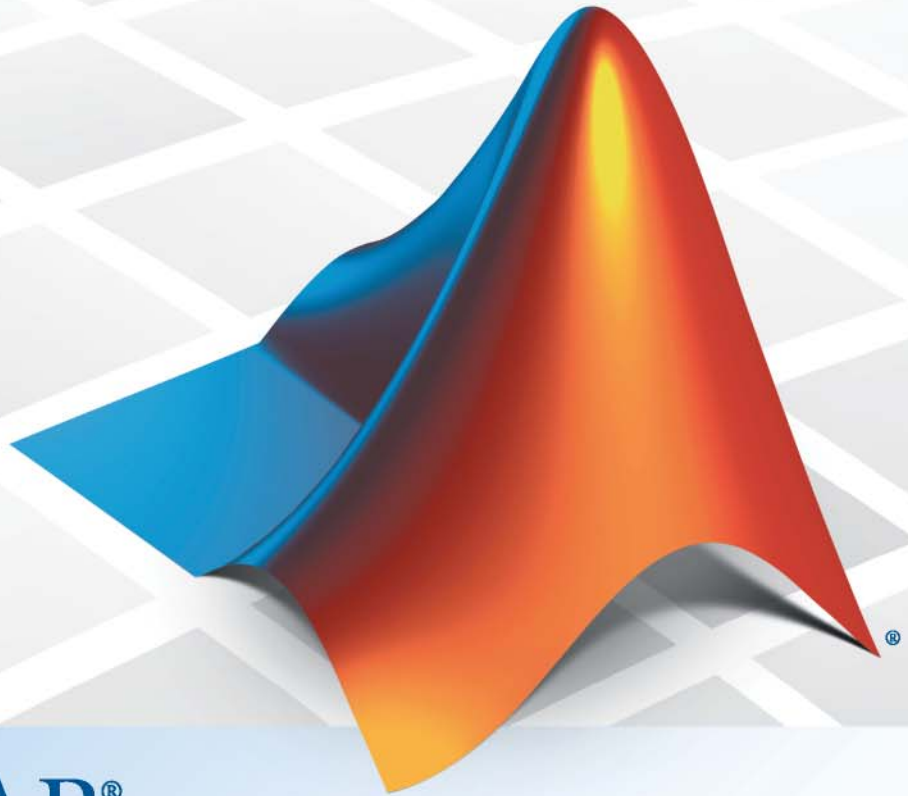


Simulink® 7

Graphical User Interface



MATLAB®
& SIMULINK®

How to Contact MathWorks



www.mathworks.com Web
comp.soft-sys.matlab Newsgroup
www.mathworks.com/contact_TS.html Technical Support



suggest@mathworks.com Product enhancement suggestions
bugs@mathworks.com Bug reports
doc@mathworks.com Documentation error reports
service@mathworks.com Order status, license renewals, passcodes
info@mathworks.com Sales, pricing, and general information



508-647-7000 (Phone)



508-647-7001 (Fax)



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

For contact information about worldwide offices, see the MathWorks Web site.

Simulink® Graphical User Interface

© COPYRIGHT 1990–2010 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

September 2007 Online only
March 2008 Online only
October 2008 Online only
March 2009 Online only
September 2009 Online only
March 2010 Online only
September 2010 Online only

New for Simulink 7.0 (Release 2007b)
Revised for Simulink 7.1 (Release 2008a)
Revised for Simulink 7.2 (Release 2008b)
Revised for Simulink 7.3 (Release 2009a)
Revised for Simulink 7.4 (Release 2009b)
Revised for Simulink 7.5 (Release 2010a)
Revised for Simulink 7.6 (Release 2010b)

Configuration Parameters Dialog Box

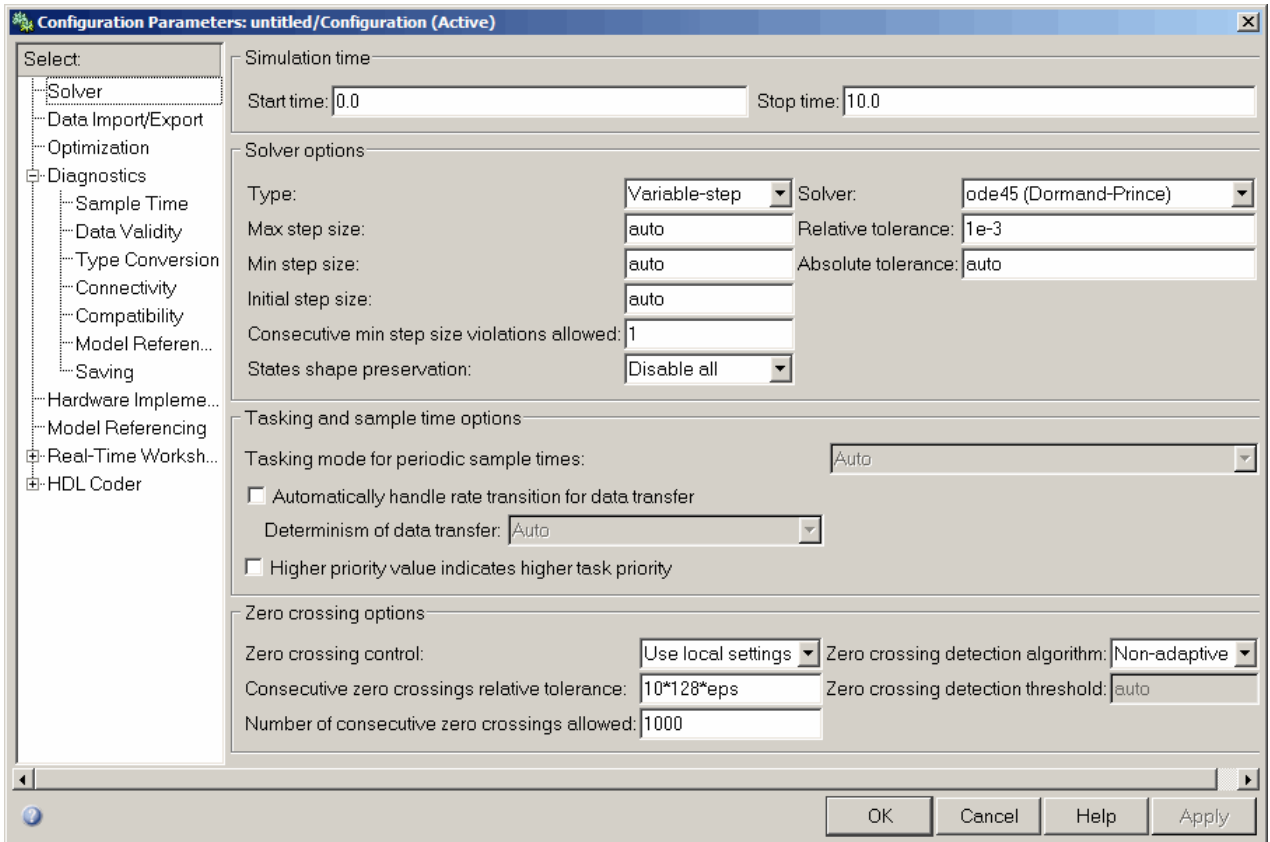
- “Configuration Parameters Dialog Box Overview” on page 1-2
- “Model Configuration Pane” on page 1-5
- “Solver Pane” on page 1-8
- “Data Import/Export Pane” on page 1-79
- “Optimization Pane” on page 1-116
- “Diagnostics Pane: Solver” on page 1-195
- “Diagnostics Pane: Sample Time” on page 1-221
- “Diagnostics Pane: Data Validity” on page 1-237
- “Diagnostics Pane: Type Conversion” on page 1-292
- “Diagnostics Pane: Connectivity” on page 1-305
- “Diagnostics Pane: Compatibility” on page 1-334
- “Diagnostics Pane: Model Referencing” on page 1-338
- “Diagnostics Pane: Saving” on page 1-352
- “Diagnostics Pane: Stateflow” on page 1-358
- “Hardware Implementation Pane” on page 1-368
- “Model Referencing Pane” on page 1-451
- “Simulation Target Pane: General” on page 1-475
- “Simulation Target Pane: Symbols” on page 1-491
- “Simulation Target Pane: Custom Code” on page 1-495

Configuration Parameters Dialog Box Overview

The **Configuration Parameters** dialog box specifies the settings for a model's active *configuration set*. These parameters determine the type of solver used, import and export settings, and other values that determine how the model runs. See Configuration Sets for more information.

Note You can also use the Model Explorer to modify settings for the active configuration set or any other configuration set. See “The Model Explorer: Overview” for more information.

To display the dialog box, select **Simulation > Configuration Parameters** in the Model Editor, or press **Ctrl+E**. The dialog box appears.



The dialog box groups the configuration parameters into various categories. To display the parameters for a specific category, click the category in the **Select** tree on the left side of the dialog box.

In most cases, Simulink® software does not apply changes until you click **OK** or **Apply** at the bottom of the dialog box. The **OK** button applies your changes and dismisses the dialog box. The **Apply** button applies your changes but leaves the dialog box open.

Note Each of the parameters in the **Configuration Parameters** dialog box can also be set via the `sim` command. Each parameter description includes the corresponding command line information.

Model Configuration Pane

In this section...
“Model Configuration Overview” on page 1-5
“Name” on page 1-6
“Description” on page 1-7

Model Configuration Overview

View or edit the name and description of your configuration set.

In the Model Explorer you can edit the name and description of your configuration sets.

In the Model Explorer or Simulink Preferences window you can edit the description of your template configuration set, Model Configuration Preferences. Go to the Model Configuration Preferences to edit the template Configuration Parameters to be used as defaults for new models.

When editing the Model Configuration preferences, you can click **Restore to Default Preferences** to restore the default configuration settings for creating new models. These underlying defaults cannot be changed.

Name

Specify the name of your configuration set.

Settings

Default: Configuration (for Active configuration set) or Configuration Preferences (for default configuration set).

Edit the name of your configuration set.

In the Model Configuration Preferences, the name of the default configuration is always Configuration Preferences, and cannot be changed.

Description

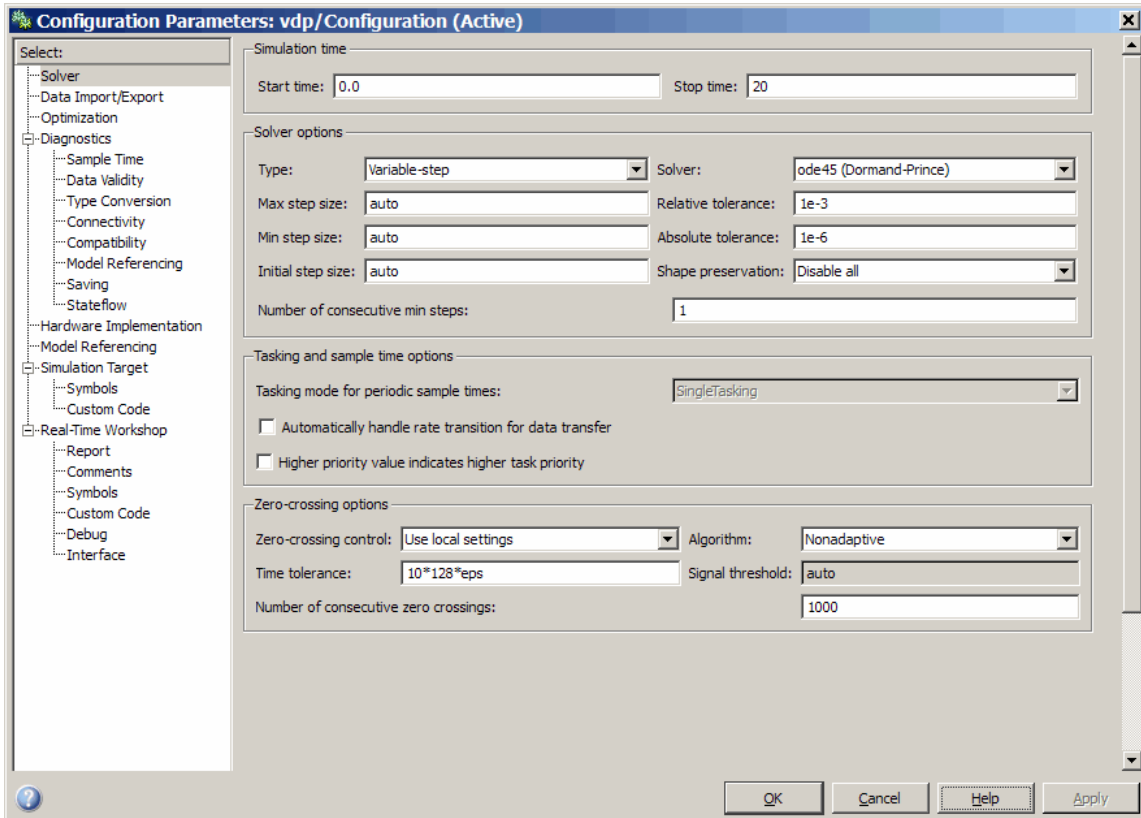
Specify a description of your configuration set.

Settings

No Default

Enter text to describe your configuration set.

Solver Pane



In this section...

“Solver Overview” on page 1-10

“Start time” on page 1-12

“Stop time” on page 1-14

“Type” on page 1-16

“Solver” on page 1-19

“Max Step Size” on page 1-27

In this section...

- “Initial Step Size” on page 1-29
- “Min Step Size” on page 1-31
- “Relative tolerance” on page 1-33
- “Absolute tolerance” on page 1-35
- “Shape preservation” on page 1-38
- “Maximum order” on page 1-40
- “Solver reset method” on page 1-42
- “Number of consecutive min steps” on page 1-44
- “Number of consecutive min steps” on page 1-45
- “Solver Jacobian Method” on page 1-47
- “Tasking mode for periodic sample times” on page 1-49
- “Automatically handle rate transition for data transfer” on page 1-51
- “Deterministic data transfer” on page 1-53
- “Higher priority value indicates higher task priority” on page 1-55
- “Zero-crossing control” on page 1-57
- “Time tolerance” on page 1-59
- “Number of consecutive zero crossings” on page 1-61
- “Algorithm” on page 1-63
- “Signal threshold” on page 1-65
- “Periodic sample time constraint” on page 1-67
- “Fixed-step size (fundamental sample time)” on page 1-70
- “Sample time properties” on page 1-72
- “Extrapolation order” on page 1-75
- “Number Newton’s iterations” on page 1-77

Solver Overview

Specify the simulation start and stop time, and the solver configuration for the simulation. Use the Solver pane to set up a solver for a model's active configuration set.

A solver computes a dynamic system's states at successive time steps over a specified time span, using information provided by the model.

Configuration

- 1 Select a solver type from the **Type** list.
- 2 Select a solver from the **Solver** list.
- 3 Set the parameters displayed for the selected type and solver combination.
- 4 Apply the changes.

Tips

- Simulation time is not the same as clock time. For example, running a simulation for 10 seconds usually does not take 10 seconds. Total simulation time depends on factors such as model complexity, solver step sizes, and computer speed.
- Fixed-step solver type is required for code generation, unless you use an S-function or RSim target.
- Variable-step solver type can significantly shorten the time required to simulate models in which states change rapidly or which contain discontinuities.

See Also

- Solvers
- Choosing a Solver
- Specifying a Simulation Start and Stop Time

- Configuration Parameters Dialog Box
- Solver Pane

Start time

Specify the start time for the simulation or generated code as a double-precision value, scaled to seconds.

Settings

Default: 0.0

- A start time other than 0.0 is an offset, and must be less than or equal to the stop time. An example of when you might use an offset is to set up a delay to accommodate some type of initialization.
- The values of block parameters with initial conditions must match the initial condition settings at the specified start time.
- Simulation time is not the same as clock time. For example, running a simulation for 10 seconds usually does not take 10 seconds. Total simulation time depends on factors such as model complexity, solver step sizes, and computer speed.

Command-Line Information

Parameter: StartTime

Type: string

Value: any valid value

Default: '0.0'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	0.0

See Also

- [Specifying a Simulation Start and Stop Time](#)
- [Configuration Parameters Dialog Box](#)
- [Solver Pane](#)

Stop time

Specify the stop time for the simulation or generated code as a double-precision value, scaled to seconds.

Settings

Default: 10

- Stop time must be greater than or equal to the start time.
- Specify `inf` to run a simulation or generated program until you explicitly pause or stop it.
- If the stop time is the same as the start time, the simulation or generated program runs for one step.
- Simulation time is not the same as clock time. For example, running a simulation for 10 seconds usually does not take 10 seconds. Total simulation time depends on factors such as model complexity, solver step sizes, and computer speed.
- If your model includes blocks that depend on absolute time and you are creating a design that runs indefinitely, see [Blocks That Depend on Absolute Time](#).

Command-Line Information

Parameter: `StopTime`

Type: `string`

Value: any valid value

Default: `'10.0'`

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Any positive value

See Also

- [Blocks That Depend on Absolute Time](#)
- [Using Blocks to Stop or Pause a Simulation](#)
- [Specifying a Simulation Start and Stop Time](#)
- [Configuration Parameters Dialog Box](#)
- [Solver Pane](#)

Type

Select the type of solver you want to use to simulate your model.

Settings

Default: Variable-step

Variable-step

Step size varies from step to step, depending on model dynamics. A variable-step solver:

- Reduces step size when model states change rapidly, to maintain accuracy.
- Increases step size when model states change slowly, to avoid unnecessary steps.

Variable-step is recommended for models in which states change rapidly or that contain discontinuities. In these cases, a variable-step solver requires fewer time steps than a fixed-step solver to achieve a comparable level of accuracy. This can significantly shorten simulation time.

Fixed-step

Step size remains constant throughout the simulation.

Required for code generation, unless you use an S-function or RSim target.

Note The solver computes the next time as the sum of the current time and the step size.

Dependencies

Selecting Variable-step enables the following parameters:

- Solver
- Max step size
- Min step size

- **Initial step size**
- **Relative tolerance**
- **Absolute tolerance**
- **Shape preservation**
- **Initial step size**
- **Number of consecutive min steps**
- **Zero-crossing control**
- **Time tolerance**
- **Number of consecutive zero crossings**
- **Algorithm**

Selecting Fixed-step enables the following parameters:

- **Solver**
- **Periodic sample time constraint**
- **Fixed-step size (fundamental sample time)**
- **Tasking mode for periodic sample times**
- **Higher priority value indicates higher task priority**
- **Automatically handle rate transitions for data transfers**

Command-Line Information

Parameter: SolverType

Type: string

Value: 'Variable-step' | 'Fixed-step'

Default: 'Variable-step'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Fixed-step

See Also

- Solvers
- Choosing a Solver
- “Purely Discrete Systems”
- Configuration Parameters Dialog Box
- Solver Pane

Solver

Select the solver you want to use to compute the model's states during simulation or code generation.

Settings

The available solvers change depending on which solver Type you selected:

- “Fixed-step Solvers” on page 1-19
- “Variable-step Solvers” on page 1-21

Fixed-step Solvers. Default: ode3 (Bogacki-Shampine)

ode3 (Bogacki-Shampine)

Computes the model's state at the next time step as an explicit function of the current value of the state and the state derivatives, using the Bogacki-Shampine Formula integration technique to compute the state derivatives. In the following example, X is the state, DX is the state derivative, and h is the step size:

$$X(n+1) = X(n) + h * DX(n)$$

Discrete (no continuous states)

Computes the time of the next time step by adding a fixed step size to the current time.

Use this solver for models with no states or discrete states only, using a fixed step size. Relies on the model's blocks to update discrete states.

The accuracy and length of time of the resulting simulation depends on the size of the steps taken by the simulation: the smaller the step size, the more accurate the results but the longer the simulation takes.

Note The fixed-step discrete solver cannot be used to simulate models that have continuous states.

ode5 (Dormand-Prince)

Computes the model's state at the next time step as an explicit function of the current value of the state and the state derivatives. Uses the Dormand-Prince Formula integration technique to compute the state derivatives. In the following example, X is the state, DX is the state derivative, and h is the step size:

$$X(n+1) = X(n) + h * DX(n)$$

ode4 (Runge-Kutta)

Uses the Fourth-Order Runge-Kutta (RK4) Formula integration technique to compute the model's state at the next time step as an explicit function of the current value of the state and the state derivatives. In the following example, X is the state, DX is the state derivative, and h is the step size:

$$X(n+1) = X(n) + h * DX(n)$$

ode2 (Heun)

Uses the Heun's Method integration technique to compute the model's state at the next time step as an explicit function of the current value of the state and the state derivatives. In the following example, X is the state, DX is the state derivative, and h is the step size:

$$X(n+1) = X(n) + h * DX(n)$$

ode1 (Euler)

Uses the Euler's Method integration technique to compute the model's state at the next time step as an explicit function of the current value of the state and the state derivatives. In the following example, X is the state, DX is the state derivative, and h is the step size:

$$X(n+1) = X(n) + h * DX(n)$$

ode14x (extrapolation)

Uses a combination of Newton's method and extrapolation from the current value to compute the model's state at the next time step, as an *implicit* function of the state and the state derivative at the next time step. In the following example, X is the state, DX is the state derivative, and h is the step size:

$$X(n+1) - X(n) - h * DX(n+1) = 0$$

This solver requires more computation per step than an explicit solver, but is more accurate for a given step size.

Variable-step Solvers. Default: ode45 (Dormand-Prince)

ode45 (Dormand-Prince)

Computes the model's state at the next time step using an explicit Runge-Kutta (4,5) formula (the Dormand-Prince pair) for numerical integration.

ode45 is a one-step solver, and therefore only needs the solution at the preceding time point.

Use ode45 as a first try for most problems.

Discrete (no continuous states)

Computes the time of the next step by adding a step size that varies depending on the rate of change of the model's states.

Use this solver for models with no states or discrete states only, using a variable step size.

ode23 (Bogacki-Shampine)

Computes the model's state at the next time step using an explicit Runge-Kutta (2,3) formula (the Bogacki-Shampine pair) for numerical integration.

ode23 is a one-step solver, and therefore only needs the solution at the preceding time point.

ode23 is more efficient than ode45 at crude tolerances and in the presence of mild stiffness.

ode113 (Adams)

Computes the model's state at the next time step using a variable-order Adams-Bashforth-Moulton PECE numerical integration technique.

ode113 is a multistep solver, and thus generally needs the solutions at several preceding time points to compute the current solution.

ode113 can be more efficient than ode45 at stringent tolerances.

`ode15s (stiff/NDF)`

Computes the model's state at the next time step using variable-order numerical differentiation formulas (NDFs). These are related to, but more efficient than the backward differentiation formulas (BDFs), also known as Gear's method.

`ode15s` is a multistep solver, and thus generally needs the solutions at several preceding time points to compute the current solution.

`ode15s` is efficient for stiff problems. Try this solver if `ode45` fails or is inefficient.

`ode23s (stiff/Mod. Rosenbrock)`

Computes the model's state at the next time step using a modified Rosenbrock formula of order 2.

`ode23s` is a one-step solver, and therefore only needs the solution at the preceding time point.

`ode23s` is more efficient than `ode15s` at crude tolerances, and can solve stiff problems for which `ode15s` is ineffective.

`ode23t (Mod. stiff/Trapezoidal)`

Computes the model's state at the next time step using an implementation of the trapezoidal rule with a "free" interpolant.

`ode23t` is a one-step solver, and therefore only needs the solution at the preceding time point.

Use `ode23t` if the problem is only moderately stiff and you need a solution with no numerical damping.

`ode23tb (stiff/TR-BDF2)`

Computes the model's state at the next time step using a multistep implementation of TR-BDF2, an implicit Runge-Kutta formula with a trapezoidal rule first stage, and a second stage consisting of a backward differentiation formula of order two. By construction, the same iteration matrix is used in evaluating both stages.

`ode23tb` is more efficient than `ode15s` at crude tolerances, and can solve stiff problems for which `ode15s` is ineffective.

Tips

- Identifying the optimal solver for a model requires experimentation, for an in-depth discussion, see Choosing a Solver.
- The optimal solver balances acceptable accuracy with the shortest simulation time.
- Simulink software uses a discrete solver for any model with no states or discrete states only, even if you specify a continuous solver.
- A smaller step size increases accuracy, but also increases simulation time.
- The degree of computational complexity increases for ode_n , as n increases.
- As computational complexity increases, the accuracy of the results also increases.

Dependencies

Selecting the `ode1` (Euler), `ode2` (Huen), `ode3` (Bogacki-Shampine), `ode4` (Runge-Kutta), `ode5` (Dormand-Prince), or `Discrete` (no continuous states) fixed-step solvers enables the following parameters:

- **Fixed-step size (fundamental sample time)**
- **Periodic sample time constraint**
- **Tasking mode for periodic sample times**
- **Automatically handle rate transition for data transfers**
- **Higher priority value indicates higher task priority**

Selecting `ode14x` (extrapolation) enables the following parameters:

- **Fixed-step size (fundamental sample time)**
- **Extrapolation order**
- **Number Newton's iterations**
- **Periodic sample time constraint**
- **Tasking mode for periodic sample times**
- **Automatically handle rate transition for data transfers**

- **Higher priority value indicates higher task priority**

Selecting the Discrete (no continuous states) variable-step solver enables the following parameters:

- **Max step size**
- **Automatically handle rate transition for data transfers**
- **Higher priority value indicates higher task priority**
- **Zero-crossing control**
- **Time tolerance**
- **Number of consecutive zero crossings**
- **Algorithm**

Selecting ode45 (Dormand-Prince), ode23 (Bogacki-Shampine), ode113 (Adams), or ode23s (stiff/Mod. Rosenbrock) enables the following parameters:

- **Max step size**
- **Min step size**
- **Initial step size**
- **Relative tolerance**
- **Absolute tolerance**
- **Shape preservation**
- **Number of consecutive min steps**
- **Automatically handle rate transition for data transfers**
- **Higher priority value indicates higher task priority**
- **Zero-crossing control**
- **Time tolerance**
- **Number of consecutive zero crossings**
- **Algorithm**

Selecting ode15s (stiff/NDF), ode23t (Mod. stiff/Trapezoidal), or ode23tb (stiff/TR-BDF2) enables the following parameters:

- **Max step size**
- **Min step size**
- **Initial step size**
- **Solver reset method**
- **Number of consecutive min steps**
- **Relative tolerance**
- **Absolute tolerance**
- **Shape preservation**
- **Maximum order**
- **Automatically handle rate transition for data transfers**
- **Higher priority value indicates higher task priority**
- **Zero-crossing control**
- **Time tolerance**
- **Number of consecutive zero crossings**
- **Algorithm**

Command-Line Information

Parameter: Solver

Type: string

Value: 'VariableStepDiscrete' | 'ode45' | 'ode23' | 'ode113' | 'ode15s' | 'ode23s' | 'ode23t' | 'ode23tb' | 'FixedStepDiscrete' | 'ode5' | 'ode4' | 'ode3' | 'ode2' | 'ode1' | 'ode14x'

Default: 'ode45'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Discrete (no continuous states)

See Also

- Solvers
- Choosing a Solver
- “Purely Discrete Systems”
- Configuration Parameters Dialog Box
- Solver Pane

Max Step Size

Specify the largest time step that the solver can take.

Settings

Default: auto

- For the discrete solver, the default value (auto) is the model's shortest sample time.
- For continuous solvers, the default value (auto) is determined from the start and stop times. If the stop time equals the start time or is `inf`, Simulink software chooses 0.2 seconds as the maximum step size. Otherwise, it sets the maximum step size to

$$h_{\max} = \frac{t_{\text{stop}} - t_{\text{start}}}{50}$$

Tips

- Generally, the default maximum step size is sufficient. If you are concerned about the solver missing significant behavior, change the parameter to prevent the solver from taking too large a step.
- If the time span of the simulation is very long, the default step size might be too large for the solver to find the solution.
- If your model contains periodic or nearly periodic behavior and you know the period, set the maximum step size to some fraction (such as 1/4) of that period.
- In general, for more output points, change the refine factor, not the maximum step size. For more information, see [Specifying Output Options](#).

Dependencies

This parameter is enabled only if the solver **Type** is set to `Variable-step`.

Command-Line Information

Parameter: MaxStep
Type: string
Value: any valid value
Default: 'auto'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- “Purely Discrete Systems”
- Specifying Output Options
- Configuration Parameters Dialog Box
- Solver Pane

Initial Step Size

Specify the size of the first time step that the solver takes.

Settings

Default: auto

By default, the solver selects an initial step size by examining the derivatives of the states at the start time.

Tips

- Be careful when increasing the initial step size. If the first step size is too large, the solver might step over important behavior.
- The initial step size parameter is a *suggested* first step size. The solver tries this step size but reduces it if error criteria are not satisfied.

Dependencies

This parameter is enabled only if the solver **Type** is set to Variable-step.

Command-Line Information

Parameter: InitialStep

Type: string

Value: any valid value

Default: 'auto'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- “Purely Discrete Systems”
- Improving Simulation Performance and Accuracy
- Configuration Parameters Dialog Box
- Solver Pane

Min Step Size

Specify the smallest time step that the solver can take.

Settings

Default: auto

- The default value (auto) sets an unlimited number of warnings and a minimum step size on the order of machine precision.
- You can specify either a real number greater than zero, or a two-element vector for which the first element is the minimum step size and the second element is the maximum number of minimum step size warnings before an error was issued.

Tips

- If the solver takes a smaller step to meet error tolerances, it issues a warning indicating the current effective relative tolerance.
- Setting the second element to zero results in an error the first time the solver must take a step smaller than the specified minimum. This is equivalent to changing the **Min step size violation** diagnostic to error on the **Diagnostics** pane (see Min step size violation).
- Setting the second element to -1 results in an unlimited number of warnings. This is also the default if the input is a scalar.

Dependencies

This parameter is enabled only if the solver **Type** is set to **Variable-step**.

Command-Line Information

Parameter: MinStep
Type: string
Value: any valid value
Default: 'auto'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- “Purely Discrete Systems”
- Min step size violation
- Configuration Parameters Dialog Box
- Solver Pane

Relative tolerance

Specify the largest acceptable solver error, relative to the size of each state during each time step. If the relative error exceeds this tolerance, the solver reduces the time step size.

Settings

Default: 1e-3

- The relative tolerance is a percentage of the state's value.
- The default value (1e-3) means that the computed state is accurate to within 0.1%.

Tips

- The acceptable error at each time step is a function of both the **Relative tolerance** and the **Absolute tolerance**. For more information about how these settings work together, see [Specifying Variable-Step Solver Error Tolerances](#).
- During each time step, the solver computes the state values at the end of the step and also determines the local error – the estimated error of these state values. If the error is greater than the acceptable error for any state, the solver reduces the step size and tries again.
- The default relative tolerance value is sufficient for most applications. Decreasing the relative tolerance value can slow down the simulation.
- To check the accuracy of a simulation after you run it, you can reduce the relative tolerance to 1e-4 and run it again. If the results of the two simulations are not significantly different, you can feel confident that the solution has converged.

Dependencies

This parameter is enabled only if you set:

- Solver **Type** to **Variable-step**.
- **Solver** to a continuous variable-step solver.

This parameter works along with **Absolute tolerance** to determine the acceptable error at each time step. For more information about how these settings work together, see [Specifying Variable-Step Solver Error Tolerances](#).

Command-Line Information

Parameter: RelTol
Type: string
Value: any valid value
Default: '1e-3'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- [Specifying Variable-Step Solver Error Tolerances](#)
- [Improving Simulation Performance and Accuracy](#)
- [Configuration Parameters Dialog Box](#)
- [Solver Pane](#)

Absolute tolerance

Specify the largest acceptable solver error, as the value of the measured state approaches zero. If the absolute error exceeds this tolerance, the solver reduces the time step size.

Settings

Default: auto

- The default value (auto) initially sets the absolute tolerance for each state to 1e-6. As the simulation progresses, the absolute tolerance for each state is reset to the maximum value that the state has thus far assumed times the relative tolerance for that state.

For example, if a state goes from 0 to 1 and the **Relative tolerance** is 1e-3, then by the end of the simulation the **Absolute tolerance** is set to 1e-3.

- If the computed setting is not suitable, you can determine an appropriate setting yourself.

Tips

- The acceptable error at each time step is a function of both the **Relative tolerance** and the **Absolute tolerance**. For more information about how these settings work together, see [Specifying Variable-Step Solver Error Tolerances](#).
- The Integrator, Transfer Fcn, State-Space, and Zero-Pole blocks allow you to specify absolute tolerance values for solving the model states that they compute or that determine their output. The absolute tolerance values that you specify in these blocks override the global setting in the Configuration Parameters dialog box.
- You might want to override the **Absolute tolerance** setting using blocks if the global setting does not provide sufficient error control for all of your model's states, for example if they vary widely in magnitude.
- If you set the **Absolute tolerance** too low, the solver may take too many steps around near-zero state values, slowing down the simulation.
- To check the accuracy of a simulation after you run it, you can reduce the absolute tolerance and run it again. If the results of the two simulations

are not significantly different, you can feel confident that the solution has converged.

- If your simulation results do not seem accurate, and your model has states whose values approach zero, the **Absolute tolerance** may be too large. Reduce the **Absolute tolerance** to force the simulation to take more steps around areas of near-zero state values.

Dependencies

This parameter is enabled only if you set:

- Solver **Type** to *Variable-step*.
- **Solver** to a continuous variable-step solver.

This parameter works along with **Relative tolerance** to determine the acceptable error at each time step. For more information about how these settings work together, see *Specifying Variable-Step Solver Error Tolerances*.

Command-Line Information

Parameter: AbsTol

Type: string

Value: any valid value

Default: 'auto'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- [Specifying Variable-Step Solver Error Tolerances](#)
- [Improving Simulation Performance and Accuracy](#)
- [Configuration Parameters Dialog Box](#)
- [Solver Pane](#)

Shape preservation

At each time step use derivative information to improve integration accuracy.

Settings

Default: Disable all

Disable all

Do not perform Shape preservation on any signals.

Enable all

Perform Shape preservation on all signals.

Tips

- The default setting (Disable all) usually provides good accuracy for most models.
- Setting to Enable all will increase accuracy in those models having signals whose derivative exhibits a high rate of change, but simulation time may be increased.

Dependencies

This parameter is enabled only if you use a continuous-step solver.

Command-Line Information

Parameter: ShapePreserveControl

Type: string

Value: 'EnableAll | 'DisableAll

Default: 'DisableAll

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact

See Also

- “Zero-Crossing Detection”
- Solver Pane

Maximum order

Select the order of the numerical differentiation formulas (NDFs) used in the `ode15s` solver.

Settings

Default: 5

- 5 Specifies that the solver uses fifth order NDFs.
- 1 Specifies that the solver uses first order NDFs.
- 2 Specifies that the solver uses second order NDFs.
- 3 Specifies that the solver uses third order NDFs.
- 4 Specifies that the solver uses fourth order NDFs.

Tips

- Although the higher order formulas are more accurate, they are less stable.
- If your model is stiff and requires more stability, reduce the maximum order to 2 (the highest order for which the NDF formula is A-stable).
- As an alternative, you can try using the `ode23s` solver, which is a lower order (and A-stable) solver.

Dependencies

This parameter is enabled only if **Solver** is set to `ode15s`.

Command-Line Information

Parameter: MaxOrder

Type: integer

Value: 1 | 2 | 3 | 4 | 5

Default: 5

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- [Specifying Variable-Step Solver Error Tolerances](#)
- [Improving Simulation Performance and Accuracy](#)
- [Configuration Parameters Dialog Box](#)
- [Solver Pane](#)

Solver reset method

Select how the solver behaves during a reset, such as when it detects a zero crossing.

Settings

Default: Fast

Fast

Specifies that the solver will not recompute the Jacobian matrix at a solver reset.

Robust

Specifies that the solver will recompute the Jacobian matrix needed by the integration step at every solver reset.

Tips

- Selecting **Fast** speeds up the simulation. However, it can result in incorrect solutions in some cases.
- If you suspect that the simulation is giving incorrect results, try the **Robust** setting. If there is no difference in simulation results between the fast and robust settings, revert to the fast setting.

Dependencies

This parameter is enabled only if you select one of the following solvers:

- ode15s (Stiff/NDF)
- ode23t (Mod. Stiff/Trapezoidal)
- ode23tb (Stiff/TR-BDF2)

Command-Line Information

Parameter: SolverResetMethod

Type: string

Value: 'Fast' | 'Robust'

Default: 'Fast'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Choosing a Solver
- Configuration Parameters Dialog Box
- Solver Pane

Number of consecutive min steps

Specify the maximum number of consecutive minimum step size violations allowed during simulation.

Settings

Default: 1

- A minimum step size violation occurs when a variable-step continuous solver takes a smaller step than that specified by the **Min step size** property (see Min step size).
- Simulink software counts the number of consecutive violations that it detects. If the count exceeds the value of **Number of consecutive min steps**, Simulink software displays either a warning or error message as specified by the **Min step size violation** diagnostic (see Min step size violation).

Dependencies

This parameter is enabled only if you set:

- Solver **Type** to Variable-step.
- **Solver** to a continuous variable step solver.

Command-Line Information

Parameter: MaxConsecutiveMinStep

Type: string

Value: any valid value

Default: '1'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact

See Also

- Choosing a Solver
- Min step size violation
- Min step size
- Configuration Parameters Dialog Box
- Solver Pane

Number of consecutive min steps

Specify the maximum number of consecutive minimum step size violations allowed during simulation.

Settings

Default: 1

- A minimum step size violation occurs when a variable-step continuous solver takes a smaller step than that specified by the **Min step size** property (see Min step size).
- Simulink software counts the number of consecutive violations that it detects. If the count exceeds the value of **Number of consecutive min steps**, Simulink software displays either a warning or error message as specified by the **Min step size violation** diagnostic (see Min step size violation).

Dependencies

This parameter is enabled only if you set:

- Solver **Type** to Variable-step.
- **Solver** to a continuous variable step solver.

Command-Line Information

Parameter: MaxConsecutiveMinStep

Type: string

Value: any valid value

Default: '1'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Choosing a Solver
- Min step size violation
- Min step size
- Configuration Parameters Dialog Box
- Solver Pane

Solver Jacobian Method

Settings

Default: Auto

auto

Sparse perturbation

Full perturbation

Sparse analytical

Full analytical

Tips

- The default setting (Auto) usually provides good accuracy for most models.

Dependencies

This parameter is enabled only if an implicit solver is used.

Command-Line Information

Parameter: SolverJacobianMethodControl

Type: string

Value: 'auto' | 'SparsePerturbation' | 'FullPerturbation' |
'SparseAnalytical' | 'FullAnalytical'

Default: 'auto'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- “Choosing a Solver”
- Solver Pane

Tasking mode for periodic sample times

Select how blocks with periodic sample times execute.

Settings

Default: Auto

Auto

Specifies that single-tasking execution is used if:

- Your model contains one sample time.
- Your model contains a continuous and a discrete sample time, and the fixed-step size is equal to the discrete sample time.

Selects multitasking execution for models operating at different sample rates.

SingleTasking

Specifies that all blocks are processed through each stage of simulation together (for example, calculating output and updating discrete states).

MultiTasking

Specifies that groups of blocks with the same execution priority are processed through each stage of simulation (for example, calculating output and updating discrete states) based on task priority. Multitasking mode helps to create valid models of real-world multitasking systems, where sections of your model represent concurrent tasks.

Tip

The **Multitask rate transition** parameter on the **Diagnostics > Sample Time** pane allows you to adjust error checking for sample rate transitions between blocks that operate at different sample rates.

Dependency

This parameter is enabled by selecting Fixed-step solver type.

Command-Line Information

Parameter: SolverMode

Type: string

Value: 'Auto' | 'SingleTasking' | 'MultiTasking'

Default: 'Auto'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Rate Transition block
- Model Execution and Rate Transitions
- Single-Tasking and Multitasking Execution Modes
- Sample Rate Transitions
- Single-Tasking and Multitasking Execution of a Model: an Example
- Configuration Parameters Dialog Box
- Solver Pane

Automatically handle rate transition for data transfer

Specify whether Simulink software automatically inserts hidden Rate Transition blocks between blocks that have different sample rates to ensure: the integrity of data transfers between tasks; and optional determinism of data transfers for periodic tasks.

Settings

Default: Off



On

Inserts hidden Rate Transition blocks between blocks when rate transitions are detected. Handles rate transitions for asynchronous and periodic tasks. Simulink software adds the hidden blocks configured to ensure data integrity for data transfers. Selecting this option also enables the parameter **Deterministic data transfer**, which allows you to control the level of data transfer determinism for periodic tasks.



Off

Does not insert hidden Rate Transition blocks when rate transitions are detected. If Simulink software detects invalid transitions, you must adjust the model such that the sample rates for the blocks in question match or manually add a Rate Transition block.

See Rate Transition Block Options in the Real-Time Workshop® documentation for further details.

Tips

- Selecting this parameter allows you to handle rate transition issues automatically. 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, Simulink software configures the inserted blocks to ensure data integrity but not determinism during data transfers.

Command-Line Information

Parameter: AutoInsertRateTranBlk

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact (for simulation and during development) Off (for production code generation)
Efficiency	No impact
Safety precaution	Off

See Also

- Rate Transition Block Options
- Configuration Parameters Dialog Box
- Solver Pane

Deterministic data transfer

Control whether the Rate Transition block parameter **Ensure deterministic data transfer (maximum delay)** is set for auto-inserted Rate Transition blocks

Default: Whenever possible

Always

Specifies that the block parameter **Ensure deterministic data transfer (maximum delay)** is always set for auto-inserted Rate Transition blocks.

If Always is selected and if a model needs to auto-insert a Rate Transition block to handle a rate transition that is *not* between two periodic sample times related by an integer multiple, Simulink errors out.

Whenever possible

Specifies that the block parameter **Ensure deterministic data transfer (maximum delay)** is set for auto-inserted Rate Transition blocks whenever possible. If an auto-inserted Rate Transition block handles data transfer between two periodic sample times that are related by an integer multiple, **Ensure deterministic data transfer (maximum delay)** is set; otherwise, it is cleared.

Never (minimum delay)

Specifies that the block parameter **Ensure deterministic data transfer (maximum delay)** is never set for auto-inserted Rate Transition blocks.

Note Clearing the Rate Transition block parameter **Ensure deterministic data transfer (maximum delay)** can provide reduced latency for models that do not require determinism. See the description of **Ensure deterministic data transfer (maximum delay)** on the Rate Transition block reference page for more information.

Dependencies

This parameter is enabled only if **Automatically handle rate transition for data transfer** is checked.

Command-Line Information

Parameter: InsertRTBMode

Type: string

Value: 'Always' | 'Whenever possible' | 'Never (minimum delay)'

Default: 'Whenever possible'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	'Whenever possible'

See Also

- Rate Transition Block Options
- Configuration Parameters Dialog Box
- Solver Pane

Higher priority value indicates higher task priority

Specify whether the real-time system targeted by the model assigns higher or lower priority values to higher priority tasks when implementing asynchronous data transfers

Settings

Default: Off



On

Real-time system assigns higher priority values to higher priority tasks, for example, 8 has a higher task priority than 4. Rate Transition blocks treat asynchronous transitions between rates with lower priority values and rates with higher priority values as low-to-high rate transitions.



Off

Real-time system assigns lower priority values to higher priority tasks, for example, 4 has a higher task priority than 8. Rate Transition blocks treat asynchronous transitions between rates with lower priority values and rates with higher priority values as high-to-low rate transitions.

Command-Line Information

Parameter: PositivePriorityOrder

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Rate Transitions and Asynchronous Blocks
- Configuration Parameters Dialog Box
- Solver Pane

Zero-crossing control

Enables zero-crossing detection during variable-step simulation of the model. For most models, this speeds up simulation by enabling the solver to take larger time steps.

Settings

Default: Use local settings

Use local settings

Specifies that zero-crossing detection be enabled on a block-by-block basis. For a list of applicable blocks, see “Simulating Dynamic Systems”

To specify zero-crossing detection for one of these blocks, open the block’s parameter dialog box and select the **Enable zero-crossing detection** option.

Enable all

Enables zero-crossing detection for all blocks in the model.

Disable all

Disables zero-crossing detection for all blocks in the model.

Tips

- For most models, enabling zero-crossing detection speeds up simulation by allowing the solver to take larger time steps.
- If a model has extreme dynamic changes, disabling this option can speed up the simulation but can also decrease the accuracy of simulation results. See Zero-crossing Detection for more information.
- Selecting **Enable all** or **Disable all** overrides the local zero-crossing detection setting for individual blocks.

Dependencies

This parameter is enabled only if the solver **Type** is set to **Variable-step**.

Selecting either **Use local settings** or **Enable all** enables the following parameters:

- **Time tolerance**
- **Number of consecutive zero crossings**
- **Algorithm**

Command-Line Information

Parameter: ZeroCrossControl

Type: string

Value: 'UseLocalSettings' | 'EnableAll' | 'DisableAll'

Default: 'UseLocalSettings'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Zero-Crossing Detection
- Number of consecutive zero crossings
- Consecutive zero-crossings violation
- Time tolerance
- Configuration Parameters Dialog Box
- Solver Pane

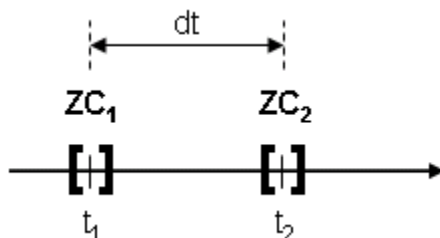
Time tolerance

Specify a tolerance factor that controls how closely zero-crossing events must occur to be considered consecutive.

Settings

Default: $10 \cdot 128 \cdot \text{eps}$

- Simulink software defines zero crossings as consecutive if the time between events is less than a particular interval. The following figure depicts a simulation timeline during which Simulink software detects zero crossings ZC_1 and ZC_2 , bracketed at successive time steps t_1 and t_2 .



Simulink software determines that the zero crossings are consecutive if

$$dt < \text{RelTolZC} * t_2$$

where dt is the time between zero crossings and RelTolZC is the **Time tolerance**.

- Simulink software counts the number of consecutive zero crossings that it detects. If the count exceeds the value of **Number of consecutive zero crossings** allowed, Simulink software displays either a warning or error as specified by the **Consecutive zero-crossings violation** diagnostic (see Consecutive zero-crossings violation).

Tips

- Simulink software resets the counter each time it detects nonconsecutive zero crossings (successive zero crossings that fail to meet the relative tolerance setting); therefore, decreasing the relative tolerance value may afford your model's behavior more time to recover.

- If your model experiences excessive zero crossings, you can also increase the **Number of consecutive zero crossings** to increase the threshold at which Simulink software triggers the **Consecutive zero-crossings violation** diagnostic.

Dependencies

This parameter is enabled only if **Zero-crossing control** is set to either `Use local settings` or `Enable all`.

Command-Line Information

Parameter: ConsecutiveZCsStepRelTol

Type: string

Value: any valid value

Default: '10*128*eps'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Zero-crossing Detection
- Zero-crossing Control
- Number of consecutive zero crossings
- Consecutive zero-crossings violation
- Configuration Parameters Dialog Box
- Solver Pane

Number of consecutive zero crossings

Specify the number of consecutive zero crossings that can occur before Simulink software displays a warning or an error.

Settings

Default: 1000

- Simulink software counts the number of consecutive zero crossings that it detects. If the count exceeds the specified value, Simulink software displays either a warning or an error as specified by the **Consecutive zero-crossings violation** diagnostic (see Consecutive zero-crossings violation).
- Simulink software defines zero crossings as consecutive if the time between events is less than a particular interval (see Time tolerance).

Tips

- If your model experiences excessive zero crossings, you can increase this parameter to increase the threshold at which Simulink software triggers the **Consecutive zero-crossings violation** diagnostic. This may afford your model's behavior more time to recover.
- Simulink software resets the counter each time it detects nonconsecutive zero crossings; therefore, decreasing the relative tolerance value may also afford your model's behavior more time to recover.

Dependencies

This parameter is enabled only if **Zero-crossing control** is set to either Use local settings or Enable all.

Command-Line Information

Parameter: MaxConsecutiveZCs

Type: string

Value: any valid value

Default: '1000'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Zero-Crossing Detection
- Zero-Crossing Control
- Consecutive zero-crossings violation
- Time tolerance
- Configuration Parameters Dialog Box
- Solver Pane

Algorithm

Specifies the algorithm to detect zero crossings when a variable-step solver is used.

Settings

Default: Nonadaptive

Adaptive

Use an improved zero-crossing algorithm which dynamically activates and deactivates zero-crossing bracketing. With this algorithm you can set a zero-crossing tolerance. See “Signal threshold” on page 1-65 to learn how to set the zero-crossing tolerance.

Nonadaptive

Use the nonadaptive zero-crossing algorithm present in the Simulink software prior to Version 7.0 (R2008a). This option is provided for backward compatibility.

Tips

- The adaptive zero-crossing algorithm is especially useful in systems having strong “chattering”, or Zeno behavior. In such systems, this algorithm yields shorter simulation run times compared to the nonadaptive algorithm. See Zero-Crossing Detection for more information.

Dependencies

- This parameter is enabled only if the solver **Type** is set to **Variable-step**.
- Selecting **Adaptive** enables the **Signal threshold** parameter.

Command-Line Information

Parameter: ZeroCrossAlgorithm

Type: string

Value: 'Nonadaptive' | 'Adaptive'

Default: 'Nonadaptive'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Zero-Crossing Detection
- Number of consecutive zero crossings
- Consecutive zero-crossings violation
- Time tolerance
- Configuration Parameters Dialog Box
- Solver Pane

Signal threshold

Specifies the deadband region used during the detection of zero crossings. Signals falling within this region are defined as having crossed through zero.

