated code and integrate it with other code. The information in the report depends on the type of code interface configuration that you use. For models that use service interface code configuration, the code interface report also includes information about services, as described in “Analyze Generated Service Code Interface Report” (Embedded Coder).

For code configured with data interfaces, the code interface report includes the following subsections:

- **Entry-Point Functions** — interface information about each model entry-point function, including `model_initialize`, `model_step`, and (if applicable) `model_reset` and `model_terminate`.
- **Inports and Outports** — interface information about each model inport and output.
- **Interface Parameters** — interface information about tunable parameters that are associated with the model.
- **Data Stores** — interface information about global data stores and data stores with non-auto storage that are associated with the model.

To generate a code interface report, select the configuration parameters **Create code generation report** and **Code Interface Report**, as described in “Generate Code Generation Report” on page 25-5. For limitations that apply to code interface reports, see “Code Interface Report Limitations” on page 25-29.

For illustration purposes, this section uses the following models:

- `rtwdemo_configinterface` for examples of report subsections
- `rtwdemo_mrmtbb` for examples of timing information
- `rtwdemo_fcncnprotctrl` for examples of function argument and return value information

Navigating Code Interface Report Subsections

To help you navigate code interface descriptions, the code interface report provides collapse/expand tokens and hyperlinks.

- For a large subsection, the report provides [-] and [+] symbols that allow you to collapse or expand that section.
- The code interface report provides several forms of hyperlink navigation. For example:

- The **Table of Contents** located at the top of the code interface report provides links to each subsection.
- You can click each function name to go to its definition in *model.c*.
- You can click the header file name for each function to go to the header file source listing.
- If you selected the configuration parameter **Code-to-model**, to go to the corresponding location in the model display, you can click hyperlinks for these constructs:
 - Function argument
 - Function return value
 - Variable representing an inport
 - Variable representing an outport
 - Interface parameter (if the parameter source is a block)
 - Variable representing a data store (if the data store source is a Data Store Memory block)

For backward and forward navigation within the HTML code generation report, use the **Back** and **Forward** buttons above the **Contents** section in the upper-left corner of the report.

Interpreting the Entry-Point Functions Subsection

The **Entry-Point Functions** subsection of the code interface report provides the following interface information about each model entry-point function, including `model_initialize`, `model_step`, and (if applicable) `model_reset` and `model_terminate`.

Field	Description
Function	The function name. You can click the function name to go to its definition in <i>model.c</i> .
Prototype	Displays the function prototype, including the function return value, name, and arguments.
Description	Provides a text description of the purpose of the function in the application.
Timing	Describes the timing characteristics of the function, such as how many times the function is called, or if it is called periodically, and at what time interval. For a multirate timing example, see the following <code>rtwdemo_mrmtbb</code> report excerpt.
Arguments	If the function has arguments, displays the number, name, data type, and Simulink description for each argument. For argument examples, see the <code>rtwdemo_fcnprotoctrl</code> report excerpt below.
Return value	If the function has a return value, this field displays the return value data type and Simulink description. If you selected the Code-to-model parameter for your model, you can click the hyperlink in the description to go to the block corresponding to the return value in the model display. For a return value example, see the following <code>rtwdemo_fcnprotoctrl</code> report excerpt.
Header file	The name of the header file for the function. You can click the header file name to go to the header file source listing.

For example, here is the **Entry-Point Functions** subsection for the model `rtwdemo_configinterface`.

Entry-Point Functions

Function: [rtwdemo_configinterface_initialize](#)

Prototype	<code>void rtwdemo_configinterface_initialize(void)</code>
Description	Initialization entry point of generated code
Timing	Must be called exactly once
Arguments	None
Return value	None
Header file	rtwdemo_configinterface.h

Function: [rtwdemo_configinterface_step](#)

Prototype	<code>void rtwdemo_configinterface_step(void)</code>
Description	Output entry point of generated code
Timing	Must be called periodically, every 1 second
Arguments	None
Return value	None
Header file	rtwdemo_configinterface.h

Function: [rtwdemo_configinterface_terminate](#)

Prototype	<code>void rtwdemo_configinterface_terminate(void)</code>
Description	Termination entry point of generated code
Timing	Must be called exactly once
Arguments	None
Return value	None
Header file	rtwdemo_configinterface.h

To illustrate how timing information might be listed for a multirate model, here are the **Entry-Point Functions** and **Inports** subsections for the model `rtwdemo_mrrmtbb`. This multirate, discrete-time, multitasking model contains Inport blocks 1 and 2, which specify 1-second and 2-second sample times, respectively. The sample times are constrained to the specified times by the **Periodic sample time constraint** option on the **Solver** pane of the Configuration Parameters dialog box.

Entry-Point Functions

Function: [rtwdemo_mrmtbb_initialize](#)

Prototype	void rtwdemo_mrmtbb_initialize(void)
Description	Initialization entry point of generated code
Timing	Must be called exactly once
Arguments	None
Return value	None
Header file	rtwdemo_mrmtbb.h

Function: [rtwdemo_mrmtbb_step0](#)

Prototype	void rtwdemo_mrmtbb_step0(void)
Description	Output entry point of generated code
Timing	Must be called periodically, every 1 second
Arguments	None
Return value	None
Header file	rtwdemo_mrmtbb.h

Function: [rtwdemo_mrmtbb_step1](#)

Prototype	void rtwdemo_mrmtbb_step1(void)
Description	Output entry point of generated code
Timing	Must be called periodically, every 2 seconds
Arguments	None
Return value	None
Header file	rtwdemo_mrmtbb.h

Inports

Block Name	Code Identifier	Data Type	Dimension
<Root>/In1_1s	rtU.In1_1s	real_T	1
<Root>/In2_2s	rtU.In2_2s	real_T	1

To illustrate how function arguments and return values are displayed in the report, here is the entry-point function description of the model step function for model `rtwdemo_fcncnprotctrl`.

Function: [rtwdemo_fcncnprotctrl_step_custom](#)

Prototype	boolean_T rtwdemo_fcncnprotctrl_step_custom(const real_T argIn1, const BusObject *const argIn2, BusObject *argOut2, const BusObject *const argIn3, uint8_T *argIn4)		
Description	Output entry point of generated code		
Timing	Can be called at any time		
Arguments	[-]		
	#	Name	Data Type
	1	argIn1	const real_T
	2	argIn2	const BusObject *const
	3	argOut2	BusObject *
	4	argIn3	const BusObject *const
	5	argIn4	uint8_T *
			Description
			<Root>/In1
			<Root>/In2
			<Root>/Out2
			<Root>/In3
			<Root>/In4
Return value	Data Type	Description	
	boolean_T	<Root>/Out1	
Header file	rtwdemo_fcncnprotctrl.h		

Interpreting the Inports and Outports Subsections

The **Inports** and **Outports** subsections of the code interface report provide the following interface information about each inport and output in the model.

Field	Description
Block Name	Displays the Simulink block name of the inport or output. If you selected the Code-to-model parameter for your model, you can click on each inport or output Block Name value to go to its location in the model display.
Code Identifier	Lists the identifier associated with the inport or output data in the generated code, as follows: <ul style="list-style-type: none"> • If the data is defined in the generated code, the field displays the identifier text. • If the data is declared but not defined in the generated code — for example, if the data is resolved with an imported storage class — the field displays the identifier text prefixed with the label 'Imported data:'. • If the data is neither defined nor declared in the generated code — for example, if model configuration parameter Code interface packaging is set to Reusable function — the field displays the text 'Defined externally'.
Data Type	Lists the data type of the inport or output.
Scaling	For fixed-point entries, lists the data type and fraction length using Simulink fixed-point data type notation. <p>Note You must have a Fixed-Point Designer license to see fixed-point scaling information in the report. For more information on how scaling is represented in the table, see “Fixed-Point Data Type and Scaling Notation” (Fixed-Point Designer).</p>
Dimension	Lists the dimensions of the inport or output (for example, 1 or [4, 5]).

For example, here are the **Inports** and **Outports** subsections for the model `rtwdemo_configinterface`.

Inports

[-]

Block Name	Code Identifier	Data Type	Dimension
<Root>/In1	Imported data: input1	MYTYPE	1
<Root>/In2	Imported data: input2	MYTYPE	1
<Root>/In3	Imported data: input3	MYTYPE	1
<Root>/In4	Imported data: input4	MYTYPE	1

Outports

Block Name	Code Identifier	Data Type	Dimension
<Root>/Out1	output	MYTYPE	1

Interpreting the Interface Parameters Subsection

The **Interface Parameters** subsection of the code interface report provides the following interface information about tunable parameters that are associated with the model.

Field	Description
Parameter Source	Lists the source of the parameter value, as follows: <ul style="list-style-type: none"> If the source of the parameter value is a block, the field displays the block name, such as <Root>/Gain2 or <S1>/Lookup1. If you selected the Code-to-model parameter for your model, you can click the Parameter Source value to go to the parameter's location in the model display. If the source of the parameter value is a workspace variable, the field displays the name of the workspace variable.
Code Identifier	Lists the identifier associated with the tunable parameter data in the generated code, as follows: <ul style="list-style-type: none"> If the data is defined in the generated code, the field displays the identifier text. If the data is declared but not defined in the generated code — for example, if the data is resolved with an imported storage class — the field displays the identifier text prefixed with the label 'Imported data:'. If the data is neither defined nor declared in the generated code — for example, if model configuration parameter Code interface packaging is set to Reusable function — the field displays the text 'Defined externally'.
Data Type	Lists the data type of the tunable parameter.
Scaling	For fixed-point entries, lists the data type and fraction length using Simulink fixed-point data type notation. <p>Note You must have a Fixed-Point Designer license to see fixed-point scaling information in the report. For more information on how scaling is represented in the table, see “Fixed-Point Data Type and Scaling Notation” (Fixed-Point Designer).</p>
Dimension	Lists the dimensions of the tunable parameter (for example, 1 or [4, 5, 6]).

For example, here is the **Interface Parameters** subsection for the model `rtwdemo_configinterface`.

Interface Parameters

[-]

Parameter Source	Code Identifier	Data Type	Dimension
'Table1' in model workspace	Table1	const volatile Table1_Type	1
'Table2' in model workspace	Table2	const volatile Table2_Type	1
'K1' in model workspace	K1	const volatile int8_T	1

Interpreting the Data Stores Subsection

The **Data Stores** subsection of the code interface report provides the following interface information about global data stores and data stores with non-auto storage that are associated with the model.

Field	Description
Data Store Source	Lists the source of the data store memory, as follows: <ul style="list-style-type: none"> If the data store is defined using a Data Store Memory block, the field displays the block name, such as <Root>/DS1. If you selected the Code-to-model parameter for your model, you can click on the Data Store Source value to go to the data store's location in the model display. If the data store is defined using a Simulink.Signal object, the field displays the name of the Simulink.Signal object.
Code Identifier	Lists the identifier associated with the data store data in the generated code, as follows: <ul style="list-style-type: none"> If the data is defined in the generated code, the field displays the identifier text. If the data is declared but not defined in the generated code — for example, if the data is resolved with an imported storage class — the field displays the identifier text prefixed with the label 'Imported data:'. If the data is neither defined nor declared in the generated code — for example, if model configuration parameter Code interface packaging is set to Reusable function — the field displays the text 'Defined externally'.
Data Type	Lists the data type of the data store.
Scaling	For fixed-point entries, lists the data type and fraction length using Simulink fixed-point data type notation. <p>Note You must have a Fixed-Point Designer license to see fixed-point scaling information in the report. For more information on how scaling is represented in the table, see “Fixed-Point Data Type and Scaling Notation” (Fixed-Point Designer).</p>
Dimension	Lists the dimensions of the data store (for example, 1 or [1, 2]).

For example, here is the **Data Stores** subsection for the model `rtwdemo_configinterface`.

Data Stores

Data Store Source	Code Identifier	Data Type	Dimension
<Root>/Data Store Memory	mode	volatile boolean_T	1

Code Interface Report Limitations

These limitations apply to the code interface section of the HTML code generation reports.

- The code interface report supports data resolved with most storage classes, except when the storage class properties are set in one of these ways:
 - The storage class property **Type** is set to `FlatStructure`. For example, when the `BitField` and `Struct` storage classes in the Simulink package have **Type** set to `FlatStructure`.
 - The storage class property **Type** is set to `Other`. For example, when the `GetSet` storage class in the Simulink package has **Type** set to `Other`.
 - The storage class property **Data access** is set to `Pointer`, indicating that the code declares imported symbols as pointer variables rather than simple variables. This property is accessible only when the storage class property **Data scope** is set to `Imported` or `Instance-specific`.

In these cases, the report displays empty **Data Type** and **Dimension** fields.

- For outputs, if you optimize memory, the code interface report cannot describe the associated memory (data type and dimensions). In these cases, the report displays empty **Data Type** and **Dimension** fields.
- The code interface report does not support data type replacement using the **Code Generation > Data Type Replacement** pane of the Configuration Parameters dialog box. The data types in the report link to built-in data types rather than their specified replacement data types.

See Also

Related Examples

- “How Generated Code Exchanges Data with an Environment” (Embedded Coder)

Code Replacement in Simulink Coder

- “What Is Code Replacement?” on page 26-2
- “Choose a Code Replacement Library” on page 26-6
- “Replace Code Generated from Simulink Models” on page 26-8

What Is Code Replacement?

Code replacement is a technique to change the code that the code generator produces for functions and operators to meet application code requirements. For example, you can replace generated code to meet requirements such as:

- Optimization for a specific run-time environment, including, but not limited to, specific target hardware.
- Integration with existing application code.
- Compliance with a standard, such as AUTOSAR.
- Modification of code behavior, such as enabling or disabling nonfinite or inline support.
- Application- or project-specific code requirements, such as:
 - Elimination of `math.h`.
 - Elimination of system header files.
 - Elimination of calls to `memcpy` or `memset`.
 - Use of BLAS.
 - Use of a specific BLAS.

To apply this technique, configure the code generator to apply a code replacement library (CRL) during code generation. By default, the code generator does not apply a code replacement library. You can choose from libraries that MathWorks provides and that you create and register by using the Embedded Coder product. The list of available libraries depends on:

- Installed support packages.
- System target file, language, language standard, and device vendor configuration.
- Whether you have created and registered libraries, using the Embedded Coder product.

Libraries that include GNU99 extensions are intended for use with the GCC compiler. If you use one of those libraries with another compiler, generated code might not compile.

Code Replacement Libraries

A code replacement library consists of one or more code replacement tables that specify application-specific implementations of functions and operators. For example, a library for a specific embedded processor specifies function and operator replacements that optimize generated code for that processor.

A code replacement table contains one or more code replacement entries, with each entry representing a potential replacement for a function or operator. Each entry maps a conceptual representation of a function or operator to an implementation representation and priority.

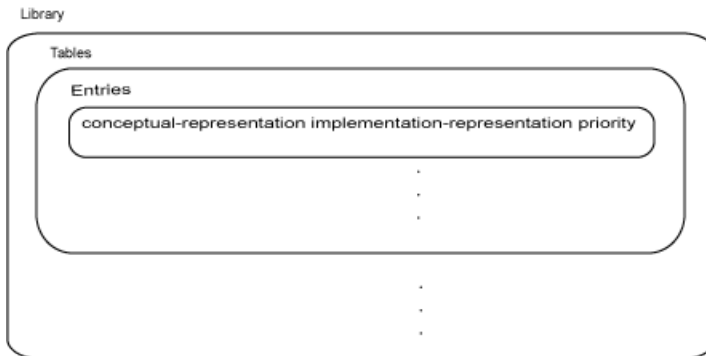


Table Entry Component	Description
Conceptual representation	<p>Identifies the table entry and contains match criteria for the code generator. Consists of:</p> <ul style="list-style-type: none"> • Function name or a key. The function name identifies most functions. For operators and some functions, a series of characters, called a key identifies a function or operator. For example, function name 'cos' and operator key 'RTW_OP_ADD'. • Conceptual arguments that observe code generator naming ('y1', 'u1', 'u2', ...), with corresponding I/O types (output or input) and data types. • Other attributes, such as an algorithm, fixed-point saturation, and rounding modes, which identify matching criteria for the function or operator.
Implementation representation	<p>Specifies replacement code. Consists of:</p> <ul style="list-style-type: none"> • Function name. For example, 'cos_dbl' or 'u8_add_u8_u8'. • Implementation arguments, with corresponding I/O types (output or input) and data types. • Parameters that provide additional implementation details, such as header and source file names and paths of build resources.
Priority	<p>Defines the entry priority relative to other entries in the table. The value can range from 0 to 100, with 0 being the highest priority. If multiple entries have the same priority, the code generator uses the first match with that priority.</p>

When the code generator looks for a match in a code replacement library, it creates and populates a call site object with the function or operator conceptual representation. If a match exists, the code generator uses the matched code replacement entry populated with the implementation representation and uses it to generate code.

The code generator searches the tables in a code replacement library for a match in the order that the tables appear in the library. If the code generator finds multiple matches within a table, the priority determines the match. The code generator uses a higher-priority entry over a similar entry with a lower priority.

Code Replacement Terminology

Term	Definition
Cache hit	A code replacement entry for a function or operator, defined in the specified code replacement library, for which the code generator finds a match.
Cache miss	A conceptual representation of a function or operator for which the code generator does not find a match.
Call site object	Conceptual representation of a function or operator that the code generator uses when it encounters a call site for a function or operator. The code generator uses the object to query the code replacement library for a conceptual representation match. If a match exists, the code generator returns a code replacement object, fully populated with the conceptual representation, implementation representation, and priority, and uses that object to generate replacement code.
Code replacement library	One or more code replacement tables that specify application-specific implementations of functions and operators. When configured to use a code replacement library, the code generator uses criteria defined in the library to search for matches. If a match is found, the code generator replaces code that it generates by default with application-specific code defined in the library.
Code replacement table	One or more code replacement table entries. Provides a way to group related or shared entries for use in different libraries.
Code replacement entry	Represents a potential replacement for a function or operator. Maps a conceptual representation of a function or operator to an implementation representation and priority.
Conceptual argument	Represents an input or output argument for a function or operator being replaced. Conceptual arguments observe naming conventions ('y1', 'u1', 'u2', ...) and data types familiar to the code generator.
Conceptual representation	Represents match criteria that the code generator uses to qualify functions and operators for replacement. Consists of: <ul style="list-style-type: none"> • Function or operator name or key • Conceptual arguments with type, dimension, and complexity specification for inputs and output • Attributes, such as an algorithm and fixed-point saturation and rounding modes
Implementation argument	Represents an input or output argument for a C or C++ replacement function. Implementation arguments observe C/C++ name and data type specifications.

Term	Definition
Implementation representation	Specifies C or C++ replacement function prototype. Consists of: <ul style="list-style-type: none"> • Function name (for example, 'cos_dbl' or 'u8_add_u8') • Implementation arguments specifying type, type qualifiers, and complexity for the function inputs and output • Parameters that provide build information, such as header and source file names and paths of build resources and compile and link flags
Key	Identifies a function or operator that is being replaced. A function name or key appears in the conceptual representation of a code replacement entry. The key RTW_OP_ADD identifies the addition operator.
Priority	Defines the match priority for a code replacement entry relative to other entries, which have the same name and conceptual argument list, within a code replacement library. The priority can range from 0 to 100, with 0 being the highest priority. The default is 100. If a library provides two implementations for a function or operator, the implementation with the higher priority shadows the one with the lower priority.

Code Replacement Limitations

Code replacement verification — It is possible that code replacement behaves differently than you expect. For example, data types that you observe in code generator input might not match what the code generator uses as intermediate data types during an operation. Verify code replacements by examining generated code.

Code replacement for matrices — Code replacement libraries do not support Dynamic and Symbolic sized matrices.

See Also

Related Examples

- “Choose a Code Replacement Library” on page 26-6
- “Replace Code Generated from Simulink Models” on page 26-8

Choose a Code Replacement Library

In this section...

“About Choosing a Code Replacement Library” on page 26-6

“Explore Available Code Replacement Libraries” on page 26-6

“Explore Code Replacement Library Contents” on page 26-6

About Choosing a Code Replacement Library

By default, the code generator does not use a code replacement library.

If you are considering using a code replacement library:

- 1 Explore available libraries. Identify one that best meets your application needs.
 - Consider the lists of application code replacement requirements and libraries that MathWorks provides in “What Is Code Replacement?” on page 26-2.
 - See “Explore Available Code Replacement Libraries” on page 26-6.
- 2 Explore the contents of the library. See “Explore Code Replacement Library Contents” on page 26-6.

If you do not find a suitable library and you have an Embedded Coder license, you can create a custom code replacement library. For more information, see “What Is Code Replacement Customization?” (Embedded Coder).

Explore Available Code Replacement Libraries

Select the Code replacement library to use for code generation from the **Configuration Parameters > Code Generation > Interface** pane . To view a description of a library, select and hover your cursor over the library name. A tooltip describes the library and lists the tables that it contains. The tooltip lists the tables in the order that the code generator searches for a function or operator match.

Explore Code Replacement Library Contents

Use the **Code Replacement Viewer** to explore the content of a code replacement library.

- 1 At the command prompt, type `crviewer`.


```
>> crviewer
```

The viewer opens. To view the content of a specific library, specify the name of the library as an argument in single quotes. For example:

```
>> crviewer('GNU C99 extensions')
```
- 2 In the left pane, select the name of a library. The viewer displays information about the library in the right pane.
- 3 In the left pane, expand the library, explore the list of tables it contains, and select a table from the list. In the middle pane, the viewer displays the function and operator entries that are in that table, along with abbreviated information for each entry.

- 4 In the middle pane, select a function or operator. The viewer displays information from the table entry in the right pane.

If you select an operator entry that specifies net slope fixed-point parameters (instantiated from entry class `RTW.TfLCOperationEntryGenerator` or `RTW.TfLCOperationEntryGenerator_NetSlope`), the viewer displays an additional tab that shows fixed-point settings.

See **Code Replacement Viewer** for details on what the viewer displays.

See Also

Related Examples

- “What Is Code Replacement?” on page 26-2
- “Replace Code Generated from Simulink Models” on page 26-8

Replace Code Generated from Simulink Models

This example shows how to replace generated code, using a code replacement library. Code replacement is a technique you can use to change the code that the code generator produces for functions and operators to meet application code requirements. For example, you can replace generated code to meet requirements such as:

- Optimization for a specific run-time environment, including, but not limited to, specific target hardware.
- Integration with existing application code.
- Compliance with a standard, such as AUTOSAR.
- Modification of code behavior, such as enabling or disabling nonfinite or inline support.
- Application- or project-specific code requirements, such as elimination of `math.h`, system header files, or calls to `memcpy` or `memset`, or use of BLAS.

Prepare for Code Replacement

1. Make sure that MATLAB®, Simulink®, Simulink Coder™, and a C compiler are installed on your system. Some code replacement libraries available in your development environment can require Embedded Coder®.

To install MathWorks® products, see the MATLAB installation documentation. If you have installed MATLAB and want to see which other MathWorks products are installed, in the Command Window, enter `ver`.

2. Identify an existing Simulink model or create a model for which you want the code generator to replace code.

Choose a Code Replacement Library

By default, the code generator does not apply a code replacement library. You can choose from libraries that MathWorks® provides and that you create and register by using the Embedded Coder® product. The list of available libraries depends on:

- Installed support packages.
- System target file, language, standard math library, and device vendor configuration.
- Whether you have created and registered libraries, using the Embedded Coder® product.

Libraries that include GNU99 extensions are intended for use with the GCC compiler. If you use one of those libraries with another compiler, generated code might not compile.

Depending on the product licenses that you have, other libraries might be available. If you have an Embedded Coder license, you can view and choose from other libraries and you can create custom code replacement libraries.

Configure Code Generator to Use Code Replacement Library

1. Configure the code generator to apply a code replacement library during code generation for the model. Do one of the following:

- In the Configuration Parameters dialog box, on the **Code Generation > Interface** pane, select a library for the Code replacement library parameter.

- Set the `CodeReplacementLibrary` parameter at the command line or programmatically.
2. Configure the code generator to produce code only (not build an executable program) so you can verify your code replacements before building an executable program. Do one of the following:
 - In the Configuration Parameters dialog box, on the **Code Generation** pane, select Generate code only.
 - Set the `GenCodeOnly` parameter at the command line or programmatically.

Include Code Replacement Information in Code Generation Report

If you have an Embedded Coder license, you can configure the code generator to include a code replacement section in the code generation report. The additional information can help you verify code replacements.

1. Configure the code generator to generate a report. In the Configuration Parameters dialog box, on the **Code Generation > Report** pane, select Create code generation report. Consider having the report open automatically. Select Open report automatically.
2. Include the code replacement section in the report. Select Summarize which blocks triggered code replacements (Embedded Coder).

Generate Replacement Code

Generate C/C++ code from the model and, if you configured the code generator accordingly, a code generation report. For example, in the model window, press **Ctrl+B**.

The code generator produces the code and displays the report.

Verify Code Replacements

Verify code replacements by examining the generated code. It is possible that code replacement behaves differently than you expect. For example, data types that you observe in the code generator input might not match what the code generator uses as intermediate data types during an operation.

More About

- “What Is Code Replacement?” on page 26-2
- “Choose a Code Replacement Library” on page 26-6
- “Code Generation Configuration” on page 20-2

Guidelines and Standards for Simulink Coder

- “Modeling Guidelines for Generated Code” on page 27-2
- “Industry Standards” on page 27-4
- “Modeling Guidelines and Model Advisor Checks for Verifying Compliance with Industry Standards” on page 27-7

Modeling Guidelines for Generated Code

Use modeling guidelines when you develop models and generate code for embedded systems using Model-Based Design with MathWorks products. The guidelines provide model setting, block usage, and parameter considerations that impact code generation.

Code Generation Modeling Guidelines

You can use code generation guidelines when developing models and generating code for embedded systems. The guidelines provide guidance on modeling patterns, block usage, and configuration parameter settings that impact code generation and component model deployment.

Guidelines for blocks include:

- “cgsl_0101: Zero-based indexing”
- “cgsl_0102: Evenly spaced breakpoints in lookup tables”
- “cgsl_0103: Precalculated signals and parameters”
-
-
- “cgsl_0201: Redundant Unit Delay and Memory blocks”

Guidelines for modeling patterns include:

- “cgsl_0202: Usage of For, While, and For Each subsystems with vector signals”
- “cgsl_0204: Vector and bus signals crossing into atomic subsystems or Model blocks”
-
-

Guidelines for configuration parameters include:

- “cgsl_0301: Prioritization of code generation objectives for code efficiency”
-

Guidelines for component deployment using a service interface configuration:

- “cgsl_0401: Modeling styles for component deployment”
- “cgsl_0402: Signal interfaces for component deployment”
- “cgsl_0404: Model startup and shutdown events by using Initialize Function and Terminate Function blocks for component deployment”
- “cgsl_0405: Data receive for component deployment”
- “cgsl_0406: Data send for component deployment”
- “cgsl_0408: Partial data send for component deployment”
- “cgsl_0409: Data transfer for component deployment”
- “cgsl_0411: Access nonvolatile memory by using Initialize Function and Terminate Function blocks”
- “cgsl_0413: Reuse memory between component state and output for component deployment”
- “cgsl_0414: Configure service interface for component model”

For information about verifying the readiness of your model for code generation, see “Check Model and Configuration for Code Generation” (Embedded Coder).

See Also

“Check Your Model Using the Model Advisor” | “Modeling Guidelines and Model Advisor Checks for Verifying Compliance with Industry Standards” (Embedded Coder) | “Model Advisor Checks for Verifying Compliance with Secure Coding Standards” (Embedded Coder)

Industry Standards

Applying Model-Based Design to a safety-critical system requires extra consideration and rigor so that the system adheres to defined safety standards, including:

- “IEC 61508: 2010” (Embedded Coder)
- “ISO 26262: 2018” (Embedded Coder)
- “AUTOSAR Standard” (Embedded Coder)
- “IEC 62304: 2015” (Embedded Coder)
- “EN 50128 / EN 50128: 2011” (Embedded Coder)
- “ISO 25119 Standard” (Embedded Coder)

If you have a Simulink Check product license, you can check that your Simulink model or subsystem and the code that you generate from it complies with the industry standard. For more information, see “Modeling Guidelines and Model Advisor Checks for Verifying Compliance with Industry Standards” on page 27-7.

IEC 61508: 2010

Development of high-integrity systems within the automotive industry is characterized by demonstrating compliance with IEC 61508: 2010, *Functional safety of electrical/electronic/programmable electronic safety related systems*. This standard addresses E/E systems⁴ in commercial vehicles – especially off-highway vehicles. IEC 61508: 2010 was published when most software was coded by hand, therefore the standard needs to be mapped to Model-Based Design technologies. For further information about MathWorks support for IEC 61508, see IEC 61508 Support in MATLAB and Simulink.

ISO 26262: 2018

Development of high-integrity systems within the automotive industry is characterized by demonstrating compliance with ISO 26262. ISO 26262: 2018 *Road vehicles — Functional safety*. This standard, which is an adaptation of IEC 61508, addresses *E/E systems* in passenger road vehicles.

For additional information about MathWorks support for ISO 26262, see ISO 26262 Support in MATLAB and Simulink.

IEC 62304: 2015

IEC 62304: 2015, *Medical device software - Software life cycle processes* is an international standard that defines the requirements of the software life-cycle processes for medical device software development. The standard specifies a set of processes, activities, and tasks that establish a common framework for designing safe and tested software for medical devices. For more information about MathWorks support for IEC 62304, see MATLAB and Simulink for Medical Devices.

If you have a Simulink Check product license, you can check that your Simulink model or subsystem and the code that you generate from it complies with selected aspects of the IEC 62304 standard. For

⁴ Systems that consists of electrical and electronic elements, including: programmable electronic elements, power supplies, input devices, communication paths, and output devices.

more information, see “Model Advisor Checks for IEC 61508, IEC 62304, ISO 26262, ISO 25119, and EN 50128/EN 50657 Standards Compliance” (Simulink Check).

EN 50128 / EN 50128: 2011

High-integrity systems developed within the railway industry need to demonstrate compliance with:

- EN 50128: 2011, *Railway applications — Software for railway control and protection systems*
- EN 50657:2017, *Railways Applications - Rolling stock applications - Software on Board Rolling Stock*

These European standards specify procedures and technical requirements for the development of programmable electronic systems for use in railway control and protection applications. For more information about MathWorks support for EN 50128, see EN 50128/EN 50657.

If you have a Simulink Check product license, you can check that your Simulink model or subsystem and the code that you generate from it complies with selected aspects of the EN 50128 standard. For more information, see “Model Advisor Checks for IEC 61508, IEC 62304, ISO 26262, ISO 25119, and EN 50128/EN 50657 Standards Compliance” (Simulink Check).

ISO 25119 Standard

Applying Model-Based Design to a safety-critical system requires extra consideration and rigor so that the system adheres to defined functional safety standards. ISO 25119, *Tractors and machinery for agriculture and forestry — Safety-related parts of control systems*, is such a standard.

MathWorks provides an IEC Certification Kit product that you can use to qualify MathWorks code generation and verification tools for projects based on the ISO 25119 standard.

If you have a Simulink Check product license, you can check that your Simulink model or subsystem and the code that you generate from it complies with selected aspects of the ISO 25119 standard. For more information, see “Model Advisor Checks for IEC 61508, IEC 62304, ISO 26262, ISO 25119, and EN 50128/EN 50657 Standards Compliance” (Simulink Check).

DO-178C Standard

Applying Model-Based Design to a high-integrity system requires extra consideration and rigor so that the system adheres to defined safety standards. DO-178C, *Software Considerations in Airborne Systems and Equipment Certification*, is such a standard. A supplement to DO-178C, DO-331, provides guidance on the use of Model-Based Design technologies. MathWorks provides a DO Qualification Kit product that you can use to qualify MathWorks verification tools for projects based on the DO-178C, DO-331, and related standards. For more information, see <https://www.mathworks.com/products/do-178/>.

For information about Model-Based Design and MathWorks support of aerospace and defense industry standards, see DO Qualification Kit.

If you have a Simulink Check product license, you can check that your Simulink model or subsystem and the code that you generate from it complies with selected aspects of the DO-178C standard. For more information, see “Model Advisor Checks for DO-178C/DO-331 Standards Compliance” (Simulink Check).

AUTOSAR Standard

Simulink software supports *AUTomotive Open System ARchitecture* (AUTOSAR), an open and standardized automotive software architecture. Automobile manufacturers, suppliers, and tool developers jointly develop AUTOSAR components.

The AUTOSAR standard addresses:

- Architecture - Application, run-time environment, and service layers, which serve to decouple AUTOSAR software components from the execution platform. Standard interfaces between software components and the run-time environment allow reuse or relocation of components within the Electronic Control Unit (ECU) topology of a vehicle.

The standard defines two AUTOSAR platforms:

- Classic Platform (CP), with Application, Runtime Environment (RTE), and Basic Software (BSW) layers
- Adaptive Platform (AP), with Application, AUTOSAR Runtime for Adaptive Applications (ARA), Services, and Basis layers
- Methodology - Specification of code formats and description file templates, for example.
- Foundation - Requirements and specifications shared between AUTOSAR platforms, supporting platform interoperability.
- Application Interfaces - Specification of interfaces for typical automotive applications.

For more information, see:

- www.autosar.org for details on the AUTOSAR standard.
- “AUTOSAR Blockset” for information on modeling and simulating AUTOSAR software, from which Embedded Coder can generate code.
- MATLAB and Simulink for AUTOSAR to learn about using MathWorks products and third-party tools for AUTOSAR.

If you have an AUTOSAR Blockset product license, you can check that your Simulink model or subsystem and the code that you generate from it complies with selected aspects of the AUTOSAR standard. For more information, see “AUTOSAR Blockset Checks” (AUTOSAR Blockset).

See Also

“Modeling Guidelines and Model Advisor Checks for Verifying Compliance with Industry Standards” on page 27-7

External Websites

- International Organization for Standardization
- International Electrotechnical Commission
- IEC functional safety website
- European Committee for Electrotechnical Standardization
- AUTOSAR Development Partnership
- Radio Technical Commission for Aeronautics (RTCA)

Modeling Guidelines and Model Advisor Checks for Verifying Compliance with Industry Standards

Use modeling guidelines when you develop models and generate code for embedded systems that are intended for use in projects that must comply with industry standards.

High-Integrity System Modeling Guidelines

“High-Integrity System Modeling” guidelines provide model setting, block usage, and block parameter considerations for creating models that are complete, unambiguous, robust, and verifiable. Use these guidelines when you develop models and generate code for safety-critical systems, such as in projects that need to comply with the DO-178C / DO-331, IEC 61508, IEC 62304, ISO 26262, or EN 50128 industry standards.

If you have a Simulink Check product license, you can use the Model Advisor to check for compliance with the high-integrity modeling guidelines. To check your model or subsystem:

- 1 Open the Model Advisor.
- 2 Navigate to **By Product > Simulink Check > Modeling Standards > <Standard> Checks**.
- 3 Run the checks in the folder.

To review the High-Integrity System Modeling guidelines and their corresponding Model Advisor checks, see “Model Advisor Checks for High Integrity Systems Modeling Guidelines” (Simulink Check).

MAB Modeling Guidelines

MathWorks Advisory Board (MAB) guidelines stipulate important basic rules for modeling in Simulink and Stateflow. The overall purpose of these modeling guidelines is to allow for a simple, common understanding by modelers and consumers of control system models. The main objectives of these guidelines are readability, simulation and verification, and code generation.

If you have a Simulink Check license, you can check that your Simulink model or subsystem, and the code that you generate from it, complies with MAB guidelines. For more information, see “Model Advisor Checks for MAB Modeling Guidelines” (Simulink Check)

JMAAB Modeling Guidelines

Similar to MAB modeling guidelines, the Japan MathWorks Automotive Advisor Board (JMAAB) modeling guidelines stipulate rules for modeling in Simulink and Stateflow. The overall purpose of these modeling guidelines is to allow for a simple, common understanding by modelers and consumers of control system models. The main objectives of these guidelines are readability, simulation and verification, and code generation.

The JMAAB modeling guidelines are published in:

- *Version 5.1 — Control Algorithm Modeling Guidelines - Using MATLAB, Simulink, and Stateflow, and*
- *Version 6.0 — Control Algorithm Modeling Guidelines - Using MATLAB, Simulink, and Stateflow*

If you have a Simulink Check license, you can check that your Simulink model or subsystem, and the code that you generate from it, complies with JMAAB guidelines. For more information, see “Model Advisor Checks for JMAAB Modeling Guidelines” (Simulink Check).

See Also

“Check Your Model Using the Model Advisor” | “Industry Standards” on page 27-4

External Websites

- MathWorks Advisory Board (MAB) Guidelines
- Version 5.1 — Control Algorithm Modeling Guidelines - Using MATLAB, Simulink, and Stateflow
- Version 6.0 — Control Algorithm Modeling Guidelines - Using MATLAB, Simulink, and Stateflow

Deployment

Real-Time Systems in Simulink Coder

- “Deploy Algorithm Model for Real-Time Rapid Prototyping” on page 28-2
- “Deploy Environment Model for Real-Time Hardware-In-the-Loop (HIL) Simulation” on page 28-4
- “Untrusted Custom Code, Custom Targets, and Callbacks” on page 28-6

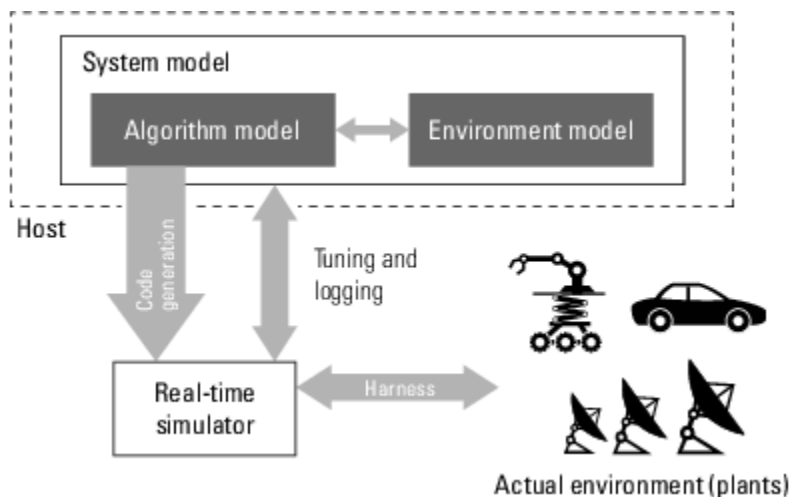
Deploy Algorithm Model for Real-Time Rapid Prototyping

Use the code generator to deploy algorithm models for real-time rapid prototyping.

About Real-Time Rapid Prototyping

Real-time rapid prototyping requires the use of a real-time simulator, potentially connected to system hardware (for example, physical plant or vehicle) being controlled. You generate, deploy, and tune code as it runs on the real-time simulator or embedded microprocessor. This design step is crucial for verifying whether a component can adequately control the system, and allows you to assess, interact with, and optimize code.

The following figure shows a typical approach for real-time rapid prototyping.



Goals of Real-Time Rapid Prototyping

Assuming that you have documented functional requirements, refined concept models, system hardware for the physical plant or vehicle being controlled, and access to target products you intend to use (for example, for example, the Simulink Real-Time or Simulink Desktop Real-Time product), you can use real-time prototyping to:

- Refine component and environment model designs by rapidly iterating between algorithm design and prototyping
- Validate whether a component can adequately control the physical system in real time
- Evaluate system performance before laying out hardware, coding production software, or committing to a fixed design
- Test hardware

Refine Code With Real-Time Rapid Prototyping

To perform real-time rapid prototyping:

- 1 Create or acquire a real-time system that runs in real time on rapid prototyping hardware. The Simulink Real-Time product facilitates real-time rapid prototyping. This product provides a real-

time operating system that makes PCs run in real time. It also provides device driver blocks for numerous hardware I/O cards. You can then create a rapid prototyping system using inexpensive commercial-off-the-shelf (COTS) hardware. In addition, third-party vendors offer products based on the Simulink Real-Time product or other code generation technology that you can integrate into a development environment.

- 2 Use provided system target files to generate code that you can deploy onto a real-time simulator. See the following information.

Engineering Tasks	Related Product Information	Examples
Generate code for real-time rapid prototyping	<p>“Compare System Target File Support Across Products” on page 21-17</p> <p>“Event-Based Scheduling”</p> <p>Embedded Coder</p> <p>“Support for Standards and Guidelines” (Embedded Coder)</p>	<p>rtwdemo_counter</p> <p>rtwdemo_counter_msvc</p>
Generate code for rapid prototyping in hard real time, using PCs	<p>Simulink Real-Time</p> <p>“Simulink Real-Time Options Pane” (Simulink Real-Time)</p>	slrt_ex_osc (and others)
Generate code for rapid prototyping in soft real time, using PCs	<p>Simulink Desktop Real-Time</p> <p>“Simulink Desktop Real-Time Pane” (Simulink Desktop Real-Time)</p>	sldrtex_vdp (and others)

- 3 Monitor signals, tune parameters, and log data.

See Also

More About

- “Access Signal, State, and Parameter Data During Execution” on page 15-17
- “Real-Time Simulation and Testing” (Simulink Real-Time)

Deploy Environment Model for Real-Time Hardware-In-the-Loop (HIL) Simulation

In this section...

“About Hardware-In-the-Loop Simulation” on page 28-4

“Set Up and Run HIL Simulations” on page 28-4

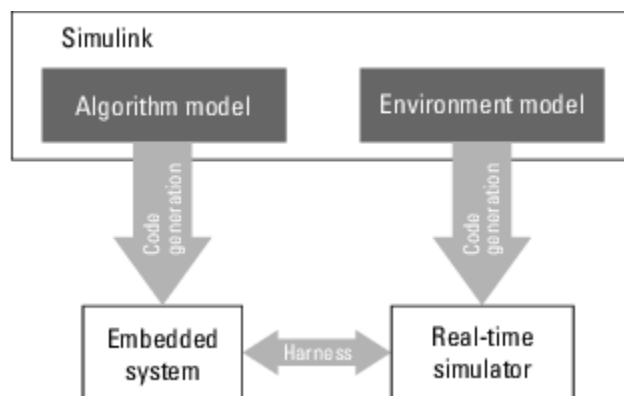
About Hardware-In-the-Loop Simulation

Hardware-in-the-loop (HIL) simulation tests and verifies an embedded system or control unit in the context of a software test platform. Examples of test platforms include real-time target systems and instruction set simulators (IISs). You use Simulink software to develop and verify a model that represents the test environment. Using the code generator, you produce, build, and download an executable program for the model to the HIL simulation platform. After you set up the environment, you can run the executable to validate the embedded system or control unit in real time.

During HIL simulation, you gradually replace parts of a system environment with hardware components as you refine and fabricate the components. HIL simulation offers an efficient design process that eliminates costly iterations of part fabrication.

The code that you build for the system simulator provides real-time system capabilities. For example, the code can include VxWorks from Wind River or another real-time operating system (RTOS).

The following figure shows a typical HIL setup.



The HIL platform available from MathWorks is the Simulink Real-Time product. Several third-party products are also available for use as HIL platforms. The Simulink Real-Time product offers hard real-time performance for PCs with Intel® or AMD® 32-bit processors functioning as your real-time target. The Simulink Real-Time product enables you to add I/O interface blocks to your models and automatically generate code with code generation technology. The Simulink Real-Time product can download the code to a second PC running the Simulink Real-Time real-time kernel. System integrator solutions that are based on Simulink Real-Time are also available.

Set Up and Run HIL Simulations

To set up and run HIL simulations iterate through the following steps:

- 1 Develop a model that represents the environment or system under development.

For more information, see “Compare System Target File Support Across Products” on page 21-17.

- 2 Generate an executable for the environment model.
- 3 Download the executable for the environment model to the HIL simulation platform.
- 4 Replace software representing a system component with corresponding hardware.
- 5 Test the hardware in the context of the HIL system.
- 6 Repeat steps 4 and 5 until you can simulate the system after including components that require testing.

See Also

More About

- “Access Signal, State, and Parameter Data During Execution” on page 15-17
- “Real-Time Simulation and Testing” (Simulink Real-Time)

Untrusted Custom Code, Custom Targets, and Callbacks

You can use custom code in:

- Simulations on your development computer.
- Applications that you deploy on external hardware.

Untrusted custom code, custom targets, or callbacks have risks. For example, unintended modification of your design, unauthorized acquisition of sensitive code and data by third parties, and unapproved destruction of data.

Best practices for exchanging sensitive code and data with a supplier or customer are:

- For code packaging, use a secure container that supports encryption such as an Open Packaging Conventions (OPC) container.
- For file transfer, use a secure transfer protocol such as HTTPS or SFTP.

Desktops in Simulink Coder

- “Accelerate, Refine, and Test Hybrid Dynamic System on Host Computer by Using RSim System Target File” on page 29-2
- “Run Rapid Simulations over a Range of Parameter Values” on page 29-24
- “Run Batch Simulations Without Recompiling Generated Code” on page 29-30
- “Use MAT-Files to Feed Data to Inport Blocks for Rapid Simulations” on page 29-35
- “Use S-Function Target for Model or Subsystem” on page 29-40

Accelerate, Refine, and Test Hybrid Dynamic System on Host Computer by Using RSim System Target File

After you create a model, you can use the rapid simulation (RSim) system target file to characterize model behavior. The executable program that results from the build process is for non-real-time execution on your development computer. The executable program is highly optimized for simulating models of hybrid dynamic systems, including models that use variable-step solvers and zero-crossing detection. The speed of the generated code makes the RSim system target file ideal for building programs for batch or Monte Carlo simulations.

About Rapid Simulation

Use the RSim target to generate an executable that runs fast, standalone simulations. You can repeat simulations with varying data sets, interactively or programmatically with scripts, without rebuilding the model. This can accelerate the characterization and tuning of model behavior and code generation testing.

Using command-line options:

- Define parameter values and input signals in one or more MAT-files that you can load and reload at the start of simulations without rebuilding your model.
- Redirect logging data to one or more MAT-files that you can then analyze and compare.
- Control simulation time.
- Specify external mode options.

Note To run an RSim executable, configure your computer to run MATLAB and have the MATLAB and Simulink installation folders accessible.

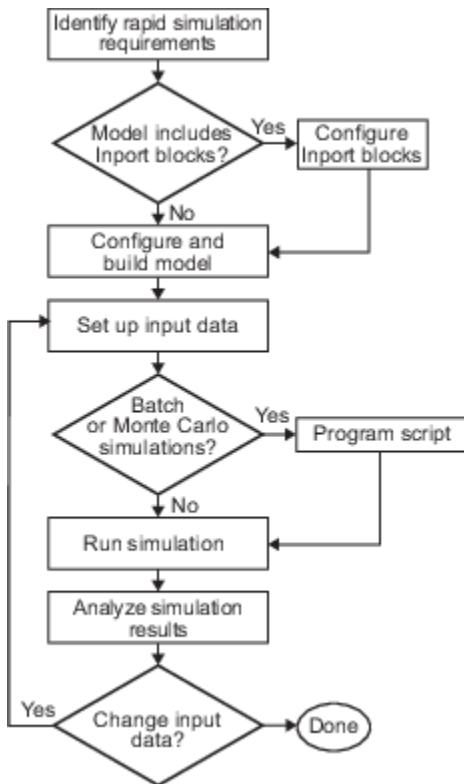
Rapid Simulation Advantage

The advantage that you gain from rapid simulation varies. Larger simulations achieve speed improvements of up to 10 times faster than standard Simulink simulations. Some models might not show noticeable improvement in simulation speed. To determine the speed difference for your model, time your standard Simulink simulation and compare the results with a rapid simulation. In addition, test the model simulation in Rapid Accelerator simulation mode.

General Rapid Simulation Workflow

Like other stages of Model-Based Design, characterization and tuning of model behavior is an iterative process, as shown in the general workflow diagram in the figure. Tasks in the workflow are:

- 1 Identify your rapid simulation requirements. on page 29-3
- 2 Configure Inport blocks on page 29-4 that provide input source data for rapid simulations.
- 3 Configure the model on page 29-4 for rapid simulation.
- 4 Set up simulation input data. on page 29-6
- 5 Run the rapid simulations. on page 29-13



Identify Rapid Simulation Requirements

The first step to setting up a rapid simulation is to identify your simulation requirements.

Question...	For More Information, See...
How long do you want simulations to run?	"Configure and Build Model for Rapid Simulation" on page 29-4
Are there solver requirements? Do you expect to use the same solver for which the model is configured for your rapid simulations?	"Configure and Build Model for Rapid Simulation" on page 29-4
Do your rapid simulations need to accommodate flexible custom code interfacing? Or, do your simulations need to retain storage class settings?	"Configure and Build Model for Rapid Simulation" on page 29-4
Will you be running simulations with multiple data sets?	"Set Up Rapid Simulation Input Data" on page 29-6
Will the input data consist of global parameters, signals, or both?	"Set Up Rapid Simulation Input Data" on page 29-6
What type of source blocks provide input data to the model — From File, Inport, From Workspace?	"Set Up Rapid Simulation Input Data" on page 29-6
Will the model's parameter vector (<i>model_P</i>) be used as input data?	"Create a MAT-File That Includes a Model Parameter Structure" on page 29-6

Question...	For More Information, See...
What is the data type of the input parameters and signals?	"Set Up Rapid Simulation Input Data" on page 29-6
Will the source data consist of one variable or multiple variables?	"Set Up Rapid Simulation Input Data" on page 29-6
Does the input data include tunable parameters?	"Create a MAT-File That Includes a Model Parameter Structure" on page 29-6
Do you need to gain access to tunable parameter information — model checksum and parameter data types, identifiers, and complexity?	"Create a MAT-File That Includes a Model Parameter Structure" on page 29-6
Will you have a need to vary the simulation stop time for simulation runs?	"Configure and Build Model for Rapid Simulation" on page 29-4 and "Override a Model Simulation Stop Time" on page 29-15
Do you want to set a time limit for the simulation? Consider setting a time limit if your model experiences frequent zero crossings and has a small minor step size.	"Set a Clock Time Limit for a Rapid Simulation" on page 29-15
Do you need to preserve the output of each simulation run?	"Specify a New Output File Name for a Simulation" on page 29-21 and "Specify New Output File Names for To File Blocks" on page 29-21
Do you expect to run the simulations interactively or in batch mode?	"Scripts for Batch and Monte Carlo Simulations" on page 29-13

Configure Inports to Provide Simulation Source Data

You can use Inport blocks as a source of input data for rapid simulations. To do so, configure the blocks so that they can import data from external MAT-files. By default, the Inport block inherits parameter settings from downstream blocks. In most cases, to import data from an external MAT-file, you must explicitly set these block parameters to match the source data in the MAT-file.

- **Main > Interpolate data**
- **Signal Attributes > Port dimensions**
- **Signal Attributes > Data type**
- **Signal Attributes > Signal type**

If you do not have control over the model content, you might need to modify the data in the MAT-file to conform to what the model expects for input. Input data characteristics and specifications of the Inport block that receives the data must match.

For details on adjusting these parameters and on creating a MAT-file for use with an Inport block, see "Create a MAT-File for an Inport Block" on page 29-10. For descriptions of the preceding block parameters, see the block description of Inport.

Configure and Build Model for Rapid Simulation

After you identify your rapid simulation requirements, configure the model for rapid simulation.

- 1 Set model configuration parameter **System target file** to `rsim.tlc` (Rapid Simulation Target).
- 2 Click **RSim Target** to view the **RSim Target** pane.

The screenshot shows the RSim Target configuration pane with the following settings:

- Parameter loading:** Enable RSim executable to load parameters from a MAT-file
- Solver:** Solver selection: auto
- Storage classes:** Force storage classes to AUTO

- 3 Set the RSim target configuration parameters for your rapid simulation requirements.

If You Want to...	Then...
Generate code that allows the RSim executable to load parameters from a MAT-file	Select Enable RSim executable to load parameters from a MAT-file (default).
Let the target choose a solver based on the solver already configured for the model	Set Solver selection to <code>auto</code> (default). The code generator uses a built-in solver if a fixed-step solver is specified on the Solver pane or calls the Simulink solver module (a shared library) if a variable-step solver is specified.
Explicitly instruct the target to use a fixed-step solver	Set Solver selection to <code>Use fixed-step solvers</code> . In the Configuration Parameters dialog box, on the Solver pane, specify a fixed-step solver.
Explicitly instruct the target to use a variable-step solver	Set Solver selection to <code>Use Simulink solver module</code> . In the Configuration Parameters dialog box, on the Solver pane, specify a variable-step solver.
Force storage classes to <code>Auto</code> for flexible custom code interfacing	Select Force storage classes to AUTO (default).
Retain storage class settings, such as <code>ExportedGlobal</code> or <code>ImportedExtern</code> , due to application requirements	Clear Force storage classes to AUTO .

- 4 Set up data import and export options. On the **Data Import/Export** pane, in the **Save to Workspace** section, select the **Time**, **States**, **Outputs**, and **Final States** parameters, as they apply. By default, the code generator saves simulation logging results to a file named `model.mat`. For more information, see “Save Simulation Data”.
- 5 If you are using external mode communication, set up the interface, using the **Code Generation > Interface** pane. See “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” on page 32-2 for details.
- 6 Press **Ctrl+B**. The code generator builds a highly optimized executable program that you can run on your development computer with varying data, without rebuilding.

For more information on compilers that are compatible with the Simulink Coder product, see “Select and Configure C or C++ Compiler” on page 31-3 and “Template Makefiles and Make Options” on page 31-36 .

Set Up Rapid Simulation Input Data

- “About Rapid Simulation Data Setup” on page 29-6
- “Create a MAT-File That Includes a Model Parameter Structure” on page 29-6
- “Create a MAT-File for a From File Block” on page 29-9
- “Create a MAT-File for an Inport Block” on page 29-10

About Rapid Simulation Data Setup

The format and setup of input data for a rapid simulation depends on your requirements.

If the Input Data Source Is...	Then...
The model's global parameter vector (<i>model_P</i>)	Use the <code>rsimgetrtp</code> function to get the vector content and then save it to a MAT-file.
The model's global parameter vector and you want a mapping between the vector and tunable parameters	Call the <code>rsimgetrtp</code> function to get the global parameter structure and then save it to a MAT-file.
Provided by a From File block	Create a MAT-file that a From File block can read.
Provided by an Inport block	Create a MAT-file that adheres to one of the three data file formats that the Inport block can read.
Provided by a From Workspace block	Create structure variables in the MATLAB workspace.

The RSim target requires that MAT-files used as input for From File and Inport blocks contain data. The gRT target inserts MAT-file data directly into the generated code, which is then compiled and linked as an executable. In contrast, RSim allows you to replace data sets for each successive simulation. A MAT-file containing From File or Inport block data must be present if a From File block or Inport block exists in your model.

Create a MAT-File That Includes a Model Parameter Structure

To create a MAT-file that includes a model global parameter structure (*model_P*),

- 1 Get the structure by calling the function `rsimgetrtp`.
- 2 Save the parameter structure to a MAT-file.

If you want to run simulations over varying data sets, consider converting the parameter structure to a cell array and saving the parameter variations to a single MAT-file.

Get the Parameter Structure for a Model

Get the global parameter structure (*model_P*) for a model by calling the function `rsimgetrtp`.

```
param_struct = rsimgetrtp('model')
```

Argument	Description
<i>model</i>	The model for which you are running the rapid simulations.

The `rsimgetrtp` function forces an update diagram action for the specified model and returns a structure that contains the following fields.

Field	Description
<code>modelChecksum</code>	A four-element vector that encodes the structure of the model. The code generator uses the checksum to check whether the structure of the model has changed since the RSim executable was generated. If you delete or add a block, and then generate a new <code>model_P</code> vector, the new checksum does not match the original checksum anymore. The RSim executable detects this incompatibility in parameter vectors and exits to avoid returning incorrect simulation results. If the model structure changes, you must regenerate the code for the model.
<code>parameters</code>	A structure that contains the model's global parameters.

The parameter structure contains the following information.

Field	Description
<code>dataTypeName</code>	The name of the parameter data type, for example, <code>double</code>
<code>dataTypeID</code>	Internal data type identifier used by the code generator
<code>complex</code>	The value 0 if real; 1 if complex
<code>dtTransIdx</code>	Internal data index used by the code generator
<code>values</code>	A vector of the parameter values associated with this structure
<code>map</code>	This field contains the mapping information that correlates the 'values' to the tunable parameters of the model. This mapping information, in conjunction with <code>rsimsetrtpparam</code> , is useful for creating subsequent <code>rtP</code> structures without compiling the block diagram.

The code generator reports a tunable fixed-point parameter according to its stored value. For example, an `sfix(16)` parameter value of 1.4 with a scaling of 2^{-8} has a value of 358 as an `int16`.

In the following example, `rsimgetrtP` returns the parameter structure for the example model `rtwdemo_rsimtf` to `param_struct`.

```
param_struct = rsimgetrtP('rtwdemo_rsimtf')

param_struct =

    modelChecksum: [1.7165e+009 3.0726e+009 2.6061e+009 2.3064e+009]
    parameters: [1x1 struct]
```

Save the Parameter Structure to a MAT-File

After you issue a call to `rsimgetrtP`, save the return value of the function call to a MAT-file. Using a command-line option, you can then specify that MAT-file as input for rapid simulations.

The following example saves the parameter structure returned for `rtwdemo_rsimtf` to the MAT-file `myrsimdemo.mat`.

```
save myrsimdemo.mat param_struct;
```

For information on using command-line options to specify required files, see “Run Rapid Simulations” on page 29-13.

Convert the Parameter Structure for Running Simulations on Varying Data Sets

To use rapid simulations to test changes to specific parameters, you can convert the model parameter structure to a cell array. You can then access a specific parameter set by using the @ operator to specify the index for a specific parameter set in the file.

To convert the structure to a cell array:

- 1 Use the function `rsimgetrtp` to get a structure containing parameter information for the example model `rtwdemo_rsimtf`. Store the structure in a variable `param_struct`.

```
param_struct = rsimgetrtp('rtwdemo_rsimtf');
```

The `parameters` field of the structure is a substructure that contains parameter information. The `values` field of the `parameters` substructure contains the numeric values of the parameters that you can tune during execution of the simulation code.

- 2 Use the function `rsimsetrtpparam` to expand the structure so that it contains more parameter sets. In this case, create two more parameter sets (for a total of three sets).

```
param_struct = rsimsetrtpparam(param_struct,3);
```

The function converts the `parameters` field to a cell array with three elements. Each element contains information for a single parameter set. By default, the function creates the second and third elements of the cell array by copying the first element. Therefore, all of the parameter sets use the same parameter values.

- 3 Specify new values for the parameters in the second and third parameter sets.

```
param_struct.parameters{2}.values = [-150 -5000 0 4950];  
param_struct.parameters{3}.values = [-170 -5500 0 5100];
```

- 4 Save the structure containing the parameter set information to a MAT-file.

```
save rtwdemo_rsimtf.mat param_struct;
```

Alternatively, you can modify the block parameters in the model, and use `rsimgetrtp` to create multiple parameter sets:

- 1 Use the function `rsimgetrtp` to get a structure containing parameter information for the example model `rtwdemo_rsimtf`. Store the structure in a variable `param_struct`.

```
param_struct = rsimgetrtp('rtwdemo_rsimtf');
```

- 2 Use the function `rsimsetrtpparam` to expand the structure so that it contains more parameter sets. In this case, create two more parameter sets (for a total of three sets).

```
param_struct = rsimsetrtpparam(param_struct,3);
```

The function converts the `parameters` field to a cell array with three elements. Each element contains information for a single parameter set. By default, the function creates the second and third elements of the cell array by copying the first element. Therefore, all of the parameter sets use the same parameter values.

- 3 Change the values of block parameters or workspace variables. For example, change the value of the variable `w` from 70 to 72.

```
w = 72;
```


- 4 Use `rimsgetrtp` to get another structure containing parameter information. Store the structure in a temporary variable `rtp_temp`.

```
rtp_temp = rsimgetrtp('rtwdemo_rsimtf');
```

- 5 Assign the value of the `parameters` field of `rtp_temp` to the structure `param_struct` as a second parameter set.

```
param_struct.parameters{2} = rtp_temp.parameters;
```

- 6 Change the value of the variable `w` from 72 to 75.

```
w = 75;
```

- 7 Use `rimsgetrtp` to get another structure containing parameter information. Then, assign the value of the `parameters` field to `param_struct` as a third parameter set.

```
rtp_temp = rsimgetrtp('rtwdemo_rsimtf');  
param_struct.parameters{3} = rtp_temp.parameters;
```

- 8 Save the structure containing the parameter set information to a MAT-file.

```
save 'rtwdemo_rsimtf.mat' param_struct;
```

For more information on how to specify each parameter set when you run the simulations, see “Change Block Parameters for an RSim Simulation” on page 29-20.

Create a MAT-File for a From File Block

You can use a MAT-file as the input data source for a From File block. The format of the data in the MAT-file must match the data format expected by that block. For example, if you are using a matrix as an input for the MAT file, this cannot be different from the matrix size for the executable.

To create a MAT-file for a From File block:

- 1 For array format data, in the workspace create a matrix that consists of two or more rows. The first row must contain monotonically increasing time points. Other rows contain data points that correspond to the time point in that column. The time and data points must be data of type `double`.

For example:

```
t=[0:0.1:2*pi]';  
Ina1=[2*sin(t) 2*cos(t)];  
Ina2=sin(2*t);  
Ina3=[0.5*sin(3*t) 0.5*cos(3*t)];  
var_matrix=[t Ina1 Ina2 Ina3]';
```

For other supported data types, such as `int16` or fixed-point, the time data points must be of type `double`, just as for array format data. However, the sample data can be of any dimension.

For more information on setting up the input data, see the block description of From File.

- 2 Save the matrix to a MAT-file.

The following example saves the matrix `var_matrix` to the MAT-file `myrsimdemo.mat` in Version 7.3 format.

```
save '-v7.3' myrsimdemo.mat var_matrix;
```

Using a command-line option, you can then specify that MAT-file as input for rapid simulations.

Create a MAT-File for an Inport Block

You can use a MAT-file as the input data source for an Inport block.

The format of the data in the MAT-file must adhere to one of the three column-based formats listed in the following table. The table lists the formats in order from least flexible to most flexible.

Format	Description
Single time/data matrix	<ul style="list-style-type: none"> Least flexible. One variable. Two or more <i>columns</i>. Number of columns must equal the sum of the dimensions of all root Inport blocks plus 1. First column must contain monotonically increasing time points. Other columns contain data points that correspond to the time point in a given row. Data of type <code>double</code>. <p>For an example, see Single time/data matrix in the following procedure, step 4. For more information, see “Loading Data Arrays to Root-Level Inputs”.</p>
Signal-and-time structure	<ul style="list-style-type: none"> More flexible than the single time/data matrix format. One variable. Must contain two top-level fields: <code>time</code> and <code>signals</code>. The <code>time</code> field contains a <i>column</i> vector of the simulation times. The <code>signals</code> field contains an array of substructures, each of which corresponds to an Inport block. The substructure index corresponds to the Inport block number. Each <code>signals</code> substructure must contain a field named <code>values</code>. The <code>values</code> field must contain an array of inputs for the corresponding Inport block, where each input corresponds to a time point specified by the <code>time</code> field. If the <code>time</code> field is set to an empty value, clear the check box for the Inport block Interpolate data parameter. Data type must match Inport block settings. <p>For an example, see Signal-and-time structure in the following procedure, step 4. For more information on this format, see “Loading Data Structures to Root-Level Inputs”.</p>

Format	Description
Per-port structure	<ul style="list-style-type: none"> • Most flexible • Multiple variables. Number of variables must equal the number of Inport blocks. • Consists of a separate structure-with-time or structure-without-time for each Inport block. Each Inport block data structure has only one <code>signals</code> field. To use this format, in the Input text field, enter the names of the structures as a comma-separated list, <code>in1, in2, ..., inN</code>, where <code>in1</code> is the data for your model's first port, <code>in2</code> for the second port, and so on. • Each variable can have a different time vector. • If the <code>time</code> field is set to an empty value, clear the check box for the Inport block Interpolate data parameter. • Data type must match Inport block settings. • To save multiple variables to the same data file, you must save them in the order expected by the model, using the <code>-append</code> option. <p>For an example, see Per-port structure in the following procedure, step 4. For more information, see “Loading Data Structures to Root-Level Inputs”.</p>

The supported formats and the following procedure are illustrated in `rtwdemo_rsim_i`.

To create a MAT-file for an Inport block:

- 1 Choose one of the preceding data file formats.
- 2 Update Inport block parameter settings and specifications to match specifications of the data to be supplied by the MAT-file.

By default, the Inport block inherits parameter settings from downstream blocks. To import data from an external MAT-file, explicitly set the following parameters to match the source data in the MAT-file.

- **Main > Interpolate data**
- **Signal Attributes > Port dimensions**
- **Signal Attributes > Data type**
- **Signal Attributes > Signal type**

If you choose to use a structure format for workspace variables and the `time` field is empty, clear block parameter **Interpolate data** or modify the parameter so that it is set to a nonempty value. Interpolation requires time data.

For descriptions of the preceding block parameters, see the block description of Inport.

- 3 Build an RSim executable program for the model. The build process creates and calculates a structural checksum for the model and embeds it in the generated executable. The RSim target uses the checksum to verify that data being passed into the model is consistent with what the model executable expects.
- 4 Create the MAT-file that provides the source data for the rapid simulations. You can create the MAT-file from a workspace variable. Using the specifications in the preceding format comparison table, create the workspace variables for your simulations.

An example of each format follows:

Single time/data matrix

```
t=[0:0.1:2*pi]';
Ina1=[2*sin(t) 2*cos(t)];
Ina2=sin(2*t);
Ina3=[0.5*sin(3*t) 0.5*cos(3*t)];
var_matrix=[t Ina1 Ina2 Ina3];
```

Signal-and-time structure

```
t=[0:0.1:2*pi]';
var_single_struct.time=t;
var_single_struct.signals(1).values(:,1)=2*sin(t);
var_single_struct.signals(1).values(:,2)=2*cos(t);
var_single_struct.signals(2).values=sin(2*t);
var_single_struct.signals(3).values(:,1)=0.5*sin(3*t);
var_single_struct.signals(3).values(:,2)=0.5*cos(3*t);
v=[var_single_struct.signals(1).values...
var_single_struct.signals(2).values...
var_single_struct.signals(3).values];
```

Per-port structure

```
t=[0:0.1:2*pi]';
Inb1.time=t;
Inb1.signals.values(:,1)=2*sin(t);
Inb1.signals.values(:,2)=2*cos(t);
t=[0:0.2:2*pi]';
Inb2.time=t;
Inb2.signals.values(:,1)=sin(2*t);
t=[0:0.1:2*pi]';
Inb3.time=t;
Inb3.signals.values(:,1)=0.5*sin(3*t);
Inb3.signals.values(:,2)=0.5*cos(3*t);
```

- 5 Save the workspace variables to a MAT-file.

Single time/data matrix

The following example saves the workspace variable `var_matrix` to the MAT-file `rsim_i_matrix.mat`.

```
save rsim_i_matrix.mat var_matrix;
```

Signal-and-time structure

The following example saves the workspace structure variable `var_single_struct` to the MAT-file `rsim_i_single_struct.mat`.

```
save rsim_i_single_struct.mat var_single_struct;
```

Per-port structure

To order data when saving per-port structure variables to a single MAT-file, use the `save` command's `-append` option. Be sure to append the data in the order that the model expects it.

The following example saves the workspace variables `Inb1`, `Inb2`, and `Inb3` to MAT-file `rsim_i_multi_struct.mat`.

```
save rsim_i_multi_struct.mat Inb1;
save rsim_i_multi_struct.mat Inb2 -append;
save rsim_i_multi_struct.mat Inb3 -append;
```

The `save` command does not preserve the order in which you specify your workspace variables in the command line when saving data to a MAT-file. For example, if you specify the variables `v1`, `v2`, and `v3`, in that order, the order of the variables in the MAT-file could be `v2 v1 v3`.

Using a command-line option, you can then specify the MAT-files as input for rapid simulations.

Scripts for Batch and Monte Carlo Simulations

The RSim target is for batch simulations in which parameters and input signals vary for multiple simulations. New output file names allow you to run new simulations without overwriting prior simulation results. You can set up a series of simulations to run by creating a `.bat` file for use on a Microsoft Windows platform.

Create a file for the Windows platform with a text editor and execute it by typing the file name, for example, `mybatch`, where the name of the text file is `mybatch.bat`.

```
rtwdemo_rsimtf -f rtwdemo_rsimtf.mat=run1.mat -o results1.mat -tf 10.0
rtwdemo_rsimtf -f rtwdemo_rsimtf.mat=run2.mat -o results2.mat -tf 10.0
rtwdemo_rsimtf -f rtwdemo_rsimtf.mat=run3.mat -o results3.mat -tf 10.0
rtwdemo_rsimtf -f rtwdemo_rsimtf.mat=run4.mat -o results4.mat -tf 10.0
```

In this case, batch simulations run using four sets of input data in files `run1.mat`, `run2.mat`, and so on. The RSim executable saves the data to the files specified with the `-o` option.

The variable names containing simulation results in each of the files are identical. Therefore, loading consecutive sets of data without renaming the data once it is in the MATLAB workspace results in overwriting the prior workspace variable with new data. To avoid overwriting, you can copy the result to a new MATLAB variable before loading the next set of data.

You can also write MATLAB scripts to create new signals and new parameter structures, as well as to save data and perform batch runs using the bang command (`!`).

For details on running simulations and available command-line options, see “Run Rapid Simulations” on page 29-13. For an example of a rapid simulation batch script, see the example “Run Batch Simulations Without Recompiling Generated Code” on page 29-30.

Run Rapid Simulations

- “Rapid Simulations” on page 29-14
- “Requirements for Running Rapid Simulations” on page 29-15
- “Set a Clock Time Limit for a Rapid Simulation” on page 29-15
- “Override a Model Simulation Stop Time” on page 29-15
- “Read the Parameter Vector into a Rapid Simulation” on page 29-16
- “Specify New Signal Data File for a From File Block” on page 29-16
- “Specify Signal Data File for an Inport Block” on page 29-18
- “Change Block Parameters for an RSim Simulation” on page 29-20

- “Specify a New Output File Name for a Simulation” on page 29-21
- “Specify New Output File Names for To File Blocks” on page 29-21

Rapid Simulations

Using the RSim target, you can build a model once and run multiple simulations to study effects of varying parameter settings and input signals. You can run a simulation directly from your operating system command line, redirect the command from the MATLAB command line by using the bang (!) character, or execute commands from a script.

From the operating system command line, use

```
rtwdemo_rsimtf
```

From the MATLAB command line, use

```
!rtwdemo_rsimtf
```

The following table lists ways you can use RSim target command-line options to control a simulation.

To...	Use...
Read input data for a From File block from a MAT-file other than the MAT-file used for the previous simulation	<code>model -f oldfilename.mat=newfilename.mat</code>
Print a summary of the options for RSim executable targets	<code>executable filename -h</code>
Read input data for an Inport block from a MAT-file	<code>model -i filename.mat</code>
Time out after n clock time seconds, where n is a positive integer value	<code>model -L n</code>
Write MAT-file logging data to file <code>filename.mat</code>	<code>model -o filename.mat</code>
Read a parameter vector from file <code>filename.mat</code>	<code>model -p filename.mat</code>
Override the default TCP port (17725) for external mode	<code>model -port TCPport</code>
Write MAT-file logging data to a MAT-file other than the MAT-file used for the previous simulation	<code>model -t oldfilename.mat=newfilename.mat</code>
Run the simulation until the time value <code>stoptime</code> is reached	<code>model -tf stoptime</code>
Run in verbose mode	<code>model -v</code>
Wait for the Simulink engine to start the model in external mode	<code>model -w</code>

The following sections use the `rtwdemo_rsimtf` example model in examples to illustrate some of these command-line options. In each case, the example assumes you have already done the following:

- Created or changed to a working folder.
- Opened the example model.
- Copied the data file `matlabroot/toolbox/rtw/rtwdemos/rsimdemos/rsim_tfdata.mat` to your working folder. You can perform this operation using the command:

```
copyfile(fullfile(matlabroot,'toolbox','rtw','rtwdemos',...
'rsimdemos','rsim_tfdata.mat'),pwd);
```

Requirements for Running Rapid Simulations

The following requirements apply to both fixed and variable step executables.

- You must run the RSim executable on a computer configured to run MATLAB. Also, the RSim.exe file must be able to access the MATLAB and Simulink installation folders on this machine. To obtain that access, your PATH environment variable must include /bin and /bin/(\$ARCH), where (\$ARCH) represents your operating system architecture. For example, for a personal computer running on a Windows platform, (\$ARCH) is “win64”, whereas for a Linux machine, (\$ARCH) is “glnxa64”.
- On GNU Linux platforms, to run an RSim executable, define the LD_LIBRARY_PATH environment variable to provide the path to the MATLAB installation folder, as follows:

```
% setenv LD_LIBRARY_PATH /matlab/sys/os/glnx64:$LD_LIBRARY_PATH
```

- On the Apple Macintosh OS X platform, to run RSim target executables, you must define the environment variable DYLD_LIBRARY_PATH to include the folders bin/mac and sys/os/mac under the MATLAB installation folder. For example, if your MATLAB installation is under /MATLAB, add /MATLAB/bin/mac and /MATLAB/sys/os/mac to the definition for DYLD_LIBRARY_PATH.

Set a Clock Time Limit for a Rapid Simulation

If a model experiences frequent zero crossings and the model's minor step size is small, consider setting a time limit for a rapid simulation. To set a time limit, specify the -L option with a positive integer value. The simulation aborts after running for the specified amount of clock time (not simulation time). For example,

```
!rtwdemo_rsimmtf -L 20
```

Based on your clock, after the executable runs for 20 seconds, the program terminates. You see a message similar to one of the following:

- On a Microsoft Windows platform,

```
Exiting program, time limit exceeded
Logging available data ...
```

- On The Open Group UNIX platform,

```
** Received SIGALRM (Alarm) signal @ Fri Jul 25 15:43:23 2003
** Exiting model 'rtwdemo_rsimmtf' @ Fri Jul 25 15:43:23 2003
```

You do not need to do anything to your model or to its configuration to use this option.

Override a Model Simulation Stop Time

By default, a rapid simulation runs until the simulation time reaches the time specified the Configuration Parameters dialog box, on the **Solver** pane. You can override the model simulation stop time by using the -tf option. For example, the following simulation runs until the time reaches 6.0 seconds.

```
!rtwdemo_rsimmtf -tf 6.0
```

The RSim target stops and logs output data using MAT-file data logging rules.

If the model includes a From File block, the end of the simulation is regulated by the stop time setting specified in the Configuration Parameters dialog box, on the **Solver** pane, or with the RSim target option `-tf`. The values in the block time vector are ignored. However, if the simulation time exceeds the endpoints of the time and signal matrix (if the final time is greater than the final time value of the data matrix), the signal data is extrapolated to the final time value.

Read the Parameter Vector into a Rapid Simulation

To read the model parameter vector into a rapid simulation, you must first create a MAT-file that includes the parameter structure as described in “Create a MAT-File That Includes a Model Parameter Structure” on page 29-6. You can then specify the MAT-file in the command line with the `-p` option.

For example:

- 1 Build an RSim executable for the example model `rtwdemo_rsimtf`.
- 2 Modify parameters in your model and save the parameter structure.

```
param_struct = rsimgetrtp('rtwdemo_rsimtf');
save myrsimdata.mat param_struct
```

- 3 Run the executable with the new parameter set.

```
!rtwdemo_rsimtf -p myrsimdata.mat
```

```
** Starting model 'rtwdemo_rsimtf' @ Tue Dec 27 12:30:16 2005
** created rtwdemo_rsimtf.mat **
```

- 4 Load workspace variables and plot the simulation results by entering the following commands:

```
load rtwdemo_rsimf.mat
plot(rt_yout)
```

Specify New Signal Data File for a From File Block

If your model's input data source is a From File block, you can feed the block with input data during simulation from a single MAT-file or you can change the MAT-file from one simulation to the next. Each MAT-file must adhere to the format described in “Create a MAT-File for a From File Block” on page 29-9.

To change the MAT-file after an initial simulation, you specify the executable with the `-f` option and an `oldfile.mat=newfile.mat` parameter, as shown in the following example.

- 1 Set some parameters in the MATLAB workspace. For example:

```
w = 100;
theta = 0.5;
```

- 2 Build an RSim executable for the example model `rtwdemo_rsimtf`.
- 3 Run the executable.

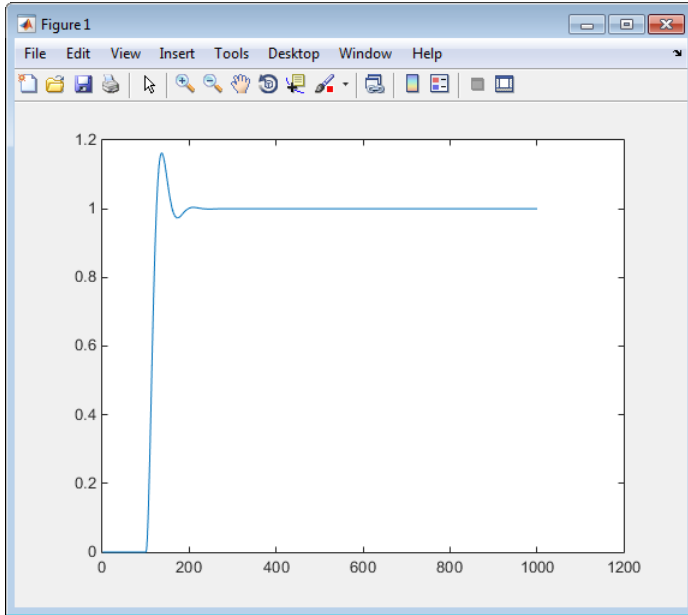
```
!rtwdemo_rsimtf
```

The RSim executable runs a set of simulations and creates output MAT-files containing the specific simulation result.

- 4 Load the workspace variables and plot the simulation results by entering the following commands:


```
load rtwdemo_rsimgtf.mat
plot(rt_yout)
```

The resulting plot shows simulation results based on default input data.



- 5 Create a new data file, `newfrom.mat`, that includes the following data:

```
t=[0:.001:1];
u=sin(100*t.*t);
tu=[t;u];
save newfrom.mat tu;
```

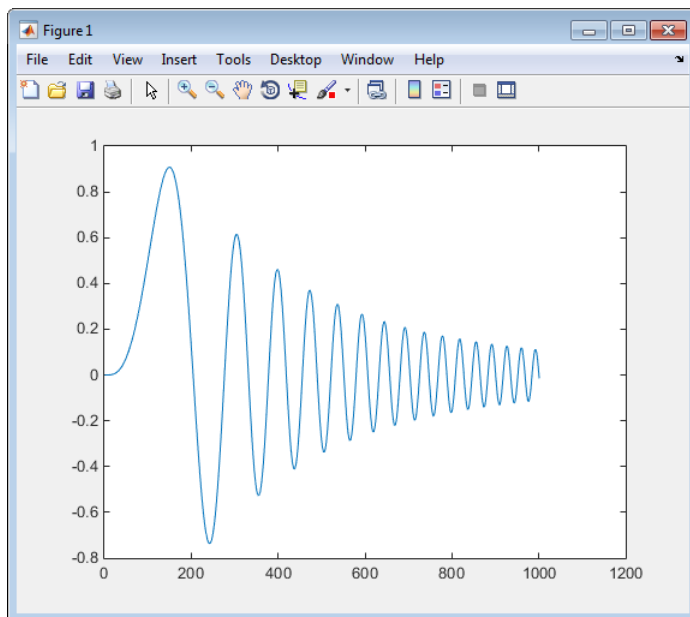
- 6 Run a rapid simulation with the new data by using the `-f` option to replace the original file, `rsim_tfdata.mat`, with `newfrom.mat`.

```
!rtwdemo_rsimgtf -f rsim_tfdata.mat=newfrom.mat
```

- 7 Load the data and plot the new results by entering the following commands:

```
load rtwdemo_rsimgtf.mat
plot(rt_yout)
```

The next figure shows the resulting plot.



From File blocks require input data of type `double`. If you need to import signal data of a data type other than `double`, use an Inport block (see “Create a MAT-File for an Inport Block” on page 29-10) or a From Workspace block with the data specified as a structure.

Workspace data must be in the format:

```
variable.time
variable.signals.values
```

If you have more than one signal, use the following format:

```
variable.time
variable.signals(1).values
variable.signals(2).values
```

Specify Signal Data File for an Inport Block

If your model's input data source is an Inport block, you can feed the block with input data during simulation from a single MAT-file or you can change the MAT-file from one simulation to the next. Each MAT-file must adhere to one of the three formats described in “Create a MAT-File for an Inport Block” on page 29-10.

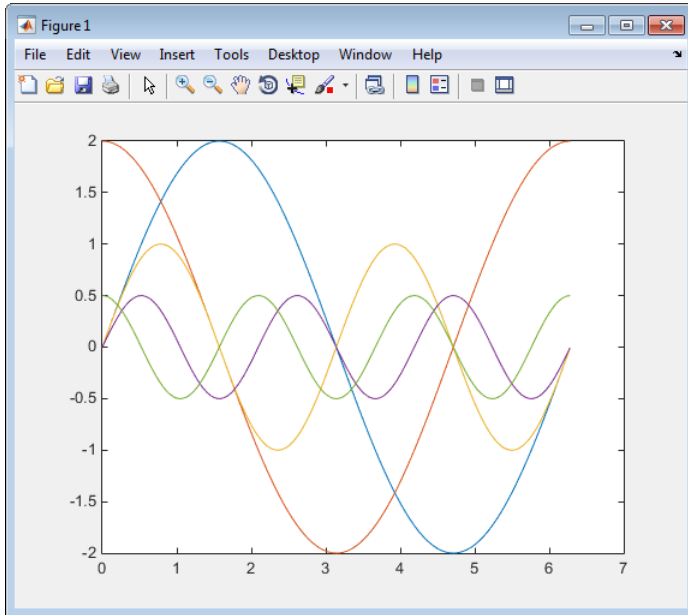
To specify the MAT-file after a simulation, you specify the executable with the `-i` option and the name of the MAT-file that contains the input data. For example:

- 1 Open the model `rtwdemo_rsim_i`.
- 2 Check the Inport block parameter settings. The following Inport block data parameter settings and specifications that you specify for the workspace variables must match settings in the MAT-file, as indicated in “Configure Inports to Provide Simulation Source Data” on page 29-4:
 - **Main > Interpolate data**
 - **Signal Attributes > Port dimensions**
 - **Signal Attributes > Data type**

- **Signal Attributes > Signal type**

- 3 Build the model.
- 4 Set up the input signals. For example:

```
t=[0:0.01:2*pi]';
s1=[2*sin(t) 2*cos(t)];
s2=sin(2*t);
s3=[0.5*sin(3*t) 0.5*cos(3*t)];
plot(t, [s1 s2 s3])
```



- 5 Prepare the MAT-file by using one of the three available file formats described in “Create a MAT-File for an Inport Block” on page 29-10. The following example defines a signal-and-time structure in the workspace and names it `var_single_struct`.

```
t=[0:0.1:2*pi]';
var_single_struct.time=t;
var_single_struct.signals(1).values(:,1)=2*sin(t);
var_single_struct.signals(1).values(:,2)=2*cos(t);
var_single_struct.signals(2).values=sin(2*t);
var_single_struct.signals(3).values(:,1)=0.5*sin(3*t);
var_single_struct.signals(3).values(:,2)=0.5*cos(3*t);
v=[var_single_struct.signals(1).values...
var_single_struct.signals(2).values...
var_single_struct.signals(3).values];
```

- 6 Save the workspace variable `var_single_struct` to MAT-file `rsim_i_single_struct`.
- 7 Run a rapid simulation with the input data by using the `-i` option. Load and plot the results.

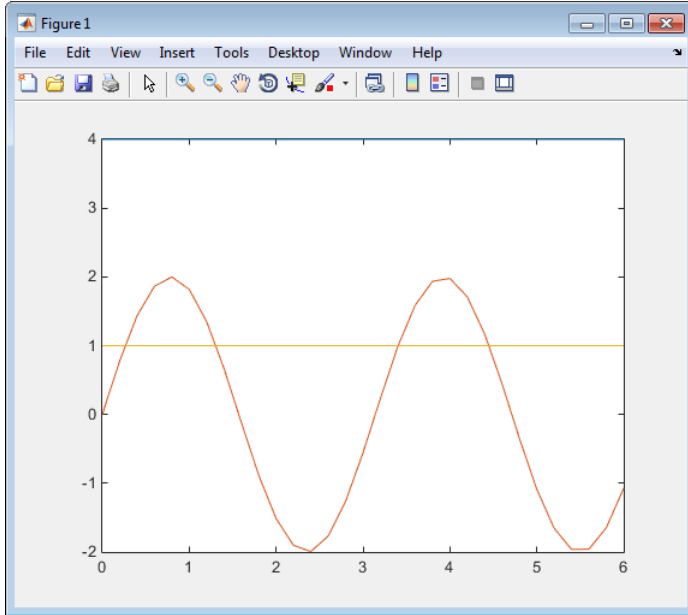
```
!rtwdemo_rsim_i -i rsim_i_single_struct.mat

** Starting model 'rtwdemo_rsim_i' @ Tue Aug 19 10:26:53 2014
*** rsim_i_single_struct.mat is successfully loaded! ***
** created rtwdemo_rsim_i.mat **

** Execution time = 0.02024185130718954s
```

- 8 Load and plot the results.

```
load rtwdemo_rsim_i.mat
plot(rt_tout, rt_yout);
```



Change Block Parameters for an RSim Simulation

As described in “Create a MAT-File That Includes a Model Parameter Structure” on page 29-6, after you alter one or more parameters in a Simulink block diagram, you can extract the parameter vector, *model_P*, for the entire model. You can then save the parameter vector, along with a model checksum, to a MAT-file. This MAT-file can be read directly by the standalone RSim executable, allowing you to replace the entire parameter vector or individual parameter values, for running studies of variations of parameter values representing coefficients, new data for input signals, and so on.

RSim can read the MAT-file and replace the entire *model_P* structure whenever you change one or more parameters, without recompiling the entire model.

For example, assume that you changed one or more parameters in your model, generated the new *model_P* vector, and saved *model_P* to a new MAT-file called *mylogfile.mat*. To run the same *rtwdemo_rsimtf* model and use these new parameter values, use the *-p* option, as shown in the following example:

```
!rtwdemo_rsimtf -p mylogfile.mat
load rtwdemo_rsimtf
plot(rt_yout)
```

If you have converted the parameter structure to a cell array for running simulations on varying data sets, as described in “Convert the Parameter Structure for Running Simulations on Varying Data Sets” on page 29-8, you must add an *@n* suffix to the MAT-file specification. *n* is the element of the cell array that contains the specific input that you want to use for the simulation.

The following example converts `param_struct` to a cell array, changes parameter values, saves the changes to MAT-file `mymatfile.mat`, and then runs the executable using the parameter values in the second element of the cell array as input.

```
param_struct = rsimgetrtf('rtwdemo_rsimtf');
param_struct = rsimsetrtpparam(param_struct,2);
param_struct.parameters{1}

ans =

    dataTypeName: 'double'
    dataTypeId: 0
    complex: 0
    dtTransIdx: 0
    values: [-140 -4900 0 4900]
    map: []
    structParamInfo: []

param_struct.parameters{2}.values=[-150 -5000 0 4950];
save mymatfile.mat param_struct;
!rtwdemo_rsimtf -p mymatfile.mat@2 -o rsim2.mat
```

Specify a New Output File Name for a Simulation

If you have specified one or more of the **Save to Workspace** parameters — **Time**, **States**, **Outputs**, or **Final States** — in the Configuration Parameters dialog box, on the **Data Import/Export** pane, the default is to save simulation logging results to the file `model.mat`. For example, the example model `rtwdemo_rsimtf` normally saves data to `rtwdemo_rsimtf.mat`, as follows:

```
!rtwdemo_rsimtf
created rtwdemo_rsimtf.mat
```

You can specify a new output file name for data logging by using the `-o` option when you run an executable.

```
!rtwdemo_rsimtf -o rsim1.mat
```

In this case, the set of parameters provided at the time of code generation, including From File block data parameters, is run.

Specify New Output File Names for To File Blocks

In much the same way as you can specify a new system output file name, you can also provide new output file names for data saved from one or more To File blocks. To do this, specify the original file name at the time of code generation with a new name, as shown in the following example:

```
!rtwdemo_rsimtf -t rtwdemo_rsimtf_data.mat=mynewsimdata.mat
```

In this case, assume that the original model wrote data to the output file `rtwdemo_rsimtf_data.mat`. Specifying a new file name forces RSim to write to the file `mynewsimdata.mat`. With this technique, you can avoid overwriting an existing simulation run.

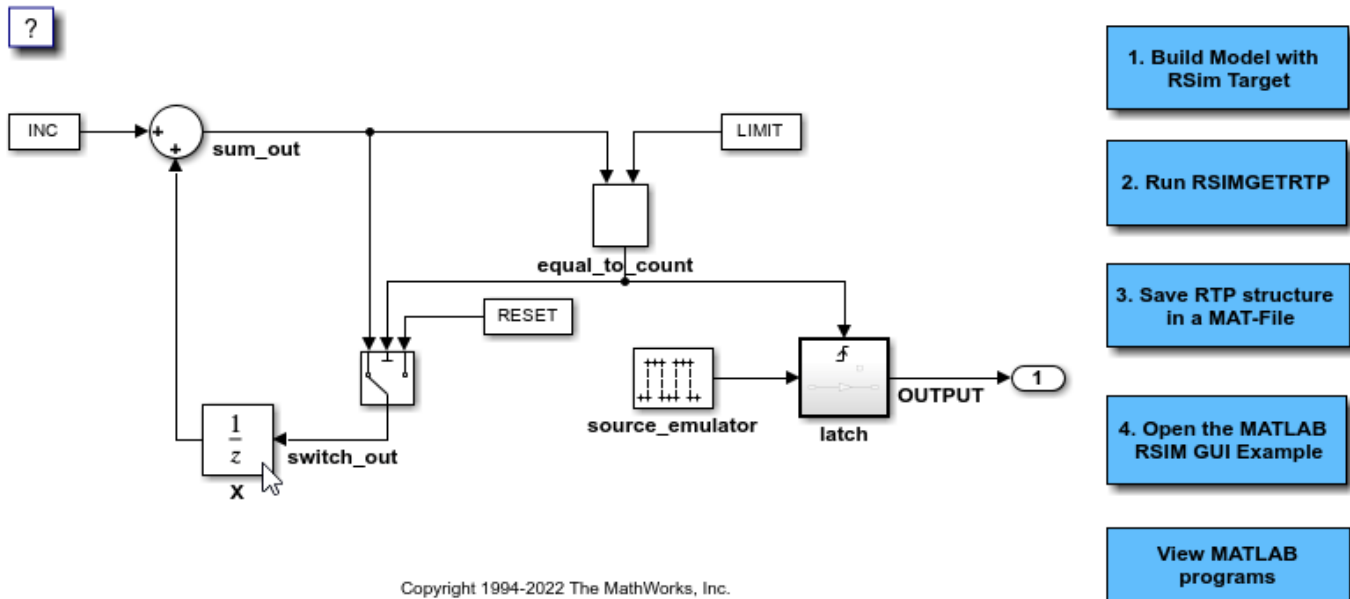
Tune Parameters Interactively During Rapid Simulation

The RSim system target file was designed to let you run batch simulations at the fastest possible speed. Using variable-step or fixed-step solvers with that system target file and a tunable parameter

data structure, whether you set **Default parameter behavior** to Tunable or to Inlined, you can create multiple parameter sets. You can run the data sets with the generated executable program (.exe on Windows). Each invocation of the executable program allows specification of the file name to use for results.

Open the example model RsimParamTuning.

```
open_system('RsimParamTuning.slx');
```



For this model, **Default parameter behavior** is set to Inlined. The model declares workspace variables as tunable parameters. To use the RSim system target file with **Default parameter behavior** set to Tunable, and without explicitly declaring tunable parameters, see “Run Batch Simulations Without Recompiling Generated Code” on page 29-30.

This model uses the RSim system target file and the `rsimgetrtp` function to allow a non real time executable to be passed new data without the need to recompile the Simulink model. This feature allows you to get a map of the tunable parameters declared in a model and save it in a MAT-file. You can then create your own MATLAB graphical user interface (GUI) or a standalone GUI (independent of MATLAB) to read and write the MAT-file and rerun the executable program to produce new output files.

In the model, double-click the buttons sequentially to run the example.

To review the code used to create the MATLAB and standalone GUIs, double-click the **View MATLAB programs** button.

For more information, see “Accelerate, Refine, and Test Hybrid Dynamic System on Host Computer by Using RSim System Target File” on page 29-2.

Rapid Simulation Target Limitations

The RSim target has the following limitations:

- Does not support algebraic loops.
- Does not support Interpreted MATLAB Function blocks.
- Does not support noninlined MATLAB language or Fortran S-functions.
- If an RSim build includes referenced models (by using Model blocks), set up these models to use fixed-step solvers to generate code for them. The top model, however, can use a variable-step solver as long as the blocks in the referenced models are discrete.
- In certain cases, changing block parameters can result in structural changes to your model that change the model checksum. An example of such a change is changing the number of delays in a DSP simulation. In such cases, you must regenerate the code for the model.

See Also

More About

- “Acceleration”
- “Use S-Function Target for Model or Subsystem” on page 29-40

Run Rapid Simulations over a Range of Parameter Values

This example shows how to use the RSim system target file to run simulations over a range of parameter values. The example uses the Van der Pol oscillator and performs a parameter sweep over a range of initial state values to obtain the phase diagram of a nonlinear system.

You can customize this example for your own application by modifying the MATLAB® script used to build this example. Click the link in the top left corner of this page to edit the MATLAB script. Click the link in the top right corner to run this example from MATLAB. Run the example from writable location. The example creates files that you may want to investigate later.

For this example, set model configuration parameter **Default parameter behavior** to **Inlined**. In the example, you create `Simulink.Parameter` objects as tunable parameters. To use RSim with **Default parameter behavior** set to **Tunable**, and without explicitly declaring tunable parameters, see “Run Batch Simulations Without Recompiling Generated Code” on page 29-30.

To run multiple simulations in the Simulink® environment, consider using rapid accelerator instead of RSim. For information about rapid accelerator, see “What Is Acceleration?”. To sweep parameter values, see “Optimize, Estimate, and Sweep Block Parameter Values”.

Preparation Model

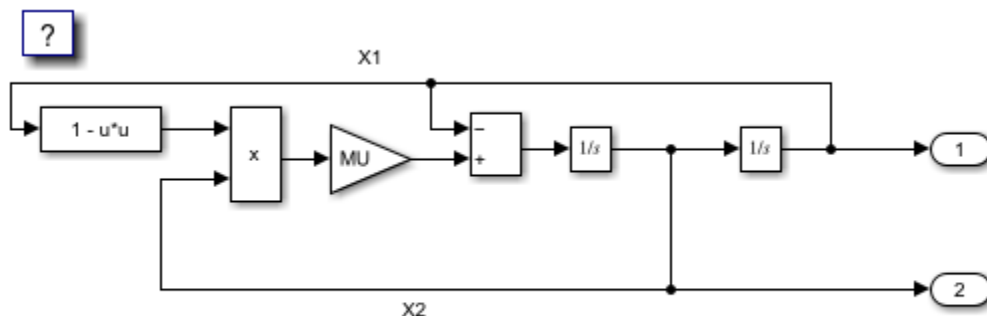
Make sure the current folder is writable. The example creates files.

```
[stat, fa] = fileattrib(pwd);
if ~fa.UserWrite
    disp('This script must be run in a writable directory');
    return
end
```

Open the model. Then, open the **Simulink Coder** app. Set model configuration parameter **System target file** to `rsim.tlc`.

For more information on doing this graphically and setting up other RSim target related options, see “Configure a System Target File” on page 21-2.

```
mdlName = 'rtwdemo_rsim_vdp';
open_system(mdlName);
cs = getActiveConfigSet(mdlName);
cs.switchTarget('rsim.tlc', []);
```



Copyright 2005-2020 The MathWorks, Inc.

Specify variables INIT_X1 (the initial condition for state x1), INIT_X2 (the initial condition for state x2), and MU (the gain value) as tunable. To create tunable parameters, convert the variables to Simulink.Parameter objects, and configure the objects to use a storage class other than Auto. In this example you investigate how the state trajectories evolve from different initial values for states x1 and x2 in the model.

```
INIT_X1 = Simulink.Parameter(INIT_X1);
INIT_X1.StorageClass = 'Model default';
```

```
INIT_X2 = Simulink.Parameter(INIT_X2);
INIT_X2.StorageClass = 'Model default';
```

```
MU = Simulink.Parameter(MU);
MU.StorageClass = 'Model default';
```

Define the names of files that will be created.

```
prmFileName = [mdlName, '_prm_sets.mat'];
logFileName = [mdlName, '_run_scr.log'];
batFileName = [mdlName, '_run_scr'];
exeFileName = mdlName;
if ispc
    exeFileName = [exeFileName, '.exe'];
    batFileName = [batFileName, '.bat'];
end
aggDataFile = [mdlName, '_results'];
startTime = cputime;
```

Build Model

Build the RSim executable program for the model. During the build process, a structural checksum is calculated for the model and embedded into the generated executable program. This checksum is used to check that a parameter set passed to the executable program is compatible with the program.

```
slbuild(mdlName);

### Starting build procedure for: rtwdemo_rsim_vdp
### Successful completion of build procedure for: rtwdemo_rsim_vdp
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_rsim_vdp	Code generated and compiled	Code generation information file does not exist.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 37.492s
```

Get Default Parameter Set for Model

Get the default rtP structure (parameter set) for the model. The modelChecksum field in the rtP structure is the structural checksum of the model. This must match the checksum embedded in the RSim executable program. If the two checksums do not match, the executable program produces an error. rsimgetrtP generates an rtP structure with entries for the named tunable variables INIT_X1, INIT_X2, and MU.

```

rtp = rsimgetrtp mdlName

rtp =
  struct with fields:
      modelChecksum: [410569968 1.5914e+09 3.4411e+09 ... ]
      parameters: [1x1 struct]
      globalParameterInfo: [1x1 struct]
      collapsedBaseWorkspaceVariables: []
      nonTunableVariables: [1x0 struct]

```

Create Parameter Sets

Using the `rtp` structure, build a structure array with different values for the tunable variables in the model. As mentioned earlier, in this example you want to see how the state trajectories evolve for different initial values for states `x1` and `x2` in the model. To do this, generate different parameter sets with different values for `INIT_X1` and `INIT_X2` and leave tunable variable `MU` set to the default value.

```

INIT_X1_vals = -5:1:5;
INIT_X2_vals = -5:1:5;
MU_vals = 1;
nPrmSets = length(INIT_X1_vals)*length(INIT_X2_vals)*length(MU_vals)

```

```

nPrmSets =
  121

```

This example has `nPrmSets` parameter sets. You need to run that many simulations. Initialize `aggData`, which is a structure array that holds the parameter set and corresponding results.

```

aggData = struct('tout', [], 'yout', [], ...
                'prms', struct('INIT_X1', [], 'INIT_X2', [], 'MU', []))
aggData = repmat(aggData, nPrmSets, 1);

```

```

aggData =
  struct with fields:
      tout: []
      yout: []
      prms: [1x1 struct]

```

The utility function `rsimsetrtpparam` builds the `rtP` structure by adding parameter sets one at a time with different parameter values.

```

idx = 1;
for iX1 = INIT_X1_vals
    for iX2 = INIT_X2_vals
        for iMU = MU_vals
            rtp = rsimsetrtpparam(rtp, idx, 'INIT_X1', iX1, 'INIT_X2', iX2, 'MU', iMU);
        end
    end
end

```

```

        aggData(idx).prms.INIT_X1 = iX1;
        aggData(idx).prms.INIT_X2 = iX2;
        aggData(idx).prms.MU      = iMU;
        idx = idx + 1;
    end
end
end

```

Save the `rtP` structure array with the parameter sets to a MAT-file.

```
save(prmFileName, 'rtp');
```

Create Batch File

Create a batch script file to run the RSim executable program over the parameter sets. Each run reads the specified parameter set from the parameter MAT-file and writes the results to the specified output MAT-file. Use the time out option so that if a run hangs (for example, because the model has a singularity for that parameter set), abort the run after the specified time limit and proceed to the next run.

For example, this command (on Windows®) specifies using the third parameter set from the `rtP` structure in `prm.mat`, writes the results to `run3.mat`, and aborts execution if a run takes longer than 3600 seconds of CPU time. While running, the script pipes messages from `model.exe` to `run.log`, which you can use to debug issues.

```

    model.exe -p prm.mat@3 -o run3.mat -L 3600 2>&1>> run.log

fid = fopen(batFileName, 'w');
idx = 1;
for iX1 = INIT_X1_vals
    for iX2 = INIT_X2_vals
        for iMU = MU_vals
            outMatFile = [mdlName, '_run', num2str(idx), '.mat'];
            cmd = [exeFileName, ...
                  '-p ', prmFileName, '@', int2str(idx), ...
                  '-o ', outMatFile, ...
                  '-L 3600'];
            if ispc
                cmd = [cmd, ' 2>&1>> ', logFileName];
            else % (unix)
                cmd = ['./ filesep cmd, ' 1> ', logFileName, ' 2>&1'];
            end
            fprintf(fid, ['echo "', cmd, '"\n']);
            fprintf(fid, [cmd, '\n']);
            idx = idx + 1;
        end
    end
end
if isunix,
    system(['touch ', logFileName]);
    system(['chmod +x ', batFileName]);
end
fclose(fid);

```

Creating a batch/script file to run simulations enables you to call the system command once to run the simulations (or even run the batch script outside MATLAB) instead of calling the system command in a loop for each simulation. This improves performance because the system command has significant overhead.

Execute Batch File to Run Simulations

Run the batch/script file, which runs the RSim executable program once for each parameter set and saves the results to a different MAT-file each time. You can run the batch file from outside MATLAB.

```
[stat, res] = system(['.' filesep batFileName]);
if stat ~= 0
    error(['Error running batch file ', batFileName, ' :', res]);
end
```

This example logs simulation run results in one batch file as it runs the batch file to sequentially run n simulations over n parameter sets. You can modify the script to generate multiple batch files and run them in parallel by distributing them across multiple computers. Also, you can run the batch files without launching MATLAB.

Load Output MAT-Files and Collate Results

Collect the simulation results from the output MAT-files into the structure `aggData`. If the output MAT-file corresponding to a particular run is not found, set the results corresponding to that run to NaN (not a number). This situation occurs if a simulation run with a particular set of parameters encounters singularities in the model.

```
idx = 1;
for iX1 = INIT_X1_vals
    for iX2 = INIT_X2_vals
        for iMU = MU_vals
            outMatFile = [mdlName, '_run', num2str(idx), '.mat'];
            if exist(outMatFile, 'file')
                load(outMatFile);
                aggData(idx).tout = rt_tout;
                aggData(idx).yout = rt_yout;
            else
                aggData(idx).tout = nan;
                aggData(idx).yout = nan;
            end
            idx = idx + 1;
        end
    end
end
```

Save the `aggData` structure to the results MAT-file. At this point, you can delete the other MAT-files, as the `aggData` data structure contains the aggregation of input (parameters sets) and output data (simulation results).

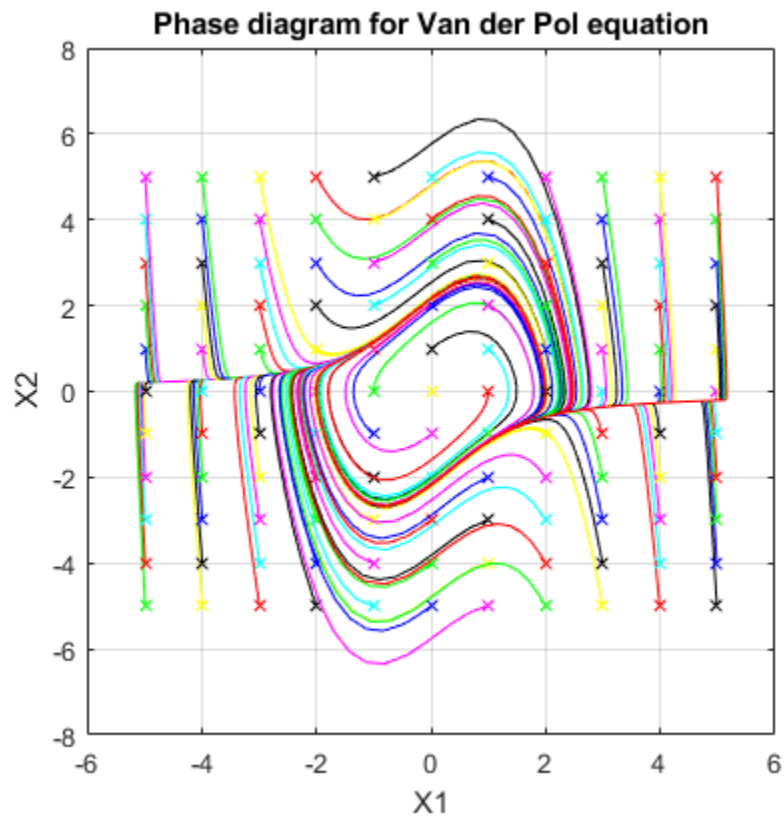
```
save(aggDataFile, 'aggData');
disp(['Took ', num2str(cputime-startTime), ...
    ' seconds to generate results from ', ...
    num2str(nPrmSets), ' simulation runs (Steps 2 to 7).']);
```

Took 29.0781 seconds to generate results from 121 simulation runs (Steps 2 to 7).

Analyze Simulation Results

Plot the phase diagram (X2 versus X1) with different initial values for X1 and X2. The diagram shows that, irrespective of the initial condition, the Van der Pol oscillator converges to its natural oscillator mode.

```
colors = {'b','g','r','c','m','y','k'}; nColors = length(colors);
for idx = 1:nPrmSets
    col = colors{idx - nColors*floor(idx/nColors) + 1};
    plot(aggData(idx).prms.INIT_X1, aggData(idx).prms.INIT_X2, [col,'x'], ...
        aggData(idx).yout(:,1), aggData(idx).yout(:,2),col);
    hold on
end
grid on
xlabel('X1');
ylabel('X2');
axis('square');
title('Phase diagram for Van der Pol equation');
```



Run Batch Simulations Without Recompiling Generated Code

This example shows how to run batch simulations without recompiling the generated code. The example modifies input signal data and model parameters by reading data from a MAT-file. In the first part (steps 1-5), ten parameter sets are created from the Simulink® model by changing the transfer function damping factor. The ten parameter sets are saved to a MAT-file, and the RSim executable reads the specified parameter set from the file. In the second part (step 6-7) of this example, five sets of signal data chirps are created with increasingly high frequencies. In both parts, the RSim executable runs the set of simulations and creates output MAT-files containing the specific simulation result. Finally, a composite of runs appears in a MATLAB® figure.

To quickly run multiple simulations in the Simulink environment, consider using rapid accelerator instead of RSim. See “What Is Acceleration?”.

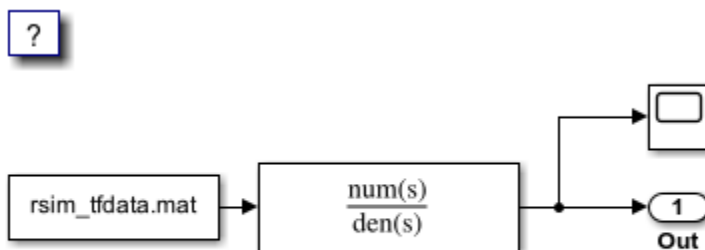
Step 1. Preparation

Make sure the current directory is writable because this example will be creating files.

```
[stat, fa] = fileattrib(pwd);
if ~fa.UserWrite
    disp('This script must be run in a writable directory');
    return;
end
```

Open the model and configure it to use the RSim target. For more information on doing this graphically and setting up other RSim target related options, see “Configure a System Target File” (Embedded Coder).

```
mdlName = 'rtwdemo_rsimgtf';
open_system(mdlName);
cs = getActiveConfigSet(mdlName);
cs.switchTarget('rsim.tlc', []);
```



Copyright 2005-2020 The MathWorks, Inc.

The MAT-file rsim_tfdata.mat is required in the local directory.

```
if ~isempty(dir('rsim_tfdata.mat')),
    delete('rsim_tfdata.mat');
end
str1 = fullfile(matlabroot, 'toolbox', 'rtw', 'rtwdemos', 'rsimgdemos', 'rsim_tfdata.mat');
str2 = ['copyfile('', str1, '', 'rsim_tfdata.mat', 'writable')'];
eval(str2);
```

Step 2. Build the Model

Build the RSim executable for the model. During the build process, a structural checksum is calculated for the model and embedded into the generated executable. This checksum is used to check that a parameter set passed to the executable is compatible with it.

```
evalin('base','w = 70;')
evalin('base','theta = 1.0;')
disp('Building compiled RSim simulation.')
slbuild mdlName);

Building compiled RSim simulation.
### Starting build procedure for: rtwdemo_rsimtf
### Successful completion of build procedure for: rtwdemo_rsimtf
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_rsimtf	Code generated and compiled.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 9.7221s

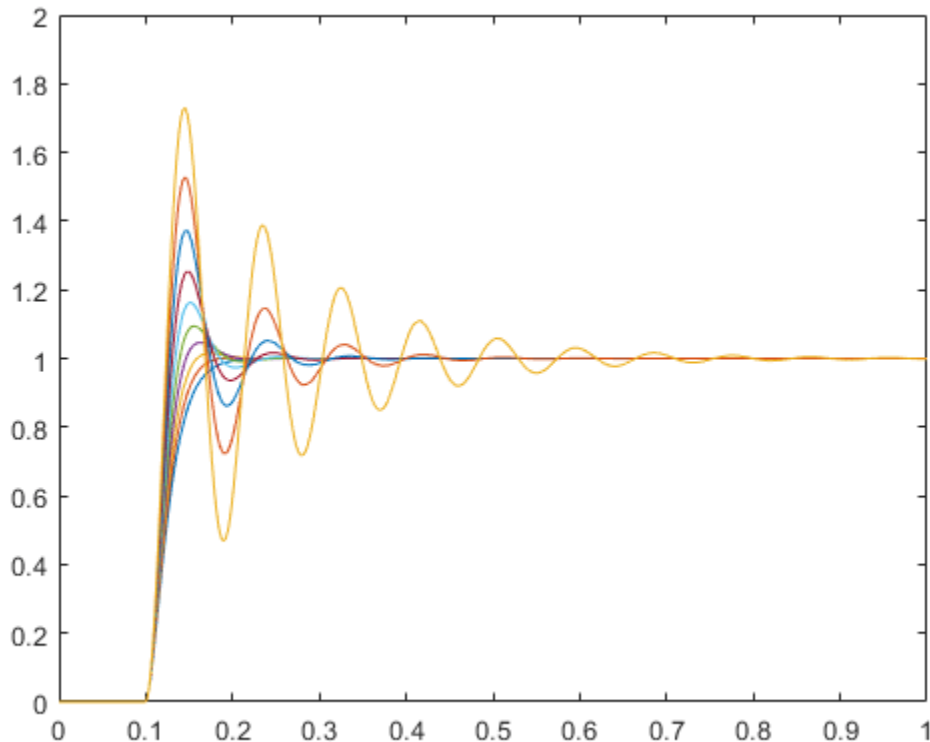
Step 3. Get the Default Parameter Set and Create 10 Parameters Sets

```
disp('Creating rtP data files')
for i=1:10
    % Extract current rtP structure using new damping factor.
    [rtpstruct]=evalin('base','rsimgetrtP(''rtwdemo_rsimtf'')');
    savestr = strcat('save_params',num2str(i),'.mat rtpstruct');
    eval(savestr);
    evalin('base','theta = theta - .1;');
end
disp('Finished creating parameter data files.')
```

Creating rtP data files
Finished creating parameter data files.

Step 4. Run 10 RSim Simulations Using New Parameter Sets and Plot the Results

```
figure
for i=1:10
    % Bang out and run a simulation using new parameter data
    runstr = ['./', filesep, 'rtwdemo_rsimtf -p params',num2str(i),'.mat', ' -v'];
    [status, result] = system(runstr);
    if status ~= 0, error(result); end
    % Load simulation data into MATLAB for plotting.
    load rtwdemo_rsimtf.mat;
    axis([0 1 0 2]);
    plot(rt_tout, rt_yout)
    hold on
end
```



The plot shows 10 simulations, each using a different damping factor.

Step 5. Set Up a Time Vector and an Initial Frequency Vector

The time vector has 4096 points in the event we want to do windowing and spectral analysis on simulation results.

```
dt = .001;
nn = [0:1:4095];
t = dt*nn; [m,n] = size(t);
wlo = 1; whi = 4;
omega = [wlo:((whi-wlo)/n):whi - (whi-wlo)/n];
```

Step 6. Create 5 Sets of Signal Data in MAT Files

Creating .mat files with chirp data.

```
disp('This part of the example illustrates a sequence of 5 plots. Each')
disp('plot shows an input chirp signal of certain frequency range.')
for i = 1:5
    wlo = whi; whi = 3*whi; % keep increasing frequencies
    omega = [wlo:((whi-wlo)/n):whi - (whi-wlo)/n];
    % In a real application we recommend shaping the chirp using
    % a windowing function (hamming or hanning window, etc.)
    % This example does not use a windowing function.
    u = sin(omega.*t);
    tudata = [t;u];
    % At each pass, save one more set of tudata to the next
```

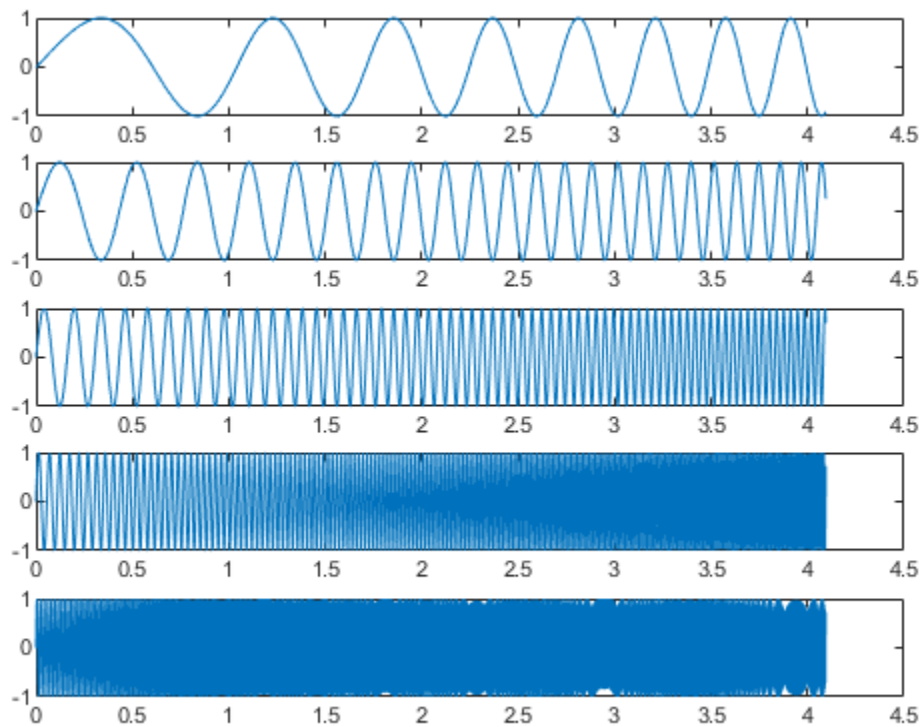


```

% .mat file.
savestr = strcat('save sweep',num2str(i),'.mat tudata');
eval(savestr);
% Display each chirp. Note that this is only input data.
% Simulations have not been run yet.
plotstr = strcat('subplot(5,1,',num2str(i),')');
eval(plotstr);
plot(t,u)
pause(0.3)
end

```

This part of the example illustrates a sequence of 5 plots. Each plot shows an input chirp signal of certain frequency range.



Step 7. Run the RSim Compiled Simulation Using New Signal Data

Replace the original signal data (rsim_tfdata.mat) with the files sweep1.mat, sweep2.mat, and so on.

```

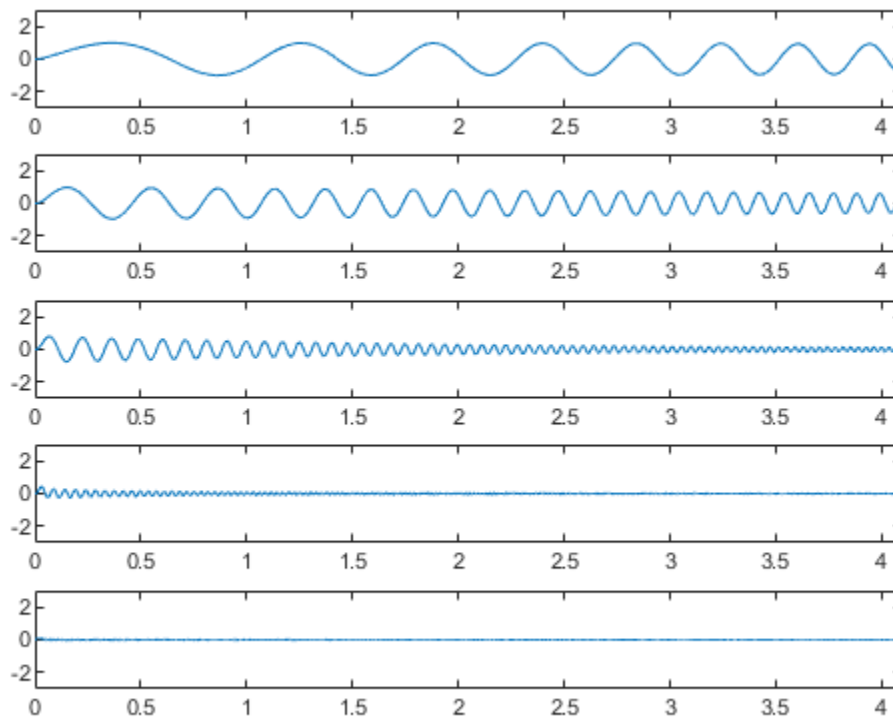
disp('Starting batch simulations.')
for i = 1:5
% Bang out and run the next set of data with RSim
runstr = ['. ', filesep, 'rtwdemo_rsimtf -f rsim_tfdata.mat=sweep', ...
num2str(i), '.mat -v -tf 4.096'];
[status, result] = system(runstr);
if status ~= 0, error(result); end
% Load the data to MATLAB and plot the results.
load rtwdemo_rsimtf.mat
plotstr = strcat('subplot(5,1,',num2str(i),')');

```

```
eval(plotstr);  
plot(rt_tout, rt_yout); axis([0 4.1 -3 3]);  
end  
zoom on  
% cleanup  
evalin('base','clear w theta')  
disp('This part of the example illustrates a sequence of 5 plots. Each plot')  
disp('shows the simulation results for the next frequency range. Using the')  
disp('mouse, zoom in on each signal to observe signal amplitudes.')  
close_system mdlName, 0);
```

Starting batch simulations.

This part of the example illustrates a sequence of 5 plots. Each plot shows the simulation results for the next frequency range. Using the mouse, zoom in on each signal to observe signal amplitudes.



Use MAT-Files to Feed Data to Inport Blocks for Rapid Simulations

This example shows how the code generator RSim -i option lets you use a MAT-file as the input data source for Inport blocks for rapid simulations. The data in such a MAT-file can be presented in the following formats:

- One variable that defines a time/input data matrix of double values.
- One variable that defines a structure that uses a combination of Simulink® data types.
- Multiple variables, each defining a structure that uses a combination of Simulink® data types.

This flexibility lends itself well to applications for which you must run simulations over a range of input data stored in different data files. This example explains how to use this feature.

To quickly run multiple simulations in the Simulink environment, consider using rapid accelerator instead of RSim. See “What Is Acceleration?”.

Prepare Model

Open the model and save a copy to a writable location. Then, open the **Simulink Coder** app. Set model configuration parameter *System target file to `rsim.tlc`.

For more information on doing this graphically and setting up other model configuration parameters relate to the RSim system target file, see “Configure a System Target File” (Embedded Coder).

```
mdlName = 'rtwdemo_rsim_i';
open_system(mdlName);
cs = getActiveConfigSet(mdlName);
cs.switchTarget('rsim.tlc',[]);
```

Configure Inport Blocks

To use the RSim -i option, you must configure each Inport block properly. You can double-click an Inport block to view its properties. By default, Inport blocks inherit their properties from downstream blocks. Before you can import data from external MAT-files, you must set the parameters of each Inport block to match the data in the MAT file. In most cases, these parameters of an Inport block must be set:

- **Interpolate Data**
- **Port Dimensions**
- **Data Type**
- **Signal Type**

For more information on these parameters, click the Help button. This example model includes three Inport blocks. A requirement for the example is for Inport 1 and Inport 2 to interpolate between data, and for Inport 3 to not interpolate. The dimension of the Inport blocks are 2, 1, and 2, respectively. Signals are real. The settings are as follows:

```
for i =1:3
    portName      =['/In', num2str(i)];
    Interp        = get_param(strcat(mdlName,portName), 'Interpolate');
    PortDimension = get_param(strcat(mdlName,portName), 'PortDimensions');
```

```

    DataType      = get_param(strcat mdlName, portName), 'OutDataTypeStr');
    SignalType    = get_param(strcat mdlName, portName), 'SignalType');
end

```

- For inports In1 and In2, interpolation is selected. For inport In3, interpolation is cleared.
- For inports In1 and In3, port dimension is set to 2. For inport In2, port dimension is set to 1.
- For inports In1, In2, and In3, data type is double and signal type is real.

Build Model

Build an executable for the model. During the build process, a structural checksum is calculated and embedded into the generated executable program. This checksum is used to check that a parameter set passed to the executable program is compatible with the program.

```
evalc('slbuild mdlName');
```

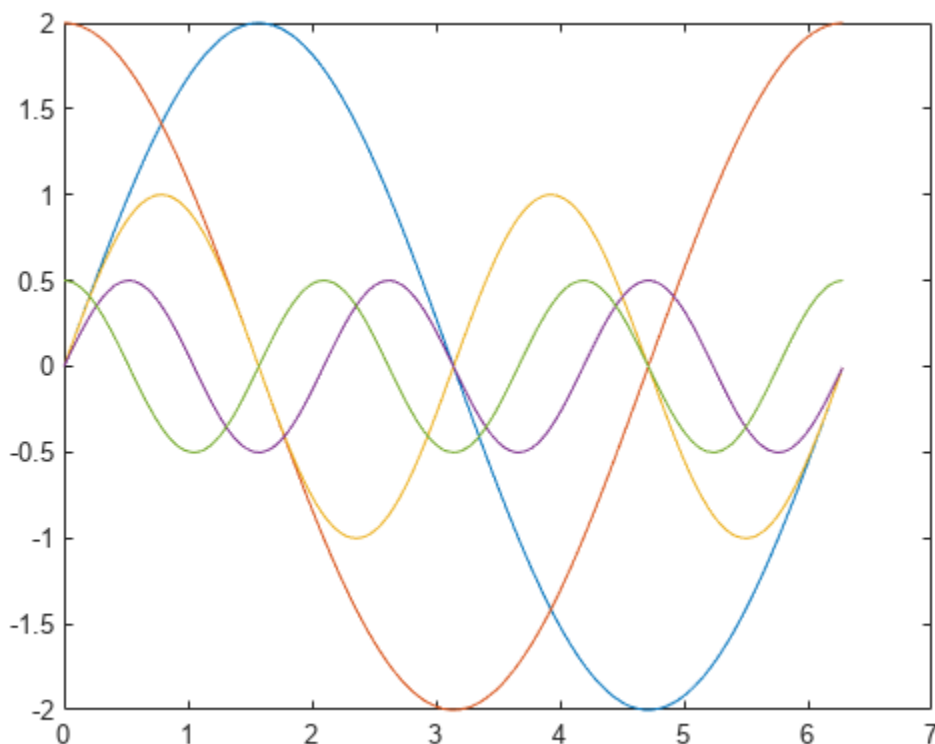
Identify Signals for Data File

After you configure the Inport block, prepare the data file based on the Inport blocks. This figure shows the input signals to be used.

```

t=[0:0.01:2*pi]';
s1 = [2*sin(t) 2*cos(t)];
s2 = sin(2*t);
s3 = [0.5*sin(3*t) 0.5*cos(3*t)];
figure;
plot(t, [s1 s2 s3]);

```



This figure displays the input signals.

Prepare MAT-File

Generally, you can create the MAT-file from a workspace variable. The RSim -i option supports three data file formats:

One variable in TU matrix format of doubles

For this format, the first column is the time vector and the remaining columns are input vectors. The number of columns in the TU matrix equals the sum of the dimensions of the root Inport blocks plus 1. This MATLAB® code generates a MAT-file containing one variable `var_matrix` in TU matrix format. You can use this format only if the input ports in the model have the same data type.

```
t=[0:0.1:2*pi]';
Ina1 = [2*sin(t) 2*cos(t)];
Ina2 = sin(2*t);
Ina3 = [0.5*sin(3*t) 0.5*cos(3*t)];
var_matrix = [t Ina1 Ina2 Ina3];
save rsim_i_matrix.mat var_matrix;
```

MAT-file `rsim_i_matrix.mat` contains one variable `var_matrix` in TU matrix format.

One variable in structure format

For this format, the variable must contain two fields: time and signals. If you set the block parameter **Interpolate data** for one of the Inport blocks, the time field of the variable must not be an empty vector and the width of the signals must equal the total width of the Inport blocks. This code generates a MAT-file that contains one variable `var_matrix` in signal variable structure format. This format is more flexible than the TU matrix format because it can support input ports with different data types.

```
t= [0:0.1:2*pi]';
var_single_struct.time = t;
var_single_struct.signals(1).values(:,1) = 2*sin(t);
var_single_struct.signals(1).values(:,2) = 2*cos(t);
var_single_struct.signals(2).values = sin(2*t);
var_single_struct.signals(3).values(:,1) = 0.5*sin(3*t) ;
var_single_struct.signals(3).values(:,2) = 0.5*cos(3*t) ;
v=[var_single_struct.signals(1).values var_single_struct.signals(2).values ...
   var_single_struct.signals(3).values ];
save rsim_i_single_struct.mat var_single_struct;
```

MAT-file `rsim_i_single_struct.mat` contains one variable `var_single_struct` in struct format.

Multiple variables in structure format

For this format, the number of variables equals the number of Inport blocks. Different variables can have different time vectors. This code generates a MAT-file that contains multiple variables, each in structure format. This is the most flexible format because it allows each Inport block to have its own time vector.

```
t= [0:0.1:2*pi]';
Inb1.time = t;
Inb1.signals.values(:,1) = 2*sin(t);
Inb1.signals.values(:,2) = 2*cos(t);
```

```

t= [0:0.2:2*pi]';
Inb2.time = t;
Inb2.signals.values(:,1) = sin(2*t);
t= [0:0.1:2*pi]';
Inb3.time = t;
Inb3.signals.values(:,1) = 0.5*sin(3*t);
Inb3.signals.values(:,2) = 0.5*cos(3*t);
save rsim_i_multi_struct.mat Inb1;
save rsim_i_multi_struct.mat Inb2 -append;
save rsim_i_multi_struct.mat Inb3 -append;

```

MAT-file `rsim_i_multi_struct.mat` contains three variables `Inb1`, `Inb2`, and `Inb3` in struct format. Use the `save` command with the `-append` option to preserve the order of variables in the generated MAT-file.

Run RSim Simulations and Plot Results

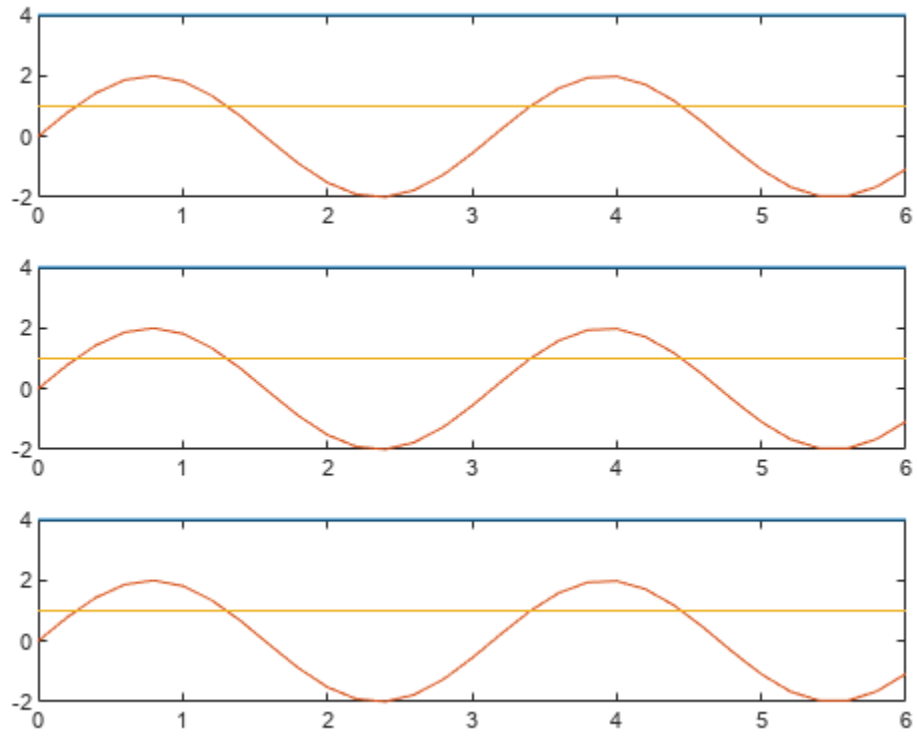
You can use RSim `-i` options for batch mode simulation. Prepare different MAT-files and run the RSim program executable with them.

This part of the example shows three plots. Each plot shows simulation results for a MAT-file with a different variable format. The model is compiled once.

```

figure
fileName = ({'rsim_i_matrix', 'rsim_i_single_struct', 'rsim_i_multi_struct'});
for i=1:3
    % bang out and run a simulation using new parameter data
    name = fileName(i);
    runstr = ['./', filesep, 'rtwdemo_rsim_i -i ',char(name),'.mat', ' -v'];
    evalc('system(runstr)');
    pause(0.5);
    % load simulation data into MATLAB(R) for plotting.
    load rtwdemo_rsim_i.mat;
    subplot(3,1,i);
    axis([0,6, -5, 5]);
    plot(rt_tout, rt_yout);
    hold on
end

```



```
close_system mdlName, 0;
```

Use S-Function Target for Model or Subsystem

S-functions are a class of system target file for which the code generator can produce code. An S-function target can encapsulate a subsystem to increase its execution efficiency, facilitate code reuse, and protect its intellectual property.

Note While you can use the S-function target to deploy an application component for reuse while shielding its internal logic from inspection and modification, the preferred solutions for protecting intellectual property in distributed components are:

- The protected model, a referenced model that hides block and line information. For more information, see “Protect Models to Conceal Contents” on page 12-2.
- The shared library system target file, used to generate a shared library for a model or subsystem for use in a system simulation external to Simulink. For more information, see “Package Generated Code as Shared Libraries” (Embedded Coder).

These solutions also increase execution efficiency and facilitate code reuse.

You can use a generated S-function target with a Generated S-Function block.

The 'S-Function' value for the CodeFormat TLC variable used by the S-function target generates code that conforms to the Simulink C MEX S-function application programming interface (API).

Required Files for S-Function Deployment

There are different files required to deploy a generated S-Function block for simulation versus code generation.

To deploy your generated S-Function block for inclusion in other models *for simulation*, you need only provide the binary MEX-file object that was generated in the current working folder when the S-Function block was created. The required file is:

- `subsys_sf.mexext`

where `subsys` is the subsystem name and `mexext` is a platform-dependent MEX-file extension (see `mexext`). For example, `SourceSubsys_sf.mexw64`.

To deploy your generated S-Function block for inclusion in other models *for code generation*, provide the files that were generated in the current working folder when the S-Function block was created. The required files are:

- `subsys_sf.c` or `.cpp`, where `subsys` is the subsystem name (for example, `SourceSubsys_sf.c`)
- `subsys_sf.h`
- `subsys_sf.mexext`, where `mexext` is a platform-dependent MEX-file extension (see `mexext`)
- Subfolder `subsys_sfcn_rtw` and its contents

The `rtwsfcn.tlc` system target file is provided for use with the S-function target.

The generated S-function code uses **Configuration Parameters > Hardware Implementation** parameter values that match the host system on which the function was built. When you use the S-function in a model for code generation, make sure that these parameter values for the model match the parameter values of the S-function.

Sample Time Propagation in Generated S-Functions

A generated S-Function block can inherit its sample time from the model in which it is placed if certain criteria are met. Conditions that govern sample time propagation for both Model blocks and generated S-Function blocks are described in “Referenced Model Sample Times” and “S-Functions That Specify Sample Time Inheritance Rules” on page 14-77.

To generate an S-Function block that meets the criteria for inheriting sample time, you must constrain the solver for the model from which the S-Function block is generated. Set model configuration parameter **Type** to **Fixed-step** and **Periodic sample time constraint** to **Ensure sample time independent**. If the model is unable to inherit sample times, this setting causes the Simulink software to display an error message when building the model. For more information about this option, see [Periodic sample time constraint](#).

Solver Type for Top Models with Generated S-Functions

The table shows the possible combinations of top model solver types as these types relate to whether the model has discrete or continuous sample times and solver types for generated S-functions.

Top Model Solver Options and Sample Times

	Model Configuration Parameters: Top-level model configuration	
Sample Times	Solver Options, Type: Variable-step	Solver Options, Type: Fixed-step
Discrete	Generated S-function requires a variable-step solver	Generated S-function can have a variable-step solver or a fixed-step solver
Continuous	Generated S-function requires a variable-step solver	Generated S-function requires a fixed-step solver

S-functions generated from a subsystem have parameters that are hardcoded into the block. Simulink calculates parameters such as sample time when it generates the block, not during simulation run time. It is important to verify whether the generated S-Function block works as expected in the destination model.

Tunable Parameters in Generated S-Functions

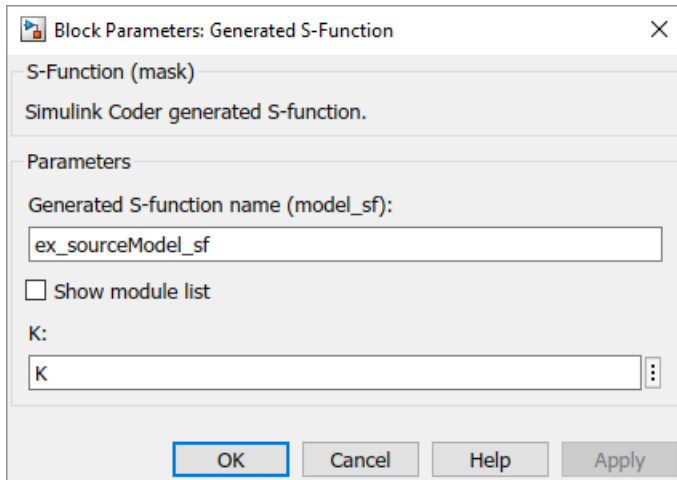
To use tunable parameters in generated S-functions, use model configuration parameters to declare desired block parameters tunable. See “[Declare Workspace Variables as Tunable Parameters Using the Model Parameter Configuration Dialog Box](#)” on page 15-198.

Block parameters that you declare as tunable with the `auto` storage class in the source model become tunable parameters of the generated S-function. These parameters do not become part of a generated `model_P` (formerly `rtP`) parameter data structure, as they would in code generated from

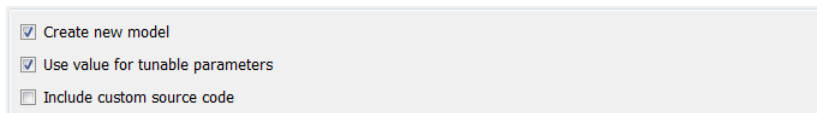
other system target files. Instead, the generated code accesses these parameters by using MEX API calls such as `mxGetPr` or `mxGetData`. Your code should access these parameters in the same way.

For more information on MEX API calls, see “About C MEX S-Functions” and “Integrate MATLAB with External Programming Languages and Systems”.

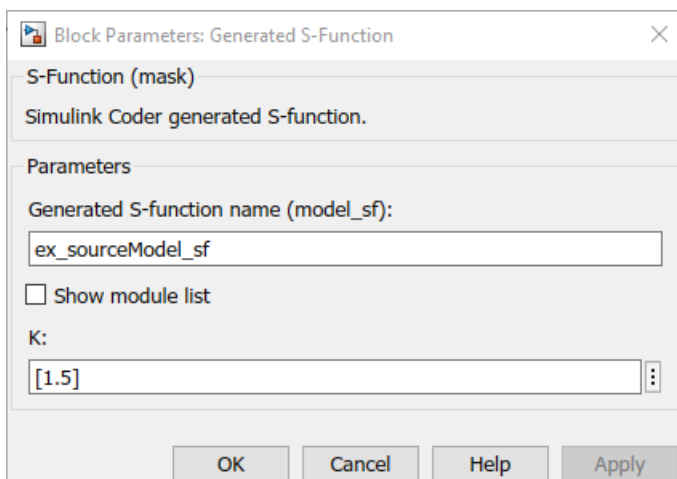
S-Function blocks created by using the S-function target are automatically masked. The mask displays each tunable parameter in an edit field. By default, the edit field displays the parameter by variable name, as in the following example.



You can choose to display the value of the parameter rather than its variable name by selecting model configuration parameter **Use value for tunable parameters**.



When select this parameter, the value of the variable (at code generation time) is displayed in the edit field.



Macro Parameters

Suppose that you apply a storage class such as `Define` to a `Simulink.Parameter` object so that the parameter appears as a macro in the generated code. If you use the parameter object inside a subsystem from which you generate an ERT S-function, you cannot select the parameter object as a tunable parameter. Instead, the S-function code generator applies the custom storage class to the parameter object. This generation of macros in the S-function code enables you to generate S-functions from subsystems that contain variant elements, such as Variant Subsystem blocks, that you configure to produce preprocessor conditionals in the generated code. You cannot change the value of the parameter during simulation of the S-function.

To select the parameter object as a tunable parameter, apply a different storage class or create your own storage class. Storage classes that treat parameters as macros include `Define`, `ImportedDefine`, `CompilerFlag`, and storage classes that you create by setting **Data initialization** to `Macro` in the Custom Storage Class Designer. If you use a non-macro storage class, you cannot use the parameter object as a variant control variable and generate preprocessor conditionals.

If you apply a storage class that treats the parameter object as an imported macro, before you generate the ERT S-function, provide the macro definition. For example, suppose that you apply the storage class `ImportedDefine` to a `Simulink.Parameter` object and use the parameter object as a variant control variable in the subsystem. If you set the custom attribute `HeaderFile` to `'myHdr.h'`, when you generate the S-function, place the custom header file `myHdr.h` in the current folder. The generated S-function uses the macro value from your header file instead of the value from the `Value` property of the parameter object.

To use a macro that you define through a compiler option, for example by applying the storage class `CompilerFlag`, use the model configuration parameter **Code Generation > Custom Code > Code information > Defines** to specify the compiler option. For more information, see Code Generation Pane: Custom Code: Additional Build Information: Defines.

Checksums and the S-Function Target

The code generator creates a checksum for a model and uses the checksum during the build process for code reuse, model reference, and external mode features.

The code generator calculates a model checksum by

- 1 Calculating a checksum for each subsystem in the model. A subsystem's checksum is the combination of properties (data type, complexity, sample time, port dimensions, and so forth) of the subsystem blocks.
- 2 Combining the subsystem checksums and other model-level information.

An S-function can add additional information, not captured during the block property analysis, to a checksum by calling the function `ssSetChecksumVal`. For the S-Function target, the value that gets added to the checksum is the checksum of the model or subsystem from which the S-function is generated.

The code generator applies the subsystem and model checksums as follows:

- Code reuse — If two subsystems in a model have the same checksum, the code generator produces code for one function only.

- Model reference — If the current model checksum matches the checksum when the model was built, the build process does not rebuild referenced models.
- External mode — If the current model checksum does not match the checksum of the code that is running on the target hardware, the build process generates an error.

Generated S-Function Compatibility

When you build a MEX S-function from your model, the code generator builds a level 2 noninlined S-function. Cross-release usage limitations on the generated code and binary MEX file (for example, *.mexw64) include:

- S-function target generated code from previous MATLAB release software is not compatible with newer releases. Do not recompile the generated code from a previous release with newer MATLAB release software. Use the same MATLAB release software to generate code for the S-function target and compile the code into a MEX file.
- You can use binary S-function MEX files generated from previous MATLAB release software with the same or newer releases with the same compatibility considerations as handwritten S-functions. For more information, see “S-Function Compatibility”.
- The code generator can generate code and build an executable from a model that contains generated S-functions. This support requires that the S-functions are built with the same MATLAB release software that builds the model. It is not possible to incorporate a generated S-function MEX file from previous MATLAB release software into a model and build the model with newer releases.

S-Function Target Limitations

- “Inport and Outport Blocks with Continuous Sample Time” on page 29-45
- “Tunable Variables in Expressions” on page 29-45
- “Parameter Tuning” on page 29-45
- “Run-Time Parameters and S-Function Compatibility Diagnostics” on page 29-45
- “Goto and From Blocks” on page 29-45
- “Building and Updating S-Functions” on page 29-46
- “Unsupported Blocks” on page 29-47
- “Model Operating Point not Supported for Code Generation” on page 29-47
- “Nesting S-Functions” on page 29-47
- “User-Defined Data Types” on page 29-47
- “Right-Click Generation of an S-Function Target” on page 29-48
- “Bus Input and Output” on page 29-48
- “Subsystems with Function-Call Input and Output Signals” on page 29-48
- “Function-Call Subsystem Modeling Pattern” on page 29-48
- “Data Store Access” on page 29-49
- “Inport or Outport Block Parameters Through Subsystem Mask” on page 29-49
- “MEX S-Function Wrappers” on page 29-49

Inport and Outport Blocks with Continuous Sample Time

The right-click build method for generating an S-function does not preserve continuous sample time for Inport and Outport blocks. Because the sample time is not preserved, simulation of the generated S-Function block can be affected.

Tunable Variables in Expressions

Certain limitations apply to the use of tunable variables in expressions. When the code generator encounters an unsupported expression while producing code, a warning appears and the equivalent numeric value is generated in the code. For a list of the limitations, see “Tunable Expression Limitations” on page 15-122.

Parameter Tuning

The S-Function block does not support tuning of tunable parameters with:

- Complex values.
- Values or data types that are transformed to a constant (by setting the model configuration parameter **Optimization > Default parameter behavior** to Inlined).
- Data types that are not built-in.
- Floating-point data types that are not equivalent to a built-in type.

Run-Time Parameters and S-Function Compatibility Diagnostics

If you set model configuration parameter **S-function upgrades needed** to warning or error, the code generator instructs you to upgrade S-functions generated from subsystems. The S-function system target file does not register run-time parameters. Run-time parameters are only supported for inlined S-functions, and the generated S-function supports features that prevent it from being inlined, for example, it can call or contain other noninlined S-functions.

To work around this limitation, set parameter **S-function upgrades needed** to none.

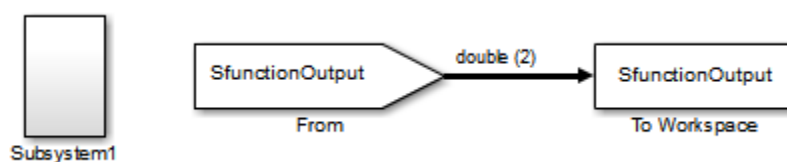
Goto and From Blocks

When using the S-function system target file, the code generator restricts input and output to correspond to the root model Inport and Outport blocks (or the Inport and Outport blocks of the Subsystem block from which the S-function target was generated). The code generator does not produce code for Goto or From blocks.

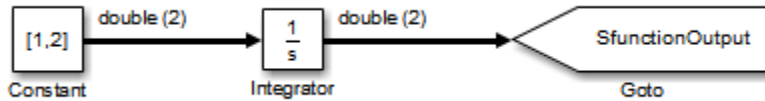
To work around this restriction, create your model and subsystem with the required Inport and Outport blocks, instead of using Goto and From blocks to pass data between the root model and subsystem. In the model that incorporates the generated S-function, you would then add Goto and From blocks.

Example Before Work Around

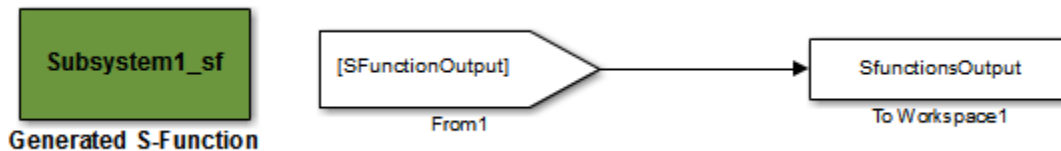
- Root model that includes a From block and subsystem, Subsystem1



- Subsystem1 that includes a Goto block that has global visibility and passes its input to the From block in the root model

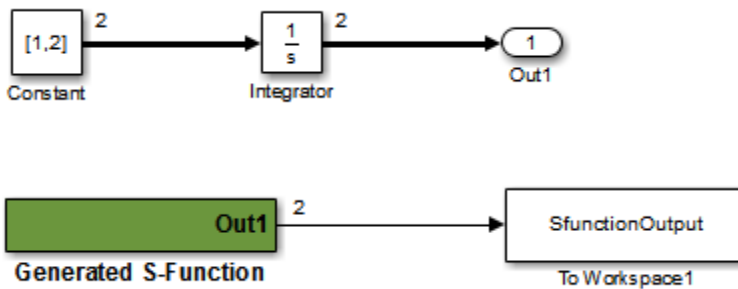


- Subsystem1 replaced with an S-function generated with the S-Function system target file — a warning results when you run the model because the generated S-function does not implement the Goto block



Example After Work Around

An Outport block replaces the GoTo block in Subsystem1. When you plug the generated S-function into the root model, its output connects directly to the To Workspace block.



Building and Updating S-Functions

The following limitations apply to building and updating S-functions using the S-function system target file:

- You cannot build models that contain Model blocks using the S-function system target file. This also means that you cannot build a subsystem by using the right-click context menu if the subsystem contains Model blocks. This restriction applies only to S-functions generated using the S-function target, not to ERT S-functions.
- You can build a model with:
 - A toolchain only if the toolchain supports MEX-file generation.
 - A template makefile only if the template makefile is associated with a toolchain on page 31-26 that supports MEX-file generation.
- If you modify the model that generated an S-Function block, the build process does not automatically rebuild models containing the generated S-Function block. This is in contrast to the

practice of automatically rebuilding models referenced by Model blocks when they are modified (depending on the Model Reference **Rebuild** configuration setting).

- Handwritten S-functions without corresponding TLC files must contain exception-free code. For more information on exception-free code, see “Exception Free Code”.

Unsupported Blocks

The S-function format does not support the following built-in blocks:

- Interpreted MATLAB Function block
- S-Function blocks containing any of the following:
 - MATLAB language S-functions (unless you supply a TLC file for C code generation)
 - Fortran S-functions (unless you supply a TLC file for C code generation)
 - C/C++ MEX S-functions that call into the MATLAB environment
- Simulink Function block
- Function Caller block
- Scope block
- To Workspace block

The S-function format does not support blocks from the `embeddedtargetslib` block library.

Model Operating Point not Supported for Code Generation

You can write C-MEX and Level-2 MATLAB S-functions that leverage the model operating point, which is used to save and restore the simulation state, as described in “S-Function Compliance with the ModelOperatingPoint”. The model operating point is not supported for code generation, including with the S-function system target file.

When you generate code for an S-function that specifies default operating point compliance or no operating point support, the generated code does not include the model operating point functionality. You cannot generate code for an S-function that implements custom operating point functionality.

Nesting S-Functions

The following limitations apply to nesting a generated S-Function block in a model or subsystem from which you generate another S-function:

- The software does not support nonvirtual bus input and output for a nested S-function.
- Avoid nesting an S-function in a model or subsystem having the same name as the S-function (possibly several levels apart). In such situations, the S-function can be called recursively. The software currently does not detect such loops in S-function dependency, which can result in aborting or hanging your MATLAB session. To prevent this from happening, name the subsystem or model to be generated as an S-function target uniquely to avoid duplicating existing MEX filenames on the MATLAB path.

User-Defined Data Types

The S-function system target file does not support the `HeaderFile` property that can be specified on user-defined data types, including those based on `Simulink.AliasType`, `Simulink.Bus`, and `Simulink.NumericType` objects. If a user-defined data type in your model uses the `HeaderFile`

property to specify an associated header file, code generation with the S-function system target file disregards the value and does not generate a corresponding include statement.

Right-Click Generation of an S-Function Target

If you generate an S-function target by right-clicking a Function-Call Subsystem block, the original subsystem and the generated S-function might not be consistent. An inconsistency occurs when the **States when enabling** parameter of the Trigger Port block inside the Function-Call Subsystem block is set to **inherit**. You must set the **States when enabling** parameter to **reset** or **held**, otherwise Simulink reports an error.

Bus Input and Output

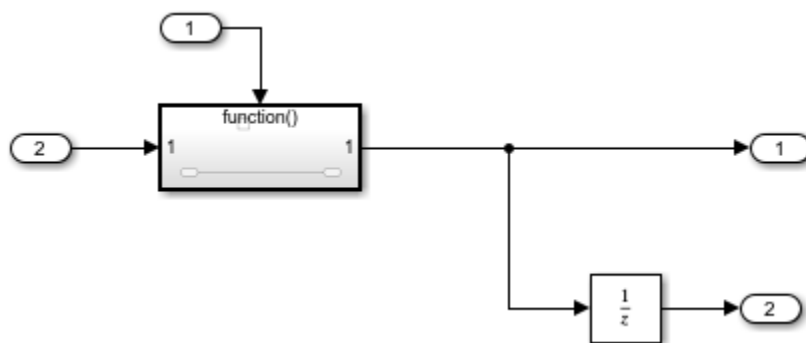
If an S-function generated using the S-function target has bus input or output, the generated bus data structures might include padding to align fields of the bus elements with the Simulink representation used during simulation. However, if you insert the S-function in a model and generate code using a model target such as `gRT.tlc`, the bus structure alignment generated for the model build might be incompatible with the padding generated for the S-function and might impact the numerical results of code execution. To make the structure alignment consistent between model simulation and execution of the model code, for each `Simulink.Bus` object, you can modify the `HeaderFile` property to remove the unpadding bus structure header file. This will cause the bus typedefs generated for the S-function to be reused in the model code.

Subsystems with Function-Call Input and Output Signals

The S-function target does not support creating an S-Function block from a subsystem that has a function-call trigger input or a function-call output.

Function-Call Subsystem Modeling Pattern

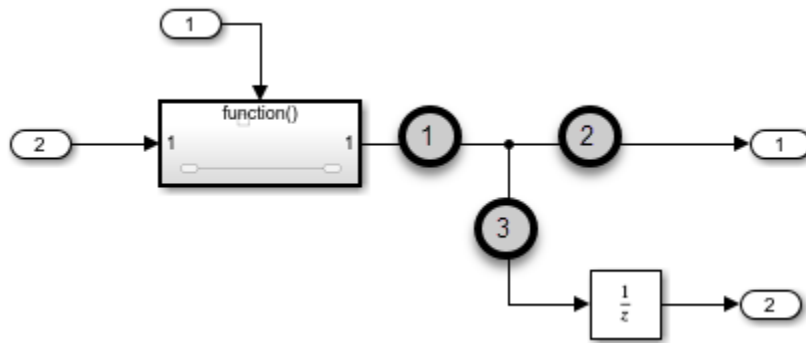
When you add a generated S-Function block to a model and then simulate that model, MATLAB might crash if the generated S-function uses this function-call subsystem modeling pattern.



The function-call subsystem connects directly to an Output block. The signal line that connects the subsystem to the Output block branches to a block that has an `Update` function, that is, a block that updates at each major time step, such as Unit Delay and Memory. The branch can be inside or outside of the subsystem.

In the model that includes the function-call subsystem, a Signal Conversion block in one of these locations can work around the problem with the generated S-function:

- 1 Between the subsystem block and the branch
- 2 Between the branch and the root Output block
- 3 Between the branch and the block that has the Update function



The Signal Conversion block in the workaround has these parameter settings:

- **Output** set to `Signal copy`
- **Exclude this block from 'Block reduction' optimization** selected

Data Store Access

When an S-Function block in your model accesses a data store during simulation, Simulink disables data store diagnostics.

- If you created the S-Function block from a model, the diagnostic is disabled for global data stores as well.
- If you created the S-Function block from a subsystem, the diagnostic is disabled for the following data stores:
 - Global data stores
 - Data stores placed outside the subsystem, but accessed by Data Store Read or Data Store Write blocks.

Inport or Output Block Parameters Through Subsystem Mask

S-functions generated from a subsystem with Inport or Output block parameters specified by subsystem mask variables produce an error when you try to run a simulation that uses the S-Function block.

