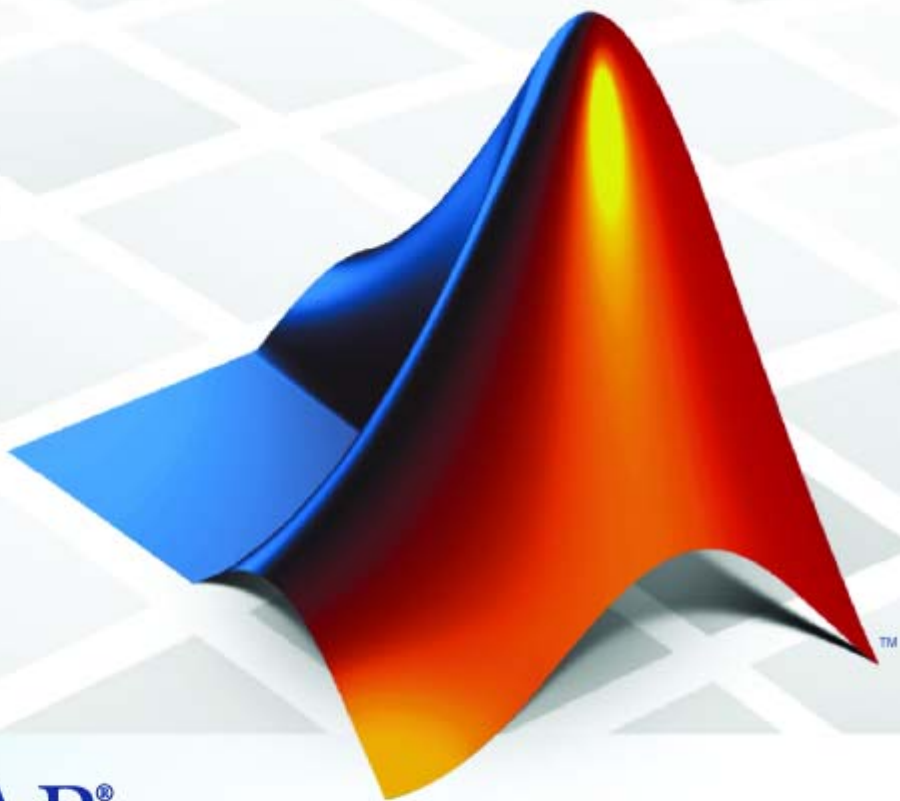


# Target Support Package™ IC1 1

## User's Guide



**MATLAB®**  
& **SIMULINK®**

## How to Contact The MathWorks



[www.mathworks.com](http://www.mathworks.com)  
[comp.soft-sys.matlab](mailto:comp.soft-sys.matlab)  
[www.mathworks.com/contact\\_TS.html](http://www.mathworks.com/contact_TS.html)

Web  
Newsgroup  
Technical Support



[suggest@mathworks.com](mailto:suggest@mathworks.com)  
[bugs@mathworks.com](mailto:bugs@mathworks.com)  
[doc@mathworks.com](mailto:doc@mathworks.com)  
[service@mathworks.com](mailto:service@mathworks.com)  
[info@mathworks.com](mailto:info@mathworks.com)

Product enhancement suggestions  
Bug reports  
Documentation error reports  
Order status, license renewals, passcodes  
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.

*Target Support Package™ IC1 User's Guide*

© COPYRIGHT 2002–2009 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](http://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

The MathWorks products are protected by one or more U.S. patents. Please see [www.mathworks.com/patents](http://www.mathworks.com/patents) for more information.

**Revision History**

November 2002 Online only  
June 2004 Online only  
October 2004 Online only  
March 2005 Online only  
September 2005 Online only  
September 2006 Online only  
March 2007 Online only  
September 2007 Online only  
March 2008 Online only  
October 2008 Online only  
March 2009 Online only

Version 1.0 (Release 13+)  
Version 1.1 (Release 14)  
Version 1.1.1 (Release 14SP1)  
Version 1.2 (Release 14SP2)  
Version 1.2.1 (Release 14SP3)  
Version 1.3 (Release 2006b)  
Version 1.4 (Release 2007a)  
Revised for Version 1.5 (Release 2007b)  
Revised for Version 1.5.1 (Release 2008a)  
Revised for Version 1.5.2 (Release 2008b)  
Revised for Version 1.5.3 (Release 2009a)



## Getting Started

# 1

<b>Product Overview</b> .....	1-2
Introduction .....	1-2
Feature Summary .....	1-2
<b>Prerequisites</b> .....	1-4
<b>Using This Guide</b> .....	1-5
<b>Installation</b> .....	1-6
<b>Hardware and Software Requirements</b> .....	1-7
Host Platform .....	1-7
Hardware Requirements .....	1-7
Software Requirements .....	1-8
Switching Between Hardware Variants .....	1-9
<b>Setting Up and Verifying Your Installation</b> .....	1-13
Setting Up Software .....	1-13
Verifying MiniMon Settings .....	1-14
<b>Setting Up Your Target Hardware</b> .....	1-18
Jumper Settings for the phyCore-167 Development Board .....	1-18
Setting Up XC164CM Hardware .....	1-18
Jumper Settings for the STMicroelectronics MB449 ST10F25x EVA Board .....	1-19
<b>Setting Target Preferences</b> .....	1-21
<b>Code Generation Configuration for Nondefault Processors</b> .....	1-23

<b>Supported Blocks and Data Types</b> .....	<b>1-27</b>
<b>Overview of C166 Options in the Configuration</b>	
<b>Parameters Dialog Box</b> .....	<b>1-30</b>

## **Tutorial: Simple Example Applications for C166 Microcontrollers**

### **2**

<b>Introduction</b> .....	<b>2-2</b>
<b>Tutorial: Creating a New Application</b> .....	<b>2-3</b>
Tutorial Overview .....	<b>2-3</b>
Before You Begin .....	<b>2-3</b>
Example Model 1: c166_serial_transmit .....	<b>2-4</b>
Generating and Downloading Code .....	<b>2-7</b>
Example 2: c166_serial_io .....	<b>2-9</b>
<b>Debugging and Using The Code Profile Report</b> .....	<b>2-12</b>
Starting the Debugger on Completion of the Build	
Process .....	<b>2-12</b>
RAM / ROM Code Profile Report .....	<b>2-14</b>
<b>Parameter Tuning and Signal Logging</b> .....	<b>2-18</b>
Methods For Parameter Tuning and Signal Logging .....	<b>2-18</b>
Using External Mode .....	<b>2-18</b>
Using a Third Party Calibration Tool .....	<b>2-27</b>

## **Integrating Your Own Device Drivers**

### **3**

<b>Integrating Hand-Coded Device Drivers with a Simulink Model</b> .....	<b>3-2</b>
--	------------

<b>Preparing Input and Output Signals to the Device</b>	
<b>Driver Functions</b> .....	3-4
<b>Calling the Device Driver Functions from</b>	
<b>c166_main.c</b> .....	3-7
<b>Adding the I/O Driver Source to the List of Files to</b>	
<b>Build</b> .....	3-10
<b>Tutorial: Using the Example Driver Functions</b> .....	3-12

## **Custom Storage Class for C166 Microcontroller Bit-Addressable Memory**

### **4**

<b>Specifying C166 Microcontroller Bit-Addressable</b>	
<b>Memory</b> .....	4-2
<b>Using the Bitfield Example Model</b> .....	4-3

## **Execution Profiling**

### **5**

<b>Overview of Execution Profiling</b> .....	5-2
Introducing Execution Profiling .....	5-2
The Profiling Command .....	5-3
Definitions .....	5-5
Execution Profiling Blocks .....	5-5
<b>Real-Time Workshop Options for Execution</b>	
<b>Profiling</b> .....	5-6
Execution Profiling .....	5-6
Number of Data Points .....	5-7
Task Scheduler Overrun Options .....	5-7

<b>Multitasking Demo Model</b> .....	<b>5-10</b>
Introducing the Multitasking Demo .....	<b>5-10</b>
Running the Multitasking Demo .....	<b>5-11</b>
Interpreting the MATLAB Graphic .....	<b>5-13</b>
The Generated HTML Report .....	<b>5-14</b>

## Block Reference

### 6

<b>C166 Drivers</b> .....	<b>6-2</b>
Top-Level Blocks .....	<b>6-2</b>
Asynchronous/Synchronous Serial Interface .....	<b>6-2</b>
CAN Interface .....	<b>6-3</b>
C-CAN Interface .....	<b>6-3</b>
Execution Profiling .....	<b>6-4</b>
TwinCAN Interface .....	<b>6-4</b>
Interrupts .....	<b>6-5</b>
Utilities .....	<b>6-5</b>
Digital Input/Output .....	<b>6-5</b>
<b>CAN Message Blocks and CAN Drivers</b> .....	<b>6-6</b>

## Blocks — Alphabetical List

### 7

## Configuration Parameters

### 8

<b>Real-Time Workshop Pane: C166 Options</b> .....	<b>8-2</b>
C166 Options Tab Overview .....	<b>8-3</b>
Include input/output driver function hooks .....	<b>8-4</b>
Maximum number of concurrent base-rate overruns: .....	<b>8-5</b>
Maximum number of concurrent sub-rate overruns: .....	<b>8-7</b>
Execution profiling .....	<b>8-9</b>



Number of data points: ..... 8-10

## Examples

### A

Simple Example Applications .....	A-2
Real-Time Target .....	A-2
Integrating Hand-Coded Device Drivers .....	A-2
Bit-Addressable Memory .....	A-2
Execution Profiling .....	A-2

## Index



# Getting Started

---

This section contains the following topics:

- “Product Overview” on page 1-2
- “Prerequisites” on page 1-4
- “Using This Guide” on page 1-5
- “Installation” on page 1-6
- “Hardware and Software Requirements” on page 1-7
- “Setting Up and Verifying Your Installation” on page 1-13
- “Setting Up Your Target Hardware” on page 1-18
- “Setting Target Preferences” on page 1-21
- “Code Generation Configuration for Nondefault Processors” on page 1-23
- “Supported Blocks and Data Types” on page 1-27
- “Overview of C166 Options in the Configuration Parameters Dialog Box” on page 1-30

## Product Overview

In this section...
“Introduction” on page 1-2
“Feature Summary” on page 1-2

### Introduction

The Target Support Package™ IC1 product is an add-on product for use with the Embedded IDE Link™ TS and Real-Time Workshop® applications. It provides a set of tools for developing embedded applications for the C166® family of processors from Infineon® (<http://www.infineon.com/>). This includes derivatives such as Infineon C167 and XC16x, and ST Microelectronics ST10 (<http://www.us.st.com>).

Used in conjunction with the Simulink®, Stateflow®, and the Embedded IDE Link TS products, the Target Support Package IC1 product lets you

- Design and model your system and algorithms.
- Compile, download, run and debug generated code on the target hardware, seamlessly integrating with industry-standard compilers and development tools for the C166 microcontroller.
- Deploy production code on the target hardware.