The signal threshold is a real number, greater than or equal to zero.

Settings

Default: Auto

Auto

The signal threshold is determined automatically by the adaptive algorithm.

String

Use the specified value for the signal threshold. The value must be a real number equal to or greater than zero.

Tips

- Entering too small of a value for the **Signal Threshold** parameter will result in long simulation run times.
- Entering a large **Signal Threshold** value may improve the simulation speed (especially in systems having extensive chattering). However, making the value too large may reduce the simulation accuracy.

Dependency

This parameter is enabled if the zero-crossing **Algorithm** is set to **Adaptive**.

Command-Line Information

Parameter: ZCThreshold

Type: string

Value: 'auto' | any real number greater than or equal to zero

Default: 'auto'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Zero-Crossing Detection
- Number of consecutive zero crossings
- Consecutive zero-crossings violation
- Time tolerance
- Configuration Parameters Dialog Box
- Solver Pane

Periodic sample time constraint

Select constraints on the sample times defined by this model. If the model does not satisfy the specified constraints during simulation, Simulink software displays an error message.

Settings

Default: Unconstrained

Unconstrained

Specifies no constraints. Selecting this option causes Simulink software to display a field for entering the solver step size.

Use the **Fixed-step size (fundamental sample time)** option to specify solver step size.

Ensure sample time independent

Specifies that Model blocks inherit sample time from the context in which they are used. You cannot use a referenced model that has intrinsic sample times in a triggered subsystem or iterator subsystem. If you plan on referencing this model in a triggered or iterator subsystem, you should select **Ensure sample time independent** so that Simulink can detect sample time problems while unit testing this model.

- Model Block Sample Times
- Inherited Sample Time for Referenced Models
- Function Call Models

Simulink software checks to ensure that this model can inherit its sample times from a model that references it without altering its behavior. Models that specify a step size (i.e., a base sample time) cannot satisfy this constraint. For this reason, selecting this option causes Simulink software to hide the group's step size field (see **Fixed-step size (fundamental sample time)**).

Specified

Specifies that Simulink software check to ensure that this model operates at a specified set of prioritized periodic sample times. Use the **Sample time properties** option to specify and assign priorities to model sample times.

Executing Multitasking Models explains how to use this option for multitasking models.

Tips

During simulation, Simulink software checks to ensure that the model satisfies the constraints. If the model does not satisfy the specified constraint, then Simulink software displays an error message.

Dependencies

This parameter is enabled only if the solver **Type** is set to Fixed-step.

Selecting Unconstrained enables the following parameters:

- **Fixed-step size (fundamental sample time)**
- **Tasking mode for periodic sample times**
- **Higher priority value indicates higher task priority**
- **Automatically handle rate transitions for data transfers**

Selecting Specified enables the following parameters:

- **Sample time properties**
- **Tasking mode for periodic sample times**
- **Higher priority value indicates higher task priority**
- **Automatically handle rate transitions for data transfers**

Command-Line Information

Parameter: SampleTimeConstraint

Type: string

Value: 'unconstrained' | 'STIndependent' | 'Specified'

Default: 'unconstrained'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Specified or Ensure sample time independent

See Also

- Model Block Sample Times
- Inherited Sample Time for Referenced Models
- Function Call Models
- Fixed-step size (fundamental sample time)
- Executing Multitasking Models
- Configuration Parameters Dialog Box
- Solver Pane

Fixed-step size (fundamental sample time)

Specify the step size used by the selected fixed-step solver.

Settings

Default: auto

- Entering auto (the default) in this field causes Simulink software to choose the step size.
- If the model specifies one or more periodic sample times, Simulink software chooses a step size equal to the least common denominator of the specified sample times. This step size, known as the fundamental sample time of the model, ensures that the solver will take a step at every sample time defined by the model.
- If the model does not define any periodic sample times, Simulink software chooses a step size that divides the total simulation time into 50 equal steps.

Dependencies

This parameter is enabled only if the **Periodic sample time constraint** is set to Unconstrained.

Command-Line Information

Parameter: FixedStep

Type: string

Value: any valid value

Default: 'auto'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Modeling Dynamic Systems
- Configuration Parameters Dialog Box
- Solver Pane

Sample time properties

Specify and assign priorities to the sample times that this model implements.

Settings

No Default

- Enter an Nx3 matrix with rows that specify the model's discrete sample time properties in order from fastest rate to slowest rate.
- Faster sample times must have higher priorities.

Format.

[period, offset, priority]

period	The time interval (sample rate) at which updates occur during the simulation.
offset	A time interval indicating an update delay. The block is updated later in the sample interval than other blocks operating at the same sample rate.
priority	Execution priority of the real-time task associated with the sample rate.

See Specifying Sample Time for more details and options for specifying sample time.

Example.

```
[[0.1, 0, 10]; [0.2, 0, 11]; [0.3, 0, 12]]
```

- Declares that the model should specify three sample times.
- Sets the fundamental sample time period to 0.1 second.
- Assigns priorities of 10, 11, and 12 to the sample times.
- Assumes higher priority values indicate lower priorities — the **Higher priority value indicates higher task priority** option is not selected.

Tips

- If the model's fundamental rate differs from the fastest rate specified by the model, specify the fundamental rate as the first entry in the matrix followed by the specified rates, in order from fastest to slowest. See “Purely Discrete Systems”.
- If the model operates at one rate, enter the rate as a three-element vector in this field — for example, [0.1, 0, 10].
- When you update a model, Simulink software displays an error message if what you specify does not match the sample times defined by the model.
- If **Periodic sample time constraint** is set to Unconstrained, Simulink software assigns priority 40 to the model base sample rate. If **Higher priority value indicates higher task priority** is selected, Simulink software assigns priorities 39, 38, 37, and so on, to subrates of the base rate. Otherwise, it assigns priorities 41, 42, 43, and so on, to the subrates.
- Continuous rate is assigned a higher priority than is the discrete base rate regardless of whether **Periodic sample time constraint** is Specified or Unconstrained.

Dependencies

This parameter is enabled by selecting Specified from the **Periodic sample time constraint** list.

Command-Line Information

Parameter: SampleTimeProperty

Type: structure

Value: any valid matrix

Default: []

Note If you specify `SampleTimeProperty` at the command line, you must enter the sample time properties as a structure with the following fields:

- `SampleTime`
 - `Offset`
 - `Priority`
-

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Period, offset, and priority of each sample time in the model; faster sample times must have higher priority than slower sample times

See Also

- “Purely Discrete Systems”
- Specifying Sample Time
- Configuration Parameters Dialog Box
- Solver Pane

Extrapolation order

Select the extrapolation order used by the `ode14x` solver to compute a model's states at the next time step from the states at the current time step.

Settings

Default: 4

- 1 Specifies first order extrapolation.
- 2 Specifies second order extrapolation.
- 3 Specifies third order extrapolation.
- 4 Specifies fourth order extrapolation.

Tip

Selecting a higher order produces a more accurate solution, but is more computationally intensive per step size.

Dependencies

This parameter is enabled by selecting `ode14x` (extrapolation) from the **Solver** list.

Command-Line Information

Parameter: `ExtrapolationOrder`

Type: integer

Value: 1 | 2 | 3 | 4

Default: 4

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Choosing a Fixed-Step Solver
- Configuration Parameters Dialog Box
- Solver Pane

Number Newton's iterations

Specify the number of Newton's method iterations used by the ode14x solver to compute a model's states at the next time step from the states at the current time step.

Settings

Default: 1

Minimum: 1

Maximum: 2147483647

More iterations produce a more accurate solution, but are more computationally intensive per step size.

Dependencies

This parameter is enabled by selecting ode14x (extrapolation) from the Solver list.

Command-Line Information

Parameter: NumberNewtonIterations

Type: integer

Value: any valid number

Default: 1

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Choosing a Fixed-Step Solver
- Configuration Parameters Dialog Box
- Solver Pane
- “Purely Discrete Systems”

Data Import/Export Pane

The screenshot shows the Data Import/Export Pane interface, which is organized into three main sections:

- Load from workspace:** Contains two checkboxes, both unchecked. The first is labeled "Input:" with an empty text field. The second is labeled "Initial state:" with an empty text field.
- Save to workspace:** Contains several checkboxes and text fields. "Time:" is checked with the value "tout". "States:" is unchecked with the value "xout". "Output:" is checked with the value "yout". "Final states:" is checked with the value "xFinal", and there is an unchecked checkbox for "Save complete SimState in final state". "Signal logging:" is checked with the value "logout", and there is an unchecked checkbox for "Inspect signal logs when simulation is paused/stopped". "Data stores:" is checked with the value "dsmout".
- Save options:** Contains several controls. "Limit data points to last:" is unchecked with the value "1000". "Decimation:" is a text field with the value "1". "Format:" is a dropdown menu set to "Array". "Output options:" is a dropdown menu set to "Refine output". "Refine factor:" is a text field with the value "1". "Return as single object:" is unchecked with the value "out".

In this section...

“Data Import/Export Overview” on page 1-81

“Input” on page 1-82

“Initial state” on page 1-84

“Time” on page 1-86

“States” on page 1-88

“Output” on page 1-90

“Final states” on page 1-92

“Save complete SimState in final state” on page 1-94

“Signal logging” on page 1-97

In this section...

“Inspect signal logs when simulation is paused/stopped” on page 1-99

“Data stores” on page 1-101

“Limit data points to last” on page 1-102

“Decimation” on page 1-105

“Format” on page 1-107

“Output options” on page 1-109

“Refine factor” on page 1-111

“Output times” on page 1-113

“Return as Single Object” on page 1-114

Data Import/Export Overview

The Data Import/Export pane allows you to import input signal and initial state data from a workspace and export output signal and state data to the MATLAB® workspace during simulation. This capability allows you to use standard or custom MATLAB functions to generate a simulated system's input signals and to graph, analyze, or otherwise postprocess the system's outputs.

Configuration

- 1 Specify the data to load from a workspace before simulation begins.
- 2 Specify the data to save to the MATLAB workspace after simulation completes.

Tips

- For more information on using this pane, see [Importing and Exporting Simulation Data](#).
- See the documentation of the `sim` command for some capabilities that are available only for programmatic simulation.

See Also

- [Importing Data from a Workspace](#)
- [Exporting Data to the MATLAB Workspace](#)
- [Configuration Parameters Dialog Box](#)
- [Data Import/Export Pane](#)

Input

Loads input data from a workspace before the simulation begins.

Settings

Default: Off, [t,u]



On

Loads data from a workspace.

Specify a MATLAB expression for the data to be imported from a workspace. The Simulink software resolves symbols used in this specification as described in “Resolving Symbols”. The input data can take any of the following forms:

- Time series
- Data array
- Time expression
- Data structure

See Importing Data from a Workspace for information on how to use this field.



Off

Does not load data from a workspace.

Tips

- You must select the **Input** check box before entering input data.
- Simulink software linearly interpolates or extrapolates input values as necessary if the Interpolate data option is selected for the corresponding Inport.
- The use of the **Input** box is independent of the setting of the **Format** list on the **Data Import/Export** pane.

Command-Line Information

Parameter: LoadExternalInput

Type: string

Value: 'on' | 'off'

Default: 'off'

Parameter: ExternalInput

Type: scalar or vector

Value: any valid value

Default: [t,u]

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Importing Data from a Workspace
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Initial state

Loads the model's initial states from a workspace before simulation begins.

Settings

Default: Off, xInitial



On

Simulink software loads initial states from a workspace.

Specify the name of a variable that contains the initial state values, for example, a variable containing states saved from a previous simulation.

Use the structure or structure-with-time option to specify initial states if you want to accomplish any of the following:

- Associate initial state values directly with the full path name to the states. This eliminates errors that could occur if Simulink software reorders the states, but the initial state array is not correspondingly reordered.
- Assign a different data type to each state's initial value.
- Initialize only a subset of the states.

See [Importing and Exporting States](#) for more information.



Off

Simulink software does not load initial states from a workspace.

Tips

- You must select the **Initial State** check box before entering initial state data.
- The initial values specified by the workspace variable override the initial values specified by the model itself (the values specified by the initial condition parameters of those blocks in the model that have states).
- You must use the structure or structure-with-time format to initialize the states of a top model and the models that it references.

Command-Line Information

Parameter: LoadInitialState

Type: string

Value: 'on' | 'off'

Default: 'off'

Parameter: InitialState

Type: variable (string) or vector

Value: any valid value

Default: 'xInitial'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Importing Data from a Workspace
- Importing and Exporting States
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Time

Saves simulation time data to the specified variable during simulation.

Settings

Default: On, tout



On

Simulink software exports time data to the MATLAB workspace during simulation.

Specify the name of the MATLAB variable used to store time data. See [Exporting Data to the MATLAB Workspace](#) for more information.



Off

Simulink software does not export time data to the MATLAB workspace during simulation.

Tips

- You must select the **Time** check box before entering the time variable.
- Simulink software saves the output to the MATLAB workspace at the base sample rate of the model. Use a To Workspace block if you want to save output at a different sample rate.
- The **Save options** area enables you to specify the format and restrict the amount of output saved.

Command-Line Information

Parameter: SaveTime

Type: string

Value: 'on' | 'off'

Default: 'on'

Parameter: TimeSaveName

Type: string

Value: any valid value

Default: 'tout'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Exporting Data to the MATLAB Workspace
- Configuration Parameters Dialog Box
- Data Import/Export Pane

States

Saves state data to the specified MATLAB variable during a simulation.

Settings

Default: Off, xout



On

Simulink software exports state data to the MATLAB workspace during simulation.

Specify the name of the MATLAB variable used to store state data. See [Importing and Exporting States](#) for more information.



Off

Simulink does not export state data during simulation.

Tips

- You must select the **States** check box before entering the states variable.
- Simulink saves the states in a MATLAB workspace variable having the specified name.
- The saved data has the format that you specify in the **Save options** area.
- If you select the **States** check box, Simulink does not log fixed-point states.
- Simulink creates empty variables for state logging (xout) if both of these conditions apply:
 - You enable **States**.
 - A model has no states.

Command-Line Information

Parameter: SaveState

Type: string

Value: 'on' | 'off'

Default: 'off'

Parameter: StateSaveName

Type: string

Value: any valid value

Default: 'xout'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Importing and Exporting States
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Output

Saves signal data to the specified MATLAB variable during simulation.

Settings

Default: On, you



On

Simulink software exports signal data to the MATLAB workspace during simulation.

Specify the name of the MATLAB variable used to store signal data. See [Exporting Data to the MATLAB Workspace](#) for more information.



Off

Simulink software does not export signal data during simulation.

Tips

- You must select the **Output** check box before entering the output variable.
- Simulink software saves the output to the MATLAB workspace at the base sample rate of the model. Use a To Workspace block if you want to save output at a different sample rate.
- The **Save options** area enables you to specify the format and restrict the amount of output saved.
- If you select the **Output** check box, Simulink logs fixed-point data as double.

Command-Line Information

Parameter: SaveOutput

Type: string

Value: 'on' | 'off'

Default: 'on'

Parameter: OutputSaveName

Type: string

Value: any valid value

Default: 'yout'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Exporting Data to the MATLAB Workspace
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Final states

Saves the logged states of the model at the end of a simulation to the specified MATLAB variable.

Settings

Default: Off, xFinal



On

Simulink software exports final logged state data to the MATLAB workspace during simulation.

Specify the name of the MATLAB variable in which to store the values of these final states. See [Importing and Exporting States](#) for more information.



Off

Simulink software does not export the final state data during simulation.

Tips

- You must select the **Final states** check box before entering the final states variable.
- Simulink software saves the final states in a MATLAB workspace variable having the specified name.
- The saved data has the format that you specify in the **Save options** area.
- Simulink creates empty variables for final state logging (xfinal) if both of these conditions apply:
 - You enable **Final states**.
 - A model has no states.

Command-Line Information

Parameter: SaveFinalState

Type: string

Value: 'on' | 'off'

Default: 'off'

Parameter: FinalStateName

Type: string

Value: any valid value

Default: 'xFinal'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Importing and Exporting States
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Save complete SimState in final state

At the end of a simulation, Simulink saves the complete set of states of the model, including logged states, to the specified MATLAB variable.

Settings

Default: Off, xFinal



On

Simulink software exports the complete set of final state data (i.e., the SimState) to the MATLAB workspace during simulation.

Specify the name of the MATLAB variable in which to store the values of the final states. See [Importing and Exporting States](#) for more information.



Off

Simulink software exports the final logged states during simulation.

Tips

- You must select the **Final states** check box to enable the **Save complete SimState in final state** option.
- Simulink saves the final states in a MATLAB workspace variable having the specified name.
- The saved data has the format that you specify in the **Save options** area.

Dependencies

This parameter is enabled by **Final states**.

Command-Line Information

Parameter: SaveCompleteFinalSimState

Type: string

Value: 'on' | 'off'

Default: 'off'

Parameter: FinalStateName

Type: string
Value: any valid value
Default: 'xFinal'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Importing and Exporting States
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Signal logging

Globally enable or disable signal logging for this model.

Settings

Default: On, logstdout



On

Enables signal logging to the MATLAB workspace during simulation.

Specify the name of the signal logging object used to record logged signal data in the MATLAB workspace. See Logging Signals for more information.



Off

Disables signal logging to the MATLAB workspace during simulation.

Tips

- You must select the **Signal logging** check box before entering the signal logging variable.
- Simulink software saves the signal data in a MATLAB workspace variable having the specified name.
- The saved data has the format that you specify in the **Save options** area.
- Simulink software does not support signal logging for the following types of signals:
 - Output of a Function-Call Generator block
 - Signal connected to the input of a Merge block
 - Outputs of Trigger and Enable blocks

Dependencies

This parameter enables **Inspect signal logs when simulation is paused/stopped**.

Command-Line Information

Parameter: SignalLogging

Type: string

Value: 'on' | 'off'

Default: 'on'

Parameter: SignalLoggingName

Type: string

Value: any valid value

Default: 'logout'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Logging Signals
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Inspect signal logs when simulation is paused/stopped

Specify whether Simulink software displays logged signals in the Simulation Data Inspector at the end of a simulation or whenever you pause the simulation.

Settings

Default: Off



On

Simulink displays logged signals in the Simulation Data Inspector at the end of a simulation or whenever you pause the simulation.



Off

Simulink does not display logged signals at the end of a simulation.

Tip

If this option is off, in the Model Editor you can select **Tools > Inspect Logged Signals** to open the Simulation Data Inspector.

Dependencies

This parameter is enabled by “Signal logging” on page 1-97.

Command-Line Information

Parameter: InspectSignalLogs

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Logging Signals
- “Understanding the Simulation Data Inspector Interface”
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Data stores

Globally enable or disable logging of Data Store Memory block variables for this model.

Settings

Default: On, dsmsout



On

Enables data store logging to the MATLAB workspace during simulation.

Specify the name of the data store logging object to use for recording logged data store data. The data store logging object must be in the MATLAB workspace.



Off

Disables data store logging to the MATLAB workspace during simulation.

Tips

- Simulink saves the data in a MATLAB workspace variable having the specified name.
- The saved data has the format that you specify in the **Save options** area.
- See “Supported Data Types, Dimensions, and Complexity for Logging Data Stores” and “Data Store Logging Limitations”.

Dependencies. Select the **Data stores** check box before entering the data store logging variable.

Command-Line Information

Parameter: DSMLogging

Type: string

Value: 'on' | 'off'

Default: 'on'

Parameter: DSMLoggingName

Type: string
Value: any valid value
Default: 'dsmOut'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- “Logging Data Stores”
- Logging Signals
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Limit data points to last

Limit the number of data points to export to the MATLAB workspace.

Settings

Default: On, 1000

On

Limits the number of data points exported to the MATLAB workspace to the number that you specify.

Specify the maximum number of data points to export to the MATLAB workspace. At the end of the simulation, the MATLAB workspace contains the last N points generated by the simulation.



Off

Does not limit the number of data points.

Tips

- You must select the **Limit data points to last** check box before specifying the number of data points.
- Saving data to the MATLAB workspace can slow down the simulation and consume memory. Use this parameter to limit the number of samples saved to help avoid this problem.
- You can also apply a **Decimation** factor to skip a selected number of samples.

Command-Line Information

Parameter: LimitDataPoints

Type: string

Value: 'on' | 'off'

Default: 'on'

Parameter: MaxDataPoints

Type: string

Value: any valid value

Default: '1000'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Exporting Data to the MATLAB Workspace
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Decimation

Specify that Simulink software output only every N points, where N is the specified decimation factor.

Settings

Default: 1

- The default value (1) specifies that all data points are saved.
- Simulink software outputs data only at the specified number of data points. For example, specifying 2 saves every other data point, while specifying 10 saves just one in ten data points.
- At the end of the simulation, the total number of data points is reduced by the factor specified.

Tips

- Saving data to the MATLAB workspace can slow down the simulation and consume memory. Use this parameter to limit the number of samples saved to help avoid this problem.
- You can also use the **Limit data points to last** parameter to help resolve this problem.

Command-Line Information

Parameter: Decimation

Type: string

Value: any valid value

Default: '1'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Exporting Data to the MATLAB Workspace
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Format

Select the format of state and output data saved to the MATLAB workspace.

Settings

Default: Array

Array

The format of the data is a matrix each row of which corresponds to a simulation time step.

Structure

The format of the data is a structure that contains substructures for each port. Each port substructure contains signal data for the corresponding port.

Structure with time

The format of the data is a structure that has two fields: a time field and a signals field. The time field contains a vector of simulation times. The signals field contains a substructure for each model input port (for imported data) or output port (for exported data). Each port substructure contains signal data for the corresponding port.

Tips

- You can use array format to save your model's outputs and states only if the outputs are either all scalars or all vectors (or all matrices for states), are either all real or all complex, and are all of the same data type. Use the Structure or Structure with time output formats (see Structure with time) if your model's outputs and states do not meet these conditions.
- Simulink software can read back simulation data saved to the workspace in the Structure with time output format. See Importing Signal-and-Time Data Structures for more information.

Command-Line Information

Parameter: SaveFormat

Type: string

Value: 'Array' | 'Structure' | 'StructureWithTime'

Default: 'Array'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Exporting Data to the MATLAB Workspace
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Output options

Select options for generating additional output signal data for variable-step solvers.

Settings

Default: Refine output

Refine output

Generates data output between as well as at simulation times steps.

Use **Refine factor** to specify the number of points to generate between simulation time steps. For more information, see Refining Output.

Produce additional output

Generates additional output at specified times. Use **Output times** to specify the simulation times at which Simulink software should generate additional output.

Produce specified output only

Generates output only at specified times. Use **Output times** to specify the simulation times at which Simulink software should generate output.

Tips

- These settings can force the solver to calculate output values for times that it would otherwise have omitted because the calculations were not needed to achieve accurate simulation results. These extra calculations can cause the solver to locate zero crossings that it would otherwise have missed.
- For additional information on how Simulink software calculates outputs for these three options, see Specifying Output Options.

Dependencies

This parameter is enabled only if the model specifies a variable-step solver (see Solver Type).

Selecting Refine output enables the **Refine factor** parameter.

Selecting Produce additional output or Produce specified output only enables the **Output times** parameter.

Command-Line Information

Parameter: OutputOption

Type: string

Value: 'RefineOutputTimes' | 'AdditionalOutputTimes' | 'SpecifiedOutputTimes'

Default: 'RefineOutputTimes'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Specifying Output Options
- Refine factor
- Refining Output
- Exporting Data to the MATLAB Workspace
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Refine factor

Specify how many points to generate between time steps to refine the output.

Settings

Default: 1

- The default refine factor is 1, meaning that no extra data points are generated.
- A refine factor of 2 provides output midway between the time steps, as well as at the steps.

Tip

Simulink software ignores this option for discrete models. This is because the value of data between time steps is undefined for discrete models.

Dependency

This parameter is enabled only if you select **Refine** output as the value of **Output options**.

Command-Line Information

Parameter: Refine
Type: string
Value: any valid value
Default: '1'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Refining Output
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Output times

Specify times at which Simulink software should generate output in addition to, or instead of, the times of the simulation steps taken by the solver used to simulate the model.

Settings

Default: []

- Enter a matrix containing the times at which Simulink software should generate output in addition to, or instead of, the simulation steps taken by the solver.
- If the value of **Output options** is `Produce additional output`, the default value [] specifies that no additional data points are generated.
- If the value of **Output options** is `Produce specified output only`, the default value [] specifies that no data points are generated.

Tips

- The `Produce additional output` option generates output at the specified times as well as the regular simulation steps.
- The `Produce specified output only` option generates output only at the specified times.
- Discrete models define outputs only at major time steps. Therefore, Simulink software logs output for discrete models only at major time steps. If the **Output times** field specifies other times, Simulink software displays a warning at the MATLAB command line.

Dependency

This parameter is enabled only if the value of **Output options** is `Produce additional output` or `Produce specified output only`.

Command-Line Information

Parameter: OutputTimes

Type: string

Value: any valid value

Default: '[]'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development Off for production code generation

See Also

- Refining Output
- Configuration Parameters Dialog Box
- Data Import/Export Pane

Return as Single Object

Enable the single-output format of the sim command.

Settings

Default: off

- Turning this option 'on' causes Simulink to return all simulation outputs within a single Simulink.SimulationOutput object, providing that you simulate by choosing **Simulation > Start** from the model window.
- Enabling this option makes the sim command compatible with the parfor command, in terms of transparency issues.

Tips

- If you select this option and you simulate by entering the `sim` command at the command line of the MATLAB command window, then the output variables will not be stored in the object `'out'`. Instead, they will be stored in their respective variable names. This design is necessary to avoid workspace issues when `sim` is called from within a `parfor` loop.
- The method `who` of the `Simulink.SimulationOutput` object returns the list of variables that the object contains.
- Use the `get` method of the `Simulink.SimulationOutput` object to access the variables that the object contains.

Command-Line Information

Parameter: `ReturnWorkspaceOutputs`

Type: `string`

Value: `'on'` | `'off'` |

Default: `'off'`

Recommended Settings

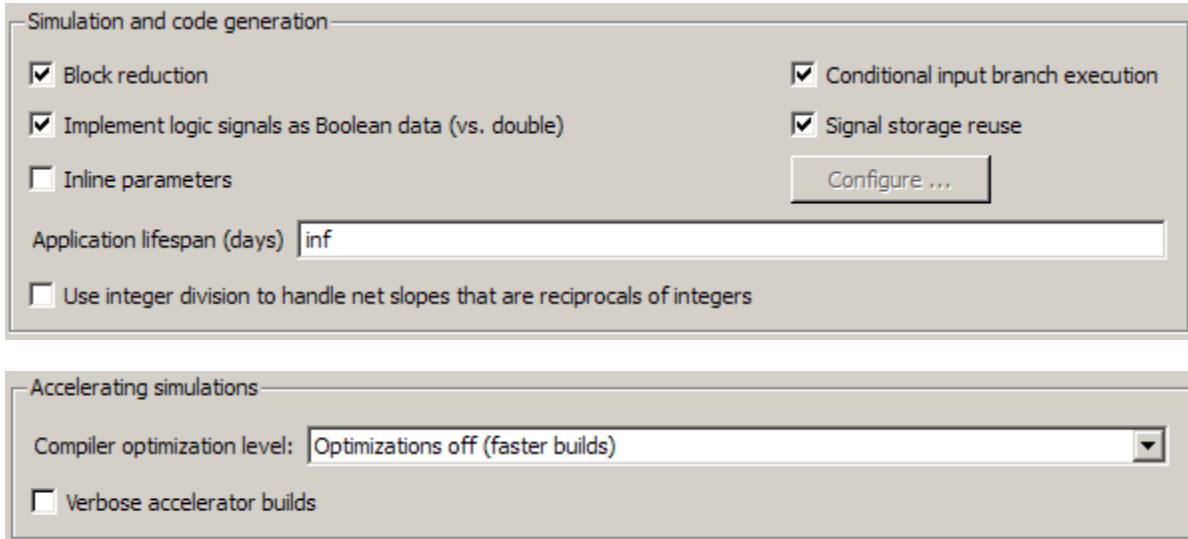
Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- “Configuration Parameters Dialog Box Overview” on page 1-2
- “Data Import/Export Pane” on page 1-79
- “Using the `sim` Command”
- “Running Parallel Simulations”

Optimization Pane

The Optimization pane includes the following parameters:



Simulation and code generation

- Block reduction
- Implement logic signals as Boolean data (vs. double)
- Inline parameters
- Conditional input branch execution
- Signal storage reuse

Application lifespan (days)

Use integer division to handle net slopes that are reciprocals of integers

Configure ...

Accelerating simulations

Compiler optimization level:

Verbose accelerator builds

The Optimization pane includes the following parameters when the Real-Time Workshop product is installed on your system and you select a GRT-based target.

Code generation

Signals

- Enable local block outputs
- Reuse block outputs
- Inline invariant signals
- Eliminate superfluous local variables (Expression folding)
- Minimize data copies between local and global variables

Loop unrolling threshold: Maximum stack size (bytes):

Use memcpy for vector assignment Memcpy threshold (bytes):

Data initialization

- Use memset to initialize floats and doubles to 0.0

Integer and fixed-point

- Remove code from floating-point to integer conversions that wraps out-of-range values
- Remove code from floating-point to integer conversions with saturation that maps NaN to zero

The Optimization pane includes the following parameters when the Real-Time Workshop product is installed on your system and you select an ERT-based target. ERT-based target optimizations require a Real-Time Workshop® Embedded Coder™ license when generating code.

Code generation

Parameter structure: Hierarchical

Optimize using the specified minimum and maximum values

Signals

Enable local block outputs Reuse block outputs

Inline invariant signals

Eliminate superfluous local variables (Expression folding) Simplify array indexing

Minimize data copies between local and global variables

Pack Boolean data into bitfields

Loop unrolling threshold: 5 Maximum stack size (bytes): Inherit from target

Use memcpy for vector assignment Memcpy threshold (bytes): 64

Pass reusable subsystem outputs as: Structure reference

Data initialization

Remove root level I/O zero initialization Use memset to initialize floats and doubles to 0.0

Remove internal data zero initialization Optimize initialization code for model reference

Integer and fixed-point

Remove code from floating-point to integer conversions that wraps out-of-range values

Remove code from floating-point to integer conversions with saturation that maps NaN to zero

Remove code that protects against division arithmetic exceptions

The Optimization pane includes the following parameters when Real-Time Workshop and Stateflow® products are both installed on your system. Your model need not include Stateflow charts or Truth Table blocks.

Stateflow

 Use bitsets for storing state configuration Use bitsets for storing Boolean data**In this section...**

“Optimization Overview” on page 1-121

“Block reduction” on page 1-122

“Conditional input branch execution” on page 1-126

“Inline parameters” on page 1-128

“Implement logic signals as Boolean data (vs. double)” on page 1-131

“Signal storage reuse” on page 1-133

“Application lifespan (days)” on page 1-135

“Use integer division to handle net slopes that are reciprocals of integers” on page 1-138

“Parameter structure” on page 1-140

“Optimize using the specified minimum and maximum values” on page 1-142

“Enable local block outputs” on page 1-145

“Eliminate superfluous local variables (Expression folding)” on page 1-148

“Minimize data copies between local and global variables” on page 1-150

“Simplify array indexing” on page 1-152

“Reuse block outputs” on page 1-154

“Inline invariant signals” on page 1-157

“Pack Boolean data into bitfields” on page 1-159

“Bitfield declarator type specifier” on page 1-161

“Loop unrolling threshold” on page 1-163

“Maximum stack size (bytes)” on page 1-165

“Use memcopy for vector assignment” on page 1-166

“Memcpy threshold (bytes)” on page 1-168

In this section...

“Pass reusable subsystem outputs as” on page 1-170

“Remove root level I/O zero initialization” on page 1-172

“Use memset to initialize floats and doubles to 0.0” on page 1-174

“Remove internal data zero initialization” on page 1-176

“Optimize initialization code for model reference” on page 1-178

“Remove code from floating-point to integer conversions that wraps out-of-range values” on page 1-180

“Remove code from floating-point to integer conversions with saturation that maps NaN to zero” on page 1-182

“Remove code that protects against division arithmetic exceptions” on page 1-184

“Use bitsets for storing state configuration” on page 1-186

“Use bitsets for storing Boolean data” on page 1-188

“Model Parameter Configuration Dialog Box” on page 1-190

“Compiler optimization level” on page 1-192

“Verbose accelerator builds” on page 1-194

Optimization Overview

Set up optimizations for a model's active configuration set. Optimizations are set for both simulation and code generation.

Configuration

Set the parameters displayed.

Tips

- Real-Time Workshop optimizations appear only when the Real-Time Workshop product is installed on your system. Selecting a GRT-based or ERT-based system target file changes the available options. ERT-based target optimizations require a Real-Time Workshop Embedded Coder license when generating code.
- Stateflow software optimizations appear only when Real-Time Workshop and Stateflow products are both installed on your system and the model includes Stateflow charts or Embedded MATLAB® Function blocks. The settings you make for the Stateflow software options also apply to all Embedded MATLAB Function blocks in the model. You do not need a Stateflow license to use Embedded MATLAB Function blocks.

See Also

- [Optimizing a Model for Code Generation](#)
- [Configuration Parameters Dialog Box](#)
- [Optimization Pane](#)
- [“Performing Acceleration”](#)

Block reduction

Reduce execution time by collapsing or removing groups of blocks.

Settings

Default: On



Simulink software searches for and reduces the following block patterns:

- **Accumulators** — A group consisting of a Constant block, a Sum block, and feedback through a Unit Delay block.
- **Redundant type conversions** — Unnecessary type conversion blocks, such as an int type conversion block with an input and output of type int.
- **Dead code** — Blocks or signals in an unused code path.
- **Fast-to-slow Rate Transition block in a single-tasking system** — Rate Transition blocks with an input frequency faster than its output frequency.



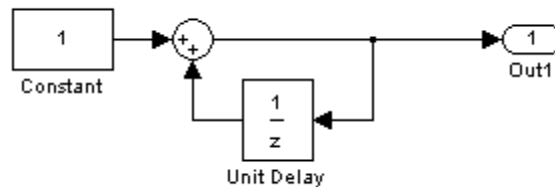
Simulink software does not search for block patterns that can be optimized. Simulation and generated code are not optimized.

Tips

- When you select **Block reduction**, Simulink software collapses certain groups of blocks into a single, more efficient block, or removes them entirely. This results in faster execution during model simulation and in generated code.
- Block reduction does not change the appearance of the source model.
- Tunable parameters do not prevent a block from being reduced by dead code elimination.
- Once block reduction takes place, Simulink software does not display the sorted order for blocks that have been removed.

- Block reduction is intended to remove only the generated code that represents execution of a block. Other supporting data, such as definitions for sample time and data types might remain in the generated code.

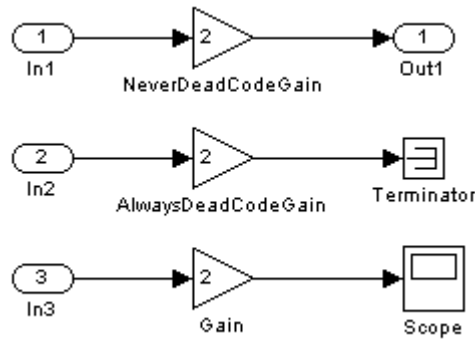
Accumulators. Simulink software recognizes the block diagram shown in the following figure as an accumulator:



- An accumulator construct is recognized anywhere across a block diagram, or within subsystems at lower levels.
- With the **Block reduction** option enabled, Simulink software creates a synthesized block, `Sum_synth_accum`. The synthesized block replaces the previous block diagram, resulting in a simple increment calculation.

Dead Code Elimination. Any blocks or signals in an *unused code path* are eliminated from generated code.

- The following conditions need to be met for a block to be considered part of an unused code path:
 - All signal paths for the block end with a block that does not execute. Examples of blocks that do not execute include Terminator blocks, disabled Assertion blocks, S-Function blocks configured for block reduction, and To Workspace blocks when MAT-file logging is disabled for code generation.
 - No signal paths for the block include global signal storage downstream from the block.
- Tunable parameters do not prevent a block from being reduced by dead code elimination.
- Consider the signal paths in the following block diagram.



If you check **Block reduction**, Real-Time Workshop software responds to each signal path as follows:

For Signal Path...	Real-Time Workshop Software...
In1 to Out1	Always generates code because dead code elimination conditions are not met.
In2 to Terminator	Never generates code because dead code elimination conditions are met.
In3 to Scope	Generates code if MAT-file logging is enabled and eliminates code if MAT-file logging is disabled.

Command-Line Information

Parameter: BlockReduction

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	Off (for simulation and during development) No impact (for production code generation)
Traceability	Off
Efficiency	On
Safety precaution	Off

See Also

- Block Reduction
- Single-Tasking Execution
- Optimizing a Model for Code Generation
- Configuration Parameters Dialog Box
- Optimization Pane

Conditional input branch execution

Improve model execution when the model contains Switch and Multiport Switch blocks.

Settings

Default: On



On

Executes only the blocks required to compute the control input and the data input selected by the control input. This optimization speeds execution of code generated from the model. Limits to Switch block optimization:

- Only blocks with -1 (inherited) or `inf` (Constant) sample time can participate.
- Blocks with outputs flagged as test points cannot participate.
- No multirate block can participate.
- Blocks with states cannot participate.
- Only S-functions with option `SS_OPTION_CAN_BE_CALLED_CONDITIONALLY` set can participate.



Off

Executes all blocks driving the Switch block input ports at each time step.

Command-Line Information

Parameter: `ConditionallyExecuteInputs`

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	On
Efficiency	On (execution), No impact (ROM, RAM)
Safety precaution	Off

See Also

- Expression Folding
- Conditional Input Execution
- Optimizing a Model for Code Generation
- Configuration Parameters Dialog Box
- Optimization Pane

Inline parameters

Transform tunable parameters into constant values.

Settings

Default: Off



Selecting **Inline parameters** has the following effects:

- Real-Time Workshop software uses the numerical values of model parameters, instead of their symbolic names, in generated code.
- Reduces global RAM usage, because parameters are not declared in the global parameters structure.
- Enables the **Configure** button. Clicking the **Configure** button opens the Model Parameter Configuration dialog box.



Uses symbolic names for model parameters in generated code.

Tips

- Simulink software allows you to override the **Inline parameters** option for parameters whose values are defined by variables in the MATLAB workspace. To specify that such a parameter remain tunable, specify the parameter as global in the Model Configuration Parameters dialog box (see Model Parameter Configuration Dialog Box). To display the dialog box, click the adjacent **Configure** button.
- To tune a global parameter, change the value of the corresponding workspace variable and select **Update Diagram (Ctrl+D)** from the Simulink **Edit** menu.
- You cannot tune inlined parameters in code generated from a model. However, when simulating a model, you can tune an inlined parameter if its value derives from a workspace variable. For example, suppose that a model has a Gain block whose **Gain** parameter is inlined and equals a , where a is a variable defined in the model's workspace. When simulating the model, Simulink software disables the **Gain** parameter field, preventing you from using the block's dialog box to change the gain.

However, you can still tune the gain by changing the value of a at the MATLAB command line and updating the diagram.

- When a top model uses referenced models:
 - All referenced models must specify **Inline parameters** to be on.
 - The top model can specify **Inline parameters** to be on or off.

See Inline Parameter Requirements for more information.

- If your model contains an Environment Controller block, you can suppress code generation for the branch connected to the Sim port if you select **Inline parameters** and the branch does not contain external signals.
- Simulink Scope and Signal Viewer blocks with Constant and Ground blocks — If you turn on inline parameters, the value of a Constant or Ground block does not change during the simulation. The constant value is determined before a simulation and plotted on a scope graph as a single point. To see a scope trace, change the sample time for a Constant or Ground block from `inf` to a value.

Dependencies

This parameter enables:

- **Configure** button
- “**Parameter structure**” on page 1-140
- “**Inline invariant signals**” on page 1-157

Command-Line Information

Parameter: `InlineParams`

Type: `string`

Value: `'on' | 'off'`

Default: `'off'`

Recommended Settings

Application	Setting
Debugging	Off (for simulation and during development) On (for production code generation)
Traceability	On
Efficiency	On
Safety precaution	No impact

See Also

- Model Parameter Configuration Dialog Box
- Parameter Storage, Interfacing, and Tuning
- Model Referencing Inline Parameters
- Optimizing a Model for Code Generation
- Configuration Parameters Dialog Box
- Optimization Pane

Implement logic signals as Boolean data (vs. double)

Controls the output data type of blocks that generate logic signals.

Settings

Default: On



On

Blocks that generate logic signals output a signal of **boolean** data type. This reduces the memory requirements of generated code.



Off

Blocks that generate logic signals output a signal of **double** data type. This ensures compatibility with models created by earlier versions of Simulink software.

Tips

- Setting this option **on** reduces the memory requirements of generated code, because a Boolean signal typically requires one byte of storage compared to eight bytes for a double signal.
- Setting this option **off** allows the current version of Simulink software to run models that were created by earlier versions of Simulink software that supported only signals of type **double**.
- This optimization affects the following blocks:
 - **Logical Operator block** – This parameter affects only those Logical Operator blocks whose **Output data type** parameter specifies **Logical**. If this parameter is selected, such blocks output a signal of **boolean** data type; otherwise, such blocks output a signal of **double** data type.
 - **Relational Operator block** – This parameter affects only those Relational Operator blocks whose **Output data type** parameter specifies **Logical**. If this parameter is selected, such blocks output a signal of **boolean** data type; otherwise, such blocks output a signal of **double** data type.
 - **Combinatorial Logic block** – If this parameter is selected, Combinatorial Logic blocks output a signal of **boolean** data type;

otherwise, they output a signal of `double` data type. See Combinatorial Logic in the *Simulink Reference* for an exception to this rule.

- **Hit Crossing block** – If this parameter is selected, Hit Crossing blocks output a signal of `boolean` data type; otherwise, they output a signal of `double` data type.

Dependencies

- This parameter is disabled for models created with a version of Simulink software that supports only signals of type `double`.

Command-Line Information

Parameter: `BooleanDataType`
Type: `string`
Value: `'on' | 'off'`
Default: `'on'`

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	On
Safety precaution	On

See Also

- [Optimizing a Model for Code Generation](#)
- [Configuration Parameters Dialog Box](#)
- [Optimization Pane](#)

Signal storage reuse

Reuse signal memory.

Settings

Default: On



On

Simulink software reuses memory buffers allocated to store block input and output signals, reducing the memory requirement of your real-time program.



Off

Simulink software allocates a separate memory buffer for each block's outputs. This makes all block outputs global and unique, which in many cases significantly increases RAM and ROM usage.

Tips

- This option applies only to signals with storage class Auto.
- Signal storage reuse can occur only among signals that have the same data type.
- Clearing this option can substantially increase the amount of memory required to simulate large models.
- Clear this option if you need to:
 - Debug a C-MEX S-function
 - Use a Floating Scope or a Display block with the **Floating display** option selected to inspect signals in a model that you are debugging
- Simulink software opens an error dialog if **Signal storage reuse** is enabled and you attempt to use a Floating Scope or floating Display block to display a signal whose buffer has been reused.

Dependencies

This parameter enables:

- “Enable local block outputs” on page 1-145

- “Reuse block outputs” on page 1-154
- “Eliminate superfluous local variables (Expression folding)” on page 1-148
- “Minimize data copies between local and global variables” on page 1-150

Command-Line Information

Parameter:OptimizeBlockIOStorage

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	Off
Traceability	Off
Efficiency	On
Safety precaution	No impact

See Also

- Signal Storage, Optimization, and Interfacing
- Optimizing a Model for Code Generation
- Configuration Parameters Dialog Box
- Optimization Pane

Application lifespan (days)

Specify how long (in days) an application that contains blocks depending on elapsed or absolute time should be able to execute before timer overflow.

Settings

Default: `inf`

Min: Must be greater than zero

Max: `inf`

Enter a positive (nonzero) scalar value (for example, `0.5`) or `inf`.

If you are licensed for the Real-Time Workshop Embedded Coder product and select an ERT target for your model, the default value for **Application lifespan (days)** is 1.

This parameter is ignored when you are operating your model in external mode, have **Mat-file logging** enabled, or have a continuous sample time because a 64 bit timer is required in these cases.

Tips

- Specifying a lifespan, along with the simulation step size, determines the data type used by blocks to store absolute time values.
- For simulation, setting this parameter to a value greater than the simulation time will ensure time does not overflow.
- Simulink software evaluates this parameter first against the model workspace. If this does not resolve the parameter, Simulink software then evaluates it against the base workspace.
- The Application lifespan also determines the word size used by timers in the generated code, which can lower RAM usage. For more information, see *Timing Services* in the Real-Time Workshop documentation.
- Application lifespan, when combined with the step size of each task, determines the data type used for integer absolute time for each task, as follows:
 - If your model does not require absolute time, this option affects neither simulation nor the generated code.

- If your model requires absolute time, this option optimizes the word size used for storing integer absolute time in generated code. This ensures that timers do not overflow within the lifespan you specify. If you set **Application lifespan** to `inf`, two `uint32` words are used.
- If your model contains fixed-point blocks that require absolute time, this option affects both simulation and generated code.

For example, using 64 bits to store timing data enables models with a step size of 0.001 microsecond (10E-09 seconds) to run for more than 500 years, which would rarely be required. To run a model with a step size of one millisecond (0.001 seconds) for one day would require a 32-bit timer (but it could continue running for 49 days).

- A timer will allocate 64 bits of memory if you specify a value of `inf`.
- To minimize the amount of RAM used by time counters, specify a lifespan no longer than necessary.
- Must be the same for top and referenced models.
- Optimize the size of counters used to compute absolute and elapsed time.

Command-Line Information

Parameter: LifeSpan

Type: string

Value: positive (nonzero) scalar value or `inf`

Default: 'inf'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	Finite value
Safety precaution	<code>inf</code>

See Also

- [Timing Services](#)
- [Using Timers in Asynchronous Tasks](#)
- [Optimizing a Model for Code Generation](#)
- [Configuration Parameters Dialog Box](#)
- [Optimization Pane](#)

Use integer division to handle net slopes that are reciprocals of integers

The Simulink® Fixed Point™ software performs net slope correction using integer division to handle net slopes that are reciprocals of integers when simplicity and accuracy conditions are met.

Settings

Default: Off

- On
Perform net slope correction using integer division when simplicity and accuracy conditions are met.
- Off
Perform net slope correction using integer multiplication followed by shifts.

Tips

- This optimization affects both simulation and code generation.
- When a change of fixed-point slope is not a power of two, net slope correction is necessary. Normally, net slope correction uses an integer multiplication followed by shifts. Enabling this new optimization replaces the multiplication and shifts with an integer division under certain simplicity and accuracy conditions.
- Performing net slope correction using integer division is not always more efficient than using multiplication followed by shifts. Ensure that the target hardware supports efficient division.
- To ensure that this optimization occurs, you must:
 - Set the word length of the block to ensure that the software can perform division using the production target long data type. This avoids using multiword operations.
 - Set the **Signed integer division rounds to** configuration parameter setting on the **Hardware Implementation > Embedded Hardware** subpane to **Zero** or **Floor**. The optimization does not occur if this parameter is set to **Undefined**.

- Set the **Integer rounding mode** parameter of the block to **Simplest** or to the value of the **Signed integer division rounds to** configuration parameter setting on the **Hardware Implementation > Embedded Hardware** subpane.

Dependency

This parameter requires a Simulink Fixed Point license.

Command-Line Information

Parameter: UseIntDivNetSlope

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	On (when target hardware supports efficient division) Off (otherwise)
Safety precaution	No impact

See Also

- Use Integer Division for Net Slope Correction
- Configuration Parameters Dialog Box
- Optimization Pane

Parameter structure

Control how parameter data is generated for reusable subsystems.

Settings

Default: Hierarchical

Hierarchical

Generates a separate header file, defining an independent parameter structure, for each subsystem that meets the following conditions:

- The subsystem's **Real-Time Workshop system code** parameter is set to `Reusable` function.
- The subsystem does not violate any code reuse limitations.
- The subsystem does not access parameters other than its own (such as parameters of the root-level model).

Each subsystem parameter structure is referenced as a substructure of the root-level parameter data structure, creating a structure hierarchy.

NonHierarchical

Generates a single, flat parameter data structure. Subsystem parameters are defined as fields within the structure. A nonhierarchical data structure can reduce compiler padding between word boundaries, producing more efficient compiled code.

Dependencies

- This parameter appears only for ERT-based targets.
- This parameter requires a Real-Time Workshop Embedded Coder license when generating code.
- This parameter is enabled by “**Inline parameters**” on page 1-128.

Command-Line Information

Parameter: `InlinedParameterPlacement`

Type: `string`

Value: `'Hierarchical' | 'NonHierarchical'`

Default: 'Hierarchical'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	Hierarchical
Efficiency	NonHierarchical
Safety precaution	No impact

See Also

- Nonvirtual Subsystem Code Generation
- Optimizing a Model for Code Generation
- Configuration Parameters Dialog Box
- Optimization Pane

Optimize using the specified minimum and maximum values

Optimize generated code using the specified minimum and maximum values for signals and parameters in the model.

Settings

Default: Off



On

Optimizes the generated code using range information derived from the minimum and maximum specified values for signals and parameters in the model.



Off

Ignores specified minimum and maximum values when generating code.

Tips

- Ensure that specified minimum and maximum values for signals and parameters are accurate and trustworthy before using this option. Otherwise, optimization might result in numerical mismatch with simulation.