```
Invalid setting in 'testSystem/Subsystem/__OutputSSForSFun__/Out2'
for parameter 'PortDimensions'
...
```

MEX S-Function Wrappers

Use a MEX S-function wrapper only in the MATLAB version in which the wrapper is created.

See Also

More About

- “Protect Models to Conceal Contents” on page 12-2
- “Accelerate, Refine, and Test Hybrid Dynamic System on Host Computer by Using RSim System Target File” on page 29-2
- “C/C++ S-Function Basics”
- “S-Functions and Code Generation” on page 14-2
- “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6
- “Use a Bus with S-Function Builder to Create an S-Function”

External Code Integration in Simulink Coder

The code generator includes multiple approaches for integrating legacy or custom code with generated code. Legacy code is existing handwritten code or code for environments that you integrate with code produced by the code generator. Custom code is legacy code or other user-specified lines of code that you include in the code generator build process. Collectively, legacy and custom code are called external code.

You integrate external code by importing existing external code into code produced by the code generator, exporting generated code into an existing external code base, or you can do both. For example, you can import code by calling an external function, by using the Legacy Code Tool, or place external code at specific locations in generated code by including Custom Code blocks in a model. When you import external code, the resulting generated code interfaces with generated scheduling code.

You can export generated code as a plug-in function for use in an external development environment. When you export generated code, you intend to interface that code manually with a scheduling mechanism in your application run-time environment.

For guidance on choosing an approach based on your application, see “Choose an External Code Integration Workflow” on page 30-2 .

test

- “Choose an External Code Integration Workflow” on page 30-2
- “Configure Target Hardware Characteristics” on page 30-9
- “Check Code Generation Assumptions” on page 30-12
- “Call Reusable External Algorithm Code for Simulation and Code Generation” on page 30-15
- “Place External C/C++ Code in Generated Code” on page 30-23
- “Call External Device Drivers” on page 30-31
- “Apply Function and Operator Code Replacements” on page 30-32
- “Build Integrated Code Within the Simulink Environment” on page 30-33
- “Generate Component Source Code for Export to External Code Base” on page 30-40
- “Generate Shared Library for Export to External Code Base” on page 30-54
- “Exchange Data Between External C/C++ Code and Simulink Model or Generated Code” on page 30-59
- “Exchange Data Between External Calling Code and Generated Code” on page 30-67
- “Generate Code That Matches Appearance of External Code” on page 30-70

Choose an External Code Integration Workflow

In this section...

“Choose a Software Execution Framework for Scheduling Code Execution” on page 30-3

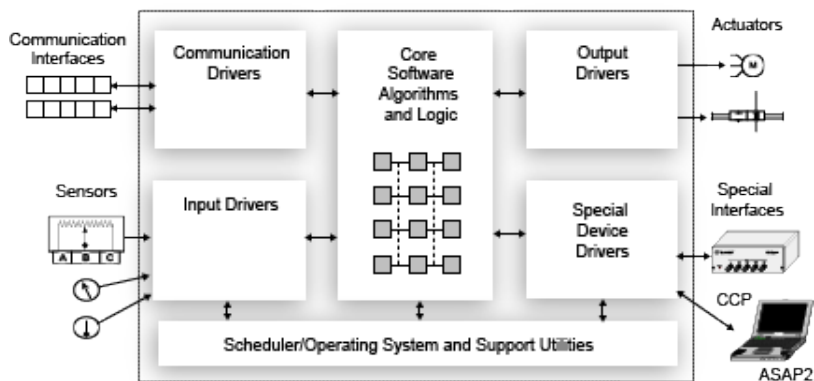
“Evaluate Characteristics of External Code” on page 30-4

“Identify Integration Requirements” on page 30-6

“Choose a Workflow” on page 30-7

Software projects typically involve combining code from multiple sources. A typical system structure for a code generation application consists of a framework that combines code from multiple sources, including external code and code generated from Simulink models.

This figure shows an application that requires integration of existing driver code for hardware devices. The core software algorithms and logic can be a combination of code modules for external reusable algorithms integrated into the Simulink environment and code generated as part of an overall model design.



Completing these tasks helps you choose external code integration workflows and tooling that align with your project.

Task	Action	More Information
1	Partition your application, map algorithms to components, and identify integration points.	“Divide Software Problem” on page 1-2, “Map System Architecture to Simulink Modeling Environment” on page 1-3
2	Determine whether you can rely on scheduling code that the code generator produces, or whether you must integrate generated code with scheduling mechanisms that are specific to your run-time environment.	“Choose a Software Execution Framework for Scheduling Code Execution” on page 30-3
3	Evaluate the characteristics of the external code that you are importing.	“Evaluate Characteristics of External Code” on page 30-4

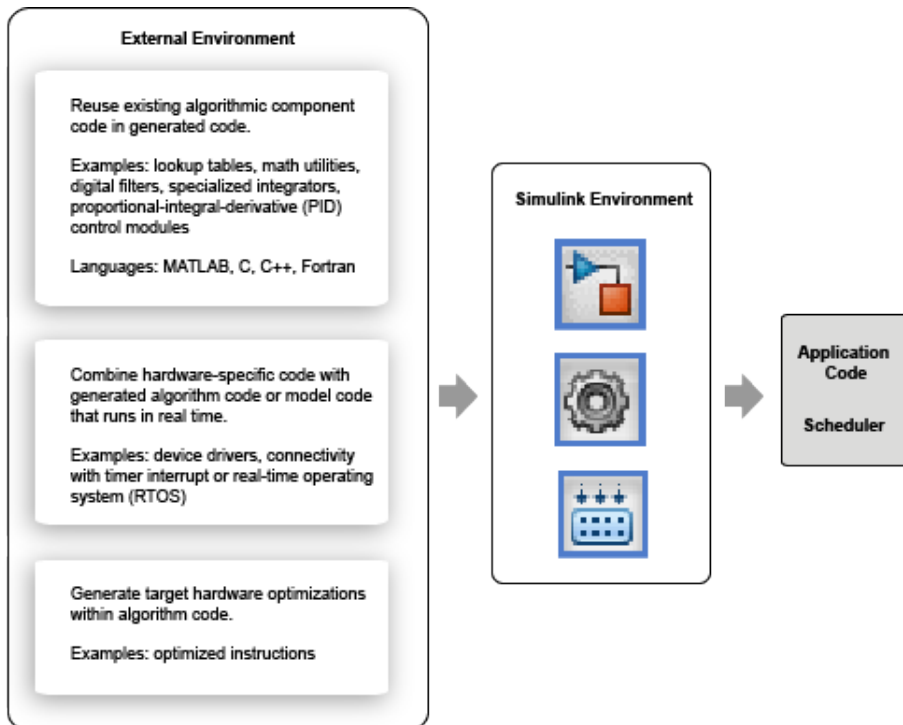
Task	Action	More Information
4	Identify integration requirements, which assists with choosing optimal tooling for your integration.	“Identify Integration Requirements” on page 30-6
5	Based on the results of tasks 1-4, choose a workflow.	“Choose a Workflow” on page 30-7

Choose a Software Execution Framework for Scheduling Code Execution

The code generator supports two types of software execution frameworks—single top model and multiple top-level. The first question to answer concerns which of the two frameworks meets the scheduling and other needs of your project. For example, you can import external code into a single, rate-based top model. You can export code from a single top model or multiple top-level models for integration with custom (external) scheduling mechanisms.

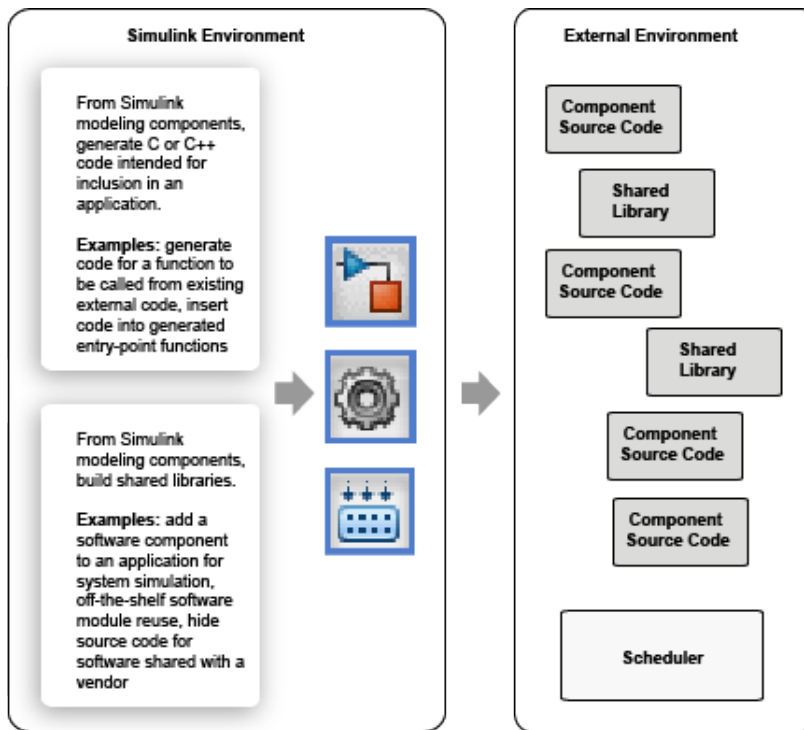
- Single top model

Generate one set of application code files from external code and code that the Simulink C/C++ code generator produces. The generated code includes a scheduler. In this case, you *import* code into the Simulink code generation environment.



- Single top model or multiple top-level models

Integrate C or C++ code that the code generator produces from model components with external application code and an external scheduler. You *export* generated code from the Simulink code generation environment.



Importing calls to external device driver code into a model and generating code for that model for export involves importing and exporting code.

Based on goals and requirements, external code integration is characterized in several ways, requiring different workflows and integration tooling:

- Import existing external code into generated code.
 - Call reusable external algorithm code for simulation and code generation.
 - Place external C/C++ code in generated code.
 - Call external device drivers.
 - Apply function and operator code replacements.
 - Interface with external timer interrupt or scheduler.
 - Generate replacement code for specific run-time environment.
- Export generated code for inclusion in external code base. Requires Embedded Coder .
 - Generate component source code for export.
 - Generate shared library for export.

For more information on exporting generated code, see “Callable Function Integration” (Embedded Coder).

Next, see “Evaluate Characteristics of External Code” (Embedded Coder).

Evaluate Characteristics of External Code

Before choosing an external integration workflow, evaluate these characteristics of the external code. To interface with external code, generated C or C++ code handles one or more of the external code

characteristics. An understanding of these characteristics and your requirements for modeling, simulation, and code generation helps you choose the optimal workflow for your integration scenario. (See “Identify Integration Requirements” (Embedded Coder).)

Characteristic	What to Consider
Hardware dependency	<p>Is the external code hardware-dependent? Utility functions, lookup tables, and filters are examples of hardware-independent code.</p> <p>Device drivers interact directly with hardware. They depend on characteristics of the hardware. For example, a device driver for an analog-to-digital converter initializes, reads data from, and writes data to hardware registers. Hardware differences and dependencies concern data type size, endianness, shift operations, compiler directives, and optimized function and operator support. Other code interfaces with device drivers by using an API and data mapped to specific memory addresses. Typically, simulation on a development computer is not possible. Reading from and writing to a register during simulation on a development computer produces unexpected and unwanted results.</p>
Reusable	<p>Is the external code a reusable software module? Examples include utility functions, lookup tables, filters, specialized integrators, and proportional-integral-derivative (PID) control modules.</p>
Dependency on data persistence between function calls	<p>Does the external code require persistent data? For example, a call to a first order filter function uses the output of the previous call to the function to calculate a new output value. You have the option of defining the data as global or using shared memory outside the context of the function.</p>
Data typing and interface	<p>How complex is the data that the external code uses? What does the data interface look like? It consists of arguments, a return value, global variables, and access functions. What data types does the code use? Are the types limited to basic ANSI C integers, floating-point types, arrays of integers or floating-point types, and pointers to these types? Does the interface include structures or pointers to structures?</p>
Fixed-point code	<p>Is the external code designed to run on integer-only processors? If yes, the code exchanges and uses data represented as integers only. Data can be associated with fixed-point scaling or offsets.</p>
External resource dependencies	<p>Does the external code use data, functions, or macros defined outside the scope of the code? For example, the function can use a standard ANSI function, a shared library, or predefined constants. In these cases, you must inform the compiler and linker of the paths and file names of the external resources.</p>
External solver required	<p>Are you using the external function for advanced development or rapid prototyping to describe a system with a continuous transfer function or a set of differential equations? If yes, the external code relies on an external solver.</p>

Next, see “Identify Integration Requirements” (Embedded Coder).

Identify Integration Requirements

Before choosing an external integration workflow, review these integration requirements. An understanding of these requirements and the characteristics of your external code helps you choose the optimal workflow for your integration scenario. (See “Evaluate Characteristics of External Code” (Embedded Coder).)

Requirement	What to Consider
Effort	What level of effort is planned for the integration project—low, medium, or high?
Learning effort	What is the programming experience of assigned project resources? How much experience do assigned resources have with Simulink and MathWorks C/C++ code generation products?
Simulation and code generation behaviors	Do you want to take advantage of Model-Based Design? To take full advantage of Model-Based Design, convert code to modeling elements, which you can then use in the Simulink and Stateflow simulation environment. Then, simulate and generate code for the integrated component. Use software-in-the-loop (SIL) or processor-in-the-loop (PIL) testing to verify whether algorithm behavior is the same in both environments.
Data interface and typing	<ul style="list-style-type: none"> • Does your model or generated code need to exchange data with the external function? If so, map inputs, outputs, and parameters to the external function interface. Typical function interfaces involve function arguments and return values, global variables, and access functions, such as <code>getRPM</code>. • Do you want to represent arrays, structures, or enumerated types? In the Simulink environment, you can represent these types as vectors, buses, and <code>IntEnum</code>, respectively. • Is fixed-point support required? If you use the Simulink fixed-point interface, you can scale and specify offsets. • Does the external code use company-specific data types? If yes and you have Embedded Coder software, create alias types to represent those external types. The code generator uses the alias types in the code that it produces. For example, once defined, you can specify an alias type in a function prototype, for a temporary variable, or for block output. • Does the code exchange data with shared memory? If yes, define and use memory sections.
Direct function call	Do you want to call C external code directly from a model? You can choose from mechanisms, such as the C Function block, the C Caller block, Legacy Code Tool, Stateflow external code interface and chart action language, and the MATLAB Function block.
Insertion of external code into generated code	Do you want to control the placement of external code within generated code? Do you want to insert code into generated entry-point functions? You can place code within generated code by using model configuration parameters or custom code blocks.

Requirement	What to Consider
Code generation optimization support	Do you want to optimize the code that the code generator produces? If so, you can configure the model for the code generator to optimize the code it produces based on application objectives, such as execution, ROM, and RAM efficiency. You also have the option of using code replacement libraries.
Files required	Do you want to minimize the number of files that you maintain? Some external code integration tools require that you maintain separate files for defining simulation and code generation.

Next, see “Choose a Workflow” (Embedded Coder).

Choose a Workflow

To choose a workflow for integrating external code into the Simulink environment, use the following table with links to more information.

Goal	Action	More Information
Call hardware-independent reusable external algorithmic code from generated code. For example: utility functions, lookup tables, and digital filters.	Choose an approach for integrating external code into the Simulink environment for code generation based on your programming language and other integration requirements.	“Call Reusable External Algorithm Code for Simulation and Code Generation” on page 30-15
Call hardware-dependent external reusable external algorithmic code from generated code. For example: device drivers.	Call external driver code from the Simulink environment.	“Call External Device Drivers” on page 30-31
Place external C/C++ code in generated code.	Augment functions with external code using model configuration parameters or custom code blocks.	“Place External C/C++ Code in Generated Code” on page 30-23
Change the code that the code generator produces for functions and operators to meet application code requirements.	Use code replacements to specify application-specific implementations of functions and operators.	“Apply Function and Operator Code Replacements” on page 30-32

See Also

More About

- “Call Reusable External Algorithm Code for Simulation and Code Generation” on page 30-15
- “Place External C/C++ Code in Generated Code” on page 30-23
- “Call External Device Drivers” on page 30-31
- “Deploy Generated Component Software to Application Target Platforms” (Embedded Coder)

- “Code Replacement”
- “Generate Component Source Code for Export to External Code Base” (Embedded Coder)
- “Generate Shared Library for Export to External Code Base” (Embedded Coder)

Configure Target Hardware Characteristics

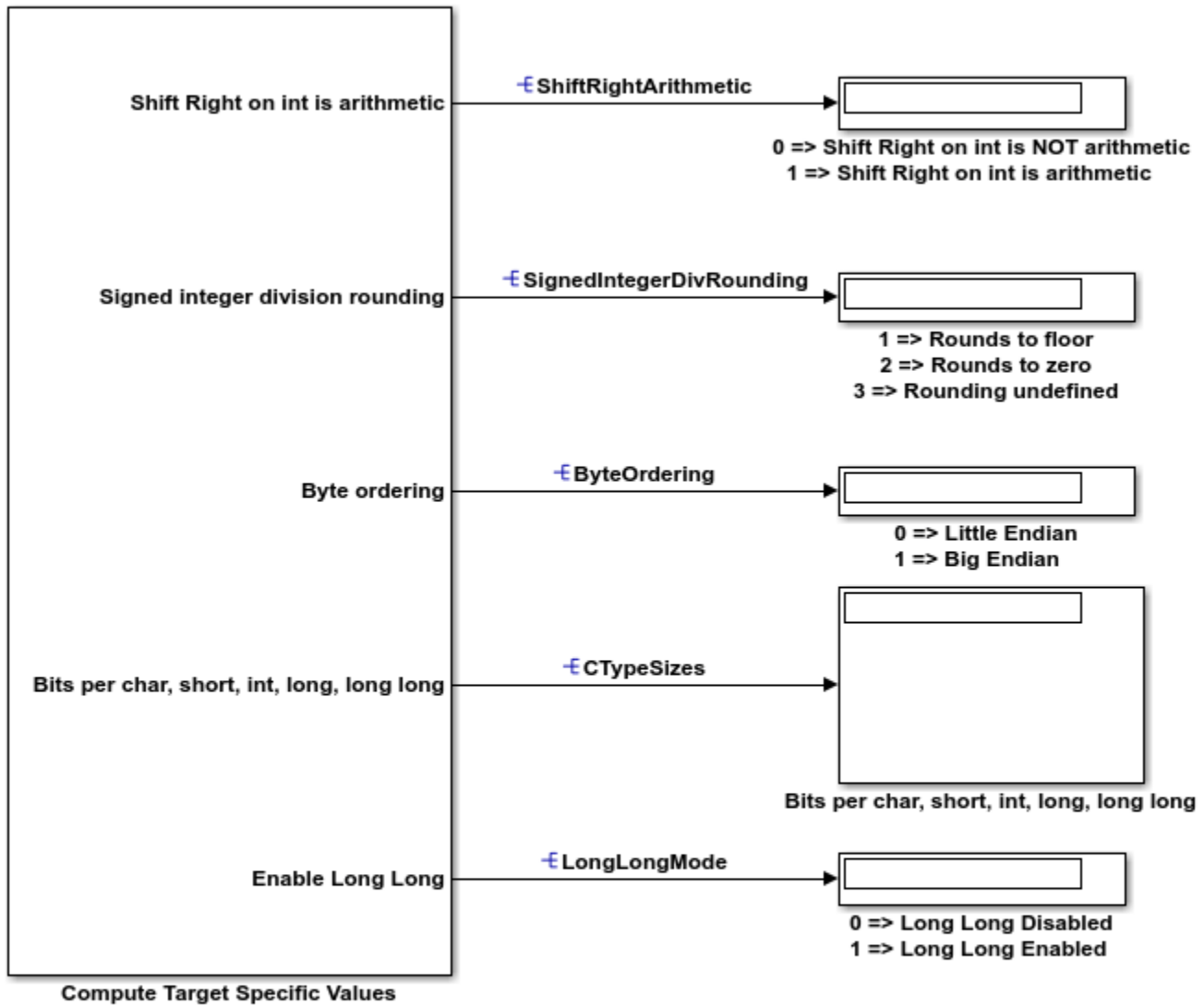
Simulink® allows you to configure the hardware implementation characteristics of your target system. Failure to do so can result in code that is incorrect or inefficient.

Open Example Model

Open the example model `rtwdemo_targetsettings`.

```
open_system('rtwdemo_targetsettings');
```

?



View Hardware Configuration (double-click)

Generate Code Using Simulink Coder (double-click)

Generate Code Using Embedded Coder (double-click)

My microprocessor is not listed ...

Open the **Hardware Implementation** pane of the Configuration Parameters dialog box by double-clicking the yellow button in the model.

Hardware Board Configuration

By default, the model is configured for the code generator to determine the hardware board configuration based on the specified system target file. The default system target file setting is `grt.tlc`.

If you have installed support for a hardware board, that board should appear in the **Hardware board** menu. If you select it, the dialog box displays parameters that are relevant to your hardware board.

My Hardware Board Is Not Listed

If you are using a hardware board that is not listed, add it to the menu by installing the corresponding target support package. Start the Support Package Installer by setting **Hardware board** to `Get hardware Support Packages`, or by entering `supportPackageInstaller` in the MATLAB® Command Window.

After installing support for a hardware board, reopen the Configuration Parameters dialog box and select the hardware board.

Cross-Development Systems That Include Source Level Debuggers

If you use a cross-development system that includes a source level debugger:

1. Configure the model for your hardware board.
2. Generate code and build an executable using your target development system. For details about retrieving file and path information, see the documentation on the Simulink Coder® Build Information API.
3. Download the executable to the target hardware and set breakpoints at the model output code for each output of the model.
4. Run a debugging session and observe the behavior of the code as it executes on the target hardware. As you step through the code, it computes the values for target hardware device characteristics.

If your target development system does not support source level debugging, generate code and add `printf` statements or other code that gets the results.

5. On the **Hardware Implementation** pane of the Configuration Parameters dialog box, set **Device vendor** to `Custom processor`, click the **Device details** arrow, and adjust the microprocessor device settings manually.

Check Code Generation Assumptions

The code generator implements assumptions that depend, for example, on the hardware implementation settings for your model. It is important to check that the assumptions are valid for your target hardware. Use the `buildStandaloneCoderAssumptions` function to create an application that performs assumption checks on your target hardware.

When you configure your model, try to specify hardware implementation and build configuration settings that match your target hardware. Then, to check code generator assumptions for your target hardware, use this workflow:

- 1 With `GenerateReport` set to 'on', build (**Ctrl+B**) your model.
- 2 To view the list of code generator assumptions that you can check, open the code generation report and click the **Coder Assumptions** link.

For more information, see “Coder Assumptions List” on page 30-13.

- 3 Run `buildStandaloneCoderAssumptions`, which uses the generated code in the build folder to create an application that runs code generation assumption checks.
- 4 Download the application onto the target hardware or target environment, and then run the application.
- 5 While the application runs, use a debug tool to view check results that are in a data structure.

Check Code Generator Assumptions for Development Computer

If the target hardware is, for example, your Windows development computer, you can use Microsoft Visual Studio® to run and debug the application:

- 1 To open a model, in the Command Window, enter:

```
openExample('ecoder/SILPILVerificationExample', ...
    supportingFile='SILTopModel.slx')
```

- 2 In the Model Configuration Parameters dialog box, in the **Hardware Implementation** pane, specify settings to match the target hardware. For example, for a 64-bit Windows computer specify these parameter settings:

- **Device vendor** -- Intel
- **Device type** -- x86-64 (Windows 64)

- 3 Clear model configuration parameter **Enable portable word sizes**.
- 4 Build the model (**Ctrl+B**).
- 5 To view the list of code generator assumptions, open the code generation report and click the **Coder Assumptions** link.
- 6 From the Command Window, in the working folder, run:

```
buildStandaloneCoderAssumptions('SILTopModel_ert_rtw')
```

In the build folder, the function creates the `coderassumptions\standalone` subfolder, which contains the target application, `SILTopModel_ca`.

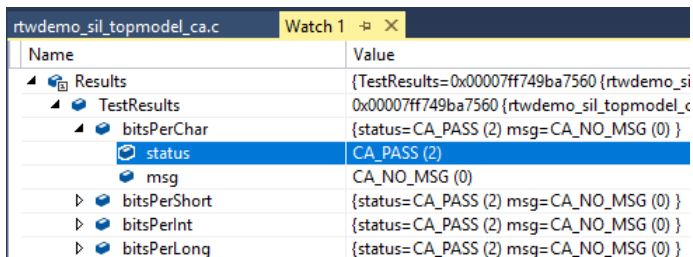
- 7 Open Microsoft Visual Studio and select **File > Open > Project/Solution**

- 8 Using the Open Project dialog box, navigate to the `coderasumptions\standalone` subfolder and select `SILTopModel_ca`. Then, click **Open**.
- 9 Select **File > Open > File**. Using the Open File dialog box, select `coderasumptions\standalone\SILTopModel_ca.c`
- 10 At the return statement, insert a break point.
- 11 Select **Debug > Start Debugging**.
- 12 To verify code generator assumptions, use a Watch window to inspect the Results data structure:
 - a In the `SILTopModel_ca.c` code, right-click `Results`.
 - b From the context menu, select **Add Watch**.

The variables in the data structure contain:

- Check outcomes (`TestResults`)
- Target hardware truths (`ActualValues`)
- Code generator assumptions (`ExpectedValues`)

For example, if the code generator assumption for `bitsPerChar` is accurate, you see the `CA_PASS` value in the `status` variable.



Coder Assumptions List

The contents section of the code generation report has a link to the Coder Assumptions page. The page provides a list of:

- Code generation assumptions that you can check
- Expected results for the assumption checks

This table describes the labels that you see in the list.

Category	Label	Assumption You Can Check
C/C++ Language Configuration for Target Hardware or Development Computer	BitPerChar	Number of bits per char (ProdBitPerChar)
	BitPerShort	Number of bits per short (ProdBitPerShort)
	BitPerInt	Number of bits per int (ProdBitPerInt)

Category	Label	Assumption You Can Check
	BitPerLong	Number of bits per long (ProdBitPerLong)
	BitPerFloat	Size of float (Only if PurelyIntegerCode is 'off')
	BitPerDouble	Size of double (Only if PurelyIntegerCode is 'off')
	BitPerPointer	Number of bits per pointer (ProdBitPerPointer)
	BitPerSizeT	Number of bits per size_t (ProdBitPerSizeT)
	BitPerPtrDiffT	Number of bits per ptrdiff_t (ProdBitPerPtrDiffT)
	Endianess	Byte ordering (ProdEndianess)
	Shift right for signed integer is arithmetic shift	Sign bit behavior (ProdShiftRightIntArith)
	Signed integer division rounds to	Rounding behavior for integer division (ProdIntDivRoundTo)
C/C++ Language Standard	Initial value of a global integer variable and dynamically allocated memory is zero	Zero initialization of memory (ZeroExternalMemoryAtStartup, ZeroInternalMemoryAtStartup, and GenerateAllocFcn)
Floating-Point Numbers	Flush-to-zero computed subnormal values (FTZ)	Flush-to-zero subnormal number processing
	Flush-to-zero incoming subnormal values (DAZ)	Denormals-are-zero subnormal number processing

See Also

buildStandaloneCoderAssumptions

More About

- “Verification of Code Generation Assumptions” (Embedded Coder)

Call Reusable External Algorithm Code for Simulation and Code Generation

Code reuse offers business and technological advantages. From a business perspective, code reuse saves time and resources. From a technological perspective, code reuse promotes consistency and reduces memory requirements. Other considerations include:

- Modularizing an application
- Reusing an optimized algorithm
- Interfacing with a predefined dataset
- Developing application variants

Examples of reusable hardware-independent algorithmic code to consider importing into the Simulink environment for simulation and code generation include:

- Utility functions
- Lookup tables
- Digital filters
- Specialized integrators
- Proportional-integral-derivative (PID) control modules

Workflow

To call reusable external algorithm code for simulation and code generation, iterate through the tasks listed in this table.

Task	Action	More Information
1	Review your assessment of external code characteristics and integration requirements.	"Choose an External Code Integration Workflow" on page 30-2
2	Based on the programming language of the external code, choose an integration approach to add the external code to a Simulink model.	"Choose an Integration Approach" on page 30-16
3	Verify algorithm behavior and performance by simulating the model.	"Simulation"
4	Define the representation of model data for code generation.	"Code Interface Configuration" and "Exchange Data Between External C/C++ Code and Simulink Model or Generated Code" (Embedded Coder)
5	Configure the model for code generation.	"Generate Code That Matches Appearance of External Code" (Embedded Coder)
6	Generate code and a code generation report.	"Code Generation"
7	Review the generated code interface and static code metrics.	"Analyze Generated Data Code Interface" on page 25-23

Task	Action	More Information
8	Build an executable program from the model.	“Build Integrated Code Within the Simulink Environment” on page 30-33
9	Verify that executable program behaves as expected.	“Verification and Testing”

Choose an Integration Approach

Several approaches are available for integrating reusable algorithmic code into the Simulink environment for code generation. Some approaches integrate external code directly. Other approaches convert the external code to Simulink or Stateflow modeling elements for simulation, and later for code generation from the modeled design. The integration approach that you choose depends on:

- Programming language of the external code — MATLAB, C, C++, or Fortran
- Your programming language experience and preference
- Performance requirements
- Whether the algorithm must model continuous time dynamics or you are integrating the algorithm into an application that uses discrete and continuous time
- Whether you want to take advantage of Model-Based Design
- Level of control required over the code that the code generator produces

To choose an approach for a reusable algorithm, see the subsection that matches the programming language of your external algorithm code.

- “Integration Approaches for External MATLAB Code” on page 30-16
- “Integration Approaches for External C or C++ Code” on page 30-18
- “Integration Approaches for External Fortran Code” on page 30-21

Integration Approaches for External MATLAB Code

Multiple approaches are available for integrating external MATLAB code into the Simulink environment. The following table helps you choose the best integration approach for your application based on integration requirements.

Condition or Requirement	Action	More Information
<ul style="list-style-type: none"> • External code complies with the MATLAB code generation subset • You want to call MATLAB code from a Simulink model 	Add a MATLAB Function block to the model. Embed the MATLAB code in that block.	<ul style="list-style-type: none"> • “Integrate C Code by Using the MATLAB Function Block” • MATLAB Function

Condition or Requirement	Action	More Information
<ul style="list-style-type: none"> External code complies with the MATLAB code generation subset You want to call MATLAB code from a Simulink model Your algorithm includes iterative computations that process large streams of data 	<p>Add a MATLAB System block to the model. Embed the MATLAB code in that block as a System object™.</p>	<ul style="list-style-type: none"> “Integrate C Code by Using the MATLAB Function Block” MATLAB System
<ul style="list-style-type: none"> External code complies with the MATLAB code generation subset You want to call MATLAB code from a Simulink model Your algorithm includes design logic that is based on state machines and flow charts 	<p>Add a Stateflow chart to the model. Call the external code from the chart, using MATLAB as the action language.</p>	<ul style="list-style-type: none"> “Chart Programming Basics” (Stateflow) “Insert External Code into Stateflow Charts” on page 30-21
<p>You want to use the <code>parfor</code> function for parallel computing or interface data types that are available to MATLAB Coder, Simulink Coder, and Embedded Coder. To use <code>parfor</code>, Parallel Computing Toolbox must be installed.</p>	<p>Use software to generate C code. Then, call that generated code as external C code.</p>	<ul style="list-style-type: none"> “Generate C Code by Using the MATLAB Coder App” “Get Started with MATLAB Coder”
<p>You have C or C++ programming experience and the external MATLAB code is compact and primarily uses C or C++ constructs.</p>	<p>Manually convert the MATLAB code to C or C++ code. Choose an integration approach for C or C++ code.</p>	<p>“Integration Approaches for External C or C++ Code” on page 30-18</p>
<p>Sections of the external MATLAB code map to built-in blocks.</p>	<p>Develop the algorithm in the context of a model, using the applicable built-in blocks.</p>	<ul style="list-style-type: none"> “Interactive Model Editing” and “Project Management” “Use Products and Blocks Supported for Code Generation” on page 3-2

Condition or Requirement	Action	More Information
The algorithm must model continuous state dynamics.	Write a MATLAB S-function and, for generating code, a corresponding TLC file for the algorithm. Add the S-function to your model.	<ul style="list-style-type: none"> • “Create and Configure MATLAB S-Functions” • “Code Generation for Custom Blocks” • “Target Language Compiler” • “Target Language Compiler” (Embedded Coder)

To embed external MATLAB code in a MATLAB Function block or generate C or C++ code from MATLAB code with the MATLAB Coder software, the MATLAB code must use functions and classes supported for C/C++ code generation.

- “Functions and Objects Supported for C/C++ Code Generation”

Integration Approaches for External C or C++ Code

Multiple approaches are available for integrating external C or C++ code into the Simulink environment. The following table helps you choose the best integration approach for your application based on integration requirements.

Condition or Requirement	Action	More Information
You want to integrate external C code with generated C++ code or conversely	Match the language choice for the generated code by modifying the language of the external code.	“Modify Programming Language of External Code to Match Generated Code” on page 30-20
<ul style="list-style-type: none"> • You have existing C code that models custom functionality. You want to integrate the code into Simulink blocks. • Your functions do not read or write global/static variables. • You want to easily integrate with Simulink Coverage, Simulink Test, and Simulink Design Verifier™ 	For simple algorithms written in C, consider using the Simulink C Caller block.	“Integrate C Code Using C Caller Blocks”

Condition or Requirement	Action	More Information
<ul style="list-style-type: none"> You have existing C code that models custom functionality. You want to call multiple C functions from a Simulink block, or call C functions conditionally. You want to include C code to preprocess and postprocess data before and after calling C functions. You want to specify different code for simulation and code generation. 	<p>Add a C Function block to your model.</p>	<p>“Integrate C Code by Using the MATLAB Function Block”</p>
<ul style="list-style-type: none"> The algorithm must model discrete and continuous state dynamics for simulation and rapid prototyping. The external code requires a fixed-point interface. Programming experience is limited. You want ease of use with basic flexibility for controlling what code the code generator produces for rapid prototyping. 	<p>Generate S-function and TLC files by using the S-Function Builder. If necessary, refine the generated code manually to meet application requirements. (If you change the generated code, you lose the changes if you regenerate the S-function and TLC files.)</p>	<ul style="list-style-type: none"> “Use a Bus with S-Function Builder to Create an S-Function”
<ul style="list-style-type: none"> You want to simulate and generate external code for a discrete time application. Optimizing generated code is essential. You want ease of use with moderate flexibility for controlling what code the code generator produces. You have C or C++ programming experience, but you prefer to generate the files for adding the code to a model. 	<p>Generate S-function and TLC files by using the Legacy Code Tool. If necessary, refine the generated code manually to meet application requirements. (If you change the generated code, you lose the changes if you regenerate the S-function and TLC files.)</p>	<ul style="list-style-type: none"> “Integrate C Functions Using Legacy Code Tool” “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6 “Integrate C Code Using C Caller Blocks”

Condition or Requirement	Action	More Information
<ul style="list-style-type: none"> The application requires more entry-point functions than the code generator typically produces—for example, more than <i>model_step</i>, <i>model_initialize</i>, and <i>model_terminate</i>. <p>You want maximum flexibility for controlling what code the code generator produces.</p>	Manually write an S-function and TLC file.	<ul style="list-style-type: none"> “C/C++ S-Function Basics” “S-Functions and Code Generation” on page 14-2 “C S-Function Examples” and “C++ S-Function Examples”
Your algorithm includes design logic that is based on state machines and flow charts. Or, a function that you want to integrate must exchange data with a model by using global variables. The function defines the global variables and uses them to write output rather than returning a value (return) or writing output to an argument.	Add a Stateflow chart to the model. Call the external code from the chart, using C as the action language. In the chart, write code that calls the external function and reads from and writes to the global variables. To perform calculations with output of the external code, the model must read from the global variable during execution.	“Insert External Code into Stateflow Charts” on page 30-21
You want to include external C or C++ code in a Stateflow chart for simulation and code generation.	Configure the model that contains the chart to apply the external C or C++ code.	<ul style="list-style-type: none"> “Custom Code” (Stateflow) “Call C Library Functions in C Charts” (Stateflow) “Insert External Code into Stateflow Charts” on page 30-21
You quickly want to embed a call to external C or C++ code in a model. Performance is not an issue.	Call the C or C++ code with the <code>coder.ceval</code> function from within a MATLAB Function block.	<ul style="list-style-type: none"> <code>coder.ceval</code> function “Integrate C Code by Using the MATLAB Function Block” MATLAB Function block

Modify Programming Language of External Code to Match Generated Code

To integrate external C code with generated C++ code or conversely, modify the language of the external code to match the programming language choice for the generated code. Options for making the programming language match include:

- Writing or rewriting the external code in the language choice for the generated code.
- If you are generating C++ code and the external code is C code, for each C function, create a header file that prototypes the function. Use this format:

```
#ifdef __cplusplus
extern "C" {
#endif
int my_c_function_wrapper();
```

```
#ifdef __cplusplus
}
#endif
```

The prototype serves as a function wrapper. If your compiler supports C++ code, the value `__cplusplus` is defined. The linkage specification `extern "C"` specifies C linkage without name mangling.

- If you are generating C code and the external code is C++ code, include an `extern "C"` linkage specification in each `.cpp` file. For example, the following example shows C++ code in the file `my_func.cpp`:

```
extern "C" {

int my_cpp_function()
{
    ...
}
}
```

Integration Approaches for External Fortran Code

To integrate external Fortran code:

- 1 Write an S-function and corresponding TLC file.
- 2 Add the S-function to your model.
- 3 If necessary, add support files and control model code generation and builds within the Simulink environment.

See “C/C++ S-Function Basics”, “Implement Algorithms Using Fortran Code”, “S-Functions and Code Generation” on page 14-2, and “Fortran S-Function Examples”.

Insert External Code into Stateflow Charts

- “Integrate External Code for Library Charts” on page 30-21
- “Integrate External Code for All Charts” on page 30-22

Integrate External Code for Library Charts

To integrate external code that applies only to Stateflow library charts for code generation, for each library model that contributes a chart to your main model, complete these steps. Then, generate code.

- 1 In the Stateflow Editor, open the Model Configuration Parameters dialog box. Select parameter **Use local custom code settings (do not inherit from main model)**,

The library model retains its own custom code settings during code generation.

- 2 Specify your custom code in the subpanes.

Follow the guidelines in “Specify Relative Paths to Your Custom Code” (Stateflow).

If you specified custom code settings for simulation, you can apply these settings to code generation. To avoid entering the same information twice, select **Use the same custom code settings as Simulation Target**.

- 3 Click **OK**.

After completing these steps for each library model, generate code.

Integrate External Code for All Charts

To integrate external code that applies to all charts for code generation:

- 1** Specify custom code options for code generation of your main model.
 - a** In the Model Configuration Parameters dialog box, select **Code Generation > Custom Code**.
 - b** In the custom code text fields, specify your custom code.

Follow the guidelines in “Specify Relative Paths to Your Custom Code” (Stateflow).

If you specified custom code settings for simulation, you can apply these settings to code generation. To avoid entering the same information twice, select **Use the same custom code settings as Simulation Target**.

- 2** Configure code generation for each library model that contributes a chart to your main model. In the Stateflow Editor, open the Model Configuration Parameters dialog box. Clear parameter **Use local custom code settings (do not inherit from main model)**. The library charts inherit the custom code settings of your main model. Click **OK**.
- 3** Generate code.

See Also

More About

- “Integrate C Functions Using Legacy Code Tool”
- “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6
- “Custom Code” (Stateflow)
- “Integrate C Code by Using the MATLAB Function Block”
- “C/C++ S-Function Basics”
- “S-Functions and Code Generation” on page 14-2

Place External C/C++ Code in Generated Code

In this section...
“Workflow” on page 30-23
“Choose an Integration Approach” on page 30-24
“Integrate External Code by Using Custom Code Blocks” on page 30-25
“Integrate External Code by Using Model Configuration Parameters” on page 30-27
“Integrate External C Code into Generated Code by Using Custom Code Blocks and Model Configuration Parameters” on page 30-28

You can customize code that the code generator produces for a model by specifying external code with custom code blocks or model configuration parameters.

- Place code at the start and end of the generated code for the root model.
- Place declaration, body, and exit code in generated function code for blocks in the root model or nonvirtual subsystems.

The functions that you can augment with external code depends on the functions that the code generator produces for blocks that are in the model. For example, if a model or atomic subsystem includes blocks that have states, you can specify code for a disable function. Likewise, if you need the code for a block to save data, free memory, or reset target hardware, specify code for a terminate function. For more information, see “Block Target File Methods”.

Workflow

To place external C or C++ code at specific locations in code that the code generator produces for root models and subsystems, iterate through the tasks listed in this table.

Task	Action	More Information
1	If you want to integrate external C code with generated C++ code or conversely, modify the language of the external code to match the language choice for the generated code.	“Modify Programming Language of External Code to Match Generated Code” on page 30-20
2	Review your assessment of external code characteristics and integration requirements.	“Choose an External Code Integration Workflow” on page 30-2
3	If necessary, rewrite code in C or C++.	
4	Choose an integration approach to add the external code to a Simulink model.	“Choose an Integration Approach” on page 30-24
5	Define the representation of model data for code generation.	“Exchange Data Between External C/C++ Code and Simulink Model or Generated Code” on page 30-59
6	Configure the model for code generation.	“Generate Code That Matches Appearance of External Code” on page 30-70 “Model Configuration Set Customization”

Task	Action	More Information
7	Generate code and a code generation report.	“Code Generation”
8	Review the generated code interface.	“Analyze Generated Data Code Interface” on page 25-23
9	Build an executable program from the model.	“Build Integrated Code Within the Simulink Environment” on page 30-33
10	Verify that executable program performs as expected.	“Verification and Testing”

Choose an Integration Approach

Within the Simulink modeling environment, two approaches are available for placing external C or C++ code into sections of code that the code generator produces:

- Add blocks from the Custom Code library to a root model or atomic subsystem.
- Set model configuration parameters on the **Code Generation > Custom Code** pane.

The following table compares the two approaches. Choose the approach that aligns best with your integration requirements. For more information about how to apply each approach, see “Integrate External Code by Using Custom Code Blocks” (Embedded Coder) and “Integrate External Code by Using Model Configuration Parameters” (Embedded Coder).

Requirement	Blocks	Model Configuration Parameters
Include a representation of your external code in the modeling canvas	✓	
Place code in functions generated for root models	✓	✓
Place code in functions generated for atomic subsystems	✓	
Save code placement in a model configuration set		✓
Place code at the top and bottom of the header and source files generated for a model	✓	✓
Place code within declaration, execution, and exit sections of the <code>SystemInitialize</code> and <code>SystemTerminate</code> functions that the code generator creates	✓	✓
Place code within declaration, execution, and exit sections of the <code>SystemStart</code> , <code>SystemEnable</code> , <code>SystemDisable</code> , <code>SystemOutputs</code> , <code>SystemUpdate</code> , or <code>SystemDerivatives</code> functions that the code generator creates	✓	
Add preprocessor macro definitions to generated code		✓
Use the custom code settings that are specified for the simulation target		✓

Requirement	Blocks	Model Configuration Parameters
Configure a library model to use custom code settings of the parent model to which the library is linked		✓

Integrate External Code by Using Custom Code Blocks

- “Custom Code Block Library” on page 30-25
- “Add Custom Code Blocks to the Modeling Canvas” on page 30-26
- “Add External Code to Generated Start Function” on page 30-26

Custom Code Block Library

The Custom Code block library contains blocks that you can use to place external C or C++ code into specific locations and functions within code that the code generator produces. The library consists of 10 blocks that add your code to the model header (*model.h*) and source (*model.c* or *model.cpp*) files that the code generator produces.

The Model Header and Model Source blocks add external code at the top and bottom of header and source files that the code generator produces for a root model. These blocks display two text fields into which you can type or paste code. One field specifies code that you want to place at the top of the generated header or source file. The second field specifies code that you want to place at the bottom of the file.

The remaining blocks add external code to functions that the code generator produces for the root model or atomic subsystem that contains the block. The blocks display text fields into which you can type or paste code that customizes functions that the code generator produces. The text fields correspond to the declaration, execution, and exit sections of code for a given function.

To Customize Code That	Use This Block
Computes continuous states	System Derivatives
Disables state	System Disable
Enables state	System Enable
Resets state	System Initialize
Produces output	System Outputs
Executes once	System Start
Saves data, free memory, reset target hardware	System Terminate
Requires updates at each major time step	System Update

The block and its location within a model determines where the code generator places the external code. For example, if the System Outputs block is at the root model level, the code generator places the code in the model `Outputs` function. If the block resides in a triggered or enabled subsystem, the code generator places the code in the subsystem `Outputs` function.

If the code generator does not need to generate a function that corresponds to a Custom Code block that you include in a model, the code generator does one of the following:

- Omits the external code that you specify in the Custom Code block.
- Returns an error, indicating that the model does not include a relevant block. In this case, remove the Custom Code block from the model.

For more information, see “Block Target File Methods”.

Note To avoid a potential mismatch between simulation and code generation results, do not use custom code blocks to read from and write to global Simulink data (signals, states, and block parameters). Instead, use the proper modeling pattern (for example, Data Store Read, Data Store Write, State Reader, and State Writer blocks.)

Add Custom Code Blocks to the Modeling Canvas

To add the Custom Code library blocks to a model:

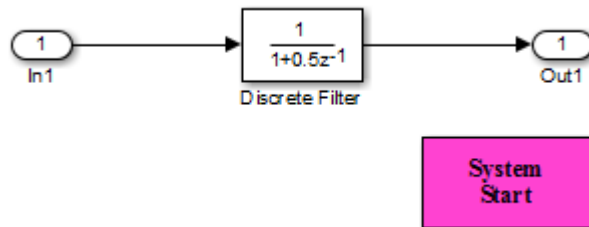
- 1 In the Simulink Library Browser, expand the Simulink Coder node then select the Custom Code block library.
- 2 Drag the blocks that you want into your model or subsystem. Drag Model Header and Model Source blocks into root models only. Drag function-based Custom Code blocks into root models or atomic subsystems.

You can use models that contain Custom Code blocks as referenced models. The code generator ignores the blocks when producing code for a simulation target. When producing code for a code generation target, the code generator includes and compiles the custom code.

Add External Code to Generated Start Function

This example shows how to use the System Start block to place external C code in the code that the code generator produces for a model that includes a discrete filter.

- 1 Create the following model.



- 2 Configure the model for code generation.
- 3 Double-click the System Start block.
- 4 In the block parameters dialog box, in the **System Start Function Declaration Code** field, enter this code:

```
unsigned int *ptr = 0xFFEE;
```

- 5 In the **System Start Function Execution Code** field, enter this code:

```
/* Initialize hardware */
*ptr = 0;
```

- 6 Click **OK**.
- 7 Generate code and a code generation report.
- 8 View the generated `model.c` file. Search for the string `start` function. You should find the following code, which includes the external code that you entered in steps 4 and 5.

```

{
  {
    /* user code (Start function Header) */
    /* System '<Root>' */
    unsigned int *ptr = 0xFFEE;

    /* user code (Start function Body) */
    /* System '<Root>' */
    /* Initialize hardware */
    *ptr = 0;
  }
}

```

For another example, see “Integrate External C Code into Generated Code by Using Custom Code Blocks and Model Configuration Parameters” on page 30-28.

Integrate External Code by Using Model Configuration Parameters

Model configuration parameters provide a way to place external C or C++ code into specific locations and functions within code that the code generator produces.

To	Select
Insert external code near the top of the generated <code>model.c</code> or <code>model.cpp</code> file	Additional code , and enter the external code to insert. If you generate subsystem code into separate files, that code does not have access to external code that you specify with the Additional code parameter. For example, if you specify an include file as a Additional code setting, the code generator inserts the <code>#include</code> near the top of the <code>model.c</code> or <code>model.cpp</code> file. The subsystem code that the code generator places in a separate file does not have access to declarations inside your included file. In this case, consider specifying your external code with the Include headers parameter.
Insert external code near the top of the generated <code>model.h</code> file	Include headers , and enter the external code to insert.
Insert external code inside the model initialize function in the <code>model.c</code> or <code>model.cpp</code> file	Initialize code , and enter the external code to insert.
Insert external code inside the model terminate function in the <code>model.c</code> or <code>model.cpp</code> file	Terminate code , and enter the external code to insert. Also select the Terminate function required parameter on the Interface pane.

To	Select
Add preprocessor macro definitions	Defines , and enter a space-separated list of preprocessor macro definitions to add to the generated code. The list can include simple definitions (for example, -DEF1) and definitions with a value (for example, -DDEF2=1). Definitions can omit the -D (for example, -DF00=1 and F00=1 are equivalent). If a definition includes -D, the toolchain can override the flag if the toolchain uses a different flag for defines.
Use the same custom code parameter settings as the settings specified for simulation of MATLAB Function blocks, Stateflow charts, and Truth Table blocks	Use the same custom code settings as Simulation Target This parameter refers to the Simulation Target pane in the Configuration Parameters dialog box.
Enable a library model to use custom code settings unique from the parent model to which the library is linked	Use local custom code settings (do not inherit from main model) This parameter is available only for library models that contain MATLAB Function blocks, Stateflow charts, or Truth Table blocks.

To include a header file in an external header file, add `#ifndef` code. Using this code avoids multiple inclusions. For example, in `rtwtypes.h`, the following `#include` guards are added:

```
#ifndef RTW_HEADER_rtwtypes_h_
#define RTW_HEADER_rtwtypes_h_
...
#endif /* RTW_HEADER_rtwtypes_h_ */
```

For more information on how to add files names and locations of header, source, and shared library files to the build process, see “Build Integrated Code Within the Simulink Environment” on page 30-33.

Note The code generator includes external code that you include in a configuration set when generating code for software-in-the-loop (SIL) and processor-in-the-loop (PIL) simulations. However, the code generator ignores external code that you include in a configuration set when producing code with the S-function, rapid simulation, or simulation system target file.

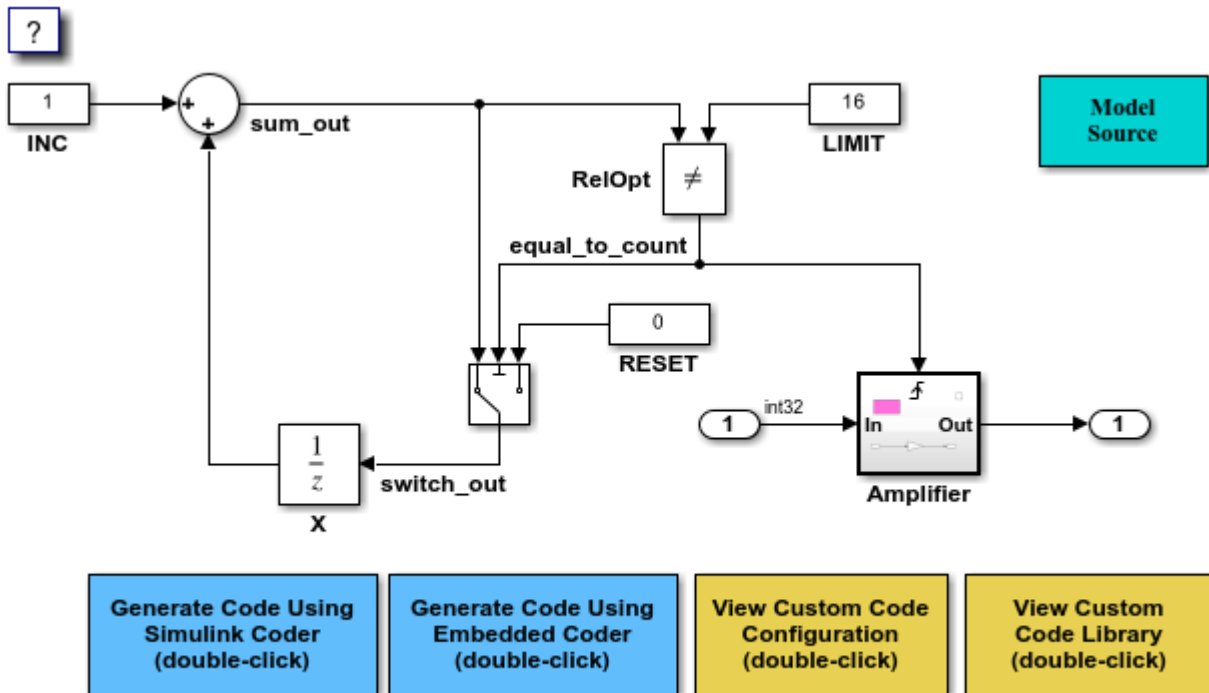
For more information about **Custom Code** parameters, see “Model Configuration Parameters: Code Generation Custom Code”. For an example, see “Integrate External C Code into Generated Code by Using Custom Code Blocks and Model Configuration Parameters” on page 30-28.

Integrate External C Code into Generated Code by Using Custom Code Blocks and Model Configuration Parameters

This example shows how to place external code in generated code by using custom code blocks and model configuration parameters.

1. Open the model `rtwdemo_slcustcode`.

```
open_system('rtwdemo_slcustcode')
```



Copyright 1994-2022 The MathWorks, Inc.

2. Open the **Simulink Coder** or **Embedded Coder** app.
3. Open the Model Configuration Parameters dialog box and navigate to the **Custom Code** pane.
4. On the **Additional source code** tab, examine the settings for model configuration parameters **Additional code** and **Initialize code**.
 - **Additional code** specifies a comment and sets the variable `GLOBAL_INT2` to -1.
 - **Initialize code** initializes the variable `GLOBAL_INT2` to 1.
5. Close the dialog box.
6. Double-click the Model Source block. Block parameter **Top of Model Source** specifies that the code generator declare the variable `GLOBAL_INT1` and set it to 0 at the top of the generated file `rtwdemo_slcustcode.c`.
7. Open the triggered subsystem **Amplifier**. The subsystem includes the System Outputs block. The code generator places code that you specify in that block in the generated code for the nearest parent atomic subsystem. In this case, the code generator places the external code in the generated code for the **Amplifier** subsystem. The external code:
 - Declares the pointer variable `*intPtr` and initializes it with the value of variable `GLOBAL_INT1`.
 - Sets the pointer variable to -1 during execution.
 - Resets the pointer variable to 0 before exiting.
8. Generate code and a code generation report.

9. Examine the code in the generated source file `rtwdemo_slcustcode.c`. At the top of the file, after the `#include` statements, you find the following declaration code. The example specifies the first declaration with the **Additional code** configuration parameter and the second declaration with the Model Source block.

```
int_T GLOBAL_INT2 = -1;
```

```
int_T GLOBAL_INT1 = 0;
```

The Output function for the `Amplifier` subsystem includes the following code, which shows the external code integrated with generated code that applies the gain. The example specifies the three lines of code for the pointer variable with the System Outputs block in the `Amplifier` subsystem.

```
int_T *intPtr = &GLOBAL_INT1;
```

```
*intPtr = -1;
```

```
rtwdemo_slcustcode_Y.Output = rtwdemo_slcustcode_U.Input << 1;
```

```
*intPtr = 0;
```

The following assignment appears in the model initialize entry-point function. The example specifies this assignment with the **Initialize code** configuration parameter.

```
GLOBAL_INT2 = 1;
```

See Also

More About

- “Configure Model and Generate Code” on page 24-11

Call External Device Drivers

Device drivers for protocols and target hardware are essential to many real-time development projects. For example, you can have a working device driver that you want to integrate with algorithmic code that has to read data from and write data to the I/O device that the driver supports. The code generator can produce a single set of application source files from an algorithm model and integrated driver code written in C or C++.

To call external device driver code from the Simulink environment, iterate through the tasks in this table.

Task	Action	More Information
1	Review your assessment of external code characteristics and integration requirements.	"Choose an External Code Integration Workflow" on page 30-2
2	Define the representation of model data for code generation.	"Exchange Data Between External C/C++ Code and Simulink Model or Generated Code" on page 30-59
3	Generate S-function and TLC files by using the Legacy Code Tool. If necessary, refine the generated code manually to meet application requirements.	<ul style="list-style-type: none"> "Integrate C Functions Using Legacy Code Tool" "Import Calls to External Code into Generated Code with Legacy Code Tool" on page 14-6
4	Verify algorithm behavior and performance by simulating the model.	"Simulation"
5	Configure the model for code generation.	"Generate Code That Matches Appearance of External Code" (Embedded Coder)
6	Generate code and a code generation report.	"Code Generation"
7	Review the generated code interface.	"Analyze Generated Data Code Interface" on page 25-23
8	Build an executable program from the model.	"Build Integrated Code Within the Simulink Environment" on page 30-33
9	Verify that executable program behaves and performs as expected.	"Verification and Testing"

See Also

More About

- "Integrate C Functions Using Legacy Code Tool"
- "Import Calls to External Code into Generated Code with Legacy Code Tool" on page 14-6
- "Custom Targets" on page 46-2

Apply Function and Operator Code Replacements

If your generated code must use functions and operators that are consistent with external code, configure the code generator to use a code replacement library (CRL). By default, the code generator does not apply a code replacement library. You can choose from several libraries that MathWorks provides, including GNU C99 extensions, AUTOSAR 4.0, and several Intel platform-specific IPP and IPP/SSE libraries. Depending on the products that you have, other libraries might be available. If you have Embedded Coder software, you can view and choose from additional libraries and you can create custom code replacement libraries.

See Also

More About

- “What Is Code Replacement?” on page 26-2
- “What Is Code Replacement Customization?” (Embedded Coder)
- **Code Replacement Viewer**
- **Code Replacement Tool**

Build Integrated Code Within the Simulink Environment

Workflow

To build executable programs that integrate generated code and external C or C++ code, iterate through the tasks in this table.

Task	Action	More Information
1	Choose a build approach.	<p>“Approaches for Building Code Generated from Simulink Models” (Embedded Coder)</p> <p>For an example, see “Build Process Workflow for Real-Time Systems” (Embedded Coder).</p>
2	Configure build process support for your external code.	“Configure Parameters for Integrated Code Build Process” (Embedded Coder)
3	Configure S-Function build support for your external code.	<p>“Build Support for S-Functions” (Embedded Coder)</p> <p>“Use makecfg to Customize Generated Makefiles for S-Functions” (Embedded Coder)</p> <p>For examples, see “Call External C Code from Model and Generated Code” (Embedded Coder) and “Call Reusable External Algorithm Code for Simulation and Code Generation” (Embedded Coder).</p>
4	Configure build process to find the external code source, library, and header files.	<p>“Manage Build Process File Dependencies” (Embedded Coder)</p> <p>“Control Library Location and Naming During Build” (Embedded Coder)</p>
5	Set up custom build processing required for your external code integration.	<p>For the build process customization workflow, see “Customize Post-Code-Generation Build Processing” (Embedded Coder).</p> <p>To automate applying build customizations to a toolchain approach build, see “Customize Build Process with sl_customization.m” (Embedded Coder).</p> <p>To automate applying build customizations to a template makefile approach build, see “Customize Build Process with STF_make_rtw_hook File” (Embedded Coder).</p>

Configure Parameters for Integrated Code Build Process

The table provides a guide to model configuration parameters on **Code Generation > Custom Code** pane of the Model Configuration Parameters dialog box that support the build process for external code integration. For information about folders for your external code, see “Manage Build Process

Folders” (Embedded Coder). If you choose to place your external code in the “Code generation folder”, see “Preserve External Code Files in Build Folder” (Embedded Coder).

To	Select
Add include folders, which contain header files, to the build process	<p>Code information > Include directories, and enter the absolute or relative paths to the folders.</p> <p>If you specify relative paths, the paths must be relative to the folder containing your model files, not relative to the build folder. The order in which you specify the folders is the order in which they are searched for header, source, and library files.</p>
Add source files to be compiled and linked	<p>Code information > Source files, and enter the full paths or just the file names for the files.</p> <p>Enter just the file name if the file is in the current MATLAB folder or in one of the include folders. For each additional source that you specify, the build process expands a generic rule in the template makefile for the folder in which the source file is located. For example, if a source file is located in folder <code>inc</code>, the build process adds a rule similar to the following:</p> <pre>%.obj: builddir\inc\%.c \$(CC) -c -Fo\$(@F) \$(CFLAGS) \$<</pre> <p>The build process adds the rules in the order that you list the source files.</p>
Add libraries to be linked	<p>Code information > Libraries, and enter the full paths or just the file names for the libraries.</p> <p>Enter just the file name if the library is located in the current MATLAB folder or in one of the include folders.</p>
Use the same custom code settings as those specified for simulation of MATLAB Function blocks, Stateflow charts, and Truth Table blocks	<p>Use the same custom code settings as Simulation Target</p> <p>This parameter refers to the Simulation Target pane in the Configuration Parameters dialog box.</p>
Enable a library model to use custom code settings unique from the parent model to which the library is linked	<p>Use local custom code settings (do not inherit from main model)</p> <p>This parameter is available only for library models that contain MATLAB Function blocks, Stateflow charts, or Truth Table blocks.</p>

Preserve External Code Files in Build Folder

By default, the build process deletes foreign source files. You can preserve foreign source files by following these guidelines.

If you put a `.c/.cpp` or `.h` source file in a build folder, and you want to prevent the code generator from deleting it during the TLC code generation process, insert the text `target specific file` in the first line of the `.c/.cpp` or `.h` file. For example:

```

/* COMPANY-NAME target specific file
 *
 * This file is created for use with the
 * COMPANY-NAME target.
 * It is used for ...
 */
...

```

Make sure that you spell the text “target specific file” as shown in the preceding example, and that the text is in the first line of the source file. Other text can appear before or after this text.

Flagging user files in this manner prevents postprocessing these files to indent them with generated source files. Auto-indenting occurred in previous releases to build folder files with names having the pattern *model_*.c/.cpp* (where * was text). The indenting is harmless, but can cause differences detected by source control software that can potentially trigger unnecessary updates.

Build Support for S-Functions

User-written S-Function blocks provide a powerful way to incorporate external code into the Simulink development environment. In most cases, you use S-functions to integrate existing external code with generated code. Several approaches to writing S-functions are available:

- “Write Noninlined S-Function” on page 14-91
- “Write Wrapper S-Function and TLC Files” on page 14-93
- “Write Fully Inlined S-Functions” on page 14-100
- “Write Fully Inlined S-Functions with mdlRTW Routine” on page 14-102
- “S-Functions for Code Reuse” on page 14-81
- “S-Functions for Multirate Multitasking Environments” on page 14-86

S-functions also provide the most flexible and capable way of including build information for legacy and custom code files in the build process.

There are different ways of adding S-functions to the build process.

Implicit Build Support

When building models with S-functions, the build process adds rules, include paths, and source file names to the generated makefile. The source files (*.h*, *.c*, and *.cpp*) for the S-function must be in the same folder as the S-function MEX-file. Whether using the toolchain approach or template makefile approach for builds, the build process propagates this information through the toolchain or template makefile.

- If the file *sfcname.h* exists in the same folder as the S-function MEX-file (for example, *sfcname.mexext*), the folder is added to the include path.
- If the file *sfcname.c* or *sfcname.cpp* exists in the same folder as the S-function MEX-file, the build process adds a makefile rule for compiling files from that folder.
- When an S-function is not inlined with a TLC file, the build process must compile the S-function source file. To determine the name of the source file to add to the list of files to compile, the build process searches for *sfcname.cpp* on the MATLAB path. If the source file is found, the build process adds the source file name to the makefile. If *sfcname.cpp* is not found on the path, the build process adds the file name *sfcname.c* to the makefile, whether or not it is on the MATLAB path.

Note For the Simulink engine to find the MEX-file for simulation and code generation, it must exist on the MATLAB path or exist in our current MATLAB working folder.

Specify Additional Source Files for an S-Function

If your S-function has additional source file dependencies, you must add the names of the additional modules to the build process. Specify the file names:

- In the **S-function modules** field in the S-Function block parameter dialog box
- With the `SFunctionModules` parameter in a call to the `set_param` function

For example, suppose you build your S-function with multiple modules.

```
mex sfun_main.c sfun_module1.c sfun_module2.c
```

You can then add the modules to the build process by doing one of the following:

- In the S-function block dialog box, specify `sfun_main`, `sfun_module1`, and `sfun_module2` in the **S-function modules** field.
- At the MATLAB command prompt, enter:

```
set_param(sfun_block, 'SFunctionModules', 'sfun_module1 sfun_module2')
```

Alternatively, you can define a variable to represent the parameter value.

```
modules = 'sfun_module1 sfun_module2'
set_param(sfun_block, 'SFunctionModules', modules)
```

The **S-function modules** field and `SFunctionModules` parameter do not support complete source file path specifications. To use the parameter, the code generator must find the additional source files when executing the makefile. For the code generator to locate the additional files, place them in the same folder as the S-function MEX-file. You can then leverage the implicit build support described in “Implicit Build Support” on page 30-35.

When you are ready to generate code, force the code generator to rebuild the top model, as described in “Control Regeneration of Top Model Code” on page 31-49.

For more complicated S-function file dependencies, such as specifying source files in other locations or specifying libraries or object files, use the `rtwmakecfg.m` API, as described in “Use `rtwmakecfg.m` API to Customize Generated Makefiles” on page 44-17.

Use TLC Library Functions

If you inline your S-function by writing a TLC file, you can add source file names to the build process by using the TLC library function `LibAddToModelSources`. For details, see “`LibAddSourceFileCustomSection(file, builtInSection, newSection)`”.

Note This function does not support complete source file path specifications. The function assumes that the code generator can find the additional source files when executing the makefile.

Another useful TLC library function is `LibAddToCommonIncludes`. Use this function in a `#include` statement to include S-function header files in the generated `model.h` header file. For details, see “`LibAddToCommonIncludes(incFileName)`”.

For more complicated S-function file dependencies, such as specifying source files in other locations or specifying libraries or object files, use the `rtwmakecfg.m` API, as described in “Use `rtwmakecfg.m` API to Customize Generated Makefiles” on page 44-17.

Precompile S-Function Libraries

You can precompile new or updated S-function libraries (MEX-files) for a model by using the MATLAB language function `rtw_precompile_libs`. Using a specified model and a library build specification, this function builds and places the libraries in a precompiled library folder.

By precompiling S-function libraries, you can optimize system builds. Once your precompiled libraries exist, the build process can omit library compilation from subsequent builds. For models that use numerous libraries, the time savings for build processing can be significant.

To use `rtw_precompile_libs`:

- 1 Set the library file suffix, including the file type extension, based on your system platform.

Consider determining the type of platform, and then use the `TargetLibSuffix` parameter to set the library suffix accordingly. For example, when applying a suffix for a GRT target, you can set the suffix to `_std.a` for a UNIX platform and `_vcx64.lib` for a Windows platform.

```
if isunix
    suffix = '_std.a';
else
    suffix = '_vcx64.lib';
end

set_param(my_model,'TargetLibSuffix', suffix);
```

There are a number of factors that influence the precompiled library suffix and extension. The following table provides examples for typical selections of system target file, the compiler toolchain, and other options that affect your choice of suffix and extension. For more information, examine the template make files in the `matlab/rtw/c/grt` folder or `matlab/rtw/c/ert` folder.

TMF File	COMPILER _TOOL_CHAIN Value	Precompiler Libraries (PRECOMP_LIBRARIES)			
		Library Suffix S-Function (EXPAND _LIBRARY _NAME Value)	Library Suffix Integer- Only Code (EXPAND _LIBRARY _NAME Value)	Library Suffix Optimize for Speed (EXPAND _LIBRARY _NAME Value)	Library Extension (EXPAND _LIBRARY _NAME Value)
<code>ert_lcc64.tmf</code>	<code>lcc</code>	<code>_rtwsfcn_lcc</code>	<code>_int_ert_lcc</code>	<code>_ert_lcc</code>	<code>.lib</code>
<code>ert_vcx64.tmf</code>	<code>vcx64</code>	<code>_rtwsfcn_vcx64</code>	<code>_int_ert_vcx64</code>	<code>_ert_vcx64</code>	<code>.lib</code>
<code>ert_unix.tmf</code>	<code>unix</code>	<code>_rtwsfcn</code>	<code>_int_ert</code>	<code>_ert</code>	<code>.a</code>
<code>grt_lcc64.tmf</code>	<code>lcc</code>	<code>n/a</code>	<code>n/a</code>	<code>_lcc</code>	<code>.lib</code>
<code>grt_vcx64.tmf</code>	<code>vcx64</code>	<code>n/a</code>	<code>n/a</code>	<code>_vcx64</code>	<code>.lib</code>
<code>grt_unix.tmf</code>	<code>unix</code>	<code>n/a</code>	<code>n/a</code>	<code>_std</code>	<code>.a</code>

- 2 Set the precompiled library folder.

Use one of the following methods to set the precompiled library folder:

- Set the `TargetPreCompLibLocation` parameter, as described in “Specify the Location of Precompiled Libraries” on page 44-4.

- Set the `makeInfo.precompile` field in an `rtwmakecfg.m` function file. (For more information, see “Use `rtwmakecfg.m` API to Customize Generated Makefiles” on page 44-17.)

If you set `TargetPreCompLibLocation` and `makeInfo.precompile`, the setting for `TargetPreCompLibLocation` takes precedence.

The following command sets the precompiled library folder for model `my_model` to folder `lib` under the current working folder.

```
set_param(my_model,'TargetPreCompLibLocation', fullfile(pwd,'lib'));
```

Note If you set both the target folder for the precompiled library files and a target library file suffix, the build process detects whether any precompiled library files are missing while processing builds.

3 Define a build specification.

Set up a structure that defines a build specification. The following table describes fields that you can define in the structure. These fields are optional, except for `rtwmakecfgDirs`.

Field	Description
<code>rtwmakecfgDirs</code>	<p>A cell array of character vectors that name the folders containing <code>rtwmakecfg</code> files for libraries to be precompiled. The function uses the <code>Name</code> and <code>Location</code> elements of <code>makeInfo.library</code>, as returned by <code>rtwmakecfg</code>, to specify the name and location of the precompiled libraries. If you set the <code>TargetPreCompLibLocation</code> parameter to specify the library folder, that setting overrides the <code>makeInfo.library.Location</code> setting.</p> <p>Note: The specified model must contain blocks that use precompiled libraries specified by the <code>rtwmakecfg</code> files because the TMF-to-makefile conversion generates the library rules only if the build process uses the libraries.</p>
<code>libSuffix</code>	<p>A character vector that specifies the suffix, including the file type extension, to be appended to the name of each library (for example, <code>.a</code> or <code>_vc.lib</code>). The character vector must include a period (<code>.</code>). You must set the suffix with either this field or the <code>TargetLibSuffix</code> parameter. If you specify a suffix with both mechanisms, the <code>TargetLibSuffix</code> setting overrides the setting of this field.</p>
<code>intOnlyBuild</code>	<p>A Boolean flag. When set to true, the flag indicates the libraries are to be optimized such that they are compiled from integer code only. This field applies to ERT targets only.</p>
<code>makeOpts</code>	<p>A character vector that specifies an option to be included in the <code>rtwMake</code> command line.</p>
<code>addLibs</code>	<p>A cell array of structures that specify libraries to be built that are not specified by an <code>rtwmakecfg</code> function. Each structure must be defined with two fields that are character arrays:</p> <ul style="list-style-type: none"> • <code>libName</code> — the name of the library without a suffix • <code>libLoc</code> — the location for the precompiled library <p>The target makefile (TMF) can specify other libraries and how those libraries are built. Use this field to precompile those libraries.</p>

The following commands set up build specification `build_spec`, which indicates that the files to be compiled are in folder `src` under the current working folder.

```
build_spec = [];  
build_spec.rtwmakecfgDirs = {fullfile(pwd,'src')};
```

4 Issue a call to `rtw_precompile_libs`.

The call must specify the model for which you want to build the precompiled libraries and the build specification. For example:

```
rtw_precompile_libs(my_model,build_spec);
```

See Also

More About

- “Call Reusable External Algorithm Code for Simulation and Code Generation” (Embedded Coder)
- “Place External C/C++ Code in Generated Code” (Embedded Coder)
- “Call External Device Drivers” (Embedded Coder)
- “Deploy Applications to Target Hardware” (Embedded Coder)
- “Deploy Generated Component Software to Application Target Platforms” (Embedded Coder)

Generate Component Source Code for Export to External Code Base

In this section...

“Modeling Options” on page 30-40

“Requirements” on page 30-40

“Limitations for Export-Function Subsystems” on page 30-42

“Workflow” on page 30-42

“Choose an Integration Approach” on page 30-43

“Generate C Function Code for Export-Function Model” on page 30-44

“Generate C++ Function and Class Code for Export-Function Model” on page 30-48

“Generate Code for Export-Function Subsystems” on page 30-52

If you have Embedded Coder software, you can generate function source code from modeling components to use in an external code base. The generated code does not include supporting scheduling code (for example, a step function). Controlling logic outside of the Simulink environment invokes the generated function code.

Modeling Options

You can generate function code to export for these modeling components:

- Export-function models (model containing functional blocks that consist exclusively of function-call subsystems, function-call model blocks, or other export-function models, as described in “Export-Function Models Overview”)
- Export-function subsystems (virtual subsystem that contains function-call subsystems)

To export code that the code generator produces for these modeling components, the modeling components must meet specific requirements (Embedded Coder).

For models designed in earlier releases, the code generator can export functions from triggered subsystems. The requirements stated for export-function subsystems also apply to exporting functions from triggered subsystems, with the following exceptions:

- Encapsulate triggered subsystems from which you intend to export functions in a top-level virtual subsystem.
- Triggered subsystems do not have to meet requirements and limitations identified for virtual subsystems that contain function-call subsystem.
- “Export Functions That Use Absolute or Elapsed Time” on page 30-41 does not apply to exporting functions from triggered subsystems.

Requirements

- Model solver must be a fixed-step discrete solver.
- You must configure each root-level Inport block that triggers a function-call subsystem to output a function-call trigger. These Inport blocks cannot connect to an Asynchronous Task Specification block.

- Model or subsystem, must contain only the following blocks at the root level:
 - Function-call blocks (such as Function-Call Subsystem, Simulink Function, S-Functions, and Function-Call Model blocks at the root level if the solver model configuration parameter **Tasking and sample time options > Periodic sample time constraint** is set to Ensure sample time independent)
 - Inport and Outport blocks (ports)
 - Constant blocks (including blocks that resolve to constants, such as Add)
 - Blocks with a sample time of Inf
 - Merge and data store memory blocks
 - Virtual connection blocks (such as, Function-Call Split, Mux, Demux, Bus Creator, Bus Selector, Signal Specification, and virtual subsystems that contain these blocks)
 - Signal-viewer blocks, such as Scope blocks (export-function subsystems only)
- When a constant block appears at the top level of the model or subsystem, you must set the model configuration parameter **Optimization > Default parameter behavior** for the model or containing model to **Inlined**.
- Blocks inside the model or subsystem must support code generation.
- Blocks that use absolute or elapsed time must be inside a periodic function-call subsystem with a discrete sample time specified on the corresponding function-call root-level Inport block. See “Export Functions That Use Absolute or Elapsed Time” on page 30-41.
- Data signals that cross the boundary of an exported system cannot be a virtual bus and cannot be implemented as a Goto-From connection. Data signals that cross the export boundary must be scalar, muxed, or a nonvirtual bus.

In addition, for export-function models, you cannot generate code for a rate-based model that includes multiple instances of an export-function model. For example, you cannot generate code for a test harness model that you use for scheduling reusable export-function models during simulation.

For export-function subsystems, the following additional requirements apply:

- A trigger signal that crosses the boundary of an export-function subsystem must be scalar. Input and output data signals that do not act as triggers do not have to be scalar.
- When a constant signal drives an output port of an export-function subsystem, the signal must specify a storage class.

Export Functions That Use Absolute or Elapsed Time

If you want to export function code for a modeling component with blocks that use absolute or elapsed time, those blocks must be inside a function-call subsystem that:

- You configure for periodic execution
- You configure the root-level Inport block with a discrete sample time

To configure a function-call subsystem for periodic execution:

- 1** In the function-call subsystem, right-click the Trigger block and choose **Block Parameters** from the context menu.
- 2** Set parameter **Sample time type** to **periodic**.
- 3** Set **Sample time** to the same granularity specified (directly or by inheritance) in the function-call initiator.

4 Click **OK** or **Apply**.

For more information, see “Absolute and Elapsed Time Computation” on page 4-2.

Limitations for Export-Function Subsystems

- Subsystem block parameters do not control the names of the files containing the generated code. The file names begin with the name of the exported subsystem.
- Subsystem block parameters do not control the names of top-level functions in the generated code. Each function name reflects the name of the signal that triggers the function or (for an unnamed signal) reflects the block from which the signal originates.
- You can export function-call systems for the C++ class code interface packaging only when its function specification is set to **Default step method**. See “Interactively Configure C++ Interface” (Embedded Coder). The exported function is compatible with single-threaded execution. To avoid potential data race conditions for shared signals, invoke all members for the class from the same execution thread.
- The code generator supports a SIL or PIL block in accelerator mode only if its function-call initiator is noninlined in accelerator mode. Examples of noninlined initiators include Stateflow charts.
- A Level-2 S-function initiator block, such as a Stateflow chart or the built-in Function-Call Generator block, must drive a SIL block.
- You can export an asynchronous (sample-time) function-call system, but the software does not support the SIL or PIL block for an asynchronous system.
- The use of the TLC function `LibIsFirstInit` has been removed for export-function subsystems.

Workflow

To generate code for an exported function, iterate through the tasks listed in this table.

Task	Action	More Information
1	Review your assessment of external code characteristics and integration requirements.	“Choose an External Code Integration Workflow” on page 30-2
2	Verify that the model or subsystem that you are exporting satisfies function exporting requirements.	“Requirements” on page 30-40
3	Address data interface requirements by modifying the model or subsystem.	“Exchange Data Between External C/C++ Code and Simulink Model or Generated Code” on page 30-59
4	If necessary, configure function prototype.	“Configure Entry-Point Function Interfaces for Simulink Function and Function Caller Blocks” on page 17-2 and, for fixed-step rate-based models, “Configure Generated C Function Interface for Model Entry-Point Functions” on page 16-79 or “Interactively Configure C++ Interface” (Embedded Coder)

Task	Action	More Information
5	If necessary , update the model to place external application-specific code in generated system functions.	“Place External C/C++ Code in Generated Code” on page 30-23
6	Verify that the functions behave and perform as expected during simulation by creating and using a test harness model. The test harness model schedules execution of the functions during simulation.	“Configure Model, Generate Code, and Simulate” on page 24-2 and, if you have Simulink Test software, “Test Authoring” (Simulink Test)
7	Configure the model or subsystem for code generation.	“Generate Code That Matches Appearance of External Code” on page 30-70 and “Model Configuration Set Customization”
8	Generate code and a code generation report.	“Code Generation”
9	Review the generated code interface and static code metrics.	“Analyze Generated Data Code Interface” on page 25-23 and “Static Code Metrics” (Embedded Coder)
10	Build an executable program that includes the exported function code.	“Build Integrated Code Outside the Simulink Environment” (Embedded Coder)
11	Verify that executable program behaves and performs as expected.	“Verification and Testing”

Choose an Integration Approach

Multiple approaches are available for generating function code for export to an external development environment. The following table compares approaches. Choose the approach that aligns best with your integration requirements. For more information on how to create export-function models, see “Export-Function Models Overview”. For more information on generating code for function call subsystems, see “Generate Component Source Code for Export to External Code Base” (Embedded Coder).

Condition or Requirement	Use	More Information
<ul style="list-style-type: none"> Traceability between modeling elements and generated code Local inputs (Inport block) and outputs (Outport block) 	Function-call subsystem	<ul style="list-style-type: none"> “Generate C Function Code for Export-Function Model” on page 30-44 “Generate C++ Function and Class Code for Export-Function Model” on page 30-48 “Using Function-Call Subsystems” Function-Call Subsystem

Condition or Requirement	Use	More Information
<ul style="list-style-type: none"> • Control over generated function prototype • Formal input arguments (Argument Inport blocks) and output arguments (Argument Outport blocks) • Local inputs (Inport block) and outputs (Outport block) 	Simulink Function block	<ul style="list-style-type: none"> • “Configure Entry-Point Function Interfaces for Simulink Function and Function Caller Blocks” on page 17-2 • “Simulink Function Blocks and Code Generation” on page 10-6 • “Generate Code for Simulink Function and Function Caller” on page 10-9 • “Simulink Functions” • Simulink Function
Code responds to an initialization event	Initialize Function block	<ul style="list-style-type: none"> • “Generate C Function Code for Export-Function Model” on page 30-44 • “Generate C++ Function and Class Code for Export-Function Model” on page 30-48 • “Startup, Reset, and Shutdown Function Interfaces” on page 2-8 • “Using Initialize, Reinitialize, Reset, and Terminate Functions”
Code responds to a reset event	Reset Function block	<ul style="list-style-type: none"> • “Generate C Function Code for Export-Function Model” on page 30-44 • “Generate C++ Function and Class Code for Export-Function Model” on page 30-48 • “Startup, Reset, and Shutdown Function Interfaces” on page 2-8 • “Using Initialize, Reinitialize, Reset, and Terminate Functions”
Code includes entry-point functions beyond what the code generator produces by default (<i>model_initialize</i> , <i>model_step</i> , and <i>model_terminate</i>)	S-function	“S-Functions and Code Generation” on page 14-2
Single-model execution framework to use as test harness and to export code generated for portions of a model	Export-function subsystem	<ul style="list-style-type: none"> • “Simulink Modeling Components” • “Using Function-Call Subsystems” • Function-Call Subsystem

Generate C Function Code for Export-Function Model

This example shows how to generate function code for individual function-call subsystems in a model without generating scheduling code.

To generate function code for export:

- 1 Create a model that contains the functions for export.
- 2 Configure the code interface.

- 3 Create a test harness model that schedules execution of the functions during simulation.
- 4 Simulate the model that contains the functions by using the test harness model.
- 5 Generate code for the model that contains the functions.

Create Model That Contains Functions for Export

The model with functions for export must satisfy architectural constraints at the model root level. At the root level, valid blocks are:

- Inport
- Outport
- Function-Call Subsystem
- Simulink Function
- Goto
- From
- Merge

For this example, the code generator produces function code for Initialize Function, Terminate Function, and Function-Call Subsystem blocks. An Initialize Function block is executed on a model initialize event and a Terminate Function block is executed on a terminate event. For a Function-call Subsystem block, you connect the block input ports to root Inport blocks that assert function-call signals. The subsystem is executed based on the function-call signal that it receives.

For exporting functions, model `ComponentDeploymentFcn` contains two function-call subsystems (`Integrator` and `Accumulator`). Function-call subsystem `Integrator` represents an aperiodic integrator function. The function applies a forward integration method and a gain value of 1.25 to the value of state variable `x` each time the function executes. Function-call subsystem `Accumulator` represents a periodic function that increments the value of state variable `x` by 1 and applies a tunable gain value `k`. The function executes every second.

```
open_system('ComponentDeploymentFcn')
```

Configure Model Code Interface

Model `ComponentDeploymentFcn` is configured to use the service code interface that is defined in `ComponentDeploymentCoderDictionary.sldd`. The code generator uses the linked dictionary to establish default code mappings that you can view and change by using the Code Mappings Editor or code mappings programming interface.

For elements for which the default interface aligns with the requirements, you do not need to make changes. For this example, to meet code interface requirements, these adjustments were made:

- On the **Functions** tab, the default function customization template for the aperiodic integrator and periodic accumulator functions produces function names `CD_Aperiodic` and `CD_Periodic`. To align with target platform requirements for these functions, the code mappings override the default functions names, by specifying the function names `CD_integrator` and `CD_accumulator`.
- An alternate timer service interface was associated with the integrator function. The coder dictionary defines two timer service interfaces: the default interface and an alternate interface `get_CD_tick`, which aligns with the target platform timer service requirements for function names. To see the timer service setting, on the **Functions** tab, select the row for Exported

Function: `Aperiodic` and click the pencil icon. In the dialog box that appears, see that the **Timer Service** property is set to `get_CD_tick`. Alternatively, you can set the property in the Property Inspector.

- On the **Parameters** tab, a code identifier was specified for the Gain block gain parameter `k`. To see the identifier setting, under **Model Parameters**, select the row for the parameter and click the pencil icon. In the dialog box that appears, see the value specified for the **Identifier** property, `k`. The identifier informs the code generator how to represent the variable in the generated code. Alternatively, you can set the property in the Property Inspector.
- On the **Signals/States** tab, code identifiers were specified for the states associated with the Discrete-Time Integrator and Unit Delay blocks. To see the identifier settings (`dti` and `delay`), select the row for the state of interest and click the pencil icon. In the dialog box that appears, see the value specified for the **Identifier** property. The identifiers inform the code generator how to represent corresponding variables in the generated code. Alternatively, you can set the property in the Property Inspector.

To review the code mappings, open the Embedded Coder app. Then, select **Code Interface > Component Interface**. The Code Mappings editor appears at the bottom of the app display.

In the Code Mappings editor, you can change the interfaces for the sender, receiver, data transfer, and timer services. To change the sender, receiver, and data transfer service, click the corresponding tab and, in the service column, select an alternative service. To change the timer service, click the **Functions** tab, select the row for the integrator function, click the pencil icon, and in the dialog that appears, select an alternative interface. The default interfaces use the outside-execution data communication method. The alternative interfaces use during-execution and direct-access data communication. Consider changing the interfaces to view differences in the generated code. If you choose to change the interfaces, for each service called along the data path of a signal, set the interfaces such that they apply the same data communication method.

For more information about model code mappings for service interfaces, see “C Service Interfaces” (Embedded Coder) and Code Mappings Editor - C (Embedded Coder).

Create Test Harness Model for Simulation

When you export functions, the generated code does not include a scheduler. Create a test harness model to handle scheduling during simulation. Do not use the test harness model to generate code that you deploy.

Model `ComponentDeploymentFcnHarness` is a test harness. The model provides function-call signals to other models in this example to schedule the model contents, including the model initialize and terminate events.

```
open_system('ComponentDeploymentFcnHarness')
```

For this model, in the Embedded Coder app, **Output** is set to `Simulation Only`. This setting allows you to switch focus between the test harness model and models from which you intend to generate code without opening a separate Simulink Editor window.

Simulate the Test Harness Model

Verify that the model containing the functions that you want to export is executed as you expect by simulating the test harness model. For example, simulate `ComponentDeploymentFcnHarness`.

```
sim('ComponentDeploymentFcnHarness')
```


Generate Function Code

In the **Embedded Coder** app, switch focus to component model by double clicking `ComponentDeploymentFcn`. Then, generate code for the functions that you want to export.

```
slbuild('ComponentDeploymentFcn')

### Starting build procedure for: ComponentDeploymentFcn
### Successful completion of code generation for: ComponentDeploymentFcn

Build Summary

Top model targets built:

Model                Action                Rebuild Reason
=====
ComponentDeploymentFcn  Code generated.      Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 27.562s
```

Review Generated Code

Review the generated code. The code generator creates the folder `ComponentDeploymentFcn_ert_rtw` in the current working folder and places source code files in that folder. The generated code is in two primary files: header file `ComponentDeploymentFcn.h` and source code file `ComponentDeploymentFcn.c`. `ComponentDeploymentFcn.c` calls the initialize function, exported functions `Integrator` and `Accumulator`, and terminate function. Model data and entry-point functions are accessible to a caller by including the header file.

The code generator also produces `rtwtypes.h` and a services header file, which by default the coder generator names `services.h`. Header file `rtwtypes.h` defines data types, structures, and macros that the generated code requires. Header file `services.h` is an interface header file that resides in subfolder `services`.

To inspect the generated code, use the generated Code Interface Report or, in the Embedded Coder app, use the Code view. The report is helpful for verifying that the generated interface code aligns with the target platform requirements. For more information, see “Analyze Generated Service Code Interface Report” (Embedded Coder).

Write Interface Code

Open and review the Code Interface Report. To write the interface code for your execution framework, use the information in that report.

- 1 Include the generated header files by adding directives `#include ComponentDeploymentFcn.h`, `#include rtwtypes.h`, and `#include services.h`.
- 2 Write input data to the generated code for model Inport blocks.
- 3 Call the generated entry-point functions.
- 4 Read data from the generated code for model Outport blocks.

Entry-point functions:

- Initialize entry-point function, `void CD_initialize(void)`. At startup, call this function once. The function uses a target environment receiver service to read data from nonvolatile memory by calling interface function `get_CD_initialize_InBus_NVM`.

- Terminate entry-point function, `void CD_terminate(void)`. At shutdown, call this function once. The function uses a target environment sender service to write data to nonvolatile memory by calling interface function `getref_CD_terminate_OutBus_NVM`.
- Exported periodic function, `void CD_accumulator(void)`. Call this function as needed. The function uses a target environment sender service to write the function output by calling interface function `getref_CD_accumulator_OutBus_y`.
- Exported aperiodic function, `void CD_integrator(void)`. Call this function as needed. The function uses a target environment receiver service to read input into the function by calling interface function `get_CD_integrator_inBus_u`. The `CD_integrator` function also uses a target environment timer service to get the target environment function clock tick by calling interface function `get_tick_outside_CD_integrator`.

More About

- “Generate Component Source Code for Export to External Code Base” (Embedded Coder)
- “Deploy Applications to Target Hardware” (Embedded Coder)
- “Customize Code Organization and Format” (Embedded Coder)
- “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder)
- “Service Interfaces” (Embedded Coder)

Generate C++ Function and Class Code for Export-Function Model

This example shows how to generate function code for an export-function model that includes a function-call subsystem. The code generator produces function and class code that does not include scheduling code.

To generate function code for export:

- 1 Create a model that contains the functions for export.
- 2 Create a test harness model that schedules execution of the functions during simulation.
- 3 Simulate the model that contains the functions by using the test harness model.
- 4 Generate code for the model that contains the functions.

Create Model That Contains Functions and C++ Class Interface for Export

The model with functions for export with a C++ model class interface must satisfy architectural constraints at the model root level. For C++ class generation, blocks that are valid at the root level are:

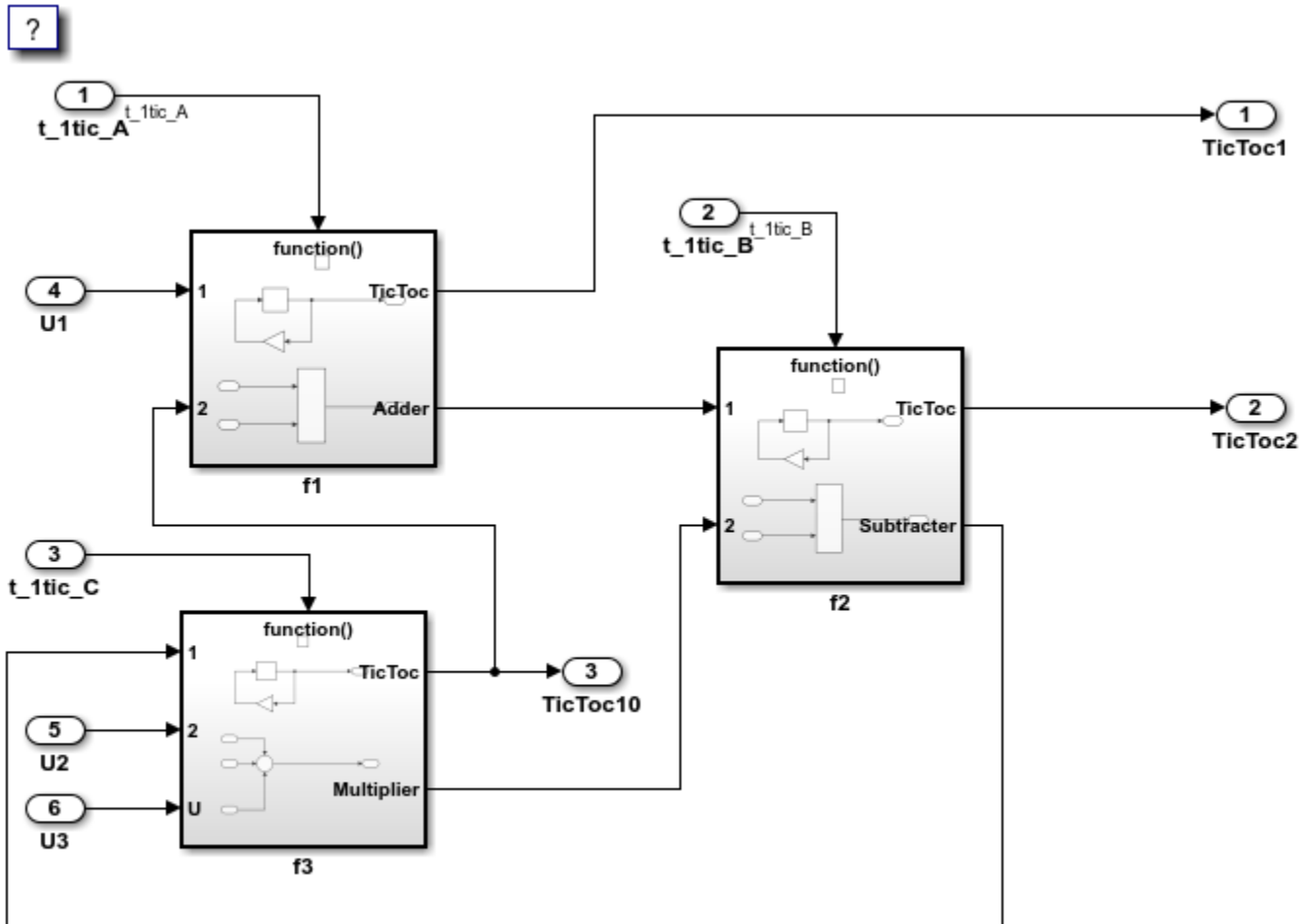
- Inport
- Outport
- Function-Call Subsystem
- Goto
- From
- Merge

Note: Export Function-Call Subsystem with C++ class interface does not support Simulink Function blocks.

The code generator produces function code for the Function-Call Subsystem block. For a Function-Call Subsystem block, you connect the block input ports to root Inport blocks that assert function-call signals. The subsystem is executed based on the function-call signal that it receives.

Model `rtwdemo_cppclass_functions` contains function-call subsystems `f1`, `f2`, and `f3` for exporting functions.

```
open_system('rtwdemo_cppclass_functions')
```



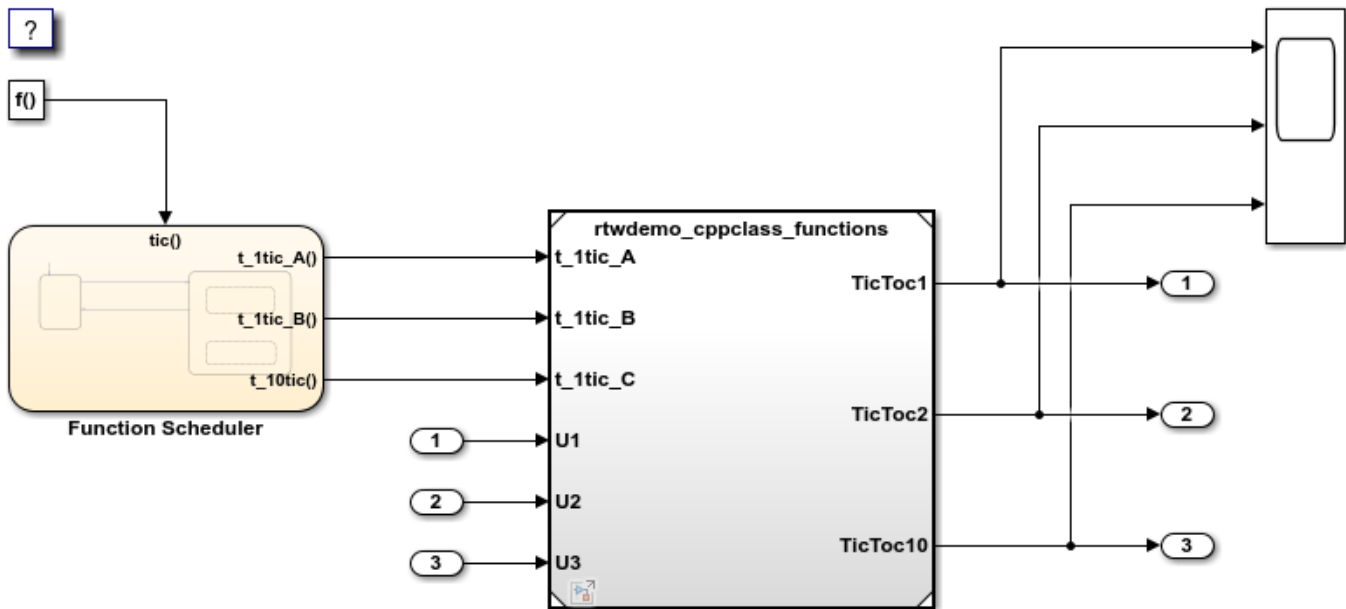
Copyright 2016-2020 The MathWorks, Inc.

Create Test Harness Model for Simulation

When you export functions, the generated code does not include a scheduler. Create a test harness model to handle scheduling during simulation. Do not use the test harness model to generate code that you deploy.

Model `rtwdemo_cppclass_export_functions` is a test harness. The model provides function-call signals to other models in this example to schedule the model contents.

```
open_system('rtwdemo_cppclass_export_functions')
```



Copyright 2014-2020 The MathWorks, Inc.

Simulate the Test Harness Model

Verify that the model containing the functions that you want to export is executed as you expect by simulating the test harness model. For example, simulate `rtwdemo_cppclass_export_functions`.

```
sim('rtwdemo_cppclass_export_functions')
```

Generate Function Code and Report

Generate code and a code generation report for the functions that you want to export. For example, generate code for `rtwdemo_cppclass_functions`.

```
slbuild('rtwdemo_cppclass_functions')
```

```
### Starting build procedure for: rtwdemo_cppclass_functions
### Successful completion of build procedure for: rtwdemo_cppclass_functions
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_cppclass_functions	Code generated and compiled.	Code generation information file does not exist.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.682s
```

Review Generated Code

From the code generation report, review the generated code.

- `ert_main.cpp` is an example main program (execution framework) for the model. This code shows how to call the exported functions. The code also shows how to initialize and execute the generated code.
- `rtwdemo_cppclass_functions.cpp` calls the initialization function, including `InitializeFunction`, and exported functions for model subsystem components `f1`, `f2`, and `f3`.
- `rtwdemo_cppclass_functions.h` declares model data structures and a public interface to the exported entry-point functions and data structures.
- `rtwtypes.h` defines data types, structures, and macros that the generated code requires.

Write Interface Code

Open and review the Code Interface Report. To write the interface code for your execution framework, use the information in that report.

- 1 Include the generated header files by adding directives `#include rtwdemo_cppclass_functions.h` and `#include rtwtypes.h`.
- 2 Write input data to the generated code for model Inport blocks.
- 3 Call the generated entry-point functions.
- 4 Read data from the generated code for model Outport blocks.

Input ports:

- `rtU.U1` of type `real_T` with dimension 1
- `rtU.U2` of type `real_T` with dimension 1
- `rtU.U3` of type `real_T` with dimension 1

Entry-point functions:

- Initialize entry-point function, `void initialize(void)`. At startup, call this function once.
- Exported function, `void t_ltic_A(void)`. Call this function as needed.
- Exported function, `void t_ltic_B(void)`. Call this function as needed.
- Exported function, `void t_ltic_C(void)`. Call this function as needed.

Output ports:

- `rtY.TicToc1` of type `int8_T` with dimension [2]
- `rtY.TicToc2` of type `int8_T` with dimension [2]
- `rtY.TicToc10` of type `int8_T` with dimension 1

More About

- “Generate Component Source Code for Export to External Code Base” (Embedded Coder)
- “Deploy Applications to Target Hardware” (Embedded Coder)
- “Customize Code Organization and Format” (Embedded Coder)
- “Generate Code for Simulink Function and Function Caller” on page 10-9

Close Example Models

```
bdclose('rtwdemo_cppclass_export_functions')
bdclose('rtwdemo_cppclass_functions')
```

Generate Code for Export-Function Subsystems

- “Specify a Custom Initialize Function Name” on page 30-52
- “Specify a Custom Description” on page 30-53
- “Optimize Code Generated for Export-Function Subsystems” on page 30-53

To generate code for an export-function subsystem:

- 1 Verify that the subsystem for which you are generating code satisfies exporting requirements (Embedded Coder).
- 2 In the Configuration Parameters dialog box:
 - a Set the parameter **System target file** to an ERT-based system target file, such as `ert.tlc`.
 - b If you want a SIL block with the generated code, for verification purposes, set model configuration parameter **Create block** to SIL.
 - c Click **OK** or **Apply**.
- 3 Right-click the subsystem block and select **C/C++ Code > Export Functions** from the context menu.

The operation creates and builds a new model, `subsystem.slx`, that contains the content of the original subsystem and creates a `ScratchModel` that contains a `Model` block. This block references the newly created `subsystem.slx` model.

The code generator produces code and places it in the working folder.

If you set **Create block** to SIL in step 2b, Simulink opens a new window that contains an S-function block that represents the generated code. This block has the same size, shape, and connectors as the original subsystem.

Code generation and optional block creation are now complete. You can test and use the code and optional block as you do for generated ERT code and S-function block. For optional workflow tasks, see “Specify a Custom Initialize Function Name” on page 30-52 and “Specify a Custom Description” on page 30-53.

Specify a Custom Initialize Function Name

You can specify a custom name for the initialize function of your exported function as an argument to the `slbuild` command. The command takes the following form:

```
blockHandle = slbuild('subsystem', 'Mode', 'ExportFunctionCalls',...
    'ExportFunctionInitializeFunctionName', 'fcname')
```

`fcname` specifies the function name. For example, if you specify the name `'myinitfcn'`, the build process emits code similar to:

```
/* Model initialize function */
void myinitfcn(void){
```

```
...
}
```

Specify a Custom Description

You can enter a custom description for an exported function by using the Block Properties dialog box of an Inport block.

- 1 Right-click the Inport block that drives the control port of the subsystem for which you are exporting code.
- 2 Select **Properties**.
- 3 In the **General** tab, in the **Description** field, enter your descriptive text.

During function export, the text you enter is emitted to the generated code in the header for the Inport block. For example, if you open the example program `rtwdemo_exporting_functions` and enter a description in the Block Properties dialog box for port `t_ltic_A`, the code generator produces code that is similar to:

```
/*
 * Output and update for exported function: t_ltic_A
 *
 * My custom description of the exported function
 */
void t_ltic_A(void)
{
...
}
```

Optimize Code Generated for Export-Function Subsystems

To optimize the code generated for an export-function subsystem, specify a separate storage class for each input signal and output signal that crosses the boundary of the subsystem.

For each function-call subsystem that you are exporting:

- 1 Right-click the subsystem.
- 2 From the context menu, choose **Block Parameters (Subsystem)**.
- 3 Select the **Code Generation** tab.
- 4 Set **Function packaging** to Auto.
- 5 Click **OK** or **Apply**.

See Also

More About

- “Using Triggered Subsystems”
- “Using Function-Call Subsystems”
- “Export-Function Models Overview”

Generate Shared Library for Export to External Code Base

In this section...

“About Generated Shared Libraries” on page 30-54

“Workflow” on page 30-54

“Generate Shared Libraries” on page 30-55

“Create Application Code That Uses Generated Shared Libraries” on page 30-55

“Limitations” on page 30-58

About Generated Shared Libraries

If you have Embedded Coder software, you can generate a shared library—Windows dynamic link library (.dll), UNIX shared object (.so), or Macintosh OS X dynamic library (.dylib)— from a model component. You or others can integrate the shared library into an application that runs on a Windows, UNIX, or Macintosh OS X development computer. Uses of shared libraries include:

- Adding a software component to an application for system simulation
- Reusing software modules among applications on a development computer
- Hiding intellectual property associated with software that you share with vendors

When producing a shared library, the code generator exports:

- Variables and signals of type `ExportedGlobal` as data
- Real-time model structure (*model_M*) as data
- Functions essential to executing the model code

Workflow

To generate a shared library from a model component and use the library, complete the tasks listed in this table.

Task	Action	More Information
1	Review your assessment of external code characteristics and integration requirements.	<ul style="list-style-type: none"> • “Choose an External Code Integration Workflow” on page 30-2 • “Shared Library Limitations” (Embedded Coder)
2	Configure the model for code generation.	“Generate Code That Matches Appearance of External Code” on page 30-70 “Model Configuration Set Customization”
3	Configure the model for the code generator to produce a shared library and initiate code generation.	“Generate Shared Libraries” on page 30-55

Task	Action	More Information
4	<p>Verify that the generated shared library meets requirements. For example, review the code generation report and view the list of symbols in the library.</p> <ul style="list-style-type: none"> • On Windows, use the Dependency Walker utility, downloadable from www.dependencywalker.com • On UNIX, use <code>nm -D model.so</code> • On Macintosh OS X, use <code>nm -g model.dylib</code> 	
5	Use the shared library in application code.	“Create Application Code That Uses Generated Shared Libraries” on page 30-55
6	Compile and link application code that loads and uses the generated shared library.	“Build Integrated Code Within the Simulink Environment” on page 30-33
7	Verify that executable program behaves and performs as expected.	“Verification and Testing”

Generate Shared Libraries

- 1 When configuring the model for code generation, select the system target file `ert_shrlib.tlc`.
- 2 Build the model. The code generator produces source code for the model and a shared library version of the code. The code generator places the source code in the code generation folder and the shared library (`.dll`, `.so`, or `.dylib` file) in your current working folder. The code generator also produces and retains a `.lib` file to support implicit linking.

Create Application Code That Uses Generated Shared Libraries

This example application code is generated for the example “Interface to a Development Computer Simulator by Using a Shared Library”.

- 1 Create an application header file that contains type declarations for model external input and output. For example:

```
#ifndef _APP_MAIN_HEADER_
#define _APP_MAIN_HEADER_

typedef struct {
    int32_T Input;
} ExternalInputs_rtwdemo_shrlib;

typedef struct {
    int32_T Output;
} ExternalOutputs_rtwdemo_shrlib;

#endif /* _APP_MAIN_HEADER_ */
```

- 2 In the application C source code, dynamically load the shared library. Use preprocessing conditional statements to invoke platform-specific commands. For example:

```

#if defined(_WIN32)||defined(_WIN64) /* WINDOWS */
#include <windows.h>
#define GETSYMBOLADDR GetProcAddress
#define LOADLIB LoadLibrary
#define CLOSELIB FreeLibrary

#else /* UNIX */
#include <dlfcn.h>
#define GETSYMBOLADDR dlsym
#define LOADLIB dlopen
#define CLOSELIB dlclose

#endif

int main()
{
    void* handleLib;
    ...
#if defined(_WIN64)
    handleLib = LOADLIB("./rtwdemo_shrlib_win64.dll");
#else #if defined(_WIN32)
    handleLib = LOADLIB("./rtwdemo_shrlib_win32.dll");
#else /* UNIX */
    handleLib = LOADLIB("./rtwdemo_shrlib.so", RTLD_LAZY);
#endif
#endif
    ...
    return(CLOSELIB(handleLib));
}

```

- 3 From the application C source code, access exported data and functions generated from the model. The code uses hooks to add user-defined initialization, step, and termination code.

```

int32_T i;
...
void (*mdl_initialize)(boolean_T);
void (*mdl_step)(void);
void (*mdl_terminate)(void);

ExternalInputs_rtwdemo_shrlib (*mdl_Uptr);
ExternalOutputs_rtwdemo_shrlib (*mdl_Yptr);

uint8_T (*sum_outptr);
...
#if defined(LCCDLL)||defined(BORLANDCDLL)
/* Exported symbols contain leading underscores
when DLL is linked with LCC or BORLANDC */
mdl_initialize =
    (void (*)(boolean_T))GETSYMBOLADDR(handleLib ,
    "_rtwdemo_shrlib_initialize");
mdl_step =
    (void (*)(void))GETSYMBOLADDR(handleLib ,
    "_rtwdemo_shrlib_step");
mdl_terminate =
    (void (*)(void))GETSYMBOLADDR(handleLib ,
    "_rtwdemo_shrlib_terminate");
mdl_Uptr =
    (ExternalInputs_rtwdemo_shrlib*)GETSYMBOLADDR(handleLib ,

```

```

        "_rtwdemo_shrplib_U");
mdl_Yptr =
    (ExternalOutputs_rtwdemo_shrplib*)GETSYMBOLADDR(handleLib ,
    "_rtwdemo_shrplib_Y");
sum_outptr =
    (uint8_T*)GETSYMBOLADDR(handleLib , "_sum_out");
#else
mdl_initialize =
    (void*)(boolean_T)GETSYMBOLADDR(handleLib ,
    "rtwdemo_shrplib_initialize");
mdl_step =
    (void*)(void)GETSYMBOLADDR(handleLib ,
    "rtwdemo_shrplib_step");
mdl_terminate =
    (void*)(void)GETSYMBOLADDR(handleLib ,
    "rtwdemo_shrplib_terminate");
mdl_Uptr =
    (ExternalInputs_rtwdemo_shrplib*)GETSYMBOLADDR(handleLib ,
    "rtwdemo_shrplib_U");
mdl_Yptr =
    (ExternalOutputs_rtwdemo_shrplib*)GETSYMBOLADDR(handleLib ,
    "rtwdemo_shrplib_Y");
sum_outptr =
    (uint8_T*)GETSYMBOLADDR(handleLib , "sum_out");
#endif

if ((mdl_initialize &&
    mdl_step &&
    mdl_terminate &&
    mdl_Uptr &&
    mdl_Yptr &&
    sum_outptr)) {
    /* user application
       initialization function */
    mdl_initialize(1);
    /* insert other user defined
       application initialization code here */

    /* user application
       step function */
    for(i=0;i<=12;i++){
        mdl_Uptr->Input = i;
        mdl_step();
        printf("Counter out(sum_out):
            %d\tAmplifier in(Input):
            %d\tout(Output):
            %d\n", *sum_outptr, i,
            mdl_Yptr->Output);
        /* insert other user defined
           application step function
           code here */
    }

    /* user application
       terminate function */
    mdl_terminate();
    /* insert other user defined
       application termination

```

```
        code here */
    }
    else {
        printf("Cannot locate the specified
              reference(s) in the shared library.\n");
        return(-1);
    }
}
```

Limitations

- Code generation for the `ert_shrlib.tlc` system target file exports the following as data:
 - Variables and signals of type `ExportedGlobal`
 - Real-time model structure (`model_M`)
- Code generation for the `ert_shrlib.tlc` system target file supports the C language only (not C++). When you select `ert_shrlib.tlc`, language selection is unavailable on the **Code Generation** pane in the Configuration Parameters dialog box.
- To reconstruct a model simulation by using a generated shared library, the application author must maintain the timing between system and shared library function calls in the original application. The timing must be consistent so that you can compare the simulation and integration results. Additional simulation considerations apply if generating a shared library from a model that enables model configuration parameters **Support: continuous time** and **Single output/update function**. For more information, see [Single output/update function \(Simulink Coder\) dependencies](#).

See Also

More About

- “Interface to a Development Computer Simulator by Using a Shared Library”
- “Map System Architecture to Simulink Modeling Environment” on page 1-3
- “Configure a System Target File” on page 21-2
- “Manage Build Process Files” on page 24-35
- “Model Protection”

Exchange Data Between External C/C++ Code and Simulink Model or Generated Code

Whether you import your external code into a Simulink model (for example, by using the Legacy Code Tool) or export the generated code to an external environment, the model or the generated code typically exchange data (signals, states, and parameters) with your code.

Functions in C or C++ code, including your external functions, can exchange data with a caller or a called function through:

- Arguments (formal parameters) of functions. When a function exchanges data through arguments, an application can call the function multiple times. Each instance of the called function can manipulate its own independent set of data so that the instances do not interfere with each other.
- Direct access to global variables. Global variables can:
 - Enable different algorithms (functions) and instances of the same algorithm to share data such as calibration parameters and error status.
 - Enable the different rates (functions) of a multitasking system to exchange data.
 - Enable different algorithms to exchange data asynchronously.

In Simulink, you can organize and configure data so that a model uses these exchange mechanisms to provide, extract, and share data with your code.

Before you attempt to match data interfaces, to choose an overall integration approach, see “Choose an External Code Integration Workflow” on page 30-2.

Import External Code into Model

To exchange data between your model and your external function, choose an exchange mechanism based on the technique that you chose to integrate the function.

- To exchange data through the arguments of your external function, construct and configure your model to create and package the data according to the data types of the arguments. Then, you connect and configure the block that calls or represents your function to accept, produce, or refer to the data from the model.

For example, if you use the Legacy Code Tool to generate an S-Function block that calls your function, the ports and parameters of the block correspond to the arguments of the function. You connect the output signals of upstream blocks to the input ports and set parameter values in the block mask. Then, you can create signal lines from the output ports of the block and connect those signals to downstream blocks.

- To exchange data through global variables that your external code already defines, a best practice is to use a Stateflow chart to call your function and to access the variables. You write algorithmic C code in the chart so that during simulation or execution of the generated code, the model reads and writes to the variables.

To use such a global variable as an item of parameter data (not signal or state data) elsewhere in a model, you can create a numeric MATLAB variable or `Simulink.Parameter` object that represents the variable. If you change the value of the C-code variable in between simulation runs, you must manually synchronize the value of the Simulink variable or object. If your algorithmic

code (function) changes the value of the C-code variable during simulation, the corresponding Simulink variable or object does not change.

If you choose to create a Simulink representation of the C-code variable, you can configure the Simulink representation so that the generated code reads and writes to the variable but does not duplicate the variable definition. Apply a storage class to the Simulink representation.

Technique for Integrating External Function	Mechanism to Exchange Data with Model	Examples and More Information
S-Function block	Function arguments	To call your function through an S-function that you create by using the Legacy Code Tool, see “Integrate C Functions into Simulink Models with Legacy Code Tool”.
Stateflow chart	Function arguments and direct access to global variables	To call your function and access global variables in a Stateflow chart, see “Access Custom Code Variables and Functions in Stateflow Charts” (Stateflow). For information about creating data items in a chart (which you can pass to your function as arguments), see “Add Stateflow Data” (Stateflow).
<code>coder.ceval</code> in MATLAB Function block	Function arguments	To call your function in a MATLAB Function block by using <code>coder.ceval</code> , see “Integrate C Code by Using the MATLAB Function Block”. For information about creating data items in a MATLAB Function block (which you can pass to your function as arguments), see “Create and Define MATLAB Function Block Variables”.

Export Generated Code to External Environment

To export the generated code into your external code, see “Exchange Data Between External Calling Code and Generated Code” on page 30-67.

Simulink Representations of C Data Types and Constructs

To model and reuse your custom C data such as structures, enumerations, and typedef aliases, use the information in these tables.

Modeling Patterns for Matching Data in External C Code

C Data Type or Construct	Example C Code	Simulink Equivalent	More Information
Primitive type alias (typedef)	typedef float mySinglePrec_T;	<p>Create a <code>Simulink.AliasType</code> object. Use the object to:</p> <ul style="list-style-type: none"> Set the data types of signals and block parameters in a model. Configure data type replacements for code generation. <p>Generating code that uses an alias data type requires Embedded Coder.</p>	<p>For information about defining custom data types for your model, see <code>Simulink.AliasType</code> and “Manage Replacement of Simulink Data Types in Generated Code” (Embedded Coder).</p> <p>For an example that shows how to export the generated code into your external code, see “Replace and Rename Simulink Coder Data Types to Conform to Coding Standards” (Embedded Coder).</p>
Array	int myArray[6];	<p>Specify signal and parameter dimensions as described in “Determine Signal Dimensions”.</p> <p>The generated code defines and accesses multidimensional data, including matrices, as column-major serialized vectors. If your external code uses a different format, consider using alternative techniques to integrate the generated code. See “Code Generation of Matrices and Arrays” on page 24-63.</p>	<p>For information about how the generated code stores nonscalar data (including limitations), see “Code Generation of Matrices and Arrays” on page 24-63.</p> <p>For an example that shows how to export the generated code into your external code, see “Reuse Parameter Data in Different Data Type Contexts” on page 15-143.</p> <p>To model lookup tables, see <code>Simulink.LookupTable</code>.</p>

C Data Type or Construct	Example C Code	Simulink Equivalent	More Information
Enumeration	<pre>typedef enum myColorsType { Red = 0, Yellow, Blue } myColorsType;</pre>	Define a Simulink enumeration that corresponds to your enumeration definition. Use the Simulink enumeration to set data types in a model.	<p>To use enumerated data in a Simulink model, see “Use Enumerated Data in Generated Code” on page 15-86.</p> <p>For an example that shows how to generate enumerated parameter data, see “Enumeration” (Embedded Coder).</p> <p>For an example that shows how to export the generated code into your external code by exchanging enumerated data, see “Exchange Structured and Enumerated Data Between Generated and External Code” (Embedded Coder).</p>

C Data Type or Construct	Example C Code	Simulink Equivalent	More Information
Structure	<pre>typedef struct myStructType { int count; double coeff; } myStructType;</pre>	<p>Create a <code>Simulink.Bus</code> object that corresponds to your structure type.</p> <p>To create structured signal or state data, package multiple signal lines in a model into a single nonvirtual bus signal.</p> <p>To create structured parameter data, create a parameter object (such as <code>Simulink.Parameter</code>) that stores a MATLAB structure. Use the bus object as the data type of the parameter object.</p> <p>To package lookup table data into a structure, use <code>Simulink.LookupTable</code> and, optionally, <code>Simulink.Breakpoint</code> objects.</p>	<p>For information about bus signals, see “Group Signals or Messages into Virtual Buses”. For information about structures of parameters, see “Organize Related Block Parameter Definitions in Structures”.</p> <p>For an example that shows how to import your external code into a model by using the Legacy Code Tool, see “Integrate C Function Whose Arguments Are Pointers to Structures”.</p> <p>For examples that show how to export the generated code into your external code, see “Exchange Structured and Enumerated Data Between Generated and External Code” (Embedded Coder) and “Access Structured Data Through a Pointer That External Code Defines” (Embedded Coder).</p> <p>To package lookup table data into a structure, <code>Simulink.LookupTable</code>.</p> <p>For general information about creating structures in the generated code, see “Organize Data into Structures in Generated Code” on page 15-147.</p>

Additional Modeling Patterns for Code Generation (Embedded Coder)

C Data Type or Construct	Example C Code	Simulink Equivalent	More Information
Macro	<pre>#define myParam 9.8</pre>	<p>Apply the storage classes <code>Define</code> and <code>ImportedDefine</code> to parameters. With macros, you can reuse a parameter value in multiple locations in an algorithm and change the parameter value between code compilations without consuming memory to store the value. Typically, macros represent engineering constants that you do not expect to change during code execution.</p> <p>This technique requires Embedded Coder.</p>	“Macro Definitions (<code>#define</code>)” (Embedded Coder)
Storage type qualifiers such as <code>const</code> and <code>volatile</code>	<pre>const myParam = 9.8;</pre>	Apply the storage classes <code>Const</code> , <code>Volatile</code> , and <code>ConstVolatile</code> to data items.	“Protect Global Data with Keywords <code>const</code> and <code>volatile</code> ” on page 30-68
Bit field	<pre>typedef struct myBitField { unsigned short int MODE : 1; unsigned short int FAIL : 1; unsigned short int OK : 1; } myBitField</pre>	<ul style="list-style-type: none"> Apply the storage class <code>BitField</code> to signals, states, and parameters whose data type is <code>boolean</code>. Use a model configuration parameter to aggregate Boolean data into bit fields. <p>These techniques require Embedded Coder.</p>	<p>“Bitfields” (Embedded Coder)</p> <p>“Optimize Generated Code by Packing Boolean Data into Bitfields” (Embedded Coder)</p>

C Data Type or Construct	Example C Code	Simulink Equivalent	More Information
Call to custom external function that reads or writes to data	External code: <pre> /* Call this function to acquire the value of the signal. */ double get_inSig(void) { return myBigGlobalStruct.inSig; } </pre> Generated algorithmic code: <pre> algorithmInput = get_inSig(); </pre>	Apply the storage class <code>GetSet</code> to signals, states, and parameters. Each data item appears in the generated code as a call to your custom functions that read and write to the target data. This technique requires Embedded Coder.	“Access Data Through Functions with Storage Class <code>GetSet</code> ” (Embedded Coder)

Considerations for Other Modeling Goals

Goal	Considerations and More Information
Use Simulink to run and interact with the generated code	<p>You can use SIL, PIL, and external mode simulations to connect your model to the corresponding generated application for simulation. When you import parameter data from your external code:</p> <ul style="list-style-type: none"> At the time that you begin an external mode simulation, the external executable uses the value that your code uses to initialize the parameter data. However, when you change the corresponding value in Simulink during the simulation (for example by modifying the <code>Value</code> property of the corresponding parameter object), Simulink downloads the new value to the executable. SIL and PIL simulations do not import the parameter value from your code. Instead, the simulations use the parameter value from Simulink. If you include your external code in the Simulink Coder build process, duplicate data definitions can prevent the generated code from compiling. <p>For information about SIL and PIL, see “Choose a SIL or PIL Approach” (Embedded Coder). For information about external mode simulation, see “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” on page 32-2.</p>
Generate code comments that describe attributes of exported data including physical units, real-world initial value, and data type	<p>Generating these comments can help you match data interfaces while handwriting integration code. See “Add Custom Comments for Variables in the Generated Code” (Embedded Coder).</p>

See Also

Related Examples

- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “Generate Code That Matches Appearance of External Code” on page 30-70

- “Design Data Interface by Configuring Inport and Outport Blocks” on page 15-170
- “Configure Generated Code According to Interface Control Document Specifications” (Embedded Coder)
- “Generate Code That Dereferences Data from a Literal Memory Address” (Embedded Coder)

Exchange Data Between External Calling Code and Generated Code

To export the generated code into your external code, you configure the generated code to match the data interface of your external code. For example, if your external code defines a global variable for storing output data and you need the generated code to read that data as input, you can configure a corresponding Inport block so that the generated code interacts with the existing variable.

- You can generate reentrant code from a model. The generated entry-point functions typically accept data through arguments. Your calling code passes data through these arguments. You can call the functions multiple times in a single application—the application can maintain multiple instances of the model. See “Data Exchange for Reentrant Generated Code” on page 30-67.
- When you generate nonreentrant code, by default, the entry-point functions exchange data between the generated code and your code through global variables. You can generate variable definitions for your code to use or you can share and reuse existing variables that your code already defines. See “Data Exchange for Nonreentrant Generated Code” on page 30-67.

Alternatively, you can configure the entry-point functions to exchange system inputs and outputs (root-level Inport and Outport blocks) through arguments instead of global variables. Apply function prototype control to the model, which requires Embedded Coder. For more information about function prototype control, see “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder).

For general information about exchanging data between generated and external code, including how to match specific C code patterns, see “Exchange Data Between External C/C++ Code and Simulink Model or Generated Code” on page 30-59.

Data Exchange for Reentrant Generated Code

When you generate reentrant code, the model entry-point functions exchange data through arguments. You can control some of the characteristics of the arguments. For more information, see “Control Data and Function Interface in Generated Code” on page 15-44.

For information about creating a separate set of data for each call site in your external code, see “Modify Static Main to Allocate and Access Model Instance Data” (Embedded Coder).

Data Exchange for Nonreentrant Generated Code

To make the generated code read or write to an item of signal, state, or parameter data as a global variable, apply a storage class to the data in the model. The storage class also determines whether the generated code exports the variable definition to your external code or imports the definition from your code. For general information about controlling the data interface of a model that you configure to generate nonreentrant code, see “Control Data Interface for Nonreentrant Code” on page 15-44. For examples, see “Design Data Interface by Configuring Inport and Outport Blocks” on page 15-170 and “Integrate External Application Code with Code Generated from PID Controller” (Embedded Coder).

When you generate code that defines (allocates memory for) global data, the generated code exports that data. When your external code defines data, the generated code imports that data. Typically, storage classes that import data have the word `Import` in the storage class name, for example, `ImportedExternPointer`.

Control File Placement of Exported Global Data (Embedded Coder)

When you export data from the generated code by using storage classes, the code generator creates an extern declaration. By default, this declaration typically appears in the generated header file `model.h`. You can include (`#include`) this header file in your external code.

By default, the definition (memory allocation) of exported data typically appears in `model.c`.

You can control the file placement of the declarations and definitions to:

- Create separate object files that store only global parameter data.
- Modularize the generated code by organizing declarations into separate files.

For more information about controlling file placement of declarations and definitions, see “Control Placement of Global Data Definitions and Declarations in Generated Files” (Embedded Coder). For an example, see “Definition, Initialization, and Declaration of Parameter Data” (Embedded Coder).

Prevent Duplicate Initialization Code for Global Variables

When your external code defines global variables, you can generate code that interacts with those variables. For example, use the storage class `ImportFromFile` (see “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75).

For imported variables that represent signal or state data, the code generator can produce initialization code as described in “Initialization of Signal, State, and Parameter Data in the Generated Code” on page 15-188. If your code already initializes a variable, consider preventing the generation of duplicate initialization code. Create your own storage class and, for that storage class, set **Data initialization** to `None`. For information about creating and applying your own storage class, see “Define Service Interfaces, Storage Classes, Memory Sections, and Function Templates for Software Architecture” (Embedded Coder).

Protect Global Data with Keywords `const` and `volatile`

When your external code and the generated code exchange data through global variables, you can generate code that uses C type qualifiers `const` and `volatile` to protect data integrity and improve the safety of your application. For example:

- Apply `const` to a calibration parameter.
- Apply `volatile` to a global variable that stores the output of a hardware device operating asynchronously.
- Apply `const` and `volatile` to signals, states, and parameters that represent data defined by your external code. Then, the generated code declares and interacts with the external data by using the corresponding storage type qualifiers.

You must have Embedded Coder. For more information, see “Protect Global Data with `const` and `volatile` Type Qualifiers” (Embedded Coder).

See Also

Related Examples

- “Exchange Data Between External C/C++ Code and Simulink Model or Generated Code” on page 30-59

- “Control Data and Function Interface in Generated Code” on page 15-44
- “Design Data Interface by Configuring Inport and Outport Blocks” on page 15-170
- “Integrate External Application Code with Code Generated from PID Controller” (Embedded Coder)

Generate Code That Matches Appearance of External Code

A key aspect of code integration, especially for larger projects, is adherence to guidelines and standards for code appearance. If code appearance requirements apply to your project, review the requirements in this table. To learn more, see the relevant information.

Requirement	More Information
Thoroughly document code or document code in a specific way.	<ul style="list-style-type: none"> • “Configure Code Comments” (Embedded Coder) • “Annotate Code for Justifying Polyspace Checks” (Embedded Coder) • “Add Custom Comments for Variables in the Generated Code” (Embedded Coder) • “Add Global Comments in the Generated Code” (Embedded Coder)
Control the length and naming of code identifiers (symbols), including the use of reserved names.	<ul style="list-style-type: none"> • “Identifier Format Control” (Embedded Coder) • “Specify Identifier Length to Avoid Naming Collisions” (Embedded Coder) • “Specify Reserved Names for Generated Identifiers” (Embedded Coder) • “Customize Generated Identifier Naming Rules” (Embedded Coder) • “Avoid Identifier Name Collisions with Referenced Models” (Embedded Coder) • “Control Name Mangling in Generated Identifiers” (Embedded Coder) • “Specify Boolean and Data Type Limit Identifiers” (Embedded Coder)

Requirement	More Information
Control style aspects of code: <ul style="list-style-type: none"> • Level of parenthesization • Order of operations in expressions • Empty primary condition expressions in <code>if</code> statements • <code>if-elseif-else</code> or <code>switch-case</code> statements for decision logic • Use of <code>extern</code> keyword in function declarations • Use of <code>static</code> keyword in function declarations • Use of <code>default</code> cases for <code>switch-case</code> statements • Use multiplications by powers of two or signed bitwise shifts • Cast expressions • Indent style (Kernighan and Ritchie or Allman) and size • Newline style 	“Model Configuration Parameters: Code Style” (Embedded Coder)
Placement of data definitions and declarations, including location of global identifiers and global data declarations (<code>extern</code>)	“Control Placement of Global Data Definitions and Declarations in Generated Files” (Embedded Coder)
Appearance of code for flow charts	“Enhance Readability of Code for Flow Charts” (Embedded Coder)
Apply code templates to control organization of code and use of banners	<ul style="list-style-type: none"> • “Customize Code Organization and Format” (Embedded Coder) • “Template Symbols and Rules” (Embedded Coder) • “Specify Templates For Code Generation” (Embedded Coder) • “Generate Custom File and Function Banners” (Embedded Coder) • “Change the Organization of a Generated File” (Embedded Coder) • “Generate Source and Header Files with a Custom File Processing (CFP) Template” (Embedded Coder) • “Customize Generated File Names” (Embedded Coder)

See Also

More About

- “Choose an External Code Integration Workflow” (Embedded Coder)

- “Model Configuration Set Customization”

Program Building, Interaction, and Debugging in Simulink Coder

- “Select C or C++ Programming Language” on page 31-2
- “Select and Configure C or C++ Compiler” on page 31-3
- “Troubleshoot Compiler Issues” on page 31-7
- “Approaches for Building Code Generated from Simulink Models” on page 31-11
- “Configure CMake Build Process” on page 31-13
- “Create Custom CMake Toolchain Definition” on page 31-17
- “Configure Toolchain (ToolchainInfo) or Template Makefile Build Process” on page 31-20
- “Define Custom Makefile-Based Toolchains Using Target Framework” on page 31-30
- “Template Makefiles and Make Options” on page 31-36
- “Build Process Workflow for Real-Time Systems” on page 31-39
- “Build Models from a Windows Command Prompt Window” on page 31-45
- “Rebuild a Model” on page 31-48
- “Control Regeneration of Top Model Code” on page 31-49
- “Reduce Build Time for Referenced Models by Using Parallel Builds” on page 31-51
- “Relocate or Share Generated Code” on page 31-56
- “Compile and Debug Generated C Code with Microsoft Visual Studio” on page 31-61
- “Executable Program Generation” on page 31-62

Select C or C++ Programming Language

The default programming language selection for code generation is C language. In the code generation workflow, after the steps in “Select a Solver That Supports Code Generation” on page 21-2 and “Select a System Target File from STF Browser” on page 21-2, an optional step is to change the programming language selection for code generation.

To change the programming language setting:

- 1 In the Configuration Parameters dialog box, from the **Language** list, select C or C++. Alternatively, set the `TargetLang` parameter at the command line.

The code generator produces `.c` or `.cpp` files, depending on your selection, and places the generated files in your build folder.

For more information, see [Language](#).

- 2 Check whether you must choose and configure a compiler. If you select C++, you must choose and configure a compiler. For details, see “Select and Configure C or C++ Compiler” on page 31-3.
- 3 Check whether the language standard is configured for your compiler. By default, the code generator uses the ISO/IEC 9899:1999 C (C99 (ISO)) library for the C language and the ISO/IEC 14882:2003 C++ (C++03 (ISO)) library for the C++ language.

For more information, see [Language standard](#).

See Also

More About

- “Select and Configure C or C++ Compiler” on page 31-3
- “Troubleshoot Compiler Issues” on page 31-7

Select and Configure C or C++ Compiler

The build process requires a supported compiler. *Compiler*, in this context, refers to a development environment containing a linker and make utility, and a high-level language compiler. For details on supported compiler versions, see:

https://www.mathworks.com/support/compilers/current_release

When creating an executable program, the build process must be able to access a supported compiler. The build process can find a compiler to use based on your default MEX compiler.

The build process also requires the selection of a toolchain or template makefile. The toolchain or template makefile determines which compiler runs, during the make phase of the build. For more information, see “Configure Toolchain (ToolchainInfo) or Template Makefile Build Process” on page 31-20.

To determine which templates makefiles are available for your compiler and system target file, see “Compare System Target File Support Across Products” on page 21-17.

For both generated files and user-supplied files, the file extension, `.c` or `.cpp`, determines whether the build process uses a C or a C++ compiler. If the file extension is `.c`, the build process uses C compiler to compile the file, and the symbols use the C linkage convention. If the file extension is `.cpp`, the build process uses a C++ compiler to compile the file, and the symbols use the C++ linkage specification.

Language Standards Compliance

The code generator produces code that is compliant with the following standards:

Language	Supported Standard	Other Names
C	ISO/IEC 9899:1990	C89/C90
	ISO/IEC 9899:1999	C99
C++	ISO/IEC 14882:2003	C++03
	ISO/IEC 14882:2011	C++11

Code that the code generator produces from these sources is ANSI C/C++ compliant:

- Simulink built-in block algorithmic code
- Generated system-level code (task ID [TID] checks, management, functions, and so on)
- Code from other blocksets, including the Fixed-Point Designer product and the Communications Toolbox product
- Code from other code generators, such as MATLAB functions

Also, the code generator can incorporate code from:

- Embedded system target files (for example, startup code, device driver blocks)
- Custom S-functions or TLC files

Note Coding standards for these two sources are beyond the control of the code generator. These standards can be a source for compliance problems, such as code that uses C99 features not supported in the ANSI C, C89/C90 subset.

Programming Language Considerations

The code generator produces C and C++ code. Consider the following as you choose a programming language:

- Does your project require you to configure the code generator to use a specific compiler? C/C++ code generation on Windows requires this selection.
- Does your project require you to change the default language configuration setting for the model? See “Select C or C++ Programming Language” on page 31-2.
- Does your project require you to integrate legacy or custom code with generated code? For a summary of integration options, see “Choose an External Code Integration Workflow” (Embedded Coder).
- Does your project require you to integrate C and C++ code? If so, see “Choose an External Code Integration Workflow” (Embedded Coder).

Note You can mix C and C++ code when integrating generated code with custom code. However, you must be aware of the differences between C and default C++ linkage conventions, and add the `extern "C"` linkage specifier where required. For the details of the differing linkage conventions and how to apply `extern "C"`, refer to a C++ programming language reference book.

- Does your project require code generation support from other products? See “C++ Language Support Limitations” on page 31-4.

For C++ code generation examples with Stateflow, see the `sfcdemo_cppcount` model or `sf_cpp` model.

C++ Language Support Limitations

To use C++ language support, you could need to configure the code generator to use a specific compiler. For example, if a supported compiler is not installed on your Microsoft Windows computer, the default compiler is the `lcc` C compiler shipped with the MATLAB product. This compiler does not support C++. If you do not configure the code generator to use a C++ compiler before you specify C++ for code generation, the software produces an error message.

Code generator limitations on C++ support include:

- The code generator does not support C++ code generation for the following:
 - Simscape Driveline
 - Simscape Multibody First Generation (Simscape Multibody Second Generation is supported)
 - Simscape Electrical Power Systems
 - Simulink Real-Time
- For ERT and ERT-based system target files with **Code interface packaging** set to **Nonreusable function**, the following fields currently do not support the `.cpp` extension.

- **Data definition filename**
- **Definition file** for a data object, which is an object of the `Simulink.Signal` or `Simulink.Parameter` classes or their subclasses.

If you specify a file name with a `.c` extension or with no extension and specify C++ for the code generation language, the code generator produces a `.cpp` file.

Code Generator Assumes Wrap on Signed Integer Overflows

The code generator reduces memory usage and enhances generated code execution by assuming signed integer C operations wrap on overflow. A signed integer overflow occurs when the result of an arithmetic operation is outside the range of values that the output data type can represent. The C programming language does not define the results of such operations. Some C compilers aggressively optimize signed operations for in-range values at the expense of overflow conditions. Other compilers preserve the full wrap-on-overflow behavior. For example, the gcc and MinGW compilers provide an option to wrap on overflow reliably for signed integer overflows. The generated program image for a model can produce results that differ from model simulation results because the handling of overflows varies, depending on your compiler.

When you generate code, if you use a supported compiler with the default options configured by the code generator, the compiler preserves the full wrap-on-overflow behavior. If you change the compiler options or compile the code in another development environment, it is possible that the compiler does not preserve the full wrap-on-overflow behavior. In this case, the executable program can produce unpredictable results.

If this issue is a concern for your application, consider one or more of the following actions:

- Verify that the compiled code produces expected results.
- If your compiler can force wrapping behavior, turn it on. For example, for the gcc compiler or a compiler based on gcc, such as MinGW, configure the build process to use the compiler option `-fwrapv`.
- Choose a compiler that wraps on integer overflow.
- If you have Embedded Coder installed, develop and apply a custom code replacement library to replace code generated for signed integers. For more information, see “Code Replacement Customization” (Embedded Coder).

Choose and Configure Compiler

To view the installed compilers and select the default compiler, in the Command Window, type:

```
mex -setup
```

On a Windows computer, you can install supported compilers and select a default compiler.

On a UNIX platform, the default compiler is GNU `gcc/g++` for GNU or Xcode for Mac.

Unless the build approach configuration selects a specific compiler, the code generator uses the default compiler for the build process.

Primarily, the specified system target file determines the compiler that the code generator requires:

- If you select a toolchain-based system target file such as `grt.tlc` (Generic Real-Time Target), `ert.tlc` (Embedded Coder), or `autosar.tlc` (Embedded Coder for AUTOSAR), the **Build process** subpane displays configuration parameters for the build process. Use the **Toolchain** configuration parameter to select a compiler and associated tools for your model build. To validate the selected toolchain, click the **Validate Toolchain**.
- If you select a template makefile (TMF) based system target file, such as `rsim.tlc`, the **Build process** subpane displays template makefile parameters for configuring the build process. The **Template makefile** parameter displays the default TMF file for the selected system target file. If the system target file supports compiler-specific template makefiles (for example, Rapid Simulation or S-Function system target files), you can set **Template makefile** to a compiler-specific TMF. For valid TMF names, see “Compare System Target File Support Across Products” on page 21-17.

If the generated code exceeds the 2 Gb static memory limit, a compiler error occurs. To resolve this issue:

- For GRT-based system target files, set model configuration parameter **Code interface packaging** to `Reusable function`.
- For ERT-based system target files, set model configuration parameter **Code interface packaging** to `Reusable function` and select parameter **Use dynamic memory allocation for model initialization**.

Include S-Function Source Code

When the code generator builds models with S-functions, source code for the S-functions can be either in the current folder or in the same folder as their MEX-file. The code generator adds an include path to the generated makefiles whenever it finds a file named `sfcnname.h` in the same folder as the S-function MEX-file. This folder must be on the MATLAB path.

Similarly, the code generator adds a rule for the folder when it finds a file `sfcnname.c` (or `.cpp`) in the same folder as the S-function MEX-file is in.

See Also

More About

- “Configure Run-Time Environment Options” on page 11-2
- “Compare System Target File Support Across Products” on page 21-17
- “Select C or C++ Programming Language” on page 31-2
- “Troubleshoot Compiler Issues” on page 31-7

External Websites

- https://www.mathworks.com/support/compilers/current_release

Troubleshoot Compiler Issues

In this section...

“Compiler Version Mismatch Errors” on page 31-7

“Results for Model Simulation and Program Execution Differ” on page 31-7

“Generates Expected Code and Produces Unexpected Results” on page 31-8

“Compile-Time Issues” on page 31-8

“LCC Compiler Does Not Support Ampersands in Source Folder Paths” on page 31-9

“LCC Compiler Might Not Support Line Lengths of Rapid Accelerator Code” on page 31-9

Compiler Version Mismatch Errors

Description

The build process produces a compiler version mismatch error.

Action

- 1 Check the list of supported and compatible compilers available at www.mathworks.com/support/compilers/current_release/.
- 2 Upgrade or change your compiler. For more information, see “Choose and Configure Compiler” on page 31-5.
- 3 Rebuild the model.

Results for Model Simulation and Program Execution Differ

Description

The program generated for the model produces different results from model simulation results. The generated source code includes an arithmetic operation that produces a signed integer overflow. It is possible that your compiler does not implement wrapping behavior for signed integer overflow conditions. Or, if you are using a compiler that supports wrapping, it is possible that you did not configure it to use the `-fwrapv` option.

For more information, see “Code Generator Relies on Undefined Behavior of C Language for Integer Overflows.”

Action

- If your compiler can force wrapping behavior, turn it on. For example, for the gcc compiler or a compiler based on gcc, such as MinGW, specify the compiler option `-fwrapv`.
- Choose a compiler that checks for integer overflows.
- If you have Embedded Coder, develop and apply a code replacement library to replace code generated for signed integers.

Generates Expected Code and Produces Unexpected Results

Description

The build process generates expected source code, but the executable program produces unexpected results. The generated source code appears as expected. However, the executable program produces unexpected results.

Action

Do one of the following:

- Lower the compiler optimization level.
 - 1 Set the **Compiler optimization level** configuration parameter to Custom.
 - 2 In the **Custom compiler optimization flags** field, specify a lower optimization level.
 - 3 Rebuild the model.
- Disable compiler optimizations.
 - 1 Set **Compiler optimization level** to Optimizations off (faster builds).
 - 2 Rebuild the model.

For more information, see “Control Compiler Optimizations” on page 40-6 and your compiler documentation.

Compile-Time Issues

Issue	Action
Error is present in the compiler configuration.	Make sure that MATLAB supports the compiler and version that you want to use. For a list of currently supported and compatible compilers, see www.mathworks.com/support/compilers/current_release/ . If necessary, upgrade or change your compiler (see “Choose and Configure Compiler” on page 31-5 or “Choose and Configure Compiler” on page 31-5).
Environment variables are incorrectly set up for your make utility, compiler, or linker. For example, installation of Cygwin tools on a Windows platform affects environment variables used by other compilers.	Review the environment variable settings for your system by using the <code>set</code> command on a Windows platform or <code>setenv</code> on a Linux platform. Make sure that the settings match what is required for the tools you are using.
Error is present in custom code specified as an S-Function block or in Custom Code . For example, the code refers to a header file that the compiler cannot find.	To isolate the source of the problem, remove the custom code from the model, debug, and rebuild the model.
The model includes a block, such as a device driver block, which is not intended for use with the currently selected system target file.	Remove the system target file-specific block or configure the model for use with another system target file.

Issue	Action
<p>A linker error about an undefined reference to the data appears when the model build generates an executable program from the model reference hierarchy and these conditions are true:</p> <ul style="list-style-type: none"> • You represent signal, state, or parameter data by creating a data object such as <code>Simulink.Signal</code>. You use the object in a model reference hierarchy. • You use a custom storage class with the data object. Custom storage classes require Embedded Coder. • You set the owner of the object to a model that does not directly access the data. • You use the toolchain <code>lcc-win64</code>. 	<p>To resolve the issue, choose one of these methods:</p> <ul style="list-style-type: none"> • In the data object, clear the <code>Owner</code> property. Alternatively, set the owner to a model that directly accesses the data. • Use a different toolchain, such as <code>gcc</code>, instead of <code>lcc</code>.

LCC Compiler Does Not Support Ampersands in Source Folder Paths

Description

If you use the LCC compiler and your model folder path contains an ampersand (&), the build process produces an error.

Action

Remove the ampersand from the model folder path. Then, rebuild the model.

LCC Compiler Might Not Support Line Lengths of Rapid Accelerator Code

Description

If you are compiling Rapid Accelerator code, the LCC compiler might produce an error related to line limits. Rapid Accelerator code can have longer line lengths due to obfuscation.

Action

Compile your Rapid Accelerator code using a compiler that supports longer code lines.

See Also

More About

- “Choose and Configure Compiler” on page 31-5
- “Select C or C++ Programming Language” on page 31-2
- “Approaches for Building Code Generated from Simulink Models” on page 31-11
- “Build Process Workflow for Real-Time Systems” on page 31-39

- “Rebuild a Model” on page 31-48
- “Reduce Build Time for Referenced Models by Using Parallel Builds” on page 31-51
- “Control Regeneration of Top Model Code” on page 31-49
- “Relocate or Share Generated Code” on page 31-56
- “Control Compiler Optimizations” on page 40-6
- “Select and Configure C or C++ Compiler” on page 31-3
- “Executable Program Generation” on page 31-62

External Websites

- www.mathworks.com/support/compilers/current_release/

Approaches for Building Code Generated from Simulink Models

Use one of these approaches for building (compiling and linking) code that you generate from Simulink models:

- CMake — CMake is a third-party, open-source tool for build process management, which uses configuration (CMakeLists.txt) files to generate standard build files for native build environments, for example, makefiles, Ninja files, or Microsoft Visual Studio and Xcode projects. You can:
 - Specify a generic CMake toolchain -- see “Deploy Component Algorithm as Component Model Library by Using CMake” (Embedded Coder)
 - Specify a shipped CMake toolchain definition -- see “Configure CMake Build Process” on page 31-13
 - Provide a custom CMake toolchain definition by using the `target` package -- see “Create Custom CMake Toolchain Definition” on page 31-17.

MATLAB ships the CMake executable file. Simulink Coder and Embedded Coder support CMake versions from 3.12.0 onwards.

- Makefile-based toolchain — The build process generates makefiles and supports custom toolchains. You can define and register custom makefile-based toolchains by using the Target Framework or `ToolchainInfo` objects. See:
 - “Define Custom Makefile-Based Toolchains Using Target Framework” on page 31-30
 - “Register Custom Toolchain and Build Executable” (Embedded Coder)
- Template makefile — The build process uses a template makefile with a toolchain that you specify. Optionally, you can control the build process through a toolchain information object. See “Template Makefile Approach” on page 31-24.

Each build approach supports:

- Simulink model configuration parameters.
- `slbuild`, **Ctrl+B**, and `codebuild`
- “SIL and PIL Simulations” (Embedded Coder)
- “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” (Embedded Coder)

For information about customizing a build approach, see “Code Compilation Customization”.

See Also

`slbuild` | `codebuild`

Related Examples

- “Deploy Component Algorithm as Component Model Library by Using CMake” (Embedded Coder)
- “Configure CMake Build Process” on page 31-13
- “Create Custom CMake Toolchain Definition” on page 31-17
- “Define Custom Makefile-Based Toolchains Using Target Framework” on page 31-30

- “Configure Toolchain (ToolchainInfo) or Template Makefile Build Process” on page 31-20
- “Compile Code in Another Development Environment” on page 24-93
- “Build Library or Executable from AUTOSAR Adaptive Model” (AUTOSAR Blockset)

External Websites

- <https://cmake.org/>

Configure CMake Build Process

CMake is a third-party, open-source tool for build process management. To build code generated from Simulink models, the software provides CMake toolchain definitions for:

- Microsoft Visual C++ and MinGW on Windows, GCC on Linux, and Xcode on Mac computers, using Ninja and makefile generators.
- Microsoft Visual Studio and Xcode project builds.

If a supported toolchain is installed on your development computer, you can specify the corresponding CMake toolchain definition for your model. When you run `slbuild`, press **Ctrl+B**, or run a software-in-the-loop (SIL), processor-in-the-loop (PIL), or external mode simulation, CMake:

- 1 Uses configuration (`CMakeLists.txt`) files to generate standard build files.
- 2 Runs the compiler and other build tools to create executable code.

You can also:

- Provide custom CMake toolchain definitions by using the `target` package. For more information, see “Create Custom CMake Toolchain Definition” on page 31-17.
- Generate only code and a configuration file for algorithm export.

Specify CMake Toolchain Definition

In the Configuration Parameters dialog box:

- 1 On the Hardware Implementation pane, specify parameter values for your target device, which determines the selection of toolchain definitions that is available. For more information, see “Available CMake Toolchain Definitions” on page 31-14.
- 2 On the Code Generation pane, specify these parameters:
 - **Toolchain** — From the drop-down list, select a CMake toolchain definition.
 - **Build configuration** — Select one of these values:
 - `Release` — Optimizations enabled.
 - `Debug` — Optimizations disabled, debug symbols generated.
 - `RelWithDebInfo` — Optimizations enabled, debug symbols generated.
 - `MinSizeRel` — Optimized for size, debug symbols not generated.
 - `Specify` — Enables you to specify toolchain details through the **CMake Configure** and **CMake Build** fields.
 - **Toolchain details** — If **Build configuration** is `Specify`, populate these fields:
 - **CMake Configure** — Additional command-line arguments to pass to CMake when generating the project build system builder artifacts for the downstream tools.
 - **CMake Build** — Additional command-line arguments to pass to CMake when using it in build mode.

For more information, see <https://cmake.org/cmake/help/latest/manual/cmake.1.html>.

Available CMake Toolchain Definitions

Platform	Shipped Toolchain Definition	Device Vendor and Type that Make Definition Available	
Windows	Microsoft Visual C++ 2017 v15.0 CMake/nmake (64-bit Windows)	Intel-x86-64 (Windows64)	
	Microsoft Visual C++ 2017 v15.0 CMake/Ninja (64-bit Windows)	AMD-x86-64 (Windows64)	
	Microsoft Visual Studio Project 2017 CMake (64-bit Windows)		
	Microsoft Visual Studio Project 2019 CMake (64-bit Windows)		
	Microsoft Visual C++ 2019 v16.0 CMake/Ninja (64-bit Windows)		
	Microsoft Visual C++ 2022 v17.0 CMake/nmake (64-bit Windows)		
	Microsoft Visual C++ 2022 v17.0 CMake/Ninja (64-bit Windows)		
	MinGW64 CMake/gmake (64-bit Windows)		
	MinGW64 CMake/Ninja (64-bit Windows)		
	Microsoft Visual C++ 2017 v15.0 CMake/nmake (32-bit Windows)	Intel-x86-32 (Windows32)	
	Microsoft Visual C++ 2017 v15.0 CMake/Ninja (32-bit Windows)	AMD-x86-32 (Windows32)	
	Microsoft Visual Studio Project 2017 CMake (32-bit Windows)		
	Microsoft Visual Studio Project 2019 CMake (32-bit Windows)		
	Microsoft Visual C++ 2019 v16.0 CMake/nmake (32-bit Windows)		
	Microsoft Visual C++ 2019 v16.0 CMake/Ninja (32-bit Windows)		
	Microsoft Visual C++ 2022 v17.0 CMake/nmake (32-bit Windows)		
	Microsoft Visual C++ 2022 v17.0 CMake/Ninja (32-bit Windows)		
	Linux	GNU gcc/g++ CMake/gmake (64-bit Linux)	Intel-x86-64 (Linux 64)
		GNU gcc/g++ CMake/Ninja (64-bit Linux)	AMD-x86-64 (Linux 64)

Platform	Shipped Toolchain Definition	Device Vendor and Type that Make Definition Available
Mac	Xcode with Clang CMake/gmake (64-bit Mac)	Intel-x86-64 (Mac OS X)
	Xcode with Clang CMake/Ninja (64-bit Mac)	AMD-x86-64 (Mac OS X)
	Xcode with Clang Project CMake (64-bit Mac)	

Note If the **Test hardware is the same as production hardware** check box is not selected, the **Test hardware** settings for device vendor and type control the availability of shipped toolchain definitions.

Generate Only Code and CMake Configuration File for Algorithm Export

To export generated code to another development environment, use this workflow:

- 1 In the Configuration Parameters dialog box, from the **Toolchain** drop-down list, select CMake. For this option, the code generator:
 - Generates C and C++ source code from the Simulink model but does not build the code.
 - Creates a CMake configuration (CMakeLists.txt) file that does not depend on specific build tools.
- 2 Select the **Package code and artifacts** check box.
- 3 Generate code, which produces a ZIP file that contains the source code and CMake configuration file.
- 4 Relocate the ZIP file to your development environment.
- 5 Unzip the packaged files.
- 6 Use the CMake build tool and configuration file to create an executable file.

If the ZIP file contains `mLrFiles.zip`, static dependencies from your `matlabroot` folder tree, you must specify the location of the unzipped files for the CMake build tool. In your development environment, set the `MATLAB_ROOT` CMake cache entry by running this command:

```
cmake -S pathToSource -B pathToProposedLocationOfDerivedFiles
-DMATLAB_ROOT=pathToUnzippedmLrFiles
```

For more information, see “Deploy Component Algorithm as Component Model Library by Using CMake” (Embedded Coder).

See Also

`slbuild` | `packNGo` | `codebuild`

Related Examples

- “Create Custom CMake Toolchain Definition” on page 31-17

- “Deploy Component Algorithm as Component Model Library by Using CMake” (Embedded Coder)
- “Relocate or Share Generated Code” on page 31-56
- “Compile Code in Another Development Environment” on page 24-93
- “SIL and PIL Simulations” (Embedded Coder)
- “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” on page 32-2

External Websites

- <https://www.mathworks.com/support/requirements/supported-compilers.html>
- <https://cmake.org/cmake/help/latest/index.html>

Create Custom CMake Toolchain Definition

CMake is a third-party, open-source tool for build process management. To build code generated from Simulink models, the software provides CMake toolchain definitions that you can select from the **Toolchain** drop-down list. You can also create custom CMake toolchain definitions by using the `target` package. When you add a custom CMake toolchain definition to an internal database, the toolchain is available for building generated code. This example shows how you can create a CMake toolchain definition using the Ninja generator. You can use the toolchain definition on computers with these operating systems:

- Windows (provided MATLAB Support for MinGW-w64 C/C++ Compiler is installed).
- Linux
- Mac

Create a `target.Toolchain` object.

```
tc = target.create('Toolchain', 'Name', 'Example Custom CMake Toolchain');
```

Use the `Builder` property and `target.CMakeBuilder` object to configure how CMake builds generated code.

```
tc.Builder = target.create('CMakeBuilder');
tc.Builder.Generator = 'Ninja';
tc.Builder.ToolchainFile = fullfile(pwd, 'ExampleCMakeToolchain.cmake');
tc.Builder.SupportedBuildTypes(end+1) = ...
    target.create('CMakeBuildType', ...
        'Name', 'FastMath', ...
        'GeneratesDebugSymbols', false, ...
        'DebugBuildType', 'FastMathWithDebug');
tc.Builder.SupportedBuildTypes(end+1) = ...
    target.create('CMakeBuildType', ...
        'Name', 'FastMathWithDebug', ...
        'GeneratesDebugSymbols', true);
```

Using an editor, create `ExampleCMakeToolchain.cmake`, a file that contains these lines.

```
# Example CMake toolchain

# Configure to use GCC
if(WIN32)
    list(APPEND CMAKE_PROGRAM_PATH "$ENV{MW_MINGW64_LOC}")
endif()
set(CMAKE_C_COMPILER "gcc")
set(CMAKE_CXX_COMPILER "g++")

# Add some custom build configurations: FastMath, and FastMathWithDebug
set(CMAKE_C_FLAGS_FASTMATH "-Ofast")
set(CMAKE_CXX_FLAGS_FASTMATH "-Ofast")
set(CMAKE_C_FLAGS_FASTMATHWITHDEBUG "-Ofast -g")
set(CMAKE_CXX_FLAGS_FASTMATHWITHDEBUG "-Ofast -g")
```

For information about CMake toolchain files, see <https://cmake.org/cmake/help/book/mastering-cmake/chapter/Cross%20Compiling%20With%20CMake.html>.

Use the first element of the `EnvironmentConfiguration` property to provide system environment setup commands and paths for the Windows operating system.

```
tc.EnvironmentConfiguration(1).HostOperatingSystemSupport.Linux = false;
tc.EnvironmentConfiguration(1).HostOperatingSystemSupport.Windows = true;
tc.EnvironmentConfiguration(1).HostOperatingSystemSupport.Mac = false;
tc.EnvironmentConfiguration(1).SystemPaths = ...
    {'$(MW_MINGW64_LOC)/bin', '$(MATLAB_ROOT)/toolbox/shared/coder/ninja/$(ARCH)'};
```

Use the second element of the `EnvironmentConfiguration` property to provide system environment setup commands and paths for Linux and Mac operating systems.