### Feature Summary

The Target Support Package IC1 product is integrated with (and dependent on) the Embedded IDE Link TS product. This integration provides capabilities to the Target including:

- A flexible build process, which allows you to automatically create and build projects in the TASKING EDE using code generated by the Real-Time Workshop® Embedded Coder™ product.
- Customizable project templates for targeting embedded hardware or instruction set simulator.
- Processor-in-the-Loop (PIL) cosimulation techniques to verify generated code running in an instruction set simulator or real embedded hardware

environment. You can set breakpoints, step through the code, and watch variables during cosimulation.

- MATLAB® commands to rapidly and easily interact with projects in the TASKING EDE or debug generated code in the CrossView Pro debugger.
- Execution profiling and code coverage reports are returned from the TASKING EDE to MATLAB for your review.

See “Product Overview”.

The Target Support Package IC1 product also provides these features:

- Automatic generation of the main program including singletasking or preemptive multitasking scheduler
- Scheduler is configurable to allow temporary overruns
- Automated build procedure including starting debugger or download utility
- Support for integer, floating-point, or fixed-point code
- Driver blocks for serial transmit and receive
- Driver blocks for CAN message transmit and receive
- Examples to show you how to integrate your own driver code
- Enhanced HTML report generation provides analysis of RAM/ROM usage; this is in addition to the standard HTML report generation that shows optimization settings and hyperlinks to generated code files
- Support for CAN Calibration Protocol
- External mode for parameter tuning and signal logging

## Prerequisites

---

**Note** You should familiarize yourself with the Embedded IDE Link TS documentation, especially “Getting Started”.

---

This document assumes you are experienced with the MATLAB, Simulink, Real-Time Workshop, and the Real-Time Workshop Embedded Coder products.

Minimally, you should read the following from the “Getting Started” section of the Real-Time Workshop documentation:

- “What You Can Accomplish Using Real-Time Workshop Technology” This section introduces general concepts and terminology related to Real Time Workshop software.
- “Working with the Real-Time Workshop Software” This section provides several hands-on exercises that demonstrate the Real-Time Workshop user interface, code generation and build process, and other essential features.

In addition, if you want to understand and use the device driver blocks in the Target Support Package IC1 library, you should have at least a basic understanding of the architecture of the C166. The C166 User’s Manual (or corresponding document for your C166 derivative processor) is required reading. The MathWorks recommends that you read the introduction to the C166 microcontroller. You can find this document by searching the Infineon Web site for the C166 family of microcontrollers, at the following URL:

<http://www.infineon.com/>

## Using This Guide

Follow this path to get acquainted with the Target Support Package IC1 product and gain hands-on experience with the features most relevant to your interests:

- Read in its entirety, paying particular attention to “Setting Up and Verifying Your Installation” on page 1-13.
- If you are interested in using the device driver blocks supplied with the Target Support Package IC1 product, and in deploying stand-alone, real-time applications on the C166 microcontroller, read Chapter 2, “Tutorial: Simple Example Applications for C166 Microcontrollers” Work through the “Tutorial: Creating a New Application” on page 2-3.
- Then, if you are interested in using the Target Support Package IC1 product for integrating automatically generated code with your own hand-written device driver code, see “Integrating Hand-Coded Device Drivers with a Simulink Model” on page 3-2. Work through the example provided in “Tutorial: Using the Example Driver Functions” on page 3-12.
- See Chapter 4, “Custom Storage Class for C166 Microcontroller Bit-Addressable Memory” to find out how to use the Target Support Package IC1 product to take advantage of C166 bit-addressable memory. This can significantly reduce code size and increase execution speed. There are examples provided in “Using the Bitfield Example Model” on page 4-3.
- For in-depth information about the device drivers and other blocks supplied with the Target Support Package IC1 product, see Chapter 6, “Block Reference” It is particularly important to read C166 Resource Configuration, as the C166 Resource Configuration block is required to use the device driver blocks.
- To browse the demos available, select **Start > Links and Targets > Target Support Package IC1 > Demos**.

We recommend you work through the tutorials in this User’s Guide with step-by-step instructions for using and understanding these demos.

## **Installation**

Your platform-specific MATLAB Installation Guide provides all of the information you need to install the Target Support Package IC1 product.

As the installation process proceeds, it displays a dialog box where you can select which products to install.



# Hardware and Software Requirements

In this section...
“Host Platform” on page 1-7
“Hardware Requirements” on page 1-7
“Software Requirements” on page 1-8
“Switching Between Hardware Variants” on page 1-9

## Host Platform

The Target Support Package IC1 product supports only the PC platform: Windows XP only.

You can see the MATLAB system requirements online at

<http://www.mathworks.com/products/system.shtml/Windows>

## Hardware Requirements

The Target Support Package IC1 product may be used to generate programs that can run on any development board or Electronic Control Unit (ECU) that is based on the C166 microcontroller.

The Target Support Package IC1 product is supplied with default configurations that have been tested on the following hardware:

- Phytex phyCORE-167 C167CS
- Phytex phyCORE-167 ST10F269
- Phytex kitCON-167 C167CR
- Infineon XC167CI Starter Kit
- Infineon XC164CM U CAN start kit
- STMicroelectronics MB449 ST10F25x EVA Board

You can switch easily between these configurations. For other hardware variants, you will need to change the default configuration settings. For part

numbers and other details see “Switching Between Hardware Variants” on page 1-9.

For other hardware variants you need to create a new Embedded IDE Link TS template project file. If the processor variant selected in the project is not on the list above then you need to configure the code generation process.

This guide assumes that you are working with the Phytex phyCORE-167CS development board, and documents specific settings and procedures for use with the Phytex phyCORE-167CS board, in conjunction with specific cross-development environments.

If you use a different development board, you may need to adapt these settings and procedures for your development board.

## **CAN Hardware**

If you want to use CAN to transmit or receive CAN messages between your host PC and your target, you require Vector-Informatik CAN hardware supported by the Vector CAN Driver Library. You must install the correct driver libraries to support profiling, downloading, and external mode.

---

**Note** For CANcaseXL, you must install *both* the Vector XL-driver library and Vector CAN Driver Library `vcand32.d11`.

For older CAN hardware, you must install the Vector CAN Driver Library `vcand32.d11`.

---

Make sure that the library, `vcand32.d11`, is placed in the Windows® `system32` directory.

## **Software Requirements**

### **Required and Related MathWorks Products**

The Target Support Package IC1 product *requires* these products:

- MATLAB

- Simulink
- Real-Time Workshop
- Embedded IDE Link TS
- Optional: Real-Time Workshop Embedded Coder
  - Required for bit-addressable memory feature.
  - Required for CCP Data Acquisition (DAQ) List mode of operation.

Simulink® Fixed Point™ software is strongly recommended but not essential, except for one of the demos: `c166_fuelsys`.

For more information about any of these products, see either

- The online documentation for that product, if it is installed
- The MathWorks Web site, at [http://www.mathworks.com/products/target\\_c166/](http://www.mathworks.com/products/target_c166/)

### **Supported Cross-Development Tools**

In addition to the required MathWorks software, a supported cross-development environment is required.

- See “Supported Altium® TASKING® Toolsets” in the Embedded IDE Link TS documentation for the currently supported cross-development tools for use with the Target Support Package IC1 product.
- MiniMon freeware download and monitor utility

Before using the Target Support Package IC1 product with the above cross-development tools, please be sure to read and follow the instructions in “Setting Up and Verifying Your Installation” on page 1-13.

### **Switching Between Hardware Variants**

There are many different members of the C166 microcontroller family, e.g., C167CS, ST10, XC167CI. For each of these processors, it is appropriate to use different compiler switches and link libraries. Even if you are working with a single processor variant, you may need to build for different memory configurations, for example, depending on whether the application will run

from RAM or flash memory. The compilation settings are captured in the project file.

The Target Support Package IC1 product is supplied with preconfigured projects for targeting the hardware and simulator for the following list of processor variants. Part numbers, product codes and other useful details are included.

- **Phytec phyCORE-C167CS**

phyCORE®-167 HS/E Rapid Development Kit

PART number KPCM-018-C1UE

Includes:

- phyCORE-167CS HS/E Single Board Computer module (C167CS, 40 MHz, 512 KB Fast-SRAM, 256 KB Flash, 4 KB EEPROM, dual 2.0B CAN, Ethernet, supp. UART) (PCM-018-C1UE)

Controller: SAK-C167CS-L40M

phyCORE Development Board HD200 (PCM-997-V3)

Ethernet Interface Adapter (EAD-001)

Bare PCB Expansion Board (PCM-988)

DB-9 serial cable

5V AC adapter

PHYTEC Spectrum CD with electronic documentation, demos, evaluation software

Infineon DAVe CD

Schematics

- **Infineon XC167CI Starter Kit**

XC167CI Easy Kit V3

Product code # 37.96.0038.0

- **Phytec phyCORE-ST10F269**

phyCORE®-ST10F269 HS/E Rapid Development Kit

<http://www.phytec.com/products/rdk/C166-xc166-st10-xa/phyCORE-ST10F269HS-E.html>

PART number: KPCM-018-140C3UE

Controller: ST10F269Z2Q6

- **Phytec kitCON-167 C167CR**

Part number: KC-167-KSM04

Includes:

- C167CR, 20 MHz, 64 KB SRAM, 256 KB Flash

Controller: SAK-C167CR-LM

- **Infineon XC164CM U CAN**

U CAN XC164CM Start Kit

Product code # 37.96.0024.0

- **STMicroelectronics MB449 ST10F25x**

ST10F252 Eva board

ST10F252-EVA/

If your hardware variant is not on this list, then you need to create a new Embedded IDE Link TS template project (see “Tutorial: Creating New Template Projects”) and set the C166 code generation options (see “Code Generation Configuration for Nondefault Processors” on page 1-23).

When switching between target configurations, you should review your Embedded IDE Link TS option set and ensure that options are set appropriately for the new configuration.

Additionally, for each model that you build, you must check, and, if necessary, change the following settings in the C166 Resource Configuration block:

- `System_frequency`
- `External_oscillator_frequency`

To determine the correct value of these parameters, consult your hardware documentation.

It is possible to make all the required changes programmatically: a convenience function `c166switchconfig` is provided for this purpose. This function can be run by double-clicking the block Switch Target Processor Variant inside any of the demo models.

## Setting Up and Verifying Your Installation

### In this section...

“Setting Up Software” on page 1-13

“Verifying MiniMon Settings” on page 1-14

### Setting Up Software

Install the Tasking C Cross-Compiler and CrossView Pro Debugger by following the instructions provided by Altium Limited.

If the CrossView connection to your target hardware requires a serial connection, install the MiniMon download utility. By using MiniMon instead of CrossView to launch your application, the serial connection will be available for other purposes, if required. If your CrossView connection is via a debug interface (for example, on XC16x hardware) then it is not necessary to install MiniMon.