Before generating code, test the specified values by simulating your model with simulation range checking enabled using the **Diagnostics > Data Validity > Simulation range checking** configuration parameter. If errors or warnings occur, fix these issues before generating code.

- Specify minimum and maximum values for signals and parameters in the model for:
 - Inport and Outport blocks.
 - Block outputs.
 - Block inputs, for example, for the Embedded MATLAB Function and Stateflow Chart blocks.
 - Simulink.Signal objects.
- This optimization does not take into account minimum and maximum values specified for:

- Merge block inputs. To work around this, use a `Simulink.Signal` object on the Merge block output and specify the range on this object
- `Simulink.Parameter` objects.
- Bus elements.
- If you use the Polyspace® software to verify code generated using this optimization, it might mark code that was previously green as orange. For example, if your model contains a division where the range of the denominator does not include zero, the generated code does not include protection against division by zero. Polyspace might mark this code orange because it does not have information about the minimum and maximum values specified for the inputs to the division.

The Polyspace Model link products do automatically capture some minimum and maximum values specified in the MATLAB workspace, for example, for `Simulink.Signal` and `Simulink.Parameter` objects. In this example, to provide range information to the Polyspace software, use a `Simulink.Signal` object on the input of the division and specify a range that does not include zero.

The link products store these values in a Data Range Specification (DRS) file. However, they do not capture all minimum and maximum values specified in your Simulink model. To provide additional min/max information to Polyspace, you can manually define a DRS file. For more information, see the Polyspace Model Link documentation.

- If you are using double-precision data types and the **Real-Time Workshop > Interface > Support non-finite numbers** configuration parameter is selected, this optimization does not occur.
- If your model contains multiple instances of a reusable subsystem and each instance uses input signals with different specified minimum and maximum values, this optimization might result in different generated code for each subsystem so code reuse does not occur. Without this optimization, the Real-Time Workshop software generates code once for the subsystem and shares this code among the multiple instances of the subsystem.
- The Model Advisor **Check safety-related optimization settings** check generates a warning if this option is selected. For many safety critical applications, it is not acceptable to remove dead code automatically because this might result in requirements without traceable code. For more information, see Check safety-related optimization settings.

- Enabling this optimization improves the ability of the Simulink Fixed Point software to eliminate unnecessary utility functions and saturation code from the generated code.

Dependencies

- This parameter appears for ERT-based targets only.
- This parameter requires a Real-Time Workshop Embedded Coder license when generating code.

Command-Line Information

Parameter: UseSpecifiedMinMax

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	Off
Traceability	Off
Efficiency	On
Safety precaution	Off

See Also

- “Optimizing Generated Code Using Specified Minimum and Maximum Values” in the Real-Time Workshop Embedded Coder documentation.
- “Optimize Generated Code Using Specified Minimum and Maximum Values” in the Simulink Fixed Point documentation.

Enable local block outputs

Specify whether block signals are declared locally or globally.

Settings

Default: On

On
Block signals are declared locally in functions.

Off
Block signals are declared globally.

Tips

- If it is not possible to declare an output as a local variable, the generated code declares the output as a global variable.
- If you are constrained by limited stack space, you can turn **Enable local block outputs** off and still benefit from memory reuse.

Dependencies

- This parameter requires a Real-Time Workshop license.
- This parameter is enabled by **Signal storage reuse**.

Command-Line Information

Parameter: LocalBlockOutputs

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	Off
Traceability	No impact
Efficiency	On
Safety precaution	No impact

See Also

- Signal Storage, Optimization, and Interfacing
- Signals with Auto Storage Class
- Optimizing a Model for Code Generation
- Configuration Parameters Dialog Box
- Optimization Pane

Eliminate superfluous local variables (Expression folding)

Collapse block computations into single expressions.

Settings

Default: On

On

- Enables expression folding.
- Eliminates local variables, incorporating the information into the main code statement.
- Improves code readability and efficiency.

Off

Disables expression folding.

Dependencies

- This parameter requires a Real-Time Workshop license.
- This parameter is enabled by **Signal storage reuse**.

Command-Line Information

Parameter: ExpressionFolding

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	Off
Traceability	No impact (for simulation and during development) Off (for production code generation)
Efficiency	On
Safety precaution	No impact

See Also

- Expression Folding
- Optimizing a Model for Code Generation
- Configuration Parameters Dialog Box
- Optimization Pane

Minimize data copies between local and global variables

Reuse existing global variables to store temporary results.

Settings

Default: Off



On

Writes data for block outputs to global variables, reducing RAM consumption and execution time.



Off

Writes data for block outputs to local variables.

Dependencies

- This parameter requires a Real-Time Workshop license.
- This parameter is enabled by “**Signal storage reuse**” on page 1-133.

Command-Line Information

Parameter: EnhancedBackFolding

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	Off
Traceability	Off
Efficiency	No impact (execution), On (ROM, RAM)
Safety precaution	No impact

See Also

- “Signal Considerations”
- “Optimizing Generated Code”
- Configuration Parameters Dialog Box
- Optimization Pane

Simplify array indexing

Replace multiply operations in array indices when accessing arrays in a loop.

Settings

Default: Off



In array indices, replace multiply operations with add operations when accessing arrays in a loop in the generated code. When the original signal is multidimensional, the Real-Time Workshop Embedded Coder generates one-dimensional arrays, resulting in multiply operations in the array indices. Using this setting eliminates costly multiply operations when accessing arrays in a loop in the C/C++ program. This optimization (commonly referred to as strength reduction) is particularly useful if the C/C++ compiler on the target platform does not have similar functionality. No appearance of multiply operations in the C/C++ program does not imply that the C/C++ compiler does not generate multiply instructions.



Leave multiply operations in array indices when accessing arrays in a loop.

Dependencies

This parameter:

- Requires a Real-Time Workshop Embedded Coder license to generate code.
- Appears only for ERT-based targets.

Command-Line Information

Parameter: StrengthReduction

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- “Optimizing Generated Code”
- “Preparing Models for Code Generation”

Reuse block outputs

Specify whether Real-Time Workshop software reuses signal memory.

Settings

Default: On

On

- Real-Time Workshop software reuses signal memory whenever possible, reducing stack size where signals are being buffered in local variables.
- Selecting this parameter trades code traceability for code efficiency.

Off

Signals are stored in unique locations.

Dependencies

- This parameter requires a Real-Time Workshop license.
- This parameter is enabled by **Signal storage reuse**.

Command-Line Information

Parameter: BufferReuse

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	Off
Traceability	Off
Efficiency	On
Safety precaution	No impact

See Also

- [Signal Storage, Optimization, and Interfacing](#)
- [Signals with Auto Storage Class](#)
- [Optimizing a Model for Code Generation](#)
- [Configuration Parameters Dialog Box](#)
- [Optimization Pane](#)

Inline invariant signals

Transform symbolic names of invariant signals into constant values.

Settings

Default: Off

On

Real-Time Workshop software uses the numerical values of model parameters, instead of their symbolic names, in generated code. An invariant signal is not inlined if it is nonscalar, complex, or the block inport the signal is attached to takes the address of the signal.

Off

Uses symbolic names of model parameters in generated code.

Dependencies

- This parameter requires a Real-Time Workshop license.
- This parameter is enabled by **Inline parameters**.

Command-Line Information

Parameter: InlineInvariantSignals

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	Off
Traceability	Off
Efficiency	On
Safety precaution	No impact

See Also

Inlining Invariant Signals

Pack Boolean data into bitfields

Specify whether Boolean signals are stored as one-bit bitfields or as a Boolean data type.

Note You cannot use this optimization when you generate code for a target that specifies an explicit structure alignment.

Settings

Default: Off



On

Stores Boolean signals into one-bit bitfields in global block I/O structures or DWork vectors. This will reduce RAM, but might cause more executable code.



Off

Stores Boolean signals as a Boolean data type in global block I/O structures or DWork vectors.

Dependencies

This parameter:

- Requires a Real-Time Workshop Embedded Coder license.
- Appears only for ERT-based targets.

Command-Line Information

Parameter: BooleansAsBitfields

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	Off (execution, ROM), On (RAM)
Safety precaution	No impact

See Also

- “Optimization Pane” on page 1-116
- “Bitfield declarator type specifier” on page 1-161

Bitfield declarator type specifier

Specify the bitfield type when selecting configuration parameter “Pack Boolean data into bitfields” on page 1-159.

Note The optimization benefit is dependent upon your choice of target.

Settings

Default: `uint_T`

`uint_T`

The type specified for a bitfield declaration is an unsigned `int`.

`uchar_T`

The type specified for a bitfield declaration is an unsigned `char`.

Tip

The “Pack Boolean data into bitfields” on page 1-159 configuration parameter default setting uses unsigned integers. This might cause an increase in RAM if the bitfields are small and distributed. In this case, `uchar_T` might use less RAM depending on your target.

Dependency

Pack Boolean data into bitfields enables this parameter.

Command-Line Information

Parameter: `BitfieldContainerType`

Type: `string`

Value: `uint_T | uchar_T`

Default: `uint_T`

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	Target dependent
Safety precaution	No impact

See Also

“Pack Boolean data into bitfields” on page 1-159

Loop unrolling threshold

Specify the minimum signal or parameter width for which a for loop is generated.

Settings

Default: 5

Specify the array size at which the code generator begins to use a for loop instead of separate assignment statements to assign values to the elements of a signal or parameter array.

When there are perfectly nested loops, the code generator uses a for loop if the product of the loop counts for all loops in the perfect loop nest is greater than or equal to the threshold.

Dependency

This parameter requires a Real-Time Workshop license.

Command-Line Information

Parameter: RollThreshold

Type: string

Value: any valid value

Default: '5'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	>0
Safety precaution	>1

See Also

- [Configuring a Loop Unrolling Threshold](#)
- [Real-Time Workshop Target Language Compiler](#)

Maximum stack size (bytes)

Specify the maximum stack size in bytes for your model.

Settings

Default:Inherit from target

Inherit from target

The Real-Time Workshop software assigns the maximum stack size to the smaller value of the following:

- The default value (200,000 bytes) set by the Real-Time Workshop software
- Value of the TLC variable `MaxStackSize` in the system target file

<Specify a value>

Specify a positive integer value. Real-Time Workshop software assigns the maximum stack size to the specified value.

Note If you specify a maximum stack size for a model, the estimated required stack size of a referenced model must be less than the specified maximum stack size of the parent model.

Tips

- If you specify the maximum stack size to be zero, then the generated code implements all variables as global data.
- If you specify the maximum stack to be `inf`, then the generated code contains the least number of global variables.

Command-Line Information

Parameter: `MaxStackSize`

Type: `int`

Value: Any valid value

Default: Inherit from target

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

“Using Stack Space Allocation” in the Real-Time Workshop documentation

Use memcpy for vector assignment

Optimize code generated for vector assignment by replacing for loops with memcpy.

Settings

Default: On

On

Enables use of memcpy for vector assignment based on the associated threshold parameter **Memcpy threshold (bytes)**. memcpy is used in the generated code if the number of array elements times the number of bytes per element is greater than or equal to the specified value for **Memcpy threshold (bytes)**. One byte equals the width of a character in this context.

Off

Disables use of memcpy for vector assignment.

Dependencies

- This parameter requires a Real-Time Workshop license.
- When selected, this parameter enables the associated parameter **Memcpy threshold (bytes)**.

Command-Line Information

Parameter: EnableMemcpy

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	On
Safety precaution	No impact

See Also

- “Optimizing Code Generated for Vector Assignments”
- Optimizing a Model for Code Generation
- Configuration Parameters Dialog Box
- Optimization Pane

Memcpy threshold (bytes)

Specify the minimum array size in bytes for which memcpy function calls should replace for loops in the generated code for vector assignments.

Settings

Default: 64

Specify the array size, in bytes, at which the code generator begins to use memcpy instead of for loops for vector assignments.

Dependencies

- This parameter requires a Real-Time Workshop license.
- This parameter is enabled when you select **Use memcpy for vector assignment**.

Command-Line Information

Parameter: MemcpyThreshold

Type: integer

Value: any valid quantity of bytes

Default: 64

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	Accept default or determine target-specific optimal value
Safety precaution	No impact

See Also

- “Optimizing Code Generated for Vector Assignments”
- Optimizing a Model for Code Generation
- Configuration Parameters Dialog Box
- Optimization Pane

Pass reusable subsystem outputs as

Specify how a reusable subsystem passes outputs.

Settings

Default: Structure reference

Structure reference

Passes reusable subsystem outputs as a pointer to a structure stored in global memory.

Individual arguments

Passes each reusable subsystem output argument as an address of a local, instead of as a pointer to an area of global memory containing all output arguments. This option reduces global memory usage and eliminates copying local variables back to global block I/O structures. When the signals are allocated as local variables, there may be an increase in stack size. If the stack size increases beyond a level that you want, use the default setting. The maximum number of output arguments passed individually is 12.

Note The default option is used for reusable subsystems that have signals with variable dimensions.

Dependencies

This parameter:

- Requires a Real-Time Workshop Embedded Coder license.
- Appears only for ERT-based targets.

Command-Line Information

Parameter: PassReuseOutputArgsAs

Type: string

Value: 'Structure reference' | 'Individual arguments'

Default: 'Structure reference'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact (execution), Structure reference (ROM), Individual arguments (RAM)
Safety precaution	No impact

See Also

- “Optimizing Generated Code”
- “Preparing Models for Code Generation”

Remove root level I/O zero initialization

Specify whether to generate initialization code for root-level inports and outports set to zero.

Settings

Default: Off (GUI), 'on' (command-line)



On

Does not generate initialization code for root-level inports and outports set to zero.



Off

Generates initialization code for all root-level inports and outports. You should use the default:

- To ensure that memory allocated for C MEX S-function wrappers is initialized to zero
- For safety-critical applications that require that all internal and external data be initialized to zero

Dependencies

- This parameter appears only for ERT-based targets.
- This parameter requires a Real-Time Workshop Embedded Coder license when generating code.

Command-Line Information

Parameter: ZeroExternalMemoryAtStartup

Type: string

Value: 'off' | 'on'

Default: 'on'

Note The command-line values are reverse of the settings values. Therefore, 'on' in the command line corresponds to the description of “Off” in the settings section, and 'off' in the command line corresponds to the description of “On” in the settings section.

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	On (GUI), off (command line) (execution, ROM), No impact (RAM)
Safety precaution	Off (GUI), on (command line)

See Also

- Optimizing a Model for Code Generation
- Configuration Parameters Dialog Box
- Optimization Pane

Use memset to initialize floats and doubles to 0.0

Specify whether to generate code that explicitly initializes floating-point data to 0.0.

Settings

Default: On (GUI), 'off' (command-line)



On

Uses `memset` to clear internal storage for floating-point data to integer bit pattern 0 (all bits 0), regardless of type. An example of a case for selecting this option is to gain compiler efficiency when the compiler and target CPU both represent floating-point zero with the integer bit pattern 0.



Off

Generates code to explicitly initialize storage for data of types `float` and `double` to 0.0. The resulting code is slightly less efficient than code generated when you select the option.

You should not select this option if you need to ensure that memory allocated for C MEX S-function wrappers is initialized to zero.

Dependency

This parameter requires a Real-Time Workshop license.

Command-Line Information

Parameter: `InitFltsAndDblsToZero`

Type: string

Value: 'on' | 'off'

Default: 'off'

Note The command-line values are reverse of the settings values. Therefore, 'on' in the command line corresponds to the description of "Off" in the settings section, and 'off' in the command line corresponds to the description of "On" in the settings section.

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	On (GUI), 'off' (command-line) (execution, ROM), No impact (RAM)
Safety precaution	No impact

See Also

- [Optimizing a Model for Code Generation](#)
- [Configuration Parameters Dialog Box](#)
- [Optimization Pane](#)

Remove internal data zero initialization

Specify whether to generate initialization code for internal work structures, such as block states and block outputs, to zero.

Settings

Default: Off (GUI), 'on' (command-line)



On

Does not generate code that initializes internal work structures to zero. An example of when you might select this parameter is to test the behavior of a design during warm boot—a restart without full system reinitialization.

Selecting this parameter does not guarantee that memory is in a known state each time the generated code begins execution. When you run a model or generated S-function multiple times, each run can produce a different answer, even when calling the model initialization function in an attempt to reset memory.

If want to get the same answer on every run from a generated S-function, enter the command `clear SFcnNam` or `clear mex` in the MATLAB Command Window before each run.



Off

Generates code that initializes internal work structures to zero. You should use the default:

- To ensure that memory allocated for C MEX S-function wrappers is initialized to zero
- For safety critical applications that require that all internal and external data be initialized to zero

Dependencies

- This parameter appears only for ERT-based targets.
- This parameter requires a Real-Time Workshop Embedded Coder license when generating code.

Command-Line Information

Parameter: ZeroInternalMemoryAtStartup

Type: string

Value: 'off' | 'on'

Default: 'on'

Note The command-line values are reverse of the settings values. Therefore, 'on' in the command line corresponds to the description of “Off” in the settings section, and 'off' in the command line corresponds to the description of “On” in the settings section.

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	On (GUI), off (command line), (execution, ROM), No impact (RAM)
Safety precaution	Off (GUI), on (command line)

See Also

- Optimizing a Model for Code Generation
- Configuration Parameters Dialog Box
- Optimization Pane

Optimize initialization code for model reference

Specify whether to generate initialization code for blocks that have states.

Settings

Default: on



On

Suppresses generation of initialization code for blocks that have states unless the blocks are in a system that can reset its states, such as an enabled subsystem. This results in more efficient code.



Off

Generates initialization code for all blocks that have states. Disable this option if the current model includes a subsystem that resets states, such as an enabled subsystem, and the model is referred to from another model with a Model block.

Tips

The following restrictions apply to using the **Optimize initialization code for model reference** parameter. However, these restrictions do not apply to a Model block that references a function-call model.

- In a subsystem that resets states, do not include a Model block that references a model that has this parameter set to on. For example, in an enabled subsystem with the **States when enabling** block parameter set to reset, do not include a Model block that references a model that has the **Optimize initialization code for model reference** parameter set to on.
- If you set the **Optimize initialization code for model reference** parameter to off in a model that includes a Model block that directly references a submodel, do not set the **Optimize initialization code for model reference** parameter for the submodel to on.

Dependencies

- This parameter appears only for ERT-based targets.
- This parameter requires a Real-Time Workshop Embedded Coder license when generating code.

Command-Line Information

Parameter: OptimizeModelRefInitCode

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	On (execution, ROM), No impact (RAM)
Safety precaution	No impact

See Also

- [Optimizing a Model for Code Generation](#)
- [Configuration Parameters Dialog Box](#)
- [Optimization Pane](#)

Remove code from floating-point to integer conversions that wraps out-of-range values

Remove wrapping code that handles out-of-range floating-point to integer conversion results.

Settings

Default: Off



Removes code when out-of-range conversions occur. Select this check box if code efficiency is critical to your application and the following conditions are true for at least one block in the model:

- Computing the outputs or parameters of a block involves converting floating-point data to integer or fixed-point data.
- The **Saturate on integer overflow** check box is cleared in the Block Parameters dialog box.

Caution Execution of generated code might not produce the same results as simulation.



Results for simulation and execution of generated code match when out-of-range conversions occur. The generated code is larger than when you select this check box.

Tips

- Selecting this check box reduces the size and increases the speed of the generated code at the cost of potentially producing results that do not match simulation in the case of out-of-range values.
- Selecting this check box affects code generation results only for out-of-range values and cannot cause code generation results to differ from simulation results for in-range values.

Dependency

This parameter requires a Real-Time Workshop license.

Command-Line Information

Parameter: EfficientFloat2IntCast

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	Off
Traceability	Off
Efficiency	On (execution, ROM), No impact (RAM)
Safety precaution	Off (for simulation and during development) On (for production code generation)

See Also

- Removing Code That Wraps Out-of-Range Values
- Configuration Parameters Dialog Box
- Optimization Pane

Remove code from floating-point to integer conversions with saturation that maps NaN to zero

Remove code that handles floating-point to integer conversion results for NaN values.

Settings

Default: On



On

Removes code when mapping from NaN to integer zero occurs. Select this check box if code efficiency is critical to your application and the following conditions are true for at least one block in the model:

- Computing outputs or parameters of a block involves converting floating-point data to integer or fixed-point data.
- The **Saturate on integer overflow** check box is selected in the Block Parameters dialog box.

Caution Execution of generated code might not produce the same results as simulation.



Off

Results for simulation and execution of generated code match when mapping from NaN to integer zero occurs. The generated code is larger than when you select this check box.

Tips

- Selecting this check box reduces the size and increases the speed of the generated code at the cost of producing results that do not match simulation in the case of NaN values.
- Selecting this check box affects code generation results only for NaN values and cannot cause code generation results to differ from simulation results for any other values.

Dependencies

- This parameter requires a Real-Time Workshop license.
- For ERT-based targets, this parameter is enabled when you select the **floating-point numbers** and **non-finite numbers** check boxes in the **Real-Time Workshop > Interface** pane.

Command-Line Information

Parameter: EfficientMapNaN2IntZero

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	Off
Traceability	Off
Efficiency	On
Safety precaution	Off (for simulation and during development) On (for production code generation)

See Also

- Removing Code That Maps NaN Values to Integer Zero
- Configuration Parameters Dialog Box
- Optimization Pane

Remove code that protects against division arithmetic exceptions

Specify whether to generate code that guards against division by zero for fixed-point data.

Settings

Default: On



On

Does not generate code that guards against division by zero for fixed-point data. When you select this option, simulation results and results from generated code might not be in bit-for-bit agreement.



Off

Generates code that guards against division by zero for fixed-point data.

Dependencies

- This parameter appears only for ERT-based targets.
- This parameter requires a Real-Time Workshop Embedded Coder license when generating code.

Command-Line Information

Parameter: NoFixptDivByZeroProtection

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	On
Safety precaution	Off

See Also

- [Optimizing a Model for Code Generation](#)
- [Configuration Parameters Dialog Box](#)
- [Optimization Pane](#)

Use bitsets for storing state configuration

Use bitsets to reduce the amount of memory required to store state configuration variables.

Settings

Default: Off



On

Stores state configuration variables in bitsets. Potentially reduces the amount of memory required to store the variables. Potentially requires more instructions to access state configuration, which can result in less optimal code.



Off

Stores state configuration variables in unsigned bytes. Potentially increases the amount of memory required to store the variables. Potentially requires fewer instructions to access state configuration, which can result in more optimal code.

Tips

- Selecting this check box can significantly reduce the amount of memory required to store the variables. However, it can increase the amount of memory required to store target code if the target processor does not include instructions for manipulating bitsets.
- Select this check box for Stateflow charts that have a large number of sibling states at a given level of the hierarchy.
- Clear this check box for Stateflow charts with a small number of sibling states at a given level of the hierarchy.

Dependency

This parameter requires a Stateflow license.

Command-Line Information

Parameter: StateBitsets

Type: string
Value: 'on' | 'off'
Default: 'off'

Recommended Settings

Application	Setting
Debugging	Off
Traceability	Off
Efficiency	Off (execution, ROM), On (RAM)
Safety precaution	No impact

See Also

- Optimizing Generated Code
- Configuration Parameters Dialog Box
- Optimization Pane

Use bitsets for storing Boolean data

Use bitsets to reduce the amount of memory required to store Boolean data.

Settings

Default: Off



On

Stores Boolean data in bitsets. Potentially reduces the amount of memory required to store the data. Potentially requires more instructions to access the data, which can result in less optimal code.



Off

Stores Boolean data in unsigned bytes. Potentially increases the amount of memory required to store the data. Potentially requires fewer instructions to access the data, which can result in more optimal code.

Tips

- Select this check box for Stateflow charts that reference Boolean data infrequently.
- Clear this check box for Stateflow charts that reference Boolean data frequently.

Dependency

This parameter requires a Stateflow license.

Command-Line Information

Parameter: DataBitsets

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

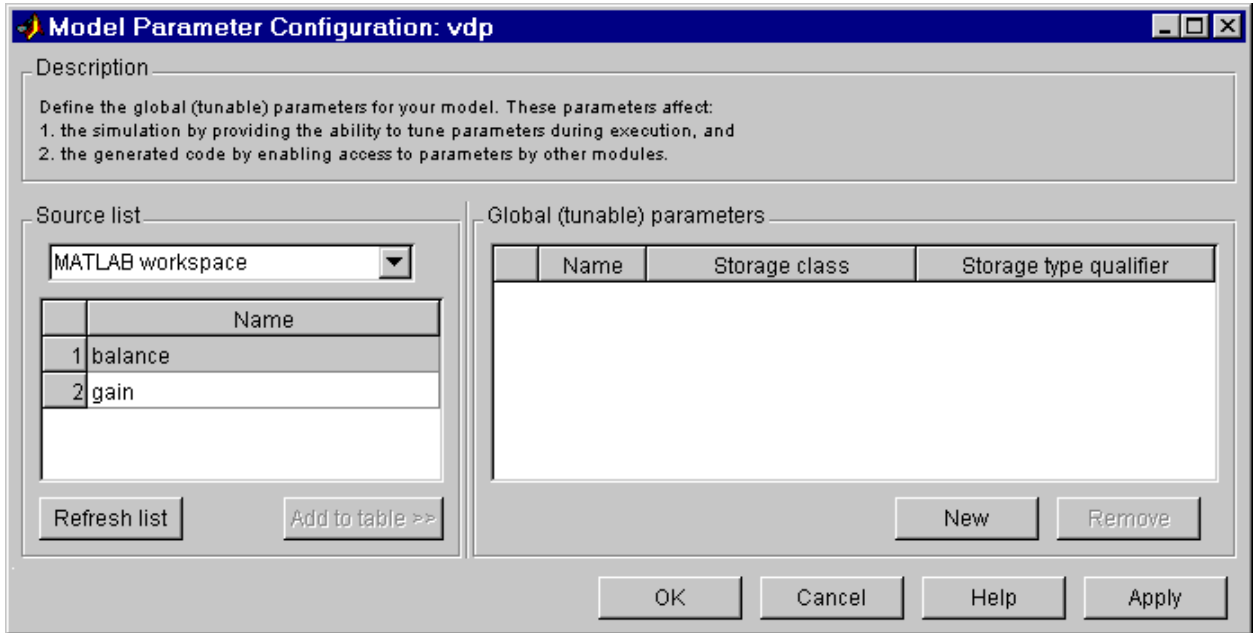
Application	Setting
Debugging	Off
Traceability	Off
Efficiency	Off (execution, ROM), On (RAM)
Safety precaution	No impact

See Also

- [Optimizing Generated Code](#)
- [Configuration Parameters Dialog Box](#)
- [Optimization Pane](#)

Model Parameter Configuration Dialog Box

The **Model Parameter Configuration** dialog box allows you to override the **Inline parameters** option (see Inline parameters) for selected parameters.



Note Simulink software ignores the settings of this dialog box if a model contains references to other models. However, you can still tune parameters of such models, using `Simulink.Parameter` objects (see “Inline Parameter Requirements” for more information).

The dialog box has the following controls.

Source list

Displays a list of workspace variables. The options are:

- MATLAB workspace

Lists all variables in the MATLAB workspace that have numeric values.

- **Referenced workspace variables**

Lists only those variables referenced by the model.

Refresh list

Updates the source list. Click this button if you have added a variable to the workspace since the last time the list was displayed.

Add to table

Adds the variables selected in the source list to the adjacent table of tunable parameters.

New

Defines a new parameter and adds it to the list of tunable parameters. Use this button to create tunable parameters that are not yet defined in the MATLAB workspace.

Note This option does not create the corresponding variable in the MATLAB workspace. You must create the variable yourself.

Storage class

Used for code generation. See the *Real-Time Workshop User's Guide* for more information.

Storage type qualifier

Used for code generation. See the *Real-Time Workshop User's Guide* for more information.

Compiler optimization level

Sets the degree of optimization used by the compiler when generating code for acceleration.

Settings

Default: Optimizations off (faster builds)

Optimizations off (faster builds)

Specifies the compiler not to optimize code. This results in faster build times.

Optimizations on (faster runs)

Specifies the compiler to generate optimized code. The generated code will run faster, but the model build will take longer than if optimizations are off.

Tips

- The default `Optimizations off` is a good choice for most models. This quickly produces code that can be used with acceleration.
- Set `Optimizations on` to optimize your code. The fast running code produced by optimization can be advantageous if you will repeatedly run your model with the accelerator.

Command-Line Information

Parameter: `SimCompilerOptimization`

Type: `string`

Value: `'on' | 'off'`

Default: `'off'`

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact

See Also

- “Accelerating Models”
- “Interacting with the Acceleration Modes Programmatically”
- “Customizing the Acceleration Build Process”

Verbose accelerator builds

Select the amount of information displayed during code generation for Simulink Accelerator mode, referenced model Accelerator mode, and Rapid Accelerator mode.

Settings

Default: Off

Off
Display limited amount of information during the code generation process.

On
Display progress information during code generation, and show the compiler options in use.

Command-Line Information

Parameter: AccelVerboseBuild

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

For more information about AccelVerboseBuild, see “Controlling Verbosity During Code Generation”.

Diagnostics Pane: Solver

Solver	
Algebraic loop:	warning
Minimize algebraic loop:	warning
Block priority violation:	warning
Min step size violation:	warning
Sample hit time adjusting:	none
Consecutive zero crossings violation:	error
Unspecified inheritability of sample time:	warning
Solver data inconsistency:	none
Automatic solver parameter selection:	warning
Extraneous discrete derivative signals:	error
State name clash:	warning

In this section...

“Solver Diagnostics Overview” on page 1-197

“Algebraic loop” on page 1-198

“Minimize algebraic loop” on page 1-200

“Block priority violation” on page 1-202

“Min step size violation” on page 1-204

“Sample hit time adjusting” on page 1-206

“Consecutive zero-crossings violation” on page 1-208

“Unspecified inheritability of sample time” on page 1-210

“Solver data inconsistency” on page 1-212

“Automatic solver parameter selection” on page 1-214

“Extraneous discrete derivative signals” on page 1-216

In this section...

“State name clash” on page 1-218

“SimState interface checksum mismatch:” on page 1-219

Solver Diagnostics Overview

Specify what diagnostic actions Simulink software should take, if any, when it detects an abnormal condition with the solver.

Configuration

Set the parameters displayed.

Tips

- The options are typically to do nothing or to display a warning or an error message.
- A warning does not terminate a simulation, but an error does.

See Also

- Diagnosing Simulation Errors
- Sample Time Diagnostics
- Data Validity Diagnostics
- Type Conversion Diagnostics
- Connectivity Diagnostics
- Compatibility Diagnostics
- Model Referencing Diagnostics
- Saving Diagnostics
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

Algebraic loop

Select the diagnostic action to take if Simulink software detects an algebraic loop while compiling the model.

Settings

Default: warning

none

When the Simulink software detects an algebraic loop, the software tries to solve the algebraic loop. If the software cannot solve the algebraic loop, it reports an error and the simulation terminates.

warning

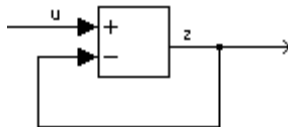
When Simulink software detects an algebraic loop, it displays a warning and tries to solve the algebraic loop. If the software cannot solve the algebraic loop, it reports an error and the simulation terminates.

error

When Simulink software detects an algebraic loop, it terminates the simulation, displays an error message, and highlights the portion of the block diagram that comprises the loop.

Tips

- An *algebraic loop* generally occurs when an input port with direct feedthrough is driven by the output of the same block, either directly, or by a feedback path through other blocks with direct feedthrough. An example of an algebraic loop is this simple scalar loop.



- When a model contains an algebraic loop, Simulink software calls a loop-solving routine at each time step. The loop solver performs iterations to determine the solution to the problem (if it can). As a result, models with algebraic loops run slower than models without them.

- Use the **error** option to highlight algebraic loops when you simulate a model. This causes Simulink software to display an error dialog (the Diagnostics Viewer) and recolor portions of the diagram that represent the first algebraic loop that it detects. Simulink software uses red to color the blocks and lines that constitute the loop. Closing the error dialog restores the diagram to its original colors.
- See Algebraic Loops for more information.

Command-Line Information

Parameter: AlgebraicLoopMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	error
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Algebraic Loops
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

Minimize algebraic loop

Select the diagnostic action to take if algebraic loop minimization cannot be performed for a subsystem because an input port of that subsystem has direct feedthrough.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- If the port is involved in an algebraic loop, Simulink software can remove the loop only if at least one other input port in the loop lacks direct feedthrough.
- Simulink software cannot minimize algebraic loops containing signals designated as test points (see Working with Test Points).

Command-Line Information

Parameter: ArtificialAlgebraicLoopMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Algebraic Loops
- Diagnosing Simulation Errors
- Working with Test Points
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

Block priority violation

Select the diagnostic action to take if Simulink software detects a block priority specification error.

Settings

Default: warning

warning

When Simulink software detects a block priority specification error, it displays a warning.

error

When Simulink software detects a block priority specification error, it terminates the simulation and displays an error message.

Tips

- Simulink software allows you to assign update priorities to blocks. Simulink software executes the output methods of higher priority blocks before those of lower priority blocks.
- Simulink software honors the block priorities that you specify only if they are consistent with the Simulink block sorting algorithm. If Simulink software is unable to honor a user specified block priority, it generates a block priority specification error.

Command-Line Information

Parameter: BlockPriorityViolationMsg

Type: string

Value: 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Controlling and Displaying the Sorted Order
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

Min step size violation

Select the diagnostic action to take if Simulink software detects that the next simulation step is smaller than the minimum step size specified for the model.

Settings

Default: warning

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- A minimum step size violation can occur if the specified error tolerance for the model requires a step size smaller than the specified minimum step size. See *Min step size* and *Maximum order* for more information.
- Simulink software allows you to specify the maximum number of consecutive minimum step size violations permitted (see *Number of consecutive min steps*).

Command-Line Information

Parameter: MinStepSizeMsg

Type: string

Value: 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact

See Also

- Min step size
- Maximum order
- Number of consecutive min steps
- “Purely Discrete Systems”
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

Sample hit time adjusting

Select the diagnostic action to take if Simulink software makes a minor adjustment to a sample hit time while running the model.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

Tips

- Simulink software might change a sample hit time if that hit time is close to the hit time for another task. If Simulink software considers the difference to be due only to numerical errors (for example, precision issues or roundoff errors), it changes the sample hits of the faster task or tasks to exactly match the time of the slowest task that has that hit.
- Over time, these sample hit changes might cause a discrepancy between the numerical simulation results and the actual theoretical results.
- When this option is set to warning, the MATLAB Command Window displays a warning like the following when Simulink software detects a change in the sample hit time:

Warning: Timing engine warning: Changing the hit time for ...

Command-Line Information

Parameter: TimeAdjustmentMsg

Type: string

Value: 'none' | 'warning'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

Consecutive zero-crossings violation

Select the diagnostic action to take when Simulink software detects that the number of consecutive zero crossings exceeds the specified maximum.

Settings

Default: error

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- If you select warning or error, Simulink software reports the current simulation time, the number of consecutive zero crossings counted, and the type and name of the block in which Simulink software detected the zero crossings.
- For more information, see [Preventing Excessive Zero Crossings](#).

Dependency

This diagnostic applies only when you are using a variable-step solver and the zero-crossing control is set to either `Enable all` or `Use local settings`.

Command-Line Information

Parameter: MaxConsecutiveZCsMsg

Type: string

Value: 'none' | 'warning'

Default: 'error'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	warning or error

See Also

- Zero-Crossing Detection
- Zero-Crossing Control
- Number of consecutive zero crossings
- Time tolerance
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

Unspecified inheritability of sample time

Select the diagnostic action to take if this model contains S-functions that do not specify whether they preclude this model from inheriting their sample times from a parent model.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- Not specifying an inheritance rule may lead to incorrect simulation results.
- Simulink software checks for this condition only if the solver used to simulate this model is a fixed-step discrete solver and the periodic sample time constraint for the solver is set to ensure sample time independence
- For more information, see Periodic sample time constraint.

Command-Line Information

Parameter: UnknownTsInhSupMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Periodic sample time constraint
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

Solver data inconsistency

Select the diagnostic action to take if Simulink software detects S-functions that have continuous sample times, but do not produce consistent results when executed multiple times.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- Consistency checking can cause a significant decrease in performance (up to 40%).
- Consistency checking is a debugging tool that validates certain assumptions made by Simulink ODE solvers. Use this option to:
 - Validate your S-functions and ensure that they adhere to the same rules as Simulink built-in blocks.
 - Determine the cause of unexpected simulation results.
 - Ensure that blocks produce constant output when called with a given value of t (time).
- Simulink software saves (caches) output, the zero-crossing, the derivative, and state values from one time step for use in the next time step. The value at the end of a time step can generally be reused at the start of the next time step. Solvers, particularly stiff solvers such as `ode23s` and `ode15s`, take advantage of this to avoid redundant calculations. While calculating the Jacobian matrix, a stiff solver can call a block's output functions many times at the same value of t .

- When consistency checking is enabled, Simulink software recomputes the appropriate values and compares them to the cached values. If the values are not the same, a consistency error occurs. Simulink software compares computed values for these quantities:
 - Outputs
 - Zero crossings
 - Derivatives
 - States

Command-Line Information

Parameter: ConsistencyChecking
Type: string
Value: 'none' | 'warning' | 'error'
Default: 'warning'

Recommended Settings

Application	Setting
Debugging	warning
Traceability	No impact
Efficiency	none
Safety precaution	No impact

See Also

- Diagnosing Simulation Errors
- Choosing a Solver
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

Automatic solver parameter selection

Select the diagnostic action to take if Simulink software changes a solver parameter setting.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

When enabled, this option notifies you if:

- Simulink software changes a user-modified parameter to make it consistent with other model settings.
- Simulink software automatically selects solver parameters for the model, such as `FixedStepSize`.

For example, if you simulate a discrete model that specifies a continuous solver, Simulink software changes the solver type to discrete and displays a warning about this change at the MATLAB command line.

Command-Line Information

Parameter: `SolverPrmCheckMsg`

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Choosing a Solver
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

Extraneous discrete derivative signals

Select the diagnostic action to take when a discrete signal appears to pass through a Model block to the input of a block with continuous states.

Settings

Default: error

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- This error can occur if a discrete signal passes through a Model block to the input of a block with continuous states, such as an Integrator block. In this case, Simulink software cannot determine with certainty the minimum rate at which it needs to reset the solver to solve this model accurately.
- If this diagnostic is set to `none` or `warning`, Simulink software resets the solver whenever the value of the discrete signal changes. This ensures accurate simulation of the model if the discrete signal is the source of the signal entering the block with continuous states. However, if the discrete signal is not the source of the signal entering the block with continuous states, resetting the solver at the rate the discrete signal changes can lead to the solver being reset more frequently than necessary, slowing down the simulation.
- If this diagnostic is set to `error`, Simulink software halts when compiling this model and displays an error.

Dependency

This diagnostic applies only when you are using a variable-step ode solver and the block diagram contains Model blocks.

Command-Line Information

Parameter: ModelReferenceExtraNoncontSigs

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'error'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Diagnosing Simulation Errors
- Choosing a Solver
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

State name clash

Select the diagnostic action to take when a name is used for more than one state in the model.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

Tips

- This diagnostic applies for continuous and discrete states during simulation.
- This diagnostic applies only if you save states to the MATLAB workspace using the format **Structure** or **Structure with time**. If you do not save states in structure format, the state names are not used, and therefore the diagnostic will not warn you about a naming conflict.

Command-Line Information

Parameter: StateNameClashWarn

Type: string

Value: 'none' | 'warning'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact

See Also

- Diagnosing Simulation Errors
- Data Import/Export Pane
- Save to workspace: States
- Save options: Format
- Exporting Data to the MATLAB Workspace
- Configuration Parameters Dialog Box
- Diagnostics Pane: Solver

SimState interface checksum mismatch:

Use this check to ensure that the interface checksum is identical to the model checksum before loading the SimState.

Settings

Default: warning

none

Simulink software does not compare the interface checksum to the model checksum.

warning

The interface checksum in the SimState is different than the model checksum.

error

When Simulink detects that a change in the configuration settings occurred after saving the SimState, it does not load the SimState and reports an error.

Command-Line Information

Parameter: SimStateInterfaceChecksumMismatchMsg

Type: string

Value: 'warning' | 'error' | 'none'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- “Saving and Restoring the Simulation State as the SimState”
-

Diagnostics Pane: Sample Time

Sample Time	
Source block specifies -1 sample time:	warning
Discrete used as continuous:	warning
Multitask rate transition:	error
Single task rate transition:	none
Multitask conditionally executed subsystem:	error
Tasks with equal priority:	warning
Enforce sample times specified by Signal Specification blocks:	warning

In this section...

“Sample Time Diagnostics Overview” on page 1-222

“Source block specifies -1 sample time” on page 1-223

“Discrete used as continuous” on page 1-225

“Multitask rate transition” on page 1-227

“Single task rate transition” on page 1-229

“Multitask conditionally executed subsystem” on page 1-231

“Tasks with equal priority” on page 1-233

“Enforce sample times specified by Signal Specification blocks” on page 1-235

Sample Time Diagnostics Overview

Specify what diagnostic actions Simulink software should take, if any, when it detects a compilation error related to model sample times.

Configuration

Set the parameters displayed.

Tips

- The options are typically to do nothing or to display a warning or an error message.
- A warning does not terminate a simulation, but an error does.

See Also

- Diagnosing Simulation Errors
- Solver Diagnostics
- Data Validity Diagnostics
- Type Conversion Diagnostics
- Connectivity Diagnostics
- Compatibility Diagnostics
- Model Referencing Diagnostics
- Saving Diagnostics
- Configuration Parameters Dialog Box
- Diagnostics Pane: Sample Time

Source block specifies -1 sample time

Select the diagnostic action to take if a source block (such as a Sine Wave block) specifies a sample time of -1.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- The Random Source block does not obey this parameter. If its **Sample time** parameter is set to -1, the Random Source block inherits its sample time from its output port and never produces warnings or errors.
- Some Communications Blockset™ blocks internally inherit sample times, which can be a useful and valid modeling technique. Set this parameter to none for these types of models.

Command-Line Information

Parameter: InheritedTsInSrcMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Sample Time

Discrete used as continuous

Select the diagnostic action to take if a discrete block (such as the Unit Delay block), inherits a continuous sample time from the block connected to its input.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Command-Line Information

Parameter: DiscreteInheritContinuousMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors

- Configuration Parameters Dialog Box
- Diagnostics Pane: Sample Time

Multitask rate transition

Select the diagnostic action to take if an invalid rate transition occurred between two blocks operating in multitasking mode.

Settings

Default: error

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- This parameter allows you to adjust error checking for sample rate transitions between blocks that operate at different sample rates.
- Use this option for models of real-time multitasking systems to ensure detection of illegal rate transitions between tasks that can result in a task's output being unavailable when needed by another task. You can then use Rate Transition blocks to eliminate such illegal rate transitions from the model.

Command-Line Information

Parameter: MultiTaskRateTransMsg

Type: string

Value: 'warning' | 'error'

Default: 'error'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	error

See Also

- Rate Transition block
- Model Execution and Rate Transitions
- Single-Tasking and Multitasking Execution Modes
- Sample Rate Transitions
- Tasking mode for periodic sample times
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Sample Time

Single task rate transition

Select the diagnostic action to take if a rate transition occurred between two blocks operating in single-tasking mode.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- This parameter allows you to adjust error checking for sample rate transitions between blocks that operate at different sample rates.
- Use this parameter when you are modeling a single-tasking system. In such systems, task synchronization is not an issue.

Command-Line Information

Parameter: SingleTaskRateTransMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	none or error

See Also

- Rate Transition block
- Model Execution and Rate Transitions
- Single-Tasking and Multitasking Execution Modes
- Sample Rate Transitions
- Tasking mode for periodic sample times
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Sample Time

Multitask conditionally executed subsystem

Select the diagnostic action to take if Simulink software detects a subsystem that may cause data corruption or non-deterministic behavior.

Settings

Default: error

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- These types of subsystems can be caused by either of the following conditions:
 - Your model uses multitasking solver mode and it contains an enabled subsystem that operates at multiple rates.
 - Your model contains a conditionally executed subsystem that can reset its states and that contains an asynchronous subsystem.These types of subsystems can cause corrupted data or nondeterministic behavior in a real-time system that uses code generated from the model.
- For models that use multitasking solver mode and contain an enabled subsystem that operates at multiple rates, consider using single-tasking solver mode or using a single-rate enabled subsystem instead.
- For models that contain a conditionally executed subsystem that can reset its states and that contains an asynchronous subsystem, consider moving the asynchronous subsystem outside the conditionally executed subsystem.

Command-Line Information

Parameter: MultiTaskCondExecSysMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'error'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Tasking mode for periodic sample times
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Sample Time

Tasks with equal priority

Select the diagnostic action to take if Simulink software detects two tasks with equal priority that can preempt each other in the target system.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- This condition can occur when one asynchronous task of the target represented by this model has the same priority as one of the target's asynchronous tasks.
- This option must be set to Error if the target allows tasks having the same priority to preempt each other.

Command-Line Information

Parameter: TasksWithSamePriorityMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	none or error

See Also

- Diagnosing Simulation Errors
- Rate Transitions and Asynchronous Blocks
- Configuration Parameters Dialog Box
- Diagnostics Pane: Sample Time

Enforce sample times specified by Signal Specification blocks

Select the diagnostic action to take if the sample time of the source port of a signal specified by a Signal Specification block differs from the signal's destination port.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- The Signal Specification block allows you to specify the attributes of the signal connected to its input and output ports. If the specified attributes conflict with the attributes specified by the blocks connected to its ports, Simulink software displays an error when it compiles the model, for example, at the beginning of a simulation. If no conflict exists, Simulink software eliminates the Signal Specification block from the compiled model.
- You can use the Signal Specification block to ensure that the actual attributes of a signal meet desired attributes, or to ensure correct propagation of signal attributes throughout a model.

Command-Line Information

Parameter: SigSpecEnsureSampleTimeMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Signal Specification block
- Configuration Parameters Dialog Box
- Diagnostics Pane: Sample Time

Diagnostics Pane: Data Validity

Data Validity

Signals

Signal resolution: Detect overflow:

Division by singular matrix: Inf or NaN block output:

Underspecified data types: "rt" prefix for identifiers:

Simulation range checking:

Parameters

Detect downcast: Detect overflow:

Detect underflow: Detect precision loss:

Detect loss of tunability:

Data Store Memory Block

Detect read before write: Multitask data store:

Detect write after read: Duplicate data store names:

Detect write after write:

Merge Block

Detect multiple driving blocks executing at the same time step:

Model Initialization

Underspecified initialization detection:

Check undefined subsystem initial output

Check preactivation output of execution context

Check runtime output of execution context

Debugging

Array bounds exceeded:

Model Verification block enabling:

In this section...

- “Data Validity Diagnostics Overview” on page 1-239
- “Signal resolution” on page 1-240
- “Division by singular matrix” on page 1-242
- “Underspecified data types” on page 1-244
- “Simulation range checking” on page 1-246
- “Detect overflow” on page 1-248
- “Inf or NaN block output” on page 1-250
- “rt” prefix for identifiers” on page 1-252
- “Detect downcast” on page 1-254
- “Detect overflow” on page 1-256
- “Detect underflow” on page 1-258
- “Detect precision loss” on page 1-260
- “Detect loss of tunability” on page 1-262
- “Detect read before write” on page 1-264
- “Detect write after read” on page 1-266
- “Detect write after write” on page 1-268
- “Multitask data store” on page 1-270
- “Duplicate data store names” on page 1-272
- “Detect multiple driving blocks executing at the same time step” on page 1-274
- “Underspecified initialization detection” on page 1-276
- “Check undefined subsystem initial output” on page 1-279
- “Check preactivation output of execution context” on page 1-282
- “Check runtime output of execution context” on page 1-285
- “Array bounds exceeded” on page 1-288
- “Model Verification block enabling” on page 1-290

Data Validity Diagnostics Overview

Specify what diagnostic action Simulink software should take, if any, when it detects a condition that could compromise the integrity of data defined by the model, as well as the Data Validity parameters that pertain to code generation, and are used to debug a model.

Configuration

Set the parameters displayed.

Tips

- The options are typically to do nothing or to display a warning or an error message.
- A warning does not terminate a simulation, but an error does.

See Also

- Diagnosing Simulation Errors
- Solver Diagnostics
- Sample Time Diagnostics
- Type Conversion Diagnostics
- Connectivity Diagnostics
- Compatibility Diagnostics
- Model Referencing Diagnostics
- Saving Diagnostics
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Signal resolution

Select how Simulink software resolves signals to Simulink.Signal objects. See “Explicit and Implicit Symbol Resolution” for more information.

Settings

Default: Explicit only

Explicit only

Do not perform implicit signal resolution. Only explicitly specified signal resolution occurs. This is the recommended setting.

Explicit and implicit

Perform implicit signal resolution wherever possible, without posting any warnings about the implicit resolutions.

Explicit and warn implicit

Perform implicit signal resolution wherever possible, posting a warning of each implicit resolution that occurs.

Tips

- Use the Signal Properties dialog box (see Signal Properties Dialog Box) to specify explicit resolution for signals.
- Use the **State Attributes** pane on dialog boxes of blocks that have discrete states, e.g., the Discrete-Time Integrator block, to specify explicit resolution for discrete states.
- Multiple signals can resolve to the same signal object and have the properties that the object specifies.
- MathWorks discourages using implicit signal resolution except for fast prototyping, because implicit resolution slows performance, complicates model validation, and can have nondeterministic effects.
- Simulink software provides the `disableimplicitsignalresolution` function, which you can use to change settings throughout a model so that it does not use implicit signal resolution.