```
tc.EnvironmentConfiguration(2) = target.create('EnvironmentConfiguration');
tc.EnvironmentConfiguration(2).HostOperatingSystemSupport.Linux = true;
tc.EnvironmentConfiguration(2).HostOperatingSystemSupport.Windows = false;
tc.EnvironmentConfiguration(2).HostOperatingSystemSupport.Mac = true;
tc.EnvironmentConfiguration(2).SystemPaths = ...
    {'$(MATLAB_ROOT)/toolbox/shared/coder/ninja/$(ARCH)'};
```

By default, the software uses the CMake executable file that ships with MATLAB. To specify another version of CMake, you can create a `target.CMake` object that provides the required path.

Optionally, associate the toolchain definition with your target devices, for example:

```
windowsProc = target.get('Processor', 'Intel-x86-64 (Windows64)');
tc.SupportedHardware(end+1) = ...
    target.create('HardwareComponentSupport', 'Component', windowsProc);

linuxProc = target.get('Processor', 'Intel-x86-64 (Linux 64)');
tc.SupportedHardware(end+1) = ...
    target.create('HardwareComponentSupport', 'Component', linuxProc);

macProc = target.get('Processor', 'Intel-x86-64 (Mac OS X)');
tc.SupportedHardware(end+1) = ...
    target.create('HardwareComponentSupport', 'Component', macProc);
```

Add the toolchain definition to the internal database:

```
target.add(tc);
```

To use the custom toolchain definition for building generated code, in the Configuration Parameters dialog box:

- 1 On the **Hardware Implementation** pane, select your target device by setting **Device vendor** to Intel and **Device type** to x86-64 (Windows64).
- 2 On the **Code Generation** pane, from the **Toolchain** list, select Example Custom CMake Toolchain. The **Build configuration** list contains these values:
 - Release
 - Debug
 - RelWithDebInfo
 - MinSizeRel
 - FastMath
 - FastMathWithDebug
 - Specify
- 3 Under **Advanced parameters**, click **Validate Toolchain**, which generates a validation report.

Validation Report for: Example Custom CMake Toolchain

Toolchain Validation Result: Passed

Validation report:

Validating definition of toolchain "Example Custom CMake Toolchain":

Passed.

Validating that the location of the CMake tool is uniquely defined for your development computer:

Passed.

Validation of build tool "CMake"

Configuring test CMakeLists.txt file:

Passed.

Validating that CMake generator "Ninja" is a single-configuration generator:

Passed.

When you run, for example, the `slbuild` function or a software-in-the-loop (SIL) simulation, the build process uses the custom toolchain to build generated code.

If you want to remove the custom toolchain definition from the internal database, run:

```
customToolChainDef = target.get('Toolchain', 'Example Custom CMake Toolchain');
target.remove(customToolChainDef);
```

`target.remove` summary:

```
Objects removed from internal database:
  target.Toolchain    "Example Custom CMake Toolchain"
```

See Also

`target.create` | `target.Toolchain` | `target.CMake` | `target.CMakeBuilder` |
`target.CMakeBuildType` | `target.CMakeCacheEntry` | `target.EnvironmentConfiguration`
| `target.HardwareComponentSupport`

Related Examples

- "Approaches for Building Code Generated from Simulink Models" on page 31-11
- "Configure CMake Build Process" on page 31-13

External Websites

- <https://cmake.org/cmake/help/book/mastering-cmake/chapter/Cross%20Compiling%20With%20CMake.html>

Configure Toolchain (ToolchainInfo) or Template Makefile Build Process

Specify the toolchain (ToolchainInfo) or template makefile approach for building code generated from a model.

If you set the System target file configuration parameter to `grt.tlcert.tlc`, `ert_shrlib.tlc`, or another toolchain-compliant system target file, the build process uses the toolchain approach on page 31-20. If you set the parameter to a non-toolchain-compliant system target file, the build process uses the template makefile approach (Embedded Coder).

You can switch from the toolchain approach to the template makefile approach by using this command:

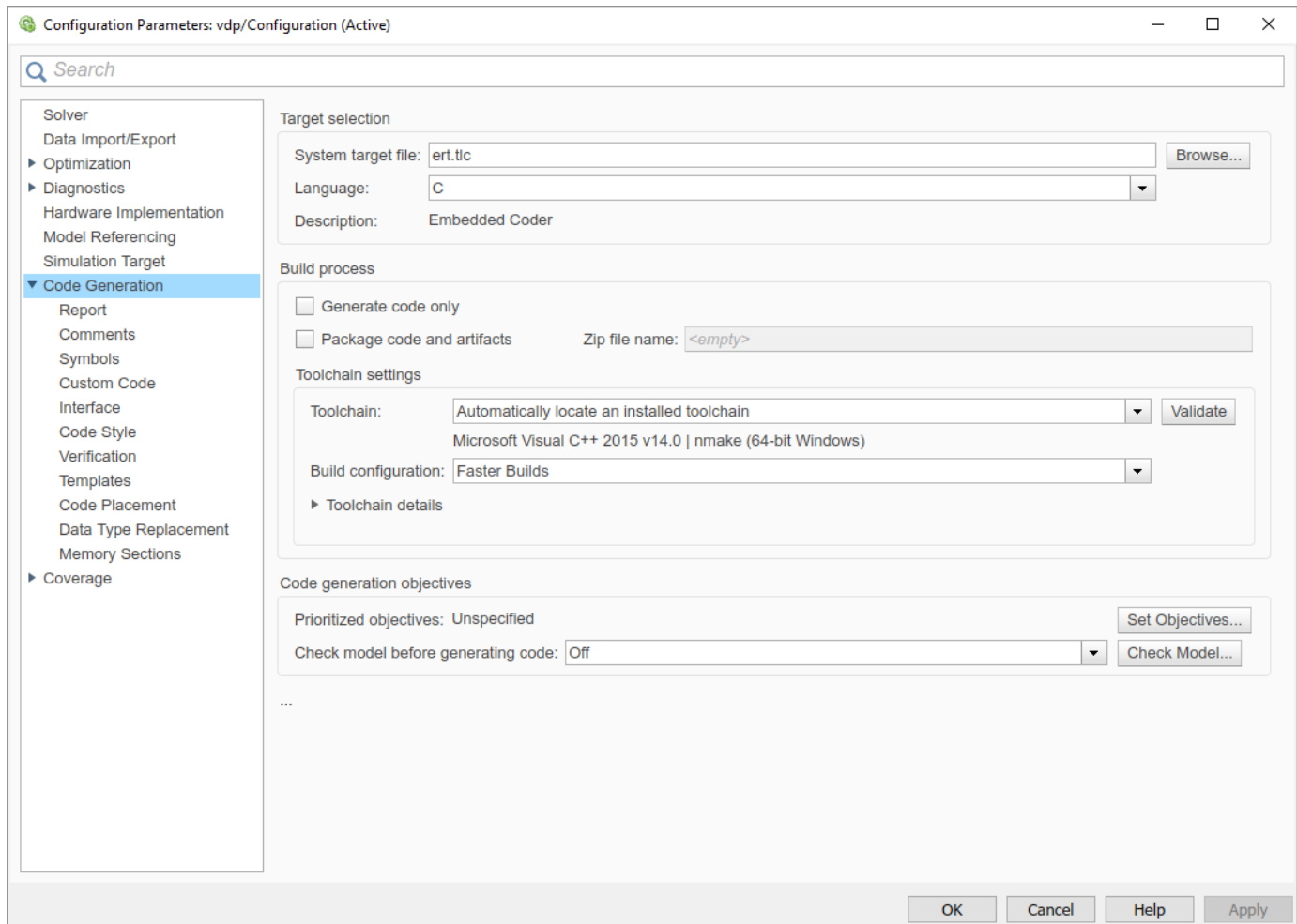
```
set_param(model, 'MakeCommand', 'make_rtw TMF=1')
```

Toolchain Approach

Toolchain settings appear under **Build process** when you set **System target file** to:

- `grt.tlc` – Generic Real-Time Target
- `ert.tlc` – Embedded Coder
- `ert_shrlib.tlc` – Embedded Coder (host-based shared library target)
- A toolchain-compliant system target file

For more information about toolchain-compliant system target files, see “Support Toolchain Approach with Custom Target” on page 46-59.



The **Toolchain settings** include these configuration parameters:

- **Toolchain** -- Specifies the collection of third-party software tools that builds the generated code. A toolchain can include a compiler, linker, and archiver, and other prebuild or postbuild tools.

The default value of **Toolchain** is **Automatically locate an installed toolchain**. The **Toolchain** parameter displays name of the located toolchain just below **Automatically locate an installed toolchain**.

To check that the toolchain is present and validate that the code generator has the information required to use the toolchain, in the **Advanced Parameters** section, click **Validate Toolchain**. The resulting Validation Report gives a pass/fail value for the selected toolchain, and shows issues that require resolution.

- **Build configuration** -- Specifies the optimization settings. By default, **Build Configuration** is set to **Faster Builds**. You can also select **Faster Runs**, **Debug**, and **Specify**. If you select **Specify**, you can customize the options for the toolchain. When you click **Apply**, the custom toolchain settings apply to the current model.

Note The following system target files, which use the template makefile approach, have the same names but different descriptions from system target files that use the toolchain approach:

- `ert.tlc` – Create Visual C/C++ Solution File for Embedded Coder
- `grt.tlc` – Create Visual C/C++ Solution File for Simulink Coder

To avoid confusion, click **Browse** to select the system target file and look at the description of each file.

Upgrade Model to Use Toolchain Approach

When you open a model created before R2013b that uses the following system target files, the software tries to upgrade the model. The upgrade changes the configuration from using template makefile settings to using the toolchain settings:

- `ert.tlc` – Embedded Coder
- `ert_shrlib.tlc` – Embedded Coder (host-based shared library target)
- `grt.tlc` – Generic Real-Time Target

Note To upgrade models using a custom system target file to use the toolchain approach, see “Support Toolchain Approach with Custom Target” on page 46-59.

Some model configuration parameter values prevent the software from upgrading a model to use toolchain settings. The following instructions show you ways to complete the upgrade process.

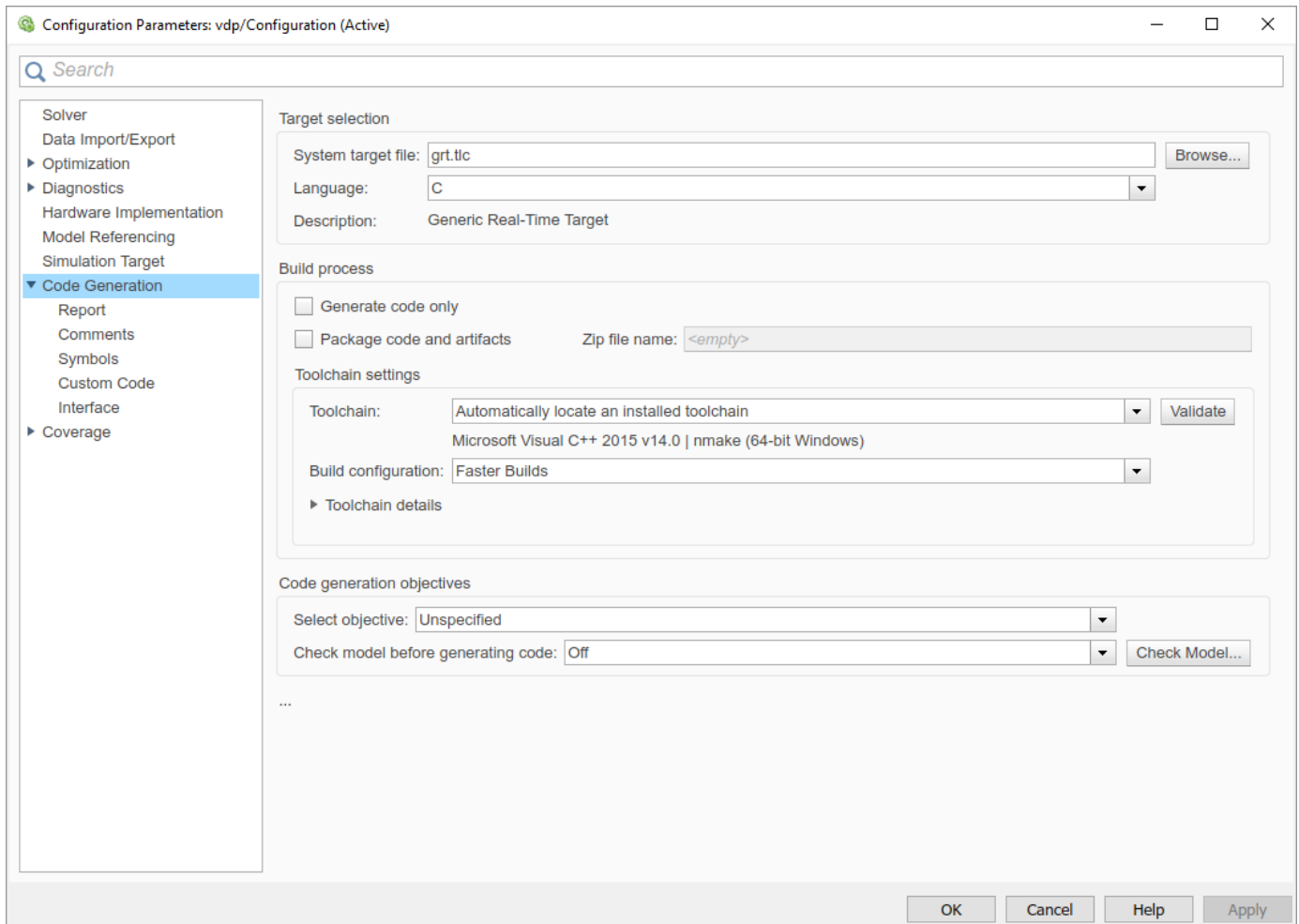
Consider upgrading your models and use the toolchain build approach. Doing so is not required. You can continue generating code from a model that has not been upgraded.

Note The software does not upgrade models that use the following system target files:

- `ert.tlc` – Create Visual C/C++ Solution File for Embedded Coder
 - `grt.tlc` – Create Visual C/C++ Solution File for Simulink Coder
-

To see if a model has been upgraded:

- 1 If the **Toolchain** and **Build configuration** configuration parameters are specified, the model has been upgraded.



If makefile configuration parameters such as **Generate makefile**, **Make command**, and **Template makefile** are specified, the model has not been upgraded.

Start by creating a working copy of the model using **File > Save As**. This action preserves the original model and configuration parameters for reference.

Try to upgrade the model using Upgrade Advisor:

- 1 On the **Modeling** tab, in the **Evaluate & Manage** section, select **Upgrade Advisor**.
- 2 In Upgrade Advisor, select **Check and update model to use toolchain approach to build generated code** and click **Run This Check**.
- 3 Perform the suggested actions and/or click **Update Model**.

When you cannot upgrade the model using Upgrade Advisor, one or more of the following parameters is not set to its default value, shown here:

- **Compiler optimization level** — Optimizations off (faster builds)
- **Generate makefile** — Enabled
- **Template makefile** — System target file-specific template makefile

- **Make command** — `make_rtw` without arguments

Sometimes, a model cannot be upgraded. Try the following procedure:

- If **Generate makefile** is disabled, this case cannot be upgradable. However, you can try enabling it and try upgrading the model using Upgrade Advisor.
- If **Compiler optimization level** is set to `Optimizations on (faster runs)`:
 - 1 Set **Compiler optimization level** is to `Optimizations off (faster builds)`.
 - 2 Upgrade the model using Upgrade Advisor.
 - 3 Set **Build configuration** to `Faster Runs`.
- If **Compiler optimization level** is set to `Custom`:
 - 1 Copy the **Custom compiler optimization flags** to a text file.
 - 2 Set **Compiler optimization level** to `Optimizations off (faster builds)`.
 - 3 Upgrade the model using Upgrade Advisor.
 - 4 Set **Build configuration** to `Specify`.
 - 5 To perform the same optimizations, edit the compiler options.
- If **Template makefile** uses a customized template makefile, this case cannot be upgradable. However, you can try the following:
 - 1 Update **Template makefile** to use the default makefile for the system target file.

Note To get the default makefile name, change the **System target file**, click **Apply**, change it back, and click **Apply** again.

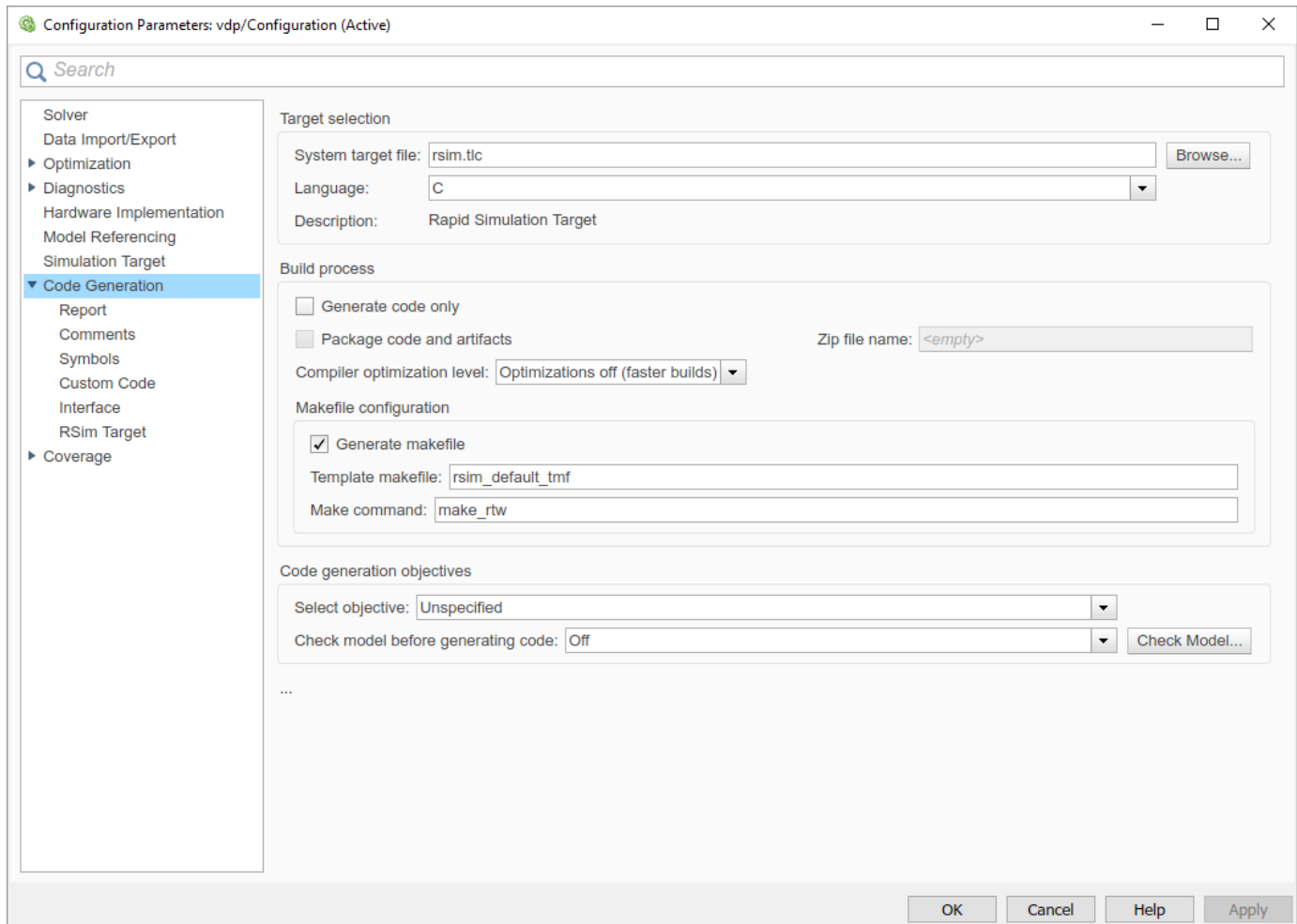
- 2 Upgrade the model using Upgrade Advisor.
- 3 If the template makefile contains build tool options, such as compiler optimization flags, set **Build configuration** to `Specify` and update the options.
- 4 If the template makefile uses custom build tools, create and register a custom toolchain, as described in “Custom Toolchain Registration” . Then, set the **Toolchain** parameter to use the custom toolchain.

Note After registering the custom toolchain, update **Toolchain** to use the custom toolchain.

- 5 If the template makefile contains custom rules and logic, these customizations cannot be applied to the upgraded model.

Template Makefile Approach

When the **System target file** is set to a `tlc` file that uses the template makefile approach, the software displays **Compiler optimization level**, **Generate makefile**, **Make command**, and **Template makefile** parameters.



Specify Whether to Generate a Makefile

The **Generate makefile** option specifies whether the build process is to generate a makefile for a model. By default, the build process generates a makefile. Suppress generation of a makefile, for example in support of custom build processing that is not based on makefiles, by clearing **Generate makefile**. When you clear this parameter:

- The **Make command** and **Template makefile** options are unavailable.
- Set up post code generation build processing using a user-defined command, as explained in “Customize Post-Code-Generation Build Processing” on page 44-7.

Specify a Make Command

Each template makefile-based system target file has an associated **make** command. The code generator uses this internal MATLAB command to control the build process. The command appears in the **Make command** field and runs when you start a build.

Most system target files use the default command, `make_rtw`. Third-party system target files could supply another make command. See the documentation from the vendor.

In addition to the name of the **make** command, you can supply makefile options in the **Make command** field. These options could include compiler-specific options, include paths, and other

parameters. When the build process invokes the `make` utility, these options are passed on the `make` command line, which adds them to the overall flags passed to the compiler.

“Template Makefiles and Make Options” on page 31-36 lists the **Make command** options you can use with each supported compiler.

Specify the Template Makefile

The **Template makefile** field has these functions:

- If you selected a system target file with the System Target File Browser, this field displays the name of a MATLAB language file that selects a template makefile for your development environment. For example, in “Model Configuration Parameters: Code Generation”, the **Template makefile** field displays `grt_default_tmf`, indicating that the build process invokes `grt_default_tmf.m`.

“Template Makefiles and Make Options” on page 31-36 gives a detailed description of the logic by which the build process selects a template makefile.

- Alternatively, you can explicitly enter the name of a specific template makefile (including the extension) or a MATLAB language file that returns a template makefile in this field. Use this approach if you are using a system target file that does not appear in the System Target File Browser. For example, use this approach if you have written your own template makefile for a custom system target file.

If you specify your own template makefile, be sure to include the file name extension. If you omit the extension, the build process attempts to find and execute a file with the extension `.m` (that is, a MATLAB language file). The template make file (or a MATLAB language file that returns a template make file) must be on the MATLAB path. To determine whether the file is on the MATLAB path, enter the following command in the MATLAB Command Window:

```
which tmf_filename
```

Associate the Template Makefile with a Toolchain

Specify a toolchain for the template makefile build process. This example provides a definition for a minimal toolchain and shows how you can associate the toolchain with a template makefile that is a copy of `ert_unix.tmf` or `ert_vcx64.tmf`.

- 1 Use a copy of `ert_unix.tmf` or `ert_vcx64.tmf` as the template makefile.

```
copyfile(fullfile(matlabroot, 'toolbox', 'coder', ...
    'compile', 'tmf', 'ert_vcx64.tmf'), ...
    'ert_copy.tmf')
```

- 2 In the template makefile, i.e. `ert_copy.tmf`, edit the `TOOLCHAIN_NAME` macro.

```
TOOLCHAIN_NAME = "Minimal Toolchain for TMF Build"
```

Note The `ert_unix.tmf` and `ert_vcx64.tmf` files from the current release are simplified compared to the corresponding files from releases before R2018a. The software determines whether your template makefile uses the simplified approach by checking for the presence of the variable `TOOLCHAIN_NAME`. A template makefile that is based on `ert_unix.tmf` and `ert_vcx64.tmf` files from releases before R2018a does not specify the `TOOLCHAIN_NAME` variable.

Through the `|>MODEL_MODULES<|` token, the simplified approach specifies the main, *model*, and other source files that require compiling. The template makefile that is based on `ert_unix.tmf` and `ert_vcx64.tmf` files from releases before R2018a explicitly adds the main and *model* source files to the list of source files that require compiling.

To update a template makefile that originates from a release before R2018a:

- a Remove references to `ert_main.c`, `classic_main.c` or `rt_main.c`. For example, literal references like `ert_main.c` or tokenized references like `ert_main.$(TARGET_LANG_EXT)`.
- b Remove references to `$(MODEL).c` or `$(MODEL).$(TARGET_LANG_EXT)`.

3 Associate the toolchain with the template make file.

- a Create a toolchain definition file. For this example, use this code:

Toolchain Definition File

```
function tc = minimalToolchainForTMF
% Create a toolchain object
tc = coder.make.ToolchainInfo('Name', 'Minimal Toolchain for TMF Build');

% Specify that the linker requires a response file instead of
% including all object file modules on the linker command line
tc.addAttribute('RequiresCommandFile', true);

if ispc
    objExt = '.obj';
else
    objExt = '.o';
end

% Specify source file and object file extension used by the C compiler
tool = tc.getBuildTool('C Compiler');
tool.setFileExtension('Source', '.c');
tool.setFileExtension('Header', '.h');
tool.setFileExtension('Object', objExt);

% Specify source file and object file extension used by the C++ compiler
tool = tc.getBuildTool('C++ Compiler');
tool.setFileExtension('Source', '.cpp');
tool.setFileExtension('Header', '.hpp');
tool.setFileExtension('Object', objExt);
```

- b Run the toolchain definition file, which generates a `ToolchainInfo` object, `tc`.

```
tc = minimalToolchainForTMF;
```

- c Save the `ToolchainInfo` object to a MAT file.

```
save('tcMinimal', 'tc')
```

- d Register the toolchain in `RTW.TargetRegistry`.

- i Place this code in an `rtwTargetInfo.m` file.

```
function rtwTargetInfo(tr)
tr.registerTargetInfo(@loc_createToolchain);
end

function config = loc_createToolchain
config(1) = coder.make.ToolchainInfoRegistry;
config(1).Name = 'Minimal Toolchain for TMF Build';
config(1).FileName = fullfile(fileparts(mfilename('fullpath'))),...
```

```

        'tcMinimal.mat');
    config(1).TargetHWDeviceType = {'*'};
    config(1).Platform = {computer('arch')};
end

```

- ii Save the file in the current working folder or in a folder that is on the MATLAB search path.
- iii Reset the TargetRegistry.

```

    RTW.TargetRegistry.getInstance('reset');

```

- 4 Associate the template makefile with your model and build the model.

```

set_param(model, 'TemplateMakefile', 'ert_copy.tmf');
slbuild(model);

```

Specify TLC for Code Generation

Target Language Compiler (TLC) is an integral part of the code generator. It enables you to customize generated code. Through customization, you can produce platform-specific code, or you can incorporate your own algorithmic changes for execution speed, code size, or compatibility with existing methods that you prefer to maintain. For additional information, see “Target Language Compiler Overview”.

TLC options that you specify for code generation appear in the summary section of the generated HTML code generation report.

Note Specifying TLC command-line options does not add flags to the make command line.

You can specify Target Language Compiler (TLC) command-line options and arguments for code generation using the model parameter `TLCOptions` in a `set_param` function call. For example,

```
>> set_param(gcs, 'TLCOptions', '-p0 -aWarnNonSaturatedBlocks=0')
```

Some common uses of TLC options include the following:

- `-aVarName=1` to declare a TLC variable and/or assign a value to it
- `-IC:\Work` to specify an include path
- `-v` to obtain verbose output from TLC processing (for example, when debugging)

Use of Checksums in Recompilation of Generated Code

When you build a model with `GenerateMakefile` set to 'on' and `GenCodeOnly` set to 'off', the software generates a makefile and runs the makefile to compile the generated code. When the compilation is complete, the software creates and saves checksums for source files and configuration data.

If checksums from a previous build are available when you build the model:

- 1 The software uses the checksums to determine whether the source files and configuration data are different from the previous build.

2 If the software detects a difference, the software:

- Regenerates the makefile.
- Recomiles the generated code.

If the build process uses a template makefile, you can disable the use of checksums by adding this text to the template makefile:

```
DO_NOT_USE_CHECKSUMS=1
```

In this case, the software uses file timestamps to determine whether it should recompile the generated code.

See Also

More About

- “Approaches for Building Code Generated from Simulink Models” on page 31-11
- “Support Toolchain Approach with Custom Target” on page 46-59
- “Build Process Workflow for Real-Time Systems” on page 31-39
- “Custom Toolchain Registration”
- “Register Custom Toolchain and Build Executable” on page 44-21
- “Template Makefiles and Make Options” on page 31-36
- “Target Language Compiler Overview”
- “Executable Program Generation” on page 31-62
- “Configure CMake Build Process” on page 31-13

Define Custom Makefile-Based Toolchains Using Target Framework

To build code that you generate from Simulink models, you can specify a shipped makefile-based toolchain definition – see “Toolchain Approach” on page 31-20. Using the Target Framework, you can define and register custom makefile-based toolchains. This topic provides toolchain definition examples that you can adapt to build code for your target hardware.

Overview of Makefile-Based Toolchain Definition

To define a toolchain, create a `target.Toolchain` object. Use the object to:

- Associate the toolchain with your target hardware.
- Specify the platforms on which the toolchain can run.
- Provide system environment setup commands and paths for the operating system.

To specify that the toolchain is makefile-based, attach a `target.MakefileBuilder` object to the `Builder` property of the `target.Toolchain` object. The `target.MakefileBuilder` object specifies the type of makefile (GNU Make or NMAKE) that the software generates.

To specify the tools (assembler, compiler, linker, etc) for the toolchain, use `target.BuildTool` objects. Associate each `target.BuildTool` object with a predefined `target.BuildToolType` object that specifies the directives and file types that the `target.BuildTool` object must define. If you specify the name of the `target.BuildToolType` when creating `target.BuildTool`, the software uses default GCC values for directives and file types. If you use a toolchain that is not GCC-based, you must update the prepopulated directives and file-type settings with values for the toolchain.

When you add the `target.Toolchain` object to the internal database, the software uses specifications in the `target.BuildToolType` objects to check that:

- Directive names are valid and the directives have a specification.
- Names for valid file types are associated with file extensions, and the required file extensions are set.

For each `target.BuildTool` object, you can use the `showDirectives` and `showFileExtensions` methods to visualize the build tool configuration.

Define GCC-Based Toolchain for Windows Development Computer

- 1 Create a `target.Toolchain` object and specify the operating system.

```
mingwtc = target.create('Toolchain', ...
    'Name', 'Example MinGW Toolchain', ...
    'HostOperatingSystemSupport', target.HostOperatingSystemSupport.WindowsOnly);
```

- 2 Place the binary files folder of the MinGW toolchain on the system search path.

```
mingwtc.EnvironmentConfiguration.SystemPaths{end+1} = '${MW_MINGW64_LOC}/bin';
```

- 3 Associate the toolchain with the target hardware, which is your Windows development computer.

```
mingwtc.SupportedHardware = target.create('HardwareComponentSupport', ...
    'Component', target.get('Processor', 'Intel-x86-64 (Windows64)'));
```


4 Specify that the toolchain is based on a GNU Make makefile.

```
mingwtc.Builder = target.create('MakefileBuilder', 'GMake');
```

5 Specify the assembler.

```
assembler = target.create('BuildTool', 'Assembler', 'as', ...
    'Name', 'GNU Assembler');
mingwtc.Tools(end+1) = assembler;
```

6 For the Windows operating system, specify C and C++ compilers that use command files and generate object files with the `.obj` extension.

```
cCompiler = target.create('BuildTool', 'C Compiler', 'gcc', ...
    'Name', 'MinGW GCC C Compiler');
cCompiler.setDirective('CommandFile', '@');
cCompiler.setFileExtensions('Object', {'obj'});
mingwtc.Tools(end+1) = cCompiler;
```

```
cppCompiler = target.create('BuildTool', 'C++ Compiler', 'g++', ...
    'Name', 'MinGW GCC C++ Compiler');
cppCompiler.setDirective('CommandFile', '@');
cppCompiler.setFileExtensions('Object', {'obj'});
mingwtc.Tools(end+1) = cppCompiler;
```

7 Through directives and file extension settings, specify C and C++ linkers that:

- Support the generation of dynamic link libraries (DLLs)
- Group libraries that have circular dependencies
- Use command files
- Create files for Windows

```
cLinker = target.create('BuildTool', 'Linker', 'gcc', ...
    'Name', 'MinGW Linker', ...
    'HostOperatingSystemSupport', target.HostOperatingSystemSupport.WindowsOnly);
cLinker.setDirective('Shared', '-shared -WL,--out-implib,$(notdir $(basename $(PRODUCT))).lib');
cLinker.setDirective('LibraryGroup', '-WL,--start-group', '-WL,--end-group');
cLinker.setDirective('CommandFile', '@');
cLinker.setFileExtensions('Object', {'obj'});
cLinker.setFileExtensions('Executable', {'exe'});
cLinker.setFileExtensions('Shared Library', {'dll'});
mingwtc.Tools(end+1) = cLinker;
```

```
cppLinker = target.create('BuildTool', 'Copy', cLinker, ...
    'Name', 'MinGW C++ Linker', ...
    'BuildToolType', target.get('BuildToolType', 'C++ Linker'), ...
    'Command', target.create('Command', 'g++'));
mingwtc.Tools(end+1) = cppLinker;
```

In this example, you create the `cppLinker` object by creating a copy of `cLinker` and modifying properties of the copy.

8 Specify an archiver.

```
archiver = target.create('BuildTool', 'Archiver', 'ar ruvs', ...
    'Name', 'GNU Archiver');
archiver.setFileExtensions('Object', {'obj'});
mingwtc.Tools(end+1) = archiver;
```

9 Specify a make tool.

```
maketool = target.create('BuildTool', 'Make Tool', 'mingw32-make -j$(NUM_CORES)', ...
    'Name', 'MinGW GNU Make', ...
    'HostOperatingSystemSupport', target.HostOperatingSystemSupport.WindowsOnly);
mingwtc.Tools(end+1) = maketool;
```

The `$(NUM_CORES)` token is expanded to the number of cores on your system, and enables parallel processing of the `make` command.

- 10 Specify basic system tools like `echo`, `del`, and `move`, which the makefile uses to display, delete, and move files. The predefined `target.Toolset` object contains the tool definitions.

```
basictools = target.get('Toolset', 'Windows system tools for Makefiles');
mingwtc.Tools(end+1) = basictools;
```

- 11 Specify standard dependencies:

- C math and Winsock libraries to link with the compiled generated code
- Compiler flags that are always passed to the C and C++ compilers

```
mingwtc.BuildRequirements.SharedLibraries{end+1} = 'm';
mingwtc.BuildRequirements.SharedLibraries{end+1} = 'ws2_32';

mingwtc.BuildRequirements.CompilerFlags{end+1} = '-fwrapv';
mingwtc.BuildRequirements.CompilerFlags{end+1} = '-fpic';

mingwtc.BuildRequirements.LinkerFlags{end+1} = '-static';
mingwtc.BuildRequirements.LinkerFlags{end+1} = '-m64';
```

- 12 Add the toolchain definition to the internal database.

```
target.add(mingwtc);
```

To use the custom toolchain definition for building generated code, in the Configuration Parameters dialog box:

- 1 On the **Hardware Implementation** pane, select your target device by setting **Device vendor** to Intel and **Device type** to x86-64 (Windows64).
- 2 On the **Code Generation** pane, from the **Toolchain** list, select Example MinGW Toolchain.
- 3 Click **OK**.

When you run, for example, the `slbuild` function or a software-in-the-loop (SIL) simulation, the build process uses the custom toolchain to build generated code.

If you want to remove the custom toolchain definition from the internal database, run:

```
customToolChainDef = target.get('Toolchain', 'Example MinGW Toolchain');
target.remove(customToolChainDef);
```

Define Microsoft Visual Studio Toolchain for Building 32-Bit Application

- 1 Create a toolchain object and specify the operating system.

```
msvctc = target.create('Toolchain', ...
    'Name', '32-bit Visual Studio Toolchain', ...
    'HostOperatingSystemSupport', target.HostOperatingSystemSupport.WindowsOnly);
```

- 2 Create a command that runs `vcvarsall.bat`, which is required to set up environment variables for Microsoft Visual C++. This command assumes that Microsoft Visual Studio 2019 is installed on your development computer. If another version is installed, provide the full path to `vcvarsall.bat`.

```
msvctc.EnvironmentConfiguration.SetupCommand = target.create('Command', ...
    '$(MSVC160_INSTALL_DIR)\VC\Auxiliary\Build\VCVARSALL.BAT x86');
```

- 3 Associate the toolchain with the target hardware, which is your Windows development computer.

```
msvctc.SupportedHardware = target.create('HardwareComponentSupport', ...
    'Component', target.get('Processor', 'Intel-x86-32 (Windows32)');
```

- 4 Specify that the toolchain is based on an NMAKE makefile.

```
msvctc.Builder = target.create('MakefileBuilder', 'NMake');
```

5 Using target.BuildTool objects, specify C and C++ compilers:

- Update default GCC directives with Microsoft Visual C++ values.
- Use command files and generate object files with the .obj extension.
- Specify additional command-line flags that only apply to the C++ compiler.

```
cCompiler = target.create('BuildTool', 'C Compiler', 'cl', ...
    'Name', 'Microsoft Visual C Compiler', ...
    'HostOperatingSystemSupport', target.HostOperatingSystemSupport.WindowsOnly);
cCompiler.setDirective('OutputFlag', '-Fo');
cCompiler.setDirective('Debug', '-Zi');
cCompiler.setDirective('EnableOptimization', '/O2 /Oy-');
cCompiler.setDirective('DisableOptimization', '/Od /Oy-');
cCompiler.setDirective('CommandFile', '@');
cCompiler.setFileExtensions('Object', {'.obj'});
msvctc.Tools(end+1) = cCompiler;
cppCompiler = target.create('BuildTool', 'C++ Compiler', 'cl -EHs /wd4251 /Zc:__cplusplus', ...
    'Name', 'Microsoft Visual C++ Compiler', ...
    'HostOperatingSystemSupport', target.HostOperatingSystemSupport.WindowsOnly);
cppCompiler.setDirective('OutputFlag', '-Fo');
cppCompiler.setDirective('Debug', '-Zi');
cppCompiler.setDirective('EnableOptimization', '/O2 /Oy-');
cppCompiler.setDirective('DisableOptimization', '/Od /Oy-');
cppCompiler.setDirective('CommandFile', '@');
cppCompiler.setFileExtensions('Object', {'.obj'});
msvctc.Tools(end+1) = cppCompiler;
```

6 Through directives and file extension settings, specify C and C++ linkers.

```
cLinker = target.create('BuildTool', 'Linker', 'link', ...
    'Name', 'Microsoft Visual C Linker', ...
    'HostOperatingSystemSupport', target.HostOperatingSystemSupport.WindowsOnly);
cLinker.setDirective('Library', '-L');
cLinker.setDirective('LibrarySearchPath', '/LIBPATH');
cLinker.setDirective('OutputFlag', '-out:');
cLinker.setDirective('Debug', '/DEBUG');
cLinker.setDirective('Shared', '-dll'); % Allow shared library
cLinker.setDirective('DefFile', '-def:');
cLinker.setDirective('CommandFile', '@');
cLinker.setFileExtensions('Object', {'.obj'});
cLinker.setFileExtensions('Executable', {'.exe'});
cLinker.setFileExtensions('Shared Library', {'.dll'});
msvctc.Tools(end+1) = cLinker;
```

```
cppLinker = target.create('BuildTool', 'C++ Linker', 'link', ...
    'Name', 'Microsoft Visual C++ Linker', ...
    'HostOperatingSystemSupport', target.HostOperatingSystemSupport.WindowsOnly);
cppLinker.setDirective('Library', '-L');
cppLinker.setDirective('LibrarySearchPath', '/LIBPATH');
cppLinker.setDirective('OutputFlag', '-out:');
cppLinker.setDirective('Debug', '/DEBUG /DEBUGTYPE:cv');
cppLinker.setDirective('Shared', '-dll'); % Allow shared library
cppLinker.setDirective('DefFile', '-def:');
cppLinker.setDirective('CommandFile', '@');
cppLinker.setFileExtensions('Object', {'.obj'});
cppLinker.setFileExtensions('Executable', {'.exe'});
cppLinker.setFileExtensions('Shared Library', {'.dll'});
msvctc.Tools(end+1) = cppLinker;
```

7 Specify an archiver.

```
archiver = target.create('BuildTool', 'Archiver', 'lib /nologo', ...
    'Name', 'Microsoft Visual C/C++ Archiver');
archiver.setDirective('OutputFlag', '-out:');
archiver.setFileExtensions('Object', {'.obj'});
archiver.setFileExtensions('Static Library', {'.lib'});
msvctc.Tools(end+1) = archiver;
```

8 Specify a make tool.

```

maketool = target.create('BuildTool', 'Make Tool', 'nmake', ...
    'Name', 'Microsoft NMake', ...
    'HostOperatingSystemSupport', target.HostOperatingSystemSupport.WindowsOnly);
msvctc.Tools(end+1) = maketool;

```

- 9 Specify basic system tools like `echo`, `del`, and `move`, which the makefile uses to display, delete, and move files. The predefined `target.Toolset` object contains the tool definitions.

```

basictools = target.get('Toolset', 'Windows system tools for Makefiles');
msvctc.Tools(end+1) = basictools;

```

- 10 Specify standard dependencies:

- Libraries to link with the compiled generated code
- Preprocessor directives that must be set
- Compiler flags that are always passed to the C and C++ compilers
- Linker flags

```

msvctc.BuildRequirements.SharedLibraries{end+1} = 'kernel32.lib';
msvctc.BuildRequirements.SharedLibraries{end+1} = 'ws2_32.lib';
msvctc.BuildRequirements.SharedLibraries{end+1} = 'msock.lib';
msvctc.BuildRequirements.SharedLibraries{end+1} = 'advapi32.lib';

msvctc.BuildRequirements.Defines{end+1} = '_CRT_SECURE_NO_WARNINGS';
msvctc.BuildRequirements.Defines{end+1} = 'WIN32';
msvctc.BuildRequirements.Defines{end+1} = '_MT';

msvctc.BuildRequirements.CompilerFlags{end+1} = '-nologo';
msvctc.BuildRequirements.CompilerFlags{end+1} = '-GS';
msvctc.BuildRequirements.CompilerFlags{end+1} = '-W4';
msvctc.BuildRequirements.CompilerFlags{end+1} = '-MT';

msvctc.BuildRequirements.LinkerFlags{end+1} = '/INCREMENTAL:NO';
msvctc.BuildRequirements.LinkerFlags{end+1} = '/NOLOGO';

```

- 11 Add the toolchain definition to the internal database.

```
target.add(msvctc);
```

To use the custom toolchain definition for building generated code, in the Configuration Parameters dialog box:

- 1 On the **Hardware Implementation** pane, select your target device by setting **Device vendor** to Intel and **Device type** to x86-32 (Windows32).
- 2 On the **Code Generation** pane, from the **Toolchain** list, select 32-bit Visual Studio Toolchain.
- 3 Click OK.

When you run, for example, the `slbuild` function or a software-in-the-loop (SIL) simulation, the build process uses the custom toolchain to build generated code.

If you want to remove the custom toolchain definition from the internal database, run:

```

customToolChainDef = target.get('Toolchain', '32-bit Visual Studio Toolchain');
target.remove(customToolChainDef);

```

See Also

`target.create` | `target.Toolchain` | `target.MakefileBuilder` | `target.BuildTool`

Related Examples

- “Approaches for Building Code Generated from Simulink Models” on page 31-11

Template Makefiles and Make Options

The code generator includes a set of built-in template makefiles that build programs for specific system target files.

Types of Template Makefiles

There are two types of template makefiles:

- *Compiler-specific* template makefiles are for a particular compiler or development system.

By convention, compiler-specific template makefiles names correspond to the system target file and compiler (or development system). For example, `grt_vcx64.tmf` is the template makefile for building a generic real-time program under the Visual C++ compiler; `ert_lcc64.tmf` is the template makefile for building an Embedded Coder program under the `lcc` compiler.

- *Default* template makefiles make your model designs more portable, by choosing the compiler-specific makefile and compiler for your installation. “Select and Configure C or C++ Compiler” on page 31-3 describes the operation of default template makefiles in detail.

Default template makefiles have names that follow the pattern `target_default_tmf`. They are MATLAB language files that, when run, select the TMF for the specified system target file configuration. For example, `grt_default_tmf` is the default template makefile for building a generic real-time program; `ert_default_tmf` is the default template makefile for building an Embedded Coder program.

For details on the structure of template makefiles, see “Customize Template Makefiles” on page 46-44. This section describes compiler-specific template makefiles and common options you can use with each.

Specify Template Makefile Options

You can specify template makefile options through the **Make command** configuration parameter. Append the options after `make_rtw` (or other make command), as in the following example:

```
make_rtw OPTS="-DMYDEFINE=1"
```

The syntax for make command options differs slightly for different compilers.

Note To control compiler optimizations for a makefile build, use the **Compiler optimization level** configuration parameter, which provides:

- System target file-independent values `Optimizations on` (faster runs) and `Optimizations off` (faster builds), which easily allow you to toggle compiler optimizations on and off during code development
- The value `Custom` for entering custom compiler optimization flags at Simulink GUI level (rather than at other levels of the build process)

If you specify compiler options for your makefile build using `OPT_OPTS`, `MEX_OPTS` (except `MEX_OPTS="-v"`), or `MEX_OPT_FILE`, the value of **Compiler optimization level** is ignored and a warning is issued about the ignored parameter.

Template Makefiles for UNIX Platforms

The template makefiles for UNIX platforms are for the Free Software Foundation's GNU Make. These makefiles conform to the guidelines specified in the IEEE^{®5} Std 1003.2-1992 (POSIX) standard:

- `ert_unix.tmf`
- `grt_unix.tmf`

You can supply options to makefiles through the **Make command** configuration parameter. Options specified in **Make command** are passed to the command-line invocation of the `make` utility, which adds them to the overall flags passed to the compiler. The following options can be used to modify the behavior of the build:

- `OPTS` — User-specific options, for example,


```
OPTS="-DMYDEFINE=1"
```
- `OPT_OPTS` — Optimization options. Default is `-O`. To enable debugging, specify the option as `OPT_OPTS=-g`. Because of optimization problems in `IBM_RS`, the default is to not optimize.
- `CPP_OPTS` — C++ compiler options.
- `USER_SRCS` — Additional user sources, such as files used by S-functions.
- `USER_INCLUDES` — Additional include paths, for example,


```
USER_INCLUDES="-Iwhere-ever -Iwhere-ever2"
```
- `DEBUG_BUILD` — Add debug information to generated code, for example,


```
DEBUG_BUILD=1
```

These options are also documented in the comments at the head of the respective template makefiles.

Template Makefiles for the Microsoft Visual C++ Compiler

To build an executable file using the Visual C++ compiler, use one of the `target_vcx64.tmf` template makefiles:

- `ert_vcx64.tmf`
- `grt_vcx64.tmf`

You can supply options to makefiles through the **Make command** configuration parameter. Options specified in **Make command** are passed to the command-line invocation of the `make` utility, which adds them to the overall flags passed to the compiler. The following options can be used to modify the behavior of the build:

- `OPT_OPTS` — Optimization option. Default is `-O2`. To enable debugging, specify the option as `OPT_OPTS=-Zi`.
- `OPTS` — User-specific options. For example:


```
OPTS="/D MYDEFINE=1"
```
- `CPP_OPTS` — C++ compiler options.
- `USER_SRCS` — Additional user sources, such as files used by S-functions.

5 IEEE is a registered trademark of The Institute of Electrical and Electronics Engineers, Inc.

- `USER_INCLUDES` — Additional include paths, for example,
`USER_INCLUDES="-Iwhere-ever -Iwhere-ever2"`
- `DEBUG_BUILD` — Add debug information to generated code, for example,
`DEBUG_BUILD=1`

These options are also documented in the comments at the head of the respective template makefiles.

Template Makefiles for the LCC Compiler

The code generator provides template makefiles to create an executable for the Windows platform using Lcc compiler Version 2.4 and GNU Make (gmake).

- `ert_lcc64.tmf`
- `grt_lcc64.tmf`

You can supply options to makefiles by using the **Make command** configuration parameter. Options specified in the **Make command** field are passed to the command-line invocation of the `make` utility, which adds them to the overall flags passed to the compiler. The following options can be used to modify the behavior of the build:

- `OPTS` — User-specific options, for example,
`OPTS="-DMYDEFINE=1"`
- `OPT_OPTS` — Optimization options. Default is to not use options. To enable debugging, specify `-g4`:
`OPT_OPTS="-g4"`
- `CPP_OPTS` — C++ compiler options.
- `USER_SRCS` — Additional user sources, such as files used by S-functions.
- `USER_INCLUDES` — Additional include paths. For example:
`USER_INCLUDES="-Iwhere-ever -Iwhere-ever2"`

For `lcc`, use `/` as file separator before the file name instead of `\`, for example, `d:\work\proj1\myfile.c`.

- `DEBUG_BUILD` — Add debug information to generated code, for example,
`DEBUG_BUILD=1`

These options are also documented in the comments at the head of the respective template makefiles.

See Also

More About

- “Configure a System Target File” on page 21-2
- “Customize System Target Files” on page 46-19
- “Associate the Template Makefile with a Toolchain” on page 31-26

Build Process Workflow for Real-Time Systems

The building process includes generating code in C or C++ from a model and building an executable program from the generated code. This example can use a generic real-time (GRT) or an embedded real-time (ERT) system target file (STF) for code generation. The resulting standalone program runs on your development computer, independent of external timing and events.

Working Folder

This example uses a local copy of the `slexAircraftExample` model, stored in its own folder, `aircraftexample`. Set up your *working folder* as follows:

- 1 In the MATLAB Current Folder browser, navigate to a folder to which you have write access.
- 2 To create the working folder, enter the following MATLAB command:

```
mkdir aircraftexample
```

- 3 Make `aircraftexample` your working folder:

```
cd aircraftexample
```

- 4 Open the `slexAircraftExample` model:

```
openExample('slexAircraftExample')
```

The model appears in the Simulink Editor model window.

- 5 In the model window, choose **File > Save As**. Navigate to your working folder, `aircraftexample`. Save a copy of the `slexAircraftExample` model as `myAircraftExample`.

Build Folder and Code Generation Folders

While producing code, the code generator creates a *build folder* within your working folder. The build folder name is `model_target_rtw`, derived from the name of the source model and the chosen system target file. The build folder stores generated source code and other files created during the build process. Examine the build folder contents at the end of this example.

When a model contains Model blocks (references to other models), the model build creates special subfolders in your “Code generation folder” to organize code for the referenced models. These code generation folders exist alongside product build folders and are named `slprj`. For more information, see “Generate Code for Model Reference Hierarchy” on page 8-2.

Under the `slprj` folder, a subfolder named `_sharedutils` contains generated code that can be shared between models.

Set Model Parameters for Code Generation

To generate code from your model, you must change some of the model configuration parameters. In particular, the generic real-time (GRT) system target file and most other system target files require that the model specifies a fixed-step solver.

Note For models that specify variable-step solvers, the code generator produces code only if the models also specify rapid simulation (`rsim`) or S-function system target files.

- 1 Open the `myAircraftExample` model if it is not already open.
- 2 In the Configuration Parameters dialog box, specify configuration parameter values for the solver:
 - **Start time:** `0.0`
 - **Stop time:** `60`
 - **Type:** Fixed-step
 - **Solver:** `ode5` (Dormand-Prince)
 - **Fixed step size (fundamental sample time):** `0.1`
 - **Treat each discrete rate as a separate task:** `Off`
- 3 Click **Apply**.
- 4 Save the model.

Configure Build Process

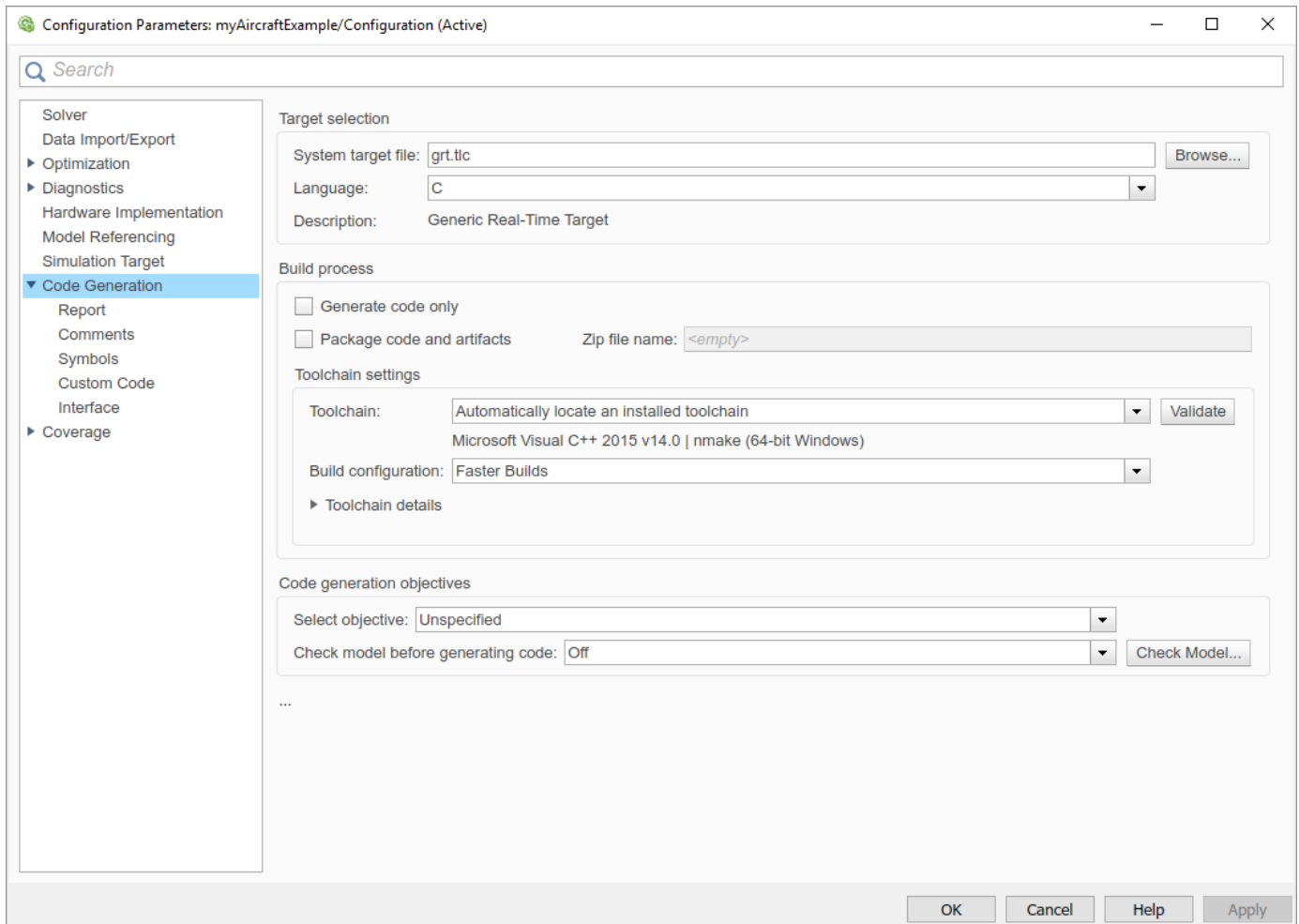
To configure the build process for your model, choose a system target file, a toolchain or template makefile, and a make command.

In these examples and in most applications, you do not need to specify these parameters individually. The examples use the ready-to-run generic real-time target (GRT) configuration. The GRT system target file builds a standalone executable program that runs on your desktop computer.

To select the GRT system target file:

- 1 Open the `myAircraftExample` model if it is not already open.
- 2 In the Configuration Parameters dialog box, in the **System target file** field, enter `grt.tlc`. Then click **Apply**.

You see selections for **Toolchain** (Automatically locate an installed toolchain), and **Build Configuration** (Faster Builds).



3 Save the model.

Note If you click **Browse**, a System Target File Browser opens and displays the system target files on the MATLAB path. Some system target files require additional products. For example, `ert.tlc` requires Embedded Coder.

Set Code Generation Parameters

- 1 Open the `myAircraftExample` model if it is not already open.
- 2 In the Configuration Parameters dialog box, specify settings:
 - a Use the default settings for the **Code Generation** advanced parameters, which control build verbosity and debugging:
 - **Verbose build** (RTWVerbose)
 - **Retain .rtw file** (RetainRTWFile)
 - **Profile TLC** (ProfileTLC)
 - **Start TLC debugger when generating code** (TLCDebug)

- **Start TLC coverage when generating code** (TLCCoverage)
 - **Enable TLC assertion** (TLCAssert)
- b** Use the default **Code Generation > Comments** settings.
 - c** The **Code Generation > Identifiers** options control the look and feel of generated code. Use the default settings.
 - d** Select **Code Generation > Interface**.
 - i** From the **Shared code placement** list, select **Shared** location. The build process places generated code for utilities in a subfolder within your “Code generation folder”.
 - ii** Under the **Advanced parameters**, clear the **Classic call interface** check box.
 - iii** Under the **Advanced parameters**, select the **Single output/update function** check box.
 - e** In **Code Generation > Report**, select **Create code generation report** and **Open report automatically**. This action enables the software to create and display a code generation report for the myAircraftExample model.
- 3** Click **Apply** and save the model.

Build and Run a Program

The build process generates C code from the model. It then compiles and links the generated program to create an executable image. To build and run the program:

- 1** With the myAircraftExample model open, perform one of these actions:
 - On the **Apps** tab, open the **Simulink Coder** app. In the **C Code** tab, click **Build**.
 - Press **Ctrl+B**.
 - Run the `slbuild` command from the MATLAB command line.

You see code generation and compilation messages in the Command Window. The initial message is:

```
### Starting build procedure for model: myAircraftExample
```

The contents of many of the succeeding messages depends on your compiler and operating system. The final messages include:

```
### Created executable myAircraftExample.exe
### Successful completion of build procedure for model: myAircraftExample
### Creating HTML report file index.html
```

The code generation folder now contains an executable, myAircraftExample.exe (Microsoft Windows platforms) or myAircraftExample (UNIX platforms). In addition, the build process has created an `slprj` folder and a myAircraftExample_grt_rtw folder in your “Code generation folder”.

Note After generating the code for the myAircraftExample model, the build process displays a code generation report. See “Report Generation” for more information about how to create and use a code generation report.

- 2** To see the contents of the working folder after the build, enter the `dir` or `ls` command:

```
>> dir
.                myAircraftExample.slx          slprj
..               myAircraftExample.slx.autosave
myAircraftExample.exe  myAircraftExample_grt_rtw
```

- 3 To run the executable from the Command Window, type `!myAircraftExample`. The `!` character passes the command that follows it to the operating system, which runs the standalone `myAircraftExample` program.

```
>> !myAircraftExample
** starting the model **
** created myAircraftExample.mat **
```

- 4 To see the files created in the build folder, use the `dir` or `ls` command again. The exact list of files produced varies among MATLAB platforms and versions. Here is a sample list from a Windows platform:

```
>> dir myAircraftExample_grt_rtw
.                rt_main.obj                myAircraftExample_data.c
..               rtmodel.h                 myAircraftExample_data.obj
buildInfo.mat    rtw_proj.tmw                myAircraftExample_private.h
codeInfo.mat     myAircraftExample.bat           myAircraftExample_ref.rsp
myAircraftExample.c  myAircraftExample_types.h    html
myAircraftExample.h  myAircraftExample.mk
rt_logging.obj     myAircraftExample.obj
```

Contents of the Build Folder

The build process creates a build folder and names it `model_target_rtw`, where *model* is the name of the source model and *target* is the system target file selected for the model. In this example, the build folder is named `myAircraftExample_grt_rtw`.

The build folder includes the following generated files.

File	Description
<code>myAircraftExample.c</code>	Standalone C code that implements the model
<code>myAircraftExample.h</code>	An include header file containing definitions of parameters and state variables
<code>myAircraftExample_private.h</code>	Header file containing common include definitions
<code>myAircraftExample_types.h</code>	Forward declarations of data types used in the code
<code>rtmodel.h</code>	Header file for including generated code in the static main program (its name does not change, and it simply includes <code>myAircraftExample.h</code>)

The code generation report that you created for the `myAircraftExample` model displays a link for each of these files. You can click the link [explore the file contents](#).

The build folder contains other files used in the build process. They include:

- `myAircraftExample.mk` — Makefile for building executable using the specified Toolchain.
- Object (`.obj`) files
- `myAircraftExample.bat` — Batch control file
- `rtw_proj.tmw` — Marker file

- `buildInfo.mat` — Build information for relocating generated code to another development environment
- `myAircraftExample_ref.rsp` — Data to include as command-line arguments to `mex` (Windows systems only)

The build folder also contains a subfolder, `html`, which contains the files that make up the code generation report. For more information, see “Reports for Code Generation” on page 25-2.

Customized Makefile Generation

After producing code, the code generator produces a customized makefile, `model.mk`. The generated makefile instructs the `make` system utility to compile and link source code generated from the model, any required harness program, libraries, or user-provided modules. The code generator produces the file `model.mk` regardless of the approach that you use for build process control:

- If you use the toolchain approach, the code generator creates `model.mk` based on the model **Toolchain settings**. You can modify generation of the makefile through the `rtwmakecfg.m` API.
- If you use the template makefile approach, the code generator creates `model.mk` from a system template file, `system.tmf` (where `system` stands for the selected system target file name). The system template makefile is designed for your system target file. You can modify the template makefile to specify compilers, compiler options, and additional information for the creation of the executable.

For more information, see “Configure Toolchain (ToolchainInfo) or Template Makefile Build Process” on page 31-20.

See Also

More About

- “Configure Toolchain (ToolchainInfo) or Template Makefile Build Process” on page 31-20
- “Manage Build Process Folders” on page 24-30
- “Reports for Code Generation” on page 25-2
- “Select and Configure C or C++ Compiler” on page 31-3

Build Models from a Windows Command Prompt Window

This example shows how to build models by using a batch file, entering commands at the Command Prompt in Windows®.

About MATLAB Command-Line (Start Up) Arguments

When you start MATLAB® from a Command Prompt in Windows (as done in a batch file), you can control MATLAB start up with a number of command-line arguments.

For a description of these command-line arguments, in the Command Prompt window, type `matlab -help`.

To start MATLAB from a Command Prompt window, use these steps:

- 1 From the Windows Start menu, open a Command Prompt window.
- 2 From the Windows Command Prompt, type: `matlab`.

Tip: To display the path to the MATLAB root folder, at the MATLAB command prompt type: `matlabroot`.

Run MATLAB with a Batch File

When you run MATLAB with a batch file, you can:

- Control MATLAB start up with command-line arguments
- Run a series of operating system commands (such as source control checkout/commit)
- Run a series of MATLAB scripts

A batch approach also lets you automate your overall build process. Such a process can generate code from one or more Simulink® models, then use your makefile to compile custom code and generated code.

This batch file sets the `MATLABROOT` environment variable, sets the `PATH` environment variable to include `MATLABROOT`, and starts MATLAB with an input script argument `%1` and a logfile argument.

Note: Customize the `MATLABROOT` value in the batch file to match your system. The batch file assumes that a `c:\temp` folder exists on your system.

Create a batch file named `mat.bat`

```
SET MATLABROOT="C:\Program Files\MATLAB\R2019a"  
PATH=%MATLABROOT%;%PATH%  
START matlab.exe -batch %1 -logfile c:\temp\logfile  
PAUSE
```

Create a MATLAB script `myFilesToBuild.m`

```
my_rtwdemo_counter_builder  
my_rtwdemo_rtwintro_builder  
exit
```

Create a MATLAB script `my_rtwdemo_counter_builder.m`

```
open_system('rtwdemo_counter');  
save_system('rtwdemo_counter','my_rtwdemo_counter')  
slbuild('my_rtwdemo_counter');  
close_system('my_rtwdemo_counter');
```

Create a MATLAB script my_rtwdemo_rtwintrou_builder.m

```
open_system('rtwdemo_rtwintrou');  
save_system('rtwdemo_rtwintrou','my_rtwdemo_rtwintrou')  
slbuild('my_rtwdemo_rtwintrou');  
close_system('my_rtwdemo_rtwintrou');
```

Run the batch file

From the Windows Start menu, open a Command Prompt window, change folders to the folder containing the batch file, and type:

```
mat myFilesToBuild
```

When you run the batch file with the input MATLAB script, the batch file runs MATLAB and loads, builds, and closes each of the example Simulink models.

Observe the log of MATLAB operations

After the batch file runs, view the `c:\temp\logfile` file.

Omitting the semicolon (;) from the `slbuild` line in each script provides more build information in the log file.

Optimize Your Batch File

Use the MATLAB command-line arguments to optimize the batch file. For example:

- Suppress the MATLAB splash screen on startup with the `-nosplash` argument.
- Provide command-line input to the input script or function selected with the `-batch` argument.

You can call a function `myfile.m`, which accepts two arguments:

```
matlab -batch myfile(arg1,arg2)
```

To pass numeric values into `myfile.m`, replace `arg1` and `arg2` with numeric values.

To pass string or character values into `myfile.m`, replace `arg1` and `arg2` with the string or character values surrounded in single quotes. For example, to pass the string values `hello` and `world` into `myfile.m`, in the Command Prompt window, type:

```
matlab -batch myfile('hello','world')
```

Copyright 2007-2019 The MathWorks, Inc.

See Also

More About

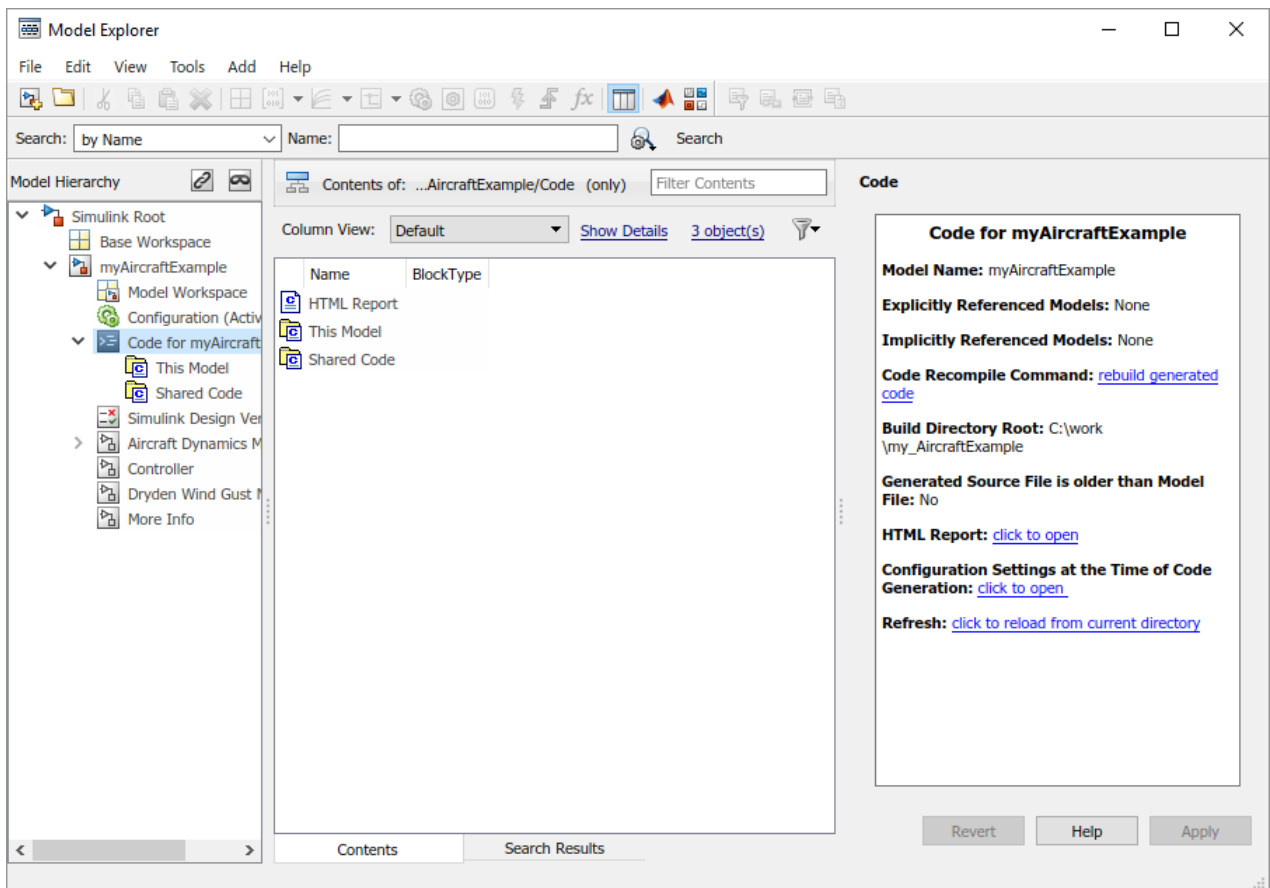
- `matlab` (Windows)

- “Compile and Debug Generated C Code with Microsoft Visual Studio” on page 31-61

Rebuild a Model

If you update generated source code or makefiles manually to add customizations, you can rebuild the files with the `rtwrebuild` command. This command recompiles the modified files by invoking the generated makefile. Alternatively, you can rebuild from the Model Explorer:

- 1 Open the Model Explorer
- 2 In the **Model Hierarchy** pane, expand the node for the model.
- 3 Click the **Code for *model*** node.
- 4 In the **Code** pane, next to **Code Recompile Command**, click **rebuild generated code**.



See Also

More About

- `rtwrebuild`
- “Build Process Workflow for Real-Time Systems” on page 31-39

Control Regeneration of Top Model Code

When you rebuild a model, by default, the build process performs checks to determine whether changes to the model or relevant settings require regeneration of the top model code. The model build regenerates top model code if any of the following conditions is true:

- The structural checksum of the model has changed.
- The top-model-only checksum has changed. The top-model-only checksum provides information about top model parameters, such as application lifespan, maximum stack size, make command, verbose and `.rtw` file debug settings, and `TLCOptions`.
- Any of the following TLC debugging configuration parameters is selected:
 - **Start TLC debugger when generating code** (`TLCDebug`)
 - **Start TLC coverage when generating code** (`TLCCoverage`)
 - **Enable TLC assertion** (`TLCAssert`)
 - **Profile TLC** (`ProfileTLC`)

Whether the top model code is regenerated, the build process calls the build process hooks and reruns the makefile. The hooks include the `STF_make_rtw_hook` functions and the post code generation command. This process recompiles and links the external dependencies.

System target file authors can perform actions related to code regeneration in the `STF_make_rtw_hook` functions that the build process calls. These actions include forcing or reacting to code regeneration. For more information, see “Control Code Regeneration Using `STF_make_rtw_hook.m`” on page 44-27.

Regeneration of Top Model Code

If the checks determine that top model code generation is required, the build process fully regenerates and compiles the model code. An example check is whether previously generated code is not current due to a model update.

The build process omits regeneration of the top model code when the checks indicate both:

- The top model generated code is current for the model.
- Model settings do not require full regeneration.

This omission can significantly reduce model build times.

With an Embedded Coder license, if you modify a code generation template (CGT) file then rebuild your model, the code generation process does not force a top model build. In this case, see “Force Regeneration of Top Model Code” on page 31-49.

Force Regeneration of Top Model Code

If you want to control or override the default top model build behavior, use one of the following command-line options:

- To ignore the checksum and force regeneration of the top model code:

- `slbuild(model, 'ForceTopModelBuild', true)`
- `slbuild(model, 'StandaloneCoderTarget', 'ForceTopModelBuild', true)`
- To clean the model build area enough to trigger regeneration of the top model code at the next build (slbuild only):
 `slbuild(model, 'CleanTopModel')`

You can also force regeneration of the top model code by deleting the `slprj` folder in the “Code generation folder”. If you delete the `slprj` folder, delete the `model.slxc` file too.

See Also

`codebuild`

More About

- “Control Code Regeneration Using `STF_make_rtw_hook.m`” on page 44-27
- “Approaches for Building Code Generated from Simulink Models” on page 31-11
- “Rebuild a Model” on page 31-48
- “Reduce Build Time for Referenced Models by Using Parallel Builds” on page 31-51

Reduce Build Time for Referenced Models by Using Parallel Builds

For models that contain large model reference hierarchies, you can reduce code generation and compilation time by building the referenced models in parallel. With Parallel Computing Toolbox software, you can distribute the code generation and compilation for referenced models across a parallel pool of MATLAB workers. If you also have MATLAB Parallel Server™ software, you can distribute the code generation and compilation across remote workers in your MATLAB Parallel Server configuration.

Build Referenced Models in Parallel

To build referenced models in parallel:

- 1 Open the Configuration Parameters dialog box for the top model of the model hierarchy.
- 2 Select the **Enable parallel model reference builds** check box.
- 3 For each MATLAB worker, you can set up a MATLAB environment that is consistent with the MATLAB environment of the client. From the **MATLAB worker initialization for builds** drop-down list, select one of these values:
 - **None** -- Simulink does not initialize workers.
 - **Copy base workspace** -- Simulink attempts to copy the base workspace to each MATLAB worker.
 - **Load top model** -- Simulink loads the top model onto each MATLAB worker.
- 4 Build your model from the command line or the Simulink Editor Code perspective:
 - **Command line** -- Use, for example, `slbuild`. The build process displays build log messages in the Command Window.
 - **Simulink Editor Code perspective** -- Click the **Build** button. The build process displays build log messages in the Diagnostic Viewer.

If the Parallel Computing Toolbox parallel preference **Automatically create a parallel pool** is selected, a parallel pool of MATLAB workers is automatically started. For more information, see “Specify Your Parallel Preferences” (Parallel Computing Toolbox).

If a parallel pool of MATLAB workers is not running when you build your model, MATLAB automatically opens a parallel pool of workers by using the default cluster profile. To change the default behaviour of the worker cluster, you can modify properties of the cluster profile. If you have not touched your parallel preferences, the default profile is `local`. Control parallel behavior with the parallel preferences, including scaling up to a cluster, automatic pool creation, and preferred number of workers. For more information, see “Discover Clusters and Use Cluster Profiles” (Parallel Computing Toolbox).

For more general information about parallel computing, see “Run MATLAB Functions with Automatic Parallel Support” (Parallel Computing Toolbox).

Monitor Parallel Building of Referenced Models

Through the Build Status window, you can:

- View the overall build progress for a model hierarchy.
- View the build status of individual models in the model hierarchy.
- Cancel the parallel build process.

Note The Build Status window supports only parallel building of model hierarchies. Do not use the Build Status window for serial builds.

To open the Build Status window, use one of these line commands:

- `coder.buildstatus.open('model')`
- `slbuild('model', ... 'OpenBuildStatusAutomatically',true)`
- `slbuild('model','StandaloneCoderTarget', ... 'OpenBuildStatusAutomatically',true)`

This example shows how you can monitor the build process for a model hierarchy. In the hierarchy, `rtwdemo_parabuild_a_1` references `rtwdemo_parabuild_b_1`, `rtwdemo_parabuild_b_2`, and `rtwdemo_parabuild_b_3`.

- 1 Copy the model files to a local folder.

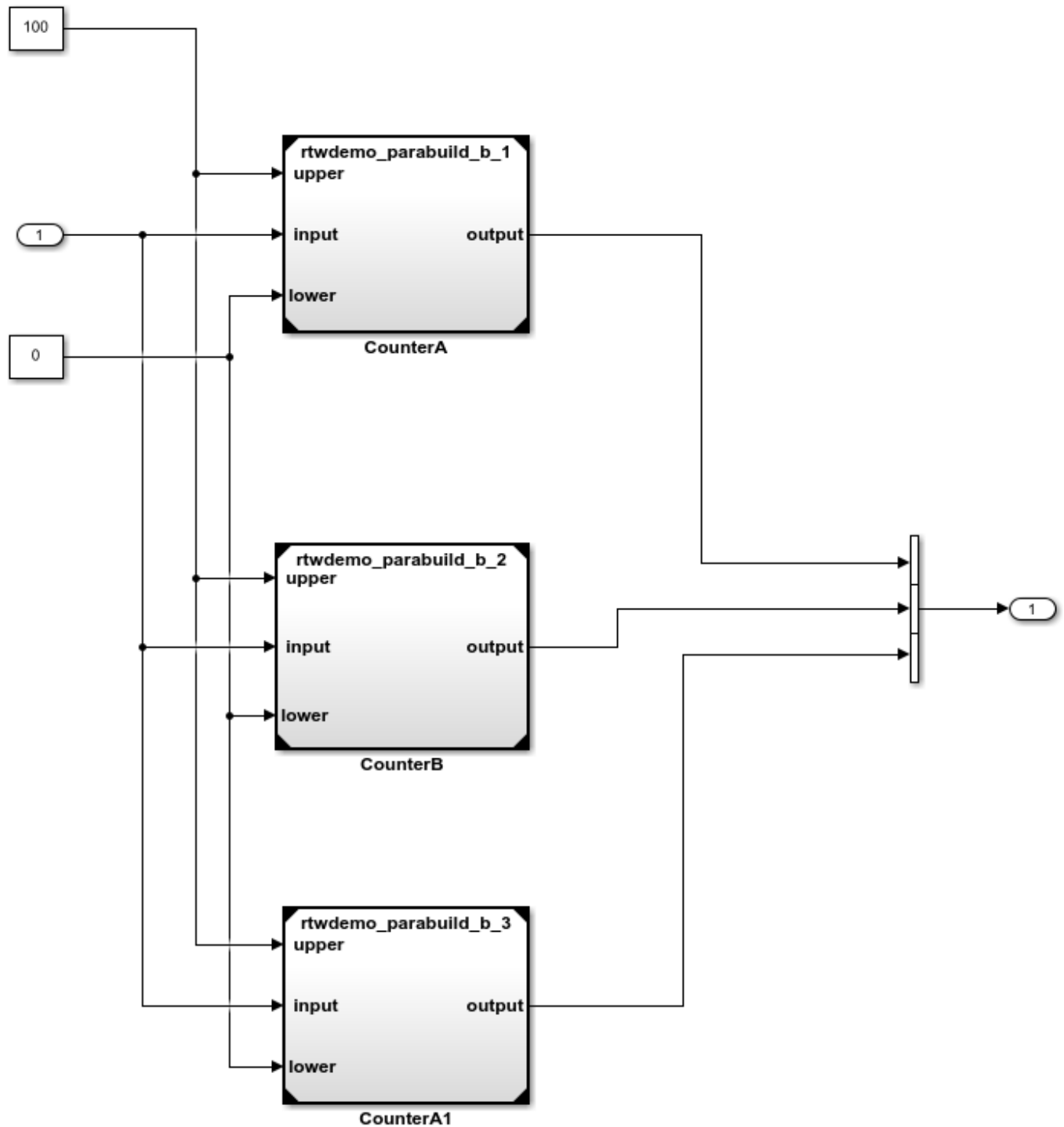
```
src_dir = ...
    fullfile(matlabroot,'toolbox','rtw','rtwdemos');
if exist(fullfile('.', 'myTempFolder'),'dir')
    rmdir('myTempFolder','s')
end
mkdir myTempFolder

copyfile(fullfile(src_dir,'rtwdemo_parabuild_a_1.slx'), ...
    'myTempFolder');
copyfile(fullfile(src_dir,'rtwdemo_parabuild_b_1.slx'), ...
    'myTempFolder');
copyfile(fullfile(src_dir,'rtwdemo_parabuild_b_2.slx'), ...
    'myTempFolder');
copyfile(fullfile(src_dir,'rtwdemo_parabuild_b_3.slx'), ...
    'myTempFolder');

cd myTempFolder
```

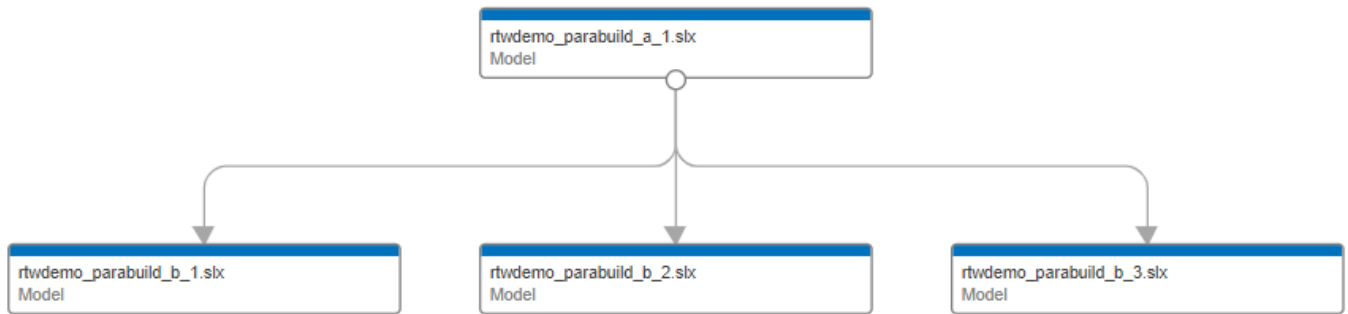
- 2 Open the top model.

```
open_system('rtwdemo_parabuild_a_1')
```



- 3 View the model dependencies with the Dependency Analyzer.

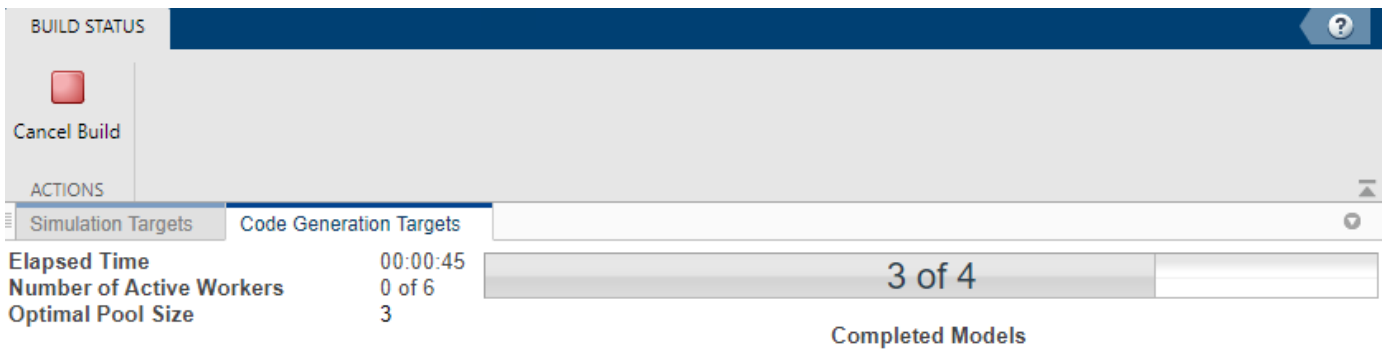
On the **Modeling** tab, click on the down arrow to expand the **Design** gallery. Under **Dependencies**, click **Dependency Analyzer**.



- 4 Start parallel building of the model hierarchy and open the Build Status window.

```
slbuild('rtwdemo_parabuild_a_1','OpenBuildStatusAutomatically',true)
```

The Build Status window displays the overall build progress and the build status for models as the code generator works through the model hierarchy.



Model Name	Status	Elapsed Time
rtwdemo_parabuild_a_1	Building	-
rtwdemo_parabuild_b_1	Completed	00:00:23
rtwdemo_parabuild_b_2	Completed	00:00:23
rtwdemo_parabuild_b_3	Completed	00:00:23

This table explains the **Status** column values.

Status	Description
Blocked	The code generator cannot schedule the model build because the build is blocked by dependencies. For example, a child model build is not complete.
Scheduled	In parallel building, the code generator schedules a model build when the build is not blocked by dependencies. The status of the model build is Scheduled until a parallel pool worker becomes available.
Building	The model is being built on a worker.
Completed	When a model is built without errors, the status changes to Completed.
Error	If the model build fails, the status changes to Error.

Status	Description
Up To Date	Code generation is not required for the model because the current generated code is up to date.
Canceling	When you click the Cancel Build button, the build process changes the status of incomplete builds to Canceling .
Canceled	The canceling operation is complete.

The **Elapsed Time** column shows the build times for models in the hierarchy. To reduce the total build time, analyse the build times for referenced models together with model dependency information. Then, consider, for example:

- Restructuring the referenced model hierarchy.
- Increasing the number of workers in the parallel pool.

See Also

`slbuild | coder.buildstatus.open | coder.buildstatus.close`

More About

- “Control Regeneration of Top Model Code” on page 31-49
- “Reduce Update Time for Referenced Models by Using Parallel Builds”
- “Build Process Workflow for Real-Time Systems” on page 31-39
- “Support Model Referencing” on page 46-61

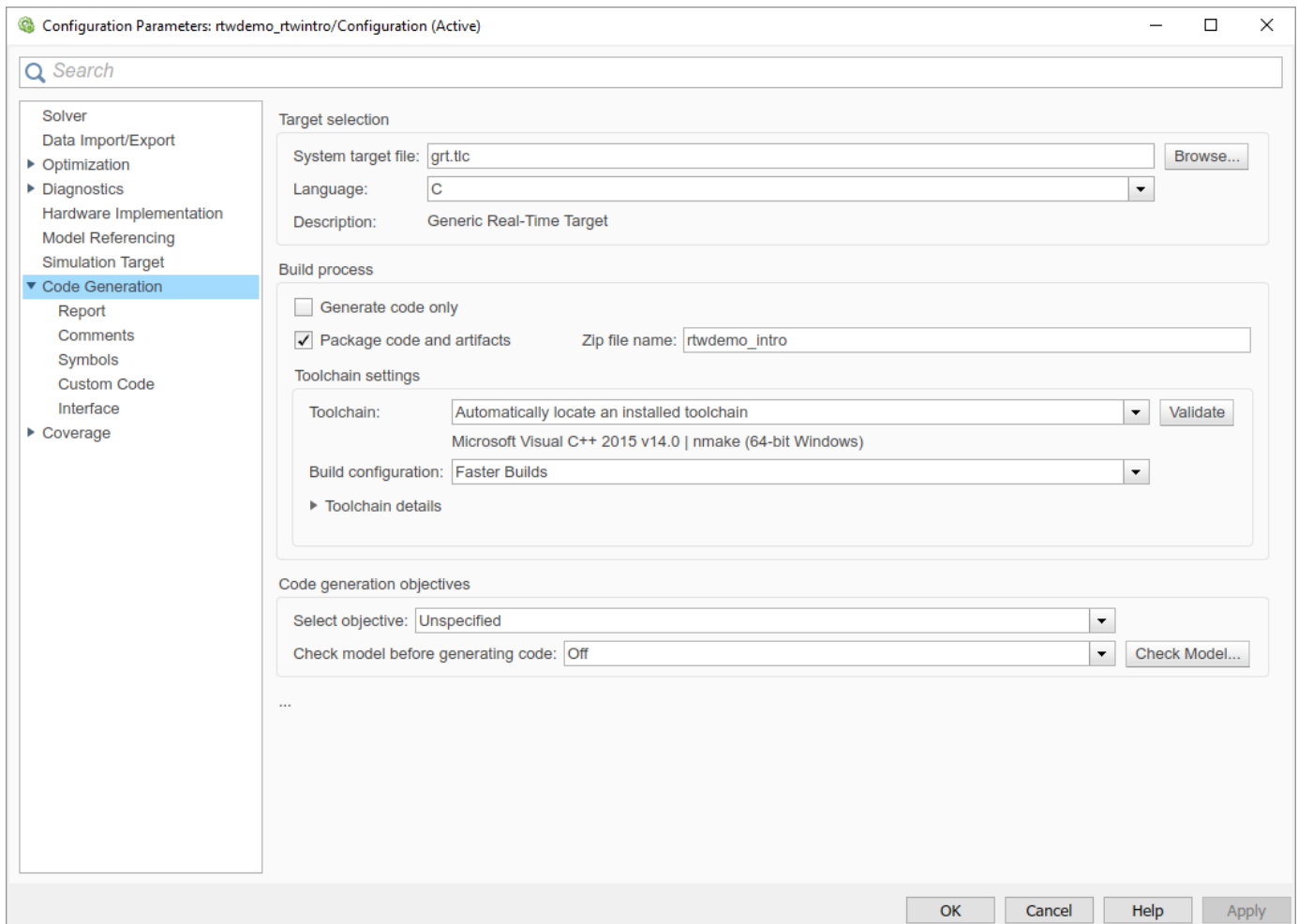
Relocate or Share Generated Code

To relocate or share static and generated code files, you can use the `packNGo` function. For example, use the function when you want to relocate generated code to a development environment that does not provide MATLAB and Simulink products.

Package and Relocate Generated Code

To package generated code:

- 1 In the Configuration Parameters dialog box, select **Package code and artifacts**. This option configures the build process to run the `packNGo` function after code generation.
- 2 In the **Zip file name** field, enter the name of the ZIP file in which you want to package the generated code and artifacts. You can specify the file name with or without the `.zip` extension. If you do not specify an extension or an extension other than `.zip`, the ZIP utility adds the `.zip` extension. If you do not specify a value, the build process uses the name `model.zip`, where `model` is the name of the top model.



- 3 Apply the changes and then generate code for your model.

- 4 Verify that the packaged code is ready for relocation by using a ZIP tool to inspect the resulting ZIP file.

Relocate the ZIP file to the destination development environment and use a ZIP tool to unpack the file. To compile the unpacked code, use a build tool.

If the destination environment provides Simulink Coder or Embedded Coder, you can use `codebuild` to compile the unpacked code.

Package and Relocate Generated Code From Command Line

To package and relocate generated code by using line commands, use this workflow:

- 1 Select a structure for the zip file. on page 31-57
- 2 Select a name for the zip file. on page 31-58
- 3 Package the model code files in the zip file. on page 31-58
- 4 Inspect the generated zip file. on page 31-59
- 5 Relocate and unpack the zip file. on page 31-59

Select a Structure for the ZIP File

Before you generate and package the files for a model build, decide whether you want the files to be packaged in a flat or hierarchical folder structure. By default, the `packNGo` function packages the files in a single, flat folder structure.

If...	Then Use a...
You are relocating files to an IDE that does not use the generated makefile, or the code is not dependent on the relative location of required static files	Single, flat folder structure
The destination development environment must maintain the folder structure of the source environment because it uses the generated makefile, or the code depends on the relative location of files	Hierarchical structure

If you use a hierarchical structure, the `packNGo` function creates two levels of zip files, a primary zip file, which in turn contains the following secondary zip files:

- `mlrFiles.zip` — files in your *matlabroot* folder tree
- `sDirFiles.zip` — files in and under your build folder where you initiated code generation for the model
- `otherFiles.zip` — required files not in the *matlabroot* or *start* folder trees

Paths for the secondary zip files are relative to the root folder of the primary zip file, maintaining the source development folder structure.

Select a Name for the Zip File

By default, the `packNGo` function names the primary zip file `model`. You have the option of specifying a different name. If you specify a file name and omit the file type extension, the function appends `.zip` to the name that you specify.

Package Model Code in a Zip File

Package model code files by using the `PostCodeGenCommand` configuration parameter, `packNGo` function, and build information object for the model. You can set up the packaging operation to use:

- A system generated build information object.

In this case, before generating the model code, use `set_param` to set the configuration parameter `PostCodeGenCommand` to an explicit call to the `packNGo` function. For example:

```
set_param(bdroot, 'PostCodeGenCommand', 'packNGo(buildInfo);');
```

After generating and writing the model code to disk and before generating a makefile, this command instructs the build process to evaluate the call to `packNGo`. This command uses the system generated build information object for the currently selected model.

- A build information object that you construct programmatically.

In this case, you could use other build information functions to include paths and files selectively in the build information object that you then specify with the `packNGo` function. For example:

```
.
.
.
myModelBuildInfo = RTW.BuildInfo;
addSourceFiles(myModelBuildInfo, {'test1.c' 'test2.c' 'driver.c'});
.
.
.
packNGo(myModelBuildInfo);
```

The following examples show how you can change the default behavior of `packNGo`.

To...	Specify...
Change the structure of the file packaging to hierarchical	<code>packNGo(buildInfo, 'packType' 'hierarchical');</code>
Rename the primary zip file	<code>packNGo(buildInfo, 'fileName' 'zippedsrcs');</code>
Change the structure of the file packaging to hierarchical and rename the primary zip file	<code>packNGo(buildInfo, 'packType' 'hierarchical'... 'fileName' 'zippedsrcs');</code>
Include header files found on the include path in the zip file	<code>packNGo(buildInfo, 'minimalHeaders' false);</code>
Generate warnings for parse errors and missing files	<code>packNGo(buildInfo, 'ignoreParseError' true... 'ignoreFileMissing' true);</code>

Note The `packNGo` function can modify the build information in the `RTW.BuildInfo` object that is passed as the first argument of the function. The function might find additional files from source and include paths recorded in build information for the model and add that information to the `RTW.BuildInfo` object.

Inspect Generated ZIP File

To verify that the generated ZIP file is ready for relocation, use a ZIP tool to inspect the file. Some ZIP tools allow you to view the file contents without unpacking the file. If unpacking is required and the model code files are packaged as a hierarchical structure, you have to unpack the primary and secondary ZIP files. When you unpack the secondary ZIP files, relative paths of the files are preserved.

Relocate and Unpack ZIP File

Relocate the ZIP file to the destination development environment and use a ZIP tool to unpack the file. To compile the unpacked code, use a build tool.

If the destination environment provides Simulink Coder or Embedded Coder, you can use `codebuild` to compile the unpacked code.

Code Packaging Example

This example shows how to package code files generated for the example model `rtwdemo_rtwinintro`:

- 1 Set your working folder to a writable folder.
- 2 Open the model `rtwdemo_rtwinintro` and save a copy to your working folder.
- 3 In the Command Window, enter:


```
set_param('rtwdemo_rtwinintro', 'PostCodeGenCommand',...
'packNGo(buildInfo, 'packType' 'hierarchical'));
```
- 4 Generate code for the model.
- 5 Inspect the generated ZIP file, `rtwdemo_rtwinintro.zip`. The ZIP file contains two secondary ZIP files, `mLrFiles.zip` and `sDirFiles.zip`.
- 6 Inspect `mLrFiles.zip` and `sDirFiles.zip`.
- 7 Relocate the ZIP file to your destination folder and unpack it.
- 8 To compile the unpacked code, use the `codebuild` function.

Limitations

packNGo Function

For information about limitations that apply to this function, see `packNGo`.

Executable File with Nondefault Extension

If a build process uses the template makefile approach, then `packNGo` uses the executable file extension specified by the linker tool to determine binary artifacts that require packaging.

If you generate an executable file with an extension that is not a default value, check that the extension is saved in the toolchain associated with the template makefile. For more information, see “Associate the Template Makefile with a Toolchain” on page 31-26.

If the build process generates an executable file with an extension that is different from the extension saved in the toolchain, `packNGo` does not package the executable file.

See Also

packNGo | codebuild

More About

- “Approaches for Building Code Generated from Simulink Models” on page 31-11
- “Configure CMake Build Process” on page 31-13
- “Deploy Component Algorithm as Component Model Library by Using CMake” (Embedded Coder)
- “Compile Code in Another Development Environment” on page 24-93
- “Build Process Workflow for Real-Time Systems” on page 31-39
- “Executable Program Generation” on page 31-62

Compile and Debug Generated C Code with Microsoft Visual Studio

This example shows how to generate code from a model and produce a Visual Studio® Solution. For the base example, see `rtwdemo_counter`.

About This Example

Building the example model generates a Visual Studio Solution. These model configuration parameters control the code generation:

- **System target file**
- **Toolchain**

When you build the model with this configuration, Simulink® Coder™ generates code in a Visual Studio Solution. Add this solution to a Visual Studio C/C++ project to integrate generated code with your custom Visual Studio code.

Before You Start

- This example builds code for Microsoft® Visual Studio running on Microsoft Windows® platforms.
- Simulink Coder uses the GRT code format, which is intended for rapid prototyping.
- Embedded Coder® uses the ERT code format, which is intended for production deployment.

Example Steps

- 1 Open the example model `rtwdemo_counter_msvc`. In the Command Window, type:
`open_system('rtwdemo_counter_msvc');`
- 2 Set **System target file** to `grt.tlc` (Generic Real-Time Target).
- 3 Set **Toolchain** to a value of the form `Microsoft Visual Studio Project <year> | CMake (64-bit Windows)` that specifies an installed version of Microsoft Visual Studio. For example, `Microsoft Visual Studio Project 2019 | CMake (64-bit Windows)`.
- 4 From the **Apps** gallery, open the Simulink Coder app.
- 5 To generate code and build a program executable, on the **C Code** tab, click **Build**.
- 6 When the build process is complete, in the `rtwdemo_counter_msvc_grt_rtw > build` folder, you see a generated Visual Studio Solution.
- 7 In Microsoft Visual Studio, open the `rtwdemo_counter_msvc.sln` solution file.

Use the solution to build and debug the generated code in Visual Studio.

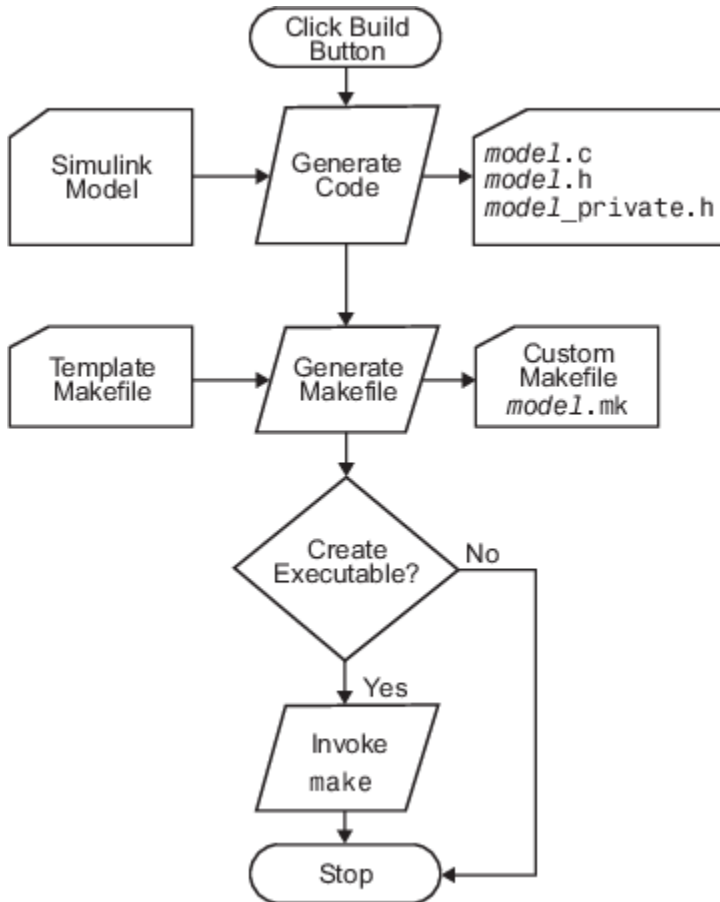
See Also

Related Examples

- “Choose an External Code Integration Workflow” on page 30-2

Executable Program Generation

The following figure shows how the process for program building.



During the final stage of processing, the build process invokes the generated makefile, *model.mk*, which in turn compiles and links the generated code. On PC platforms, a batch file is created to invoke the generated makefile. The batch file sets up the environment for invoking the make utility and related compiler tools. To avoid recompiling C files, the make utility performs date checking on the dependencies between the object and C files; only out-of-date source files are compiled. Optionally, the makefile can download the resulting executable image to your target hardware.

This stage is optional, as illustrated by the control logic in the preceding figure. You could choose to omit this stage (for example, if you are targeting an embedded microcontroller board).

To omit this stage of processing, in the Configuration Parameters dialog box, select the **Generate code only** check box. You can then cross-compile your code and download it to your target hardware.

If you also select the **Create code generation report** check box, the code generator produces an HTML report. To view the report, on the **C Code** tab, select **Open Latest Report**. This example is a code generation report for a generic real-time (GRT) system target file.

Code Generation Report for 'rtwdemo_rtwintr0'

Contents

- [Summary](#)
- [Subsystem Report](#)
- [Code Interface Report](#)

Generated Code

- [-] Model files**
 - [rtwdemo_rtwintr0.c](#)
 - [rtwdemo_rtwintr0.h](#)
 - [rtwdemo_rtwintr0_private.h](#)
 - [rtwdemo_rtwintr0_types.h](#)
- [+] Shared files (2)**
- [+] Interface files (1)**

Model Information

Author	The MathWorks, Inc.
Last Modified By	The MathWorks, Inc.
Model Version	1.256
Tasking Mode	MultiTasking

[Configuration settings at time of code generation](#)

Code Information

System Target File	grt.tlc
Hardware Device Type	Intel->x86-64 (Windows64)
Simulink Coder Version	8.13 (R2017b) 22-May-2017
Timestamp of Generated Source Code	Tue Jun 20 15:37:35 2017
Location of Generated Source Code	C:\work\my_rtwdemo_rtwintr0\rtwdemo_rtwintr0_grt_rtw\
Type of Build	Model
Objectives Specified	Unspecified

Additional Information

OK Help

See Also

More About

- “Approaches for Building Code Generated from Simulink Models” on page 31-11
- “Build Process Workflow for Real-Time Systems” on page 31-39

Host/Target Communication in Simulink Coder

- “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” on page 32-2
- “External Mode Simulation by Using XCP Communication” on page 32-8
- “Graphical Controls for XCP External Mode Simulations” on page 32-27
- “Set Up External Mode Connectivity Between Simulink and Target Hardware” on page 32-35
- “Register Custom Launcher” on page 32-43
- “Customize XCP Server Software” on page 32-45
- “External Mode Simulation with TCP/IP or Serial Communication” on page 32-55
- “Create a Transport Layer for TCP/IP or Serial External Mode Communication” on page 32-94

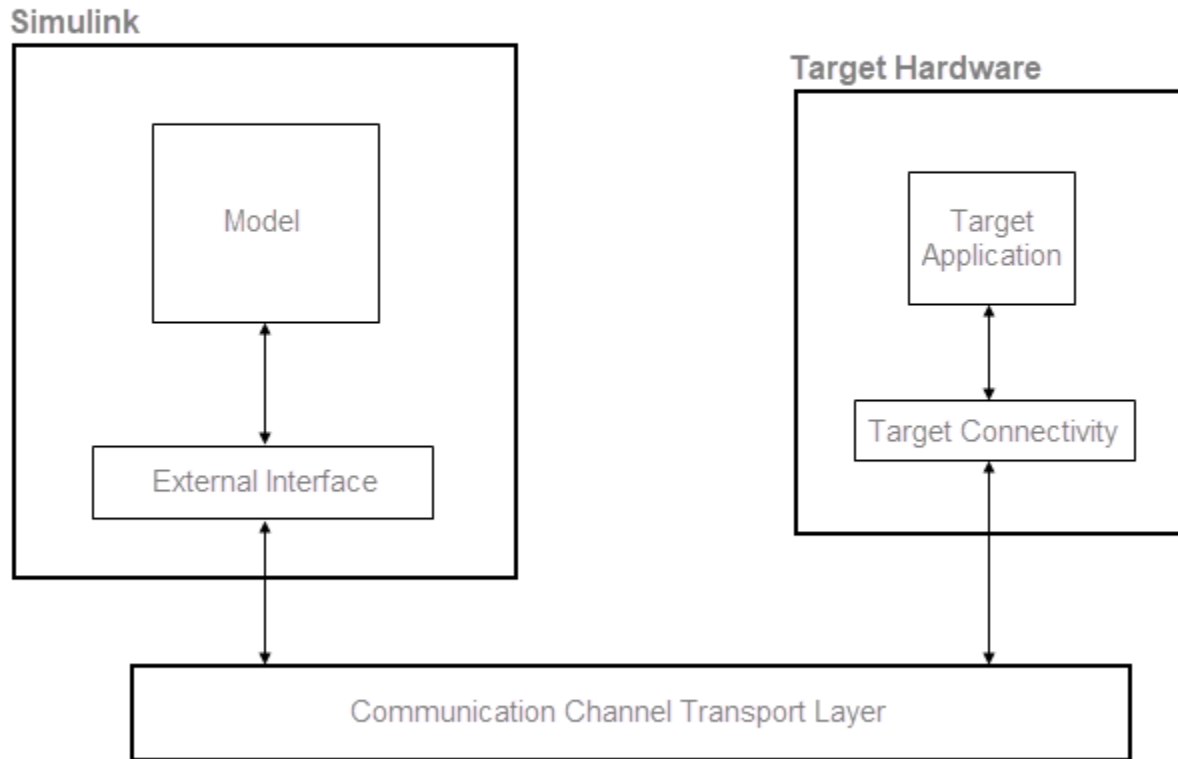
External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling

You can use external mode simulations for rapid prototyping. An external mode simulation establishes a communication channel between Simulink on your development computer (host) and the target hardware that runs the executable file created by the code generation and build process.

Through the communication channel, you can:

- Modify or tune block parameters in real time. When you change parameters in your model, Simulink downloads the new values to the executing target application.
- Monitor and save signal data from the executing target application.
- Retrieve execution-time metrics if code execution profiling is enabled for the external mode simulation. An Embedded Coder license is required.

The low-level transport layer of the channel handles the transmission of messages. Simulink and the generated model code are independent of this layer. The transport layer and its interface code are isolated in separate modules that format, transmit, and receive messages and data packets.



Communication Mechanisms for External Mode

Simulink supports two communication mechanisms for external mode simulation. To decide which mechanism to use, see the information in this table.

Communication Mechanism	Supports	Reasons for Selecting Communication Mechanism
XCP, the Universal Measurement and Calibration Protocol	<p>ERT and GRT system target files.</p> <p>Some Simulink support packages. For details, see https://www.mathworks.com/hardware-support.html?q=&page=1.</p>	<p>XCP external mode:</p> <ul style="list-style-type: none"> • Uses a standard communication protocol. • Requires only a lightweight communication software stack on the target hardware. • Supports signal logging and streaming for Dashboard blocks and the Simulation Data Inspector. You can stream signals from within a referenced model hierarchy. • Provides documented <code>ext_mode.h</code> API for external mode target connectivity. • Supports streaming of execution-time metrics to the Simulation Data Inspector for host-based and support package external mode simulations. <p>For information about running XCP external mode simulations, see:</p> <ul style="list-style-type: none"> • “External Mode Simulation by Using XCP Communication” on page 32-8 • “Customize XCP Server Software” on page 32-45
TCP/IP and serial (RS-232)	<p>ERT, GRT, and RSim system target files.</p> <p>Simulink support packages.</p>	<p>XCP external mode simulations do not support your target hardware because of known limitations on page 32-21.</p> <p>For information about running TCP/IP and serial external mode simulations, see:</p> <ul style="list-style-type: none"> • “External Mode Simulation with TCP/IP or Serial Communication” on page 32-55 • “Create a Transport Layer for TCP/IP or Serial External Mode Communication” on page 32-94

Simulink Feature Support for External Mode

This table summarizes feature support for both forms of external mode simulations.

Feature		XCP Support	TCP/IP and Serial Support
Parameter tuning	With Dashboard blocks	Yes	Yes






Feature		XCP Support	TCP/IP and Serial Support
	Of tunable block parameters	Yes	Yes
Simulation Data Inspector		Yes. Includes signals within referenced models.	No
Logic Analyzer		Yes. Includes signals within referenced models.	No
Blocks that receive and display signals from target application	Dashboard library	Yes	No
	Floating Scope, Scope	Yes	Yes
	Spectrum Analyzer, Time Scope (DSP System Toolbox)	Yes	Yes
	Display	Yes	Yes
	To Workspace	Yes	Yes
	User-written S-Function. A method, which enables user-written blocks to support external mode, is built into the S-function API. See <code>matlabroot/simulink/include/simstruc.h</code> .	Yes	Yes
	Record, XY Graph	Yes	Yes
Signal Viewing Subsystem on page 32-74		Yes	Yes

External Mode Simulations by Using the Run on Custom Hardware App

To run external mode simulations on your development computer or target hardware that is not supported by MathWorks support packages, use the Run on Custom Hardware app. For an external mode simulation, you:

- 1 Build the target application on your development computer.
- 2 Deploy the target application to the target hardware.
- 3 Connect Simulink to the target application that runs on the target hardware.
- 4 Start execution of generated code on the target hardware.

With the app, you can perform the steps separately or by using one-click. This table shows the workflow support that the app provides for your target hardware.

Workflow Feature or Task		Target Hardware	
		Development Computer	Custom Hardware
One-click 		Yes	Yes, if you set up connectivity on page 32-35 between Simulink and your target hardware.
Step-by-step	Build for Monitoring 	Yes	Yes. Software builds target application on development computer by using the toolchain that you specify.
	Deploy 	Yes	Yes, if you set up connectivity on page 32-35 between Simulink and your target hardware.
	Connect 	Yes	Yes
	Start 	Yes	Yes

External Mode Control Panel

Using the External Mode Control Panel, you can perform some of the functions that the Run on Custom Hardware app provides, for example, **Connect**, **Start**, and **Disconnect**. You can also open the External Signal & Triggering dialog box, which enables you to:

- Select logged signals that you want to monitor.
- Configure a trigger for uploading target application data.

For details, see XCP panel on page 32-28 or TCP/IP and serial panel on page 32-61.

Security for External Mode Simulations

Ensure that access to your development computer and the target hardware is secure.

Provide a secure communication channel between your development computer and the target application by applying the security measures listed in this table.

Communication Protocol	Security Measure
XCP on TCP/IP, TCP/IP	Run your development computer and target application within a trusted private network or virtual private network (VPN).
XCP on serial, serial	Use a point-to-point link between your development computer and target application, for example, a serial cable.

Before generating code for production, to reduce the risk of unintended access to the target application, disable external mode settings.

See Also

More About

- “External Mode Simulation by Using XCP Communication” on page 32-8
- “Set Up External Mode Connectivity Between Simulink and Target Hardware” on page 32-35
- “Customize XCP Server Software” on page 32-45
- “External Mode Simulation with TCP/IP or Serial Communication” on page 32-55
- “Create a Transport Layer for TCP/IP or Serial External Mode Communication” on page 32-94
- “Create Execution-Time Profile for Generated Code” (Embedded Coder)

External Mode Simulation by Using XCP Communication

In an external mode simulation, you can tune parameters in real time and monitor target application signals. Using the Run on Custom Hardware app, you can set up and run on your development computer or custom hardware external mode simulations on page 32-2 that use an XCP communication channel.

For an external mode simulation, you:

- 1 Build the target application on your development computer.
- 2 Deploy the target application to the target hardware.
- 3 Connect Simulink to the target application that runs on the target hardware.
- 4 Start execution of generated code on the target hardware.

Run XCP External Mode Simulation on Development Computer

Configure and run an external mode simulation on page 32-2 that uses the XCP communication protocol. During the simulation:

- Monitor a signal by using a Scope block, a Dashboard block, and the Simulation Data Inspector.
- Tune a parameter by using a Dashboard block.

Configure Signal Monitoring and Parameter Tuning for XCP

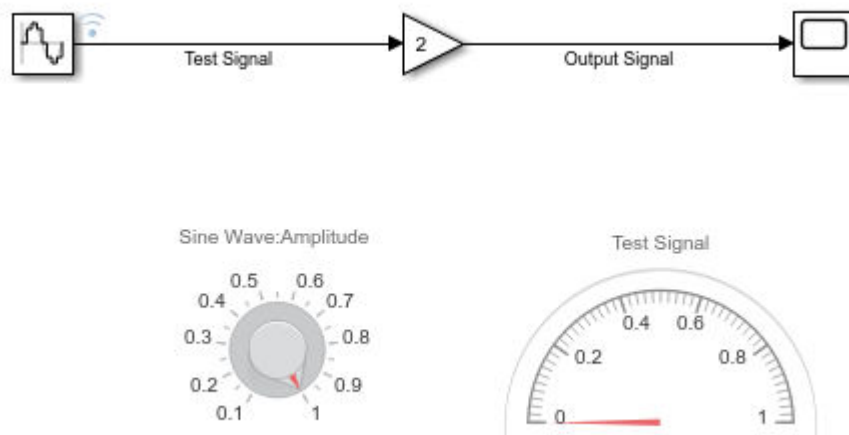
- 1 Create a folder for this example.

```
mkdir ext_mode_xcp_example
cd ext_mode_xcp_example
```

- 2 Open Simulink and create a simple model, `xcpExample`, which contains these blocks:
 - Sine Wave
 - Gain
 - Scope
 - Half Gauge
 - Knob
- 3 Double-click the Sine Wave block. Set **Sample time** to 0.1, and then click **OK**.
- 4 Double-click the Gain block. Set **Gain** to 2, and then click **OK**.
- 5 Connect the Sine Wave block to the Gain block and name the connection, for example, `Test Signal`.
- 6 Connect the Gain block to the Scope block and name the connection, for example, `Output Signal`.
- 7 Configure `Test Signal` for logging:
 - a Select `Test Signal`.
 - b On the **Signal** tab, in the **Monitor** section, select **Log Signals**.

If you do not enable signal logging, you cannot stream `Test Signal` to the Simulation Data Inspector.

- 8 Configure the Half Gauge block to monitor the value of Test Signal:
 - a Double-click the Half Gauge block.
 - b In the Simulink Editor, select Test Signal.
 - c In the Block Parameters dialog box:
 - Connect the block to Test Signal.
 - In the **Maximum** field, enter a value, for example, 1.
 - d Click **OK**.
- 9 Configure the Knob block to tune the **Amplitude** parameter of the Sine Wave block:
 - a Double-click the Knob block.
 - b In the Simulink Editor, select the Sine Wave block.
 - c In the Block Parameters dialog box:
 - Connect the block to the **Amplitude** parameter of the Sine Wave block.
 - In the **Minimum** and **Maximum** fields, enter values, for example, 0.1 and 1 respectively.
 - d Click **OK**.
- 10 Save the model as xcpExample.




Specify Hardware and Prepare Model


- 1 From the **Apps** tab on the Simulink toolstrip, in the **Setup to Run on Hardware** section, click **Run on Custom Hardware**. On the Run on Custom Hardware pop-up dialog box, click **Finish**.
- 2 In the **Hardware** section, specify the system target file. For this example, use the default, `grt.tlc`. If you have Embedded Coder, you can specify `ert.tlc`.
- 3 In the **Prepare** gallery, under **Model Configuration**, click **Hardware Settings**. The Configuration Parameters dialog box opens, displaying **Hardware Implementation** settings that are determined by the system target file.
 - a On the **Solver** pane, specify Fixed-step in the **Type** field. Then, under **Solver details**, specify 0.1 in the **Fixed-step size** field.

- b On the **Code Generation > Optimization** pane, set **Default parameter behavior** to Tunable.
 - c On the **Code Generation > Interface** pane, select the **External mode** check box. Then set **Transport layer** to XCP on TCP/IP. Selecting XCP on TCP/IP automatically specifies `ext_xcp` for **MEX-file name**, selects the **Automatically allocate static memory** check box, and enables **Maximum duration**. For this example, use the default value of **Maximum duration**.
 - d If **System target file** is `ert.tlc`, on the **Code Generation > Templates** pane select the **Generate an example main program** check box, and set **Target operating system** to BareBoardExample. The code generator uses the documented APIs of the external mode abstraction layer that are available in `matlabroot\toolbox\coder\xcp\src\target\ext_mode\include\ext_mode.h` to create an example that shows how you can provide XCP external mode target connectivity for a bare board that does not run an operating system.
 - e Click **OK** to apply the changes and close the dialog box.
- 4 Save the model.

Build and Run Target Application

In the **Run on Hardware** section:

- 1 To specify a nondefault value for the simulation stop time, in the **Stop Time** field, specify your value, for example, `inf`.
- 2 Click **Monitor & Tune**  to start the process. The software:
 - a Builds the target application files including the executable file. The debugging symbols for signals are created as part of the executable file or in a separate file, depending on the configuration of Code Generation. To learn more about different configurations for Code Generation, see “Approaches for Building Code Generated from Simulink Models” on page 31-11.
 - b Deploys the target application as a separate process on your development computer.
 - c Connects Simulink to the target application.
 - d Starts the generated model code.

To perform the steps separately, click the following buttons under **Monitor & Tune**  > **Step by Step Commands**, in this order:

- a **Build for Monitoring**
- b **Deploy**
- c **Connect**
- d **Start**

Monitor Signal and Tune Parameter

You can monitor Test Signal through the:

- Scope block -- Double-click the block.
- Simulation Data Inspector -- Click the Simulation Data Inspector button. When the Simulation Data Inspector opens, select the **Test Signal** check box, which displays streamed data.

- Half-Gauge block.

To change the amplitude of the sine wave, rotate the pointer on the Knob block to the required value.

This table describes more ways of tuning tunable block parameters during a simulation.

Approach	Details
Model Data Editor	<p>To tune parameters through the Model Data Editor:</p> <ol style="list-style-type: none"> 1 On the Hardware tab of the Simulink Editor, in the Prepare section, click Tune Parameters, which opens the Model Data Editor. 2 If you want to update multiple tunable parameters simultaneously, in the Prepare section, toggle on Hold Updates. 3 On the Parameters tab of the Model Data Editor, in the Value column, specify new values for your tunable parameters. 4 Toggle off Hold Updates or click Update All Parameters (Ctrl+D). Simulink downloads the new values to the target application simultaneously. <p>If Hold Updates is off, then immediately after you specify a new value, Simulink downloads the new value to the target application.</p> <p>For more information, see Model Data Editor.</p>

Approach	Details
Block Parameter dialog box	<p>To tune parameters through the Block Parameter dialog box:</p> <ol style="list-style-type: none"> 1 For each block that you want to update: <ol style="list-style-type: none"> a Double-click the block, which opens the Block Parameter dialog box. b In the parameter fields, specify new parameter values. c Click Apply or OK. 2 If Hold Updates is on, toggle off Hold Updates or click Update All Parameters (Ctrl+D). Simulink downloads the new values to the target application simultaneously. <p>If Hold Updates is off, then immediately after you click Apply or OK, Simulink downloads the new block values to the target application.</p>
MATLAB workspace	<p>If block parameters are MATLAB workspace variables:</p> <ol style="list-style-type: none"> 1 In the Command Window, assign new values to the variables. 2 On the Hardware tab of the Simulink Editor, in the Prepare section, click Update All Parameters (Ctrl+D). Simulink downloads the new values to the target application. <p>For more information, see “Create and Edit Variables”.</p>

For more information about parameter tuning with generated code, see “External Mode Simulation with TCP/IP or Serial Communication” on page 32-55 and “Create Tunable Calibration Parameter in the Generated Code” (Embedded Coder).

If your model contains a Stateflow chart, you can view state activity. For more information, see “Animate Stateflow Charts” (Stateflow).

Stop Target Application

To stop the execution and disconnect the target application before **Stop Time** is reached, in the **Run on Hardware** section of the **Hardware** tab, click **Stop** .

If you want to disconnect the target application from Simulink without stopping code execution, click **Stop** ▾ > **Step by Step Commands > Disconnect**.

Triggered Signal Monitoring for XCP External Mode Simulations

For XCP external mode simulations, you can configure a trigger that starts the uploading of target application data for signal monitoring in Simulink.

1 Create a simple model in a local folder.

In the Command Window, enter these commands:

```
% Create work folder
if mkdir('workFolder')
    rmdir('workFolder' 's
    mkdir('workFolder')
end
cd workFolder
open_system(new_system('myModel'));

% Add, configure, and connect blocks
add_block('simulink/Sources/Sine Wave','myModel/Signal',...
          'Bias','5',...
          'Amplitude','2',...
          'SampleTime','0.01');
set_param('myModel/Signal','position',[140,80,180,120]);

add_block('simulink/Math Operations/Gain','myModel/Signal Gain',...
          'Gain','2');
set_param('myModel/Signal Gain','position',[340,80,380,120]);
add_line('myModel','Signal/1','Signal Gain/1');

% Label signal
signal=get_param('myModel/Signal','PortHandles');
set_param(signal.Outport(1),'Name','test_signal');

% And so on
add_block('simulink/Math Operations/Product','myModel/Product');
set_param('myModel/Product','position',[540,160,580,200]);
add_line('myModel','Signal Gain/1','Product/1');

add_block('simulink/Sources/Sine Wave','myModel/Carrier',...
          'Bias','5',...
          'Amplitude','10',...
          'Frequency','3',...
          'SampleTime','0.02');

set_param('myModel/Carrier','position',[140,240,180,280]);

add_block('simulink/Math Operations/Gain','myModel/Carrier Gain');
set_param('myModel/Carrier Gain','position',[340,240,380,280]);
add_line('myModel','Carrier/1','Carrier Gain/1');

signal=get_param('myModel/Carrier','PortHandles');
set_param(signal.Outport(1),'Name','test_carrier');

add_block('simulink/Signal Attributes/Rate Transition',...
          'myModel/Rate Transition');
set_param('myModel/Rate Transition','position',[440,200,480,240]);
add_line('myModel','Carrier Gain/1','Rate Transition/1');
add_line('myModel','Rate Transition/1','Product/2');

add_block('simulink/Sources/Constant','myModel/Constant',...
          'Value','0');
set_param('myModel/Constant','position',[340,320,380,360]);

add_block('simulink/Logic and Bit Operations/Relational Operator',...
          'myModel/Relational Operator','Operator','>=');
set_param('myModel/Relational Operator','position',[540,300,580,340]);
add_line('myModel','Carrier Gain/1','Relational Operator/1');
add_line('myModel','Constant/1','Relational Operator/2');

signal=get_param('myModel/Carrier Gain','PortHandles');
```

```

set_param(signal.Outport(1), 'Name', 'pos_carrier');

add_block('simulink/Sinks/Scope', 'myModel/ScopeA');
set_param('myModel/ScopeA', 'position', [740, 160, 780, 200]);
add_line('myModel', 'Product/1', 'ScopeA/1');

signal=get_param('myModel/Product', 'PortHandles');
set_param(signal.Outport(1), 'Name', 'test_output');

add_block('simulink/Sinks/Scope', 'myModel/ScopeB');
set_param('myModel/ScopeB', 'position', [740, 300, 780, 340]);
add_line('myModel', 'Relational Operator/1', 'ScopeB/1');

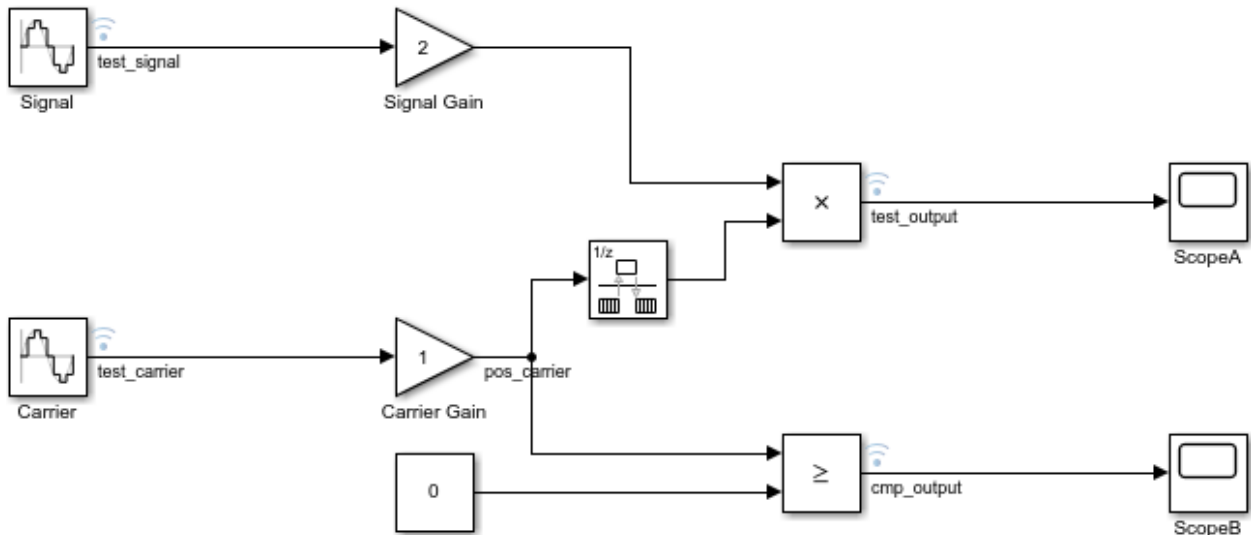
signal=get_param('myModel/Relational Operator', 'PortHandles');
set_param(signal.Outport(1), 'Name', 'cmp_output');

% Finally, save the model
save_system('myModel');

```

- In the Simulink Editor, configure these signals for logging by right-clicking each signal and then selecting **Log Selected Signals**:

- test_signal
- test_carrier
- test_output
- cmp_output



- From the **Apps** tab on the Simulink toolstrip, in the **Setup to Run on Hardware** section, click **Run on Custom Hardware**. On the Run on Custom Hardware pop-up dialog box, click **Finish**.
- In the **Prepare** gallery, under **Model Configuration**, click **Hardware Settings**.
- On the **Solver** pane, in the **Type** field, specify Fixed-step.
- On the **Code Generation** > **Interface** pane, select the **External mode** check box.
- Set **Transport layer** to XCP on TCP/IP, which specifies ext_xcp for **MEX-file name**.
- Click **OK**. Then save the model.
- To open the External Signal & Triggering dialog box, from the **Prepare** gallery, under **Signal Monitoring & Tracing**, click **Control Panel**.

- 10** In the **Configuration** section, click **Signal & Triggering**, which opens the External Signal & Triggering dialog box.

Signal selection

Trigger	Selected	Signal	Source Port
	X	test_carrier	myModel/Carrier:1
	X	test_output	myModel/Product:1
	X	cmp_output	myModel/Relational Operator:1
	X	test_signal	myModel/Signal:1

Select all
 Clear all
 on
 off
 Trigger Signal
 Go To Signal

Trigger options

Source: Mode: Duration: Delay:

Arm when connecting to target Send multiple contiguous samples in same packet

Trigger signal

Direction: Level: Hold-off:

OK Cancel Help Apply

Through this dialog box, you can select signals for monitoring and configure the trigger for uploading data from the target application. For this example, use the default settings.

- The **Select all** check box is selected — The dialog box displays an X in each row of the **Selected** column. You can monitor all logged signals during the simulation.
 - **Source** is set to **manual** -- The External Mode Control Panel provides manual control of data uploading, for example, the **Arm Trigger** button.
 - The **Arm when connecting to target** check box is selected — When you connect Simulink to the target application, the trigger is armed.
- 11** Click **OK** to close the External Signal & Triggering dialog box, then click **OK** to close the External Mode Control Panel.
- 12** In the Simulink Editor, go to the **Hardware** tab.
- 13** In the **Stop Time** field of the **Run on Hardware** section, specify your value, for example, `inf`.
- 14** Save the model.
- 15** Click **Monitor & Tune** to start the process. The software:
- a Builds the target application.

- b** Deploys the target application as separate process on your development computer.
- c** Connects Simulink to the target application.
- d** Starts the generated model code.

To view the scope displays, double-click ScopeA and ScopeB.

To monitor signals in the Simulation Data Inspector, in the **Review Results** section, click **Data Inspector**.

From the External Mode Control Panel, you can suspend or resume signal logging and monitoring. In the **Connection and triggering** section, click **Cancel Trigger** or **Arm Trigger** respectively.

To stop the execution of the target application and disconnect Simulink from the target environment, click **Stop Real-Time Code**.

For information about signal monitoring and trigger options, see “XCP External Mode Control Panel” on page 32-28 and “XCP External Signal & Triggering Dialog Box” on page 32-29.

Run XCP External Mode Simulation From Command Line

You can use commands to run XCP external mode simulations. To retrieve and set the values of model parameters, use the `get_param` and `set_param` commands.

To run these commands, you must have a Simulink model open and a target application running.

- 1 Set the model simulation mode to external mode.

```
set_param(gcs, 'SimulationMode', 'external');
```

- 2 Connect Simulink to the target application.

```
set_param(gcs, 'SimulationCommand', 'connect')
```

- 3 Run generated model code.

```
set_param(gcs, 'SimulationCommand', 'start');
```

- 4 To tune a parameter, change its workspace variable value through a line command. For example, if a block parameter value is specified as a `Simulink.Parameter` object, assign the new value to the `Value` property.

```
myParamObj.Value = 5.23;
```

- 5 To download the new value to the target application, update the model.

```
set_param(gcs, 'SimulationCommand', 'update');
```

- 6 Stop the target application and disconnect Simulink from the target environment.

```
set_param(gcs, 'SimulationCommand', 'stop');
```

To disconnect Simulink from the target application without stopping execution of generated code, use this command:

```
set_param(gcs, 'SimulationCommand', 'disconnect');
```

The `set_param` commands that use the `'SimulationCommand'` argument are asynchronous. If you run the commands successively from a script, each command starts without waiting for the previous

command to complete. To check that each command is complete, in the script, use the `get_param` command with the 'SimulationStatus' argument. For example, for steps 1 to 3, specify these commands in the script:

```
set_param(gcs, 'SimulationMode', 'external');
set_param(gcs, 'SimulationCommand', 'connect');

isExternalSimulationActive = false;
while ~isExternalSimulationActive
    simStatus = get_param(gcs, 'SimulationStatus');
    isExternalSimulationActive = strcmp(simStatus, 'external');
end
set_param(gcs, 'SimulationCommand', 'start');
```

For more information, see “Run Simulations Programmatically”.

The Diagnostic Viewer displays error messages produced by the `get_param` and `set_param` commands.

This table gives parameters that you can use in `get_param` and `set_param` commands.

XCP External Mode Command-Line Parameters

Parameter and Values	Dialog Box Equivalent	Description
ExtModeArmWhenConnect off, on	External Signal & Triggering: Arm when connecting to target check box	Arm the trigger as soon as the Simulink Coder software connects to the target.
ExtModeBatchMode off, on	External Mode Control Panel: Batch download check box	Enable or disable downloading of parameters in batch mode.
ExtModeConnected off, on	External Mode Control Panel: Connect/Disconnect button	Indicate the state of the connection with the target application.
ExtModeEnableFloating off, on	External Mode Control Panel: Enable data uploading check box	Enable or disable the arming and canceling of triggers when a connection is established with floating scopes.
ExtModeLogAll off, on	External Signal & Triggering: Select all check box	Upload available signals from the target to the host.
ExtModeSendContiguousSamples off, on	External Signal & Triggering: Send multiple contiguous samples in same packet check box	Specify XCP DAQ packed mode for streaming signals to Simulink. Target application sends samples from multiple time steps in a packet. ExtModeTrigDuration determines number of samples sent in the packet.
ExtModeTrigDelay integer (0)	External Signal & Triggering: Delay text field	Specify the amount of time (expressed in base rate steps) that elapses between a trigger occurrence and the start of data collection.
ExtModeTrigDirection character vector - rising , falling, either	External Signal & Triggering: Direction menu	Specify the direction in which the signal must be traveling when it crosses the threshold value.

Parameter and Values	Dialog Box Equivalent	Description
ExtModeTrigDuration <i>integer (1000)</i>	External Signal & Triggering: Duration text field	Specify the number of base rate steps for which external mode logs data after a trigger event. If ExtModeSendContiguousSamples is on, the number of samples sent in a packet from the target application to Simulink is equal to: $\text{ExtModeTrigDuration}/n$ where $n = (\text{base rate})/(\text{sample rate})$ For $n=1$, number of samples sent in a packet is exactly ExtModeTrigDuration value. For subrates in multirate models, number of samples sent in a packet is rounded down to the nearest integer. The target application discards buffer samples that are not sent to Simulink.
ExtModeTrigHoldOff <i>integer (0)</i>	External Signal & Triggering: Hold-off text field	Specify the base rate steps between when a trigger event terminates and the trigger is rearmed.
ExtModeTrigLevel <i>integer (0)</i>	External Signal & Triggering: Level text field	Specify the threshold value the trigger signal must cross to fire the trigger.
ExtModeTrigMode <i>character vector - normal, oneshot</i>	External Signal & Triggering: Mode menu	Specify whether the trigger is to rearm automatically after each trigger event or whether only one buffer of data is to be collected each time the trigger is armed.
ExtModeTrigSignalBlockPath <i>character vector</i>	External Signal & Triggering: In Signal selection view, select signal and then click Trigger Signal button.	Specify the path to the block that contains the output port that provides the trigger signal.
ExtModeTrigSignalOutputPortIndex <i>integer</i>	External Signal & Triggering: In Signal selection view, select signal and then click Trigger Signal button.	Specify the index of the trigger signal output port for the block specified by ExtModeTrigSignalBlockPath.

Parameter and Values	Dialog Box Equivalent	Description
ExtModeTrigType <i>character vector</i> - <code>manual</code> , <code>signal</code>	External Signal & Triggering: Source menu	Specify whether to start logging data when the trigger is armed or when a specified trigger signal satisfies trigger conditions.

Memory Allocation for Communication Buffers During XCP External Mode Simulation

You do not have to manually determine memory requirements for XCP-based external mode simulations with models that, for example, stream a large amount of data to Simulink or run on low-memory target devices.

Setting **Transport layer** to `XCP` on `TCP/IP` or `XCP` on `Serial` enables these configuration parameters:

- **Automatically allocate static memory** (ExtModeAutomaticAllocSize)
- **Maximum duration** (ExtModeMaxTrigDuration) — Default is 10 base-rate steps.

The software uses model information, including the **Maximum duration** value, to allocate memory for:

- Internal data structures used by the XCP stack
- Storage of XCP packets

If you clear the **Automatically allocate static memory** check box:

- Memory allocated for internal data structures is a default value.
- You can use the **Static memory buffer size** (ExtModeStaticAllocSize) field to specify memory allocated for XCP packets. The default value is 1,000,000 bytes.

If insufficient memory is allocated, the external mode simulation issues an out-of-memory error when it tries to reserve memory for internal data structures or signal streaming. In this case, reselect the **Automatically allocate static memory** check box to enable the software to compute the amount of memory required.

Target Application Arguments

You can run your target application with optional arguments. You can pass the optional arguments to your target application through your custom launcher implementation.

Argument	Description
-w	Specify that the target application enters and stays in a wait state until it receives a message from Simulink. If you do not specify -w, the target application executes model code immediately. The model code runs with parameter values from the time when you built the model.

Argument	Description
<code>-tf <i>time</i></code>	<p>Override the model parameter <code>StopTime</code>.</p> <p>Specify <i>time</i> in:</p> <ul style="list-style-type: none"> • Seconds if <code>PurelyIntegerCode</code> is set to 'off'. • Base rate steps if <code>PurelyIntegerCode</code> is set to 'on'. <p><code>-tf inf</code> specifies that the model runs indefinitely when model code is executed.</p>

For host-based external mode simulations, you can specify additional `rtiostream` (Embedded Coder) arguments.

Argument	Description
<code>-verbose <i>level</i></code>	<p>Specify verbosity level:</p> <ul style="list-style-type: none"> • 0 -- No information • 1 -- Detailed information
<code>-port <i>number</i></code>	<p>For XCP on TCP/IP transport layer, specify the port number of the TCP/IP server. Use an integer value between 256 and 65535. Default is 17725.</p> <p>For XCP on Serial transport layer, specify the serial port ID. For example:</p> <p>On Windows, 'COM1' or 1 for COM1, 'COM2' or 2 for COM2, and so on.</p> <p>On Linux, '/dev/ttyS0', and so on.</p>
<code>-baud <i>value</i></code>	<p>For XCP on Serial transport layer, specify the baud value: 1200, 2400, 4800, 9600, 14400, 19200, 38400, 57600 (default), or 115200.</p>

XCP External Mode Limitations

This table describes limitations that apply to external mode simulations that use XCP communication.

Feature	Details
Parameter updates that change model structure	<p>You cannot change:</p> <ul style="list-style-type: none"> • The number of states, inputs, or outputs of a block • The sample time or the number of sample times • The integration algorithm for continuous systems • The name of the model or of a block <p>If you make parameter updates that change the model structure, you must rebuild the target application.</p> <p>You can change numerator and denominator polynomial parameters for the Transfer Fcn, Discrete Transfer Fcn, and Discrete Filter blocks if the number of states does not change.</p> <p>You cannot change zero entries in the State-Space, Zero-Pole, and Discrete Zero-Pole blocks in the user-specified or computed parameters, that is, the A, B, C, and D matrices obtained by a zero-pole to state-space transformation.</p> <p>In the State-Space block, if you specify the matrices in the controllable canonical realization, then changes to the A, B, C, and D matrices that preserve this realization and the dimensions of the matrices are allowed.</p> <p>If the Simulink block diagram does not match the target application, Simulink produces an error stating that the checksums do not match. The checksums take into account the top models, but not referenced models. To rebuild the target application, use the updated block diagram.</p>
Signal value display	<p>The graphical display of signal values during a simulation is not supported. For example, you cannot use the Data Display in Simulation menu items Show Value Labels When Hovering, Toggle Value Labels When Clicked, and Show Value Label of Selected Port. For more information, see “Display Signal Values in Model Diagrams”.</p>
Data archiving	<p>Data Archiving features available on the External Mode Control Panel are not supported. You can use the Simulation Data Inspector to collect and export data.</p>
Overriding signal logging settings	<p>If you use the Signal Logging Selector to override signal logging settings, the controls to override the Decimation and Limit data points settings are not supported.</p>

Feature	Details
Subsystem signal logging	<p>For external mode simulations, signal logging occurs in response to an event on the target hardware. Currently, the only supported logging event is a new sample time tick. Some subsystems are enabled or disabled by control signals. For these subsystems, the software logs signal values even when the control signals for the subsystems are not enabled.</p> <p>For example, if signals in an Enabled Subsystem block are configured for logging, the software logs the signal values even when the control signal for the subsystem is not greater than zero. In this case, the logged values are the values the signals had when the control signal was last greater than zero.</p> <p>For more information about the subsystems for which this limitation applies, see “Simulink Subsystem Semantics”.</p>
Compiler debug symbol format	<p>Your toolchain must generate debug information in one of these formats:</p> <ul style="list-style-type: none"> • DWARF • PDB
Global variables	<p>Signals, parameters, and states must be specified as global variables. The target memory locations where the variables are stored must lie in the range 0 - 4294967295.</p>
Parameter structures	<p>You cannot tune parameters that are structures.</p>
Pure integer code	<p>Pure integer code is supported. For code generation, if <code>PurelyIntegerCode</code> is 'on', specify <code>FixedStep</code> with a resolution that is greater or equal to 1 microsecond. For example, specify 1.000001, but not 1.0000001.</p> <p>If you do not specify <code>-tf finalTime</code> in the execution command, the target application runs the generated model code indefinitely, ignoring <code>StopTime</code>.</p> <p>If you specify <code>-tf finalTime</code> in the execution command:</p> <ul style="list-style-type: none"> • The <code>finalTime</code> value represents base rate steps, not seconds. • The maximum value for <code>finalTime</code>, in ticks, is <code>MAX_int32_T</code>.
Variable-size signals	<p>Uploading of variable-size signals is not supported.</p>
Compiler support	<p><code>lcc-win64</code> is not supported.</p>
Address granularity	<p>Target hardware that uses word addresses is not supported.</p>
Portable word sizes	<p>When the production target hardware is word-addressable, the generation of a host-based target application by using portable word sizes is not supported. The production target hardware must support 8-bit, 16-bit, and 32-bit native data types.</p>

Feature	Details
Scope blocks	If your model uses Scope blocks and the amount of data logged during an external mode simulation becomes very large, Simulink might slow down. Instead of Scope blocks, you can use the Simulation Data Inspector or equivalent blocks from the Dashboard library. For more information, see “Decide How to Visualize Simulation Data”.
Scope and Floating Scope blocks, and Scope Viewer	Some signal data types are not supported. The simulation produces a warning.
Floating Scope blocks and Scope Viewer	You cannot use Floating Scope blocks and Scope Viewers to monitor signals in referenced models during external mode simulations. To monitor referenced model signals, enable signal logging and use the Simulation Data Inspector.
Scopes in referenced models	In a model hierarchy, if the top model runs in external mode, scopes in the referenced models are not displayed. To monitor signals within a referenced model, enable signal logging and use the Simulation Data Inspector.
Scope and Display blocks connected to Simulink message blocks	<p>During external mode simulations, Scope and Display blocks connected to Simulink message blocks do not provide displays.</p> <p>To visualize messages, use the Simulation Data Inspector or Dashboard blocks.</p>
Nonzero simulation start time	Nonzero simulation start times are not supported. Use the default value for Solver > Start time , 0.0.
Intermediate step values	<p>Some Simulink blocks can generate multiple values in a simulation time step. For example:</p> <ul style="list-style-type: none"> • A Simulink Function or Function-Call Subsystem block that is invoked multiple times in a time step. • Iterator subsystem blocks that execute multiple times in a time step. <p>For each time step in an external mode simulation, Simulink uploads from the target application only the final values of such blocks. Simulink does not upload the intermediate values generated during the step.</p>
Service code interface	Using XCP communication for tuning parameters and monitoring signals as generated code executes in a target environment is not supported for code generated from component models configured with an ERT-based system target file and service code interface.
File-scoped data	<p>File-scoped data is not supported. For example, data items to which you apply the built-in custom storage class FileScope. The simulation produces a warning.</p> <p>For more information about the FileScope storage class, see “Choose Storage Class for Controlling Data Representation in Generated Code” (Embedded Coder).</p>
Row-major code generation	Code generated with the row-major format is not supported.

Feature	Details																																										
Concurrent execution	<p>On Windows and Linux computers, concurrent execution is supported.</p> <p>On Mac computers, concurrent execution using the native threads example is not supported. If the External mode and Allow tasks to execute concurrently on target check boxes are selected, you cannot build the target application unless you are using a target hardware support package that supports concurrent execution for external mode simulations.</p> <p>On a Windows computer, at run time, the base sample time in the target application is quantized to the nearest multiple of the system timer interrupt interval (usually 15.625 ms). Sampling subrates are the expected multiples of the base sample rate. Logged timestamps in the Simulation Data Inspector (SDI) correspond to simulation times rather than wall-clock times. As an example, this table shows the step times for a Simulink model that has a base sample time of 10 ms and a subrate sample time of 20 ms.</p> <table border="1" data-bbox="605 850 1466 1218"> <thead> <tr> <th colspan="3" data-bbox="605 850 1036 892">Base Rate</th> <th colspan="3" data-bbox="1036 850 1466 892">Subrate</th> </tr> <tr> <th data-bbox="605 892 678 1003">Step</th> <th data-bbox="678 892 824 1003">Wall-Clock Time (ms)</th> <th data-bbox="824 892 1036 1003">Logged Timestamp (ms)</th> <th data-bbox="1036 892 1109 1003">Step</th> <th data-bbox="1109 892 1255 1003">Wall-Clock Time (ms)</th> <th data-bbox="1255 892 1466 1003">Logged Timestamp (ms)</th> </tr> </thead> <tbody> <tr> <td data-bbox="605 1003 678 1045">0</td> <td data-bbox="678 1003 824 1045">0</td> <td data-bbox="824 1003 1036 1045">0</td> <td data-bbox="1036 1003 1109 1045">0</td> <td data-bbox="1109 1003 1255 1045">0</td> <td data-bbox="1255 1003 1466 1045">0</td> </tr> <tr> <td data-bbox="605 1045 678 1087">1</td> <td data-bbox="678 1045 824 1087">15.625</td> <td data-bbox="824 1045 1036 1087">10</td> <td data-bbox="1036 1045 1109 1087"></td> <td data-bbox="1109 1045 1255 1087"></td> <td data-bbox="1255 1045 1466 1087"></td> </tr> <tr> <td data-bbox="605 1087 678 1129">2</td> <td data-bbox="678 1087 824 1129">31.250</td> <td data-bbox="824 1087 1036 1129">20</td> <td data-bbox="1036 1087 1109 1129">1</td> <td data-bbox="1109 1087 1255 1129">31.250</td> <td data-bbox="1255 1087 1466 1129">20</td> </tr> <tr> <td data-bbox="605 1129 678 1171">3</td> <td data-bbox="678 1129 824 1171">46.875</td> <td data-bbox="824 1129 1036 1171">30</td> <td data-bbox="1036 1129 1109 1171"></td> <td data-bbox="1109 1129 1255 1171"></td> <td data-bbox="1255 1129 1466 1171"></td> </tr> <tr> <td data-bbox="605 1171 678 1218">4</td> <td data-bbox="678 1171 824 1218">62.500</td> <td data-bbox="824 1171 1036 1218">40</td> <td data-bbox="1036 1171 1109 1218">2</td> <td data-bbox="1109 1171 1255 1218">62.500</td> <td data-bbox="1255 1171 1466 1218">40</td> </tr> </tbody> </table>	Base Rate			Subrate			Step	Wall-Clock Time (ms)	Logged Timestamp (ms)	Step	Wall-Clock Time (ms)	Logged Timestamp (ms)	0	0	0	0	0	0	1	15.625	10				2	31.250	20	1	31.250	20	3	46.875	30				4	62.500	40	2	62.500	40
Base Rate			Subrate																																								
Step	Wall-Clock Time (ms)	Logged Timestamp (ms)	Step	Wall-Clock Time (ms)	Logged Timestamp (ms)																																						
0	0	0	0	0	0																																						
1	15.625	10																																									
2	31.250	20	1	31.250	20																																						
3	46.875	30																																									
4	62.500	40	2	62.500	40																																						
Simulation data export	<p>You can export logged signal and top-model output data to the MATLAB workspace:</p> <ul style="list-style-type: none"> • For logged signal data, select the Signal logging check box. • For top-model output data, select the Output check box and, from the Format list, select Dataset. <p>If you want to export the data to a MAT file instead of the MATLAB workspace, also select the Log Dataset data to file check box.</p> <p>The export of simulation state data is not supported.</p> <p>Saving the simulation output as a single object is not supported. The simulation produces a warning.</p>																																										

See Also

More About

- “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” on page 32-2

- “Mark Signals for Logging”
- “Set Up External Mode Connectivity Between Simulink and Target Hardware” on page 32-35
- “Customize XCP Server Software” on page 32-45
- Simulation Data Inspector

External Websites

- <https://www.asam.net/standards/detail/mcd-1-xcp/wiki/>

Graphical Controls for XCP External Mode Simulations


In this section...

“Equivalent Controls on Hardware Tab and External Mode Control Panel” on page 32-27

“XCP External Mode Control Panel” on page 32-28







“XCP External Signal & Triggering Dialog Box” on page 32-29



You can control an XCP external mode simulation through:

- The **Hardware** tab on the Simulink Editor toolbar. To display the **Hardware** tab, from the **Apps** tab on the Simulink toolstrip, click **Run on Custom Hardware**.
- The External Mode Control Panel. To open this dialog box, on the **Hardware** tab, in the **Prepare** section, click . Then, under **Signal Monitoring & Tracing**, click **Control Panel**.

Equivalent Controls on Hardware Tab and External Mode Control Panel

This table lists external mode actions that you can perform by using controls that are available in the **Hardware** tab and the External Mode Control Panel.

External Mode Action	Hardware Tab	External Mode Control Panel
Build target application.	Run on Hardware > Build for Monitoring 	N/A
Run application on target hardware.	Run on Hardware > Deploy 	N/A
Connect Simulink to a waiting or running target application.	Run on Hardware > Connect 	Connect
Start real-time execution of generated code in the target environment.	Run on Hardware > Start 	Start Real-Time Code
Disconnect Simulink from the target environment, but do not stop real-time execution of code.	Run on Hardware > Disconnect 	Disconnect
Stop target application execution and disconnect Simulink from the target environment. If you want to reestablish the external mode connection after stopping the target application, you must perform the Deploy step.	In Run on Hardware section, Stop button  .	Stop Real-Time Code

External Mode Action	Hardware Tab	External Mode Control Panel
Tune batch of block parameters. You can modify values of tunable parameters in Simulink, and then update the parameters in the target application simultaneously. See “Monitor Signal and Tune Parameter” on page 32-10.	In Prepare section, Hold Updates  and Update All Parameters  buttons.	Batch download and Download

XCP External Mode Control Panel

The External Mode Control Panel provides these controls for XCP external mode simulations.

Section	Control	Description
Connection and triggering	Connect	Connect Simulink to a waiting or running target application. When Simulink is connected to the target application, Connect changes to Disconnect .
	Start Real-Time Code	Start real-time execution of generated code in the target environment. When the generated code starts executing, the button changes to Stop Real-Time Code .
	Arm Trigger	Arm the trigger that you specify through the External Signal & Triggering dialog box. If Simulink is not connected to the target application, the button appears dimmed. If Simulink is connected to the target application and the trigger is armed, the button becomes Cancel Trigger .
	Cancel Trigger	Disarm the trigger. Once the trigger is disarmed, the button changes to Arm Trigger .
Floating scope	Enable data uploading	Selecting the check box arms the trigger for uploading target application data to Floating Scope blocks.
	Duration	Not supported.
Parameter tuning	Batch download	To tune a batch of block parameters: <ol style="list-style-type: none"> In the External Mode Control Panel, select Batch download. In the Simulink Editor, modify the required block parameters. When you modify parameters, the External Mode Control Panel displays this message next to Download: Parameter changes pending... Simulink stores the modified parameters locally. Click Download. Simulink downloads the batch of modified parameters to the target application.

Section	Control	Description
	Download	If the Batch download check box is not selected, the button appears dimmed.
Configuration	Signal & Triggering	Open the External Signal & Triggering dialog box.
	Data Archiving	XCP external mode simulations do not support this button, which appears dimmed.

XCP External Signal & Triggering Dialog Box

For an XCP external mode simulation, you can use the External Signal & Triggering dialog box to:

- Specify the logged signals that you want to monitor.
- Configure a trigger that starts the uploading of data from the target application.

The External Signal & Triggering dialog box provides these controls.

Section	Control	Description
Signal selection	Select all	The Signal selection view lists top-model and referenced-model signals that are configured for logging. Simulink uploads a logged signal for monitoring only if the signal is selected in the Signal selection view. By default, this check box is selected, which enables monitoring of all logged signals.
	Clear all	Deselect the signals that are selected for monitoring. If the Select all check box is selected, this button appears dimmed.
	on, off	If you select a row, clicking on selects the corresponding signal for monitoring. The Selected column displays the letter X. Clicking off deselects the signal. If the Select all check box is selected, these radio buttons appear dimmed.

Section	Control	Description
	<p>Trigger Signal</p>	<p>If you select a row, clicking this button sets the corresponding signal as the trigger for uploading target application data. The Trigger column displays the letter T. If the signal is already selected, clicking this button deselects the signal.</p> <p>If the configuration parameter <code>PurelyIntegerCode</code> is set to 'on', the trigger signal must be a non-complex, <code>int32</code> scalar.</p> <p>If the configuration parameter <code>PurelyIntegerCode</code> is set to 'off', the trigger signal must be a non-complex, <code>double</code> scalar.</p>
	<p>Go To Signal</p>	<p>To locate a signal in the model, select the corresponding row in the Signal selection view and then click this button.</p>
<p>Trigger options</p>	<p>Source</p>	<p>Specify how the trigger is armed:</p> <ul style="list-style-type: none"> • <code>manual</code> (default) — Arm the trigger by selecting the Arm when connecting to target check box or by clicking the Arm Trigger button on the External Mode Control Panel. • <code>signal</code> — Use specific characteristics of the trigger signal to start uploading data.

Section	Control	Description
	Mode	Specify trigger rearming after a trigger event is complete: <ul style="list-style-type: none"><li data-bbox="1065 386 1466 512">• normal (default) — The external mode simulation rearms the trigger after each trigger event.<li data-bbox="1065 527 1466 682">• one-shot— The external mode simulation collects only one buffer of data for each time that you arm the trigger.

Section	Control	Description
	<p>Duration</p>	<p>Specify the number of base rate steps for which the external mode simulation uploads data after a trigger event (default is 1000). For example, if Duration is set to 1000, and the base (fastest) rate of the model is one second:</p> <ul style="list-style-type: none"> • For a signal sampled at the base rate, one second (1.0 Hz), the external mode simulation collects 1000 contiguous samples during a trigger event. • For a signal sampled at two seconds (0.5 Hz), the external mode simulation collects 500 samples during a trigger event. <p>If Send multiple contiguous samples in same packet is also selected, on the target hardware, the target application:</p> <ol style="list-style-type: none"> 1 Stores data samples in a buffer. 2 Sends samples from multiple time steps in a packet. <p>The Duration value, say <code>durationFieldValue</code>, determines the number of samples sent in a packet:</p> $\text{numOfSamplesSent} = \text{durationFieldValue} / n$ <p>where $n = (\text{base rate}) / (\text{sample rate})$</p> <p>When the sample rate is the base rate ($n=1$), <code>numOfSamples</code> is exactly <code>durationFieldValue</code>. For subrates ($n=2, 3, \dots$), <code>numOfSamplesSent</code> is rounded down to the nearest integer. The target application discards</p>

Section	Control	Description
		buffer samples that are not sent to Simulink.
	Delay	<p>Apply a delay to data collection, which is the time between a trigger event and the start of data collection expressed in base rate steps. You can specify a positive value or zero (default).</p> <p>An XCP external mode simulation does not support a negative delay (pretriggering).</p>
	Arm when connecting to target	<p>This check box is selected by default. Connecting Simulink to the target application arms the trigger.</p> <ul style="list-style-type: none"> • If the trigger Source is manual, data uploading begins immediately. • If the trigger Source is signal, monitoring of the trigger signal begins immediately. Data uploading begins when the trigger signal satisfies trigger conditions (as defined in the Trigger signal section). <p>If you clear the check box, you can arm the trigger by clicking the Arm Trigger button on the External Mode Control Panel.</p>
	Send multiple contiguous samples in same packet	<p>This check box is not selected by default.</p> <p>If you select the check box, the XCP server uses the XCP DAQ packed mode for streaming signals to Simulink, which improves the ratio of useful data to control data in streamed signals.</p> <p>The target application sends samples from multiple time steps in a packet. The Duration value determines the number of samples in the packet.</p>

Section	Control	Description
Trigger signal	Direction	<p>To enable the uploading of data from the target application when the trigger signal crosses the threshold value (specified by Level), specify the required direction of the signal:</p> <ul style="list-style-type: none"> • rising (default) — Signal must be increasing in value. • falling — Signal must be decreasing in value. • either — Signal can be increasing or decreasing in value.
	Level	Specify the threshold value that the signal must cross to start the uploading of target application data. Default is 0.
	Hold-off	Specify, in base rate steps, the time between the termination of one trigger event and the rearming of the trigger. Applies only when Mode value is normal. Default is 0.

See Also

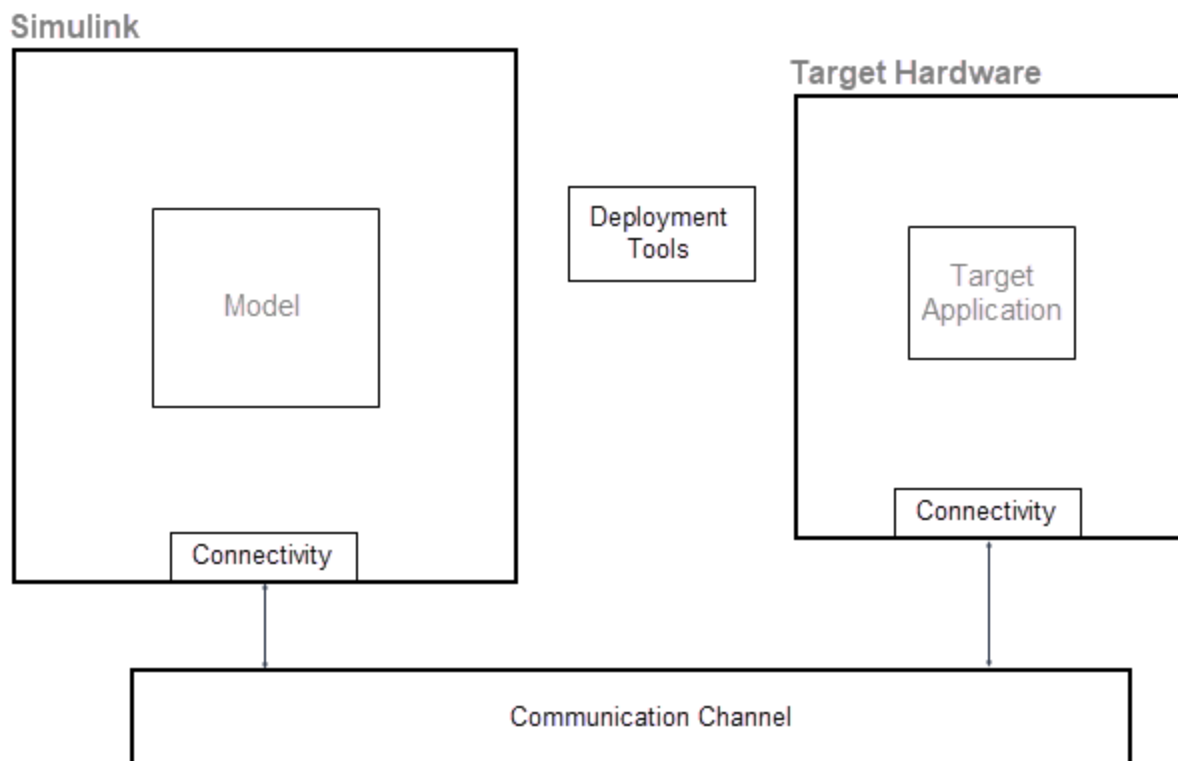
More About

- “External Mode Simulation by Using XCP Communication” on page 32-8
- “Triggered Signal Monitoring for XCP External Mode Simulations” on page 32-13

Set Up External Mode Connectivity Between Simulink and Target Hardware

For external mode simulations, you can use the `target` Package to provide connectivity between Simulink and your target hardware.

This diagram gives an overview of the components of an external mode simulation.



The `target` package provides classes for the implementation of components. This table lists the main classes.

Component	Class	Purpose
Target hardware	<code>target.Board</code>	Provide MATLAB with a description of target hardware.
Deployment tools	<code>target.SystemCommandExecutionTool</code>	You can use the classes to:

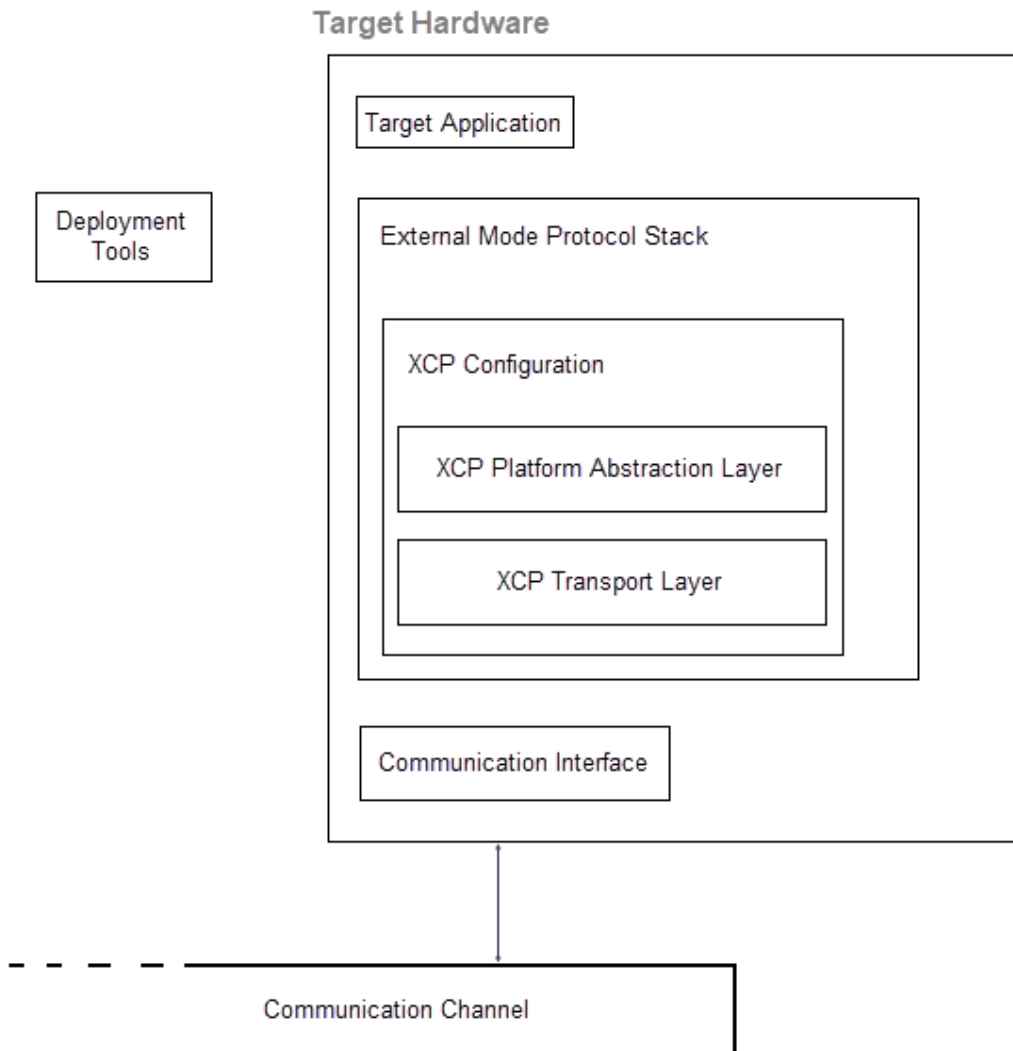
Component	Class	Purpose
	target.ExecutionService target.ExecutionTool	<ul style="list-style-type: none"> • Capture system commands for running the target application from your development computer. • Describe the execution service implementation for the target application. • Provide the MATLAB service interface for the tool that manages the target application execution. <p>To provide Monitor & Tune, Deploy, Connect, and Start functionality, the Run on Custom Hardware app requires the use of target.ExecutionTool.</p>
Connectivity	target.ExternalMode	Provide communication protocol for data transfer between Simulink and target hardware.
	target.CommunicationInterface	Provide target hardware with details of the communication channel and the rtiostream API (Embedded Coder) implementation.

Customise Connectivity for XCP External Mode Simulations

For code that is generated by using ERT (ert.tlc) and GRT (grt.tlc) system target files, you can run external mode simulations that use the XCP communication protocol:

- On your development computer.
- On other target hardware by using support packages.

If your system target file for custom target hardware is derived from the ERT or GRT system target files, use classes from the target Package to customise connectivity. For example, target.ExternalMode and target.CommunicationInterface.



This example shows how you can customise connectivity for XCP-based external mode simulations. To set up connectivity between Simulink and the target hardware:

1 Create board description

Create a `target.Board` object that provides MATLAB with a description of your target hardware. To support code generation, associate the object with a processor.

```
processor = target.get('Processor', 'Intel-x86-64 (Windows64)');
board = target.create('Board', ...
    'Name', 'Example Intel Board', ...
    'Processors', processor);
```

2 Choose deployment tool

Use one of these classes:

- `target.ExecutionTool` -- Provides the `getApplicationStatus` method, which is required by the Run on Custom Hardware app for **Monitor & Tune**, **Deploy**, **Connect**, and **Start** functionality.
- `target.SystemCommandExecutionTool` -- The `getApplicationStatus` method is not available with this class. The **Monitor & Tune**, **Deploy**, **Connect**, and **Start** functionality of the Run on Custom Hardware app is not fully supported.

In step 3 and step 4, the availability of a tool that can download the target application onto the target hardware is assumed.

3 Using `target.ExecutionTool`

Implement the MATLAB service interface in a class called `MyImplementationClass`. For an example pseudocode implementation, see “Execution Tool Template” on page 32-41.

```
classdef MyImplementationClass < target.ExecutionTool
    properties (Access=private)
        ...
    end

    methods
        ...
    end

    methods (Access=private)
        ...
    end
end
```

Create a `target.ExecutionService` object that contains details about executing the target application. The object describes the tool that is required to run the target application on the target hardware.

```
matlabExecution = ...
    target.create('ExecutionService', ...
        'Name', 'My Application Launcher');
matlabExecution.APIImplementation = ...
    target.create('APIImplementation', ...
        'Name', 'MyServiceImplementation');
matlabExecution.APIImplementation.BuildDependencies = ...
    target.create('MATLABDependencies');
matlabExecution.APIImplementation.BuildDependencies.Classes = ...
    {'MyImplementationClass'};
matlabExecution.APIImplementation.API = ...
    target.get('API', 'ExecutionTool');
```

Associate the `target.ExecutionService` object with the `target.Board` object.

```
board.Tools.ExecutionTools = matlabExecution;
```

4 Using `target.SystemCommandExecutionTool`

Create a `target.SystemCommandExecutionTool` object that captures system commands for running the target application from your development computer.


```

executionTool = target.create('SystemCommandExecutionTool', ...
    'Name', 'My Execution Tool');
executionTool.StartCommand.String = 'customDownloadTool';
executionTool.StartCommand.Arguments = {'$(EXE)'};

```

Associate the `target.SystemCommandExecutionTool` object with the `target.Board` object.

```
board.Tools.ExecutionTools = executionTool;
```

5 Create communication interface for target hardware

Create a `target.CommunicationInterface` object that provides the target hardware with details of the communication channel and the `rtiostream` API (Embedded Coder) implementation.

For the `rtiostream` API, use a `BuildDependencies` object to specify the source files that are compiled with the target application. This example uses the shipped TCP/IP `rtiostream` implementation source file, which enables you to run external mode simulations on your development computer. For your target hardware, provide your own `rtiostream` implementation.

```

buildDependencies = target.create('BuildDependencies', ...
    'SourceFiles', ...
    {fullfile('$(MATLAB_ROOT)', ...
        'toolbox', 'coder', ...
        'rtiostream', ...
        'src', ...
        'rtiostreamtcpip', ...
        'rtiostream_tcpip.c')});

apiImplementation = target.create('APIImplementation', ...
    'Name', 'TCP x86 RTIOStream Implementation', ...
    'API', target.get('API', 'rtiostream'), ...
    'BuildDependencies', buildDependencies);

tcpComms = target.create('CommunicationInterface', ...);
    'Name', 'Windows TCP Interface', ...
    'Channel', 'TCPChannel', ...
    'APIImplementations', apiImplementation);

```

To specify a TCP channel, this example uses the name-value argument `'Channel'`, `'TCPChannel'`. You can specify a serial communication channel by using the name-value argument `'Channel'`, `'RS232Channel'`.

Associate the `target.CommunicationInterface` object with the `target.Board` object.

```
board.CommunicationInterfaces = tcpComms;
```

6 Specify communication protocol stack

Create an object that provides the communication protocol for data transfer between Simulink and target hardware.

- a Use the `target.XCPPlatformAbstraction` class to utilize your custom implementation of the XCP platform abstraction layer. The layer provides a static memory allocator on page 32-50 and other platform abstraction functionality on page 32-50.

```

xcpPlatformAbstraction = target.create('XCPPlatformAbstraction', ...
    'Name', 'XCP Platform Abstraction');

xcpPlatformAbstraction.BuildDependencies.Defines = {'XCP_CUSTOM_PLATFORM'};
customPlatformAbstractionPath = 'pathToImplementationFolder';
xcpPlatformAbstraction.BuildDependencies.SourceFiles = ...
    {fullfile(customPlatformAbstractionPath, 'myXCPPlatform.c')};
xcpPlatformAbstraction.BuildDependencies.IncludePaths = ...
    {customPlatformAbstractionPath};

```

- b** Use the `target.XCPTCPIPTransport` class to implement the XCP transport layer, which transmits and receives messages from the communication medium according to ASAM specifications. In this example, the transport protocol is TCP/IP. For serial transport, replace `target.XCPTCPIPTransport` with `target.XCPSerialTransport`.

```
xcpTransport = target.create('XCPTCPIPTransport', ...
    'Name', 'XCP Transport');
```

- c** Create a `target.XCP` object that represents the XCP protocol stack for the target hardware.