At the time of writing, you can obtain the MiniMon download utility for monitoring the serial interface from the Infineon Web site at this URL:

<http://www.infineon.com/>

To download the MiniMon utility:

- 1 Go to the Infineon Web site, and click the sitemap.
- 2 Select Product Categories > Microcontrollers > Development Tools, Software and Training -> C166/XC166 Development Tools and Software > Software Downloads.

Find MiniMon in the table, and download and install Minimon. At the time of writing, version 2.2.33 has been verified with this product.

Minimon may need to be configured for your target processor.

After you install, you must specify the location of MiniMon in the `BootstrapLoaderExe` target preference, as detailed in “Setting Target Preferences” on page 1-21. Check that MiniMon is correctly configured for


your target, as detailed in the next section, “Verifying MiniMon Settings” on page 1-14.

The next sections describe how to configure your development environment (compiler, debugger, etc.) for use with the Target Support Package IC1 product, and how to verify correct operation. The initial configuration steps are described in the following sections:

- “Setting Up Your Target Hardware” on page 1-18
- “Setting Target Preferences” on page 1-21

## Verifying MiniMon Settings

You must check that MiniMon has the correct target settings. This section describes MiniMon configuration settings that work for the C167CR processor and for the C167CS processor. Settings for the C167CS board also work successfully with the ST10F269. You may be able to use MiniMon to download onto other processors, however, you must establish a corresponding MiniMon configuration.

To check settings, start MiniMon, then click Configure Hardware (  ) in the toolbar (or select **Target > Configuration**) and make sure the settings are as in the following illustrations.

In general, you should choose configuration settings that are consistent with the values specified in the Tasking EDE project.

Select **Settings > Interface** and ensure that the settings for the serial interface match those in the Resource Configuration block of your model.

To set up a configuration for a C167CR:

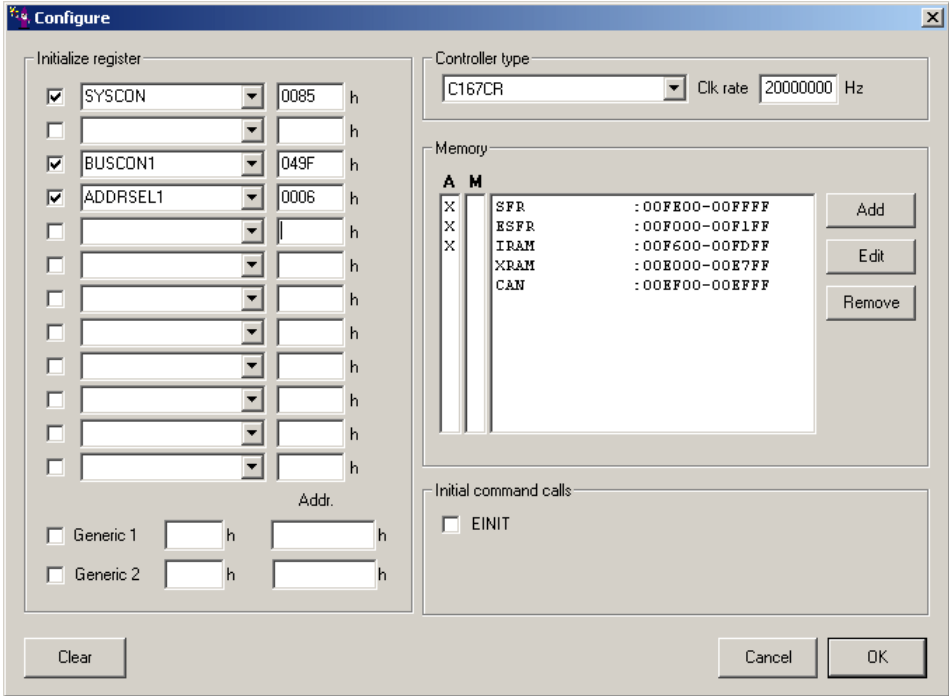
- 1** Select **C167CR** from the **Controller type** drop-down list.
- 2** Click **Yes** three times when prompted by the dialog boxes asking the following questions:
  - a** Do you want to load default memory units for this Type?
  - b** Do you want to activate the default kernel for this Type?



- c Do you want to load default initialization registers of this Type?
- 3 Perform the following steps on the Initialize register settings:
  - a Set SYSCON to 0085.
  - b Set BUSCON1 to 049F.
  - c Set ADDRSEL1 to 0006.
  - d Clear all the other check boxes.

The register settings should look as shown.

This configuration has been verified with a Phytex kc167 (C167CR).



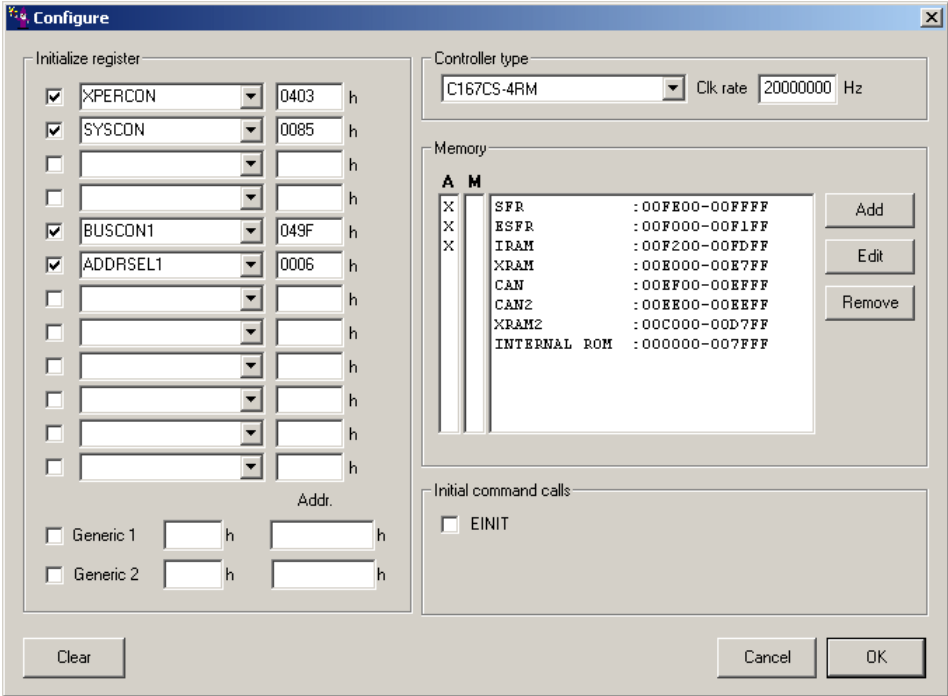
To set up a configuration for a C167CS or ST10F269:

- 1 Select C167CS-4RM from the **Controller type** drop-down list.

- 2** Click **Yes** three times when prompted by the dialog boxes asking the following questions:
  - a** Do you want to load default memory units for this Type?
  - b** Do you want to activate the default kernel for this Type?
  - c** Do you want to load default initialization registers of this Type?
- 3** Perform the following steps on the Initialize register settings:
  - a** Change SYSCON to XPERCON, and set the value to 0403.
  - b** Change SYSCON1 to SYSCON, and set the value to 0085.  
  
The order is important: XPERCON must be above SYSCON.
  - c** Set BUSCON1 to 049F.
  - d** Set ADDRSEL1 to 0006.
  - e** Clear all the other check boxes.

The register settings should look as shown.

This second configuration has been verified with phyCORE-C167CS and on phyCORE-ST10F269 hardware.



## Setting Up Your Target Hardware

In this section...
“Jumper Settings for the phyCore-167 Development Board” on page 1-18
“Setting Up XC164CM Hardware” on page 1-18
“Jumper Settings for the STMicroelectronics MB449 ST10F25x EVA Board” on page 1-19

### Jumper Settings for the phyCore-167 Development Board

This section describes the required connections and jumper settings for the phyCORE-167CS module with HD200 development board.

After setting up your board, you must configure target settings associated with the Target Support Package IC1 product, as described in the next section.

- 1 Configure jumpers as detailed in the instructions found in the phyCORE QuickStart documentation. Note that these settings can be markedly different from the configuration fresh out of the box.
- 2 If you are running applications from RAM only, it is useful if the board starts up in bootloader rather than execution mode. There is one jumper setting that needs to be changed to achieve this: close pins 1 and 2 on JP10. This is optional; if you do not close this jumper, then when you download to the target, you need to keep the Boot switch depressed while pressing the Reset button.

Connect the supplied power cable to the board, and use the serial cable to connect the serial port P1 on the board to the serial port of your PC.

### Setting Up XC164CM Hardware

See the Embedded IDE Link TS documentation for information on software installation.

If you need to profile on XC164CM hardware via the serial port, this is possible when using CrossView. Check which COM port is assigned to USB

COM Port. To access the Device Manager where you can see this information, select Windows **Start > Settings > Control Panel**, double-click System, select the **Hardware** tab, and click **Device Manager**.

This information can be passed to the `profile_c166` command as follows:

```
% assuming that it was assigned to COM4
profile_c166('serial','SerialPort','COM4')
```

The MiniMon hyperlink may not be provided at the end of builds for hardware (e.g. XC164CM U CAN and xc167ci\_hw\_usb) that has a JTAG interface available. This is because this hardware uses the JTAG debug interface instead of a serial connection to ASC0. For hardware with a JTAG interface there is no conflict between using the CrossView debugger simultaneously with the ASC0 serial interface: in these cases it is recommended always to use CrossView for downloading and running applications.

## Jumper Settings for the STMicroelectronics MB449 ST10F25x EVA Board

Settings not listed here should be as default, as specified by the board manual.