Command-Line Information

Parameter: SignalResolutionControl

Type: string

Value: 'UseLocalSettings' | 'TryResolveAll' |
'TryResolveAllWithWarning'

Default: 'UseLocalSettings'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Explicit only

See Also

- Diagnosing Simulation Errors
- Simulink.Signal
- Signal Properties Dialog Box
- Discrete-Time Integrator block
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Division by singular matrix

Select the diagnostic action to take if the Product block detects a singular matrix while inverting one of its inputs in matrix multiplication mode.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Command-Line Information

Parameter: CheckMatrixSingularityMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors

- Product block
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Underspecified data types

Select the diagnostic action to take if Simulink software could not infer the data type of a signal during data type propagation.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Command-Line Information

Parameter: UnderSpecifiedDataTypeMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors

- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Simulation range checking

Select the diagnostic action to take when signals exceed specified minimum or maximum values.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- Use a block's **Output minimum** or **Minimum** parameter to specify the minimum value that the block should output.
- Use a block's **Output maximum** or **Maximum** parameter to specify the maximum value that the block should output.
- Enable this diagnostic to check whether block outputs exceed the minimum or maximum values that you specified.
- When **Simulation range checking** is enabled, Simulink software performs signal range checking at every time step during a simulation. Setting this diagnostic to warning or error can cause a decrease in simulation performance.

Command-Line Information

Parameter: SignalRangeChecking

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	warning or error
Traceability	warning or error
Efficiency	none
Safety precaution	error

See Also

- Checking Signal Ranges
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Detect overflow

Select the diagnostic action to take if the value of a signal or parameter is too large to be represented by the signal or parameter's data type.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- This diagnostic applies only to overflows for integer and fixed-point data types.
- To check for floating-point overflows (e.g., Inf or NaN) for double or single data types, select the **Inf or NaN block output** diagnostic. (See “Inf or NaN block output” on page 1-250 for more information.)

Command-Line Information

Parameter: IntegerOverflowMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Working with Data Types
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Inf or NaN block output

Select the diagnostic action to take if the value of a block output is Inf or NaN at the current time step.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- This diagnostic applies only to floating-point overflows for double or single data types.
- To check for integer and fixed-point overflows, select the **Detect overflow** diagnostic. (See “Detect overflow” on page 1-248 for more information.)

Command-Line Information

Parameter: SignalInfNanChecking

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

"rt" prefix for identifiers

Select the diagnostic action to take during code generation if a Simulink object name (the name of a parameter, block, or signal) begins with `rt`.

Settings

Default: error

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- The default setting (error) causes code generation to terminate with an error if it encounters a Simulink object name (parameter, block, or signal), that begins with `rt`.
- This is intended to prevent inadvertent clashes with generated identifiers whose names begins with `rt`.

Command-Line Information

Parameter: RTPrefix

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'error'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Detect downcast

Select the diagnostic action to take when a parameter downcast occurs during simulation.

Settings

Default: error

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- A parameter downcast occurs if the computation of block output required converting the parameter's specified type to a type having a smaller range of values (for example, from `uint32` to `uint8`).
- This diagnostic applies only to named tunable parameters.

Command-Line Information

Parameter: `ParameterDowncastMsg`

Type: `string`

Value: `'none' | 'warning' | 'error'`

Default: `'error'`

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Detect overflow

Select the diagnostic action to take if a parameter overflow occurs during simulation.

Settings

Default: error

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- A parameter overflow occurs if Simulink software encounters a parameter whose data type's range is not large enough to accommodate the parameter's ideal value (the ideal value is either too large or too small to be represented by the data type). For example, suppose that the parameter's ideal value is 200 and its data type is `int8`. Overflow occurs in this case because the maximum value that `int8` can represent is 127.
- Parameter overflow differs from parameter precision loss, which occurs when the ideal parameter value is within the range of the data type and scaling being used, but cannot be represented exactly.
- Both parameter overflow and precision loss are quantization errors, and the distinction between them can be a fine one. The **Detect overflow** diagnostic reports all quantization errors greater than one bit. For very small parameter quantization errors, precision loss will be reported rather than an overflow when

$$(Max + Slope) \geq V_{ideal} > (Min - Slope)$$

where

- *Max* is the maximum value representable by the parameter data type

- *Min* is the minimum value representable by the parameter data type
- *Slope* is the slope of the parameter data type (slope = 1 for integers)
- V_{ideal} is the ideal value of the parameter

Command-Line Information

Parameter: ParameterOverflowMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'error'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Detect underflow

Select the diagnostic action to take when a parameter underflow occurs during simulation.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- Parameter underflow occurs when Simulink software encounters a parameter whose data type does not have enough precision to represent the parameter's ideal value because the ideal value is too small.
- When parameter underflow occurs, casting the ideal value to the data type causes the parameter's modeled value to become zero, and therefore to differ from its ideal value.

Command-Line Information

Parameter: ParameterUnderflowMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Detect precision loss

Select the diagnostic action to take when parameter precision loss occurs during simulation.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- Precision loss occurs when Simulink software encounters a parameter whose data type does not have enough precision to represent the parameter's value exactly. As a result, the modeled value differs from the ideal value.
- Parameter precision loss differs from parameter overflow, which occurs when the range of the parameter's data type, i.e., that maximum value that it can represent, is smaller than the ideal value of the parameter.
- Both parameter overflow and precision loss are quantization errors, and the distinction between them can be a fine one. The **Detect Parameter overflow** diagnostic reports all parameter quantization errors greater than one bit. For very small parameter quantization errors, precision loss will be reported rather than an overflow when

$$(Max + Slope) \geq V_{ideal} > (Min - Slope)$$

where

- *Max* is the maximum value representable by the parameter data type.
- *Min* is the minimum value representable by the parameter data type.

- $Slope$ is the slope of the parameter data type (slope = 1 for integers).
- V_{ideal} is the full-precision, ideal value of the parameter.

Command-Line Information

Parameter: ParameterPrecisionLossMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Detect loss of tunability

Select the diagnostic action to take when an expression with tunable variables is reduced to its numerical equivalent.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tip

If a tunable workspace variable is modified by Mask Initialization code, or is used in an arithmetic expression with unsupported operators or functions, the expression is reduced to a numeric expression and therefore cannot be tuned.

Command-Line Information

Parameter: ParameterTunabilityLossMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- [Diagnosing Simulation Errors](#)
- [Tunable Expressions](#)
- [Configuration Parameters Dialog Box](#)
- [Diagnostics Pane: Data Validity](#)

Detect read before write

Select the diagnostic action to take if the model attempts to read data from a data store to which it has not written data in this time step.

Settings

Default: Use local settings

Use local settings

For each local data store (defined by a Data Store Memory block or Simulink.Signal object in a model workspace) use the setting specified by the block. For each global data store (defined by a Simulink.Signal object in the base workspace) disable the diagnostic.

Disable All

Disables this diagnostic for all data stores accessed by the model.

Enable All As Warnings

Displays diagnostic as a warning at the MATLAB command line.

Enable All As Errors

Halts the simulation and displays the diagnostic in an error dialog box.

Command-Line Information

Parameter: ReadBeforeWriteMsg

Type: string

Value: 'UseLocalSettings' | 'DisableAll' | 'EnableAllAsWarning' | 'EnableAllAsError'

Default: 'UseLocalSettings'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Enable all as errors

See Also

- [Diagnosing Simulation Errors](#)
- [Working with Data Stores](#)
- [Data Store Memory block](#)
- [Simulink.Signal object](#)
- [Configuration Parameters Dialog Box](#)
- [Diagnostics Pane: Data Validity](#)

Detect write after read

Select the diagnostic action to take if the model attempts to write data to a data store after previously reading data from it in the current time step.

Settings

Default: Use local settings

Use local settings

For each local data store (defined by a Data Store Memory block or Simulink.Signal object in a model workspace) use the setting specified by the block. For each global data store (defined by a Simulink.Signal object in the base workspace) disable the diagnostic.

Disable All

Disables this diagnostic for all data stores accessed by the model.

Enable All As Warnings

Displays diagnostic as a warning at the MATLAB command line.

Enable All As Errors

Halts the simulation and displays the diagnostic in an error dialog box.

Command-Line Information

Parameter: WriteAfterReadMsg

Type: string

Value: 'UseLocalSettings' | 'DisableAll' | 'EnableAllAsWarning' | 'EnableAllAsError'

Default: 'UseLocalSettings'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Enable all as errors

See Also

- Diagnosing Simulation Errors
- Working with Data Stores
- Data Store Memory block
- Simulink.Signal object
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Detect write after write

Select the diagnostic action to take if the model attempts to write data to a data store twice in succession in the current time step.

Settings

Default: Use local settings

Use local settings

For each local data store (defined by a Data Store Memory block or Simulink.Signal object in a model workspace) use the setting specified by the block. For each global data store (defined by a Simulink.Signal object in the base workspace) disable the diagnostic.

Disable All

Disables this diagnostic for all data stores accessed by the model.

Enable All As Warnings

Displays diagnostic as a warning at the MATLAB command line.

Enable All As Errors

Halts the simulation and displays the diagnostic in an error dialog box.

Command-Line Information

Parameter: WriteAfterWriteMsg

Type: string

Value: 'UseLocalSettings' | 'DisableAll' | 'EnableAllAsWarning' | 'EnableAllAsError'

Default: 'UseLocalSettings'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Enable all as errors

See Also

- Diagnosing Simulation Errors
- Working with Data Stores
- Data Store Memory block
- Simulink.Signal object
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Multitask data store

Select the diagnostic action to take when one task reads data from a Data Store Memory block to which another task writes data.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- Such a situation is safe only if one of the tasks cannot interrupt the other, such as when the data store is a scalar and the writing task uses an atomic copy operation to update the store or the target does not allow the tasks to preempt each other.
- You should disable this diagnostic (set it to none) only if the application warrants it, such as if the application uses a cyclic scheduler that prevents tasks from preempting each other.

Command-Line Information

Parameter: MultiTaskDSMMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Working with Data Stores
- Data Store Memory block
- Simulink.Signal object
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Duplicate data store names

Select the diagnostic action to take when the model contains multiple data stores that have the same name. The data stores can be defined with Data Store Memory blocks or `Simulink.Signal` objects.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tip

This diagnostic is useful for detecting errors that can occur when a lower-level data store unexpectedly shadows a higher-level data store that has the same name.

Command-Line Information

Parameter: UniqueDataStoreMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact

See Also

- Diagnosing Simulation Errors
- Working with Data Stores
- Data Store Memory block
- `Simulink.Signal` object
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Detect multiple driving blocks executing at the same time step

Select the diagnostic action to take when the software detects a Merge block with more than one driving block executing at the same time step.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- Connecting the inputs of a Merge block to multiple driving blocks that execute at the same time step can lead to inconsistent results for both simulation and generated code. Set **Detect multiple driving blocks executing at the same time step** to error to avoid such situations.
- If **Underspecified initialization detection** is set to Simplified, this parameter is disabled, and Simulink software automatically uses the strictest setting (error) for this diagnostic. Multiple driving blocks executing at the same time step always result in an error.

Dependency

This parameter is enabled only if **Underspecified initialization detection** is set to Classic.

Command-Line Information

Parameter: MergeDetectMultiDrivingBlocksExec

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	error
Traceability	error
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Merge block
- “Check consistency of initialization parameters for Outport and Merge blocks”
- “Underspecified initialization detection” on page 1-276
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Underspecified initialization detection

Select how Simulink software handles initialization of initial conditions for conditionally executed subsystems, Merge blocks, subsystem elapsed time, and Discrete-Time Integrator blocks.

Settings

Default: Classic

Classic

Initial conditions are initialized the same way they were prior to R2008b.

Simplified

Initial conditions are initialized using the enhanced behavior, which can improve the consistency of simulation results.

Tips

- Use **Classic** to ensure compatibility with previous releases of Simulink.
- Use **Simplified** to improve the consistency of simulation results, especially for models that do not specify initial conditions for conditional subsystem output ports, and for models that have conditionally executed subsystem output ports connected to S-functions.
- When using **Simplified** initialization mode, you must set both **Mux blocks used to create bus signals**, and **Bus signal treated as vector** to error on the Connectivity Diagnostics pane.
- For existing models, MathWorks recommends using the Model Advisor to migrate your model to the new settings. To migrate your model, run the following Model Advisor checks:
 - **Check for proper bus usage**
 - **Check consistency of initialization parameters for Outport and Merge blocks**

For more information, see “Check consistency of initialization parameters for Outport and Merge blocks” in the *Simulink Reference*.

Dependencies

Selecting **Classic** enables the following parameters:

- **Detect multiple driving blocks executing at the same time step**
- **Check undefined subsystem initial output**
- **Check preactivation output of execution context**
- **Check runtime output of execution context**

Selecting **Simplified** disables these parameters, and automatically sets **Detect multiple driving blocks executing at the same time step** to error.

Command-Line Information

Parameter: UnderspecifiedInitializationDetection

Type: string

Value: 'classic' | 'simplified'

Default: 'classic'

Recommended Settings

Application	Setting
Debugging	Simplified
Traceability	Simplified
Efficiency	Simplified
Safety precaution	Simplified

See Also

- “Check consistency of initialization parameters for Outport and Merge blocks”
- Merge block
- Discrete-Time Integrator block

- “Creating Conditional Subsystems”
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Check undefined subsystem initial output

Specify whether to display a warning if the model contains a conditionally executed subsystem in which a block with a specified initial condition drives an Outport block with an undefined initial condition

Settings

Default: On



Displays a warning if the model contains a conditionally executed subsystem in which a block with a specified initial condition drives an Outport block with an undefined initial condition.

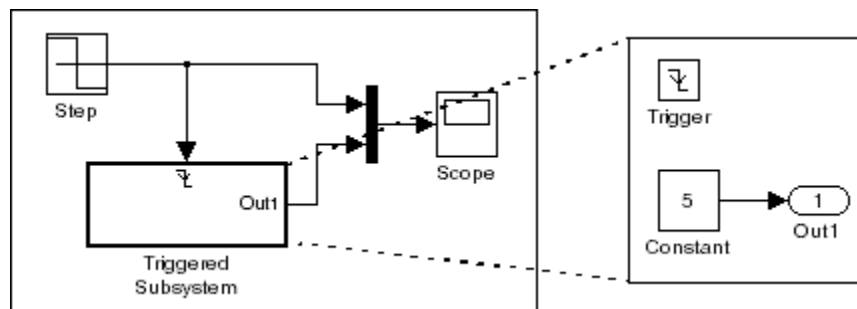


Does not display a warning.

Tips

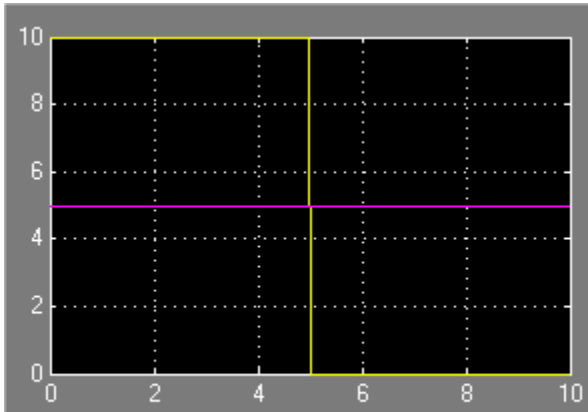
- This situation occurs when a block with a specified initial condition, such as a Constant, Initial Condition, or Delay block, drives an Outport block with an undefined initial condition (**Initial output** parameter is set to []).
- Models with such subsystems can produce initial results (i.e., before initial activation of the conditionally executed subsystem) in the current release that differ from initial results produced in Release 13 or earlier releases.

Consider for example the following model.

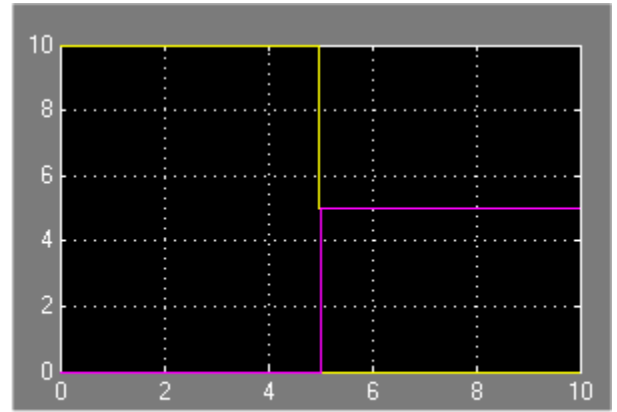


This model does not define the initial condition of the triggered subsystem's output port.

The following figure compares the superimposed output of this model's Step block and the triggered subsystem in Release 13 and the current release.



Release 13



Current Release

Notice that the initial output of the triggered subsystem differs between the two releases. This is because Release 13 and earlier releases use the initial output of the block connected to the output port (i.e., the Constant block) as the triggered subsystem's initial output. By contrast, this release outputs 0 as the initial output of the triggered subsystem because the model does not specify the port's initial output.

Dependency

This parameter is enabled only if **Underspecified initialization detection** is set to **Classic**.

Command-Line Information

Parameter: CheckSSInitialOutputMsg

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	On

See Also

- Diagnosing Simulation Errors
- “Creating Conditional Subsystems”
- “Underspecified initialization detection” on page 1-276
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Check preactivation output of execution context

Specify whether to display a warning if Simulink software detects potential initial output differences from previous releases.

Settings

Default: Off



On

Displays a warning if Simulink software detects potential initial output differences from previous releases.



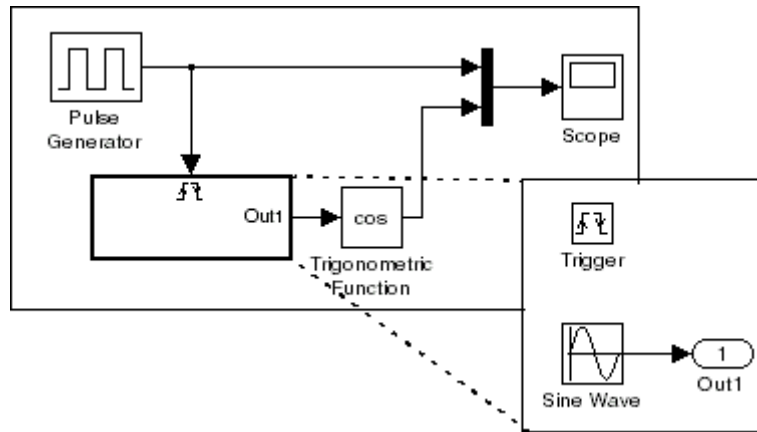
Off

Does not display a warning.

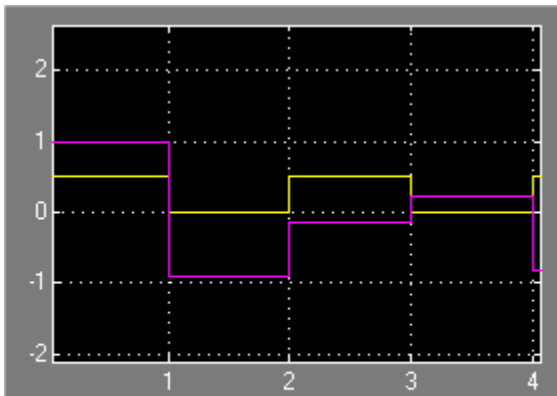
Tips

- This diagnostic is triggered if the model contains a block that meets the following conditions:
 - The block produces nonzero output for zero input (e.g., a Cosine block).
 - The block is connected to an output of a conditionally executed subsystem.
 - The block inherits its execution context from that subsystem.
 - The Output to which it is connected has an undefined initial condition, i.e., the Output block's **Initial output** parameter is set to [].
- Models with blocks that meet these criteria can produce initial results (i.e., before the conditionally executed subsystem is first activated in the current release that differ from initial results produced in Release 13 or earlier releases.

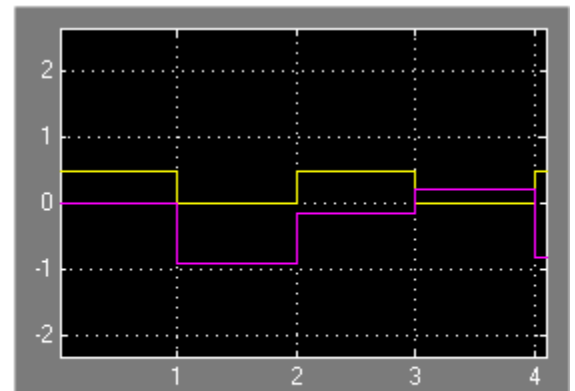
Consider for example the following model.



The following figure compares the superimposed output of the Pulse Generator and cos block in Release 13 and the current release.



Release 13



Current Release

Note that the initial output of the cos block differs between the two releases. This is because in Release 13, the cos block belongs to the execution context of the root system and hence executes at every time step whereas in the current release, the cos block belongs to the execution context of the triggered subsystem and hence executes only when the triggered subsystem executes.

Dependency

This parameter is enabled only if **Underspecified initialization detection** is set to **Classic**.

Command-Line Information

Parameter: CheckExecutionContextPreStartOutputMsg

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	On

See Also

- Diagnosing Simulation Errors
- “Underspecified initialization detection” on page 1-276
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Check runtime output of execution context

Specify whether to display a warning if Simulink software detects potential output differences from previous releases.

Settings

Default: Off



On

Displays a warning if Simulink software detects potential output differences from previous releases.



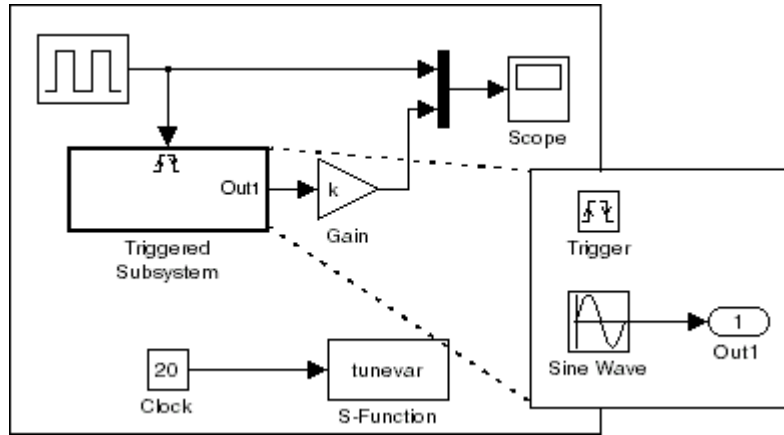
Off

Does not display a warning.

Tips

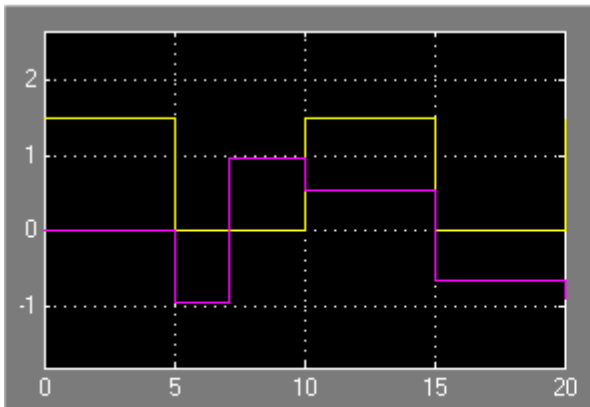
- This diagnostic is triggered if the model contains a block that meets the following conditions:
 - The block has a tunable parameter.
 - The block is connected to an output of a conditionally executed subsystem.
 - The block inherits its execution context from that subsystem.
 - The Output to which it is connected has an undefined initial condition, i.e., the Output block's **Initial output** parameter is set to [].
- Models with blocks that meet these criteria can produce results when the parameter is tuned in the current release that differ from results produced in Release 13 or earlier releases.

Consider for example the following model.

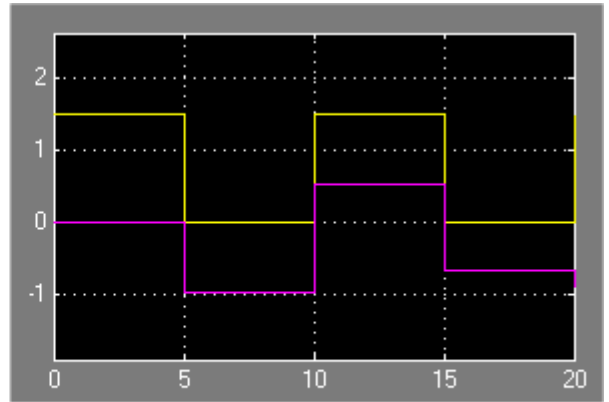


In this model, the tunevar S-function changes the value of the Gain block's k parameter and updates the diagram at simulation time 7 (i.e., it simulates tuning the parameter).

The following figure compares the superimposed output of the model's Pulse Generator block and its Gain block in Release 13 and the current release.



Release 13



Current Release

Note that the output of the Gain block changes at time 7 in Release 13 but does not change in the current release. This is because in Release 13, the Gain block belongs to the execution context of the root system and hence executes at every time step whereas in the current release, the Gain

block belongs to the execution context of the triggered subsystem and hence executes only when the triggered subsystem executes, i.e., at times 5, 10, 15, and 20.

Dependency

This parameter is enabled only if **Underspecified initialization detection** is set to **Classic**.

Command-Line Information

Parameter: CheckExecutionContextRuntimeOutputMsg

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	On

See Also

- Diagnosing Simulation Errors
- “Underspecified initialization detection” on page 1-276
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Array bounds exceeded

Select the diagnostic action to take when blocks write data to locations outside the memory allocated to them.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- Use this option to check whether execution of each instance of a block during model simulation writes data to memory locations not allocated to the block. This can happen only if your model includes a user-written S-function that has a bug.
- Enabling this option slows down model execution considerably. Thus, you should enable it only if you suspect that your model contains a user-written S-function that has a bug.
- This option causes Simulink software to check whether a block writes outside the memory allocated to it during simulation. Typically this can happen only if your model includes a user-written S-function that has a bug.
- See Writing S-Functions for more information on using this option.

Command-Line Information

Parameter: ArrayBoundsChecking

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	warning
Traceability	No impact
Efficiency	none
Safety precaution	No impact

See Also

- Diagnosing Simulation Errors
- Writing S-Functions
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Model Verification block enabling

Enable model verification blocks in the current model either globally or locally.

Settings

Default: Use local settings

Use local settings

Enables or disables blocks based on the value of the `Enable assertion` parameter of each block. If a block's `Enable assertion` parameter is on, the block is enabled; otherwise, the block is disabled.

Enable All

Enables all model verification blocks in the model regardless of the settings of their `Enable assertion` parameters.

Disable All

Disables all model verification blocks in the model regardless of the settings of their `Enable assertion` parameters.

Dependency

Simulation and code generation ignore the **Model Verification block enabling** parameter when model verification blocks are inside a S-function.

Command-Line Information

Parameter: AssertControl

Type: string

Value: 'UseLocalSettings' | 'EnableAll' | 'DisableAll'

Default: 'UseLocalSettings'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	Disable all

See Also

- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Data Validity

Diagnostics Pane: Type Conversion

The screenshot shows a configuration dialog box titled "Type Conversion". It contains several settings, each with a dropdown menu:

- Unnecessary type conversions: none
- Vector/matrix block input conversion: none
- 32-bit integer to single precision float conversion: warning

Below these is a section titled "Fixed-point Constants" with three settings:

- Detect underflow: none
- Detect overflow: none
- Detect precision loss: none

In this section...

“Type Conversion Diagnostics Overview” on page 1-293

“Unnecessary type conversions” on page 1-294

“Vector/matrix block input conversion” on page 1-295

“32-bit integer to single precision float conversion” on page 1-297

“Detect underflow” on page 1-299

“Detect precision loss” on page 1-301

“Detect overflow” on page 1-303

Type Conversion Diagnostics Overview

Specify the diagnostic actions that Simulink software should take when it detects a data type conversion problem while compiling the model.

Configuration

Set the parameters displayed.

Tips

- The options are typically to do nothing or to display a warning or an error message.
- A warning does not terminate a simulation, but an error does.

See Also

- Diagnosing Simulation Errors
- Solver Diagnostics
- Sample Time Diagnostics
- Data Validity Diagnostics
- Connectivity Diagnostics
- Compatibility Diagnostics
- Model Referencing Diagnostics
- Saving Diagnostics
- Configuration Parameters Dialog Box
- Diagnostics Pane: Type Conversion

Unnecessary type conversions

Select the diagnostic action to take when Simulink software detects a Data Type Conversion block used where no type conversion is necessary.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

Command-Line Information

Parameter: UnnecessaryDatatypeConvMsg

Type: string

Value: 'none' | 'warning'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	warning

See Also

- Diagnosing Simulation Errors
- Data Type Conversion block
- Configuration Parameters Dialog Box
- Diagnostics Pane: Type Conversion

Vector/matrix block input conversion

Select the diagnostic action to take when Simulink software detects a vector-to-matrix or matrix-to-vector conversion at a block input.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

Simulink software converts vectors to row or column matrices and row or column matrices to vectors under the following circumstances:

- If a vector signal is connected to an input that requires a matrix, Simulink software converts the vector to a one-row or one-column matrix.
- If a one-column or one-row matrix is connected to an input that requires a vector, Simulink software converts the matrix to a vector.
- If the inputs to a block consist of a mixture of vectors and matrices and the matrix inputs all have one column or one row, Simulink software converts the vectors to matrices having one column or one row, respectively.

Command-Line Information

Parameter: VectorMatrixConversionMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Determining Output Signal Dimensions
- Configuration Parameters Dialog Box
- Diagnostics Pane: Type Conversion

32-bit integer to single precision float conversion

Select the diagnostic action to take if Simulink software detects a 32-bit integer value was converted to a floating-point value.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

Tip

Converting a 32-bit integer value to a floating-point value can result in a loss of precision. See *Working with Data Types* for more information.

Command-Line Information

Parameter: Int32ToFloatConvMsg

Type: string

Value: 'none' | 'warning'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	warning

See Also

- Diagnosing Simulation Errors

- Working with Data Types
- Configuration Parameters Dialog Box
- Diagnostics Pane: Type Conversion

Detect underflow

Specifies diagnostic action to take when a fixed-point constant underflow occurs during simulation.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- This diagnostic applies only to fixed-point constants (net slope and net bias).
- Fixed-point constant underflow occurs when Simulink software encounters a fixed-point constant whose data type does not have enough precision to represent the ideal value of the constant because the ideal value is too small.
- When fixed-point constant underflow occurs, casting the ideal value to the data type causes the value of the fixed-point constant to become zero, and therefore to differ from its ideal value.

Dependency

This parameter requires a Simulink Fixed Point license.

Command-Line Information

Parameter: FixptConstUnderflowMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Net Slope and Net Bias Precision Issues
- Configuration Parameters Dialog Box
- Diagnostics Pane: Type Conversion

Detect precision loss

Specifies diagnostic action to take when a fixed-point constant precision loss occurs during simulation.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- This diagnostic applies only to fixed-point constants (net slope and net bias).
- Precision loss occurs when Simulink software converts a fixed-point constant to a data type which does not have enough precision to represent the exact value of the constant. As a result, the quantized value differs from the ideal value.
- Fixed-point constant precision loss differs from fixed-point constant overflow. Overflow occurs when the range of the parameter's data type, that is, the maximum value that it can represent, is smaller than the ideal value of the parameter.

Dependency

This parameter requires a Simulink Fixed Point license.

Command-Line Information

Parameter: FixptConstPrecisionLossMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Net Slope and Net Bias Precision Issues
- Configuration Parameters Dialog Box
- Diagnostics Pane: Type Conversion

Detect overflow

Specifies diagnostic action to take when a fixed-point constant overflow occurs during simulation.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- This diagnostic applies only to fixed-point constants (net slope and net bias).
- Overflow occurs when the Simulink software converts a fixed-point constant to a data type whose range is not large enough to accommodate the ideal value of the constant. The ideal value is either too large or too small to be represented by the data type. For example, suppose that the ideal value is 200 and the converted data type is `int8`. Overflow occurs in this case because the maximum value that `int8` can represent is 127.
- Fixed-point constant overflow differs from fixed-point constant precision loss. Precision loss occurs when the ideal fixed-point constant value is within the range of the data type and scaling being used, but cannot be represented exactly.

Dependency

This parameter requires a Simulink Fixed Point license.

Command-Line Information

Parameter: `FixptConstOverflowMsg`

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

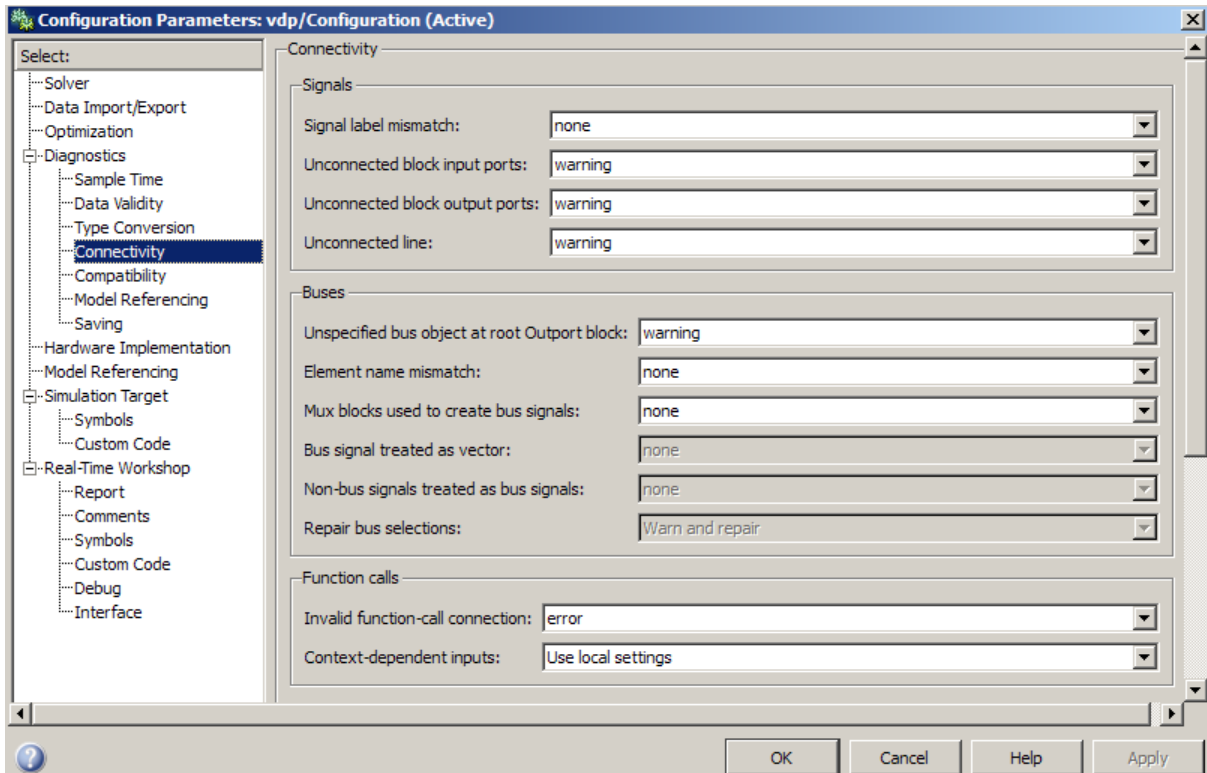
Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Net Slope and Net Bias Precision Issues
- Configuration Parameters Dialog Box
- Diagnostics Pane: Type Conversion

Diagnostics Pane: Connectivity



In this section...

- “Connectivity Diagnostics Overview” on page 1-307
- “Signal label mismatch” on page 1-308
- “Unconnected block input ports” on page 1-310
- “Unconnected block output ports” on page 1-312
- “Unconnected line” on page 1-314
- “Unspecified bus object at root Outport block” on page 1-316
- “Element name mismatch” on page 1-318

In this section...

“Mux blocks used to create bus signals” on page 1-320

“Bus signal treated as vector” on page 1-323

“Non-bus signals treated as bus signals” on page 1-326

“Repair bus selections” on page 1-328

“Invalid function-call connection” on page 1-330

“Context-dependent inputs” on page 1-332

Connectivity Diagnostics Overview

Specify the diagnostic actions that Simulink software should take when it detects a problem with block connections while compiling the model.

Configuration

Set the parameters displayed.

Tips

- The options are typically to do nothing or to display a warning or an error message.
- A warning does not terminate a simulation, but an error does.

See Also

- Diagnosing Simulation Errors
- Solver Diagnostics
- Sample Time Diagnostics
- Data Validity Diagnostics
- Type Conversion Diagnostics
- Compatibility Diagnostics
- Model Referencing Diagnostics
- Saving Diagnostics
- Configuration Parameters Dialog Box
- Diagnostics Pane: Connectivity

Signal label mismatch

Select the diagnostic action to take when Simulink software detects virtual signals that have a common source signal but different labels.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Command-Line Information

Parameter: SignalLabelMismatchMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors

- Virtual Signals
- Configuration Parameters Dialog Box
- Diagnostics Pane: Connectivity

Unconnected block input ports

Select the diagnostic action to take when the model contains a block with an unconnected input.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Command-Line Information

Parameter: UnconnectedInputMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors

- Configuration Parameters Dialog Box
- Diagnostics Pane: Connectivity

Unconnected block output ports

Select the diagnostic action to take when the model contains a block with an unconnected output.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Command-Line Information

Parameter: UnconnectedOutputMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors

- Configuration Parameters Dialog Box
- Diagnostics Pane: Connectivity

Unconnected line

Select the diagnostic action to take when the Model contains an unconnected line or an unmatched Goto or From block.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Command-Line Information

Parameter: UnconnectedLineMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors

- Goto block
- From block
- Configuration Parameters Dialog Box
- Diagnostics Pane: Connectivity

Unspecified bus object at root Output block

Select the diagnostic action to take while generating a simulation target for a referenced model if any of the model's root Output blocks is connected to a bus but does not specify a bus object (see `Simulink.Bus`).

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Command-Line Information

Parameter: `RootOutputRequireBusObject`

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors

- Outport block
- Simulink.Bus
- Configuration Parameters Dialog Box
- Diagnostics Pane: Connectivity

Element name mismatch

Select the diagnostic action to take if the name of a bus element does not match the name specified by the corresponding bus object.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tip

You can use this diagnostic along with bus objects to ensure that your model meets bus element naming requirements imposed by some blocks, such as the Switch block.

Command-Line Information

Parameter: BusObjectLabelMismatch

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- [Diagnosing Simulation Errors](#)
- [Configuration Parameters Dialog Box](#)
- [Diagnostics Pane: Connectivity](#)

Mux blocks used to create bus signals

Select the diagnostic action to take if Simulink detects a Mux block that creates a virtual bus.

Settings

Default: warning

none

Simulink software takes no action.

This option disables checking for Mux blocks used to create virtual bus signals.

warning

Simulink software displays a warning.

This option does not enforce strict bus behavior. However, if it detects a Mux block that creates a virtual bus during model update or simulation, it displays a message in the MATLAB Command Window that identifies the offending block. It does this for the first ten Mux blocks that it encounters that violate strict bus behavior.

error

Simulink terminates the simulation and displays an error message identifying the first Mux block it encounters that is used to create a virtual bus. If this option is selected, a Mux block with more than one input is allowed to output only a vector signal, and a Mux block with only one input is allowed to output only a scalar, vector, or matrix signal.

Tips

- This diagnostic detects use of Mux blocks to create virtual buses. The diagnostic considers a signal created by a Mux block to be a virtual bus if the signal meets either or both of the following conditions:
 - A Bus Selector block individually selects one or more of the signal elements (as opposed to the entire signal).
 - The signal components have differing data types, numeric types (complex or real), dimensionality, storage classes (see the *Real-Time Workshop*

User's Guide for information on storage classes), or sampling modes (see the Signal Processing Blockset™ documentation for information on frame-based sampling).

- Using a Mux block to create a virtual bus does not support strong type checking and increases the likelihood of run-time errors. In new applications, do not use Mux blocks to create bus signals. Consider upgrading existing applications to that use of Mux blocks.
 - Simulink generates a warning when you load a model created in a release prior to R2010a, if that model contains a Mux block to create a bus signal. For new models, Simulink generates an error.
- If you are using simplified initialization mode, you must set this diagnostic to error. For more information, see Underspecified initialization detection.
- You can identify Mux blocks used to create virtual buses using the Model Advisor check **Check for proper bus usage**. For more information, see “Check for proper bus usage”.
- You can avoid strict bus behavior errors and warnings by using `slreplace_mux` to remove Mux blocks that violate strict bus behavior from your model. Before executing the command, you should set this diagnostic to warning or none.
- See Avoiding Mux/Bus Mixtures for more information.

Dependency

Selecting error enables the following parameter:

- **Bus signal treated as vector**

Command-Line Information

Parameter: StrictBusMsg

Type: string

Value: 'none' | 'warning' | 'ErrorLevel1' |
'WarnOnBusTreatedAsVector' | 'ErrorOnBusTreatedAsVector'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Avoiding Mux/Bus Mixtures
- Diagnosing Simulation Errors
- Mux block
- Bus Creator block
- Bus Selector block
- Underspecified initialization detection
- “Check for proper bus usage”
- Configuration Parameters Dialog Box
- Diagnostics Pane: Connectivity

Bus signal treated as vector

Select the diagnostic action to take when Simulink software detects a virtual bus signal that is used as a mux signal.

Settings

Default: warning

none

Disables checking for virtual buses used as muxes.

warning

Simulink software displays a warning if it detects a virtual bus used as a mux. This option does not enforce strict bus behavior.

error

Simulink software terminates the simulation and displays an error message when it builds a model that uses any virtual bus as a mux.

Tips

- This diagnostic detects the use of virtual bus signals used to specify muxes. The diagnostic considers a virtual bus signal to be used as a mux if it is input to a Demux block or to any block that can input a mux or a vector but is not formally defined as bus-capable. See Bus-Capable Blocks for details.
- Virtual buses can be used as muxes only when they contain no nested buses and all constituent signals have the same attributes. This practice is deprecated as of R2007a (V6.6) and may cease to be supported at some future time. MathWorks, therefore, discourages mixing virtual buses with muxes in new applications, and encourages upgrading existing applications to avoid such mixtures.
- If you are using simplified initialization mode, you must set this diagnostic to **error**. For more information, see Underspecified initialization detection.
- You can identify bus signals that are treated as a vectors using the Model Advisor check **Check for proper bus usage**. For more information, see “Check for proper bus usage”.
- You can eliminate warnings and errors about virtual buses used as muxes by using `Simulink.BlockDiagram.addBusToVector` to insert a Bus to

Vector block into any virtual bus signal that is used as a mux. Before executing the command, you should set this diagnostic to warning or none.

- See Avoiding Mux/Bus Mixtures for more information.

Dependency

This parameter is enabled only when **Mux blocks used to create bus signals** is set to error.

Command-Line Information

Parameter: StrictBusMsg

Type: string

Value: 'none' | 'warning' | 'ErrorLevel1' | 'WarnOnBusTreatedAsVector' | 'ErrorOnBusTreatedAsVector'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Avoiding Mux/Bus Mixtures
- Diagnosing Simulation Errors
- Bus-Capable Blocks
- Demux block
- Bus to Vector block
- Underspecified initialization detection

- “Check for proper bus usage”
- `Simulink.BlockDiagram.addBusToVector`
- Configuration Parameters Dialog Box
- Diagnostics Pane: Connectivity

Non-bus signals treated as bus signals

Detect when Simulink implicitly converts a non-bus signal to a bus signal to support connecting the signal to a Bus Assignment or Bus Selector block.

Settings

Default: none

none

Implicitly converts non-bus signals to bus signals to support connecting the signal to a Bus Assignment or Bus Selector block.

warning

Simulink displays a warning, indicating that it has converted a non-bus signal to a bus signal. The warning lists the non-bus signals that Simulink converts.

error

Simulink terminates the simulation without performing converting non-bus signals to bus signals. The error message lists the non-bus signal that is being treated as a bus signal.

Tips

- Using a Mux block to create a virtual bus does not support strong type checking and increases the likelihood of run-time errors. In new applications, do not use Mux blocks to create bus signals. Consider upgrading existing applications to that use of Mux blocks.
 - Simulink generates a warning when you load a model created in a release prior to R2010a, if that model contains a Mux block to create a bus signal. For new models, Simulink generates an error.
- See Avoiding Mux/Bus Mixtures for more information.

Dependency

This parameter is enabled only when **Mux blocks used to create bus signals** is set to error.

Command-Line Information

Parameter: NonbusSignalTreatedAsBus

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- [Avoiding Mux/Bus Mixtures](#)
- [Diagnosing Simulation Errors](#)
- [Bus-Capable Blocks](#)
- [Demux block](#)
- [Bus to Vector block](#)
- [Simulink.BlockDiagram.addBusToVector](#)
- [Configuration Parameters Dialog Box](#)
- [Diagnostics Pane: Connectivity](#)

Repair bus selections

Repair broken selections in the Bus Selector and Bus Assignment block parameter dialogs due to upstream bus hierarchy changes.

Settings

Default: Warn and repair

Warn and repair

Simulink displays a warning, indicating the block parameters for Bus Selector and Bus Assignment blocks that Simulink repaired to reflect upstream bus hierarchy changes.

Error without repair

Simulink terminates the simulation and displays an error message indicating the block parameters that you need to repair for Bus Selector and Bus Assignment blocks to reflect upstream bus hierarchy changes.

Tips

- See Avoiding Mux/Bus Mixtures for more information.

Dependency

This parameter is enabled only when **Mux blocks used to create bus signals** is set to error.

Command-Line Information

Parameter: BusNameAdapt

Type: string

Values: 'WarnAndRepair' | 'ErrorWithoutRepair'

Default: 'WarnAndRepair'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Warn and repair

See Also

- [Avoiding Mux/Bus Mixtures](#)
- [“Nesting Buses”](#)
- [Diagnosing Simulation Errors](#)
- [Bus-Capable Blocks](#)
- [Configuration Parameters Dialog Box](#)
- [Diagnostics Pane: Connectivity](#)

Invalid function-call connection

Select the diagnostic action to take if Simulink software detects incorrect use of a function-call subsystem.

Settings

Default: error

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- See the "Function-call systems" examples in the Simulink "Subsystem Semantics" library for examples of invalid uses of function-call subsystems.
- Setting this parameter to none or warning can lead to invalid simulation results.
- Setting this parameter to none or warning may cause Simulink software to insert extra delay operations.

Command-Line Information

Parameter: InvalidFcnCallConnMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'error'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors
- Subsystem Semantics library
- Configuration Parameters Dialog Box
- Diagnostics Pane: Connectivity

Context-dependent inputs

Select the diagnostic action to take when Simulink software has to compute any of a function-call subsystem's inputs directly or indirectly during execution of a call to a function-call subsystem.

Settings

Default: Use local settings

Use local settings

Issues a warning only if the corresponding diagnostic is selected on the function-call subsystem's parameters dialog box (see the documentation for the Subsystem block's parameter dialog box for more information).

Enable all

Enables this diagnostic for all function-call subsystems in this model.

Disable all

Disables this diagnostic for all function-call subsystems in this model.

Tips

- This situation occurs when executing a function-call subsystem can change its inputs.
- See the "Function-call systems" examples in the Simulink "Subsystem Semantics" library for examples of such function-call subsystems).

Command-Line Information

Parameter: FcnCallInpInsideContextMsg

Type: string

Value: 'Use local settings' | 'Enable All' | 'Disable All'

Default: 'Use local settings'

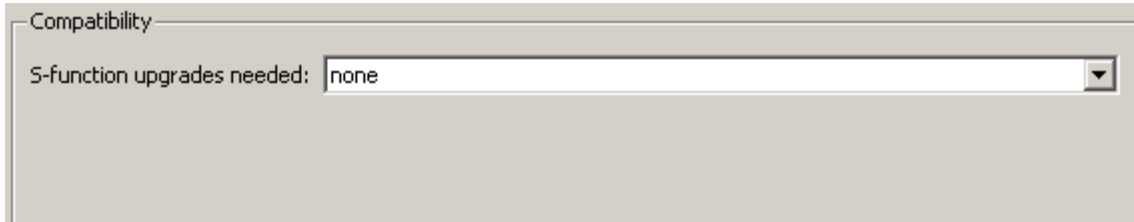
Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Enable all

See Also

- Subsystem Semantics library
- Subsystem block
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Connectivity

Diagnostics Pane: Compatibility



In this section...
“Compatibility Diagnostics Overview” on page 1-335
“S-function upgrades needed” on page 1-336

Compatibility Diagnostics Overview

Specify the diagnostic actions that Simulink software should take when it detects an incompatibility between the current version of Simulink software and the model.

Configuration

Set the parameters displayed.

Tips

- The options are typically to do nothing or to display a warning or an error message.
- A warning does not terminate a simulation, but an error does.

See Also

- Diagnosing Simulation Errors
- Solver Diagnostics
- Sample Time Diagnostics
- Data Validity Diagnostics
- Type Conversion Diagnostics
- Connectivity Diagnostics
- Compatibility Diagnostics
- Model Referencing Diagnostics
- Saving Diagnostics
- Configuration Parameters Dialog Box
- Diagnostics Pane: Compatibility

S-function upgrades needed

Select the diagnostic action to take if Simulink software encounters a block that has not been upgraded to use features of the current release.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Command-Line Information

Parameter: SfunCompatibilityCheckMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Diagnosing Simulation Errors

- Configuration Parameters Dialog Box
- Diagnostics Pane: Compatibility

Diagnostics Pane: Model Referencing

Model Referencing	
Model block version mismatch:	none
Port and parameter mismatch:	none
Model configuration mismatch:	none
Invalid root Inport/Outport block connection:	none
Unsupported data logging:	warning

In this section...