```
xcpConfiguration = target.create('XCP', ...
    'Name', 'XCP Configuration', ...
    'XCPTransport', xcpTransport, ...
    'XCPPlatformAbstraction', xcpPlatformAbstraction);
```

- d** Create an object that represents the XCP external mode connectivity options that are available in the XCP protocol stack.

```
extModeConnectivity = ...
    target.create('XCPEXternalModeConnectivity', ...
        'Name', 'External Mode Connectivity', ...
        'XCP', xcpConfiguration);
```

- e** Create an external mode protocol stack object.

```
externalMode = target.create('ExternalMode', ...
    'Name', 'External Mode', ...
    'Connectivities', extModeConnectivity);
```

Note You can create the object in a single call to `target.create`.

```
externalMode = target.create('ExternalMode', ...
    'Name', 'External Mode', ...
    'XCPTransportLayer', 'TCP', ...
    'Defines', {'XCP_CUSTOM_PLATFORM'}, ...
    'SourceFiles', {fullfile('pathToImplementationFolder', 'myXCPPlatform.c')}, ...
    'IncludePaths', {'pathToImplementationFolder'});
```

- f** Add the object to the list of communication protocols that the target hardware supports.

```
board.CommunicationProtocolStacks = externalMode;
```

7 Make board object persist across MATLAB sessions

Add the board object to an internal database and make the object persist over MATLAB sessions.

```
target.add(board, 'UserInstall', true);
```

8 Select board for model

In the Configuration Parameters dialog box, set **Hardware board** to Example Intel Board.

9 Specify connection between Simulink and target hardware

In the Configuration Parameters dialog box:

- From the **Transport layer** list, select XCP on TCP/IP. The transport protocol is in the list because of the connectivity customization.
- In the **Mex-file arguments** field, enter 'localhost' 1 0. For more information, see MEX-file arguments.

Customize Connectivity for TCP/IP or Serial External Mode Simulations

For TCP/IP or serial external mode simulations, you can customize connectivity through a workflow that:

- Implements transport and communication protocols.
- Specifies the execution tool for the target application by using the `target` Package package.

To set up connectivity between Simulink and the target hardware, use the workflow described in “Customise Connectivity for XCP External Mode Simulations” on page 32-36 with these differences:

- After step 1, using the information in “Choose Communication Protocol for Client and Server” on page 32-78 and “Create a Transport Layer for TCP/IP or Serial External Mode Communication” on page 32-94, implement the client and server sides of external mode communication for TCP/IP or serial protocols.
- Do not perform step 5 and step 6.

Execution Tool Template

This section provides a pseudocode example for a `target.ExecutionTool` service interface. The tool starts and tracks an application on the target hardware.

```
classdef MyExecutionTool < target.ExecutionTool
    methods
        function errFlag = startApplication(this)
            % Call "customDownloadTool" to download the application.
            [status, result] = ...
                system(sprintf('customDownloadTool %s', this.Application));
            if status == 0
                errFlag = false;
            else
                disp(result);
                errFlag = true;
            end
        end

        function errFlag = stopApplication(~)
            % Add code here to stop the application, if possible.
            errFlag = false;
        end

        function [status, errFlag] = getApplicationStatus(~)
            % Add code here to return the application status, if known.
            status = target.ApplicationStatus.Unknown;
            errFlag = false;
        end
    end
end
```

See Also

`target` Package

Related Examples

- “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” on page 32-2
- “External Mode Simulation by Using XCP Communication” on page 32-8
- “Choose Communication Protocol for Client and Server” on page 32-78
- “Create a Transport Layer for TCP/IP or Serial External Mode Communication” on page 32-94
- “Set Up PIL Connectivity by Using `target` Package” (Embedded Coder)

External Websites

- <https://www.asam.net/standards/detail/mcd-1-xcp/wiki/>

Register Custom Launcher

Note The use of a custom launcher to set up connectivity between Simulink and your target hardware is not recommended. Use the `target` package instead. For more information, see “Set Up External Mode Connectivity Between Simulink and Target Hardware” on page 32-35

To support the one-click workflow or **Deploy** step for custom hardware, register your custom launcher for the target application.

- 1 Create an `rtw.connectivity.Config` subclass, which requires `myCustomLauncher`, a launcher that you provide. `myCustomLauncher` is a subclass of `rtw.connectivity.Launcher`. In `myCustomLauncher`, to support one-click functionality, provide an implementation of the `getApplicationStatus` method. For the communicator and builder subclasses, you can use dummy classes.

```
classdef myTargetConnectivityConfig < rtw.connectivity.Config
    methods
        % Constructor
        function this = myTargetConnectivityConfig(componentArgs)

            % Launcher with port 0 argument to start TCP/IP server on free port
            modelName = componentArgs.getModelName();
            bDirInfo = RTW.getBuildDir(modelName);
            exeName = fullfile(bDirInfo.CodeGenFolder, modelName);
            if ispc
                exeName = [exeName, '.exe'];
            end
            launcher = myCustomLauncher(componentArgs);
            launcher.setExe(exeName);

            % Create dummy communicator and builder
            communicator = rtw.connectivity.RtIOStreamHostCommunicator.empty();
            builder = rtw.connectivity.MakefileBuilder.empty();

            % Call super class constructor to register components
            this@rtw.connectivity.Config(componentArgs, builder, launcher, communicator);
        end
    end
end
```

- 2 To register the implementation in Simulink, create an `sl_customization.m` file.

```
function sl_customization(cm)
    cm.registerTargetInfo(@loc_createConfig);
end
% local function
function config = loc_createConfig
    config = rtw.connectivity.ConfigRegistry;
    config.ConfigClass = 'myTargetConnectivityConfig';

    % Specify compatible model configuration. For example,
    % through SystemTargetFile and TargetHWDeviceType properties.
    config.SystemTargetFile = {'ert.tlc'};
    config.TargetHWDeviceType = { 'ARM Compatible->ARM Cortex' };
end
```

See “Create `sl_customization.m` File” (Embedded Coder).

- 3 Add the folder containing `sl_customization.m` to the search path and refresh your customizations.

```
addpath(sl_customization_path);
sl_refresh_customizations;
```

See Also

Related Examples

- “Customize XCP Server Software” on page 32-45

Customize XCP Server Software

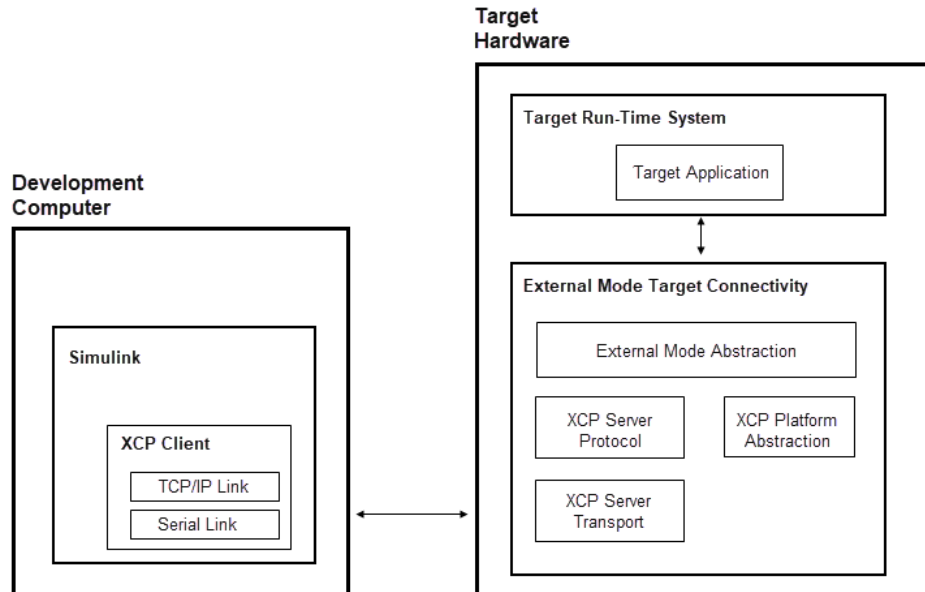
The XCP communication protocol for Simulink external mode simulations is a client-server communication protocol. By default, the software supports XCP external mode simulations:

- On your development computer for code that is generated by using ERT (`ert.tlc`) and GRT (`grt.tlc`) system target files.
- For some support packages on page 32-2.

If your system target file for custom target hardware is derived from the ERT or GRT system target files, you can use supplied APIs to provide XCP target connectivity. XCP external mode limitations on page 32-21 apply.

The external mode target connectivity software comprises:

- External mode abstraction layer
- XCP server protocol layer
- XCP server transport layer
- XCP platform abstraction layer



External Mode Abstraction Layer

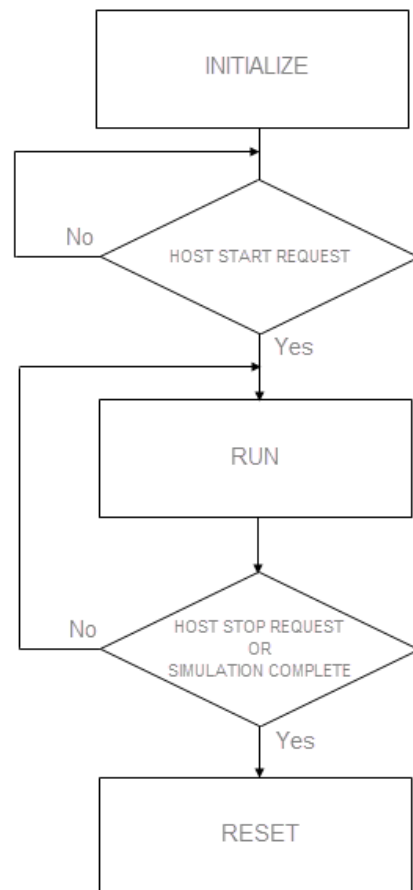
To communicate with Simulink during an external mode simulation, your target application must invoke functions from the external mode abstraction layer.

The parent folder for source code files is:

```
matlabroot\toolbox\coder\xcp\src\target\ext_mode
```

The layer APIs are declared in `include\ext_mode.h` and implemented in `src\xcp_ext_mode.c`.

To communicate with Simulink, the target application uses services exposed by the external mode abstraction layer. This flow chart shows the required target application steps to establish communication with Simulink.



This table lists the functions that your target application must invoke at each stage.

Stage	Function	Purpose
INITIALIZE	<code>extmodeParseArgs</code>	Extract external mode command-line arguments.
	<code>modelName_initialize</code>	Initialize generated Simulink model code.
	<code>extmodeInit</code>	Initialize external mode target connectivity.
HOST START REQUEST	<code>extmodeWaitForHostRequest</code>	Wait for a start request from development computer, which is created when you click Run button.
RUN	<code>modelName_step</code>	Run single step of generated Simulink model code.
	<code>extmodeEvent</code>	For sample time ID of the model, sample signals generated by model step function and pass packet content to transport layer of communication protocol for transmission to development computer.
	<code>extmodeBackgroundRun</code>	Transmit and receive packet from physical communication interface and process packet content in accordance with selected communication protocol.
HOST STOP REQUEST OR SIMULATION COMPLETE	<code>extmodeStopRequested</code>	Check whether a request to stop external mode simulation is received from the model on the development computer. The request is created when you click the Stop button.
	<code>extmodeSimulationComplete</code>	Check whether execution of generated model code is complete.
RESET	<code>modelName_terminate</code>	Terminate generated Simulink model code.
	<code>extmodeReset</code>	Reset the external mode target connectivity to initial state.

This pseudo-code example shows how you can implement the various flowchart stages in your target application. During the RUN stage, the external mode run-time environment requires at least two threads:

- A periodic thread that is responsible for the execution of the generated model code.
- A background thread that is responsible for running the external mode communication stack and transmitting and receiving packets.

The pseudo-code simulates multithreading by running `periodicThread` and `extmodeBackgroundRun` sequentially within the same `while()` loop.

```

/*----- Pseudo-Code Example -----*/
/* Define periodic thread */
void periodicThread(void)
{
    /* Run model step function */

```

```

    modelName_step();
    /* Notify external mode abstraction layer about periodic event */
    extmodeEvent(PERIODIC_EVENT_ID, currentSimulationTime);
}

/* Main function for target application */
main(int argc, char *argv[])
{
    /*----- INITIALIZE -----*/
    /* Parse external mode command line arguments */
    extmodeParseArgs(argc, argv);

    /* Initialize model */
    modelName_initialize();

    /* Initialize external mode target connectivity */
    extmodeInit(extModeInfo, finalSimulationTime);

    /*----- HOST START REQUEST -----*/
    /* Wait until a start request is received from development computer */
    extmodeWaitForHostRequest(EXTMODE_WAIT_FOREVER);

    /*----- HOST STOP REQUEST OR SIMULATION COMPLETE -----*/
    /* While simulation is not complete and stop request is not received */
    while (!extmodeSimulationComplete() && !extmodeStopRequested()) {

        /*----- RUN -----*/
        periodicThread();
        extmodeBackgroundRun();
    }

    /*----- RESET -----*/
    /* Terminate model */
    modelName_terminate();

    /* Reset external mode target connectivity */
    extmodeReset();
    return 0;
}

```

To see code that invokes the functions, complete the example in “External Mode Simulation by Using XCP Communication” on page 32-8 with **System target file** set to `ert.tlc`. Then, from the code generation folder, open `ert_main.c`.

Note If your system target file is `grt.tlc` or GRT-based, the code generator creates `rt_main.c`, and, by default, uses API defined by the internal file `ext_work.h`. To enable the use of `ext_mode.h` API, before building the target application, run:

```
set_param(model, 'TLCOptions', '-aExtModeXCPClassicInterface=0');
```

If your system target file is `ert.tlc` or ERT-based and `GenerateSampleERTMain` is set to 'on', the code generator creates `ert_main.c` and, by default, uses API defined by `ext_mode.h`. To enable the use of `ext_work.h` API, before building the target application, run:

```
set_param(model, 'TLCOptions', '-aExtModeXCPClassicInterface=1');
```

XCP Server Protocol Layer

The XCP server protocol layer interprets XCP commands and data according to the Association for Standardisation of Automation and Measuring Systems (ASAM) standard, ASAM MCD-1 XCP.

The source code folder is:

```
matlabroot\toolbox\coder\xcp\src\target\slave\protocol\src
```

In an external mode simulation, the build process automatically adds the required files to the build information object on page 44-7.

XCP Server Transport Layer

The XCP server transport layer transmits and receives messages from the communication medium according to ASAM specifications.

The source folder is:

```
matlabroot\toolbox\coder\xcp\src\target\slave\transport\src
```

In an external mode simulation, the build process automatically adds the required files to the build information object on page 44-7.

XCP Platform Abstraction Layer

The XCP platform abstraction layer provides:

- An XCP driver on page 32-49 for sending and receiving raw data through the physical communication interface.
- The implementation of a static memory allocator on page 32-50.
- Other target hardware-specific functionality on page 32-50.

You can use the `target` package to provide connectivity between Simulink and your target hardware. To specify:

- The XCP driver, use the `target.CommunicationInterface` class.
- Other parts of the XCP platform abstraction layer, use the `target.XCPPlatformAbstraction` class.

If you do not specify the XCP platform abstraction layer, the software uses the default platform abstraction layer, which supports Linux, Windows, and Mac platforms. For more information, see “Customise Connectivity for XCP External Mode Simulations” on page 32-36.

For a customization example, see “Create Custom Abstraction Layer” on page 32-53.

XCP Driver

The XCP driver sends and receives XCP messages through the communication channel. In an external mode simulation, the build process automatically adds the driver files to build information.

The XCP driver is based on the `rtiostream` API. For example, host-based external mode simulations use these `rtiostream` files:

- *matlabroot*\toolbox\coder\rtiostream\src\rtiostreamtcpip\rtiostream_tcpip.c
- *matlabroot*\toolbox\coder\rtiostream\src\rtiostreamserial\rtiostream_serial.c

For details on implementing and testing an `rtiostream` communications channel, see:

- “Communications `rtiostream` API” (Embedded Coder)

- `rtiostreamtest`

To provide `rtiostream` information:

- If you use the `target` package for connectivity, use the `target.CommunicationInterface` class. See “Customise Connectivity for XCP External Mode Simulations” on page 32-36.
- If you do not use the `target` package, add your `rtiostream` files to a build information object on page 44-7.

For more information, see “Create Custom Abstraction Layer” on page 32-53.

Memory Allocator

The XCP server software requires the dynamic allocation of variable-size, contiguous memory blocks to hold internal data structures. In an external mode simulation, the build process automatically adds memory allocator files to the build information object.

Configure alignment for the memory allocator through the `XCP_MEM_ALIGNMENT` preprocessor macro. For example:

```
#define XCP_MEM_ALIGNMENT 8
```

If you use the `target` package for connectivity, use the `BuildDependencies` property of the `target.XCPPlatformAbstraction` class. For example:

```
xcpPlatformAbstraction.BuildDependencies.Defines = {'XCP_MEM_ALIGNMENT=8'};
```

See “Customise Connectivity for XCP External Mode Simulations” on page 32-36.

Other Platform Abstraction Layer Functionality

This file defines the platform abstraction layer interface:

```
matlabroot\toolbox\coder\xcp\src\target\slave\platform\include\xcp_platform.h
```

This table describes the functionality that you must provide for the XCP server software that you deploy on your target hardware.

Functionality	Details
Mutual exclusion	<p>To access system data structures with <i>mutual exclusion</i> support, the XCP driver depends on basic APIs for definition, initialization, lock, and unlock.</p> <p>To support mutual exclusion for your target hardware, provide a custom implementation.</p> <p>The default implementation for Linux and Windows supports mutual exclusion.</p> <pre data-bbox="589 590 1266 919"> #if defined(_WIN32) defined(__WIN32__) defined(WIN32) ... #define XCP_MUTEX_DEFINE(lock) HANDLE lock #define XCP_MUTEX_INIT(lock) lock = CreateMutex(0, FALSE, 0) #define XCP_MUTEX_LOCK(lock) WaitForSingleObject((lock), INFINITE) #define XCP_MUTEX_UNLOCK(lock) ReleaseMutex(lock) #else ... #include <pthread.h> #define XCP_MUTEX_DEFINE(lock) pthread_mutex_t lock #define XCP_MUTEX_INIT(lock) pthread_mutex_init(&(lock), NULL) #define XCP_MUTEX_LOCK(lock) pthread_mutex_lock(&(lock)) #define XCP_MUTEX_UNLOCK(lock) pthread_mutex_unlock(&(lock)) ... #endif </pre>
Sleep	<p>Provide a sleep API that makes the calling thread sleep until a specified time has elapsed. The default implementation for Linux and Windows systems is:</p> <pre data-bbox="589 1041 1450 1392"> #if defined(_WIN32) defined(__WIN32__) defined(WIN32) ... #define XCP_SLEEP(seconds,microseconds) Sleep((seconds) * 1000 \ + ((microseconds) + 1000 - 1) / 1000) #else ... #if _POSIX_C_SOURCE >= 199309L #define XCP_SLEEP(seconds,microseconds) do {struct timespec t;\ t.tv_sec = seconds; t.tv_nsec = microseconds * 1000; nanosleep(&t, NULL);} while(0) #else /* nanosleep is not available. Use select instead. */ #define XCP_SLEEP(seconds,microseconds) do {struct timeval t; t.tv_sec = seconds;\ t.tv_usec = microseconds; select(0, NULL, NULL, NULL, &t);} while(0) #endif /* _POSIX_C_SOURCE >= 199309L */ ... #endif </pre>
Logging	<p>To generate diagnostic messages, the XCP server software requires a custom print API that supports logging services provided by the target hardware. If you do not define the <code>XCP_PRINTF</code> preprocessor macro, the default implementation is empty.</p>

Functionality	Details
Address conversion	<p>The XCP standard expresses the address of a variable in memory as a 32-bit address with an 8-bit extension.</p> <p>The XCP client extracts the addresses of signals and parameters of the model by parsing the debug information that the build process creates. The debug information is in DWARF or PDB format.</p> <p>The XCP client requests access to parameters and signals by sending an XCP command that contains the arguments <code>addressExtension</code> and <code>address</code>.</p> <p>When <code>addressExtension</code> and <code>address</code> information is received by the target hardware, the XCP server must convert the address to the location of the variable in the target hardware memory. The variable location is assigned by the loader and is target-specific.</p> <p>Use the <code>XCP_ADDRESS_GET()</code> macro to specify the conversion logic. The default implementation for Linux and Windows systems is:</p> <pre data-bbox="589 863 1122 1003"> #if defined(__MINGW32__) defined(__MINGW64__) #define XCP_ADDRESS_GET(addressExtension, address) \ (uint8_T*) ((uintptr_t) address) #else #define XCP_ADDRESS_GET(addressExtension, address) \ (uint8_T*) ((address) + (uint8_T*)&__ImageBase) #endif </pre> <p>Currently, only 32-bit hardware architectures are supported, so <code>addressExtension</code> is 0.</p>
Set and copy memory	<p>You can specify an optimized version of copy memory and set memory operations.</p> <p>If you do not define the <code>XCP_MEMCPY</code> and <code>XCP_MEMSET</code> preprocessor macros, the default implementation specifies the standard C functions, <code>memcpy</code> and <code>memset</code>.</p> <pre data-bbox="589 1304 846 1444"> #ifndef XCP_MEMCPY #define XCP_MEMCPY memcpy #endif #ifndef XCP_MEMSET #define XCP_MEMSET memset #endif </pre>
Parsing command-line arguments	<p>If your target hardware does not support the parsing of command-line arguments, define the preprocessor macro <code>EXTMODE_DISABLE_ARGS_PROCESSING</code>.</p> <p>To replace the <code>-w</code> option, you can use this command to specify that the target application enters and stays in a wait state until it receives a Connect message from Simulink:</p> <pre data-bbox="589 1696 1321 1728"> set_param(modelName, 'OnTargetWaitForStart', 'on'); </pre> <p>The build process provides the required option (<code>-DON_TARGET_WAIT_FOR_START=1</code>) to the compiler.</p>

Functionality	Details
Structure packing and alignment	<p>The XCP software stack implementation requires the packing and alignment of some internal C structures. In the platform abstraction layer, provide packing and alignment information by using preprocessor directives that are compiler-specific.</p> <p>To pack and align C structures, provide these preprocessor directives:</p> <pre>#define XCP_PRAGMA_PACK_BEGIN(n) #define XCP_PRAGMA_PACK_END() #define XCP_ATTRIBUTE_ALIGNED(n) #define XCP_ATTRIBUTE_PACKED</pre> <p>For example:</p> <pre>XCP_PRAGMA_PACK_BEGIN(1) typedef struct XCP_ATTRIBUTE_PACKED XcpSetDaqListModeCmdPacketFrame { uint8_T PID; /* Packet identifier, always XCP_PID_SET_DAQ_LIST_MODE */ uint8_T mode; /* DAQ list mode (XCP DAQ Mode bit masks) */ uint16_T daqListId; /* DAQ list Id */ uint16_T eventId; /* Event Channel Id */ uint8_T prescaler; /* Transmission rate prescaler */ uint8_T priority; /* DAQ list priority (0xFF -> highest) */ } XcpSetDaqListModeCmdPacketFrame; XCP_PRAGMA_PACK_END()</pre> <pre>XCP_PRAGMA_PACK_BEGIN(XCP_MEM_ALIGNMENT) struct XCP_ATTRIBUTE_ALIGNED(XCP_MEM_ALIGNMENT) xcpFifoEntry { struct xcpFifoEntry *next; size_t msgFrameSize; /* in AG units */ }; XCP_PRAGMA_PACK_END()</pre> <p>For a GCC compiler, use these directives:</p> <pre>#define XCP_PRAGMA_PACK_BEGIN(n) #define XCP_PRAGMA_PACK_END() #define XCP_ATTRIBUTE_ALIGNED(n) __attribute__((aligned(n))) #define XCP_ATTRIBUTE_PACKED __attribute__((packed))</pre> <p>For a Microsoft Visual C++ compiler, use these directives:</p> <pre>#define PRAGMA(n) _Pragma(#n) #define XCP_PRAGMA_PACK_BEGIN(n) PRAGMA(pack(push, n)) #define XCP_PRAGMA_PACK_END() PRAGMA(pack(pop)) #define XCP_ATTRIBUTE_ALIGNED(n) #define XCP_ATTRIBUTE_PACKED</pre>

Create Custom Abstraction Layer

To create a custom platform abstraction layer:

- 1 Specify the header file `xcp_platform_custom.h`. This Linux example defines the required functions.

```
#ifndef XCP_PLATFORM_CUSTOM_H
#define XCP_PLATFORM_CUSTOM_H

/* XCP_ADDRESS_GET */
#include <stdint.h>
#define XCP_ADDRESS_GET(addressExtension, address) (uint8_T*) ((uintptr_t) address)

/* XCP_MUTEX */
#include <pthread.h>
#define XCP_MUTEX_DEFINE(lock)    pthread_mutex_t lock
```

```

#define XCP_MUTEX_INIT(lock)      pthread_mutex_init(&(lock), NULL)
#define XCP_MUTEX_LOCK(lock)     pthread_mutex_lock(&(lock))
#define XCP_MUTEX_UNLOCK(lock)   pthread_mutex_unlock(&(lock))

/* XCP_SLEEP */
#include <sys/time.h>      /* gettimeofday */
#if _POSIX_C_SOURCE >= 199309L
#include <time.h>         /* for nanosleep */
#else
#include <stddef.h>
#include <sys/select.h> /* for select */
#endif

/* _POSIX_C_SOURCE >= 199309L */
#if _POSIX_C_SOURCE >= 199309L
#define XCP_SLEEP(seconds,microseconds) do {struct timespec t;\
    t.tv_sec = seconds; t.tv_nsec = microseconds * 1000; nanosleep(&t, NULL);} while(0)
#else
/* nanosleep is not available. Use select instead. */
#define XCP_SLEEP(seconds,microseconds) do {struct timeval t; t.tv_sec = seconds;\
    t.tv_usec = microseconds; select(0, NULL, NULL, NULL, &t);} while(0)
#endif /* _POSIX_C_SOURCE >= 199309L */
#endif

```

2 Register the header file:

- If you use the `target` package for connectivity, use the `BuildDependencies` property of the `target.XCPPlatformAbstraction` class to specify the required information. See “Customise Connectivity for XCP External Mode Simulations” on page 32-36.
- If you do not use the `target` package for connectivity, define a post-code generation command:

```

function myXCPTargetPostCodeGenCommand(buildInfo)

    buildInfo.addDefines('-DXCP_CUSTOM_PLATFORM', 'OPTS');

    % Add my rtiostream
    buildInfo.addSourceFiles(customRtIOStreamFileName, ...
        customRtIOStreamSrcPath);

    % If the target hardware does not support parsing of command
    % line arguments
    buildInfo.addDefines('-DEXTMODE_DISABLE_ARGS_PROCESSING', 'OPTS');

end

```

See Also

[extmodeBackgroundRun](#) | [extmodeEvent](#) | [extmodeGetFinalSimulationTime](#) | [extmodeInit](#) | [extmodeParseArgs](#) | [extmodeReset](#) | [extmodeSetFinalSimulationTime](#) | [extmodeSimulationComplete](#) | [extmodeStopRequested](#) | [extmodeWaitForHostRequest](#)

More About

- “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” on page 32-2
- “External Mode Simulation by Using XCP Communication” on page 32-8
- “Set Up External Mode Connectivity Between Simulink and Target Hardware” on page 32-35

External Websites

- <https://www.asam.net/standards/detail/mcd-1-xcp/wiki/>

External Mode Simulation with TCP/IP or Serial Communication

Set up and run an external mode simulation that uses a TCP/IP or serial (RS-232) communication channel.

- 1 Create and configure a simple model.
- 2 Build the target executable file.
- 3 Run the target application.
- 4 Tune parameters.

The example, which uses the GRT target, does not require external hardware. The generated executable file runs:

- On the development computer that hosts Simulink and Simulink Coder.
- As a separate process from MATLAB and Simulink.

Create and Configure Model

In this part of the example, you create a simple model, `ex_extModeExample`. You also create a folder called `ext_mode_example` to store the model and the generated executable.

To create the folder and the model:

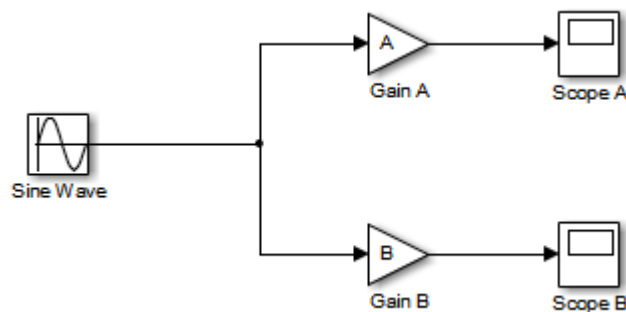
- 1 From the MATLAB command line, type:

```
mkdir ext_mode_example
```

- 2 Make `ext_mode_example` your working folder:

```
cd ext_mode_example
```

- 3 Create a model in Simulink with a Sine Wave block for the input signal, two Gain blocks in parallel, and two Scope blocks. Be sure to label the Gain and Scope blocks as shown.

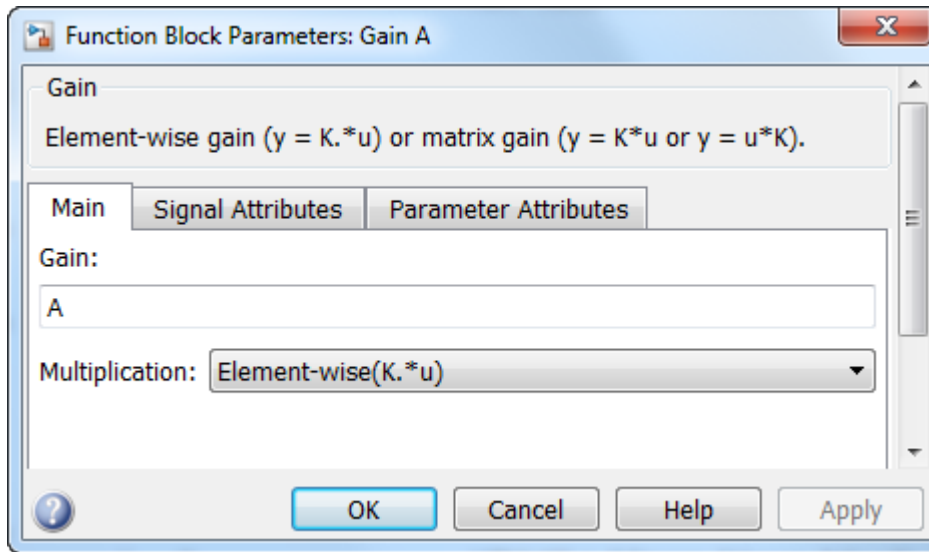


- 4 Define and assign two MATLAB workspace variables, A and B:

```
A = 2;
```

```
B = 3;
```

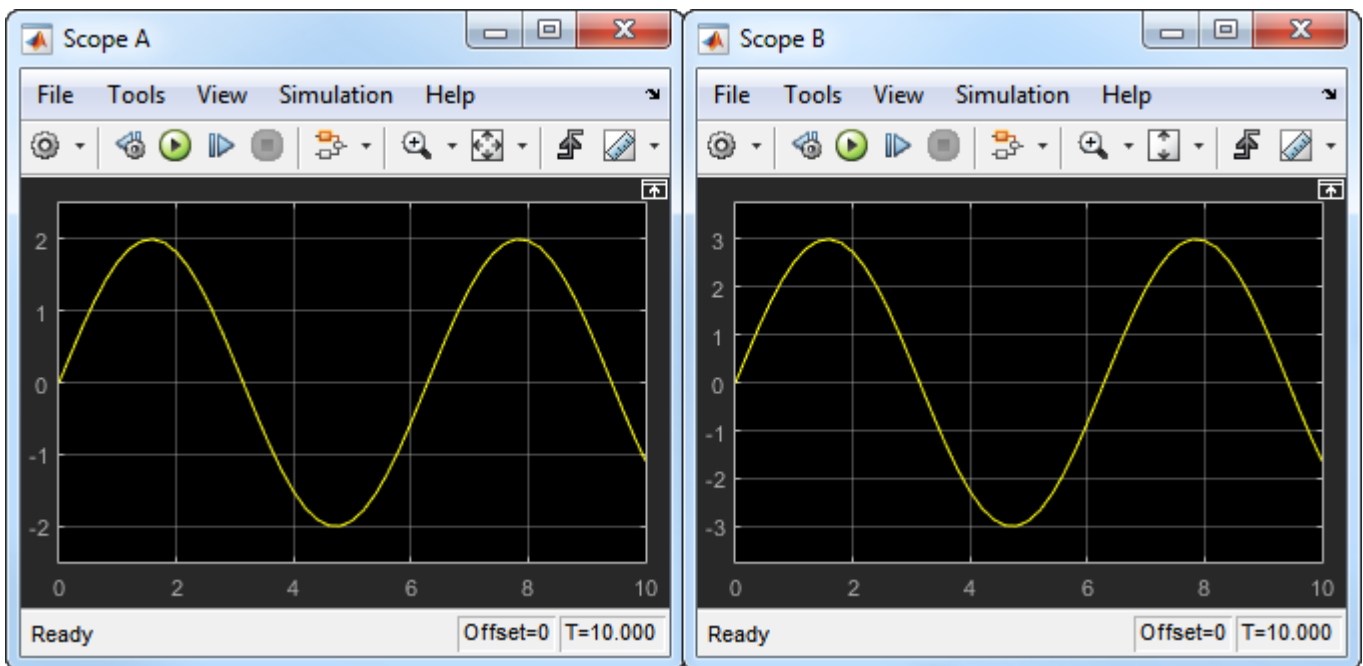
- 5 Open Gain block A and set its **Gain** parameter to the variable A.



- 6 Open Gain block B and set its **Gain** parameter to the variable B.

When the target application is built and connected to Simulink in external mode, you can download new gain values to the executing target application. To do this, you can assign new values to workspace variables A and B, or edit the values in the block parameters dialog box. For more information, see “Tune Parameters” on page 32-59.

- 7 Verify operation of the model. Open the Scope blocks and run the model. When $A = 2$ and $B = 3$, the output appears as shown.



- 8 Save the model as ex_extModeExample.

Build Target Executable

Set up the model and code generation parameters required for an external mode target application. Then, generate code and build the target application.

- 1 From the **Apps** tab on the Simulink toolstrip, in the **Setup to Run on Hardware** section, click **Run on Custom Hardware**.
- 2 In the **Hardware** section of the **Hardware** tab, in the **System Target File Description** list, select **Generic Real-Time Target (grt.tlc)**.
- 3 In the **Prepare** section, click **Hardware Settings**. The Configuration Parameters dialog box opens, displaying **Hardware Implementation** settings that are determined by the system target file.
- 4 On the **Solver** pane:
 - a In the **Type** field, select **Fixed-step**.
 - b In the **Solver** field, select **discrete (no continuous states)**.
 - c Click **Solver details**. In the **Fixed-step size** field, specify **0.1**. (Otherwise, when you generate code, the Simulink Coder build process produces a warning and supplies a value.)
 - d Click **Apply**.
- 5 On the **Data Import/Export** pane, clear the **Time** and **Output** check boxes. In this example, data is not logged to the workspace or to a MAT-file. Click **Apply**.
- 6 On the **Code Generation > Optimization** pane, check that **Default parameter behavior** is set to **Tunable**. If you make a change, click **Apply**.
- 7 On the **Code Generation > Interface** pane, in the **Data exchange interface** section, select **External mode**.
- 8 In the **External mode configuration** section, make sure that the default value **tcpip** is selected for the **Transport layer** parameter.

External mode

External mode configuration

Transport layer: tcpip ▼ MEX-file name: ext_comm

MEX-file arguments:

Static memory allocation

External mode supports communication via TCP/IP, serial, and custom transport protocols. The **MEX-file name** specifies the name of a MEX-file that implements host-target communication. The default for TCP/IP is `ext_comm`, a MEX-file provided with the Simulink Coder software. You can override this default by supplying other files. If you need to support other transport layers, see “Create a Transport Layer for TCP/IP or Serial External Mode Communication” on page 32-94.

The **MEX-file arguments** field lets you specify arguments, such as a TCP/IP server port number, to be passed to the external interface program. These arguments are specific to the external interface that you are using. For information on setting these arguments, see “MEX-File Optional Arguments for TCP/IP Transport” on page 32-80 and “MEX-File Optional Arguments for Serial Transport” on page 32-81.

This example uses the default arguments. Leave the **MEX-file arguments** field blank.

The **Static memory allocation** check box controls how memory is allocated for external mode communication buffers in the target. For this example, do not select the check box. For more information, see “Control Memory Allocation for Communication Buffers in Target” on page 32-61.

- 9 Click **Apply** to save the external mode settings.
- 10 Save the model.
- 11 Select the **Code Generation** pane. Make sure that **Generate code only** is cleared.
- 12 To generate code and create the target application, in the model window, press **Ctrl+B**. Or, on the **Hardware** tab, in the **Run on Hardware** section, click **Monitor & Tune**. Then, under **Step By Step Commands**, click **Build for Monitoring**.

The software creates the `ex_extModeExample` executable file in your working folder.

Run Target Application


You now run the `ex_extModeExample` target executable and use Simulink as an interactive front end to the running target application. The executable file is in your working folder. Run the target application and establish communication between Simulink and the target.

Note An external mode program like `ex_extModeExample` is a host-based executable. Its execution is not tied to a real-time operating system (RTOS) or a periodic timer interrupt, and it does not run in real time. The program just runs as fast as possible, and the time units it counts off are simulated time units that do not correspond to time in the world outside the program.

To run the target application:

- 1 On the **Hardware** tab, in the **Run on Hardware** section:
 - a In the **Stop Time** field, specify `inf`, which makes the model run until the target application receives a stop message from Simulink
 - b Click **Monitor & Tune**. Then, under **Step By Step Commands**, click **Deploy**.

The target application begins execution, and enters a wait state.

- 2 Open the Scope blocks in the model. Signals are not visible on the scopes until you begin model execution.
- 3 On the **Hardware** tab, in the **Run on Hardware** section, click **Monitor & Tune**. Then, under **Step By Step Commands**, click **Connect**. This action initiates a handshake between Simulink and the target application. When Simulink and the target are connected, the **Connect** button changes to **Disconnect**.
- 4 In the **Run on Hardware** section, click , which starts execution of the generated model code. The two scopes in your model display the outputs of Gain blocks A and B.

You have established communication between Simulink and the running target application. You can now tune block parameters in Simulink and observe the effects of parameter changes.

Tune Parameters

This table describes how you can tune tunable block parameters during a simulation.

Approach	Details
Model Data Editor	<p>To tune parameters through the Model Data Editor:</p> <ol style="list-style-type: none"> 1 On the Hardware tab of the Simulink Editor, in the Prepare section, click Tune Parameters, which opens the Model Data Editor. 2 If you want to update multiple tunable parameters simultaneously, in the Prepare section, toggle on Hold Updates. 3 On the Parameters tab of the Model Data Editor, in the Value column, specify new values for your tunable parameters. 4 Toggle off Hold Updates or click Update All Parameters (Ctrl+D). Simulink downloads the new values to the target application simultaneously. <p>If Hold Updates is off, then immediately after you specify a new value, Simulink downloads the new value to the target application.</p> <p>For more information, see Model Data Editor.</p>
Block Parameter dialog box	<p>To tune parameters through the Block Parameter dialog box:</p> <ol style="list-style-type: none"> 1 For each block that you want to update: <ol style="list-style-type: none"> a Double-click the block, which opens the Block Parameter dialog box. b In the parameter fields, specify new parameter values. c Click Apply or OK. 2 If Hold Updates is on, toggle off Hold Updates or click Update All Parameters (Ctrl+D). Simulink downloads the new values to the target application simultaneously. <p>If Hold Updates is off, then immediately after you click Apply or OK, Simulink downloads the new block values to the target application.</p>

Approach	Details
MATLAB workspace	<p>If block parameters are MATLAB workspace variables:</p> <ol style="list-style-type: none"> 1 In the Command Window, assign new values to the variables. 2 On the Hardware tab of the Simulink Editor, in the Prepare section, click Update All Parameters (Ctrl+D). Simulink downloads the new values to the target application. <p>For more information, see “Create and Edit Variables”.</p>

You cannot change the sample time of the Sine Wave block during simulation. Block sample times are part of the structural definition of the model and are part of the generated code. Therefore, if you want to change a block sample time, you must stop the external mode simulation, reset the sample time of the block, and rebuild the executable.

Block parameter tunability during external mode simulation depends on the way that the generated code represents block parameters.

For example, in the Gain A block dialog box, you cannot change the expression A in the **Gain** parameter during simulation. Instead, you must change the value of the variable A in the base workspace. You cannot change the expression because the generated code does not allocate storage in memory for the **Gain** parameter. Instead, the code creates a field A in a structure:

```

/* Parameters (auto storage) */
struct P_ex_extModeExample_T_ {
    real_T A;                /* Variable: A
                           */
    real_T B;                /* Variable: B
                           */
    real_T SineWave_Amp;     /* Expression: 1
                           */
    real_T SineWave_Bias;    /* Expression: 0
                           */
    real_T SineWave_Freq;    /* Expression: 1
                           */
    real_T SineWave_Phase;   /* Expression: 0
                           */
};

```

The generated code algorithm uses that field in the code that represents the block Gain A. In this case, the global structure variable `ex_extModeExample_P` uses the type `P_ex_extModeExample_T_`:

```
ex_extModeExample_B.GainA = ex_extModeExample_P.A * rtb_SineWave;
```

When you change the value of A in the base workspace, the simulation downloads the new value to the field A in the target application.

You can change the expressions in the Sine Wave block parameters during simulation because the generated code creates a field in the global structure `ex_extModeExample_P` to represent each

parameter in the block. When you change an expression in the block dialog box, the simulation first evaluates the new expression. The simulation then downloads the resulting numeric value to the corresponding structure field in the target application.

See “Create Tunable Calibration Parameter in the Generated Code” on page 15-112.

Stop Target Application

To simultaneously disconnect Simulink from the host/target communication and end execution of the target application, on the **Hardware** tab, in the **Run on Hardware** section, click **Stop**.

Control Memory Allocation for Communication Buffers in Target

If you select the **Code Generation > Interface > Static memory allocation** check box (for GRT and ERT targets), the code generator produces code for external mode that uses only static memory allocation (“malloc-free” code). Selecting **Static memory allocation** enables the **Static memory buffer size** parameter. Use this parameter to specify the size of the external mode static memory buffer. The default value is 1,000,000 bytes. If you enter too small a value for your program, the external mode simulation issues an out-of-memory error when it tries to allocate more memory than specified. In such cases, increase the value in the **Static memory buffer size** field and regenerate the code.

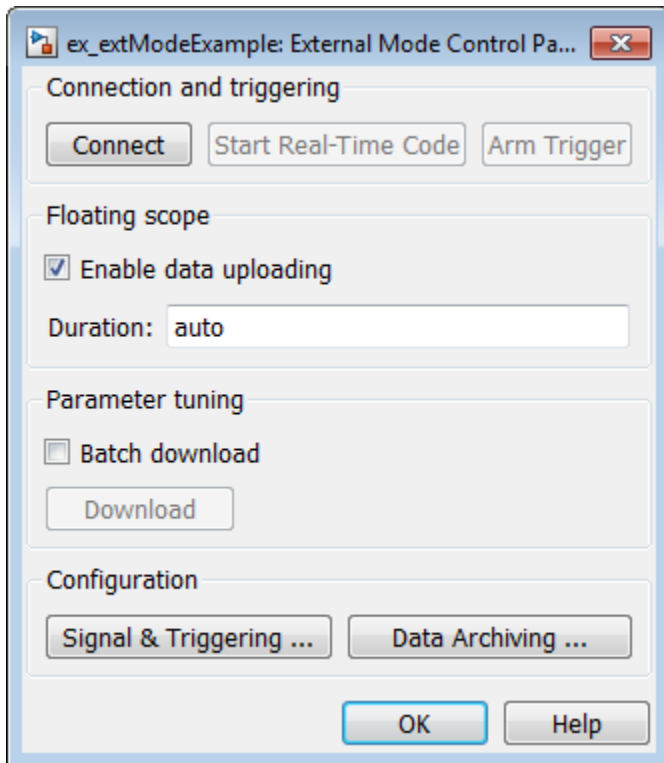
To determine how much memory to allocate, enable verbose mode on the target (by including `OPTS="-DVERBOSE"` on the make command line). As it executes, external mode displays the amount of memory it tries to allocate and the amount of memory available to it each time it attempts an allocation. If an allocation fails, you can use this console log to determine the required value for the **Static memory buffer size** field.

TCP/IP or Serial External Mode Control Panel

The External Mode Control Panel provides controls for TCP/IP or serial external mode operations, including:

- “Connect, Start, and Stop” on page 32-62
- “Upload Target Application Signal Data to Host” on page 32-62
- “Download Parameters to Target Application” on page 32-63
- “Configure Host Monitoring of Target Application Signal Data” on page 32-64
- “Configure Host Archiving of Target Application Signal Data” on page 32-70

To open the External Mode Control Panel dialog box, on the **Hardware** tab, in the **Prepare** section, click **Control Panel**.



Connect, Start, and Stop

The External Mode Control Panel performs the same connect/disconnect and start/stop functions found on the **Hardware** tab on the Simulink toolstrip (see “Summary of Graphical Controls for TCP/IP or Serial External Mode Simulation” on page 32-72).

Clicking the **Connect** button connects your model to a waiting or running target application. While you are connected, the button changes to a **Disconnect** button. **Disconnect** disconnects your model from the target environment, but does not halt real-time code running in the target environment.

Clicking the **Start Real-Time Code** button commands the target to start running real-time code. While real-time code is running in the target environment, the button changes to a **Stop Real-Time Code** button. **Stop Real-Time Code** stops target application execution and disconnects your model from the target environment.

Upload Target Application Signal Data to Host

The External Mode Control Panel allows you to trigger and cancel data uploads to the host. The destination for the uploaded data can be a scope block, Display block, To Workspace block, or another block or subsystem listed in “Blocks and Subsystems Compatible with External Mode” on page 32-73.

The **Arm Trigger** and **Cancel Trigger** buttons provide manual control of data uploading to compatible blocks or subsystems, except floating scopes. (For floating scopes, use the **Floating scope** section of the External Mode Control Panel.)

- To trigger data uploading to compatible blocks or subsystems, click the **Arm Trigger** button. The button changes to **Cancel Trigger**.

- To cancel data uploading, click the **Cancel Trigger** button. The button reverts to **Arm Trigger**.

You can trigger data uploads manually or automatically. To configure signals and triggers for data uploads, see “Configure Host Monitoring of Target Application Signal Data” on page 32-64.

A subset of external mode compatible blocks, including Scope, Time Scope, and To Workspace, allow you to log uploaded data to disk. To configure data archiving, see “Configure Host Archiving of Target Application Signal Data” on page 32-70.

The **Floating scope** section of the External Mode Control Panel controls when and for how long data is uploaded to Floating Scope blocks. When used in external mode, floating scopes:

- Do not appear in the External Signal & Triggering dialog box.
- Do not log data to external mode archiving.
- Support manual triggering only.

The **Floating scope** section contains the following parameters:

- **Enable data uploading** option, which functions as an **Arm Trigger** button for floating scopes. When the target is disconnected, the option controls whether to arm the trigger when connecting the floating scopes. When the target is connected, the option acts as a toggle button to arm or cancel the trigger.
 - To trigger data uploading to floating scopes, select **Enable data uploading**.
 - To cancel data uploading to floating scopes, clear **Enable data uploading**.
- **Duration** edit field, which specifies the number of base-rate steps for which external mode logs floating scopes data after a trigger event. By default, it is set to **auto**, which causes the duration value set in the External Signal & Triggering dialog box to be used. The default duration value is 1000 base rate steps.

Download Parameters to Target Application

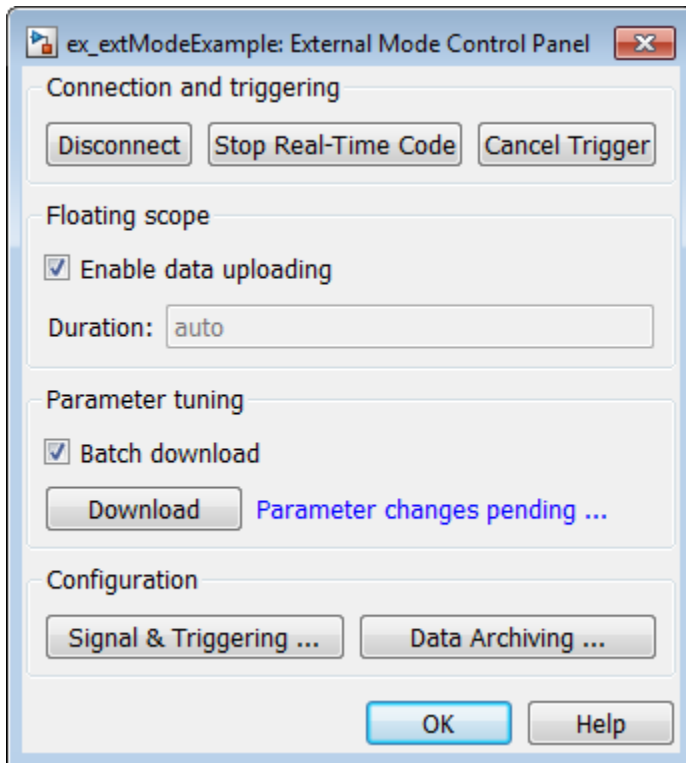
The **Batch download** option on the External Mode Control Panel enables or disables batch parameter changes.

By default, batch download is disabled. If batch download is disabled, when you click **OK** or **Apply**, changes made directly to block parameters by editing block parameter dialog boxes are sent to the target. When you perform an **Update Diagram**, changes to MATLAB workspace variables are sent.

If you select **Batch download**, the **Download** button is enabled. Until you click **Download**, changes made to block parameters are stored locally. When you click **Download**, the changes are sent in a single transmission.

When parameter changes are awaiting batch download, the External Mode Control Panel displays the message **Parameter changes pending...** to the right of the **Download** button. This message remains visible until the Simulink engine receives notification that the new parameters have been installed in the parameter vector of the target system.

The next figure shows the External Mode Control Panel with the **Batch download** option activated and parameter changes pending.



Configure Host Monitoring of Target Application Signal Data

- “Role of Trigger in Signal Data Uploading” on page 32-64
- “Configure Signal Data Uploading” on page 32-65
- “Default Trigger Options” on page 32-66
- “Select Signals to Upload” on page 32-66
- “Configure Trigger Options” on page 32-66
- “Select Trigger Signal” on page 32-68
- “Set Trigger Conditions” on page 32-68
- “Modify Signal and Triggering Options While Connected” on page 32-69

Role of Trigger in Signal Data Uploading

In external mode, uploading target application signal data to the host depends on a *trigger*. The trigger is a set of conditions that must be met for data uploading to begin. The trigger can be armed or not armed.

- When the trigger is armed, the software checks for the trigger conditions that allow data uploading to begin.
- If the trigger is not armed, the software does not check for the trigger conditions and data uploading cannot begin.
- The trigger can be armed automatically, when the host connects to the target, or manually, by clicking the **Arm Trigger** button on the External Mode Control Panel.

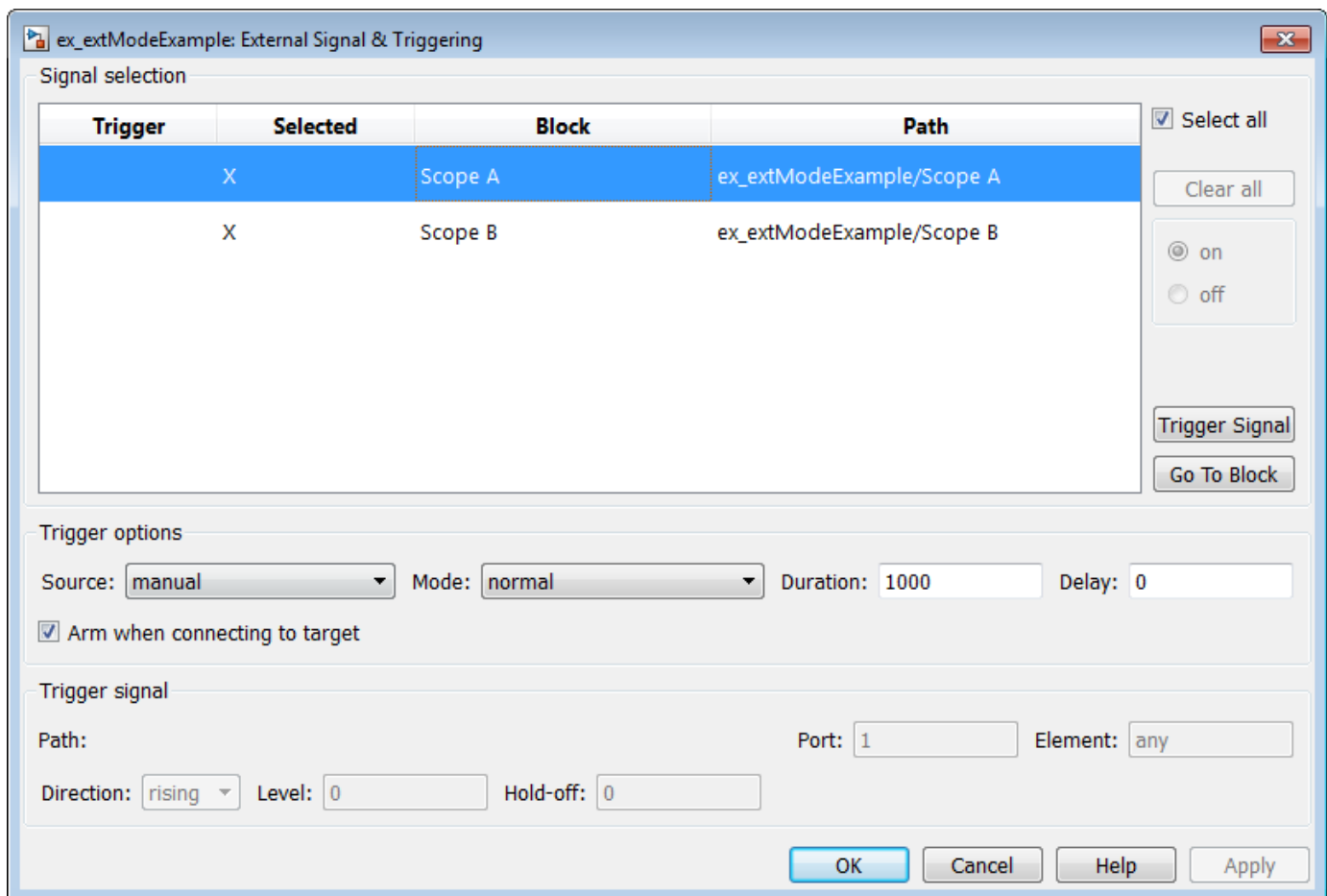
When the trigger is armed and the trigger conditions are met, the trigger fires and data uploading begins.

When data has been collected for a defined duration, the trigger event completes and data uploading stops. The trigger can then rearm, or remain unarmed until you click the **Arm Trigger** button.

To select the target application signals to upload and configure how uploads are triggered, see “Configure Signal Data Uploading” on page 32-65.

Configure Signal Data Uploading

Clicking the **Signal & Triggering** button of the External Mode Control Panel opens the External Signal & Triggering dialog box.



The External Signal & Triggering dialog box displays a list of blocks and subsystems in your model that support external mode signal uploading. For information on which types of blocks are external mode compatible, see “Blocks and Subsystems Compatible with External Mode” on page 32-73.

In the External Signal & Triggering dialog box, you can select the signals that are collected from the target system and viewed in external mode. You can also select a trigger signal, which triggers uploading of data based on meeting certain signal conditions, and define the triggering conditions.

Default Trigger Options

The preceding figure shows the default settings of the External Signal & Triggering dialog box. The default operation of the External Signal & Triggering dialog box simplifies monitoring the target application. If you use the default settings, you do not need to preconfigure signals and triggers. You start the target application and connect the Simulink engine to it. External mode compatible blocks are selected and the trigger is armed. Signal uploading begins immediately upon connection to the target application.

The default configuration of trigger options is:

- **Select all:** on
- **Source:** manual
- **Mode:** normal
- **Duration:** 1000
- **Delay:** 0
- **Arm when connecting to target:** on

Select Signals to Upload

External mode compatible blocks in your model appear in the **Signal selection** list of the External Signal & Triggering dialog box. You use this list to select signals that you want to view. In the **Selected** column, an X appears for each selected block.

The **Select all** check box selects all signals. By default, **Select all** is selected.

If **Select all** is cleared, you can select or clear individual signals using the **on** and **off** options. To select a signal, click its list entry and select the **on** option. To clear a signal, click its list entry and select the **off** option.

The **Clear all** button clears all signals.

Configure Trigger Options

As described in “Role of Trigger in Signal Data Uploading” on page 32-64, signal data uploading depends on a trigger. The trigger defines conditions that must be met for uploading to begin. Also, the trigger must be armed for data uploading to begin. When the trigger is armed and trigger conditions are met, the trigger fires and uploading begins. When data has been collected for a defined duration, the trigger event completes and data uploading stops.

To control when and how signal data is collected (uploaded) from the target system, configure the following **Trigger options** in the External Signal & Triggering dialog box.

- **Source:** manual or signal. Controls whether a button or a signal triggers data uploading.

Selecting **manual** directs external mode to use the **Arm Trigger** button on the External Mode Control Panel as the trigger to start uploading data. When you click **Arm Trigger**, data uploading begins.

Selecting **signal** directs external mode to use a trigger signal as the trigger to start uploading data. When the trigger signal satisfies trigger conditions (that is, the signal crosses the trigger level in the specified direction), a *trigger event* occurs. (Specify trigger conditions in the **Trigger signal** section.) If the trigger is *armed*, external mode monitors for the occurrence of a trigger event. When a trigger event occurs, data uploading begins.

- **Mode:** normal or one-shot. Controls whether the trigger rearms after a trigger event completes.

In normal mode, external mode automatically rearms the trigger after each trigger event. The next data upload begins when the trigger fires.

In one-shot mode, external mode collects only one buffer of data each time you arm the trigger.

For more information on the **Mode** setting, see “Configure Host Archiving of Target Application Signal Data” on page 32-70.

- **Duration:** Specifies the number of base rate steps for which external mode uploads data after a trigger event (default is 1000). For example, if **Duration** is set to 1000, and the base (fastest) rate of the model is one second:
 - For a signal sampled at the base rate, one second (1.0 Hz), external mode collects 1000 contiguous samples during a trigger event.
 - For a signal sampled at two seconds (0.5 Hz), external mode collects 500 samples during a trigger event.
- **Delay:** Specifies a delay to be applied to data collection. The delay represents the amount of time that elapses between a trigger event and the start of data collection. The delay is expressed in base rate steps. It can be positive or negative (default is 0). A negative delay corresponds to pretriggering. When the delay is negative, data from the time preceding the trigger event is collected and uploaded.
- **Arm when connecting to target:** Selected or cleared. Whether a button or a signal triggers data uploading (as defined by **Source**), the trigger must be armed to allow data uploading to begin.

If you select this option, connecting to the target arms the trigger.

- If the trigger **Source** is manual, data uploading begins immediately.
- If the trigger **Source** is signal, monitoring of the trigger signal begins immediately. Data uploading begins when the trigger signal satisfies trigger conditions (as defined in the **Trigger signal** section).

If you clear **Arm when connecting to target**, manually arm the trigger by clicking the **Arm Trigger** button on the External Mode Control Panel.

When simulating in external mode, each rate in the model creates a buffer on the target. Each entry in the buffer is big enough to hold all of the data required of every signal in that rate for one time step (time plus data plus external mode indices identifying the signal). The number of entries in the circular buffer is determined by the external mode trigger **Duration** parameter (ExtModeTrigDuration). The memory allocated on the target for buffering signals is proportional to the **Duration** and the number of signals uploading. The **Duration** also provides an indication of the number of base rate steps with log data after a trigger event in external mode.

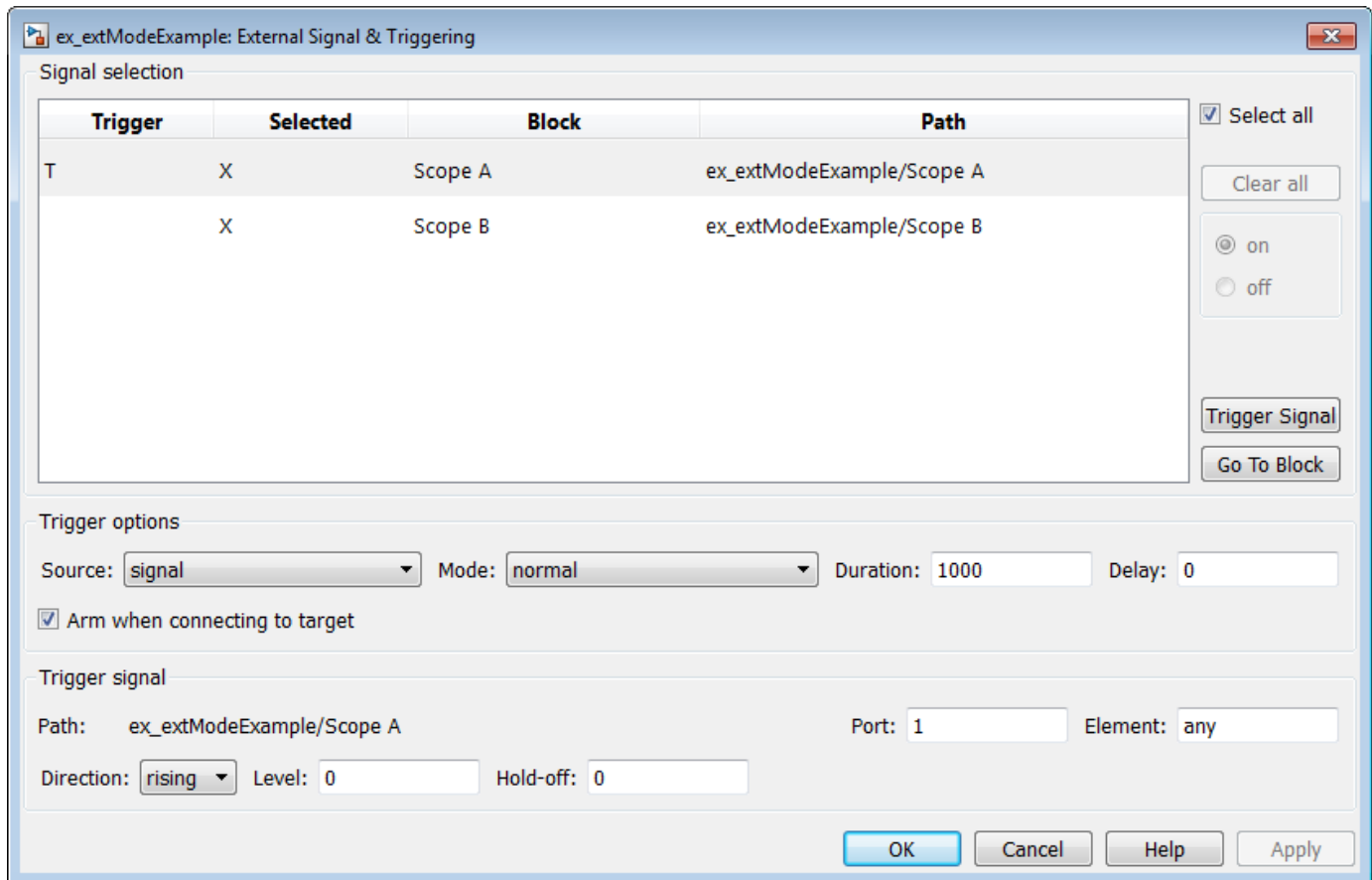
The **Duration** value specifies the number of contiguous points of data to be collected in each buffer of data. You should enter a **Duration** value equal to the number of continuous sample points that you need to collect rather than relying on a series of buffers to be continuous. If you enter a value less than the total number of sample points, you may lose sample points during the time spent transferring values from the data buffer to the MATLAB workspace. The Simulink software maintains point continuity only within one buffer. Between buffers, because of transfer time, some samples may be omitted.

The **Duration** value can affect the **Limit data points to last** value of Scope and To Workspace blocks. The number of sample points that the blocks save to the MATLAB workspace is the smaller of the two values. To set the number of sample points that the blocks save, clear **Limit data points to last**. Then, use **Duration** to specify the number of sample points saved.

Select Trigger Signal

You can designate one signal as a trigger signal. To select a trigger signal, from the **Source** menu in the **Trigger options** section, select **signal**. This action enables the parameters in the **Trigger signal** section. Then, select a signal in the **Signal selection** list, and click the **Trigger Signal** button.

When you select a signal to be a trigger, a T appears in the **Trigger** column of the **Signal selection** list. In the next figure, the **Scope A** signal is the trigger. **Scope B** is also selected for viewing, as indicated by the X in the **Selected** column.



After selecting the trigger signal, you can use the **Trigger signal** section to define the trigger conditions and set the trigger signal **Port** and **Element** parameters.

Set Trigger Conditions

Use the **Trigger signal** section of the External Signal & Triggering dialog box to set trigger conditions and attributes. **Trigger signal** parameters are enabled only when the trigger parameter **Source** is set to **signal** in the **Trigger options** section.

By default, any element of the first input port of a specified trigger block can cause the trigger to fire (that is, Port 1, any element). You can modify this behavior by adjusting the **Port** and **Element** values in the **Trigger signal** section. The **Port** field accepts a number or the keyword `last`. The **Element** field accepts a number or the keywords `any` or `last`.

In the **Trigger signal** section, you also define the conditions under which a trigger event occurs.

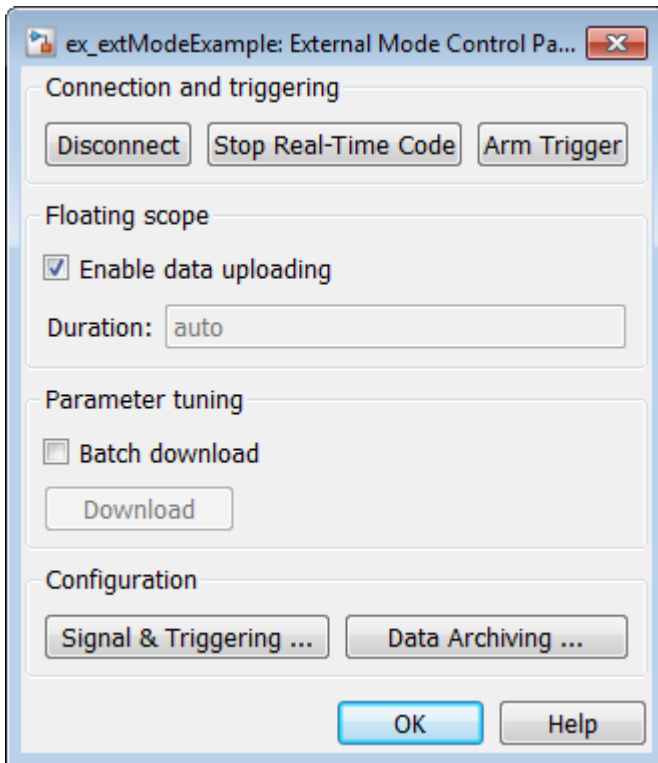
- **Direction:** `rising`, `falling`, or `either`. The direction in which the signal must be traveling when it crosses the threshold value. The default is `rising`.
- **Level:** A value indicating the threshold the signal must cross in a designated direction to fire the trigger. By default, the level is 0.
- **Hold-off:** Applies only to `normal` mode. Expressed in base rate steps, **Hold-off** is the time between the termination of one trigger event and the rearming of the trigger.

Modify Signal and Triggering Options While Connected

After you configure signal data uploading, and connect Simulink to a running target executable, you can modify signal and triggering options without disconnecting from the target.

If the trigger is armed (for example, if the trigger option **Arm when connecting to the target** is selected, which is the default), the External Signal & Triggering dialog box cannot be modified. To modify signal and triggering options:

- 1 Open the External Mode Control Panel.
- 2 Click **Cancel Trigger**. Triggering and display of uploaded data stops.
- 3 Open the External Signal & Triggering dialog box and modify signal and trigger options as required. For example, in the **Signal selection** section, you can enable or disable a scope, and in the **Trigger options** section, change the trigger **Mode**, for example, from `normal` to `one-shot`.
- 4 Click **Arm Trigger**. Triggering and display of uploaded data resumes, with your modifications.



Configure Host Archiving of Target Application Signal Data

In external mode, you can use the Simulink Scope and To Workspace blocks to archive data to disk.

To understand how the archiving features work, consider the handling of data when archiving is not enabled. There are two cases, one-shot mode and normal mode.

- In one-shot mode, after a trigger event occurs, each selected block writes its data to the workspace, as it would at the end of a simulation. If another one-shot is triggered, the existing workspace data is overwritten.
- In normal mode, external mode automatically rearms the trigger after each trigger event. Consequently, you can think of normal mode as a series of one-shots. Each one-shot in this series, except for the last, is referred to as an *intermediate result*. Because the trigger can fire at any time, writing intermediate results to the workspace can result in unpredictable overwriting of the workspace variables. For this reason, the default behavior is to write only the results from the final one-shot to the workspace. The intermediate results are discarded. If you know that enough time exists between triggers for inspection of the intermediate results, you can override the default behavior by selecting the **Write intermediate results to workspace** option. This option does not protect the workspace data from being overwritten by subsequent triggers.

If you use a Simulink Scope block to archive data to disk, open the Scope parameters dialog box and select the option **Log data to workspace**. The option is required for these reasons:

- The data is first transferred from the scope data buffer to the MATLAB workspace, before being written to a MAT-file.

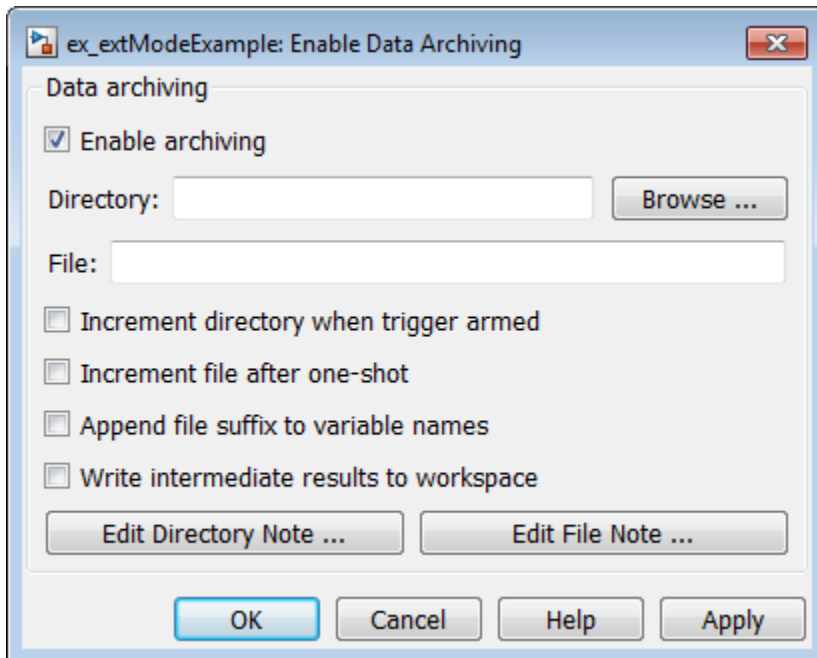
- The **Variable name** entered in the Scope parameters dialog box is the same as the one in the MATLAB workspace and the MAT-file. Enabling the data to be saved enables a variable named with the **Variable name** parameter to be saved to a MAT-file.

Note If you do not select the Scope block option **Log data to workspace**, the MAT-files for data logging are created, but they are empty.

The Enable Data Archiving dialog box supports:

- Folder notes
- File notes
- Automated data archiving

On the External Mode Control Panel, click the **Data Archiving** button to open the Enable Data Archiving dialog box. If your model is connected to the target environment, disconnect it while you configure data archiving. To enable the other controls in the dialog box, select **Enable archiving**.



These operations are supported by the Enable Data Archiving dialog box.

Folder Notes

To add annotations for a collection of related data files in a folder, in the Enable Data Archiving dialog box, click **Edit Directory Note**. The MATLAB editor opens. Place comments that you want saved to a file in the specified folder in this window. By default, the comments are saved to the folder last written to by data archiving.

File Notes

To add annotations for an individual data file, in the Enable Data Archiving dialog box, click **Edit File Note**. A file finder window opens, which by default is set to the last file to which you have written.

Selecting a MAT-file opens an edit window. In this window, add or edit comments that you want saved with your individual MAT-file.


Automated Data Archiving

To configure automatic writing of logging results to disk, optionally including intermediate results, use the **Enable archiving** option and the controls it enables. The dialog box provides the following related controls:

- **Directory:** Specifies the folder in which data is saved. If you select **Increment directory when trigger armed**, external mode appends a suffix.
- **File:** Specifies the name of the file in which data is saved. If you select **Increment file after one-shot**, external mode appends a suffix.
- **Increment directory when trigger armed:** Each time that you click the **Arm Trigger** button, external mode uses a different folder for writing log files. The folders are named incrementally, for example, `dirname1`, `dirname2`, and so on.
- **Increment file after one-shot:** New data buffers are saved in incremental files: `filename1`, `filename2`, and so on. File incrementing happens automatically in normal mode.
- **Append file suffix to variable names:** Whenever external mode increments file names, each file contains variables with identical names. Selecting **Append file suffix to variable name** results in each file containing unique variable names. For example, external mode saves a variable named `xdata` in incremental files (`file_1`, `file_2`, and so on) as `xdata_1`, `xdata_2`, and so on. This approach supports loading the MAT-files into the workspace and comparing variables at the MATLAB command prompt. Without the unique names, each instance of `xdata` would overwrite the previous one in the MATLAB workspace.
- **Write intermediate results to workspace:** If you want the Simulink Coder software to write intermediate results to the workspace, select this option.










Summary of Graphical Controls for TCP/IP or Serial External Mode Simulation

You can control a TCP/IP or serial external mode simulation through:

- The **Hardware** tab on the Simulink toolstrip. To display the **Hardware** tab, from the **Apps** tab on the Simulink toolstrip, click **Run on Custom Hardware**.
- The External Mode Control Panel. To open this dialog box, in Simulink Editor, on the **Hardware** tab, in the **Prepare** section, click . Then under **Signal Monitoring & Tracing**, click **Control Panel**.

This table lists external mode actions that you can perform by using controls that are available in the **Hardware** tab and the External Mode Control Panel.

External Mode Action	Hardware Tab	External Mode Control Panel
Build target application.	Run on Hardware > Build for Monitoring 	N/A

External Mode Action	Hardware Tab	External Mode Control Panel
Run application on target hardware.	Run on Hardware > Deploy 	N/A
Connect Simulink to a waiting or running target application.	Run on Hardware > Connect  When Simulink is connected to the target application, Connect appears dimmed and Disconnect  is active.	Connect When Simulink is connected to the target application, Connect changes to Disconnect .
Start real-time execution of generated code in the target environment.	Run on Hardware > Start 	Start Real-Time Code When the generated code starts executing, the button changes to Stop Real-Time Code .
Disconnect Simulink from the target environment, but do not stop real-time execution of code.	Run on Hardware > Disconnect  When Simulink is disconnected from the target application, Disconnect appears dimmed and Connect  is active.	Disconnect
Stop target application execution and disconnect Simulink from the target environment.	In Run on Hardware section, Stop button  .	Stop Real-Time Code
Tune batch of block parameters.	In Prepare section, Hold Updates  and Update All Parameters  buttons.	Batch download and Download

Blocks and Subsystems Compatible with External Mode

- “Compatible Blocks” on page 32-73
- “Signal Viewing Subsystems” on page 32-74
- “Supported Blocks for Data Archiving” on page 32-75

Compatible Blocks

In external mode, you can use the following types of blocks to receive and view signals uploaded from the target application:

- Floating Scope and Scope blocks
- Spectrum Analyzer and Time Scope blocks from the DSP System Toolbox product
- Display blocks
- To Workspace blocks
- User-written S-Function blocks

An external mode method is built into the S-function API. This method allows user-written blocks to support external mode. See `matlabroot/simulink/include/simstruc.h`.

- Record, XY Graph blocks

You can designate certain subsystems as Signal Viewing Subsystems and use them to receive and view signals uploaded from the target application. See “Signal Viewing Subsystems” on page 32-74 for more information.

You select external mode compatible blocks and subsystems, and arm the trigger, by using the External Signal & Triggering dialog box. By default, such blocks in a model are selected, and a manual trigger is set to be armed when connected to the target application.

Signal Viewing Subsystems

A Signal Viewing Subsystem is an atomic subsystem that encapsulates processing and viewing of signals received from the target system. A Signal Viewing Subsystem runs only on the host, and does not generate code in the target system. Signal Viewing Subsystems run in normal, accelerator, rapid accelerator, and external simulation modes.

Note Signal Viewing Subsystems are inactive if placed inside a SIL or PIL component, such as a top model in SIL or PIL mode, a Model block in SIL or PIL mode, or a SIL or PIL block. However, a SIL or PIL component can feed a Signal Viewing Subsystem running in a supported mode.

Signal Viewing Subsystems are useful in situations where you want to process or condition signals before viewing or logging them, but you do not want to perform these tasks on the target system. By using a Signal Viewing Subsystem, you can generate smaller and more efficient code on the target system.

Like other external mode compatible blocks, Signal Viewing Subsystems are displayed in the External Signal & Triggering dialog box.

To declare a subsystem to be a Signal Viewing Subsystem:

- 1 In the Block Parameters dialog box, select the **Treat as atomic unit** option.

For more information on atomic subsystems, see “Simulink Modeling Components”.

- 2 To turn the `SimViewingDevice` property on, use the `set_param` command:

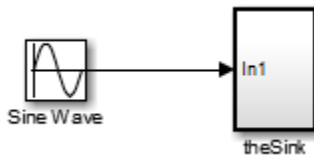
```
set_param('blockname', 'SimViewingDevice', 'on')
```

'blockname' is the name of the subsystem.

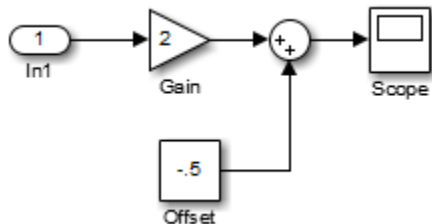
- 3 Make sure the subsystem meets the following requirements:

- It must be a pure Sink block. That is, it must not contain Outport blocks or Data Store blocks. It can contain Goto blocks only if the corresponding From blocks are contained within the subsystem boundaries.
- It must not have continuous states.

The following model, `sink_examp`, contains an atomic subsystem, `theSink`.



The subsystem `theSink` applies a gain and an offset to its input signal and displays it on a Scope block.



If `theSink` is declared as a Signal Viewing Subsystem, the generated target application includes only the code for the Sine Wave block. If `theSink` is selected and armed in the External Signal & Triggering dialog box, the target application uploads the sine wave signal to `theSink` during simulation. You can then modify the parameters of the blocks within `theSink` and observe the uploaded signal.

If `theSink` were not declared as a Signal Viewing Subsystem, its Gain, Constant, and Sum blocks would run as subsystem code on the target system. The Sine Wave signal would be uploaded to the Simulink engine after being processed by these blocks, and viewed on `sink_examp/theSink/Scope2`. Processing demands on the target system would be increased by the additional signal processing, and by the downloading of changes in block parameters from the host.

Supported Blocks for Data Archiving

In external mode, you can use the following types of blocks to archive data to disk:

- Scope blocks
- To Workspace blocks

You configure data archiving by using the Enable Data Archiving dialog box, as described in “Configure Host Archiving of Target Application Signal Data” on page 32-70.

External Mode Mechanism for Downloading Tunable Parameters

- “Download Mechanism” on page 32-76

- “Inlined and Tunable Parameters” on page 32-77

Depending on the setting of the **Default parameter behavior** option when the target application is generated, there are differences in the way parameter updates are handled. “Download Mechanism” on page 32-76 describes the operation of external mode communication with **Default parameter behavior** set to **Tunable**. “Inlined and Tunable Parameters” on page 32-77 describes the operation of external mode with **Default parameter behavior** set to **Inlined**.

Download Mechanism

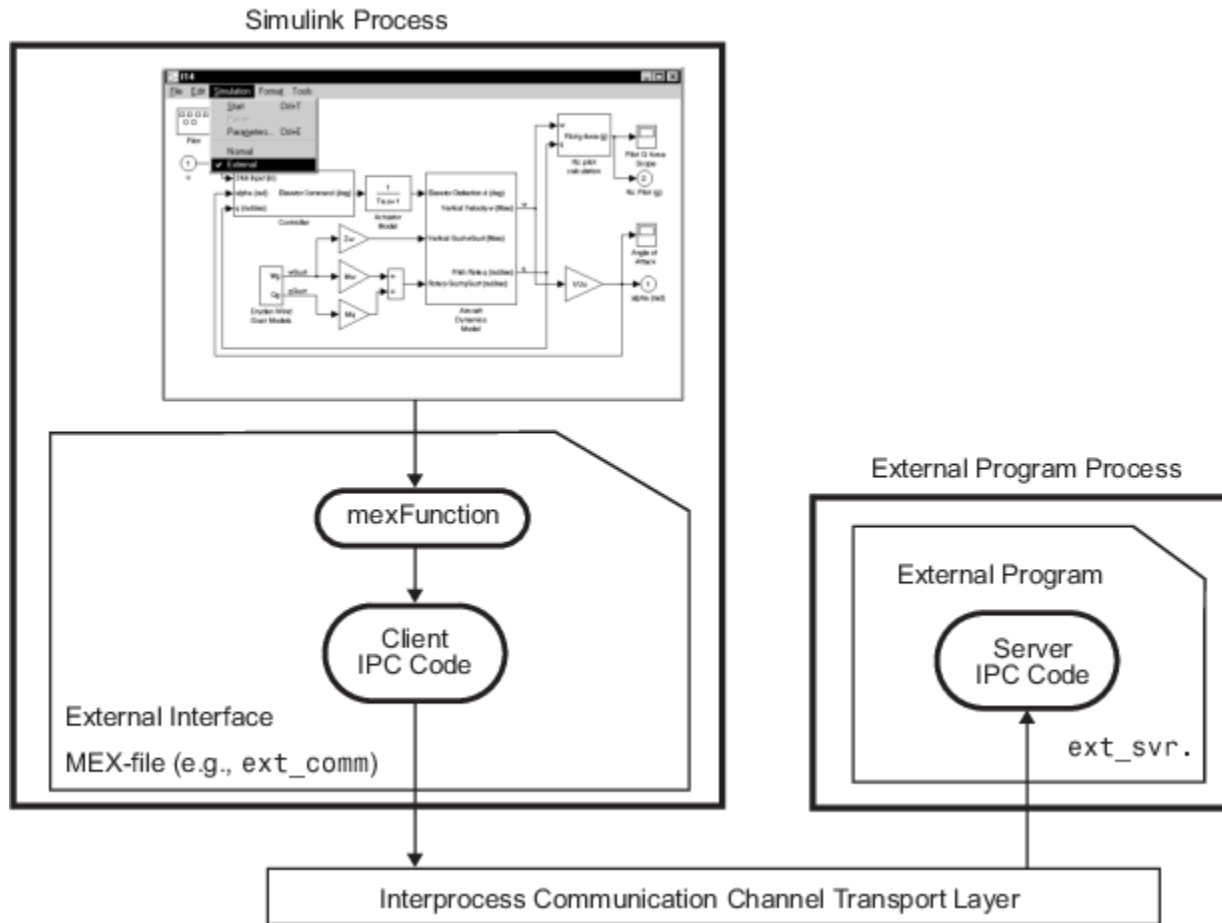
In external mode, the Simulink engine does not simulate the system represented by the block diagram. By default, when external mode is enabled, the Simulink engine downloads parameters to the target system. After the initial download, the engine remains in a waiting mode until you change parameters in the block diagram or until the engine receives data from the target.

When you change a parameter in the block diagram, the Simulink engine calls the external interface MEX-file, passing new parameter values (along with other information) as arguments. The external interface MEX-file contains code that implements one side of the interprocess communication (IPC) channel. This channel connects the Simulink process (where the MEX-file executes) to the process that is executing the external program. The MEX-file transfers the new parameter values by using this channel to the external program.

The other side of the communication channel is implemented within the external program. This side writes the new parameter values into the target's parameter structure (*model_P*).

The Simulink side initiates the parameter download operation by sending a message containing parameter information to the external program. In the terminology of client/server computing, the Simulink side is the client and the external program is the server. The two processes can be remote, or they can be local. Where the client and server are remote, a protocol such as TCP/IP is used to transfer data. Where the client and server are local, a serial connection or shared memory can be used to transfer data.

The next figure shows this relationship. The Simulink engine calls the external interface MEX-file whenever you change parameters in the block diagram. The MEX-file then downloads the parameters to the external program by using the communication channel.



External Mode Architecture

Inlined and Tunable Parameters

By default, parameters (except those listed in “TCP/IP and Serial External Mode Limitations” on page 32-90) in an external mode program are tunable; that is, you can change them by using the download mechanism described in this section.

If you set **Default parameter behavior** to **Inlined** (on the **Optimization** pane of the Configuration Parameters dialog box), the Simulink Coder code generator embeds the numerical values of model parameters (constants), instead of symbolic parameter names, in the generated code. Inlining parameters generates smaller and more efficient code. However, inlined parameters, because they effectively become constants, are not tunable.

The Simulink Coder software lets you improve overall efficiency by inlining most parameters, while at the same time retaining the flexibility of run-time tuning for selected parameters that are important to your application. When you inline parameters, you can use `Simulink.Parameter` objects to remove individual parameters from inlining and declare them to be tunable. In addition, you can use these objects to control how parameters are represented in the generated code.

For more information on tunable parameters, see “Create Tunable Calibration Parameter in the Generated Code” on page 15-112.

Automatic Parameter Uploading on Host/Target Connection

Each time the Simulink engine connects to a target application that was generated with **Default parameter behavior** set to **Inlined**, the target application uploads the current value of its tunable parameters to the host. These values are assigned to the corresponding MATLAB workspace variables. This procedure synchronizes the host and target with respect to parameter values. If the tunable parameters are stored in the model workspace or a data dictionary, the values are not uploaded and Simulink produces a warning.

Workspace variables required by the model must be initialized at the time of host/target connection. Otherwise the uploading cannot proceed and an error results. Once the connection is made, these variables are updated to reflect the current parameter values on the target system.

Automatic parameter uploading takes place only if the target application was generated with **Default parameter behavior** set to **Inlined**. “Download Mechanism” on page 32-76 describes the operation of external mode communication with **Default parameter behavior** set to **Tunable**.

Choose Communication Protocol for Client and Server

- “Introduction” on page 32-78
- “Using the TCP/IP Implementation” on page 32-78
- “Using the Serial Implementation” on page 32-81
- “Run the External Program” on page 32-82
- “Implement an External Mode Protocol Layer” on page 32-84

Introduction

The Simulink Coder product provides code to implement both the client and server side of external mode communication using either TCP/IP or serial protocols. You can use the socket-based external mode implementation provided by the Simulink Coder product with the generated code, provided that your target system supports TCP/IP. If not, use or customize the serial transport layer option provided.

A low-level *transport layer* handles physical transmission of messages. Both the Simulink engine and the model code are independent of this layer. Both the transport layer and code directly interfacing to the transport layer are isolated in separate modules that format, transmit, and receive messages and data packets.

Using the TCP/IP Implementation

You can use TCP/IP-based client/server implementation of external mode with real-time programs on The Open Group UNIX or PC systems. For help in customizing external mode transport layers, see “Create a Transport Layer for TCP/IP or Serial External Mode Communication” on page 32-94.

To use Simulink external mode over TCP/IP:

- Make sure that the external interface MEX-file for your target's TCP/IP transport is specified.

Targets provided by MathWorks specify the name of the external interface MEX-file in `matlabroot/toolbox/simulink/simulink/extmode_transports.m`. The name of the interface appears as uneditable text in the **External mode configuration** section of the **Interface** pane of the Configuration Parameters dialog box. The TCP/IP default is `ext_comm`.

To specify a TCP/IP transport for a custom target, you must add an entry of the following form to an `sl_customization.m` file on the MATLAB path:

```
function sl_customization(cm)
    cm.ExtModeTransports.add('stf.tlc', 'transport', 'mexfile', 'Level1');
%end function
```

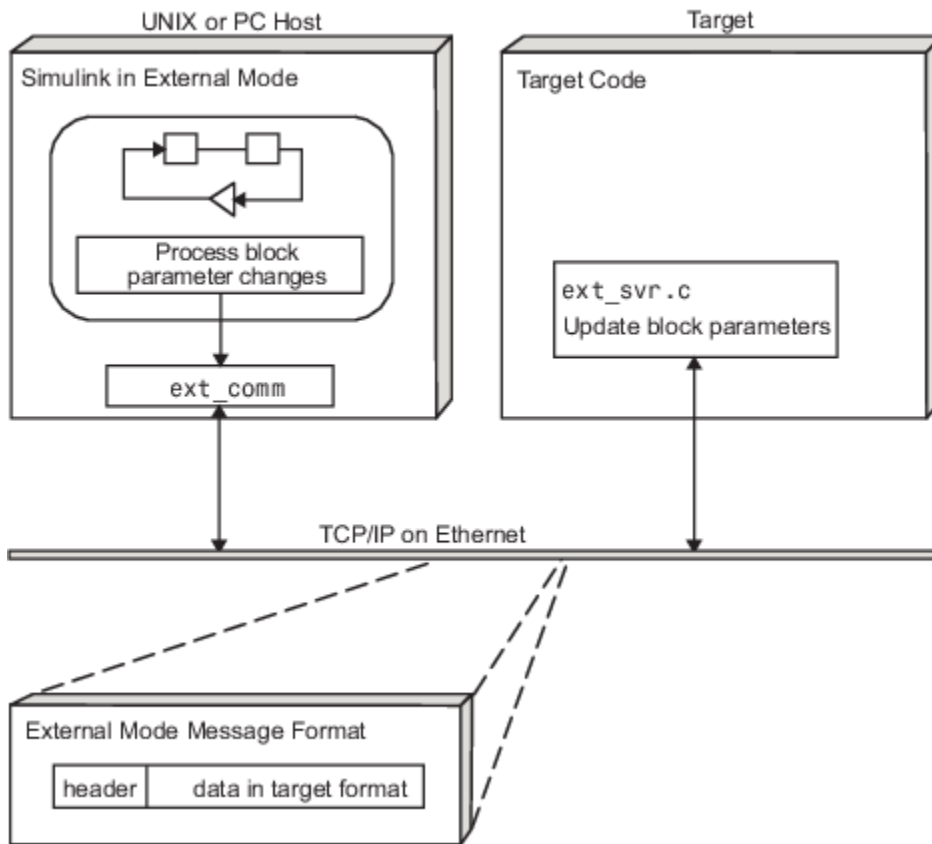
- `stf.tlc` is the name of the system target file for which you are registering the transport (for example, 'mytarget.tlc')
- `transport` is the transport name to display in the **Transport layer** menu on the **Interface** pane of the Configuration Parameters dialog box (for example, 'tcpip')
- `mexfile` is the name of the transport's associated external interface MEX-file (for example, 'ext_comm')

You can specify multiple targets and/or transports with additional `cm.ExtModeTransports.add` lines, for example:

```
function sl_customization(cm)
    cm.ExtModeTransports.add('mytarget.tlc', 'tcpip', 'ext_comm', 'Level1');
    cm.ExtModeTransports.add('mytarget.tlc', 'serial', ...
        'ext_serial_win32_comm', 'Level1');
%end function
```

- Be sure that the template makefile is configured to link the source files for the TCP/IP server code and that it defines the compiler flags when building the generated code.
- Build the external program.
- Run the external program.
- Set the Simulink model to external mode and connect to the target.

The next figure shows the structure of the TCP/IP-based implementation.



TCP/IP-Based Client/Server Implementation for External Mode

MEX-File Optional Arguments for TCP/IP Transport

In the External Target Interface dialog box, you can specify optional arguments that are passed to the external mode interface MEX-file for communicating with executing targets.

- Target network name: the network name of the computer running the external program. By default, this is the computer on which the Simulink product is running, for example, 'myComputer'. You can also use the IP address, for example, '148.27.151.12'.
- Verbosity level: controls the level of detail of the information displayed during the data transfer. The value is either 0 or 1 and has the following meaning:
 - 0 — No information
 - 1 — Detailed information
- TCP/IP server port number: The default value is 17725. You can change the port number to a value between 256 and 65535 to avoid a port conflict.

The arguments are positional and must be specified in the following order:

<TargetNetworkName> <VerbosityLevel> <ServerPortNumber>

For example, if you want to specify the verbosity level (the second argument), then you must also specify the target network name (the first argument). Arguments can be delimited by white space or commas. For example:

```
'148.27.151.12' 1 30000
```

You can specify command-line options to the external program when you launch it. See “Run the External Program” on page 32-82.

Using the Serial Implementation

Controlling host/target communication on a serial channel is similar to controlling host/target communication on a TCP/IP channel.

To use Simulink external mode over a serial channel, you must:

- Make sure that the external interface MEX-file for your target's serial transport is specified.

Targets provided by MathWorks specify the name of the external interface MEX-file in *matlabroot/toolbox/simulink/simulink/extmode_transports.m*. The name of the interface appears as uneditable text in the **External mode configuration** section of the **Interface** pane of the Configuration Parameters dialog box. The serial default is `serial`.

To specify a serial transport for a custom target, you must add an entry of the following form to an *sl_customization.m* file on the MATLAB path:

```
function sl_customization(cm)
    cm.ExtModeTransports.add('stf.tlc', 'transport', 'mexfile', 'Level1');
%end function
```

- `stf.tlc` is the name of the system target file for which you are registering the transport (for example, 'mytarget.tlc')
- `transport` is the transport name to display in the **Transport layer** menu on the **Interface** pane of the Configuration Parameters dialog box (for example, 'serial')
- `mexfile` is the name of the transport's associated external interface MEX-file (for example, 'ext_serial_win32_comm')

You can specify multiple targets and/or transports with additional `cm.ExtModeTransports.add` lines, for example:

```
function sl_customization(cm)
    cm.ExtModeTransports.add('mytarget.tlc', 'tcpip', 'ext_comm', 'Level1');
    cm.ExtModeTransports.add('mytarget.tlc', 'serial', ...
        'ext_serial_win32_comm', 'Level1');
%end function
```

- Be sure that the template makefile is configured to link the source files for the serial server code and that it defines the compiler flags when building the generated code.
- Build the external program.
- Run the external program.
- Set the Simulink model to external mode and connect to the target.

MEX-File Optional Arguments for Serial Transport

In the **MEX-file arguments** field of the **Interface** pane of the Configuration Parameters dialog box, you can specify arguments that are passed to the external mode interface MEX-file for communicating with the executing targets. For serial transport, the optional arguments to `ext_serial_win32_comm` are as follows:

- **Verbosity level:** This argument controls the level of detail of the information displayed during data transfer. The value of this argument is:

- 0 (no information), or
- 1 (detailed information)
- Serial port ID: The port ID of the host, specified as an integer or character vector. For example, specify the port ID of a USB to serial converter as `'/dev/ttyusb0'`. Simulink Coder prefixes integer port IDs with `\\.COM` on Windows and by `/dev/ttyS` on Unix.

When you start the target application using a serial connection, you must specify the port ID to use to connect it to the host. Do this by including the `-port` command-line option. For example:

```
mytarget.exe -port 2 -w
```

- Baud rate: Specify an integer value. The default value is 57600.

The MEX-file options arguments are positional and must be specified in the following order:

```
<VerbosityLevel> <SerialPortID> <BaudRate>
```

For example, if you want to specify the serial port ID (the second argument), then you must also specify the verbosity level (the first argument). Arguments can be delimited by white space or commas. For example:

```
1 '/dev/ttyusb0' 57600
```

When you launch the external program, you can specify command-line options.

Run the External Program

Before you can use the Simulink product in external mode, the external program must be running.

If the target application is executing on the same machine as the host and communication is through a loopback serial cable, the target's port ID must differ from that of the host (as specified in the **MEX-file arguments** edit field).

To run the external program, you type a command of the form:

```
model -opt1 ... -optN
```

`model` is the name of the external program and `-opt1 ... -optN` are options. (See “Command-Line Options for the External Program” on page 32-83.) In the examples in this section, the name of the external program is `ext_example`.

Running the External Program in the Windows Environment

In the Windows environment, you can run the external programs in either of the following ways:

- Open a Command Prompt window. At the command prompt, type the name of the target executable, followed by possible options, such as:

```
ext_example -tf inf -w
```

- Alternatively, you can launch the target executable from the MATLAB Command Window. The command must be preceded by an exclamation point (!) and followed by an ampersand (&), as in the following example:

```
!ext_example -tf inf -w &
```

The ampersand (&) causes the operating system to spawn another process to run the target executable. If you do not include the ampersand, the program still runs, but you cannot enter commands at the MATLAB command prompt or manually terminate the executable.

Running the External Program in the UNIX Environment

In the UNIX environment, you can run the external programs in either of the following ways:

- Open an Xterm window. At the command prompt, type the name of the target executable, followed by possible options, such as:

```
./ext_example -tf inf -w
```

- Alternatively, you can launch the target executable from the MATLAB Command Window. You must run it in the background so that you can still access the Simulink environment. The command must be preceded by an exclamation point (!), dot slash (./ indicating the current directory), and followed by an ampersand (&), as in the following example:

```
!./ext_example -tf inf -w &
```

The ampersand (&) causes the operating system to spawn another process to run the target executable.

Command-Line Options for the External Program

External mode target executables generated by the Simulink Coder code generator support the following command-line options:

- `-tf n`

The `-tf` option overrides the stop time set in the Simulink model. The argument `n` specifies the number of seconds the program will run. The value `inf` directs the model to run indefinitely. In this case, the model code runs until the target application receives a stop message from the Simulink engine.

The following example sets the stop time to 10 seconds.

```
ext_example -tf 10
```

When integer-only ERT targets are built and executed in external mode, the stop time parameter (`-tf`) is interpreted by the target as the number of base rate steps rather than the number of seconds to execute.

- `-w`

Instructs the target application to enter a wait state until it receives a message from the host. At this point, the target is running, but not executing the model code. The start message is sent when you select **Start Real-Time Code** from the **Simulation** menu or click the **Start Real-Time Code** button in the External Mode Control Panel.

Use the `-w` option if you want to view target application execution data from time step 0, or if you want to modify parameters before the target application begins execution of model code.

- `-port n`

Specifies the TCP/IP port number or the serial port ID, `n`, for the target application. The port number of the target application must match that of the host for TCP/IP transport. The port number depends on the type of transport.

- For TCP/IP transport: Port number is an integer between 256 and 65535, with the default value being 17725. The selected port must not be used by another TCP/IP service that runs on the target hardware.
- For serial transport: Port ID is an integer or a character vector. For example, specify the port ID of a USB to serial converter as `'/dev/ttyusb0'`
- `-baud r`

Specified as an integer, this option is only available for serial transport.

Implement an External Mode Protocol Layer

If you want to implement your own transport layer for external mode communication, you must modify certain code modules provided by the Simulink Coder product and create a new external interface MEX-file. See “Create a Transport Layer for TCP/IP or Serial External Mode Communication” on page 32-94.

Use External Mode Programmatically

You can run external mode simulations from the MATLAB command line or programmatically in scripts. Use the `get_param` and `set_param` commands to retrieve and set the values of model simulation command-line parameters, such as `SimulationMode` and `SimulationCommand`, and external mode command-line parameters, such as `ExtModeCommand` and `ExtModeTrigType`.

The following model simulation commands assume that a Simulink model is open and that you have loaded a target application to which the model will connect using external mode.

- 1 Change the Simulink model to external mode:

```
set_param(gcs, 'SimulationMode', 'external')
```

- 2 Connect the open model to the loaded target application:

```
set_param(gcs, 'SimulationCommand', 'connect')
```

- 3 Start running the target application:

```
set_param(gcs, 'SimulationCommand', 'start')
```

- 4 Stop running the target application:

```
set_param(gcs, 'SimulationCommand', 'stop')
```

- 5 Disconnect the target application from the model:

```
set_param(gcs, 'SimulationCommand', 'disconnect')
```

The `set_param` commands that use the `'SimulationCommand'` argument are asynchronous. If you run the commands successively from a script, each command starts without waiting for the previous command to complete. To check that each command is complete, in the script, use the `get_param` command with the `'SimulationStatus'` argument. For example, for steps 1 to 3, specify these commands in the script:

```
set_param(gcs, 'SimulationMode', 'external');
set_param(gcs, 'SimulationCommand', 'connect');
```

```
isExternalSimulationActive = false;
while ~isExternalSimulationActive
```

```

    simStatus = get_param(gcs, 'SimulationStatus');
    isExternalSimulationActive = strcmp(simStatus, 'external');
end
set_param(gcs, 'SimulationCommand', 'start');

```

For more information, see “Run Simulations Programmatically”.

The Diagnostic Viewer displays error messages produced by the `get_param` and `set_param` commands.

To tune a workspace parameter, change its value at the command prompt. If the workspace parameter is a `Simulink.Parameter` object, assign the new value to the `Value` property.

```

myVariable = 5.23;
myParamObj.Value = 5.23;

```

To download the workspace parameter in external mode, you update the model diagram. The following model simulation command initiates a model update:

```
set_param(gcs, 'SimulationCommand', 'update')
```

To trigger or cancel data uploading to scopes, use the `ExtModeCommand` values `armFloating` and `cancelFloating`, or `armWired` and `cancelWired`. For example, to trigger and then cancel data uploading to wired (nonfloating) scopes:

```

set_param(gcs, 'ExtModeCommand', 'armWired')
set_param(gcs, 'ExtModeCommand', 'cancelWired')

```

The next table lists external mode command-line parameters that you can use in `get_param` and `set_param` commands. The table provides brief descriptions, valid values (bold type highlights defaults), and a mapping to External Mode dialog box equivalents. For external mode parameters that are equivalent to **Interface** pane options in the Configuration Parameters dialog box, see “Model Configuration Parameters: Code Generation Interface”.

External Mode Command-Line Parameters

Parameter and Values	Dialog Box Equivalent	Description
ExtModeAddSuffixToVar off , on	Enable Data Archiving: Append file suffix to variable names check box	Increment variable names for each incremented filename.
ExtModeArchiveDirName <i>character vector</i>	Enable Data Archiving: Directory text field	Save data in specified folder.
ExtModeArchiveFileName <i>character vector</i>	Enable Data Archiving: File text field	Save data in specified file.
ExtModeArchiveMode <i>character vector</i> - off , auto, manual	Enable Data Archiving: Enable archiving check box	Activate automated data archiving features. To specify manual, run <code>set_param(gcs, 'ExtModeArchiveMode', 'manual')</code> . Note that if you specify auto , <code>ExtModeAutoIncOneShot</code> is set to on .
ExtModeArmWhenConnect off , on	External Signal & Triggering: Arm when connecting to target check box	Arm the trigger as soon as the Simulink Coder software connects to the target.
ExtModeAutoIncOneShot off , on	Enable Data Archiving: Increment file after one-shot check box	Save new data buffers in incremental files.
ExtModeAutoUpdateStatusClock (Windows platforms only) off , on	Not available	Continuously upload and display target time on the model window status bar.
ExtModeBatchMode off , on	External Mode Control Panel: Batch download check box	Enable or disable downloading of parameters in batch mode.
ExtModeChangesPending off , on	Not available	When <code>ExtModeBatchMode</code> is enabled, indicates whether parameters remain in the queue of parameters to be downloaded to the target.

Parameter and Values	Dialog Box Equivalent	Description
ExtModeCommand <i>character vector</i> - armFloating, armWired, cancelFloating, cancelWired	<ul style="list-style-type: none"> armFloating and cancelFloating are equivalent to selecting and clearing External Mode Control Panel check box Floating scope > Enable data uploading armWired and cancelWired are equivalent to External Mode Control Panel buttons Arm Trigger and Cancel Trigger 	Issue an external mode command to the target application.
ExtModeConnected off , on	External Mode Control Panel: Connect/Disconnect button	Indicate the state of the connection with the target application.
ExtModeEnableFloating off, on	External Mode Control Panel: Enable data uploading check box	Enable or disable the arming and canceling of triggers when a connection is established with floating scopes.
ExtModeIncDirWhenArm off , on	Enable Data Archiving: Increment directory when trigger armed check box	Write log files to incremental folders each time the trigger is armed.
ExtModeLogAll off, on	External Signal & Triggering: Select all check box	Upload available signals from the target to the host.
ExtModeParamChangesPending off , on	Not available	When the Simulink Coder software is connected to the target and ExtModeBatchMode is enabled, indicates whether parameters remain in the queue of parameters to be downloaded to the target. More efficient than ExtModeChangesPending, because it checks for a connection to the target.
ExtModeSkipDownloadWhenConnect off , on	Not available	Connect to the target application without downloading parameters.
ExtModeTrigDelay <i>integer (0)</i>	External Signal & Triggering: Delay text field	Specify the amount of time (expressed in base rate steps) that elapses between a trigger occurrence and the start of data collection.
ExtModeTrigDirection <i>character vector</i> - rising , falling, either	External Signal & Triggering: Direction menu	Specify the direction in which the signal must be traveling when it crosses the threshold value.

Parameter and Values	Dialog Box Equivalent	Description
ExtModeTrigDuration <i>integer (1000)</i>	External Signal & Triggering: Duration text field	Specify the number of base rate steps for which external mode is to log data after a trigger event.
ExtModeTrigDurationFloating <i>character vector - integer (auto)</i>	External Mode Control Panel: Duration text field	Specify the duration for floating scopes. If auto is specified, the value of ExtModeTrigDuration is used.
ExtModeTrigElement <i>character vector - integer, any, last</i>	External Signal & Triggering: Element text field	Specify the elements of the input port of the specified trigger block that can cause the trigger to fire.
ExtModeTrigHoldOff <i>integer (0)</i>	External Signal & Triggering: Hold-off text field	Specify the base rate steps between when a trigger event terminates and the trigger is rearmed.
ExtModeTrigLevel <i>integer (0)</i>	External Signal & Triggering: Level text field	Specify the threshold value the trigger signal must cross to fire the trigger.
ExtModeTrigMode <i>character vector - normal, oneshot</i>	External Signal & Triggering: Mode menu	Specify whether the trigger is to rearm automatically after each trigger event or whether only one buffer of data is to be collected each time the trigger is armed.
ExtModeTrigPort <i>character vector - integer (1), last</i>	External Signal & Triggering: Port text field	Specify the input port of the specified trigger block for which elements can cause the trigger to fire.
ExtModeTrigType <i>character vector - manual, signal</i>	External Signal & Triggering: Source menu	Specify whether to start logging data when the trigger is armed or when a specified trigger signal satisfies trigger conditions.
ExtModeUploadStatus <i>character vector - inactive, armed, uploading</i>	Not available	Return the status of the external mode upload mechanism — inactive, armed, or uploading.
ExtModeWriteAllDataToWs off, on	Enable Data Archiving: Write intermediate results to workspace check box	Write intermediate results to the workspace.

Animate Stateflow Charts in External Mode

If you have Stateflow, you can animate a chart in external mode. In external mode, you can animate states in a chart, and view test point signals in a floating scope or signal viewer.

- “Animate States During Simulation in External Mode” on page 32-89
- “View Test Point Data in Floating Scopes and Signal Viewers” on page 32-89

Animate States During Simulation in External Mode

To animate states in a chart in external mode:

- 1 Load the chart you want to animate to the target machine.
- 2 In the Configuration Parameters dialog box, select the **External mode** check box.
- 3 Open the External Mode Control Panel dialog box.
- 4 Click **Signal & Triggering**.
- 5 In the External Signal & Triggering dialog box, set these parameters.

In:	Select:
Signal selection pane	Chart you want to animate
Trigger options pane	Arm when connecting to target check box
Trigger options pane	normal from drop-down menu in Mode field

- 6 Build the model to generate an executable file.
- 7 Deploy the target application.
- 8 Connect Simulink to the target application.
- 9 Start execution of the generated model code. The chart highlights states as they execute.

View Test Point Data in Floating Scopes and Signal Viewers

When you simulate a chart in external mode, you can designate chart data of local scope to be test points and view the test point data in floating scopes and signal viewers.

To view test point data during simulation in external mode:

- 1 Open the Model Explorer and for each data you want to view, follow these steps:
 - a In the middle **Contents** pane, select the state or local data of interest.
 - b In the right **Dialog** pane, select the **Logging** tab and select **Test point** check box.
- 2 From a floating scope or signal viewer, click the signal selection button:



The Signal Selector dialog box opens.

- 3 In the Signal Selector **Model hierarchy** pane, select the chart.
- 4 In the Signal Selector **List contents** menu, select **Testpointed/Logged signals only** and then select the signals you want to view.
- 5 Simulate the model in external mode as described in “Animate States During Simulation in External Mode” on page 32-89.

The scope or viewer displays the values of the test point signals as the simulation runs.

For more information, see “Behavior of Scopes and Viewers with Rapid Accelerator Mode”.

TCP/IP and Serial External Mode Limitations

Feature	Details
Service code interface	Using external mode for tuning parameters and monitoring signals as generated code executes in a target environment is not supported for code generated from component models configured with an ERT-based system target file and service code interface.
Changing parameters	<p>In general, you cannot change a parameter if doing so results in a change in the structure of the model. For example, you cannot change</p> <ul style="list-style-type: none"> • The number of states, inputs, or outputs of a block • The sample time or the number of sample times • The integration algorithm for continuous systems • The name of the model or of a block <p>If you make these changes to the block diagram, then you must rebuild the program with newly generated code.</p> <p>You can change parameters in transfer function and state space representation blocks in specific ways:</p> <ul style="list-style-type: none"> • The parameters (numerator and denominator polynomials) for the Transfer Fcn (continuous and discrete) and Discrete Filter blocks can be changed (as long as the number of states does not change). • Zero entries in the State-Space and Zero Pole (both continuous and discrete) blocks in the user-specified or computed parameters (that is, the A, B, C, and D matrices obtained by a zero-pole to state-space transformation) cannot be changed once external simulation is started. • In the State-Space block, if you specify the matrices in the controllable canonical realization, then all changes to the A, B, C, D matrices that preserve this realization and the dimensions of the matrices are allowed. <p>If the Simulink block diagram does not match the external program, Simulink produces an error stating that the checksums do not match. The checksums take into account the top models, but not referenced models. Use the updated block diagram to rebuild the target application.</p>
Uploading data	For parameters with fixed-point values, uploading of parameter values from the target application is not supported.
Uploading variable-size signals	<p>Uploading of variable-size signals is not supported for these targets:</p> <ul style="list-style-type: none"> • Simulink Real-Time • Texas Instruments® C2000™

Feature	Details
Signal value display in simulation	Graphical display of signal values in models (described in “Display Signal Values in Model Diagrams”) is not supported. For example, you cannot use the Data Display in Simulation menu selections Show Value Labels When Hovering , Toggle Value Labels When Clicked , and Show Value Label of Selected Port .
Tunable structure parameters	Uploading or downloading of tunable structure parameters is not supported.
C++ code	Generated C++ code is not supported. The simulation produces an error if you specify these configuration parameter settings: <ul style="list-style-type: none"> • Language — C++ • Code interface packaging — C++ class
Pure integer code	Pure integer code is supported. If you do not specify <code>-tf finalTime</code> in the execution command, the target application runs the generated model code indefinitely, ignoring <code>StopTime</code> . If you specify <code>-tf finalTime</code> in the execution command: <ul style="list-style-type: none"> • The <i>finalTime</i> value represents base rate steps, not seconds. • The maximum value for <i>finalTime</i>, in ticks, is <code>MAX_int32_T</code>. • When the 16-bit or 32-bit tick counter overflows, the simulation time in Scope blocks returns to zero.
Half precision	Half-precision data types are not supported.
Simulink.ImageType	Simulink.ImageType data type is not supported.
Archiving data	For archiving data to disk, Scope and To Workspace blocks are supported. However, other scopes are not supported for data archiving. For example, you cannot use Floating Scope blocks or Viewers and Generators Manager viewer objects to archive data.
Scopes in referenced models	In a model hierarchy, if the top model simulates in external mode and a referenced model simulates in normal or accelerator mode, scopes in the referenced model are not displayed. However, if the top model is changed to simulate in normal mode, the behavior of scopes in the referenced models differs between normal and accelerator mode. Scopes in a referenced model simulating in normal mode are displayed, while scopes in a referenced model simulating in accelerator mode are not displayed.
Scope and Display blocks connected to Simulink message blocks	During external mode simulations, Scope and Display blocks connected to Simulink message blocks do not provide displays.
Simulation start time	Nonzero simulation start times are not supported. In the Configuration Parameters dialog box, Solver pane, leave Start time set to the default value of <code>0.0</code> .

Feature	Details
Intermediate step values	<p>Some Simulink blocks can generate multiple values in a simulation time step. For example:</p> <ul style="list-style-type: none"> • A Simulink Function or Function-Call Subsystem block that is invoked multiple times in a time step. • Iterator subsystem blocks that execute multiple times in a time step. <p>For each time step in an external mode simulation, Simulink uploads from the target application only the final values of such blocks. Simulink does not upload the intermediate values generated during the step.</p>
File-scoped data	File-scoped data are not supported, for example, data items to which you apply the built-in custom storage class, FileScope. File-scoped data are not externally accessible.
Signals with custom storage classes	Uploading of signals with custom storage classes (CSC) is not supported.
Use of printf Statements	To show target application error and information messages on the target hardware display, you can use printf calls. For some target hardware, the use of printf statements can increase the external mode binary file size. To disable printf calls, specify the preprocessor macro definition EXTMODE_DISABLEPRINTF for your target application compiler.
Command-line arguments	<p>You can use command-line arguments for running target applications. These limitations apply:</p> <ul style="list-style-type: none"> • Parsing of the command-line arguments requires the sscanf function, which increases the program size for some target hardware. • Some target applications do not accept command-line arguments. <p>If your target hardware does not support the parsing of command-line arguments, specify the preprocessor macro definition EXTMODE_DISABLE_ARGS_PROCESSING=1 for your target application compiler.</p> <p>To replace the -w option, you can use this command to specify that the target application enters and stays in a wait state until it receives a Connect message from Simulink:</p> <pre>set_param(modelName, 'OnTargetWaitForStart', 'on');</pre> <p>The build process provides the required option (-DON_TARGET_WAIT_FOR_START=1) to the compiler.</p>
Row-major code generation	Code generated with the row-major format is not supported.
Simulation data export	The export of simulation output, signal, and state data is not supported.

See Also

More About

- “External Mode Simulation by Using XCP Communication” on page 32-8
- “Create a Transport Layer for TCP/IP or Serial External Mode Communication” on page 32-94
- “Customize Connectivity for TCP/IP or Serial External Mode Simulations” on page 32-40

Create a Transport Layer for TCP/IP or Serial External Mode Communication

In this section...

“Design of External Mode” on page 32-94

“External Mode Communications Overview” on page 32-96

“External Mode Source Files” on page 32-97

“Implement a Custom Transport Layer” on page 32-100

This section helps you to connect your custom target by using external mode using your own low-level communications layer. The topics include:

- An overview of the design and operation of external mode
- A description of external mode source files
- Guidelines for modifying the external mode source files and building an executable to handle the tasks of the default `ext_comm` MEX-file

This section assumes that you are familiar with the execution of Simulink Coder programs, and with the basic operation of external mode.

Design of External Mode

External mode communication between the Simulink engine and a target system is based on a client/server architecture. The client (the Simulink engine) transmits messages requesting the server (target) to accept parameter changes or to upload signal data. The server responds by executing the request.

A low-level *transport layer* handles physical transmission of messages. Both the Simulink engine and the model code are independent of this layer. Both the transport layer and code directly interfacing to the transport layer are isolated in separate modules that format, transmit, and receive messages and data packets.

This design makes it possible for different targets to use different transport layers. The GRT, ERT, and RSim targets support host/target communication by using TCP/IP and RS-232 (serial) communication. The Simulink Desktop Real-Time target supports shared memory communication. The Wind River Systems Tornado® target supports TCP/IP only.

The Simulink Coder product provides full source code for both the client and server-side external mode modules, as used by the GRT, ERT, Rapid Simulation, and Tornado targets, and the Simulink Desktop Real-Time and Simulink Real-Time products. The main client-side module is `ext_comm.c`. The main server-side module is `ext_svr.c`.

These two modules call the specified transport layer through the following source files.

Built-In Transport Layer Implementations

Protocol	Client or Server?	Source Files
TCP/IP	Client (host)	<ul style="list-style-type: none"> <i>matlabroot/toolbox/coder/simulinkcoder_core/ext_mode/host/common/rtiostream_interface.c</i> <i>matlabroot/toolbox/coder/rtiostream/src/rtiostreamtcpip/rtiostream_tcpip.c</i>
	Server (target)	<ul style="list-style-type: none"> <i>matlabroot/rw/c/src/ext_mode/common/rtiostream_interface.c</i> <i>matlabroot/toolbox/coder/rtiostream/src/rtiostreamtcpip/rtiostream_tcpip.c</i>
Serial	Client (host)	<ul style="list-style-type: none"> <i>matlabroot/toolbox/coder/simulinkcoder_core/ext_mode/host/serial/ext_serial_transport.c</i> <i>matlabroot/toolbox/coder/rtiostream/src/rtiostreamserial/rtiostream_serial.c</i>
	Server (target)	<ul style="list-style-type: none"> <i>matlabroot/rw/c/src/ext_mode/serial/ext_svr_serial_transport.c</i> <i>matlabroot/toolbox/coder/rtiostream/src/rtiostreamserial/rtiostream_serial.c</i>

For serial communication, the modules *ext_serial_transport.c* and *rtiostream_serial.c* implement the client-side transport functions and the modules *ext_svr_serial_transport.c* and *rtiostream_serial.c* implement the corresponding server-side functions. For TCP/IP communication, the modules *rtiostream_interface.c* and *rtiostream_tcpip.c* implement both client-side and server-side functions. You can edit copies of these files (but do not modify the originals). You can support external mode using your own low-level communications layer by creating similar files using the following templates:

- Client (host) side: *matlabroot/toolbox/coder/rtiostream/src/rtiostreamtcpip/rtiostream_tcpip.c* (TCP/IP) or *matlabroot/toolbox/coder/rtiostream/src/rtiostreamserial/rtiostream_serial.c* (serial)
- Server (target) side: *matlabroot/toolbox/coder/rtiostream/src/rtiostreamtcpip/rtiostream_tcpip.c* (TCP/IP) or *matlabroot/toolbox/coder/rtiostream/src/rtiostreamserial/rtiostream_serial.c* (serial)

The *rtiostream_interface.c* file is an interface between the external mode protocol and an *rtiostream* communications channel. For more details on implementing and testing an *rtiostream* communications channel, see:

- “Communications *rtiostream* API” (Embedded Coder)
- *rtiostreamtest*

Implement your *rtiostream* communications channel by using the documented interface to avoid having to make changes to the file *rtiostream_interface.c* or other external mode related files.

Note Do not modify working source files. Use the templates provided in the */custom* or */rtiostream* folder as starting points, guided by the comments within them.

You need only provide code that implements low-level communications. You need not be concerned with issues such as data conversions between host and target, or with the formatting of messages. The Simulink Coder software handles these functions.

On the client (Simulink engine) side, communications are handled by `ext_comm` (for TCP/IP) and `ext_serial_win32_comm` (for serial) MEX-files.

On the server (target) side, external mode modules are linked into the target executable. This takes place automatically at code generation time if you select **External mode** and is based on the **Transport layer** option that you specify. The modules, called from the main program and the model execution engine, are independent of the generated model code.

The general procedure for implementing your own client-side low-level transport protocol is as follows:

- 1 Edit the template `rtiostream_tcpip.c` to replace low-level communication calls with your own communication calls.
- 2 Generate a MEX-file executable for your custom transport.
- 3 Register your new transport layer with the Simulink software, so that the transport can be selected for a model using the **Interface** pane of the Configuration Parameters dialog box.

For more details, see “Create a Custom Client (Host) Transport Protocol” on page 32-100.

The general procedure for implementing your own server-side low-level transport protocol is as follows:

- 1 Edit the template `rtiostream_tcpip.c` to replace low-level communication calls with your own communication calls. Typically this involves writing or integrating device drivers for your target hardware.
- 2 Modify template makefiles to support the new transport.

For more details, see “Create a Custom Server (Target) Transport Protocol” on page 32-103.

External Mode Communications Overview

This section gives a high-level overview of how a Simulink Coder generated program communicates with Simulink external mode. This description is based on the TCP/IP version of external mode that ships with the Simulink Coder product.

For communication to take place, both the server (target) program and the Simulink software must be executing. This does not mean that the model code in the server system must be executing. The server can be waiting for the Simulink engine to issue a command to start model execution.

The client and server communicate by using bidirectional sockets carrying packets. Packets consist either of *messages* (commands, parameter downloads, and responses) or *data* (signal uploads).

If the target application was invoked with the `-w` command-line option, the program enters a wait state until it receives a message from the host. Otherwise, the program begins execution of the model. While the target application is in a wait state, the Simulink engine can download parameters to the target and configure data uploading.

When the user chooses the **Connect to Target** option from the **Simulation** menu, the host initiates a handshake by sending an `EXT_CONNECT` message. The server responds with information about itself. This information includes

- Checksums. The host uses model checksums to determine that the target code is an exact representation of the current Simulink model.
- Data format information. The host uses this information when formatting data to be downloaded, or interpreting data that has been uploaded.

At this point, host and server are connected. The server is either executing the model or in the wait state. (In the latter case, the user can begin model execution by selecting **Start Real-Time Code** from the **Simulation** menu.)

During model execution, the message server runs as a background task. This task receives and processes messages such as parameter downloads.

Data uploading comprises both foreground execution and background servicing of the signal packets. As the target computes model outputs, it also copies signal values into data upload buffers. This occurs as part of the task associated with each task identifier (`tid`). Therefore, data collection occurs in the foreground. Transmission of the collected data, however, occurs as a background task. The background task sends the data in the collection buffers to the Simulink engine by using data packets.

The host initiates most exchanges as messages. The target usually sends a response confirming that it has received and processed the message. Examples of messages and commands are:

- Connection message / connection response
- Start target simulation / start response
- Parameter download / parameter download response
- Arm trigger for data uploading / arm trigger response
- Terminate target simulation / target shutdown response

Model execution terminates when the model reaches its final time, when the host sends a terminate command, or when a Stop Simulation block terminates execution. On termination, the server informs the host that model execution has stopped, and shuts down its socket. The host also shuts down its socket, and exits external mode.

External Mode Source Files

- “Client (Host) MEX-file Interface Source Files” on page 32-97
- “Server (Target) Source Files” on page 32-99
- “Other Files in the Server Folder” on page 32-100

Client (Host) MEX-file Interface Source Files

The source files for the MEX-file interface component are located in the folder `matlabroot/toolbox/coder/simulinkcoder_core/ext_mode/host` (open), except as noted:

- `common/ext_comm.c`

This file is the core of external mode communication. It acts as a relay station between the target and the Simulink engine. `ext_comm.c` communicates to the Simulink engine by using a shared data structure, `ExternalSim`. It communicates to the target by using calls to the transport layer.

Tasks carried out by `ext_comm.c` include establishment of a connection with the target, downloading of parameters, and termination of the connection with the target.

- `common/rtiostream_interface.c`

This file is an interface between the external mode protocol and an `rtiostream` communications channel. For more details on implementing an `rtiostream` communications channel, see “Communications `rtiostream` API” (Embedded Coder). Implement your `rtiostream` communications channel using the documented interface to avoid having to change the file `rtiostream_interface.c` or other external mode related files.

- `matlabroot/toolbox/coder/rtiostream/src/rtiostreamtcpip/rtiostream_tcpip.c`

This file implements required TCP/IP transport layer functions. The version of `rtiostream_tcpip.c` shipped with the Simulink Coder software uses TCP/IP functions including `recv()`, `send()`, and `socket()`.

- `matlabroot/toolbox/coder/rtiostream/src/rtiostreamserial/rtiostream_serial.c`

This file implements required serial transport layer functions. The version of `rtiostream_serial.c` shipped with the Simulink Coder software uses serial functions including `ReadFile()`, `WriteFile()`, and `CreateFile()`.

- `serial/ext_serial_transport.c`

This file implements required serial transport layer functions. `ext_serial_transport.c` includes `ext_serial_utils.c`, which is located in `matlabroot/rtw/c/src/ext_mode/serial` (open) and contains functions common to client and server sides.

- `common/ext_main.c`

This file is a MEX-file wrapper for external mode. `ext_main.c` interfaces to the Simulink engine by using the standard `mexFunction` call. (See the `mexFunction` reference page and “Integrate MATLAB with External Programming Languages and Systems” for more information.) `ext_main.c` contains a function dispatcher, `esGetAction`, that sends requests from the Simulink engine to `ext_comm.c`.

- `common/ext_convert.c` and `ext_convert.h`

This file contains functions used for converting data from host to target formats (and vice versa). Functions include byte-swapping (big to little- endian), conversion from non-IEEE floats to IEEE doubles, and other conversions. These functions are called both by `ext_comm.c` and directly by the Simulink engine (by using function pointers).

Note You do not need to customize `ext_convert` to implement a custom transport layer. However, you might want to customize `ext_convert` for the intended target. For example, if the target represents the `float` data type in Texas Instruments format, `ext_convert` must be modified to perform a Texas Instruments to IEEE conversion.

- `common/extsim.h`

This file defines the `ExternalSim` data structure and access macros. This structure is used for communication between the Simulink engine and `ext_comm.c`.

- `common/extutil.h`

This file contains only conditionals for compilation of the `assert` macro.

- `common/ext_transport.h`

This file defines functions that must be implemented by the transport layer.

Server (Target) Source Files

These files are linked into the `model.exe` executable. They are located within `matlabroot/rtw/c/src/ext_mode` (open) except as noted.

- `common/ext_svr.c`

`ext_svr.c` is analogous to `ext_comm.c` on the host, but generally is responsible for more tasks. It acts as a relay station between the host and the generated code. Like `ext_comm.c`, `ext_svr.c` carries out tasks such as establishing and terminating connection with the host. `ext_svr.c` also contains the background task functions that either write downloaded parameters to the target model, or extract data from the target data buffers and send it back to the host.

- `common/rtiostream_interface.c`

This file is an interface between the external mode protocol and an `rtiostream` communications channel. For more details on implementing an `rtiostream` communications channel, see “Communications `rtiostream` API” (Embedded Coder). Implement your `rtiostream` communications channel by using the documented interface to avoid having to change the file `rtiostream_interface.c` or other external mode related files.

- `matlabroot/toolbox/coder/rtiostream/src/rtiostreamtcpip/rtiostream_tcpip.c`

This file implements required TCP/IP transport layer functions. The version of `rtiostream_tcpip.c` shipped with the Simulink Coder software uses TCP/IP functions including `recv()`, `send()`, and `socket()`.

- `matlabroot/toolbox/coder/rtiostream/src/rtiostreamserial/rtiostream_serial.c`

This file implements required serial transport layer functions. The version of `rtiostream_serial.c` shipped with the software uses serial functions including `ReadFile()`, `WriteFile()`, and `CreateFile()`.

- `matlabroot/toolbox/coder/rtiostream/src/rtiostream.h`

This file defines the `rtIOStream*` functions implemented in `rtiostream_tcpip.c`.

- `serial/ext_svr_serial_transport.c`

This file implements required serial transport layer functions. `ext_svr_serial_transport.c` includes `serial/ext_serial_utils.c`, which contains functions common to client and server sides.

- `common/updown.c`

`updown.c` handles the details of interacting with the target model. During parameter downloads, `updown.c` does the work of installing the new parameters into the model's parameter vector. For data uploading, `updown.c` contains the functions that extract data from the model's `blockio` vector and write the data to the upload buffers. `updown.c` provides services both to `ext_svr.c` and to the model code (for example, `grt_main.c`). It contains code that is called by using the

background tasks of `ext_svr.c` as well as code that is called as part of the higher priority model execution.

- `matlabroot/rtw/c/src/dt_info.h` (included by generated model build file `model.h`)

These files contain data type transition information that allows access to multi-data type structures across different computer architectures. This information is used in data conversions between host and target formats.

- `common/updown_util.h`

This file contains only conditionals for compilation of the `assert` macro.

- `common/ext_svr_transport.h`

This file defines the Ext* functions that must be implemented by the server (target) transport layer.

Other Files in the Server Folder

- `common/ext_share.h`

Contains message code definitions and other definitions required by both the host and target modules.

- `serial/ext_serial_utils.c`

Contains functions and data structures for communication, MEX link, and generated code required by both the host and target modules of the transport layer for serial protocols.

- The serial transport implementation includes the additional files
 - `serial/ext_serial_pkt.c` and `ext_serial_pkt.h`
 - `serial/ext_serial_port.h`

Implement a Custom Transport Layer

- “Requirements for Custom Transport Layers” on page 32-100
- “Create a Custom Client (Host) Transport Protocol” on page 32-100
- “MATLAB Commands to Rebuild `ext_comm` and `ext_serial_win32` MEX-Files” on page 32-101
- “Register a Custom Client (Host) Transport Protocol” on page 32-102
- “Create a Custom Server (Target) Transport Protocol” on page 32-103
- “Serial Receive Buffer Smaller than 64 Bytes” on page 32-104

Requirements for Custom Transport Layers

- By default, `ext_svr.c` and `updown.c` use `malloc` to allocate buffers in target memory for messages, data collection, and other purposes, although there is also an option to preallocate static memory. If your target uses another memory allocation scheme, you must modify these modules.
- The target is assumed to support both `int32_T` and `uint32_T` data types.

Create a Custom Client (Host) Transport Protocol

To implement the client (host) side of your low-level transport protocol,

- 1 Edit the template file `matlabroot/toolbox/coder/rtiostream/src/rtiostreamtcpip/rtiostream_tcpip.c` to replace low-level communication calls with your own communication calls.
 - a Copy and rename the file to `rtiostream_name.c` (replacing *name* with a name meaningful to you).
 - b Replace the functions `rtIOStreamOpen`, `rtIOStreamClose`, `rtIOStreamSend`, and `rtIOStreamRecv` with functions (of the same name) that call your low-level communication primitives. These functions are called from other external mode modules via `rtiostream_interface.c`. For more information, see “Communications rtiostream API” (Embedded Coder).
 - c Build your `rtiostream` implementation into a shared library that exports the `rtIOStreamOpen`, `rtIOStreamClose`, `rtIOStreamRecv` and `rtIOStreamSend` functions.
- 2 Build the customized MEX-file executable using the MATLAB `mex` function. See “MATLAB Commands to Rebuild `ext_comm` and `ext_serial_win32` MEX-Files” on page 32-101 for examples of `mex` invocations.

Do not replace the existing `ext_comm` MEX-file if you want to preserve its functionality. Instead, use the `-output` option to name the new executable file, for example, `my_ext_comm`. For more information, see `mex`.

- 3 Register your new client transport layer with the Simulink software, so that the transport can be selected for a model using the **Interface** pane of the Configuration Parameters dialog box. For details, see “Register a Custom Client (Host) Transport Protocol” on page 32-102.

Sample commands for rebuilding external mode MEX-files are listed in “MATLAB Commands to Rebuild `ext_comm` and `ext_serial_win32` MEX-Files” on page 32-101.

MATLAB Commands to Rebuild `ext_comm` and `ext_serial_win32` MEX-Files

The following table lists the commands for building the standard `ext_comm` and `ext_serial_win32` modules on PC and UNIX platforms.

Platform	Commands
Windows, TCP/IP	<pre>cd (matlabroot) mex toolbox/coder/simulinkcoder_core/ext_mode/host/common/ext_comm.c ... toolbox/coder/simulinkcoder_core/ext_mode/host/common/ext_convert.c ... toolbox/coder/simulinkcoder_core/ext_mode/host/common/rtiostream_interface.c ... toolbox/coder/simulinkcoder_core/ext_mode/host/common/ext_util.c ... -R2018a ... -Ittoolbox/coder/rtiostream/src ... -Ittoolbox/coder/rtiostream/src/utils_host ... -Ittoolbox/coder/simulinkcoder_core/ext_mode/host/common/include ... -Irtw/c/src/ext_mode/common ... -lmwrtiostreamutils -lmwsl_services ... -DEXTMODE_TCPIP_TRANSPORT ... -DSL_EXT_DLL -output my_ext_comm</pre> <p>Note The <code>rtiostream_interface.c</code> function defines <code>RTIOSTREAM_SHARED_LIB</code> as <code>libmwrtiostreamtcpip</code> and dynamically loads the MathWorks TCP/IP <code>rtiostream</code> shared library. Modify this file if you need to load a different <code>rtiostream</code> shared library.</p>

Platform	Commands
Linux, TCP/IP	Use the Windows commands, with these changes: <ul style="list-style-type: none"> • Change -DSL_EXT_DLL to -DSL_EXT_S0. • Replace back slashes with forward slashes.
Mac, TCP/IP	Use the Windows commands, with these changes: <ul style="list-style-type: none"> • Change -DSL_EXT_DLL to -DSL_EXT_DYLIB. • Replace back slashes with forward slashes.
Windows, serial	<pre>cd (matlabroot) mex toolbox\coder\simulinkcoder_core\ext_mode\host\common\ext_comm.c ... toolbox\coder\simulinkcoder_core\ext_mode\host\common\ext_convert.c ... toolbox\coder\simulinkcoder_core\ext_mode\host\serial\ext_serial_transport.c ... toolbox\coder\simulinkcoder_core\ext_mode\host\serial\ext_serial_pkt.c ... toolbox\coder\simulinkcoder_core\ext_mode\host\serial\rtiostream_serial_interface.c ... toolbox\coder\simulinkcoder_core\ext_mode\host\common\ext_util.c ... -R2018a ... -Itoolbox\coder\rtiostream\src ... -Itoolbox\coder\rtiostream\src\utils_host ... -Itoolbox\coder\simulinkcoder_core\ext_mode\host\common ... -Itoolbox\coder\simulinkcoder_core\ext_mode\host\common\include ... -Irtw\c\src\ext_mode\common ... -Irtw\c\src\ext_mode\serial ... -lmwrtiostreamutils -lmwsl_services ... -DEXTMODE_SERIAL_TRANSPORT -DSL_EXT_DLL ... -output my_ext_serial_comm</pre> <p>Note The <code>rtiostream_interface.c</code> function defines <code>RTIOSTREAM_SHARED_LIB</code> as <code>libmwrtiostreamserial</code> and dynamically loads the MathWorks serial <code>rtiostream</code> shared library. Modify this file if you need to load a different <code>rtiostream</code> shared library.</p>
Linux, serial	Use the Windows commands, with these changes: <ul style="list-style-type: none"> • Change -DSL_EXT_DLL to -DSL_EXT_S0. • Replace back slashes with forward slashes.
Mac, serial	Use the Windows commands, with these changes: <ul style="list-style-type: none"> • Change -DSL_EXT_DLL to -DSL_EXT_DYLIB. • Replace back slashes with forward slashes.

Note `mex` requires a compiler supported by the MATLAB API. See the `mex` reference page and “Integrate MATLAB with External Programming Languages and Systems” for more information about the `mex` function.

Register a Custom Client (Host) Transport Protocol

To register a custom client transport protocol with the Simulink software, you must add an entry of the following form to an `sl_customization.m` file on the MATLAB path:

```
function sl_customization(cm)
    cm.ExtModeTransports.add('stf.tlc', 'transport', 'mexfile', 'Level1');
% -- end of sl_customization
```

where

- *stf.tlc* is the name of the system target file for which the transport will be registered (for example, 'grt.tlc')
- *transport* is the transport name to display in the **Transport layer** menu on the **Interface** pane of the Configuration Parameters dialog box (for example, 'mytcpip')
- *mexfile* is the name of the transport's associated external interface MEX-file (for example, 'ext_mytcpip_comm')

You can specify multiple targets and/or transports with additional `cm.ExtModeTransports.add` lines, for example:

```
function sl_customization(cm)
    cm.ExtModeTransports.add('grt.tlc', 'mytcpip', 'ext_mytcpip_comm', 'Level1');
    cm.ExtModeTransports.add('ert.tlc', 'mytcpip', 'ext_mytcpip_comm', 'Level1');
% -- end of sl_customization
```

If you place the `sl_customization.m` file containing the transport registration information on the MATLAB path, your custom client transport protocol will be registered with each subsequent Simulink session. The name of the transport will appear in the **Transport layer** menu on the **Interface** pane of the Configuration Parameters dialog box. When you select the transport for your model, the name of the associated external interface MEX-file will appear in the noneditable **MEX-file name** field, as shown in the following figure.

External mode

External mode configuration

Transport layer: MEX-file name: ext_mytcpip_comm

MEX-file arguments:

Static memory allocation

Create a Custom Server (Target) Transport Protocol

The `rtIOStream*` function prototypes in `matlabroot/toolbox/coder/rtiostream/src/rtiostream.h` define the calling interface for both the server (target) and client (host) side transport layer functions.

- The TCP/IP implementations are in `matlabroot/toolbox/coder/rtiostream/src/rtiostreamtcpip/rtiostream_tcpip.c`.
- The serial implementations are in `matlabroot/toolbox/coder/rtiostream/src/rtiostreamserial/rtiostream_serial.c`.

Note The `Ext*` function prototypes in `matlabroot/rtw/c/src/ext_mode/common/ext_svr_transport.h` are implemented in `matlabroot/rtw/c/src/ext_mode/common/rtiostream_interface.c` or `matlabroot/rtw/c/src/ext_mode/serial/rtiostream_serial_interface.c`. In most cases you will not need to modify `rtiostream_interface.c` or `rtiostream_serial_interface.c` for your custom TCP/IP or serial transport layer.

To implement the server (target) side of your low-level TCP/IP or serial transport protocol:

- 1 Edit the template `matlabroot/toolbox/coder/rtiostream/src/rtiostreamtcpip/rtiostream_tcpip.c` or `matlabroot/toolbox/coder/rtiostream/src/rtiostreamserial/rtiostream_serial.c` to replace low-level communication calls with your own communication calls.
 - a Copy and rename the file to `rtiostream_name.c` (replacing *name* with a name meaningful to you).
 - b Replace the functions `rtIOStreamOpen`, `rtIOStreamClose`, `rtIOStreamSend`, and `rtIOStreamRecv` with functions (of the same name) that call your low-level communication drivers.

You must implement the functions defined in `rtiostream.h`, and your implementations must conform to the prototypes defined in that file. Refer to the original `rtiostream_tcpip.c` or `rtiostream_serial.c` for guidance.

- 2 Incorporate the external mode source files for your transport layer into the model build process. Use a build process mechanism such as a post code generation command or a `before_make` hook function to make the transport files available to the build process. For more information on the build process mechanisms, see “Customize Post-Code-Generation Build Processing” on page 44-7, “Customize Build Process with `STF_make_rtw_hook` File” on page 44-25, and “Customize Build Process with `sl_customization.m`” on page 44-29.

For example:

- Add the file created in the previous step to the build information:

```
path/rtiostream_name.c
```

- For TCP/IP, add the following file to the build information:

```
matlabroot/rtw/c/src/ext_mode/common/rtiostream_interface.c
```

- For serial, add the following files to the build information:

```
matlabroot/rtw/c/src/ext_mode/serial/ext_serial_pkt.c
matlabroot/rtw/c/src/ext_mode/serial/rtiostream_serial_interface.c
matlabroot/rtw/c/src/ext_mode/serial/ext_svr_serial_transport.c
```

Note For external mode, check that `rtIOStreamRecv` is not a blocking implementation. Otherwise, it might cause the external mode server to block until the host sends data through the `comm` layer.

Serial Receive Buffer Smaller than 64 Bytes

For serial communication, if the serial receive buffer of your target is smaller than 64 bytes:

- 1 Update the following macro with the actual target buffer size:

```
#define TARGET_SERIAL_RECEIVE_BUFFER_SIZE 64
```

Implement the change in the following files:

```
matlabroot/rtw/c/src/ext_mode/serial/ext_serial_utils.c
matlabroot/toolbox/coder/simulinkcoder_core/ext_mode/host/serial/ext_serial_utils.c
```

- 2 Run the command to rebuild the `ext_serial_win32` MEX-file. See “MATLAB Commands to Rebuild `ext_comm` and `ext_serial_win32` MEX-Files” on page 32-101.

See Also

Related Examples

- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49
- “External Mode Simulation with TCP/IP or Serial Communication” on page 32-55
- “Customize Connectivity for TCP/IP or Serial External Mode Simulations” on page 32-40
- “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” on page 32-2
- “Customize XCP Server Software” on page 32-45

Logging in Simulink Coder

Log Program Execution Results

Multiple techniques are available by which a program generated by the Simulink Coder software can save data to a MAT-file for analysis. A generated executable can save system states, outputs, and simulation time at each model execution time step. The data is written to a MAT-file, named (by default) *model.mat*, where *model* is the name of your model. See “Log Data for Analysis” on page 33-2 for a data logging tutorial.

Note Data logging is available only for system target files that have access to a file system. In addition, only the RSim target executables are capable of accessing MATLAB workspace data.

For MAT-file logging limitations, see the configuration parameter MAT-file logging.

Log Data for Analysis

- “Set Up and Configure Model” on page 33-2
- “Data Logging During Simulation” on page 33-3
- “Data Logging from Generated Code” on page 33-5

Set Up and Configure Model

This example shows how data generated by a copy of the model `slexAircraftExample` is logged to the file `myAircraftExample.mat`. Refer to “Build Process Workflow for Real-Time Systems” on page 31-39 for instructions on setting up a copy of `slexAircraftExample` as `myAircraftExample` in a working folder if you have not done so already.

Note When you configure the code generator to produce code that includes support for data logging during execution, the code generator can include text for block names in the block paths included in the log file. If the text includes characters that are unrepresented in the character set encoding for the model, the code generator replaces the characters with XML escape sequences. For example, the code generator replaces the Japanese full-width Katakana letter `ア` with the escape sequence `ア`. For more information, see “Internationalization and Code Generation” on page 22-2.

To configure data logging, open the Configuration Parameters dialog box and select the **Data Import/Export** pane. The process is the same as configuring a Simulink model to save output to the MATLAB workspace. For each workspace return variable you define and enable, the Simulink Coder software defines a parallel MAT-file variable. For example, if you save simulation time to the variable `tout`, your generated program logs the same data to a variable named `rt_tout`. You can change the prefix `rt_` to a suffix (`_rt`), or eliminate it entirely. You do this by setting model configuration parameter **MAT-file variable name modifier**.

To log signals in generated code, use the **Data Import/Export** model configuration parameters described below or include To File or To Workspace blocks in your model.

Note MAT-file logging for code generation does not support signal logging. If you enable MAT-file and signal logging by using **Data Import/Export** model configuration parameters and select signals for logging in the Simulink Editor, when you build the model, the code generator displays this warning:

Warning: MAT-file logging does not support signal logging.
When your model code executes, the signal logging variable 'rt_logout' will not be saved to the MAT-file.

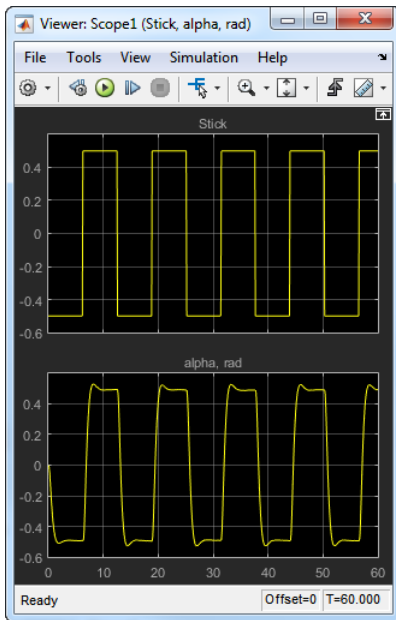
To avoid this warning, clear the model configuration parameter **Signal logging**.

In this example, you modify the `myAircraftExample` model so that the generated program saves the simulation time and system outputs to the file `myAircraftExample.mat`. Then you load the data into the base workspace and plot simulation time against one of the outputs. The `myAircraftExample` model should be configured as described in “Build Process Workflow for Real-Time Systems” on page 31-39.

Data Logging During Simulation

To use the data logging feature:

- 1 Open the `myAircraftExample` model if it is not already open.
- 2 Open the Configuration Parameters dialog box.
- 3 Select the **Data Import/Export** pane. The **Data Import/Export** pane lets you specify which output data is to be saved to the workspace and what variable names to use for it.
- 4 Set parameter **Format** to **Structure with time**. When you select this format, Simulink saves the model states and outputs in structures that have their names specified in the **Save to workspace or file** area. By default, the structures are `xout` for states and `yout` for output. The structure used to save output has two top-level fields: `time` and `signals`. The `time` field contains a vector of simulation times and `signals` contains an array of substructures, each of which corresponds to a model output port.
- 5 Select **Output**. This tells Simulink to save output signal data during simulation as a variable named `yout`. Selecting **Output** enables the code generator to create code that logs the root Output block (`alpha`, `rad`) to a MAT-file.
- 6 Set **Decimation** to 1.
- 7 If other parameters are selected, clear them.
- 8 Click **Apply** and **OK** to register your changes and close the dialog box.
- 9 Save the model.
- 10 In the model window, double-click the scope symbol next to the Aircraft Dynamics Model block, then simulate the model. The resulting scope display is shown below.



- 11 Verify that the simulation time and outputs have been saved to the base workspace in MAT-files. At the MATLAB prompt, type:

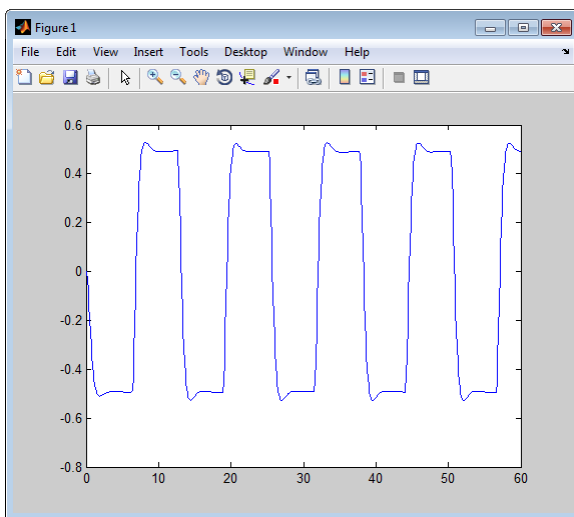
```
whos yout
```

Simulink displays:

Name	Size	Bytes	Class	Attributes
yout	1x1	10756	struct	

- 12 Verify that alpha, rad was logged by plotting simulation time versus that variable. In the Command Window, type:

```
plot(yout.time,yout.signals.values)
```



Data Logging from Generated Code

In the second part of this example, you build and run an executable program built by the code generator for model `myAircraftExample`. The program outputs a MAT-file containing the simulation time and output you previously examined. Even though you have already generated code for the `myAircraftExample` model, you must now regenerate that code because you have changed the model by enabling data logging. The steps below explain this procedure.

To avoid overwriting workspace data with data from simulation runs, the code generator modifies identifiers for variables logged by Simulink. You can control these modifications.

- 1 Set model configuration parameter **MAT-file variable name modifier** to `_rt`. This adds the suffix `_rt` to each variable that you selected to be logged in the first part of this example.
- 2 Click **Apply** and **OK** to register your changes and close the dialog box.
- 3 Save the model.
- 4 Build an executable.
- 5 When the build concludes, run the executable program with the command:

```
!myAircraftExample
```

- 6 The program now produces two message lines, indicating that the MAT-file has been written.