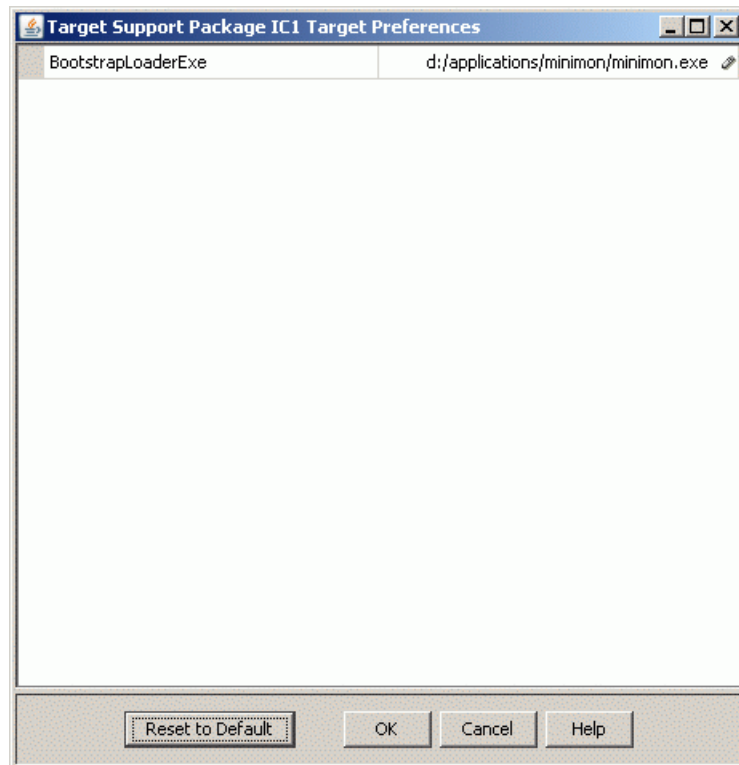
Type	Jumper Settings
Boot / Configuration Mode > SW4	Switch 4-5-6: CLKCFGSwitch State: on-on-off.  Note: With fCPU = 5* fXTAL and oscillator frequency of 8MHz, the system frequency, fCPU, is 40MHz
	Switch 2-3: SALSEL Switch State: off-off
	Switch 1: WRCSwitch State: off
Boot / Configuration Mode > SW3	Switch 7-8: BUSTYPSwitch State: off-off
	Switch 5-6: BSLSwitch State: on-off
	Switch 2: ADPSwitch State: off

<b>Type</b>	<b>Jumper Settings</b>
External Memory	J1, 1-2: Closed Note: Enables external memory
Reset and Vstby EA jumpers	J4, 2-3: Closed Note: Forces EA pin to Vcc level
CAN	J11, 1-2: Closed Note: Connects onboard CAN transceiver
	J11, 3-4: Closed Note: Connects onboard CAN transceiver

## Setting Target Preferences

This section describes configuration settings associated with the Target Support Package IC1 product. These settings, which persist across MATLAB sessions and different models, are referred to as *target preferences*. Target preferences let you specify the location of your cross-compiler and other parameters affecting the generation, building, and downloading of code.

- 1 First you must set up your Embedded IDE Link TS target preferences to specify the location of your cross-compiler and other settings. See “Setting Target Preferences” in the Embedded IDE Link TS documentation.
- 2 Open the Target Support Package IC1 Target Preferences dialog box by selecting **Start > Links and Targets > Target Support Package IC1 > Target Preferences** .



### 3 Edit the settings for your cross-development environment:

- `BootstrapLoaderExe` specifies the path to your download utility (MiniMon).

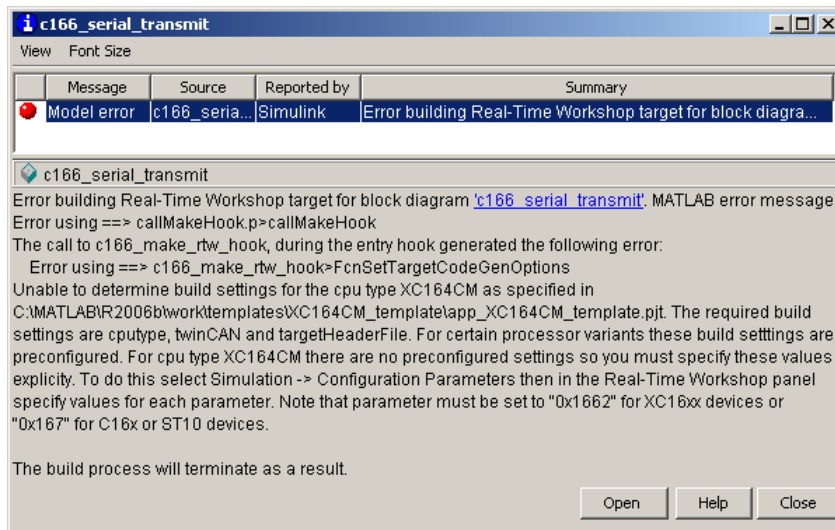
You must check this path and also verify that the Embedded IDE Link TS target preferences are correct for your machine. You may need to localize these paths to suit your PC. You can edit a path by clicking on it. The drive designated in the path must be either an actual hard drive on your PC, or a mapped drive. Do not use a Universal Naming Convention (UNC).



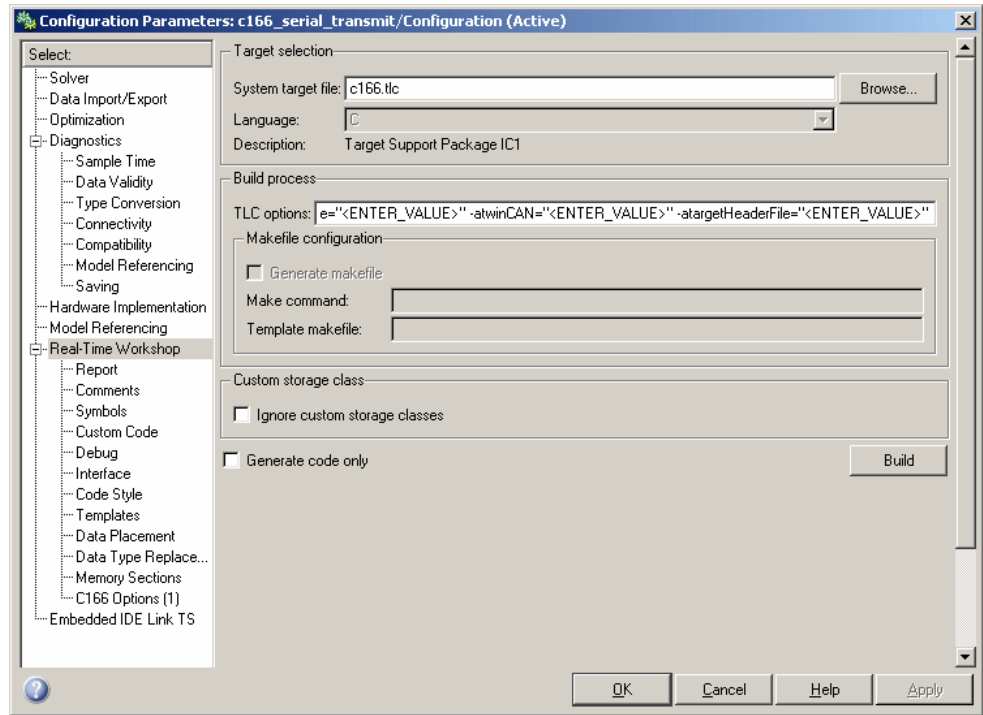
## Code Generation Configuration for Nondefault Processors

If you wish to target nondefault processor types, then you need to set some code generation options in the **TLC Options** of your model's configuration parameters.

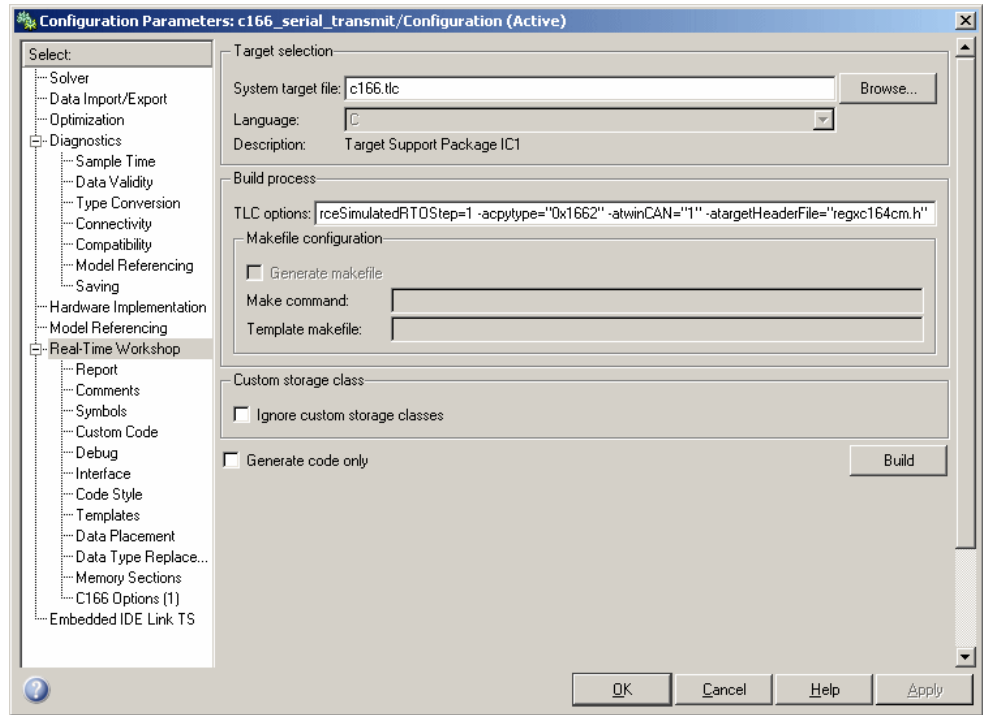
If you are using a template that specifies a nondefault processor type (see “Template Projects” in the Embedded IDE Link TS documentation), when you try to build the model, you see a build error message similar to the one in the following figure.



When you open the Configuration Parameters dialog box, the parameters you need to set now appear in the TLC Options field. You must replace the string <ENTER VALUE> for each of the parameters `cpuType`, `twinCAN`, and `targetHeaderFile`. The following example shows these parameters before the strings <ENTER VALUE> are replaced.



An example configured for an XC164CM is shown in the following figure.



## Summary of Parameters

The **TLC Options** edit box includes the following parameters:

cpuType

- 0x167, for C16x and ST10 type processors
- 0x1662, for XC16x type processors

twinCAN

- 0 – disabled, for use with processors without TwinCAN support
- 1 – enabled, for use with processors with TwinCAN support

`targetHeaderFile` — The file name of the header file for your processor type. These are found in *TASKING\_ROOT*\include directory.

### Typical Parameter Configuration

The following table shows a configuration matrix for the parameters `cpuType`, `twinCAN` and the typical configurations used for the processor variants supported by the product.

Processor Type	CPU type	TwinCAN
16x, ST	0x167	0 - disabled
XC	0x1662	1 – enabled

---

**Note** Driver blocks may not work on unsupported processors.

---

## Supported Blocks and Data Types

The Target Support Package IC1 product supports the same blocks and data types as the Real-Time Workshop Embedded Coder product.

Note however

- You should not use IEEE values Inf or NaN in your model: these are not supported and result in an error.
- Floating point support is implemented in the software; if speed and ROM usage are of concern, you should select the option for integer code and avoid the use of floating-point values in your model. This is detailed in step 9 of “Tutorial: Using the Example Driver Functions” on page 3-12.

The Target Support Package IC1 product provides one block library, containing seven sublibraries that support different functions, as follows:

- C166 Drivers Library
  - Asynchronous/Synchronous Serial Interface Sublibrary
  - CAN Interface Sublibrary
  - Execution Profiling Sublibrary
  - TwinCAN Interface Sublibrary
  - Interrupts Sublibrary
  - Utilities Sublibrary
  - Digital Input/Output Sublibrary

See Chapter 6, “Block Reference” for details of each block. You can click **Help** on the Block Parameters dialog box for the block or access the block reference page through Help.