- “Model Referencing Diagnostics Overview” on page 1-339
- “Model block version mismatch” on page 1-340
- “Port and parameter mismatch” on page 1-342
- “Model configuration mismatch” on page 1-344
- “Invalid root Inport/Outport block connection” on page 1-346
- “Unsupported data logging” on page 1-350

Model Referencing Diagnostics Overview

Specify the diagnostic actions that Simulink software should take when it detects an incompatibility relating to a model reference hierarchy.

Configuration

Set the parameters displayed.

Tips

- The options are typically to do nothing or to display a warning or an error message.
- A warning does not terminate a simulation, but an error does.

See Also

- Referencing Models
- Diagnosing Simulation Errors
- Solver Diagnostics
- Sample Time Diagnostics
- Data Validity Diagnostics
- Type Conversion Diagnostics
- Connectivity Diagnostics
- Compatibility Diagnostics
- Saving Diagnostics
- Configuration Parameters Dialog Box
- Diagnostics Pane: Model Referencing

Model block version mismatch

Select the diagnostic action to take when loading or updating this model if Simulink software detects a mismatch between the version of the model used to create or refresh a Model block in this model and the referenced model's current version.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning and refreshes the Model block.

error

Simulink software displays an error message and does not refresh Model block.

Tip

If you have enabled display of referenced model version numbers on Model blocks for this model (see [Displaying Referenced Model Version Numbers](#)), Simulink software displays a version mismatch on the Model block icon, for example: Rev:1.0 != 1.2.

Command-Line Information

Parameter: ModelReferenceVersionMismatchMessage

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	none

See Also

- Referencing Models
- Diagnosing Simulation Errors
- Displaying Referenced Model Version Numbers
- Configuration Parameters Dialog Box
- Diagnostics Pane: Model Referencing

Port and parameter mismatch

Select the diagnostic action to take if Simulink software detects a port or parameter mismatch during model loading or updating.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning and refreshes the Model block.

error

Simulink software displays an error message and does not refresh the Model block.

Tips

- Port mismatches occur when there is a mismatch between the I/O ports of a Model block and the root-level I/O ports of the model it references.
- Parameter mismatches occur when there is a mismatch between the parameter arguments recognized by the Model block and the parameter arguments declared by the referenced model.
- Model block icons can display a message indicating port or parameter mismatches. To enable this feature, select **Block displays > Model Block I/O Mismatch** from the parent model's **Format** menu.

Command-Line Information

Parameter: ModelReferenceIOMismatchMessage

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Referencing Models
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Model Referencing

Model configuration mismatch

Select the diagnostic action to take if the configuration parameters of a model referenced by this model do not match this model's configuration parameters or are inappropriate for a referenced model.

Settings

Default: none

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tip

Set this diagnostic to warning or error if you suspect that an inappropriate or mismatched configuration parameter may be causing your model to give the wrong result.

Command-Line Information

Parameter: ModelReferenceCSMismatchMessage

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	warning

See Also

- Referencing Models
- Configuration Parameter Requirements
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Model Referencing

Invalid root Inport/Outport block connection

Select the diagnostic action to take if Simulink software detects invalid internal connections to this model's root-level Output port blocks.

Settings

Default: none

none

Simulink software silently inserts hidden blocks to satisfy the constraints wherever possible.

warning

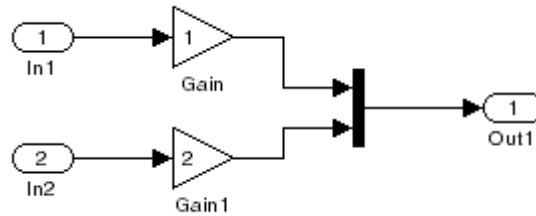
Simulink software warns you that a connection constraint has been violated and attempts to satisfy the constraint by inserting hidden blocks.

error

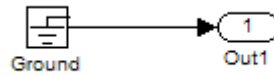
Simulink software terminates the simulation or code generation and displays an error message.

Tips

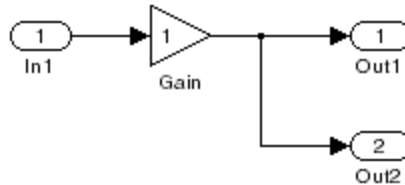
- In some cases (such as function-call feedback loops), automatically inserted hidden blocks may introduce delays and thus may change simulation results.
- Auto-inserting hidden blocks to eliminate root I/O problems stops at subsystem boundaries. Therefore, you may need to manually modify models with subsystems that violate any of the constraints below.
- The types of invalid internal connections are:
 - A root Output port is connected directly or indirectly to more than one nonvirtual block port:



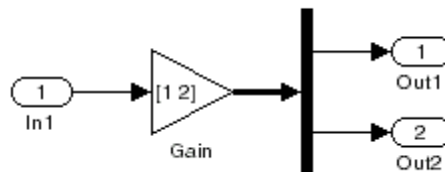
- A root Output port is connected to a Ground block:



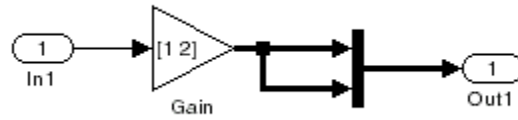
- Two root Output blocks are connected to the same block port:



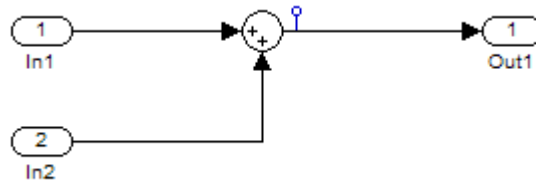
- An Output port block is connected to some elements of a block output and not others:



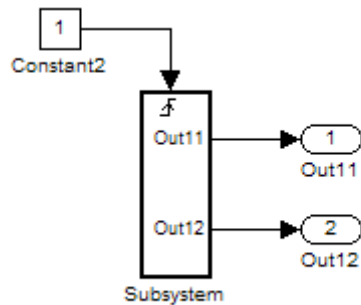
- An Output block is connected more than once to the same element:



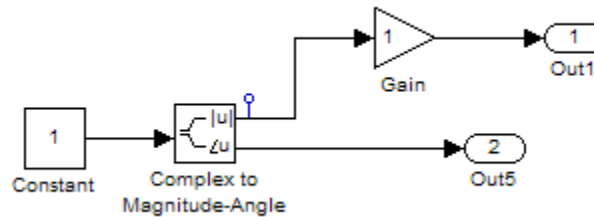
- The signal driving the root outputport is a test point:



- The output port has a constant sample time, but the driving block has a non-constant sample time:



- The driving block has a constant sample time and multiple output ports, and one of the other output ports of the block is a test point.



Command-Line Information

Parameter: ModelReferenceIOMsg
Type: string
Value: 'none' | 'warning' | 'error'
Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error

See Also

- Referencing Models
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Diagnostics Pane: Model Referencing

Unsupported data logging

Select the diagnostic action to take if this model contains To Workspace blocks or Scope blocks with data logging enabled.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning.

error

Simulink software terminates the simulation and displays an error message.

Tips

- The default action warns you that Simulink software does not support use of these blocks to log data from referenced models.
- See Logging Referenced Model Signals for information on how to log signals from a reference to this model.

Command-Line Information

Parameter: ModelReferenceDataLoggingMessage

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

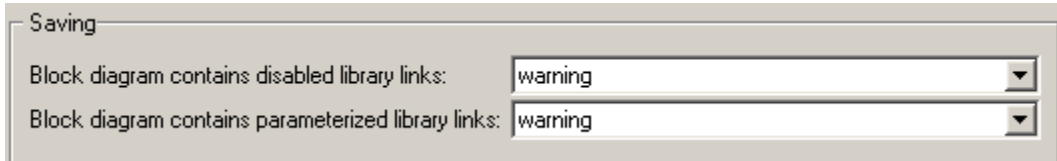
Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	error

See Also

- Referencing Models
- Diagnosing Simulation Errors
- Logging Referenced Model Signals
- To Workspace block
- Scope block
- Configuration Parameters Dialog Box
- Diagnostics Pane: Model Referencing

Diagnostics Pane: Saving



In this section...

“Saving Tab Overview” on page 1-353

“Block diagram contains disabled library links” on page 1-354

“Block diagram contains parameterized library links” on page 1-356

Saving Tab Overview

Specify the diagnostic actions that Simulink software takes when saving a block diagram containing disabled library links or parameterized library links.

Configuration

Set the parameters displayed.

Tips

- The options are typically to do nothing or to display a warning or an error message.
- A warning does not terminate a simulation, but an error does.

See Also

- Saving a Model
- Model Parameters
- Diagnosing Simulation Errors
- Solver Diagnostics
- Sample Time Diagnostics
- Data Validity Diagnostics
- Type Conversion Diagnostics
- Connectivity Diagnostics
- Compatibility Diagnostics
- Model Referencing Diagnostics
- Configuration Parameters Dialog Box
- Diagnostics Pane: Saving

Block diagram contains disabled library links

Select the diagnostic action to take when saving a model containing disabled library links.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning and saves the block diagram. The diagram may not contain the information you had intended.

error

Simulink software displays an error message. The model is not saved.

Tip

Use the Model Advisor Identify disabled library links check to find disabled library links.

Command-Line Information

Parameter: SaveWithDisabledLinksMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- [Disabling Library Links](#)
- [Identify disabled library links](#)
- [Saving a Model](#)
- [Model Parameters](#)
- [Configuration Parameters Dialog Box](#)
- [Diagnostics Pane: Saving](#)

Block diagram contains parameterized library links

Select the diagnostic action to take when saving a model containing parameterized library links.

Settings

Default: warning

none

Simulink software takes no action.

warning

Simulink software displays a warning and saves the block diagram. The diagram may not contain the information you had intended.

error

Simulink software displays an error message. The model is not saved.

Tips

- Use the Model Advisor Identify parameterized library links check to find parameterized library links.

Command-Line Information

Parameter: SaveWithParameterizedLinksMsg

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'none'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Identify parameterized library links
- Configuration Parameters Dialog Box
- Diagnostics Pane: Saving

Diagnostics Pane: Stateflow

Stateflow	
Unused data and events:	warning
Unexpected backtracking:	warning
Invalid input data access in chart initialization:	warning
No unconditional default transitions:	warning
Transition outside natural parent:	warning

In this section...

“Stateflow Diagnostics Overview” on page 1-359

“Unused data and events” on page 1-359

“Unexpected backtracking” on page 1-362

“Invalid input data access in chart initialization” on page 1-363

“No unconditional default transitions” on page 1-365

“Transition outside natural parent” on page 1-366

Stateflow Diagnostics Overview

Specify the diagnostic actions to take for detection of undesirable chart designs.

Configuration

Set the parameters displayed.

Tips

- The options are typically to do nothing or to display a warning or an error message.
- A warning does not terminate a simulation, but an error does.

See Also

- Saving a Model
- Model Parameters
- Diagnosing Simulation Errors
- Solver Diagnostics
- Sample Time Diagnostics
- Data Validity Diagnostics
- Type Conversion Diagnostics
- Connectivity Diagnostics
- Compatibility Diagnostics
- Model Referencing Diagnostics
- Saving Diagnostics
- Configuration Parameters Dialog Box

Unused data and events

Select the diagnostic action to take for detection of unused data and events in a chart. By removing unused data and events in a chart, you can minimize the size of your model.

Settings

Default: warning

none

No warning or error appears.

warning

A warning appears, with a link to delete the unused data or event in your chart.

error

An error appears and stops the simulation.

Tip

This diagnostic does not detect the following types of data and events:

- Machine-parented data
- Inputs and outputs of Embedded MATLAB functions
- Input events

Command-Line Information

Parameter: SFUnusedDataAndEventsDiag

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	warning
Traceability	No impact
Efficiency	No impact (for simulation) none (for production code generation)
Safety precaution	warning

See Also

- Configuration Parameters Dialog Box
- Stateflow Diagnostics
- “Diagnostic for Detecting Unused Data”
- “Diagnostic for Detecting Unused Events”

Unexpected backtracking

Select the diagnostic action to take when a chart junction:

- Does not have an unconditional transition path to a state or a terminal junction
- Has multiple transition paths leading to it

This chart configuration can lead to undesired backtracking behavior during simulation.

Settings

Default: warning

none

No warning or error appears.

warning

A warning appears, with a link to examples of undesired backtracking behavior.

error

An error appears and stops the simulation.

Tip

To avoid this behavior, consider adding an unconditional transition from that chart junction to a terminal junction.

Command-Line Information

Parameter: SFUnexpectedBacktrackingDiag

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	warning
Traceability	No impact
Efficiency	No impact (for simulation) No impact (for production code generation)
Safety precaution	error

See Also

- Configuration Parameters Dialog Box
- Stateflow Diagnostics
- “Best Practices for Creating Flow Graphs”
- “Backtracking Behavior in Flow Graphs Example”

Invalid input data access in chart initialization

Select the diagnostic action to take when a chart:

- Has the `ExecuteAtInitialization` property set to `true`
- Accesses input data on a default transition or associated state entry actions, which execute at chart initialization

This chart configuration is not recommended because blocks connected to chart input ports might not initialize their outputs during the initialization phase.

Settings

Default: warning

none

No warning or error appears.

warning

A warning appears.

error

An error appears and stops the simulation.

Tip

In charts that do not contain states, the `ExecuteAtInitialization` property has no effect.

Command-Line Information

Parameter: `SFInvalidInputDataAccessInChartInitDiag`

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	warning
Traceability	No impact
Efficiency	No impact (for simulation) No impact (for production code generation)
Safety precaution	error

See Also

- Configuration Parameters Dialog Box
- Stateflow Diagnostics
- “Execution of a Chart at Initialization”

No unconditional default transitions

Select the diagnostic action to take when a chart does not have an unconditional default transition to a state or a junction.

This chart configuration is not recommended because an inconsistency error can occur.

Settings

Default: warning

none

No warning or error appears.

warning

A warning appears.

error

An error appears and stops the simulation.

Command-Line Information

Parameter: SFNoUnconditionalDefaultTransitionDiag

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	warning
Traceability	No impact
Efficiency	No impact (for simulation) none (for production code generation)
Safety precaution	error

See Also

- Configuration Parameters Dialog Box
- Stateflow Diagnostics
- “Debugging State Inconsistencies in a Chart”

Transition outside natural parent

Select the diagnostic action to take when a chart contains a transition that loops outside the parent state or junction.

Settings

Default: warning

none

No warning or error appears.

warning

A warning appears.

error

An error appears and stops the simulation.

Command-Line Information

Parameter: SFTransitionOutsideNaturalParentDiag

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'warning'

Recommended Settings

Application	Setting
Debugging	warning
Traceability	No impact

Application	Setting
Efficiency	No impact (for simulation) none (for production code generation)
Safety precaution	error

See Also

- Configuration Parameters Dialog Box
- Stateflow Diagnostics

Hardware Implementation Pane

The screenshot shows a configuration dialog box titled "Embedded hardware (simulation and code generation)". It contains several sections for setting hardware parameters:

- Device vendor:** A dropdown menu set to "Generic".
- Device type:** A dropdown menu set to "Unspecified (assume 32-bit Generic)".
- Number of bits:** A group box containing six input fields: char (8), short (16), int (32), long (32), float (32), double (64), native (32), and pointer (32).
- Largest atomic size:** A group box containing two dropdown menus: integer (set to "Char") and floating-point (set to "None").
- Byte ordering:** A dropdown menu set to "Unspecified".
- Signed integer division rounds to:** A dropdown menu set to "Undefined".
- Shift right on a signed integer as arithmetic shift.
- Emulation hardware (code generation only):** A group box containing None.

In this section...

- “Hardware Implementation Overview” on page 1-370
- “Device vendor” on page 1-371
- “Device type” on page 1-374
- “Number of bits: char” on page 1-386
- “Number of bits: short” on page 1-388
- “Number of bits: int” on page 1-390
- “Number of bits: long” on page 1-392
- “Number of bits: float” on page 1-394
- “Number of bits: double” on page 1-395
- “Number of bits: native” on page 1-396
- “Number of bits: pointer” on page 1-398
- “Largest atomic size: integer” on page 1-399
- “Largest atomic size: floating-point” on page 1-401

In this section...

“Byte ordering” on page 1-403

“Signed integer division rounds to” on page 1-405

“Shift right on a signed integer as arithmetic shift” on page 1-408

“None” on page 1-410

“Device vendor” on page 1-412

“Device type” on page 1-415

“Number of bits: char” on page 1-427

“Number of bits: short” on page 1-429

“Number of bits: int” on page 1-431

“Number of bits: long” on page 1-433

“Number of bits: float” on page 1-435

“Number of bits: double” on page 1-436

“Number of bits: native” on page 1-437

“Number of bits: pointer” on page 1-439

“Largest atomic size: integer” on page 1-440

“Largest atomic size: floating-point” on page 1-442

“Byte ordering” on page 1-444

“Signed integer division rounds to” on page 1-446

“Shift right on a signed integer as arithmetic shift” on page 1-449

Hardware Implementation Overview

Describe the hardware characteristics for the modelled system, including how to set up embedded and emulation hardware settings for both simulation and code generation.

Note Hardware Implementation pane options do not control hardware or compiler behavior: their purpose is to solely describe hardware and compiler properties to MATLAB software, which uses the information to generate code that is correct for the platform, runs as efficiently as possible, and gives bit-true agreement for the results of integer and fixed-point operations in simulation, production code, and test code.

Configuration

- 1 Choose the **Device type** in the Embedded hardware subpane.
- 2 Set the parameters displayed for the selected device type.
- 3 Apply the changes.
- 4 Repeat as needed for Emulation hardware.

Tips

- This pane applies to models of computer-based systems, such as embedded controllers.
- Specifying hardware characteristics enables simulation of the model to detect error conditions that could arise when executing code, such as hardware overflow.

See Also

- [Configuring Hardware Properties](#)
- [Configuration Parameters Dialog Box](#)
- [Hardware Implementation Pane](#)

Device vendor

Select the manufacturer of the hardware you will use to implement the production version of the system represented by this model.

Settings

Default: Generic

- AMD
- ARM Compatible
- ASIC/FPGA (**Embedded hardware** subpane only)
- Analog Devices
- Atmel
- Freescale
- Infineon
- Intel
- Microchip
- NEC
- Renesas
- SGI
- STMicroelectronics
- Texas Instruments
- Generic

Tips

- Select the device vendor before you specify the hardware device used to define your system's constraints.
- If your production hardware does not match any of the listed vendors, select **Generic**.

- The **Device vendor** and **Device type** fields both share the same command line parameter: `ProdHWDeviceType`. When specifying this parameter from the command line, separate the device vendor and device type values using the characters `>`. For example: `'Intel->8051 Compatible'`.
- To add **Device vendor** and **Device type** values to the default set that is displayed on the **Hardware Implementation** pane, see “Registering Additional Device Vendor and Device Type Values” in the Real-Time Workshop documentation.

Dependencies

Selecting a value for this parameter allows you to view a list of supported devices from the selected vendor in the **Device type** drop-down menu.

Command-Line Information

Parameter: `ProdHWDeviceType`

Type: string

Value: any valid value (see tips)

Default: `'Generic->Unspecified (assume 32-bit Generic)'`

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Device type (production hardware)
- Device vendor (test hardware)
- Hardware Implementation Options

- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Device type

Select the type of hardware you will use to implement the production version of the system represented by this model.

Settings

Default: Unspecified (assume 32 bit Generic)

Generic options:

- 16-bit Embedded Processor
- 32-bit Embedded Processor
- 32-bit Real-Time Simulator
- 32-bit x86 compatible
- 8-bit Embedded Processor
- Custom
- MATLAB Host Computer (available as an **Emulation hardware** device and, for Embedded MATLAB Coder configuration, as an **Embedded hardware** device)
- Unspecified (assume 32-bit Generic)

AMD® options:

- K5/K6/Athlon

ARM® Compatible options:

- ARM 10
- ARM 11
- ARM 7
- ARM 8
- ARM 9
- ARM Cortex

ASIC/FPGA options: (**Embedded hardware** subpane only)

- ASIC/FPGA

Analog Devices™ options:

- Blackfin
- SHARC
- TigerSHARC

Atmel® options:

- AVR

Freescale™ options:

- 32-bit PowerPC
- 68332
- 68HC08
- 68HC11
- ColdFire
- DSP563xx (16-bit mode)
- HC(S)12
- MPC52xx
- MPC5500
- MPC55xx
- MPC5xx
- MPC7400
- MPC7xxx
- MPC82xx
- MPC83xx

- MPC85xx
- MPC86xx
- MPC8xx
- S12x

Infineon® options:

- C16x, XC16x
- TriCore

Intel® options:

- 8051 Compatible
- x86/Pentium

Microchip:

- PIC18
- dsPIC

NEC® options:

- V850

Renesas® options:

- M16C
- M32C
- R8C/Tiny
- SH-2/3/4

SGI:

- UltraSPARC IIi

STMicroelectronics:

- ST10/Super10

Texas Instruments™ options:

- C2000
- C5000
- C6000
- MSP430
- TMS470

Tips

- Select the device vendor before you specify the hardware device type.
- Selecting a device type specifies the hardware device to define your system's constraints:
 - Default hardware properties appear as the initial values.
 - Parameters with only one possible value cannot be changed.
 - Parameters with more than one possible value provide a list of legal values.
 - Static values for each device type are displayed in the following table.
 - Parameters that you can modify are identified with an **x**.

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
Generic											
Unspecified (assume 32-bit Generic) (default)	8	16	32	32	32	x	x	Unspecified	x	Set	
Custom	x	x	x	x	x	x	x	x	x	x	
16-bit Embedded Processor	8	16	16	32	16	x	x	x	x	Set	
32-bit Embedded Processor	8	16	32	32	32	x	x	x	x	Set	
32-bit Real Time Simulator	8	16	32	32	32	x	x	x	x	Set	
32-bit x86 compatible	8	16	32	32	32	x	x	Little Endian	Zero	Set	
8-bit Embedded Processor	8	16	16	32	8	x	x	x	x	Set	

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
MATLAB Host Computer	8	16	32	Host specific value (32 or 64)	Host specific value (32 or 64)	x	x	Little Endian	x	Set	
AMD											
K5/K6/Athlon	8	16	32	32	32	x	x	Little Endian	x	Set	
ARM Compatible											
ARM 7/8/9/10	8	16	32	32	32	Long	Float	x	x	x	
ARM 11	8	16	32	32	32	Long	Double	x	x	x	
ARM Cortex	8	16	32	32	32	Char	None	x	x	x	
ASIC/FPGA											
ASIC/FPGA	NA	NA	NA	NA	NA	NA	NA	NA	NA	NA	
Analog Devices											
Blackfin	8	16	32	32	32	Long	Double	Little Endian	Zero	Set	
SHARC	32	32	32	32	32	Long	Double	Big Endian	Zero	Set	
TigerSHARC	32	32	32	32	32	Long	Double	Little Endian	Zero	Set	
Atmel											

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
AVR	8	16	16	32	8	x	x	Little Endian	Zero	Set	
Freescale											
32-bit PowerPC	8	16	32	32	32	Long	Double	Big Endian	Zero	Set	
68332	8	16	32	32	32	x	x	Big Endian	x	Set	
68HC08	8	16	16	32	8	x	x	Big Endian	x	Set	
68HC11	8	16	16	32	8	x	x	Big Endian	x	Set	
ColdFire	8	16	32	32	32	x	x	Big Endian	Zero	Set	
DSP563xx (16-bit mode)	8	16	16	32	16	x	x	x	x	Set	
HC(S)12	8	16	16	32	16	x	x	Big Endian	x	Set	

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
MPC52xx, MPC5500, MPC55xx, MPC5xx, MPC7400, MPC7xxx, MPC82xx, MPC83xx, MPC86xx, MPC8xx	8	16	32	32	32	Long	Double	x	Zero	Set	
MPC85xx	8	16	32	32	32	Long	Float	x	Zero	Set	
S12x	8	16	16	32	16	x	x	Big Endian	x	Set	
Infineon											
C16x, XC16x	8	16	16	32	16	x	x	Little Endian	Zero	Set	
TriCore	8	16	32	32	32	x	x	Little Endian	x	Set	
Intel											
8051 Compatible	8	16	16	32	8	x	x	x	x	Clear	
x86/Pentium	8	16	32	32	32	x	x	Little Endian	x	Set	
Microchip											

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
PIC18	8	16	16	32	8	x	x	Little Endian	Zero	Set	
dsPIC	8	16	16	32	16	x	x	Little Endian	Zero	Set	
NEC											
V850	8	16	32	32	32	x	x	x	x	x	
Renesas											
M16C	8	16	16	32	16	x	x	Little Endian	x	x	
M32C	8	16	x	32	x	x	x	Little Endian	x	x	
R8C/Tiny	8	16	16	32	16	x	x	Little Endian	x	x	
SH-2/3/4	8	16	32	32	32	x	x	x	x	x	
SGI											
UltraSPARC Iii	8	16	32	32	32	x	x	Big Endian	x	Set	
STMicroelectronics											
ST10/Super10	8	16	16	32	16	x	x	Little Endian	Zero	Set	
Texas Instruments											
C2000	16	16	16	32	16	Int	None	x	Zero	Set	

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
C5000	16	16	16	32	16	Int	None	Big Endian	Zero	Set	
C6000	8	16	32	40	32	Int	None	x	Zero	Set	
MSP430	8	16	16	32	16	x	x	Little Endian	Zero	Set	
TMS470	8	16	32	32	x	x	x	x	x	x	

- If your production hardware does not match any of the listed types, select **Unspecified** (assume 32-bit Generic) if it has the characteristics of a generic 32-bit microprocessor; otherwise select **Custom**.
- The **Device vendor** and **Device type** fields both share the same command line parameter: `ProdHWDeviceType`. When specifying this parameter from the command line, separate the device vendor and device type values using the characters `>`. For example: `'Intel->8051 Compatible'`.
- To add **Device vendor** and **Device type** values to the default set that is displayed on the **Hardware Implementation** pane, see “Registering Additional Device Vendor and Device Type Values” in the Real-Time Workshop documentation.

Dependencies

The options available in the drop-down menu are determined by the **Device vendor** parameter.

Selecting **ASIC/FPGA** enables the **Emulation hardware (code generation only)** subpane.

Selecting any other device type sets the following device-specific parameters:

- **Number of bits: char**
- **Number of bits: short**
- **Number of bits: int**
- **Number of bits: long**
- **Number of bits: float**
- **Number of bits: double**
- **Number of bits: native**
- **Number of bits: pointer**
- **Largest atomic size: integer**
- **Largest atomic size: floating-point**
- **Byte ordering**
- **Signed integer division rounds to**
- **Shift right on a signed integer as arithmetic shift**

Whether you can modify a device-specific parameter varies according to device type. Parameters that cannot be modified for a device are greyed out in the GUI display.

Command-Line Information

Parameter: ProdHWDeviceType

Type: string

Value: any valid value (see tips)

Default: 'Generic->Unspecified (assume 32-bit Generic)'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact

See Also

- Device vendor (production hardware)
- Device type (test hardware)
- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: char

Describe the character bit length for the production hardware.

Settings

Default: 8

Minimum: 8

Maximum: 32

Enter a value between 8 and 32.

Tip

All values must be a multiple of 8.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: ProdBitPerChar

Type: integer

Value: any valid value

Default: 8

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: short

Describe the data bit length for the production hardware.

Settings

Default: 16

Minimum: 8

Maximum: 32

Enter a value between 8 and 32.

Tip

All values must be a multiple of 8.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: ProdBitPerShort

Type: integer

Value: any valid value

Default: 16

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: int

Describe the data integer bit length for the production hardware.

Settings

Default: 32

Minimum: 8

Maximum: 32

Enter a number between 8 and 32.

Tip

All values must be a multiple of 8.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: ProdBitPerInt

Type: integer

Value: any valid value

Default: 32

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: long

Describe the data bit lengths for the production hardware.

Settings

Default: 32

Minimum: 32

Maximum: 64

Enter a value between 32 and 64.

Tip

All values must be a multiple of 8 and between 32 and 64.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: ProdBitPerLong

Type: integer

Value: any valid value

Default: 32

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: float

Describe the bit length of floating-point data for the production hardware (read-only)

Settings

Default: 32

Always equals 32.

Command-Line Information

Parameter: ProdBitPerFloat

Type: integer

Value: 32 (read-only)

Default: 32

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: double

Describe the bit-length of double data for the production hardware (read-only).

Settings

Default: 64

Always equals 64.

Command-Line Information

Parameter: ProdBitPerDouble

Type: integer

Value: 64 (read only)

Default: 64

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: native

Describe the microprocessor native word size for the production hardware.

Settings

Default: 32

Minimum: 8

Maximum: 64

Enter a value between 8 and 64.

Tip

All values must be a multiple of 8.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: ProdWordSize

Type: integer

Value: any valid value

Default: 32

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: pointer

Describe the bit-length of pointer data for the production hardware (read-only).

Settings

Default: matches the setting for “Number of bits: int” on page 1-390

Minimum: 8

Maximum: 32

Command-Line Information

Parameter: ProdBitPerPointer

Type: integer

Value: any valid value (read-only)

Default: matches the setting for ProdBitPerInt

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Largest atomic size: integer

Specify the largest integer data type that can be atomically loaded and stored on the production hardware.

Settings

Default: Char

Char

Specifies that `char` is the largest integer data type that can be atomically loaded and stored on the production hardware.

Short

Specifies that `short` is the largest integer data type that can be atomically loaded and stored on the production hardware.

Int

Specifies that `int` is the largest integer data type that can be atomically loaded and stored on the production hardware.

Long

Specifies that `long` is the largest integer data type that can be atomically loaded and stored on the production hardware.

Tip

This parameter is used, where possible, to optimize away unnecessary double-buffering or unnecessary semaphore protection, based on data size, in generated multirate code.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: ProdLargestAtomicInteger

Type: string

Value: 'Char' | 'Short' | 'Int' | 'Long'

Default: 'Char'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Largest atomic size: floating-point

Specify the largest floating-point data type that can be atomically loaded and stored on the production hardware.

Settings

Default: None

Float

Specifies that `float` is the largest floating-point data type that can be atomically loaded and stored on the production hardware.

Double

Specifies that `double` is the largest floating-point data type that can be atomically loaded and stored on the production hardware.

None

Specifies that there is no applicable setting or not to use this parameter in generating multirate code.

Tip

This parameter is used, where possible, to optimize away unnecessary double-buffering or unnecessary semaphore protection, based on data size, in generated multirate code.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: `ProdLargestAtomicFloat`

Type: `string`

Value: `'Float' | 'Double' | 'None'`

Default: `'None'`

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Byte ordering

Describe the byte ordering for the production hardware.

Settings

Default: Unspecified

Unspecified

Specifies that the code determines the endianness of the hardware.
This is the least efficient choice.

Big Endian

The most significant byte appears first.

Little Endian

The least significant byte appears first.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: ProdEndianness

Type: string

Value: 'Unspecified' | 'LittleEndian' | 'BigEndian'

Default: 'Unspecified'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Signed integer division rounds to

Describe how to produce a signed integer quotient for the production hardware.

Settings

Default: Undefined

Undefined

Choose this option if neither Zero nor Floor describes the compiler's behavior, or if that behavior is unknown.

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.

Tips

- Use the **Integer rounding mode** parameter on your model's blocks to simulate the rounding behavior of the C compiler that you 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 block.
- 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" in the *Simulink Fixed Point User's Guide*.
- See Hardware Implementation Options in the Real-Time Workshop documentation for information on how this option affects code generation.
- The following table illustrates the compiler behavior described by the options for this parameter.

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

Dependency

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: ProdIntDivRoundTo

Type: string

Value: 'Floor' | 'Zero' | 'Undefined'

Default: 'Undefined'

Recommended settings

Application	Setting
Debugging	No impact for simulation and during development. Undefined for production code generation.
Traceability	No impact for simulation and during development. Zero or Floor for production code generation.

Application	Setting
Efficiency	No impact for simulation and during development. Zero for production code generation.
Safety precaution	No impact for simulation and during development. Floor for production code generation.

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Shift right on a signed integer as arithmetic shift

Describe how your compiler rounds the result of two signed integers for the production hardware.

Settings

Default: On



On

Generates simple efficient code whenever the Simulink model performs arithmetic shifts on signed integers.



Off

Generates fully portable but less efficient code to implement right arithmetic shifts.

Tips

- Select this parameter if the C compiler implements a signed integer right shift as an arithmetic right shift.
- 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 twos complement notation.

Dependency

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: ProdShiftRightIntArith

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	On
Safety precaution	No impact

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

None

Specify whether the test hardware differs from the deployment hardware.

Settings

Default: On



Specifies that the hardware used to test the code generated from the model is the same as the production hardware, or has the same characteristics.



Specifies that the hardware used to test the code generated from the model has different characteristics than the production hardware.

Tips

- You can generate code that runs on the test hardware but behaves as if it had been generated for and executed on the deployment hardware.
- The **Embedded hardware (simulation and code generation)** subpane specifies the deployment hardware properties. The **Emulation hardware (code generation only)** subpane is used to specify the test hardware properties.

Dependency

Enables the Emulation hardware subpane.

Command-Line Information

Parameter: ProdEqTarget

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

More information

- [Specifying Emulation Hardware Characteristics](#)
- [Hardware Implementation Options](#)
- [Configuration Parameters Dialog Box](#)
- [Hardware Implementation Pane](#)

Device vendor

Select the manufacturer of the hardware that will be used to test the code generated from the model.

Settings

Default: Generic

- AMD
- ARM Compatible
- ASIC/FPGA (**Embedded hardware** subpane only)
- Analog Devices
- Atmel
- Freescale
- Infineon
- Intel
- Microchip
- NEC
- Renesas
- SGI
- STMicroelectronics
- Texas Instruments
- Generic

Tips

- Select the device vendor before you specify the hardware device used to define your system's constraints.
- If your test hardware does not match any of the listed vendors, select Generic.

- The **Device vendor** and **Device type** fields both share the same command line parameter: `TargetHWDeviceType`. When specifying this parameter from the command line, separate the device vendor and device type values using the characters `>`. For example: `'Intel->8051 Compatible'`.
- To add **Device vendor** and **Device type** values to the default set that is displayed on the **Hardware Implementation** pane, see “Registering Additional Device Vendor and Device Type Values” in the Real-Time Workshop documentation.

Dependencies

Selecting a value for this parameter allows you to view a list of supported devices from the selected vendor in the **Device type** drop-down menu.

Command-Line Information

Parameter: `TargetHWDeviceType`

Type: string

Value: any valid value (see tips)

Default: `'Generic->Unspecified (assume 32-bit Generic)'`

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Device type (test hardware)
- Specifying Emulation Hardware Characteristics
- Hardware Implementation Options

- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Device type

Select the type of hardware that will be used to test the code generated from the model.

Settings

Default: Unspecified (assume 32 bit Generic)

Generic options:

- 16-bit Embedded Processor
- 32-bit Embedded Processor
- 32-bit Real-Time Simulator
- 32-bit x86 compatible
- 8-bit Embedded Processor
- Custom
- MATLAB Host Computer (available as an **Emulation hardware** device and, for Embedded MATLAB Coder configuration, as an **Embedded hardware** device)
- Unspecified (assume 32-bit Generic)

AMD options:

- K5/K6/Athlon

ARM Compatible options:

- ARM 10
- ARM 11
- ARM 7
- ARM 8
- ARM 9
- ARM Cortex

ASIC/FPGA options: (**Embedded hardware** subpane only)

- ASIC/FPGA

Analog Devices options:

- Blackfin
- SHARC
- TigerSHARC

Atmel options:

- AVR

Freescale options:

- 32-bit PowerPC
- 68332
- 68HC08
- 68HC11
- ColdFire
- DSP563xx (16-bit mode)
- HC(S)12
- MPC52xx
- MPC5500
- MPC55xx
- MPC5xx
- MPC7400
- MPC7xxx
- MPC82xx
- MPC83xx

- MPC85xx
- MPC86xx
- MPC8xx
- S12x

Infineon options:

- C16x, XC16x
- TriCore

Intel options:

- 8051 Compatible
- x86/Pentium

Microchip:

- PIC18
- dsPIC

NEC options:

- V850

Renesas options:

- M16C
- M32C
- R8C/Tiny
- SH-2/3/4

SGI:

- UltraSPARC IIi

STMicroelectronics:

- ST10/Super10

Texas Instruments options:

- C2000
- C5000
- C6000
- MSP430
- TMS470

Tips

- Select the device vendor before you specify the hardware device type.
- Selecting a device type specifies the hardware device to define your system's constraints:
 - Default hardware properties appear as the initial values.
 - Parameters with only one possible value cannot be changed.
 - Parameters with more than one possible value provide a list of legal values.
 - Static values for each device type are displayed in the following table.
 - Parameters that you can modify are identified with an x.

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
Generic											
Unspecified (assume 32-bit Generic) (default)	8	16	32	32	32	x	x	Unspecified	x	Set	
Custom	x	x	x	x	x	x	x	x	x	x	
16-bit Embedded Processor	8	16	16	32	16	x	x	x	x	Set	
32-bit Embedded Processor	8	16	32	32	32	x	x	x	x	Set	
32-bit Real Time Simulator	8	16	32	32	32	x	x	x	x	Set	
32-bit x86 compatible	8	16	32	32	32	x	x	Little Endian	Zero	Set	
8-bit Embedded Processor	8	16	16	32	8	x	x	x	x	Set	

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
MATLAB Host Computer	8	16	32	Host specific value (32 or 64)	Host specific value (32 or 64)	x	x	Little Endian	x	Set	
AMD											
K5/K6/Athlon	8	16	32	32	32	x	x	Little Endian	x	Set	
ARM Compatible											
ARM 7/8/9/10	8	16	32	32	32	Long	Float	x	x	x	
ARM 11	8	16	32	32	32	Long	Double	x	x	x	
ARM Cortex	8	16	32	32	32	Char	None	x	x	x	
ASIC/FPGA											
ASIC/FPGA	NA	NA	NA	NA	NA	NA	NA	NA	NA	NA	
Analog Devices											
Blackfin	8	16	32	32	32	Long	Double	Little Endian	Zero	Set	
SHARC	32	32	32	32	32	Long	Double	Big Endian	Zero	Set	
TigerSHARC	32	32	32	32	32	Long	Double	Little Endian	Zero	Set	
Atmel											

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
AVR	8	16	16	32	8	x	x	Little Endian	Zero	Set	
Freescale											
32-bit PowerPC	8	16	32	32	32	Long	Double	Big Endian	Zero	Set	
68332	8	16	32	32	32	x	x	Big Endian	x	Set	
68HC08	8	16	16	32	8	x	x	Big Endian	x	Set	
68HC11	8	16	16	32	8	x	x	Big Endian	x	Set	
ColdFire	8	16	32	32	32	x	x	Big Endian	Zero	Set	
DSP563xx (16-bit mode)	8	16	16	32	16	x	x	x	x	Set	
HC(S)12	8	16	16	32	16	x	x	Big Endian	x	Set	

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
MPC52xx, MPC5500, MPC55xx, MPC5xx, MPC7400, MPC7xxx, MPC82xx, MPC83xx, MPC86xx, MPC8xx	8	16	32	32	32	Long	Double	x	Zero	Set	
MPC85xx	8	16	32	32	32	Long	Float	x	Zero	Set	
S12x	8	16	16	32	16	x	x	Big Endian	x	Set	
Infineon											
C16x, XC16x	8	16	16	32	16	x	x	Little Endian	Zero	Set	
TriCore	8	16	32	32	32	x	x	Little Endian	x	Set	
Intel											
8051 Compatible	8	16	16	32	8	x	x	x	x	Clear	
x86/Pentium	8	16	32	32	32	x	x	Little Endian	x	Set	
Microchip											

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
PIC18	8	16	16	32	8	x	x	Little Endian	Zero	Set	
dsPIC	8	16	16	32	16	x	x	Little Endian	Zero	Set	
NEC											
V850	8	16	32	32	32	x	x	x	x	x	
Renesas											
M16C	8	16	16	32	16	x	x	Little Endian	x	x	
M32C	8	16	x	32	x	x	x	Little Endian	x	x	
R8C/Tiny	8	16	16	32	16	x	x	Little Endian	x	x	
SH-2/3/4	8	16	32	32	32	x	x	x	x	x	
SGI											
UltraSPARC IIIi	8	16	32	32	32	x	x	Big Endian	x	Set	
STMicroelectronics											
ST10/Super10	8	16	16	32	16	x	x	Little Endian	Zero	Set	
Texas Instruments											
C2000	16	16	16	32	16	Int	None	x	Zero	Set	

Key:	float and double (not listed) always equal 32 and 64, respectively										
	pointer (not listed) matches the value of int										
	Rounds to = Signed integer division rounds to										
	Shift right = Shift right on a signed integer as arithmetic shift										
Device vendor / Device type	Number of bits					Largest atomic size		Byte ordering	Rounds to	Shift right	
	char	short	int	long	native	int	float				
C5000	16	16	16	32	16	Int	None	Big Endian	Zero	Set	
C6000	8	16	32	40	32	Int	None	x	Zero	Set	
MSP430	8	16	16	32	16	x	x	Little Endian	Zero	Set	
TMS470	8	16	32	32	x	x	x	x	x	x	

- If your test hardware does not match any of the listed types, select **Unspecified** (assume 32-bit **Generic**) if it has the characteristics of a generic 32-bit microprocessor; otherwise select **Custom**.
- The **Device vendor** and **Device type** fields both share the same command line parameter: `TargetHWDeviceType`. When specifying this parameter from the command line, separate the device vendor and device type values using the characters `>`. For example: `'Intel->8051 Compatible'`.
- To add **Device vendor** and **Device type** values to the default set that is displayed on the **Hardware Implementation** pane, see “Registering Additional Device Vendor and Device Type Values” in the Real-Time Workshop documentation.

Dependencies

The options available in the drop-down menu are determined by the **Device vendor** parameter.

Selecting a device type sets the following device-specific parameters:

- **Number of bits: char**

- **Number of bits: short**
- **Number of bits: int**
- **Number of bits: long**
- **Number of bits: float**
- **Number of bits: double**
- **Number of bits: native**
- **Number of bits: pointer**
- **Largest atomic size: integer**
- **Largest atomic size: floating-point**
- **Byte ordering**
- **Signed integer division rounds to**
- **Shift right on a signed integer as arithmetic shift**

Whether you can modify a device-specific parameter varies according to device type. Parameters that cannot be modified for a device are greyed out in the GUI display.

Command-Line Information

Parameter: TargetHWDeviceType

Type: string

Value: any valid value (see tips)

Default: 'Generic->Unspecified (assume 32-bit Generic)'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Device vendor (test hardware)
- Specifying Emulation Hardware Characteristics
- Hardware Implementation Options
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: char

Describe the character bit length for the hardware used to test code.

Settings

Default: 8

Minimum: 8

Maximum: 32

Enter a value between 8 and 32.

Tip

All values must be a multiple of 8.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: TargetBitPerChar

Type: integer

Value: any valid value

Default: 8

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Specifying Emulation Hardware Characteristics
- Hardware Implementation Options
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: short

Describe the data bit length for the hardware used to test code.

Settings

Default: 16

Minimum: 8

Maximum: 32

Enter a value between 8 and 32.

Tip

All values must be a multiple of 8.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: TargetBitPerShort

Type: integer

Value: any valid value

Default: 16

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Specifying Emulation Hardware Characteristics
- Hardware Implementation Options
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: int

Describe the data integer bit length of the hardware used to test code.

Settings

Default: 32

Minimum: 8

Maximum: 32

Enter a number between 8 and 32.

Tip

All values must be a multiple of 8.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: TargetBitPerInt

Type: integer

Value: any valid value

Default: 32

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Specifying Emulation Hardware Characteristics
- Hardware Implementation Options
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: long

Describe the data bit lengths for the hardware used to test code.

Settings

Default: 32

Minimum: 32

Maximum: 64

Enter a value between 32 and 64. (The value 64 is selected by default if you run MATLAB software on a 64-bit host computer and select the MATLAB host as the test hardware — that is, `TargetHWDeviceType` equals `'Generic->MATLAB Host Computer'`.)

Tip

All values must be a multiple of 8 and between 32 and 64.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: `TargetBitPerLong`

Type: integer

Value: any valid value

Default: 32

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Specifying Emulation Hardware Characteristics
- Hardware Implementation Options
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: float

Describe the bit length of floating-point data for the hardware used to test code (read-only).

Settings

Default: 32

Always equals 32.

Command-Line Information

Parameter: TargetBitPerFloat

Type: integer

Value: 32 (read-only)

Default: 32

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: double

Describe the bit-length of double data for the hardware used to test code (read-only).

Settings

Default: 64

Always equals 64.

Command-Line Information

Parameter: TargetBitPerDouble

Type: integer

Value: 64 (read only)

Default: 64

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: native

Describe the microprocessor native word size for the hardware used to test code.

Settings

Default: 32

Minimum: 8

Maximum: 64

Enter a value between 8 and 64. (The value 64 is selected by default if you run MATLAB software on a 64-bit host computer and select the MATLAB host as the test hardware — that is, TargetHWDeviceType equals 'Generic->MATLAB Host Computer'.)

Tip

All values must be a multiple of 8.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: TargetWordSize

Type: integer

Value: any valid value

Default: 32

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Specifying Emulation Hardware Characteristics
- Hardware Implementation Options
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Number of bits: pointer

Describe the bit-length of pointer data for the hardware used to test code (read-only).

Settings

Default: matches the setting for “Number of bits: int” on page 1-431

Minimum: 8

Maximum: 32

Command-Line Information

Parameter: TargetBitPerPointer

Type: integer

Value: any valid value (read only)

Default: matches the setting for TargetBitPerInt

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Largest atomic size: integer

Specify the largest integer data type that can be atomically loaded and stored on the hardware used to test code.

Settings

Default: Char

Char

Specifies that `char` is the largest integer data type that can be atomically loaded and stored on the hardware used to test code.

Short

Specifies that `short` is the largest integer data type that can be atomically loaded and stored on the hardware used to test code.

Int

Specifies that `int` is the largest integer data type that can be atomically loaded and stored on the hardware used to test code.

Long

Specifies that `long` is the largest integer data type that can be atomically loaded and stored on the hardware used to test code.

Tip

This parameter is used, where possible, to optimize away unnecessary double-buffering or unnecessary semaphore protection, based on data size, in generated multirate code.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: TargetLargestAtomicInteger

Type: string

Value: 'Char' | 'Short' | 'Int' | 'Long'

Default: 'Char'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Largest atomic size: floating-point

Specify the largest floating-point data type that can be atomically loaded and stored on the hardware used to test code.

Settings

Default: None

Float

Specifies that `float` is the largest floating-point data type that can be atomically loaded and stored on the hardware used to test code.

Double

Specifies that `double` is the largest floating-point data type that can be atomically loaded and stored on the hardware used to test code.

None

Specifies that there is no applicable setting or not to use this parameter in generating multirate code.

Tip

This parameter is used, where possible, to optimize away unnecessary double-buffering or unnecessary semaphore protection, based on data size, in generated multirate code.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: `TargetLargestAtomicFloat`

Type: `string`

Value: `'Float' | 'Double' | 'None'`

Default: `'None'`

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	Target specific
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Hardware Implementation Options
- Specifying Embedded Hardware Characteristics
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Byte ordering

Describe the byte ordering for the hardware used to test code.

Settings

Default: Unspecified

Unspecified

Specifies that the code determines the endianness of the hardware.
This is the least efficient choice.

Big Endian

The most significant byte comes first.

Little Endian

The least significant byte comes first.

Note For guidelines to observe when configuring **Embedded hardware** controls for code generation, see Hardware Implementation Options in the Real-Time Workshop documentation.

Dependencies

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: TargetEndianness

Type: string

Value: 'Unspecified' | 'LittleEndian' | 'BigEndian'

Default: 'Unspecified'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact for simulation and during development. Match operation of compiler and hardware for code generation.

See Also

- Specifying Emulation Hardware Characteristics
- Hardware Implementation Options
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Signed integer division rounds to

Describe how to produce a signed integer quotient for the hardware used to test code.

Settings

Default: Undefined

Undefined

Choose this option if neither Zero nor Floor describes the compiler's behavior, or if that behavior is unknown.

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.

Tips

- Use the **Integer rounding mode** parameter on your model's blocks to simulate the rounding behavior of the C compiler that you 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 block.
- 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" in the *Simulink Fixed Point User's Guide*.
- See Hardware Implementation Options in the Real-Time Workshop documentation for information on how this option affects code generation.
- The following table illustrates the compiler behavior described by the options for this parameter.

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

Dependency

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: TargetIntDivRoundTo

Type: string

Value: 'Floor' | 'Zero' | 'Undefined'

Default: 'Undefined'

Recommended settings

Application	Setting
Debugging	No impact for simulation and during development. Undefined for production code generation.
Traceability	No impact for simulation and during development. Zero or Floor for production code generation.

Application	Setting
Efficiency	No impact for simulation and during development. Zero for production code generation.
Safety precaution	No impact for simulation and during development. Floor for production code generation.

See Also

- Specifying Emulation Hardware Characteristics
- Hardware Implementation Options
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Shift right on a signed integer as arithmetic shift

Describe how your compiler rounds the result of two signed integers for the hardware used to test code.

Settings

Default: On



On

Generates simple efficient code whenever the Simulink model performs arithmetic shifts on signed integers.