```
** starting the model **
** created myAircraftExample.mat **
```

- 7 Load the MAT-file data created by the executable and look at the workspace variables from simulation and the generated program by typing:

```
load myAircraftExample.mat
whos yout*
```

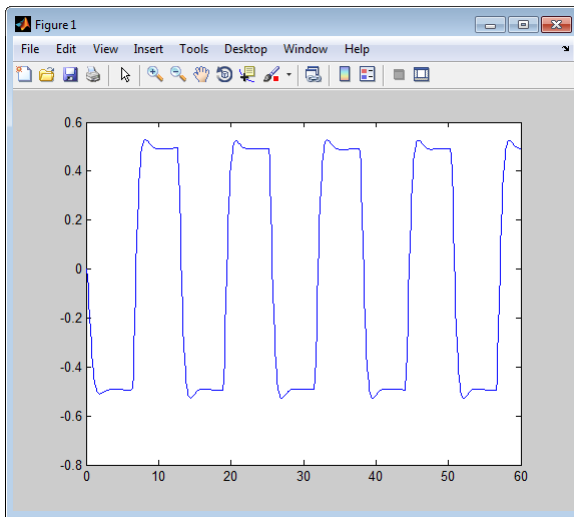
Simulink displays:

Name	Size	Bytes	Class	Attributes
yout	1x1	10756	struct	
yout_rt	1x1	10756	struct	

Note the size and bytes of the structures resulting from the simulation run and generated code are the same.

- 8 Plot the generated code output by entering the following command in the Command Window:

```
plot(yout_rt.time,yout_rt.signals.values)
```



The plot should be identical to the plot that you produced in the previous part of this example.

Tip For UNIX platforms, run the executable program in the Command Window with the syntax `!./executable_name`. If preferred, run the executable program from an OS shell with the syntax `./executable_name`. For more information, see “Run External Commands, Scripts, and Programs”.

Configure State, Time, and Output Logging

The **Data Import/Export** pane enables a generated program to save system states, outputs, and simulation time at each model execution time step. The data is written to a MAT-file, named (by default) `model.mat`.

Before using this data logging feature, you should learn how to configure a Simulink model to return output to the MATLAB workspace. This is discussed in “Save Simulation Data”.

For each workspace return variable that you define and enable, the code generator defines a MAT-file variable. For example, if your model saves simulation time to the workspace variable `tout`, your generated program logs the same data to a variable named (by default) `rt_tout`.

The code generated by the code generator logs the following data:

- Root Output blocks

The default MAT-file variable name for system outputs is `rt_yout`.

The sort order of the `rt_yout` array is based on the port number of the Output block, starting with 1.

- Continuous and discrete states in the model

The default MAT-file variable name for system states is `rt_xout`.

- Simulation time

The default MAT-file variable name for simulation time is `rt_tout`.

- “Override Default MAT-File Variable Names” on page 33-7
- “Override Default MAT-File Name or Buffer Size” on page 33-7

Override Default MAT-File Variable Names

By default, the code generator prefixes the text `rt_` to the variable names for system outputs, states, and simulation time to form MAT-file variable names. To change this prefix for a model, set model configuration parameter **MAT-file variable name modifier** to `(rt_)`, a suffix `(_rt)`, or no modifier (none). Other system target files might not support this parameter.

Override Default MAT-File Name or Buffer Size

You can specify compiler options to override the following MAT-file attributes in generated code:

MAT-File Attribute	Default	Compiler Option
Name	<code>model.mat</code>	<code>-DSAVEFILE=<i>filename</i></code>
Size of data logging buffer	1024 bytes	<code>-DDEFAULT_BUFFER_SIZE=<i>n</i></code>

Note Valid option syntax can vary among compilers. For example, Microsoft Visual C++ compilers typically accept `/DSAVEFILE=filename` as well as `-DSAVEFILE=filename`.

For a template makefile (TMF) based target, set model configuration parameter **Make command** to the compiler option. For example:

```
Make command: make_rtw OPTS="-DSAVEFILE=myCodeLog.mat"
```

For a toolchain-based system target file such as GRT or ERT, add the compiler option to model configuration. Set parameter **Build configuration** to `Specify`, and add the compiler option to the **C Compiler** row of the **Tool/Options** table. For example:

Tool	Options
C Compiler	<code>\$(cflags) \$(CVARSFLAG) \$(CFLAGS_ADDITIONAL) -DSAVEFILE=myCodeLog.mat /Od /Oy-</code>

To add the compiler option to a custom toolchain, you can modify and reregister the custom toolchain using the procedures shown in the example “Add Custom Toolchains to MATLAB® Coder™ Build Process”. For example, to add the compiler option to the MATLAB source file for the custom toolchain, you could define `myCompilerOpts` as follows:

```
optimsOff0pts = {'/c /Od'};
optimsOn0pts  = {'/c /O2'};
cCompilerOpts = '$(cflags) $(CVARSFLAG) $(CFLAGS_ADDITIONAL)';
cppCompilerOpts = '$(cflags) $(CVARSFLAG) $(CPPFLAGS_ADDITIONAL)';
myCompilerOpts = {' -DSAVEFILE=myCodeLog.mat '};
...
```

Then you can add `myCompilerOpts` to the flags for each configuration and compiler to which it applies, for example:

```
cfg = tc.getBuildConfiguration('Faster Builds');
cfg.setOption('C Compiler', horzcat(cCompilerOpts, myCompilerOpts, optimsOff0pts));
```

As shown in “Add Custom Toolchains to MATLAB® Coder™ Build Process”, after modifying the custom toolchain, you save the configuration to a MAT-file and refresh the target registry.

Log Data with Scope and To Workspace Blocks

The code generated by the code generator also logs data from these sources:

- Scope blocks that have the block parameter **Log data to workspace** enabled

You must specify the variable name and data format in each Scope block's dialog box.

- To Workspace blocks in the model

You must specify the variable name and data format in each To Workspace block's dialog box.

The variables are written to `model.mat`, along with variables logged from the **Workspace I/O** pane.

Log Data with To File Blocks

You can also log data to a To File block. The generated program creates a separate MAT-file (distinct from `model.mat`) for each To File block in the model. The file contains the block time and input data. You must specify the file name, variable name, decimation, and sample time in the To File block dialog box.

Note Models referenced by Model blocks do not perform data logging in that context except for states, which you can include in the state logged for top models. Code generated by the Simulink Coder software for referenced models does not perform data logging to MAT-files.

Data Logging Differences Between Single- and Multitasking

When logging data in single-tasking and multitasking systems, you will notice differences in the logging of

- Noncontinuous root Output blocks
- Discrete states

In multitasking mode, the logging of states and outputs is done after the first task execution (and not at the end of the first time step). In single-tasking mode, the code generated by the build procedure logs states and outputs after the first time step.

See *Data Logging in Single-Tasking and Multitasking Model Execution* on page 17-20 for more details on the differences between single-tasking and multitasking data logging.

Note The rapid simulation target (RSim) provides enhanced logging options. See “Accelerate, Refine, and Test Hybrid Dynamic System on Host Computer by Using RSim System Target File” on page 29-2 for more information.

See Also

Related Examples

- “How Generated Code Exchanges Data with an Environment” on page 15-35

- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Data Interchange Using the C API in Simulink Coder

- “Exchange Data Between Generated and External Code Using C API” on page 34-2
- “Use C API to Access Model Signals and States” on page 34-19
- “Use C API to Access Model Parameters” on page 34-24

Exchange Data Between Generated and External Code Using C API

Some Simulink Coder applications must interact with signals, states, root-level inputs/outputs, or parameters in the generated code for a model. For example, calibration applications monitor and modify parameters. Signal monitoring or data logging applications interface with signal, state, and root-level input/output data. Using the Simulink Coder C API, you can build target applications that log signals, states, and root-level inputs/outputs, monitor signals, states, and root-level inputs/outputs, and tune parameters, while the generated code executes.

The C API minimizes its memory footprint by sharing information common to signals, states, root-level inputs/outputs, and parameters in smaller structures. Signal, state, root-level input/output, and parameter structures include an index into the structure map, allowing multiple signals, states, root-level inputs/outputs, or parameters to share data.

To get started with an example, see “Use C API to Access Model Signals and States” on page 34-19 or “Use C API to Access Model Parameters” on page 34-24.

In this section...

“Generated C API Files” on page 34-2

“Generate C API Files” on page 34-4

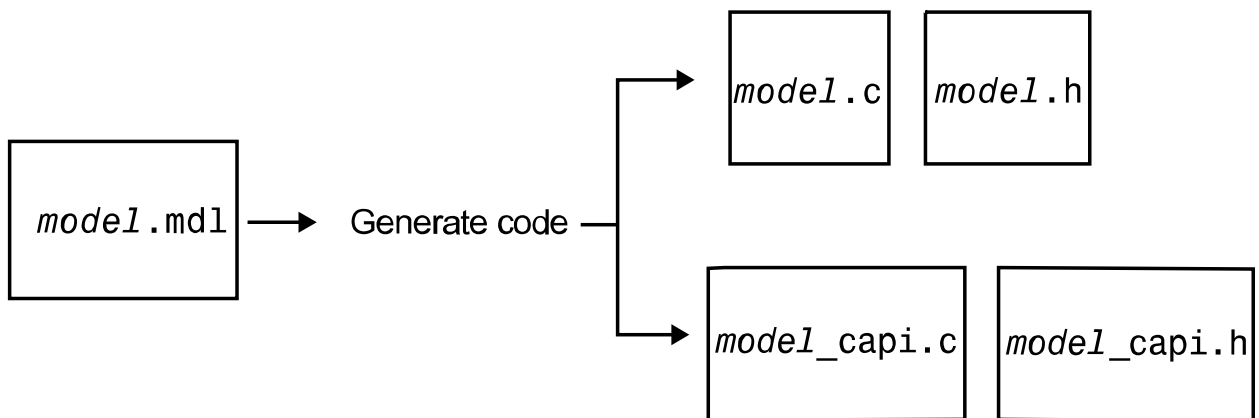
“Description of C API Files” on page 34-5

“Generate C API Data Definition File for Exchanging Data with a Target System” on page 34-16

“C API Limitations” on page 34-17

Generated C API Files

When you configure a model to use the C API, the Simulink Coder code generator generates two additional files, *model_capi.c* (or *.cpp*) and *model_capi.h*, where *model* is the name of the model. The code generator places the two C API files in the build folder, based on settings in the Configuration Parameters dialog box. The C API source code file contains information about global block output signals, states, root-level inputs/outputs, and global parameters defined in the generated code model source code. The C API header file is an interface header file between the model source code and the generated C API. You can use the information in these C API files to create your application. Among the files generated are those shown in the next figure.



Note When you configure the code generator to produce code that includes support for the C API interface and data logging, the code generator can include text for block names in the block paths logged to C API files *model_capi.c* (or *.cpp*) and *model_capi.h*. If the text includes characters that are unrepresented in the character set encoding for the model, the code generator replaces the characters with XML escape sequences. For example, the code generator replaces the Japanese full-width Katakana letter ア with the escape sequence `ア`; . For more information, see “Internationalization and Code Generation” on page 22-2.

Generate C API Files

To generate C API files for your model:

- 1 Select the C API interface for your model. There are two ways to select the C API interface for your model, as described in the following sections.
 - “Select C API with Configuration Parameters Dialog” on page 34-4
 - “Select C API from the Command Line” on page 34-4
- 2 Generate code for your model.

After generating code, you can examine the files *model_capi.c* (or *.cpp*) and *model_capi.h* in the model build folder.

Select C API with Configuration Parameters Dialog

- 1 Open your model, and open the Configuration Parameters dialog box.
- 2 In the **Code Generation > Interface** pane, in the **Data exchange interface** subgroup, select one or more C API options. Based on the options you select, support for accessing signals, parameters, states, and root-level I/O will appear in the C API generated code.
 - If you want to generate C API code for global block output signals, select **Generate C API for: signals**.
 - If you want to generate C API code for global block parameters, select **Generate C API for: parameters**.
 - If you want to generate C API code for discrete and continuous states, select **Generate C API for: states**.
 - If you want to generate C API code for root-level inputs and outputs, select **Generate C API for: root-level I/O**.

Select C API from the Command Line

From the MATLAB command line, you can use the `set_param` function to select or clear C API model configuration parameters. At the MATLAB command line, enter one or more of the following commands, where `modelName` is the name of your model.

To select **Generate C API for: signals**, enter:

```
set_param('modelName', 'RTWCAPISignals', 'on')
```

To clear **Generate C API for: signals**, enter:

```
set_param('modelName', 'RTWCAPISignals', 'off')
```

To select **Generate C API for: parameters**, enter:

```
set_param('modelName', 'RTWCAPIParams', 'on')
```

To clear **Generate C API for: parameters**, enter:

```
set_param('modelName', 'RTWCAPIParams', 'off')
```

To select **Generate C API for: states**, enter:

```
set_param('modelName', 'RTWCAPIStates', 'on')
```

To clear **Generate C API for: states**, enter:

```
set_param('modelName', 'RTWCAPIStates', 'off')
```

To select **Generate C API for: root-level I/O**, enter:

```
set_param('modelName', 'RTWCAPIRootIO', 'on')
```

To clear **Generate C API for: root-level I/O**, enter:

```
set_param('modelName', 'RTWCAPIRootIO', 'off')
```

Description of C API Files

- “About C API Files” on page 34-5
- “Structure Arrays Generated in C API Files” on page 34-7
- “Generate Example C API Files” on page 34-9
- “C API Signals” on page 34-9
- “C API States” on page 34-12
- “C API Root-Level Inputs and Outputs” on page 34-12
- “C API Parameters” on page 34-14
- “Map C API Data Structures to rtModel” on page 34-15

About C API Files

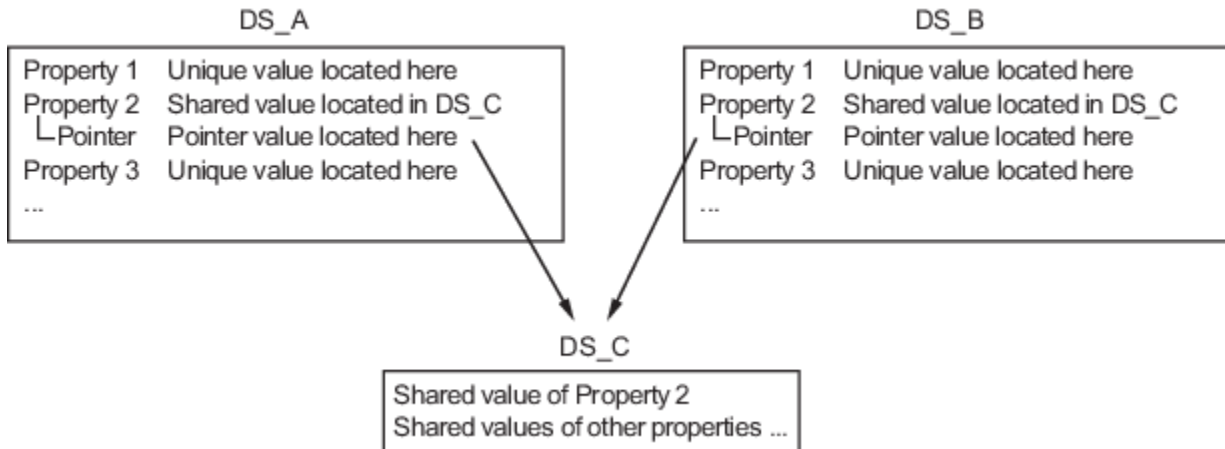
The `model_capi.c` (or `.cpp`) file provides external applications with a consistent interface to model data. Depending on your configuration settings, the data could be a signal, state, root-level input or output, or parameter. In this document, the term data item refers to either a signal, a state, a root-level input or output, or a parameter. The C API uses structures that provide an interface to the data item properties. The interface packages the properties of each data item in a data structure. If the model contains multiple data items, the interface generates an array of data structures. The members of a data structure map to data properties.

To interface with data items, an application requires the following properties for each data item:

- Name
- Block path
- Port number (for signals and root-level inputs/outputs only)
- Address
- Data type information: native data type, data size, complexity, and other attributes
- Dimensions information: number of rows, number of columns, and data orientation (scalar, vector, matrix, or n -dimensional)

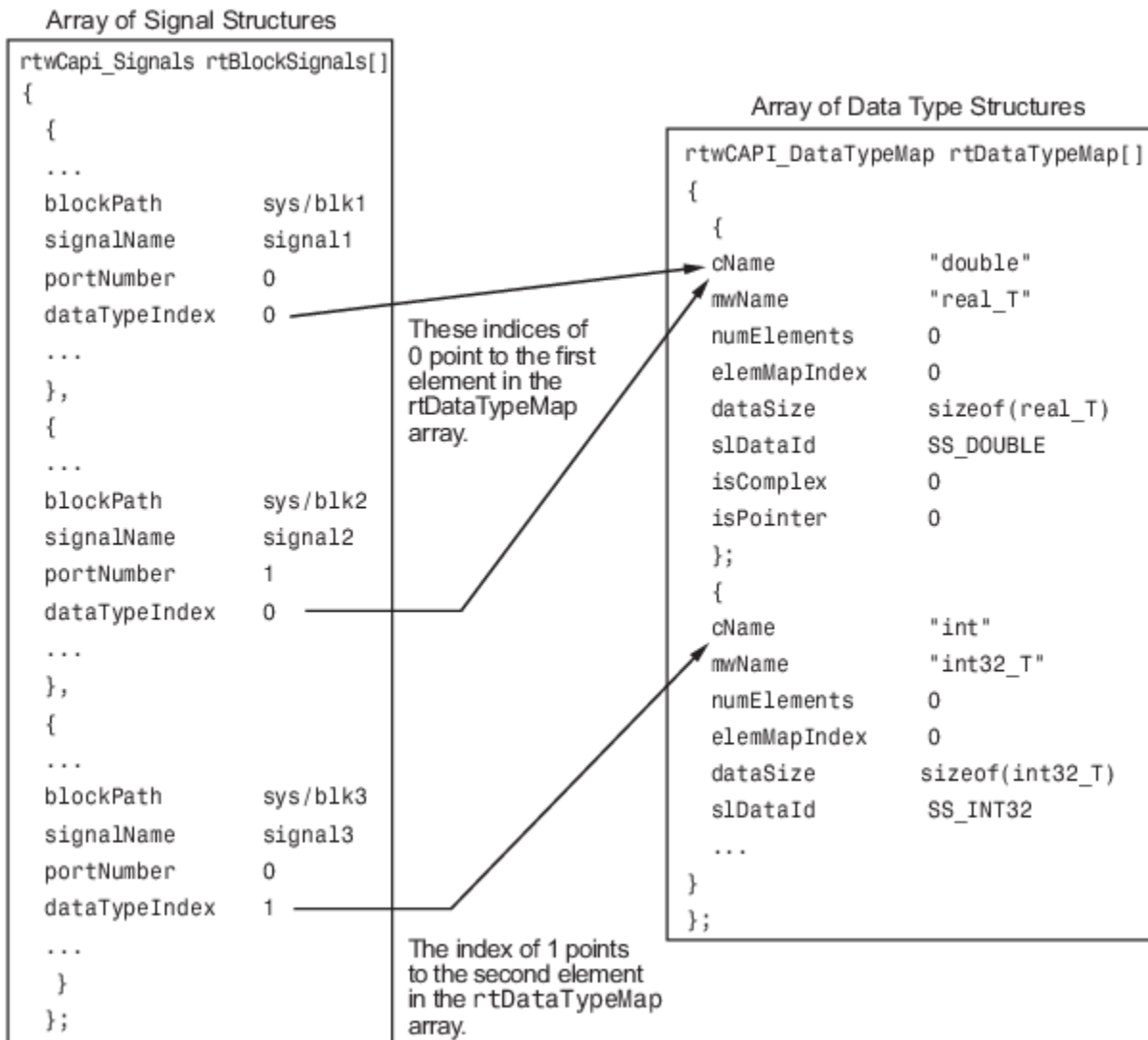
- Fixed-point information: slope, bias, scale type, word length, exponent, and other attributes
- Sample-time information (for signals, states, and root-level inputs/outputs only): sample time, task identifier, frames

As illustrated in the next figure, the properties of data item A, for example, are located in data structure DS_A. The properties of data item B are located in data structure DS_B.



Some property *values* can be unique to each data item, and there are some property values that several data items can share in common. Name, for example, has a unique value for each data item. The interface places the unique property values directly in the structure for the data item. The name value of data item A is in DS_A, and the name value of data item B is in DS_B.

But data type could be a property whose value several data items have in common. The ability of some data items to share a property allows the C API to have a reuse feature. In this case, the interface places only an index value in DS_A and an index value in DS_B. These indices point to a different data structure, DS_C, that contains the actual data type value. The next figure shows this scheme with more detail.



The figure shows three signals. `signal1` and `signal2` share the same data type, `double`. Instead of specifying this data type value in each signal data structure, the interface provides only an index value, 0, in the structure. "double" is described by entry 0 in the `rtDataTypeMap` array, which is referenced by both signals. Additionally, property values can be shared between signals, states, root-level inputs/outputs, and parameters, so states, root-level inputs/outputs, and parameters also might reference the `double` entry in the `rtDataTypeMap` array. This reuse of information reduces the memory size of the generated interface.

Structure Arrays Generated in C API Files

As with data type, the interface maps other common properties (such as address, dimension, fixed-point scaling, and sample time) into separate structures and provides an index in the structure for the data item. For a complete list of structure definitions, refer to the file `matlabroot/rtw/c/src/rtw_capi.h`. This file also describes each member in a structure. The structure arrays generated in the `model_capi.c` (or `.cpp`) file are of structure types defined in the `rtw_capi.h` file. Here is a brief description of the structure arrays generated in `model_capi.c` (or `.cpp`):

- **rtBlockSignals** is an array of structures that contains information about global block output signals in the model. Each element in the array is of type `struct rtwCAPI_Signals`. The members of this structure provide the signal name, block path, block port number, address, and indices to the data type, dimension, fixed-point, and sample-time structure arrays.
- **rtBlockParameters** is an array of structures that contains information about the tunable block parameters in the model by block name and parameter name. Each element in the array is of type `struct rtwCAPI_BlockParameters`. The members of this structure provide the parameter name, block path, address, and indices to data type, dimension, and fixed-point structure arrays.
- **rtBlockStates** is an array of structures that contains information about discrete and continuous states in the model. Each element in the array is of type `struct rtwCAPI_States`. The members of this structure provide the state name, block path, type (continuous or discrete), and indices to the address, data type, dimension, fixed-point, and sample-time structure arrays.
- **rtRootInputs** is an array of structures that contains information about root-level inputs in the model. Each element in the array is of type `struct rtwCAPI_Signals`. The members of this structure provide the root-level input name, block path, block port number, address, and indices to the data type, dimension, fixed-point, and sample-time structure arrays.
- **rtRootOutputs** is an array of structures that contains information about root-level outputs in the model. Each element in the array is of type `struct rtwCAPI_Signals`. The members of this structure provide the root-level output name, block path, block port number, address, and indices to the data type, dimension, fixed-point, and sample-time structure arrays.
- **rtModelParameters** is an array of structures that contains information about workplace variables that one or more blocks or Stateflow charts in the model reference as block parameters. Each element in the array is of data type `rtwCAPI_ModelParameters`. The members of this structure provide the variable name, address, and indices to data type, dimension, and fixed-point structure arrays.
- **rtDataAddrMap** is an array of base addresses of signals, states, root-level inputs/outputs, and parameters that appear in the `rtBlockSignals`, `rtBlockParameters`, `rtBlockStates`, and `rtModelParameters` arrays. Each element of the `rtDataAddrMap` array is a pointer to `void` (`void*`).
- **rtDataTypeMap** is an array of structures that contains information about the various data types in the model. Each element of this array is of type `struct rtwCAPI_DataTypeMap`. The members of this structure provide the data type name, size of the data type, and information on whether or not the data is complex.
- **rtDimensionMap** is an array of structures that contains information about the various data dimensions in the model. Each element of this array is of type `struct rtwCAPI_DimensionMap`. The members of this structure provide information on the number of dimensions in the data, the orientation of the data (whether it is scalar, vector, or a matrix), and the actual dimensions of the data.
- **rtFixPtMap** is an array of structures that contains fixed-point information about the signals, states, root-level inputs/outputs, and parameters. Each element of this array is of type `struct rtwCAPI_FixPtMap`. The members of this structure provide information about the data scaling, bias, exponent, and whether or not the fixed-point data is signed. If the model does not have fixed-point data (signal, state, root-level input/output, or parameter), the Simulink Coder software assigns NULL or zero values to the elements of the `rtFixPtMap` array.
- **rtSampleTimeMap** is an array of structures that contains sampling information about the global signals, states, and root-level inputs/outputs in the model. (This array does not contain information about parameters.) Each element of this array is of type `struct rtwCAPI_SampleTimeMap`. The members of this structure provide information about the sample period, offset, and whether or not the data is frame-based or sample-based.

Generate Example C API Files

Subtopics “C API Signals” on page 34-9, “C API States” on page 34-12, “C API Root-Level Inputs and Outputs” on page 34-12, and “C API Parameters” on page 34-14 discuss generated C API structures using the example model `rtwdemo_capi`. To generate code from the example model, do the following:

- 1 Open the model by clicking the `rtwdemo_capi` link above or by typing `rtwdemo_capi` on the MATLAB command line.
- 2 If you want to generate C API structures for root-level inputs/outputs in `rtwdemo_capi`, select model configuration parameter **Generate C API for: root-level I/O**.

The setting this parameter must match between the top model and the referenced model. If you modify the parameter setting, save the top model and the referenced model to the same writable work folder.

- 3 Generate code for the model.

The C API code examples in the next subtopics are generated with C as the target language.

This model has three global block output signals that will appear in C API generated code:

- `top_sig1`, which is a test point at the output of the Gain1 block in the top model
- `sig2_eg`, which appears in the top model and is defined in the base workspace as a `Simulink.Signal` object having storage class `ExportedGlobal`
- `bot_sig1`, which appears in the referenced model `rtwdemo_capi_bot` and is defined as a `Simulink.Signal` object having storage class `Model default`

The model also has two discrete states that will appear in the C API generated code:

- `top_state`, which is defined for the Delay1 block in the top model
- `bot_state`, which is defined for the Discrete Filter block in the referenced model

The model has root-level inputs/outputs that will appear in the C API generated code if you select model configuration parameter **Generate C API for: root-level I/O**:

- Four root-level inputs, `In1` through `In4`
- Six root-level outputs, `Out1` through `Out6`

Additionally, the model has five global block parameters that will appear in C API generated code:

- `Kp` (top model Gain1 block and referenced model Gain2 block share)
- `Ki` (referenced model Gain3 block)
- `p1` (lookup table `lu1d`)
- `p2` (lookup table `lu2d`)
- `p3` (lookup table `lu3d`)

C API Signals

The `rtwC_API_Signals` structure captures signal information including the signal name, address, block path, output port number, data type information, dimensions information, fixed-point information, and sample-time information.

Here is the section of code in `rtwdemo_capi_capi.c` that provides information on C API signals for the top model in `rtwdemo_capi`:

```
/* Block output signal information */
static const rtwC_API_Signals rtBlockSignals[] = {
  /* addrMapIndex, sysNum, blockPath,
   * signalName, portNumber, dataTypeIndex, dimIndex, fxpIndex, sTimeIndex
   */
  { 0, 0, "rtwdemo_capi/Gain1",
    "top_sig1", 0, 0, 0, 0, 0 },

  { 1, 0, "rtwdemo_capi/lu2d",
    "sig2_eg", 0, 0, 1, 0, 0 },

  {
    0, 0, (NULL), (NULL), 0, 0, 0, 0, 0
  }
};
```

Note To better understand the code, read the comments in the file. For example, notice the comment that begins on the third line in the preceding code. This comment lists the members of the `rtwC_API_Signals` structure, in order. This tells you the order in which the assigned values for each member appear for a signal. In this example, the comment tells you that `signalName` is the fourth member of the structure. The following lines describe the first signal:

```
{ 0, 0, "rtwdemo_capi/Gain1",
  "top_sig1", 0, 0, 0, 0, 0 },
```

From these lines you infer that the name of the first signal is `top_sig1`.

Each array element, except the last, describes one output port for a block signal. The final array element is a sentinel, with all elements set to null values. For example, examine the second signal, described by the following code:

```
{ 1, 0, "rtwdemo_capi/lu2d",
  "sig2_eg", 0, 0, 1, 0, 0 },
```

This signal, named `sig2_eg`, is the output signal of the first port of the block `rtwdemo_capi/lu2d`. (This port is the first port because the zero-based index for `portNumber` displayed on the second line is assigned the value 0.)

The address of this signal is given by `addrMapIndex`, which, in this example, is displayed on the first line as 1. This provides an index into the `rtDataAddrMap` array, found later in `rtwdemo_capi_capi.c`:

```
/* Declare Data Addresses statically */
static void* rtDataAddrMap[] = {
  &rtwdemo_capi_B.top_sig1,          /* 0: Signal */
  &sig2_eg[0],                      /* 1: Signal */
  &rtwdemo_capi_DWork.top_state,    /* 2: Discrete State */
  &rtP_Ki,                          /* 3: Model Parameter */
  &rtP_Kp,                          /* 4: Model Parameter */
  &rtP_p1[0],                       /* 5: Model Parameter */
  &rtP_p2[0],                       /* 6: Model Parameter */
  &rtP_p3[0],                       /* 7: Model Parameter */
};
```

The index of 1 points to the second element in the `rtDataAddrMap` array. From the `rtDataAddrMap` array, you can infer that the address of this signal is `&sig2_eg[0]`.

This level of indirection supports multiple code instances of the same model. For multiple instances, the signal information remains constant, except for the address. In this case, the model is a single instance. Therefore, the `rtDataAddrMap` is declared statically. If you choose to generate reusable code, an initialize function is generated that initializes the addresses dynamically per instance. For details on generating reusable code, see “Configure Generated C Function Interface for Model Entry-Point Functions” on page 16-79 and see “Configure Code Reuse Support” (Embedded Coder).

The `dataTypeIndex` provides an index into the `rtDataTypeMap` array, found later in `rtwdemo_capi_capi.c`, indicating the data type of the signal:

```
/* Data Type Map - use dataTypeMapIndex to access this structure */
static const rtwCAPI_DataTypeMap rtDataTypeMap[] = {
    /* cName, mwName, numElements, elemMapIndex, dataSize, slDataId, *
     * isComplex, isPointer */
    { "double", "real_T", 0, 0, sizeof(real_T), SS_DOUBLE, 0, 0 }
};
```

Because the index is 0 for `sig2_eg`, the index points to the first structure element in the array. You can infer that the data type of the signal is `double`. The value of `isComplex` is 0, indicating that the signal is not complex. Rather than providing the data type information directly in the `rtwCAPI_Signals` structure, a level of indirection is introduced. The indirection allows multiple signals that share the same data type to point to one map structure, saving memory for each signal.

The `dimIndex` (dimensions index) provides an index into the `rtDimensionMap` array, found later in `rtwdemo_capi_capi.c`, indicating the dimensions of the signal. Because this index is 1 for `sig2_eg`, the index points to the second element in the `rtDimensionMap` array:

```
/* Dimension Map - use dimensionMapIndex to access elements of ths structure*/
static const rtwCAPI_DimensionMap rtDimensionMap[] = {
    /* dataOrientation, dimArrayIndex, numDims, vardimsIndex */
    { rtwCAPI_SCALAR, 0, 2, 0 },

    { rtwCAPI_VECTOR, 2, 2, 0 },
    ...
};
```

From this structure, you can infer that this is a nonscalar signal having a dimension of 2. The `dimArrayIndex` value, 2, provides an index into `rtDimensionArray`, found later in `rtwdemo_capi_capi.c`:

```
/* Dimension Array- use dimArrayIndex to access elements of this array */
static const uint_T rtDimensionArray[] = {
    1, /* 0 */
    1, /* 1 */
    2, /* 2 */
    ...
};
```

The `fxpIndex` (fixed-point index) provides an index into the `rtFixPtMap` array, found later in `rtwdemo_capi_capi.c`, indicating fixed-point information about the signal. Your code can use the scaling information to compute the real-world value of the signal, using the equation $V = SQ + B$, where V is “real-world” (that is, base-10) value, S is user-specified slope, Q is “quantized fixed-point value” or “stored integer,” and B is user-specified bias. For details, see “Scaling” (Fixed-Point Designer).

Because this index is 0 for `sig2_eg`, the signal does not have fixed-point information. A fixed-point map index of zero means that the signal does not have fixed-point information.

The `sTimeIndex` (sample-time index) provides the index to the `rtSampleTimeMap` array, found later in `rtwdemo_capi_capi.c`, indicating task information about the signal. If you log multirate signals or conditionally executed signals, the sampling information can be useful.

Note `model_capi.c` (or `.cpp`) includes `rtw_capi.h`. A source file that references the `rtBlockSignals` array also must include `rtw_capi.h`.

C API States

The `rtwCAPI_States` structure captures state information including the state name, address, block path, type (continuous or discrete), data type information, dimensions information, fixed-point information, and sample-time information.

Here is the section of code in `rtwdemo_capi_capi.c` that provides information on C API states for the top model in `rtwdemo_capi`:

```
/* Block states information */
static const rtwCAPI_States rtBlockStates[] = {
  /* addrMapIndex, contStateStartIndex, blockPath,
   * stateName, pathAlias, dWorkIndex, dataTypeIndex, dimIndex,
   * fixPtIdx, sTimeIndex, isContinuous
   */
  { 2, -1, "rtwdemo_capi/Delay1",
    "top_state", "", 0, 0, 0, 0, 0, 0 },

  {
    0, -1, (NULL), (NULL), (NULL), 0, 0, 0, 0, 0, 0
  }
};
```

Each array element, except the last, describes a state in the model. The final array element is a sentinel, with all elements set to null values. In this example, the C API code for the top model displays one state:

```
{ 2, -1, "rtwdemo_capi/Delay1",
  "top_state", "", 0, 0, 0, 0, 0, 0 },
```

This state, named `top_state`, is defined for the block `rtwdemo_capi/Delay1`. The value of `isContinuous` is zero, indicating that the state is discrete rather than continuous. The other fields correspond to the like-named signal equivalents described in “C API Signals” on page 34-9, as follows:

- The address of the signal is given by `addrMapIndex`, which, in this example, is 2. This is an index into the `rtDataAddrMap` array, found later in `rtwdemo_capi_capi.c`. Because the index is zero based, 2 corresponds to the third element in `rtDataAddrMap`, which is `&rtwdemo_capi_DWork.top_state`.
- The `dataTypeIndex` provides an index into the `rtDataTypeMap` array, found later in `rtwdemo_capi_capi.c`, indicating the data type of the parameter. The value 0 corresponds to a double, noncomplex parameter.
- The `dimIndex` (dimensions index) provides an index into the `rtDimensionMap` array, found later in `rtwdemo_capi_capi.c`. The value 0 corresponds to the first entry, which is `{ rtwCAPI_SCALAR, 0, 2, 0 }`.
- The `fixPtIndex` (fixed-point index) provides an index into the `rtFixPtMap` array, found later in `rtwdemo_capi_capi.c`, indicating fixed-point information about the parameter. As with the corresponding signal attribute, a fixed-point map index of zero means that the parameter does not have fixed-point information.

C API Root-Level Inputs and Outputs

The `rtwCAPI_Signals` structure captures root-level input/output information including the input/output name, address, block path, port number, data type information, dimensions information, fixed-

point information, and sample-time information. (This structure also is used for block output signals, as previously described in “C API Signals” on page 34-9.)

Here is the section of code in `rtwdemo_capi_capi.c` that provides information on C API root-level inputs/outputs for the top model in `rtwdemo_capi`:

```

/* Root Inputs information */
static const rtwC_API_Signals rtRootInputs[] = {
  /* addrMapIndex, sysNum, blockPath,
   * signalName, portNumber, dataTypeIndex, dimIndex, fxpIndex, sTimeIndex
   */
  { 3, 0, "rtwdemo_capi/In1",
    "", 1, 0, 0, 0, 0, 0 },

  { 4, 0, "rtwdemo_capi/In2",
    "", 2, 0, 0, 0, 0, 0 },

  { 5, 0, "rtwdemo_capi/In3",
    "", 3, 0, 0, 0, 0, 0 },

  { 6, 0, "rtwdemo_capi/In4",
    "", 4, 0, 0, 0, 0, 0 },

  {
    0, 0, (NULL), (NULL), 0, 0, 0, 0, 0, 0
  }
};

/* Root Outputs information */
static const rtwC_API_Signals rtRootOutputs[] = {
  /* addrMapIndex, sysNum, blockPath,
   * signalName, portNumber, dataTypeIndex, dimIndex, fxpIndex, sTimeIndex
   */
  { 7, 0, "rtwdemo_capi/Out1",
    "", 1, 0, 0, 0, 0, 0 },

  { 8, 0, "rtwdemo_capi/Out2",
    "", 2, 0, 0, 0, 0, 0 },

  { 9, 0, "rtwdemo_capi/Out3",
    "", 3, 0, 0, 0, 0, 0 },

  { 10, 0, "rtwdemo_capi/Out4",
    "", 4, 0, 0, 0, 0, 0 },

  { 11, 0, "rtwdemo_capi/Out5",
    "sig2_eg", 5, 0, 1, 0, 0, 0 },

  { 12, 0, "rtwdemo_capi/Out6",
    "", 6, 0, 1, 0, 0, 0 },

  {
    0, 0, (NULL), (NULL), 0, 0, 0, 0, 0, 0
  }
};

```

Note When you generate C++ code, the code generator does not provide information on C API root-level inputs/outputs.

For information about interpreting the values in the `rtwC_API_Signals` structure, see the previous section “C API Signals” on page 34-9.

C API Parameters

The `rtwCAPI_BlockParameters` and `rtwCAPI_ModelParameters` structures capture parameter information including the parameter name, block path (for block parameters), address, data type information, dimensions information, and fixed-point information.

The `rtModelParameters` array contains entries for workspace variables that are referenced as tunable Simulink block parameters or Stateflow data of machine scope. For example, tunable parameters include `Simulink.Parameter` objects that use a storage class other than `Auto`. The Simulink Coder software assigns its elements only `NULL` or zero values in the absence of such data.

The setting that you select for the model configuration parameter **Default parameter behavior** determines how information is generated into the `rtBlockParameters` array in `model_capi.c` (or `.cpp`).

- If you set **Default parameter behavior** to `Tunable`, the `rtBlockParameters` array contains an entry for every modifiable parameter of every block in the model. However, if you use a MATLAB variable or a tunable parameter to specify a block parameter, the block parameter does not appear in `rtBlockParameters`. Instead, the variable or tunable parameter appears in `rtModelParameters`.
- If you set **Default parameter behavior** to `Inlined`, the `rtBlockParameters` array is empty. The Simulink Coder software assigns its elements only `NULL` or zero values.

The last member of each array is a sentinel, with all elements set to null values.

Here is the `rtBlockParameters` array that is generated by default in `rtwdemo_capi_capi.c`:

```
/* Individual block tuning is not valid when inline parameters is *
 * selected. An empty map is produced to provide a consistent *
 * interface independent of inlining parameters. *
 */
static const rtwCAPI_BlockParameters rtBlockParameters[] = {
    /* addrMapIndex, blockPath,
     * paramName, dataTypeIndex, dimIndex, fixPtIdx
     */
    {
        0, (NULL), (NULL), 0, 0, 0
    }
};
```

In this example, only the final, sentinel array element is generated, with all members of the structure `rtwCAPI_BlockParameters` set to `NULL` and zero values. This is because **Default parameter behavior** is set to `Inlined` by default for the `rtwdemo_capi` example model. If you set **Default parameter behavior** to `Tunable`, the block parameters are generated in the `rtwCAPI_BlockParameters` structure. However, MATLAB variables and tunable parameters appear in the `rtwCAPI_ModelParameters` structure.

Here is the `rtModelParameters` array that is generated by default in `rtwdemo_capi_capi.c`:

```
/* Tunable variable parameters */
static const rtwCAPI_ModelParameters rtModelParameters[] = {
    /* addrMapIndex, varName, dataTypeIndex, dimIndex, fixPtIndex */
    { 2, TARGET_STRING("Ki"), 0, 0, 0 },
    { 3, TARGET_STRING("Kp"), 0, 0, 0 },
    { 4, TARGET_STRING("p1"), 0, 2, 0 },
    { 5, TARGET_STRING("p2"), 0, 3, 0 },
    { 6, TARGET_STRING("p3"), 0, 4, 0 },
};
```

```
{ 0, (NULL), 0, 0, 0 }
};
```

In this example, the `rtModelParameters` array contains entries for each variable that is referenced as a tunable Simulink block parameter.

For example, the `varName` (variable name) of the fourth parameter is `p2`. The other fields correspond to the like-named signal equivalents described in “C API Signals” on page 34-9, as follows:

- The address of the fourth parameter is given by `addrMapIndex`, which, in this example, is 5. This is an index into the `rtDataAddrMap` array, found later in `rtwdemo_capi_capi.c`. Because the index is zero based, 5 corresponds to the sixth element in `rtDataAddrMap`, which is `rtP_p2`.
- The `dataTypeIndex` provides an index into the `rtDataTypeMap` array, found later in `rtwdemo_capi_capi.c`, indicating the data type of the parameter. The value 0 corresponds to a double, noncomplex parameter.
- The `dimIndex` (dimensions index) provides an index into the `rtDimensionMap` array, found later in `rtwdemo_capi_capi.c`. The value 3 corresponds to the fourth entry, which is `{ rtwCAPI_MATRIX_COL_MAJOR, 6, 2, 0 }`.
- The `fixPtIndex` (fixed-point index) provides an index into the `rtFixPtMap` array, found later in `rtwdemo_capi_capi.c`, indicating fixed-point information about the parameter. As with the corresponding signal attribute, a fixed-point map index of zero means that the parameter does not have fixed-point information.

For more information about tunable parameter storage in the generated code, see “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49.

Map C API Data Structures to `rtModel`

The real-time model data structure encapsulates model data and associated information that describes the model fully. When you select the C API feature and generate code, the Simulink Coder code generator adds another member to the real-time model data structure generated in `model.h`:

```
/*
 * DataMapInfo:
 * The following substructure contains information regarding
 * structures generated in the model's C API.
 */
struct {
    rtwCAPI_ModelMappingInfo mmi;
} DataMapInfo;
```

This member defines `mmi` (for model mapping information) of type `struct rtwCAPI_ModelMappingInfo`. The structure is located in `matlabroot/rtw/c/src/rtw_modelmap.h`. The `mmi` substructure defines the interface between the model and the C API files. More specifically, members of `mmi` map the real-time model data structure to the structures in `model_capi.c` (or `.cpp`).

Initializing values of `mmi` members to the arrays accomplishes the mapping, as shown in “Map Model to C API Arrays of Structures” on page 34-16. Each member points to one of the arrays of structures in the generated C API file. For example, the address of the `rtBlockSignals` array of structures is allocated to the first member of the `mmi` substructure in `model.c` (or `.cpp`), using the following code in the `rtw_modelmap.h` file:

```
/* signals */
struct {
```

```

rtwC_API_Signals const *signals;    /* Signals Array */
uint_T                numSignals;   /* Num Signals   */
rtwC_API_Signals const *rootInputs; /* Root Inputs array */
uint_T                numRootInputs; /* Num Root Inputs */
rtwC_API_Signals const *rootOutputs; /* Root Outputs array */
uint_T                numRootOutputs; /* Num Root Outputs */
} Signals;

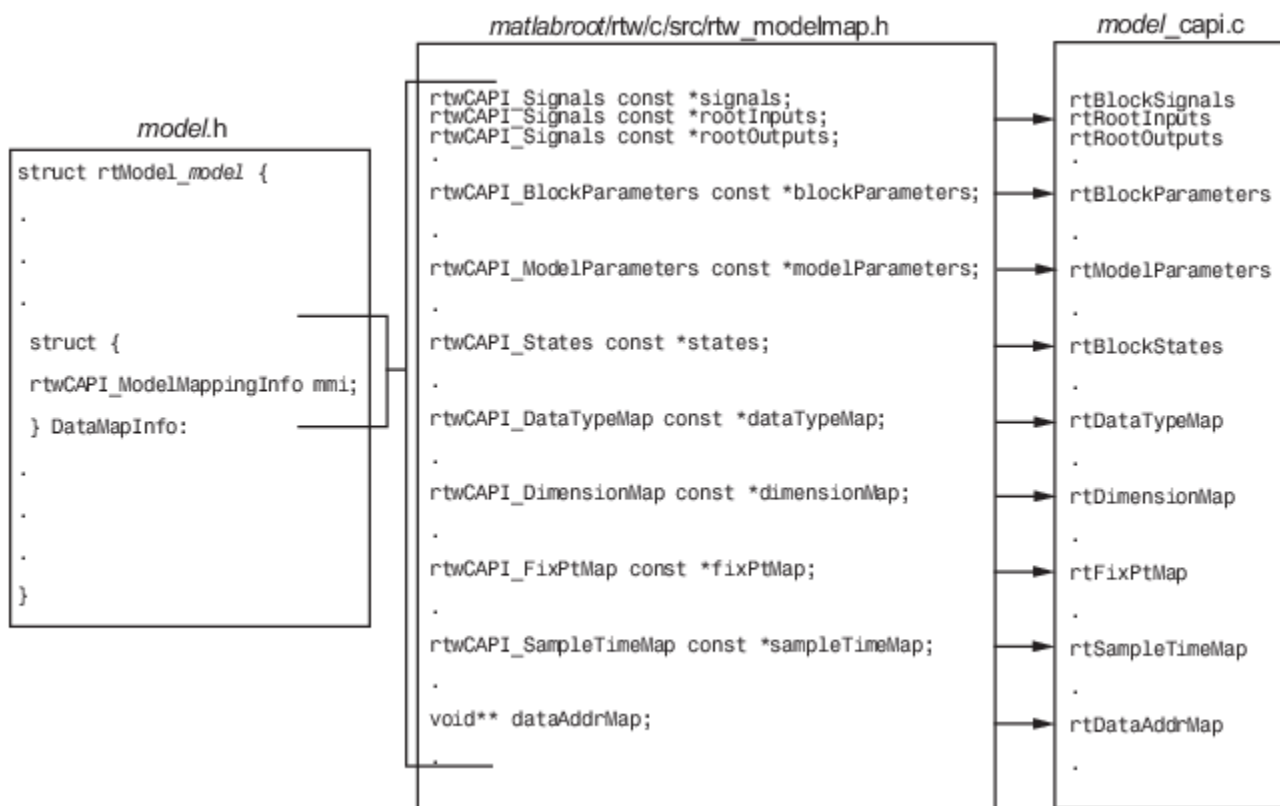
```

The model initialize function in *model.c* (or *.cpp*) performs the initializing by calling the C API initialize function. For example, the following code is generated in the model initialize function for example model *rtwdemo_capi*:

```

/* Initialize DataMapInfo substructure containing ModelMap for C API */
rtwdemo_capi_InitializeDataMapInfo(rtwdemo_capi_M);

```



Map Model to C API Arrays of Structures

Note This figure lists the arrays in the order that their structures appear in *rtw_modelmap.h*, which differs slightly from their generated order in *model_capi.c*.

Generate C API Data Definition File for Exchanging Data with a Target System

This example shows how to use the target-based C API to interface with generated code that represents signals, states, parameters, and root-level I/O.

Open Example Model

Open the example model `rtwdemo_capi`.

```
open_system('rtwdemo_capi');
```

The C API is useful for interacting with application data in generated code, without stopping program execution or recompiling the generated code. To use the C API interface, for a top model and its referenced models:

1. Set up a client/server protocol (such as TCP/IP or a dual-port memory connection) between your development computer and the target computer.
2. Select at least one of the C API model configuration parameters: **signals**, **parameters**, **states**, and **root-level I/O**.
3. Configure the data elements that you want to access with the C API with addressable storage classes.

C API configuration settings for the top model and referenced models must match.

The code generator places the C API interface in the file `model_capi.c`. Depending on your configuration settings, the data can represent signals, states, parameters, and root-level I/O configured with addressable storage classes. The file includes structures that provide an interface to data properties.

C API Limitations

The C API feature has the following limitations.

- The C API does not support the following values for the CodeFormat TLC variable:
 - `S-Function`
 - `Accelerator_S-Function` (for accelerated simulation)
- For ERT-based targets, the C API requires that support for floating-point code be enabled.
- Local block output signals are not supported.
- Local Stateflow parameters are not supported.
- The following custom storage class objects are not supported:
 - Objects without the package `csc_registration` file
 - Grouped custom storage classes
 - Objects defined by using macros
 - `BitField` objects
 - `FileScope` objects
- Customized data placement is disabled when you are using the C API. The interface looks for global data declaration in `model.h` and `model_private.h`. Declarations placed in any other file by customized data placement result in code that does not compile.

Note Custom Storage Class objects work in code generation, only if you use the ERT system target file and clear the model configuration parameter **Ignore custom storage classes**.

See Also

Related Examples

- “Access Signal, State, and Parameter Data During Execution” on page 15-17
- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Use C API to Access Model Signals and States

This example helps you get started writing application code to interact with model signals and states. To get started writing application code to interact with model parameters, see “Use C API to Access Model Parameters” on page 34-24.

The C API provides you with the flexibility of writing your own application code to interact with model signals, states, root-level inputs/outputs, and parameters. Your target-based application code is compiled with the Simulink Coder generated code into an executable. The target-based application code accesses the C API structure arrays in *model_capi.c* (or *.cpp*). You might have host-based code that interacts with your target-based application code. Or, you might have other target-based code that interacts with your target-based application code. The files *rtw_modelmap.h* and *rtw_capi.h*, located in *matlabroot/rtw/c/src* (open), provide macros for accessing the structures in these arrays and their members.

Here is an example application that logs global signals and states in a model to a text file. This code is intended as a starting point for accessing signal and state addresses. You can extend the code to perform signal logging and monitoring, state logging and monitoring, or both.

This example uses the following macro and function interfaces:

- `rtmGetDataMapInfo` macro

Accesses the model mapping information (MMI) substructure of the real-time model structure. In the following macro call, `rtM` is the pointer to the real-time model structure in *model.c* (or *.cpp*):

```
rtwCAPI_ModelMappingInfo* mmi = &(rtmGetDataMapInfo(rtM).mmi);
```

- `rtmGetTPtr` macro

Accesses the absolute time information for the base rate from the timing substructure of the real-time model structure. In the following macro call, `rtM` is the pointer to the real-time model structure in *model.c* (or *.cpp*):

```
rtmGetTPtr(rtM)
```

- Custom functions `capi_StartLogging`, `capi_UpdateLogging`, and `capi_TerminateLogging`, provided via the files *rtwdemo_capi_data_log.h* and *rtwdemo_capi_data_log.c*. These files are located in *matlabroot/toolbox/rtw/rtwdemos* (open).
 - `capi_StartLogging` initializes signal and state logging.
 - `capi_UpdateLogging` logs a signal and state value at each time step.
 - `capi_TerminateLogging` terminates signal and state logging and writes the logged values to a text file.

You can integrate these custom functions into generated code for a model by using these methods:

- **Custom Code** of the Model Configuration Parameters dialog box.
- Custom Code library blocks
- TLC custom code functions

This tutorial uses the **Custom Code** pane of the Model Configuration Parameters dialog box and the System Outputs block from the Custom Code library to insert calls to the custom functions into *model.c* (or *.cpp*), as follows:

- `capi_StartLogging` is called in the `model_initialize` function.
- `capi_UpdateLogging` is called in the `model_step` function.
- `capi_TerminateLogging` is called in the `model_terminate` function.

The following excerpts of generated code from `model.c` (rearranged to reflect their order of execution) show how the function interfaces are used.

```
void rtwdemo_capi_initialize(void)
{
...
/* user code (Initialize function Body) */

/* C API Custom Logging Function: Start Signal and State logging via C API.
 * capi_StartLogging: Function prototype in rtwdemo_capi_dataolog.h
 */
{
    rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo(rtwdemo_capi_M).mmi);
    printf("*** Started state/signal logging via C API **\n");
    capi_StartLogging(MMI, MAX_DATA_POINTS);
}
...
}
...
/* Model step function */
void rtwdemo_capi_step(void)
{
...
/* user code (Output function Trailer) */

/* System '<Root>' */

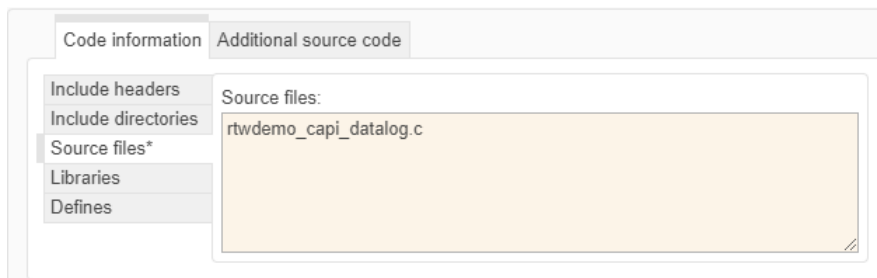
/* C API Custom Logging Function: Update Signal and State logging buffers.
 * capi_UpdateLogging: Function prototype in rtwdemo_capi_dataolog.h
 */
{
    rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo(rtwdemo_capi_M).mmi);
    capi_UpdateLogging(MMI, rtmGetTPtr(rtwdemo_capi_M));
}
...
}
...
/* Model terminate function */
void rtwdemo_capi_terminate(void)
{
    /* user code (Terminate function Body) */

    /* C API Custom Logging Function: Dump Signal and State buffers into a text file.
     * capi_TerminateLogging: Function prototype in rtwdemo_capi_dataolog.h
     */
    {
        capi_TerminateLogging("rtwdemo_capi_ModelLog.txt");
        printf("*** Finished state/signal logging. Created rtwdemo_capi_ModelLog.txt **\n");
    }
}
}
```

The following procedure illustrates how you can use the C API macro and function interfaces to log global signals and states in a model to a text file.

- 1 At the MATLAB command line, enter `rtwdemo_capi` to open the example model.
- 2 Save the top model `rtwdemo_capi` and the referenced model `rtwdemo_capi_bot` to the same writable work folder.
- 3 Open the Configuration Parameters dialog box.
- 4 If you are licensed for Embedded Coder software and you want to use the `ert.tlc` system target file instead of the default `grt.tlc`, change the setting of model configuration parameter **System target file**. Make sure that you also select `ert.tlc` for the referenced model `rtwdemo_capi_bot`.
- 5 For the top model, confirm these model configuration parameter settings:

- a Select parameters **Generate C API for signals**, **Generate C API for states**, and **Generate C API for parameters**.
 - b If you are using the `ert.tlc` system target file, select **Support complex numbers**
 - c Select **MAT-file logging**.
 - d Click **Apply**.
 - e Update configuration parameter settings in the referenced model, `rtwdemo_capi_bot`, to match changes you made in the top model.
- 6 Use the **Custom Code** pane to embed your custom application code in the generated code. Select the **Custom Code** pane. On the **Code information** tab, click **Include directories**. The **Include directories** input field is displayed.
 - 7 In the **Include directories** field, type `matlabroot/toolbox/rtw/rtwdemos`, where `matlabroot` represents the root of your MATLAB installation folder. (If you are specifying a Windows path that contains a space, place the text inside double quotes.)
 - 8 On the same tab, click **Source files**, and type `rtwdemo_capi_data_log.c`.



- 9 On the **Additional source code** tab, click **Additional code**, and type or copy and paste the following include statement:


```
#include "rtwdemo_capi_data_log.h"
```
- 10 On the same tab, in the **Initialize code** field, type or copy and paste the following application code:

```
/* C API Custom Logging Function: Start Signal and State logging via C API.
 * capi_StartLogging: Function prototype in rtwdemo_capi_data_log.h
 */
{
    rtwC_API_ModelMappingInfo *MMI = &(rtmGetDataMapInfo(rtwdemo_capi_M).mmi);
    printf("*** Started state/signal logging via C API **\n");
    capi_StartLogging(MMI, MAX_DATA_POINTS);
}
```

Note If you renamed the top model `rtwdemo_capi`, update the name `rtwdemo_capi_M` in the application code to reflect the new model name.

- 11 On the same tab, in the **Terminate code** field, type or copy and paste the following application code:

```
/* C API Custom Logging Function: Dump Signal and State buffers into a text file.
 * capi_TerminateLogging: Function prototype in rtwdemo_capi_data_log.h
 */
{
    capi_TerminateLogging("rtwdemo_capi_ModelLog.txt");
    printf("*** Finished state/signal logging. Created rtwdemo_capi_ModelLog.txt **\n");
}
```

Click **Apply**.

- 12** In the MATLAB Command Window, enter `custcode` to open the Simulink Coder Custom Code library. At the top level of the `rtwdemo_capi` model, add a System Outputs block.
- 13** Double-click the System Outputs block to open the System Outputs Function Custom Code dialog box. In the **System Outputs Function Exit Code** field, type or copy and paste the following application code:

```
/* C API Custom Logging Function: Update Signal and State logging buffers.
 * capi_UpdateLogging: Function prototype in rtwdemo_capi_dataLog.h
 */
{
    rtwCAPI_ModelMappingInfo *MMI = &(rtmGetDataMapInfo(rtwdemo_capi_M).mmi);
    capi_UpdateLogging(MMI, rtmGetTPtr(rtwdemo_capi_M));
}
```

If you renamed the top model `rtwdemo_capi`, update two instances of the name `rtwdemo_capi_M` in the application code to reflect the new model name.

Click **OK**.

- 14** Clear model configuration parameter **Generate code only**.

Build the model and generate an executable program. For example, on a Windows system, the build generates the executable file `rtwdemo_capi.exe` in your current working folder.

- 15** In the MATLAB Command Window, enter the command `!rtwdemo_capi` to run the executable file. During execution, signals and states are logged using the C API and then written to the text file `rtwdemo_capi_ModelLog.txt` in your current working folder.

```
>> !rtwdemo_capi

** starting the model **
** Started state/signal logging via C API **
** Logging 2 signal(s) and 1 state(s). In this demo, only scalar named
    signals/states are logged **
** Finished state/signal logging. Created rtwdemo_capi_ModelLog.txt **
```

- 16** Examine the text file in the MATLAB editor or other text editor. Here is an excerpt of the signal and state logging output.

```
***** Signal Log File *****

Number of Signals Logged: 2
Number of points (time steps) logged: 51

Time          bot_sig1 (Referenced Model)      top_sig1
0             70                          4
0.2           70                          4
0.4           70                          4
0.6           70                          4
0.8           70                          4
1             70                          4
1.2           70                          4
1.4           70                          4
1.6           70                          4
1.8           70                          4
2             70                          4
...

***** State Log File *****

Number of States Logged: 1
Number of points (time steps) logged: 51

Time          bot_state (Referenced Model)
0             0
0.2           70
0.4           35
0.6           52.5
```

0.8	43.75
1	48.13
1.2	45.94
1.4	47.03
1.6	46.48
1.8	46.76
2	46.62
...	

See Also

Related Examples

- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Use C API to Access Model Parameters

This example helps you get started writing application code to interact with model parameters. To get started writing application code to interact with model signals and states, see “Use C API to Access Model Signals and States” on page 34-19.

The C API provides you with the flexibility of writing your own application code to interact with model signals, states, root-level inputs/outputs, and parameters. Your target-based application code is compiled with generated code into an executable program. The target-based application code accesses the C API structure arrays in *model_capi.c* (or *.cpp*). You might have host-based code that interacts with your target-based application code. Or, you might have other target-based code that interacts with your target-based application code. The files *rtw_modelmap.h* and *rtw_capi.h*, located in *matlabroot/rtw/c/src* (open), provide macros for accessing the structures in these arrays and their members.

Here is an example application that prints the parameter values of tunable parameters in a model to the standard output. This code is intended as a starting point for accessing parameter addresses. You can extend the code to perform parameter tuning. The application:

- Uses the `rtmGetDataMapInfo` macro to access the mapping information in the `mmi` substructure of the real-time model structure

```
rtwCAPI_ModelMappingInfo* mmi = &(rtmGetDataMapInfo(rtM).mmi);
```

where `rtM` is the pointer to the real-time model structure in *model.c* (or *.cpp*).

- Uses `rtwCAPI_GetNumModelParameters` to get the number of model parameters in mapped C API:

```
uint_T nModelParams = rtwCAPI_GetNumModelParameters(mmi);
```

- Uses `rtwCAPI_GetModelParameters` to access the array of model parameter structures mapped in C API:

```
rtwCAPI_ModelParameters* capiModelParams = \
    rtwCAPI_GetModelParameters(mmi);
```

- Loops over the `capiModelParams` array to access individual parameter structures. A call to the function `capi_PrintModelParameter` displays the value of the parameter.

The example application code is provided below:

```
{
/* Get CAPI Mapping structure from Real-Time Model structure */
rtwCAPI_ModelMappingInfo* capiMap = \
&(rtmGetDataMapInfo(rtwdemo_capi_M).mmi);

/* Get number of Model Parameters from capiMap */
uint_T nModelParams = rtwCAPI_GetNumModelParameters(capiMap);
printf("Number of Model Parameters: %d\n", nModelParams);

/* If the model has Model Parameters, print them using the
application capi_PrintModelParameter */
if (nModelParams == 0) {
    printf("No Tunable Model Parameters in the model \n");
}
else {
```

```

unsigned int idx;

for (idx=0; idx < nModelParams; idx++) {
    /* call print utility function */
    capi_PrintModelParameter(capiMap, idx);
}
}
}

```

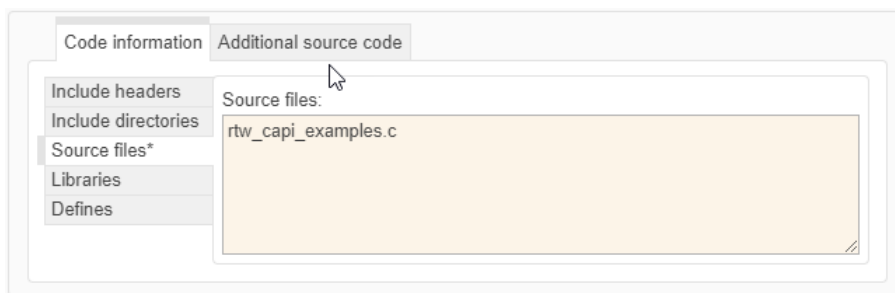
The print utility function is located in `matlabroot/rtw/c/src/rtw_capi_examples.c`. This file contains utility functions for accessing the C API structures.

To become familiar with the example code, try building a model that displays the tunable block parameters and MATLAB variables. You can use `rtwdemo_capi`, the C API example model. The following steps apply to both `grt.tlc` and `ert.tlc` system target files, unless otherwise indicated.

- 1 At the MATLAB command line, enter `rtwdemo_capi` to open the example model.
- 2 Save the top model `rtwdemo_capi` and the referenced model `rtwdemo_capi_bot` to the same writable work folder.
- 3 If you are licensed for Embedded Coder software and you want to use the `ert.tlc` system target file instead of the default `grt.tlc`, use model configuration parameter **System target file** to select an `ert.tlc` system target file. Make sure that you also select `ert.tlc` for the referenced model `rtwdemo_capi_bot`.
- 4 Confirm these model configuration parameter settings:
 - a Select **Generate C API for parameters**.
 - b If you are using the `ert.tlc` system target file, select **Support complex numbers**.
 - c Select **MAT-file logging**.
 - d Click **Apply**.
 - e Update configuration parameter settings in the referenced model, `rtwdemo_capi_bot`, to match changes you made in the top model.
- 5 Use the **Custom Code** pane to embed your custom application code in the generated code. Select the **Custom Code** pane. On the **Additional source code** tab, click **Initialize code**. The **Initialize code** input field is displayed.
- 6 In the **Initialize code** input field, type or copy and paste the example application code shown above step 1. This embeds the application code in the `model_initialize` function.

Note If you renamed the top model `rtwdemo_capi`, update the name `rtwdemo_capi_M` in the application code to reflect the new model name.

- 7 On the **Code information** tab, click **Include directories**, and type `matlabroot/rtw/c/src`, where `matlabroot` represents the root of your MATLAB installation folder. (If you are specifying a Windows path that contains a space, place the text inside double quotes.)
- 8 On the same tab, click **Source files**, and type `rtw_capi_examples.c`.



Click **Apply**.

- 9 Clear model configuration parameter **Generate code only**.

Build the model and generate an executable program. For example, on a Windows system, the build generates the executable file `rtwdemo_capi.exe` in your current working folder.

- 10 In the MATLAB Command Window, enter `!rtwdemo_capi` to run the executable file. Running the program displays parameter information in the Command Window.

```
>> !rtwdemo_capi

** starting the model **
Number of Model Parameters: 5
Ki =
    7
Kp =
    4
p1 =
    0.15
    0.36
    0.81
p2 =
    0.09    0.75    0.57
    0.13    0.96    0.059
p3 =
ans(:,:,1) =
    0.23    0.82    0.04    0.64
    0.35    0.01    0.16    0.73

ans(:,:,2) =
    0.64    0.54    0.74    0.68
    0.45    0.29    0.18    0.18
```

See Also

Related Examples

- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

ASAP2 Data Measurement and Calibration in Simulink Coder

- “Export ASAP2 File for Data Measurement and Calibration” on page 35-2
- “Calibrate Generated Code and Monitor Signals” on page 35-11

Export ASAP2 File for Data Measurement and Calibration

“Generate ASAP2 and CDF Calibration Files” (Embedded Coder) is recommended way of generating the ASAP2 file, because it allows you to generate the latest version of ASAP2 file as per ASAM standards and provides the **Generate Calibration Files** tool.

Generation of ASAP2 File Using Generate Calibration Files Tool

- “Open Generate Calibration Files Tool” on page 35-2
- “ASAP2 File Generation” on page 35-3

Open Generate Calibration Files Tool

- 1 Open the model.
- 2 Open the Embedded Coder app.
- 3 Generate code for the model.
- 4 On the **C Code** tab, select **Share > Generate Calibration Files**.

Note For AUTOSAR models, select **Autosar > Share > Generate Calibration Files**.

Using the tool, you can generate A2L and cdfx files at the same time for a model. You can use the tool for models configured with an ERT-based, GRT-based, Simulink Real-Time, Classic AUTOSAR, or Adaptive AUTOSAR system target file. The tool remembers the last used settings, such as the version of the ASAP2 file, include or exclude comments, turn off or on the ASAP2 file and CDF file generation. Simulink saves these settings in the MATLAB preferences. The settings will override the default settings.

For Adaptive AUTOSAR and Simulink Real-Time system target files, you can generate the ASAP2 file for the C++ language. For GRT- and ERT- based system target files, you can generate an ASAP2 file only for C.

For classic AUTOSAR models, the tool allows you to include or exclude RTE elements into the ASAP2 file by using the `Include AUTOSAR RTE elements` option.

Generate Calibration Files (ClassicAutosar_DemoModel)

Generate .a2l and .cdfx files according to "ASAM MCD-2 MC (ASAP2/A2L)" and "ASAM AE CDF" standards, respectively.

Generate ASAP2 Off On

File name :

Folder path : Browse...

Address replacement

Symbol file : Browse...

Version : 1.71 ▼

Include comments

Include AUTOSAR RTE elements

Generate CDF Off On

File name :

Folder path : Browse...

Schema type : DTD ▼

Generate Close Help

ASAP2 File Generation

ASAP2 defines the description format of the internal ECU variables used for tuning parameters and measuring signal data as an application executes. To generate an ASAP2 file:

- 1 By default the **Generate ASAP2** option is enabled. If it is not enabled, slide the **Generate ASAP2** slider to the on position to enable A2L generation.
- 2 To change the name of the A2L file, in the **File name** field, specify the name.
- 3 By default, the code generator uses the build folder for the folder path of the selected model. If you want to specify a different folder path for the A2L file, you can do that in the **Folder path** field.
- 4 To replace ECU addresses in the A2L file, in the **Symbol file** field, provide a symbol file associated with the application executable. Otherwise, clear the **Address replacement** field.

The tool supports an ELF, PDB, DWARF, or executable file as a symbol file to replace the ECU addresses.

- 5 Set **Version** to the required version of the A2L file to be generated. The default version is 1.71.
- 6 To exclude comments in the A2L file, clear **Include comments**.

- 7 For Classic AUTOSAR models, to exclude RTE elements in the A2L file, clear **Include AUTOSAR RTE elements**.
- 8 Click **Generate**.

To generate the A2L file programmatically, use the `coder.asap2.export` function.

Generation of ASAP2 File Using ASAP2Interface Parameter

- “What You Should Know” on page 35-4
- “System Target Files Supporting ASAP2” on page 35-4
- “Define ASAP2 Information” on page 35-5
- “Generate an ASAP2 File” on page 35-8
- “Structure of ASAP2 File” on page 35-10

The `ASAP2Interface` parameter is not present in the configuration parameters anymore. In a coming release the parameter will be completely removed. You can generate the ASAP2 file using **Generate Calibration Files** tool, or `coder.asap2.export` function. For more information, see “Generate ASAP2 and CDF Calibration Files” (Embedded Coder).

The ASAM MCD-2 MC standard, also known as ASAP2, is a data definition standard proposed by the Association for Standardization of Automation and Measuring Systems (ASAM). ASAP2 is a non-object-oriented description of the data used for measurement, calibration, and diagnostic systems. For more information on ASAM and the ASAM MCD-2 MC (ASAP2) standard, see the ASAM website at www.asam.net.

The code generator enables you to export an ASAP2 file containing information about your model during the code generation process.

Simulink Coder support for ASAP2 file generation is version-neutral. By default, the software generates ASAP2 version 1.31 format, but the generated model information is generally compatible with other ASAP2 versions. ASAP2 file generation is also neutral with respect to the specific needs of ASAP2 measurement and calibration tools. The software provides customization APIs that you can use to customize ASAP2 file generation to generate an ASAP2 version and to meet the specific needs of your ASAP2 tools.

What You Should Know

To make use of ASAP2 file generation, you should become familiar with:

- ASAM and the ASAP2 standard and terminology. See the ASAM Web site at www.asam.net.
- Configuring model data elements for code generation. See **Code Mappings Editor - C**.
- Storage and representation of signals and parameters in generated code. See “Data Representation in Generated Code”.
- If you have Embedded Coder, see “Data Representation in Generated Code” (Embedded Coder).

System Target Files Supporting ASAP2

You can use ASAP2 file generation with varying system target file configurations. For example, GRT- and ERT- based configurations enables you to generate an ASAP2 file as part of the code generation and build process.

For Adaptive AUTOSAR target and Simulink Real-Time targets, you can generate the ASAP2 file for C++ language. For ERT and GRT targets, you can generate ASAP2 file only for language C and language C++ is not supported.

For procedures for generating ASAP2 files by using the system target file configurations, see ASAP2 interface.

Define ASAP2 Information

- “Define ASAP2 Information for Parameters and Signals” on page 35-5
- “Memory Address Attribute” on page 35-5
- “Automatic ECU Address Replacement for ASAP2 Files (Embedded Coder)” on page 35-6
- “Define ASAP2 Information for Lookup Tables” on page 35-7

Define ASAP2 Information for Parameters and Signals

The ASAP2 file generation process requires information about parameters and signals in your model. Some of this information is contained in the model itself. Use Simulink data objects in a workspace or data dictionary to complete the ASAP2 file generation process. In some cases, the use of workspace objects is optional.

Use the Model Data Editor and built-in Simulink data objects to provide information, such as using `Simulink.Signal` objects to provide MEASUREMENT information and `Simulink.Parameter` objects to provide CHARACTERISTIC information. You can use data objects from data classes that are derived from `Simulink.Signal` and `Simulink.Parameter` to provide the information. See “Data Objects” and **Model Data Editor**.

This table contains the minimum set of data attributes required for ASAP2 file generation. Some data attributes are defined in the model. Others attributes are supplied in the properties of objects. For attributes that are defined in `Simulink.Signal` or `Simulink.Parameter` objects, the table lists the associated property name.

Data Attribute	Defined In	Property Name
Name (symbol)	Model and data object	Inherited from the handle of the data object to which parameter or signal name resolves
Description	Data object	Description
Data type	Model or data object	Data Type
Scaling (if fixed-point data type)	Model or data object	Data Type
Minimum allowable value	Model or data object	Min
Maximum allowable value	Model or data object	Max
Unit	Model or data object	Unit
Memory address (optional)	Model or data object	MemoryAddress_ASAP2

Memory Address Attribute

If the memory address attribute is unknown before code generation, the code generator inserts ECU Address placeholder text in the generated ASAP2 file. You can substitute an actual address for the placeholder by postprocessing the generated file. For an example, see the file `matlabroot/`

`toolbox/rtw/targets/asap2/asap2/asap2post.m`. `asap2post.m` parses through the linker map file that you provide and replaces the ECU Address placeholders in the ASAP2 file with the actual memory addresses. Because linker map files vary from compiler to compiler, you might need to modify the regular expression code in `asap2post.m` to match the format of the linker map that you use.

If you have Embedded Coder installed on your system, and if you are generating Executable and Linkable Format (ELF) or Program Database (PDB) files for your embedded target, you can use the `rtw.asap2SetAddress` function to automate ECU address replacement. For more information, see “Automatic ECU Address Replacement for ASAP2 Files (Embedded Coder)” (Embedded Coder).

If the memory address attribute is known before code generation, it can be defined in the data item or data object. By default, the `MemoryAddress_ASAP2` property does not exist in the `Simulink.Signal` or `Simulink.Parameter` data object classes. If you want to add the attribute, add a property called `MemoryAddress_ASAP2` to a custom class that is a subclass of the `Simulink` or `ASAP2` class. For information on subclassing Simulink data classes, see “Define Data Classes”.

This table indicates the Simulink object properties that have replaced the ASAP2 object properties of the previous release.

Differences Between ASAP2 and Simulink Parameter and Signal Object Properties

ASAP2 Object Properties (Previous)	Simulink Object Properties (Current)
<code>LONGID_ASAP2</code>	Description
<code>PhysicalMin_ASAP2</code>	Min
<code>PhysicalMax_ASAP2</code>	Max
<code>Units_ASAP2</code>	Unit

Automatic ECU Address Replacement for ASAP2 Files (Embedded Coder)

If you have Embedded Coder installed on your system, and if you are generating Executable and Linkable Format (ELF) or Program Database (PDB) files for your embedded target, you can use the `rtw.asap2SetAddress` function to automate the replacement of ECU Address placeholder memory address values with actual addresses in the generated ASAP2 file.

If the memory address attribute is unknown before code generation, the code generator inserts ECU Address placeholder text in the generated ASAP2 file, as shown

```
/begin CHARACTERISTIC
/* Name          */ Ki
/* Long Identifier */ ""
/* Type          */ VALUE
/* ECU Address   */ 0x0000 /* @ECU_Address@Ki@ */
```

To substitute actual addresses for the ECU Address placeholders, process the generated ASAP2 file by using the `rtw.asap2SetAddress` function. The syntax is:

```
rtw.asap2SetAddress(ASAP2File,InfoFile)
```

The arguments are character vectors specifying the name of the generated ASAP2 file and the name of the generated executable ELF file, PDB file from Microsoft toolchain, or DWARF debug information files for the model. When called, `rtw.asap2SetAddress` extracts the actual ECU address from the specified ELF, PDB, or DWARF file and replaces the placeholder in the ASAP2 file with the actual address, for example:

```

/begin CHARACTERISTIC
/* Name          */ Ki
/* Long Identifier */ ""
/* Type         */ VALUE
/* ECU Address   */ 0x40009E60

```

Define ASAP2 Information for Lookup Tables

Simulink Coder software generates ASAP2 descriptions for lookup table data and its breakpoints. The software represents 1-D table data as **CURVE** information, 2-D table data as **MAP** information, and breakpoints as **AXIS_DESCR** and **AXIS_PTS** information. You can model lookup tables by using one of the following Simulink Lookup Table blocks:

- Interpolation Using Prelookup — 1 and 2 dimensions
- 1-D Lookup Table
- 2-D Lookup Table
- n-D Lookup Table — 1 and 2 dimensions

The software supports these types of lookup table breakpoints (axis points).

Breakpoint Type	Information Generated
Tunable and shared among multiple table axes (common axis)	COM_AXIS
Fixed and nontunable (fixed axis)	One of these variants of FIX_AXIS: <ul style="list-style-type: none"> • FIX_AXIS_PAR if breakpoints are integers with equidistant spacing and the equidistant spacing is a power of two • FIX_AXIS_PAR_DIST if breakpoints are integers with equidistant spacing • FIX_AXIS_PAR_LIST if breakpoints are integers with non-equidistant spacing
Tunable but not shared among multiple tables (standard axis)	STD_AXIS

When you configure the blocks for ASAP2 code generation:

- For table data, use a `Simulink.Parameter` data object with a non-Auto storage class.
- For tunable breakpoint data that is shared among multiple table axes (COM_AXIS), use a `Simulink.Parameter` data object that has a non-Auto storage class.
- For fixed, nontunable breakpoint data (FIX_AXIS), use workspace variables or arrays specified in the block parameters dialog box. The breakpoints must be stored as integers in the code. The data type must be a built-in integer type (`int8`, `int16`, `int32`, `uint8`, `uint16`, or `uint32`), a fixed-point data type, or an equivalent alias type.
- For tunable breakpoint data that is not shared among multiple tables (STD_AXIS):
 - 1 Create a `Simulink.Bus` object to define the `struct` packaging (names and order of the fields). The fields of the parameter structure must correspond to the lookup table data and each axis of the lookup table block. For example, in an n-D Lookup Table block that has 2 dimensions, the structure must contain only three fields. This bus object describes the record layout for the lookup characteristic.

- 2 Create a Simulink.Parameter object to represent a tunable parameter.
- 3 Create table and axis values.
- 4 Optionally, specify the **Units**, **Minimum**, and **Maximum** properties for the parameter object.

Here is an example of an n-D Lookup Table record generated into an ASAP2 file in Standard Axis format:

```

/begin CHARACTERISTIC
/* Name */  STDAxisParam
...
/* Record Layout */  Lookup1D_X_WORD_Y_FLOAT32_IEEE
...
begin AXIS_DESCR
/* Description of X-Axis Points */
/* Axis Type */      STD_AXIS
...
/end AXIS_DESCR
/end CHARACTERISTIC

/begin RECORD_LAYOUT Lookup1D_X_WORD_Y_FLOAT32_IEEE
  AXIS_PTS_X 1 WORD INDEX_INCR DIRECT
  FNC_VALUES 2 FLOAT32_IEEE COLUMN_DIR DIRECT
/end RECORD_LAYOUT

```

Generate an ASAP2 File

- “Generating ASAP2 Files” on page 35-8
- “Use GRT- or ERT- Based System Target File” on page 35-8
- “Generate ASAP2 Files for Referenced Models” on page 35-9
- “Merge ASAP2 Files for Top and Referenced Models” on page 35-9

Generating ASAP2 Files

Use the Generic Real-Time system target file or a Embedded Coder system target file to generate an ASAP2 file as part of the code generation and build process.

Generate ASAP2 file by using the system target files that have built-in ASAP2 support.

Use GRT- or ERT- Based System Target File

To generate the ASAP2 data definition for a model by using a Generic Real-Time(GRT) system target file or Embedded Coder system target file:

- 1 Use the **Code Mappings Editor - C** to apply storage classes to signals, block states, and model parameters, as described in “C Code Generation Configuration for Model Interface Elements” on page 16-2.

Parameters are not represented in the ASAP2 file if one or more of the following conditions exist:

- You apply storage class Auto, FileScope, Struct, or Default (through Model default).
- You apply storage class through Embedded Coder Dictionary.
- You apply a storage class that causes the code generator to generate a macro or non-addressable variable.

- 2 Use the Property Inspector to configure the remaining storage class properties for each data element.
- 3 Set the model configuration parameter **System target file** to a GRT- or ERT- based system target file.
- 4 Enable the configuration parameter **ASAP2 interface** with the command


```
set_param('model_Name','GenerateASAP2',true);
```

Note You can no longer enable generation of ASAP2 files from the Model Configuration Parameter dialog box. As “Generate ASAP2 and CDF Calibration Files” on page 36-2 describes, use the Generate Calibration Files tool, or use the command line model configuration parameter `GenerateASAP2`.

- 5 Select parameter **Generate code only**.
- 6 Click **Apply**.
- 7 Generate code.

The code generator writes the ASAP2 file to the build folder. By default, the file is named `model.a2l`, where `model` is the name of the model. The ASAP2 setup file controls the ASAP2 file name. For details, see “Customize Generated ASAP2 File” on page 45-2.

Generate ASAP2 Files for Referenced Models

The build process can generate an ASAP2 file for each referenced model in a model reference hierarchy. In the generated ASAP2 file, MEASUREMENT records represent signals and states inside the referenced model.

To generate ASAP2 files for referenced models, select ASAP2 file generation for the top model and for each referenced model in the reference hierarchy.

Merge ASAP2 Files for Top and Referenced Models

Use function `rtw.asap2MergeMdlRefs` to merge the ASAP2 files generated for top and referenced models. The function has this syntax:

```
[status,info] = rtw.asap2MergeMdlRefs(topModelName,asap2FileName)
```

- `topModelName` is the name of the model containing one or more referenced models.
- `asap2FileName` is the name that you specify for the merged ASAP2 file.
- *Optional:* `status` returns false (logical 0) if the merge completed and returns true (logical 1) otherwise.
- *Optional:* `info` returns additional information about merge failure if `status` is true. Otherwise, it returns an empty character vector.

Consider the following example.

```
[status,info] = rtw.asap2MergeMdlRefs('myTopMdl','merged.a2l')
```

This command merges the ASAP2 files generated for the top model `myTopMdl` and its referenced models in the file `merged.a2l`.

Structure of ASAP2 File

This table outlines the basic structure of the ASAP2 file and describes the Target Language Compiler (TLC) functions and files used to create each part of the file:

- Static parts of the ASAP2 file are shown in **bold**.
- Function calls are indicated by %<FunctionName (>).

File Section	Contents of asap2main.tlc	TLC File Containing Function Definition
File header	%<ASAP2UserFcnWriteFileHead(>	asap2userlib.tlc
/begin PROJECT ""	/begin PROJECT "%<ASAP2ProjectName>"	asap2setup.tlc
/begin HEADER "" HEADER contents	/begin HEADER "%<ASAP2HeaderName>" %<ASAP2UserFcnWriteHeader(>	asap2setup.tlc asap2userlib.tlc
/end HEADER	/end HEADER	asap2userlib.tlc
/begin MODULE "" MODULE contents:	/begin MODULE "%<ASAP2ModuleName>" }	asap2setup.tlc asap2userlib.tlc
- A2ML - MOD_PAR - MOD_COMMON ...	%<ASAP2UserFcnWriteHardwareInterface(>	asap2userlib.tlc
Model-dependent MODULE contents:	%<SLibASAP2WriteDynamicContents(> Calls user-defined functions:	asap2lib.tlc
- RECORD_LAYOUT - CHARACTERISTIC - ParameterGroups - ModelParameters	...WriteRecordLayout_TemplateName() ...WriteCharacteristic_TemplateName() ...WriteCharacteristic_Scalar()	user/templates/...
- MEASUREMENT - ExternalInputs - BlockOutputs	...WriteMeasurement()	asap2userlib.tlc
- COMPU_METHOD	...WriteCompuMethod()	asap2userlib.tlc
/end MODULE	/end MODULE	
File footer/tail	%<ASAP2UserFcnWriteFileTail(>	asap2userlib.tlc

See Also

Related Examples

- “Generate ASAP2 and CDF Calibration Files” on page 36-2
- “Create Tunable Calibration Parameter in the Generated Code” on page 15-112
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Calibrate Generated Code and Monitor Signals

When you create an external mode target application that uses an ASAM MCD-1 XCP communication channel, you can also generate an ASAP2 file that provides a description of the Simulink Coder Universal Measurement and Calibration Protocol (XCP) server configuration. The description enables third-party calibration tools, such as ETAS® INCA and Vector CANape®, to connect to the external mode target application. You can use the third-party tools to calibrate generated code and monitor signals in environments supported by the third-party tools, for example, a vehicle, a test bench, or a virtual environment on a PC.

Note This feature requires an Embedded Coder license.

Configure your model to generate an ASAP2 file that provides a description of the XCP server configuration:

- 1 On the **Apps** tab, click **Run on Custom Hardware**, which displays the **Hardware** tab.
- 2 In the **Hardware** section, in the **System Target File Description** field, select Embedded Coder (`ert.tlc`).
- 3 In the **Prepare** section, click **Hardware Settings**, which opens the Configuration Parameters dialog box.
- 4 Clear **Generate code only**.
- 5 Specify **Toolchain settings** for your development computer.
- 6 Select **External mode**.
- 7 Set **Transport layer** to XCP on TCP/IP or XCP on Serial.
- 8 For **Mex-file arguments**, specify the required arguments. For example:
 - For XCP on TCP/IP, specify the IP address of the target processor and the port number of the TCP/IP server.
 - For XCP on Serial, specify the serial port ID and the baud value.
- 9 Click **OK**.

To generate the target application and the ASAP2 file:

- 1 On the **Hardware** tab, in the **Run on Hardware** section, click **Monitor & Tune**.
- 2 Under **Step By Step Commands**, click **Build for Monitoring**.
- 3 Use **C Code > Share > Generate Calibration Files** tool to generate the ASAP2 file. Or generate the file by using this command:

```
coder.asap2.export('modelName', 'MapFile', 'symbolFileName');
```

- 4 In the code generation folder, the build process generates the XCP external mode target application `modelName.exe` and the **Generate Calibration Files** tool generates ASAP2 file `modelName.a2l`. The ASAP2 file contains an IF_DATA XCP block that describes the Simulink Coder XCP server configuration.

```
...
/begin IF_DATA XCP
  /begin PROTOCOL_LAYER
  ...
```

```
    /end PROTOCOL_LAYER
    ...
    /begin XCP_ON_TCP_IP
    ...
    /end XCP_ON_TCP_IP
/end IF_DATA
...
```

Import the ASAP2 file into your third-party tool and manually run the external mode application on your target hardware. Through XCP, connect the third-party tool to the target application. Then, using that tool, perform data acquisition and calibration of the generated code.

See Also

`coder.asap2.export`

More About

- “Generate ASAP2 and CDF Calibration Files” on page 36-2
- “External Mode Simulation by Using XCP Communication” on page 32-8
- “Customize XCP Server Software” on page 32-45

Generate ASAP2 and CDF Calibration Files

Generate ASAP2 and CDF Calibration Files

Generate and customize an ASAP2 file and a CDF (Calibration Data Format) file for a model by using the **Generate Calibration Files** tool. By default, the tool generates the latest supported version of the A2L file according to ASAM MCD-2 MC standards. You can choose to generate a different version. You can customize the options for calibration files, such as the version for the A2L file and the schema type, for the CDF file. For more information on the ASAM MCD-2 MC (ASAP2) standard and the ASAM CDF Standards, see the ASAM website at www.asam.net.

Open Generate Calibration Files Tool

- 1 Open the model.
- 2 Open the Simulink Coder app.

Note For AUTOSAR models, open the AUTOSAR Component Designer app. And for Simulink Real-Time models, open the Simulink Coder app.

- 3 Generate code for the model.
- 4 On the **C Code** tab, select **Share > Generate Calibration Files**.

Note For AUTOSAR models, select **Autosar > Share > Generate Calibration Files**.

Using the tool, you can generate A2L and cdfx files at the same time for a model. You can use the tool for models configured with an ERT-based, GRT-based, Simulink Real-Time, Classic AUTOSAR, or Adaptive AUTOSAR system target file. The tool remembers the last used settings, such as the version of the ASAP2 file, include or exclude comments, turn off or on the ASAP2 file and CDF file generation. Simulink saves these settings in the MATLAB preferences. The settings will override the default settings.

For Adaptive AUTOSAR and Simulink Real-Time system target files, you can generate the ASAP2 file for the C++ language. For GRT- and ERT- based system target files, you can generate an ASAP2 file only for C.

For classic AUTOSAR models, the tool allows you to include or exclude RTE elements into the ASAP2 file by using the **Include AUTOSAR RTE elements** option.

Generate Calibration Files (ClassicAutosar_DemoModel)

Generate .a2l and .cdfx files according to "ASAM MCD-2 MC (ASAP2/A2L)" and "ASAM AE CDF" standards, respectively.

Generate ASAP2 Off On

File name :

Folder path : Browse...

Address replacement

Symbol file : Browse...

Version : 1.71 ▼

Include comments

Include AUTOSAR RTE elements

Generate CDF Off On

File name :

Folder path : Browse...

Schema type : DTD ▼

Generate Close Help

ASAP2 File Generation

ASAP2 defines the description format of the internal ECU variables used for tuning parameters and measuring signal data as an application executes. To generate an ASAP2 file:

- 1 By default the **Generate ASAP2** option is enabled. If it is not enabled, slide the **Generate ASAP2** slider to the on position to enable A2L generation.
- 2 To change the name of the A2L file, in the **File name** field, specify the name.
- 3 By default, the code generator uses the build folder for the folder path of the selected model. If you want to specify a different folder path for the A2L file, you can do that in the **Folder path** field.
- 4 To replace ECU addresses in the A2L file, in the **Symbol file** field, provide a symbol file associated with the application executable. Otherwise, clear the **Address replacement** field.

The tool supports an ELF, PDB, DWARF, or executable file as a symbol file to replace the ECU addresses.

- 5 Set **Version** to the required version of the A2L file to be generated. The default version is 1.71.
- 6 To exclude comments in the A2L file, clear **Include comments**.

- 7 For Classic AUTOSAR models, to exclude RTE elements in the A2L file, clear **Include AUTOSAR RTE elements**.
- 8 Click **Generate**.

To generate the A2L file programmatically, use the `coder.asap2.export` function.

CDF File Generation

ASAM CDF defines the data description format for storing the ECU parameter values and the associated meta data. The ASAM CDF files use XML, which can be easily validated, edited, imported, and exported by calibration tools and XML editors. To generate a CDF file by using the tool:

- 1 Slide the **Generate CDF** slider to the on position.
- 2 To change the name of the CDF file, specify it in the **File name** field.
- 3 By default, the code generator selects the build folder for the folder path of the selected model. If you want to specify a different folder path for the CDF file, in the **Folder path** field, specify the path.
- 4 Use the **Schema type** field to set to **XSD** (XML Schema Definition) or **DTD** (Document Type Definition). The **DTD** schema type is the default.

To generate the CDF file programmatically, use `coder.cdf.export` function.

Note CDF file generation is disabled for Simulink Real-Time system target file.

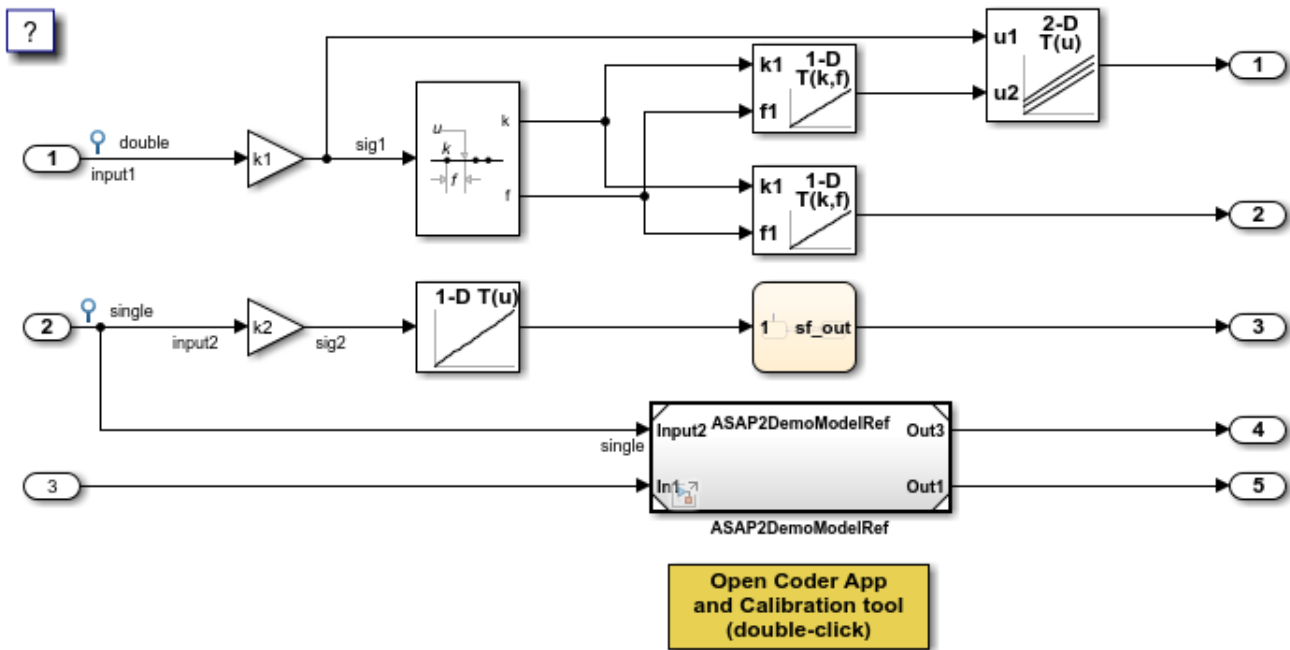
Create a Host-Based ASAM-ASAP2 Data Definition File for Data Measurement and Calibration

This model shows ASAP2 data export. ASAP2 is a data definition standard proposed by the Association for Standardization of Automation and Measuring Systems (ASAM).

Open Example Model

Open the example model ASAP2Demo.

```
open_system('ASAP2Demo');
```

Copyright 1994-2022 The MathWorks, Inc.

See the “ASAP2 File Generation” on page 36-3 to customize and generate the ASAP2 file.

See Also

`coder.cdf.export` | `coder.asap2.export` | **Generate Calibration Files**

More About

- “External Mode Simulation by Using XCP Communication” on page 32-8
- “Customize XCP Server Software” on page 32-45
- “Configure AUTOSAR Adaptive Data for Run-Time Calibration and Measurement” (AUTOSAR Blockset)

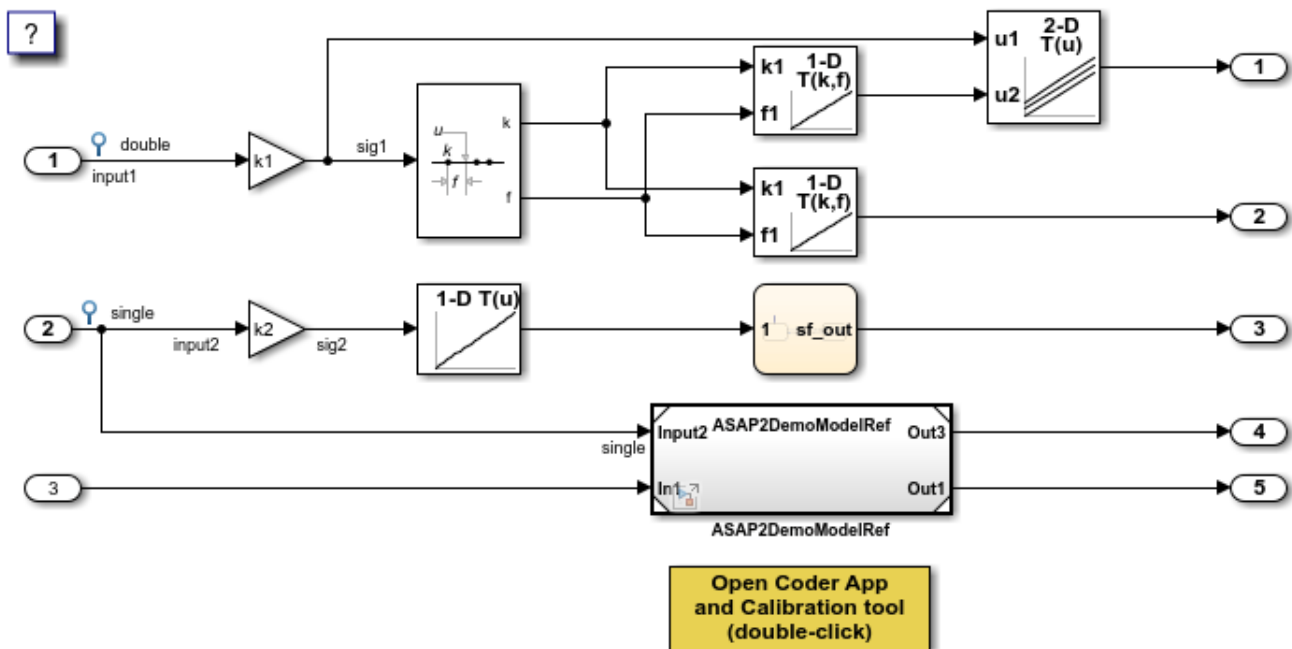
Create a Host-Based ASAM-ASAP2 Data Definition File for Data Measurement and Calibration

This model shows ASAP2 data export. ASAP2 is a data definition standard proposed by the Association for Standardization of Automation and Measuring Systems (ASAM).

Open Example Model

Open the example model ASAP2Demo.

```
open_system('ASAP2Demo');
```



Copyright 1994-2022 The MathWorks, Inc.

See the “ASAP2 File Generation” on page 36-3 to customize and generate the ASAP2 file.

Direct Memory Access to Generated Code for Simulink Coder

Access Memory in Generated Code Using Global Data Map

Simulink Coder provides a Target Language Compiler (TLC) function library that lets you create a global data map record. The global data map record, when generated, is added to the `CompiledModel` structure in the `model.rtw` file. The global data map record is a database containing information required for accessing memory in the generated code, including

- Signals (Block I/O)
- Parameters
- Data type work vectors (DWork)
- External inputs
- External outputs

Use of the global data map requires knowledge of the Target Language Compiler and of the structure of the `model.rtw` file. See “Target Language Compiler Overview” for information on these topics.

The TLC functions that are required to generate and access the global data map record are contained in `matlabroot/rtw/c/tlc/mw/globalmaplib.tlc`. The comments in the source code fully document the global data map structures and the library functions.

The global data map structures and functions might be modified or enhanced in future releases.

See Also

Related Examples

- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Customize Generated ASAP2 File

Customize Generated ASAP2 File

In this section...

“Customize Computation Method Names” on page 38-2
 “Export Characteristics and Measurement Objects in Groups” on page 38-3
 “Generate Record Layouts” on page 38-4
 “Exclude Structures and 64-Bit Integers” on page 38-4
 “Generate ASAP2 Containing Default Event List” on page 38-5
 “Specify ECU Address Extension” on page 38-6
 “Merge ASAP2 Files” on page 38-6

You can customize the fields of an ASAP2 file by using the code generator in any of these ways:

Customize Computation Method Names

In generated ASAP2 files, computation methods translate the electronic control unit (ECU) internal representation of measurement and calibration quantities into a physical model-oriented representation. Using Simulink Coder software, you can customize the names of computation methods. You can provide names that are more intuitive, enhancing ASAP2 file readability, or names that meet organizational requirements.

To customize computation method names, use the MATLAB function `getCompuMethodName`, which is defined in `matlabroot/toolbox/rtw/targets/asap2/asap2/user/getCompuMethodName.m`.

The `getCompuMethodName` function constructs a computation method name. The function prototype is:

```
cmName = getCompuMethodName(dataTypeName, cmUnits)
```

where

- *dataTypeName* is the name of the data type associated with the computation method.
- *cmUnits* are the units as specified in the Unit property of a `Simulink.Parameter` or `Simulink.Signal` object (for example, rpm or m/s).
- *cmName* returns the constructed computation method name.

The default constructed name returned by the function has the format:

```
<localPrefix><datatype>_<cmUnits>
```

where

- `<local_Prefix>` is a local prefix, `CM_`, defined in `matlabroot/toolbox/rtw/targets/asap2/asap2/user/getCompuMethodName.m`.
- `<datatype>` and `<cmUnits>` are the arguments that you specified to the `getCompuMethodName` function.

Special characters in *cmUnits* are not supported in the computation method name in ASAP2 file. The special characters are converted as shown in this table.

Special Character	Conversion
%	_percent_
-	_dash_
/	_per_
*	_into_
°	_deg_

For example, if you call the `getCompuMethodName` function by using the *dataTypeName* argument 'int16' and the *cmUnits* argument 'm/s', and then generate an ASAP2 file for a model named `myModel`, the computation method name appears in the generated file:

```
/begin COMPU_METHOD
  /* Name of CompuMethod */ myModel_CM_int16_m_per_s
  /* Units */ "m/s"
  ...
/end COMPU_METHOD
```

Export Characteristics and Measurement Objects in Groups

Starting in R2021b, you can group the characteristic and measurement objects in the ASAP2 file based on their properties. By default, the generated ASAP2 file contains GROUPS and SUB_GROUPS based on the subsystems present in the model.

- 1 Open the `sldemo_fuelsys` model.
- 2 Use **Embedded Coder Quick Start** app by entering into code perspective of the model and clicking **C Code > Quick Start**.
- 3 Build the model.
- 4 Generate the ASAP2 file by using the **Generate Calibration Files** tool or by using this command:

```
coder.asap2.export("sldemo_fuelsys")
```

The data elements are grouped based on the model architecture. The top model is referred to as **GROUP** and subsystems in the model are referred to as **SUB_GROUPS**. The generated file contains these lines:

```
/begin GROUP
  /* Name */ sldemo_fuelsys
  /* Long identifier */ "sldemo_fuelsys"
  /* Root */ ROOT
  /begin SUB_GROUP
    rtB.es_o
    sldemo_fuelsys_Dashboard
    sldemo_fuelsys_Engine_Gas_Dynamics
    sldemo_fuelsys_Throttle_Command
    sldemo_fuelsys_To_Controller
    sldemo_fuelsys_To_Plant
    sldemo_fuelsys_fuel_rate_control
  /end SUB_GROUP
/end GROUP
```

- To group the scalar objects into a group named GROUP_TYPE_SCALAR in the ASAP2 file, use this command:

```
coder.asap2.export("sldemo_fuelsys",CustomizeGroupsBy={'SCALAR'})
```

You can also group the objects of type array, and lookup tables into MAP, CURVE, CUBOID, CUBE_4, and CUBE_5 based on dimensions of the table.

Generate Record Layouts

- Open the sldemo_fuelsys model.

```
open_system("sldemo_fuelsys")
```

- Build the model.

- Generate the ASAP2 file by using this command:

```
coder.asap2.export("sldemo_fuelsys",IncludeAllRecordLayouts=true)
```

- The RecordLayouts.a2l file is generated according to the version of the ASAP2 file. In the ASAP2 file, verify this line:

```
/include "RecordLayouts.a2l"
```

- The generated RecordLayouts.a2l file contains the record layouts of the data types present in the model.

Note The RecordLayouts.a2l file is required to be in the same folder as the ASAP2 file.

- coder.asap2.export("sldemo_fuelsys",IncludeAllRecordLayouts=false) does not generate separate record layouts file, but includes the record layouts of the data types of exported data objects in the ASAP2 file.

```
/begin RECORD_LAYOUT Scalar_FLOAT32_IEEE
    FNC_VALUES 1 FLOAT32_IEEE COLUMN_DIR DIRECT
/end RECORD_LAYOUT

/begin RECORD_LAYOUT Scalar_FLOAT64_IEEE
    FNC_VALUES 1 FLOAT64_IEEE COLUMN_DIR DIRECT
/end RECORD_LAYOUT
...
```

Tip In the ASA2P file, to preserve the shape of a lookup table that is used as a model argument, make sure that the LookUp Table block is in the referenced model. Pass the LookUp table object from the top model as an instance parameter value.

Exclude Structures and 64-Bit Integers

Starting in R2022a, you can exclude the structure elements and 64-bit integers in the generated A2L file. By default, structures and 64-bit integers are included in the A2L file.

- Open the sldemo_fuelsys model.

```
open_system("sldemo_fuelsys")
```


- 2 Build the model.
- 3 Generate the ASAP2 file by using this command:

```
coder.asap2.export("sldemo_fuelsys", SupportStructureElements=false, ...
Support64bitIntegers=false)
```

The generated ASAP2 file does not have 64-bit integers and structure elements in it.

Note Starting in R2022a, the pointer variables will not be generated in the ASAP2 file.

Generate ASAP2 Containing Default Event List

Starting in R2022a, for a multirate model, the ASAP2 file contains the IF_DATA section that has the DEFAULT_EVENT_LIST, enabling you to view the respective raster values for the data elements in the calibration tools.

- 1 Open the sldemo_fuelsys model.

```
open_system("sldemo_fuelsys")
```

- 2 Build the model.
- 3 Generate the ASAP2 file by using Generate Calibration Files tool or by using this command:

```
coder.asap2.export("sldemo_fuelsys")
```

- 4 The model has a single rate of 0.01. The generated file defines measurements as:

```
/begin MEASUREMENT
/* Name */ rtB.es_o.throttle
/* Long identifier */ ""
/* Data type */ FLOAT32_IEEE
/* Conversion method */ sldemo_fuelsys_CM_single
/* Resolution (Not used) */ 0
/* Accuracy (Not used) */ 0
/* Lower limit */ -3.4E+38
/* Upper limit */ 3.4E+38
ECU_ADDRESS 0x0000 /* @ECU_Address@rtB.es_o.throttle@ */
/end MEASUREMENT
```

- 5 Convert the model into multirate model by using this command:

```
blockName = ['sldemo_fuelsys' '/EGO Fault Switch'];
set_param(blockName, 'SampleTime', '0.02');
```

- 6 Build the model.
- 7 Generate the ASAP2 file by using this command:

```
coder.asap2.export("sldemo_fuelsys")
```

- 8 The generated file now defines the measurements as:

```
/begin MEASUREMENT
/* Name */ rtB.es_o.throttle
/* Long identifier */ ""
/* Data type */ FLOAT32_IEEE
/* Conversion method */ sldemo_fuelsys_CM_single
/* Resolution (Not used) */ 0
/* Accuracy (Not used) */ 0
/* Lower limit */ -3.4E+38
/* Upper limit */ 3.4E+38
ECU_ADDRESS 0x0000 /* @ECU_Address@rtB.es_o.throttle@ */
/begin IF_DATA XCP
```

```

    /begin DAQ_EVENT VARIABLE
        /begin DEFAULT_EVENT_LIST
            EVENT 0x0001
        /end DEFAULT_EVENT_LIST
    /end DAQ_EVENT
/end IF_DATA
/end MEASUREMENT

```

- 9 To generate the ASAP2 file without DEFAULT_EVENT_LIST, use the command:

```
coder.asap2.export("sldemo_fuelsys", IncludeDefaultEventList=false);
```

Specify ECU Address Extension

Starting in R2022a, you can specify additional address information in the ASAP2 file by specifying an integer value for the EcuAddressExtension property. By default, no additional address information is provided.

- 1 Open the sldemo_fuelsys model.
- 2 Build the model.
- 3 Generate the ASAP2 file by using this command:

```
coder.asap2.export("sldemo_fuelsys", EcuAddressExtension=4)
```

The ECU_ADDRESS_EXTENSION key word for the characteristics, measurements, and axis points are in the ASAP2 file, enabling you to specify additional address information.

```

/begin MEASUREMENT
/* Name                */      rtB.MinMax
/* Long identifier     */      ""
/* Data type           */      FLOAT64_IEEE
/* Conversion method   */      sldemo_fuelsys_CM_double
/* Resolution (Not used) */      0
/* Accuracy (Not used) */      0
/* Lower limit         */      -1.7E+308
/* Upper limit         */      1.7E+308
ECU_ADDRESS            0x0000 /* @ECU_Address@rtB.MinMax@ */
ECU_ADDRESS_EXTENSION 4
/end MEASUREMENT

```

Merge ASAP2 Files

Starting in R2023a, you can merge the data elements of multiple A2L files to a Simulink model. Use the coder.asap2.merge function to create the merged descriptor object and generate an A2L file containing the data elements of the model and merged A2L files.

Vehicle Network Toolbox is required to merge A2L files using the coder.asap2.merge function.

- 1 Open the sldemo_fuelsys model.
- 2 Build the model.
- 3 Merge the data elements of an A2L file to the model using the command:

```
mergeObj = coder.asap2.merge("sldemo_fuelsys", {"ASAP2File1.a2l"})
```

To merge more than one A2L file, specify the files in comma separated format. For example:

```
mergeObj = coder.asap2.merge("sldemo_fuelsys",{ASAP2File1.a2l,"ASAP2File2.a2l"})
```

- 4 Generate the A2L file using the mergeObj.

```
coder.asap2.export("sldemo_fuelsys",CustomEcuDescriptions=mergeObj)
```

The generated A2L file contains the data elements present in sldemo_fuelsys model and the ASAP2File1 A2L file.

Note Lookup table objects will not be merged.

See Also

coder.asap2.export

More About

- “Generate ASAP2 and CDF Calibration Files” on page 36-2
- “External Mode Simulation by Using XCP Communication” on page 32-8

Create and Add Custom Data Elements

Generate an ASAP2 file by adding custom characteristics, measurements, axis-points, functions, record layouts, and compu methods. By default, an ASAP2 file contains the data elements present in model that were configured to export.

Using the Simulink Coder you can:

- Create and add custom data elements such as characteristics, measurements, axis-points, functions, record layouts, and compu methods.
- Remove data elements from the ASAP2 file.
- Filter and filter data elements based on their properties.
- Get or update the properties of data elements.

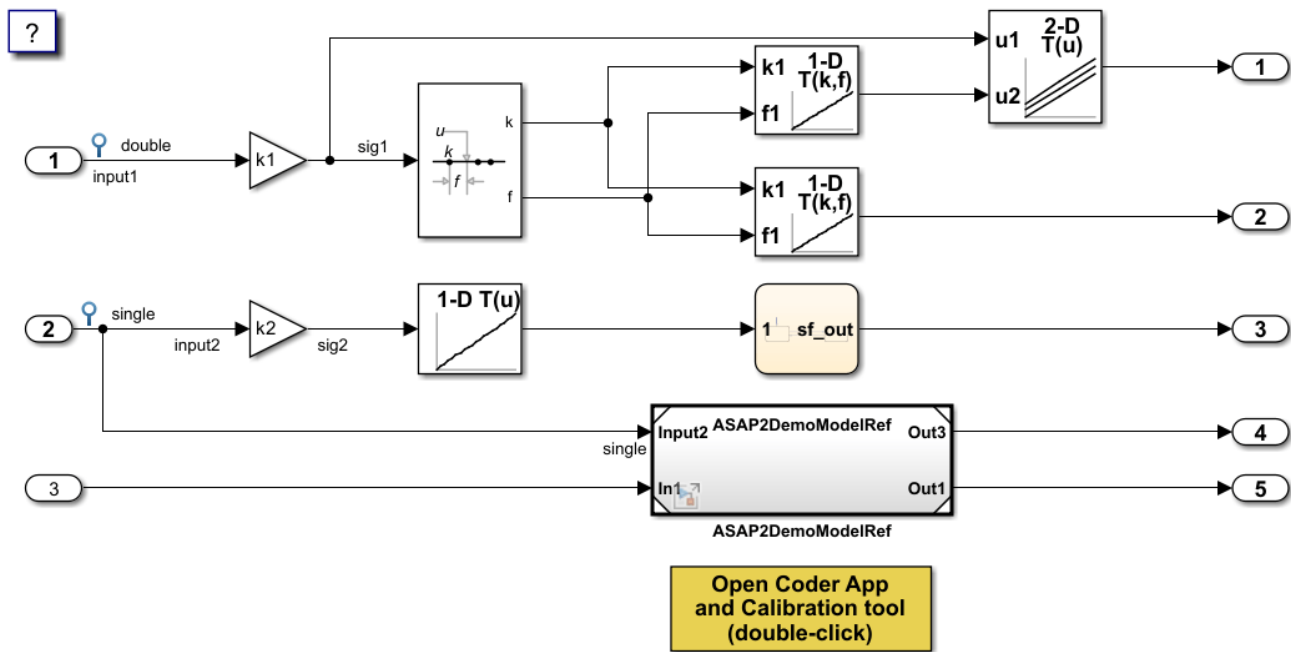
Customize an ASAP2 File

Add, update, filter, find, and remove ECU descriptions in an ASAP2 file.

Open and Build Example Model

Open the example model *ASAP2Demo*

```
open_system("ASAP2Demo");
```



Copyright 1994-2022 The MathWorks, Inc.

Build the model.

```
rtwbuild("ASAP2Demo");
```

```
### Starting serial model reference code generation build.
### Successfully updated the model reference code generation target for: ASAP2DemoModelRef
### Starting build procedure for: ASAP2Demo
### Successful completion of build procedure for: ASAP2Demo
```

Build Summary

Code generation targets built:

Model	Action	Rebuild Reason
ASAP2DemoModelRef	Code generated and compiled.	ASAP2DemoModelRef.c does not exist.

Top model targets built:

Model	Action	Rebuild Reason
-------	--------	----------------

```
=====
ASAP2Demo Code generated and compiled. Code generation information file does not exist.
```

```
2 of 2 models built (0 models already up to date)
Build duration: 0h 0m 30.188s
```

Create the ECU description object for the model.

```
descObj = coder.asap2.getEcuDescriptions("ASAP2Demo");
```

Get the list of available computation methods in the description object.

```
find(descObj, "CompuMethod")
```

```
ans = 1x10 string
      "ASAP2DemoModelRef_CM_double"      "ASAP2DemoModelRef_CM_int16_rpm"      "ASAP2Demo_CM_double"
```

Filter the computation methods and get a list of computation methods that use *rpm* as the unit.

```
find(descObj, "CompuMethod", Units='rpm')
```

```
ans = 1x3 string
      "ASAP2DemoModelRef_CM_int16_rpm"      "ASAP2Demo_CM_double_rpm"      "ASAP2Demo_CM_single_rpm"
```

Create and Add Custom Computation Method

To add a new computation method to the ASAP2 file, create a custom computation method.

```
CompuMethod_CM1 = coder.asap2.CompuMethod;
CompuMethod_CM1.Name = 'CompuMethod_1';
CompuMethod_CM1.ConversionType = 'LINEAR';
CompuMethod_CM1.Coefficients = [2 3];
CompuMethod_CM1.LongIdentifier = 'longIdentifierTest';
CompuMethod_CM1.Format = '%2.3';
CompuMethod_CM1.Units = 's';
```

Add the custom computation method to the ECU description object.

```
add(descObj, CompuMethod_CM1);
```

Get the properties of the newly added computation method.

```
get(descObj, "CompuMethod", "CompuMethod_1")
```

```
ans =
  CompuMethod with properties:
      Name: 'CompuMethod_1'
 LongIdentifier: 'longIdentifierTest'
      Format: '%2.3'
      Units: 's'
  Coefficients: [2 3]
  ConversionType: 'LINEAR'
 CompuVTabValues: [1x1 struct]
  CustomData: ""
```

To modify a property of the computation method, use *set* function. Update the *LongIdentifier* field of the computation method.

```
set(descObj, "CompuMethod", "CompuMethod_1", LongIdentifier="longIdentifierCM1")
```

Generate the ASAP2 file using the updated ECU description object and verify that the ASAP2 file contains the computation method *CompuMethod_1*.

```
coder.asap2.export("ASAP2Demo", CustomEcuDescriptions=descObj);
```

Following Characteristics or Measurements with unsupported data types are not exported in ASAP2

Delete Computation Method

Remove the newly added computation method from the description object.

```
delete(descObj, "CompuMethod", "CompuMethod_1");
```

Create and Add Custom Characteristic

To add a new characteristic to the ASAP2 file, create a custom characteristic.

```
Parameter_1 = coder.asap2.Characteristic;
Parameter_1.Name = 'Custom_parameter1';
Parameter_1.LongIdentifier = 'longIdentifierParam';
Parameter_1.UpperLimit = 255;
Parameter_1.LowerLimit = 0;
```

Add the custom characteristic to the ECU description object.

```
add(descObj, Parameter_1);
```

Get the properties of the newly added characteristic.

```
get(descObj, "Characteristic", "Custom_parameter1")
```

```
ans =
  Characteristic with properties:

      Name: 'Custom_parameter1'
 LongIdentifier: 'longIdentifierParam'
      Type: 'VALUE'
   EcuAddress: '0x0000'
 CompuMethodName: 'NO_COMPU_METHOD'
   LowerLimit: 0
   UpperLimit: 255
 EcuAddressComment: ""
 EcuAddressExtension: []
 CalibrationAccess: 'Calibration'
 DisplayIdentifier: ""
      Format: ""
      BitMask: []
      AxisInfo: []
 RecordLayout: ""
   Dimensions: []
      Export: 1
   MaxRefresh: [1x1 struct]
   SymbolLink: [1x1 struct]
```

```
CustomData: ""
```

To modify a property of the characteristic, use *set* function. Update the *UpperLimit* field of the characteristic.

```
set(descObj, "Characteristic", "Custom_parameter1", UpperLimit=128)
```

Generate the ASAP2 file using the updated ECU description object and verify that the ASAP2 file contains the characteristic *Custom_parameter1*.

```
coder.asap2.export("ASAP2Demo", CustomEcuDescriptions=descObj);
```

Following Characteristics or Measurements with unsupported data types are not exported in ASAP2 .

Remove the newly added characteristic from the description object.

```
delete(descObj, "Characteristic", "Custom_parameter1");
```

Create and Add Custom Measurement

To add a new measurement to the ASAP2 file, create a custom measurement.

```
Signal_1 = coder.asap2.Measurement;
Signal_1.Name = 'Custom_signal1';
Signal_1.LongIdentifier = 'longIdentifierSignal';
Signal_1.UpperLimit = 255;
Signal_1.LowerLimit = 0;
```

Add the custom measurement to the ECU description object.

```
add(descObj, Signal_1);
```

Get the properties of the newly added measurement.

```
get(descObj, "Measurement", "Custom_signal1")
```

```
ans =
  Measurement with properties:
      Name: 'Custom_signal1'
 LongIdentifier: 'longIdentifierSignal'
   DataType: 'UBYTE'
   EcuAddress: '0x0000'
 CompuMethodName: ''
   LowerLimit: 0
   UpperLimit: 255
      Raster: [1x1 struct]
 EcuAddressComment: ''
 EcuAddressExtension: []
 CalibrationAccess: 'NoCalibration'
 DisplayIdentifier: ''
      Format: ''
      BitMask: []
   Dimensions: []
      Export: 1
   MaskData: [1x1 struct]
   MaxRefresh: [1x1 struct]
   SymbolLink: [1x1 struct]
```



```
CustomData: ""
```

To modify a property of the measurement, use *set* function. Update the *CalibrationAccess* field of the measurement.

```
set(descObj, "Measurement", "Custom_signal1", CalibrationAccess='Calibration')
```

Generate the ASAP2 file using the updated ECU description object and verify that the ASAP2 file contains the measurement with name *Custom_signal1*.

```
coder.asap2.export("ASAP2Demo", CustomEcuDescriptions=descObj);
```

Following Characteristics or Measurements with unsupported data types are not exported in ASAP2

Remove the newly added measurement from the description object.

```
delete(descObj, "Measurement", "Custom_signal1");
```

Create and Add Lookup Table Parameter

Add a lookup table parameter.

```
LUT_Parameter = coder.asap2.Characteristic;
LUT_Parameter.Name = "custom_lookup_table";
LUT_Parameter.Type = "MAP";
```

Create and add axis information to the parameter.

```
axis_data = coder.asap2.AxisInfo;
axis_data(1).Name = 'BP3';
axis_data(2).Name = 'Bp4';
axis_data(1).CompuMethodName = 'ASAP2Demo_CM_double';
axis_data(2).CompuMethodName = 'ASAP2Demo_CM_double';
axis_data(1).MaxAxisPoints = '3';
axis_data(2).MaxAxisPoints = '3';
axis_data(1).AxisType = 'STD_AXIS';
axis_data(2).AxisType = 'STD_AXIS';
LUT_Parameter.AxisInfo = axis_data;
```

Add the lookup table parameter to the description object.

```
add(descObj, LUT_Parameter)
```

Generate the ASAP2 file using the updated ECU description object and verify that the ASAP2 file contains the lookup table parameter *custom_lookup_table*.

```
coder.asap2.export("ASAP2Demo", CustomEcuDescriptions=descObj);
```

Following Characteristics or Measurements with unsupported data types are not exported in ASAP2

See Also

[coder.asap2.export](#) | [coder.asap2.getEcuDescriptions](#) | [coder.asap2.Characteristic](#) | [coder.asap2.Measurement](#) | [coder.asap2.CompuMethod](#) | [coder.asap2.Function](#) | [coder.asap2.RecordLayout](#)

Related Examples

- “Generate ASAP2 and CDF Calibration Files” on page 36-2

Customize Using Base Files

You can add custom A2L file fragments such as PROJECT, MODULE, MOD, MOD_PAR, and MOD_COMMON settings to the ASAP2 file. You can do it in the following ways:

Customize ASAP2 Fields by Using Base Class

- 1 Create an object from the base class using the `coder.asap2.UserCustomizeBase` function.

```
obj = coder.asap2.UserCustomizeBase;
```

- 2 Define the object fields.

```
obj.HeaderComment = 'Header comment';
obj.ModParComment = 'Mod Par comment';
obj.ModCommonComment = 'Mod Common comment';
obj.ASAP2FileName = 'File_name';
obj.ByteOrder = 'BYTEORDER MSB_LAST';
```

- 3 Pass the object to the function and generate the ASAP2 file.

```
coder.asap2.export(modelName,CustomizationObject=obj);
```

Customize ASAP2 Sections by Deriving from Base Class

- 1 Derive from the base class `coder.asap2.UserCustomizeBase`. You can place the derived file anywhere on the MATLAB path.
- 2 To edit the Header section, use the `writeHeader` function.
- 3 Use the `writeHardwareInterface` function to edit the hardware interface section consisting of the MOD_PAR and MOD_COMMON fields of the ASAP2 file.
- 4 To add text at the beginning of the ASAP2 file, use the `writeFileHead` function.
- 5 To add text at the end of the ASAP2 file, use the `writeFileTail` function.

For example, the code for generating a derived class `TestASAP2Customization` might look like this code:

```
classdef TestASAP2Customization < coder.asap2.UserCustomizeBase
    % Customization class for asap2 file
    methods
        function header = writeHeader(obj)
            header = sprintf([' /begin HEADER ' 'test Header text' '\n',...
                'header test comments from user\n',...
                ' /end HEADER']);
        end

        function hardwareInterface = writeHardwareInterface(obj)
            hardwareInterface = sprintf([' /begin MOD_PAR "' 'UserDefined' '"\n',...
                ' test comments from user\n',...
                '/end MOD_PAR\n\n',...
                '/begin MOD_COMMON "' 'UserDefined MOD_COMMON' '"\n',...
                ' user defined values for MOD_COMMON\n',...
                ' obj.ByteOrder '\n',...
                '/end MOD_COMMON']);
        end

        function fileTail = writeFileTail(obj)
            fileTail = sprintf(['/* This File can be used for ' ,...
                'calibration .*/\n'...
                '/* EOF ',obj.ASAP2FileName, '*/']);
        end
    end
end
```

```

function fileHeader = writeFileHead(obj)
    fileHeader = sprintf(['/*****\n
        ' This is a a2l file which can be used for calibration with INCA or CANAPE\n',...
        ' *****/']);
    end
end
end

```

- 6 Create an object from the derived class. For example, this command creates an object from `TestASAP2Customization`.

```
obj = TestASAP2Customization;
```

- 7 Pass the object to the function that generates the ASAP2 file.

```
coder.asap2.export(modelName,CustomizationObject=obj);
```

See Also

`coder.asap2.export` | `coder.asap2.UserCustomizeBase`

More About

- “Generate ASAP2 and CDF Calibration Files” on page 36-2
- “External Mode Simulation by Using XCP Communication” on page 32-8

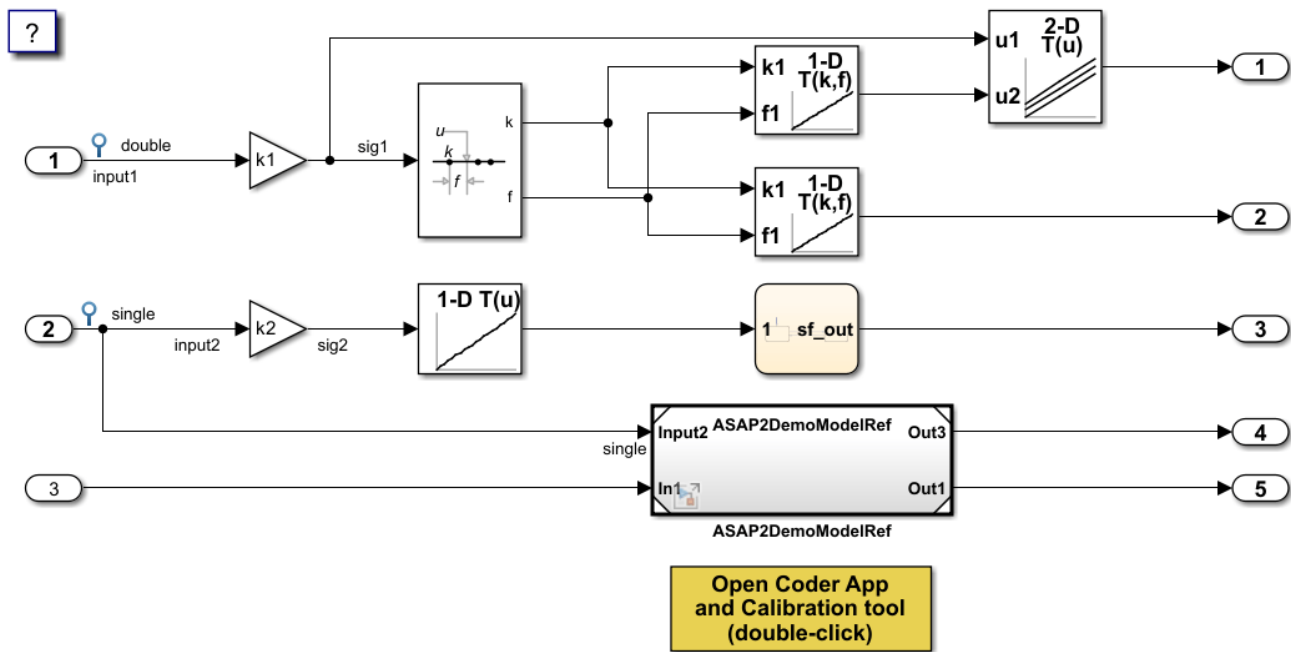
Customize an ASAP2 File

Add, update, filter, find, and remove ECU descriptions in an ASAP2 file.

Open and Build Example Model

Open the example model *ASAP2Demo*

```
open_system("ASAP2Demo");
```



Copyright 1994-2022 The MathWorks, Inc.

Build the model.

```
rtwbuild("ASAP2Demo");
```

```
### Starting serial model reference code generation build.
### Successfully updated the model reference code generation target for: ASAP2DemoModelRef
### Starting build procedure for: ASAP2Demo
### Successful completion of build procedure for: ASAP2Demo
```

Build Summary

Code generation targets built:

Model	Action	Rebuild Reason
ASAP2DemoModelRef	Code generated and compiled.	ASAP2DemoModelRef.c does not exist.

Top model targets built:

Model	Action	Rebuild Reason
-------	--------	----------------

```
=====
ASAP2Demo Code generated and compiled. Code generation information file does not exist.
```

```
2 of 2 models built (0 models already up to date)
Build duration: 0h 0m 30.188s
```

Create the ECU description object for the model.

```
descObj = coder.asap2.getEcuDescriptions("ASAP2Demo");
```

Get the list of available computation methods in the description object.

```
find(descObj, "CompuMethod")
```

```
ans = 1x10 string
      "ASAP2DemoModelRef_CM_double"      "ASAP2DemoModelRef_CM_int16_rpm"      "ASAP2Demo_CM_double"
```

Filter the computation methods and get a list of computation methods that use *rpm* as the unit.

```
find(descObj, "CompuMethod", Units='rpm')
```

```
ans = 1x3 string
      "ASAP2DemoModelRef_CM_int16_rpm"      "ASAP2Demo_CM_double_rpm"      "ASAP2Demo_CM_single_rpm"
```

Create and Add Custom Computation Method

To add a new computation method to the ASAP2 file, create a custom computation method.

```
CompuMethod_CM1 = coder.asap2.CompuMethod;
CompuMethod_CM1.Name = 'CompuMethod_1';
CompuMethod_CM1.ConversionType = 'LINEAR';
CompuMethod_CM1.Coefficients = [2 3];
CompuMethod_CM1.LongIdentifier = 'longIdentifierTest';
CompuMethod_CM1.Format = '%2.3';
CompuMethod_CM1.Units = 's';
```

Add the custom computation method to the ECU description object.

```
add(descObj, CompuMethod_CM1);
```

Get the properties of the newly added computation method.

```
get(descObj, "CompuMethod", "CompuMethod_1")
```

```
ans =
  CompuMethod with properties:
      Name: 'CompuMethod_1'
 LongIdentifier: 'longIdentifierTest'
      Format: '%2.3'
      Units: 's'
  Coefficients: [2 3]
 ConversionType: 'LINEAR'
 CompuVTabValues: [1x1 struct]
  CustomData: ""
```

To modify a property of the computation method, use *set* function. Update the *LongIdentifier* field of the computation method.

```
set(descObj, "CompuMethod", "CompuMethod_1", LongIdentifier="longIdentifierCM1")
```

Generate the ASAP2 file using the updated ECU description object and verify that the ASAP2 file contains the computation method *CompuMethod_1*.

```
coder.asap2.export("ASAP2Demo", CustomEcuDescriptions=descObj);
```

Following Characteristics or Measurements with unsupported data types are not exported in ASAP2

Delete Computation Method

Remove the newly added computation method from the description object.

```
delete(descObj, "CompuMethod", "CompuMethod_1");
```

Create and Add Custom Characteristic

To add a new characteristic to the ASAP2 file, create a custom characteristic.

```
Parameter_1 = coder.asap2.Characteristic;
Parameter_1.Name = 'Custom_parameter1';
Parameter_1.LongIdentifier = 'longIdentifierParam';
Parameter_1.UpperLimit = 255;
Parameter_1.LowerLimit = 0;
```

Add the custom characteristic to the ECU description object.

```
add(descObj, Parameter_1);
```

Get the properties of the newly added characteristic.

```
get(descObj, "Characteristic", "Custom_parameter1")
```

```
ans =
  Characteristic with properties:
      Name: 'Custom_parameter1'
 LongIdentifier: 'longIdentifierParam'
      Type: 'VALUE'
      EcuAddress: '0x0000'
 CompuMethodName: 'NO_COMPU_METHOD'
      LowerLimit: 0
      UpperLimit: 255
 EcuAddressComment: ''
 EcuAddressExtension: []
 CalibrationAccess: 'Calibration'
 DisplayIdentifier: ''
      Format: ''
      BitMask: []
      AxisInfo: []
 RecordLayout: ''
      Dimensions: []
      Export: 1
 MaxRefresh: [1x1 struct]
 SymbolLink: [1x1 struct]
```

```
CustomData: ""
```

To modify a property of the characteristic, use *set* function. Update the *UpperLimit* field of the characteristic.

```
set(descObj, "Characteristic", "Custom_parameter1", UpperLimit=128)
```

Generate the ASAP2 file using the updated ECU description object and verify that the ASAP2 file contains the characteristic *Custom_parameter1*.

```
coder.asap2.export("ASAP2Demo", CustomEcuDescriptions=descObj);
```

Following Characteristics or Measurements with unsupported data types are not exported in ASAP2 .

Remove the newly added characteristic from the description object.

```
delete(descObj, "Characteristic", "Custom_parameter1");
```

Create and Add Custom Measurement

To add a new measurement to the ASAP2 file, create a custom measurement.

```
Signal_1 = coder.asap2.Measurement;
Signal_1.Name = 'Custom_signal1';
Signal_1.LongIdentifier = 'longIdentifierSignal';
Signal_1.UpperLimit = 255;
Signal_1.LowerLimit = 0;
```

Add the custom measurement to the ECU description object.

```
add(descObj, Signal_1);
```

Get the properties of the newly added measurement.

```
get(descObj, "Measurement", "Custom_signal1")
```

```
ans =
  Measurement with properties:
      Name: 'Custom_signal1'
 LongIdentifier: 'longIdentifierSignal'
      DataType: 'UBYTE'
      EcuAddress: '0x0000'
 CompuMethodName: ''
      LowerLimit: 0
      UpperLimit: 255
      Raster: [1x1 struct]
 EcuAddressComment: ''
 EcuAddressExtension: []
 CalibrationAccess: 'NoCalibration'
 DisplayIdentifier: ''
      Format: ''
      BitMask: []
      Dimensions: []
      Export: 1
      MaskData: [1x1 struct]
      MaxRefresh: [1x1 struct]
      SymbolLink: [1x1 struct]
```



```
CustomData: ""
```

To modify a property of the measurement, use *set* function. Update the *CalibrationAccess* field of the measurement.

```
set(descObj, "Measurement", "Custom_signal1", CalibrationAccess='Calibration')
```

Generate the ASAP2 file using the updated ECU description object and verify that the ASAP2 file contains the measurement with name *Custom_signal1*.

```
coder.asap2.export("ASAP2Demo", CustomEcuDescriptions=descObj);
```

Following Characteristics or Measurements with unsupported data types are not exported in ASAP2

Remove the newly added measurement from the description object.

```
delete(descObj, "Measurement", "Custom_signal1");
```

Create and Add Lookup Table Parameter

Add a lookup table parameter.

```
LUT_Parameter = coder.asap2.Characteristic;
LUT_Parameter.Name = "custom_lookup_table";
LUT_Parameter.Type = "MAP";
```

Create and add axis information to the parameter.

```
axis_data = coder.asap2.AxisInfo;
axis_data(1).Name = 'BP3';
axis_data(2).Name = 'Bp4';
axis_data(1).CompuMethodName = 'ASAP2Demo_CM_double';
axis_data(2).CompuMethodName = 'ASAP2Demo_CM_double';
axis_data(1).MaxAxisPoints = '3';
axis_data(2).MaxAxisPoints = '3';
axis_data(1).AxisType = 'STD_AXIS';
axis_data(2).AxisType = 'STD_AXIS';
LUT_Parameter.AxisInfo = axis_data;
```

Add the lookup table parameter to the description object.

```
add(descObj, LUT_Parameter)
```

Generate the ASAP2 file using the updated ECU description object and verify that the ASAP2 file contains the lookup table parameter *custom_lookup_table*.

```
coder.asap2.export("ASAP2Demo", CustomEcuDescriptions=descObj);
```

Following Characteristics or Measurements with unsupported data types are not exported in ASAP2

Configure Model Data Elements for ASAP2 File Generation

Configure Model Data Elements for ASAP2 File Generation

When you generate an ASAP2 file from a model, the ASAP2 file includes information about the data elements in your model such as characteristics and measurement variables. Each data element is described by information such as the memory address, storage structure, data types and conversion rules. In Simulink you can now customize how the data elements are described in an ASAP2 file. You can choose only those characteristics and measurements that you are interested in exporting into ASAP2 file.


To configure data elements for ASAP2 file generation in the **Code Mappings Editor - C** display, use the tabs:

- Inports
- Outports
- Parameters
- Data Stores
- Signals/States

View Calibration Attributes

- 1 Open a model with multiple data elements.
`open_system('rtwdemo_configrpinterface')`
- 2 Open the Simulink Coder app. On the **C Code** tab, select **Code Interface + Individual Element Code Mappings**.

Configure Inport for ASAP2 File Generation

- 1 On the **Inports** tab, select an inport and click the  icon.
- 2 Specify the measurement property fields for the inport.

Copyright 1994-2020 The MathWorks

Source	Storage Class
In1	ImportedExtern
In2	Model default: ImportedExternPointer
In3	Model default: ImportedExternPointer

3 Alternatively, at the command line, you can enter these commands:

```
setInport(cm, 'In1', 'CalibrationAccess', 'Calibration');
setInport(cm, 'In1', 'CompuMethod', 'CM_Inport');
setInport(cm, 'In1', 'DisplayIdentifier', 'inport_display_name');
setInport(cm, 'In1', 'Format', '%4.2');
```

4 Build the model.

5 Generate the ASAP2 file by using this command:

```
coder.asap2.export('rtwdemo_configrinterface');
```

- 6 Verify the exported inport in the ASAP2 file and verify the fields that you specified are present.

```


/begin MEASUREMENT
/* Name */ input1
/* Long identifier */ ""
/* Data type */ FLOAT64_IEEE
/* Conversion method */ CM_Inport
/* Resolution (Not used) */ 0
/* Accuracy (Not used) */ 0
/* Lower limit */ -20
/* Upper limit */ 20
ECU_ADDRESS 0x0000 /* @ECU_Address@input1@ */
DISPLAY_IDENTIFIER inport_display_name
FORMAT "%4.2"
READ_WRITE
/end MEASUREMENT

```

You can configure an outport in the same way. By default, all the inports and outports are exported to A2L file. To avoid exporting an inport or an outport to A2L file, set the `Export` property to false using the command:

```
setInport(cm, 'In1', 'Export', false);
```

Configure Parameter for ASAP2 File Generation

- 1 On the **Parameters** tab, select a parameter and click the  icon.
- 2 Specify the calibration property fields for the parameter.

Copyright 1994-2020 The MathWorks, Inc.

Code Mappings - C

Source	Storage Class
External Parameter Objects...	
Model Parameter Arguments (2)	
LOWER	Auto

3 Alternatively, at the command line, you can enter these commands:

```
setModelParameter(cm, 'LOWER', 'CalibrationAccess', 'NoCalibration');
setModelParameter(cm, 'LOWER', 'CompuMethod', 'CM_param');
setModelParameter(cm, 'LOWER', 'DisplayIdentifier', 'param_lower');
setModelParameter(cm, 'LOWER', 'Format', '%4.2');
```

4 Build the model.

5 Generate the ASAP2 file by using this command:

```
coder.asap2.export('rtwdemo_configrinterface');
```

- 6 Verify the exported parameter in the ASAP2 file and verify the fields that you specified are present.

```

/begin CHARACTERISTIC
/* Name          */      rtwdemo_configrpinterface_InstP.LOWER
/* Long Identifier */      ""
/* Type          */      VALUE
/* ECU Address   */      0x0000 /* @ECU_Address@rtwdemo_configrpinterface_InstP.LOWER@ */
/* Record Layout */      Scalar_FLOAT64_IEEE
/* Maximum Difference */  0
/* Conversion Method */   CM_param
/* Lower Limit   */      -20.000000
/* Upper Limit   */      0.000000
CALIBRATION_ACCESS NO_CALIBRATION
DISPLAY_IDENTIFIER param_lower
FORMAT "%4.2"
/end CHARACTERISTIC

```

You can configure data stores, states in the same way. By default, signals, states, and data stores are exported to A2L file. To avoid exporting a data element to A2L file, set the Export property to false using the command:

```
setModelParameter(cm, 'LOWER', 'Export', false);
```

Note To export signals, states, data stores, and parameters to the A2L file, configure the data element to a non-auto storage class.

See Also

setSignal | setInport | setOutport | setModelParameter | setDataStore | setState | coder.asap2.export | coder.mapping.api.CodeMapping | **Code Mappings Editor - C**

More About

- “Generate ASAP2 and CDF Calibration Files” on page 36-2
- “Customize Generated ASAP2 File” on page 38-2
- “C Code Generation Configuration for Model Interface Elements” on page 16-2
- “Choose Storage Class for Controlling Data Representation in Generated Code” on page 15-75
- “Control Data and Function Interface in Generated Code” on page 15-44

Performance

Optimizations for Generated Code in Simulink Coder

- “Increase Code Generation Speed” on page 40-3
- “Control Compiler Optimizations” on page 40-6
- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Control Memory Allocation for Time Counters” on page 40-10
- “Execution Profiling for Generated Code” on page 40-11
- “Optimize Generated Code by Combining Multiple for Constructs” on page 40-13
- “Subnormal Number Execution Speed” on page 40-16
- “Remove Code for Out-of-Range Floating Point to Integer Conversions” on page 40-20
- “Remove Code That Maps NaN to Integer Zero” on page 40-23
- “Disable Nonfinite Checks or Inlining for Math Functions” on page 40-26
- “Fold Expressions” on page 40-31
- “Minimize Computations and Storage for Intermediate Results at Block Outputs” on page 40-35
- “Inline Invariant Signals” on page 40-38
- “Inline Numeric Values of Block Parameters” on page 40-41
- “Configure Loop Unrolling Threshold” on page 40-46
- “Use memcpy Function to Optimize Generated Code for Vector Assignments” on page 40-48
- “Generate Target Optimizations Within Algorithm Code” on page 40-51
- “Remove Code for Blocks That Have No Effect on Computational Results” on page 40-52
- “Eliminate Dead Code Paths in Generated Code” on page 40-56
- “Floating-Point Multiplication to Handle a Net Slope Correction” on page 40-59
- “Use Conditional Input Branch Execution” on page 40-62
- “Optimize Generated Code for Complex Signals” on page 40-66
- “Speed Up Linear Algebra in Code Generated from a MATLAB Function Block” on page 40-68
- “Speed Up Matrix Operations in Code Generated from a MATLAB Function Block” on page 40-71
- “Speed Up Fast Fourier Transforms in Code Generated from a MATLAB Function Block” on page 40-75
- “Synchronize Multithreaded FFTW Planning in Code Generated from a MATLAB Function Block” on page 40-78
- “Control Memory Allocation for Variable-Size Arrays in a MATLAB Function Block” on page 40-82
- “Optimize Memory Usage for Time Counters” on page 40-84
- “Optimize Generated Code Using Boolean Data for Logical Signals” on page 40-89
- “Reduce Memory Usage for Boolean and State Configuration Variables” on page 40-92
- “Customize Stack Space Allocation” on page 40-93
- “Optimize Generated Code Using memset Function” on page 40-95

- “Vector Operation Optimization” on page 40-99
- “Enable and Reuse Local Block Outputs in Generated Code” on page 40-102
- “Generate SIMD Code from Simulink Blocks” on page 40-105
- “Optimize Code for Reduction Operations by Using SIMD” on page 40-110

Increase Code Generation Speed

In this section...

“Build a Model in Increments” on page 40-3

“Build Large Model Reference Hierarchies in Parallel” on page 40-3

“Minimize Memory Requirements During Code Generation” on page 40-4

“Generate Only Code” on page 40-4

“Suppress Creation of a Code Generation Report” on page 40-4

The amount of time it takes to generate code for a model depends on the size and configuration settings of the model. For instance, if you are working with a large model, it can take awhile to generate code. To decrease the amount of time for code generation of a model, try one or more of the following methods:

- Build a model in increments
- Build large model reference hierarchies in parallel
- Minimize memory requirements during code generation
- Generate only code
- Disable the creation of a code generation report

Build a Model in Increments

You can use the `slbuild` command to build a model and generate code. By default, when rebuilding a model, `slbuild` provides an incremental model build, which only rebuilds a model or submodels that have changed since the most recent model build. Incremental model build saves code generation time. Use the **Rebuild** parameter on the **Model Referencing** pane to change the method that Simulink uses to determine when to rebuild code for referenced models. For more information on the **Rebuild** parameter, see [Rebuild](#).

Build Large Model Reference Hierarchies in Parallel

In a parallel computing environment, whenever conditions allow, you can increase the speed of code generation and compilation by building models containing large model reference hierarchies in parallel. For example, if you have Parallel Computing Toolbox software, you can distribute code generation and compilation for each referenced model across the cores of a multicore host computer. If you have MATLAB Parallel Server software, you can distribute code generation and compilation for each referenced model across remote workers in your MATLAB Parallel Server configuration.

The execution speed improvement realized by using parallel builds for referenced models depends on several factors, including:

- How many models can be built in parallel for a given model referencing hierarchy
- The size of the referenced models
- Parallel computing resources such as the number of local and remote workers available
- The hardware attributes of the local and remote machines (amount of RAM, number of cores, and so on)

For more information, see “Reduce Build Time for Referenced Models by Using Parallel Builds” on page 31-51.

Minimize Memory Requirements During Code Generation

Models that have large amounts of parameter and constant data (such as lookup tables) can tax memory resources and slow code generation. The code generator copies this data to the *model.rtw* file. The *model.rtw* file is a partial representation of the model that the Target Language Compiler parses to transform block computations, parameters, signals, and constant data into a high-level language (for example, C). The Target Language Compiler (TLC) is an integral part of the code generator. The code generator copies parameters and data into *model.rtw*, whether they originate in the model or come from variables or objects in a workspace.

You can improve code generation speed by specifying the maximum number of elements that data vectors can have for the code generator to copy this data to *model.rtw*. When a data vector exceeds the specified size, the code generator places a reference key in *model.rtw*. The TLC uses this key to access the data from Simulink and format it into the generated code. Reference keys result in maintaining only one copy of large data vectors in memory.

The default value above which the code generator uses reference keys in place of actual data values is 10 elements. You can verify this value. In the Command Window, type the following command:

```
get_param(0, 'RTWDataReferencesMinSize')
```

To set the threshold to a different value, in the Command Window, type the following `set_param` function:

```
set_param(0, 'RTWDataReferencesMinSize', <size>)
```

Provide an integer value for `size` that specifies the number of data elements above which the code generator uses reference keys in place of actual data values.

Generate Only Code

You can increase code generation speed by specifying that the build process generate code and a makefile, but not invoke the make command. When the code generator invokes the make command, the build process takes longer because the code generator generates code, compiles code, and creates an executable file.

On the **Code Generation** pane in the Model Configuration Parameters dialog box, you can specify that the build process generate only code by selecting the **Generate code only** parameter. You can specify that the code generation process build a makefile by selecting the **Configuration Parameters > Code Generation > Build process > Makefile configuration > Generate makefile** parameter.

Suppress Creation of a Code Generation Report

You can speed up code generation by not generating a code generation report as a part of the build process. To disable the creation of a code generation report, on the **Code Generation > Report** pane, clear the **Create code generation report** parameter. To generate a code generation report after the build process, see “Generate Code Generation Report” (Embedded Coder).

See Also

Related Examples

- Enable parallel model reference builds
- MATLAB worker initialization for builds
- “Reduce Build Time for Referenced Models by Using Parallel Builds” on page 31-51

Control Compiler Optimizations

To control compiler optimizations for a makefile build at the GUI level, use the **Compiler optimization level** parameter. The **Compiler optimization level** parameter provides

- Target-independent values `Optimizations on (faster runs)` and `Optimizations off (faster builds)`, which allow you to easily toggle compiler optimizations on and off during code development
- The value `Custom` for entering custom compiler optimization flags at the Simulink GUI level, rather than editing compiler flags into template makefiles (TMFs) or supplying compiler flags to build process make commands

The default setting is `Optimizations off (faster builds)`. Selecting the value `Custom` enables the **Custom compiler optimization flags** field, in which you can enter custom compiler optimization flags (for example, `-O2`).

If you specify compiler options for your makefile build using `OPT_OPTS`, `MEX_OPTS` (except `MEX_OPTS="-v"`), or `MEX_OPT_FILE`, the value of **Compiler optimization level** is ignored and a warning is issued about the ignored parameter.

For a rapid simulation on page 29-2 that uses the MinGW compiler:

- If `RTWCompilerOptimization` is set to `'on'`, the build process customizes compiler optimization to minimize run time. The build process ignores the `BuildConfiguration` and `CustomToolchainOptions` settings.
- If `RTWCompilerOptimization` is set to `'custom'` or `'off'`, the build process uses `BuildConfiguration` and `CustomToolchainOptions` settings. The build process ignores the `RTWCompilerOptimization` setting.

See Also

Related Examples

- “Template Makefiles and Make Options” on page 31-36
- “Configure a System Target File” on page 21-2
- “Support Compiler Optimization Level Control” on page 46-65

Design Techniques to Optimize Models for Efficient Code Generation

Use the Model Advisor to Optimize a Model

You can use the Model Advisor to analyze a model for code generation and identify aspects of your model that impede production deployment or limit code efficiency. You can select from a set of checks to run on a model's current configuration. The Model Advisor analyzes the model and generates check results providing suggestions for improvements in each area. Most Model Advisor diagnostics do not require the model to be in a compiled state; those that do are noted.

Before running the Model Advisor, select the target you plan to use for code generation. The Model Advisor works most effectively with ERT and ERT-based system target files.

Use the following examples to investigate optimizing models for code generation using the Model Advisor:

- `rtwdemo_advisor1`
- `rtwdemo_advisor2`
- `rtwdemo_advisor3`

Note Example models `rtwdemo_advisor2` and `rtwdemo_advisor3` require Stateflow and Fixed-Point Designer software.

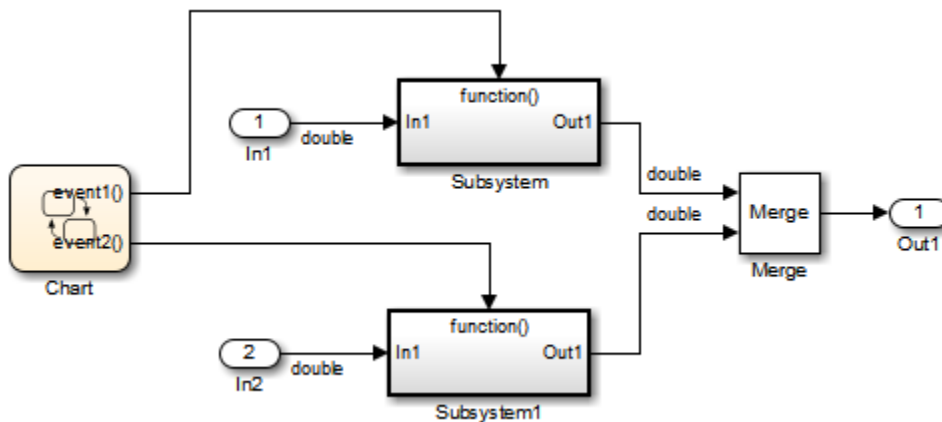
For more information on using the Model Advisor, see “Run Model Advisor Checks”. For more information about the checks, see “Model Advisor Checks”.

Design Techniques to Optimize a Model for Code Generation

You can apply the following techniques to optimize a model for code generation:

- For Embedded Coder users, if your application uses only integer arithmetic, clear the **Support floating-point numbers** parameter in the **Software environment** section of the **Interface** pane so that the generated code does not contain floating-point data or operations. When this parameter is cleared, an error is raised if noninteger data or expressions are encountered during code generation. The error message reports the offending blocks and parameters.
- Disable the **Configuration Parameters > Code Generation > Interface > Advanced parameters > MAT-file logging** parameter. Deselecting this parameter eliminates extra code and memory usage for initializing, updating, and cleaning-up logging variables. In addition, the code generated to support MAT-file logging invokes `malloc`, which can be undesirable for your application.
- Use the Upgrade Advisor to upgrade older models (saved by prior versions or the current version) to use current features. For details, see “Model Upgrades”.
- Before building, set optimization flags for the compiler (for example, `-O2` for `gcc`, `-Ot` for the Microsoft Visual C++ compiler).

- Directly inline C/C++ S-functions into the generated code by writing a TLC file for the S-function. For more information, see “Use S-Function Target for Model or Subsystem” on page 29-40 and see “Inline C MEX S-Functions”.
- Use a Simulink data type other than `double` when possible. The available data types are `Boolean`, signed and unsigned 8-, 16-, and 32-bit integers, and 32- and 64-bit floats (a `double` is a 64-bit float). For more information, see “About Data Types in Simulink”. For a block-by-block summary, click `showblockdatatypetable` or type the command in the Command Window.
- For tunable block parameters that you configure to store in memory in the generated code, you can match parameter data types with signal data types to eliminate unnecessary typecasts and C shifts. Where possible, store parameter values in small integer data types. See “Parameter Data Types in the Generated Code” on page 15-131.
- Remove repeated values in lookup table data.
- Use the Merge block to merge the output of signals wherever possible. This block is particularly helpful when you need to control the execution of function-call subsystems with a Stateflow chart. The following model shows an example of how to use the Merge block.



When more than one signal connected to a Merge block has a non-Auto storage class, all non-Auto signals connected to that block must *be identically labeled* and *have the same storage class*. When Merge blocks connect directly to one another, these rules apply to the signals connected to any of the Merge blocks in the group.

Design Techniques to Optimize Stateflow Objects for Code Generation

Be Explicit About the Inline Option of a Graphical Function

When you use a graphical function in a Stateflow chart, select `Inline` or `Function` for the property **Function Inline Option**. Otherwise, the code generated for a graphical function may not appear as you want. For more information, see “Specify Properties of Graphical Functions” (Stateflow).

Avoid Using Multiple Edge-Triggered Events in Stateflow Charts

If you use more than one trigger, you generate multiple code statements to handle rising or falling edge detections. If multiple triggers are required, use function-call events instead. For more information, see “Activate a Stateflow Chart by Sending Input Events” (Stateflow).

Combine Input Signals of a Chart Into a Single Bus Object

When you use a bus object, you reduce the number of parameters in the parameter list of a generated function. This guideline also applies to output signals of a chart. For more information, see “Define Stateflow Structures” (Stateflow).

Use Charts with Discrete Sample Times

The code generated for discrete charts that are not inside a triggered or enabled subsystem uses integer counters to track time instead of Simulink provided time. This allows for more efficient code generation in terms of overhead and memory, as well as enabling this code for use in software-in-the-loop (SIL) and processor-in-the-loop (PIL) simulations.

See Also

Related Examples

- “Increase Code Generation Speed” on page 40-3
- “Execution Profiling for Generated Code” on page 40-11
- “Model Configuration Parameters: Code Generation Optimization”
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Control Memory Allocation for Time Counters

The **Application lifespan (days)** parameter lets you control the allocation of memory for absolute and elapsed time counters. Such counters exist in the code for blocks that use absolute or elapsed time. For a list of such blocks, see “Absolute Time Limitations” on page 4-14.

The size of the time counters in generated code is 8, 16, 32, or 64 bits. The size is set automatically to the minimum that can accommodate the duration value specified by **Application lifespan (days)** given the step size specified in the Configuration Parameters **Solver** pane. To minimize the amount of RAM used by time counters, specify the smallest lifespan possible and the largest step size possible.

An application runs to its specified lifespan. It may be able to run longer. For example, running a model with a step size of one millisecond (0.001 seconds) for one day requires a 32-bit timer, which could continue running without overflow for 49 days more.

To maximize application lifespan, specify **Application lifespan (days)** as `inf`. This value allocates 64 bits (two `uint32` words) for each timer. Using 64 bits to store timing data would allow a model with a step size of 0.001 microsecond (10E-09 seconds) to run for more than 500 years, which would rarely be required. 64-bit counters do not violate the usual code generator length limitation of 32 bits because the value of a time counter does not provide the value of a signal, state, or parameter.

See Also

“Application lifespan (days)”

Related Examples

- “Absolute and Elapsed Time Computation” on page 4-2
- “Timers in Asynchronous Tasks” on page 6-38

Execution Profiling for Generated Code

Use code execution profiling to:

- Determine whether the generated code meets execution time requirements for real-time deployment on your target hardware.
- Identify code sections that require execution speed improvements.

The following tasks represent a general workflow that uses code execution profiling:

- 1 With the Simulink model, design and optimize your algorithm.
- 2 Configure the model for code execution profiling, and generate code.
- 3 Execute generated code on target hardware. For example, you can perform real-time execution with Simulink Real-Time or a target support package.

If you are using Embedded Coder, you can run software-in-the-loop (SIL), processor-in-the-loop (PIL), and XCP-based external model simulations on your development computer or embedded target hardware.

- 4 Analyze execution speed through code execution profiling plots and reports. For example, check that the algorithm code satisfies execution time requirements for real-time deployment:
 - If the algorithm code easily meets the requirements, consider enhancing your algorithm to exploit available processing power.
 - If the code cannot be executed in real time, look for ways to reduce execution time.

Identify the tasks that require the most time. For these tasks, investigate whether trade-offs between functionality and speed are possible.

If your target is a multicore processor, distribute the execution of algorithm code across available cores.

- 5 If required, refine the model and return to step 2.

To find information about code execution profiling with Simulink products, use the following table.

Type of Execution	Target	Relevant Products	See
Software-in-the-loop (SIL)	Development computer	Embedded Coder	<ul style="list-style-type: none"> • “Create Execution-Time Profile for Generated Code” (Embedded Coder) • “View and Compare Code Execution Times” (Embedded Coder) • “Analyze Code Execution Data” (Embedded Coder)
Processor-in-the-loop (PIL)	Embedded hardware or instruction set simulator	Embedded Coder	<ul style="list-style-type: none"> • “Create Execution-Time Profile for Generated Code” (Embedded Coder) • “View and Compare Code Execution Times” (Embedded Coder) • “Analyze Code Execution Data” (Embedded Coder)

Type of Execution	Target	Relevant Products	See
XCP external mode	Development computer or custom hardware	Embedded Coder	<ul style="list-style-type: none"> • “Create Execution-Time Profile for Generated Code” (Embedded Coder) • “Analyze Code Execution Data” (Embedded Coder)
Real-time execution	Simulink Real-Time	Simulink Coder, Simulink Real-Time	<ul style="list-style-type: none"> • Execution Profiling for Real-Time Applications (Simulink Real-Time)

See Also

Related Examples

- “Optimize and Deploy on a Multicore Target”
- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “SIL and PIL Simulations” (Embedded Coder)
- “External Mode Simulations for Parameter Tuning, Signal Monitoring, and Code Execution Profiling” (Embedded Coder)

Optimize Generated Code by Combining Multiple for Constructs

This example shows how the code generator combines for loops. The generated code uses for constructs to represent a variety of modeling patterns, such as a matrix signal or Iterator blocks. Using data dependency analysis, the code generator combines for constructs to reduce static code size and runtime branching.

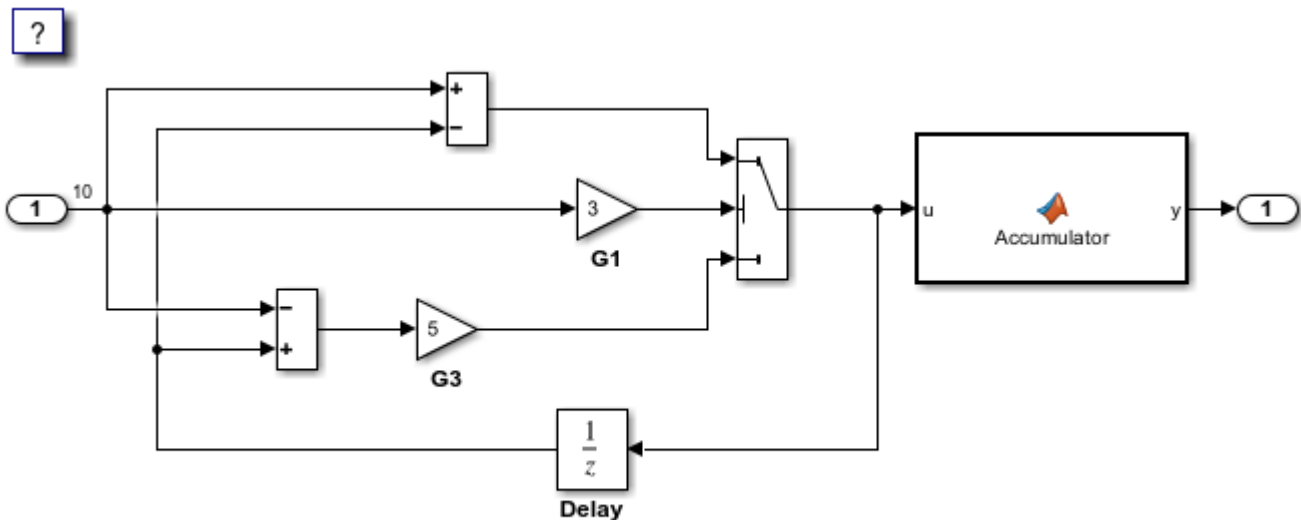
The benefits of optimizing for loops are:

- Reducing ROM and RAM consumption.
- Increasing execution speed.

for Loop Modeling Patterns

Open the example model ForLoopConstruct. The Switch block and MATLAB Function block represent for constructs. In the In1 Block Parameters dialog box, the **Port dimensions** parameter is set to 10 .

```
model = 'ForLoopConstruct';
open_system(model);
```



Copyright 1994-2020 The MathWorks, Inc.

Generate Code

The model does not contain data dependencies across the for loop iterations. Therefore, the code generator combines for loops into one loop.

Build the model.

```
slbuild(model)
```

```
### Starting build procedure for: ForLoopConstruct
### Successful completion of build procedure for: ForLoopConstruct
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
ForLoopConstruct	Code generated and compiled.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)

Build duration: 0h 0m 13.915s

View the optimized code in `ForLoopConstruct.c`.

```
cfile = fullfile('ForLoopConstruct_grt_rtw','ForLoopConstruct.c');
rtwdemodbtype(cfile,'/* Model step', '/* Model initialize', 1, 0);
```

```
/* Model step function */
void ForLoopConstruct_step(void)
{
    real_T In1;
    int32_T k;

    /* MATLAB Function: '<Root>/Accum' */
    /* MATLAB Function 'Accum': '<S1>:1' */
    /* '<S1>:1:3' */
    /* '<S1>:1:4' */
    ForLoopConstruct_Y.Out1 = 0.0;

    /* '<S1>:1:5' */
    for (k = 0; k < 10; k++) {
        /* Gain: '<Root>/G1' incorporates:
         * Inport: '<Root>/In1'
         */
        In1 = ForLoopConstruct_U.In1[k];

        /* Switch: '<Root>/Switch' incorporates:
         * Gain: '<Root>/G1'
         * Gain: '<Root>/G3'
         * Sum: '<Root>/Sum1'
         * Sum: '<Root>/Sum2'
         * UnitDelay: '<Root>/Delay'
         */
        if (3.0 * In1 >= 0.0) {
            In1 -= ForLoopConstruct_DW.Delay_DSTATE[k];
        } else {
            In1 = (ForLoopConstruct_DW.Delay_DSTATE[k] - In1) * 5.0;
        }

        ForLoopConstruct_DW.Delay_DSTATE[k] = In1;

        /* End of Switch: '<Root>/Switch' */

        /* MATLAB Function: '<Root>/Accum' */
        /* '<S1>:1:6' */
        ForLoopConstruct_Y.Out1 += ((real_T)k + 1.0) + In1;
    }
}
```



```
    }  
}
```

The generated code contains code for the single `for` loop.

Close the model

```
bdclose(model)  
rtwdemoclean;
```

See Also

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Configure Loop Unrolling Threshold” on page 40-46
- “For Loop” (Embedded Coder)

Subnormal Number Execution Speed

Subnormal numbers, formerly known as denormal numbers in floating-point literature, fill the underflow gap around zero in floating-point arithmetic. Subnormal values are a special category of floating-point values that are too close to 0.0 to be represented as a normalized value. The leading significand (mantissa) of a subnormal number is zero. When adding and subtracting floating-point numbers, subnormal numbers prevent underflow.

Using subnormal numbers provides precision beyond the normal representation by using leading zeros in the significand to represent smaller values after the representation reaches the minimum exponent. As the value approaches 0.0, you trade off precision for extended range. Subnormal numbers are useful if your application requires extra range.

However, in a real-time system, using subnormal numbers can dramatically increase execution latency, resulting in excessive design margins and real-time overruns. If the simulation or generated code performs calculations that produce or consume subnormal numbers, the execution of these calculations can be up to 50 times slower than similar calculations on normal numbers. The actual simulation or code execution time for subnormal number calculations depends on your computer operating environment. Typically, for desktop processors, the execution time for subnormal number calculations is five times slower than similar calculations on normal numbers.

To minimize the possibility of execution slowdowns or overruns due to subnormal number calculation latency, do one of the following:

- In your model, manually flush to zero any incoming or computed subnormal values at inputs and key operations, such as washouts and filters. For an example, see “Flush Subnormal Numbers to Zero” on page 40-18.

To detect a subnormal value for a single precision, 32-bit floating-point number:

- 1 Find the smallest normalized number on a MATLAB host. In the Command Window, type:

```
>> SmallestNormalSingle = realmin('single')
```

In the C language, `FLT_MIN`, defined in `float.h`, is equivalent to `realmin('single')`.

- 2 Look for values in range:

$$0 < \text{fabsf}(x) < \text{SmallestNormalSingle}$$

To detect a subnormal value for a double precision, 64-bit floating-point number:

- 1 Find the smallest normalized number on a MATLAB host. In the Command Window, type:

```
>> SmallestNormalDouble = realmin('double')
```

In the C language, `DBL_MIN`, defined in `float.h`, is equivalent to `realmin('double')`.

- 2 To detect a subnormal value, look for values in this range:

$$0 < \text{fabs}(x) < \text{SmallestNormalDouble}$$

- Set the **Simulation behavior for denormal numbers** parameter to **Flush to zero (FTZ)** to emulate flush-to-zero behavior for all denormal results from arithmetic operations. For more information, see “Simulation behavior for denormal numbers”.

- On your processor, set flush-to-zero mode or, with your compiler, specify an option to disable subnormal numbers. Flush-to-zero modes treat a subnormal number as 0 when it is an input to a floating-point operation. Underflow exceptions do not occur in flush-to-zero mode.

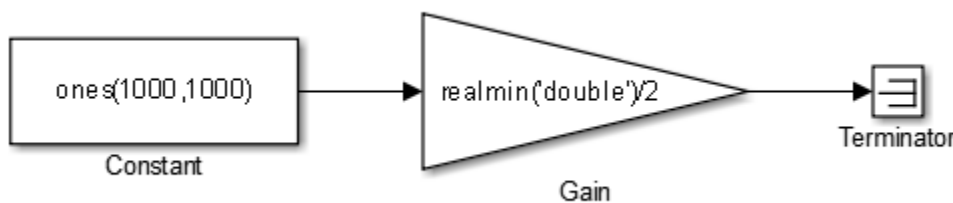
For example, in Intel processors, the flush-to-zero (FTZ) and denormals-are-zero (DAZ) flags in the MXCSR register control floating-point calculations. For the gcc compiler on Linux, `-ffast-math` sets abrupt underflow (FTZ), flush-to-zero, while `-O3 -ffast-math` reverts to gradual underflow, using subnormal numbers.

For more information, see the IEEE Standard 754, *IEEE Standard for Floating-Point Arithmetic*.

Simulation Time with and Without Subnormal Numbers

This example shows how using subnormal numbers increases simulation time by ~5 times.

For this example, create a simple model `ex_subnormal` that has a Constant and a Gain block. The Gain is set to subnormal value `realmin('double')/2`.



To run a simulation, in the Command Window, type `for k=1:5, tic; sim('ex_subnormal'); toc, end`. Observe the elapsed times for simulation using subnormals, similar to the following:

```
>> for k=1:5, tic; sim('ex_subnormal'); toc, end
Elapsed time is 9.909326 seconds.
Elapsed time is 9.617966 seconds.
Elapsed time is 9.797183 seconds.
Elapsed time is 9.702397 seconds.
Elapsed time is 9.893946 seconds.
```

Set the Gain to a number, 2, that is not a subnormal value:

```
>> set_param('ex_subnormal/Gain', 'Gain', '2');
```

To run a simulation, in the Command Window, type `for k=1:5, tic; sim('ex_subnormal'); toc, end`. Observe elapsed times for simulations that do not use subnormal values, similar to the following:

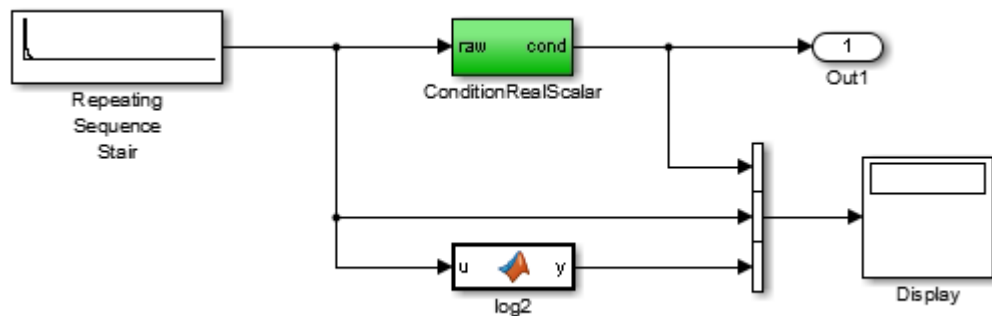
```
>> for k=1:5, tic; sim('ex_subnormal'); toc, end
Elapsed time is 2.045123 seconds.
Elapsed time is 1.796598 seconds.
Elapsed time is 1.758458 seconds.
```

Elapsed time is 1.721721 seconds.
 Elapsed time is 1.780569 seconds.

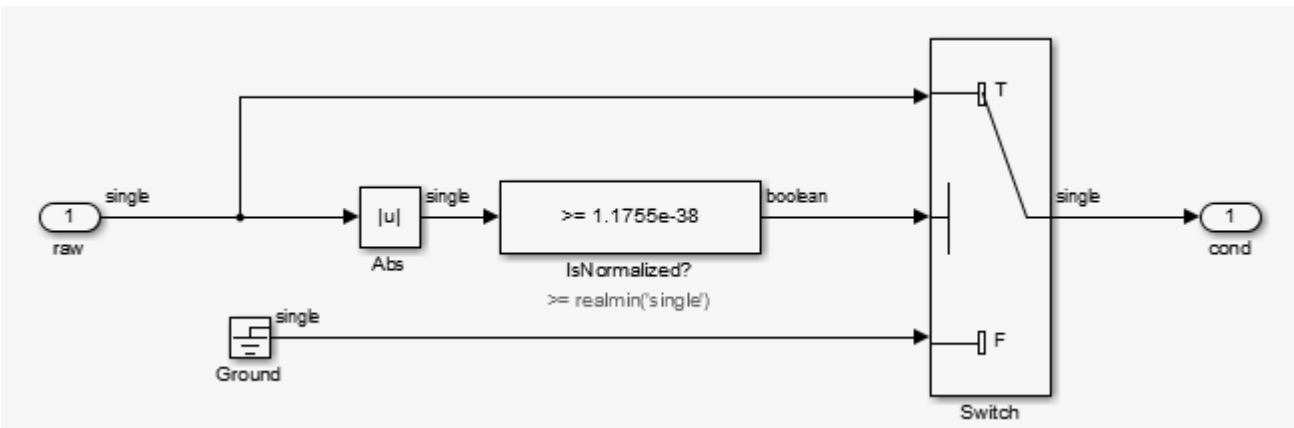
Flush Subnormal Numbers to Zero

This example shows how to flush single precision subnormal numbers to zero.