The top-level C166 Drivers library contains the C166 Resource Configuration block. This block supports driver configuration for C166 microcontrollers and is required if there are device driver blocks in the model. See C166 Resource Configuration.

The C166 Resource Configuration block provides information required for generating timer interrupt code. If you do not include a C166 Resource Configuration block in your model, the code simply executes as fast as possible. That is, it is not synchronized to real time. This behavior may be desirable if you are running code on the debugger or hardware simulator.

---

**Caution** When using device driver blocks from the Target Support Package IC1 libraries with the C166 Resource Configuration block, do not disable or break library links on the driver blocks. If library links are disabled or broken, the C166 Resource Configuration block operates incorrectly. See the C166 Resource Configuration reference page for further information.

---

## Model Reference and Driver Blocks

Referenced sub-models that contain driver blocks (including the C166 Resource Configuration block) cause build failures. All driver blocks from the Target Support Package IC1 product must be placed in the top level model. It is not possible to include driver blocks in any of the referenced sub-models.

## Configuration Class Blocks

Each sublibrary of Target Support Package IC1 library contains a *configuration class block* that has an icon similar to the one shown in this picture.



---

**Caution** Configuration class blocks exist only to provide information to other blocks. *Do not copy these objects into a model.* If you do you see an error dialog box to warn you. This causes build failures.

---

## Overview of C166 Options in the Configuration Parameters Dialog Box

When you select a C166 system target file in the Real-Time Workshop configuration parameters dialog box, additional options appear in the tree: C166 Options, and Embedded IDE Link TS.

Select **C166 Options** (under **Real-Time Workshop** in the Configuration Parameters dialog box) as shown in the following figure, to see the following options:

### **Include input/output driver function hooks**

Use this option to integrate your own device driver code. This is described in “Calling the Device Driver Functions from `c166_main.c`” on page 3-7.

The following are all execution profiling controls. See “Overview of Execution Profiling” on page 5-2.

### **Maximum number of concurrent base-rate overruns**

Option for task execution profiling. See “Task Scheduler Overrun Options” on page 5-7.

### **Maximum number of concurrent sub-rate overruns**

Option for task execution profiling. See “Task Scheduler Overrun Options” on page 5-7.

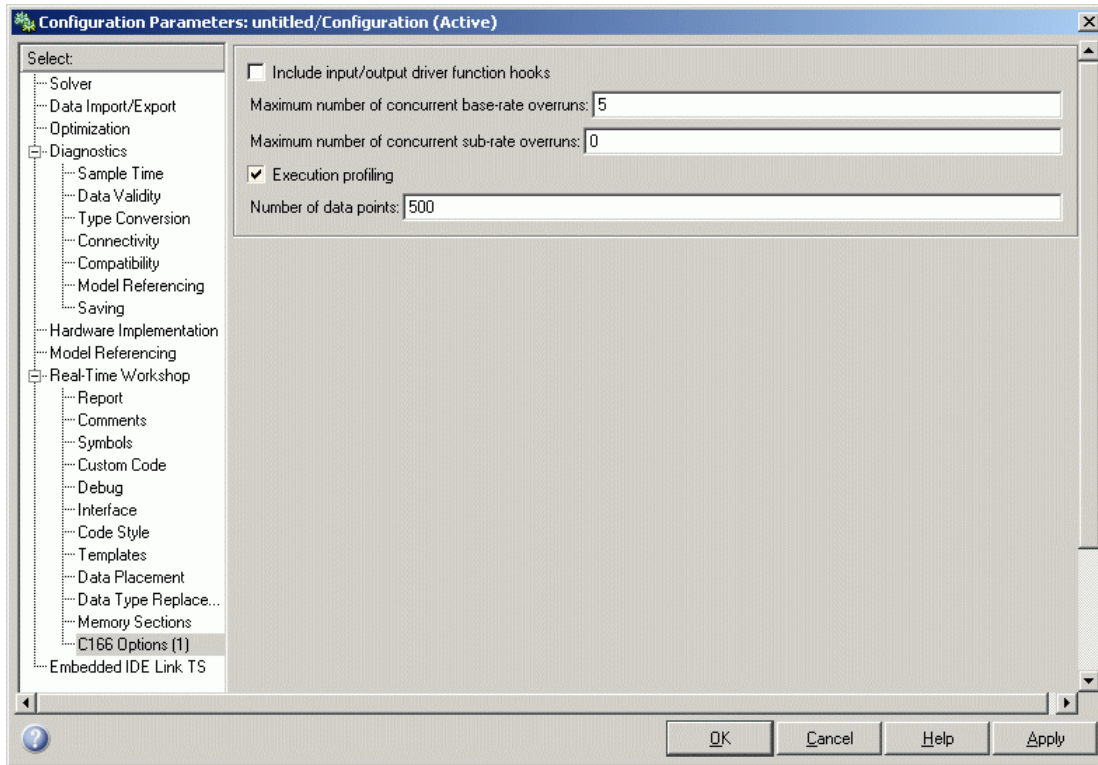
### **Execution profiling**

Option for task execution profiling. See “Real-Time Workshop Options for Execution Profiling” on page 5-6.

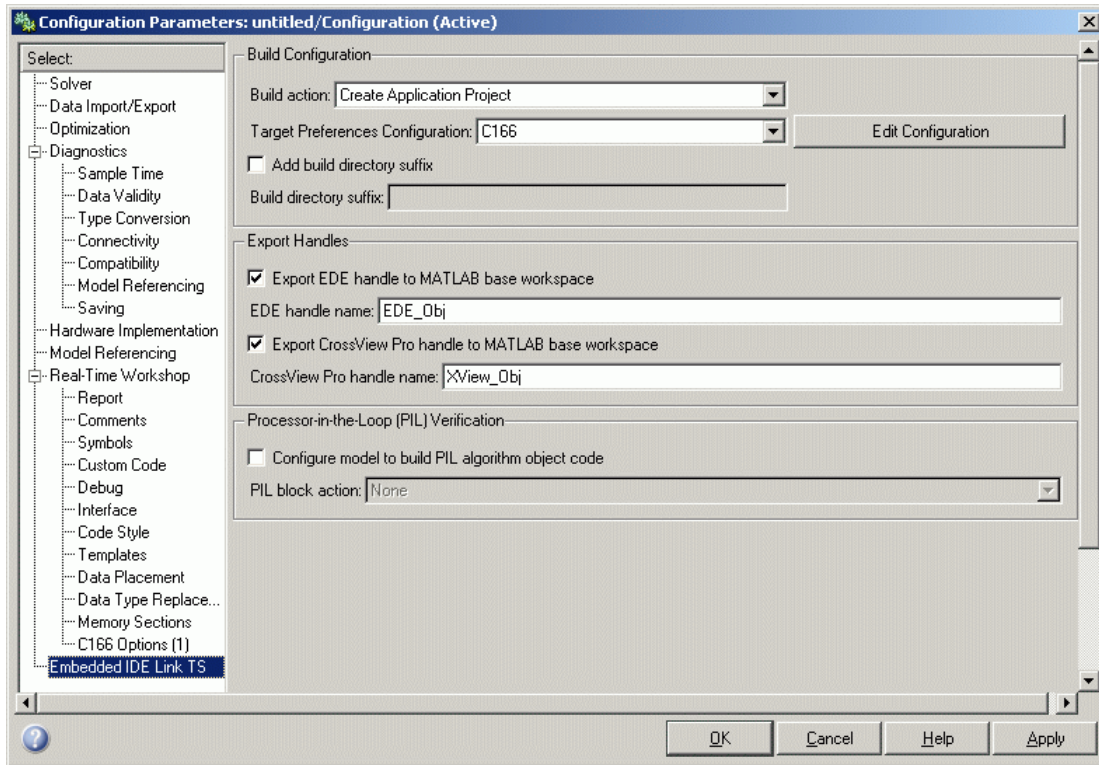
### **Number of data points**

Option for task execution profiling. See “Real-Time Workshop Options for Execution Profiling” on page 5-6.





When you select a C166 system target file, the Embedded IDE Link TS Configuration Parameters component is automatically added to the model, as shown following.



The **Target Preferences Configuration** description is automatically set to C166 to use the predefined C166 project templates. The Embedded IDE Link TS options contain settings for configuring the Embedded IDE Link TS build process. See the Embedded IDE Link TS documentation for more information on these options.

# Tutorial: Simple Example Applications for C166 Microcontrollers

---

This section includes the following topics:

- “Introduction” on page 2-2
- “Tutorial: Creating a New Application” on page 2-3
- “Debugging and Using The Code Profile Report” on page 2-12
- “Parameter Tuning and Signal Logging” on page 2-18

# Introduction

This section describes how to use two example models to generate, download and run stand-alone real-time applications for the C166 microcontroller. The components required to generate stand-alone code are

- The Target Support Package IC1 real-time target
- The example models provided: `c166_serial_transmit` and `c166_serial_io`
- The Tasking C Cross-Compiler and Tasking CrossView Pro Debugger for compiling and downloading generated code to the target hardware

As an alternative to CrossView, you can use the MiniMon utility for downloading an application to your target hardware.

Using these, you can build the complete application. You do not need to hand-write any C code to integrate the generated code into a final application.

The tutorial “Tutorial: Creating a New Application” on page 2-3 uses two blocks from the Target Support Package IC1 library. For complete information on the Target Support Package IC1 library blocks, see Chapter 6, “Block Reference”.

## Tutorial: Creating a New Application

In this section...
“Tutorial Overview” on page 2-3
“Before You Begin” on page 2-3
“Example Model 1: c166_serial_transmit” on page 2-4
“Generating and Downloading Code” on page 2-7
“Example 2: c166_serial_io” on page 2-9

### Tutorial Overview

In this tutorial, you build stand-alone real-time applications from models incorporating blocks from the Target Support Package IC1 library.

In the following sections, you will

- Examine two models
- Generate code from the models
- Download and run the code automatically as part of the build process
- Use MiniMon to monitor the code executing on the target

### Before You Begin

We assume that you are already familiar with Simulink software and with the Real-Time Workshop code generation and build process. This tutorial requires the following specific hardware and software in addition to the Target Support Package IC1 product:

- Phytex phyCORE-167CS development board, connected via serial port to your PC
- Tasking C Cross-Compiler and CrossView Pro Debugger
- MiniMon download utility

You must make sure the target preferences have been set correctly. See “Setting Target Preferences” on page 1-21.