Off

Generates fully portable but less efficient code to implement right arithmetic shifts.

Tips

- Select this parameter if your C compiler implements a signed integer right shift as an arithmetic right shift.
- 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 twos complement notation. It is equivalent to dividing the number by 2.
- This setting affects only code generation

Dependency

- Selecting a device using the **Device vendor** and **Device type** parameters sets a device-specific value for this parameter.
- This parameter is enabled only if it can be modified for the currently selected device.

Command-Line Information

Parameter: TargetShiftRightIntArith

Type: string

Value: 'on' | 'off'

Default: 'on'

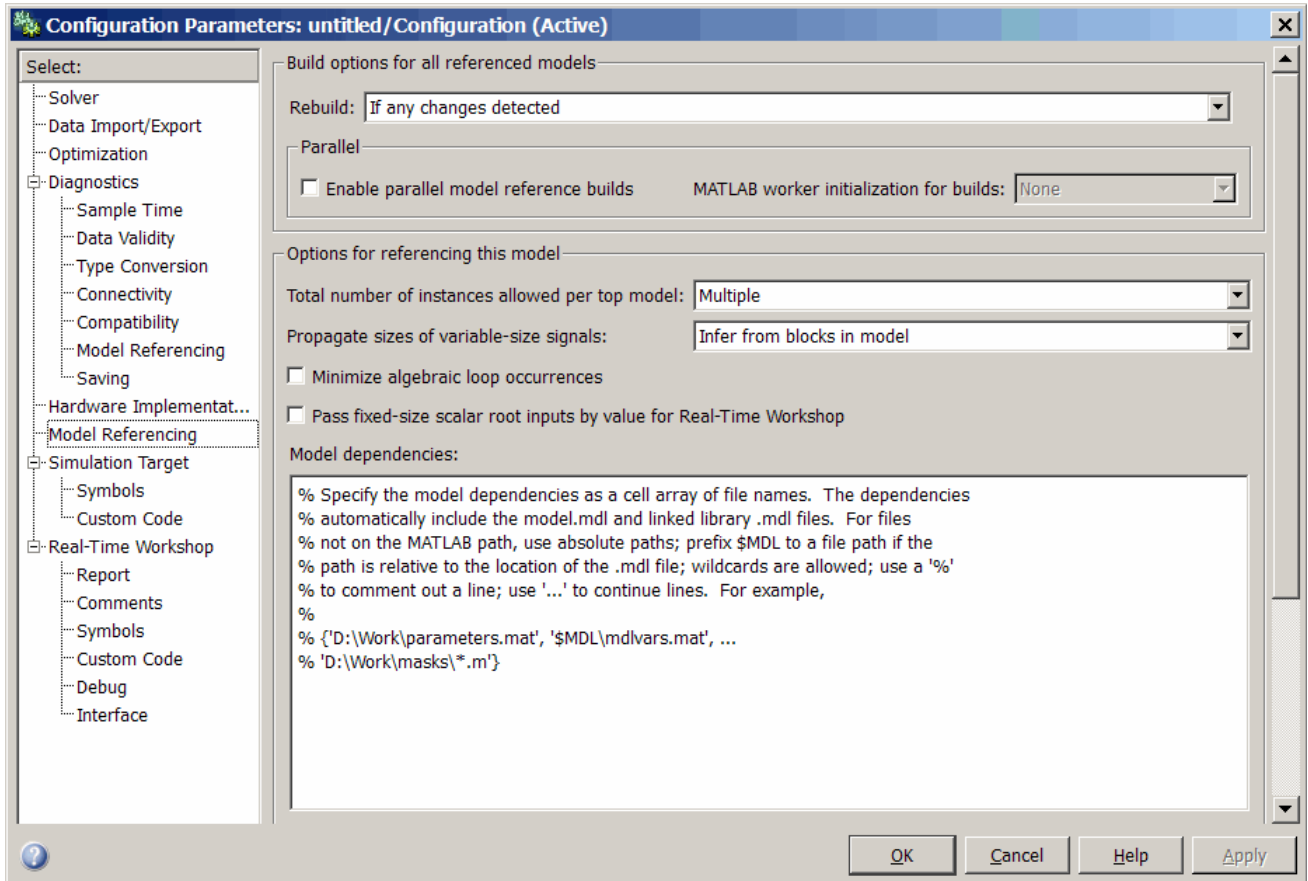
Recommended settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	On
Safety precaution	No impact

See Also

- Specifying Emulation Hardware Characteristics
- Hardware Implementation Options
- Configuration Parameters Dialog Box
- Hardware Implementation Pane

Model Referencing Pane



In this section...

“Model Referencing Pane Overview” on page 1-453

“Rebuild options” on page 1-454

“Never rebuild targets diagnostic” on page 1-457

“Enable parallel model reference builds” on page 1-459

“MATLAB worker initialization for builds” on page 1-461

In this section...

“Total number of instances allowed per top model” on page 1-463

“Pass fixed-size scalar root inputs by value for Real-Time Workshop” on page 1-465

“Minimize algebraic loop occurrences” on page 1-467

“Propagate sizes of variable-size signals” on page 1-470

“Model dependencies” on page 1-473

Model Referencing Pane Overview

Specify the options for including other models in this model, this model in other models, and for building simulation and code generation targets.

Configuration

Set the parameters displayed.

Tips

- The Model Referencing pane allows you to specify options for:
 - Including other models in this model.
 - Including the current model in other models.
- The option descriptions use the term *this model* to refer to the model that you are configuring and the term *referenced model* to designate models referenced by *this model*.

See Also

- Model Dependencies
- Configuration Parameters Dialog Box
- Model Referencing Pane

Rebuild options

Select whether to rebuild simulation and Real-Time Workshop targets for referenced models before updating, simulating, or generating code from this model.

Settings

Default: If any changes detected

Always

Rebuilds all targets referenced by this model before simulating, updating, or generating code from it.

If any changes detected

Rebuilds the target for a referenced model if Simulink software detects any changes of any kind in the target's dependencies. This option also checks for changes in the compiled form of the referenced model. Checking the compiled model can detect some changes that occur even in dependencies that you do not specify.

If any changes in known dependencies detected

Rebuilds a target if Simulink software detects any changes in known target dependencies (as listed under "Tips" on page 1-454) since the target was last built. This option ignores cosmetic changes, such as annotation changes, in the referenced model and in any block library dependencies, thus preventing unnecessary rebuilds.

Never

Never rebuild targets before simulating or generating code from this model. If you are certain that your targets are up-to-date, you can use this option to avoid time-consuming target dependency checking when simulating, updating, or generating code from a model.

Tips

- Simulink software detects target dependencies in:
 - The referenced model's model file
 - Block library files used by the referenced model
 - Targets of models referenced by the referenced model

- S-functions and associated TLC files used by the referenced model
- User-specified dependencies (see “Model dependencies” on page 1-473)
- Workspace variables used by the referenced model
- It is a good idea to use the **Always** option before deployment of a model to assure that all the model reference targets are up-to-date.
- Before selecting **If any changes in known dependencies detected**, you should be certain that you have specified every user-created dependency (for example, MATLAB code files or MAT-files) for this model to ensure that all targets that need to be rebuilt are rebuilt. Otherwise, invalid simulation results may occur.
- The **If any changes in known dependencies detected** option cannot detect changes in unspecified dependencies, such as MATLAB files used to initialize block masks. If you suspect that a model has such unknown dependencies, you can still guarantee valid simulation by selecting the **Always** or the **If any changes detected** option.
- Use the **Never** option with caution because it may lead to invalid results if referenced model targets are not in fact up-to-date.

Dependency

Selecting **Never** enables the **Never rebuild targets diagnostic** parameter.

Command-Line Information

Parameter: UpdateModelReferenceTargets

Type: string

Value: 'Force' | 'IfOutOfDateOrStructuralChange' | 'IfOutOfDate' | 'AssumeUpToDate'

Default: 'IfOutOfDateOrStructuralChange'

UpdateModelReferenceTargets Value	Equivalent Rebuild options Value (See “Settings” on page 1-454)
'Force'	Always
'IfOutOfDateOrStructuralChange'	If any changes detected

UpdateModelReferenceTarget Value	Equivalent Rebuild options Value (See "Settings" on page 1-454)
'IfOutOfDate'	If any changes in known dependencies detected
'AssumeUpToDate'	Never

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Never or If any changes detected

See Also

- Model Dependencies
- Configuration Parameters Dialog Box
- Model Referencing Pane

Never rebuild targets diagnostic

Select the diagnostic action that Simulink software should take if it detects a target that needs to be rebuilt.

Settings

Default: Error if targets require rebuild

none

Simulink software takes no action.

Warn if targets require rebuild

Simulink software displays a warning.

Error if targets require rebuild

Simulink software terminates the simulation and displays an error message.

Tip

Selecting None bypasses dependency checking, and thus enables faster updating, simulation, and code generation, but can cause models that are not up-to-date to malfunction or generate incorrect results.

Dependency

This parameter is enabled only if you select Never in the **Rebuild options** field.

Command-Line Information

Parameter: CheckModelReferenceTargetMessage

Type: string

Value: 'none' | 'warning' | 'error'

Default: 'error'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	error if targets require rebuild

See Also

- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Model Referencing Pane

Enable parallel model reference builds

Specify whether to use automatic parallel building of the model reference hierarchy whenever possible.

Settings

Default: Off



On

Simulink software builds the model reference hierarchy in parallel whenever possible (based on computing resources and the structure of the model reference hierarchy).



Off

Simulink never builds the model reference hierarchy in parallel.

Dependency

Selecting this option enables the **MATLAB worker initialization for builds** parameter.

Command-Line Information

Parameter: EnableParallelModelReferenceBuilds

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- “Parallel Building for Large Model Reference Hierarchies”
- “Reducing Build Time for Referenced Models” in the Real-Time Workshop documentation
- Configuration Parameters Dialog Box
- Model Referencing Pane

MATLAB worker initialization for builds

Specify how to initialize MATLAB workers for parallel builds.

Settings

Default: None

None

Simulink software takes no action. Specify this value if the child models in the model reference hierarchy do not rely on anything in the base workspace beyond what they explicitly set up (for example, with a model load function).

Copy base workspace

Simulink attempts to copy the base workspace to each MATLAB worker. Specify this value if you use a setup script to prepare the base workspace for all models to use.

Load top model

Simulink loads the top model on each MATLAB worker. Specify this value if the top model in the model reference hierarchy handles all of the base workspace setup (for example, with a model load function).

Limitation

For values other than None, limitations apply to global variables in the base workspace. Global variables are not propagated across parallel workers and do not reflect changes made by top and child model scripts.

Dependency

Selecting the option **Enable parallel model reference builds** enables this parameter.

Command-Line Information

Parameter: ParallelModelReferenceMATLABWorkerInit

Type: string

Value: 'None' | 'Copy Base Workspace' | 'Load Top Model'

Default: 'None'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- “Parallel Building for Large Model Reference Hierarchies”
- “Reducing Build Time for Referenced Models” in the Real-Time Workshop documentation
- Configuration Parameters Dialog Box
- Model Referencing Pane

Total number of instances allowed per top model

Specify how many references to this model can occur in another model.

Settings

Default: Multiple

Zero

The model cannot be referenced. An error occurs if a reference to the model occurs in another model.

One

The model can be referenced at most once in a model reference hierarchy. An error occurs if more than one reference exists.

Multiple

The model can be referenced more than once in a hierarchy, provided that it contains no constructs that preclude multiple reference. An error occurs if the model cannot be multiply referenced, even if only one reference exists.

To use multiple instances of a referenced model in Normal mode, use the Multiple setting. For details, see .

Command-Line Information

Parameter: ModelReferenceNumInstancesAllowed

Type: string

Value: 'Zero' | 'Single' | 'Multi'

Default: 'Multi'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- [Diagnosing Simulation Errors](#)
- [Configuration Parameters Dialog Box](#)
- [Model Referencing Pane](#)

Pass fixed-size scalar root inputs by value for Real-Time Workshop

Specify whether a model that calls (references) this model passes its scalar inputs to this model by value.

Settings

Default: Off (GUI), 'on' (command-line)



On

A model that calls (references) this model passes scalar inputs to this model by value.



Off

The calling model passes the inputs by reference (it passes the addresses of the inputs rather than the input values).

Tips

- This option is ignored in either of these two cases:
 - The C function prototype control is not the default.
 - The C++ encapsulation interface is not the default.
- Passing roots by value allows this model to read its scalar inputs from register or local memory which is faster than reading the inputs from their original locations.
- However, passing roots by value can lead to incorrect results if the model's root scalar inputs can change within a time step. This can happen, for instance, if this model's inputs and outputs share memory locations (as a result of a feedback loop) and the model is invoked multiple times in a time step; for example, by an Inport or Trigger block (see "Function-Call Subsystems"). In such cases, this model sees scalar input changes that occur in the same time step only if the inputs are passed by reference. That is why this option is off by default.
- If you are certain that this model is not referenced in contexts where its inputs can change within a time step, select this option to generate more efficient code for this model.

- Selecting this option can affect reuse of code generated for subsystems. See Reusable Code and Referenced Models for more information.
- For SIM targets, a model that references this model passes inputs by reference, regardless of how you set the **Pass fixed-size scalar root inputs by value for Real-Time Workshop** parameter.

Command-Line Information

Parameter:ModelReferencePassRootInputsByReference

Type: string

Value: 'on' | 'off'

Default: 'on'

Note The command-line values are reverse of the settings values. Therefore, 'on' in the command line corresponds to the description of “Off” in the settings section, and 'off' in the command line corresponds to the description of “On” in the settings section.

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Off

See Also

- “Function-Call Subsystems”
- Reusable Code and Referenced Models
- Configuration Parameters Dialog Box
- Model Referencing Pane

Minimize algebraic loop occurrences

Specify whether Simulink software tries to eliminate algebraic loops involving this model from models that reference it.

Settings

Default: Off



On

Simulink software tries to eliminate algebraic loops involving this model from models that reference it.



Off

Simulink software does not try to eliminate algebraic loops from the models that reference this model.

Tips

- Enabling this option disables:
 - Conditional input branch optimization for simulation
 - The Real-Time Workshop **Single output/update function** parameter (disabling this parameter generates separate *model_output* and *model_update* functions)
- See Algebraic Loops for more information.

Command-Line Information

Parameter: ModelReferenceMinAlgLoopOccurrences

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Off

See Also

- Model block
- Algebraic Loops
- Model Blocks and Direct Feedthrough
- Diagnosing Simulation Errors
- Configuration Parameters Dialog Box
- Model Referencing Pane

Propagate sizes of variable-size signals

Select how variable-size signals propagate through referenced models.

Settings

Default: Infer from blocks in model

Infer from blocks in model

Searches a referenced model and groups blocks into the following categories.

Category	Description	Example Blocks in This Category
1	Output signal size depends on input signal values.	Switch or Enable Subsystem block whose parameter Propagate sizes of variable-size signals is set to <code>During execution</code>
2	States require resetting when the input signal size changes.	Unit Delay block in an Enabled Subsystem whose parameter Propagate sizes of variable-size signals is set to <code>Only when enabling</code>
3	Output signal size depends only on the input signal size.	Gain block.

The search stops at the boundary of Enable, Function-Call, and Action subsystems because these subsystems can specify when to propagate the size of a variable-size signal.

Simulink sets the propagation of variable-size signals for a referenced model as follows:

- One or more blocks in category 1, and all other blocks are in category 3, select `During execution`.
- One or more blocks in category 2, and all another blocks are in category 3, select `Only when enabling`.

- Blocks in category 1 and 2, report an error.
- All blocks in category 3 with a conditionally executed subsystem that is not an Enable, Function-Call, or Action subsystem, report an error. Simulink, in this case, cannot determine when to propagate sizes of variable-size signals.
- All blocks in category 3 with only conditionally executed subsystems that are an Enable, Function-Call, or Action subsystem, support both Only when enabling and During execution.

Only when enabling

Propagates sizes of variable-size signals for the referenced model only when enabling (at Enable method).

During execution

Propagates sizes of variable-size signals for the referenced model during execution (at Outputs method).

Command-Line Information

Parameter: PropagateVarSize

Type: string

Value: 'Infer from blocks in model' | 'Only when enabling' | 'During execution'

Default: 'Infer from blocks in model'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	Off

See Also

- Configuration Parameters Dialog Box

- Model Referencing Pane

Model dependencies

Specify the files on which this model relies. They are typically MAT-files and MATLAB code files used to initialize parameters and to provide data.

Settings

No Default

- Specify the dependencies as a cell array of strings, where each cell array entry is the filename or path of a dependent file. These filenames may include spaces and must include file extensions (for example, `.m` or `.mat`).
- Prefix the token `$MDL` to a dependency to indicate that the path to the dependency is relative to the location of this model file.
- Wildcards are allowed. Use a `'%'` to comment out a line; use `'...'` to continue lines.

Tips

- If Simulink software cannot find a specified dependent file when you update or simulate a model that references this model, Simulink software displays a warning.
- The dependencies automatically include the `model.mdl` and linked library `.mdl` files.
- For files not on the MATLAB path, use absolute paths.

Command-Line Information

Parameter: ModelDependencies

Type: string

Value: any valid value

Default: ''

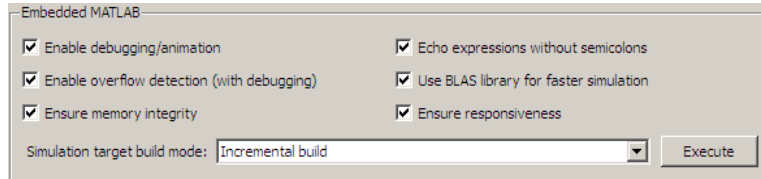
Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- “Rebuild options” on page 1-454
- Configuration Parameters Dialog Box
- Model Referencing Pane

Simulation Target Pane: General



In this section...

“Simulation Target: General Tab Overview” on page 1-476

“Enable debugging/animation” on page 1-477

“Enable overflow detection (with debugging)” on page 1-479

“Ensure memory integrity” on page 1-481

“Echo expressions without semicolons” on page 1-483

“Use BLAS library for faster simulation” on page 1-485

“Ensure responsiveness” on page 1-487

“Simulation target build mode” on page 1-489

Simulation Target: General Tab Overview

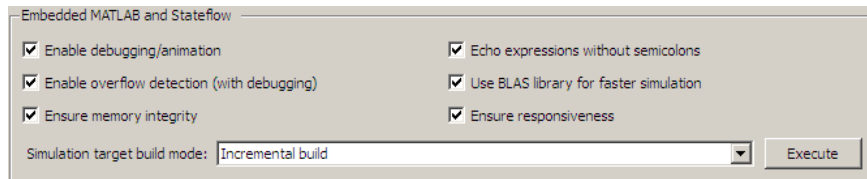
Configure the simulation target for a model that contains Embedded MATLAB Function blocks, Stateflow charts, or Truth Table blocks.

Configuration

Set the parameters that appear.

Dependency

If you have a Stateflow license, the title becomes **Embedded MATLAB and Stateflow**. This title also appears if you do not have a Stateflow license, but load a model that contains Stateflow charts or Truth Table blocks. In either case, the parameters remain the same.



See Also

- Speeding Up Simulation
- Configuration Parameters Dialog Box
- Simulation Target Pane: General

Enable debugging/animation

Enable debugging and animation during simulation of a model that contains Embedded MATLAB Function blocks, Stateflow charts, or Truth Table blocks.

Settings

Default: On



On

Enables debugging and animation of a model during simulation.

This action also enables the Stateflow Debugger.



Off

Disables debugging and animation of a model during simulation.

This action also disables the Stateflow Debugger.

Tip

- If you disable debugging and animation, faster model simulation occurs.

Command-Line Information

Parameter: SFSimEnableDebug

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	On
Traceability	No impact
Efficiency	Off
Safety precaution	On

See Also

- [Speeding Up Simulation](#)
- [Configuration Parameters Dialog Box](#)
- [Simulation Target Pane: General](#)

Enable overflow detection (with debugging)

Enable overflow detection of data during simulation of a model that contains Embedded MATLAB Function blocks, Stateflow charts, or Truth Table blocks. Overflow occurs for data when a value assigned to it exceeds the numeric capacity of the data type.

Settings

Default: On



On

Enables overflow detection of data during simulation.

You must also select the **Data Range** check box in the Stateflow Debugger window.



Off

Disables overflow detection of data during simulation.

Tips

- If your model contains fixed-point data, enable overflow detection.
- If you disable overflow detection, faster model simulation occurs.

Command-Line Information

Parameter: SFSimOverflowDetection

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	On
Traceability	No impact

Application	Setting
Efficiency	Off
Safety precaution	On

See Also

- [Speeding Up Simulation](#)
- [Overflow Detection for Fixed-Point Types](#)
- [Configuration Parameters Dialog Box](#)
- [Simulation Target Pane: General](#)

Ensure memory integrity

Detects violations of memory integrity in code generated for Embedded MATLAB function blocks and stops execution with a diagnostic.

Settings

Default: On



On

Detect violations of memory integrity in code generated for Embedded MATLAB function blocks and stops execution with a diagnostic message.



Off

Does not detect violations of memory integrity in code generated for Embedded MATLAB function blocks.

Caution Without these checks, violations will result in unpredictable behavior.

Tips

- The most likely cause of memory integrity issues is accessing an array out of bounds.
- Only disable these checks if you are sure that your code is safe and that all array bounds and dimension checking is unnecessary.

Command-Line Information

Parameter: SimIntegrity

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	On
Traceability	On
Efficiency	Off
Safety precaution	On

See Also

- “Controlling Run-Time Checks” in the *Simulink User’s Guide*.
- Configuration Parameters Dialog Box
- Simulation Target Pane: General

Echo expressions without semicolons

Enable run-time output in the MATLAB Command Window, such as actions that do not terminate with a semicolon. This behavior applies to a model that contains Embedded MATLAB Function blocks, Stateflow charts, or Truth Table blocks.

Settings

Default: On



On

Enables run-time output to appear in the MATLAB Command Window during simulation.



Off

Disables run-time output from appearing in the MATLAB Command Window during simulation.

Tip

- If you disable run-time output, faster model simulation occurs.

Command-Line Information

Parameter: SFSimEcho

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	On
Traceability	No impact
Efficiency	Off
Safety precaution	No impact

See Also

- [Speeding Up Simulation](#)
- [Configuration Parameters Dialog Box](#)
- [Simulation Target Pane: General](#)

Use BLAS library for faster simulation

Enable Embedded MATLAB Function blocks in Simulink models and Embedded MATLAB functions in Stateflow charts to speed up low-level matrix operations during simulation.

Settings

Default: On



On

Embedded MATLAB Function blocks in Simulink models and Embedded MATLAB functions in Stateflow charts use Basic Linear Algebra Subprograms (BLAS) libraries for low-level matrix operations.



Off

Embedded MATLAB Function blocks in Simulink models and Embedded MATLAB functions in Stateflow charts do not use Basic Linear Algebra Subprograms (BLAS) libraries for low-level matrix operations.

Command-Line Information

Parameter: SimBlas

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	On
Safety precaution	No impact

See Also

- Configuration Parameters Dialog Box
- Simulation Target Pane: General

Ensure responsiveness

Enables responsiveness checks in code generated for Embedded MATLAB function blocks.

Settings

Default: On



On

Enables periodic checks for Ctrl+C breaks in code generated for Embedded MATLAB function blocks. Also allows graphics refreshing.



Off

Disables periodic checks for Ctrl+C breaks in code generated for Embedded MATLAB function blocks. Also disables graphics refreshing.

Caution Without these checks the only way to end a long-running execution may be to terminate MATLAB.

Command-Line Information

Parameter: SimCtrlC

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	On
Traceability	On
Efficiency	Off
Safety precaution	On

See Also

- “Controlling Run-Time Checks” in the *Simulink User’s Guide*.
- Configuration Parameters Dialog Box
- Simulation Target Pane: General

Simulation target build mode

Specifies how you build the simulation target for a model that contains Embedded MATLAB Function blocks, Stateflow charts, or Truth Table blocks.

Settings

Default: Incremental build

Incremental build

This option rebuilds only those portions of the target that you changed since the last build.

Rebuild all (including libraries)

This option rebuilds the target, including libraries, from scratch.

Make without generating code

This option invokes the make process without generating code.

Clean all (delete generated code/executables)

This option deletes both generated source code and executable files.

Clean objects (delete executables only)

This option deletes only executable files.

Tips

- The default `Incremental build` is a good choice for most models. This action takes place whenever you simulate your model.
- Set `Rebuild all (including libraries)` if you have changed your compiler or updated your object files since the last simulation. For example, use this option to rebuild the simulation target to include custom code changes.
- Set `Make without generating code` when you have custom source files that you must recompile in an incremental build mechanism that does not detect changes in custom code files.

Command-Line Information

Parameter: SimBuildMode

Type: string

Value: 'sf_incremental_build' | 'sf_nonincremental_build' | 'sf_make' | 'sf_make_clean' | 'sf_make_clean_objects'

Default: 'sf_incremental_build'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Configuration Parameters Dialog Box
- Simulation Target Pane: General

Simulation Target Pane: Symbols



In this section...

“Simulation Target: Symbols Tab Overview” on page 1-492

“Reserved names” on page 1-493

Simulation Target: Symbols Tab Overview

Configuration

1 Enter reserved names for a model that contains Embedded MATLAB Function blocks, Stateflow charts, or Truth Table blocks.

2 Click **Apply**.

See Also

- Configuration Parameters Dialog Box
- Simulation Target Pane: Symbols

Reserved names

Enter the names of variables or functions in the generated code that match the names of variables or functions specified in custom code for a model that contains Embedded MATLAB Function blocks, Stateflow charts, or Truth Table blocks.

Settings

Default: {}

This action changes the names of variables or functions in the generated code to avoid name conflicts with identifiers in custom code. Reserved names must be shorter than 256 characters.

Tips

- Start each reserved name with a letter or an underscore to prevent error messages.
- Each reserved name must contain only letters, numbers, or underscores.
- Separate the reserved names using commas or spaces.
- You can also specify reserved names by using the command line:

```
config_param_object.set_param('SimReservedNameArray', {'abc', 'xyz'})
```

where *config_param_object* is the object handle to the model settings in the Configuration Parameters dialog box.

Command-Line Information

Parameter: SimReservedNameArray

Type: string array

Value: any reserved names shorter than 256 characters

Default: {}

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Configuration Parameters Dialog Box
- Simulation Target Pane: Symbols

Simulation Target Pane: Custom Code

The image shows a dialog box titled "Simulation Target Pane: Custom Code". It has a checked checkbox at the top left labeled "Parse custom code symbols". Below this, there are two main sections. The first section is titled "Include custom C code in generated:" and contains a list of options on the left: "Source file", "Header file", "Initialize function", and "Terminate function". To the right of this list is a large empty text area labeled "Source file:". The second section is titled "Include list of additional:" and contains a list of options on the left: "Include directories", "Source files", and "Libraries". To the right of this list is another large empty text area labeled "Include directories:".

In this section...

“Simulation Target: Custom Code Tab Overview” on page 1-497

“Parse custom code symbols” on page 1-498

“Source file” on page 1-500

In this section...

“Header file” on page 1-501

“Initialize function” on page 1-503

“Terminate function” on page 1-504

“Include directories” on page 1-505

“Source files” on page 1-507

“Libraries” on page 1-509

“Use local custom code settings (do not inherit from main model)” on page 1-511

Simulation Target: Custom Code Tab Overview

Include custom code settings for a model that contains Embedded MATLAB Function blocks, Stateflow charts, or Truth Table blocks.

Configuration

- 1 Select the type of information to include from the list on the left side of the pane.
- 2 Enter a string to identify the specific code, folder, source file, or library.
- 3 Click **Apply**.

See Also

- Including Custom C Code
- Configuration Parameters Dialog Box
- Simulation Target Pane: Custom Code

Parse custom code symbols

Specify whether or not to parse the custom code and report unresolved symbols in a model. This setting applies to all charts in the model, including library link charts.

Settings

Default: On



On

Enables parsing of custom code to report unresolved symbols in charts of your model.



Off

Disables parsing of custom code.

Tips

- When you create a new model, this check box is selected by default.
- When you load models saved as version R2010a or earlier, this check box is selected only if the mex compiler is lcc. Otherwise, the check box is not selected.

Command-Line Information

Parameter: SimParseCustomCode

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

Application	Setting
Debugging	On
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	On

See Also

- Including Custom C Code
- Resolving Symbols in Stateflow Charts
- Configuration Parameters Dialog Box
- Simulation Target Pane: Custom Code

Source file

Enter code lines to appear near the top of a generated source code file.

Settings

Default: ''

Code lines appear near the top of the generated *model.c* source file, outside of any function.

Command-Line Information

Parameter: SimCustomSourceCode

Type: string

Value: any source file name

Default: ''

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Including Custom C Code
- Configuration Parameters Dialog Box
- Simulation Target Pane: Custom Code

Header file

Enter code lines to appear near the top of a generated header file.

Settings

Default: ''

Code lines appear near the top of the generated *model.h* header file.

Tips

- When you include a custom header file, enclose the file name in double quotes. For example, `#include "sample_header.h"` is a valid declaration for a custom header file.
- You can include extern declarations of variables or functions.

Command-Line Information

Parameter: SimCustomHeaderCode

Type: string

Value: any header file name

Default: ''

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Including Custom C Code

- Configuration Parameters Dialog Box
- Simulation Target Pane: Custom Code

Initialize function

Enter code statements that execute once at the start of simulation.

Settings

Default: ''

Code appears inside the model's initialize function in the *model.c* file.

Tip

- Use this code to invoke functions that allocate memory or to perform other initializations of your custom code.

Command-Line Information

Parameter: SimCustomInitializer

Type: string

Value: any code

Default: ''

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Including Custom C Code
- Configuration Parameters Dialog Box
- Simulation Target Pane: Custom Code

Terminate function

Enter code statements that execute at the end of simulation.

Settings

Default: ''

Code appears inside the model's terminate function in the *model.c* file.

Tip

- Use this code to invoke functions that free memory allocated by the custom code or to perform other cleanup tasks.

Command-Line Information

Parameter: SimCustomTerminator

Type: string

Value: any code

Default: ''

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Including Custom C Code
- Configuration Parameters Dialog Box
- Simulation Target Pane: Custom Code

Include directories

Specify a list of folder paths that contain files you include in the compiled target.

Settings

Default: ' '

Enter a space-separated list of folder paths.

- Specify absolute or relative paths to the directories.
- Relative paths must be relative to the folder containing your model files, not relative to the build folder.
- The order in which you specify the directories is the order in which they are searched for header, source, and library files.

Note If you specify a Windows® path string containing one or more spaces, you must enclose the string in double quotes. For example, the second and third path strings in the **Include directories** entry below must be double-quoted:

```
C:\Project "C:\Custom Files" "C:\Library Files"
```

If you set the equivalent command-line parameter `SimUserIncludeDirs`, each path string containing spaces must be separately double-quoted within the single-quoted third argument string, for example,

```
>> set_param('mymodel', 'SimUserIncludeDirs', ...  
            'C:\Project "C:\Custom Files" "C:\Library Files"')
```

Command-Line Information

Parameter: `SimUserIncludeDirs`

Type: string

Value: any folder file name

Default: ' '

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Including Custom C Code
- Configuration Parameters Dialog Box
- Simulation Target Pane: Custom Code

Source files

Specify a list of source files to compile and link into the target.

Settings

Default: ''

You can separate source files with a comma, a space, or a new line.

Limitation

This parameter does not support Windows file names that contain embedded spaces.

Tip

- The file name is sufficient if the file is in the current MATLAB folder or in one of the include directories.

Command-Line Information

Parameter: SimUserSources

Type: string

Value: any source file name

Default: ''

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Including Custom C Code
- Configuration Parameters Dialog Box
- Simulation Target Pane: Custom Code

Libraries

Specify a list of static libraries that contain custom object code to link into the target.

Settings

Default: ''

Enter a space-separated list of library files.

Limitation

This parameter does not support Windows file names that contain embedded spaces.

Tip

- The file name is sufficient if the file is in the current MATLAB folder or in one of the include directories.

Command-Line Information

Parameter: SimUserLibraries

Type: string

Value: any library file name

Default: ''

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact
Efficiency	No impact
Safety precaution	No impact

See Also

- Including Custom C Code
- Configuration Parameters Dialog Box
- Simulation Target Pane: Custom Code

Use local custom code settings (do not inherit from main model)

Specify if a library model can use custom code settings that are unique from the main model.

Settings

Default: Off



On

Enables a library model to use custom code settings that are unique from the main model.



Off

Disables a library model from using custom code settings that are unique from the main model.

Dependency

This parameter is available only for library models that contain Embedded MATLAB Function blocks, Stateflow charts, or Truth Table blocks. To access this parameter, select **Tools > Open Simulation Target** in the Embedded MATLAB Editor or Stateflow Editor for your library model.

Command-Line Information

Parameter: SimUseLocalCustomCode

Type: string

Value: 'on' | 'off'

Default: 'off'

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact

See Also

- Including Custom C Code
- Configuration Parameters Dialog Box
- Simulation Target Pane: Custom Code

Library Browser

- “About the Library Browser” on page 2-2
- “Libraries Pane” on page 2-5
- “Library Pane” on page 2-8
- “Block Description Pane” on page 2-14
- “Search Toolbar” on page 2-15
- “Found Pane” on page 2-18
- “Most Frequently Used Blocks Pane” on page 2-21
- “Library Data Repository” on page 2-22
- “Library Browser Keyboard Shortcuts” on page 2-23

About the Library Browser

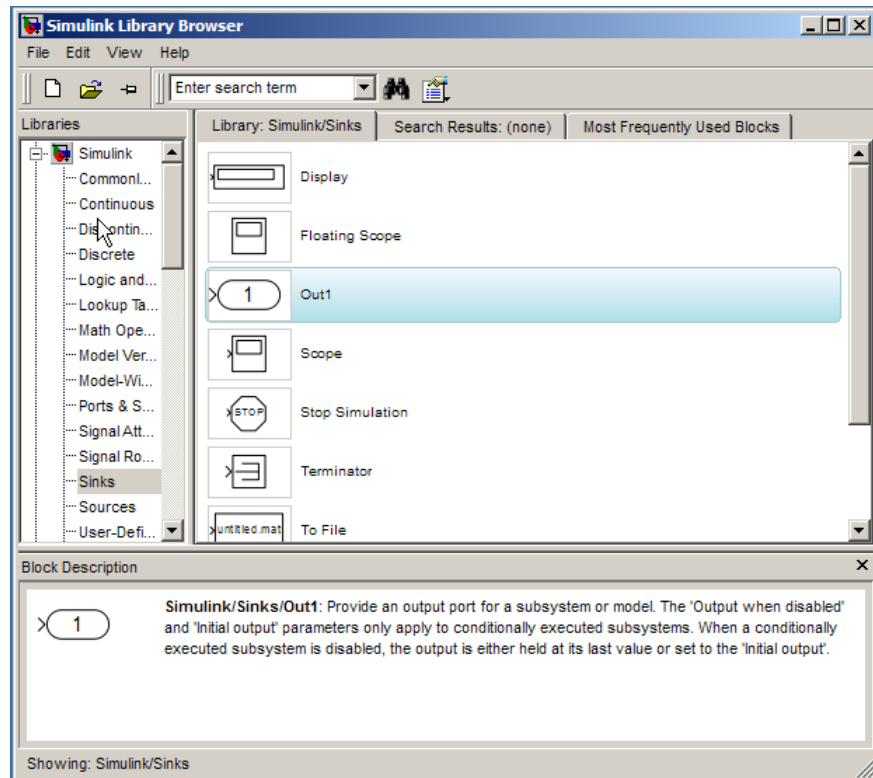
Use the Library Browser to browse and search Simulink block libraries for blocks to use in your models. You can also select from a list of your most frequently used blocks.

The following figure shows the Library Browser with:

- The Sinks library selected from the **Libraries** pane (left-most pane)
- The Out1 block selected in the **Library** pane, which lists the blocks from the library selected in the **Libraries** pane (Sinks, in this example)
- A **Block Description** pane for the Out1 block (the block selected in the **Library** pane), at the bottom of the Library browser.

The Library Browser has a search box for searching for blocks or libraries whose names that contain the search term you specify.

The **Most Frequently Used Blocks** pane displays a list of the blocks you have added the most often.



For details about Library Browser features, see:

- “Libraries Pane” on page 2-5
- “Library Pane” on page 2-8
- “Most Frequently Used Blocks Pane” on page 2-21
- “Block Description Pane” on page 2-14
- “Search Toolbar” on page 2-15
- “Found Pane” on page 2-18

For information about key tasks involving the Library Browser, see:

- “Populating a Model”

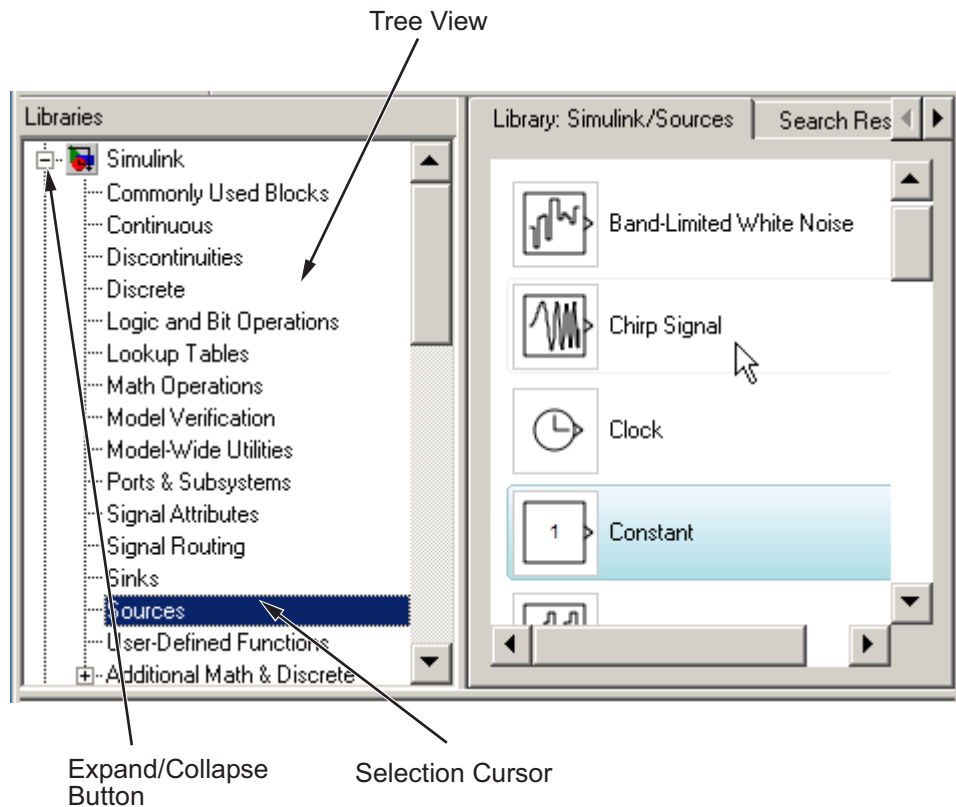
- “Adding Blocks”
- “Adding Libraries to the Library Browser”

Libraries Pane

In this section...
“About the Libraries Pane” on page 2-5
“Selecting Nodes” on page 2-6
“Expanding and Collapsing Nodes” on page 2-7
“Refreshing the Tree View” on page 2-7

About the Libraries Pane

The **Libraries** pane allows you to select block libraries for browsing. The pane displays a tree-structured directory (tree view) of libraries installed on your system, each of whose nodes you can select with your mouse or keyboard. Selecting a node displays the contents of the corresponding library in the Library Browser’s **Library** pane. The **Libraries** pane indicates the selected library via a selection cursor that highlights the corresponding node.



Selecting Nodes

You can use your keyboard or your mouse to select a node in the tree view. To select a node with the mouse, first use the **Libraries** pane's scroll bars, if necessary, to move the node into view. Then click the node. To select a node with the keyboard, use the keyboard's **Up** or **Down** arrow key to move the tree's node selection cursor to the desired node.

Note Selecting a node can cause the Library Browser to update its library data repository (see “Library Data Repository” on page 2-22). In this case, it displays an advisory dialog box while the update is in process.

Expanding and Collapsing Nodes

To facilitate navigation of large directory trees, the **Libraries** pane displays libraries containing sublibraries as expandable nodes, each containing a toggle button labeled + or -, depending on whether the node is collapsed or expanded. Clicking the button expands or collapses its node to show or hide the corresponding parent library's children. You can also expand or collapse the currently selected node by pressing the **Right** or **Left** arrow keys on your keyboard, respectively.

Refreshing the Tree View

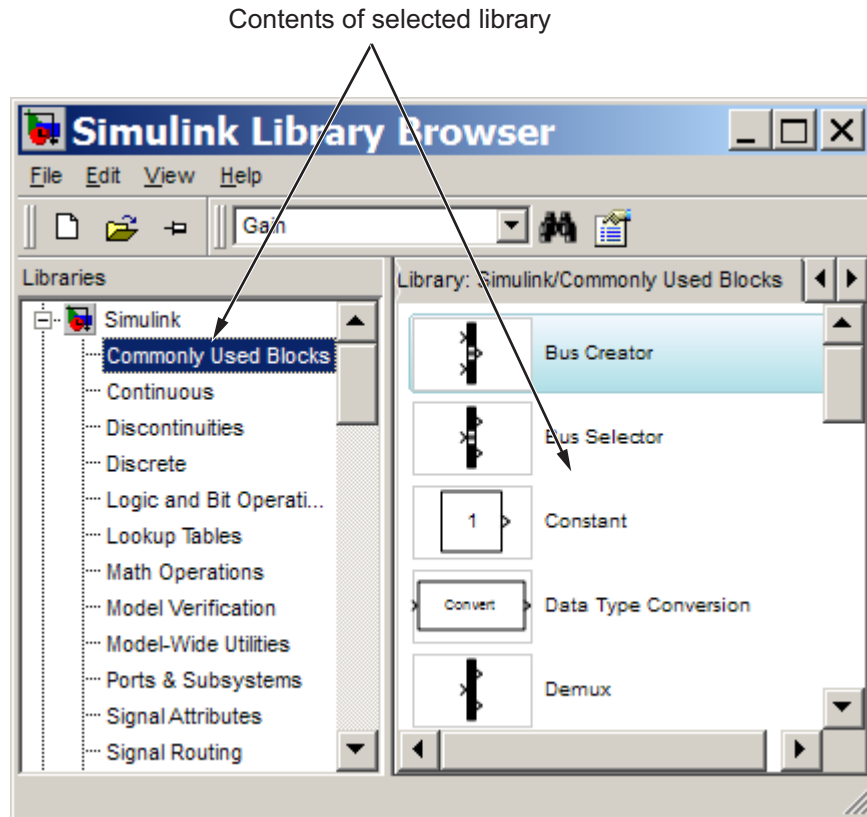
To refresh the tree view displayed in the **Libraries** pane, select **Refresh Tree View** from the Library Browser's **View** menu. The Library Browser adds or removes any libraries from the tree view that have been added or deleted, respectively, from the MATLAB path since the view was last updated.

Library Pane

In this section...
“About the Library Pane” on page 2-8
“How to Tell Blocks from Libraries in the Library Pane” on page 2-9
“Choosing the Library Pane’s Layout” on page 2-10
“Setting Icon Size” on page 2-11
“Selecting Blocks” on page 2-12
“Creating an Instance of a Library Block in a Model” on page 2-12
“Displaying a Library Block’s Parameters” on page 2-12
“Displaying Help for a Library Block” on page 2-12
“Displaying the Contents of a Sublibrary” on page 2-12
“Displaying a Block or Sublibrary’s Parent” on page 2-13

About the Library Pane

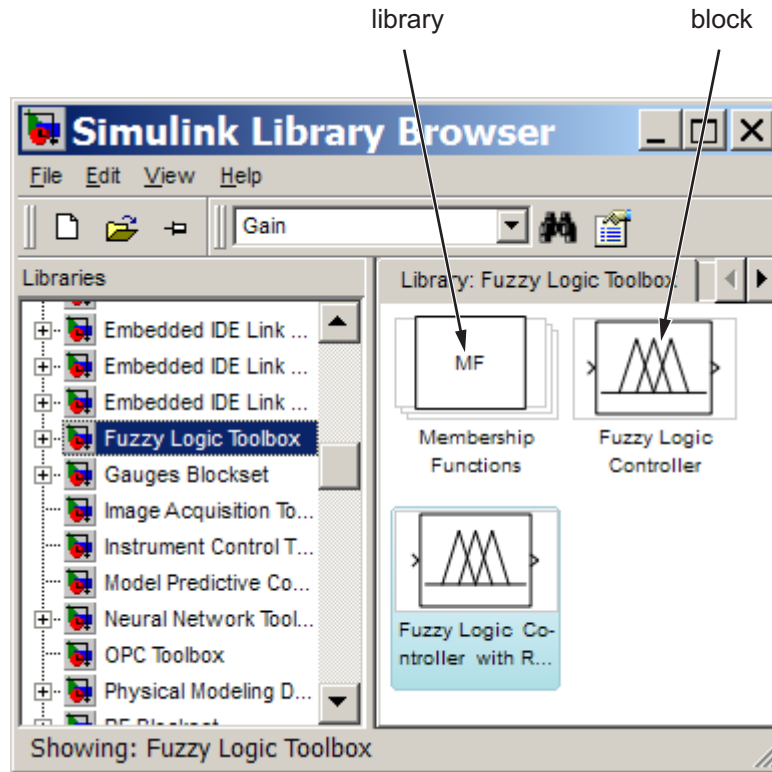
The Library Browser’s **Library** pane displays the contents of the library selected in the **Libraries** pane (see “Libraries Pane” on page 2-5).



You can use the **Library** pane to browse the contents of the selected library, to view a library block's parameters or help, and to create instances of library blocks in models.

How to Tell Blocks from Libraries in the Library Pane

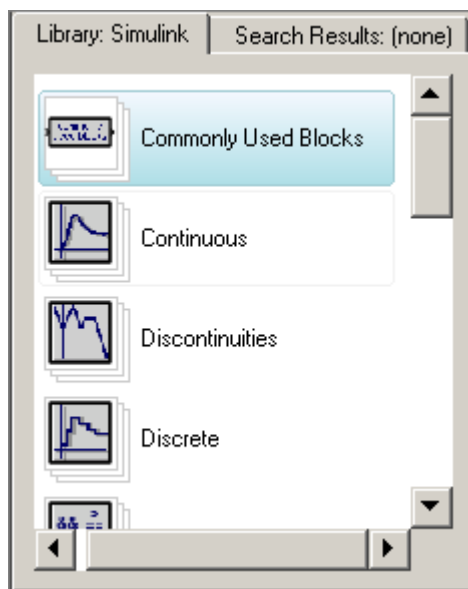
The **Library** pane displays the icon and name of each block or sublibrary contained by the library selected in the **Libraries** pane. To enable you to determine at a glance which icons represent libraries and which blocks, the Library Browser by default draws a single rectangle around an icon representing a block and a stack of three rectangles around an icon representing a library.



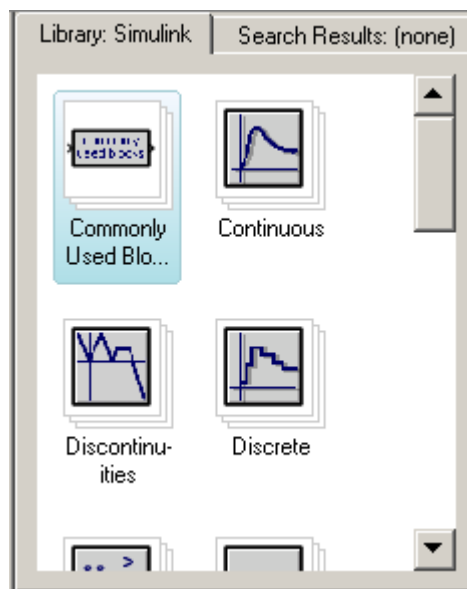
Note The Library Browser's compact icon option eliminates display of rectangles around icons (see "Setting Icon Size" on page 2-11). This allows the Library Browser to fit more blocks and libraries into the visible area of the **Library** pane.

Choosing the Library Pane's Layout

The Library Browser can display blocks in either a list (single-column) layout or a screen-saving grid layout. To choose a layout, select **List** or **Grid** from the Library Browser's **View > Layout** menu.



List Layout



Grid Layout

Tip

Setting the Library Browser's layout option to grid and its icon size to compact (see "Setting Icon Size" on page 2-11) results in the most compact display of a library's contents in the **Library** pane.

Setting Icon Size

The Library Browser's **View > Icon Size** menu allows you to select any of four icon sizes: **Compact (Ctrl+1)**, **Small (Ctrl+2)**, **Medium (Ctrl+3)**, **Large (Ctrl+4)**. The compact option eliminates the use of rectangles to distinguish block icons from library icons and thus takes up the least amount of space. Use this option if you want to maximize the number of icons visible in the Library Browser's **Library** pane.

Selecting Blocks

Many **Library** pane procedures require you to select a block displayed in the pane. To select a block, click it with the mouse. The Library Browser moves the block selection cursor to the selected block.

Creating an Instance of a Library Block in a Model

To create an instance of a library block in an existing model, select the block in the **Library** pane and drag it into the model's window.

To create an instance of a library block in a new model, select the block in the **Library** pane and then select **Add to a new model** from the library block's context menu or **Add Selected Block to a New Model** from the Library Browser's **Edit** menu.