- 1 For this example, create a simple model `ex_flush_to_zero`.



- Repeating Sequence Stair generates a sequence of numbers from two raised to the power of 0 through two raised to the power of -165. The sequence approaches zero.
- ConditionRealScalar flushes subnormal single precision values that are less than `realmin('single')` to zero.



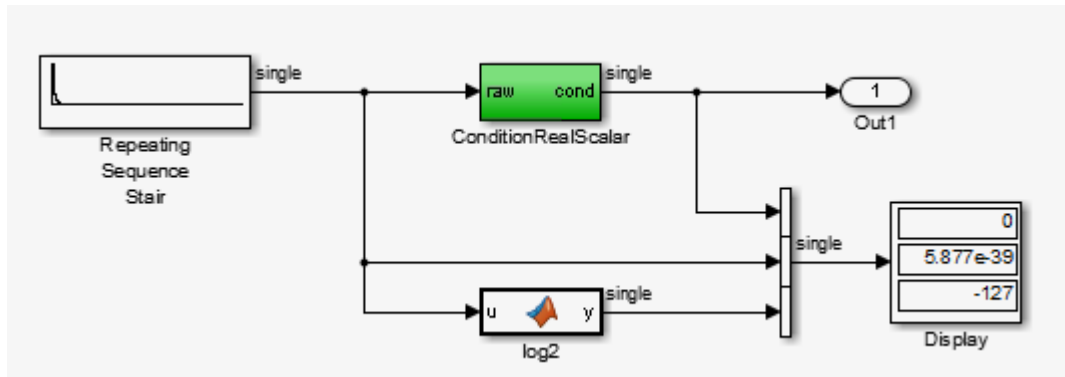
- MATLAB function block `log2` generates the base 2 logarithm of the Repeating Sequence Stair output. Specifically, `log2` generates the numbers 0 through -165.

```
function y = fcn(u)
  %#codegen

  y = log2(u);
end
```

- 2 On the **Simulation > Step Back > Configure simulation stepping** pane:

- Select **Enable stepping back**.
 - Select **Pause simulation when time reaches** and enter 121.
- 3 In the model window, run the simulation. The simulation pauses at T=121. The displayed values:
- ConditionRealScalar output approaches zero.
 - Repeating Sequence Stair output approaches zero.
- 4 Step the simulation forward to T=127. ConditionRealScalar flushes the subnormal value output from Repeating Sequence Stair to zero.



- 5 Continue stepping the simulation forward. ConditionRealScalar flushes the subnormal single precision values output from Repeating Sequence Stair to zero. When T=150, the output of Repeating Sequence Stair is itself zero.

See Also

Related Examples

- “Data Types Supported by Simulink”
- “Numerical Consistency of Model and Generated Code Simulation Results” on page 43-2
- “Specify Single-Precision Data Type for Embedded Application” on page 15-104

Remove Code for Out-of-Range Floating Point to Integer Conversions

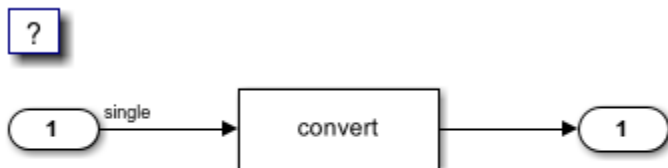
This example shows how to generate optimized code by removing code for out-of-range floating point to integer conversions. Standard C does not define the behavior of out-of-range floating point to integer conversions, while these conversions are well-defined during simulation. When you select the configuration parameter **Remove code from floating-point to integer conversions that wraps out-of-range values**, the code generator removes the wrapping code that handles out-of-range floating point to integer conversion and improves code efficiency. Without this code, there might be a mismatch between simulation and code generation results for out-of-range values.

If the input values in your application are in the range of the output type, remove code for out-of-range floating-point to integer conversions. Removing this code reduces the size and increases the speed of the generated code.

Example Model

Open the example model `FloatMultiplicationNetSlope`.

```
model = 'FloatMultiplicationNetSlope';
open_system(model);
```



Copyright 2022 The MathWorks, Inc.

The model contains a Data Type Conversion block. To configure the model to convert the input signal from a `single` to a `uint8`, open the Data Type Conversion dialog box. For the **Output data type** parameter, select `uint8`. Alternatively, use this command-line API to set the **Output data type**:

```
set_param('FloatMultiplicationNetSlope/Data Type Conversion', 'OutDataTypeStr', 'uint8');
```

A `uint8` can support values from 0 to 255. If the input signal has a value outside of this range, an out-of-range conversion occurs.

Generate Code Without Optimization

1. Open the Configuration Parameters dialog box. On the Optimization pane, clear the **Remove code from floating-point to integer conversions that wraps out-of-range values** parameter check box. Alternatively, use the command-line API to disable the optimization:

```
set_param(model, 'EfficientFloat2IntCast', 'off');
```

2. Build the model by using the `slbuild` function or by pressing **Ctrl+B**.

```
slbuild(model)
```

```
### Starting build procedure for: FloatMultiplicationNetSlope
### Successful completion of build procedure for: FloatMultiplicationNetSlope
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
FloatMultiplicationNetSlope	Code generated and compiled.	Code generation information file does

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 12.377s
```

3. Inspect the generated `FloatMultiplicationNetSlope_step` step function in the `FloatMultiplicationNetSlope.c`.

```
file = fullfile('FloatMultiplicationNetSlope_grt_rtw','FloatMultiplicationNetSlope.c');
rtwdemodbtype(file,'/* Model step function */','/* Model initialize function',1,1);
```

```
/* Model step function */
void FloatMultiplicationNetSlope_step(void)
{
    real32_T tmp;

    /* DataTypeConversion: '<Root>/Data Type Conversion' incorporates:
     * Inport: '<Root>/Input'
     */
    tmp = (real32_T)floor(FloatMultiplicationNetSlope_U.Input);
    if (rtIsNaNF(tmp) || rtIsInfF(tmp)) {
        tmp = 0.0F;
    } else {
        tmp = (real32_T)fmod(tmp, 256.0);
    }

    /* Outport: '<Root>/Output' incorporates:
     * DataTypeConversion: '<Root>/Data Type Conversion'
     */
    FloatMultiplicationNetSlope_Y.Output = (uint8_T)(tmp < 0.0F ? (int32_T)
        (uint8_T)-(int8_T)(uint8_T)-tmp : (int32_T)(uint8_T)tmp);
}
```

The code generator applies the `fmod` function to handle out-of-range-results. In this case, the generated code is larger but the results of the generated code match the simulation results.

Generate Code with Optimization

1. Open the Configuration Parameters dialog box. On the Optimization pane, select the **Remove code from floating-point to integer conversions that wraps out-of-range values** parameter. Alternatively, use the command-line API to enable the optimization:

```
set_param(model, 'EfficientFloat2IntCast', 'on')
```

2. Build the model. Inspect the generated `FloatMultiplicationNetSlope_step` step function with optimization in the `FloatMultiplicationNetSlope.c`

```

slbuild(model);
file = fullfile('FloatMultiplicationNetSlope_grt_rtw','FloatMultiplicationNetSlope.c');
rtwdemodbtype(file,'/* Model step function */','/* Model initialize function',1,1);

### Starting build procedure for: FloatMultiplicationNetSlope
### Successful completion of build procedure for: FloatMultiplicationNetSlope

Build Summary

Top model targets built:

Model                                Action                                Rebuild Reason
=====
FloatMultiplicationNetSlope  Code generated and compiled.  Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 9.0399s

/* Model step function */
void FloatMultiplicationNetSlope_step(void)
{
    /* Output: '<Root>/Output' incorporates:
     * DataTypeConversion: '<Root>/Data Type Conversion'
     * Inport: '<Root>/Input'
     */
    FloatMultiplicationNetSlope_Y.Output = (uint8_T)
        FloatMultiplicationNetSlope_U.Input;
}

```

The generated code is more efficient without the protective code, but it is possible that the execution of the generated code does not produce the same results as simulation for values not in the range of 0 to 255.

Clean Up Example Folders and Files

Close the model and remove temporary folders and files.

```

bdclose(model);
rtwdemoclean;

```

See Also

Remove code from floating-point to integer conversions that wraps out-of-range values

Related Examples

- “hisl_0053: Configuration Parameters > Code Generation > Optimization > Remove code from floating-point to integer conversions that wraps out-of-range values”
- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Remove Code That Maps NaN to Integer Zero” on page 40-23

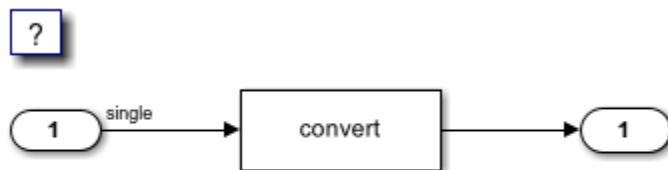
Remove Code That Maps NaN to Integer Zero

This example shows how to remove code that maps NaN to integer zero. For floating point to integer conversions involving saturation, Simulink® converts NaN to integer zero during simulation. If your model contains an input value of NaN, you can specify that the code generator produce code that maps NaN to zero. Without this code, there is a mismatch between simulation and code generation results because in Standard C, every condition involving NaN evaluates to false.

If input values of NaN do not exist in your application, you can remove code that maps NaN to integer zero. Removing this code reduces the size and increases the speed of the generated code.

Example Model

Open the example model `FloatMultiplicationNetSlope` by entering the model name in the Command Window.



Copyright 2022 The MathWorks, Inc.

The model contains a Data Type Conversion block. Configure the model to convert the input signal from a `single` to a `uint8`.

1. Open the Data Type Conversion dialog box. For the **Output data type** parameter, select `uint8`.
2. Select the **Saturate on integer overflow** parameter check box. Selecting this parameter specifies that an out-of-range signal value equals either the minimum or maximum value that the data type can represent.
3. Open the Configuration Parameters dialog box. Select the **Support: non-finite numbers** parameter check box. Selecting this parameter enables generating nonfinite data and operations on nonfinite data.

Generate Code Without Optimization

1. Open the Configuration Parameters dialog box. On the Optimization pane, clear the **Remove code from floating-point to integer conversions with saturation that maps NaN to zero** parameter check box.
2. Build the model.

```

### Starting build procedure for: FloatMultiplicationNetSlope
### Successful completion of build procedure for: FloatMultiplicationNetSlope
  
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
FloatMultiplicationNetSlope	Code generated and compiled.	Code generation information file does

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 10.645s

3. Inspect the generated `FloatMultiplicationNetSlope_step` step function in the `FloatMultiplicationNetSlope.c`.

```

/* Model step function */
void FloatMultiplicationNetSlope_step(void)
{
    /* DataTypeConversion: '<Root>/Data Type Conversion' incorporates:
     * Inport: '<Root>/Input'
     */
    if (FloatMultiplicationNetSlope_U.Input < 256.0F) {
        if (FloatMultiplicationNetSlope_U.Input >= 0.0F) {
            /* Output: '<Root>/Output' */
            FloatMultiplicationNetSlope_Y.Output = (uint8_T)
                FloatMultiplicationNetSlope_U.Input;
        } else {
            /* Output: '<Root>/Output' */
            FloatMultiplicationNetSlope_Y.Output = 0U;
        }
    } else if (FloatMultiplicationNetSlope_U.Input >= 256.0F) {
        /* Output: '<Root>/Output' */
        FloatMultiplicationNetSlope_Y.Output = MAX_uint8_T;
    } else {
        /* Output: '<Root>/Output' */
        FloatMultiplicationNetSlope_Y.Output = 0U;
    }

    /* End of DataTypeConversion: '<Root>/Data Type Conversion' */
}

```

The code generator applies condition to map NaN to interger zero. In this case, the generated code is larger but the results of the generated code match with simulation results.

Generate Code with Optimization

1. Open the Configuration Parameters dialog box.
2. On the Optimization pane, select the **Remove code from floating-point to integer conversions with saturation that maps NaN to zero** parameter.
3. Build the model.

```

### Starting build procedure for: FloatMultiplicationNetSlope
### Successful completion of build procedure for: FloatMultiplicationNetSlope

```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
-------	--------	----------------

```
=====
FloatMultiplicationNetSlope Code generated and compiled. Generated code was out of date.
```

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 9.701s
```

4. Inspect the generated `FloatMultiplicationNetSlope_step` step function with optimization in the `FloatMultiplicationNetSlope.c`.

```
/* Model step function */
void FloatMultiplicationNetSlope_step(void)
{
  /* DataTypeConversion: '<Root>/Data Type Conversion' incorporates:
   * Inport: '<Root>/Input'
   */
  if (FloatMultiplicationNetSlope_U.Input < 256.0F) {
    if (FloatMultiplicationNetSlope_U.Input >= 0.0F) {
      /* Outport: '<Root>/Output' */
      FloatMultiplicationNetSlope_Y.Output = (uint8_T)
        FloatMultiplicationNetSlope_U.Input;
    } else {
      /* Outport: '<Root>/Output' */
      FloatMultiplicationNetSlope_Y.Output = 0U;
    }
  } else {
    /* Outport: '<Root>/Output' */
    FloatMultiplicationNetSlope_Y.Output = MAX_uint8_T;
  }

  /* End of DataTypeConversion: '<Root>/Data Type Conversion' */
}
```

The generated code maps NaN to 255 and not integer zero. The generated code is more efficient without the extra code that maps NaN to integer zero. But it is possible that the execution of generated code does not produce the same results as the simulation for NaN values.

See Also

Remove code from floating-point to integer conversions with saturation that maps NaN to zero

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Remove Code for Out-of-Range Floating Point to Integer Conversions” on page 40-20

Disable Nonfinite Checks or Inlining for Math Functions

When the code generator produces code for math functions:

- If the model option **Support non-finite numbers** is selected, nonfinite number checking is generated uniformly for math functions, without the ability to specify that nonfinite number checking should be generated for some functions, but not for others.
- By default, inlining is applied uniformly for math functions, without the ability to specify that inlining should be generated for some functions, while invocations should be generated for others.

You can use code replacement library (CRL) customization entries to:

- Selectively disable nonfinite checks for math functions. This can improve the execution speed of the generated code.
- Selectively disable inlining of math functions. This can increase code readability and decrease code size.

The functions for which these customizations are supported include the following:

- Floating-point only: `atan2`, `copysign`, `fix`, `hypot`, `log`, `log10`, `round`, `sincos`, and `sqrt`
- Floating-point and integer: `abs`, `max`, `min`, `mod`, `rem`, `saturate`, and `sign`

The general workflow for disabling nonfinite number checking and/or inlining is as follows:

- 1 If you can disable nonfinite number checking for a particular math function, or if you want to disable inlining for a particular math function and instead generate a function invocation, you can copy the following MATLAB function code into a MATLAB file with an `.m` file name extension, for example, `crl_table_customization.m`.

```
function hTable = crl_table_customization

% Create an instance of the Code Replacement Library table for controlling
% function intrinsic inlining and nonfinite support

hTable = RTW.TflTable;

% Inline - true (if function needs to be inline)
%           false (if function should not be inlined)
% SNF (support nonfinite) - ENABLE (if non-finite checking should be performed)
%                               DISABLE (if non-finite checking should NOT be performed)
%                               UNSPECIFIED (Default behavior)

% registerCustomizationEntry(hTable, ...
%     Priority, numInputs, key, inType, outType, Inline, SNF);

registerCustomizationEntry(hTable, ...
    100, 2, 'atan2', 'double', 'double', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'atan2', 'single', 'single', false, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 1, 'sincos', 'double', 'double', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'sincos', 'single', 'single', false, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 1, 'abs', 'double', 'double', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'abs', 'single', 'single', true, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 1, 'abs', 'int32', 'int32', true, 'UNSPECIFIED');
```

```

registerCustomizationEntry(hTable, ...
    100, 1, 'abs', 'int16', 'int16', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'abs', 'int8', 'int8', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'abs', 'integer', 'integer', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'abs', 'uint32', 'uint32', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'abs', 'uint16', 'uint16', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'abs', 'uint8', 'uint8', true, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 2, 'hypot', 'double', 'double', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'hypot', 'single', 'single', false, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 1, 'log', 'double', 'double', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'log', 'single', 'double', false, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 1, 'log10', 'double', 'double', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'log10', 'single', 'double', false, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 2, 'min', 'double', 'double', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'min', 'single', 'single', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'min', 'int32', 'int32', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'min', 'int16', 'int16', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'min', 'int8', 'int8', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'min', 'uint32', 'uint32', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'min', 'uint16', 'uint16', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'min', 'uint8', 'uint8', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'min', 'integer', 'integer', true, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 2, 'max', 'double', 'double', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'max', 'single', 'single', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'max', 'int32', 'int32', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'max', 'int16', 'int16', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'max', 'int8', 'int8', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'max', 'uint32', 'uint32', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'max', 'uint16', 'uint16', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'max', 'uint8', 'uint8', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'max', 'integer', 'integer', true, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 2, 'mod', 'double', 'double', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'mod', 'single', 'single', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'mod', 'int32', 'int32', false, 'UNSPECIFIED');

```

```

registerCustomizationEntry(hTable, ...
    100, 2, 'mod', 'int16', 'int16', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'mod', 'int8', 'int8', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'mod', 'uint32', 'uint32', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'mod', 'uint16', 'uint16', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'mod', 'uint8', 'uint8', false, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 2, 'rem', 'double', 'double', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'rem', 'single', 'single', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'rem', 'int32', 'int32', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'rem', 'int16', 'int16', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'rem', 'int8', 'int8', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'rem', 'uint32', 'uint32', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'rem', 'uint16', 'uint16', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 2, 'rem', 'uint8', 'uint8', false, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 1, 'round', 'double', 'double', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'round', 'single', 'single', false, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 3, 'saturate', 'double', 'double', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 3, 'saturate', 'single', 'single', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 3, 'saturate', 'int32', 'int32', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 3, 'saturate', 'int16', 'int16', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 3, 'saturate', 'int8', 'int8', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 3, 'saturate', 'uint32', 'uint32', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 3, 'saturate', 'uint16', 'uint16', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 3, 'saturate', 'uint8', 'uint8', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 3, 'saturate', 'integer', 'integer', true, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 1, 'sign', 'double', 'double', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'sign', 'single', 'single', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'sign', 'int32', 'integer', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'sign', 'int16', 'integer', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'sign', 'int8', 'integer', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'sign', 'uint32', 'uint32', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'sign', 'uint16', 'uint16', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'sign', 'uint8', 'uint8', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'sign', 'integer', 'integer', true, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...

```

```

    100, 1, 'sqrt', 'double', 'double', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'sqrt', 'single', 'single', true, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 1, 'fix', 'double', 'double', false, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'fix', 'single', 'single', false, 'UNSPECIFIED');

registerCustomizationEntry(hTable, ...
    100, 1, 'copy sign', 'double', 'double', true, 'UNSPECIFIED');
registerCustomizationEntry(hTable, ...
    100, 1, 'copy sign', 'single', 'single', true, 'UNSPECIFIED');

```

- 2 To reduce the size of the file, you can delete the `registerCustomizationEntry` lines for functions for which the default nonfinite number checking and inlining behavior is acceptable.
- 3 For each remaining entry,
 - Set the `Inline` argument to `true` if the function should be inlined or `false` if it should not be inlined.
 - Set the `SNF` argument to `ENABLE` if nonfinite checking should be generated, `DISABLE` if nonfinite checking should not be generated, or `UNSPECIFIED` to accept the default behavior based on the model option settings.

Save the file.

- 4 Optionally, perform a quick check of the syntactic validity of the customization table entries by invoking the table definition file at the MATLAB command line (`>> tbl = crl_table_customization`). Fix syntax errors that are flagged.
- 5 Optionally, view the customization table entries in the **Code Replacement Viewer** (`>> crviewer(crl_table_customization)`). For more information about viewing code replacement tables, see “Choose a Code Replacement Library” on page 26-6.
- 6 To register these changes and make them appear in the **Code replacement library** drop-down list located on the **Code Generation > Interface** pane of the Configuration Parameters dialog box, first copy the following MATLAB function code into an instance of the file `rtwTargetInfo.m`.

Note For the example below, specify the argument `'RTW'` if a GRT target is selected for your model, otherwise omit the argument.

```

function rtwTargetInfo(cm)
% rtwTargetInfo function to register a code replacement library (CRL)

    % Register the CRL defined in local function locCrlRegFcn
    cm.registerTargetInfo(@locCrlRegFcn);

end % End of RTWTARGETINFO

% Local function to define a CRL containing crl_table_customization
function thisCrl = locCrlRegFcn

    % Instantiate a CRL registry entry - specify 'RTW' for GRT
    thisCrl = RTW.TflRegistry('RTW');

    % Define the CRL properties
    thisCrl.Name = 'CRL Customization Example';
    thisCrl.Description = 'Example of CRL Customization';
    thisCrl.TableList = {'crl_table_customization'};
    thisCrl.TargetHWDeviceType = {'*'};

```

```
end % End of LOCCRLREGFCN
```

You can edit the **Name** field to specify the library name that appears in the **Code replacement library** drop-down list. Also, the file name in the **TableList** field must match the name of the file you created in step 1.

To register your changes, with both of the MATLAB files you created present in the MATLAB path, enter the following command at the MATLAB command line:

```
sl_refresh_customizations
```

- 7** Create or open a model that generates function code corresponding to one of the math functions for which you specified a change in nonfinite number checking or inlining behavior.
- 8** Open the Configuration Parameters dialog box, go to the **Code Generation > Interface** pane, and use the **Code replacement library** drop-down list to select the code replacement entry you registered in step 6, for example, **CRL Customization Example**.
- 9** Generate code for the model and examine the generated code to verify that the math functions are generated as expected.

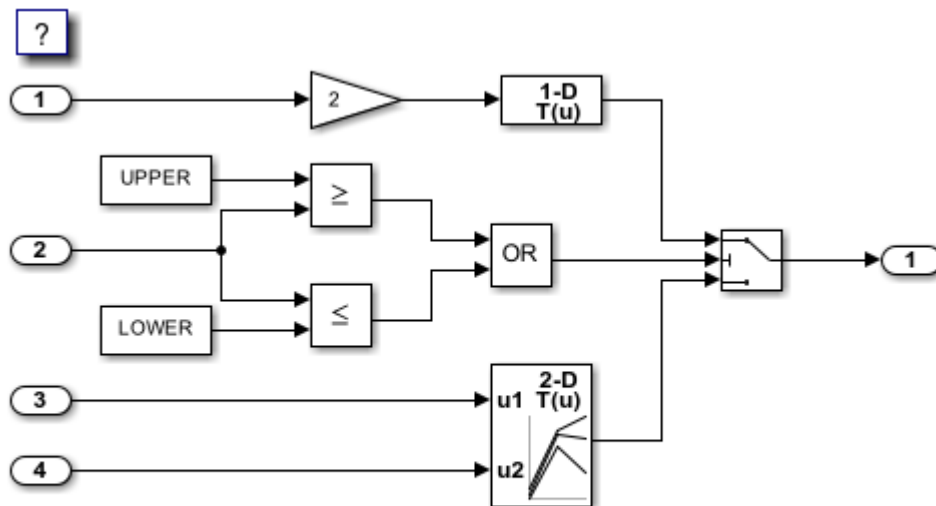
Fold Expressions

Expression folding optimizes code to minimize the computation of intermediate results at block outputs and the storage of such results in temporary buffers or variables. When expression folding is on, the code generator collapses (folds) block computations into a single expression, instead of generating separate code statements and storage declarations for each block in a model. Most Simulink® blocks support expression folding.

Expression folding improves the efficiency of generated code, frequently achieving results that compare favorably to hand-optimized code. In many cases, entire groups of model computations fold into a single, highly optimized line of code.

Example Model

```
model = 'FoldBlockComputations';
open_system(model);
```



Copyright 2022 The MathWorks, Inc.

Generate Code

Expression folding is available only when the **Signal storage reuse** parameter is set to on because expression folding operates only on expressions involving local variables. The **Signal storage reuse** and **Eliminate superfluous local variables (expression folding)** parameters are on by default. Clear the **Eliminate superfluous local variables (expression folding)** parameter or enter the following command in the MATLAB® Command Window to turn the parameter off:

```
set_param(model, 'ExpressionFolding', 'off');
```

Build the model.

```
slbuild(model)
```

```
### Starting build procedure for: FoldBlockComputations
### Successful completion of build procedure for: FoldBlockComputations
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
FoldBlockComputations	Code generated and compiled.	Code generation information file does not exist

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 14.593s

With expression folding off, in the `FoldBlockComputations.c` file, there are separate code statements before and in the Switch block operation.

```
cfile = fullfile('FoldBlockComputations_grt_rtw','FoldBlockComputations.c');
rtwdemodbtype(cfile,'/* Model step', '/* Model initialize', 1, 0);
```

```
/* Model step function */
void FoldBlockComputations_step(void)
{
    real_T rtb_Gain;
    boolean_T rtb_LogicalOperator;
    boolean_T rtb_RelationalOperator;

    /* RelationalOperator: '<Root>/Relational Operator1' incorporates:
     * Constant: '<Root>/Constant'
     * Inport: '<Root>/In2'
     */
    rtb_LogicalOperator = (FoldBlockComputations_P.UPPER >=
        FoldBlockComputations_U.In2);

    /* RelationalOperator: '<Root>/Relational Operator' incorporates:
     * Constant: '<Root>/Constant1'
     * Inport: '<Root>/In2'
     */
    rtb_RelationalOperator = (FoldBlockComputations_U.In2 <=
        FoldBlockComputations_P.LOWER);

    /* Logic: '<Root>/Logical Operator' */
    rtb_LogicalOperator = (rtb_LogicalOperator || rtb_RelationalOperator);

    /* Switch: '<Root>/Switch' */
    if (rtb_LogicalOperator) {
        /* Gain: '<Root>/Gain' incorporates:
         * Inport: '<Root>/In1'
         */
        rtb_Gain = 2.0 * FoldBlockComputations_U.In1;

        /* Lookup_n-D: '<Root>/Look-Up Table' incorporates:
         * Gain: '<Root>/Gain'
         */
        FoldBlockComputations_Y.Out1 = look1_binl(rtb_Gain,
            FoldBlockComputations_P.T1Break, FoldBlockComputations_P.T1Data, 10U);
    } else {
        /* Lookup_n-D: '<Root>/Look-Up Table' incorporates:
         * Inport: '<Root>/In3'
         */
    }
}
```

```

    * Inport: '<Root>/In4'
    * Lookup_n-D: '<Root>/Look-Up Table (2-D)'
    */
    FoldBlockComputations_Y.Out1 = look2_binlx(FoldBlockComputations_U.In3,
        FoldBlockComputations_U.In4, FoldBlockComputations_P.T2Break,
        FoldBlockComputations_P.T2Break, FoldBlockComputations_P.T2Data,
        rtCP_LookUpTable2D_maxIndex, 3U);
}

/* End of Switch: '<Root>/Switch' */
}

```

Generate Code with Optimization

Enter the following command to turn expression folding on:

```
set_param(model, 'ExpressionFolding', 'on');
```

Build the model.

```
slbuild(model);
```

```
### Starting build procedure for: FoldBlockComputations
### Successful completion of build procedure for: FoldBlockComputations
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
FoldBlockComputations	Code generated and compiled.	Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.251s

The following is a portion of `FoldBlockComputations.c`. In the optimized code, the code generator folds all computations into the Switch block operation.

```

rtwdemodbtype(cfile, '/* Model step', '/* Model initialize', 1, 0);

/* Model step function */
void FoldBlockComputations_step(void)
{
    /* Switch: '<Root>/Switch' incorporates:
    * Constant: '<Root>/Constant'
    * Constant: '<Root>/Constant1'
    * Inport: '<Root>/In2'
    * Logic: '<Root>/Logical Operator'
    * RelationalOperator: '<Root>/Relational Operator'
    * RelationalOperator: '<Root>/Relational Operator1'
    */
    if ((FoldBlockComputations_P.UPPER >= FoldBlockComputations_U.In2) ||
        (FoldBlockComputations_U.In2 <= FoldBlockComputations_P.LOWER)) {
        /* Outport: '<Root>/Out1' incorporates:
        * Gain: '<Root>/Gain'
        * Inport: '<Root>/In1'
        * Lookup_n-D: '<Root>/Look-Up Table'

```

```
    */
    FoldBlockComputations_Y.Out1 = look1_binlx(2.0 * FoldBlockComputations_U.In1,
        FoldBlockComputations_P.T1Break, FoldBlockComputations_P.T1Data, 10U);
} else {
    /* Outport: '<Root>/Out1' incorporates:
    * Inport: '<Root>/In3'
    * Inport: '<Root>/In4'
    * Lookup_n-D: '<Root>/Look-Up Table (2-D)'
    */
    FoldBlockComputations_Y.Out1 = look2_binlx(FoldBlockComputations_U.In3,
        FoldBlockComputations_U.In4, FoldBlockComputations_P.T2Break,
        FoldBlockComputations_P.T2Break, FoldBlockComputations_P.T2Data,
        rtCP_LookUpTable2D_maxIndex, 3U);
}

/* End of Switch: '<Root>/Switch' */
}
```

Close the model and code generation report.

```
bdclose(model)
rtwdemoclean;
```

Minimize Computations and Storage for Intermediate Results at Block Outputs

In this section...

“Expression Folding” on page 40-35
 “Example Model” on page 40-35
 “Generate Code” on page 40-35
 “Enable Optimization” on page 40-36
 “Generate Code with Optimization” on page 40-36

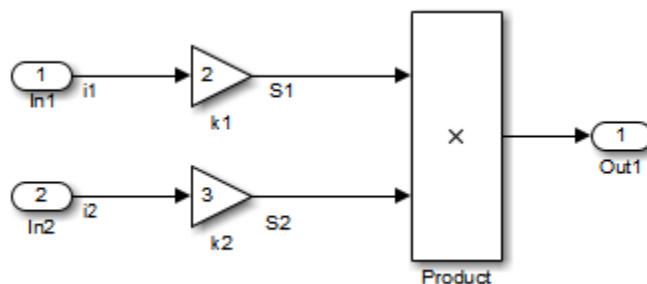
Expression Folding

Expression folding optimizes code to minimize the computation of intermediate results at block outputs and the storage of such results in temporary buffers or variables. When expression folding is on, the code generator collapses (folds) block computations into a single expression, instead of generating separate code statements and storage declarations for each block in the model. Most Simulink blocks support expression folding.

Expression folding improves the efficiency of generated code, frequently achieving results that compare favorably to hand-optimized code. In many cases, entire groups of model computations fold into a single, highly optimized line of code.

You can use expression folding in your own inlined S-function blocks. For more information, see “S-Functions That Support Expression Folding” on page 14-64.

Example Model



Generate Code

With expression folding off, in the `explfld.c` file, the code generator generates this code.

```

/* Model step function */
void exprfld_step(void)
{
  /* Gain: '<Root>/Gain' incorporates:
   * Inport: '<Root>/In1'
   */
}
  
```

```

exprfld_B.S1 = exprfld_P.Gain_Gain * exprfld_U.i1;

/* Gain: '<Root>/Gain1' incorporates:
 * Inport: '<Root>/In2'
 */
exprfld_B.S2 = exprfld_P.Gain1_Gain * exprfld_U.i2;

/* Output: '<Root>/Out1' incorporates:
 * Product: '<Root>/Product'
 */
exprfld_Y.Out1 = exprfld_B.S1 * exprfld_B.S2;
}

```

There are separate code statements for both Gain blocks. Before final output, these code statements compute temporary results for the Gain blocks.

Enable Optimization

Expression folding is on by default. To see if expression folding is on for an existing model:

- 1 Expression folding is available only when the **Configuration Parameters > Signal storage reuse** parameter is selected because expression folding operates only on expressions involving local variables. Enable the **Signal storage reuse** parameter.
- 2 When you select **Signal storage reuse**, the **Enable local block outputs**, **Reuse local block outputs**, and **Eliminate superfluous local variables (expression folding)** parameters are all on by default.

Generate Code with Optimization

With expression folding, the code generator generates a single-line output computation, as shown in the `expfld.c` file. The generated comments document the block parameters that appear in the expression.

```

/* Model step function */
void exprfld_step(void)
{
  /* Output: '<Root>/Out1' incorporates:
   * Gain: '<Root>/Gain'
   * Gain: '<Root>/Gain1'
   * Inport: '<Root>/In1'
   * Inport: '<Root>/In2'
   * Product: '<Root>/Product'
   */
  exprfld_Y.Out1 =
    exprfld_P.Gain_Gain *
    exprfld_U.i1 *
    (exprfld_P.Gain1_Gain * exprfld_U.i2);
}

```

See Also

Signal storage reuse | Reuse local block outputs | Enable local block outputs | Eliminate superfluous local variables (Expression folding)

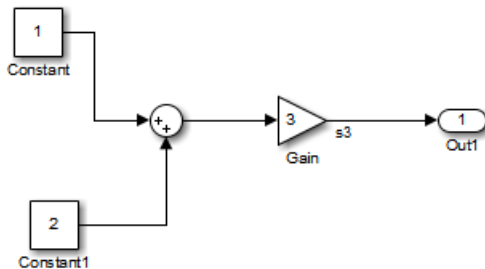
Related Examples

- “Fold Expressions” on page 40-31
- “Enable and Reuse Local Block Outputs in Generated Code” on page 40-102
- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “C Code Generation Configuration for Model Interface Elements” on page 16-2

Inline Invariant Signals

You can optimize the generated code by selecting **Inline invariant signals** on the **Optimization** pane. The generated code uses the numerical values of the invariant signals instead of their symbolic names.

An invariant signal is a block output signal that does not change during Simulink simulation. For example, the signal `S3` is an invariant signal. An *invariant signal* is not the same as an *invariant constant*. The two constants (1 and 2) and the gain value of 3 are invariant constants. To inline invariant constants, set **Default parameter behavior** to **Inlined**.



Optimize Generated Code Using Inline Invariant Signals

This example shows how to use inline invariant signals to optimize the generated code. This optimization transforms symbolic names of invariant signals into constant values.

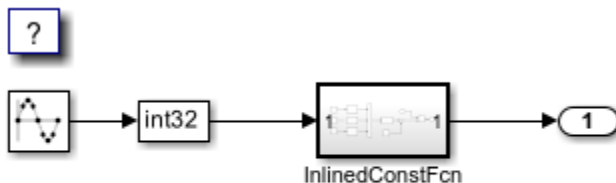
The `InlineInvariantSignals` optimization:

- Reduces ROM and RAM consumption.
- Improves execution speed.

Example Model

Consider the model `InvariantSignalsInline`.

```
model = 'InvariantSignalsInline';
open_system(model);
```



Copyright 2014-2020 The MathWorks, Inc.

Generate Code

Build the model using Simulink® Coder™.


```
slbuild(model)
```

```
### Starting build procedure for: InvariantSignalsInline
### Successful completion of build procedure for: InvariantSignalsInline
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
InvariantSignalsInline	Code generated and compiled.	Code generation information file does not e

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 13.123s

View the generated code without the optimization. These lines of code are in `InvariantSignalsInline.c`.

```
cfile = fullfile('InvariantSignalsInline_grt_rtw',...
    'InvariantSignalsInline.c');
rtwdemodtype(cfile, '/* Output and update for atomic system',...
    '/* Model output', 1, 0);

/* Output and update for atomic system: '<Root>/InlinedConstFcn' */
void InvariantSignal_InlinedConstFcn(int32_T rtu_In1,
    B_InlinedConstFcn_InvariantSi_T *localB, const ConstB_InlinedConstFcn_Invari_T
    *localC)
{
    /* Product: '<S1>/Product' */
    localB->Product = rtu_In1 * localC->Sum_p;
}
```

Enable Optimization

- 1 Open the Configuration Parameters dialog box.
- 2 On the **Optimization** pane, select **Inline Invariant Signals**.

Alternatively, you can use the command-line API to enable the optimization:

```
set_param(model, 'InlineInvariantSignals', 'on');
```

Generate Code with Optimization

The generated code uses the numerical values of the folded constants instead of creating an additional structure.

Build the model using Simulink Coder.

```
slbuild(model)
```

```
### Starting build procedure for: InvariantSignalsInline
### Successful completion of build procedure for: InvariantSignalsInline
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
=====		
InvariantSignalsInline	Code generated and compiled.	Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 9.9308s

View the generated code with the optimization in `InvariantSignalsInline.c`.

```
rtwdemodbtype(cfile,...  
    /* Output and update for atomic system', /* Model output', 1, 0);
```

```
/* Output and update for atomic system: '<Root>/InlinedConstFcn' */  
void InvariantSignal_InlinedConstFcn(int32_T rtu_In1,  
    B_InlinedConstFcn_InvariantSi_T *localB)  
{  
    /* Product: '<S1>/Product' */  
    localB->Product = rtu_In1 << 5;  
}
```

Close the model and code generation report.

```
bdclose(model)  
rtwdemoclean;
```

See Also

Inline invariant signals

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Inline Numeric Values of Block Parameters” on page 40-41

Inline Numeric Values of Block Parameters

This example shows how to optimize the generated code by inlining the numeric values of block parameters. *Block parameters* include the **Gain** parameter of a Gain block and the table data and breakpoint sets of an n-D Lookup Table block.

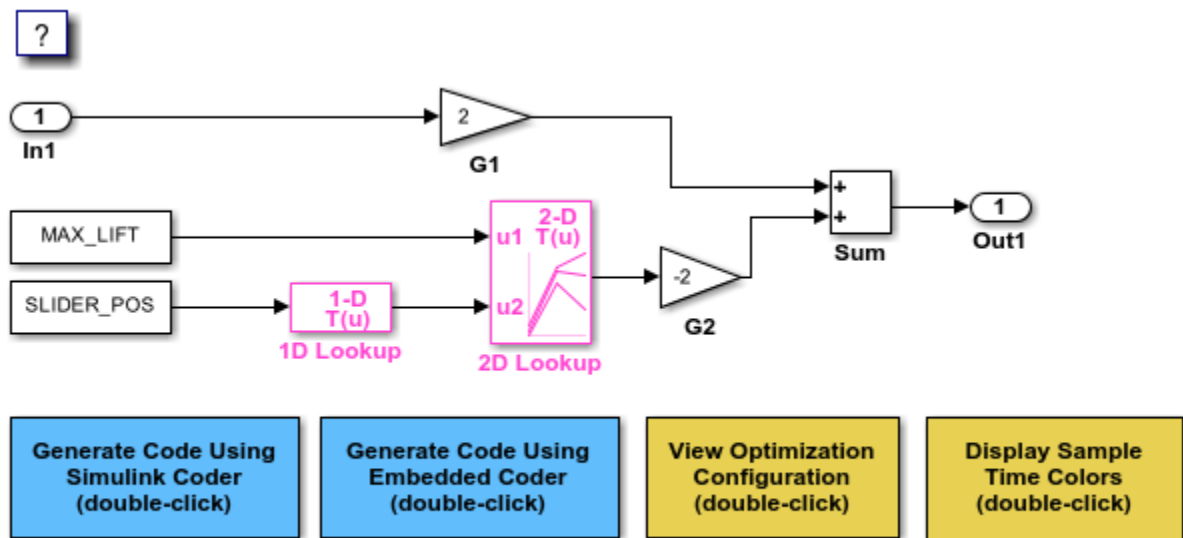
This optimization determines whether numeric block parameters occupy global memory in the generated code. The optimization can:

- Improve execution speed.
- Reduce RAM and ROM consumption.

Explore Example Model

Open the example model `rtwdemo_paraminline` and configure it to show the generated names of blocks.

```
load_system('rtwdemo_paraminline')
set_param('rtwdemo_paraminline','HideAutomaticNames','off')
open_system('rtwdemo_paraminline')
```



Copyright 1994-2020 The MathWorks, Inc.

The model contains blocks that have these numeric parameters:

- The **Gain** parameters of the Gain blocks
- The **Constant value** parameters of the Constant blocks
- The table data and breakpoint sets of the n-D Lookup Table blocks

The output of the block G2, and the outputs of blocks upstream of G2, change only if you tune the values of the block parameters during simulation or during code execution. When you update the model diagram, these blocks and signal lines appear magenta in color.

Several blocks use `Simulink.Parameter` objects in the base workspace to set the values of their parameters. The parameter objects all use the storage class `Auto`, which means that you can configure the generated code to inline the parameter values.

Generate Code Without Optimization

Create a temporary folder for the build and inspection process.

```
currentDir = pwd;
[~,cgDir] = rtwdemodir();
```

Disable the optimization by setting **Configuration Parameters > Default parameter behavior** to **Tunable**.

```
set_param('rtwdemo_paraminline','DefaultParameterBehavior','Tunable')
```

Generate code from the model.

```
slbuild('rtwdemo_paraminline')
```

```
### Starting build procedure for: rtwdemo_paraminline
### Successful completion of build procedure for: rtwdemo_paraminline
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_paraminline	Code generated and compiled.	Code generation information file does not exist

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 12.351s
```

In the code generation report, view the source file `rtwdemo_paraminline_data.c`. The code defines a global structure that contains the block parameter values. Each block parameter in the model, such as a lookup table array, breakpoint set, or gain, appears as a field of the structure.

```
cfile = fullfile(...
    cgDir,'rtwdemo_paraminline_grt_rtw','rtwdemo_paraminline_data.c');
rtwdemodbtype(cfile,'/* Block parameters (default storage) */','}',1,1);
```

```
/* Block parameters (default storage) */
P_rtwdemo_paraminline_T rtwdemo_paraminline_P = {
  /* Variable: MAX_LIFT
   * Referenced by: '<Root>/Constant'
   */
  10.0,

  /* Variable: SLIDER_POS
   * Referenced by: '<Root>/Constant1'
   */
  0.0,

  /* Variable: T1Break
   * Referenced by: '<Root>/1D Lookup'
   */
```

```

{ -5.0, -4.0, -3.0, -2.0, -1.0, 0.0, 1.0, 2.0, 3.0, 4.0, 5.0 },

/* Variable: T1Data
 * Referenced by: '<Root>/1D Lookup'
 */
{ -1.0, -0.99, -0.98, -0.96, -0.76, 0.0, 0.76, 0.96, 0.98, 0.99, 1.0 },

/* Variable: T2Break
 * Referenced by: '<Root>/2D Lookup'
 */
{ 1.0, 2.0, 3.0 },

/* Variable: T2Data
 * Referenced by: '<Root>/2D Lookup'
 */
{ 4.0, 16.0, 10.0, 5.0, 19.0, 18.0, 6.0, 20.0, 23.0 },

/* Expression: 2
 * Referenced by: '<Root>/G1'
 */
2.0,

/* Expression: -2
 * Referenced by: '<Root>/G2'
 */
-2.0,

/* Computed Parameter: uDLookup_maxIndex
 * Referenced by: '<Root>/2D Lookup'
 */
{ 2U, 2U }
};

```

You can tune the structure fields during code execution because they occupy global memory. However, at each step of the generated algorithm, the code must calculate the output of each block, including the outputs of the block G2 and the upstream blocks. View the algorithm in the model step function in the file `rtwdemo_paraminline.c`.

```

cfile = fullfile(cgDir, 'rtwdemo_paraminline_grt_rtw', 'rtwdemo_paraminline.c');
rtwdemodbtype(...
    cfile, /* Model step function */, /* Model initialize function */, 1, 0);

/* Model step function */
void rtwdemo_paraminline_step(void)
{
    /* Output: '<Root>/Out1' incorporates:
     * Constant: '<Root>/Constant'
     * Constant: '<Root>/Constant1'
     * Gain: '<Root>/G1'
     * Gain: '<Root>/G2'
     * Inport: '<Root>/In1'
     * Lookup_n-D: '<Root>/1D Lookup'
     * Lookup_n-D: '<Root>/2D Lookup'
     * Sum: '<Root>/Sum'
     */
    rtwdemo_paraminline_Y.Out1 = rtwdemo_paraminline_P.G1_Gain *
        rtwdemo_paraminline_U.In1 + rtwdemo_paraminline_P.G2_Gain * look2_binlx

```

```

    (rtwdemo_paraminline_P.MAX_LIFT, look1_binlx
    (rtwdemo_paraminline_P.SLIDER_POS, rtwdemo_paraminline_P.T1Break,
    rtwdemo_paraminline_P.T1Data, 10U), rtwdemo_paraminline_P.T2Break,
    rtwdemo_paraminline_P.T2Break, rtwdemo_paraminline_P.T2Data,
    rtwdemo_paraminline_P.udLookup_maxIndex, 3U);
}

```

Generate Code with Optimization

Set **Default parameter behavior** to Inlined.

```
set_param('rtwdemo_paraminline','DefaultParameterBehavior','Inlined')
```

Generate code from the model.

```
slbuild('rtwdemo_paraminline')
```

```
### Starting build procedure for: rtwdemo_paraminline
### Successful completion of build procedure for: rtwdemo_paraminline
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_paraminline	Code generated and compiled.	Global variable MAX_LIFT changed.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 9.6319s

In the code generation report, view the algorithm in the file `rtwdemo_paraminline.c`.

```

rtwdemodbtype(...
    cfile, /* Model step function */, /* Model initialize function */, 1, 0);

/* Model step function */
void rtwdemo_paraminline_step(void)
{
    /* Output: '<Root>/Out1' incorporates:
    * Gain: '<Root>/G1'
    * Inport: '<Root>/In1'
    * Sum: '<Root>/Sum'
    */
    rtwdemo_paraminline_Y.Out1 = 2.0 * rtwdemo_paraminline_U.In1 + 150.0;
}

```

The code does not allocate memory for block parameters or for parameter objects that use the storage class `Auto`. Instead, the code generator uses the parameter values from the model, and from the parameter objects, to calculate and inline the constant output of the block `G2`, `150.0`. The generator also inlines the value of the **Gain** parameter of the Gain block `G1`, `2.0`.

With the optimization, the generated code leaves out computationally expensive algorithmic code for blocks such as the lookup tables. The optimized code calculates the output of a block only if the output can change during execution. For this model, only the outputs of the Inport block `In1`, the Gain block `G1`, and the Sum block can change.

Close the model and the code generation report.

```
bdclose('rtwdemo_paraminline')  
rtwdemoclean;  
cd(currentDir)
```

Preserve Block Parameter Tunability

When you set **Default parameter behavior** to **Inlined**, you can preserve block parameter tunability by creating `Simulink.Parameter` objects for individual parameters. You can configure each object to appear in the code as a tunable field of the global parameter structure or as an individual global variable. You can change parameter values during code execution and interface the generated code with your own handwritten code. For more information, see “Create Tunable Calibration Parameter in the Generated Code” on page 15-112.

Inline Invariant Signals

You can select the **Inline invariant signals** code generation option (which also places constant values in the generated code) only when you set **Default parameter behavior** to **Inlined**. See “Inline Invariant Signals” on page 40-38.

See Also

Default parameter behavior

Related Examples

- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

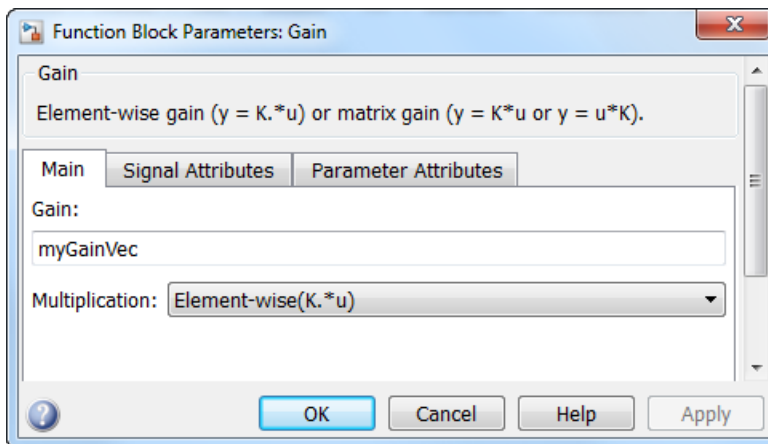
Configure Loop Unrolling Threshold

The **Loop unrolling threshold** parameter on the **Optimization** pane determines when a wide signal or parameter should be wrapped into a `for` loop and when it should be generated as a separate statement for each element of the signal. The default threshold value is 5.

For example, consider the model below:



The gain parameter of the Gain block is the vector `myGainVec`.



Assume that the loop unrolling threshold value is set to the default, 5.

If `myGainVec` is declared as

```
myGainVec = [1:10];
```

an array of 10 elements, `myGainVec_P.Gain_Gain[]`, is declared within the `Parameters_model` data structure. The size of the gain array exceeds the loop unrolling threshold. Therefore, the code generated for the Gain block iterates over the array in a `for` loop, as shown in the following code:

```
{
    int32_T i1;

    /* Gain: '<Root>/Gain' */
    for(i1=0; i1<10; i1++) {
        myGainVec_B.Gain_f[i1] = rtb_foo *
            myGainVec_P.Gain_Gain[i1];
    }
}
```

If `myGainVec` is declared as

```
myGainVec = [1:3];
```


an array of three elements, `myGainVec_P.Gain_Gain[]`, is declared within the `Parameters` data structure. The size of the gain array is below the loop unrolling threshold. The generated code consists of inline references to each element of the array, as in the code below.

```
/* Gain: '<Root>/Gain' */
myGainVec_B.Gain_f[0] = rtb_foo * myGainVec_P.Gain_Gain[0];
myGainVec_B.Gain_f[1] = rtb_foo * myGainVec_P.Gain_Gain[1];
myGainVec_B.Gain_f[2] = rtb_foo * myGainVec_P.Gain_Gain[2];
```

See “Explore Variable Names and Loop Rolling” for more information on loop rolling.

Note When a model includes Stateflow charts or MATLAB Function blocks, you can apply a set of Stateflow optimizations on the **Optimization** pane. The settings you select for the Stateflow options also apply to MATLAB Function blocks in the model. This is because the MATLAB Function blocks and Stateflow charts are built on top of the same technology and share a code base. You do not need a Stateflow license to use MATLAB Function blocks.

Note If your MATLAB Function block contains code with two nested `for`-loops:

- If the number of iterations of the inner loop is less than the threshold, the code generator first unrolls the inner loop. Subsequently, if the product of the number of iterations of the two loops is also less than the threshold, the code generator unrolls the outer loop. Otherwise the code generator produces the outer `for`-loop.
 - If the number of iterations of the inner loop is equal to or greater than the threshold, the code generator produces both `for`-loops
 - For an N-Dimensional array the code generator unrolls each contiguous loop until the product of loop iterations is less than the threshold value.
-

See Also

Loop unrolling threshold

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Optimize Generated Code by Combining Multiple `for` Constructs” on page 40-13
- “For Loop” (Embedded Coder)

Use memcpy Function to Optimize Generated Code for Vector Assignments

In this section...

“Example Model” on page 40-48

“Generate Code” on page 40-49

“Generate Code with Optimization” on page 40-50

You can use the **Use memcpy for vector assignment** parameter to optimize generated code for vector assignments by replacing `for` loops with `memcpy` function calls. The `memcpy` function is more efficient than `for`-loop controlled element assignment for large data sets. This optimization improves execution speed.

Selecting the **Use memcpy for vector assignment** parameter enables the associated parameter **Memcpy threshold (bytes)**, which allows you to specify the minimum array size in bytes for which `memcpy` function calls should replace `for` loops in the generated code. For more information, see [Use memcpy for vector assignment and Memcpy threshold \(bytes\)](#). In considering whether to use this optimization,

- Verify that your target supports the `memcpy` function.
- Determine whether your model uses signal vector assignments (such as `Y=expression`) to move large amounts of data, for example, using the Selector block.

To apply this optimization,

- 1 Consider first generating code without this optimization and measuring its execution speed, to establish a baseline for evaluating the optimized assignment.
- 2 Select **Use memcpy for vector assignment** and examine the setting of **Memcpy threshold (bytes)**, which by default specifies 64 bytes as the minimum array size for which `memcpy` function calls replace `for` loops. Based on the array sizes used in your application's signal vector assignments, and target environment considerations that might bear on the threshold selection, accept the default or specify another array size.
- 3 Generate code, and measure its execution speed against your baseline or previous iterations. Iterate on steps 2 and 3 until you achieve an optimal result.

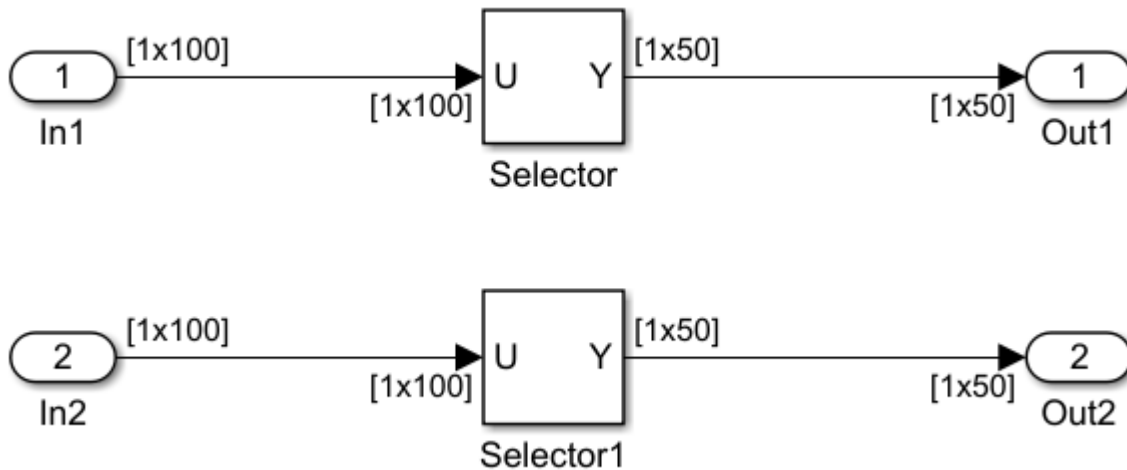
Note The `memcpy` optimization may not occur under certain conditions, including when other optimizations have a higher precedence than the `memcpy` optimization, or when the generated code is originating from Target Language Compiler (TLC) code, such as a TLC file associated with an S-function block.

Note If you are licensed for Embedded Coder software, you can use a code replacement library (CRL) to provide your own custom implementation of the `memcpy` function to be used in generated model code. For more information, see [“Memory Function Code Replacement”](#) (Embedded Coder).

Example Model

To examine the result of using the **Use memcpy for vector assignment** parameter on the generated vector assignment code, create a model that generates signal vector assignments. For example,

- 1 Use In, Out, and Selector blocks to create the following model.



- 2 Open Model Explorer and configure the **Signal Attributes** for the In1 and In2 source blocks. For each, set **Port dimensions** to [1, 100], and set **Data type** to int32. Apply the changes and save the model. In this example, the model has the name vectorassign.
- 3 For each Selector block, set the **Index** parameter to 1:50. Set the **Input port size** parameter to 100.

Generate Code

- 1 The **Use memcpy for vector assignment** parameter is on by default. To turn off the parameter, go to the **Optimization** pane and clear the **Use memcpy for vector assignment** parameter.
- 2 Go to the **Code Generation > Report** pane of the Configuration Parameters dialog box and select the **Create code generation report** parameter and the **Open report automatically** parameter. Then go to the **Code Generation** pane, select the **Generate code only** option, and generate code for the model. When code generation completes, the HTML code generation report is displayed.
- 3 In the HTML code generation report, click the vectorassign.c section and inspect the model step function. Notice that the vector assignments are implemented using for loops.

```

/* Model step function */
void vectorassign_step(void)
{
    int32_T i;
    for (i = 0; i < 50; i++) {
        /* Output: '<Root>/Out1' incorporates:
         * Inport: '<Root>/In1'
         */
        vectorassign_Y.Out1[i] = vectorassign_U.In1[i];

        /* Output: '<Root>/Out2' incorporates:
         * Inport: '<Root>/In2'
         */
        vectorassign_Y.Out2[i] = vectorassign_U.In2[i];
    }
}

```

Generate Code with Optimization

- 1 Go to the **Optimization** pane of the Configuration Parameters dialog box and select the **Use memcpy for vector assignment** option. Leave the **Memcpy threshold (bytes)** option at its default setting of 64. Apply the changes and regenerate code for the model. When code generation completes, the HTML code generation report again is displayed.
- 2 In the HTML code generation report, click the `vectorassign.c` section and inspect the model output function. Notice that the vector assignments now are implemented using `memcpy` function calls.

```
/* Model step function */
void vectorassign_step(void)
{
    /* Output: '<Root>/Out1' incorporates:
     * Inport: '<Root>/In1'
     */
    memcpy(&vectorassign_Y.Out1[0], &vectorassign_U.In1[0], 50U * sizeof(real_T));

    /* Output: '<Root>/Out2' incorporates:
     * Inport: '<Root>/In2'
     */
    memcpy(&vectorassign_Y.Out2[0], &vectorassign_U.In2[0], 50U * sizeof(real_T));
}
```

See Also

Use `memcpy` for vector assignment

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Vector Operation Optimization” on page 40-99
- “Convert Data Copies to Pointer Assignments” (Embedded Coder)

Generate Target Optimizations Within Algorithm Code

Some application components are hardware-specific and cannot simulate on a host system. For example, consider a component that includes pragmas and assembly code for enabling hardware instructions for saturate on add operations or a Fast Fourier Transform (FFT) function.

The following table lists integration options to customize generated algorithm code with target-specific optimizations.

Note Solutions marked with *EC only* require an Embedded Coder license.

If...	Then...	For More Information, See
You want to optimize the execution speed and memory of the model code by replacing default math functions and operators with target-specific code	<i>EC only</i> —Implement function and operator replacements by using the Code Replacement Tool , code replacement library (CRL) API, and Code Replacement Viewer to create, examine, validate, and register hardware-specific replacement tables	“Optimize Generated Code by Developing and Using Code Replacement Libraries - Simulink” (Embedded Coder)
You want to control how code generation technology declares, stores, and represents signals, tunable parameters, block states, and data objects in generated code	<i>EC only</i> —Design (create) and apply custom storage classes	<ul style="list-style-type: none"> • rtwdemo_cscpredef • rtwdemo_importstruct • rtwdemo_configinterface • “Organize Parameter Data into a Structure by Using Struct Storage Class” (Embedded Coder)

Note To simulate an algorithm that includes target-specific elements in a host environment, you must create code that is equivalent to the target code and can run in the host environment.

Remove Code for Blocks That Have No Effect on Computational Results

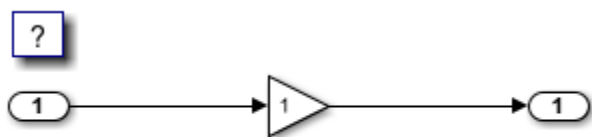
This example shows how the code generator optimizes generated code by removing code that has no effect on computational results. This optimization:

- Increases execution speed.
- Reduces ROM consumption.

Example

In the model `rtwdemo_blockreduction`, a Gain block of value `1.0` is in between Inport and Output blocks.

```
model = 'rtwdemo_blockreduction';
open_system(model);
```



Copyright 2014-2020 The MathWorks, Inc.

Generate Code

Create a temporary folder for the build and inspection process.

```
currentDir=pwd;
[~,cgDir]=rtwdemodir();
```

Build the model.

```
set_param(model, 'BlockReduction', 'off');
slbuild(model)
```

```
### Starting build procedure for: rtwdemo_blockreduction
### Successful completion of build procedure for: rtwdemo_blockreduction
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_blockreduction	Code generated and compiled.	Code generation information file does not e

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.655s
```

This code is the code from `rtwdemo_blockreduction.c`.

```

cfile = fullfile(cgDir,'rtwdemo_blockreduction_ert_rtw','rtwdemo_blockreduction.c');
rtwdemodbtype(cfile, '/* Model step function */',...
    '/* Model initialize function */', 1, 0);

/* Model step function */
void rtwdemo_blockreduction_step(void)
{
    /* Outport: '<Root>/Out1' incorporates:
    * Gain: '<Root>/Gain'
    * Inport: '<Root>/In1'
    */
    rtwdemo_blockreduction_Y.Out1 = 1.0 * rtwdemo_blockreduction_U.In1;
}

```

Enable Optimization

- 1 Open the Configuration Parameters dialog box.
- 2 If the **Block reduction** check box is not selected, select it.

Alternatively, use the command-line API to enable the optimization.

```
set_param(model, 'BlockReduction', 'on');
```

Generate Code with Optimization

```
slbuild(model)
```

```

### Starting build procedure for: rtwdemo_blockreduction
### Successful completion of build procedure for: rtwdemo_blockreduction

```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_blockreduction	Code generated and compiled.	Generated code was out of date.

1 of 1 models built (0 models already up to date)

Build duration: 0h 0m 9.7152s

Here is the optimized code from `rtwdemo_blockreduction.c`.

```

cfile = fullfile(cgDir,'rtwdemo_blockreduction_ert_rtw','rtwdemo_blockreduction.c');
rtwdemodbtype(cfile, '/* Model step function */',...
    '/* Model initialize function */', 1, 0);

/* Model step function */
void rtwdemo_blockreduction_step(void)
{
    /* Outport: '<Root>/Out1' incorporates:
    * Inport: '<Root>/In1'
    */
    rtwdemo_blockreduction_Y.Out1 = rtwdemo_blockreduction_U.In1;
}

```

Because multiplying the input signal by a value of 1.0 does not impact computational results, the code generator excludes the Gain block from the generated code. Close the model and clean up.

```
bdclose(model)
rtwdemoclean;
cd(currentDir)
```

Limitations

The code generator cannot remove blocks under these circumstances:

- The block output is configured as a test point or has a storage class and the block input and output have different data types, complexities, or widths.
- The block has a nonconstant sample time, has an output connected to a root output, and has a source that has a constant sample time.
- The block has multiple sources, some of which have constant sample times and some of which have nonconstant sample times.
- The block output has a storage class and has a source that is muxed, is a virtual bus, is connected to a root output, is configured as a test point, or has a storage class.
- The block has an output that is connected to the block input.
- The block is inside an Initialize Function subsystem, Reset Function subsystem, or Terminate Function subsystem whose output is connected to an output of the subsystem.
- The block has an output that is connected to a Subsystem Output block that has an initial condition.
- The block has an output that is connected to an output of an Initialize Function subsystem, Reset Function subsystem, or Terminate Function subsystem.
- The block has an output that is connected to an output for which the parameter **Ensure output is virtual** is selected.
- The block has an output that is connected to an output that has an initial condition parameter.
- The block has an output that is connected to a Merge, a Vector Concatenate, or a Matrix Concatenate block.
- The block has tunable run-time parameters.
- The block is inside a reusable subsystem whose output is connected to a block outside the reusable subsystem.
- The block has a continuous state.
- The block has a discrete state and the states are logged or have a nonauto storage class.
- The block is an S-Function, unless you use `ssSetBlockReduction` to enable block reduction.
- The block output is configured as a test point or has a storage class and its source is a block inside a different conditional subsystem.

See Also

“Block reduction”

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Inline Numeric Values of Block Parameters” on page 40-41

- “Minimize Computations and Storage for Intermediate Results at Block Outputs” on page 40-35

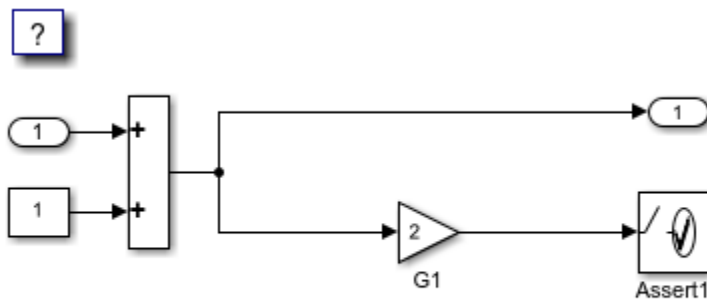
Eliminate Dead Code Paths in Generated Code

This example shows how the code generator eliminates dead (that is, unused) code paths from generated code. This optimization increases execution speed and conserves ROM and RAM consumption.

Example

In the `DeadPathElimination` model, the signal leaving the Sum block divides into two separate code paths. The top path is not a dead code path. If the user disables the Assertion block, the bottom path becomes a dead code path.

```
model = 'DeadPathElimination';
open_system(model);
```



Copyright 2019-2020 The MathWorks, Inc.

Generate Code with an Enabled Assertion Block

- 1 For the Assertion block, open the block parameters dialog box.
- 2 Select the **Enable assertion** box. Alternatively, use the command-line API to enable the Assertion block.

```
set_param([model '/Assert1'], 'Enabled', 'on');
```

Build the model.

```
slbuild(model)
```

```
### Starting build procedure for: DeadPathElimination
### Successful completion of build procedure for: DeadPathElimination
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
DeadPathElimination	Code generated and compiled.	Code generation information file does not exist.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 14.78s
```

Because the Assertion block is enabled, these lines of `DeadPathElimination.c` include code for the Gain and Assertion blocks.

```
cfile = fullfile('DeadPathElimination_grt_rtw','DeadPathElimination.c');
rtwdemodbtype(cfile,'/* Model step', '/* Model initialize function */', 0, 1);

void DeadPathElimination_step(void)
{
    /* Sum: '<Root>/Sum1' incorporates:
     * Constant: '<Root>/Constant1'
     * Inport: '<Root>/In1'
     */
    DeadPathElimination_Y.Out1 = DeadPathElimination_U.In1 + 1.0;

    /* Assertion: '<Root>/Assert1' incorporates:
     * Gain: '<Root>/G1'
     */
    utAssert(2.0 * DeadPathElimination_Y.Out1 != 0.0);
}
```

Generate Code with a Disabled Assertion Block

Disable the Assertion block to generate a dead code path. The code generator detects the dead code path and eliminates it from the generated code.

- 1 For the Assertion block, open the Block Parameters dialog box.
- 2 Deselect the **Enable assertion** box.

Alternatively, use the command-line API to disable the Assertion block.

```
set_param([model '/Assert1'], 'Enabled', 'off');
```

Build the model.

```
slbuild(model)

### Starting build procedure for: DeadPathElimination
### Successful completion of build procedure for: DeadPathElimination

Build Summary

Top model targets built:

Model                Action                Rebuild Reason
=====
DeadPathElimination  Code generated and compiled.  Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.738s
```

Because the Assertion block is disabled, these lines of `DeadpathElim.c` do not include code for the Gain and Assertion blocks.

```
rtwdemodbtype(cfile,'/* Model step', '/* Model initialize function */', 0, 1);

void DeadPathElimination_step(void)
```

```
{
  /* Outport: '<Root>/Out1' incorporates:
   * Constant: '<Root>/Constant1'
   * Inport: '<Root>/In1'
   * Sum: '<Root>/Sum1'
   */
  DeadPathElimination_Y.Out1 = DeadPathElimination_U.In1 + 1.0;
}
```

Close the Model and Clean Up

```
bdclose(model)
rtwdemoclean;
```

For another example of how the code generator eliminates dead code paths in the generated code, see `rtwdemo_deadpath`.

See Also

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Remove Code for Blocks That Have No Effect on Computational Results” on page 40-52

Floating-Point Multiplication to Handle a Net Slope Correction

This example shows how to use floating-point multiplication to handle a net slope correction. When converting floating-point data types to fixed-point data types in the generated code, a net slope correction is one method of scaling fixed-point data types. Scaling the fixed-point data types avoids overflow conditions and minimizes quantization errors.

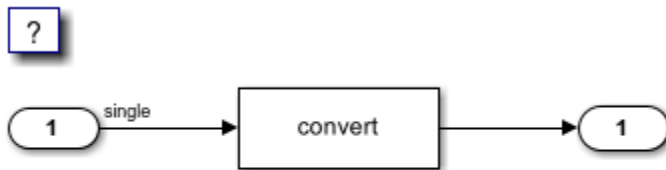
For processors that support efficient multiplication, using floating-point multiplication to handle a net slope correction improves code efficiency. If the net slope correction has a value that is not a power of two, using division improves precision.

Note: This example requires a Fixed-Point Designer™ license.

Example

In the model `FloatMultiplicationNetSlope`, a Convert block converts an input signal from a floating-point data type to a fixed-point data type. The net slope correction has a value of 3.

```
model = 'FloatMultiplicationNetSlope';
open_system(model);
```



Copyright 2022 The MathWorks, Inc.

Generate Code

Build the model.

```
slbuild(model)
```

```
### Starting build procedure for: FloatMultiplicationNetSlope
### Successful completion of build procedure for: FloatMultiplicationNetSlope
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
FloatMultiplicationNetSlope	Code generated and compiled.	Code generation information file does

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.223s
```

In these lines of `FloatMultiplicationNetSlope.c` code, the code generator divides the input signal by $3.0F$.

```

cfile = fullfile('FloatMultiplicationNetSlope_grt_rtw',...
    'FloatMultiplicationNetSlope.c');
rtwdemodbtype(cfile,'/* Model step', '/* Model initialize function */', 1, 0);

/* Model step function */
void FloatMultiplicationNetSlope_step(void)
{
    /* Outport: '<Root>/Output' incorporates:
     * DataTypeConversion: '<Root>/Data Type Conversion'
     * Inport: '<Root>/Input'
     */
    FloatMultiplicationNetSlope_Y.Output = (int16_T)(real32_T)floor
        (FloatMultiplicationNetSlope_U.Input / 3.0F);
}

```

Enable Optimization

- 1 Open the Configuration Parameters dialog box.
- 2 On the **Math and Data Types** pane, select **Use floating-point multiplication to handle net slope corrections**. This optimization is on by default.

Alternatively, you can use the command-line API to enable the optimization.

```
set_param(model, 'UseFloatMulNetSlope', 'on');
```

Generate Code with Optimization

```

slbuild(model)

### Starting build procedure for: FloatMultiplicationNetSlope
### Successful completion of build procedure for: FloatMultiplicationNetSlope

```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
FloatMultiplicationNetSlope	Code generated and compiled.	Generated code was out of date.

```

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 9.6138s

```

In the optimized code, the code generator multiplies the input signal by the reciprocal of 3.0F, that is 0.333333343F.

```

rtwdemodbtype(cfile,'/* Model step', '/* Model initialize function */', 1, 0);

/* Model step function */
void FloatMultiplicationNetSlope_step(void)
{
    /* Outport: '<Root>/Output' incorporates:
     * DataTypeConversion: '<Root>/Data Type Conversion'
     * Inport: '<Root>/Input'
     */
    FloatMultiplicationNetSlope_Y.Output = (int16_T)(real32_T)floor

```

```
    (FloatMultiplicationNetSlope_U.Input * 0.333333343F);  
}
```

Close the model and the code generation report.

```
bdclose(model)  
rtwdemoclean;
```

See Also

“Use floating-point multiplication to handle net slope corrections”

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Remove Code for Out-of-Range Floating Point to Integer Conversions” on page 40-20
- “Subnormal Number Execution Speed” on page 40-16

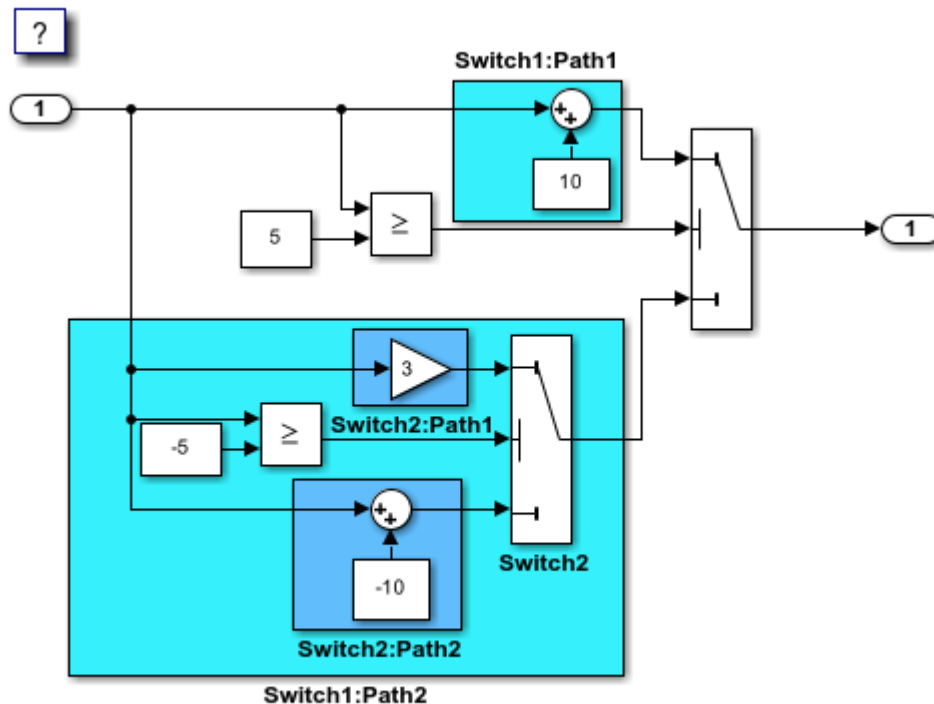
Use Conditional Input Branch Execution

This example shows how to optimize the generated code for a model that contains Switch and Multiport Switch blocks. When you select the model configuration parameter **Conditional input branch execution**, Simulink® executes only blocks that compute the control input and data input that the control input selects. This optimization improves execution speed.

Example Model

In this example, switch paths are conditionally executed. If Switch1 control input is true, Switch1 executes blocks grouped in the Switch1:Path1 branch. If Switch1 control input is false, Switch1 executes blocks grouped in the Switch1:Path2 branch. If Switch1 executes blocks in the Switch1:Path2 branch and Switch2 control input is true, Switch2 executes blocks in the Switch2:Path1 branch. If Switch2 control input is false, Switch2 executes blocks in the Switch2:Path2 branch. The pseudo code shows this logic.

```
model='ConditionalInput';
open_system(model);
```



Copyright 1994-2020 The MathWorks, Inc.

Generate Code

The **Conditional input branch execution** parameter is on by default. Enter the following command-line API to turn off the parameter.

```
set_param(model, 'ConditionallyExecuteInputs', 'off');
```


Build the model

```
slbuild(model)
```

```
### Starting build procedure for: ConditionalInput
### Successful completion of build procedure for: ConditionalInput
```

```
Build Summary
```

```
Top model targets built:
```

Model	Action	Rebuild Reason
ConditionalInput	Code generated and compiled.	Code generation information file does not exist.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.486s
```

View the generated code without the optimization. These lines of code are in the `ConditionalInput.c` file.

```
cfile = fullfile('ConditionalInput_grt_rtw','ConditionalInput.c');
rtwdemodbtype(cfile,'/* Model step', '/* Model initialize', 1, 0);

/* Model step function */
void ConditionalInput_step(void)
{
    /* Switch: '<Root>/ Switch2' incorporates:
     * Constant: '<Root>/C_5'
     * Inport: '<Root>/input'
     * RelationalOperator: '<Root>/Relational Operator'
     */
    if (ConditionalInput_U.input >= -5.0) {
        /* Switch: '<Root>/Switch1' incorporates:
         * Gain: '<Root>/ G3'
         */
        ConditionalInput_Y.output = 3.0 * ConditionalInput_U.input;
    } else {
        /* Switch: '<Root>/Switch1' incorporates:
         * Sum: '<Root>/ Sum'
         */
        ConditionalInput_Y.output = ConditionalInput_U.input - 10.0;
    }

    /* End of Switch: '<Root>/ Switch2' */

    /* Switch: '<Root>/Switch1' incorporates:
     * Constant: '<Root>/C5'
     * Inport: '<Root>/input'
     * RelationalOperator: '<Root>/Relational Operator1'
     */
    if (ConditionalInput_U.input >= 5.0) {
        /* Switch: '<Root>/Switch1' incorporates:
         * Constant: '<Root>/ C10'
         * Outport: '<Root>/output'
         * Sum: '<Root>/ Sum1'
         */
    }
}
```

```

    ConditionalInput_Y.output = ConditionalInput_U.input + 10.0;
}

/* End of Switch: '<Root>/Switch1' */
}

```

The generated code contains an `if-else` statement for the `Switch2` block and an `if` statement for the `Switch1` block. Therefore, the generated code for `Switch1:Path2` executes even if the `if` statement for `Switch1:Path1` evaluates to true.

Enable Optimization

- 1 Open the Configuration Parameters dialog box.
- 2 Select the **Conditional input branch execution** parameter. Alternatively, you can use the command-line API to enable the optimization.

```
set_param(model, 'ConditionallyExecuteInputs', 'on');
```

Generate Code with Optimization

```
slbuild(model)
cfile = fullfile('ConditionalInput_grt_rtw', 'ConditionalInput.c');
rtwdemodbtype(cfile, '/* Model step', '/* Model initialize', 1, 0);
```

```
### Starting build procedure for: ConditionalInput
### Successful completion of build procedure for: ConditionalInput
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
ConditionalInput	Code generated and compiled.	Generated code was out of date.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 9.6076s
```

```

/* Model step function */
void ConditionalInput_step(void)
{
    /* Switch: '<Root>/Switch1' incorporates:
     * Constant: '<Root>/C5'
     * Constant: '<Root>/C_5'
     * Inport: '<Root>/input'
     * RelationalOperator: '<Root>/Relational Operator'
     * RelationalOperator: '<Root>/Relational Operator1'
     * Switch: '<Root>/ Switch2'
     */
    if (ConditionalInput_U.input >= 5.0) {
        /* Output: '<Root>/output' incorporates:
         * Constant: '<Root>/ C10'
         * Sum: '<Root>/ Sum1'
         */
        ConditionalInput_Y.output = ConditionalInput_U.input + 10.0;
    } else if (ConditionalInput_U.input >= -5.0) {
        /* Switch: '<Root>/ Switch2' incorporates:
         * Gain: '<Root>/ G3'

```

```
    * Output: '<Root>/output'
    */
    ConditionalInput_Y.output = 3.0 * ConditionalInput_U.input;
} else {
    /* Output: '<Root>/output' incorporates:
    * Sum: '<Root>/ Sum'
    * Switch: '<Root>/ Switch2'
    */
    ConditionalInput_Y.output = ConditionalInput_U.input - 10.0;
}

/* End of Switch: '<Root>/Switch1' */
}
```

The generated code contains one `if` statement. The generated code for `Switch1:Path2` only executes if the `if` statement evaluates to false.

Close Model and Code Generation Report

```
bdclose(model)
rtwdemoclean;
```

See Also

“Conditional input branch execution” | Switch | Multiport Switch

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Eliminate Dead Code Paths in Generated Code” on page 40-56

Optimize Generated Code for Complex Signals

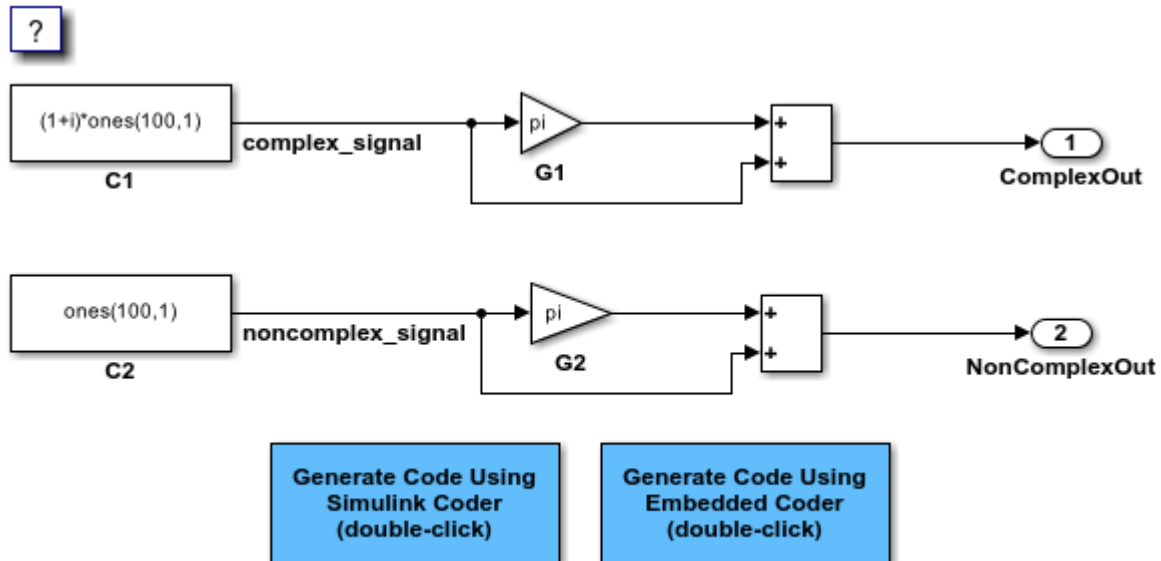
This example shows how Simulink® Coder™ handles complex signals efficiently. To view the data types of the signals, update the model. On the **Debug** menu, select **Update Model**. Complex signals are represented as structures in generated code. Simulink Coder performs various optimizations on these structures. For example:

- Expression Folding: Gain and Sum operations on the complex signal are folded into a single expression.
- For-loop fusion: Two separate for-loops, one for the complex signal and one for noncomplex signal, are combined into a single for-loop.
- Inlined block parameters: The value of Gain block "pi" is inlined in the expression of the complex Gain-Sum.

Because of optimizations such as these, the code generated for complex and noncomplex signals is equally efficient.

Example Model

```
model='rtwdemo_complex';
open_system(model);
```



Copyright 1994-2020 The MathWorks, Inc.

See Also

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Minimize Computations and Storage for Intermediate Results at Block Outputs” on page 40-35

- “Disable Nonfinite Checks or Inlining for Math Functions” on page 40-26

Speed Up Linear Algebra in Code Generated from a MATLAB Function Block

To improve the execution speed of code generated for certain linear algebra functions in a MATLAB Function block, specify that the code generator produce LAPACK calls. LAPACK is a software library for numerical linear algebra. The code generator uses the LAPACKE C interface to LAPACK. If you specify that you want to generate LAPACK calls, and the input arrays for the linear algebra functions meet certain criteria, the code generator produces the LAPACK calls. Otherwise, the code generator produces code for the linear algebra functions.

The code generator uses the LAPACK library that you specify. Specify a LAPACK library that is optimized for your execution environment. See netlib.org/lapack/faq.html#_what_and_where_are_the_lapack_vendors_implementations.

Specify LAPACK Library

To generate LAPACK calls, you must have access to a LAPACK callback class. A LAPACK callback class specifies the LAPACK library and LAPACKE header file for the LAPACK calls. To indicate that you want to generate LAPACK calls and that you want to use a specific LAPACK library, specify the name of the LAPACK callback class. In the Configuration Parameters dialog box, set **Custom LAPACK library callback** to the name of the callback class, for example, useMyLAPACK.

Write LAPACK Callback Class

To specify the locations of a particular LAPACK library and LAPACKE header file, write a LAPACK callback class. Share the callback class with others who want to use this LAPACK library for LAPACK calls in generated code.

The callback class must derive from the abstract class `coder.LAPACKCallback`. Use this example callback class as a template.

```
classdef useMyLAPACK < coder.LAPACKCallback
    methods (Static)
        function hn = getHeaderFilename()
            hn = 'mylapacke_custom.h';
        end
        function updateBuildInfo(buildInfo, buildctx)
            buildInfo.addIncludePaths(fullfile(pwd,'include'));
            libName = 'mylapack';
            libPath = fullfile(pwd,'lib');
            [~,linkLibExt] = buildctx.getStdLibInfo();
            buildInfo.addLinkObjects([libName linkLibExt], libPath, ...
                '', true, true);
            buildInfo.addDefines('HAVE_LAPACK_CONFIG_H');
            buildInfo.addDefines('LAPACK_COMPLEX_STRUCTURE');
            buildInfo.addDefines('LAPACK_ILP64');
        end
    end
end
```

You must provide the `getHeaderFilename` and `updateBuildInfo` methods. The `getHeaderFilename` method returns the LAPACKE header file name. In the example callback class, replace `mylapacke_custom.h` with the name of your LAPACKE header file. The `updateBuildInfo`

method provides the information required for the build process to link to the LAPACK library. Use code like the code in the template to specify the location of header files and the full path name of the LAPACK library. In the example callback class, replace `mylapack` with the name of your LAPACK library.

If your compiler supports only complex data types that are represented as structures, include these lines in the `updateBuildInfo` method.

```
buildInfo.addDefines('HAVE_LAPACK_CONFIG_H');
buildInfo.addDefines('LAPACK_COMPLEX_STRUCTURE');
```

You must specify the integer type that your LAPACK library uses. Not specifying this integer type can result in incorrect behaviors or crashes. Do one of the following:

- Include these lines in the `updateBuildInfo` method.

```
buildInfo.addDefines('HAVE_LAPACK_CONFIG_H');
buildInfo.addDefines('LAPACK_ILP64');
```

- Alternatively, you can directly specify the integer type that your LAPACK library uses. For example, if the integer type is `long long`, include this line in the `updateBuildInfo` method.

```
buildInfo.addDefines('lapack_int=long long');
```

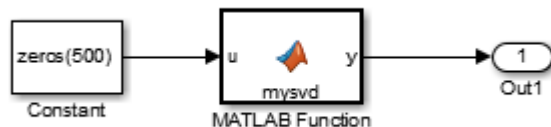
Generate LAPACK Calls by Specifying a LAPACK Callback Class

This example shows how to generate code that calls LAPACK functions in a specific LAPACK library. For this example, assume that the LAPACK callback class `useMyLAPACK` specifies the LAPACK library that you want.

- 1 Create a Simulink model.
- 2 Add a MATLAB Function block to the model.
- 3 In the MATLAB Function block, add code that calls a linear algebra function. For example, add the function `mysvd` that calls the MATLAB function `svd`.

```
function s = mysvd(A)
    %#codegen
    s = svd(A);
end
```

- 4 Add a Constant block to the left of the MATLAB Function block. Set the value to `zeros(500)`.
- 5 Add an Outport block to the right of the MATLAB Function block.
- 6 Connect the blocks.



- 7 Set the **Configuration Parameters > Code Generation > Advanced parameters > Custom LAPACK library callback** parameter to `useMyLAPACK`.

The callback class must be on the MATLAB path.

- 8 Build the model.

If the input to `mysvd` is large enough, the code generator produces a LAPACK call for `svd`. An example of a call to the LAPACK library function for `svd` is:

```
info_t = LAPACKE_dgesvd( ...  
    LAPACK_COL_MAJOR, 'N', 'N', (lapack_int)500, ...  
    (lapack_int)500, &A[0], (lapack_int)500, &S[0], ...  
    NULL, (lapack_int)1, NULL, (lapack_int)1, &superb[0]);
```

Locate LAPACK Library in Execution Environment

The LAPACK library must be available in your execution environment. If your LAPACK library is shared, use environment variables or linker options to specify the location of the LAPACK library.

- On a Windows platform, modify the `PATH` environment variable.
- On a Linux platform, modify the `LD_LIBRARY_PATH` environment variable or use the `rpath` linker option.
- On a macOS platform, modify the `DYLD_LIBRARY_PATH` environment variable or use the `rpath` linker option.

To specify the `rpath` linker option, you can use the build information `addLinkFlags` method in the `updateBuildInfo` method of your `coder.LAPACKCallback` class. For example, for a GCC compiler:

```
buildInfo.addLinkFlags(sprintf('-Wl,-rpath,"%s"', libPath));
```

See Also

`coder.LAPACKCallback`

More About

- “LAPACK Calls for Linear Algebra in a MATLAB Function Block”

External Websites

- www.netlib.org/lapack
- www.netlib.org/lapack/faq.html#_what_and_where_are_the_lapack_vendors_implementations

Speed Up Matrix Operations in Code Generated from a MATLAB Function Block

To improve the execution speed of code generated for certain low-level vector and matrix operations (such as matrix multiplication) in a MATLAB Function block, specify that you want the code generator to produce BLAS calls. BLAS is a software library for low-level vector and matrix computations that has several highly optimized machine-specific implementations. The code generator uses the CBLAS C interface to BLAS. If you specify that you want to generate BLAS calls, and the input arrays for the matrix functions meet certain criteria, the code generator produces the BLAS calls. Otherwise, the code generator produces code for the matrix functions.

The code generator uses the BLAS library that you specify. Specify a BLAS library that is optimized for your execution environment.

Specify BLAS Library

To produce BLAS calls in generated code, you must have access to a BLAS callback class. A BLAS callback class specifies the BLAS library, the CBLAS header file, certain C data types the particular CBLAS interface uses and the compiler and linker options for the build process.

To indicate that you want to generate BLAS calls and use a specific BLAS library, specify the name of the BLAS callback class. In the Configuration Parameters dialog box, set **Custom BLAS library callback** to the name of the callback class.

Write BLAS Callback Class

To generate calls to a specific BLAS library in the generated code, write a BLAS callback class. Share the callback class with others who want to use this BLAS library for BLAS calls in standalone code.

The callback class must derive from the abstract class `coder.BLASCallback`. This example is an implementation of the callback class `mklcallback` for integration with the Intel MKL BLAS library on a Windows platform.

```
classdef mklcallback < coder.BLASCallback
    methods (Static)
        function updateBuildInfo(buildInfo, ~)
            libPath = fullfile(pwd, 'mkl', 'WIN', 'lib', 'intel64');
            libPriority = '';
            libPreCompiled = true;
            libLinkOnly = true;
            libs = {'mkl_intel_ilp64.lib' 'mkl_intel_thread.lib' 'mkl_core.lib'};
            buildInfo.addLinkObjects(libs, libPath, libPriority, libPreCompiled, ...
                libLinkOnly);
            buildInfo.addLinkObjects('libiomp5md.lib', fullfile(matlabroot, 'bin', ...
                'win64'), libPriority, libPreCompiled, libLinkOnly);
            buildInfo.addIncludePaths(fullfile(pwd, 'mkl', 'WIN', 'include'));
            buildInfo.addDefines('-DMKL_ILP64');
        end
        function headerName = getHeaderFilename()
            headerName = 'mkl_cblas.h';
        end
        function intTypeName = getBLASIntTypeName()
            intTypeName = 'MKL_INT';
        end
    end
end
```

```

        end
    end
end

```

You must provide the `getHeaderFilename`, `getBLASIntTypeName`, and `updateBuildInfo` methods. The `getHeaderFilename` method returns the CBLAS header file name. If you are using a different BLAS library, replace `mkL_cblas.h` with the name of your CBLAS header file. The `getBLASIntTypeName` method returns the name of the integer data type that your CBLAS interface uses. If you are using a different BLAS library, replace `MKL_INT` with the name of the integer data type specific to your CBLAS interface. The `updateBuildInfo` method provides the information required for the build process to link to the BLAS library. Use code that is like the code in the example callback class to specify the location of header file, the full path name of the BLAS library, and the compiler and linker options. If you use the Intel MKL BLAS library, use the link line advisor to see which libraries and compiler options are recommended for your use case.

There are three other methods that are already implemented in `coder.BLASCallback`. These methods are `getBLASDoubleComplexTypeName`, `getBLASSingleComplexTypeName`, and `useEnumNameRatherThanTypedef`. By default, your callback class inherits these implementations from `coder.BLASCallback`. In certain situations, you must override these methods with your own definitions when you define your callback class.

The `getBLASDoubleComplexTypeName` method returns the type used for double-precision complex variables in the generated code. If your BLAS library takes a type other than `double*` and `void*` for double-precision complex array arguments, include this method in your callback class definition.

```

function doubleComplexTypeName = getBLASDoubleComplexTypeName()
doubleComplexTypeName = 'my_double_complex_type';
end

```

Replace `my_double_complex_type` with the type that your BLAS library takes for double-precision complex array arguments.

The `getBLASSingleComplexTypeName` method returns the type used for single-precision complex variables in the generated code. If your BLAS library takes a type other than `float*` and `void*` for single-precision complex array arguments, include this method in your callback class definition.

```

function singleComplexTypeName = getBLASSingleComplexTypeName()
doubleComplexTypeName = 'my_single_complex_type';
end

```

Replace `my_single_complex_type` with the type that your BLAS library takes for single-precision complex array arguments.

The `useEnumNameRatherThanTypedef` method returns `false` by default. If types for enumerations in your BLAS library include the `enum` keyword, redefine this method to return `true` in your callback class definition.

```

function p = useEnumNameRatherThanTypedef()
p = true;
end

```

An excerpt from generated C source code that includes the `enum` keyword is:

```

enum CBLAS_SIDE t;
enum CBLAS_UPLO b_t;
double temp;

```

```
enum CBLAS_TRANSPOSE c_t;
enum CBLAS_DIAG d_t;
```

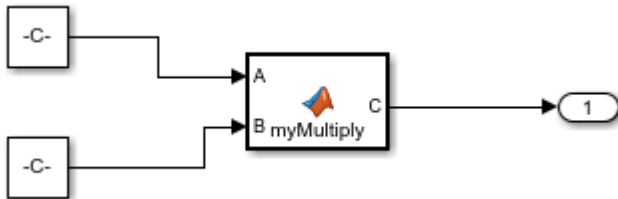
Generate BLAS Calls by Specifying a BLAS Callback Class

This example shows how to generate code that calls BLAS functions in a specific BLAS library. The BLAS callback class `useMyBLAS` specifies the BLAS library that you want to use in this example.

- 1 Create a Simulink model.
- 2 Add a MATLAB Function block to the model.
- 3 In the MATLAB Function block, add code that calls a function for a basic matrix operation. For example, add the function `myMultiply` that multiplies two matrices A and B.

```
function C = myMultiply(A,B) %#codegen
C = A*B;
end
```

- 4 Add two Constant blocks to the left of the MATLAB Function block. Set their values to `zeros(1000)`.
- 5 Add an Output block to the right of the MATLAB Function block.
- 6 Connect the blocks.



- 7 In the Configuration Parameters dialog box, set **Custom BLAS library callback** to the name of the callback class `useMyBLAS`.

The callback class must be on the MATLAB path.

- 8 Build the model.

If A and B are large enough, the code generator produces a BLAS call for the matrix multiplication function.

Locate BLAS Library in Execution Environment

The BLAS library must be available in your execution environment. If your BLAS library is shared, use environment variables or linker options to specify the location of the BLAS library.

- On a Windows platform, modify the `PATH` environment variable.
- On a Linux platform, modify the `LD_LIBRARY_PATH` environment variable or use the `rpath` linker option.
- On a macOS platform, modify the `DYLD_LIBRARY_PATH` environment variable or use the `rpath` linker option.

To specify the `rpath` linker option, use the build information `addLinkFlags` method in the `updateBuildInfo` method of your BLAS callback class. For example, for a GCC compiler:

```
buildInfo.addLinkFlags(sprintf('-Wl,-rpath,"%s"', libPath));
```

Usage Notes and Limitations for OpenBLAS Library

If you generate code that includes calls to the OpenBLAS library functions, follow these guidelines and restrictions:

- If you generate C++ code that includes calls to OpenBLAS library functions, compiling it with the `-pedantic` option produces warnings. To disable the `-pedantic` compiler option, include these lines in the `updateBuildInfo` method:

```
if ctx.getTargetLang() == 'C++'  
    buildInfo.addCompileFlags('-Wno-pedantic');  
end
```

- OpenBLAS does not support the C89/C90 standard.

See Also

`coder.BLASCallback`

More About

- “BLAS Calls for Matrix Operations in a MATLAB Function Block”

External Websites

- <https://netlib.org/blas/>
- https://netlib.org/blas/faq.html#_5_a_id_are_optimized_blas_libraries_available_where_can_i_find_vendor_supplied_blas_a_are_optimized_blas_libraries_available_where_can_i_find_optimized_blas_libraries
- <https://www.intel.com/content/www/us/en/develop/documentation/onemkl-developer-reference-c/top.html>
- <https://www.intel.com/content/www/us/en/develop/tools/oneapi/onemkl/link-line-advisor.html>

Speed Up Fast Fourier Transforms in Code Generated from a MATLAB Function Block

When you simulate a model that includes a MATLAB Function block that calls MATLAB fast Fourier transform (FFT) functions, the simulation software uses the library that MATLAB uses for FFT algorithms. If you generate C/C++ code for this model, by default, the code generator produces code for the FFT algorithms instead of producing FFTW library calls. To increase the speed of fast Fourier transforms in generated code, specify that the code generator produce calls to the FFTW library. For more information about FFTW, see www.fftw.org.

To produce calls to the FFTW library instead of generating FFT algorithms, you can choose from these configuration parameters in **Configuration Parameters > Code Generation** under the **Advanced Parameters**:

- Select **Built-in FFTW library callback** to use the FFTW library included with MATLAB. This instructs the code generator to use the built-in callback for FFTW library calls in the generated code.
- Select **Custom FFT library callback** to use the specific installed FFTW library.

The code generator produces FFTW library calls when:

- A MATLAB Function block calls one of these MATLAB functions: `fft`, `fft2`, `fftn`, `ifft`, `ifft2`, or `ifftn`.
- You generate C/C++ code from a model that includes the MATLAB Function block.

Use Built-in FFTW Library Callback for FFTW Library Usage

If you select **Built-in FFTW library callback**, the code generator produces FFTW library calls using the FFTW library version 3.3.8 included with the MATLAB.

You don't need to perform the additional steps required in **Custom FFT library callback** if you select this parameter during code generation.

Use Custom FFT Library Callback for FFTW Library Usage

If you select **Custom FFT library callback**, the code generator produces FFTW library calls when all of these conditions are true:

- You have access to an FFTW library installation, version 3.2 or later.
- You specify the FFTW library installation in an FFT library callback class that derives from `coder.fftw.StandaloneFFTW3Interface`.
- You set the **Custom FFT library callback** configuration parameter to the name of the callback class.

Install an FFTW Library

If you do not have access to an installed FFTW library, version 3.2 or later, then you must install one.

For a Linux platform or a Mac platform, consider using a package manager to install the FFTW library.

For a Windows platform, in addition to `.dll` files, you must have `.lib` import libraries, as described in the Windows installation notes on the FFTW website.

See the installation instructions for your platform on the FFTW website.

Write FFT Callback Class

To specify your installation of the FFTW library, write an FFT callback class. Share the callback class with others who want to use this FFTW library for FFTW calls.

The callback class must derive from the abstract class `coder.fftw.StandaloneFFTW3Interface`. Use this example callback class as a template.

```
% copyright 2017 The MathWorks, Inc.

classdef useMyFFTW < coder.fftw.StandaloneFFTW3Interface

    methods (Static)
        function th = getNumThreads
            coder.inline('always');
            th = int32(coder.const(1));
        end

        function updateBuildInfo(buildInfo, ctx)
            fftwLocation = '/usr/lib/fftw';
            includePath = fullfile(fftwLocation, 'include');
            buildInfo.addIncludePaths(includePath);
            libPath = fullfile(fftwLocation, 'lib');

            %Double
            libName1 = 'libfftw3-3';
            [~, libExt] = ctx.getStdLibInfo();
            libName1 = [libName1 libExt];
            addLinkObjects(buildInfo, libName1, libPath, 1000, true, true);

            %Single
            libName2 = 'libfftw3f-3';
            [~, libExt] = ctx.getStdLibInfo();
            libName2 = [libName2 libExt];
            addLinkObjects(buildInfo, libName2, libPath, 1000, true, true);
        end
    end
end
```

Implement the `updateBuildInfo` and `getNumThreads` methods. In the `updateBuildInfo` method, set `fftwLocation` to the full path for your installation of the library. Set `includePath` to the full path of the folder that contains `fftw3.h`. Set `libPath` to the full path of the folder that contains the library files. If your FFTW installation uses multiple threads, modify the `getNumThreads` method to return the number of threads that you want to use.

Optionally, you can implement these methods:

- `getPlanMethod` to specify the FFTW planning method. See `coder.fftw.StandaloneFFTW3Interface`.
- `lock` and `unlock` to synchronize multithreaded access to the FFTW planning process. See “Synchronize Multithreaded FFTW Planning in Code Generated from a MATLAB Function Block” on page 40-78.

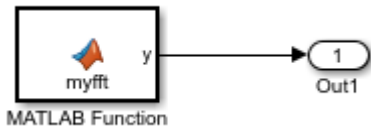
Generate FFTW Calls by Specifying an FFT Callback Class

- 1 Create a Simulink model.
- 2 Add a MATLAB Function block to the model.

- 3 In the MATLAB Function block, add code that calls a MATLAB FFT function. For example, add the function `myfft` that calls the MATLAB function `fft`.

```
function y = myfft()
t = 0:1/50:10-1/50;
x = sin(2*pi*15*t) + sin(2*pi*20*t);
y = fft(x);
end
```

- 4 Connect the blocks.



- 5 Indicate that the code generator produce calls to the FFTW library specified in the FFT library callback class `useMyFFTW`. In the Configuration Parameters dialog box, set **Custom FFT library callback** to `useMyFFTW`.

The callback class must be on the MATLAB path.

- 6 Build the model.

Locate FFTW Library in Execution Environment

The FFTW library must be available in your execution environment. If the FFTW library is shared, use environment variables or linker options to specify the location of the library.

- On a Windows platform, modify the `PATH` environment variable.
- On a Linux platform, modify the `LD_LIBRARY_PATH` environment variable or use the `rpath` linker option.
- On a macOS platform, modify the `DYLD_LIBRARY_PATH` environment variable or use the `rpath` linker option.

To specify the `rpath` linker option, you can use the build information `addLinkFlags` method in the `updateBuildInfo` method of your `coder.fftw.StandaloneFFTW3Interface` class. For example, for a GCC compiler:

```
buildInfo.addLinkFlags(sprintf('-Wl,-rpath,"%s"', libPath));
```

See Also

`coder.fftw.StandaloneFFTW3Interface`

More About

- “Synchronize Multithreaded FFTW Planning in Code Generated from a MATLAB Function Block” on page 40-78

External Websites

- www.fftw.org

Synchronize Multithreaded FFTW Planning in Code Generated from a MATLAB Function Block

This example shows how to generate code that synchronizes multithreaded access to the FFTW planning process for FFTW library calls in code generated from a MATLAB Function block.

The code generator produces FFTW library calls when all of these conditions are true:

- A MATLAB Function block calls one of these functions: `fft`, `fft2`, `fftn`, `ifft`, `ifft2`, or `ifftn`.
- You generate C/C++ code for a model that includes the MATLAB Function block.
- You have access to an FFTW library installation, version 3.2 or later.
- You specify the FFTW library installation in an FFT library callback class that derives from `coder.fftw.StandaloneFFTW3Interface`.
- You set the **Custom FFT library callback** configuration parameter to the name of the callback class.

If you integrate the code that contains the FFTW calls with external code that runs on multiple threads, then you must prevent concurrent access to the FFTW planning process. In your FFT library callback class, implement the `lock` and `unlock` methods. You must also provide C code that manages a lock or mutex. Many libraries, such as OpenMP, pthreads, and the C++ standard library (C++ 11 and later), provide locks. This example shows how to implement the `lock` and `unlock` methods and provide supporting C code. To manage a lock, this example uses the OpenMP library.

Prerequisites

Before you start, for the basic workflow for generating FFTW library calls for fast Fourier transforms in a MATLAB Function block, see “Speed Up Fast Fourier Transforms in Code Generated from a MATLAB Function Block” on page 40-75.

You must have:

- Access to an installed FFTW library.
- A compiler that supports the OpenMP library. To use a different library, such as pthreads, modify the supporting C code accordingly.

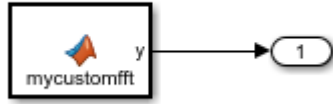
Create a Model with a MATLAB Function Block That Calls an FFT Function

- 1 Create a Simulink model and add a MATLAB Function block to it.
- 2 Add this code to the MATLAB Function block.

```
function y = mycustomfft()

t = 0:1/50:10-1/50;
x = sin(2*pi*15*t) + sin(2*pi*20*t);
y = fft(x);
for k = 1:100
    y = y + ifft(x+k);
end
```


- 3 Add an `outport` block and connect it to the MATLAB Function block.



Write Supporting C Code

Write C functions that initialize, set, and unset a lock. This example uses the OpenMP library to manage the lock. For a different library, modify the code accordingly.

- Create a file `mylock.c` that contains this C code:

```
#include "mylock.h"
#include "omp.h"

static omp_nest_lock_t lockVar;

void mylock_initialize(void)
{
    omp_init_nest_lock(&lockVar);
}

void mylock(void)
{
    omp_set_nest_lock(&lockVar);
}

void myunlock(void)
{
    omp_unset_nest_lock(&lockVar);
}
```

- Create a header file `mylock.h` that contains:

```
#ifndef MYLOCK_H
#define MYLOCK_H

void mylock_initialize(void);
void mylock(void);
void myunlock(void);

#endif
```

Create an FFT Library Callback Class

Write an FFT callback class `myfftc` that:

- Specifies the FFTW library.
- Implements `lock` and `unlock` methods that call the supporting C code to control access to the FFTW planning.

Use this class as a template. Replace `fftwLocation` with the location of your FFTW library installation.

```
classdef myfftwcb < coder.fftw.StandaloneFFTW3Interface

    methods (Static)
        function th = getNumThreads
            coder.inline('always');
            th = int32(coder.const(1));
        end

        function lock()
            coder.cinclude('mylock.h', 'InAllSourceFiles', true);
            coder.inline('always');
            coder.ceval('mylock');
        end

        function unlock()
            coder.cinclude('mylock.h', 'InAllSourceFiles', true);
            coder.inline('always');
            coder.ceval('myunlock');
        end

        function updateBuildInfo(buildInfo, ctx)
            fftwLocation = '\usr\lib\fftw';
            includePath = fullfile(fftwLocation, 'include');
            buildInfo.addIncludePaths(includePath);
            libPath = fullfile(fftwLocation, 'lib');

            %Double
            libName1 = 'libfftw3-3';
            [~, libExt] = ctx.getStdLibInfo();
            libName1 = [libName1 libExt];
            addLinkObjects(buildInfo, libName1, libPath, 1000, true, true);

            %Single
            libName2 = 'libfftw3f-3';
            [~, libExt] = ctx.getStdLibInfo();
            libName2 = [libName2 libExt];
            addLinkObjects(buildInfo, libName2, libPath, 1000, true, true);
        end
    end
end
```

Configure Code Generation Parameters and Build the Model

- 1 Configure code generation to use the FFTW callback class and the C code called by the `lock` and `unlock` methods. Configure code generation to generate a call to `mylock_initialize` in the initialization code.

In the Configuration Parameters dialog box:

- Set **Custom FFT library callback** to `myfftwcb`.
- In **Code Generation > Custom Code**, under **Additional build information**, set **Source files** to `mylock.c`.

- In **Code Generation > Custom Code**, under **Insert custom C code in generated**, set **Initialize function** to `mylock_initialize();`.

2 Build the model.

See Also

`coder.fftw.StandaloneFFTW3Interface`

More About

- “Speed Up Fast Fourier Transforms in Code Generated from a MATLAB Function Block” on page 40-75

External Websites

- www.fftw.org

Control Memory Allocation for Variable-Size Arrays in a MATLAB Function Block

Dynamic memory allocation allocates memory on the heap as needed at run time, instead of allocating memory statically on the stack. You can use dynamic memory allocation for arrays inside a MATLAB Function block.

You cannot use dynamic memory allocation for:

- Input and output signals. Variable-size input and output signals must have an upper bound.
- Parameters or global variables. Parameters and global variables must be fixed-size.
- Fields of bus arrays. Bus arrays cannot have variable-size fields.
- Discrete state properties of System objects associated with a MATLAB System block.

Dynamic memory allocation is beneficial when:

- You do not know the upper bound of an array.
- You do not want to allocate memory on the stack for large arrays.

Dynamic memory allocation and the freeing of this memory can result in slower execution of the generated code. To control the use of dynamic memory allocation for variable-size arrays in a MATLAB Function block, you can:

- Provide upper bounds for variable-size arrays on page 40-82.
- Disable dynamic memory allocation for MATLAB Function blocks on page 40-82.
- Modify the dynamic memory allocation threshold on page 40-82.

Provide Upper Bounds for Variable-Size Arrays

For an unbounded variable-size array, the code generator allocates memory dynamically on the heap. For a bounded variable-size array, if the size, in bytes, is less than the dynamic memory allocation threshold, the code generator allocates memory statically on the stack. To avoid dynamic memory allocation, provide upper bounds for the array dimensions so that the size of the array, in bytes, is less than the dynamic memory allocation threshold. See “Specify Upper Bounds for Variable-Size Arrays”.

Disable Dynamic Memory Allocation for MATLAB Function Blocks

By default, dynamic memory allocation for MATLAB Function blocks is enabled for GRT-based targets and disabled for ERT-based targets. To change the setting, in the Configuration Parameters dialog box, clear or select **Dynamic memory allocation in MATLAB functions**.

If you disable dynamic memory allocation, you must provide upper bounds for variable-size arrays.

Modify the Dynamic Memory Allocation Threshold

Instead of disabling dynamic memory allocation for all variable-size arrays, you can use the dynamic memory allocation threshold to specify when the code generator uses dynamic memory allocation.

Use the dynamic memory allocation threshold to:

- Disable dynamic memory allocation for smaller arrays. For smaller arrays, static memory allocation can speed up generated code. However, static memory allocation can lead to unused storage space. You can decide that the unused storage space is not a significant consideration for smaller arrays.
- Enable dynamic memory allocation for larger arrays. For larger arrays, when you use dynamic memory allocation, you can significantly reduce storage requirements.

The default value of the dynamic memory allocation threshold is 64 kilobytes. To change the threshold, in the Configuration Parameters dialog box, set the **Dynamic memory allocation threshold in MATLAB functions** parameter.

To use dynamic memory allocation for all variable-size arrays, set the threshold to 0.

See Also

MATLAB Function

Related Examples

- “Code Generation for Variable-Size Arrays”
- “Specify Upper Bounds for Variable-Size Arrays”
- “Use Dynamic Memory Allocation for Variable-Size Arrays in a MATLAB Function Block”

Optimize Memory Usage for Time Counters

This example shows how to optimize the amount of memory that the code generator allocates for time counters. The example optimizes the memory that stores elapsed time, the interval of time between two events.

The code generator represents time counters as unsigned integers. The word size of time counters is based on the setting of the model configuration parameter **Application lifespan (days)**, which specifies the expected maximum duration of time the application runs. You can use this parameter to prevent time counter overflows. The default size is 64 bits.

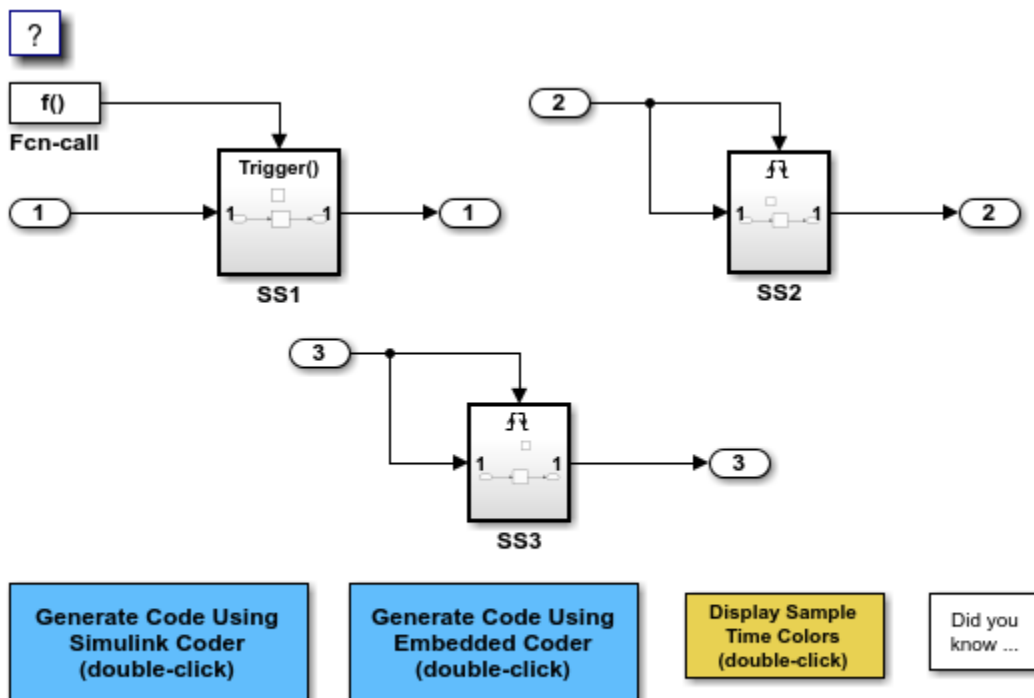
The number of bits that a time counter uses depends on the setting of the **Application lifespan (days)** parameter. For example, if a time counter increments at a rate of 1 kHz, to avoid an overflow, the counter has the following number of bits:

- Lifespan < 0.25 sec: 8 bits
- Lifespan < 1 min: 16 bits
- Lifespan < 49 days: 32 bits
- Lifespan > 50 days: 64 bits

A 64-bit time counter does not overflow for 590 million years.

Open Example Model

Open the example model `rtwdemo_abstime`.



Copyright 1994-2020 The MathWorks, Inc.

The model consists of three subsystems SS1, SS2, and SS3. Open the Model Configuration Parameters dialog box. On the **Math and Data Types** pane, the **Application lifespan (days)** parameter is set to the default, which is `inf`.

The three subsystems contain a discrete-time integrator that requires elapsed time as input to compute its output value. The subsystems vary as follows:

- SS1 - Clocked at 1 kHz. Does not require a time counter. **Sample time type** parameter for trigger port is set to `periodic`. Elapsed time is inlined as 0.001.
- SS2 - Clocked at 100 Hz. Requires a time counter. Based on a lifespan of 1 day, a 32-bit counter stores the elapsed time.
- SS3 - Clocked at 0.5 Hz. Requires a time counter. Based on a lifespan of 1 day, a 16-bit counter stores the elapsed time.

Simulate the Model

Simulate the model. By default, the model is configured to show sample times in different colors. Discrete sample times for the three subsystems appear red, green, and blue. Triggered subsystems are blue-green.

Generate Code and Report

1. Create a temporary folder for the build and inspection process.
2. Configure the model for the code generator to use the GRT system target file and a lifespan of `inf` days.
3. Build the model.

```
### Starting build procedure for: rtwdemo_abstime
### Successful completion of build procedure for: rtwdemo_abstime
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_abstime	Code generated and compiled.	Code generation information file does not exist.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.921s
```

Review Generated Code

Open the generated source file `rtwdemo_abstime.h`.

```
struct tag_RTM_rtwdemo_abstime_T {
    const char_T *errorStatus;

    /*
     * Timing:
     * The following substructure contains information regarding
     * the timing information for the model.
     */
};
```

```

struct {
    uint32_T clockTick1;
    uint32_T clockTickH1;
    uint32_T clockTick2;
    uint32_T clockTickH2;
    struct {
        uint16_T TID[3];
        uint16_T cLimit[3];
    } TaskCounters;
} Timing;
};

/* Block states (default storage) */
extern DW_rtdemo_abstime_T rtdemo_abstime_DW;

/* Zero-crossing (trigger) state */
extern PrevZCX_rtdemo_abstime_T rtdemo_abstime_PrevZCX;

/* External inputs (root inport signals with default storage) */
extern ExtU_rtdemo_abstime_T rtdemo_abstime_U;

/* External outputs (root outports fed by signals with default storage) */
extern ExtY_rtdemo_abstime_T rtdemo_abstime_Y;

/* Model entry point functions */
extern void rtdemo_abstime_initialize(void);
extern void rtdemo_abstime_step0(void);
extern void rtdemo_abstime_step1(void);
extern void rtdemo_abstime_step2(void);
extern void rtdemo_abstime_terminate(void);

/* Real-time Model object */
extern RT_MODEL_rtdemo_abstime_T *const rtdemo_abstime_M;

/*-
 * The generated code includes comments that allow you to trace directly
 * back to the appropriate location in the model. The basic format
 * is <system>/block_name, where system is the system number (uniquely
 * assigned by Simulink) and block_name is the name of the block.
 *
 * Use the MATLAB hilite_system command to trace the generated code back
 * to the model. For example,
 *
 * hilite_system('<S3>') - opens system 3
 * hilite_system('<S3>/Kp') - opens and selects block Kp which resides in S3
 *
 * Here is the system hierarchy for this model
 *
 * '<Root>' : 'rtdemo_abstime'
 * '<S1>' : 'rtdemo_abstime/SS1'
 * '<S2>' : 'rtdemo_abstime/SS2'
 * '<S3>' : 'rtdemo_abstime/SS3'
 */
#endif
/* RTW_HEADER_rtdemo_abstime_h_ */

```

Four 32-bit unsigned integers, `clockTick1`, `clockTickH1`, `clockTick2`, and `clockTickH2` are counters for storing the elapsed time of subsystems SS2 and SS3.

Enable Optimization and Regenerate Code

1. Reconfigure the model to set the lifespan to 1 day.

2. Build the model.

```
### Starting build procedure for: rtwdemo_abstime
### Successful completion of build procedure for: rtwdemo_abstime
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_abstime	Code generated and compiled.	Incremental checksum changed.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 10.449s
```

Review the Regenerated Code

```
struct tag_RTM_rtwdemo_abstime_T {
    const char_T *errorStatus;

    /*
     * Timing:
     * The following substructure contains information regarding
     * the timing information for the model.
     */
    struct {
        uint32_T clockTick1;
        uint16_T clockTick2;
        struct {
            uint16_T TID[3];
            uint16_T cLimit[3];
        } TaskCounters;
    } Timing;
};

/* Block states (default storage) */
extern DW_rtwdemo_abstime_T rtwdemo_abstime_DW;

/* Zero-crossing (trigger) state */
extern PrevZCX_rtwdemo_abstime_T rtwdemo_abstime_PrevZCX;

/* External inputs (root inport signals with default storage) */
extern ExtU_rtwdemo_abstime_T rtwdemo_abstime_U;

/* External outputs (root outports fed by signals with default storage) */
extern ExtY_rtwdemo_abstime_T rtwdemo_abstime_Y;

/* Model entry point functions */
extern void rtwdemo_abstime_initialize(void);
extern void rtwdemo_abstime_step0(void);
extern void rtwdemo_abstime_step1(void);
extern void rtwdemo_abstime_step2(void);
```

```

extern void rtwdemo_abstime_terminate(void);

/* Real-time Model object */
extern RT_MODEL_rtwdemo_abstime_T *const rtwdemo_abstime_M;

/*-
 * The generated code includes comments that allow you to trace directly
 * back to the appropriate location in the model. The basic format
 * is <system>/block_name, where system is the system number (uniquely
 * assigned by Simulink) and block_name is the name of the block.
 *
 * Use the MATLAB hilite_system command to trace the generated code back
 * to the model. For example,
 *
 * hilite_system('<S3>') - opens system 3
 * hilite_system('<S3>/Kp') - opens and selects block Kp which resides in S3
 *
 * Here is the system hierarchy for this model
 *
 * '<Root>' : 'rtwdemo_abstime'
 * '<S1>' : 'rtwdemo_abstime/SS1'
 * '<S2>' : 'rtwdemo_abstime/SS2'
 * '<S3>' : 'rtwdemo_abstime/SS3'
 */
#endif
/* RTW_HEADER_rtwdemo_abstime_h_ */

```

The new setting for the **Application lifespan (days)** parameter instructs the code generator to set aside less memory for the time counters. The regenerated code includes:

- 32-bit unsigned integer, `clockTick1`, for storing the elapsed time of the task for SS2
- 16-bit unsigned integer, `clockTick2`, for storing the elapsed time of the task for SS3

Related Information

- “Model Configuration Parameters: Code Generation Optimization”
- “Timers in Asynchronous Tasks” on page 6-38
- “Time-Based Scheduling and Code Generation” on page 5-2

See Also

More About

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Control Memory Allocation for Time Counters” on page 40-10
- “Access Timers Programmatically” on page 4-4
- “Generate Code for an Elapsed Time Counter” on page 4-11
- “Absolute Time Limitations” on page 4-14

Optimize Generated Code Using Boolean Data for Logical Signals

Optimize generated code by storing logical signals as Boolean data. When you select the model configuration parameter **Implement logic signals as Boolean data (vs. double)**, blocks that generate logic signals output Boolean signals.

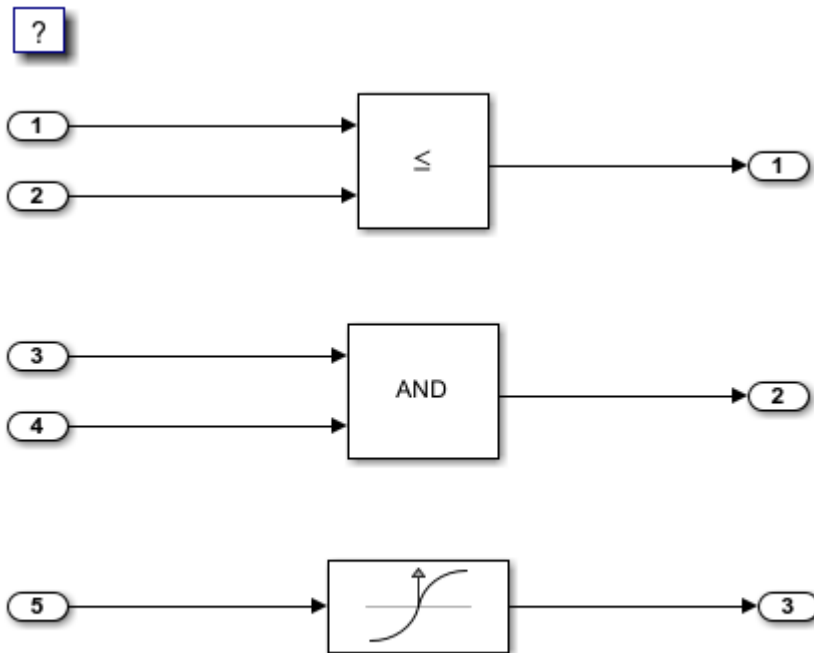
The optimization:

- Reduces the ROM and RAM consumption.
- Improves execution speed.

Example Model

Consider the model LogicalAsBoolean. The outputs of the Relational Operator, Logical Operator and HitCrossing blocks are double, even though they represent logical data.

```
model = 'LogicalAsBoolean';
open_system(model);
```



Copyright 1994-2020 The MathWorks, Inc.

Generate Code

Build the model.

```
slbuild(model)
```

```
### Starting build procedure for: LogicalAsBoolean
### Successful completion of build procedure for: LogicalAsBoolean
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
LogicalAsBoolean	Code generated and compiled.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 14.959s

View the generated code without the optimization. These lines of code are in `LogicalAsBoolean.h`.

```
hfile = fullfile('LogicalAsBoolean_grt_rtw',...
    'LogicalAsBoolean.h');
rtwdemodbtype(hfile,'/* External outputs','/* Parameters (default storage) */',1,0);

/* External outputs (root outports fed by signals with default storage) */
typedef struct {
    real_T Out1;          /* '<Root>/Out1' */
    real_T Out2;          /* '<Root>/Out2' */
    real_T Out3;          /* '<Root>/Out3' */
} ExtY_LogicalAsBoolean_T;
```

Enable Optimization

- 1 Open the Configuration Parameters dialog box.
- 2 Select the **Implement logic signals as Boolean data (vs. double)** parameter.

Alternatively, you can use the command-line API to enable the optimization:

```
set_param(model, 'BooleanDataType', 'on');
```

Generate Code with Optimization

The generated code stores the logical signal output as Boolean data.

Build the model.

```
slbuild(model)

### Starting build procedure for: LogicalAsBoolean
### Successful completion of build procedure for: LogicalAsBoolean
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
LogicalAsBoolean	Code generated and compiled.	Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 10.968s

View the generated code with the optimization. These lines of code are in `LogicalAsBoolean.h`.

```
rtwdemodbtype(hfile, '/* External outputs', '/* Parameters (default storage) */', 1, 0);

/* External outputs (root outports fed by signals with default storage) */
typedef struct {
    boolean_T Out1;          /* '<Root>/Out1' */
    boolean_T Out2;          /* '<Root>/Out2' */
    boolean_T Out3;          /* '<Root>/Out3' */
} ExtY_LogicalAsBoolean_T;
```

Close the model and code generation report.

```
bdclose(model)
rtwdemoclean;
```

See Also

“Implement logic signals as Boolean data (vs. double)”

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Data Types Supported by Simulink”
- “Use Conditional Input Branch Execution” on page 40-62
- “Bitfields” (Embedded Coder)

Reduce Memory Usage for Boolean and State Configuration Variables

- 1 Open the Model Configuration Parameters dialog box.
- 2 In the Model Configuration Parameters dialog box, select the **Optimization** pane.
- 3 Choose from these options:
 - **Use bitsets for storing state configuration** — Reduces the amount of memory that stores state configuration variables. However, it can increase the amount of memory that stores target code if the target processor does not include instructions for manipulating bitsets.
 - **Use bitsets for storing Boolean data** — Reduces the amount of memory that stores Boolean variables. However, it can increase the amount of memory that stores target code if the target processor does not include instructions for manipulating bitsets.

Note You cannot use bitsets when you generate code for these cases:

- An external mode simulation
 - A target that specifies an explicit structure alignment
-

See Also

Use bitsets for storing state configuration | Use bitsets for storing Boolean data

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Optimize Generated Code Using Boolean Data for Logical Signals” on page 40-89
- “Bitfields” (Embedded Coder)

Customize Stack Space Allocation

Your application might be constrained by limited memory. Controlling the maximum allowable size for the stack is one way to modify whether data is defined as local or global in the generated code. You can limit the use of stack space by specifying a positive integer value for the **Maximum stack size (bytes)** parameter, on the **Optimization** pane of the Configuration parameter dialog box. Specifying the maximum allowable stack size provides control over the number of local and global variables in the generated code. Specifically, lowering the maximum stack size might generate more variables into global structures. The number of local and global variables help determine the required amount of stack space for execution of the generated code.

If your model contains a variable that is larger than 4096 bytes, the code generator implements it in global memory by default. You can increase the size of variables that the code generator places in local memory by changing the value of the TLC variable `MaxStackVariableSize`. You can change this value by typing the following command in MATLAB Command Window:

```
set_param(modelName, 'TLCOptions', '-aMaxStackVariableSize=N')
```

The default setting for Maximum stack size (bytes) is `Inherit from target`. In this case, the value of the maximum stack size is the smaller value of the following: the default value set by the code generator (200,000 bytes) or the value of the TLC variable `MaxStackSize` found in the system target file (`ert.tlc`).

To specify a smaller stack size for your application, select the **Specify a value** option of the **Maximum stack size (bytes)** parameter and enter a positive integer value. To specify a smaller stack size at the command line, use:

```
set_param(model_name, 'MaxStackSize', 65000);
```

Note For overall executable stack usage metrics, you might want to do a target-specific measurement, such as using runtime (empirical) analysis or static (code path) analysis with object code.

It is recommended that you use the **Maximum stack size (bytes)** parameter to control stack space allocation instead of modifying the TLC variable, `MaxStackSize`, in the system target file. However, a target author might want to set the TLC variable, `MaxStackSize`, for a target. To set `MaxStackSize`, use `assign` statements in the system target file (`ert.tlc`), as in the following example.

```
%assign MaxStackSize = 4096
```

Write your `%assign` statements in the **Configure RTW code generation settings** section of the system target file. The `%assign` statement is described in “Target Language Compiler”.

See Also

Maximum stack size (bytes)

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Enable and Reuse Local Block Outputs in Generated Code” on page 40-102

- “Minimize Computations and Storage for Intermediate Results at Block Outputs” on page 40-35
- “Inline Numeric Values of Block Parameters” on page 40-41

Optimize Generated Code Using memset Function

This example shows how to optimize the generated code by using the `memset` function to clear the internal storage. When you select the model configuration parameter **Use memset to initialize floats and doubles to 0.0**, the `memset` function clears internal storage, to the integer bit pattern 0 (that is, all bits are off).

If your compiler and target CPU both represent floating-point zero with the integer bit pattern 0, consider setting this parameter to gain execution and ROM efficiency.

NOTE: The command-line values are the reverse of the settings values. 'on' in the command line corresponds to clearing the setting. 'off' in the command line corresponds to selecting the setting.

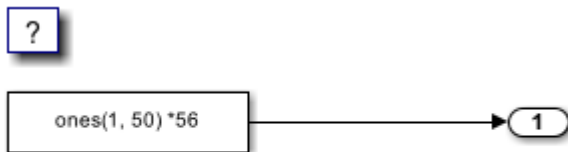
This optimization:

- Reduces ROM consumption.
- Improves execution speed.

Example Model

Consider the model `matlab:rtwdemo_memset`.

```
model = 'rtwdemo_memset';
open_system(model);
```



Copyright 2014-2020 The MathWorks, Inc.

Generate Code

The code generator uses a loop to initialize the Constant block values.

Create a temporary folder (in your system temporary folder) for the build and inspection process.

```
currentDir = pwd;
[~,cgDir] = rtwmoddir();
```

Build the model.

```
slbuild(model)
```

```
### Starting build procedure for: rtwdemo_memset
### Successful completion of build procedure for: rtwdemo_memset
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_memset	Code generated and compiled.	Code generation information file does not exist.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 13.456s

View the generated code without the optimization. These lines of code are in `rtwdemo_memset.c`.

```
cfile = fullfile(cgDir,'rtwdemo_memset_grt_rtw','rtwdemo_memset.c');
rtwdemodbtype(cfile,'/* Model initialize function */',...
    '/* Model terminate function */',1,0);
```

```
/* Model initialize function */
void rtwdemo_memset_initialize(void)
{
    /* Registration code */

    /* initialize error status */
    rtmSetErrorStatus(rtwdemo_memset_M, (NULL));

    /* external outputs */
    {
        int32_T i;
        for (i = 0; i < 50; i++) {
            rtwdemo_memset_Y.Out1[i] = 0.0;
        }
    }

    {
        int32_T i;

        /* ConstCode for Outport: '<Root>/Out1' */
        for (i = 0; i < 50; i++) {
            rtwdemo_memset_Y.Out1[i] = 56.0;
        }

        /* End of ConstCode for Outport: '<Root>/Out1' */
    }
}
```

Enable Optimization

- 1 Open the Configuration Parameters dialog box.
- 2 In the Configuration Parameter dialog box select the **Use memset to initialize floats and doubles to 0.0** parameter. Alternatively, you can use the command-line API to enable the optimization:

```
set_param(model,'InitFltsAndDblsToZero','off');
```

% # Open the Configuration Parameters dialog box.

Generate Code with Optimization

The code generator uses the `memset` function to initialize the Constant block values.

Build the model.

```
slbuild(model)

### Starting build procedure for: rtwdemo_memset
### Successful completion of build procedure for: rtwdemo_memset
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_memset	Code generated and compiled.	Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 11.593s

View the generated code with the optimization. These lines of code are in `rtwdemo_memset.c`.

```
rtwdemodbtype(cfile, /* Model initialize function */, ...
             /* Model terminate function */, 1, 0);

/* Model initialize function */
void rtwdemo_memset_initialize(void)
{
    /* Registration code */

    /* initialize error status */
    rtmSetErrorStatus(rtwdemo_memset_M, (NULL));

    /* external outputs */
    (void)memset(&rtwdemo_memset_Y, 0, sizeof(ExtY_rtwdemo_memset_T));

    {
        int32_T i;

        /* ConstCode for Output: '<Root>/Out1' */
        for (i = 0; i < 50; i++) {
            rtwdemo_memset_Y.Out1[i] = 56.0;
        }

        /* End of ConstCode for Output: '<Root>/Out1' */
    }
}
```

Close the model and the code generation report.

```
bdclose(model)
rtwdemoclean;
cd(currentDir)
```

See Also

Use `memset` to initialize floats and doubles to 0.0

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7

- “Use memcpy Function to Optimize Generated Code for Vector Assignments” on page 40-48
- “Vector Operation Optimization” on page 40-99
- “Remove Zero-Initialization Code” (Embedded Coder)

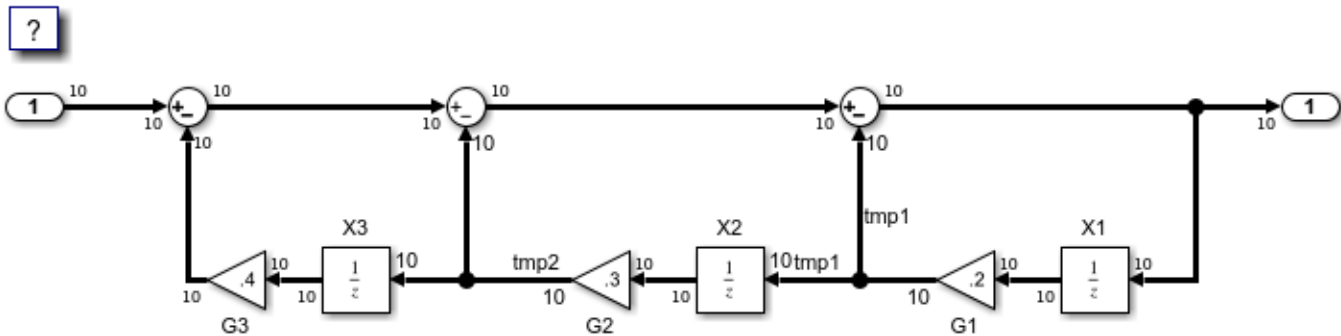
Vector Operation Optimization

This example shows how Simulink® Coder™ optimizes generated code by setting block output that generates vectors to scalars, for blocks such as the Mux, Sum, Gain, and Bus. This optimization reduces stack memory by replacing temporary local arrays with local variables.

Example Model

In the model, `rtwdemo_VectorOptimization`, the output of Gain blocks G1 and G2 are the vector signals `tmp1` and `tmp2`. These vectors have a width of 10.

```
model = 'rtwdemo_VectorOptimization';
open_system(model);
set_param(model, 'SimulationCommand', 'update')
```



Copyright 2014-2020 The MathWorks, Inc.

Generate Code

Create a temporary folder (in your system temporary folder) for the build and inspection process.

```
currentDir = pwd;
[~,cgDir] = rtwdemdir();
```

Build the model.

```
slbuild(model)
```

```
### Starting build procedure for: rtwdemo_VectorOptimization
### Successful completion of build procedure for: rtwdemo_VectorOptimization
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_VectorOptimization	Code generated and compiled.	Code generation information file does not exist

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 12.217s
```

The optimized code is in `rtwdemo_VectorOptimization.c`. The signals `tmp1` and `tmp2` are the local variables `rtb_tmp1` and `rtb_tmp2`.

```

cfile = fullfile(cgDir, 'rtwdemo_VectorOptimization_grt_rtw', ...
    'rtwdemo_VectorOptimization.c');
rtwdemodbtype(cfile, '/* Model step', '/* Model initialize', 1, 0);

/* Model step function */
void rtwdemo_VectorOptimization_step(void)
{
    real_T rtb_Sum3_0;
    real_T rtb_tmp1_0;
    real_T rtb_tmp2_0;
    int32_T i;
    for (i = 0; i < 10; i++) {
        /* Gain: '<Root>/G2' incorporates:
        * UnitDelay: '<Root>/X2'
        */
        rtb_tmp2_0 = 0.3 * rtwdemo_VectorOptimization_DW.X2_DSTATE[i];

        /* Gain: '<Root>/G1' incorporates:
        * UnitDelay: '<Root>/X1'
        */
        rtb_tmp1_0 = 0.2 * rtwdemo_VectorOptimization_DW.X1_DSTATE[i];

        /* Sum: '<Root>/Sum3' incorporates:
        * Gain: '<Root>/G3'
        * Inport: '<Root>/In2'
        * Sum: '<Root>/Sum1'
        * Sum: '<Root>/Sum2'
        * UnitDelay: '<Root>/X3'
        */
        rtb_Sum3_0 = ((rtwdemo_VectorOptimization_U.In2[i] - 0.4 *
            rtwdemo_VectorOptimization_DW.X3_DSTATE[i]) - rtb_tmp2_0) -
            rtb_tmp1_0;

        /* Outport: '<Root>/Out2' */
        rtwdemo_VectorOptimization_Y.Out2[i] = rtb_Sum3_0;

        /* Update for UnitDelay: '<Root>/X3' */
        rtwdemo_VectorOptimization_DW.X3_DSTATE[i] = rtb_tmp2_0;

        /* Update for UnitDelay: '<Root>/X2' */
        rtwdemo_VectorOptimization_DW.X2_DSTATE[i] = rtb_tmp1_0;

        /* Update for UnitDelay: '<Root>/X1' */
        rtwdemo_VectorOptimization_DW.X1_DSTATE[i] = rtb_Sum3_0;
    }
}

```

Close the model and code generation report.

```
bdclose(model)  
rtwdemoclean;  
cd(currentDir)
```

See Also

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Minimize Computations and Storage for Intermediate Results at Block Outputs” on page 40-35
- “Use memcpy Function to Optimize Generated Code for Vector Assignments” on page 40-48

Enable and Reuse Local Block Outputs in Generated Code

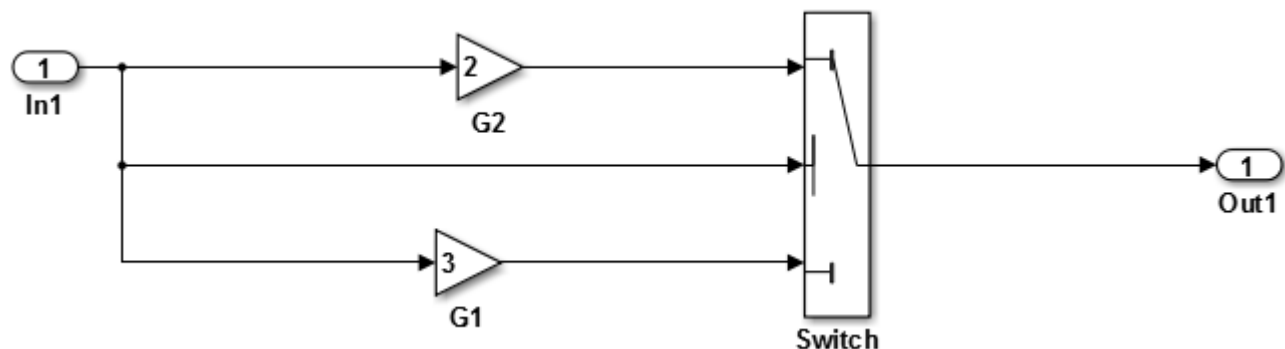
In this section...

- “Example Model” on page 40-102
- “Generate Code Without Optimization” on page 40-102
- “Enable Local Block Outputs and Generate Code” on page 40-103
- “Reuse Local Block Outputs and Generate Code” on page 40-104

This example shows how to specify block output as local variables. The code generator can potentially reuse these local variables in the generated code. Declaring block output as local variables conserves ROM consumption. Reusing local variables conserves RAM consumption, reduces data copies, and increases execution speed.

Example Model

- 1 Use Inport, Outport, Gain, and Switch blocks to create the following model. In this example, the model is named `local_variable_ex`.



- 2 For G2, open the Gain Block Parameters dialog box. Enter a value of 2.
- 3 For G1, enter a value of 3.
- 4 For the Switch block, open the Block Parameters dialog box. For the **Criteria for passing first input** parameter, select `u2 >= Threshold`.

Generate Code Without Optimization

- 1 Open the Model Configuration Parameters dialog box. Select the **Solver** pane. For the **Type** parameter, select `Fixed-step`.
- 2 Clear the **Configuration Parameters > Signal storage reuse** parameter.
- 3 Select the **Code Generation > Report** pane and select **Create code generation report**.
- 4 Select the **Code Generation** pane. Select **Generate code only**, and then, in the model window, press **Ctrl+B**. When code generation is complete, an HTML code generation report appears.
- 5 In the code generation report, select the `local_variable_ex.c` section and view the model step function. The Gain block outputs are the global variables `local_variable_ex_B.G2` and `local_variable_ex_B.G1`.


```

/* Model step function */
void local_variable_ex_step(void)
{
  /* Switch: '<Root>/Switch' incorporates:
   * Inport: '<Root>/In1'
   */
  if (local_variable_ex_U.In1 >= 0.0) {
    /* Gain: '<Root>/G2' */
    local_variable_ex_B.G2 = 2.0 * local_variable_ex_U.In1;

    /* Output: '<Root>/Out1' */
    local_variable_ex_Y.Out1 = local_variable_ex_B.G2;
  } else {
    /* Gain: '<Root>/G1' */
    local_variable_ex_B.G1 = 3.0 * local_variable_ex_U.In1;

    /* Output: '<Root>/Out1' */
    local_variable_ex_Y.Out1 = local_variable_ex_B.G1;
  }

  /* End of Switch: '<Root>/Switch' */
}

```

Enable Local Block Outputs and Generate Code

- 1 Select the **Configuration Parameters > Signal Storage Reuse** parameter. The **Signal Storage Reuse** enables the following optimization parameters:
 - **Enable local block outputs**
 - **Reuse local block outputs**
 - **Eliminate superfluous local variables (expression folding)**
- 2 Clear **Reuse local block outputs** and **Eliminate superfluous local variables (expression folding)**.
- 3 Generate code and view the model step function. There are three local variables in the model step function because you selected the optimization parameter **Enable Local Block Outputs**. The local variables `rtb_G2` and `rtb_G1` hold the outputs of the Gain blocks. The local variable `rtb_Switch` holds the output of the Switch block.

```

/* Model step function */
void local_variable_ex_step(void)
{
  real_T rtb_Switch;
  real_T rtb_G2;
  real_T rtb_G1;

  /* Switch: '<Root>/Switch' incorporates:
   * Inport: '<Root>/In1'
   */
  if (local_variable_ex_U.In1 >= 0.0) {
    /* Gain: '<Root>/G2' */
    rtb_G2 = 2.0 * local_variable_ex_U.In1;
    rtb_Switch = rtb_G2;
  } else {
    /* Gain: '<Root>/G1' */
    rtb_G1 = 3.0 * local_variable_ex_U.In1;
    rtb_Switch = rtb_G1;
  }
}

```

```
}

/* End of Switch: '<Root>/Switch' */

/* Output: '<Root>/Out1' */
local_variable_ex_Y.Out1 = rtb_Switch;
```

Reuse Local Block Outputs and Generate Code

- 1 Select the **Configuration Parameters > Reuse local block outputs** parameter.
- 2 Generate code. In the `local_variable_ex.c` section, view the model step function. There is one local variable, `rtb_G2`, that the code generator uses three times.

```
/* Model step function */
void local_variable_ex_step(void)
{
    real_T rtb_G2;

    /* Switch: '<Root>/Switch' incorporates:
     * Inport: '<Root>/In1'
     */
    if (local_variable_ex_U.In1 >= 0.0) {
        /* Gain: '<Root>/G2' */
        rtb_G2 = 2.0 * local_variable_ex_U.In1;
    } else {
        /* Gain: '<Root>/G1' */
        rtb_G2 = 3.0 * local_variable_ex_U.In1;
    }

    /* End of Switch: '<Root>/Switch' */

    /* Output: '<Root>/Out1' */
    local_variable_ex_Y.Out1 = rtb_G2;
```

The extra temporary variable `rtb_Switch` and the associated data copy is not in the generated code.

See Also

Enable local block outputs

Related Examples

- “Design Techniques to Optimize Models for Efficient Code Generation” on page 40-7
- “Customize Stack Space Allocation” on page 40-93
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Generate SIMD Code from Simulink Blocks

You can generate single instruction, multiple data (SIMD) code from certain Simulink blocks by using Intel SSE and, if you have Embedded Coder, Intel AVX technology. SIMD is a computing paradigm in which a single instruction processes multiple data. Many modern processors have SIMD instructions that, for example, perform several additions or multiplications at once. For computationally intensive operations on supported blocks, SIMD intrinsics can significantly improve the performance of the generated code on Intel platforms.

Blocks That Support SIMD Code Generation

When certain conditions are met, you can generate SIMD code by using Intel SSE or Intel AVX technology. This table lists blocks that support SIMD code generation. The table also details the conditions under which the support is available.

Block	Conditions
Add	<ul style="list-style-type: none"> • For AVX, SSE, and FMA, the input signal has a data type of <code>single</code> or <code>double</code>. • For AVX2, SSE4.1, and SSE2, the input signal has a data type of <code>single</code>, <code>double</code>, <code>int8</code>, <code>int16</code>, <code>int32</code>, or <code>int64</code>. • For AVX512F, the input signal has a data type of <code>single</code> or <code>double</code>.
Subtract	<ul style="list-style-type: none"> • For AVX, SSE, and FMA, the input signal has a data type of <code>single</code> or <code>double</code>. • For AVX2, SSE4.1, and SSE2, the input signal has a data type of <code>single</code>, <code>double</code>, <code>int8</code>, <code>int16</code>, <code>int32</code>, or <code>int64</code>. • For AVX512F, the input signal has a data type of <code>single</code> or <code>double</code>.
Sum of Elements	<ul style="list-style-type: none"> • For AVX, SSE, and FMA, the input signal has a data type of <code>single</code> or <code>double</code>. • For AVX2, SSE4.1, and SSE2, the input signal has a data type of <code>single</code>, <code>double</code>, <code>int8</code>, <code>int16</code>, <code>int32</code>, or <code>int64</code>. • For AVX512F, the input signal has a data type of <code>single</code> or <code>double</code>. • The Optimize reductions configuration parameter is set to <code>on</code>.

Block	Conditions
Product	<ul style="list-style-type: none"> • For AVX, SSE, and FMA, the input signal has a data type of <code>single</code> or <code>double</code>. • For AVX2, SSE4.1, and SSE2, the input signal has a data type of <code>single</code>, <code>double</code>, <code>int16</code> or <code>int32</code>. • For AVX512F, the input signal has a data type of <code>single</code> or <code>double</code>. • Set Multiplication parameter to <code>Element-wise(.*)</code>
Product of Elements	<ul style="list-style-type: none"> • For AVX, SSE, and FMA, the input signal has a data type of <code>single</code> or <code>double</code>. • For AVX2, SSE4.1, and SSE2, the input signal has a data type of <code>single</code>, <code>double</code>, <code>int16</code> or <code>int32</code>. • For AVX512F, the input signal has a data type of <code>single</code> or <code>double</code>. • Set Multiplication parameter to <code>Element-wise(.*)</code> • Set the Optimize reductions configuration parameter to <code>on</code>.
Gain	<ul style="list-style-type: none"> • For AVX, and SSE, the input signal has a data type of <code>single</code> or <code>double</code>. • For AVX2, SSE4.1, and SSE2, the input signal has a data type of <code>single</code>, <code>double</code>, <code>int16</code>, or <code>int32</code>. • For Intel AVX512F, the input signal has a data type of <code>single</code> or <code>double</code>. • Set Multiplication parameter to <code>Element-wise(.*)</code>
Divide	The input signal has a data type of <code>single</code> or <code>double</code> .
Sqrt	The input signal has a data type of <code>single</code> or <code>double</code> .
Ceil	<ul style="list-style-type: none"> • For AVX2, AVX, SSE4.1, SSE2, and SSE, the input signal has a data type of <code>single</code> or <code>double</code>. • AVX512F is not supported.
Floor	<ul style="list-style-type: none"> • For AVX2, AVX, SSE4.1, SSE2, and SSE, the input signal has a data type of <code>single</code> or <code>double</code>. • AVX512F is not supported.

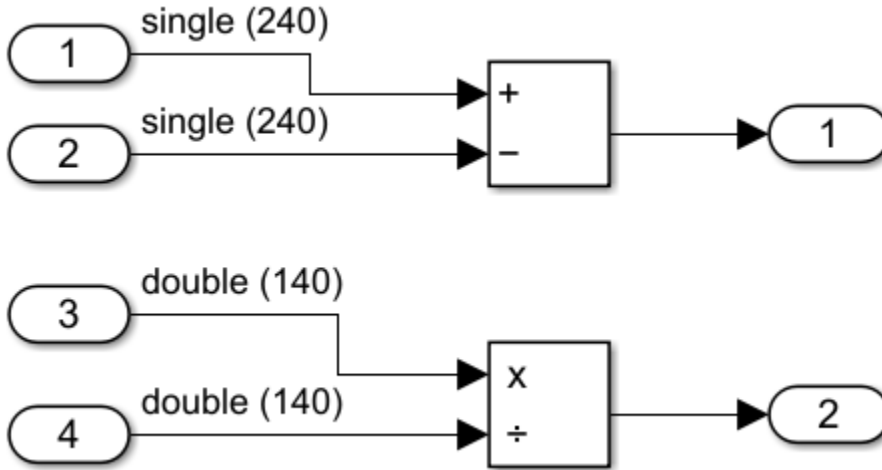
Block	Conditions
MinMax	<ul style="list-style-type: none"> The input signal has a data type of single or double. The value of the Support: non-finite numbers configuration parameter is set to off.
MinMax of Elements	<ul style="list-style-type: none"> The input signal has a data type of single or double. The value of the Support: non-finite numbers configuration parameter is set to off. The Optimize reductions configuration parameter is set to on.
MATLAB Function	MATLAB code meets the conditions specified in this topic: “Generate SIMD Code for MATLAB Functions” (Embedded Coder).
For Each Subsystem	<ul style="list-style-type: none"> The For Each Subsystem block contains a block listed in this table that meets the specified conditions. The value of the Partition Dimension block parameter must be above the value of the Loop unrolling threshold configuration parameter.
Bitwise Operator	<ul style="list-style-type: none"> The value of the Operator block parameter must be AND, OR, or XOR. For SSE2, the input signal has a data type of int8, int16, int32, or int64. For AVX2, and AVX512F, the input signal has a data type of int8, int16, int32, or int64.
Shift Arithmetic	<ul style="list-style-type: none"> The input signal has a data type of int32.

If you have DSP System Toolbox, you can also generate SIMD code from certain DSP System Toolbox blocks. For more information, see “Simulink Blocks in DSP System Toolbox that Support SIMD Code Generation” (DSP System Toolbox).

If you have the Embedded Coder Support Package for ARM[®] Cortex[®]-A Processors, you can generate SIMD code optimized for ARM Cortex A processors by setting the Code replacement libraries (Embedded Coder) parameter to GCC ARM Cortex-A. You will also need to set the **Device vendor** parameter to ARM Compatible and the **Device type** parameter to ARM Cortex-A.

Generate SIMD Code Compared to Plain C Code

For this example, create a simple model `simdDemo` that has a Subtract block and a Divide block. The Subtract block has an input signal that has a dimension of 240 and an input data type of single. The Divide block has an input signal that has a dimension of 140 and an input data type of double.



The plain generated C code for this model is:

```
void simdDemo_step(void)
{
    int32_T i;
    for (i = 0; i < 240; i++) {
        simdDemo_Y.Out1[i] = simdDemo_U.In1[i] - simdDemo_U.In2[i];
    }

    for (i = 0; i < 140; i++) {
        simdDemo_Y.Out2[i] = simdDemo_U.In3[i] / simdDemo_U.In4[i];
    }
}
```

In the plain (non-SIMD) C code, each loop iteration produces one result.

To generate SIMD code:

- 1 Open the Simulink Coder app or the Embedded Coder app.
- 2 Click **Settings > Hardware Implementation**.
- 3 Set the **Device vendor** parameter to Intel or AMD.
- 4 Set the **Device type** parameter to x86-64(Windows 64) or x86-64(Linux 64).
- 5 On the **Optimization** pane, for the **Leverage target hardware instruction set extensions** parameter, select the instruction set extension that your processor supports. For example, select SSE2. If you use Embedded Coder, you can also select from the instruction sets SSE, SSE4.1, AVX, AVX2, FMA, and AVX512F. For more information, see <https://www.intel.com/content/www/us/en/support/articles/000005779/processors.html>.
- 6 Optionally, select the **Optimize reductions** parameter to generate SIMD code for reduction operations.
- 7 Generate code from the model.

```
void simdDemo_step(void)
{
    int32_T i;
    for (i = 0; i <= 236; i += 4) {
        _mm_storeu_ps(&simdDemo_Y.Out1[i], _mm_sub_ps(_mm_loadu_ps(&simdDemo_U.In1[i]),
            _mm_loadu_ps(&simdDemo_U.In2[i])));
    }
}
```

```

}
for (i = 0; i <= 138; i += 2) {
    _mm_storeu_pd(&simdDemo_Y.Out2[i], _mm_div_pd(_mm_loadu_pd(&simdDemo_U.In3[i]),
        _mm_loadu_pd(&simdDemo_U.In4[i])));
}
}

```

This code is for the SSE2 instruction set extension. The SIMD instructions are the intrinsic functions that start with the identifier `_mm`. These functions process multiple data in a single iteration of the loop because the loop increments by four for single data types and by two for double data types. For models that process more data and are computationally more intensive than this one, the presence of SIMD instructions can significantly speed up the code execution time.

For a list of a Intel intrinsic functions for supported Simulink blocks, see <https://www.intel.com/content/www/us/en/docs/intrinsics-guide/index.html>.

Limitations

The generated code is not optimized through SIMD if:

- The code in a MATLAB Function block contains scalar data types outside the body of loops. For instance, if `a`, `b`, and `c` are scalars, the generated code does not optimize an operation such as `c=a+b`.
- The code in a MATLAB Function block contains indirectly indexed arrays or matrices. For instance if `A`, `B`, `C`, and `D` are vectors, the generated code is not vectorized for an operation such as `D(A)=C(A)+B(A)`.
- The blocks within a reusable subsystem might not be optimized.
- If the code in a MATLAB Function block contains parallel for-Loops (`parfor`), the `parfor` loop is not optimized with SIMD code, but loops within the body of the `parfor` loop can be optimized with SIMD code.
- Polyspace does not support analysis of generated code that includes SIMD instructions. Disable SIMD code generation by setting the **Leverage target hardware instruction set extensions** parameter to `None`.
- Reduction operations that operate on floating-point values when support for nonfinite numbers is enabled. To generate SIMD code for reduction operations that operate on floating-point values, set the configuration parameter **Support: non-finite numbers** to `Off`.

See Also

More About

- “Generate SIMD Code for MATLAB Functions” (Embedded Coder)
- “Generate High Performance SIMD Code on Intel from Simulink Blocks in DSP System Toolbox” (DSP System Toolbox)
- “Generate High Performance SIMD Code on Intel from MATLAB Algorithms in DSP System Toolbox” (DSP System Toolbox)

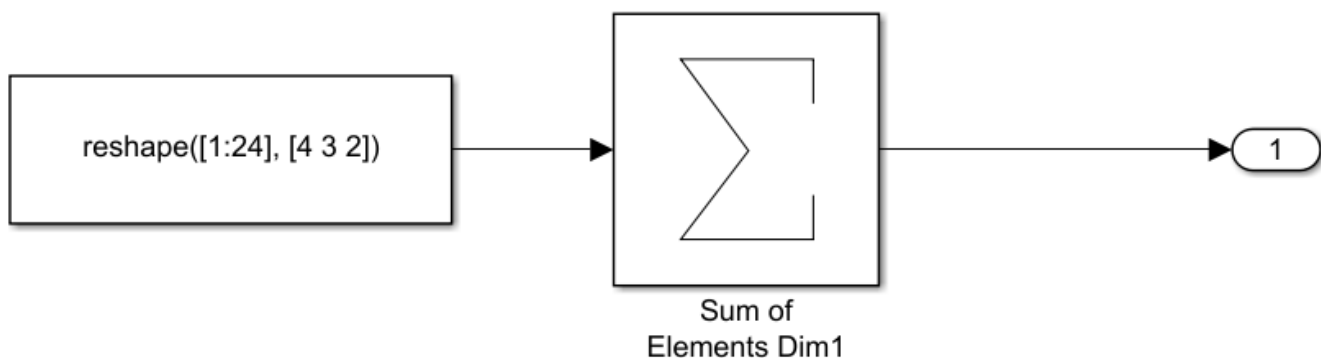
Optimize Code for Reduction Operations by Using SIMD

This example shows how to generate optimized single instruction, multiple data (SIMD) code that performs reduction operations. A reduction is an operation that reduces a set of elements, such as an array, to a single value using an associative binary operator. For example, calculating the sum of the elements in an array is a reduction operation that uses the addition operator. For processors that support SIMD instructions, you can optimize reduction operations by generating parallel code that uses the SIMD instructions for the operation.

Example Model

Open the example model `ex_rowmajor_algorithm`. The model calculates the sum of the elements in a 4 by 3 by 2 array.

```
model = 'ex_rowmajor_algorithm';
open_system(model)
```



Copyright 2021 The MathWorks, Inc.

Generate SIMD Code Without Reduction Optimization

Open the configuration parameters for the model. On the **Modeling** tab, click **Settings**. On the **Code Generation** pane, set **System target file** to `grt.tlc`. On the **Optimization** pane, set the **Leverage target hardware instruction set extensions** to SSE2. Alternatively, use the command-line.

```
set_param(model, 'SystemTargetFile', 'grt.tlc');
set_param(model, 'SaveOutput', 'off');
set_param(model, 'RTWVerbose', 'off');
set_param(model, 'InstructionSetExtensions', 'SSE2');
```

De-select the parameter **Optimize reductions**.

```
set_param(model, 'OptimizeReductions', 'off');
```

Build the model.

```
slbuild(model);
```



```
### Starting build procedure for: ex_rowmajor_algorithm
### Successful completion of build procedure for: ex_rowmajor_algorithm
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
ex_rowmajor_algorithm	Code generated and compiled.	Code generation information file does not exist

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 13.88s

Inspect the generated `ex_rowmajor_algorithm_step` function.

```
file = fullfile('ex_rowmajor_algorithm_grt_rtw','ex_rowmajor_algorithm.c');
rtwdemodbtype(file,'/* Model step function */','/* End of Sum',1,1);

/* Model step function */
void ex_rowmajor_algorithm_step(void)
{
    int32_T i;

    /* Sum: '<Root>/Sum of Elements Dim1' incorporates:
     * Constant: '<Root>/Constant1'
     */
    ex_rowmajor_algorithm_Y.Out2 = -0.0F;
    for (i = 0; i < 24; i++) {
        ex_rowmajor_algorithm_Y.Out2 +=
            ex_rowmajor_algorithm_ConstP.Constant1_Value[i];
    }
}
```

The generated code contains a non-vectorized loop that adds each array element individually.

Generate SIMD Code With Reduction Optimization

Open the configuration parameters for the model. On the **Modeling** tab, click **Settings**. On the **Optimization** pane, set the **Leverage target hardware instruction set extensions** to SSE2. Alternatively, use the command-line.

```
set_param(model, 'InstructionSetExtensions', 'SSE2');
```

Select the parameter **Optimize reductions**. If you want to optimize reduction operations that operate on floating-point values, set **Support: Non-finite numbers** to Off. Reduction optimization does not support operations on floating-point values when non-finite numbers are supported.