---

**Note** Make sure the `default.ini` file in the MiniMon directory is not read only. This can cause errors.

---

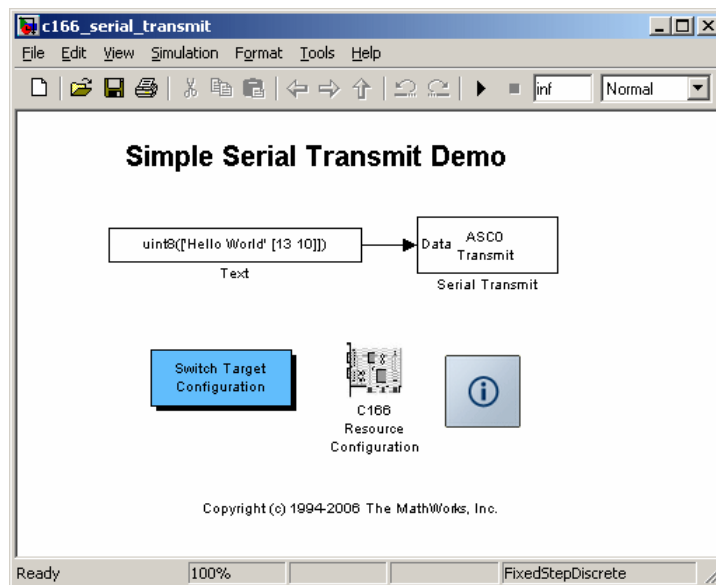
### Example Model 1: `c166_serial_transmit`

In this tutorial you start with a simple example model, `c166_serial_transmit`, from the directory `matlabroot/toolbox/rtw/targets/c166/c166demos`.

This directory is on the default MATLAB path.

- 1 Open the model by typing `c166_serial_transmit` at the command line.

This example shows the tutorial model `c166_serial_transmit` at the root level.



The model contains a C166 Resource Configuration object. When building a model with driver blocks from the Target Support Package IC1 library, you

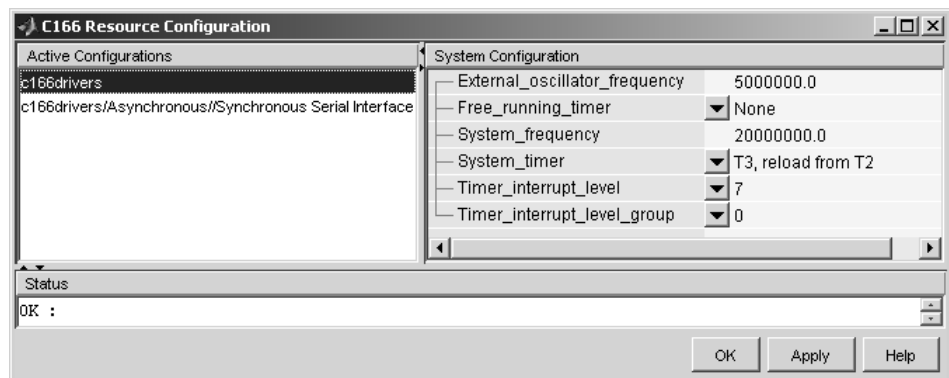
must always place a C166 Resource Configuration object into the model (or the subsystem from which you want to generate code) first.

The purpose of the C166 Resource Configuration object is to provide information to other blocks in the model. Unlike conventional blocks, the C166 Resource Configuration object is not connected to other blocks via input or output ports. Instead, driver blocks (such as the ASC0 Serial Transmit block in the example model) query the C166 Resource Configuration object for required information.

For example, a driver block may need to find the system clock speed that is configured in the C166 Resource Configuration object. The C166 microcontroller has a number of clocked subsystems; to generate correct code, driver blocks need to know the speeds at which these clock busses will run.

The C166 Resource Configuration window lets you examine and edit the C166 Resource Configuration settings.

- 2** Double click the switch target configuration block, and then select `c167cs_hw`. This selection sets the appropriate `System_frequency` and `External_oscillator_frequency` in the Resource Configuration block and the Embedded IDE Link TS option set. See “Option Sets” in the Embedded IDE Link TS documentation for more information.
- 3** To open the C166 Resource Configuration window, double-click the C166 Resource Configuration icon. The picture following shows the C166 Resource Configuration window for the `c166_serial_transmit` model.



In this tutorial, use the default C166 Resource Configuration settings.

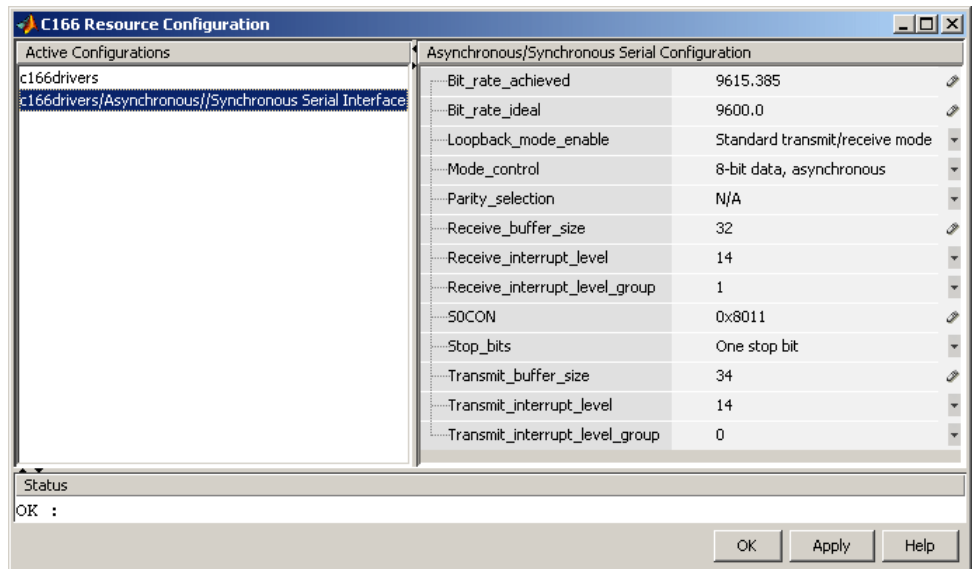
---

**Note** If hardware is running at a system frequency other than 20 MHz, you must change this parameter appropriately.

---

Otherwise, observe, but do not change, the parameters in the **C166 Resource Configuration** window. By default, the **c166drivers** configuration is selected. This shows parameters for the C166 microcontroller CPU in the **System Configuration** pane on the right.

- 4 View the settings for the serial driver block by clicking the **c166drivers/Asynchronous/Synchronous Serial Interface** option in the **Active Configurations** pane. These settings are shown in the following illustration.



The settings appear in the **Asynchronous/Synchronous Serial Configuration** pane on the right. Do not edit any of these parameters for this tutorial. To learn more about the C166 Resource Configuration object, see C166 Resource Configuration.



**5** Close the **C166 Resource Configuration** window before proceeding.

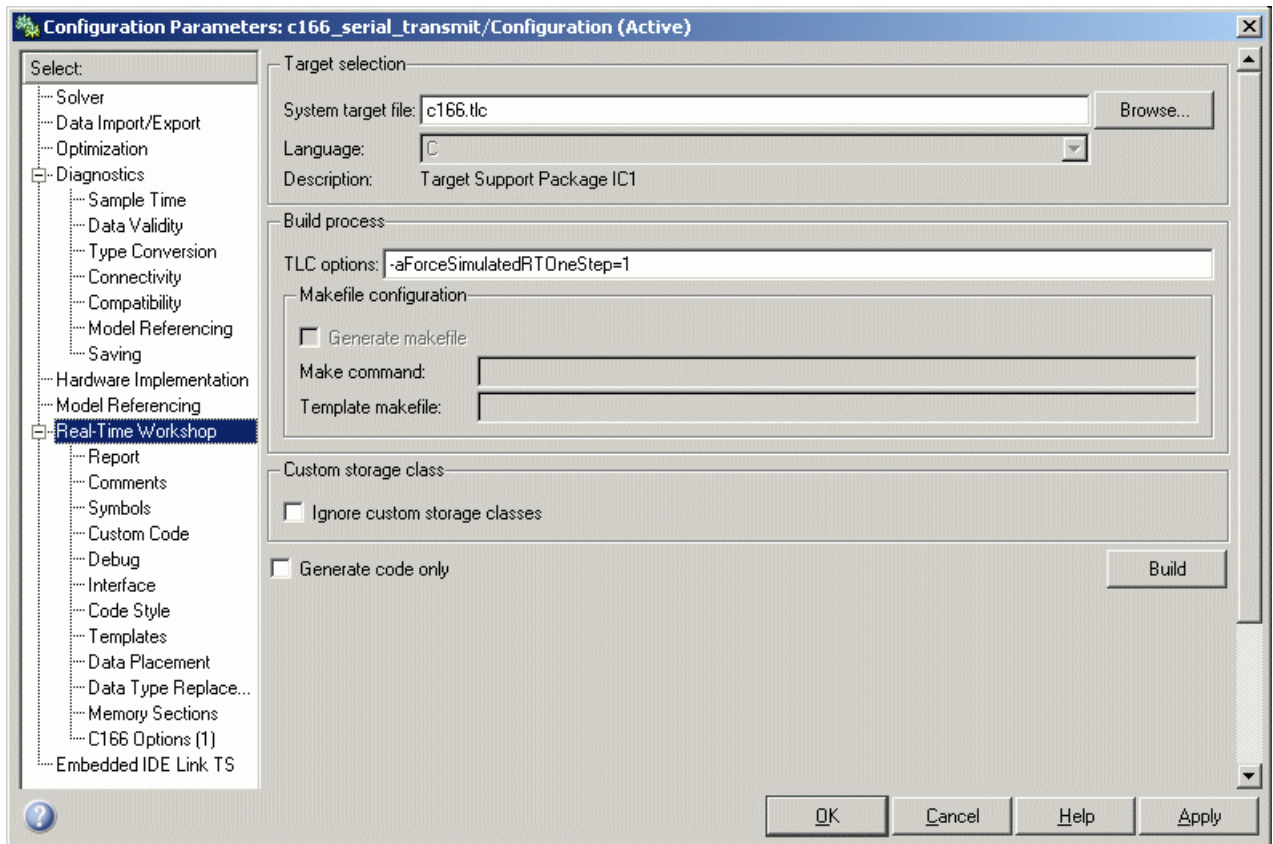
## Generating and Downloading Code

To generate code for the model:

**1** Select **Simulation > Configuration Parameters**.

The Configuration Parameters dialog box opens.

**2** Select **Real-Time Workshop** in the tree, as shown below.



**3** Click **Build**.

Alternately, you can go straight to building the model by selecting **Tools > Real-Time Workshop > Build Model** or using the shortcut **Ctrl+B**.

Watch the progress messages in the command window as code is generated.

- 4 Click the MiniMon link in the MATLAB command window to download your application via MiniMon. If the Minimon link has not been generated then your C166 Embedded IDE Link TS option set is not compatible with MiniMon downloads. This failure to generate could be because you are targeting one of the simulator configurations or your board is using OCDS (on-board wiggler) to connect to the target. The Minimon option should appear when:
  - The Embedded IDE Link TS Build Action is set to **Create and Build Application Project, Create, Build and Execute Application Project, or Create, Build and Debug Application Project**.
  - The option set is for hardware (rather than simulator).
  - You are using a serial connection to connect to your target.
  - If you have created your own template projects, the option to generate a hex file must be selected.