Displaying a Library Block's Parameters

To display a library block's parameter dialog box, double-click the block in the **Library** pane or select **Block parameter** from the block's context menu.

Note The dialog box is disabled to prevent you from using it to change a library block's parameters.

Displaying Help for a Library Block

To display help for a library block, select the block and then select **Help** from the block's context menu or from the Library Browser's **Help** menu.

Displaying the Contents of a Sublibrary

To display the contents of a sublibrary appearing in the **Library** pane, double-click the library. The library's contents replaces the parent's contents in the **Library** pane.

Displaying a Block or Sublibrary's Parent

To display the parent of an item (block or library) displayed in the **Library** pane, select **Go to parent** from the item's context menu. The contents of the parent replaces the contents of the child in the **Library** pane.

Block Description Pane

This pane displays a description of the block selected in the **Library** pane. The text of the description is the same as the text of the description that appears on the block's parameter dialog box.

To hide this pane, deselect **Show block descriptions** on the Library Browser's **View** menu.

Search Toolbar

In this section...

“About the Search Toolbar” on page 2-15

“Using the Search Toolbar to Find Blocks” on page 2-15

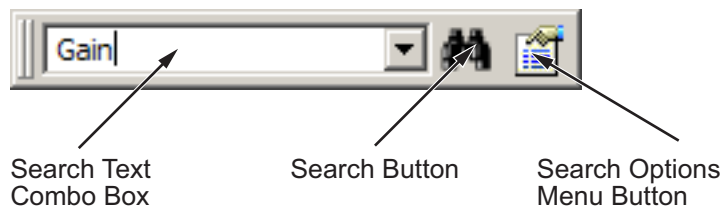
“Search Text Combo Box” on page 2-16

“Search Options Menu” on page 2-16

“Search Button” on page 2-17

About the Search Toolbar

The Library Browser’s **Search** toolbar allows you to search the block libraries installed on your system for blocks whose names contain or match a text string that you specify. The **Search** toolbar displays the blocks it finds in the Library Browser’s **Found** pane. You can use the **Found** pane to view a found block’s help or parameters or create an instance of the block in a model (see “Found Pane” on page 2-18 for more information).



Using the Search Toolbar to Find Blocks

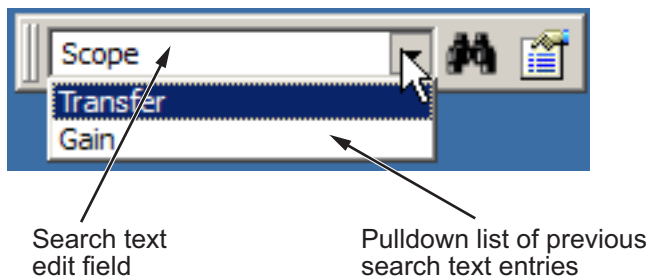
To find blocks whose names contain or match a specified text string:

- 1 Enter the string in the toolbar search text combo box (see “Search Text Combo Box” on page 2-16).
- 2 Use the toolbar search options menu to specify the search options you want to use, e.g., match whole words (see “Search Options Menu” on page 2-16).
- 3 Select the toolbar **Search** button to start the search.

The Library Browser searches the libraries installed on your system whose names contain or match the search string you specified, depending on the options you specified. It displays the results of the search in its **Found** pane (see “Found Pane” on page 2-18).

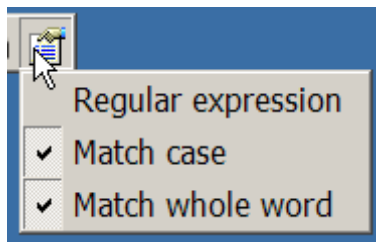
Search Text Combo Box

The search text combo box allows you to specify the text string to be used to search the block libraries on your system. The combo box consists of an edit field and a drop-down list of search strings that you previously entered in the current or previous sessions. To specify a string, enter the string in the edit field or choose a string from the drop-down list.



Search Options Menu

The **Search Options** menu allows you to specify various search options. To display the menu, select the search options menu button on the **Search** toolbar.



Regular Expression

Treat search text as a MATLAB regular expression (see “Regular Expressions”).

Match Case

Consider case when matching search text against block names.

Match Whole Word

Allow a match only if the search string matches a word, i.e., text set off by white space, in a block’s name.

Search Button

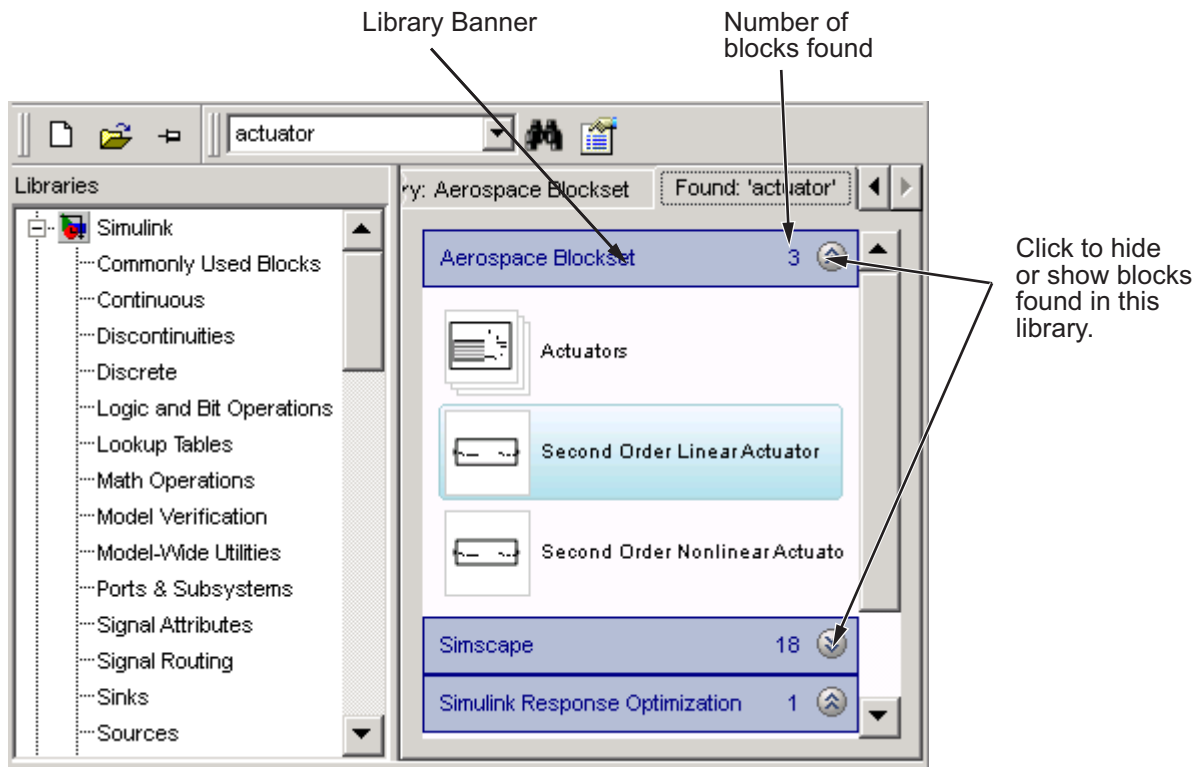
Selecting this button initiates a search of the block libraries on your system, using the text string and search options specified in the toolbar’s search text combo box (see “Search Text Combo Box” on page 2-16) and search options menu (see “Search Options Menu” on page 2-16), respectively.

Found Pane

In this section...
“About the Found Pane” on page 2-18
“Selecting a Found Block in Library View” on page 2-19
“Displaying a Found Block’s Path” on page 2-20

About the Found Pane

The **Found** pane allows you to view and select blocks found by the Library Browser’s **Search** tool. The pane displays the blocks found by the tool grouped by library. A banner at the top of each group displays the name of the top-level library containing the found blocks, the number of blocks found in the library, and a button that allows you to hide or display the search results for the library.



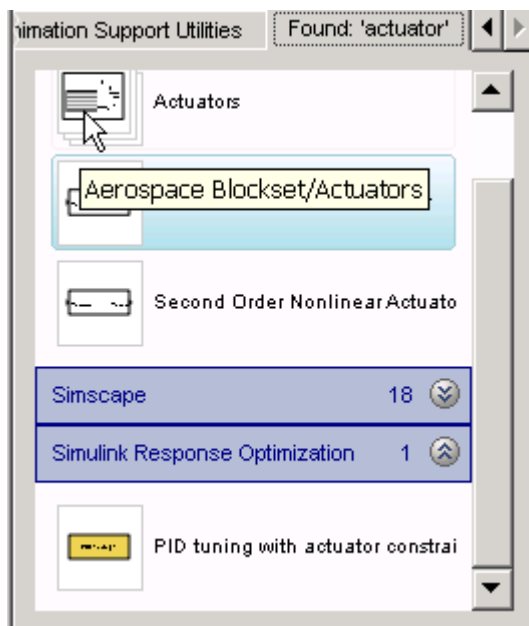
You can use the **Found** pane to perform many of the same tasks you can perform with the **Library** pane, including creating instances of found blocks in models and other libraries, displaying a found block's parameters, and displaying help for a found block. The procedures for these tasks are the same as for the **Library** pane (see “Library Pane” on page 2-8 for more information).

Selecting a Found Block in Library View

To select a found block in the **Library** pane, select the block in the **Found** pane and then select **Select in library view** from the block's context menu or enter **Ctrl+R**. This brings the **Library** pane forward in the Library Browser with the found block in view and selected.

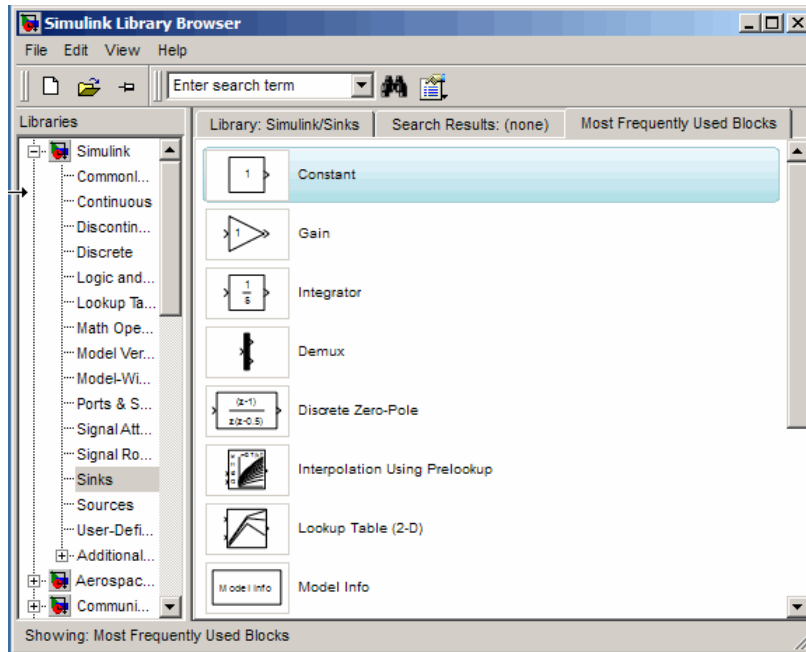
Displaying a Found Block's Path

To display the path of a found block, move the cursor over the block. The block's path appears in a tooltip.



Most Frequently Used Blocks Pane

Open the **Most Frequently Used Blocks** pane from the third tab in the Library Browser.



For details, see “Selecting Your Most Frequently Used Blocks From the Library Browser”.

Library Data Repository

To allow you to browse and search libraries without first loading them, the Library Browser maintains a repository of data about each library installed on your system. The data includes block paths and icons. The Library Browser updates the repository when you select a library whose contents have changed since the last time you selected the library.

Library Browser Keyboard Shortcuts

Task	Shortcut
Select Libraries or Library panes or tools on the Search toolbar.	Tab
Switch from the Library pane to the Found pane, or vice versa.	Ctrl+P
Open a model.	Ctrl+O
Move node selection down in the Libraries pane tree view.	Down Arrow
Move node selection up in the Libraries pane tree view.	Up Arrow
Expand a node in the Libraries pane tree view.	Right Arrow
Collapse a node in the Libraries pane tree view.	Left Arrow
Select a block found with the search tool in the Library pane.	Ctrl+R
Insert the selected block in a new model.	Ctrl+I
Increase the Library Browser's font size.	Ctrl++
Decrease the Library Browser's font size	Ctrl+-
Use compact block icons.	Ctrl+1
Use small block icons.	Ctrl+2
Use medium block icons.	Ctrl+3
Use large block icons.	Ctrl+4
Find a block.	Ctrl+F

Signal Properties Dialog Box

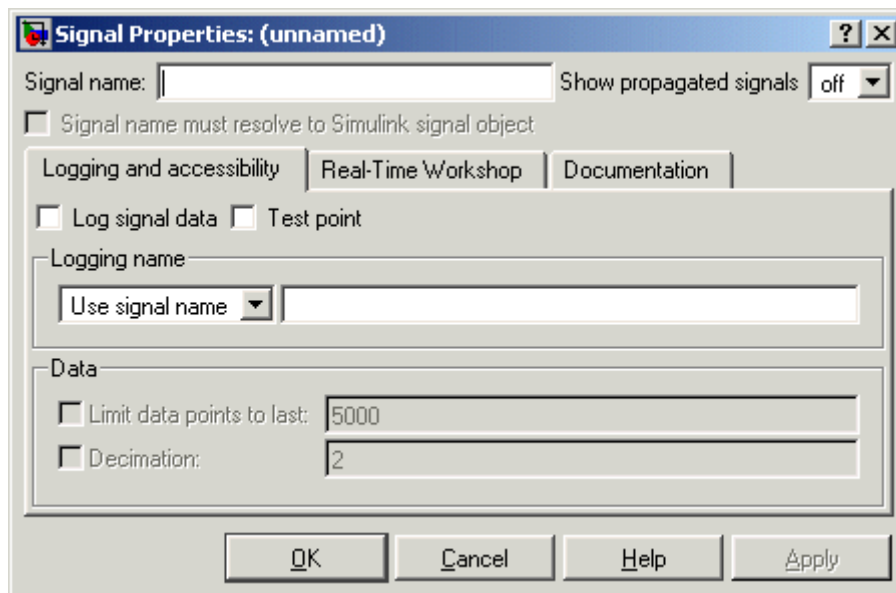
The **Signal Properties** dialog box lets you display and edit signal properties. To display the dialog box, either

- Select the line that represents the signal whose properties you want to set and then choose **Signal Properties** from the signal's context menu or from the Simulink **Edit** menu

or

- Select a block that outputs or inputs the signal and select **Port Signal Properties** from the block's context menu, then select the port to which the signal is connected from the resulting menu

The **Signal Properties** dialog box appears.



The dialog box includes the following controls.

- “Signal Properties Controls” on page 3-2
- “Logging and Accessibility Options” on page 3-4
- “Real-Time Workshop Options” on page 3-6
- “Documentation Options” on page 3-7

Signal Properties Controls

Signal name

Name of signal.

Signal name must resolve to Simulink signal object

Specifies that either the base MATLAB workspace or the model workspace must contain a `Simulink.Signal` object with the same name as this signal.

Simulink software displays an error message if it cannot find such an object when you update or simulate the model containing this signal.

Note Simulink.Signal objects in the model workspace must have their storage class set to Auto. See “Using Model Workspaces” for more information.

When **Signal name must resolve to Simulink signal object** is enabled, a signal resolution icon appears by default to the left of any label on the signal. The icon looks like this:



See “Signal Resolution Indicators” for more information.

Show propagated signals

Note This option appears only for signals that originate from a virtual block other than a Bus Selector block.

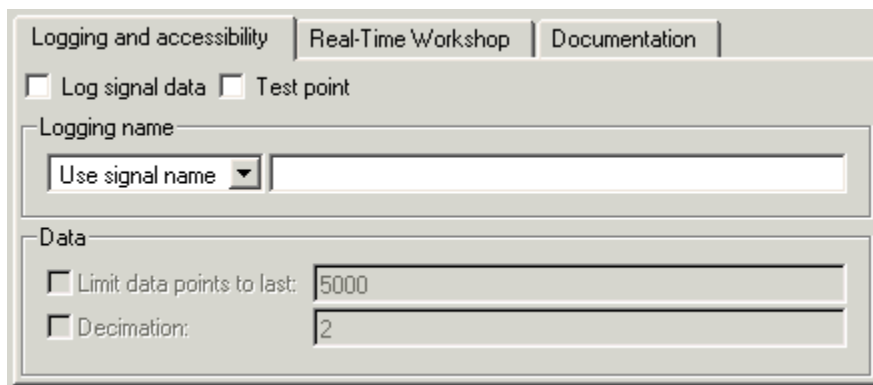
Show propagated signal names. You can select one of the following options:

Option	Description
off	Do not display signals represented by a virtual signal in the signal's label.
on	Display the virtual and nonvirtual signals represented by a virtual signal in the signal's label. For example, suppose that virtual signal <i>s1</i> represents a nonvirtual signal <i>s2</i> and a virtual signal <i>s3</i> . If this option is selected, the label for <i>s1</i> is <i>s1</i> < <i>s2</i> , <i>s3</i> >.
all	Display all the nonvirtual signals that a virtual signal represents either directly or indirectly. For example, suppose that virtual signal <i>s1</i> represents a nonvirtual signal <i>s2</i> and a virtual signal <i>s3</i> and virtual signal <i>s3</i> represents nonvirtual signals <i>s4</i> and <i>s5</i> . If this option is selected, the label for <i>s1</i> is <i>s1</i> < <i>s2</i> , <i>s4</i> , <i>s5</i> >.

See “Displaying the Nonvirtual Components of Virtual Signals” for more information.

Logging and Accessibility Options

Select the **Logging and accessibility** tab on the **Signal Properties** dialog box to display controls that enable you to specify signal logging and accessibility options for this signal.



The screenshot shows the 'Logging and accessibility' tab of the Signal Properties dialog box. It features three tabs at the top: 'Logging and accessibility' (selected), 'Real-Time Workshop', and 'Documentation'. Below the tabs are two main sections. The first section contains two checkboxes: 'Log signal data' and 'Test point'. Below these is a 'Logging name' section with a dropdown menu set to 'Use signal name' and an adjacent text input field. The second section is titled 'Data' and contains two checkboxes: 'Limit data points to last:' with a text input field containing '5000', and 'Decimation:' with a text input field containing '2'.

Log signal data

Select this option to cause Simulink software to save this signal’s values to the MATLAB workspace during simulation. See “Logging Signals” for details.

Test point

Select this option to designate this signal as a test point. See “Working with Test Points” for details.

Logging name

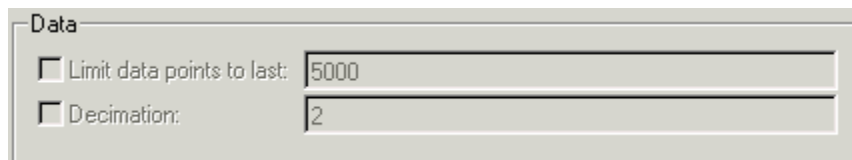
This pair of controls, consisting of a list box and an edit field, specifies the name associated with logged signal data.



Simulink software uses the signal's signal name as its logging name by default. To specify a custom logging name, select **Custom** from the list box and enter the custom name in the adjacent edit field.

Data

This group of controls enables you to limit the amount of data that Simulink software logs for this signal.



The options are as follows.

Limit data points to last

Discard all but the last N data points where N is the number entered in the adjacent edit field.

Decimation

Log every Nth data point where N is the number entered in the adjacent edit field. For example, suppose that your model uses a fixed-step solver with a step size of 0.1 s. if you select this option and accept the default decimation value (2), Simulink software records data points for this signal at times 0.0, 0.2, 0.4, etc.

Real-Time Workshop Options

The following controls set properties used by Real-Time Workshop to generate code from the model. You can ignore them if you are not going to generate code from the model.

Storage class

Select the storage class of this signal from the list. See “Interfacing Signals to External Code”, “Using the State Attributes Tab to Interface States to External Code”, and “Storage Classes for Data Store Memory Blocks” for information on how to use the listed options.

Storage type qualifier

Enter a storage type qualifier for this signal. For more information, see “Interfacing Signals to External Code”, “Using the State Attributes Tab to Interface States to External Code”, and “Storage Classes for Data Store Memory Blocks”.

Documentation Options

Description

Enter a description of the signal in this field.

Document link

Enter a MATLAB expression in the field that displays documentation for the signal. To display the documentation, click “Document Link.” For example, entering the expression

```
web(['file:/// ' which('foo_signal.html')])
```

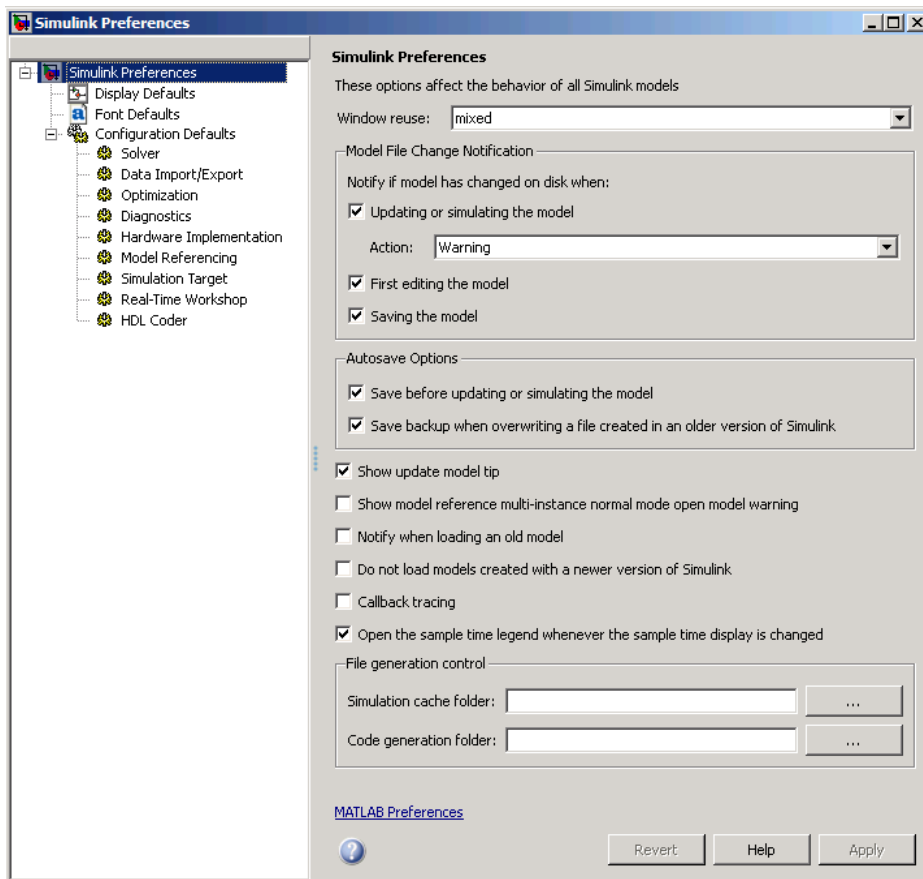
in the field causes MATLAB software’s default Web browser to display `foo_signal.html` when you click the field’s label.

3 Signal Properties Dialog Box

Simulink Preferences Window

- “Simulink Preferences Window: Main Pane” on page 4-2
- “Simulink Preferences Window: Display Defaults Pane” on page 4-25
- “Simulink Preferences Window: Font Defaults Pane” on page 4-32
- “Simulink Preferences Window: Configuration Defaults Pane” on page 4-33

Simulink Preferences Window: Main Pane



In this section...

“Simulink Preferences Window Overview” on page 4-3

“Window Reuse” on page 4-5

“Model File Change Notification” on page 4-7

“Updating or simulating the model” on page 4-8

“Action” on page 4-9

In this section...

“First editing the model” on page 4-11

“Saving the model” on page 4-12

“Autosave” on page 4-13

“Save before updating or simulating the model” on page 4-14

“Save backup when overwriting a file created in an older version of Simulink” on page 4-16

“Show model reference multi-instance normal mode open model warning” on page 4-17

“Show update model tip” on page 4-17

“Notify when loading an old model” on page 4-18

“Do not load models created with a newer version of Simulink” on page 4-19

“Callback tracing” on page 4-20

“Open the Sample Time Legend whenever Sample Time Display is changed” on page 4-21

“File generation control” on page 4-22

“Simulation cache folder” on page 4-23

“Code generation folder” on page 4-24

Simulink Preferences Window Overview

The Simulink Preferences Window comprises the following panes:

- General Preferences (root level)
Set preferences for configuring window reuse, file change and old version notifications, autosave, and callback tracing.
- Display Defaults
Configure display options for the Model Browser, block connection lines and port data types.
- Font Defaults

Configure font options for blocks, lines and annotations.

- **Configuration Defaults**

Edit the template Configuration Parameters to be used as defaults for new models.

Click items in the tree to select panes.

Configuration

- 1** On the root level pane, select the check boxes to configure preferences.
- 2** Close the window to apply your changes.

Click **Apply** to apply your changes and keep the window open.

Your settings affect the behavior of all Simulink models, including those currently open, and all subsequent models. Your preference settings are preserved for the next time you use the software.

See Also

- “Simulink Preferences Window: Main Pane” on page 4-2
- Model File Change Notification
- “Simulink Preferences Window: Display Defaults Pane” on page 4-25
- “Simulink Preferences Window: Font Defaults Pane” on page 4-32
- “Simulink Preferences Window: Configuration Defaults Pane” on page 4-33

Window Reuse

Specify whether to use the current window or open new windows to display a subsystem or its parent.

Settings

Default: Mixed

Mixed

Mixed window reuse. Subsystem opens in its own window. When you press **Escape** (go to parent), the parent window rises to the front, and the subsystem window disappears.

None

Never reuse windows. Subsystem opens in a new window. Press **Escape**: the parent window moves to the front.

Replace

Replace parent window. Subsystem opens in a new window. Parent window disappears. Press **Escape**: the parent window appears, and the subsystem window disappears.

Reuse

Reuse parent window. Subsystem replaces the parent in the current window. Press **Escape**: the parent window replaces subsystem in current window.

Tips

- Reusing windows avoids cluttering your screen.
- Creating a window for each subsystem allows you to view subsystems side by side with their parents or siblings.
- The **WindowReuse** preference does not affect the behavior of library windows.

Command-Line Information

Parameter: WindowReuse

Type: string

Value: 'none' | 'reuse' | 'replace' | 'mixed'
Default: 'mixed'

Model File Change Notification

Use these preferences to specify notifications if the model has changed on disk when you update, simulate, edit or save the model. When updating or simulating, you can choose the action to take: warn, error, reload if unmodified, or show a dialog box where you can choose to reload or ignore. For more information, see Model File Change Notification.

The frame contains these controls:

- “Updating or simulating the model” on page 4-8
- “Action” on page 4-9
- “First editing the model” on page 4-11
- “Saving the model” on page 4-12

Updating or simulating the model

Specify whether to notify if the model has changed on disk when updating or simulating the model.

Settings

Default: On



On

Notify if the model has changed on disk when updating or simulating the model. Select the action to take in the **Action** list.



Off

Do not notify if the model has changed on disk when updating or simulating the model.

Tip

To programmatically check whether the model has changed on disk since it was loaded, use the function `slIsFileChangedOnDisk`.

Dependency

This parameter enables **Action**.

Command-Line Information

Parameter: MDLFileChangedOnDiskChecks

Type: struct, field name: CheckWhenUpdating

Value: true | false | 1 | 0

Default: true

See Also

Model File Change Notification

Action

Select what action to take if the file has changed on disk since it was loaded.

Settings

Default: Warning

Warning

Displays a warning in MATLAB command window

Error

Displays an error, at the MATLAB command window if simulating from the command line, or if simulating from a menu item, in the Simulation Diagnostics window.

Reload model (if unmodified)

Reloads if the model is unmodified. If the model is modified, you see the prompt dialog.

Show prompt dialog

Shows prompt dialog. In the dialog, you can choose to close and reload, or ignore the changes.

Tip

To programmatically check whether the model has changed on disk since it was loaded, use the function `slIsFileChangedOnDisk`.

Dependencies

This parameter is enabled by the parameter **Updating or simulating the model**.

Command-Line Information

Parameter: `MdlFileChangedOnDiskHandling`

Type: string

Value: 'Warning' | 'Error' | 'Reload model (if unmodified)' | 'Show prompt dialog'

Default: 'Warning'

See Also

Model File Change Notification

First editing the model

Specify whether to notify if the file has changed on disk when editing the model.

Settings

Default: On



On

Displays a warning if the file has changed on disk when you modify the block diagram. Any graphical operation that modifies the block diagram (e.g., adding a block) causes a warning dialog to appear. Any command-line operation that causes the block diagram to be modified (e.g., a call to `set_param`) will result in a warning like this at the command line:

```
Warning: Block diagram 'mymodel' is being edited but file has
changed on disk since it was loaded. You should close and
reload the block diagram.
```



Off

Do not check for changes on disk when first editing the model.

Tip

To programmatically check whether the model has changed on disk since it was loaded, use the function `slIsFileChangedOnDisk`.

Command-Line Information

Parameter: `MDLFileChangedOnDiskChecks`

Type: struct, field name: `CheckWhenEditing`

Value: `true` | `false` | `1` | `0`

Default: `true`

See Also

Model File Change Notification

Saving the model

Specify whether to notify if the file has changed on disk when saving the model.

Settings

Default: On



On

Notify if the file has changed on disk when you save the model.

- The `save_system` function displays an error, unless the `OverwriteIfChangedOnDisk` option is used.
- Saving the model by using the menu (**File > Save**) or a keyboard shortcut causes a dialog to be shown. In the dialog, you can choose to overwrite, save with a new name, or cancel the operation.



Off

Do not check for changes on disk when saving the model.

Tip

To programmatically check whether the model has changed on disk since it was loaded, use the function `slIsFileChangedOnDisk`.

Command-Line Information

Parameter: `MDLFileChangedOnDiskChecks`

Type: struct, field name: `CheckWhenSaving`

Value: `true` | `false` | `1` | `0`

Default: `true`

See Also

Model File Change Notification

Autosave

Use the Autosave preferences to specify whether to automatically save a backup copy of the model before updating or simulating, or when overwriting with a newer version of Simulink.

For more information, see these controls:

- “Save before updating or simulating the model” on page 4-14
- “Save backup when overwriting a file created in an older version of Simulink” on page 4-16

Save before updating or simulating the model

Specify whether to automatically save a backup copy of the model before updating or simulating.

Settings

Default: On



On

If the model has unsaved changes, automatically save a backup copy of the model before updating or simulating. This autosave copy can be useful for crash recovery.

The copy is saved in the same directory as the model, with the name *MyModel.mdl.autosave*.

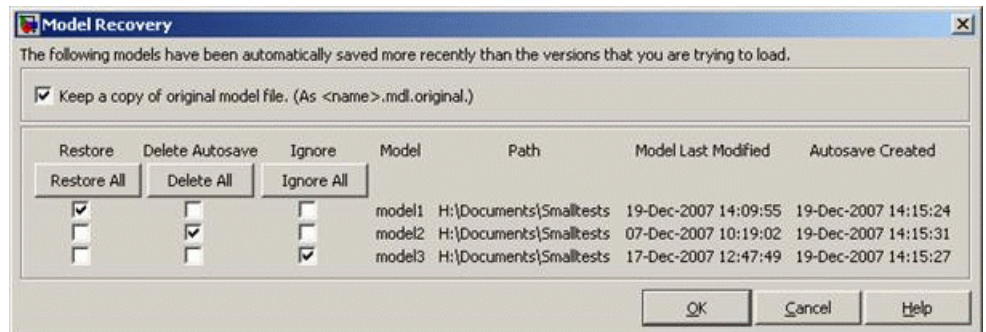


Off

Do not automatically save a copy before updating or simulating.

Tips

- If you are calling the `sim` command from within a `parfor` loop, turn the **AutoSave** option `Off`; otherwise, you may experience data concurrency issues.
- If you open or load a model that has a more recent autosave copy available, then after the model loads, the following Model Recovery dialog appears.



For each model in the list, you can select a check box to specify whether to **Restore**, **Delete Autosave**, or **Ignore**. Or you can click the **Restore All**, **Delete All** or **Ignore All** button to select that option for all listed models.

Option	Result
Restore	Overwrite the original model file with the autosave copy, and delete the autosave copy. Simulink will close the model and reload from the restored file. If you select the check box to Keep a copy of original model file , you can save copies of the original model files named <i>MyModel.mdl.original</i> .
Delete Autosave	Delete the autosave copy.
Ignore	Leave the model and the autosave copy untouched. This setting is the default. The next time you open the model, the Model Recovery dialog will reappear and you can choose to restore or delete autosave files.

- If you deliberately close a modified model, any autosave copy is deleted.
- Autosave does not occur for models that are part of the MATLAB installation, so you will not create autosave copies of the demo models.
- Autosave does not occur if the autosave file or location is read only.

Caution If a segmentation violation occurred, then the last autosave file for the model reflects the state of the autosave data prior to the segmentation violation. Because Simulink models might be corrupted by a segmentation violation, Simulink does not autosave a model after a segmentation violation occurs.

Command-Line Information

Parameter: AutoSaveOptions

Type: struct, field name: SaveOnModelUpdate

Value: true | false | 1 | 0

Default: true

Save backup when overwriting a file created in an older version of Simulink

Specify whether to automatically save a backup copy of the model when overwriting with a newer version of Simulink.

Settings

Default: On



On

If saving the model with a newer version of Simulink, automatically save a backup copy of the model. This backup copy can be useful for recovering the original file in case of accidental overwriting with a newer version.

The backup copy is saved in the same directory as the model, with the name *MyModel.mdl.Version*, where *Version* is the last version that saved the model, e.g., R2010a.



Off

Do not automatically save a backup copy when overwriting a model with a newer version of Simulink.

Tips

To recover the original model, rename the backup copy to *MyModel.mdl* by deleting the *Version* suffix.

Command-Line Information

Parameter: AutoSaveOptions

Type: struct, field name: SaveBackupOnVersionUpgrade

Value: true | false | 1 | 0

Default: true

Show model reference multi-instance normal mode open model warning

Show a warning when opening models from Model blocks that have Normal Mode Visibility set to off.

Settings

Default: On



On

After simulation, Simulink displays a warning if you try to open a referenced model from a Model block that has Normal Mode Visibility set to off.



Off

No warning displayed if, after simulation, you try to open a referenced model from a Model block that has Normal Mode Visibility set to off.

Tips

For more information, see “Normal Mode Visibility”.

Show update model tip

Specify whether or not to recommend the updating of a model from an earlier release.

Settings

Default: On



On

Display the update model tip when a model from an earlier release fails to compile on the first attempt after loading.



Off

Do not display the update model tip.

Notify when loading an old model

Specify whether to notify when loading a model last saved in a older version of Simulink software.

Settings

Default: Off



On

Print a message in the command window when loading a model last saved in an old version of Simulink software.



Off

No notification when loading old models.

Tips

- Run `slupdate('modelName')` to convert the block diagram to the format of the current version of Simulink software.
- For advice on upgrading a model to the current version of Simulink software, see “Consulting the Model Advisor”.

Command-Line Information

Parameter: `NotifyIfLoadOldModel`

Type: `string`

Value: `'on' | 'off'`

Default: `off`

Do not load models created with a newer version of Simulink

Specify whether to load a model last saved in a newer version of Simulink software.

Settings

Default: Off



On

Do not load any model last saved in a newer version of Simulink software, and print an error message in the command window.



Off

Load models last saved in a newer version of Simulink software, and print a warning message in the command window.

Tip

If possible, use the **Save As** command to convert the block diagram to the format of the desired version of Simulink software. The **Save As** command allows you to save a model created with the latest version of the Simulink software in formats used by earlier versions. See “Saving a Model in an Earlier Simulink Version”.

Command-Line Information

Parameter: ErrorIfLoadNewModel

Type: string

Value: 'on' | 'off'

Default: off

Callback tracing

Specify whether to display the model callbacks that Simulink software invokes when simulating a model.

Settings

Default: Off



On

Display the model callbacks in the MATLAB command window as they are invoked.

Callback tracing allows you to determine the callbacks the software invokes, and in what order, when you open or simulate a model.



Off

Do not display model callbacks.

Command-Line Information

Parameter: CallbackTracing

Type: string

Value: 'on' | 'off'

Default: 'off'

Open the Sample Time Legend whenever Sample Time Display is changed

Specify whether to display the Sample Time Legend whenever Sample Time Display is changed.

Settings

Default: On



On

Display the Sample Time Legend whenever you change Sample Time Display by selecting Colors, Annotations, or All from the Sample Time Display submenu. The model diagram is updated and the legend opens.



Off

Do not display the Sample Time Legend whenever Sample Time Display is changed.

Command-Line Information

Parameter: OpenLegendWhenChangingSampleTimeDisplay

Type: string

Value: 'on' | 'off'

Default: 'on'

File generation control

Use these preferences to control the locations at which model build artifacts are placed. By default, build artifacts are placed in the current working folder (pwd) at the time update diagram or code generation is initiated. For more information, see these controls:

- “Simulation cache folder” on page 4-23
- “Code generation folder” on page 4-24

Simulation cache folder

Specify root folder in which to put model build artifacts used for simulation.

Settings

Default: ' '

Enter a string specifying a valid folder path. If no path is specified, build artifacts are placed in the current working folder (pwd) at the time update diagram is initiated.

Tip

You can specify an absolute or relative path to the folder. For example:

- `C:\Work\mymodelsimcache` and `/mywork/mymodelsimcache` specify absolute paths.
- `mymodelsimcache` is a path relative to the current working folder (pwd). The software converts a relative path to a fully qualified path at the time the preference is set. For example, if `pwd` is `'/mywork'`, the result is `/mywork/mymodelsimcache`.
- `../test/mymodelsimcache` is a path relative to `pwd`. If `pwd` is `'/mywork'`, the result is `/test/mymodelsimcache`.

Command-Line Information

Parameter: CacheFolder

Type: string

Value: valid folder path

Default: ' '

See Also

“Controlling Generated File Output Locations” in the Real-Time Workshop documentation

Code generation folder

Specify root folder in which to put Real-Time Workshop code generation files.

Settings

Default: ' '

Enter a string specifying a valid folder path. If no path is specified, build artifacts are placed in the current working folder (pwd) at the time code generation is initiated.

Tip

You can specify an absolute or relative path to the folder. For example:

- `C:\Work\mymodelgencode` and `/mywork/mymodelgencode` specify absolute paths.
- `mymodelgencode` is a path relative to the current working folder (pwd). The software converts a relative path to a fully qualified path at the time the preference is set. For example, if `pwd` is `'/mywork'`, the result is `/mywork/mymodelgencode`.
- `../test/mymodelgencode` is a path relative to `pwd`. If `pwd` is `'/mywork'`, the result is `/test/mymodelgencode`.

Command-Line Information

Parameter: CodeGenFolder

Type: string

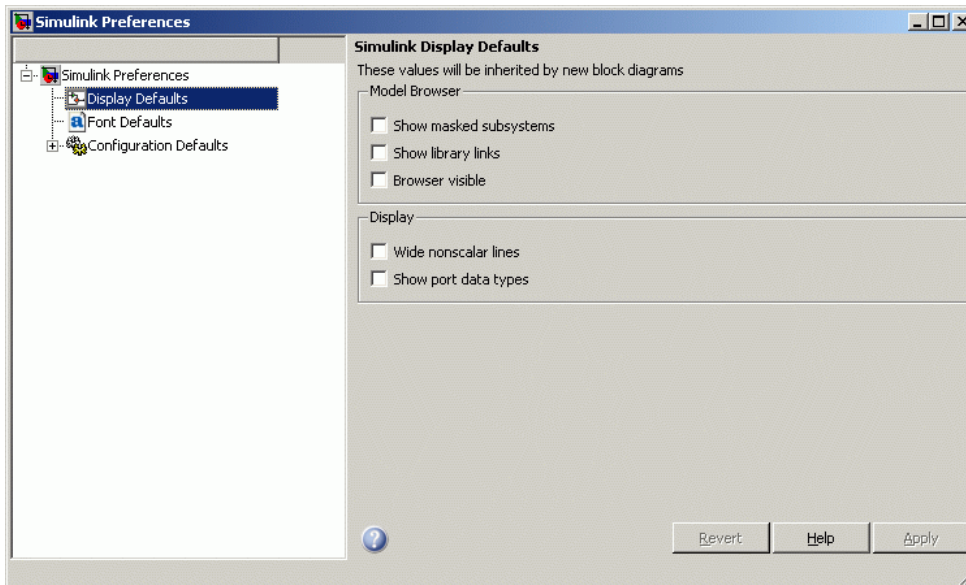
Value: valid folder path

Default: ' '

See Also

“Controlling Generated File Output Locations” in the Real-Time Workshop documentation

Simulink Preferences Window: Display Defaults Pane



In this section...

“Simulink Display Defaults Overview” on page 4-25

“Show masked subsystems” on page 4-27

“Show library links” on page 4-28

“Browser visible” on page 4-29

“Wide nonscalar lines” on page 4-30

“Show port data types” on page 4-31

Simulink Display Defaults Overview

Configure display options for the Model Browser, block connection lines and port data types.

Configuration

- 1 Select check boxes to configure display properties that will be applied to all new block diagrams.
- 2 Close the window to apply your changes.

Click **Apply** to apply your changes and keep the window open.

These values will be inherited by new block diagrams.

See Also

- “The Model Browser” (Windows only)
- Signal Display Options

Show masked subsystems

Specify whether masked subsystems and their contents are shown in the Model Browser (Windows only).

Settings

Default: Off



On

Display masked subsystems and their contents in the Model Browser.



Off

Do not display masked subsystems and their contents in the Model Browser.

Command-Line Information

Parameter: BrowserLookUnderMasks

Type: string

Value: 'on' | 'off'

Default: 'off'

Show library links

Specify whether library links and their contents are shown in the Model Browser (Windows only).

Settings

Default: Off



On

Display library links and their contents in the Model Browser.



Off

Do not display library links and their contents in the Model Browser.

Command-Line Information

Parameter: BrowserShowLibraryLinks

Type: string

Value: 'on' | 'off'

Default: 'off'

Browser visible

Specify whether the Model Browser (Windows only) is shown when you open a model.

Settings

Default: Off



On

Display the Model Browser.



Off

Do not display the Model Browser.

Command-Line Information

Parameter: ModelBrowserVisibility

Type: string

Value: 'on' | 'off'

Default: 'off'

Wide nonscalar lines

Specify whether to show thick lines for nonscalar connections between blocks.

Settings

Default: Off



On

Show thick lines for nonscalar connections between blocks



Off

Do not show thick lines for nonscalar connections between blocks

Command-Line Information

Parameter: WideVectorLines

Type: string

Value: 'on' | 'off'

Default: 'off'

Show port data types

Specify whether to show the data type on each block port

Settings

Default: Off



On

Display the data type for each port on each block.



Off

Do not display data types on block ports.

Command-Line Information

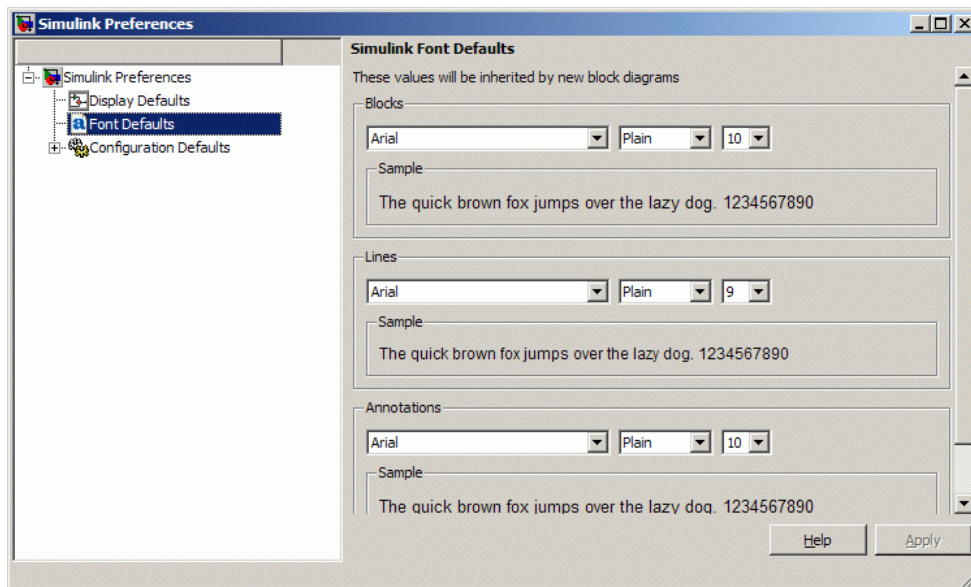
Parameter: ShowPortDataTypes

Type: string

Value: 'on' | 'off'

Default: 'off'

Simulink Preferences Window: Font Defaults Pane



Simulink Font Defaults Overview

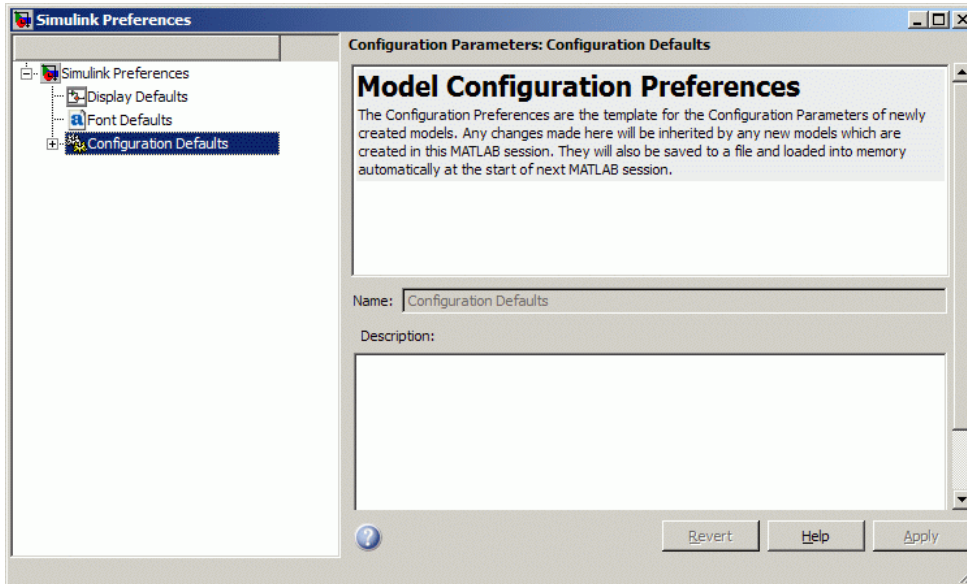
Configure font options for blocks, lines and annotations.

Configuration

- 1 Use the drop-down lists to specify font types, styles, and sizes that will be applied to all new block diagrams.
- 2 Close the window to apply your changes.
Click **Apply** to apply your changes and keep the window open.

These properties will be inherited by new block diagrams.

Simulink Preferences Window: Configuration Defaults Pane



Simulink Configuration Defaults Overview

On the main Configuration Defaults pane you can edit the description of your template configuration set.

Expand the tree under Configuration Defaults to edit the template Configuration Parameters to be used as defaults for new models.

Configuration

- 1 Expand the tree under Configuration Defaults to edit the template for default Configuration Parameters.
- 2 Edit Configuration Parameters that you want to apply to all new block diagrams.
- 3 Close the window to apply your changes.

Click **Apply** to apply your changes and keep the window open.

These values will be inherited by new block diagrams.

See Also

“Configuration Parameters Dialog Box Overview” on page 1-2

Simulink Mask Editor

- “Mask Editor Overview” on page 5-2
- “Icon & Ports Pane” on page 5-5
- “Parameters Pane” on page 5-12
- “Initialization Pane” on page 5-30
- “Documentation Pane” on page 5-33

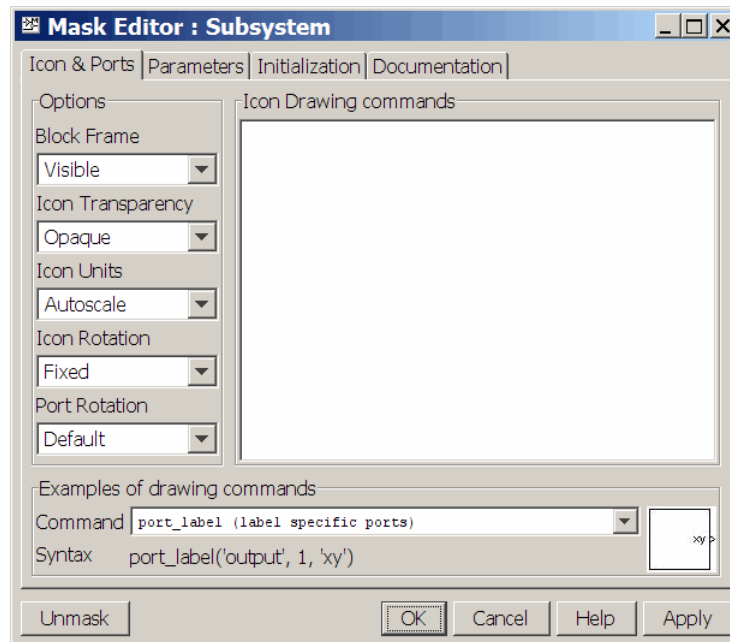
Mask Editor Overview

You can use the Simulink Mask Editor to define a custom interface for a Subsystem block, Model block, or S-Function block. This chapter provides reference information about Mask Editor capabilities, but does not describe the workflows needed to combine those capabilities to create and use masks. That information appears in “Working with Block Masks” in the Simulink User’s Guide.