```
set_param(model, 'OptimizeReductions', 'on');
set_param(model, 'SupportNonFinite', 'off');
set_param(model, 'MatFileLogging', 'off');
```

Build the model.

```
slbuild(model);

### Starting build procedure for: ex_rowmajor_algorithm
### Successful completion of build procedure for: ex_rowmajor_algorithm
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
ex_rowmajor_algorithm	Code generated and compiled.	Generated code was out of date.

1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 9.3657s

Inspect the generated `ex_rowmajor_algorithm_step` function.

```
file = fullfile('ex_rowmajor_algorithm_grt_rtw','ex_rowmajor_algorithm.c');
rtwdemodbtype(file,'/* Model step function */','/* Matfile',1,1);

/* Model step function */
void ex_rowmajor_algorithm_step(void)
{
    __m128 tmp;
    int32_T i;
    real32_T tmp_0[4];
    tmp = _mm_set1_ps(0.0F);
    for (i = 0; i <= 20; i += 4) {
        /* Sum: '<Root>/Sum of Elements Dim1' incorporates:
         * Constant: '<Root>/Constant1'
         */
        tmp = _mm_add_ps(tmp, _mm_loadu_ps
            (&ex_rowmajor_algorithm_ConstP.Constant1_Value[i]));
    }

    _mm_storeu_ps(&tmp_0[0], tmp);

    /* Output: '<Root>/Out2' */
    ex_rowmajor_algorithm_Y.Out2 = (tmp_0[0] + tmp_0[1]) + (tmp_0[2] + tmp_0[3]);
}

/* Model initialize function */
void ex_rowmajor_algorithm_initialize(void)
{
    /* Registration code */

    /* initialize error status */
    rtmSetErrorStatus(ex_rowmajor_algorithm_M, (NULL));

    /* external outputs */
    ex_rowmajor_algorithm_Y.Out2 = 0.0F;
}

/* Model terminate function */
void ex_rowmajor_algorithm_terminate(void)
{
    /* (no terminate code required) */
}
```

The vectorized loop uses the SSE2 function `mm_add_ps`, which adds four elements of the array to the sum in one step. This optimization improves the execution speed of the generated code.

Open the code generation report. On the **C Code** tab, click **Open Report**.

Additional Information

Code Generation Advisor	Not run
Optimize reductions	Reductions are optimized using SSE2 Instruction Set Extensions

On the report summary, in the **Additional Information** section, the report indicates that reduction operations are optimized using the SSE2 instruction set.

See Also

More About

- “Row-Major Algorithms for Row-Major Array Layout” on page 18-32

High-throughput Applications in Simulink Coder

- “Code Generation for Deep Learning Simulink Model That Performs Lane and Vehicle Detection” on page 41-2
- “Generate Generic C/C++ for Sequence-to-Sequence Deep Learning Simulink Models” on page 41-10
- “Generate Code for Deep Learning Networks Using MATLAB Function Block” on page 41-22
- “Generate Code for Blocks from Deep Neural Networks Library” on page 41-27
- “Workflow for Deep Learning C/C++ Code Generation for Simulink Models” on page 41-32

Code Generation for Deep Learning Simulink Model That Performs Lane and Vehicle Detection

This example shows how to generate C++ code from a Simulink® model that performs lane and vehicle detection by using convolutional neural networks (CNNs). The example takes the frames of a traffic video as an input, outputs two lane boundaries that correspond to the left and right lanes of the ego vehicle, and detects vehicles in the frame. This example uses a pretrained lane detection network and a pretrained vehicle detection network from the *Object Detection Using YOLO v2 Deep Learning* example of the Computer Vision Toolbox™. For more information, see “Object Detection Using YOLO v2 Deep Learning” (Computer Vision Toolbox).

This example illustrates the following concepts:

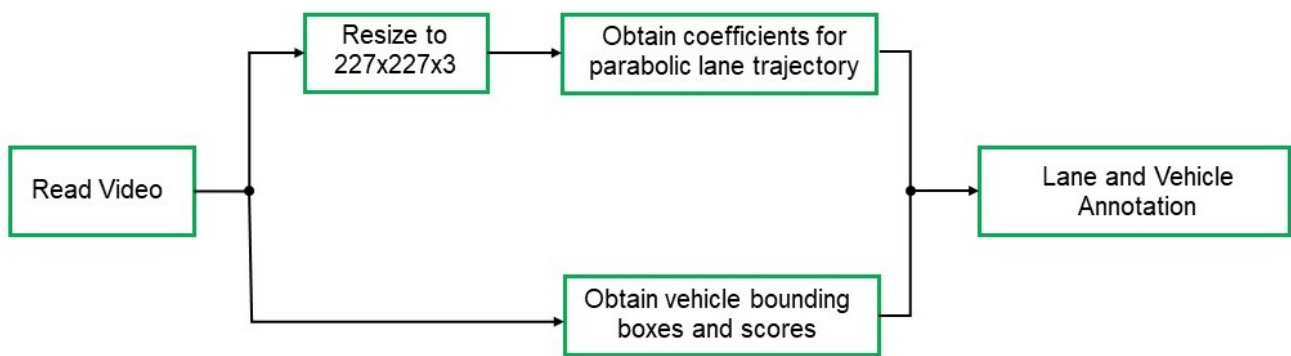
- Model the lane detection application in Simulink. First you process the traffic video by resizing to 227-by-227-by-3 and multiplying by a constant factor of 255. Subsequently, you process the traffic video by using the pretrained network loaded in the Predict block from Deep Learning Toolbox™. Finally, if the left and right lane boundaries are detected, the parabolic coefficients to model the trajectories of the lane boundaries are obtained.
- Model the vehicle detection application in Simulink by processing the traffic video using a pretrained YOLO v2 detector. This network detects vehicles in the video and outputs the coordinates of the bounding boxes for these vehicles and their confidence score.
- Configure the model for code generation.

Prerequisites

- Intel Math Kernel Library for Deep Neural Networks (MKL-DNN).
- Refer to MKLDNN CPU Support for the list of processors that support MKL-DNN library.
- Deep Learning Toolbox™ for using the DAGNetwork object.
- Computer Vision Toolbox™ for video I/O operations.

Algorithmic Workflow

The block diagram for the algorithmic workflow of the Simulink model follows.



Get Pretrained Lane and Vehicle Detection Networks

This example uses the `trainedLaneNet` and `yolov2ResNet50VehicleExample` MAT files containing the pretrained networks. The files are approximately 143 MB and 98 MB in size, respectively. Download the files.

```
lanenetFile = matlab.internal.examples.downloadSupportFile('gpcoder/cnn_models/lane_detection',
vehiclenetFile = matlab.internal.examples.downloadSupportFile('vision/data','yolov2ResNet50VehicleExample');
```

Download Test Traffic Video

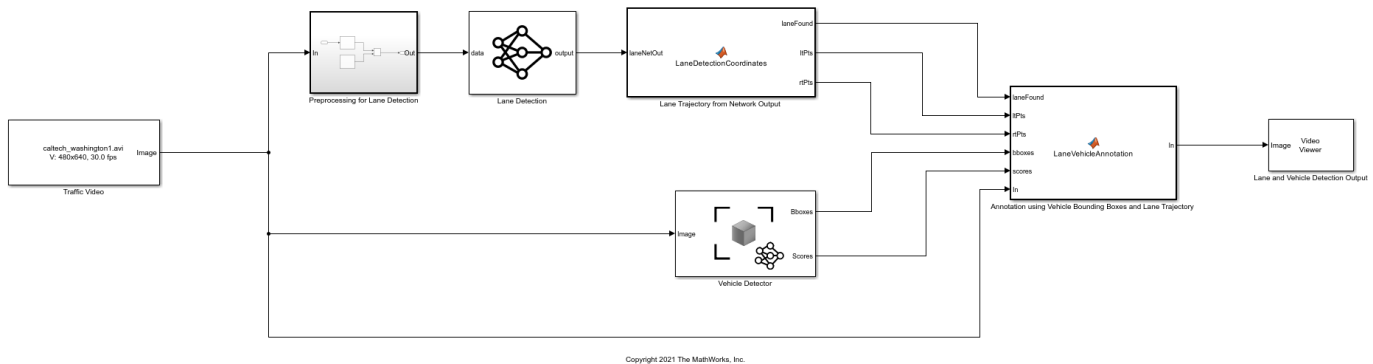
To test the model, the example uses the Caltech lanes data set. The file is approximately 16 MB in size. Download this file.

```
mediaFile = matlab.internal.examples.downloadSupportFile('gpcoder/media','caltech_washington1.avi');
```

Lane and Vehicle Detection Simulink Model

The following diagram shows the Simulink model for performing lane and vehicle detection on the traffic video. When the model runs, the Video Viewer block displays the traffic video with lane and vehicle annotations.

```
model='laneAndVehicleDetection';
open_system(model);
```



Set the file paths of the downloaded network model in the Predict and Detector blocks of the Simulink model.

```
set_param('laneAndVehicleDetection/Lane Detection','NetworkFilePath',lanenetFile)
set_param('laneAndVehicleDetection/Vehicle Detector','DetectorFilePath',vehiclenetFile)
```

Set the location of the test video that you load to the Simulink model.

```
set_param('laneAndVehicleDetection/Traffic Video','inputFileName',mediaFile)
```

Lane Detection

The Predict block loads the pretrained lane detection network from the `trainedLaneNet.mat` file. This network takes an image as an input and outputs two lane boundaries that correspond to the left and right lanes of the ego vehicle. Each lane boundary is represented by the parabolic equation:

$$y = ax^2 + bx + c$$

Here, y is the lateral offset and x is the longitudinal distance from the vehicle. The network outputs the three parameters a , b , and c per lane. The LaneDetectionCoordinates MATLAB Function block defines a function `lane_detection_coordinates` that takes the output from the predict block and outputs three parameters: `laneFound`, `ltPts`, and `rtPts`. The block uses thresholding to determine if both left and right lane boundaries are both found. If are found, `laneFound` is set to be true and the trajectories of the boundaries are calculated and stored in `ltPts` and `rtPts`.

```
type lane_detection_coordinates
```

```
function [laneFound,ltPts,rtPts] = lane_detection_coordinates(laneNetOut)

% Copyright 2020 The MathWorks, Inc.

persistent laneCoeffMeans;
if isempty(laneCoeffMeans)
    laneCoeffMeans = [-0.0002    0.0002    1.4740   -0.0002    0.0045   -1.3787];
end

persistent laneCoeffStds;
if isempty(laneCoeffStds)
    laneCoeffStds = [0.0030    0.0766    0.6313    0.0026    0.0736    0.9846];
end

params = laneNetOut .* laneCoeffStds + laneCoeffMeans;

isRightLaneFound = abs(params(6)) > 0.5; %c should be more than 0.5 for it to be a right lane
isLeftLaneFound = abs(params(3)) > 0.5;

persistent vehicleXPoints;
if isempty(vehicleXPoints)
    vehicleXPoints = 3:30; %meters, ahead of the sensor
end

ltPts = coder.nullcopy(zeros(28,2,'single'));
rtPts = coder.nullcopy(zeros(28,2,'single'));

if isRightLaneFound && isLeftLaneFound
    rtBoundary = params(4:6);
    rt_y = computeBoundaryModel(rtBoundary, vehicleXPoints);
    ltBoundary = params(1:3);
    lt_y = computeBoundaryModel(ltBoundary, vehicleXPoints);

    % Visualize lane boundaries of the ego vehicle
    tform = get_tformToImage;
    % map vehicle to image coordinates
    ltPts = tform.transformPointsInverse([vehicleXPoints', lt_y']);
    rtPts = tform.transformPointsInverse([vehicleXPoints', rt_y']);
    laneFound = true;
else
    laneFound = false;
end
end
```


Vehicle Detection

A YOLO v2 object detection network is composed of two subnetworks: a feature extraction network followed by a detection network. This pretrained network uses a ResNet -50 for feature extraction. The detection sub-network is a small CNN compared to the feature extraction network and is composed of a few convolutional layers and layers specific to YOLO v2. The Simulink model performs vehicle detection using the Object Detector block. This block takes an image as input and outputs the bounding box coordinates along with the confidence scores for vehicles in the image.

Annotation of Vehicle Bounding Boxes and Lane Trajectory in Traffic Video

The LaneVehicleAnnotation MATLAB Function block defines a function `lane_vehicle_annotation`, which annotates the vehicle bounding boxes with the confidence scores. Also, if `laneFound` is true, then the left and right lane boundaries stored in `ltPts` and `rtPts` are annotated in the traffic video.

```
type lane_vehicle_annotation

function In = lane_vehicle_annotation(laneFound, ltPts, rtPts, bboxes, scores, In)

% Copyright 2020 The MathWorks, Inc.

if ~isempty(bboxes)
    In = insertObjectAnnotation(In, 'rectangle', bboxes, scores);
end

pts = coder.nullcopy(zeros(28, 4, 'single'));
if laneFound
    prevpt = [ltPts(1,1) ltPts(1,2)];
    for k = 2:1:28
        pts(k,1:4) = [prevpt ltPts(k,1) ltPts(k,2)];
        prevpt = [ltPts(k,1) ltPts(k,2)];
    end
    In = insertShape(In, 'Line', pts, 'LineWidth', 2);
    prevpt = [rtPts(1,1) rtPts(1,2)];
    for k = 2:1:28
        pts(k,1:4) = [prevpt rtPts(k,1) rtPts(k,2)];
        prevpt = [rtPts(k,1) rtPts(k,2)];
    end
    In = insertShape(In, 'Line', pts, 'LineWidth', 2);
    In = insertMarker(In, ltPts);
    In = insertMarker(In, rtPts);
end

end
```

Get Pretrained Lane and Vehicle Detection Networks

This function downloads the `yolov2ResNet50VehicleExample.mat` file.

```
getVehicleDetectionAndLaneDetectionNetworks()
```

Run Simulation

Open Configuration Parameters dialog box. On the **Simulation Target** pane, in the **Deep Learning** group, select the Target library as MKL - DNN.

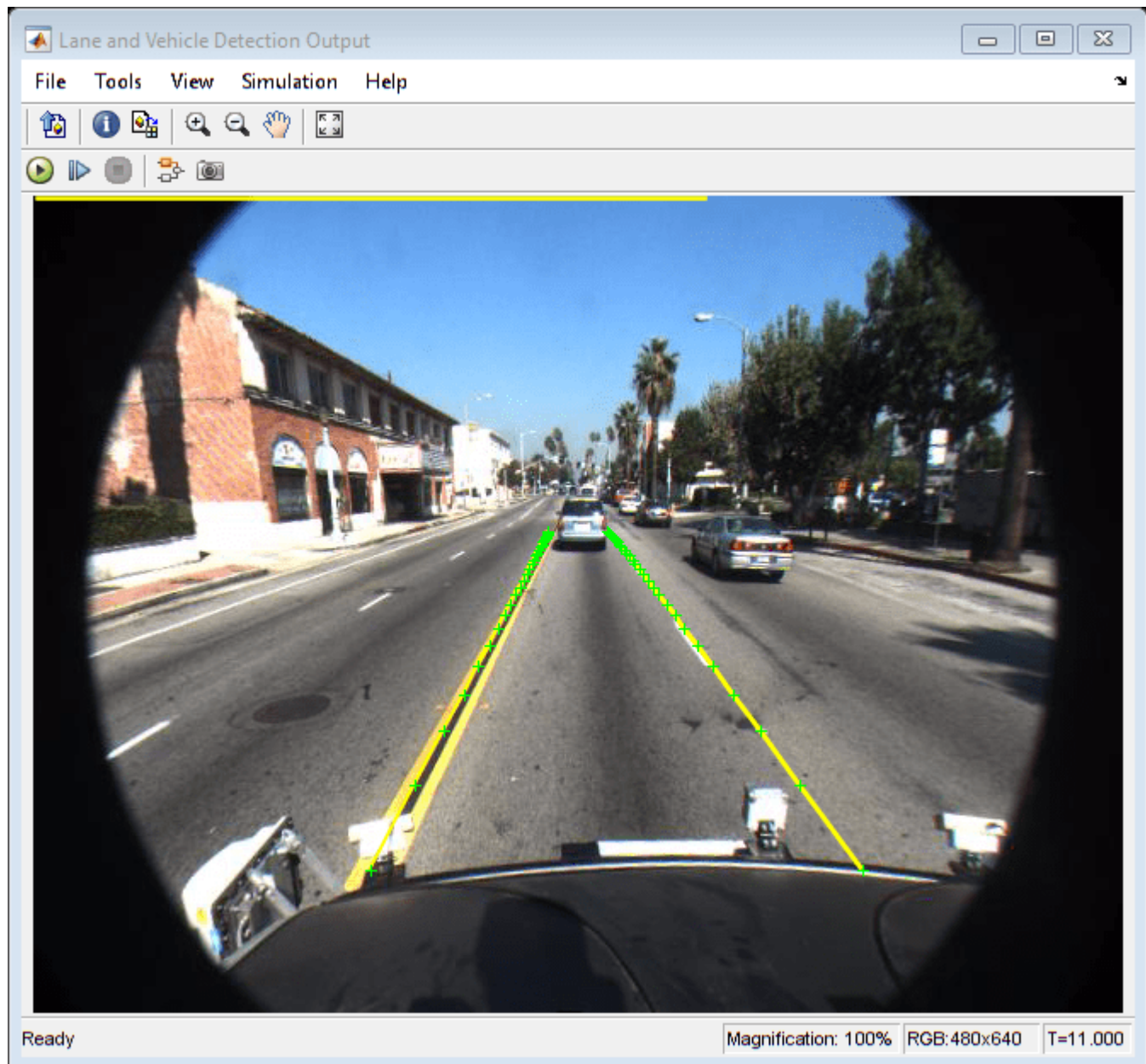
```
set_param(model, 'SimDLTargetLibrary', 'MKL-DNN');
```

On the **Interface** pane, in the **Deep Learning** group, select the **Target library** as MKL - DNN.

```
set_param(model, 'DLTargetLibrary', 'MKL-DNN');
```

To verify the lane and vehicle detection algorithms and display the lane trajectories, vehicle bounding boxes, and scores for the traffic video loaded in the Simulink model, run the simulation.

```
set_param('laneAndVehicleDetection', 'SimulationMode', 'Normal');  
sim('laneAndVehicleDetection');
```



Generate and Build Simulink Model

In the **Code Generation** pane, select the **Language** as **C++**.

```
set_param(model, 'TargetLang', 'C++');
```

Generate and build the Simulink model using the `slbuild` command. The code generator places the files in the `laneAndVehicleDetection_grt_rtw` build subfolder under your current working folder.

```
currentDir = pwd;
status = evalc("slbuild('laneAndVehicleDetection')");
```

Generated C++ Code

The subfolder named `laneAndVehicleDetection_grt_rtw` contains the generated C++ code corresponding to the different blocks in the Simulink model and the specific operations being performed in those blocks. For example, the file `trainedLaneNet0_0.h` contains the C++ class, which contains attributes and member functions representing the pretrained lane detection network.

```
hfile = fullfile(currentDir, 'laneAndVehicleDetection_grt_rtw', ...
    'trainedLaneNet0_0.h');
rtwdemodbtype(hfile, '#ifndef RTW_HEADER_trainedLaneNet0_0_h_', ...
    '#endif')
```

```
#ifndef RTW_HEADER_trainedLaneNet0_0_h_
#define RTW_HEADER_trainedLaneNet0_0_h_
#include "MWOndnnTargetNetworkImpl.hpp"
#include "rtwtypes.h"
#include "MWTensorBase.hpp"
#include "MWTensor.hpp"
#include "MWCNNLayer.hpp"
#include "MWInputLayer.hpp"
#include "MWElementwiseAffineLayer.hpp"
#include "MWFusedConvActivationLayer.hpp"
#include "MWNormLayer.hpp"
#include "MWMaxPoolingLayer.hpp"
#include "MWFCLayer.hpp"
#include "MWReLULayer.hpp"
#include "MWOutputLayer.hpp"
#include "MWConvLayer.hpp"
#include "MWYoloExtractionLayer.hpp"
#include "MWSigmoidLayer.hpp"
#include "MWExponentialLayer.hpp"
#include "MWYoloSoftmaxLayer.hpp"
#include "MWConcatenationLayer.hpp"
#include "MWActivationFunctionType.hpp"
#include "MWRNNParameterTypes.hpp"
#include "MWTargetTypes.hpp"
#include "shared_layers_export_macros.hpp"
#include "MWOndnnUtils.hpp"
#include "MWOndnnCustomLayerBase.hpp"
#include "MWOndnnCommonHeaders.hpp"
```

```
class trainedLaneNet0_0
{
public:
```

```

    boolean_T isInitialized;
    boolean_T matlabCodegenIsDeleted;
    trainedLaneNet0_0();
    void setSize();
    void resetState();
    void setup();
    void predict();
    void cleanup();
    real32_T *getLayerOutput(int32_T layerIndex, int32_T portIndex);
    int32_T getLayerOutputSize(int32_T layerIndex, int32_T portIndex);
    real32_T *getInputDataPointer(int32_T index);
    real32_T *getInputDataPointer();
    real32_T *getOutputDataPointer(int32_T index);
    real32_T *getOutputDataPointer();
    int32_T getBatchSize();
    ~trainedLaneNet0_0();
private:
    int32_T numLayers;
    MWTensorBase *inputTensors;
    MWTensorBase *outputTensors;
    MWCNNLayer *layers[18];
    MWOnednnTarget::MWTargetNetworkImpl *targetImpl;
    void allocate();
    void postsetup();
    void deallocate();
};

```

Similarly, the file `yolov2ResNet50VehicleExample0_0.h` contains the C++ class, which represents the pretrained YOLO v2 detection network.

```

hfile = fullfile(currentDir, 'laneAndVehicleDetection_grt_rtw',...
    'yolov2ResNet50VehicleExample0_0.h');
rtwdemodbtype(hfile, '#ifndef RTW_HEADER_yolov2ResNet50VehicleExample0_0_h_',...
    '#endif')

#ifndef RTW_HEADER_yolov2ResNet50VehicleExample0_0_h_
#define RTW_HEADER_yolov2ResNet50VehicleExample0_0_h_
#include "MWOnednnTargetNetworkImpl.hpp"
#include "rtwtypes.h"
#include "MWTensorBase.hpp"
#include "MWTensor.hpp"
#include "MWCNNLayer.hpp"
#include "MWInputLayer.hpp"
#include "MWElementwiseAffineLayer.hpp"
#include "MWFusedConvActivationLayer.hpp"
#include "MWNormLayer.hpp"
#include "MWMaxPoolingLayer.hpp"
#include "MWFCLayer.hpp"
#include "MWReLULayer.hpp"
#include "MWOutputLayer.hpp"
#include "MWConvLayer.hpp"
#include "MWYoloExtractionLayer.hpp"
#include "MWSigmoidLayer.hpp"
#include "MWExponentialLayer.hpp"
#include "MWYoloSoftmaxLayer.hpp"
#include "MWConcatenationLayer.hpp"
#include "MWActivationFunctionType.hpp"

```

```

#include "MWRNNParameterTypes.hpp"
#include "MWTargetTypes.hpp"
#include "shared_layers_export_macros.hpp"
#include "MW0nednnUtils.hpp"
#include "MW0nednnCustomLayerBase.hpp"
#include "MW0nednnCommonHeaders.hpp"

class yolov2ResNet50VehicleExample0_0
{
public:
    boolean_T isInitialized;
    boolean_T matlabCodegenIsDeleted;
    yolov2ResNet50VehicleExample0_0();
    void setSize();
    void resetState();
    void setup();
    void predict();
    void activations(int32_T layerIdx);
    void cleanup();
    real32_T *getLayerOutput(int32_T layerIndex, int32_T portIndex);
    int32_T getLayerOutputSize(int32_T layerIndex, int32_T portIndex);
    real32_T *getInputDataPointer(int32_T index);
    real32_T *getInputDataPointer();
    real32_T *getOutputDataPointer(int32_T index);
    real32_T *getOutputDataPointer();
    int32_T getBatchSize();
    ~yolov2ResNet50VehicleExample0_0();
private:
    int32_T numLayers;
    MWTensorBase *inputTensors;
    MWTensorBase *outputTensors;
    MWCNNLayer *layers[57];
    MW0nednnTarget::MWTargetNetworkImpl *targetImpl;
    void allocate();
    void postsetup();
    void deallocate();
};

```

Note: If the **System target file** parameter is set to `grt.tlc`, you must select the **Generate code only** model configuration parameter. If you set the **System target file** to `ert.tlc`, you can clear the **Generate code only** parameter, but to create an executable file, you must generate an example main program.

Cleanup

Close the Simulink model.

```

save_system('laneAndVehicleDetection');
close_system('laneAndVehicleDetection');

```

Generate Generic C/C++ for Sequence-to-Sequence Deep Learning Simulink Models

This example shows how to generate a generic C/C++ executable that does not depend on any third-party deep learning libraries from a deep learning Simulink® model. The example predicts the remaining useful life (RUL) of a turbofan engine, measured in cycles at each step of an input time series that represents data from various sensors in an engine.

For a video demonstration of this example, see [Generate Generic C/C++ Code for Deep Learning Networks in Simulink](#).

This example is based on the *Generate Generic C/C++ Code for Sequence-to-Sequence Regression That Uses Deep Learning* example from MATLAB® Coder™. For more information, see “Generate Generic C/C++ Code for Sequence-to-Sequence Regression That Uses Deep Learning”.

Prerequisites

For PIL verification, you will need:

- STMicroelectronics STM32F407G-Discovery, STM32F746G-Discovery, or STM32F769I-Discovery board
- USB type A to Mini-B cable
- USB TTL-232 cable - TTL-232R 3.3V (serial communication for STM32F4-Discovery board)
- Embedded Coder® Support Package for STMicroelectronics Discovery Boards. To install this support package, use the **Add-Ons** menu in the **Environment** section of the MATLAB **Home** tab.

Pretrained RUL Network

This example uses a pretrained LSTM network to predict the remaining useful life of an engine, measured in cycles. The LSTM network consists of an LSTM layer with 200 hidden units, followed by a fully connected layer of size 50 and a dropout layer with dropout probability 0.5. The network was trained using the *Turbofan Engine Degradation Simulation Data Set* as described in [1]. The training data contains simulated time series data for 100 engines. Each sequence varies in length and corresponds to a full run to failure (RTF) instance. The test data contains 100 partial sequences and corresponding values of the remaining useful life at the end of each sequence. For more information on training the network, see the example “Sequence-to-Sequence Regression Using Deep Learning” (Deep Learning Toolbox).

```
net = coder.loadDeepLearningNetwork('rulNetwork.mat')
```

```
net =
  SeriesNetwork with properties:
    Layers: [6x1 nnet.cnn.layer.Layer]
    InputNames: {'sequenceinput'}
    OutputNames: {'regressionoutput'}
```

For information on the networks, layers, and classes supported for code generation, see “Networks and Layers Supported for Code Generation”.

Download and Prepare Test Data

This section summarizes the steps to download and prepare the test data that this example uses. For more information on the Turbofan Engine Degradation Simulation data set and the preprocessing steps, see the example “Sequence-to-Sequence Regression Using Deep Learning” (Deep Learning Toolbox).

Download Data Set

Create a directory to store the Turbofan Engine Degradation Simulation data set.

```
dataFolder = fullfile(tempdir,"turbofan");
if ~exist(dataFolder,'dir')
    mkdir(dataFolder);
end
```

Download and extract the Turbofan Engine Degradation Simulation data set.

```
filename = matlab.internal.examples.downloadSupportFile("nnet","data/TurbofanEngineDegradationSimulationData");
unzip(filename,dataFolder)
```

Calculate Mean and Standard Deviation of Training Data

In the following step, you normalize the test predictors using the mean and standard deviation of the training data. So, you must first use the training data to calculate these normalization parameters.

Load the training data, each column is one observation, each row is one feature. Remove the features that have constant values.

```
filenamePredictors = fullfile(dataFolder,"train_FD001.txt");
[XTrain] = processTurboFanDataTrain(filenamePredictors);

m = min([XTrain{:}],[],2);
M = max([XTrain{:}],[],2);
idxConstant = M == m;

for i = 1:numel(XTrain)
    XTrain{i}(idxConstant,:) = [];
end
```

Calculate the mean and standard deviation over all observations.

```
mu = mean([XTrain{:}],2);
sig = std([XTrain{:}],0,2);
```

Prepare Test Data

Prepare the test data using the function `processTurboFanDataTest` attached to this example. The function `processTurboFanDataTest` extracts the data from `filenamePredictors` and `filenameResponses` and returns the cell arrays `XValidate` and `YValidate`, which contain the test predictor and response sequences, respectively.

```
filenamePredictors = fullfile(dataFolder,"test_FD001.txt");
filenameResponses = fullfile(dataFolder,"RUL_FD001.txt");
[XValidate,YValidate] = processTurboFanDataTest(filenamePredictors,filenameResponses);
```

Remove features with constant values using `idxConstant` calculated from the training data. Normalize the test predictors using the parameters `mu` and `sig` calculated from the training data.

Clip the test responses at the threshold 150. This same clipping threshold was used on the training data while training the network.

```
thr = 150;
for i = 1:numel(XValidate)
    XValidate{i}(idxConstant,:) = [];
    XValidate{i} = (XValidate{i} - mu) ./ sig;
    YValidate{i}(YValidate{i} > thr) = thr;
end
```

To make the input validation data compatible with Simulink code generation, the sequence lengths for each of the independent 100 observations are zero-padded to create uniformly-sized, 17-by-303 input arrays.

```
sequenceLengths = cellfun(@length,XValidate,'UniformOutput',true);
maxSequenceLen = max(sequenceLengths);
padFcn = @(x) [x,zeros(size(x,1),maxSequenceLen-size(x,2))];
XValidatePad = cellfun(padFcn,XValidate,'UniformOutput',false);
```

The padded values are then converted to a 17-by-303-100 numeric array. To import this data into the Simulink model, specify a structure variable containing the data values and an empty time vector. During simulation, the input for the first time step is read from the first 17-by-303 element of the array. The value for the second time step is read from the second element, and so on, for a total of 100 steps.

```
simin.time = [];
simin.signals.values = cell2mat(reshape(XValidatePad,1,1,[]));
simin.signals.dimensions = size(XValidatePad{1});
```

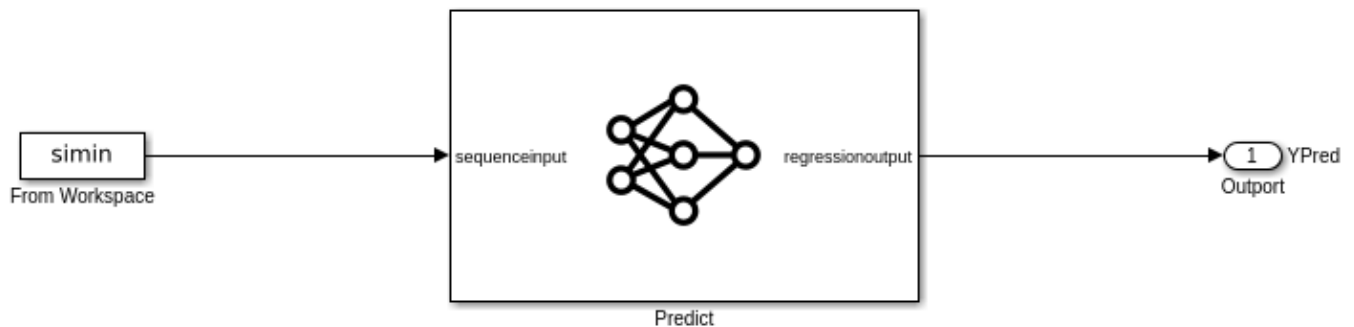
Simulink Model for Prediction

The Simulink model for predicts the remaining useful life of a turbofan engine is shown. The model uses the Predict block from the Deep Neural Networks library that imports the trained network from a rulNetwork MAT-file. Additionally, the Mini-batch size parameter of the block is set to 1.

```
model = 'rulPredict';
open_system(model)
```


?

Generate Generic C++ Code for Sequence-to-Sequence Regression Deep Learning Models



Copyright 2021 The MathWorks, Inc.

Run the Simulation

To validate the Simulink model, run the simulation.

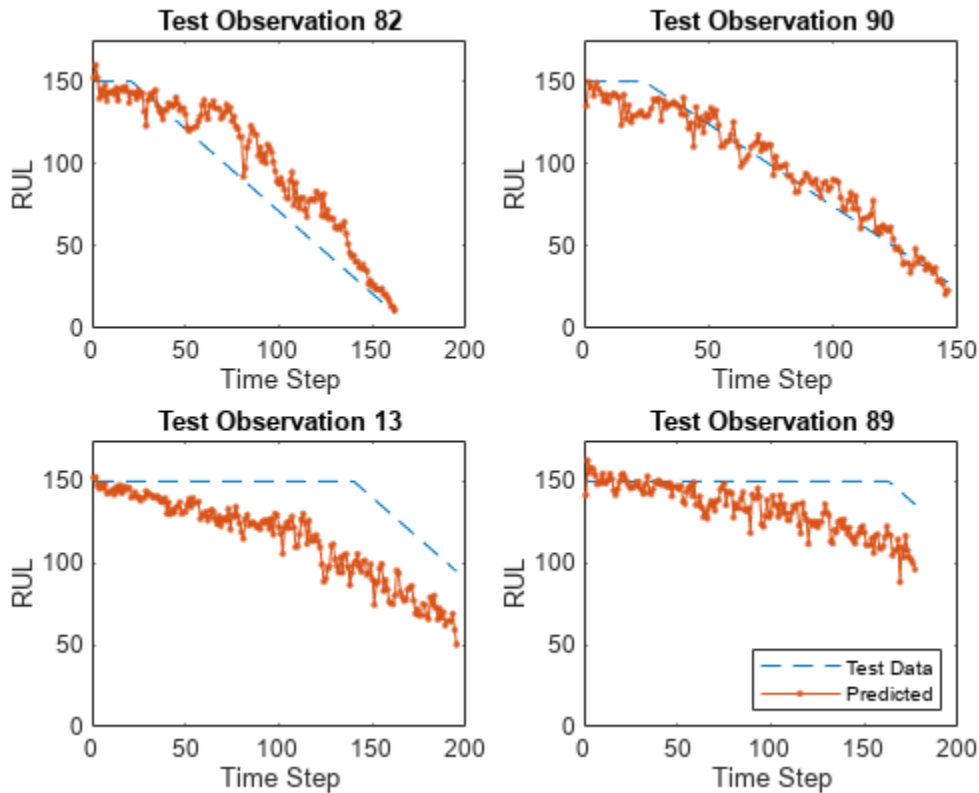
```
set_param(model, 'SimulationMode', 'Normal');
sim(model);
```

The output YPred of the Simulink model contains the predicted remaining useful life values from the network. This output is first trimmed to remove the results from the zero-padded values and then converted to a cell array.

```
YPred_cell = squeeze(mat2cell(YPred,1,maxSequenceLen,ones(1,100)));
for t = 1:length(sequenceLengths)
    YPred_cell{t}(:,sequenceLengths(t) + 1:end) = [];
end
```

Plot the predicted remaining useful life (RUL) values for four randomly-selected observations and compare them to the validation data.

```
observationIdx = randperm(100,4);
rulExamplePlots(observationIdx,YValidate,YPred_cell);
```



Configure Model for Code Generation

To generate C/C++ code that does not depend on any third-party deep learning libraries and build an executable, select the generic real-time (ERT) target from Embedded Coder™.

```
cs = getActiveConfigSet(model);
switchTarget(cs, 'ert.tlc', []);
```

Configure code generation specific parameters. Setting MAT-file logging enables the generated code to redirect simulation data to a MAT-file.

```
set_param(cs, 'TargetLang', 'C');
set_param(cs, 'Toolchain', 'Automatically locate an installed toolchain');
set_param(cs, 'ObjectivePriorities', 'Execution efficiency');
set_param(cs, 'DLTargetLibrary', 'None');
set_param(cs, 'GenerateReport', 'on');
set_param(cs, 'CodeInterfacePackaging', 'Nonreusable function');
set_param(cs, 'MatFileLogging', 'on');
```

Generate and Build the Simulink Model

Generate and build the Simulink model by using the `slbuild` command. The code generator places the files in a *build folder*, a subfolder named `rulPredict_ert_rtw` under your current working folder.

```
evalc("slbuild('rulPredict')");
```

Run the Generated Executable

Run the generated executable. After successful execution, the `rulPredict` executable creates a `rulPredict` MAT-file in the current working folder.

```
if ispc
    status = system('rulPredict.exe');
else
    status = system('./rulPredict');
end
```

```
** created rulPredict.mat **
```

```
load('rulPredict.mat')
```

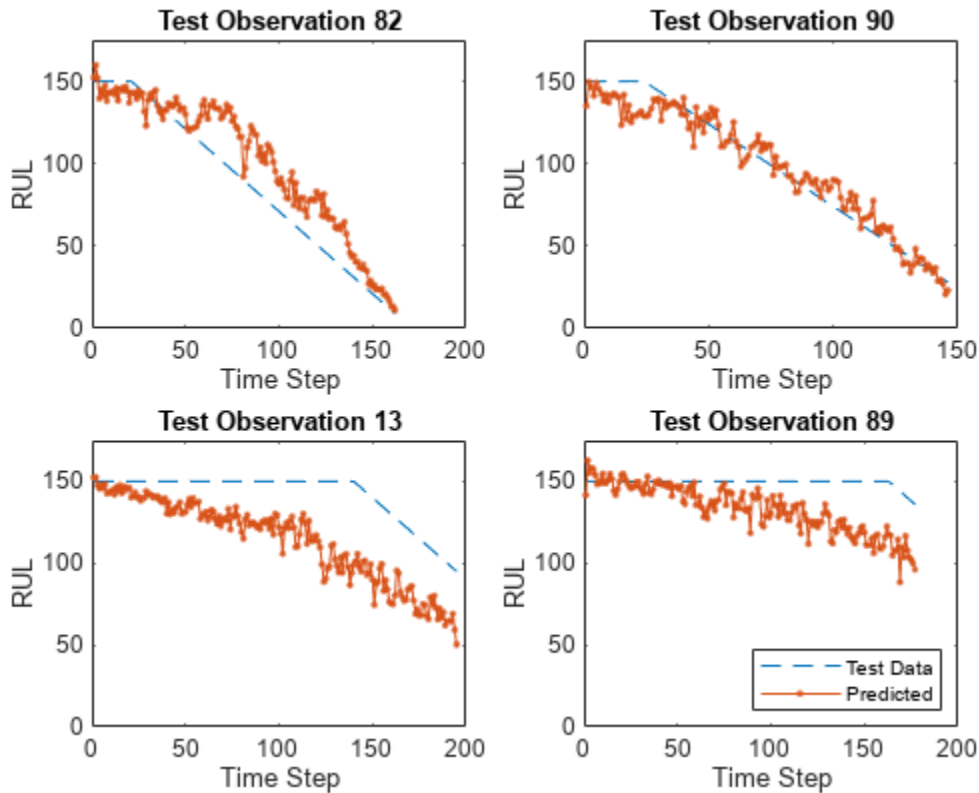
The output `rt_YPred` contains the predicted remaining useful life values from the network. This output is first trimmed to remove the results from the zero-padded values and then converted to a cell array.

```
rt_YPred_cell = squeeze(mat2cell(rt_YPred,1,maxSequenceLen,ones(1,100)));
```

```
for t = 1:length(sequenceLengths)
    rt_YPred_cell{t}(:,sequenceLengths(t) + 1:end) = [];
end
```

Plot the predicted remaining useful life (RUL) values for four randomly-selected observations and compare them to the validation data.

```
rulExamplePlots(observationIdx,YValidate,rt_YPred_cell);
```



Code Verification with Processor-in-the-Loop (PIL) Simulation

To ensure that the behavior of the deployment code matches the design, you can configure the Simulink model to run a PIL simulation on embedded boards. In a PIL simulation, the generated code runs on embedded boards such as the STMicroelectronics® Discovery. The results of the PIL simulation are transferred to Simulink to verify numerical equivalence of the simulation and the code generation results.

This example shows PIL verification on the STMicroelectronics Discovery board by using the *Embedded Coder® Support Package for STMicroelectronics Discovery Boards*. This example is pre-configured to run on the STM32F746G-Discovery board. You can configure this model to use other supported embedded boards by selecting them as the "Hardware board" on the Hardware Implementation pane of the Model Configuration Parameters.

Prepare Test Data

For PIL simulation, this example sorts the observations by sequence length and picks the first 10. These observations are zero-padded to create uniformly-sized, 17-by-54 input arrays. The padded values are then converted to a 17-by-54-10 numeric array. To import this data into the Simulink model, specify a structure variable containing the data values and an empty time vector. During simulation, the input for the first time step is read from the first 17-by-54 element of the array. The value for the second time step is read from the second element, and so on, for a total of 10 steps.

```
[~,idx] = sort(cellfun(@length,XValidate));
XValidatePIL = XValidate(idx(1:10));
YValidatePIL = YValidate(idx(1:10));
```

```

sequenceLengths = cellfun(@length,XValidatePIL,'UniformOutput',true);
maxSequenceLen = max(sequenceLengths);
padFcn = @(x) [x,zeros(size(x,1),maxSequenceLen-size(x,2))];
XValidatePILPad = cellfun(padFcn,XValidatePIL,'UniformOutput',false);
simin.time = [];
simin.signals.values = cell2mat(reshape(XValidatePILPad,1,1,[]));
simin.signals.dimensions = size(XValidatePILPad{1});

```

Top Model PIL

The `rulPredict_pil` is a modified version of the `rulPredict` model for PIL verification. To load the test vectors, the `rulPredict_pil` model replaces the From Workspace block with an Inport block. The Inport block is configured to accept a 17-by-54 array of double data type.

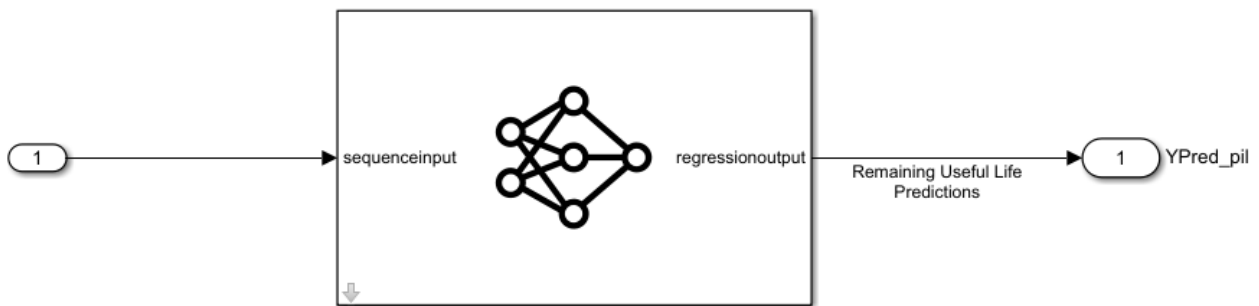
```

pilModel = 'rulPredict_pil';
open_system(pilModel);

```



Top Model PIL



Copyright 2021 The MathWorks, Inc.

Configure the `rulPredict_pil` model for the **STM32F467G-Discovery** target.

```

cs = getActiveConfigSet(pilModel);
set_param(cs,'HardwareBoard','STM32F746G-Discovery');
set_param(cs,'Toolchain','GNU Tools for ARM Embedded Processors');

```

The STM32F4-Discovery board supports two different communication interfaces for PIL: ST-LINK and serial. The ST-LINK communication interface does not require any additional cables or hardware besides a USB type A to Mini-B cable used to connect the STM32F4-Discovery board to the host computer. The serial communication interface requires a USB TTL-232 cable. Running a PIL simulation using the serial communication interface is much faster than the running a PIL simulation using ST-LINK. This example is configured to use the serial interface.

Additionally, you must set the COM port parameter in Configuration Parameters > Hardware Implementation > Target Hardware Resources > PIL to match the port number of the serial interface on your Windows computer.

Run PIL Simulation

To compare the PIL results with simulation, run the `rulPredict_pil` model in normal mode and then in PIL mode.

```
set_param(cs, 'LoadExternalInput', 'on');
set_param(cs, 'ExternalInput', 'simin');
set_param(pilModel, 'SimulationMode', 'Normal');

sim(pilModel);
YPred_ref = YPred_pil;

set_param(pilModel, 'SimulationMode', 'Processor-in-the-Loop (PIL)');
sim(pilModel);
```

Plot Results

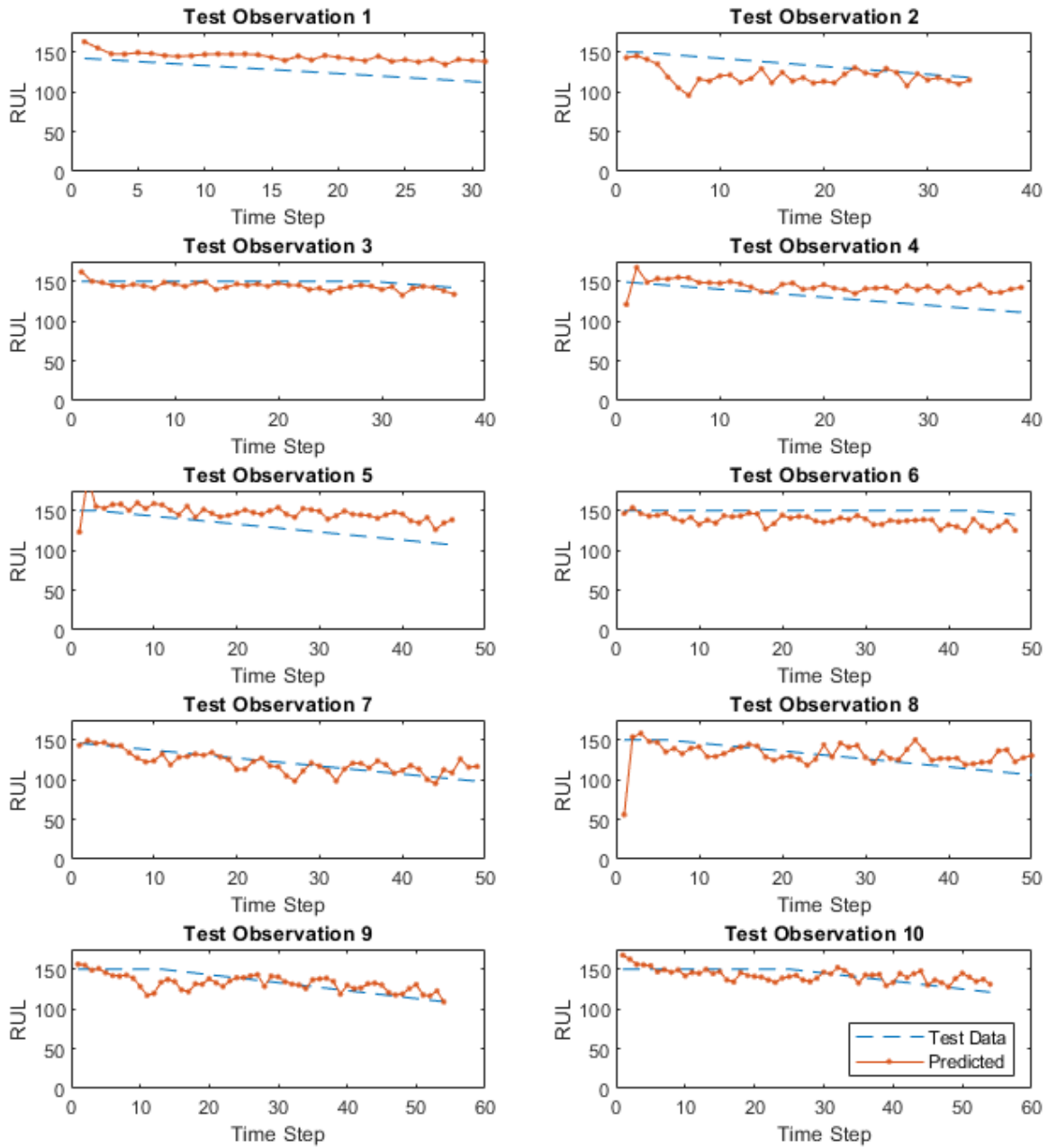
The output `YPred_pil` and `YPred_ref` contain the predicted remaining useful life values from the network. This output is first trimmed to remove the results from the zero-padded values and then converted to a cell array.

Plot the predicted remaining useful life (RUL) values for PIL and compare them to the validation data.

```
YPred_ref_cell = squeeze(mat2cell(YPred_ref,1,maxSequenceLen,ones(1,10)));
YPred_pil_cell = squeeze(mat2cell(YPred_pil,1,maxSequenceLen,ones(1,10)));

for t = 1:length(sequenceLengths)
    YPred_ref_cell{t}(:,sequenceLengths(t) + 1:end) = [];
    YPred_pil_cell{t}(:,sequenceLengths(t) + 1:end) = [];
end

rulExamplePlots([1:10],YValidatePIL,YPred_pil_cell);
```



Compute the error from the PIL simulation and plot the results.

```
YPred_error = YPred_ref-YPred_pil;
```

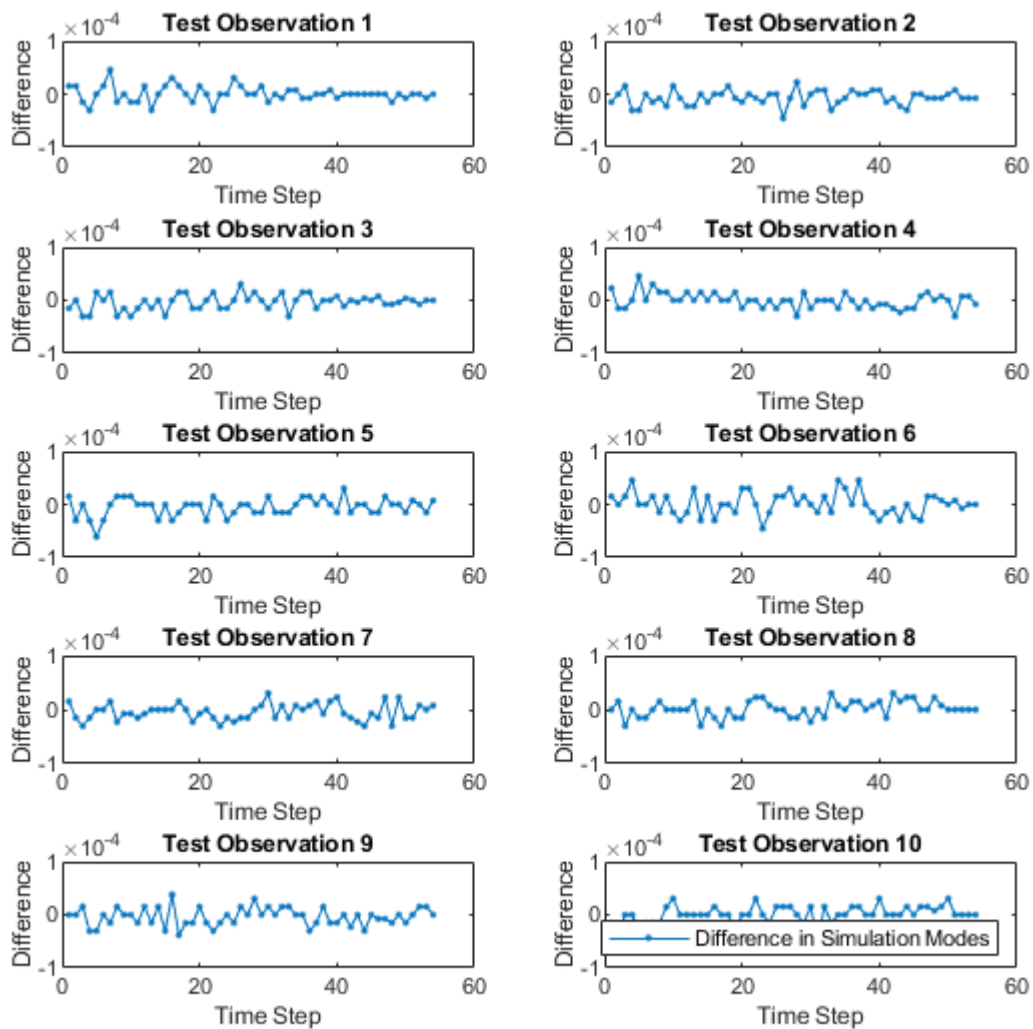
```
figure('Name', 'PIL Error')
```

```

for i = 1:10
    subplot(5,2,i)

    plot(YPred_error(:,:,i),'.-')
    maxerror = string(max(YPred_error(:,:,i)));
    ylim([-1e-4 1e-4])
    title("Test Observation " + i)
    xlabel("Time Step")
    ylabel("Difference")
end
legend(["Difference in Simulation Modes"],'Location','southeast')

```



Supporting Functions

```

function rulExamplePlots(observationIdx,YTest,YPred)
N = numel(observationIdx);

```



```
figure
for i = 1:N
    subplot(N/2,2,i)

    plot(YTest{observationIdx(i)},'--')
    hold on
    plot(YPred{observationIdx(i)},'.-')
    hold off

    ylim([0 175])
    title("Test Observation " + observationIdx(i))
    xlabel("Time Step")
    ylabel("RUL")
end
legend(["Test Data" "Predicted"],'Location','southeast')

end
```

References

- 1 Saxena, Abhinav, Kai Goebel, Don Simon, and Neil Eklund. "Damage propagation modeling for aircraft engine run-to-failure simulation." In *Prognostics and Health Management, 2008. PHM 2008. International Conference on*, pp. 1-9. IEEE, 2008.

Generate Code for Deep Learning Networks Using MATLAB Function Block

You can generate optimized code for Simulink models containing a variety of trained deep learning networks. You can implement the deep learning functionality in Simulink by using MATLAB Function blocks or by using blocks from the **Deep Neural Networks** library. When implementing deep learning functionality with MATLAB Function blocks, use the `coder.loadDeepLearningNetwork` function to load a trained deep learning network and use the object functions of the network object to obtain the desired responses. You can configure the code generator to take advantage of the Intel Math Kernel library for Deep Neural Networks (MKL-DNN). You can also generate generic C or C++ code that does not depend on third-party libraries. The generated code implements the deep convolutional neural network (CNN) by using the architecture, the layers, and parameters that you specify in a network object.

Classify Images by Using GoogLeNet

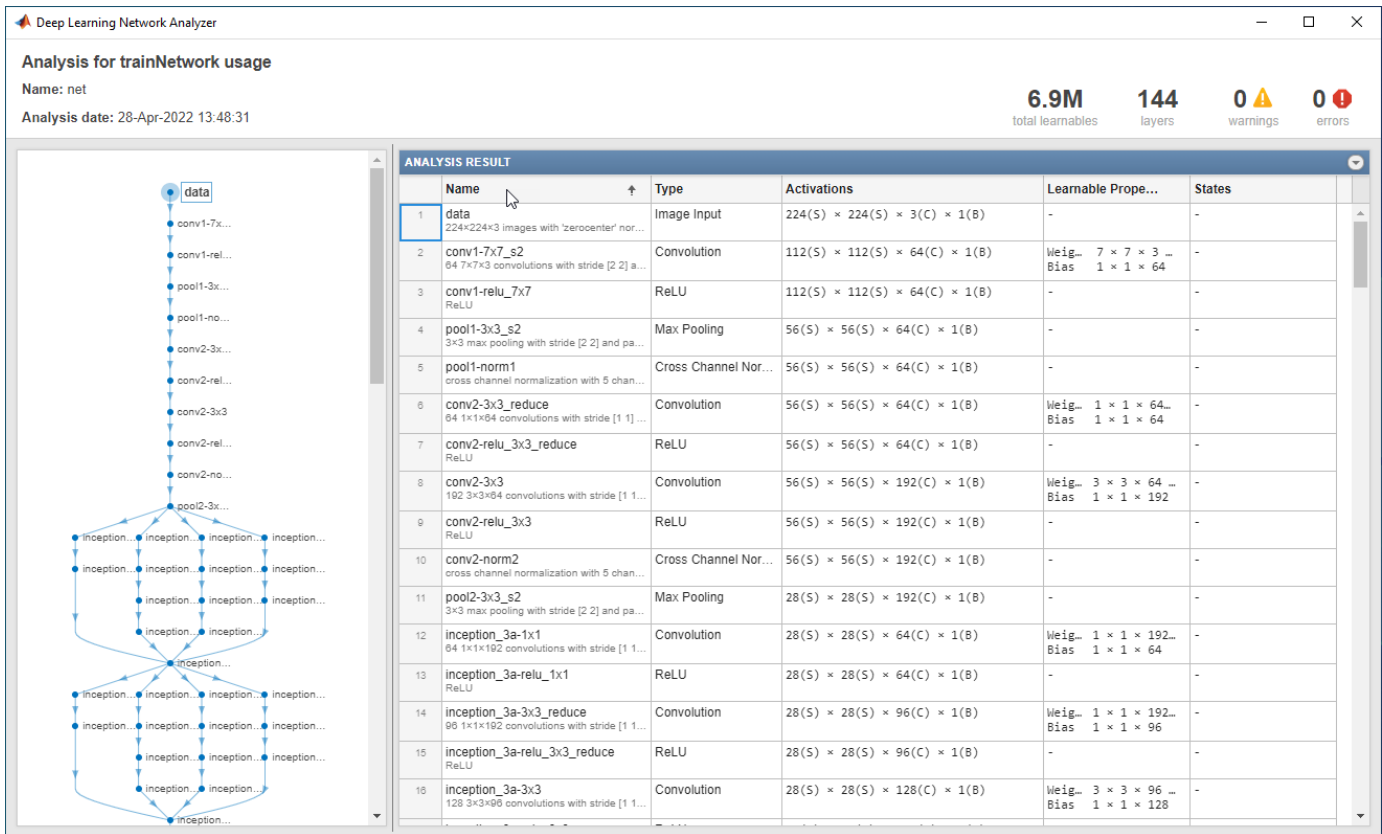
GoogLeNet is a convolutional neural network that is 22 layers deep. The network has been trained on over a million images and can classify an image into 1000 object categories (such as a keyboard, coffee mug, pencil, and animals). The network takes an image as input, and then outputs a label for the object in the image together with the probabilities for each of the object categories. This example shows you how to perform simulation and generate C++ code for the pretrained GoogLeNet deep convolutional neural network and classify an image.

- 1 Load the pretrained GoogLeNet network. You can choose to load a different pretrained network for image classification. This function requires the Deep Learning Toolbox Model for GoogLeNet Network support package. If this support package is not installed, then the function provides a download link.

```
net = googlenet;
```

- 2 The object `net` contains the `DAGNetwork` object. Use the function to display an interactive visualization of the network architecture, to detect errors and issues in the network, and to display detailed information about the network layers. The layer information includes the sizes of layer activations and learnable parameters, the total number of learnable parameters, and the sizes of state parameters of recurrent layers.

```
analyzeNetwork(net);
```



- The image that you want to classify must have the same size as the input size of the network. For GoogLeNet, the size of the `imageInputLayer` is 224-by-224-by-3. The `Classes` property of the output `classificationLayer` contains the names the names of the classes learned by the network. View 10 random class names out of the total of 1000.

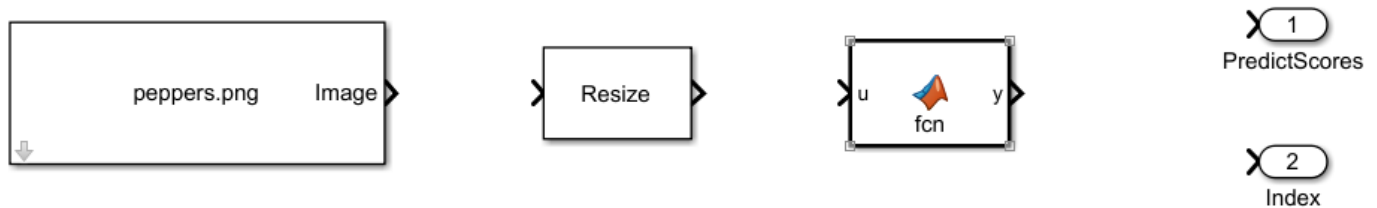
```
classNames = net.Layers(end).Classes;
numClasses = numel(classNames);
disp(classNames(randperm(numClasses, 10)))
```

```
speedboat
window screen
isopod
wooden spoon
lipstick
drake
hyena
dumbbell
strawberry
custard apple
```

Create GoogLeNet Model

- Create a Simulink model and insert a MATLAB Function block from the **User-Defined Functions** library.
- Add an Image From File block from the **Computer Vision Toolbox** library and set the File name parameter to `peppers.png`.

- 3 Add a Resize block from the **Computer Vision Toolbox** library to the model. Set the **Specify** parameter of the Resize block to **Number of output rows and columns** and enter [224 224] as the value for **Number of output rows and columns**. This block resizes the input image to that of the input layer of the network.



- 4 Double-click the MATLAB Function block. A default function signature appears in the MATLAB Function Block Editor.
- 5 Define a function called `googlenet_predict`, which implements the prediction entry-point function. The function header declares `in` as an argument to the `googlenet_predict` function, with `scores` and `indxTop` as the return value.

```
function [scores,indxTop] = googlenet_predict(in) %#codegen

persistent mynet;

if isempty(mynet)
    mynet = coder.LoadDeepLearningNetwork('googlenet');
end

% pass in input
predict_scores = predict(mynet,in);
[scores,indx] = sort(predict_scores, 'descend');
indxTop = indx(1:5);
```

A persistent object `mynet` loads the `DAGNetwork` object. At the first call to the entry-point function, the persistent object is constructed and set up. On subsequent calls to the function, the same object is reused to call `predict` on inputs, avoiding reconstructing and reloading the network object.

You can use the `activations` method to network activations for a specific layer. For example, the following line of code returns the network activations for the layer specified in `layerIdx`.

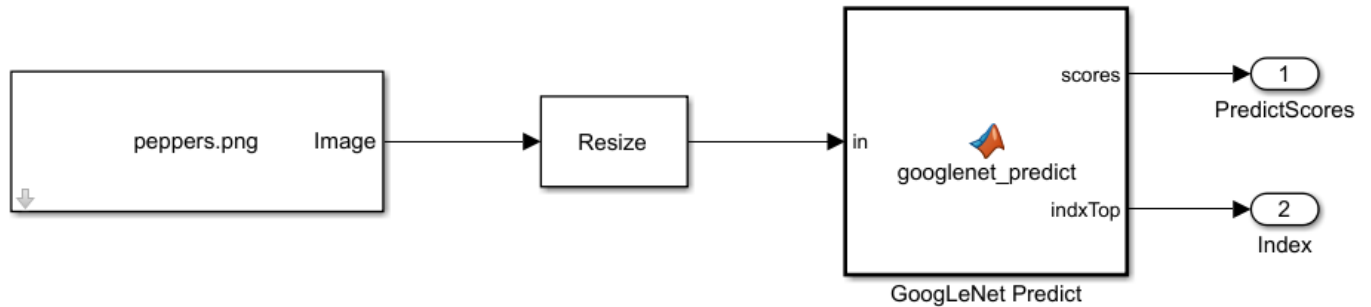
```
out = activations(mynet,in,layerIdx,'OutputAs','Channels');
```

You can use the `classify` method to predict class labels for the image data in `in` using the trained network `mynet`.

```
[out,scores] = classify(mynet,in);
```

For LSTM networks, you can use the `predictAndUpdateState` and `resetState` methods.

- 6 Open the block parameters of the MATLAB Function block. On the **Code Generation** tab, select **Reusable function** for **Function packaging**.
- 7 Connect these blocks as shown in the diagram. Save the model as `googlenetModel`.



Simulate Model

- 1 Open the Configuration Parameters dialog box. Select the **Solver** pane. Set the **Type** parameter to Fixed-step. This setting maintains a constant (fixed) step size, which is required for code generation.
- 2 Select the **Simulation Target** pane. Set the **Language** to C++.
- 3 On the **Simulation Target** pane, set the **Target Library** parameter in the **Deep learning** group to MKL-DNN.
- 4 Click **OK** to save and close the Configuration Parameters dialog box.
- 5 To build and simulate the model, use this command:

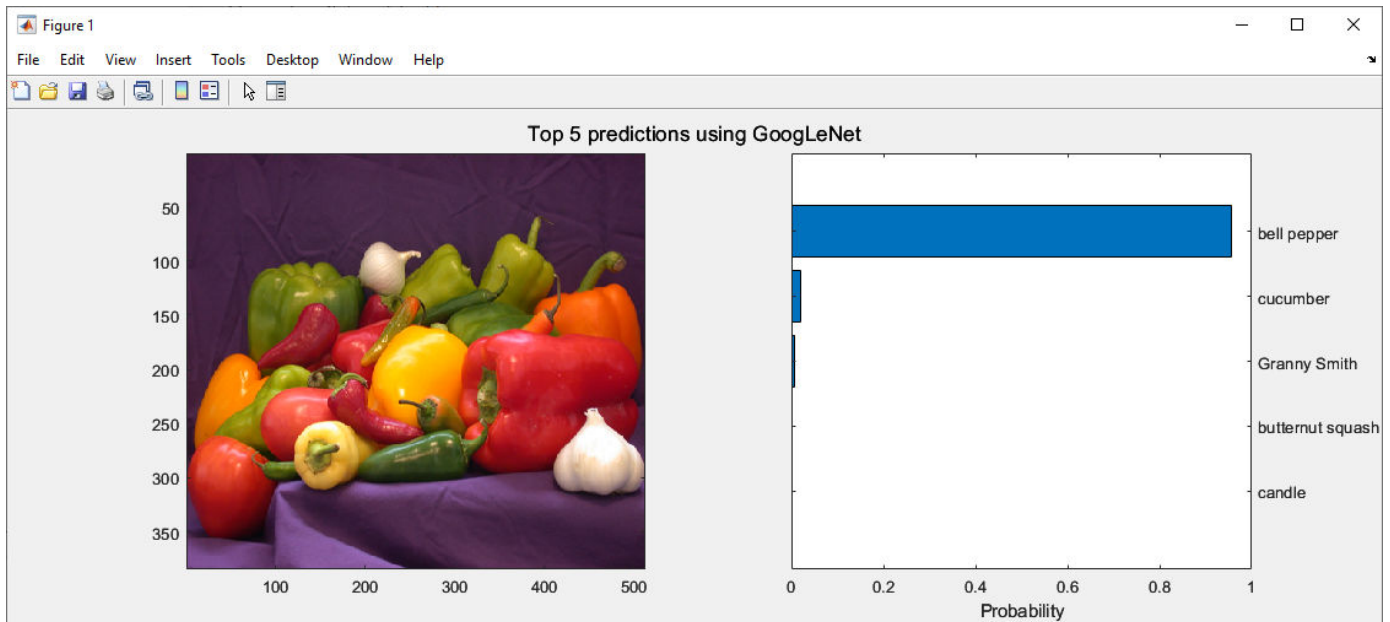
```
out = sim('googlenetModel');
```

- 6 Display the top five predicted labels and their associated probabilities as a histogram. Because the network classifies images into so many object categories, and many categories are similar, it is common to consider the top-five accuracy when evaluating networks. The network classifies the image as a bell pepper with a high probability.

```
im = imread('peppers.png');
classNamesTop = classNames(out.yout{2}.Values.Data(:,:,1))
```

```
h = figure;
h.Position(3) = 2*h.Position(3);
ax1 = subplot(1,2,1);
ax2 = subplot(1,2,2);
```

```
image(ax1,im);
barh(ax2,out.yout{1}.Values.Data(1,5:-1:1,1))
xlabel(ax2,'Probability')
yticklabels(ax2,classNamesTop(5:-1:1))
ax2.YAxisLocation = 'right';
sgtitle('Top 5 predictions using GoogLeNet')
```



Generate Code

- 1 Open the **Simulink Coder** or **Embedded Coder** app. The **C Code** tab opens.
- 2 In the Configuration Parameters dialog box, if you have a Simulink Coder license, set the **System target file** to `grt.tlc`. If you have Embedded Coder, set the **System target file** parameter to `ert.tlc`.
- 3 Set the **Language** to C++.
- 4 Select **Generate code only**.
- 5 On the **Code Generation > Interface** pane, set the **Target Library** in the **Deep learning** group to MKL - DNN.
- 6 Clear the **MAT-file logging** parameter.
- 7 Click **OK** to save and close the Configuration Parameters dialog box.
- 8 On the **C++ Code** app, click **Generate Code**.

See Also

googlenet (Deep Learning Toolbox) | Image From File (Computer Vision Toolbox) | MATLAB Function

Related Examples

- “Generate Code for Blocks from Deep Neural Networks Library” on page 41-27
- “Generate Generic C/C++ for Sequence-to-Sequence Deep Learning Simulink Models” on page 41-10
- “Code Generation for Deep Learning Simulink Model That Performs Lane and Vehicle Detection” on page 41-2

Generate Code for Blocks from Deep Neural Networks Library

You can generate optimized code for Simulink models containing a variety of trained deep learning networks. You can implement the deep learning functionality in Simulink by using MATLAB Function blocks or by using blocks from the **Deep Neural Networks** library. For information on the blocks in the **Deep Neural Networks** library, see [Blocks](#).

You can configure the code generator to take advantage of the Intel Math Kernel library for Deep Neural Networks (MKL-DNN). You can also generate generic C or C++ code that does not depend on third-party libraries. The generated code implements the deep convolutional neural network (CNN) by using the architecture, the layers, and parameters that you specify in network object.

Classify Images by Using GoogLeNet

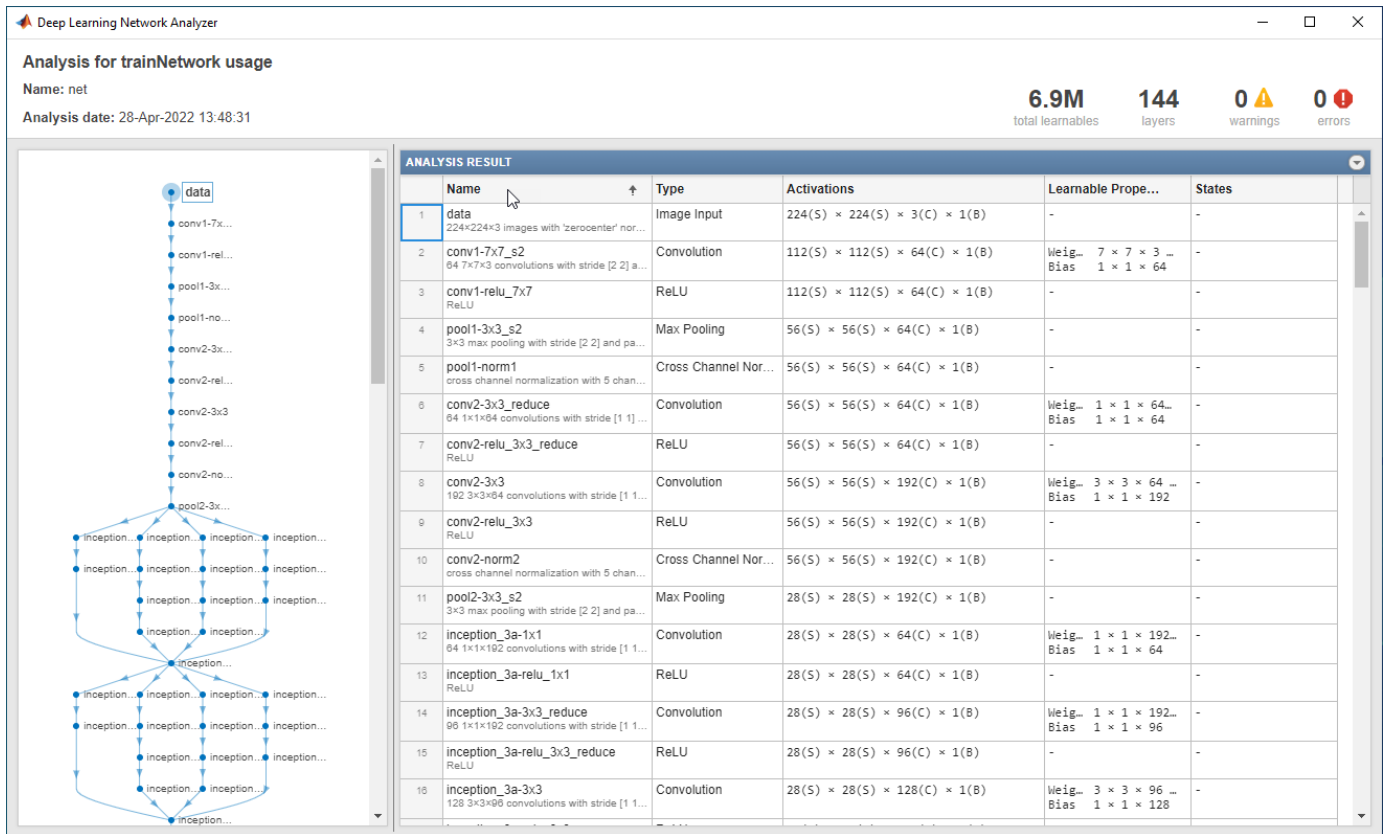
GoogLeNet has been trained on over a million images and can classify images into 1000 object categories (such as a keyboard, coffee mug, pencil, and animals). The network takes an image as input, and then outputs a label for the object in the image with the probabilities for each of the object categories. This example shows how to perform simulation and generate C++ code for the pretrained `googlenet` deep convolutional neural network and classify an image. The pretrained networks are available as support packages from the Deep Learning Toolbox.

- 1 Load the pretrained GoogLeNet network. You can choose to load a different pretrained network for image classification. This function requires the Deep Learning Toolbox Model for GoogLeNet Network support package. If this support package is not installed, then the function provides a download link.

```
net = googlenet;
```

- 2 The object `net` contains the `DAGNetwork` object. Use the function to display an interactive visualization of the network architecture, to detect errors and issues in the network, and to display detailed information about the network layers. The layer information includes the sizes of layer activations and learnable parameters, the total number of learnable parameters, and the sizes of state parameters of recurrent layers.

```
analyzeNetwork(net);
```



- 3 The image that you want to classify must have the same size as the input size of the network. For GoogLeNet, the size of the `imageInputLayer` is 224-by-224-by-3. The `Classes` property of the output `classificationLayer` contains the names of the classes learned by the network. View 10 random class names out of the total of 1000.

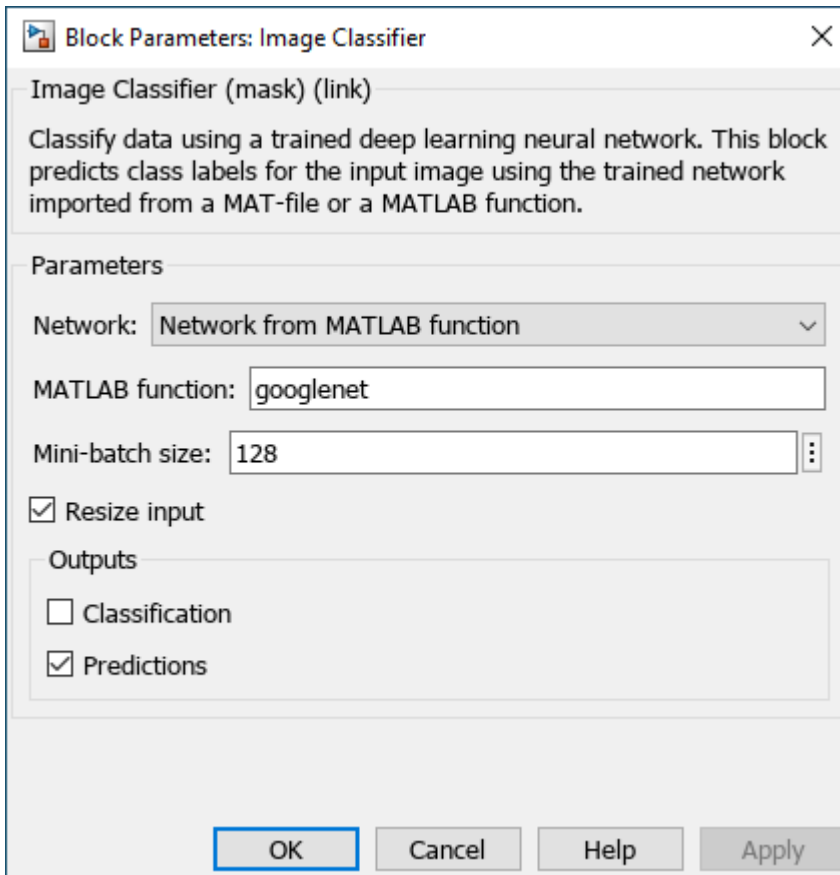
```
classNames = net.Layers(end).Classes;
numClasses = numel(classNames);
disp(classNames(randperm(numClasses, 10)))
```

```
speedboat
window screen
isopod
wooden spoon
lipstick
drake
hyena
dumbbell
strawberry
custard apple
```

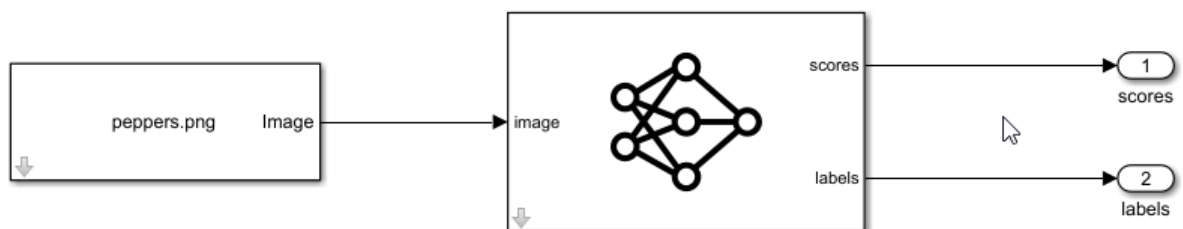
Create GoogLeNet Model

- 1 Create a Simulink model.
- 2 Add an Image From File block from the **Computer Vision Toolbox** library and verify that the `File name` parameter is set to `peppers.png`.
- 3 Insert an Image Classifier block from the **Deep Learning Toolbox > Deep Neural Networks** library. Set these block configuration parameters:

- Set the **Network** parameter to Network from MATLAB function.
- Set the **MATLAB function** parameter to googlenet.
- Select the **Resize input** parameter.
- Select the **Predictions** parameter.
- Clear the **Classifications** parameter.



- 4 Add two Output blocks and connect them to the Image Classifier block. Connect the Image Classifier block to the Image From File block.



- 5 Save the model as googlenetModel.slx.

Simulate Model

- 1 Open the Configuration Parameters dialog box. Select the **Solver** pane. Set the **Type** parameter to **Fixed-step**. This setting maintains a constant (fixed) step size, which is required for code generation.
- 2 Select the **Simulation Target** pane. Set the **Language** to C++.
- 3 On the **Simulation Target** pane, verify that the **Target Library** parameter in the **Deep learning** group is set to MKL-DNN.
- 4 Click **OK** to save and close the Configuration Parameters dialog box.
- 5 To build and simulate the model, use this command:

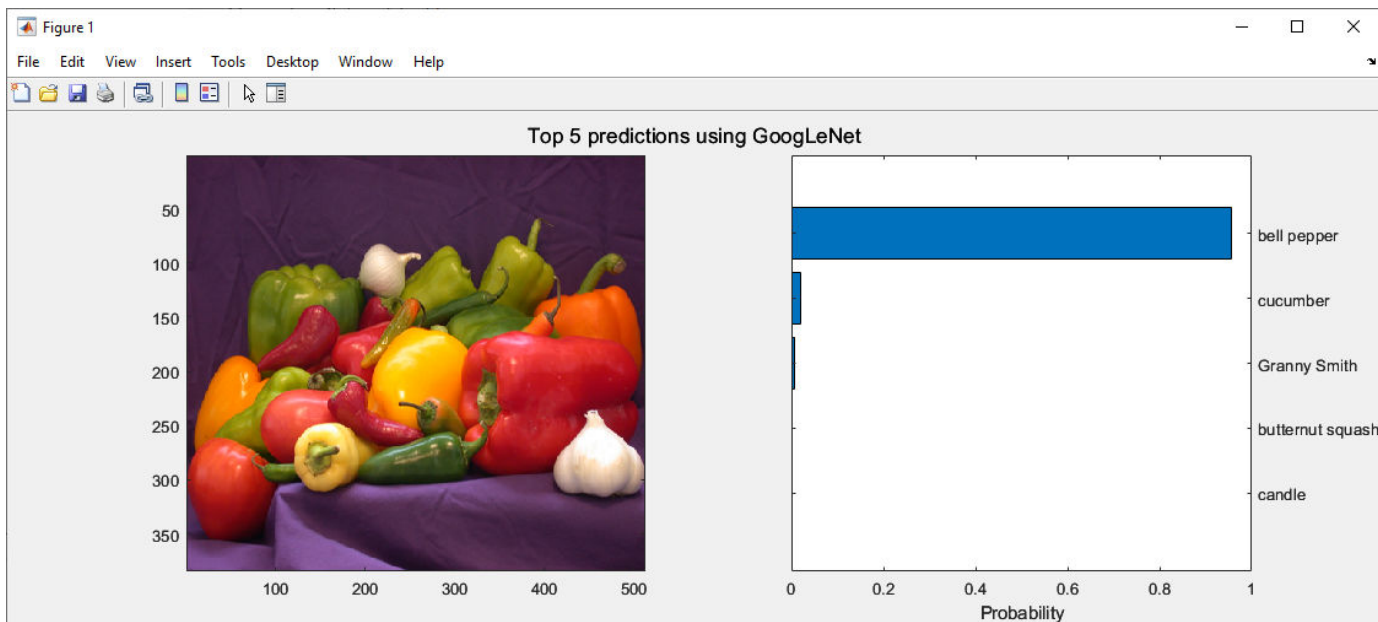
```
out = sim('googlenetModel');
```

- 6 Display the top five predicted labels and their associated probabilities as a histogram. Because the network classifies images into so many object categories, and many categories are similar, it is common to consider the top-five accuracy when evaluating networks. The network classifies the image as a bell pepper with a high probability.

```
im = imread('peppers.png');
predict_scores = out.yout{1}.Values.Data(:, :, 1);
[scores, indx] = sort(predict_scores, 'descend');
topScores = scores(1:5);
classNamesTop = classNames(indx(1:5))
```

```
h = figure;
h.Position(3) = 2*h.Position(3);
ax1 = subplot(1,2,1);
ax2 = subplot(1,2,2);
```

```
image(ax1, im);
barh(ax2, topScores(1,5:-1:1,1))
xlabel(ax2, 'Probability')
yticklabels(ax2, classNamesTop(5:-1:1))
ax2.YAxisLocation = 'right';
sgtitle('Top 5 predictions using GoogLeNet')
```



Generate Code

- 1 Open the **Simulink Coder** or **Embedded Coder** app. The **C Code** tab opens.
- 2 In the Configuration Parameters dialog box, if you have a Simulink Coder license, set the **System target file** to `grt.tlc`. If you have Embedded Coder, set the **System target file** parameter to `ert.tlc`.
- 3 Set the **Language** to C++.
- 4 Select **Generate code only**.
- 5 On the **Code Generation > Interface** pane, set the **Target Library** in the **Deep learning** group to MKL-DNN.
- 6 On the **Interface** pane, clear the **MAT-file logging** parameter.
- 7 If the **System target file** parameter is set to `ert.tlc`, on the **Interface** pane, select the **variable-size signals** parameter.
- 8 Click **OK** to save and close the Configuration Parameters dialog box.
- 9 On the **C++ Code** app, click **Build**.

See Also

googlenet (Deep Learning Toolbox) | Image From File (Computer Vision Toolbox) | MATLAB Function

Related Examples

- “Generate Code for Deep Learning Networks Using MATLAB Function Block” on page 41-22
- “Generate Generic C/C++ for Sequence-to-Sequence Deep Learning Simulink Models” on page 41-10
- “Code Generation for Deep Learning Simulink Model That Performs Lane and Vehicle Detection” on page 41-2

Workflow for Deep Learning C/C++ Code Generation for Simulink Models

With Simulink Coder, you can generate code from deep learning neural networks you design and implement using the Deep Learning Toolbox. To use Simulink Coder to generate code for deep learning networks, you must also install the MATLAB Coder Interface for Deep Learning. Deep learning uses neural networks to learn useful representations of features directly from data. You can obtain a pretrained neural network or train one yourself using the Deep Learning Toolbox. For more information, see “Train Deep Learning Network to Classify New Images” (Deep Learning Toolbox) and “Pretrained Deep Neural Networks” (Deep Learning Toolbox).

Implement the trained neural network in Simulink by using blocks from the Deep Neural Networks library or by using a MATLAB Function block. When implementing the trained neural network with a MATLAB Function block, use the `coder.loadDeepLearningNetwork` to load a trained deep learning network and use the object functions of the network object to obtain the desired responses. The network must be supported for code generation. See “Networks and Layers Supported for Code Generation”.

You can generate C++ code that targets an embedded platform that uses an Intel processor or an ARM processor. The generated code calls the Intel Math Kernel Library for Deep Neural Networks (MKL-DNN) or the ARM Compute Library to apply high performance. The hardware and software requirements depend on the target platform. To apply these libraries, in the Model Configuration Parameters dialog box, set these parameter settings.

Pane	Parameter	Setting
Simulation Target	Language	C++
Simulation Target	Target library	MKL - DNN
Code Generation	Language	C++
Interface	Target library	<ul style="list-style-type: none"> • MKL - DNN • ARM Compute

For an example that uses the MKL-DNN library, see “Code Generation for Deep Learning Simulink Model That Performs Lane and Vehicle Detection” on page 41-2.

You can also generate generic C or C++ code that does not depend on third-party libraries. To generate generic C or C++ code, set these parameter settings.

Pane	Parameter	Setting
Simulation Target	Language	C or C++
Code Generation	Language	C or C++
Interface	Target library	None

For an example, see “Generate Generic C/C++ for Sequence-to-Sequence Deep Learning Simulink Models” on page 41-10.

Deep learning models typically work on large sets of labeled data. Performing inference on these models is computationally intensive, consuming a significant amount of memory. You can use pruning in combination with network quantization to reduce the inference time and memory footprint of the deep learning network, making it easier to deploy to low-power microcontrollers and FPGAs. For more information, see:

- “Quantization of Deep Neural Networks” (Deep Learning Toolbox)
- “Generate int8 Code for Deep Learning Networks”
- “Prune Image Classification Network Using Taylor Scores” (Deep Learning Toolbox)

To perform quantization, you must install the Deep Learning Toolbox Model Quantization Library support package.

See Also

Related Examples

- “Generate Code for Deep Learning Networks Using MATLAB Function Block” on page 41-22
- “Generate Code for Blocks from Deep Neural Networks Library” on page 41-27
- “Deep Learning in MATLAB” (Deep Learning Toolbox)
- “Workflow for Deep Learning Code Generation with MATLAB Coder”

Verification

Simulation and Code Comparison in Simulink Coder

Simulation and Code Comparison

In this section...

“Configure Signal Data for Logging” on page 42-2

“Log Simulation Data” on page 42-3

“Run Executable and Load Data” on page 42-4

“Visualize and Compare Results” on page 42-5

“Compare States for Simulation and Code Generation” on page 42-7

This example shows how to verify the answers computed by code generated from the `slexAircraftExample` model. It shows how to capture and compare two sets of output data. Simulating the model produces one set of output data. Executing the generated code produces a second set of output data.

Note To obtain a valid comparison between model output and the generated code, use the same **Solver selection** and **Step size** for the simulation run and the build process.

Configure Signal Data for Logging

Configure the model for logging and recording signal data.

- 1 Make sure that `slexAircraftExample` is closed. Clear the base workspace to eliminate the results of previous simulation runs. In the Command Window, type:

```
clear
```

The clear operation clears variables created during previous simulations and all workspace variables, some of which are standard variables that the `slexAircraftExample` model requires.

- 2 To open the model, in the Command Window, enter:

```
openExample('slexAircraftExample')
```

- 3 In the model window, choose **File > Save As**, navigate to the working folder, and save a copy of the `slexAircraftExample` model as `myAircraftExample`.
- 4 Set up your model to log signal data for signals: `Stick`, `alpha`, `rad`, and `q`, `rad/sec`. For each signal:
 - a Right-click the signal. From the context menu, select **Properties**.
 - b In the Signal Properties dialog box, select **Log signal data**.
 - c In the **Logging name** section, from the drop-down list, select **Custom**.
 - d In the text field, enter the logging name for the corresponding signal.

Signal Name	Logging Name
Stick	Stick_input
alpha, rad	Alpha

Signal Name	Logging Name
q, rad/sec	Pitch_rate

- e Click **Apply** and **OK**.

For more information, see “Save Signal Data Using Signal Logging”.

- 5 In the Configuration Parameters dialog box:
 - a Set **Type** to Fixed-step.
 - b Set **Format** to Structure with time.
 - c Clear the **States** check box.
 - d Select the **Signal logging** check box.
 - e Select the **Record logged workspace data in Simulation Data Inspector** check box.
- 6 Save the model.

Proceed to “Log Simulation Data” on page 42-3.

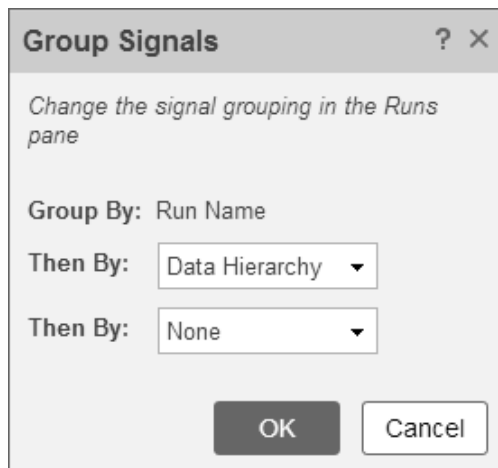
Log Simulation Data

Run the simulation, log the signal data, and view the data in the Simulation Data Inspector.

- 1 Run the model. When the simulation is done, on the Simulink Editor toolbar, the **Simulation Data Inspector** button is highlighted to indicate that new simulation output is available in the Simulation Data Inspector.

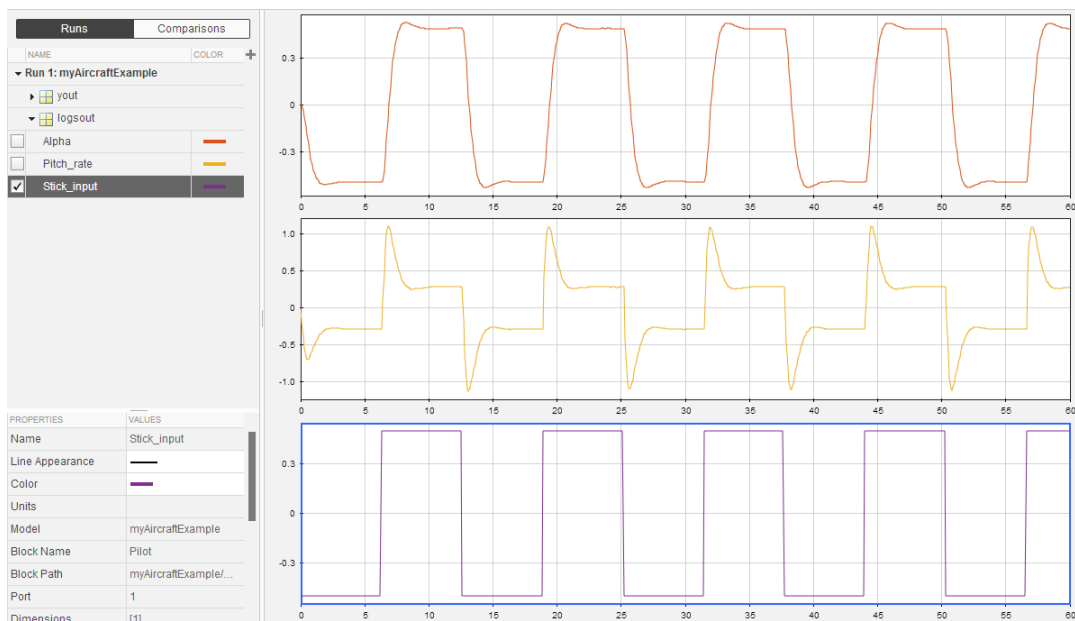


- 2 Click the **Simulation Data Inspector** button to open the Simulation Data Inspector.
- 3 Group the signals:
 - a On the **Visualize** tab, click **Group Signals**.
 - b In the Group Signals dialog box, select Data Hierarchy from the **Then By** list.



- c Click **OK**.

- 4 Click the **Logout** expander to view the logged signals.
- 5 Click the **Format** tab.
- 6 Click the **Subplots** button and select 3x1 to show three subplots.
- 7 For each signal:
 - a Click the top subplot. A blue border indicates the plot selection.
 - b Select the check box next to the Alpha signal name. The signal data appears in the subplot.
 - c Plot the Pitch_rate signal in the middle subplot.
 - d Plot the Stick_input signal in the bottom subplot.



Proceed to “Run Executable and Load Data” on page 42-4.

Run Executable and Load Data

You must rebuild and run the `myAircraftExample` executable to obtain a valid data file because you have modified the model.

- 1 In the Configuration Parameters dialog box, set the **MAT-file variable name modifier** parameter to `rt_`. `rt_` is prefixed to each variable that you selected for logging in the first part of this example.
- 2 Click **Apply** and **OK**.
- 3 Save the model.
- 4 To generate code, on the **C Code** tab, click **Build** button.
- 5 When the build is finished, run the standalone program from the Command Window.

```
!myAircraftExample
```

The executing program writes the following messages to the Command Window.

```
** starting the model **  
** created myAircraftExample.mat **
```

- 6 Load the data file `myAircraftExample.mat`.

```
load myAircraftExample
```

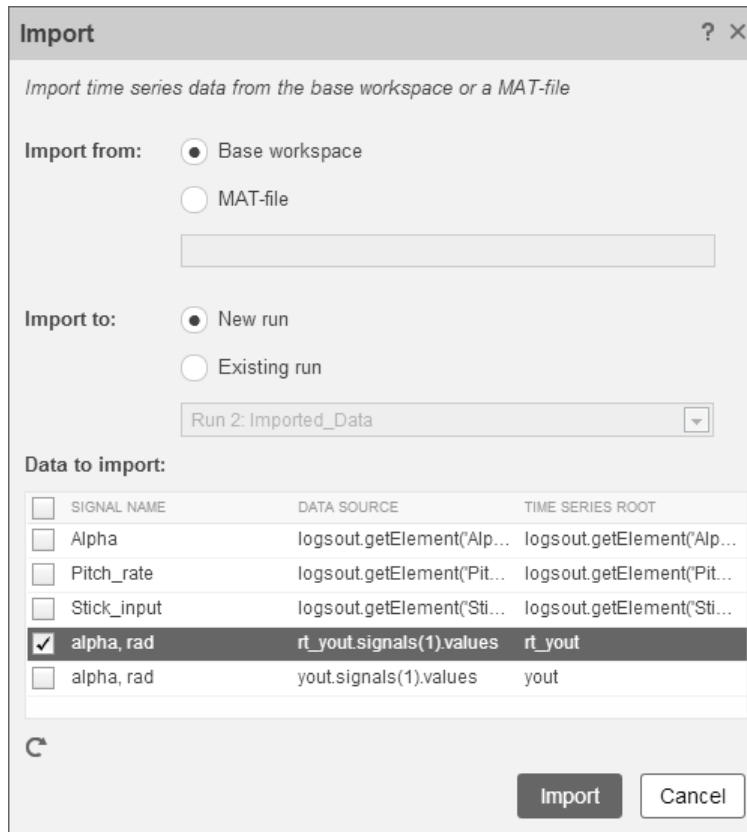
Tip For UNIX platforms, run the executable program in the Command Window with the syntax `!./executable_name`. If preferred, run the executable program from an OS shell with the syntax `./executable_name`. For more information, see “Run External Commands, Scripts, and Programs”.

Proceed to “Visualize and Compare Results” on page 42-5.

Visualize and Compare Results

When you follow the example sequence that began in “Configure Signal Data for Logging” on page 42-2, you obtain data from a Simulink run of the model and from a run of the program generated from the model.

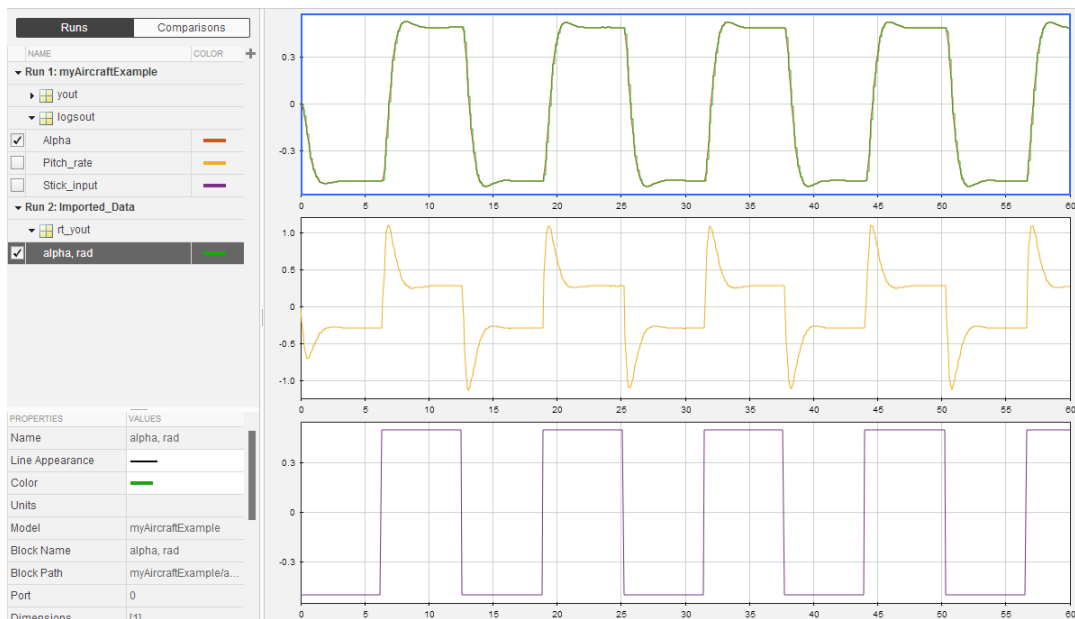
- 1 To view the execution output for `alpha , rad`, import the data into the Simulation Data Inspector.
 - a On the Simulation Data Inspector **Visualize** tab, click the **Import** button to open the Import dialog.
 - b Specify **Import from** as **Base workspace**.
 - c Specify **Import to** as **New run**.
 - d To the left of **Signal Name**, click the check mark to clear the check boxes.
 - e Select the check box for the `alpha , rad` data where the **Time Series Root** is `rt_yout`.
 - f Click **Import**.



The selected data is now under **Run 2: Imported_Data**.

- 2 View a plot of the executed data.
 - a Click the `rt_yout` expander.
 - b Click the top subplot and select the check box next to the `alpha, rad` signal name. The signal data appears in the top subplot.

The `alpha, rad` signal from Run 1 and Run 2 overlap in the subplot because the signals are equivalent.



It is possible to see a very small difference between simulation and code generation results. A slight difference can be caused by many factors, including:

- Different compiler optimizations
- Statement ordering
- Run-time libraries

For example, a function call such as `sin(2.0)` can return a slightly different value depending on which C library you use. Such variations can also cause differences between your results and these results.

Compare States for Simulation and Code Generation

The order in which Simulink logs states during simulation is different than the order in which Simulink Coder logs states during code generation. If you want to compare states between simulation and code generation, sort the states by block name.

For example, by default, Simulink exports state data to the MATLAB variable, `xout`. Simulink Coder exports state data to the variable `rt_xout`. To sort the state data for these variables, enter the following commands in the MATLAB Command Window:

```
[~,idx1]=sort({xout.signals.blockName});
xout_sorted=[xout.signals(idx1).values];
[~,idx2]=sort({rt_xout.signals.blockName});
rt_xout_sorted=[rt_xout.signals(idx2).values];
```

You can confirm that the logging order is the same between code generation and simulation by entering the following command in the MATLAB Command Window:

```
isequal(xout_sorted, rt_xout_sorted)
```

See Also

Related Examples

- “Log Program Execution Results” on page 33-2

Numerical Consistency between Model and Generated Code

Numerical Consistency of Model and Generated Code Simulation Results

In this section...

“Numerical Consistency” on page 43-2

“Numerical Consistency in Complex Systems” on page 43-2

“Reasons for Block-Level Numerical Differences” on page 43-4

Numerical Consistency

In the Model-Based Design workflow, you use MathWorks products to generate code for numerical applications that employ fixed-point and floating-point arithmetic.

- To develop models, you use MATLAB, Simulink, and Stateflow.
- To generate source code, you use Simulink Coder and Embedded Coder.
- To test numerical equivalence between your model and generated code, you compare model and generated code simulation results. For example, normal mode simulation results compared with software-in-the-loop (SIL) simulation results.

The results from the model and generated code simulations are numerically consistent if:

- In fixed-point applications, the results agree in a bit-wise comparison.
- In floating-point applications, the results agree with an error tolerance that you specify.

Use the Simulation Data Inspector to compare results. To determine whether discrepancies exist or are significant, you can specify absolute and relative tolerance values:

- For fixed-point applications, you can specify an absolute tolerance of zero.
- For floating-point applications, you can specify tolerance with respect to a reference value or signal. The choice of reference depends on your application. Consider these examples:
 - An algorithm that solves a linear algebraic equation by iterative, feed-forward error calculations. You can specify tolerance with respect to `eps`.
 - A Proportional-Integral-Derivative (PID) controller for a closed-loop system. For transient behavior, you can specify tolerance with criteria from a standard. For steady-state behavior, you can specify tolerance with reference to the PID controller characteristics.

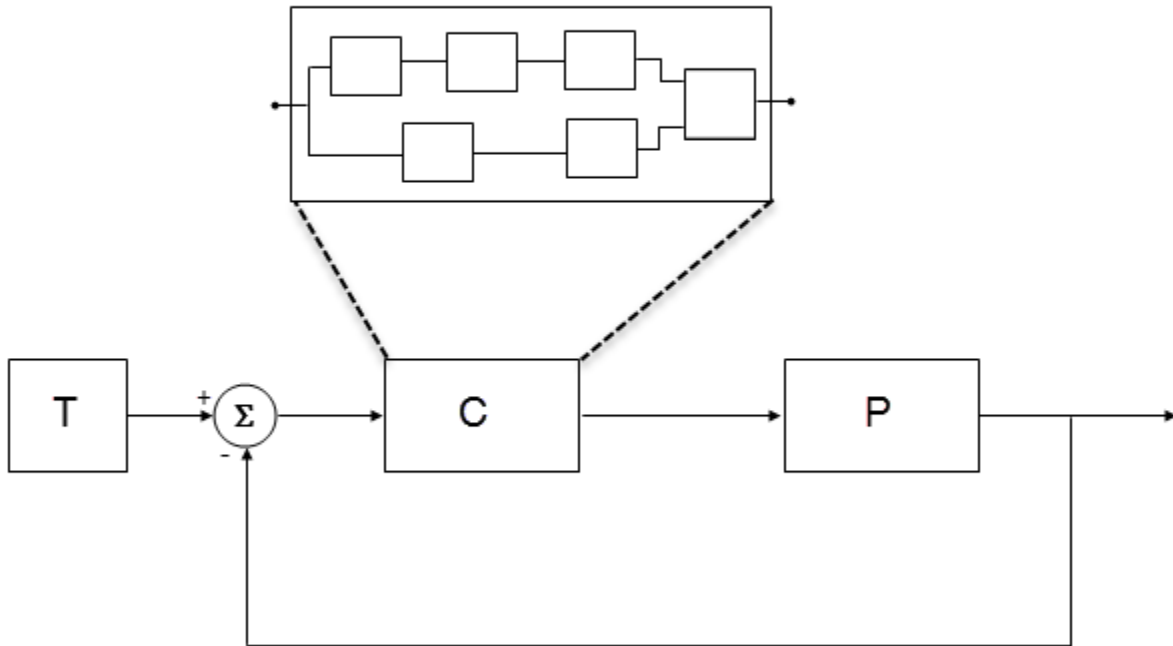
Programmatically, you can specify absolute and relative tolerance values through the `absTol` and `relTol` properties of the `Simulink.sdi.Signal` object.

Numerical Consistency in Complex Systems

For complex systems, numerical differences between model and generated code simulations can be a result of block-level differences propagating through the system. If you observe numerical differences at the system level:

- 1 Identify blocks for which block-level numerical differences exceed the error tolerance.
- 2 Investigate each identified block.

Consider the following plant-controller model.



- T produces reference or test signals.
- C is the controller component. The controller output is the plant input. C can be a Model block that comprises multiple Model blocks.
- P is the plant component. The plant output is subtracted from the reference signal to produce the controller input.

To test numerical equivalence between the model controller and the generated code version:

- 1 Run the model in normal mode, and, using the Simulation Data Inspector, record the output of C.
- 2 Specify SIL mode for C. Rerun the simulation, recording the output of C.
- 3 Using the Simulation Data Inspector, compare normal and SIL mode outputs with reference to your specified error tolerance.

If the Simulation Data Inspector comparison indicates a match, the model and generated code results are numerically consistent.

If the normal and SIL mode outputs do not match:

- 1 Within C, enable signal logging for block outputs.
- 2 Run the model in normal mode.
- 3 Rerun the simulation with C in SIL mode.

- 4 Using the Simulation Data Inspector, compare the logged output signals with reference to your specified error tolerance. See “Compare Simulation Data”.
- 5 Identify blocks for which normal and SIL mode output differences exceed the error tolerance.
- 6 Analyze each identified block and look for the cause. For example, the generated code might use a different math library than MATLAB.

Note If the comparison of a large number of signals is required, you can automate the workflow with Simulink Test. See “SIL Verification for a Subsystem” (Simulink Test).

Reasons for Block-Level Numerical Differences

In fixed-point and floating-point application development, there are factors that can affect numerical agreement between block-level results from model and generated code simulations.

Some factors can affect both fixed-point and floating-point applications. For example, the use of:

- Code generation optimization.
- Custom code.
- Code replacement library entries whose results differ from MATLAB results.
- Code replacement libraries that implement different algorithms.

Other factors affect only floating-point applications. For example:

- Numerical soundness of algorithm.
- Algorithm sensitivity to input.
- Closed loop and open loop behavior.

References

- [1] *IEEE Standard on Transitions, Pulses, and Related Waveforms*, IEEE Standard 181, 2003, pp. 15-17.

See Also

eps

Related Examples

- “Analyze Simulation Results”
- “SIL Verification for a Subsystem” (Simulink Test)
- “Differences Between Generated Code and MATLAB Code”
- “Types of In-the-Loop Testing in the V-Model” (Embedded Coder)
- MATLAB Function
- “SIL and PIL Limitations” (Embedded Coder)

Customization

Build Process Integration in Simulink Coder

- “Control Build Process Compiling and Linking” on page 44-2
- “Control Library Location and Naming During Build” on page 44-3
- “Customize Post-Code-Generation Build Processing” on page 44-7
- “Configure Generated Code with TLC” on page 44-12
- “Use makecfg to Customize Generated Makefiles for S-Functions” on page 44-15
- “Use rtwmakecfg.m API to Customize Generated Makefiles” on page 44-17
- “Register Custom Toolchain and Build Executable” on page 44-21
- “Customize Build Process with STF_make_rtw_hook File” on page 44-25
- “Customize Build Process with sl_customization.m” on page 44-29
- “Replace STF_rtw_info_hook Supplied Target Data” on page 44-33

Control Build Process Compiling and Linking

After generating code for a model, the build process determines whether to compile and link an executable program. Various factors guide this determination:

- **Generate code only** option

When you select this option, the code generator produces code for the model, including a makefile.

- **Generate makefile** option

When you clear this option, the code generator does not produce a makefile for the model. You must specify post code generation processing, including compilation and linking, as a user-defined command, as explained in “Customize Post-Code-Generation Build Processing” on page 44-7.

- **Makefile-only target**

The Microsoft Visual C++ Project Makefile versions of the `grt` and Embedded Coder target configurations generate a Visual C++ project makefile (`model.mak`). To build an executable, you must open `model.mak` in the Visual C++ IDE and compile and link the model code.

- **HOST template makefile variable**

The template makefile variable `HOST` identifies the type of system upon which your executable is intended to run. The variable can be set to one of three possible values: `PC`, `UNIX`, or `ANY`.

By default, `HOST` is set to `UNIX` in template makefiles designed for use with The Open Group UNIX platforms (such as `grt_unix.tmf`), and to `PC` in the template makefiles designed for use with development systems for the PC (such as `grt_vcx64.tmf`).

If the Simulink software is running on the same type of system as the system specified by the `HOST` variable, then the executable is built. Otherwise,

- If `HOST = ANY`, an executable is still built. This option is useful when you want to cross-compile a program for a system other than the one that the Simulink software is running on.
- Otherwise, processing stops after generating the model code and the makefile; the following message is displayed on the MATLAB command line.

```
### Make will not be invoked - template makefile is for a different host
```


Control Library Location and Naming During Build

When you generate precompiled, non-precompiled, and model reference libraries, you can control the library location and library name by using configuration parameters. These parameters control values in generated makefiles during model builds:

- For build processes that use the toolchain approach, control the generated library location by using the `TargetPreCompLibLocation` configuration parameter.
- For build processes that use the template makefile approach, control the generated library location by using the `TargetPreCompLibLocation` configuration parameter and control the generated library name by using the `TargetLibSuffix` configuration parameter.

Library Control Parameters

Use the library control parameters to:

- Specify the location of precompiled libraries, such as blockset libraries or the Simulink Coder block library. Typically, a target has cross-compiled versions of these libraries and places them in a target-specific folder.
- Control the suffix applied to library file names (for example, `_target.a` or `_target.lib`).

Targets can set the parameters inside the system target file (STF) select callback. For example:

```
function mytarget_select_callback_handler(varargin)
    hDig=varargin{1};
    hSrc=varargin{2};
    slConfigUISetVal(hDig, hSrc, 'TargetPreCompLibLocation',...
        'c:\mytarget\precomplibs');
    slConfigUISetVal(hDig, hSrc, 'TargetLibSuffix',...
        '_target.library');
```

The TMF has corresponding expansion tokens:

```
|>EXPAND_LIBRARY_LOCATION<|
|>EXPAND_LIBRARY_SUFFIX<|
```

Alternatively, you can use a call to the `set_param` function. For example:

```
set_param(model,'TargetPreCompLibLocation',...
    'c:\mytarget\precomplibs');
```

Note If your model contains referenced models, you can use the make option `USE_MDLREF_LIBPATHS` to control whether libraries used by the referenced models are copied to the build folder of the parent model. For more information, see “Location of Model Reference Libraries” on page 44-5.

Identify Library File Type for Toolchain Approach

The toolchain approach for model builds does not use the value of the `TargetLibSuffix` configuration parameter to select the library file name *suffix* and *extension*.

With the toolchain approach, the final binary name is composed of the *modelname*, the *compilername*, and the *extension* provided by the build tool description in the toolchain definition:

```
model_compilername.extension
```

You can identify the static library file name extension from the build tool description in the toolchain definition. To get this information for the default toolchain, use this procedure:

- 1 Get the default toolchain name. For example, enter:

```
tc_name = coder.make.getDefaultToolchain()
```

- 2 Get the default toolchain handle. For example, enter:

```
tc = coder.make.getToolchainInfoFromRegistry(tc_name)
```

- 3 Get the handle to the toolchain object. For example, enter:

```
tool_archiver = tc.getBuildTool('Archiver');
```

- 4 Get the extension. For example, enter:

```
ext_archiver = tool_archiver.getFileExtension('Static Library');
```

Note If you do not set the `TargetLibSuffix` parameter, template makefile and toolchain approaches produce the same static library file name *extension*.

Specify the Location of Precompiled Libraries

Use the `TargetPreCompLibLocation` configuration parameter to:

- Override the precompiled library location specified in the `rtwmakecfg.m` file (see “Use `rtwmakecfg.m` API to Customize Generated Makefiles” on page 44-17 for details)
- Precompile and distribute target-specific versions of product libraries (for example, the DSP System Toolbox product)

For a precompiled library, such as a blockset library or the Simulink Coder block library, the location specified in `rtwmakecfg.m` is typically a location specific to the blockset or the Simulink Coder product. The code generator expects that the library exists in this location and links against the library during builds.

For some applications, such as custom targets, it is preferable to locate the precompiled libraries in a target-specific or other alternate location rather than in the location specified in `rtwmakecfg.m`. For a custom target, the code generator expects that the target-specific cross-compiler creates the library, and you place the library in the target-specific location. Compile and place libraries supported by the target in the target-specific location, so they can be used during the build process.

You can set up the `TargetPreCompLibLocation` parameter in its select callback. The path that you specify for the parameter must be a fully qualified, absolute path to the library location. Relative paths are not supported. For example:

```
slConfigUISetVal(hDlg, hSrc, 'TargetPreCompLibLocation',...
'c:\mytarget\precomplibs');
```

Alternatively, you set the parameter with a call to the `set_param` function. For example:

```
set_param(model, 'TargetPreCompLibLocation',...
'c:\mytarget\precomplibs');
```

During makefile generation, the build process replaces the tokens with the location from the `rtwmakecfg.m` file. For example, if the library name in the `rtwmakecfg.m` file is `'rtwlib'`, the template makefile build approach expands the token from:

```
LIBS += |>EXPAND_LIBRARY_LOCATION<|\>EXPAND_LIBRARY_NAME<|\
_target.library
```

to:

```
LIBS += c:\mytarget\precomplibs\rtwlib_target.library
```

By default, `TargetPreCompLibLocation` is an empty character vector. The build process uses the location in `rtwmakecfg.m` for the token replacement.

Location of Model Reference Libraries

The build process assigns a path such as `../slprj/grt/sub/sub_rtwlib.lib` to `MODELREF_LINK_LIBS`. The build process uses the path to gain access to the library file at link time.

Control the Suffix Applied to Library File Names

With the template makefile approach for model builds, use the `TargetLibSuffix` configuration parameter to control the suffix applied to library names (for example, `_target.lib` or `_target.a`). The specified suffix scheme must include a period (.). You can apply `TargetLibSuffix` to the following libraries:

- Libraries on which a target depends, as specified in the `rtwmakecfg.m` API. You can use `TargetLibSuffix` to change the suffix of both precompiled and non-precompiled libraries configured from the `rtwmakecfg` API. For details, see “Use `rtwmakecfg.m` API to Customize Generated Makefiles” on page 44-17.

In this case, a target can set the parameter in its select callback. For example:

```
slConfigUISetVal(hDlg, hSrc, 'TargetLibSuffix',...
_target.library');
```

Alternatively, you can use a call to the `set_param` function. For example:

```
set_param(model, 'TargetLibSuffix', '_target.library');
```

During the TMF-to-makefile conversion, the build process replaces the token `|>EXPAND_LIBRARY_SUFFIX<|` with the specified suffix. For example, if the library name specified in the `rtwmakecfg.m` file is `'rtwlib'`, the TMF expands from:

```
LIBS += |>EXPAND_LIBRARY_LOCATION<|\>EXPAND_LIBRARY_NAME<|\
|>EXPAND_LIBRARY_SUFFIX<|
```

to:

```
LIBS += c:\mytarget\precomplibs\rtwlib_target.library
```

By default, `TargetLibSuffix` is set to an empty character vector. In this case, the build process replaces the token `|>EXPAND_LIBRARY_SUFFIX<|` with an empty character vector.

- Model libraries created with model reference. For these cases, associated makefile variables do not require the `|>EXPAND_LIBRARY_SUFFIX<|` token. Instead, the build process includes `TargetLibSuffix` implicitly. For example, for a top model named `topmodel` with referenced models named `refmodel1` and `refmodel2`, the TMF of the top model is expanded from:

```
MODELLIB          = |>MODELLIB<|
MODELREF_LINK_LIBS = |>MODELREF_LINK_LIBS<|
```

to:

```
MODELLIB          = topmodellib_target.library
MODELREF_LINK_LIBS = \
refmodell_rtwlib_target.library refmodel2_rtwlib_target.library
```

By default, the `TargetLibSuffix` parameter is an empty character vector. In this case, the build process chooses a default suffix for these three tokens using a file extension of `.lib` on Windows hosts and `.a` on UNIX hosts. For model reference libraries, the default suffix also includes `_rtwlib`. For example, on a Windows host, the expanded makefile values are:

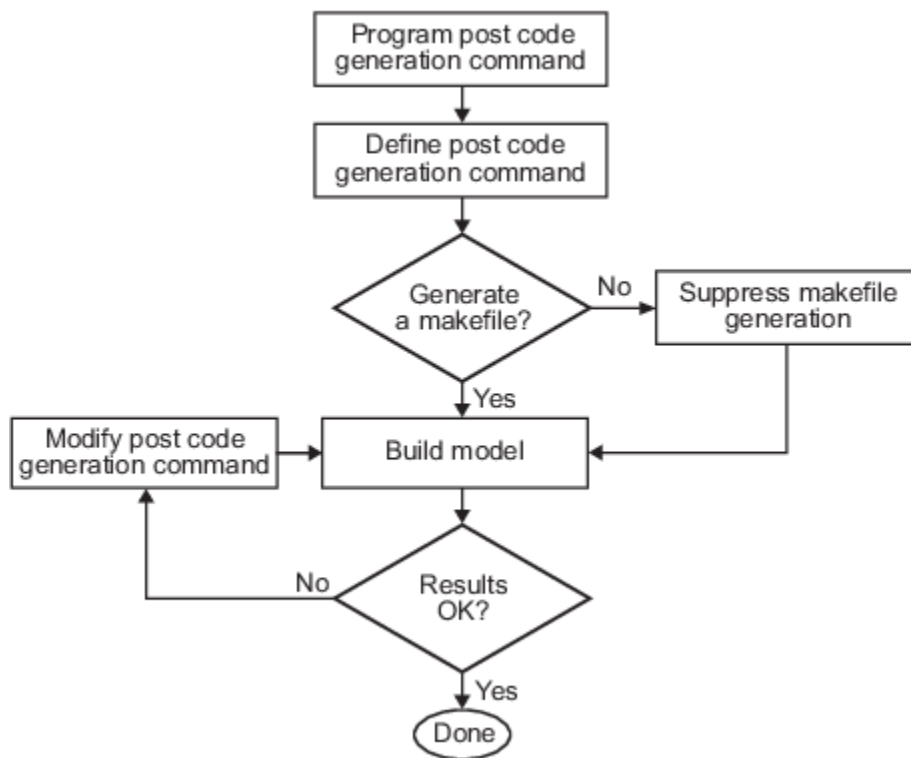
```
MODELLIB          = topmodellib.lib
MODELREF_LINK_LIBS = refmodell_rtwlib.lib refmodel2_rtwlib.lib
```

Customize Post-Code-Generation Build Processing

The code generator provides a set of tools, including a build information object, you can use to customize build processing that occurs after code generation. You can use such customizations for target development or the integration of third-party tools into your application development environment.

Workflow for Setting Up Customizations

The following figure and the steps that follow show the general workflow for setting up post-code-generation customizations.



- 1 Program the post code generation command on page 44-8.
- 2 Define the post code generation command on page 44-9.
- 3 Suppress makefile generation on page 44-11, if applicable.
- 4 Build the model.
- 5 Modify the command and rebuild the model until the build results are acceptable.

Build Information Object

At the start of a model build, the build process logs the following build option and dependency information to a temporary build information object:

- Compiler options

- Preprocessor identifier definitions
- Linker options
- Source files and paths
- Include files and paths
- Precompiled external libraries

You can retrieve information from and add information to this object by using an extensive set of functions. For a list of available functions and detailed function descriptions, see “Code Compilation Customization” “Code Compilation Customization” (Embedded Coder). “Program a Post Code Generation Command” on page 44-8 explains how to use the functions to control post code generation build processing.

Program a Post Code Generation Command

For certain applications, you could want to control aspects of the build process after the code generation. For example, you can use this approach if you develop your own target, or you want to apply an analysis tool to the generated code before continuing with the build process. You can apply this level of control to the build process by programming and then defining a post code generation command.

A post code generation command is a MATLAB language file that typically calls functions that get data from or add data to the build information object of the model. You can program the command as a script or function.

If you program the command as a: Then the:

Script	Script can gain access to the model name and the build information directly
Function	Function can pass the model name and the build information as arguments

If your post code generation command calls user-defined functions, make sure that the functions are on the MATLAB path. If the build process cannot find a function you use in your command, the build process errors out.

You can then call a combination of build information functions, as listed in “Code Compilation Customization” “Code Compilation Customization” (Embedded Coder), to customize the post code generation build processing of the model.

The following example shows a fragment of a post code generation command that gets the file names and paths of the source and include files generated for a model for analysis.

```
function analyzegeneratecode(buildInfo)
% Get the names and paths of source and include files
% generated for the model and then analyze them.

% buildInfo - build information for my model.

% Define cell array to hold data.
MyBuildInfo={};

% Get source file information.
MyBuildInfo.srcfiles=getSourceFiles(buildInfo, true, true);
MyBuildInfo.srcpaths=getSourcePaths(buildInfo, true);
```

```
% Get include (header) file information.
MyBuildInfo.incfiles=getIncludeFiles(buildInfo, true, true);
MyBuildInfo.incpaths=getIncludePaths(buildInfo, true);

% Analyze generated code.
.
.
.
```

Define a Post Code Generation Command

After you program a post code generation command, inform the build process that the command exists and to add it to the build processing of the model. Define the command with the `PostCodeGenCommand` model configuration parameter. When you define a post code generation command, the build process evaluates the command after generating and writing the generated code to disk and before generating a makefile.

As the following syntax lines show, the arguments that you specify when setting the configuration parameter varies depending on whether you program the command as a script, function, or set of functions.

Note When defining the command as a function, you can specify an arbitrary number of input arguments. To pass the name and build information of the model to the function, specify identifiers `modelName` and `buildInfo` as arguments.

Script

```
set_param(model, 'PostCodeGenCommand',...
'pcgScriptName');
```

Function

```
set_param(model, 'PostCodeGenCommand',...
'pcgFunctionName(modelName)');
```

Multiple Functions

```
pcgFunctions=...
'pcgFunction1Name(modelName);...
'pcgFunction2Name(buildInfo)';
set_param(model, 'PostCodeGenCommand',...
'pcgFunctions');
```

The following call to `set_param` defines `PostCodGenCommand` to evaluate the function `analyzegencode`.

```
set_param(model, 'PostCodeGenCommand',...
'analyzegencode(buildInfo)');
```

Customize Build Process with PostCodeGenCommand and Relocate Generated Code to an External Environment

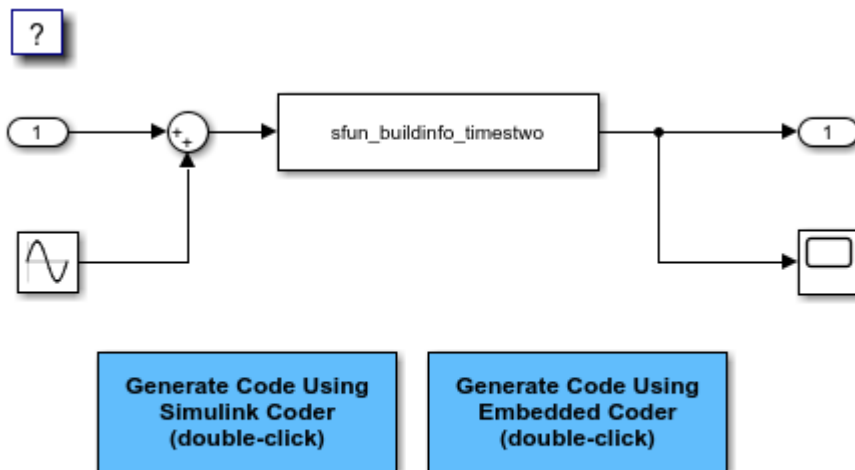
This example shows how to use the build information programming interface and the model configuration parameter **Post code generation command** (`PostCodeGenCommand`) to customize the build process. For this example, **Post code generation command** is set to `rtwdemo_buildinfo_data`. The build process invokes that function after code generation.

The example also shows how to use build configuration function `rtwmakecfg.m`.

Open Example Model

Open the example model `rtwdemo_buildinfo`.

```
open_system('rtwdemo_buildinfo');
```



Copyright 1994-2020 The MathWorks, Inc.

Generate Code from Model

Generate code. The code generator produces the file `BuildInfo.html`, which documents the build information object.

Examine the Build Process Customizations and Output

Examine the build process customizations and the post code generation query of the build information object. In a Web browser, open the file `BuildInfo.html`. The file provides hyperlinks that you can use to examine the code produced for the model. Alternatively, in the MATLAB Command Window:

- To view the function configured to execute during the post code generation stage of the build process, enter:

```
get_param('rtwdemo_buildinfo', 'PostCodeGenCommand');
```

- To study the programming interface for the build information object, enter:

```
rtwdemo_buildinfo_data(buildInfo);
```

- To study how the example uses the `rtwmakecfg` function, enter:

```
edit rtwmakecfg.m;
```

- To view the build information object data that the build process saves in the `buildInfo.mat` file, open the file `rtwdemo_<target>_rtw\buildInfo.mat`.

- To view the contents of the post code generation function `rtwdemo_buildinfo_data.m`, enter:

```
edit rtwdemo_buildinfo_data.m;
```

At the end of the file, the function calls `packNGo` to package the source and object files that the build information object identifies for relocation.

Related Topics

- “Code Compilation Customization”
- “Customize Post-Code-Generation Build Processing” on page 44-7
- “Use `rtwmakecfg.m` API to Customize Generated Makefiles” on page 44-17
- “Relocate or Share Generated Code” on page 31-56

Suppress Makefile Generation

The code generator lets you suppress makefile generation during the build process. For example, you can use this support when you integrate tools into the build process that do not use makefiles.

To instruct the code generator not to produce a makefile, do one of the following:

- Clear the **Generate makefile** option on the **Code Generation** pane of the Configuration Parameters dialog box.
- Set the value of the configuration parameter `GenerateMakefile` to `off`.

When you suppress makefile generation,

- You cannot explicitly specify a make command or template makefile.
- Specify your own instructions for a post code generation processing, including compilation and linking, in a post code generation command as explained in “Program a Post Code Generation Command” on page 44-8 and “Define a Post Code Generation Command” on page 44-9.

See Also

`RTW.BuildInfo`

Configure Generated Code with TLC

You can use the Target Language Compiler (TLC) to fine-tune your generated code. TLC supports extended code generation variables and options in addition to parameters available on the **Code Generation** pane on the Configuration Parameters dialog box. There are two ways to set TLC variables and options, as described in this section.

Note Do not customize TLC files in the folder *matlabroot/rtw/c/tlc* even though the capability exists to do so. It is possible that such TLC customizations are not applied during the code generation process. Such customizations can lead to unpredictable results.

Assigning Target Language Compiler Variables

The `%assign` statement lets you assign a value to a TLC variable, as in:

```
%assign MaxStackSize = 4096
```

This assignment is also known as creating a *parameter name/parameter value pair*.

For a description of the `%assign` statement, see “Target Language Compiler Directives”. Write your `%assign` statements in the Configure RTW code generation settings section of the system target file.

The following table lists the code generation variables you can set with the `%assign` statement.

Target Language Compiler Optional Variables

Variable	Description
MaxStackSize= <i>N</i>	<p>When the Enable local block outputs check box is selected, the total allocation size of local variables that are declared by block outputs in the model cannot exceed MaxStackSize (in bytes). MaxStackSize can be a positive integer. If the total size of local block output variables exceeds this maximum, the remaining block output variables are allocated in global, rather than local, memory. The default value for MaxStackSize is <code>rtInf</code>, that is, unlimited stack size.</p> <p>Note: Local variables in the generated code from sources other than local block outputs, such as from a Stateflow diagram or MATLAB Function block, and stack usage from sources such as function calls and context switching are not included in the MaxStackSize calculation. For overall executable stack usage metrics, do a target-specific measurement by using run-time (empirical) analysis or static (code path) analysis with object code.</p>
MaxStackVariableSize= <i>N</i>	<p>When the Enable local block outputs check box is selected, this selection limits the size of a local block output variable declared in the code to <i>N</i> bytes, where $N > 0$. A variable whose size exceeds MaxStackVariableSize is allocated in global, rather than local, memory. The default is 4096.</p>
RemoveFixptWordSizeChecks= <i>value</i>	<p>For a model that uses fixed-point data types, control the generation of preprocessor directives that perform fixed-point word size checks. If <i>value</i> is:</p> <ul style="list-style-type: none"> • 0, generate the preprocessor directives. The generated code contains fixed-point word size checks. • 1, suppress the generation of the preprocessor directives. The generated code does not contain fixed-point word size checks. <p>You can also suppress the generation of the preprocessor directives by running this command:</p> <pre>set_param(gcs, 'TLCOptions', ... '-aRemoveFixptWordSizeChecks=1')</pre> <p>If you generate code that does not contain the fixed-point word size checks, you can use processor-in-the-loop (PIL) simulations that check Hardware Implementation settings with reference to the target hardware. For more information, see “Verification of Code Generation Assumptions” (Embedded Coder).</p>

Variable	Description
WarnNonSaturatedBlocks= <i>value</i>	<p>Flag to control display of overflow warnings for blocks that have saturation capability, but have it turned off (not selected) in their dialog box. Options include:</p> <ul style="list-style-type: none">• 0 — Warning is not displayed.• 1 — Displays one warning for the model during code generation• 2 — Displays one warning that contains a list of offending blocks

Set Target Language Compiler Options

You can specify TLC command-line options for code generation using the model parameter `TLCOptions` in a `set_param` function call. For information about these options, see “Specify TLC for Code Generation” on page 31-28 and “Configure TLC”.

See Also

Related Examples

- “Target Language Compiler Directives”

Use makecfg to Customize Generated Makefiles for S-Functions

With the toolchain or template makefile approach for building code, you can customize generated makefiles for S-functions. Through the customization, you can specify additional items for the S-function build process:

- Source files and folders
- Include files and folders
- Library names
- Preprocessor macro definitions
- Compiler flags
- Link objects

To customize the generated makefile:

- 1 In the S-function MEX file folder that contains your S-function components, create a `makecfg.m` or `specificSFunction_makecfg.m` file. If you create a `makecfg.m` file, it is active for any S-function in the S-function MEX file folder that the model uses. A `specificSFunction_makecfg.m` file is active only if the model uses the corresponding S-function.
- 2 In the file that you create, use `RTW.BuildInfo` functions to specify additional items for the S-function build process. For example, you can use:
 - `addCompileFlags` to specify compiler options.
 - `addDefines` to specify preprocessor macro definitions.
- 3 Save the created file.

After code generation, in the S-function MEX file folder, the code generator searches for `makecfg.m` and `specificSFunction_makecfg.m` files. If the files are present, the code generator uses these files to customize the generated makefile, `model.mk`.

For example, consider a folder that contains `signalConvert.mexa64` (S-function binary file) and `signalConvert.tlc` (inlined S-function implementation) after the TLC phase of the build process. The S-function requires an additional source code file, `filterV1.c`, which is located in `anotherFolder`. You can create a file, `signalConvert_makecfg.m`, that uses `RTW.BuildInfo` functions to specify `filterV1.c` for the build process.

```
function signalConvert_makecfg(objBuildInfo)

absolutePath = fullfile('${START_DIR}', 'anotherFolder');
addSourceFiles(objBuildInfo, 'filterV1.c', absolutePath);
```

See Also

Related Examples

- “Build Process Workflow for Real-Time Systems” on page 31-39
- “Approaches for Building Code Generated from Simulink Models” on page 31-11
- “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6

- “Use rtwmakecfg.m API to Customize Generated Makefiles” on page 44-17

Use rtwmakecfg.m API to Customize Generated Makefiles

Note To customize generated makefiles for S-functions, MathWorks recommends the use of the `makecfg` approach instead of the `rtwmakecfg` function. For more information, see “Use `makecfg` to Customize Generated Makefiles for S-Functions” on page 44-15.

Both the toolchain approach and the template makefile approach for builds let you add the following items to generated makefiles:

- Source folders
- Include folders
- Library names
- Module objects

About the `rtwmakecfg` Function

Using an `rtwmakecfg` function, you add this information to the makefile during the build operation for S-functions. The `rtwmakecfg` function is useful when specifying added sources and libraries to build a model that contains one or more of your S-function blocks.

To add information pertaining to an S-function to the makefile:

- 1 Create the MATLAB language `rtwmakecfg` function in the `rtwmakecfg.m` file. The code generator associates this file with your S-function based on its folder location. “Create the `rtwmakecfg` Function” on page 44-17 describes the requirements for the `rtwmakecfg` function and the data it returns.
- 2 If you are using the template makefile approach, modify the TMF of your target such that it supports macro expansion for the information that the `rtwmakecfg` function returns. “Modify the Template Makefile for `rtwmakecfg`” on page 44-19 describes the required modifications. If you are using the toolchain approach, the information that the `rtwmakecfg` function returns is used by the generated makefile; no further configuration is required.

After the TLC phase of the build process, when generating a makefile, the code generator searches for an `rtwmakecfg.m` file in the folder that contains the S-function MEX file. If it finds the file, the build process calls the `rtwmakecfg` function.

Create the `rtwmakecfg` Function

Create the `rtwmakecfg.m` file containing the `rtwmakecfg` function in the same folder as your S-function component (a MEX-file with a platform-dependent extension, such as `.mexext` on Microsoft Windows systems). The function must return a structured array that contains these fields.

Field	Description
<code>makeInfo.includePath</code>	A cell array that specifies additional include folder names, organized as a row vector. The build process expands the folder names into include instructions in the generated makefile.

Field	Description
<code>makeInfo.sourcePath</code>	A cell array that specifies additional source folder names, organized as a row vector. Include the folder names of files entered into the S-function modules field on the S-Function Block Parameters dialog box or into the <code>SFunctionModules</code> parameter of the block if they are not in the same folder as the S-function. The build process expands the folder names into make rules in the generated makefile.
<code>makeInfo.sources</code>	A cell array that specifies additional source file names (C or C++), organized as a row vector. Do not include the name of the S-function or files entered into the S-function modules field on the S-Function Block Parameters dialog box or into the <code>SFunctionModules</code> parameter of the block. The build process expands the file names into make variables that contain the source files. Specify only file names (with extension). Specify path information with the <code>sourcePath</code> field.
<code>makeInfo.linkLibsObjs</code>	A cell array that specifies additional, fully qualified paths to object or library files against which the generated code links. The build process does not compile the specified objects and libraries. However, it includes them when linking the final executable. This inclusion can be useful for incorporating libraries that you do not want the build process to recompile or for which the source files are not available. You can also use this element to integrate source files from languages other than C and C++. This integration is possible if you first create a C compatible object file or library outside of the build process.
<code>makeInfo.precompile</code>	A Boolean flag that indicates whether the libraries specified in the <code>rtwmakecfg.m</code> file exist in a specified location (<code>precompile==1</code>) or if you must create the libraries in the build folder during the build process (<code>precompile==0</code>).
<code>makeInfo.library</code>	A structure array that specifies additional run-time libraries and module objects, organized as a row vector. The build process expands the information into make rules in the generated makefile. For a list of the library fields, see the next table.

The `makeInfo.library` field consists of the following elements.

Element	Description
<code>makeInfo.library(n).Name</code>	A character array that specifies the name of the library (without an extension).
<code>makeInfo.library(n).Location</code>	A character array that specifies the folder in which the library is located when precompiled. For more information, see the description of <code>makeInfo.precompile</code> in the preceding table. A target can use the <code>TargetPreCompLibLocation</code> parameter to override this value. See “Specify the Location of Precompiled Libraries” on page 44-4.
<code>makeInfo.library(n).Modules</code>	A cell array that specifies the C or C++ source file base names (without an extension) that comprise the library. Do not include the file extension. The makefile appends the object extension.

Note The `makeInfo.library` field must fully specify each library and how to build it. The modules list in the `makeInfo.library(n).Modules` element cannot be empty. To specify a link-only library, use the `makeInfo.linkLibsObjs` field instead.

Example:

```
disp(['Running rtwmakecfg from folder: ',pwd]);
makeInfo.includePath = { fullfile(pwd, 'somedir2') };
makeInfo.sourcePath = {fullfile(pwd, 'somedir2'), fullfile(pwd, 'somedir3')};
makeInfo.sources = { 'src1.c', 'src2.cpp'};
makeInfo.linkLibsObjs = { fullfile(pwd, 'somedir3', 'src3.object'),...
    fullfile(pwd, 'somedir4', 'mylib.library')};

makeInfo.precompile = 1;
makeInfo.library(1).Name = 'myprecompiledlib';
makeInfo.library(1).Location = fullfile(pwd, 'somedir2', 'lib');
makeInfo.library(1).Modules = {'srcfile1' 'srcfile2' 'srcfile3' };
```

Note If a path that you specify in the `rtwmakecfg.m` API contains spaces, the build process does not convert the path to its nonspace equivalent. If the build environments you intend to support do not support spaces in paths, refer to “Build Process Support for File and Folder Names” on page 24-57.

Modify the Template Makefile for rtwmakecfg

To expand the information that an `rtwmakecfg` function generates, modify the following sections in the TMF of your target:

- Include Path
- C Flags and/or Additional Libraries
- Rules

It is possible that these TMF code examples do not apply to your make utility. For additional examples, see the GRT or ERT TMFs located in *matlabroot/toolbox/coder/compile/tmf*.

Add Folder Names to the Makefile Include Path

The following TMF code example adds folder names to the include path in the generated makefile:

```
ADD_INCLUDES = \
|>START_EXPAND_INCLUDES<|   -I|>EXPAND_DIR_NAME<| \
|>END_EXPAND_INCLUDES<|
```

Also, the `ADD_INCLUDES` macro must be added to the `INCLUDES` line.

```
INCLUDES = -I. -I.. $(ADD_INCLUDES) $(USER_INCLUDES)
```

Add Library Names to the Makefile

The following TMF code example adds library names to the generated makefile.

```
LIBS =
|>START_PRECOMP_LIBRARIES<|
LIBS += |>EXPAND_LIBRARY_NAME<|.a |>END_PRECOMP_LIBRARIES<|
|>START_EXPAND_LIBRARIES<|
LIBS += |>EXPAND_LIBRARY_NAME<|.a |>END_EXPAND_LIBRARIES<|
```

For more information, see “Control Library Location and Naming During Build” on page 44-3.

Add Rules to the Makefile

The TMF code example adds rules to the generated makefile.

```
|>START_EXPAND_RULES<|
$(BLD)/%.o: |>EXPAND_DIR_NAME<|/%.c $(SRC)/$(MAKEFILE) rtw_proj.tmw
@$(BLANK)
@echo ### "|>EXPAND_DIR_NAME<|\$.c"
$(CC) $(CFLAGS) $(APP_CFLAGS) -o $(BLD)$(DIRCHAR)$*.o \
|>EXPAND_DIR_NAME<|$(DIRCHAR)$*.c > $(BLD)$(DIRCHAR)$*.lst
|>END_EXPAND_RULES<|

|>START_EXPAND_LIBRARIES<|MODULES_|>EXPAND_LIBRARY_NAME<| = \
|>START_EXPAND_MODULES<| |>EXPAND_MODULE_NAME<|.o \
|>END_EXPAND_MODULES<|

|>EXPAND_LIBRARY_NAME<|.a : $(MAKEFILE) rtw_proj.tmw
$(MODULES_|>EXPAND_LIBRARY_NAME<|:%.o=$(BLD)/%.o)
@$(BLANK)
@echo ### Creating $@
$(AR) -r $@ $(MODULES_|>EXPAND_LIBRARY_NAME<|:%.o=$(BLD)/%.o)
|>END_EXPAND_LIBRARIES<|

|>START_PRECOMP_LIBRARIES<|MODULES_|>EXPAND_LIBRARY_NAME<| = \
|>START_EXPAND_MODULES<| |>EXPAND_MODULE_NAME<|.o \
|>END_EXPAND_MODULES<|

|>EXPAND_LIBRARY_NAME<|.a : $(MAKEFILE) rtw_proj.tmw
$(MODULES_|>EXPAND_LIBRARY_NAME<|:%.o=$(BLD)/%.o)
@$(BLANK)
@echo ### Creating $@
$(AR) -r $@ $(MODULES_|>EXPAND_LIBRARY_NAME<|:%.o=$(BLD)/%.o)
|>END_PRECOMP_LIBRARIES<|
```

Register Custom Toolchain and Build Executable

This example shows how to register and use a toolchain to build an executable file. This example uses the Intel® compiler. However, the concepts and programming interface apply for other toolchains. Once you register a toolchain, you can configure a model such that the code generator uses the toolchain to build an executable file for the model.

Toolchain

A toolchain is a collection of tools required to compile and link code for a specified platform. A toolchain consists of multiple tools, such as a compiler, linker, and archiver. You can configure the tools in a toolchain with multiple options and group tool specifications into types of configurations.

Create a New Folder and Copy Relevant Files

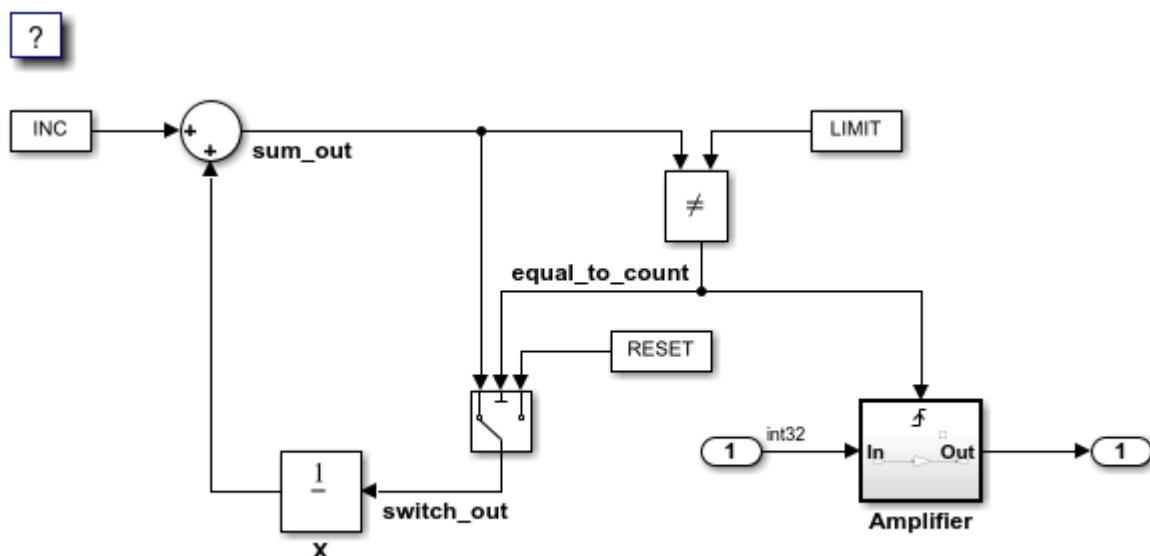
The following code creates a folder in your current working folder (pwd). The new folder contains files for this example. If you do not want to affect the current folder, or cannot generate files in this folder, change your working folder before invoking the command.

```
copyfile(fullfile(matlabroot, 'toolbox', 'rtw', 'rtwdemos', 'toolchain_demo'), 'toolchain_demo', 'f')
cd toolchain_demo
```

Open a Model

Open the `rtwdemo_counter` model. By default, this model is configured to use the Generic Real-Time Target (GRT) configuration and the toolchain approach for building an executable file. The capability demonstrated in this example is available for models configured to use the toolchain approach.

```
model = 'rtwdemo_counter';
open_system(model)
```



Copyright 1994-2021 The MathWorks, Inc.

Choose a Toolchain

Open the **Configuration Parameters** dialog box. Select **Code Generation**. The **Toolchain settings** section contains parameters for configuring a toolchain. From the **Toolchain** drop-down list, select the toolchain installed on your development system for building an executable file from the code generated from your model.

By default, the **Faster Runs** build configuration is selected. Click **Show settings** to see the toolchain flags specified for building the generated code. Choose a build configuration based on your current application development goal.

Close the **Configuration Parameters** dialog box.

Create a ToolchainInfo Object

This example shows how you can register a custom toolchain and add it as a selectable **Toolchain** option.

The first step to registering a custom toolchain is to create a `ToolchainInfo` object that contains information about the toolchain. Methods are available to set toolchain specifications. You can share a `ToolchainInfo` object across installations.

Open the toolchain definition file for an Intel compiler. This file creates a `ToolchainInfo` object that contains information about the Intel toolchain on a 64-bit Windows® platform.

```
edit intel_tc
```

You can display the errors and warnings in the diagnostics messages pane from builds generated by custom compilers. For more information, see “Diagnostic Message Pane”.

Run the toolchain definition file to generate the `ToolchainInfo` object.

```
tc = intel_tc;
```

Save the `ToolchainInfo` object `tc` to a MAT file.

```
save intel_tc tc
```

Register the Custom Toolchain

After you create the `ToolchainInfo` object for the new toolchain, register it. Register a toolchain in `RTW.TargetRegistry`. To register the toolchain, write an `rtwTargetInfo.m` file. Then, add that file to the MATLAB path so the system loads it automatically.

```
edit rtwTargetInfo
```

Reset the `TargetRegistry` to use the new `rtwTargetInfo.m` file.

```
RTW.TargetRegistry.getInstance('reset');
```

Choose the Custom Toolchain

Reopen the **Configuration Parameters** dialog box. Click **Toolchain**. You should see the new toolchain in the list. Select the Intel v18 toolchain.

Programmatically, you can accomplish the same task with the following commands:

```
cs = getActiveConfigSet(model);
set_param(cs, 'Toolchain', tc.Name)
```

Verify your selection.

```
toolchain = get_param(cs, 'Toolchain')
```

```
toolchain =
```

```
    'Intel v18 | nmake makefile (64-bit Windows)'
```

Build the Model Using the Custom Toolchain

You can now build the model with the new custom toolchain.

Note: If you do not have the Intel toolchain installed, you can use the following command to generate the code and makefile only.

```
set_param(cs, 'GenCodeOnly', 'on')
```

Build the model to generate the code and makefile that uses the new toolchain.

```
slbuild(model)
```

```
### Starting build procedure for: rtwdemo_counter
### Successful completion of code generation for: rtwdemo_counter
```

Build Summary

Top model targets built:

Model	Action	Rebuild Reason
rtwdemo_counter	Code generated	Code generation information file does not exist.

```
1 of 1 models built (0 models already up to date)
Build duration: 0h 0m 16.819s
```

Obtain the build directory information.

```
dirInfo = RTW.getBuildDir(model);
```

Examine the generated makefile.

```
edit(fullfile(dirInfo.BuildDirectory, [model '.mk']))
```

If Intel compilers are installed, then when the build process is complete, you can run the executable file.

```
if ispc
    system([model '.exe'])
else
    system(model)
end
```

Restore

You can optionally remove the folder you created earlier.

```
cd ..  
rmdir('toolchain_demo', 's')
```

Reset the TargetRegistry to remove the toolchain that you registered above.

```
RTW.TargetRegistry.getInstance('reset');
```

Close the model.

```
close_system(model, 0)
```

Clear the variables introduced in the workspace.

```
clear INC K LIMIT RESET model tc cs toolchain
```

Customize Build Process with `STF_make_rtw_hook` File

The build process lets you supply optional custom code in hook methods that are executed at specified points in the code generation and make process. You can use hook methods to add target-specific actions to the build process.

`STF_make_rtw_hook` File

You can modify hook methods in a file generically referred to as `STF_make_rtw_hook.m`, where `STF` is the name of a system target file, such as `ert` or `mytarget`. This file implements a function, `STF_make_rtw_hook`, that dispatches to a specific action, depending on the `hookMethod` argument passed in.

The build process calls `STF_make_rtw_hook`, passing in the `hookMethod` argument and other arguments. You implement only those hook methods that your build process requires.

If your model contains reference models, you can implement an `STF_make_rtw_hook.m` for each reference model as required. The build process calls each `STF_make_rtw_hook` for reference models, processing these files recursively (in dependency order).

Conventions for Using the `STF_make_rtw_hook` File

For the build process to call the `STF_make_rtw_hook`, check that the following conditions are met:

- The `STF_make_rtw_hook.m` file is on the MATLAB path.
- The file name is the name of your system target file (`STF`), appended to the text `_make_rtw_hook.m`. For example, if you generate code with a custom system target file `mytarget.tlc`, name your hook file `mytarget_make_rtw_hook.m`, and name the hook function implemented within the file `mytarget_make_rtw_hook`.
- The hook function implemented in the file uses the function prototype described in “`STF_make_rtw_hook.m` Function Prototype and Arguments” on page 44-25.

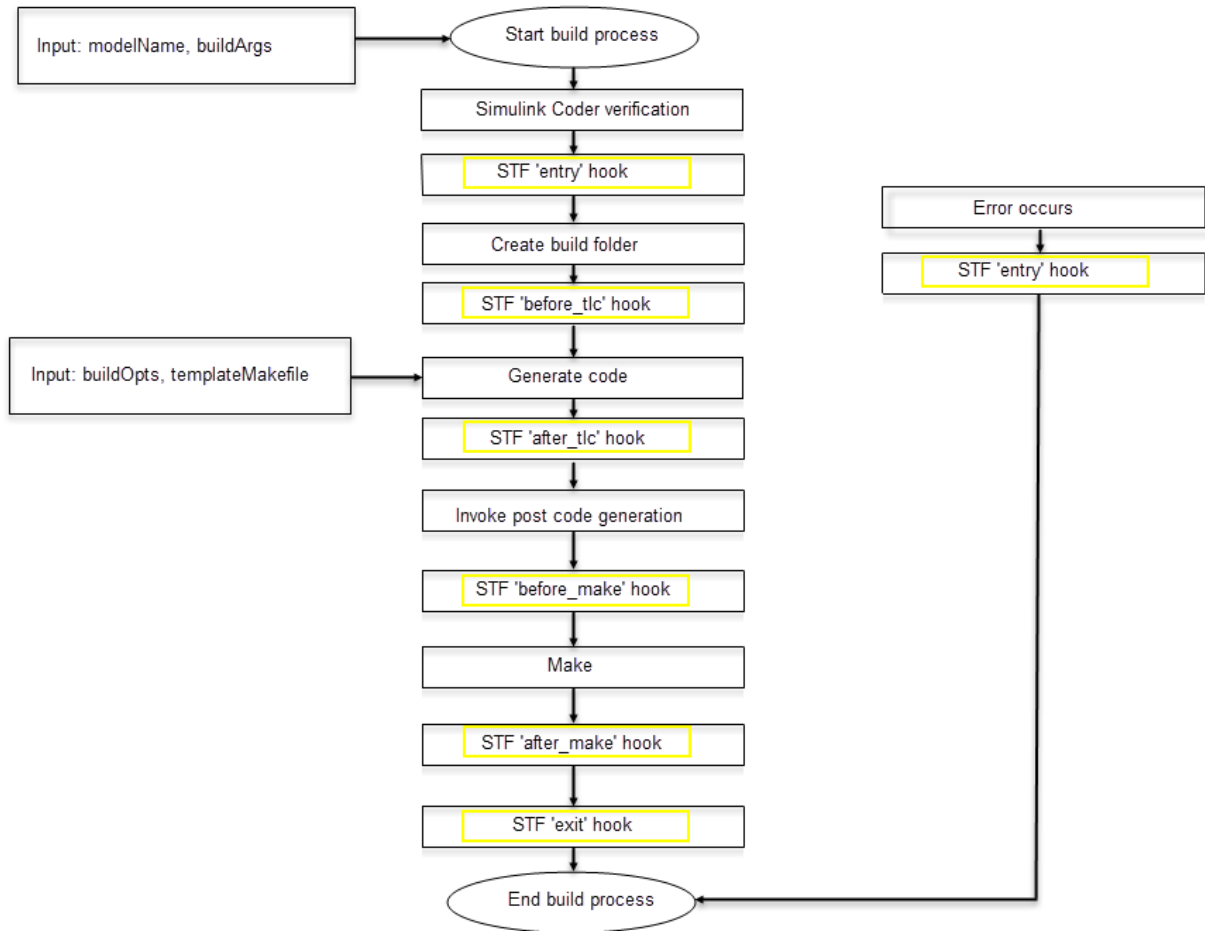
`STF_make_rtw_hook.m` Function Prototype and Arguments

The function prototype for `STF_make_rtw_hook` is:

```
function STF_make_rtw_hook(hookMethod, modelName, rtwRoot, templateMakefile,
    buildOpts, buildArgs, buildInfo)
```

The arguments are defined as:

- `hookMethod`: Character vector specifying the stage of build process from which the `STF_make_rtw_hook` function is called. The following flow chart summarizes the build process, highlighting the hook points. Valid values for `hookMethod` are `'entry'`, `'before_tlc'`, `'after_tlc'`, `'before_make'`, `'after_make'`, `'exit'`, and `'error'`. The `STF_make_rtw_hook` function dispatches to the relevant code with a `switch` statement.



- `modelName`: Character vector specifying the name of the model. Valid at all stages of the build process.
- `rtwRoot`: Reserved.
- `templateMakefile`: Name of template makefile.
- `buildOpts`: A MATLAB structure that contains the Boolean field `codeWasUpToDate`. Valid for the `'before_make'`, `'after_make'`, and `'exit'` stages only.
- `buildArgs`: Character vector containing the argument to `make_rtw`. When you invoke the build process, `buildArgs` is copied from the argument following `"make_rtw"` in the **Configuration Parameters + Code Generation + Make command** field.

For example, the following make arguments from the **Make command** field

```
make_rtw VAR1=0 VAR2=4
```

generate the following:

```
% make -f untitled.mk VAR1=0 VAR2=4
```


The `buildArgs` argument does not apply for toolchain approach builds because these builds do not allow adding make arguments to the `make_rtw` call. On the compiler command line, to provide custom definitions (for example, `VAR1=0 VAR2=4`) that apply for both TMF approach and toolchain approach builds, use the **Configuration Parameters > Code Generation > Custom Code > Defines** field.

- `buildInfo`: The `RTW.BuildInfo` object that contains information for compiling and linking generated code. Available only for the `'after_tlc'`, `'before_make'`, `'after_make'`, and `'exit'` stages. The information in the object at the end of the `'after_tlc'` stage might not be complete. In later stages, the `'before_make'` and `'after_make'` hook methods can also add information to the object. For more information about using the `RTW.BuildInfo` object, see “Code Compilation Customization”.

Applications for STF_make_rtw_hook.m

This section shows how you can use the `STF_make_rtw_hook.m` hook methods.

In general, use the `'entry'` hook to initialize the build process, for example, to change or validate settings before code is generated. One application for the `'entry'` hook is to rerun the auto-configuration script that initially ran at target selection time to compare model parameters before and after the script executes, for validation purposes.

The other hook points, `'before_tlc'`, `'after_tlc'`, `'before_make'`, `'after_make'`, `'exit'`, and `'error'` are useful for interfacing with external tool chains, source control tools, and other environment tools.

For example, you can use the `STF_make_rtw_hook.m` file at a stage after `'entry'` to obtain the path to the build folder. At the `'exit'` stage, you can then locate generated code files within the build folder and check them into your version control system. You can use `'error'` to clean up static or global data used by the hook function when an error occurs during code generation or the build process.

Note The build process temporarily changes the MATLAB working folder to the build folder for stages `'before_make'`, `'after_make'`, `'exit'`, and `'error'`. Your `STF_make_rtw_hook.m` file must not make incorrect assumptions about the location of the build folder. At a point after the `'entry'` stage, you can obtain the path to the build folder. In the following MATLAB code example, the build folder path is returned as a character vector to the variable `buildDirPath`.

```
buildDirPath = rtwprivate('get_makertwsettings',gcs,'BuildDirectory');
```

Note Do not use the `STF_make_rtw_hook.m` file to change the model configuration. For example, do not use a hook method to:

- Switch between model variants.
- Call the `set_param` function.

Changing the model configuration might produce unexpected code generation results.

Control Code Regeneration Using STF_make_rtw_hook.m

When you rebuild a model, by default, the build process performs checks to determine whether changes to the model or relevant settings require regeneration of the top model code. (For details on

the criteria, see “Control Regeneration of Top Model Code” on page 31-49.) If the checks determine that top model code generation is required, the build process fully regenerates and compiles the model code. If the checks indicate that the top model generated code is current with respect to the model, and model settings do not require full regeneration, the build process omits regeneration of the top model code.

Regardless of whether the top model code is regenerated, the build process subsequently calls the build process hooks, including *STF_make_rtw_hook* functions and the post code generation command. The following mechanisms allow you to perform actions related to code regeneration in the *STF_make_rtw_hook* functions:

- To force code regeneration, use the following function call from the 'entry' hook:


```
rtw.targetNeedsCodeGen('set', true);
```
- In hooks from 'before_tlc' through 'exit', the `buildOpts` structure passed to the hook has a Boolean field `codeWasUpToDate`. The field is set to `true` if model code was up to date and code was not regenerated, or `false` if code was not up to date and code was regenerated. You can customize hook actions based on the value of this field. For example:

```
...
case 'before_tlc'
    if buildOpts.codeWasUpToDate
        %Perform hook actions for up to date model
    else
        %Perform hook actions for full code generation
    end
end
...
```

Use *STF_make_rtw_hook.m* for Your Build Procedure

To create a custom *STF_make_rtw_hook* hook file for your build procedure, copy and edit the example *ert_make_rtw_hook.m* file, which is located in the folder *matlabroot/toolbox/coder/embeddedcoder* (open), as follows:

- 1 Copy *ert_make_rtw_hook.m* to a folder in the MATLAB path. Rename it in accordance with the naming conventions described in “Conventions for Using the *STF_make_rtw_hook* File” on page 44-25. For example, to use it with the GRT target *grt.tlc*, rename it to *grt_make_rtw_hook.m*.
- 2 Rename the *ert_make_rtw_hook* function within the file to match the file name.
- 3 Implement the hooks that you require by adding code to case statements within the `switch hookMethod` statement.

Hook Method *after_tlc*

The *after_tlc* hook method is available solely for backwards compatibility.

The format of the generated code during the *after_tlc* stage is not the final format.