---

**Caution** You must ensure the option to generate a hex file is turned on. If you do not you will see the following warning:

```
It was not possible to generate a minimon script for this
build. This is because your EDE project template is not
configured to generate a .hex file which is required by
Minimon. To generate a .hex file as part of the build
you need to check the box 'Intel HEX records' in your
EDE project template.
```

```
You can change this option via Project -> Project Options
-> Linker/Locator -> Output Format.
```

---

When MiniMon is started, a dialog box appears asking you to reset your hardware.

- 5 Press the Reset button on your phyCORE-167CS board or cycle the power, and then click **OK**.

You can see progress messages in the MiniMon window as it connects and then downloads to the target. MiniMon then disappears and the code begins executing on the target.

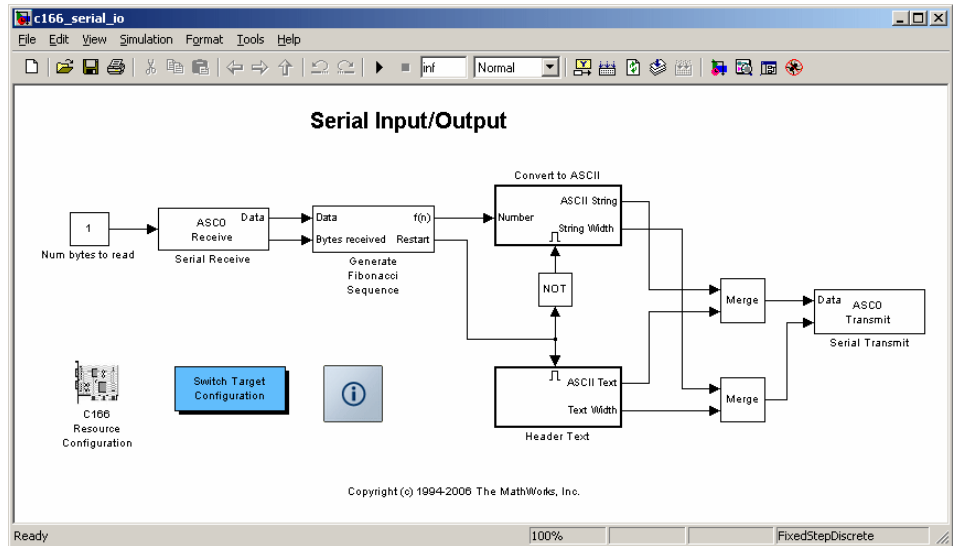
## Verifying Code Execution on the Target

- 1 Start MiniMon (select **Start > Programs > MiniMon > MiniMon** in Windows, or navigate to `MiniMon.exe` and double-click).
- 2 Watch the model output in the MiniMon window. When the application is running, it sends the text "Hello World" plus a carriage return and a linefeed over the serial interface.

## Example 2: `c166_serial_io`

This example model demonstrates how to use both serial transmit and receive blocks for the C166 microcontroller. You could use these blocks in this way with your own Simulink models.

- 1 Open the model by typing `c166_serial_io` at the command line.



**2** Double click on the switch target configuration block, then select `c167cs_hw`. This will set the `System_frequency` and `External_oscillator_frequency` in the Resource Configuration block and the Embedded IDE Link TS option set.

**3** Press **Ctrl+B** or select **Tools > Real-Time Workshop > Build Model**.

Watch the progress messages as code is generated from the model.

**4** You can download the application by clicking on the link at the end of the build log. This link launches MiniMon.

MiniMon is started to download the code to the target over the serial connection. The MiniMon dialog box appears asking you to reset your hardware.

**5** Press the Reset button on your phyCORE-167CS board or cycle the power, and then click **OK**.

You can see progress messages in the MiniMon window as it connects and then downloads to the target. MiniMon then disappears and the code begins executing on the target.

You can restart MiniMon to monitor the serial interface.

## **Verifying Code Execution on the Target**

- 1** Start MiniMon (select **Start > Programs > MiniMon > MiniMon** in Windows, or navigate to `MiniMon.exe` and double-click).
- 2** Watch the model output in the MiniMon window. When the application is running, it generates a sequence of 16-bit numbers, converts them to ASCII characters, and transmits them over the serial interface.
- 3** If you enter the character `r` in the MiniMon command line field, the application restarts at the beginning of the sequence. Examine the model to see how this works: the Serial Receive block passes the restart command through to the Generate Fibonacci Sequence subsystem. This subsystem checks for the restart command.

## Debugging and Using The Code Profile Report

<b>In this section...</b>
“Starting the Debugger on Completion of the Build Process” on page 2-12
“RAM / ROM Code Profile Report” on page 2-14

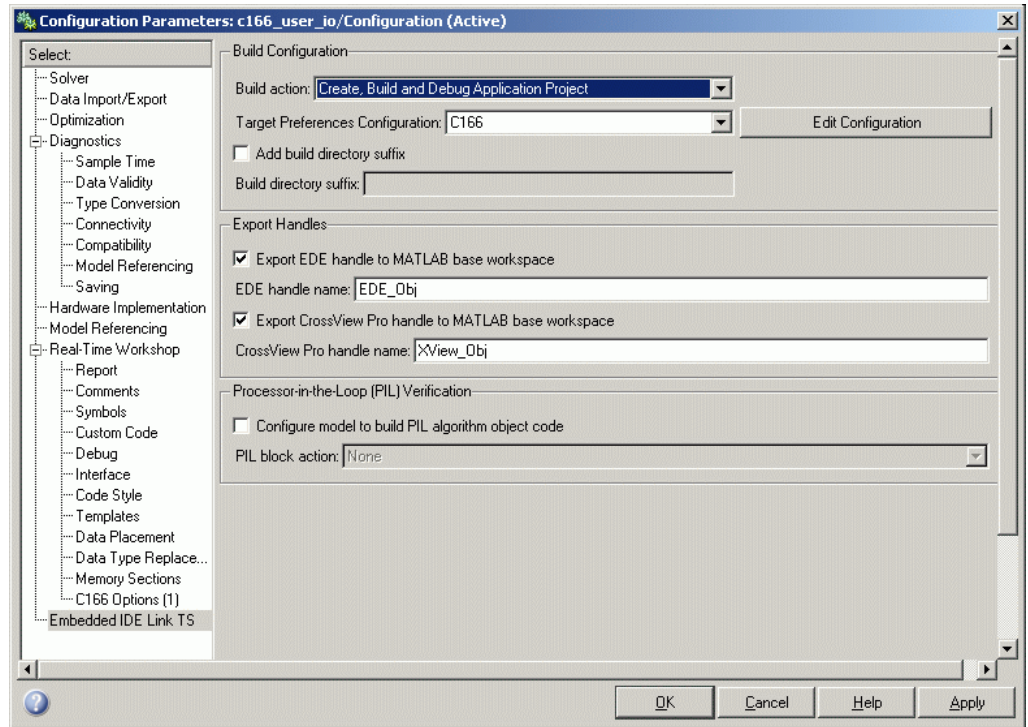
### Starting the Debugger on Completion of the Build Process

As an alternative to downloading with MiniMon at the end of the build process, you can start your debugger. Depending on the features provided by your debugger, you can debug the application either on-chip or on a hardware simulator.

For this example, you use another demo model, `c166_user_io`. This model is designed to show you how to integrate your own hand-coded device drivers with automatically generated code using the Target Support Package IC1 product. This model is covered in detail in Chapter 3, “Integrating Your Own Device Drivers”. You use it as an example here because you will typically need to use the debugger in cases where you are integrating your own code.

Also, note that running the debugger on-chip over the serial interface conflicts with the serial transmit and receive blocks. The `c166_user_io` model does not use serial blocks, so this avoids serial conflicts for this example. If you need to debug an application that includes the serial transmit and receive blocks, you must run the debugger using a hardware simulator; alternatively, it may be possible to run your debugger on-chip without using the serial interface, for example, if debugging over CAN or JTAG is available.

- 1** Open the model `c166_user_io`.
- 2** Select **Simulation > Configuration Parameters**.
- 3** Select **Embedded IDE Link TS** in the tree.



- 4 Select the **Build action** Create, Build and Debug Application Project.
- 5 Before generating code, check that your target preferences related to the debugger are correctly configured. See “Setting Target Preferences” on page 1-21.
- 6 Click **OK**.
- 7 Right-click the controller subsystem and select **Real-Time Workshop > Build Subsystem**.
- 8 Click **Build** in the next dialog box.

Watch the progress messages in the command window as code is generated. At the end of the build process, your debugger launches automatically with the application ready to run. You may now debug the application.

---

**Note** If your model contains a serial transmit or receive block, it is not possible to perform on-chip debugging over the same serial interface. If you attempt to use the debugger once your application is running, you will no longer be able to control the application from the debugger, because the application is using the serial channel.

---

### **RAM / ROM Code Profile Report**

The `c166_fuelsys` model is derived from the `fuelsys` demo model. The floating point control algorithm from the original model has been converted to fixed point to allow efficient code generation for the Infineon C166 microcontroller.

---

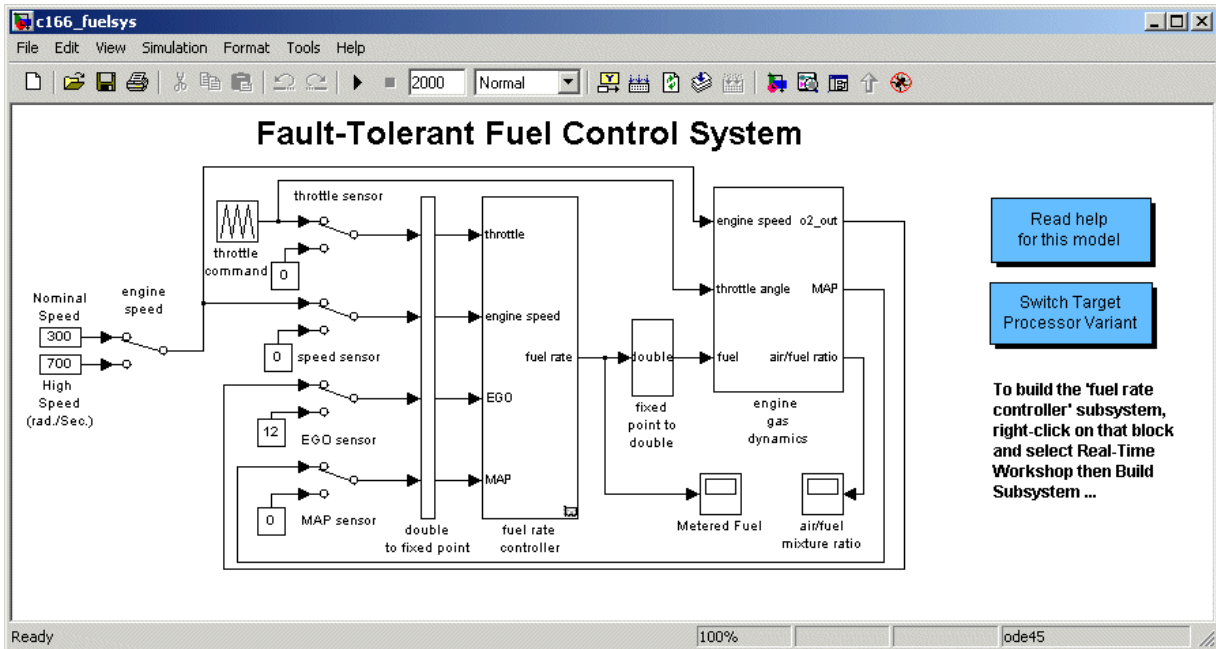
**Note** This demo requires the Simulink Fixed Point product.