To open the Mask Editor for a block, do one of the following:

- To create a new mask, select the block to be masked, then in the model window **Edit** menu or in the context menu for the block, choose **MaskBlockType**.
- To edit an existing mask, select the masked block and select **Edit > Edit Mask** or the in the context menu for the block, select **Edit Mask**.

The Mask Editor, looking similar to the figure below. If the block is already masked, the current mask specification appears in the editor, allowing you to change the mask as needed. Otherwise, the editor is initially empty:



The Mask Editor contains a set of tabbed panes, each of which enables you to define a feature of the mask:

- The **Icon & Ports** pane enables you to define the block icon. See “Icon & Ports Pane” on page 5-5.
- The **Parameters** pane enables you to define and describe Mask Parameters dialog box parameter prompts, and to name the variables associated with the parameters. See “Parameters Pane” on page 5-12.
- The **Initialization** pane enables you to specify initialization commands. See “Initialization Pane” on page 5-30.
- The **Documentation** pane enables you to define the mask type and specify the block description and the block help. See “Documentation Pane” on page 5-33.

Five buttons appear along the bottom of the Mask Editor:

- The **Unmask** button deactivates the mask and closes the Mask Editor. While the model is still active, the mask information is still retained so that you can reactivate it.

To reactivate the mask, select the block and choose **Mask Subsystem**. The Mask Editor opens, displaying the previous settings. Click **OK** or **Apply** to restore the mask.

When you close the model, the inactive mask information is discarded. If you want the mask information after this, you will need to recreate it the next time you open the model.

- The **OK** button applies the mask settings on all panes and closes the Mask Editor.
- The **Cancel** button closes the Mask Editor without applying any changes made since you last clicked the **Apply** button.
- The **Help** button displays online information about the Mask Editor.
- The **Apply** button applies the mask settings on all panes and leaves the Mask Editor open.

To see the system under the mask without unmasking it, select the Subsystem block, then select **Look Under Mask** from the **Edit** menu or the context menu of the block. This command opens the subsystem. The mask is not affected.

Icon & Ports Pane

In this section...

“About the Icon & Ports Pane” on page 5-5

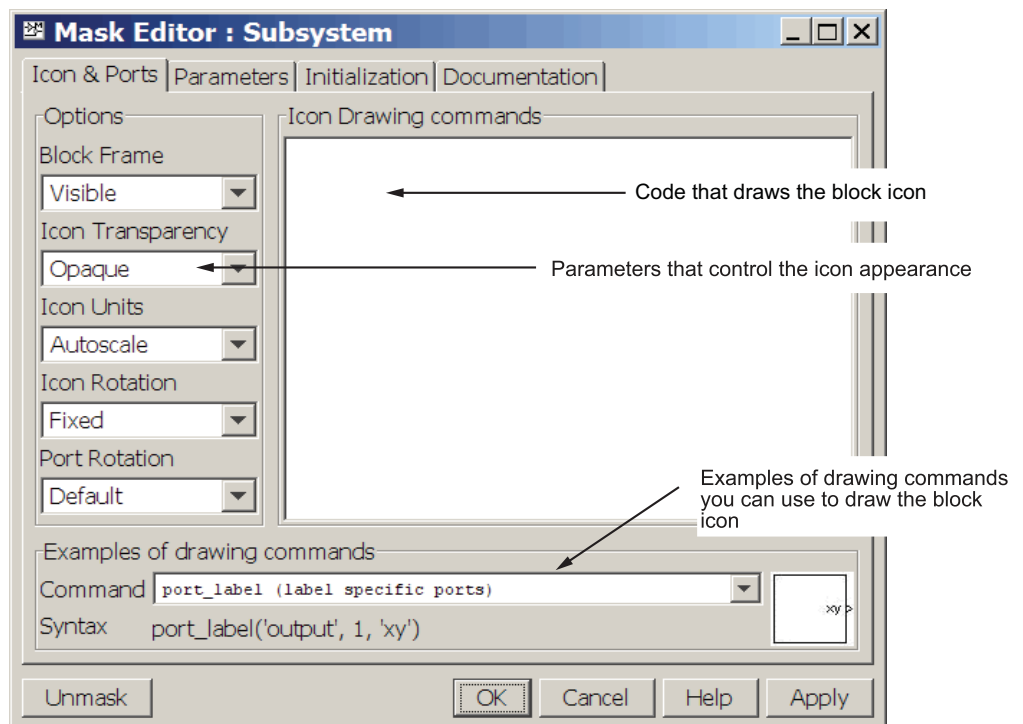
“Icon Drawing commands” on page 5-6

“Examples of drawing commands” on page 5-6

“Icon & Ports options” on page 5-7

About the Icon & Ports Pane

Use the **Icon & Ports** pane to create block icons that can contain descriptive text, state equations, images, and graphics.



The **Icon & Ports** pane contains the controls described in this section.

Icon Drawing commands

The Icon Drawing commands text box allows you to enter code that draw the block icon. Use commands from the set of commands to display text, display one or more plots, or show a transfer function, as described in “Mask Icon Drawing Commands”.

You can use only these commands to draw your icon. The drawing commands execute in the order in which they appear in this field. Drawing commands have access to all variables in the mask workspace. If any drawing command cannot successfully execute, the icon displays three question marks.

You can include MATLAB comments at the end of a drawing command or on their own line, as in the following examples:

```
image(imread('engine.jpg'))    % Use engine icon for subsystem

% Use engine icon for subsystem
image(imread('engine.jpg'))
```

The drawing commands execute when you:

- Load the model.
- Run or update the block diagram.
- Apply any changes made in the Mask Parameters dialog box, either by clicking **Apply** or **OK**.
- Apply any changes made in the Mask Editor, either by clicking **Apply** or **OK**.
- Make changes to the block diagram that affect the appearance of the block, such as rotating the block.
- Copy the masked block within the same model or between different models.

Examples of drawing commands

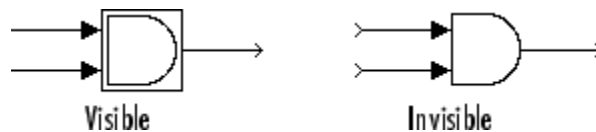
This pane demonstrates the usage of the various icon drawing commands supported by the Simulink product. To determine the syntax of a command, select the command from the **Command** list. An example of the selected command is displayed at the bottom of the pane and the icon produced by the command to the right of the list.

Icon & Ports options

These controls allow you to specify the following attributes of the block icon.

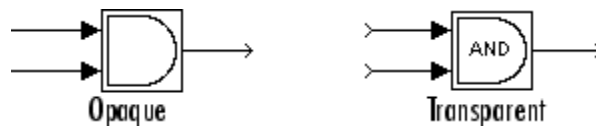
Block Frame

The icon frame is the rectangle that encloses the block. You can choose to show or hide the frame by setting the **Frame** parameter to **Visible** or **Invisible**. The default is to make the icon frame visible. For example, this figure shows visible and invisible icon frames for an AND gate block.



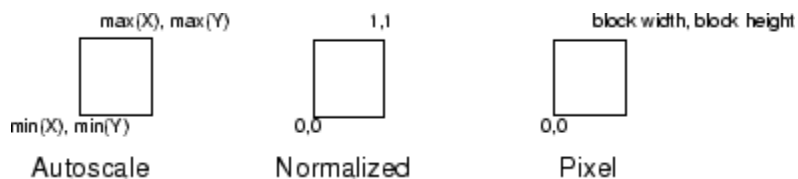
Icon Transparency

The icon can be set to **Opaque** or **Transparent**, either hiding or showing what is underneath the icon. **Opaque**, the default, covers information the Simulink software draws, such as port labels. This figure shows opaque and transparent icons for an AND gate block. Notice the text on the transparent icon.



Icon Units

This option controls the coordinate system used by the drawing commands. It applies only to plot and text drawing commands. You can select from among these choices: **Autoscale**, **Normalized**, and **Pixel**.



- **Autoscale** scales the icon to fit the block frame. When the block is resized, the icon is also resized. For example, this figure shows the icon drawn using these vectors:

$$X = [0 \ 2 \ 3 \ 4 \ 9]; \ Y = [4 \ 6 \ 3 \ 5 \ 8];$$



The lower-left corner of the block frame is (0,3) and the upper-right corner is (9,8). The range of the x -axis is 9 (from 0 to 9), while the range of the y -axis is 5 (from 3 to 8).

- **Normalized** draws the icon within a block frame whose bottom-left corner is (0,0) and whose top-right corner is (1,1). Only X and Y values between 0 and 1 appear. When the block is resized, the icon is also resized. For example, this figure shows the icon drawn using these vectors:

$$X = [.0 \ .2 \ .3 \ .4 \ .9]; \ Y = [.4 \ .6 \ .3 \ .5 \ .8];$$



- **Pixel** draws the icon with X and Y values expressed in pixels. The icon is not automatically resized when the block is resized. To force the icon to resize with the block, define the drawing commands in terms of the block size.

Icon Rotation

When the block is rotated or flipped, you can choose whether to rotate or flip the icon or to have it remain fixed in its original orientation. The default is not to rotate the icon. The icon rotation is consistent with block port rotation. This figure shows the results of choosing **Fixed** and **Rotates** icon rotation when the AND gate block is rotated.



Port Rotation

The **Icon & Ports > Port Rotation** port option lets you specify a masked block's port rotation type. The choices are:

- default

Ports are reordered after a clockwise rotation to maintain a left-to-right port numbering order for ports along the top and bottom of the block and a top-to-bottom port numbering order for ports along the left and right sides of the block.

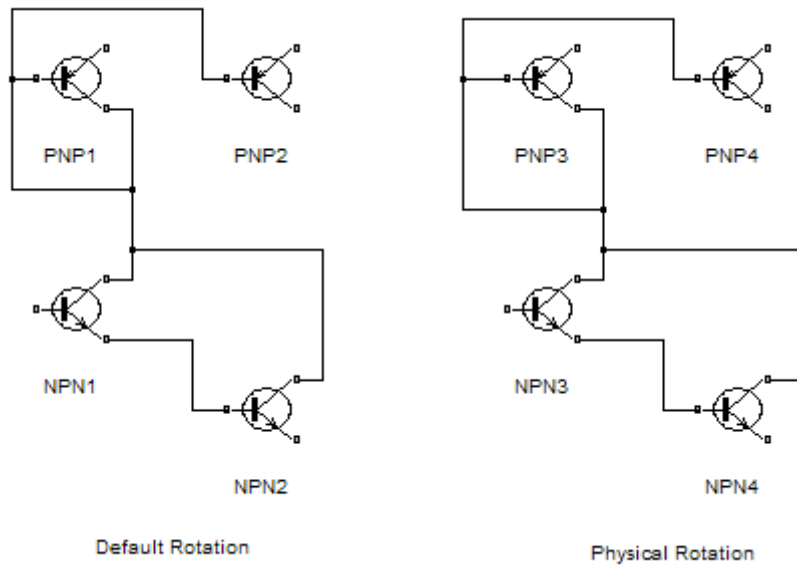
- physical

Ports rotate with the block without being reordered after a clockwise rotation.

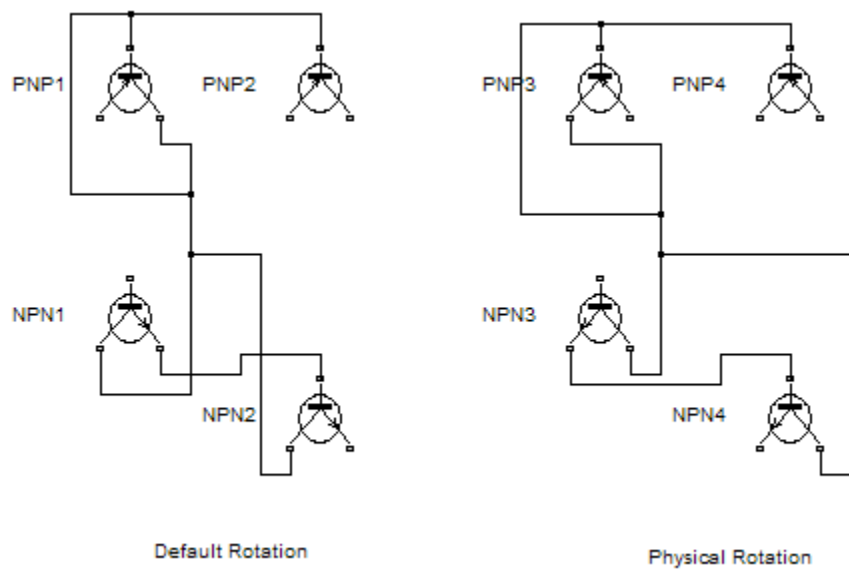
The default rotation option is appropriate for control systems and other modeling applications where block diagrams typically have a top-down and left-right orientation. It simplifies editing of diagrams by minimizing the need to reconnect blocks after rotations to preserve the standard orientation.

Similarly, the physical rotation option is appropriate for electronic, mechanical, hydraulic, and other modeling applications where blocks represent physical components and lines represent physical connections. The physical rotation option more closely models the behavior of the devices represented (that is, the ports rotate with the block as they would on a physical device). In addition, the option avoids introducing line crossings as the result of rotations, making diagrams easier to read.

For example, the following figure shows two diagrams representing the same transistor circuit. In one, the masked blocks representing transistors use default rotation; in the other, physical rotation.



Both diagrams avoid line crossings that make diagrams harder to read. The next figure shows the diagrams after a single clockwise rotation.



Note that the rotation introduces a line crossing in the diagram that uses default rotation but not in the diagram that uses physical rotation. Note also that there is no way to edit the diagram with default rotation to remove the line crossing. See “Changing a Block’s Orientation” for more information.

Parameters Pane

In this section...

“About the Parameters Pane” on page 5-12

“Dialog Parameters Pane” on page 5-13

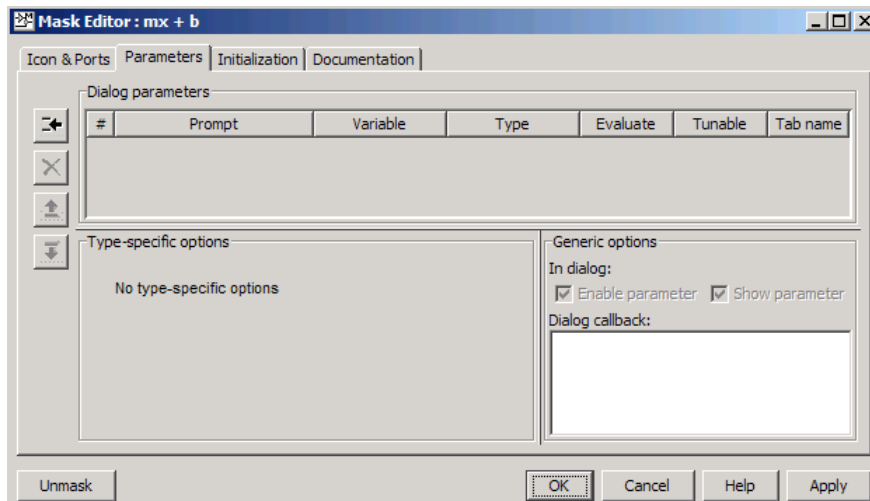
“Control Types” on page 5-16

“Generic Options Pane” on page 5-26

“Parameter Buttons” on page 5-28

About the Parameters Pane

The **Parameters** pane allows you to create and modify masked subsystem parameters (mask parameters, for short) that determine the behavior of the masked subsystem.



The **Parameters** pane contains the following elements:

- The **Dialog parameters** pane allows you to select and change the major properties of the mask parameters. See “Dialog Parameters Pane” on page 5-13 for more information.

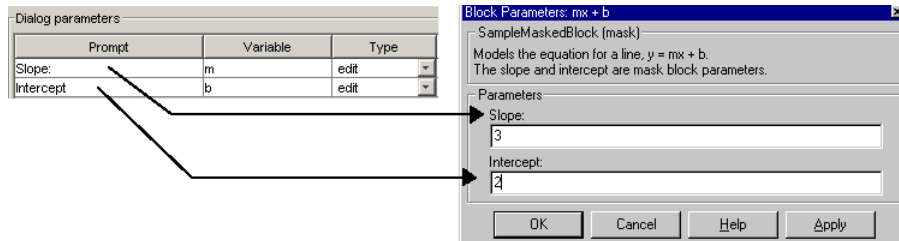
- The **Type-specific options** pane allows you to set additional options for some control types (the **Type** attribute in the **Dialog parameters** pane).
- The **Generic options** pane allows you to set additional options for the parameter.
- The buttons on the left side of the **Parameters** pane allow you to add, delete, and change the order of appearance of parameters on the Mask Parameters dialog box. See “Dialog Parameters Pane” on page 5-13 for more information.

Dialog Parameters Pane

Lists the mask parameters in tabular form. Each row displays the major attributes of one of the mask parameters.

Prompt

Text that identifies the parameter on a Mask Parameters dialog box.



Variable

Name of the variable that stores the parameter value in the mask workspace. You can use this variable as the value of parameters of blocks inside the masked subsystem, allowing the masked block user to set the parameters through the Mask Parameters dialog box.

Note The `set_param` and `get_param` commands are insensitive to case differences in mask variable names. For example, suppose a model named `MyModel` contains a masked subsystem named `A` that defines a mask variable named `Volume`. Then, the following line of code returns the value of the `Volume` parameter.

```
get_param('MyModel/A', 'voLUME')
```

However, case does matter when using a mask variable as the value of a block parameter inside the masked subsystem. For example, suppose a `Gain` block inside the masked subsystem `A` specifies `VOLUME` as the value of its `Gain` parameter. This variable name does not resolve in the masked subsystem's workspace, as it contains a mask variable named `Volume`. If the base workspace does not contain a variable named `VOLUME`, simulating `MyModel` produces an error.

The mask variable name set by the **Variable** field cannot duplicate any parameter name in or beneath the mask. Therefore, you cannot use any built-in block parameter name, such as `name`, as a mask variable name. Violating this requirement causes an error when you apply changes in the Mask Editor.

Type

Type of control that is used to specify the value of this parameter. Some examples of the types of controls you can specify include edit boxes, check boxes, and data type controls. See “Control Types” on page 5-16 for details about the control types you can specify.

The control appears on the Mask Parameters dialog box following the parameter prompt.

Evaluate

Simulink can use the value of a mask parameter as the user enters it in the Mask Parameters dialog box, or it can evaluate what the user specifies and use the result of the evaluation. Select the **Evaluate** option for a parameter to specify parameter evaluation (the default) or clear the option to suppress evaluation.

If checked, the **Evaluate** option causes Simulink to evaluate the expression that the user enters, before assigning that value to the variable. Otherwise, Simulink treats the expression as a string value and assigns that string to the variable. For example, if you check **Evaluate** option, then if the user enters the expression `gain` in an edit field, Simulink evaluates `gain` and assigns the result to the variable. Otherwise, Simulink assigns the string `'gain'` to the variable. The effect of the **Evaluate** option depends on the control type, as shown in “Control Types” on page 5-16.

If you need both the string entered *and* the evaluated value, clear the **Evaluate** option. To get the value of a base workspace variable entered as the literal value of the mask parameter, use the MATLAB `evalin` command in the mask initialization code. For example, suppose the masked block user enters the string `'gain'` as the literal value of the mask parameter `k`, where `gain` is the name of a base workspace variable. To obtain the value of the base workspace variable, use the following command in the mask initialization code:

```
value = evalin('base', k)
```

Tunable

By default, the masked block user can change a Mask Parameters dialog box parameter value during simulation. Clear the **Tunable** option to prohibit the changing of the parameter value during simulation.

If the masked block does not support parameter tuning, Simulink ignores the **Tunable** option setting of a mask parameter. See “Using Tunable Parameters” for information about parameter tuning and the blocks that support it.

Tab name

The name of the tab that contains a parameter.

To group multiple parameters in one tab, use the same tab name for each of the parameters that you want to be on a tab.

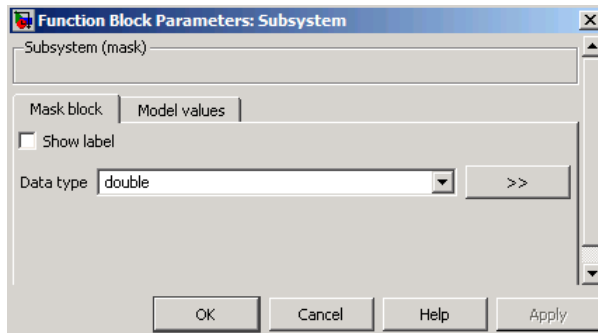
If you specify a tab name for one parameter, then any other parameters for which you do not specify a tab name appear in the Mask Parameters dialog box under a blank tab.

The parameters on a tab appear in the order in which the parameters appear in the **Dialog parameters** table. You do not have to group all the parameters for a tab together in the **Dialog parameters** table.

For example, you can specify two tabs for parameters:

#	Prompt	Variable	Type	Evaluate	Tunable	Tab name
1	Show label	label	checkbox	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Mask block
2	Data type	outdatatype	DataTypeStr	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Mask block
3	Slope	m	edit	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Model values
4	Intercept	b	edit	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Model values

That definition of the two tabs produces the following Mask Parameters dialog box:

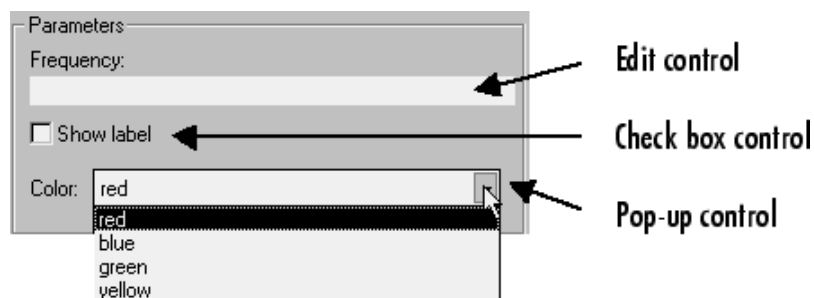


Control Types

You can choose how a masked block user selects or enters parameter values. To specify the type of control that the Mask Parameters dialog box provides for a mask parameter, set the **Type** parameter to one of the following options.

Control	Masked Block User Interaction
edit	Enters a parameter value by typing it into a field
checkbox	Chooses between two alternatives by selecting or deselecting a check box
popup	Selects a parameter value from a list of possible values
DataTypeStr	Enters a data type or selects one from a list of data types, and optionally opens the Data Type Assistant
Minimum	Specifies a minimum value
Maximum	Specifies a maximum value

For example, this figure shows the **Parameters** pane of a Mask Parameters dialog box that uses three types of controls, with the pop-up control open to show its possible values:



Edit Control

An *edit control* enables the user to enter a parameter value by typing it into a field. This figure highlights the definition of the **Slope** prompt, which uses an edit control.

Dialog parameters							
#	Prompt	Variable	Type	Evaluate	Tunable	Tab name	
1	Slope	Slope	edit	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		
2	Intercept	b	edit	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		

The **Evaluate** option determines whether Simulink evaluates the value that the user enters for this parameter.

Edit Control Evaluation

Evaluation	Edit Control Parameter Value
On	The result of evaluating the expression entered in the field
Off	The literal string entered in the field

Check Box Control

A *check box* enables the user to choose between two alternatives by selecting or clearing a check box. This figure shows the definition of the **Saturate** check box control.

#	Prompt	Variable	Type	Evaluate	Tunable	Tab name
1	Saturate	sat	checkbox	<input checked="" type="checkbox"/>	<input type="checkbox"/>	

The **Evaluate** option determines whether Simulink evaluates the value that the user enters for this parameter.

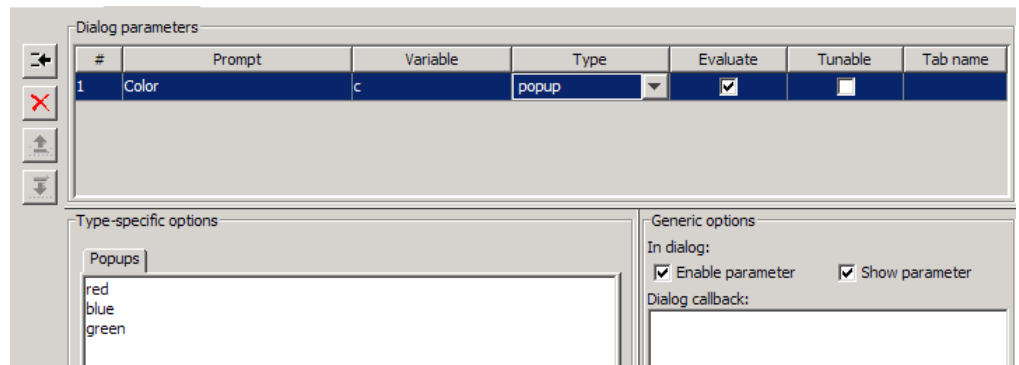
Check Box Evaluation

Evaluation	Check Box Parameter State	Check Box Parameter Value
On	Selected	1
	Cleared	0
Off	Selected	'on'
	Cleared	'off'

Pop-Up Control

A *pop-up* control enables the user to choose a parameter value from a list of possible values. When you define a pop-up control, specify the values that the

control presents to the user in the Mask Parameters dialog box. To specify the values, select the parameter in the **Dialog parameters** table, then enter the values in the **Popups** field of the **Type-specific options** below the table. The next figure shows the pop-up control definition for the **Color** prompt.



The value of the variable associated with the parameter (**Color**) depends on the item selected from the pop-up list and whether you select the **Evaluate** option.

Pop-Up Evaluation

Evaluation	Pop-Up Parameter Value
On	The index of the value selected from the list, starting with 1. For example, if the third item is selected, the parameter value is 3.
Off	The literal string that is the value selected, as it appears in the Mask Editor Popups field and the corresponding Mask Parameters dialog box list.

Data Type Control

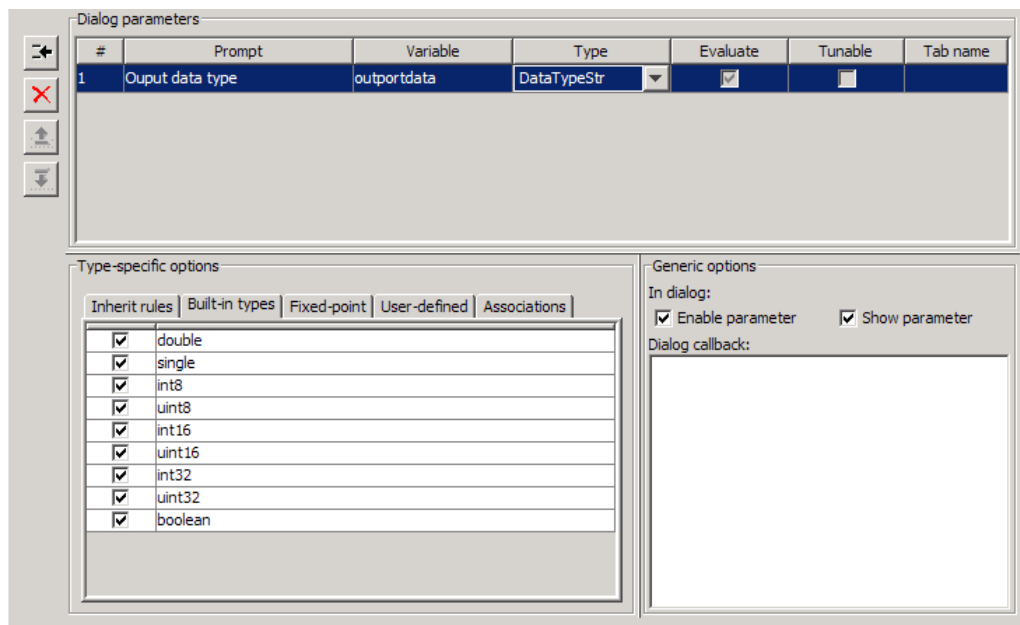
A *data type* control enables the masked block user to specify a data type for a mask parameter. The data type control provides the masked block user with a drop-down list of data types, and a button to open the Data Type Assistant. For details about the Data Type Assistant, see “Working with Data Types”.

A data type control is particularly useful when you include a masked block in a user-defined library. For information about using masked blocks in user-defined libraries, see .

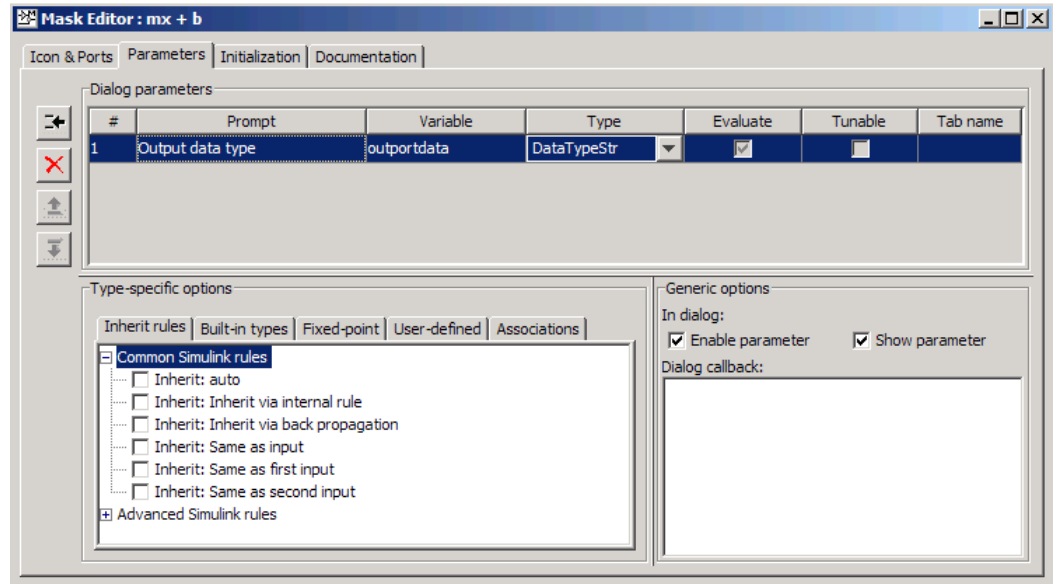
When you define a data type control, specify the data type or data types that the control presents to the user in the Mask Parameters dialog box. To specify the data type options from which a user can choose, in the **Dialog parameters** table, select the parameter, and then use one or more of the following tabs in the **Type-specific options** pane:

- **Inherit rules** — Specify inheritance rules for determining the data types.
- **Built-in types** — Specify one or more built-in Simulink data types, such as `double` or `int8`.
- **Fixed-point** — Specify the scaling and signedness modes for a fixed-point data type.
- **User-defined** — Specify a bus or enumerated (enum) data type, or both.

The next figure shows a data type control definition for an **Output Data Type** prompt that allows the masked block user to select any built-in type. (To restrict the user choices to only built-in data types, do not select any check boxes in the **Fixed-point** and **User-defined** tabs.)



Specifying Inheritance Rules. To specify one or more inheritance rules for the data type control, on the **Inherit rules** tab, select the appropriate check boxes.



By default, the **Inherit rules** tab includes two groups of rules:

- Common Simulink rules
- Advanced Simulink rules

The **Common Simulink rules** are inheritance rules that apply to many blocks in the Simulink library. The **Advanced Simulink rules** are inheritance rules that apply to one or only a few Simulink blocks.

If there are any custom inheritance rules registered on the MATLAB search path, then the **Inherit rules** tab also includes a third group of rules: **Custom Simulink rules**.

To register custom inheritance rules, use the `sl_customization` function:

- 1 Navigate to the MATLAB search path.
- 2 Create an `sl_customization.m` file, if that file does not already exist.
- 3 At the MATLAB command line, enter:

```
function sl_customization(cm)
```

4 In the `sl_customization` function definition, use an `InheritRuleCustomizer.registerCustomRules()` method, specifying a cell array of strings, with each string defining an inheritance rule to be registered.

- Each inheritance rule name must:
- Begin with `Inherit:` (for readability, include a space after `Inherit:`)
- Not include any of these characters:
 - |
 - {
 - }
 - ,
- The following is an example of an `sl_customization` function that registers two custom inheritance rules:

```
function sl_customization(cm)
    cm.InheritRuleCustomizer.registerCustomRules(
        {'Inherit: Custom rule 1',
         'Inherit: Custom rule 2'});
end
```

where `Custom rule 1` and `Custom rule 2` are the names of the custom inheritance rules that you want to appear in the Mask Editor interface.

- For more information about using the `sl_customization` function, see “About Registering User Interface Customizations”.
- 5** Use the **Initialization** pane of the Mask Editor to add mask initialization code that specifies how Simulink configures blocks in the mask when the user of the masked block specifies either `Custom rule 1` or `Custom rule 2`.
- 6** To display the custom inheritance rules group in the **Inherit rules** tab of the Mask Editor, do one the following steps:
- a** At the MATLAB command line, type `sl_refresh_customization`.
 - b** Restart MATLAB.

Specifying a Fixed-Point Data Type. To specify a fixed-point data type for a parameter:

- 1** Select the parameter in the **Dialog parameters** table.
- 2** In the **Type-specific options** pane, select the **Fixed-point** tab.
- 3** Select the appropriate scaling and signedness mode check boxes. If you do not select a mode, then the masked block user cannot choose a fixed-point data type.
- 4** Also in the **Type-specific options** pane, select the **Associations** tab.

The masked block user can use the association when specifying a fixed-point data type with the Data Type Assistant. For a value or value range for a signal, the association can help the user select the data type with the best precision for the value or the value range.

- 5** Specify the minimum, maximum, and value for the fixed-point data.

Specifying an Enumerated Data Type. To specify an enumerated (enum) data type:

- 1** Select the parameter in the **Dialog parameters** table.
- 2** In the **Type-specific options** pane, select the **User-defined** tab.
- 3** Select the **Enumerated** check box.

Specifying a Bus Data Type. To specify a bus data type:

- 1** Select the parameter in the **Dialog parameters** table.
- 2** In the **Type-specific options** pane, select the **User-defined** tab.
- 3** Select the **Bus** check box.

If you specify a bus data type as one of the data types that a masked block user can specify in the Mask Parameters dialog box, then you must add code in the **Initialization** pane. Add code to handle the way that the **Data type** parameter of an underlying block specifies the data type. See

“Initialization Pane” on page 5-30 for more information about adding code to the **Initialization** pane.

To handle cases where the **Data type** parameter of an underlying block specifies the data type by including the mask parameter variable as a literal (for example, 'outportdatatype'), add code similar to this example:

```
maskDTPrmString = get_param(gcb, MaskDTPrmName);
if is_a_bus_type(MaskDTPrmString)
    blockDTPrmString = get_param(BlockUnderMask, BlockDTPrmName);
    set_param(BlockUnderMask, BlockDTPrmName, ...
        ['Bus: ' blockDTPrmString]);
end
```

To handle cases where the **Data type** parameter of an underlying block specifies the data type by including the mask parameter variable in the Bus: <object> form, add code similar to this example:

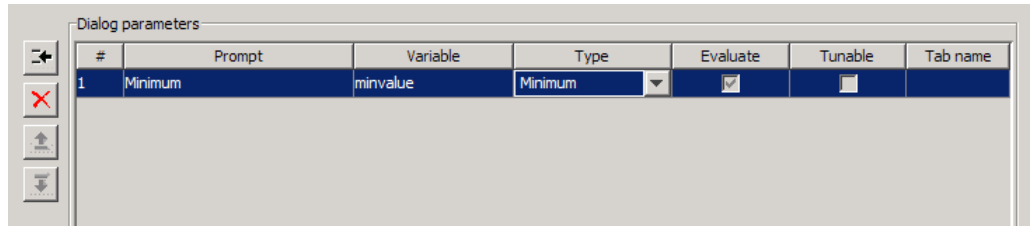
```
maskDTPrmString = get_param(gcb, MaskDTPrmName);
if ~is_a_bus_type(maskDTPrmString)
    blockDTPrmString = get_param(BlockUnderMask, BlockDTPrmName);
    set_param(BlockUnderMask, BlockDTPrmName, ...
        remove_bus_colon_prefix(blockDTPrmString));
end
```

Data Type Evaluation. Simulink enables the **Evaluate** option for data type controls. You cannot change this setting.

Minimum and Maximum Controls

A *minimum* or *maximum* control enables the masked block user to enter a value for the Minimum or Maximum parameters of an underlying block.

This figure highlights the definition of a **Minimum** prompt, which uses the Minimum control.



Minimum and Maximum Control Evaluation. Simulink enables the **Evaluate** option for Minimum and Maximum controls. You cannot change this setting.

Generic Options Pane

This pane allows you to set additional options for the parameter selected in the **Dialog parameters** table.

Enable parameter

Clearing this option grays the selected parameter prompt and disables its edit control. This means that the masked block user cannot set the value of the parameter.

Show parameter

The selected parameter appears on the masked block's parameter dialog box only if this option is checked (the default).

Dialog callback

Enter the MATLAB code that you want Simulink to execute when a masked block user applies a change to the selected parameter by selecting the **Apply** or **OK** button on the Mask Parameters dialog box. You can use such callbacks to create dialog boxes that change in appearance depending on the changes the user makes to control settings,, such as enabling an edit field when a user selects a check box.

Note Do not use callbacks to alter the contents of a masked subsystem. Altering contents in a callback, for example, by adding or deleting blocks or changing block parameter values, can trigger errors during model update or simulation. If you need to modify the contents of a masked subsystem, use the mask's initialization code to perform the modifications. See “Initialization Pane” on page 5-30 for more information.

The callback can create and reference variables only in the mask workspace. If the callback needs the value of a mask parameter, it can use `get_param` to obtain the value, for example:

```
if str2num(get_param(gcb, 'g'))<0
    error('Gain is negative.')
end
```

Simulink executes the callback commands when you:

- Open the mask parameter dialog box. Callback commands execute top down, starting with the top mask dialog parameter.
- Modify a parameter value in the mask parameter dialog box, then change the cursor focus by pressing the **Tab** key or clicking into another field in the dialog box.





Note The callback commands do not execute when the `set_param` command modifies the parameter value.

- Modify the parameter value, either in the Mask Parameters dialog box or through a call to `set_param`, then apply the change by clicking **Apply** or **OK**. Any mask initialization commands execute after the callback commands. See “Initialization Pane” on page 5-30 for more information.
- Cancel any changes that the user has made and applied in the Mask Parameters dialog box by clicking **Cancel**.
- Hover over the masked block to see its data tip, when the data tip contains parameter names and values. The callback executes again when the block data tip disappears.

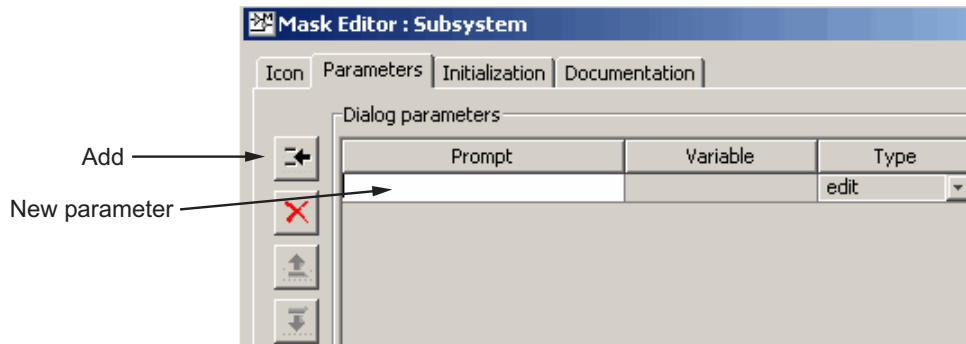
Note The callback commands do not execute if the Mask Parameters dialog box is open when the block data tip appears.

Parameter Buttons

This table explains the purpose of the buttons that appear on the **Parameters** pane.

Button	Action
 Add	Add a new mask parameter. An empty parameter appears in the adjacent Dialog parameters table. If you select an parameter, the new parameter appears below the selected parameter.
 Delete	Delete the parameter that is currently selected in the Dialog parameters table.
 Up	Move the currently selected parameter up one row in the Dialog parameters table. The prompt for this parameter moves up one slot in the Mask Parameters dialog box.
 Down	Move the currently selected parameter down one row in the Dialog parameters table. The prompt for this parameter moves down one slot in the Mask Parameters dialog box.

This figure shows the result of clicking the **Add** button to create the first parameter for a mask:



Initialization Pane

In this section...

“About the Initialization Pane” on page 5-30

“Dialog variables” on page 5-31

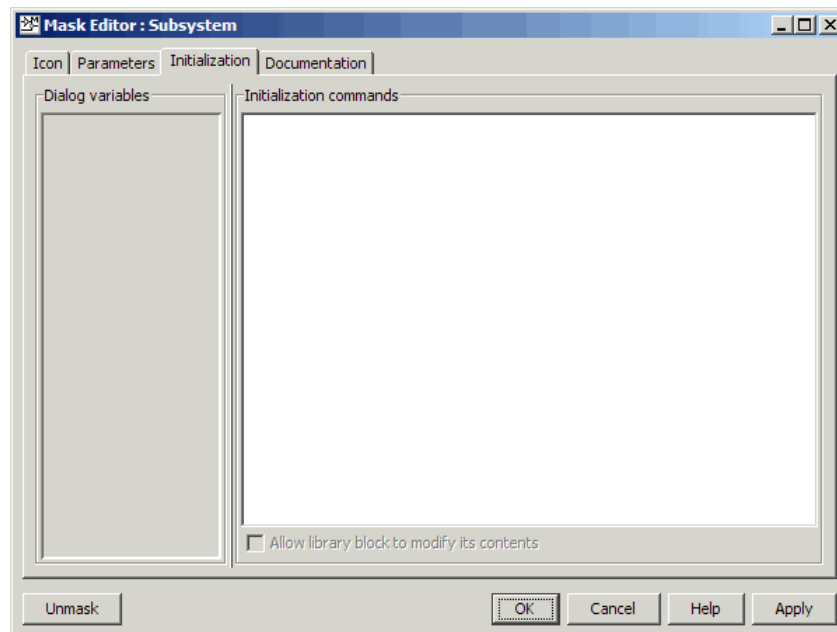
“Initialization commands” on page 5-32

“Allow library block to modify its contents” on page 5-32

“Initialization Command Limitations” on page 5-32

About the Initialization Pane

The **Initialization** pane allows you to enter MATLAB commands that initialize the masked subsystem.



When you open a model, initialization commands execute for all masked blocks that are visible because they reside at the top level of the model or in

an open subsystem. Initialization commands for masked blocks that are not initially visible execute when you open the subsystem or model that contains the blocks.

When you load a model into memory without displaying the model graphically, no initialization commands initially run for any masked blocks. See “Loading a Model” and `load_system` for information about loading a model without displaying it.

Initialization commands for all masked blocks in a model run when you:

- Update the diagram.
- Start simulation.
- Start code generation.

Initialization commands for an individual masked block run when you:

- Change any of the parameters that define the mask, such as `MaskDisplay` and `MaskInitialization`, by using the Mask Editor or `set_param`.
- Change the appearance of the masked block, for example by resizing or rotating the block.
- Change the value of a mask parameter by using the block dialog or `set_param`.
- Copy the masked block within the same model or between different models.

The **Initialization** pane contains the controls described in this section.

Dialog variables

The **Dialog variables** list displays the names of the variables associated with the subsystem’s mask parameters, which are defined in the **Parameters** pane. You can copy the name of a parameter from this list and paste it into the adjacent **Initialization commands** field, using the Simulink keyboard copy and paste commands. You can also use the list to change the names of mask parameter variables. To change a name, double-click the name in the list. An edit field containing the existing name appears. Edit the existing name and click **Enter** or click outside the edit field to confirm your changes.

Initialization commands

Enter the initialization commands in this field. You can enter any valid MATLAB expression, consisting of MATLAB functions and scripts, operators, and variables defined in the mask workspace. Initialization commands run in the mask workspace, not the base workspace. Terminate initialization commands with a semicolon to avoid echoing results to the MATLAB Command Window.

Allow library block to modify its contents

This check box is enabled only if the masked subsystem resides in a library. Checking this option allows the block's initialization code to modify the contents of the masked subsystem by adding or deleting blocks and setting the parameters of those blocks. Otherwise, an error is generated when a masked library block tries to modify its contents in any way. To set this option at the MATLAB prompt, select the self-modifying block and enter the following command.

```
set_param(gcb, 'MaskSelfModifiable', 'on');
```

Then save the block.

Initialization Command Limitations

Mask initialization commands must observe the following rules:

- Do not use initialization code to create mask dialogs whose appearance or control settings change depending on changes made to other control settings. Instead, use the mask callbacks provided specifically for this purpose.
- Avoid prefacing variable names in initialization commands with `L_` and `M_` to prevent undesirable results. These specific prefixes are reserved for use with internal variable names.
- Avoid using `set_param` commands to set parameters of blocks residing in masked subsystems that reside in the masked subsystem being initialized. See for details.

Documentation Pane

In this section...

“About the Documentation Pane” on page 5-33

“Mask Type Field” on page 5-34

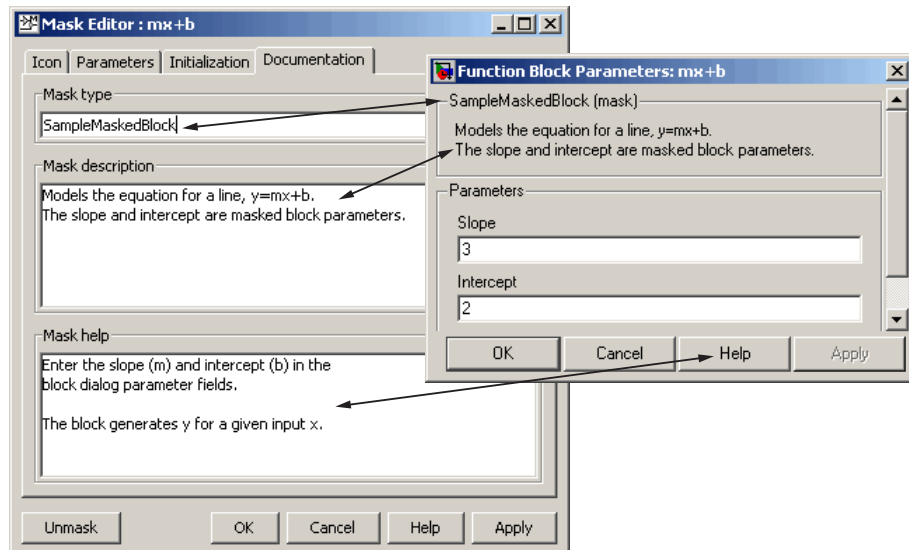
“Mask Description Field” on page 5-34

“Mask Help Field” on page 5-34

About the Documentation Pane

The **Documentation** pane enables you to define or modify the type, description, and help text for a masked block.

The next figure shows the relationship between the values in the **Documentation** pane and the appearance of the corresponding Mask Parameters dialog box and **Online Help** window.



Mask Type Field

The *mask type* is a block classification that appears in the masked block's Mask Parameters dialog box and on all Mask Editor panes for the block. When Simulink displays a masked block's Mask Parameters dialog box, it suffixes (mask) to the mask type. To define the mask type, enter it in the **Mask type** field. The text can contain any legal MATLAB character, but cannot contain line breaks.

Mask Description Field

The *mask description* is summary Help text that describe the block's purpose or function. The description appears in the masked block's Mask Parameters dialog box under the mask type. To define the mask description, enter it in the **Mask description** field. The text can contain any legal MATLAB character. Simulink automatically wraps long lines. You can force line breaks by using the **Enter** or **Return** key.

Mask Help Field

The Online Help for a masked block provides information in addition to that provided by the **Mask type** and **Mask description** fields of the block. This information appears in a separate window when the masked block user clicks the **Help** button on the masked block's Mask Parameters dialog box. To define the mask help, enter one of the following in the **Mask help** field.

- URL specification
- web or eval command
- Literal or HTML text

Providing a URL

If the first line of the **Mask help** field is a URL, Simulink passes the URL to your default web browser. The URL can begin with `http:`, `www:`, `file:`, `ftp:`, or `mailto:`. Examples:

```
http://www.mathworks.com
file:///c:/mydir/helpdoc.html
```


Once the browser is active, MATLAB and Simulink have no further control over its actions.

Providing a web Command

If the first line of the **Mask help** field is a **web** command, Simulink passes the command to MATLAB, which displays the specified file in the MATLAB Web Browser or the MATLAB Online Help browser. Example:

```
web([docroot '/MyBlockDoc/' get_param(gcb,'MaskType') '.html'])
web('hello.html', '-helpbrowser')
```

See the MATLAB **web** command documentation for details. A **web** command used for mask help cannot return values.

Providing an eval Command

If the first line of the **Mask help** field is an **eval** command, Simulink passes the command to MATLAB, which performs the specified evaluation. Example:

```
eval('!Word My_Spec.doc')
```

See the MATLAB **eval** command documentation for details. An **eval** command used for mask help cannot return values.

Providing Literal or HTML Text

If the first line of the **Mask help** field is not a URL, or a **web** or **eval** command, Simulink displays the text in the MATLAB Online Help browser under a heading that is the value of the **Mask type** field. The text can contain any legal MATLAB character, line breaks, and any standard HTML tag, including tags like **img** that display images.

Simulink first copies the text to a temporary folder, then displays the text using the **web** command. If you want the text to display an image, you can provide a URL path to the image file, or you can place the image file in the temporary folder. Use **tempdir** to find the temporary folder that Simulink uses for your system.