See Also

Related Examples

- “Customize Build Process with *sl_customization.m*” on page 44-29

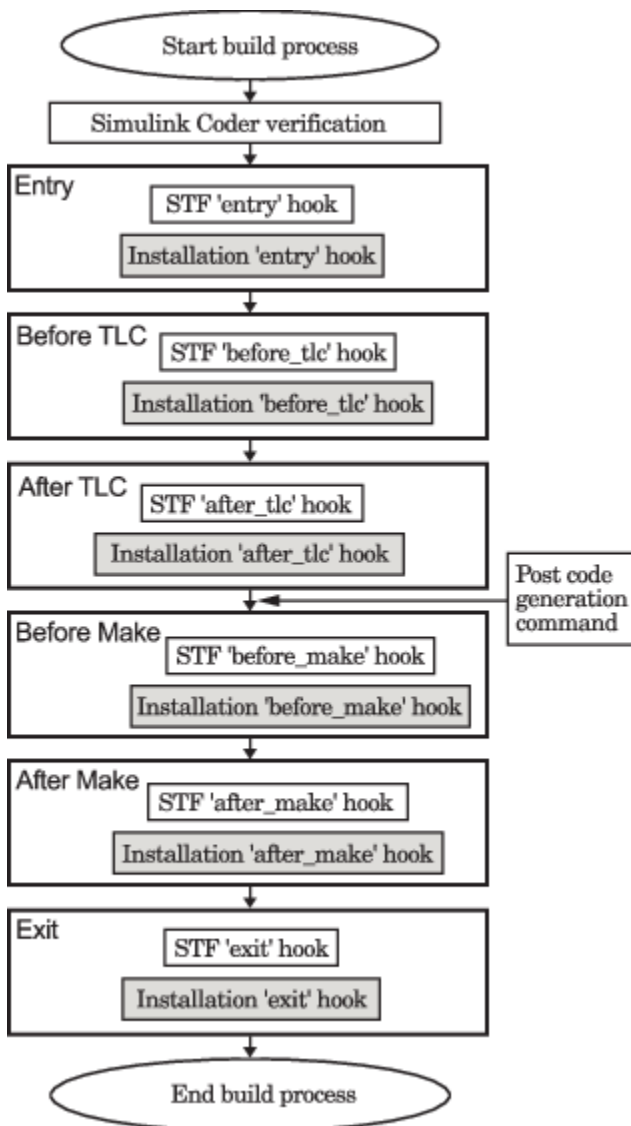
Customize Build Process with `sl_customization.m`

The Simulink customization file `sl_customization.m` is a mechanism that allows you to use MATLAB to customize the build process interface. The Simulink software reads the `sl_customization.m` file, if present on the MATLAB path, when it starts and the customizations specified in the file are applied to the Simulink session. For more information on the `sl_customization.m` customization file, see “Register Customizations with Simulink”.

The `sl_customization.m` File

The `sl_customization.m` file can be used to register installation-specific hook functions to be invoked during the build process. The hook functions that you register through `sl_customization.m` complement System Target File (STF) hooks (described in “Customize Build Process with `STF_make_rtw` hook File” on page 44-25) and post-code generation commands (described in “Customize Post-Code-Generation Build Processing” on page 44-7).

The following figure shows the relationship between installation-level hooks and the other available mechanisms for customizing the build process.



Register Build Process Hook Functions Using `sl_customization.m`

To register installation-level hook functions that are invoked during the build process, you create a MATLAB function called `sl_customization.m` and include it on the MATLAB path of the Simulink installation that you want to customize. The `sl_customization` function accepts one argument: a handle to a customization manager object. For example,

```
function sl_customization(cm)
```

As a starting point for your customizations, the `sl_customization` function must first get the default (factory) customizations, using the following assignment statement:

```
hObj = cm.RTWBuildCustomizer;
```

You then invoke methods to register your customizations. The customization manager object includes the following method for registering build process hook customizations:

- `addUserHook(hObj, hookType, hook)`

Registers the MATLAB hook script or function specified by `hook` for the build process stage represented by `hookType`. The valid values for `hookType` are 'entry', 'before_tlc', 'after_tlc', 'before_make', 'after_make', and 'exit'.

Use this method to register installation-specific hook functions in your instance of the `sl_customization` function.

The Simulink software reads the `sl_customization.m` file when it starts. If you subsequently change the file, you must restart the Simulink session or enter the following command in the Command Window to enable the changes:

```
sl_refresh_customizations
```

Variables Available for `sl_customization.m` Hook Functions

The following variables are available for `sl_customization.m` hook functions to use:

- `modelName` — The name of the Simulink model (valid for all stages)
- `dependencyObject` — An object containing the dependencies of the generated code (valid only for the 'after_make' stage)

A hook script can directly access the valid variables. A hook function can pass the valid variables as arguments to the function. For example:

```
hObj.addUserHook('after_make', 'afterMakeFunction(modelName,dependencyObject)');
```

Example of Build Process Customization with `sl_customization.m`

The `sl_customization.m` file shown in the example, **`sl_customization.m` for Build Process Customizations**, uses the `addUserHook` method to specify installation-specific build process hooks to be invoked at the 'entry' and 'after_tlc' stages of the build process. For the hook function source code, see the **`CustomRTWEntryHook.m`** and **`CustomRTWPostProcessHook.m`** examples.

Example 44.1. `sl_customization.m` for Build Process Customizations

```
function sl_customization(cm)
% Register user customizations

% Get default (factory) customizations
hObj = cm.RTWBuildCustomizer;

% Register build process hooks
hObj.addUserHook('entry', 'CustomRTWEntryHook(modelName)');
hObj.addUserHook('after_tlc', 'CustomRTWPostProcessHook(modelName)');

end
```

Example 44.2. `CustomRTWEntryHook.m`

```
function [str, status] = CustomRTWEntryHook(modelName)
str = sprintf('Custom entry hook for model ''%s.''',modelName);
disp(str)
status =1;
```

Example 44.3. `CustomRTWPostProcessHook.m`

```
function [str, status] = CustomRTWPostProcessHook(modelName)
str = sprintf('Custom post process hook for model ''%s.''',modelName);
```

```
disp(str)
status =1;
```

If you include the above three files on the MATLAB path of the Simulink installation that you want to customize, the coded hook function messages appear in the displayed output for builds. For example, if you open the ERT-based model `UserDefinedDataTypes`, open the **Code Generation** pane of the Configuration Parameters dialog box, and press **Ctrl+B** to initiate a build, the following messages are displayed:

```
>> UserDefinedDataTypes

### Starting build procedure for model: UserDefinedDataTypes
Custom entry hook for model 'UserDefinedDataTypes.'
Custom post process hook for model 'UserDefinedDataTypes.'
### Successful completion of build procedure for model: rtwdemo_udt
>>
```

See Also

Related Examples

- “Use `STF_make_rtw_hook.m` for Your Build Procedure” on page 44-28

Replace STF_rtw_info_hook Supplied Target Data

Prior to MATLAB Release 14, custom targets supplied target-specific information with a hook file (referred to as *STF_rtw_info_hook.m*). The *STF_rtw_info_hook* specified properties such as word sizes for integer data types (for example, char, short, int, and long), and C implementation-specific properties of the custom target.

The *STF_rtw_info_hook* mechanism has been replaced by the **Hardware Implementation** pane of the Configuration Parameters dialog box. Using this dialog box, you can specify properties that were formerly specified in your *STF_rtw_info_hook* file.

For backward compatibility, existing *STF_rtw_info_hook* files are available. However, you should convert your target and models to use of the **Hardware Implementation** pane. See “Configure Production and Test Hardware” on page 11-2.

Run-Time Data Interface Extensions in Simulink Coder

Customize Generated ASAP2 File

In this section...

“About ASAP2 File Customization” on page 45-2
 “ASAP2 File Structure on the MATLAB Path” on page 45-2
 “Customize the Contents of the ASAP2 File” on page 45-2
 “ASAP2 Templates” on page 45-3
 “Customize Computation Method Names” on page 45-5
 “Suppress Computation Methods for FIX_AXIS” on page 45-6

Note TLC based customization will be deprecated in a future release. To customize the ASAP2 file, see “Customize Generated ASAP2 File” (Embedded Coder).

About ASAP2 File Customization

The Embedded Coder product provides a number of Target Language Compiler (TLC) files to enable you to customize the ASAP2 file generated from a Simulink model.

ASAP2 File Structure on the MATLAB Path

The ASAP2 related files are organized within the folders identified below:

- TLC files for generating ASAP2 file

The *matlabroot*/rtw/c/tlc/mw (open) folder contains TLC files that generate ASAP2 files, *asamlib.tlc*, *asap2lib.tlc*, *asap2main.tlc*, and *asap2grouplib.tlc*. These files are included by the setting of model configuration parameter **System target file**.

- ASAP2 target files

The *matlabroot*/toolbox/rtw/targets/asap2/asap2 (open) folder contains the ASAP2 system target file and other control files.

- Customizable TLC files

The *matlabroot*/toolbox/rtw/targets/asap2/asap2/user (open) folder contains files that you can modify to customize the content of your ASAP2 files.

- ASAP2 templates

The *matlabroot*/toolbox/rtw/targets/asap2/asap2/user/templates (open) folder contains templates that define each type of CHARACTERISTIC in the ASAP2 file.

Customize the Contents of the ASAP2 File

The ASAP2 related TLC files enable you to customize the appearance of the ASAP2 file generated from a Simulink model. Most customization is done by modifying or adding to the files contained in the *matlabroot*/toolbox/rtw/targets/asap2/asap2/user (open) folder. This section refers to this folder as the *asap2/user* folder.

The user-customizable files provided are divided into two groups:

- The *static* files define the parts of the ASAP2 file that are related to the environment in which the generated code is used. They describe information specific to the user or project. The static files are not model dependent.
- The *dynamic* files define the parts of the ASAP2 file that are generated based on the structure of the source model.

The procedure for customizing the ASAP2 file is as follows:

- 1 Make a copy of the `asap2/user` folder before making modifications.
- 2 Remove the old `asap2/user` folder from the MATLAB path, or add the new `asap2/user` folder to the MATLAB path above the old folder. The MATLAB session uses the ASAP2 setup file, `asap2setup.tlc`, in the new folder.

`asap2setup.tlc` specifies the folders and files to include in the TLC path during the ASAP2 file generation process. Modify `asap2setup.tlc` to control the folders and folders included in the TLC path.

- 3 Modify the static parts of the ASAP2 file. These include
 - Project and header symbols, which are specified in `asap2setup.tlc`
 - Static sections of the file, such as file header and tail, `A2ML`, `MOD_COMMON`, and so on. These are specified in `asap2userlib.tlc`.
 - Specify the appearance of the dynamic contents of the ASAP2 file by modifying the existing ASAP2 templates or by defining new ASAP2 templates. Sections of the ASAP2 file affected include

`RECORD_LAYOUT`: modify parts of the ASAP2 template files.

`CHARACTERISTIC`: modify parts of the ASAP2 template files. For more information on modifying the appearance of `CHARACTERISTIC` records, see “ASAP2 Templates” on page 45-3.

- `MEASUREMENT`: These are specified in `asap2userlib.tlc`.
- `COMPU_METHOD`: These are specified in `asap2userlib.tlc`.

ASAP2 Templates

The appearance of `CHARACTERISTIC` records in the ASAP2 file is controlled using a different template for each type of `CHARACTERISTIC`. The `asap2/user` folder contains template definition files for scalars, 1-D Lookup Table blocks and 2-D Lookup Table blocks. You can modify these template definition files, or you can create additional templates as required.

The procedure for creating a new ASAP2 template is as follows:

- 1 Create a template definition file. See “Create Template Definition Files” on page 45-4.
- 2 Include the template definition file in the TLC path. The path is specified in the ASAP2 setup file, `asap2setup.tlc`.

Create Template Definition Files

This section describes the components that make up an ASAP2 template definition file. This description is in the form of code examples from `asap2lookup1d.tlc`, the template definition file for the Lookup1D template. This template corresponds to the Lookup1D parameter group.

Note When creating a new template, use the corresponding parameter group name in place of Lookup1D in the code shown.

Template Registration Function

The input argument is the name of the parameter group associated with this template:

```
%<LibASAP2RegisterTemplate("Lookup1D")>
```

RECORD_LAYOUT Name Definition Function

Record layout names (aliases) can be arbitrarily specified for each data type. This function is used by the other components of this file.

```
%function ASAP2UserFcnRecordLayoutAlias_Lookup1D(dtId) void
    %switch dtId
        %case tSS_UINT8
            %return "Lookup1D_UBYTE"
        ...
    %endswitch
%endfunction
```

Function to Write RECORD_LAYOUT Definitions

This function writes RECORD_LAYOUT definitions associated with this template. The function is called by the built-in functions involved in the ASAP2 file generation process. The function name must be defined as shown, with the template name after the underscore:

```
%function ASAP2UserFcnWriteRecordLayout_Lookup1D() Output
    /begin RECORD_LAYOUT
    %<ASAP2UserFcnRecordLayoutAlias_Lookup1D(tSS_UINT8)>
    ...
    /end RECORD_LAYOUT
%endfunction
```

Function to Write the CHARACTERISTIC

This function writes the CHARACTERISTIC associated with this template. The function is called by the built-in functions involved in the ASAP2 file generation process. The function name must be defined as shown, with the template name after the underscore.

The input argument to this function is a pointer to a parameter group record. The example shown is for a Lookup1D parameter group that has two members. The references to the associated x and y data records are obtained from the parameter group record as shown.

This function calls a number of built-in functions to obtain the required information. For example, `LibASAP2GetSymbol` returns the symbol (name) for the specified data record:

```
%function ASAP2UserFcnWriteCharacteristic_Lookup1D(paramGroup)
Output
```

```

%assign xParam = paramGroup.Member[0].Reference
%assign yParam = paramGroup.Member[1].Reference
%assign dtId = LibASAP2GetDataTypeId(xParam)
  /begin CHARACTERISTIC
    /* Name */           %<LibASAP2GetSymbol(xParam)>
    /* Long identifier */ %<LibASAP2GetLongID(xParam)>"
    ...
  /end CHARACTERISTIC
%endfunction

```

Customize Computation Method Names

In generated ASAP2 files, computation methods translate the electronic control unit (ECU) internal representation of measurement and calibration quantities into a physical model oriented representation. Simulink Coder software provides the ability to customize the names of computation methods. You can provide names that are more intuitive, enhancing ASAP2 file readability, or names that meet organizational requirements.

To customize computation method names, use the MATLAB function `getCompuMethodName`, which is defined in `matlabroot/toolbox/rtw/targets/asap2/asap2/user/getCompuMethodName.m`.

The `getCompuMethodName` function constructs a computation method name. The function prototype is

```
cmName = getCompuMethodName(dataTypeName, cmUnits)
```

where *dataTypeName* is the name of the data type associated with the computation method, *cmUnits* is the units as specified in the Unit property of a Simulink.Parameter or Simulink.Signal object (for example, rpm or m/s), and *cmName* returns the constructed computation method name.

The default constructed name returned by the function has the format

```
<localPrefix><datatype>_<cmUnits>
```

where

- `<local_Prefix>` is a local prefix, `CM_`, defined in `matlabroot/toolbox/rtw/targets/asap2/asap2/user/getCompuMethodName.m`.
- `<datatype>` and `<cmUnits>` are the arguments you specified to the `getCompuMethodName` function.

Additionally, in the generated ASAP2 file, the constructed name is prefixed with `<ASAP2CompuMethodName_Prefix>`, a model prefix defined in `matlabroot/toolbox/rtw/targets/asap2/asap2/user/asap2setup.tlc`.

For example, if you call the `getCompuMethodName` function with the *dataTypeName* argument 'int16' and the *cmUnits* argument 'm/s', and generate an ASAP2 file for a model named `myModel`, the computation method name would appear in the generated file as follows:

```

/begin COMPU_METHOD
  /* Name of CompuMethod */ myModel_CM_int16_m_s
  /* Units */ "m/s"
  ...
/end COMPU_METHOD

```

Suppress Computation Methods for FIX_AXIS

Versions 1.51 and later of the ASAP2 specification state that for certain cases of lookup table axis descriptions (integer data type and no doc units), a computation method is not required and the Conversion Method parameter must be set to the value `NO_COMPU_METHOD`. You can control whether or not computation methods are suppressed when not required using the Target Language Compiler (TLC) option `ASAP2GenNoCompuMethod`. This TLC option is disabled by default. If you enable the option, ASAP2 file generation does not generate computation methods for lookup table axis descriptions when not required, and instead generates the value `NO_COMPU_METHOD`. For example:

```
/begin CHARACTERISTIC
/* Name          */
luld_fix_axisTable_data
...
/begin AXIS_DESCR
    ...
    /* Conversion Method */
    NO_COMPU_METHOD
    ...
/end CHARACTERISTIC
```

The `ASAP2GenNoCompuMethod` option is defined in `matlabroot/toolbox/rtw/targets/asap2/asap2/user/asap2setup.tlc`.

See Also

Related Examples

- “How Generated Code Exchanges Data with an Environment” on page 15-35
- “How Generated Code Stores Internal Signal, State, and Parameter Data” on page 15-49

Custom Target Development in Simulink Coder

- “Custom Targets” on page 46-2
- “Sample Custom Targets” on page 46-4
- “Target Development Folders, Files, and Builds” on page 46-6
- “Customize System Target Files” on page 46-19
- “Customize Template Makefiles” on page 46-44
- “Custom Target Optional Features” on page 46-57
- “Support Toolchain Approach with Custom Target” on page 46-59
- “Support Model Referencing” on page 46-61
- “Support Compiler Optimization Level Control” on page 46-65
- “Support C Function Prototype Control” on page 46-67
- “Support C++ Class Interface Control” on page 46-69
- “Support Concurrent Execution of Multiple Tasks” on page 46-71
- “Device Drivers” on page 46-72

Custom Targets

A custom target is a software package that you create to deploy MATLAB and Simulink designs to target hardware. Using the package, you can perform tasks to optimize, prototype, verify, and deploy an application to your hardware. This table lists tasks that you can perform by using target classes and functions.

Task	See
Register new hardware devices and hardware boards for Hardware Implementation settings.	"Register New Hardware Devices" on page 11-13 <code>target.Board</code> <code>target.Processor</code> <code>target.LanguageImplementation</code>
Build a target application using a CMake-based toolchain.	"Create Custom CMake Toolchain Definition" on page 31-17 "Generated Code Compilation"
Deploy, download, and run an application on target hardware.	<code>target.ExecutionTool</code> <code>target.SystemCommandExecutionTool</code>
Tune parameters and monitor target application signals during an external mode simulation.	"Set Up External Mode Connectivity Between Simulink and Target Hardware" on page 32-35 "External Mode Simulation by Using XCP Communication" on page 32-8 "Customize XCP Server Software" on page 32-45
Test numerical equivalence between your MATLAB code or Simulink model and the generated code by running a processor-in-the-loop (PIL) simulation.	"Set Up PIL Connectivity by Using target Package" (Embedded Coder) "Configure and Run PIL Simulation" (Embedded Coder)
Perform code execution profiling through a PIL simulation.	"Set Up PIL Connectivity by Using target Package" (Embedded Coder) "Create Execution-Time Profile for Generated Code" (Embedded Coder)

This table lists additional tasks that you can perform alongside a custom target defined by target classes and functions.

Task	See
Generate or register a main file to schedule the generated code	“Deploy Applications to Target Hardware” (Embedded Coder) “Model Multicore Concurrent Tasking Application” (Embedded Coder) Target operating system (Embedded Coder) “Generate Source and Header Files with a Custom File Processing (CFP) Template” (Embedded Coder) “Support C Function Prototype Control” on page 46-67 “Support C++ Class Interface Control” on page 46-69
Register build hooks.	“Customize Build Process with sl_customization.m” on page 44-29 “Customize Post-Code-Generation Build Processing” on page 44-7
Optimize generated code by using a processor-specific code-replacement library.	“Code Replacement”
Integrate real-time operating systems and device drivers.	“Device Drivers” on page 46-72
Define a custom system target file with hooks and custom settings to add target-specific actions to the build process.	“Target Platform Device Customization” “Customize System Target Files” on page 46-19 “Customize Build Process with STF_make_rtw_hook File” on page 44-25

You can also create custom software for ARM processors, which is based on MathWorks reference targets. For example, see:

- “Embedded Coder Support Package for ARM Cortex-A Processors”
- “Embedded Coder Support Package for ARM Cortex-M Processors”

Sample Custom Targets

There are technical solutions on the MathWorks Web site that you can use as a starting point to create your own target solution. The solutions provide guides to the following tasks for creating custom targets:

- Methods of embedding code onto a custom processor
- Creating a system target file
- Customizing the makefile and main file
- Adding compiler, chip, and board specific information
- Integrating legacy code and device drivers
- Creating blocks and libraries
- Implementing processor-in-the-loop (PIL) testing.

Consider these options:

- Download the embedded targets development guide zip file from this web page:

Is there an example guide on developing an embedded target...?

The zip file provides example files and a guide to developing a custom embedded target. The guide is divided into two parts, one on creating a generic custom target and another on creating a target for the Freescale® S12X processor using the Cosmic Compiler.

Read the example guide along with this document to understand the tasks for developing embedded targets.

- For more detailed example files for specific processors, see:
 - Is there an example Freescale S12X target... using the Cosmic Compiler?
 - Is there an example Freescale S12X target... using the CodeWarrior Compiler?

These example kits contain example models, code generation files, and instruction guides on generating and testing code for the processor. The Cosmic example illustrates the use of the target connectivity API for processor-in-the-loop (PIL) testing. The CodeWarrior example does not have PIL but shows CAN Calibration Protocol (CCP) and Simulink external mode.

The intent of the example kits is to provide working examples that you can use as a base to create your own target solution. The intent is not to provide a full featured and maintained Embedded Target product like those provided by MathWorks or third-party products, as listed on the Embedded Coder Hardware Support Web page.

- You can watch a video that shows an overview of the example kit www.mathworks.com/videos/programming-the-freescale-s12x-target-68811.html.
- For an example target for the ARM9 (STR9) processor, see [Is there an example ARM9 \(STR9\) target... using the GNU ARM Compiler and Hitex STR9-comStick?](#)

If you have questions on specific targets, email mytarget@mathworks.com.

The example kits and this document describe Embedded Coder features such as customized ert system target files and processor-in-the-loop testing, but you can study the examples as a starting point for use with Simulink Coder targets.

See Also

More About

- “Custom Targets” on page 46-2
- “Customize System Target Files” on page 46-19
- “Customize Template Makefiles” on page 46-44
- “Custom Target Optional Features” on page 46-57
- “Support Toolchain Approach with Custom Target” on page 46-59
- “Device Drivers” on page 46-72

Target Development Folders, Files, and Builds

Target development mechanics work with a number of folder and file types. The following topics provide the information to develop custom targets, configure folder usage, and use custom targets in the build process.

Folder and File Naming Conventions

You can use a single folder for your custom target files, or if desired you can use subfolders, for example containing files associated with specific development environments or tools.

For a custom target implementation, the recommended folder and file naming conventions are

- Use *only* lowercase in folder names, filenames, and extensions.
- Do not embed spaces in folder names. Spaces in folder names cause errors with many third-party development environments.
- Include desired folders in the MATLAB path
- Do *not* place your custom target folder anywhere in the MATLAB folder tree (that is, in or under the *matlabroot* folder). If you place your folder under *matlabroot* you risk losing your work if you install a new MATLAB version (or reinstall the current version).

The following sections explain how to organize your target folders and files and add them to your MATLAB path. They also provide high-level descriptions of the files.

In this document, *mytarget* is a placeholder name that represents folders and files that use the target's name. The names *dev_tool1*, *dev_tool2*, and so on represent subfolders containing files associated with development environments or tools. This document describes an example structure where the folder *mytarget* contains subfolders for *mytarget*, *blocks*, *dev_tool1*, *dev_tool2*. The top level folder *mytarget* is the *target root folder*.

Components of a Custom Target

- “Overview” on page 46-6
- “Code Components” on page 46-7
- “Control Files” on page 46-8

Overview

The components of a custom target are files located in a hierarchy of folders. The top-level folder in this structure is called the *target root folder*. The target root folder and its contents are named, organized, and located on the MATLAB path according to conventions described in “Folder and File Naming Conventions” on page 46-6.

The components of a custom target include

- Code components: C source code that supervises and supports execution of generated model code.
- Control files:
 - A system target file to control the code generation process.

- File(s) to control the building of an executable from the generated code. In a traditional make-based environment, a template makefile (TMF) generates a makefile for this purpose. Another approach is to generate project files in support of a modern integrated development environment (IDE) such as the Freescale Semiconductor CodeWarrior IDE.
- Hook files: Optional TLC and MATLAB program files that can be invoked at well-defined stages of the build process. Hook files let you customize the build process and communicate information between various phases of the process.
- Other target files: Files that let you integrate your target into the MATLAB environment. For example, you can provide an `info.xml` file to make your target block libraries and examples available from a MATLAB session.

The next sections introduce key concepts and terminology you need to know to develop each component. References to more detailed information sources are provided.

Code Components

An executable program containing code generated from a Simulink model consists of a number of code modules and data structures. These fall into two categories: application components and execution support files.

Application Components

Application components are those which are specific to a particular model; they implement the functions represented by the blocks in the model. Application components are not specific to the target. Application components include

- Modules generated from the model
- User-written blocks (S-functions)
- Parameters of the model that are visible, and can be interfaced to, external code

Execution Support Files

A number of code modules and data structures, referred to collectively as the *execution support files*, are responsible for managing and supporting the execution of the generated program. The execution support files modules are not automatically generated. Depending on the requirements of your target, you must implement certain parts of the execution support files. Execution Support Files summarizes the execution support files.

Execution Support Files

You Provide...	The Code Generator Provides...
Customized main program	Generic main program
Timer interrupt handler to run model	Execution engine and integration solver (called by timer interrupt handler)
Other interrupt handlers	Example interrupt handlers (Asynchronous Interrupt blocks)
Device drivers	Example device drivers
Data logging, parameter tuning, signal monitoring, and external mode support	Data logging, parameter tuning, signal monitoring, and external mode APIs

User-Written Execution Support Files

The code generator provides most of the execution support files. Depending on the requirements of your target, you must implement some or all of the following elements:

- A timer *interrupt service routine* (ISR). The timer runs at the program's base sample rate. The timer ISR is responsible for operations that must be completed within a single clock period, such as computing the current output sample. The timer ISR usually calls the `rt_OneStep` function.

If you are targeting a real-time operating system (RTOS), your generated code usually executes under control of the timing and task management mechanisms provided by the RTOS. In this case, you may not have to implement a timer ISR.

- The *main program*. Your main program initializes the blocks in the model, installs the timer ISR, and executes a background task or loop. The timer periodically interrupts the main loop. If the main program is designed to run for a finite amount of time, it is also responsible for cleanup operations — such as memory deallocation and masking the timer interrupt — before terminating the program.

If you are targeting a real-time operating system (RTOS), your main program most likely spawns tasks (corresponding to the sample rates used in the model) whose execution is timed and controlled by the RTOS.

Your main program typically is based on a generated or static main program. For details on the structure of the execution support files, code execution, and guidelines for customizing main programs, see “Deploy Applications to Target Hardware” (Embedded Coder).

- *Device drivers*. Drivers communicate with I/O devices on your target hardware. In production code, device drivers are normally implemented as inlined S-functions.
- *Other interrupt handlers*. If your models need to support asynchronous events, such as hardware generated interrupts and asynchronous read and write operations, you must supply interrupt handlers. The Interrupt Templates library provides examples.
- *Data logging, parameter tuning, signal monitoring, and external mode support*. It is atypical to implement rapid prototyping features such as external mode support in an embedded target. However, it is possible to support these features by using standard code generator APIs. See “Generated Code Interfacing” for details.

Control Files

The code generation and build process is directed by a number of TLC and MATLAB files collectively called *control files*. This section introduces and summarizes the main control files.

Top-Level Control File (`make_rtw`)

The build process is initiated when you press **Ctrl+B**. At this point, the build process parses the **Make command** field of the **Code Generation** pane of the Configuration Parameters dialog box, expecting to find the name of a MATLAB command that controls the build process (as well as optional arguments to that command). The default command is `make_rtw`, and the default top-level control file for the build process is `make_rtw.m`.

Note `make_rtw` is an internal MATLAB command used by the build process. Normally, target developers do not need detailed knowledge of how `make_rtw` works. (The details for target developers are described in “Target Development and the Build Process” on page 46-14.) You should not invoke `make_rtw` directly from MATLAB code, and you should not customize `make_rtw.m`.

The `make_rtw.m` file contains the logic required to execute your target-specific control files, including a number of hook points for execution of your custom code. `make_rtw` does the following:

- Passes optional arguments in to the build process
- Performs required preprocessing before code generation
- Executes the system target file to perform code generation (and optional HTML report generation)
- Processes the TMF to generate a makefile
- Invokes a make utility to execute the makefile and build an executable
- Performs required post-processing (such as generating calibration data files or downloading the generated executable to the target)

System Target File

The Target Language Compiler (TLC) generates target-specific C or C++ code from a partial description of your Simulink block diagram (`model.rtw`). The Target Language Compiler reads `model.rtw` and executes a program consisting of several target files (`.tlc` files.) The system target file, at the top level of this program, controls the code generation process. The output of this process is a number of source files, which are fed to your development system's make utility.

You need to create a customized system target file to set code generation parameters for your target. You should copy, rename, and modify the standard ERT system target file (`matlabroot/rtw/c/ert/ert.tlc`).

The detailed structure of the system target file is described in “Customize System Target Files” on page 46-19.

Note The system target file selects whether the target supports the toolchain approach or template makefile approach for code generation. See “Customize System Target Files” on page 46-19.

Template Makefile (TMF)

A TMF provides information about your model and your development system. The build process uses this information to create a makefile (`.mk` file) that builds an executable program.

Some targets implement more than one TMF, in order to support multiple development environments (for example, two or more cross-compilers) or multiple modes of code generation (for example, generating a binary executable versus generating a project file for your compiler).

The Embedded Coder software provides a large number of TMFs suitable for different types of development computer systems. These TMFs are located in `matlabroot/toolbox/coder/compile/tmf`. The standard TMFs are described in “Template Makefiles and Make Options” on page 31-36.

The detailed structure of the TMF is described in “Customize Template Makefiles” on page 46-44.

Note The system target file selects whether the target supports the toolchain approach or template makefile approach for code generation. See “Customize System Target Files” on page 46-19.

Hook Files

The build process allows you to supply optional *hook files* that are executed at specified points in the code generation and make process. You can use hook files to add target-specific actions to the build process.

The hook files must follow well-defined naming and location requirements. “Folder and File Naming Conventions” on page 46-6 describes these requirements.

Key Folders Under Target Root (mytarget)

- “Target Root Folder (mytarget)” on page 46-10
- “Target Folder (mytarget/mytarget)” on page 46-10
- “Target Block Folder (mytarget/blocks)” on page 46-10
- “Development Tools Folder (mytarget/dev_tool1, mytarget/dev_tool2)” on page 46-11
- “Target Source Code Folder (mytarget/src)” on page 46-11

Target Root Folder (mytarget)

This folder contains the key subfolders for the target (see “Folder and File Naming Conventions” on page 46-6). You can also locate miscellaneous files (such as a `readme` file) in the target root folder. The following sections describe required and optional subfolders and their contents.

Target Folder (mytarget/mytarget)

This folder contains files that are central to the target, such as the system target file (STF) and template makefile (TMF). “Key Files in Target Folder (mytarget/mytarget)” on page 46-12 summarizes the files that should be stored in `mytarget/mytarget`, and provides pointers to detailed information about these files.

Note `mytarget/mytarget` should be on the MATLAB path.

Target Block Folder (mytarget/blocks)

If your target includes device drivers or other blocks, locate the block implementation files in this folder. `mytarget/blocks` contains

- Compiled block MEX-files
- Source code for the blocks
- TLC inlining files for the blocks
- Library models for the blocks (if you provide your blocks in one or more libraries)

Note `mytarget/blocks` should be on the MATLAB path.

You can also store example models and supporting files in `mytarget/blocks`. Alternatively, you can create a `mytarget/mytargetdemos` folder, which should also be on the MATLAB path.

To display your blocks in the standard Simulink Library Browser and/or integrate your example models into the MATLAB session environment, you can create the files described below and store them in `mytarget/blocks`.

mytarget/blocks/slblocks.m

This file allows a group of blocks to be integrated into the Simulink Library and Simulink Library Browser.

Example 46.1. Example slblocks.m File

```
function blkStruct = slblocks
% Information for "Blocksets and Toolboxes" subsystem
blkStruct.Name = sprintf('Embedded Target\n for MYTARGET');
blkStruct.OpenFcn = 'mytargetlib';
blkStruct.MaskDisplay = 'disp(''MYTARGET'')';

% Information for Simulink Library Browser
Browser(1).Library = 'mytargetlib';
Browser(1).Name = 'Embedded Target for MYTARGET';
Browser(1).IsFlat = 1;% Is this library "flat" (i.e. no subsystems)?

blkStruct.Browser = Browser;
```

mytarget/blocks/demos.xml

This file provides information about the components, organization, and location of example models. MATLAB software uses this information to place the example in the MATLAB session environment.

Example 46.2. Example demos.xml File

```
<?xml version="1.0" encoding="utf-8"?>
<demos>
  <name>Embedded Target for MYTARGET</name>
  <type>simulink</type>
  <icon>$toolbox/matlab/icons/boardicon.gif</icon>
  <description source = "file">mytarget_overview.html</description>

  <demosession>
    <label>Multirate model</label>
    <demoitem>
      <label>MYTARGET demo</label>
      <file>mytarget_overview.html</file>
      <callback>mytarget_model</callback>
    </demoitem>
  </demosession>
</demos>
```

Development Tools Folder (mytarget/dev_tool1, mytarget/dev_tool2)

These folders contain files associated with specific development environments or tools (`dev_tool1`, `dev_tool2`, etc.). Normally, your target supports at least one such development environment and invokes its compiler, linker, and other utilities during the build process. `mytarget/dev_tool1` includes linker command files, startup code, hook functions, and other files required to support this process.

For each development environment, you should provide a separate folder.

Target Source Code Folder (mytarget/src)

This folder is optional. If the complexity of your target requires it, you can use `mytarget/src` to store common source code and configuration code (such as boot and startup code).

Key Files in Target Folder (mytarget/mytarget)

- “Introduction” on page 46-12
- “mytarget.tlc” on page 46-12
- “mytarget.tmf” on page 46-12
- “mytarget_genfiles.tlc” on page 46-12
- “mytarget_main.c” on page 46-13
- “STF_make_rtw_hook.m” on page 46-13
- “STF_rtw_info_hook.m (obsolete)” on page 46-13
- “info.xml” on page 46-13
- “mytarget_overview.html” on page 46-13

Introduction

The target folder `mytarget/mytarget` contains key files in your target implementation. These include the system target file, template makefile, main program module, and optional M and TLC hook files that let you add target-specific actions to the build process. The following sections describe the key target folder files.

mytarget.tlc

`mytarget.tlc` is the system target file. Functions of the system target file include

- Making the target visible in the System Target File Browser
- Definition of code generation options for the target (inherited and target-specific)
- Providing an entry point for the top-level control of the TLC code generation process

You should base your system target file on `ert.tlc`, the system target file provided by Embedded Coder software.

“Customize System Target Files” on page 46-19 gives detailed information on the structure of the system target file, and also gives instructions on how to customize a system target file to

- Display your target in the System Target File Browser
- Add your own target options to the Configuration Parameters dialog box
- Tailor the code generation and build process to the requirements of your target

mytarget.tmf

`mytarget.tmf` is the template makefile for building an executable for your target.

For basic information on the structure and operation of template makefiles, see “Customize Template Makefiles” on page 46-44.

mytarget_genfiles.tlc

This file is optional. `mytarget_genfiles.tlc` is useful as a central file from which to invoke target-specific TLC files that generate additional files as part of your target build process. For example, your target may create sub-makefiles or project files for a development environment, or command scripts for a debugger to do automatic downloads. See “Using `mytarget_genfiles.tlc`” on page 46-31 for details.

mytarget_main.c

A main program module is required for your target. To provide a main module, you can either

- Modify the `rt_main.c` or `rt_cppclass_main.cpp` module provided by the software
- Generate `mytarget_main.c` or `.cpp` during the build process

For a description of the operation of main programs, see “Deploy Applications to Target Hardware” (Embedded Coder). The section also contains guidelines for generating and modifying a main program module.

STF_make_rtw_hook.m

`STF_make_rtw_hook.m` is an optional hook file that you can use to invoke target-specific functions or executables at specified points in the build process. `STF_make_rtw_hook.m` implements a function that dispatches to a specific action depending on the `method` argument that is passed into it.

“Customize Build Process with `STF_make_rtw_hook` File” on page 44-25 describes the operation of the `STF_make_rtw_hook.m` hook file in detail.

STF_rtw_info_hook.m (obsolete)

Prior to Release 14, custom targets supplied target-specific information with a hook file (referred to as `STF_rtw_info_hook.m`). The `STF_rtw_info_hook` specified properties such as word sizes for integer data types (for example, `char`, `short`, `int`, and `long`), and C implementation-specific properties of the custom target.

The `STF_rtw_info_hook` mechanism has been replaced by the **Hardware Implementation** pane of the Configuration Parameters dialog box. Using this dialog box, you can specify the properties that were formerly specified in your `STF_rtw_info_hook` file.

For backward compatibility, existing `STF_rtw_info_hook` files are still available. However, you should convert your target and models to use the **Hardware Implementation** pane. See “Configure Run-Time Environment Options” on page 11-2.

info.xml

This file provides information to MATLAB software that specifies where to display the target toolbox in the MATLAB session environment. For more information, see “Display Custom Documentation”.

mytarget_overview.html

By convention, this file serves as home page for the target examples.

The `<description>` field in `demos.xml` should point to `mytarget_overview.html` (see “mytarget/blocks/demos.xml” on page 46-11).

Example mytarget_overview.html File

```

<html>
<head><title>Embedded Target for MYTARGET</title></head><body>
<p style="color:#990000; font-weight:bold; font-size:x-large">Embedded Target
for MYTARGET Example Model</p>

<p>This example provides a simple model that allows you to generate an executable
for a supported target board. You can then download and run the executable and
set breakpoints to study and monitor the execution behavior.</p>

```

```
</body>  
</html>
```

Additional Files for Externally Developed Targets

- “Introduction” on page 46-14
- “mytarget/mytarget/mytarget_setup.m” on page 46-14
- “mytarget/mytarget/doc” on page 46-14

Introduction

If you are developing an embedded target that is not installed into the MATLAB tree, you should provide a target setup script and target documentation within mytarget/mytarget, for the convenience of your users. The following sections describe the required materials and where to place them.

mytarget/mytarget/mytarget_setup.m

This file script adds paths for your target to the MATLAB path. Your documentation should instruct users to run the script when installing the target.

You should include a call to the MATLAB function `savepath` in your `mytarget_setup.m` script. This function saves the added paths, so users need to run `mytarget_setup.m` only once.

The following code is an example `mytarget_setup.m` file.

```
function mytarget_setup()  
curpath = pwd;  
tgtpath = curpath(1:end-length('\mytarget'));  
addpath(fullfile(tgtpath, 'mytarget'));  
addpath(fullfile(tgtpath, 'dev_tool1'));  
addpath(fullfile(tgtpath, 'blocks'));  
addpath(fullfile(tgtpath, 'mytargetdemos'));  
savepath;  
disp('MYTARGET Target Path Setup Complete.');
```

mytarget/mytarget/doc

You should put the documentation related to your target in the folder `mytarget/mytarget/doc`.

Target Development and the Build Process

- “About the Build Process” on page 46-14
- “Build Process Phases and Information Passing” on page 46-15
- “Additional Information Passing Techniques” on page 46-16

About the Build Process

To develop an embedded target, you need a thorough understanding of the build process. Your embedded target uses the build process and might require you to modify or customize the process. A general overview of code generation and the build process is given in “Source Code Generation”.

This section supplements that overview with a description of the build process as customized by the Embedded Coder software. The emphasis is on points in the process where customization hooks are available and on passing information between different phases of the process.

This section concludes with “Additional Information Passing Techniques” on page 46-16, describing assorted tips and tricks for passing information during the build process.

Build Process Phases and Information Passing

It is important to understand where (and when) the build process obtains required information. Sources of information include

- The *model*.rtw file, which provides information about the generating model. The information in *model*.rtw is available to target TLC files.
- The code generation panes of the Configuration Parameters dialog box. Options (both general and target-specific) are provided through check boxes, menus, and edit fields. You can associate options with TLC variables in the *rtwoptions* data structure. Use the **Configuration Parameters > Code Generation > Custom Code > Code information > Defines** field to define makefile tokens .
- The selected toolchain (for toolchain approach builds) or selected template makefile .tmf (for template makefile approach builds); these generate the model-specific makefile.
- Environment variables on the host computer. Environment variables provide additional information about installed development tools.
- Other target-specific files such as target-related TLC files, linker command files, or project files.

It is also important to understand the several phases of the build process and how to pass information between the phases. The build process comprises several high-level phases:

- Execution of the top-level file (*slbuild.m*) to sequence through the build process for a target
- Conversion of the model into the TLC input file (*model*.rtw)
- Generation of the target code by the TLC compiler
- Compilation of the generated code with *make* or other utilities
- Transmission of the final generated executable to the target hardware with a debugger or download utility

It is helpful to think of each phase of the process as a different “environment” that maintains its own data. These environments include

- MATLAB code execution environment (MATLAB)
- Simulink
- Target Language Compiler execution environment
- makefile
- Development environments such as and IDE or debugger

In each environment, you might get information from the various sources mentioned above. For example, during the TLC phase, execute MATLAB file might execute to obtain information from the MATLAB environment. Also, a given phase may generate information for a subsequent phase.

See “Key Files in Target Folder (mytarget/mytarget)” on page 46-12 for details on the available MATLAB file and TLC hooks for information passing, with code examples.

Additional Information Passing Techniques

This section describes a number of useful techniques for passing information among different phases of the build process.

tlcvariable Field in rtoptions Structure

Parameters on the code generation panes of the Configuration Parameters dialog box can be associated with a TLC variable, and specified in the `tlcvariable` field of the option's entry in the `rtoptions` structure. The variable value is passed on the command line when TLC is invoked. This provides a way to make code generation parameters and their values available in the TLC phase.

See "System Target File Structure" on page 46-19 for further information.

makevariable Field in rtoptions Structure

You can associate code generation parameters with a template makefile token, that you specify in the `makevariable` field of the option's entry in the `rtoptions` structure. If a token of the same name as the `makevariable` name exists in the TMF, the token is updated with the option value when the final makefile is created. If the token does not exist in the TMF, the `makevariable` is passed in on the command line when `make` is invoked. Thus, in either case, the `makevariable` is available to the makefile.

See "System Target File Structure" on page 46-19 for further information.

Accessing Host Environment Variables

You can access host shell environment variables at the MATLAB command line by entering the `getenv` command. For example:

```
getenv ('MSDEVDIR')
```

```
ans =
```

```
D:\Applications\Microsoft Visual Studio\Common\MSDev98
```

To access the same information from TLC, use the `FEVAL` directive to invoke `getenv`.

```
%assign eVar = FEVAL("getenv", "<varname>")
```

Supplying Development Environment Information to Your Template Makefile

An embedded target must tie the build process to target-specific development tools installed on a host computer. For the `make` process to run these tools, the TMF must be able to determine the name of the tools, the path to the compiler, linker, and other utilities, and possibly the host operating system environment variable settings.

Require the end user to modify the target TMF. The user enters path information (such as the location of a compiler executable), and possibly host operating system environment variables, as `make` variables. This allows the TMF to be tailored to specific needs.

Using MATLAB Application Data

Application data provides a way for applications to save and retrieve data stored with the GUI. This technique enables you to create what is essentially a user-defined property for an object, and use this property to store data for use in the build process. For more information, see `setappdata` and `getappdata`.

The following code examples illustrates the use of application data to pass information to TLC.

This file, `tlc2appdata.m`, stores the data passed in as application data under the name passed in (`appDataName`).

```
function k = tlc2appdata(appDataName,data)
    disp([mfilename,': ',appDataName,' ', data]);
    setappdata(0,appDataName,data);
    k = 0; % TLC expects a return value for FEVAL.
```

The following sample TLC file uses the FEVAL directive to invoke `tlc2appdata.m` to store arbitrary application data, under the name `z80`.

```
%% test.tlc
%%
%assign myApp = "z80"
%assign myData = "314159"
%assign dummy = FEVAL("tlc2appdata",myApp,myData)
```

To test this technique:

- 1 Create the `tlc2appdata.m` file as shown. Check that `tlc2appdata.m` is stored in a folder on the MATLAB path.
- 2 Create the TLC file as shown. Save it as `test.tlc`.
- 3 Enter the following command at the MATLAB prompt to execute the TLC file:

```
tlc test.tlc
```

- 4 Get the application data at the MATLAB prompt:

```
k = getappdata(0,'z80')
```

The function returns the value 314159.

- 5 Enter the following command.

```
who
```

Note that application data is not stored in the MATLAB workspace. Also observe that the `z80` data is not visible. Using application data in this way has the advantage that it does not clutter the MATLAB workspace. Also, it helps prevent you from accidentally deleting your data, since it is not stored directly in your workspace.

A real-world use of application data might be to collect information from the `model.rtw` file and store it for use later in the build process.

Adding Block-Specific Information to the Makefile

The `rtwmakecfg` mechanism provides a method for inlined S-functions such as driver blocks to add information to the makefile. This mechanism is described in “Use `rtwmakecfg.m` API to Customize Generated Makefiles” on page 44-17.

See Also

More About

- “Custom Targets” on page 46-2

Customize System Target Files

This section provides information on the structure of the system target file, guidelines for customizing a system target file, and a basic tutorial that helps you to produce a skeletal system target file.

Control Code Generation With the System Target File

The system target file controls the code generation stage of the build process. The system target file also lets you control the presentation of your target to the end user. The system target file provides

- Definitions of variables that are fundamental to the build process, such as the value for the CodeFormat TLC variable
- The main entry point to the top-level TLC program that generates code
- Target information for display in the System Target File Browser
- A mechanism for defining target-specific code generation options (and other parameters related to the build process) and for displaying them in the Configuration Parameters dialog box
- A mechanism for inheriting options from another target (such as the Embedded Real-Time (ERT) target)

Note that, although the system target file is a Target Language Compiler (TLC) file, it contains embedded MATLAB code. Before creating or modifying a system target file, you should acquire a working knowledge of TLC and of the MATLAB language. “Target Language Compiler” and “Scripts vs. Functions” describe the features and syntax of both the TLC and MATLAB languages.

While reading this section, you may want to refer to the system target files provided with the code generator. Most of these files are stored in the target-specific folders under *matlabroot*/rtw/c (open). Additional system target files are stored under *matlabroot*/toolbox/rtw/targets (open).

System Target File Naming and Location Conventions

A system target file must be located in a folder on the MATLAB path for the target to be displayed in the System Target File Browser and invoked in the build process. Follow the location and naming conventions for system target files and related target files given in “Folder and File Naming Conventions” on page 46-6.

System Target File Structure

- “Overview” on page 46-20
- “Header Comments” on page 46-21
- “TLC Configuration Variables” on page 46-22
- “TLC Program Entry Point and Related %includes” on page 46-22
- “RTW_OPTIONS Section” on page 46-23
- “rtwgensettings Structure” on page 46-23
- “Additional Code Generation Options” on page 46-25
- “Model Reference Considerations” on page 46-25

Overview

This section is a guide to the structure and contents of a system target file. The following listing shows the general structure of a system target file. Note that this is not a complete code listing of a system target file. The listing consists of excerpts from each of the sections that make up a system target file.

```

%%-----
%% Header Comments Section
%%-----
%% SYSTLC: Example Real-Time Target
%%   TMF: my_target.tmf MAKE: make_rtw
%% Initial comments contain directives for STF Browser.
%% Documentation, date, copyright, and other info may follow.
    ...
%selectfile NULL_FILE
    ...
%%-----
%% TLC Configuration Variables Section
%%-----
%% Assign code format, language, target type.
%%
%assign CodeFormat = "Embedded-C"
%assign TargetType = "RT"
%assign Language   = "C"
%%
%%-----
%% TLC Program Entry Point
%%-----
%% Call entry point function.
%include "codegenentry.tlc"
%%
%%-----
%% (OPTIONAL) Generate Files for Build Process
%%-----
%include "mytarget_genfiles.tlc"
%%-----
%% RTW_OPTIONS Section
%%-----
/%
BEGIN_RTW_OPTIONS
%% Define rtwoptions structure array. This array defines target-specific
%% code generation variables, and controls how they are displayed.
rtwoptions(1).prompt = 'example code generation options';
    ...
rtwoptions(6).prompt = 'Show eliminated blocks';
rtwoptions(6).type = 'Checkbox';
    ...
%-----%
% Configure RTW code generation settings %
%-----%
    ...
%%-----
%% rtwgensettings Structure
%%-----
%% Define suffix text for naming build folder here.
rtwgensettings.BuildDirSuffix = '_mytarget_rtw'
%% Callback compatibility declaration
rtwgensettings.Version = '1';

%% (OPTIONAL) target inheritance declaration
rtwgensettings.DerivedFrom = 'ert.tlc';
%% (OPTIONAL) other rtwGenSettings fields...
    ...
END_RTW_OPTIONS
%/
%%-----
%% targetComponentClass - MATHWORKS INTERNAL USE ONLY
%% REMOVE NEXT SECTION FROM USER_DEFINED CUSTOM TARGETS
%%-----

```

```

/%
BEGIN_CONFIGSET_TARGET_COMPONENT
targetComponentClass = 'Simulink.ERTTargetCC';
END_CONFIGSET_TARGET_COMPONENT
%/

```

If you are creating a custom target based on an existing system target file, you must remove the `targetComponentClass` section (bounded by the directives `BEGIN_CONFIGSET_TARGET_COMPONENT` and `END_CONFIGSET_TARGET_COMPONENT`). This section is reserved for the use of targets developed internally by MathWorks.

Header Comments

These lines at the head of the file are formatted as TLC comments. They provide required information to the System Target File Browser and to the build process. Note that you must place the browser comments at the head of the file, before other comments or TLC statements.

The presence of the comments enables the code generator to detect system target files. When the System Target File Browser is opened, the code generator scans the MATLAB path for TLC files that have formatted header comments. The comments contain the following directives:

- **SYSTLC**: The descriptor that appears in the browser.
- **TMF**: Name of the template makefile (TMF) to use during build process. When the target is selected, this filename is displayed in the Template makefile field of the **Code Generation** pane of the Configuration Parameters dialog box.
- **MAKE**: make command to use during build process. When the target is selected, this command is displayed in the **Make command** field of the **Code Generation** pane of the Configuration Parameters dialog box.

The following header comments are from `matlabroot/rtw/c/ert/ert.tlc`.

```

%% SYSTLC: Embedded Coder TMF: ert_default_tmf MAKE: make_rtw
%% SYSTLC: Create Visual C/C++ Solution File for Embedded Coder
.
.
.

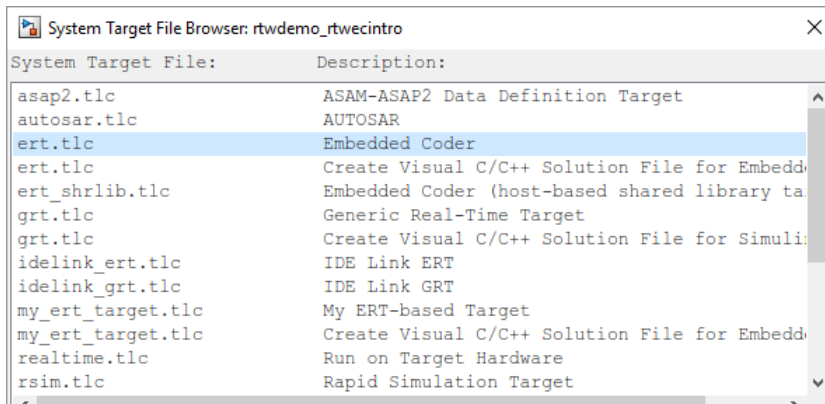
```

Each comment can only contain a maximum of two lines.

If you do not specify the TMF field in the system target file, the file is still valid. To change the value for the `TemplateMakefile` parameter, you can instead use the callback function specified by `rtwgensettings.SelectCallback`.

You can also use the callback function specified by `rtwgensettings.SelectCallback` to change the value for external mode parameters, `ExtMode`, `ExtModeTransport`, `ExtModeMexFile`, or `ExtModeIntrfLevel`.

You can specify more than one group of directives in the header comments. Each such group is displayed as a different target configuration in the System Target File Browser. In the above example, the first two lines of code specify the default configuration of the ERT target. The next two lines specify a configuration that creates and builds a Microsoft Visual C++ Solution (.sln) file. The figure below shows how these configurations appear in the System Target File Browser.



See “Create a Custom Target Configuration” on page 46-33 for an example of customized header comments.

TLC Configuration Variables

This section of the system target file assigns global TLC variables that relate to the overall code generation process.

For an embedded target, in most cases you should simply use the global TLC variable settings used by the ERT target (`ert.tlc`). It is especially important that your system target file use the 'Embedded-C' value for the `CodeFormat` TLC variable and uses the corresponding `rtwgensettings.DerivedFrom = 'ert.tlc'` in the `RTW_OPTIONS` section of the TLC file. Verify that values are assigned to the following variables:

- **CodeFormat:** The `CodeFormat` TLC variable selects generated code features. The 'Embedded-C' value for this variable is used by the ERT target. Your ERT-based target should specify 'Embedded-C' as the value for `CodeFormat`. This selection is designed for production code, minimal memory usage, static memory allocation, and a simplified interface to generated code.

For information on other values for the `CodeFormat` TLC variable, see “Compare System Target File Support Across Products” on page 21-17.

- **Language:** The only valid value is `C`, which enables support for C or C++ code generation as specified by the configuration parameter `TargetLang`.
- **TargetType:** The code generator defines the preprocessor symbols `RT` and `NRT` to distinguish simulation code from real-time code. These symbols are used in conditional compilation. The `TargetType` variable determines whether `RT` or `NRT` is defined.

Most targets are intended to generate real-time code. They assign `TargetType` as follows.

```
%assign TargetType = "RT"
```

Some targets, such as the model reference simulation target, accelerated simulation target, `RSim` target, and S-function target, generate code for use in nonreal time only. Such targets assign `TargetType` as follows.

```
%assign TargetType = "NRT"
```

TLC Program Entry Point and Related `%includes`

The code generation process normally begins with `codegenentry.tlc`. The system target file invokes `codegenentry.tlc` as follows.

```
%include "codegenentry.tlc"
```

Note `codegenentry.tlc` and the lower-level TLC files assume that `CodeFormat`, `TargetType`, and `Language` have been assigned. Set these variables before including `codegenentry.tlc`.

If you need to implement target-specific code generation features, you should include the TLC file `mytarget_genfiles.tlc` in your system target file. This file provides a mechanism for executing custom TLC code before and after invoking `codegenentry.tlc`. For information on this mechanism, see

- “Using `mytarget_genfiles.tlc`” on page 46-31 for an example of custom TLC code for execution after the main code generation entry point.
- “Target Development and the Build Process” on page 46-14 for general information on the build process, and for information on other build process customization hooks.

Another way to customize the code generation process is to call lower-level functions (normally invoked by `codegenentry.tlc`) directly, and include your own TLC functions at each stage of the process. This approach should be taken with caution. See “TLC Files” for more information.

The lower-level functions called by `codegenentry.tlc` are

- `genmap.tlc`: maps block names to corresponding language-specific block target files.
- `commonsetup.tlc`: sets up global variables.
- `commonentry.tlc`: starts the process of generating code.

RTW_OPTIONS Section

The `RTW_OPTIONS` section is bounded by the directives:

```
/%
  BEGIN_RTW_OPTIONS
  .
  .
  .
  END_RTW_OPTIONS
%/
```

The first part of the `RTW_OPTIONS` section defines an array of `rtwoptions` structures. This structure is discussed in “Using `rtwoptions` to Display Custom Target Options” on page 46-25.

The second part of the `RTW_OPTIONS` section defines `rtwgensettings`, a structure defining the build folder name and other settings for the code generation process. See “`rtwgensettings` Structure” on page 46-23 for information about `rtwgensettings`.

`rtwgensettings` Structure

The final part of the system target file defines the `rtwgensettings` structure. This structure stores information that is written to the `model.rtw` file and used by the build process. The `rtwgensettings` fields of most interest to target developers are

- `rtwgensettings.Version`: Use this property to enable `rtwoptions` callbacks and to use the Callback API in `rtwgensettings.SelectCallback`.

Note To use callbacks, you *must* set:

```
rtwgensettings.Version = '1';
```

Add the statement above to the **Configure RTW code generation settings** section of the system target file.

- `rtwgensettings.DerivedFrom`: This structure field defines the system target file from which options are to be inherited. See “Inheriting Target Options” on page 46-30.
- `rtwgensettings.SelectCallback`: This structure field specifies a `SelectCallback` function. You must set `rtwgensettings.Version = '1'`; or your callback will be ignored. `SelectCallback` is associated with the target rather than with any of its individual options. The `SelectCallback` function is triggered when you:
 - Select a target by setting the **System target file** configuration parameter.
 - Build the model.

The `SelectCallback` function is useful for setting up (or disabling) configuration parameters specific to the target.

The following code installs a `SelectCallback` function:

```
rtwgensettings.SelectCallback = 'my_select_callback_handler(hDlg,hSrc)';
```

The arguments to the `SelectCallback` function (`hDlg`, `hSrc`) are handles to private data used by the callback API functions.

Note If you have developed a custom target and you want it to be compatible with model referencing, you must implement a `SelectCallback` function to declare model reference compatibility. See “Support Model Referencing” on page 46-61.

- `rtwgensettings.ActivateCallback`: this property specifies an `ActivateCallback` function. The `ActivateCallback` function is triggered when the active configuration set of the model changes. This could happen during model loading, and also when the user changes the active configuration set.

The following code installs an `ActivateCallback` function:

```
rtwgensettings.ActivateCallback = 'my_activate_callback_handler(hDlg,hSrc)';
```

The arguments to the `ActivateCallback` function (`hDlg`, `hSrc`) are handles to private data used by the callback API functions.

- `rtwgensettings.PostApplyCallback`: this property specifies a `PostApplyCallback` function. The `PostApplyCallback` function is triggered when the user clicks the **Apply** or **OK** button after editing options in the Configuration Parameters dialog box. The `PostApplyCallback` function is called after the changes have been applied to the configuration set.

The following code installs an `PostApplyCallback` function:

```
rtwgensettings.PostApplyCallback = 'my_postapply_callback_handler(hDlg,hSrc)';
```

The arguments to the `PostApplyCallback` function (`hDlg`, `hSrc`) are handles to private data used by the callback API functions.

- `rtwgensettings.BuildDirSuffix`: To identify the build folder created by the build process, most system target files define a folder name suffix. To form the name of the build folder, the build process appends the suffix defined in the `rtwgensettings.BuildDirSuffix` field to the model name. You can modify the suffix by specifying a new string value for `rtwgensettings.BuildDirSuffix`. For example:

```
rtwgensettings.BuildDirSuffix = '_mytarget_rtw'
```

The build process creates a build folder named `model_mytarget_rtw`. For more information, see “Build Process Folders” on page 24-31.

Do *not* use a function to specify the new string value. For example:

```
rtwgensettings.BuildDirSuffix = my_get_build_suffix(bdroot);
```

Additional Code Generation Options

“Configure Generated Code with TLC” on page 44-12 describes additional TLC code generation variables. End users of a target can assign these variables by entering a MATLAB command of the form

```
set_param(modelName, 'TLCOptions', '-aVariable=val');
```

(For more information, see “Specify TLC for Code Generation” on page 31-28.)

However, the preferred approach is to assign these variables in the system target file using statements of the form:

```
%assign Variable = val
```

For readability, we recommend that you add such assignments in the section of the system target file after the comment **Configure RTW code generation settings**.

Model Reference Considerations

See “Support Model Referencing” on page 46-61 for important information on system target file and other modifications you may need to make to support the code generator model referencing features.

Define and Display Custom Target Options

- “Using `rtwoptions` to Display Custom Target Options” on page 46-25
- “Example System Target File With Customized `rtwoptions`” on page 46-29
- “Inheriting Target Options” on page 46-30

Using `rtwoptions` to Display Custom Target Options

You control the options to display in the **Code Generation** pane in the Configuration Parameters dialog box by customizing the `rtwoptions` structure in your system target file.

The fields of the `rtwoptions` structure define variables and associated user interface elements to be displayed in the Configuration Parameters dialog box. Using the `rtwoptions` structure array, you can define target-specific options displayed in the dialog box and organize options into categories. You can also write callback functions to specify how these options are processed.

When the **Code Generation** pane opens, the `rtwoptions` structure array is scanned and the listed options are displayed. Each option is represented by an assigned user interface element (check box, edit field, menu, or push button), which displays the current option value.

The user interface elements can be in an enabled or disabled (appears dimmed) state. If an option is enabled, the user can change the option value. If an option is disabled, the option uses the default value and the user cannot change the option value.

You can also use the `rtwoptions` structure array to define special NonUI elements that cause callback functions to be executed, but that are not displayed in the **Code Generation** pane. See “NonUI Elements” on page 46-29.

The elements of the `rtwoptions` structure array are organized into groups. Each group of items begins with a header element of type `Category`. The default field of a `Category` header must contain a count of the remaining elements in the category.

The `Category` header is followed by options to be displayed on the **Code Generation** pane. The header in each category is followed by one or more option definition elements.

Each category of target options corresponds to options listed under **Code Generation** in the Configuration Parameters dialog box.

The table `rtwoptions` Structure Fields Summary summarizes the fields of the `rtwoptions` structure.

Example `rtwoptions` Structure

The following `rtwoptions` structure is excerpted from an example system target file, `matlabroot/toolbox/rtw/rtwdemos/rtwoptions_demo/usertarget.tlc`. The code defines an `rtwoptions` structure array. The default field of the first (header) element is set to 4, indicating the number of elements that follow the header.

```
rtwoptions(1).prompt      = 'userPreferred target options (I)';
rtwoptions(1).type       = 'Category';
rtwoptions(1).enable     = 'on';
rtwoptions(1).default    = 4;    % number of items under this category
                             % excluding this one.
rtwoptions(1).popupstrings = ''; % At the first item, user has to
rtwoptions(1).tlcvariable = ''; % initialize all supported fields
rtwoptions(1).tooltip    = '';
rtwoptions(1).callback   = '';
rtwoptions(1).makevariable = '';

rtwoptions(2).prompt     = 'Execution Mode';
rtwoptions(2).type       = 'Popup';
rtwoptions(2).default    = 'Real-Time';
rtwoptions(2).popupstrings = 'Real-Time|UserDefined';
rtwoptions(2).tlcvariable = 'tlcvariable1';
rtwoptions(2).tooltip    = ['See this text as tooltip'];

rtwoptions(3).prompt     = 'Log Execution Time';
rtwoptions(3).type       = 'Checkbox';
rtwoptions(3).default    = 'on';
rtwoptions(3).tlcvariable = 'RL32LogTETModifier';
rtwoptions(3).tooltip    = ['']; % no tooltip

rtwoptions(4).prompt     = 'Real-Time Interrupt Source';
rtwoptions(4).type       = 'Popup';
rtwoptions(4).default    = 'Timer';
rtwoptions(4).popupstrings = 'Timer|5|6|7|8|9|10|11|12|13|14|15';
rtwoptions(4).tlcvariable = 'tlcvariable3';
rtwoptions(4).callback   = 'usertargetcallback(hDlG, hSrc, ''tlcvariable3'')';
rtwoptions(4).tooltip    = [''];
rtwoptions(4).tooltip    = ['See TLC file for how to use reserved '...'];
```

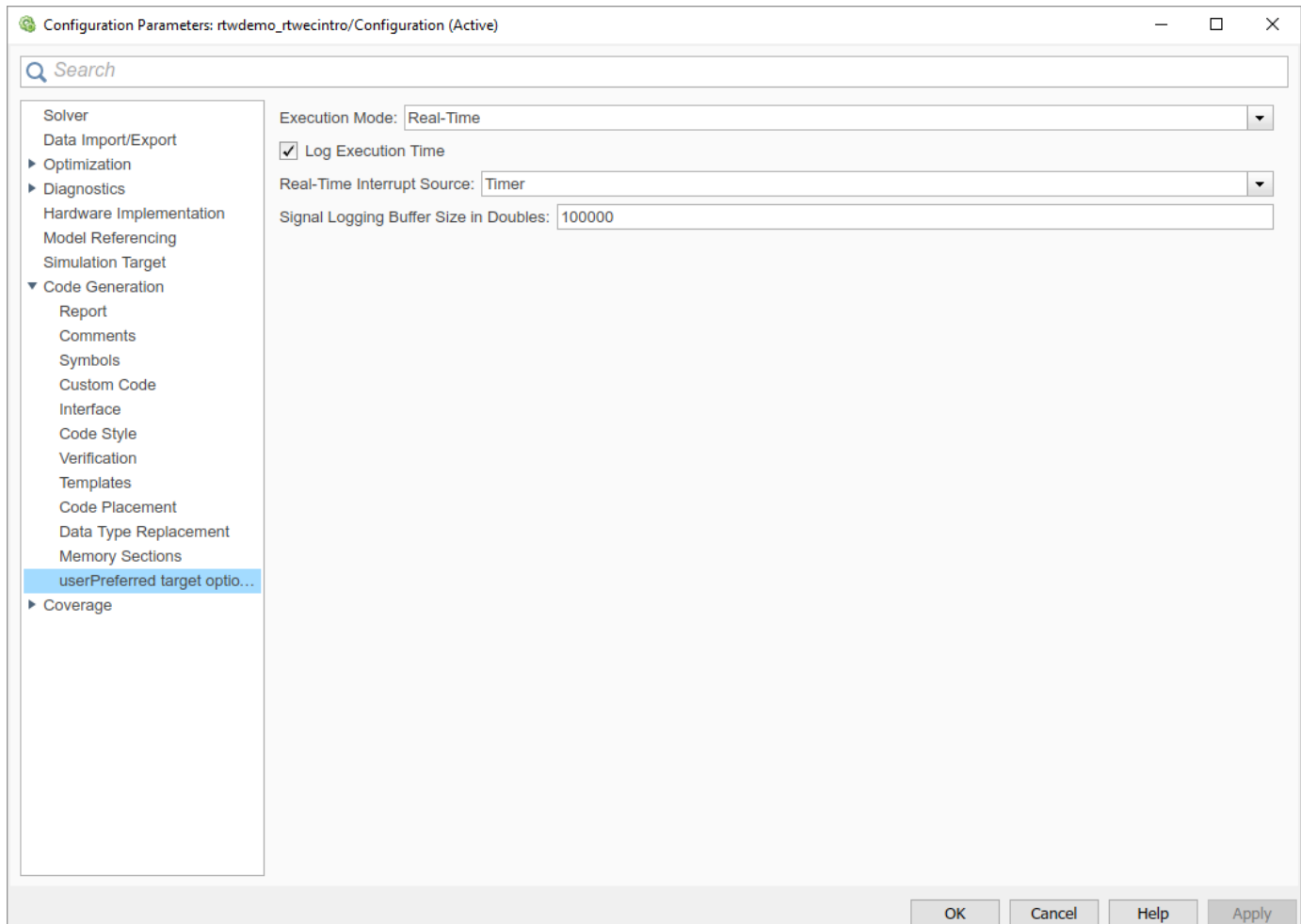


```

    ' keyword 'hdlg'', and 'hSrc''.');
...
rtwoptions(5).prompt      = 'Signal Logging Buffer Size in Doubles';
rtwoptions(5).type       = 'Edit';
rtwoptions(5).default    = '100000';
rtwoptions(5).tlcvariable = 'tlcvariable2';
rtwoptions(5).tooltip    = [''];

```

The first element adds a **userPreferred target options (I)** pane under **Code Generation** in the Configuration Parameters dialog box. The pane displays the options defined in `rtwoptions(2)`, `rtwoptions(3)`, `rtwoptions(4)`, and `rtwoptions(5)`.



If you want to define a large number of options, you can define multiple **Category** groups within a single system target file.

Note the `rtwoptions` structure and callbacks are written in MATLAB code, although they are embedded in a TLC file. To verify the syntax of your `rtwoptions` structure definitions and code, you can execute the commands at the MATLAB prompt by copying and pasting them to the MATLAB Command Window.

To learn more about `usertarget.tlc` and the example callback files provided with it, see “Example System Target File With Customized `rtwoptions`” on page 46-29. For more examples of target-specific `rtwoptions` definitions, see the `target.tlc` files under `matlabroot/rtw/c` (open).

rtwoptions Structure Fields Summary lists the fields of the rtwoptions structure.

rtwoptions Structure Fields Summary

Field Name	Description
callback	For examples of callback usage, see “Example System Target File With Customized rtwoptions” on page 46-29.
closecallback (obsolete)	Do not use closecallback. Use rtwgensettings.PostApplyCallback instead (see “rtwgensettings Structure” on page 46-23). closecallback is ignored. For examples of callback usage, see “Example System Target File With Customized rtwoptions” on page 46-29.
default	Default value of the option (empty if the type is Pushbutton).
enable	Must be 'on' or 'off'. If 'on', the option is displayed as an enabled item; otherwise, as a disabled item.
makevariable	Template makefile token (if any) associated with the option. The makevariable is expanded during processing of the template makefile. See “Template Makefile Tokens” on page 46-44.
modelReferenceParameter- Check	Specifies whether the option must have the same value in a referenced model and its parent model. If this field is unspecified or has the value 'on' the option values must be same. If the field is specified and has the value 'off' the option values can differ. See “Controlling Configuration Option Value Agreement” on page 46-63.
NonUI	Element that is not displayed, but is used to invoke a close or open callback. See “NonUI Elements” on page 46-29.
opencallback (obsolete)	Do not use opencallback. Use rtwgensettings.SelectCallback instead (see “rtwgensettings Structure” on page 46-23). For examples of callback usage, see “Example System Target File With Customized rtwoptions” on page 46-29.
popupstrings	If type is Popup, popupstrings defines the items in the menu. Items are delimited by the " " (vertical bar) character. The following example defines the items of the MAT-file variable name modifier menu used by the GRT target. 'rt_ _rt none'
prompt	Label for the option.
tlcvariable	Name of TLC variable associated with the option.
tooltip	Help text displayed when mouse is over the item.
type	Type of element: Checkbox, Edit, NonUI, Popup, Pushbutton, or Category.

NonUI Elements

Elements of the `rtwoptions` array that have type `NonUI` exist solely to invoke callbacks. A `NonUI` element is not displayed in the Configuration Parameters dialog box. You can use a `NonUI` element if you want to execute a callback that is not associated with a user interface element, when the dialog box opens or closes. See the next section, “Example System Target File With Customized `rtwoptions`” on page 46-29 for an example.

Note The default value of an element of type `NonUI` or `Edit` determines the set of values allowed for that element.

- If the default value is '0' or '1':
 - For type `NonUI`, the element stores a Boolean value.
 - For type `Edit`, the element stores a value of type `int32`.
 - If the default value contains an integer other than '0' or '1', the element stores a value of type `int32`.
 - If the default value does not contain an integer, the element is evaluated as a character vector.
-

Example System Target File With Customized `rtwoptions`

A working system target file, with MATLAB file callback functions, has been provided as an example of how to use the `rtwoptions` structure to display and process custom options on the **Code Generation** pane. The examples are compatible with the callback API.

The example target files are in the folder (open):

```
matlabroot/toolbox/rtw/rtwdemos/rtwoptions_demo
```

The example target files include:

- `usertarget.tlc`: The example system target file. This file illustrates how to define custom menus, check boxes, and edit fields. The file also illustrates the use of callbacks.
- `usertargetcallback.m`: A MATLAB file callback invoked by a menu.

Refer to the example files while reading this section. The example system target file, `usertarget.tlc`, illustrates the use of `rtwoptions` to display the following custom target options:

- The **Execution Mode** menu.
- The **Log Execution Time** check box.
- The **Real-Time Interrupt Source** menu. The menu executes a callback defined in an external file, `usertargetcallback.m`. The TLC variable associated with the menu is passed in to the callback, which displays the menu's current value.
- The edit field **Signal Logging Buffer Size in Doubles**.

Try studying the example code while interacting with the example target options in the Configuration Parameters dialog box. To interact with the example target file,

- 1 Make `matlabroot/toolbox/rtw/rtwdemos/rtwoptions_demo` (open) your working folder.
- 2 Open a model of your choice.

- 3 Open the Configuration Parameters dialog box and select the **Code Generation** pane.
- 4 Click **Browse**. The System Target File Browser opens. Select `usertarget.tlc`. Then click **OK**.
- 5 Observe that the **Code Generation** pane contains a custom sub-tab: **userPreferred target options (I)**.
- 6 As you interact with the options in this category and open and close the Configuration Parameters dialog box, observe the messages displayed in the MATLAB Command Window. These messages are printed from code in the system target file, or from callbacks invoked from the system target file.

Inheriting Target Options

`ert.tlc` provides a basic set of Embedded Coder code generation options. If your target is based on `ert.tlc`, your system target file should normally inherit the options defined in ERT.

Use the `rtwgensettings.DerivedFrom` field in the `rtwgensettings` structure to define the system target file from which options are to be inherited. You should convert your custom target to use this mechanism as follows.

Set the `rtwgensettings.DerivedFrom` field value as in the following example:

```
rtwgensettings.DerivedFrom = 'stf.tlc';
```

where `stf` is the name of the system target file from which options are to be inherited. For example:

```
rtwgensettings.DerivedFrom = 'ert.tlc';
```

When the Configuration Parameters dialog box executes this line of code, it includes the options from `stf.tlc` automatically. If `stf.tlc` is a MathWorks internal system target file that has been converted to a new layout, the dialog box displays the inherited options using the new layout.

Handling Unsupported Options

If your target does not support all of the options inherited from `ert.tlc`, you should detect unsupported option settings and display a warning or error message. In some cases, if a user has selected an option your target does not support, you may need to abort the build process. For example, if your target does not support the **Generate an example main program** option, the build process should not be allowed to proceed if that option is selected.

Even though your target may not support all inherited ERT options, it is required that the ERT options are retained in the **Code Generation** pane of the Configuration Parameters dialog box. Do not simply remove unsupported options from the `rtwoptions` structure in the system target file. Options must be in the dialog box to be scanned by the code generator when it performs optimizations.

For example, you may want to prevent users from turning off the **Single output/update function** option. It may seem reasonable to remove this option from the dialog box and simply assign the TLC variable `CombineOutputUpdateFcns` to `on`. However, if the option is not included in the dialog box, the code generator assumes that output and update functions are *not* to be combined. Less efficient code is generated as a result.

Tips and Techniques for Customizing Your System Target File

- “Introduction” on page 46-31

- “Required and Recommended %includes” on page 46-31
- “Handling Aliases for Target Option Values” on page 46-32
- “Supporting Multiple Development Environments” on page 46-33
- “Updating Your Custom System Target File” on page 46-33

Introduction

The following sections include information on techniques for customizing your system target file, including:

- How to invoke custom TLC code from your system target file
- Approaches to supporting multiple development environments
- Considerations for when you update your system target file

Required and Recommended %includes

If you need to implement target-specific code generation features, we recommend that your system target file include the TLC file `mytarget_genfiles.tlc`.

Once your system target file has set up the required TLC environment, you must include `codegenentry.tlc` to start the standard code generation process.

`mytarget_genfiles.tlc` provides a mechanism for executing custom TLC code after the main code generation entry point. See “Using `mytarget_genfiles.tlc`” on page 46-31.

Using `mytarget_genfiles.tlc`

`mytarget_genfiles.tlc` (optional) is useful as a central file from which to invoke target-specific TLC files that generate additional files as part of your target build process. For example, your target may create sub-makefiles or project files for a development environment, or command scripts for a debugger to do automatic downloads.

The build process can then invoke these generated files either directly from the make process or after the executable is created. This is done with the `STF_make_rtw_hook.m` mechanism, as described in “Customize Build Process with `STF_make_rtw_hook` File” on page 44-25.

The following TLC code shows an example `mytarget_genfiles.tlc` file.

```
%selectfile NULL_FILE

%assign ModelName = CompiledModel.Name

%% Create Debugger script
%assign model_script_file = "%<ModelName>.cfg"
%assign script_file = "debugger_script_template.tlc"

%if RTWVerbose
  %selectfile STDOUT
  ### Creating %<model_script_file>
  %selectfile NULL_FILE
%endif

%include "%<script_file>"
%openfile bld_file = "%<model_script_file>"
```

```
%<CreateDebuggerScript()>  
%closefile bld_file
```

Handling Aliases for Target Option Values

This section describes utility functions that can be used to detect and resolve alias values or legacy values when testing user-specified values for the target device type (`ProdHWDeviceType`) and the code replacement library (`CodeReplacementLibrary`).

RTW.isHWDeviceTypeEq

To test if two target device type values represent the same hardware device, invoke the following function:

```
result = RTW.isHWDeviceTypeEq(type1, type2)
```

where *type1* and *type2* are character vectors containing target device type values or aliases.

The `RTW.isHWDeviceTypeEq` function returns true if *type1* and *type2* are character vectors representing the same hardware device. For example, the following call returns true:

```
RTW.isHWDeviceTypeEq('Specified', 'Generic->Custom')
```

For a description of the target device type option `ProdHWDeviceType`, see the command-line information for the **Hardware Implementation** pane parameters “Device vendor” and “Device type”.

RTW.resolveHWDeviceType

To return the device type value for a hardware device, given a value that might be an alias or legacy value, invoke the following function:

```
result = RTW.resolveHWDeviceType(type)
```

where *type* is a character vector containing a target device type value or alias.

The `RTW.resolveHWDeviceType` function returns the device type value of the device. For example, the following calls both return 'Generic->Custom':

```
RTW.resolveHWDeviceType('Specified')  
RTW.resolveHWDeviceType('Generic->Custom')
```

For a description of the target device type option `ProdHWDeviceType`, see the command-line information for the **Hardware Implementation** pane parameters “Device vendor” and “Device type”.

RTW.isTfLEq

To test if two code replacement library (CRL) names represent the same CRL, invoke the following function:

```
result = RTW.isTfLEq(name1, name2)
```

where *name1* and *name2* are character vectors containing CRL values or aliases.

The `RTW.isTfLEq` function returns true if *name1* and *name2* are character vectors representing the same code replacement library. For example, the following call returns true:

```
RTW.isTfLEq('GNU', 'GNU C99 extensions')
```

For a description of the `CodeReplacementLibrary` parameter, see Code replacement library.

RTW.resolveTflName

To return the CRL value for a code replacement library, given a value that might be an alias or legacy value, invoke the following function:

```
result = RTW.resolveTflName(name)
```

where *name* is a character vector containing a CRL value or alias.

The `RTW.resolveTflName` function returns the value of the referenced code replacement library. For example, the following calls both return `'GNU C99 extensions'`:

```
RTW.resolveTflName('GNU')
RTW.resolveTflName('GNU C99 extensions')
```

For a description of the `CodeReplacementLibrary` parameter, see Code replacement library.

Supporting Multiple Development Environments

Your target may require support for multiple development environments (for example, two or more cross-compilers) or multiple modes of code generation (for example, generating a binary executable versus generating a project file for your compiler).

One approach to this requirement is to implement multiple system target files. Each system target file invokes a template makefile for the development environment. This amounts to providing two separate targets.

Updating Your Custom System Target File

Updating your custom system target file can impact the option values of a model that you load that uses the updated system target file. If you disable an option, the updated system target file uses the default value for that option. If a model has a different value for that option, when you load the model with the updated system target file, the value from the model is discarded and the system target file uses the default value instead.

Create a Custom Target Configuration

- “Introduction” on page 46-33
- “my_ert_target Overview” on page 46-34
- “Creating Target Folders” on page 46-35
- “Create ERT-Based, Toolchain Compliant System Target File” on page 46-36
- “Create ERT-Based Template MakeFile” on page 46-40
- “Create Test Model and S-Function” on page 46-41
- “Verify Target Operation” on page 46-42

Introduction

This tutorial can supplement the example target guides described in “Sample Custom Targets” on page 46-4. For an introduction and example files, try the example targets first.

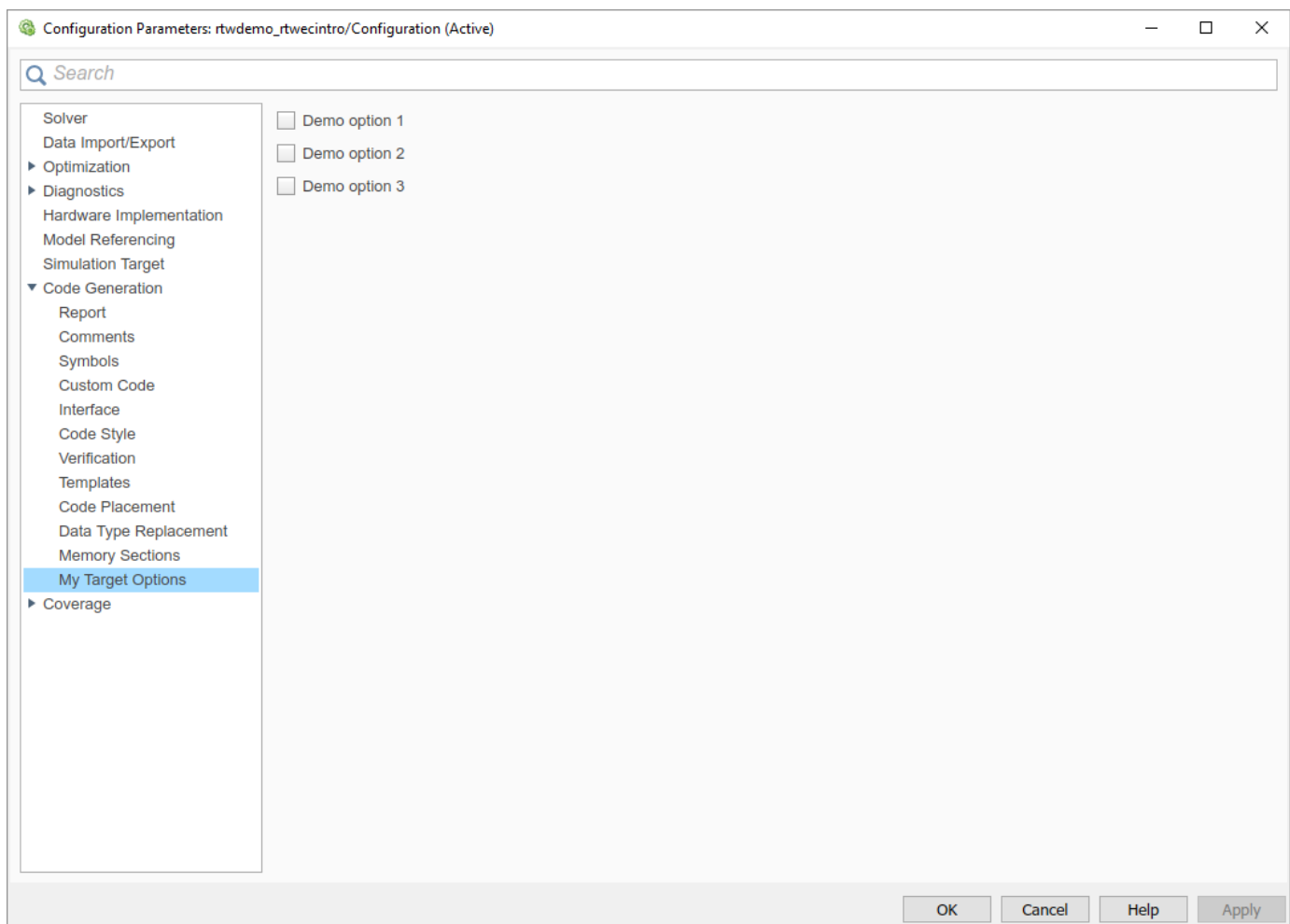
This tutorial guided you through the process of creating an ERT-based target, `my_ert_target`. This exercise illustrates several tasks, which are typical for creating a custom target:

- Setting up target folders and modifying the MATLAB path.
- Making modifications to a standard system target file and TMF such that the custom target is visible in the System Target File Browser, inherits ERT options, displays target-specific options, and generates code with the default host-based compiler.
- Testing the build process with the custom target, using a simple model that incorporates an inlined S-function.

During this exercise, you implement an operational, but skeletal, ERT-based target. This target can be useful as a starting point in a complete implementation of a custom embedded target.

my_ert_target Overview

In the following sections, you create a skeletal target, `my_ert_target`. The target inherits and supports the standard options of the ERT target and displays additional target-specific options in the Configuration Parameters dialog box (see “Target-Specific Options for `my_ert_target`” on page 46-34).



Target-Specific Options for my_ert_target

`my_ert_target` supports a toolchain-based build, generating code and executables that run on the host system. `my_ert_target` uses the `lcc` compiler on a Microsoft Windows platform. The chosen

compiler is readily available and is distributed with the code generator. On a Microsoft Windows platform, if you use a different compiler, you can set up `lcc` temporarily as your default compiler through the following MATLAB command:

```
mex -setup
```

The software displays links for supported compilers that are installed on your computer. Click the `lcc` link.

Note On Linux systems, make sure that you have an installed C compiler. If so, you can use Linux folder syntax to complete this exercise.

`my_ert_target` can also support template makefile-based builds. For more information about using this target with the template makefile approach, see “Create ERT-Based Template MakeFile” on page 46-40.

You can test `my_ert_target` with a model that is compatible with the ERT target (see “Configure a System Target File” (Embedded Coder)). Generated programs operate identically to ERT generated programs.

To simplify the testing of your target, test with `targetmodel`, a very simple fixed-step model (see “Create Test Model and S-Function” on page 46-41). The S-Function block in `targetmodel` uses the source code from the `timestwo` example, and generates fully inlined code. See “S-Function Examples” and “Inline S-Functions with TLC” for further discussion of the `timestwo` example S-function.

Creating Target Folders

Create folders to store the target files and add them to the MATLAB path, following the recommended conventions (see “Folder and File Naming Conventions” on page 46-6). You also create a folder to store the test model, S-function, and generated code.

This example assumes that your target and model folders are located within the folder `c:/work`. Do not place your target and model folders within the MATLAB folder tree (that is, in or under the `matlabroot` folder).

To create the folders and make them accessible:

- 1 Create a target root folder, `my_ert_target`. From the MATLAB Command Window on a Windows platform, enter:

```
cd c:/work
mkdir my_ert_target
```

- 2 Within the target root folder, create a subfolder to store your target files.

```
mkdir my_ert_target/my_ert_target
```

- 3 Add these folders to your MATLAB path.

```
addpath c:/work/my_ert_target
addpath c:/work/my_ert_target/my_ert_target
```

- 4 Create a folder, `my_targetmodel`, to store the test model, S-function, and generated code.

```
mkdir my_targetmodel
```

Create ERT-Based, Toolchain Compliant System Target File

Create a system target file for your target by copying and modifying the standard system target file for the ERT target. Then, validate the system target file by viewing the new target in the System Target File Browser and in the Configuration Parameters dialog box.

Editing the System Target File

To edit the system target file, use these steps:

- 1 Change your working folder to the folder you created in “Creating Target Folders” on page 46-35.

```
cd c:/work/my_ert_target/my_ert_target
```

- 2 Place a copy of `matlabroot/rtw/c/ert/ert.tlc` in `c:/work/my_ert_target/my_ert_target` and rename it to `my_ert_target.tlc`. The file `ert.tlc` is the system target file for the ERT target.
- 3 Open `my_ert_target.tlc` in a text editor of your choice.
- 4 Customize the system target file, replacing the header comment lines with directives that make your system target file visible in the System Target File Browser and define the associated TMF, make command, and external mode interface file (if any). For more information about these directives, see “Header Comments” on page 46-21 .

Replace the header comments in `my_ert_target.tlc` with the following header comments.

```
%% SYSTLC: My ERT-based Target TMF: my_ert_target_lcc.tmf MAKE: make_rtw
```

- 5 The file `my_ert_target.tlc` inherits the standard ERT options, using the mechanism described in “Inheriting Target Options” on page 46-30. Therefore, the existing `rtwoptions` structure definition is superfluous. Edit the `RTW_OPTIONS` section such that it includes only the following code.

```
/%
BEGIN_RTW_OPTIONS

%-----%
% Configure RTW code generation settings %
%-----%

rtwgenSettings.BuildDirSuffix = '_ert_rtw';

END_RTW_OPTIONS
%/
```

- 6 Delete the code after the end of the `RTW_OPTIONS` section, which is delimited by the directives `BEGIN_CONFIGSET_TARGET_COMPONENT` and `END_CONFIGSET_TARGET_COMPONENT`. This code is for use only by internal MathWorks developers.
- 7 Modify the build folder suffix in the `rtwgenSettings` structure in accordance with the conventions described in “`rtwgenSettings` Structure” on page 46-23.

To set the suffix to a character vector for the `_my_ert_target` custom target, change the line

```
rtwgenSettings.BuildDirSuffix = '_ert_rtw'
```

to

```
rtwgenSettings.BuildDirSuffix = '_my_ert_target_rtw'
```

- 8 Modify the `rtwgenSettings` structure to inherit options from the ERT target and declare Release 14 or later compatibility as described in “`rtwgenSettings` Structure” on page 46-23. Add the following code to the `rtwgenSettings` definition:

```
rtwgenSettings.DerivedFrom = 'ert.tlc';
rtwgenSettings.Version = '1';
```

- 9 Add an `rtwoptions` structure that defines a target-specific options category with three check boxes just after the `BEGIN_RTW_OPTIONS` directive. The following code shows the complete `RTW_OPTIONS` section, including the previous `rtwgenSettings` changes.

```
/%
BEGIN_RTW_OPTIONS

    rtwoptions(1).prompt      = 'My Target Options';
    rtwoptions(1).type       = 'Category';
    rtwoptions(1).enable     = 'on';
    rtwoptions(1).default    = 3;    % number of items under this category
                                % excluding this one.

    rtwoptions(1).popupstrings = '';
    rtwoptions(1).tlcvariable = '';
    rtwoptions(1).tooltip    = '';
    rtwoptions(1).callback   = '';
    rtwoptions(1).makevariable = '';

    rtwoptions(2).prompt      = 'Demo option 1';
    rtwoptions(2).type       = 'Checkbox';
    rtwoptions(2).default    = 'off';
    rtwoptions(2).tlcvariable = 'DummyOpt1';
    rtwoptions(2).makevariable = '';
    rtwoptions(2).tooltip    = ['Demo option1 (non-functional)'];
    rtwoptions(2).callback   = '';

    rtwoptions(3).prompt      = 'Demo option 2';
    rtwoptions(3).type       = 'Checkbox';
    rtwoptions(3).default    = 'off';
    rtwoptions(3).tlcvariable = 'DummyOpt2';
    rtwoptions(3).makevariable = '';
    rtwoptions(3).tooltip    = ['Demo option2 (non-functional)'];
    rtwoptions(3).callback   = '';

    rtwoptions(4).prompt      = 'Demo option 3';
    rtwoptions(4).type       = 'Checkbox';
    rtwoptions(4).default    = 'off';
    rtwoptions(4).tlcvariable = 'DummyOpt3';
    rtwoptions(4).makevariable = '';
    rtwoptions(4).tooltip    = ['Demo option3 (non-functional)'];
    rtwoptions(4).callback   = '';

    %-----%
    % Configure RTW code generation settings %
    %-----%

    rtwgenSettings.BuildDirSuffix = '_my_ert_target_rtw';
    rtwgenSettings.DerivedFrom = 'ert.tlc';
    rtwgenSettings.Version = '1';
    rtwgenSettings.SelectCallback = 'enableToolchainCompliant(hSrc, hDlg)';
    %SelectCallback provides toolchain approach support, but requires custom function
    %Omit this SelectCallback if using the template makefile approach

END_RTW_OPTIONS
%/
```

- 10 Save your changes to `my_ert_target.tlc` and close the file.

Create ToolchainCompliant Function

To enable builds using the toolchain approach, you create a function that corresponds to the `SelectCallback` near the end of the custom system target file. This function sets properties for toolchain compliance.

```
function enableToolchainCompliant(hSrc, hDlg)
    % The following parameters enable toolchain compliance.
    slConfigUISetVal(hDlg, hSrc, 'UseToolchainInfoCompliant', 'on');
```

```

slConfigUISetVal(hDlg, hSrc, 'GenerateMakefile','on');

% The following parameters are not required for toolchain compliance.
% But, it is recommended practice to set these default values and
% disable the parameters (as shown).
slConfigUISetVal(hDlg, hSrc, 'RTWCompilerOptimization','off');
slConfigUISetVal(hDlg, hSrc, 'MakeCommand','make_rtw');
slConfigUISetEnabled(hDlg, hSrc, 'RTWCompilerOptimization',false);
slConfigUISetEnabled(hDlg, hSrc, 'MakeCommand',false);
hCS = hSrc.getConfigSet();
hCS.refreshDialog;
end

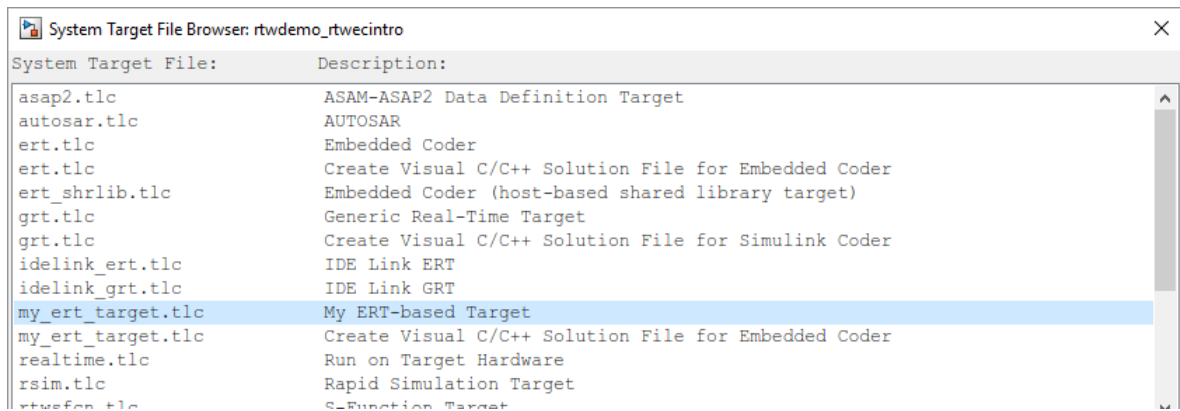
```

Note If you are using the template makefile approach, omit calling the function enabling toolchain-compliance from your system target file. Instead, use the information in “Create ERT-Based Template MakeFile” on page 46-40.

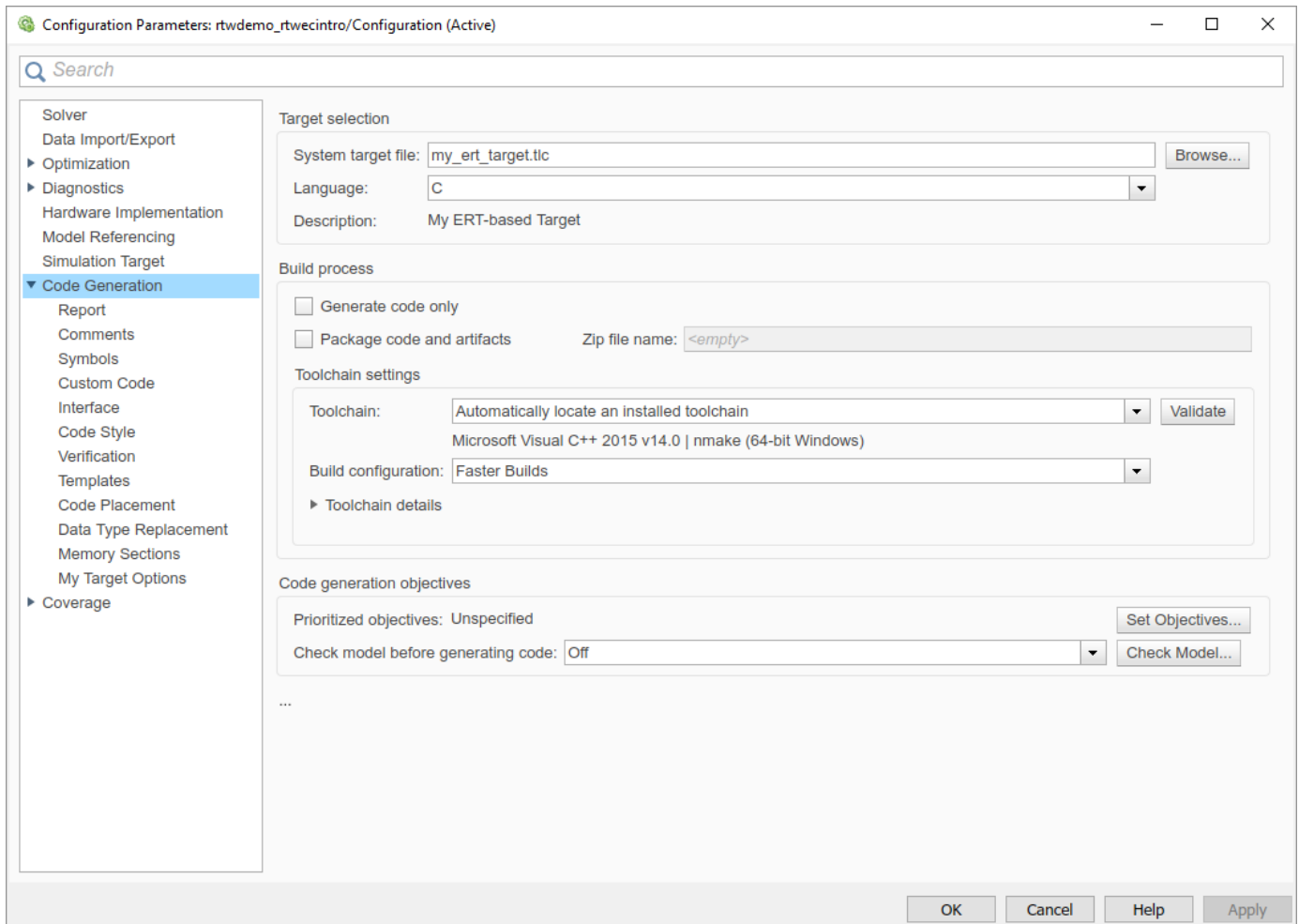
Viewing the System Target File

At this point, you can verify that the target inherits and displays ERT options as follows:

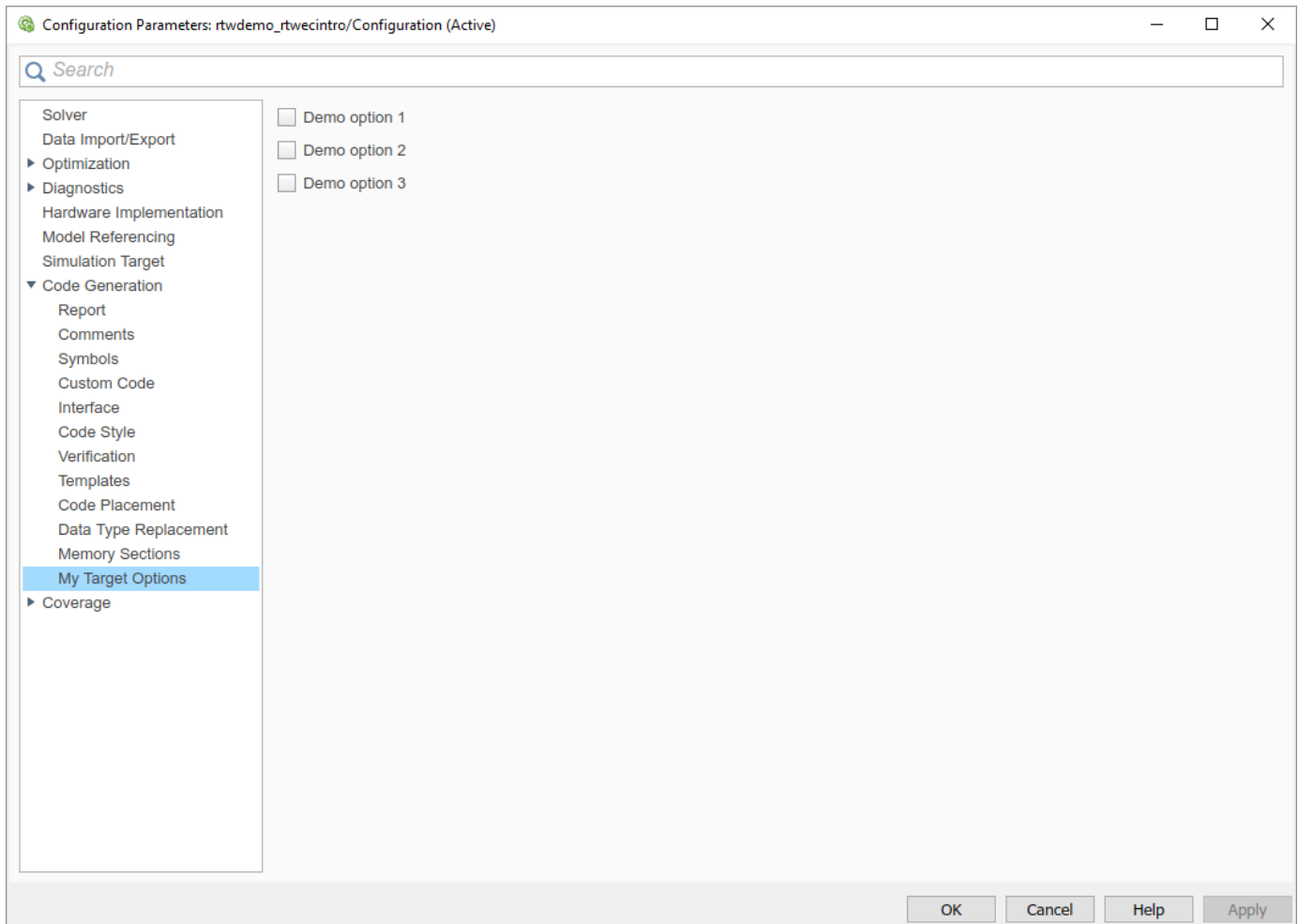
- 1 Create a new model.
- 2 Open the Model Explorer or the Configuration Parameters dialog box.
- 3 Select the **Code Generation** pane.
- 4 Click **Browse** to open the System Target File browser.
- 5 In the file browser, scroll through the list of targets to find the new target, `my_ert_target.tlc`. (This step assumes that your MATLAB path contains `c:/work/my_ert_target/my_ert_target`, as previously set in “Creating Target Folders” on page 46-35.)
- 6 Select My ERT-based Target and click **OK**.



- 7 The **Code Generation** pane now shows that the model is configured for the `my_ert_target.tlc` target. The **System target file**, **Language**, **Toolchain**, and **Build configuration** fields should appear:



- 8 Select the **My Target Options** pane. The target displays the three check box options defined in the `rtwoptions` structure.



- 9 Select the **Code Generation** pane and reopen the System Target File Browser.
- 10 Select the Embedded Coder target (`ert.tlc`). The target displays the standard ERT options.
- 11 Close the model. You do not need to save it.

The system target file for the skeletal target is complete. If you are using the toolchain approach, you are ready to invoke the build process for your target.

If you prefer to use the template makefile approach, the reference to a TMF, `my_ert_target_lcc.tmf`, in the system target file header comments prevents you from invoking the build process for your target until the TMF file is in place. First, you must create a `my_ert_target_lcc.tmf` file.

Create ERT-Based Template MakeFile

If you are using the toolchain makefile approach with a toolchain compliant custom target, omit the steps that apply to the template makefile approach. (Skip this section.)

If you are using the template makefile approach, follow the steps that apply to a template makefile and omit calling the function that enables toolchain-compliance from your system target file, which is described in “Create ERT-Based, Toolchain Compliant System Target File” on page 46-36.

Create a template makefile for your target by copying and modifying a supplied ERT template makefile. Select the template makefile that fits your target best. This example uses `ert_lcc64.tmf`, but you can also use `ert_vcx64.tmf` or `ert_unix.tmf`.

- 1 Make sure that your working folder is still set to the target file folder you created previously in “Creating Target Folders” on page 46-35.

```
c:/work/my_ert_target/my_ert_target
```

- 2 Place a copy of `matlabroot/toolbox/coder/compile/tmf/ert_lcc64.tmf` in `c:/work/my_ert_target/my_ert_target` and rename it `my_ert_target_lcc.tmf`. The file `ert_lcc64.tmf` is the ERT compiler-specific template makefile for the LCC compiler.

- 3 Open `my_ert_target_lcc.tmf` in a text editor.

- 4 Change the `SYS_TARGET_FILE` parameter so that the file reference for your `.tlc` file is generated in the make file. Change the line

```
SYS_TARGET_FILE = any
```

to

```
SYS_TARGET_FILE = my_ert_target.tlc
```

- 5 Save changes to `my_ert_target_lcc.tmf` and close the file.

Your target can now generate code and build a host-based executable. In the next sections, you create a test model and test the build process using `my_ert_target`.

Create Test Model and S-Function

In this section, you build a simple test model for later use in code generation:

- 1 Set your working folder to `c:/work/my_targetmodel`.

```
cd c:/work/my_targetmodel
```

For the remainder of this tutorial, `my_targetmodel` is assumed to be the working folder. Your target writes the output files of the code generation process into a build folder within the working folder. When inlined code is generated for the `timestwo` S-function, the build process looks for the TLC implementation of the S-function in the working folder.

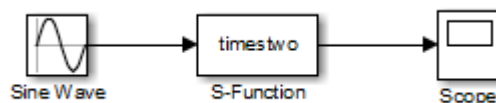
- 2 Copy the following C and TLC files for the `timestwo` S-function to your working folder:

- `matlabroot/toolbox/simulink/simdemos/simfeatures/src/timestwo.c`
- `matlabroot/toolbox/simulink/simdemos/simfeatures/tlc_c/timestwo.tlc`

- 3 Build the `timestwo` MEX-file in `c:/work/my_targetmodel`.

```
mex timestwo.c
```

- 4 Create the following model, using an S-Function block from the Simulink User-Defined Functions library. Save the model in your working folder as `targetmodel`.



- 5 Double-click the S-Function block to open the Block Parameters dialog box. Enter the S-function name `timestwo`. Click **OK**. The block is now bound to the `timestwo` MEX-file.

- 6 Open Model Explorer or the Configuration Parameters dialog box and select the **Solver** pane.
- 7 Set the solver **Type** to `fixed-step` and click **Apply**.
- 8 Save the model.
- 9 Open the scope and run a simulation. Verify that the `timestwo` S-function multiplies its input by 2.0.

Keep the `targetmodel` model open for use in the next section, in which you generate code using the test model.

Verify Target Operation

In this section you configure `targetmodel` for the `my_ert_target` custom target, and use the target to generate code and build an executable:

- 1 In the Configuration Parameters dialog box, select the **Code Generation** pane.
- 2 Click **Browse** to open the System Target File Browser.
- 3 In the Browser, select `My ERT-based Target` and click **OK**.
- 4 The Configuration Parameters dialog box now displays the **Code Generation** pane for `my_ert_target`.
- 5 Select the **Code Generation > Report** pane and select the **Create code generation report** option.
- 6 Click **Apply** and save the model. The model is configured for `my_ert_target`.
- 7 Build the model. If the build succeeds, the MATLAB Command Window displays the message below.

```
### Created executable: ../targetmodel.exe
### Successful completion of build procedure for model:
targetmodel
```

Your working folder contains the `targetmodel.exe` file and the build folder, `targetmodel_my_ert_target_rtw`, which contains generated code and other files. The working folder also contains an `s\prj` folder, used internally by the build process.

The code generator also creates a code generation report.

- 8 To view the generated code, on the **C Code** tab, click **View Code**. In the **Contents** pane of the code generation report, click the `targetmodel.c` link.
- 9 In `targetmodel.c`, locate the model step function, `targetmodel_step`. Observe the following code.

```
/* S-Function Block: <Root>/S-Function */
/* Multiply input by two */
targetmodel_B.SFunction = targetmodel_B.SineWave * 2.0;
```

The presence of this code confirms that the `my_ert_target` custom target has generated an inlined output computation for the S-Function block in the model.

See Also

More About

- “Custom Targets” on page 46-2

- “Support Toolchain Approach with Custom Target” on page 46-59
- “Support Model Referencing” on page 46-61
- “Support Compiler Optimization Level Control” on page 46-65
- “Support C Function Prototype Control” on page 46-67
- “Support C++ Class Interface Control” on page 46-69
- “Support Concurrent Execution of Multiple Tasks” on page 46-71

Customize Template Makefiles

To configure or customize a template makefile (TMF), you should be familiar with how the `make` command works and how it processes a makefile (.mk file). You should also understand makefile build rules. For information on these topics, refer to the documentation provided with the `make` utility that you use.

Template Makefiles and Tokens

A template makefile contains tokens. The build process expands the tokens and creates makefiles:

- `model.mk` -- Compiles and links code generated from model components.
- `rtwshared.mk` -- Compiles generated shared utility code.

The makefiles (`model_or_sharedutils.mk`) use commands that are specific to your development computer.

Template Makefile Tokens

The `make_rtw` command (or a different command provided with some targets) directs the process of generating `model_or_sharedutils.mk`. The `make_rtw` command processes the TMF specified on the **Code Generation** pane of the Configuration Parameters dialog box. `make_rtw` copies the TMF, line by line, expanding each token encountered. Template Makefile Tokens Expanded by `make_rtw` lists the tokens and their expansions.

These tokens are used in several ways by the expanded makefile:

- To control the conditional behavior in the makefile. The conditionals are used to control the source file lists, library names, target to be built, and other build-related information.
- To provide the macro definitions for compiling the files, for example, `-DINTEGER_CODE=1`.

Template Makefile Tokens Expanded by make_rtw

Token	Expansion
General purpose	
>ADDITIONAL_LDFLAGS<	Linker flags automatically added by blocks.
>ALT_MATLAB_BIN<	Alternate full pathname for the MATLAB executable; value is different than value for MATLAB_BIN token when the full pathname contains spaces.
>ALT_MATLAB_ROOT<	Alternate full pathname for the MATLAB installation; value is different than value for MATLAB_ROOT token when the full pathname contains spaces.
>BUILDARGS<	Options passed to make_rtw. This token is provided so that the contents of your <i>model_or_sharedutils.mk</i> file change when you change the build arguments, thus forcing an update of modules when your build options change.
>COMBINE_OUTPUT_UPDATE_FCNS<	True (1) when you select model configuration parameter Single output/update function , otherwise False (0). Used for the macro definition <code>-DONESTEPFCN=1</code> .
>COMPILE_FLAGS_OTHER<	<p>Compiler flags in a group other than 'OPTS', 'OPT_OPTS', and 'OPTIMIZATION_FLAGS'. The build process produces a warning if >COMPILE_FLAGS_OTHER< is absent and these conditions apply:</p> <ul style="list-style-type: none"> • The build information contains a compiler flag that is not in 'OPTS', 'OPT_OPTS', or 'OPTIMIZATION_FLAGS'. • The compiler flag is not in the template makefile.
>COMPUTER<	Computer type. See the MATLAB computer command.

Token	Expansion
>DEFINES_OTHER<	<p>Preprocessor macro definitions in a group other than 'OPTS', 'OPT_OPTS', 'OPTIMIZATION_FLAGS', and 'Custom'.</p> <p>A preprocessor macro definition in 'OPTS', 'OPT_OPTS', 'OPTIMIZATION_FLAGS', or 'Custom' that has a corresponding variable/token pair in the template makefile is not expanded in <code>DEFINES_OTHER</code>. This behavior helps to avoid the provision of duplicate preprocessor macro definitions.</p> <p>For example, some template makefiles contain these statements:</p> <pre>... NUMST = >NUMST< ... CPP_REQ_DEFINES1 = ... -DNUMST=\$(NUMST) ...</pre> <p>If the template makefile contains the variable/token pair <code>NUMST = >NUMST< </code> and <code>NUMST</code> is a preprocessor macro definition in <code>buildInfo</code>, then the build process does not expand it in <code>DEFINES_OTHER</code>.</p> <p>The build process produces a warning if <code> >DEFINES_OTHER< </code> is absent and these conditions apply:</p> <ul style="list-style-type: none"> • The build information contains a preprocessor macro definition that is not in 'OPTS', 'OPT_OPTS', 'OPTIMIZATION_FLAGS', or 'Custom'. • The preprocessor macro definition, for example, <code>NUMST</code>, is not in the template makefile.
>EXPAND_LIBRARY_LOCATION<	<p>Location of precompiled library file. The <code>TargetPreCompLibLocation</code> configuration parameter can override this setting. For examples, see “Control Library Location and Naming During Build” on page 44-3.</p>
>EXPAND_LIBRARY_NAME<	<p>Library name. For examples, see “Control Library Location and Naming During Build” on page 44-3 and “Modify the Template Makefile for <code>rtwmakecfg</code>” on page 44-19.</p>
>EXPAND_LIBRARY_SUFFIX<	<p>Library suffix. The <code>TargetLibSuffix</code> configuration parameter can override this setting. For examples, see “Control Library Location and Naming During Build” on page 44-3.</p>
>EXT_MODE< (Not required for R2018a and later provided template makefile specifies <code>TOOLCHAIN_NAME</code> on page 31-26)	<p>True (1) to enable generation of external mode support code, otherwise False (0).</p>
>EXTMODE_TRANSPORT< (Not required for R2018a and later provided template makefile specifies <code>TOOLCHAIN_NAME</code> on page 31-26)	<p>Index of transport mechanism (for example, <code>tcpip</code>, <code>serial</code>) for external mode.</p>

Token	Expansion
>EXTMODE_STATIC< (Not required for R2018a and later provided template makefile specifies TOOLCHAIN_NAME on page 31-26)	True (1) if static memory allocation is selected for external mode. False (0) if dynamic memory allocation is selected.
>EXTMODE_STATIC_SIZE< (Not required for R2018a and later provided template makefile specifies TOOLCHAIN_NAME on page 31-26)	Size of static memory allocation buffer for external mode.
>GENERATE_ERT_S_FUNCTION<	True (1) when model configuration parameter Create block is set to SIL, otherwise False (0). Used for control of the makefile target of the build.
>INCLUDE_MDL_TERMINATE_FCN<	True (1) when model configuration parameter Terminate function required is selected, otherwise False (0). Used for the macro definition -DTERMFCN==1.
>INTEGER_CODE<	True (1) when model configuration parameter Support floating-point numbers is cleared, otherwise False (0). INTEGER_CODE is a required macro definition when compiling the source code and is used when selecting precompiled libraries to link against.
>MAKEFILE_NAME<	<i>model_or_sharedutils.mk</i> — The name of the makefile that was created from the TMF.
>MAT_FILE<	True (1) when model configuration parameter MAT-file logging is selected, otherwise False (0). MAT_FILE is a required macro definition when compiling the source code and also is used to include logging code in the build process.
>MATLAB_BIN<	Location of the MATLAB executable program.
>MATLAB_ROOT<	Path to where MATLAB is installed.
>MEM_ALLOC<	RT_MALLOC or RT_STATIC. Indicates how memory is to be allocated.
>MEXEXT<	MEX-file extension. See the MATLAB mexext command.
>MODEL_MODULES<	Additional generated source modules. For example, you can split a large model into two files, <i>model.c</i> and <i>model1.c</i> . In this case, this token expands to <i>model1.c</i> .
>MODEL_MODULES_OBJ<	Object filenames (.obj) corresponding to additional generated source modules.
>MODEL_NAME<	Name of the Simulink block diagram currently being built.
>MULTITASKING<	True (1) if solver mode is multitasking, otherwise False (0).
>NCSTATES<	Number of continuous states.
>NUMST<	Number of sample times in the model.
>RELEASE_VERSION<	The MATLAB release version.
>S_FUNCTIONS_LIB<	List of S-function libraries available for linking.
>SOLVER<	Solver source filename, for example, <i>ode3.c</i> .
>SOLVER_OBJ<	Solver object (.obj) filename, for example, <i>ode3.obj</i> .
>START_DIR<	Code generation folder at the start of the build.

Token	Expansion
>TARGET_LANG_EXT<	c when model configuration parameter Language is set to C, cpp when Language is set to C++. Used in the makefile to control the extension on generated source files.
>TID01EQ<	True (1) if sampling rates of the continuous task and the first discrete task are equal, otherwise False (0).
S-function and build information support	
Note For examples of the tokens in this section, see “Modify the Template Makefile for rtwmakecfg” on page 44-19.	
>START_EXPAND_INCLUDES< >EXPAND_DIR_NAME< >END_EXPAND_INCLUDES<	List of folder names to add to the include path. Additionally, the ADD_INCLUDES macro must be added to the INCLUDES line.
>START_EXPAND_LIBRARIES< >EXPAND_LIBRARY_NAME< >END_EXPAND_LIBRARIES<	List of library names.
>START_EXPAND_MODULES< >EXPAND_MODULE_NAME< >END_EXPAND_MODULES<	Library module names within >START_EXPAND_LIBRARIES< and >START_PRECOMP_LIBRARIES< library lists.
>START_EXPAND_RULES< >EXPAND_DIR_NAME< >END_EXPAND_RULES<	Makefile rules.
>START_PRECOMP_LIBRARIES< >EXPAND_LIBRARY_NAME< >END_PRECOMP_LIBRARIES<	List of precompiled library names.
Model reference support	
Note For examples of the tokens in this section, see “Providing Model Referencing Support in the TMF” on page 46-62.	
>MODELLIB<	Name of the library file generated for the current model.
>MODELREFS<	List of models referenced by the top model.
>MODELREF_LINK_LIBS<	List of referenced model libraries against which the top model links.
>MODELREF_LINK_RSPFILE_NAME<	Name of a response file against which the top model links. This token is valid only for build environments that support linker response files. For an example of its use, see <i>matlabroot/toolbox/coder/compile/tmf/grt_vcx64.tmf</i> .
>MODELREF_TARGET_TYPE<	Type of target being built. Possible values are <ul style="list-style-type: none"> • NONE: Standalone model or top model referencing other models • RTW: Model reference target build • SIM: Model reference simulation target build

Token	Expansion
>RELATIVE_PATH_TO_ANCHOR<	Relative path, from the location of the generated makefile, to the MATLAB working folder.

These tokens are expanded by substitution of parameter values known to the build process. For example, if the source model contains blocks with two different sample times, the TMF statement

```
NUMST = |>NUMST<|
```

expands to:

```
NUMST = 2
```

In addition to the above, `make_rtw` expands tokens from other sources:

- Target-specific tokens defined in the target options of the Configuration Parameters dialog box
- Structures in the `rtwoptions` section of the system target file. Structures in the `rtwoptions` structure array that contain the field `makevariable` are expanded.

The following example is extracted from `matlabroot/rtw/c/grt/grt.tlc`. The section starting with `BEGIN_RTW_OPTIONS` contains MATLAB code that sets up `rtwoptions`. The following directive causes the `|>EXT_MODE<|` token to be expanded to 1 (on) or 0 (off), depending on how you set the external mode options.

```
rtwoptions(2).makevariable = 'EXT_MODE'
```

Invoke the make Utility

- “make Command” on page 46-49
- “make Utility Versions” on page 46-50

make Command

After creating `model_or_sharedutils.mk` from your TMF, the build process invokes a `make` command. To invoke `make`, the build process issues this command.

```
makecommand -f model_or_sharedutils.mk
```

`makecommand` is defined by the `MAKECMD` macro in your system target file TMF (see “Structure of the Template Makefile” on page 46-50). You can specify additional options to `make` by using the model configuration parameter **Make command**. (See the sections “Specify a Make Command” on page 31-25 and “Template Makefiles and Make Options” on page 31-36.)

For example, specifying `OPT_OPTS=-02` in the **Make command** field causes `make_rtw` to generate the following `make` command.

```
makecommand -f model_or_sharedutils.mk OPT_OPTS=-02
```

A comment at the top of the TMF specifies the available `make` command options. If these options do not provide you with enough flexibility, you can configure your own TMF.

make Utility Versions

The make utility lets you control nearly every aspect of building your real-time program. There are several different versions of make available. The code generator provides the Free Software Foundation GNU make for both UNIX⁶ and PC platforms in platform-specific subfolders under

matlabroot/bin

It is possible to use other versions of make with the code generator, although GNU Make is recommended. To be compatible with the code generator, verify that your version of make supports the following command format.

```
makecommand -f model_or_sharedutils.mk
```

Structure of the Template Makefile

A TMF has multiple sections, including the following:

- **Abstract** — Describes what the makefile targets. Here is a representative abstract from the ERT TMFs in *matlabroot/toolbox/coder/compile/tmf*:

```
# Abstract:
#   Template makefile for building a Windows-based stand-alone embedded
#   real-time version of Simulink model using generated C code and the
#   Microsoft Visual C/C++ compiler for x64.
#
#   Note that this template is automatically customized by the build
#   procedure to create "<model>.mk"
#
#   The following defines can be used to modify the behavior of the
#   build:
#   OPT_OPTS      - Optimization option. See DEFAULT_OPT_OPTS in
#                   vctools.mak for default.
#   OPTS          - User specific options.
#   CPP_OPTS      - C++ compiler options.
#   USER_SRCS     - Additional user sources, such as files needed by
#                   S-functions.
#   USER_INCLUDES - Additional include paths
#                   (i.e. USER_INCLUDES="-Iwhere-ever -Iwhere-ever2")
#
#   To enable debugging:
#   set DEBUG_BUILD = 1, which will trigger OPTS=-Zi (may vary with
#                   compiler version, see compiler doc)
#
#   This template makefile is designed to be used with a system target
#   file that contains 'rtwgensettings.BuildDirSuffix' see ert.tlc
```

- **Macros read by make_rtw section** — Defines macros that tell make_rtw how to process the TMF. Here is a representative Macros read by make_rtw section from the GRT TMFs in *matlabroot/toolbox/coder/compile/tmf*:

```
#----- Macros read by make_rtw -----
#
# The following macros are read by the build procedure:
#
# MAKECMD      - This is the command used to invoke the make utility
# HOST         - What platform this template makefile is targeted for
#               (i.e. PC or UNIX)
# BUILD        - Invoke make from the build procedure (yes/no)?
# SYS_TARGET_FILE - Name of system target file.
#
# MAKECMD      = nmake
# HOST         = PC
```

6 UNIX is a registered trademark of The Open Group in the United States and other countries.


```

BUILD          = yes
SYS_TARGET_FILE = any
BUILD_SUCCESS  = ^#^#^# Created

# Opt in to simplified format by specifying compatible Toolchain
TOOLCHAIN_NAME = [ \
  "Microsoft Visual C++ 2019 v16.0 | nmake (64-bit Windows)", \
  "Microsoft Visual C++ 2017 v15.0 | nmake (64-bit Windows)", \
  "Microsoft Visual C++ 2015 v14.0 | nmake (64-bit Windows)" ]

```

The macros in this section might include:

- **MAKECMD** — Specifies the command used to invoke the make utility. For example, if **MAKECMD** = **mymake**, then the make command invoked is

```
mymake -f model_or_sharedutils.mk
```

- **HOST** — Specifies the platform targeted by this TMF. This can be **PC**, **UNIX**, *computer_name* (see the **MATLAB computer** command), or **ANY**.
- **BUILD** — Instructs **make_rtw** whether or not it should invoke make from the build procedure. Specify **yes** or **no**.
- **SYS_TARGET_FILE** — Specifies the name of the system target file or the value **any**. This is used for consistency checking by **make_rtw** to verify the system target file specified in the **Target selection** panel of the **Code Generation** pane of the Configuration Parameters dialog box. If you specify **any**, you can use the TMF with any system target file.
- **BUILD_SUCCESS** — When the makefile creates a static library, it produces the **BUILD_SUCCESS** string as defined, for example, by this rule in **ert_vcx64.tmf**:

```

$(PRODUCT) : $(OBJS) $(LIBS) $(MODELREF_LINK_LIBS)
  @cmd /C "echo ### Linking ..."
  $(LD) $(LDFLAGS) $(LIBS) \
  @$ (CMD_FILE) @$ (MODELREF_LINK_RSPFILE) -dll -def:$(MODEL).def -out:$@
  @cmd /C "echo $(BUILD_SUCCESS) dynamically linked library $(PRODUCT)"

```

When the build process finds the **BUILD_SUCCESS** string, it polls for the existence of the updated static library file. The build process does not link the library file to the executable file until the build process can confirm the presence of the library file, with an updated timestamp, on the file system.

If the build process does not find **BUILD_SUCCESS** in the compilation log, then the build process assumes that the updated library archive is available on the file system. If required, the build process links the library to the executable file.

- **Tokens expanded by make_rtw section** — Defines the tokens that **make_rtw** expands. Here is a brief excerpt from a representative **Tokens expanded by make_rtw section** from the ERT TMFs in *matlabroot/toolbox/coder/compile/tmf/*:

```

#----- Tokens expanded by make_rtw -----
#
# The following tokens, when wrapped with "|>" and "<|" are expanded by the
# build procedure.
#
# MODEL_NAME          - Name of the Simulink block diagram
# MODEL_MODULES       - Any additional generated source modules
# MAKEFILE_NAME       - Name of makefile created from template makefile <model>.mk
# MATLAB_ROOT         - Path to where MATLAB is installed.
...

MODEL                 = |>MODEL_NAME<|
MODULES               = |>MODEL_MODULES<|
PRODUCT               = |>PRODUCT<|
MAKEFILE              = |>MAKEFILE_NAME<|

```

```
MATLAB_ROOT      = |>MATLAB_ROOT<|
...
```

For more information about TMF tokens, see `Template Makefile Tokens Expanded by make_rtw`.

- Subsequent sections vary based on compiler, host, and target. Some common sections include `Model` and `reference models`, `External mode`, `Tool Specifications` or `Tool Definitions`, `Include Path`, `C Flags`, `Additional Libraries`, and `Source Files`.
- `Rules` section — Contains the make rules used in building an executable from the generated source code. The build rules are typically specific to your version of make. The `Rules` section might be followed by related sections such as `Dependencies`.

Customize and Create Template Makefiles

- “Introduction” on page 46-52
- “Setting Up a Template Makefile” on page 46-52
- “Using Macros and Pattern Matching Expressions in a Template Makefile” on page 46-52
- “Customizing Generated Makefiles with `rtwmakecfg`” on page 46-54
- “Supporting Continuous Time in Custom Targets” on page 46-54
- “Model Reference Considerations” on page 46-55
- “Timeout Error When Building Shared Utility Code with Custom Template Makefile” on page 46-55

Introduction

This section describes the mechanics of setting up a custom template makefile (TMF) and incorporating it into the build process. It also discusses techniques for modifying a TMF and MATLAB file mechanisms associated with the TMF.

Before creating a custom TMF, you should read “Folder and File Naming Conventions” on page 46-6 to understand the folder structure and MATLAB path requirements for custom targets.

Setting Up a Template Makefile

To customize or create a new TMF, copy an existing GRT or ERT TMF from `matlabroot/toolbox/coder/compile/tmf`.

Place the copy in the same folder as the associated system target file. Usually, this is the `mytarget/mytarget` folder within the target folder structure. Then, rename your TMF (for example, `mytarget.tmf`) and modify it.

To allow the build process to locate and select your TMF, you must provide information in the system target file file header (see “System Target File Structure” on page 46-19). For a target that implements a single TMF, the standard way to specify the TMF to be used in the build process is to use the `TMF` directive of the system target file file header.

```
TMF: mytarget.tmf
```

Using Macros and Pattern Matching Expressions in a Template Makefile

This section shows, through an example, how to use macros and file-pattern-matching expressions in a TMF to generate commands in `model_or_sharedutils.mk`.

The `make` utility processes `model_or_sharedutils.mk` and generates a set of commands based upon dependency rules defined in `model_or_sharedutils.mk`. After `make` generates the set of commands for building or rebuilding `test`, `make` executes them.

For example, to build a program called `test`, `make` must link the object files. However, if the object files don't exist or are out of date, `make` must compile the source code. Thus there is a dependency between source and object files.

Each version of `make` differs slightly in its features and how rules are defined. For example, consider a program called `test` that gets created from two sources, `file1.c` and `file2.c`. Using most versions of `make`, the dependency rules would be

```
test: file1.o file2.o
    cc -o test file1.o file2.o

file1.o: file1.c
    cc -c file1.c

file2.o: file2.c
    cc -c file2.c
```

In this example, a UNIX⁷ environment is assumed. In a PC environment the file extensions and compile and link commands are different.

In processing the first rule

```
test: file1.o file2.o
```

`make` sees that to build `test`, it needs to build `file1.o` and `file2.o`. To build `file1.o`, `make` processes the rule

```
file1.o: file1.c
```

If `file1.o` doesn't exist, or if `file1.o` is older than `file1.c`, `make` compiles `file1.c`.

The format of TMFs follows the above example. Our TMFs use additional features of `make` such as macros and file-pattern-matching expressions. In most versions of `make`, a macro is defined with

```
MACRO_NAME = value
```

References to macros are made with `$(MACRO_NAME)`. When `make` sees this form of expression, it substitutes `value` for `$(MACRO_NAME)`.

You can use pattern matching expressions to make the dependency rules more general. For example, using GNU⁸ `Make`, you could replace the two "`file1.o: file1.c`" and "`file2.o: file2.c`" rules with the single rule

```
%.o : %.c
    cc -c $<
```

Note that `$<` in the previous example is a special macro that equates to the dependency file (that is, `file1.c` or `file2.c`). Thus, using macros and the "%" pattern matching character, the previous example can be reduced to

7 UNIX is a registered trademark of The Open Group in the United States and other countries.

8 GNU is a registered trademark of the Free Software Foundation.

```
SRCS = file1.c file2.c
OBJS = $(SRCS:.c=.o)

test: $(OBJS)
    cc -o $@ $(OBJS)

%.o : %.c
    cc -c $<
```

Note that the `$@` macro above is another special macro that equates to the name of the current dependency target, in this case `test`.

This example generates the list of objects (`OBJS`) from the list of sources (`SRCS`) by using the text substitution feature for macro expansion. It replaces the source file extension (for example, `.c`) with the object file extension (`.o`). This example also generalized the build rule for the program, `test`, to use the special "`$@`" macro.

Customizing Generated Makefiles with `rtwmakecfg`

TMFs provide tokens that let you add the following items to generated makefiles:

- Source folders
- Include folders
- Run-time library names
- Run-time module objects

S-functions can add this information to the makefile by using an `rtwmakecfg.m` file function. This function is particularly useful when building a model that contains one or more of your S-Function blocks, such as device driver blocks.

To add information pertaining to an S-function to the makefile,

- 1 Create the function `rtwmakecfg` in file `rtwmakecfg.m`. The code generator associates this file with your S-function based on its folder location.
- 2 Modify your target's TMF such that it supports macro expansion for the information returned by `rtwmakecfg` functions.

After the TLC phase of the build process, when generating a makefile from the TMF, the build process searches for an `rtwmakecfg.m` file in the folder that contains the S-function component. If it finds the file, the build process calls the `rtwmakecfg` function. For more information, see "Use `rtwmakecfg.m` API to Customize Generated Makefiles" on page 44-17.

Supporting Continuous Time in Custom Targets

If you want your custom ERT-based target to support continuous time, you must update your template makefile (TMF) and the static main program module (for example, `mytarget_main.c`) for your target.

Template Makefile Modifications

Add the `NCSTATES` token expansion after the `NUMST` token expansion, as follows:

```
NUMST = |>NUMST<|
NCSTATES = |>NCSTATES<|
```

In addition, add NCSTATES to the CPP_REQ_DEFINES macro, as in the following example:

```
CPP_REQ_DEFINES = -DMODEL=$(MODEL) -DNUMST=$(NUMST) -DNCSTATES=$(NCSTATES) \
-DMAT_FILE=$(MAT_FILE)
-DINTEGER_CODE=$(INTEGER_CODE) \
-DONESTEPFCN=$(ONESTEPFCN) -DTERMFCN=$(TERMFCN) \
-DHAVESTDIO
-DMULTI_INSTANCE_CODE=$(MULTI_INSTANCE_CODE) \
```

Modifications to Main Program Module

The main program module defines a static main function that manages task scheduling for the supported tasking modes of single- and multiple-rate models. NUMST (the number of sample times in the model) determines whether the main function calls multirate or single-rate code. However, when a model uses continuous time, do not rely on NUMST directly.

When the model has continuous time and the flag TID01EQ is true, both continuous time and the fastest discrete time are treated as one rate in generated code. The code associated with the fastest discrete rate is guarded by a major time step check. When the model has only two rates, and TID01EQ is true, the generated code has a single-rate call interface.

To support models that have continuous time, update the static main module to take TID01EQ into account, as follows:

- 1 Before NUMST is referenced in the file, add the following code:

```
#if defined(TID01EQ) && TID01EQ == 1 && NCSTATES == 0
#define DISC_NUMST (NUMST - 1)
#else
#define DISC_NUMST NUMST
#endif
```

- 2 Replace instances of NUMST in the file by DISC_NUMST.

Model Reference Considerations

See “Support Model Referencing” on page 46-61 for important information on TMF modifications you may need to make to support the code generator model referencing features.

Note If you are using a TMF without the variable MODELREFS, the file might have been used with a previous release of Simulink software. If you want your TMF to support model referencing, add MODELREFS to the make file.

Timeout Error When Building Shared Utility Code with Custom Template Makefile

If the build process uses a custom template makefile to create a makefile for the shared utility code, the build process produces a timeout error when all of these conditions apply:

- The configuration parameter GenCodeOnly is 'off'.
- The BUILD_SUCCESS message is output during the make call to rtwshared.mk.
- The make call to rtwshared.mk does not update the final product (typically rtwshared.lib).

To avoid the error, update the custom template makefile so that the make call generates the BUILD_SUCCESS message only when the final product is generated (as identified by the make variable PRODUCT). Alternatively, if the build process does not use the generated makefile, disable makefile generation by setting the configuration parameter GenerateMakefile to 'off'.

See Also

More About

- “Custom Targets” on page 46-2
- “Sample Custom Targets” on page 46-4
- “Customize System Target Files” on page 46-19
- “Custom Target Optional Features” on page 46-57
- “Support Toolchain Approach with Custom Target” on page 46-59

Custom Target Optional Features

This section describes how to configure a custom embedded target to support these optional features.

To ...	Use Target Configuration Parameters ...	For more information, see ...
Indicate a custom target is toolchain-compliant	UseToolchainInfoCompliant GenerateMakefile	“Support Toolchain Approach with Custom Target” on page 46-59
Build a model that includes referenced models and uses a custom target	ModelReferenceCompliant ParMdlRefBuildCompliant (parallel build support)	“Support Model Referencing” on page 46-61
Control the compiler optimization level building generated code for a custom target	CompOptLevelCompliant	“Support Compiler Optimization Level Control” on page 46-65
Control C function prototypes of initialize and step functions that are generated for a model that uses a custom target	ModelStepFunctionPrototypeControlCompliant (ERT only)	“Support C Function Prototype Control” on page 46-67
Control C++ class interfaces that are generated for a model that uses a custom target	CPPClassGenCompliant (ERT only)	“Support C++ Class Interface Control” on page 46-69
Enable concurrent execution of multiple tasks on a multicore platform for a model that uses a custom target	ConcurrentExecutionCompliant	“Support Concurrent Execution of Multiple Tasks” on page 46-71

The required configuration changes are modifications to your system target file, and in some cases also modifications to your template makefile (TMF) or your custom static main program.

The API for system target file callbacks provides a function `SelectCallback` for use in system target files. `SelectCallback` is associated with the target rather than with its individual options. If you implement a `SelectCallback` function for a target, it is triggered whenever the user selects the target in the System Target File Browser.

The API provides the functions `slConfigUIGetVal`, `slConfigUISetEnabled`, and `slConfigUISetVal` for controlling custom target configuration options from a user-written `SelectCallback` function. (For function descriptions and examples, see the function reference pages.)

The general requirements for supporting one of the optional features include:

- To support model referencing or compiler optimization level control, the target must be derived from the GRT or the ERT target. To support C function prototype control or C++ class interface control, the target must be derived from the ERT target.
- The system target file must declare feature compliance by including one of the target configuration parameters listed above in a `SelectCallback` function call.

- Additional changes such as TMF modifications or static main program modifications may be required, depending on the feature. See the detailed steps in the subsections for individual features.

For an example that shows how to configure custom target optional features, see “Customize System Target Files” on page 46-19.

See Also

More About

- “Custom Targets” on page 46-2
- “Sample Custom Targets” on page 46-4
- “Support Toolchain Approach with Custom Target” on page 46-59
- “Support Model Referencing” on page 46-61
- “Support Compiler Optimization Level Control” on page 46-65
- “Support C Function Prototype Control” on page 46-67
- “Support C++ Class Interface Control” on page 46-69
- “Support Concurrent Execution of Multiple Tasks” on page 46-71

Support Toolchain Approach with Custom Target

This section describes how to configure a custom system target file to support builds with the toolchain approach.

In the Configuration Parameters dialog box, on the Code Generation pane of, you can set the build controls for the toolchain approach or the template makefile approach. The model configuration parameters that contribute to determining which build controls appear include these parameters.

Model Parameter	Value	Notes
UseToolchainInfoCompliant	on	For toolchain approach, set this parameter to 'on'. For TMF approach, set this parameter to 'off'.
GenerateMakefile	on	For toolchain approach, set this parameter to 'on'.

When the dialog box detects that the selected target has these properties, the dialog box recognizes the target as toolchain-compliant and displays the build controls for the toolchain approach.

Because the custom target file cannot set these properties directly, use a `SelectCallback` function in the custom target file to set the properties. The `SelectCallback` function call in the `RTW_OPTION` section of the TLC file can take the form:

```
rtwgensettings.SelectCallback = 'enableToolchainCompliant(hSrc, hDlg)';
```

A corresponding callback function can contain:

```
function enableToolchainCompliant(hSrc, hDlg)
% The following parameters enable toolchain compliance.
slConfigUISetVal(hDlg, hSrc, 'UseToolchainInfoCompliant', 'on');
slConfigUISetVal(hDlg, hSrc, 'GenerateMakefile', 'on');

% The following parameters are not required for toolchain compliance.
% But, it is recommended practice to set these default values and
% disable the parameters (as shown).
slConfigUISetVal(hDlg, hSrc, 'RTWCompilerOptimization', 'off');
slConfigUISetVal(hDlg, hSrc, 'MakeCommand', 'make_rtw');
slConfigUISetEnabled(hDlg, hSrc, 'RTWCompilerOptimization', false);
slConfigUISetEnabled(hDlg, hSrc, 'MakeCommand', false);
hCS = hSrc.getConfigSet();
hCS.refreshDialog;
end
```

When you select your custom target, the configuration parameters dialog box displays the toolchain approach build controls. For an example, see “Create a Custom Target Configuration” on page 46-33.

For an example that shows how to configure custom target optional features, see “Customize System Target Files” on page 46-19.

See Also

More About

- “Customize System Target Files” on page 46-19
- “Support Model Referencing” on page 46-61

- “Support Compiler Optimization Level Control” on page 46-65
- “Support C Function Prototype Control” on page 46-67
- “Support C++ Class Interface Control” on page 46-69
- “Support Concurrent Execution of Multiple Tasks” on page 46-71

Support Model Referencing

If you want to use a custom system target file for building a model that has referenced models, you must configure the custom system target file to support model referencing.

Requirements for Model Referencing with a Custom System Target File

To build a model that references other models:

- Use a custom system target file that is derived from a GRT or ERT system target file.
- The custom system target file must declare model reference compliance -- see “Declaring Model Referencing Compliance” on page 46-61.
- The template makefile must define some entities that support model referencing -- see “Providing Model Referencing Support in the TMF” on page 46-62.

Declaring Model Referencing Compliance

To declare model reference compliance for your target, you must implement a callback function that sets the `ModelReferenceCompliant` flag, and then install the callback function in the `SelectCallback` field of the `rtwgensettings` structure in your system target file. The callback function is triggered whenever the user selects the target in the System Target File Browser. For example, the following system target file code installs a `SelectCallback` function named `custom_select_callback_handler`:

```
rtwgensettings.SelectCallback = 'custom_select_callback_handler(hDlg,hSrc)';
```

The arguments to the `SelectCallback` function (`hDlg`, `hSrc`) are handles to private data used by the callback API functions. These handles are restricted to use in system target file callback functions. They should be passed in without alteration.

Your callback function should set the `ModelReferenceCompliant` flag as follows:

```
slConfigUISetVal(hDlg,hSrc,'ModelReferenceCompliant','on');
slConfigUISetEnabled(hDlg,hSrc,'ModelReferenceCompliant',false);
hSrc.refreshDialog;
```

If you might use the target to build models containing large model reference hierarchies, consider configuring the target to support parallel builds, as discussed in “Reduce Build Time for Referenced Models by Using Parallel Builds” on page 31-51.

To configure a target for parallel builds, your callback function must also set the `ParMdlRefBuildCompliant` flag as follows:

```
slConfigUISetVal(hDlg,hSrc,'ParMdlRefBuildCompliant','on');
slConfigUISetEnabled(hDlg,hSrc,'ParMdlRefBuildCompliant',false);
hSrc.refreshDialog;
```

For more information about the system target file callback API, see the `slConfigUIGetVal`, `slConfigUISetEnabled`, and `slConfigUISetVal` function reference pages.

Providing Model Referencing Support in the TMF

To configure the template makefile (TMF) for model referencing:

- 1 Add the following make variables and tokens to be expanded when the makefile is generated:

```
MODELREFS          = |>MODELREFS<|
MODELLIB           = |>MODELLIB<|
MODELREF_LINK_LIBS = |>MODELREF_LINK_LIBS<|
MODELREF_LINK_RSPFILE = |>MODELREF_LINK_RSPFILE_NAME<|
RELATIVE_PATH_TO_ANCHOR = |>RELATIVE_PATH_TO_ANCHOR<|
MODELREF_TARGET_TYPE = |>MODELREF_TARGET_TYPE<|
```

The following code excerpt shows how makefile tokens are expanded for a referenced model.

```
MODELREFS          =
MODELLIB           = engine3200cc_rtwlib.a
MODELREF_LINK_LIBS =
MODELREF_LINK_RSPFILE =
RELATIVE_PATH_TO_ANCHOR = ../../..
MODELREF_TARGET_TYPE = RTW
```

The following code excerpt shows how makefile tokens are expanded for the top model that references the referenced model.

```
MODELREFS          = engine3200cc transmission
MODELLIB           =
MODELREF_LINK_LIBS = engine3200cc_rtwlib.a transmission_rtwlib.a
MODELREF_LINK_RSPFILE =
RELATIVE_PATH_TO_ANCHOR = ..
MODELREF_TARGET_TYPE = NONE
```

Token	Expands to
MODELREFS for the top model	List of referenced model names.
MODELLIB	Name of the library generated for the model.
MODELREF_LINK_LIBS token for the top model	List of referenced model libraries that the top model links against.
MODELREF_LINK_RSPFILE token for the top model	Name of a response file that the top model links against. This token is valid only for build environments that support linker response files. For an example of its use, see <i>matlabroot/toolbox/coder/compile/tmf/grt_vcx64.tmf</i> .
RELATIVE_PATH_TO_ANCHOR	Relative path, from the location of the generated makefile, to the MATLAB working folder.
MODELREF_TARGET_TYPE	Signifies the type of target being built. Possible values are <ul style="list-style-type: none"> • NONE: Standalone model or top model referencing other models • RTW: Model reference target build • SIM: Model reference simulation target build

- 2 Add the RELATIVE_PATH_TO_ANCHOR include path to the overall INCLUDES variable.

```
INCLUDES = -I. -I$(RELATIVE_PATH_TO_ANCHOR) $(ADD_INCLUDES) \
$(USER_INCLUDES) $(SHARED_INCLUDES)
```

- 3 Change the SRCS variable in your TMF so that it initially lists only common modules. Additional modules are then appended conditionally, as described in the next step. For example, change

```
SRCS = $(MODEL).c $(MODULES) ert_main.c $(ADD_SRCS) $(EXT_SRC)
```

to

```
SRCS = $(MODULES)
```

- 4 Create variables to define the final target of the makefile. You can remove variables that may have existed for defining the final target. For example, remove

```
PROGRAM = ../$(MODEL)
```

and replace it with

```
ifeq ($(MODELREF_TARGET_TYPE), NONE)
# Top model for RTW
PRODUCT = $(RELATIVE_PATH_TO_ANCHOR)/$(MODEL)
BIN_SETTING = $(LD) $(LDFLAGS) -o $(PRODUCT) $(SYSLIBS)
BUILD_PRODUCT_TYPE = "executable"
# ERT based targets
SRCS += $(MODEL).c ert_main.c $(EXT_SRC)
# GRT based targets
# SRCS += $(MODEL).c grt_main.c rt_sim.c $(EXT_SRC) $(SOLVER)
else
# sub-model for RTW
PRODUCT = $(MODELLIB)
BUILD_PRODUCT_TYPE = "library"
endif
```

Note If the template makefile is associated with a toolchain, remove `$(MODEL).c` or `$(MODEL).$(TARGET_LANG_EXT)` from the SRCS list.

- 5 Create rules for the final target of the makefile (replace existing final target rules). For example:

```
ifeq ($(MODELREF_TARGET_TYPE), NONE)
# Top model for RTW
$(PRODUCT) : $(OBJS) $(LIBS) $(MODELREF_LINK_LIBS)
              $(BIN_SETTING) $(LINK_OBJS) $(MODELREF_LINK_LIBS)
              $(LIBS)
              @echo "### Created $(BUILD_PRODUCT_TYPE): $@"
else
# sub-model for RTW
$(PRODUCT) : $(OBJS) $(LIBS)
              @rm -f $(MODELLIB)
              $(ar) ruvs $(MODELLIB) $(LINK_OBJS)
              @echo "### Created $(MODELLIB)"
              @echo "### Created $(BUILD_PRODUCT_TYPE): $@"
endif
```

- 6 Create a rule to allow referenced models to compile files that reside in the MATLAB working folder (`pwd`).

```
%.o : $(RELATIVE_PATH_TO_ANCHOR)/%.c
      $(CC) -c $(CFLAGS) $<
```

Note If you are using a TMF without the variable `MODELREFS`, the file might have been used with a previous release of Simulink software. If you want your TMF to support model referencing, add either variable `MODELREFS` to the make file.

Controlling Configuration Option Value Agreement

By default, the value of a configuration option defined in the system target file for a TLC-based custom target must be the same in a referenced model and its parent model. To relax this

requirement, include the `modelReferenceParameterCheck` field in the `rtwoptions` structure element that defines the configuration option, and set the value of the field to `'off'`. For example:

```
rtwoptions(2).prompt      = 'My Custom Parameter';
rtwoptions(2).type       = 'Checkbox';
rtwoptions(2).default    = 'on';
rtwoptions(2).modelReferenceParameterCheck = 'on';
rtwoptions(2).tlcvariable = 'mytlcvariable';
...
```

The configuration option **My Custom Parameter** can differ in a referenced model and its parent model. See “Customize System Target Files” on page 46-19 for information about TLC-based system target files, and `rtwoptions` Structure Fields Summary for a list of `rtwoptions` fields.

Preventing Resource Conflicts (Optional)

Hook files are optional TLC and MATLAB program files that are invoked at well-defined stages of the build process. Hook files let you customize the build process and communicate information between various phases of the process.

If you are adapting your custom target for code generation compatibility with model reference features, consider adding checks to your hook files for handling referenced models differently than top models to prevent resource conflicts.

For example, consider adding the following check to your `STF_make_rtw_hook.m` file:

```
% Check if this is a referenced model
mdlRefTargetType = get_param(codeGenModelName, 'ModelReferenceTargetType');
isNotModelRefTarget = strcmp(mdlRefTargetType, 'NONE'); % NONE, SIM, or RTW
if isNotModelRefTarget
    % code that is specific to the top model
else
    % code that is specific to the referenced model
end
```

You may need to do a similar check in your TLC code.

```
%if !IsModelReferenceTarget()
    %% code that is specific to the top model
%else
    %% code that is specific to the referenced model
%endif
```

See Also

More About

- “Custom Targets” on page 46-2
- “Sample Custom Targets” on page 46-4
- “Customize System Target Files” on page 46-19
- “Model Reference Basics”
- “Generate Code for Model Reference Hierarchy” on page 8-2

Support Compiler Optimization Level Control

This section describes how to configure a custom embedded target to support compiler optimization level control. Without the described modifications, you cannot use model configuration parameter **Compiler optimization level** to control the compiler optimization level for building generated code. For more information about compiler optimization level control, see [Compiler optimization level](#).

About Compiler Optimization Level Control and Custom Targets

The requirements for supporting compiler optimization level control are as follows:

- The target must be derived from the GRT target or the ERT target.
- The system target file must declare compiler optimization level control compliance, as described in [“Declaring Compiler Optimization Level Control Compliance”](#) on page 46-65.
- The target makefile must honor the setting for model configuration parameter **Compiler optimization level**, as described in [“Providing Compiler Optimization Level Control Support in the Target Makefile”](#) on page 46-66.

For an example that shows how to configure custom target optional features, see [“Customize System Target Files”](#) on page 46-19.

Declaring Compiler Optimization Level Control Compliance

To declare compiler optimization level control compliance for your target, you must implement a callback function that sets the `CompOptLevelCompliant` flag, and then install the callback function in the `SelectCallback` field of the `rtwgensettings` structure in your system target file. The callback function is triggered whenever the user selects the target in the System Target File Browser. For example, the following system target file code installs a `SelectCallback` function named `custom_select_callback_handler`:

```
rtwgensettings.SelectCallback = 'custom_select_callback_handler(hDlg,hSrc)';
```

The arguments to the `SelectCallback` function (`hDlg`, `hSrc`) are handles to private data used by the callback API functions. These handles are restricted to use in system target file callback functions. They should be passed in without alteration.

Your callback function should set the `CompOptLevelCompliant` flag as follows:

```
slConfigUISetVal(hDlg,hSrc,'CompOptLevelCompliant','on');
slConfigUISetEnabled(hDlg,hSrc,'CompOptLevelCompliant',false);
hSrc.refreshDialog;
```

For more information about the system target file callback API, see the `slConfigUIGetVal`, `slConfigUISetEnabled`, and `slConfigUISetVal` function reference pages.

When the `CompOptLevelCompliant` target configuration parameter is set to on, the **Compiler optimization level** parameter is displayed in the **Code Generation** pane of the Configuration Parameters dialog box for your model.

Providing Compiler Optimization Level Control Support in the Target Makefile

As part of supporting compiler optimization level control for your target, you must modify the target makefile to honor the setting for model configuration parameter **Compiler optimization level**. Use a GRT or ERT target provided by MathWorks as a model for making the modifications.

See Also

More About

- “Custom Targets” on page 46-2
- “Sample Custom Targets” on page 46-4
- “Customize System Target Files” on page 46-19

Support C Function Prototype Control

This section describes how to configure a custom embedded target to support function prototype control. Without the described modifications, you will not be able to use available interfaces for customizing entry-point function interfaces.

Function prototype control is not supported for component models configured with a service interface. For more information, see “Model Code Interfaces” (Embedded Coder) and “Configure Code Interfaces for Code Generation” (Embedded Coder).

For more information, see “Configure Generated C Function Interface for Model Entry-Point Functions” (Embedded Coder).

About C Function Prototype Control and Custom Targets

The requirements for supporting C function prototype control are as follows:

- The system target file must be derived from the ERT system target file.
- The system target file must declare C function prototype control compliance, as described in “Declaring C Function Prototype Control Compliance” on page 46-67.
- If your system target file uses a custom static main program, and if a nondefault function prototype control configuration is associated with a model, the static main program must call the function prototype controlled initialize and step functions, as described in “Providing C Function Prototype Control Support in the Custom Static Main Program” on page 46-68.

For an example that shows how to configure custom target optional features, see “Customize System Target Files” on page 46-19.

Declaring C Function Prototype Control Compliance

To declare C function prototype control compliance for your target, you must implement a callback function that sets the `ModelStepFunctionPrototypeControlCompliant` flag, and then install the callback function in the `SelectCallback` field of the `rtwgensettings` structure in your system target file. The callback function is triggered whenever the user selects the target in the System Target File Browser. For example, the following system target file code installs a `SelectCallback` function named `custom_select_callback_handler`:

```
rtwgensettings.SelectCallback = 'custom_select_callback_handler(hDlg,hSrc)';
```

The arguments to the `SelectCallback` function (`hDlg`, `hSrc`) are handles to private data used by the callback API functions. These handles are restricted to use in system target file callback functions. They should be passed in without alteration.

Your callback function should set the `ModelStepFunctionPrototypeControlCompliant` flag as follows:

```
slConfigUISetVal(hDlg,hSrc,'ModelStepFunctionPrototypeControlCompliant','on');
slConfigUISetEnabled(hDlg,hSrc,'ModelStepFunctionPrototypeControlCompliant',false);
hSrc.refreshDialog;
```

For more information about the system target file callback API, see the `slConfigUIGetVal`, `slConfigUISetEnabled`, and `slConfigUISetVal` function reference pages.

When the `ModelStepFunctionPrototypeControlCompliant` target configuration parameter is set to `on`, you can use the `Configure C Step Function Interface` dialog box to control the function prototype of the base-rate step function generated for a rate-based model.

Providing C Function Prototype Control Support in the Custom Static Main Program

If your target uses a custom static main program, and if a nondefault function prototype control configuration is associated with a model, you must update the static main program to call the function prototype controlled initialize and step functions. You can do this in either of the following ways:

- 1 Manually adapt your static main program to declare model data and call the function prototype controlled initialize and step functions.
- 2 Generate your main program by selecting model configuration parameter **Generate an example main program**. The generated main program declares model data and calls the function prototype controlled initialize and step function.

See Also

More About

- “Custom Targets” on page 46-2
- “Sample Custom Targets” on page 46-4
- “Customize System Target Files” on page 46-19

Support C++ Class Interface Control

This section describes how to configure a custom embedded target to support C++ class interface control. Without the described modifications, you will not be able to use C++ class code interface packaging to generate and configure C++ class interfaces to model code. For more information about C++ class interface control, see “Interactively Configure C++ Interface” (Embedded Coder).

About C++ Class Interface Control and Custom Targets

The requirements for supporting C++ class interface control are as follows:

- The target must be derived from the ERT system target file.
- The system target file must declare C++ class interface control compliance, as described in “Declaring C++ Class Interface Control Compliance” on page 46-69.

For an example that shows how to configure custom target optional features, see “Customize System Target Files” on page 46-19.

Declaring C++ Class Interface Control Compliance

To declare C++ class interface control compliance for your target, you must implement a callback function that sets the `CPPClassGenCompliant` flag, and then install the callback function in the `SelectCallback` field of the `rtwgensettings` structure in your system target file. The callback function is triggered whenever the user selects the target in the System Target File Browser. For example, the following system target file code installs a `SelectCallback` function named `custom_select_callback_handler`:

```
rtwgensettings.SelectCallback = 'custom_select_callback_handler(hDlg,hSrc)';
```

The arguments to the `SelectCallback` function (`hDlg`, `hSrc`) are handles to private data used by the callback API functions. These handles are restricted to use in system target file callback functions. They should be passed in without alteration.

Your callback function should set the `CPPClassGenCompliant` flag as follows:

```
slConfigUISetVal(hDlg,hSrc,'CPPClassGenCompliant','on');
slConfigUISetEnabled(hDlg,hSrc,'CPPClassGenCompliant',false);
hSrc.refreshDialog;
```

For more information about the system target file callback API, see the `slConfigUIGetVal`, `slConfigUISetEnabled`, and `slConfigUISetVal` function reference pages.

When the `CPPClassGenCompliant` target configuration parameter is set to `on`, you can use the C++ class code interface packaging to generate and configure C++ class interfaces to model code.

Providing C++ Class Interface Control Support in the Custom Static Main Program

Selecting C++ class code interface packaging for your model enables model configuration parameter **Generate an example main program**. When you select this parameter, the code generator produces an example main program, `ert_main.cpp`. The generated example main program declares model data and calls the C++ class interface configured model step method, and illustrates how the generated code can be deployed.

To customize the build process and disable generation and inclusion of an example main program, see the `setTargetProvidesMain` function. Disabling example main generation permits including a custom main program.

See Also

More About

- “Custom Targets” on page 46-2
- “Sample Custom Targets” on page 46-4
- “Customize System Target Files” on page 46-19

Support Concurrent Execution of Multiple Tasks

If a custom embedded target must support concurrent execution of multiple tasks on a multicore platform, the target must declare support for concurrent execution by setting the target configuration option `ConcurrentExecutionCompliant`. Otherwise, you will not be able to configure a multicore target model for concurrent execution.

If `ConcurrentExecutionCompliant` is not already configured for your custom target, you can set the option in the following ways:

- Include the following code directly in your system target file (*mytarget.tlc*):

```
rtwgensettings.SelectCallback = 'slConfigUISetVal(hDlg,hSrc,...
    ''ConcurrentExecutionCompliant'', 'on');';
rtwgensettings.ActivateCallback = 'slConfigUISetVal(hDlg,hSrc,...
    ''ConcurrentExecutionCompliant'', 'on');';
```

- Implement a callback function that sets the `ConcurrentExecutionCompliant` option, and then install the callback function in the `SelectCallback` field of the `rtwgensettings` structure in your system target file. The callback function is triggered whenever the user selects the target in the System Target File Browser. For example, the following system target file code installs a `SelectCallback` function named `custom_select_callback_handler`:

```
rtwgensettings.SelectCallback = 'custom_select_callback_handler(hDlg,hSrc)';
```

The arguments to the `SelectCallback` function (`hDlg`, `hSrc`) are handles to private data used by the callback API functions. These handles are restricted to use in system target file callback functions. They should be passed in without alteration.

Your callback function should set the `ConcurrentExecutionCompliant` option as follows:

```
slConfigUISetVal(hDlg,hSrc,'ConcurrentExecutionCompliant','on');
slConfigUISetEnabled(hDlg,hSrc,'ConcurrentExecutionCompliant',false);
hSrc.refreshDialog;
```

For more information about the system target file callback API, see the `slConfigUIGetVal`, `slConfigUISetEnabled`, and `slConfigUISetVal` function reference pages.

When the `ConcurrentExecutionCompliant` target configuration option is set to 'on', you can select the custom target and configure your multicore target model for concurrent execution.

For an example that shows how to configure custom target optional features, see “Customize System Target Files” on page 46-19.

See Also

More About

- “Custom Targets” on page 46-2
- “Sample Custom Targets” on page 46-4
- “Customize System Target Files” on page 46-19

Device Drivers

Device drivers that communicate with target hardware are essential to many real-time development projects.

You can integrate existing C (or C++) device driver functions into Simulink models by using the Legacy Code Tool. When you use the code generator to generate code from a model, the Legacy Code Tool can insert a call to your C function into the generated code. For details, see “Import Calls to External Code into Generated Code with Legacy Code Tool” on page 14-6 and “Integrate C Functions into Simulink Models with Legacy Code Tool”.

See Also

More About

- “Custom Targets” on page 46-2
- “Sample Custom Targets” on page 46-4
- “Customize System Target Files” on page 46-19