---

The complete model includes a plant simulation as well as a fixed-point implementation of the control algorithm. When you generate code for this example, be sure to generate code for the control algorithm subsystem only:

- 1 Open the model `c166_fuelsys`.





**2** Select **Simulation > Configuration Parameters**.

**3** Select **Real Time Workshop** in the tree. Note that the **Generate code only** option is not selected. The reason for this step is that the code generation report obtains information from MAP files that are created by your cross-compiler during the build process. If the **Generate code only** option is on, these files are not generated, which prevents the generation of the code generation report.

**4** Select **Report** in the tree, and then observe the selected check box **Create code generation report**.

Select the **Launch report automatically** check box.

**5** Select **Embedded IDE Link TS** in the tree, and observe the **Build Action** is **Create and Build Application Project**. You must have one of the Build options selected to get the code profile report (with RAM/ROM usage):

- **Create and Build Application Project**

- **Create, Build and Execute Application Project**
  - **Create, Build and Debug Application Project**
- 6** Close the Configuration Parameters dialog box.
  - 7** Right-click the fuel rate controller block.
  - 8** From the pop-up menu, select **Real Time Workshop > Build Subsystem**.
  - 9** On the following dialog box, click **Build**.

When code generation is complete, the Code Generation Report appears in your Help browser. Here you can review the RAM and ROM requirements of the model. To do this, left-click the link `Code profile` report in the left list. If you compared with the original floating-point version of the `fuel` control algorithm: you would find that using the fixed-point implementation results in a considerable reduction in both RAM and ROM. An example report is shown following.

Real-Time Workshop Report

Back Forward

**Contents**  
[Summary](#)  
[Traceability Report](#)  
[Subsystem Report](#)  
[Code profile report](#)

**Generated Source Files**  
[asc\\_serial\\_pec.c](#)  
[c166\\_main.c](#)  
[fuel.c](#)  
[fuel\\_data.c](#)  
[fuel.h](#)  
[fuel\\_private.h](#)  
[fuel\\_types.h](#)  
[profile\\_vars.h](#)

# Code Profile Report

**Compiler: Tasking**

**Compiler Options: Not Available**

- [Entire Code Summary](#)
- [Entire Code Detail](#)
- [Memory MAP](#)

## Entire Code Summary

Module	Size [in bytes]
RAM	2331
ROM	10420

## Entire Code Detail

RAM	File	Section	Size [in bytes]
ASC_SERIAL_PEC_2_IR.....		ASC_SERIAL_PEC_2_IR.....	65
C166_US.....		C166_US.....	192
DISPATCH_BITDATA_SCT.....		DISPATCH_BITDATA_SCT.....	4
DISPATCH_RAM_DATA_SCT.....		DISPATCH_RAM_DATA_SCT.....	4
FUEL_6_NB.....		FUEL_6_NB.....	44
FUEL_ID_BA.....		FUEL_ID_BA.....	2
FUEL_ID_NB.....		FUEL_ID_NB.....	2
PROFILE_5_NB.....		PROFILE_5_NB.....	2012
PROFILE_ID_NB.....		PROFILE_ID_NB.....	6
ROM	File	Section	Size [in bytes]
?INTVECT.....		?INTVECT.....	512
ASC_SERIAL_PEC_1_PR.....		ASC_SERIAL_PEC_1_PR.....	368
BINARYSEARCH_S16_1_PR.....		BINARYSEARCH_S16_1_PR.....	74
C166_BSS.....		C166_BSS.....	20
C166_INIT.....		C166_INIT.....	38
C166_MAIN_1_PR.....		C166_MAIN_1_PR.....	356
C166_MAIN_2_CO.....		C166_MAIN_2_CO.....	8
DISPATCH_SCT.....		DISPATCH_SCT.....	52
DIV_S32_SAT_FLOOR_1_PR.....		DIV_S32_SAT_FLOOR_1_PR.....	262
DOTPRODUCT_S32S16_1_PR.....		DOTPRODUCT_S32S16_1_PR.....	60
FUEL_5_PR.....		FUEL_5_PR.....	3854
FUEL_DATA_1_NC.....		FUEL_DATA_1_NC.....	2810
FUEL_IR_BA.....		FUEL_IR_BA.....	2
FUEL_IR_NB.....		FUEL_IR_NB.....	2
INTERPOLATE EVEN S16 S16 SA		INTERPOLATE EVEN S16 S16 SA	192

OK Cancel Help Apply

## Parameter Tuning and Signal Logging

In this section...
“Methods For Parameter Tuning and Signal Logging” on page 2-18
“Using External Mode” on page 2-18
“Using a Third Party Calibration Tool” on page 2-27

### Methods For Parameter Tuning and Signal Logging

The Target Support Package IC1 product supports parameter tuning and signal logging either using Simulink external mode or with a third party calibration tool. In both cases the model must include a special block, the CAN Calibration Protocol block.

### Using External Mode

The Simulink external mode feature enables you to log signals and tune parameters without requiring a calibration tool. This section describes the steps for converting a model to use external mode.

External mode is supported using the CAN Calibration Protocol block and ASAP2 interface. The CAN Calibration Protocol block is used to communicate with the target, downloading parameter updates and uploading signal information. The ASAP2 interface is used to get information about where in the target memory a parameter or signal lives.

---

**Note** You must configure the host-side CAN application channel. See “Configuring the Host Vector CAN Application Channel ” on page 2-20.

---

To prepare your model for external mode, follow these steps:

- 1 Add a CCP driver block.
- 2 Add a Switch External Mode Configuration Block (for ease of use; you can also make changes manually).

- 3** Identify signals you want to tune, and associate them with `Simulink.Parameter` objects with `ExportedGlobal` storage class. It is important to set the data type and value of the `Simulink.Parameter` object. See “Using Supported Objects and Data Types” on page 2-20.
- 4** Identify signals you want to log, and associate them with `canlib.Signal` objects. It is important to set the data type of the `canlib.Signal`. See “Using Supported Objects and Data Types” on page 2-20.

For information about visualizing logged signal data, see “Viewing and Storing Signal Data” on page 2-22.

- 5** Load the the `Simulink.Parameter` and `canlib.Signal` data objects into the base workspace.
- 6** Configure the model for building by double-clicking the Switch External Mode Configuration block. In the dialog box, select **Building an executable**, and click **OK**.
- 7** Build the model, and download the executable to the target
- 8** After downloading the executable to the target, you can switch the model to external mode by double-clicking the Switch External Mode Configuration Block. In the dialog box that appears, select **External Mode**, and click **OK**.
- 9** You can now connect to the target using external mode by clicking the **Connect** button.
- 10** If you have set up tunable parameters, you can now tune them. See “Tuning Parameters” on page 2-21.

If you do not want to use the Switch External Mode Configuration block, you can configure for building and then external mode manually. For instructions, see “Manual Configuration For External Mode” on page 2-25.

See the following topics for more information:

- “Configuring the Host Vector CAN Application Channel” on page 2-20
- “Using Supported Objects and Data Types” on page 2-20
- “Tuning Parameters” on page 2-21

- “Viewing and Storing Signal Data” on page 2-22
- “Manual Configuration For External Mode” on page 2-25
- “Limitations” on page 2-26

### **Configuring the Host Vector CAN Application Channel**

External mode expects that the host-side CAN connection is using the 'MATLAB 1' application channel. To configure the application channel used by the Vector CAN drivers, enter the following at the MATLAB command line:

```
TargetsComms_VectorApplicationChannel.configureApplicationChannels
```

The Vector CAN Configuration tool appears. Use this tool to configure your host-side CAN channel settings.

If you try to connect using an application channel other than 'MATLAB 1', then you see the following warning in the command window:

```
Warning:  
It was not possible to connect to the target using CCP.  
An error occurred when issuing the CONNECT command.
```

### **Using Supported Objects and Data Types**

Supported objects:

- Simulink.Parameter for parameter tuning
- canlib.Signal for signal logging

Supported data types:

- uint8, int8
- uint16, int16
- uint32, int32
- single

You need to define data objects for the signals and parameters of interest for ASAP 2 file generation. For ease of use, create an m-file to define the data objects, so that you only have to set up the objects once.

To set up tuneable parameters and signal logging:

- 1 Associate the parameters to be tuned with `Simulink.Parameter` objects with `ExportedGlobal` storage class. It is important to set the data type and value of the `Simulink.Parameter` object. See the following m-code for an example of how to create such a `Simulink.Parameter` object for tuning:

```
stepSize = Simulink.Parameter;  
stepSize.DataType = 'uint8';  
stepSize.RTWInfo.StorageClass = 'ExportedGlobal';  
stepSize.Value = 1;
```

- 2 Associate the signals to be logged with `canlib.Signal` objects. It is important to set the data type of the `canlib.Signal`. The following m-code example shows how to declare such a `canlib.Signal` object for logging:

```
counter = canlib.Signal;  
counter.DataType = 'uint8';
```

- 3 Associate the data objects you have defined in the m-file with parameters or signals in the model. For the previous m-code examples, you could set the **Constant value** in a Source block to `stepSize`, and set a **Signal name** to `counter` in the Signal Properties dialog box. Remember that `stepSize` and `counter` are data objects defined in the m-code.

## Tuning Parameters

To tune a parameter, follow these steps:

- 1 Set `dataobject.value` in the workspace while the model is running in external mode. For example, to tune the parameter `stepSize` (that is, to change its value) from 1 to 2, enter the following at the command line:

```
stepSize.value = 2
```

You see output similar to the following:

```
stepSize =  
  
Simulink.Parameter (handle)  
    RTWInfo: [1x1 Simulink.ParamRTWInfo]  
    Description: ''  
    DataType: 'uint8'  
    Min: -Inf  
    Max: Inf  
    DocUnits: ''  
    Value: 2  
    Complexity: 'real'  
    Dimensions: [1 1]
```

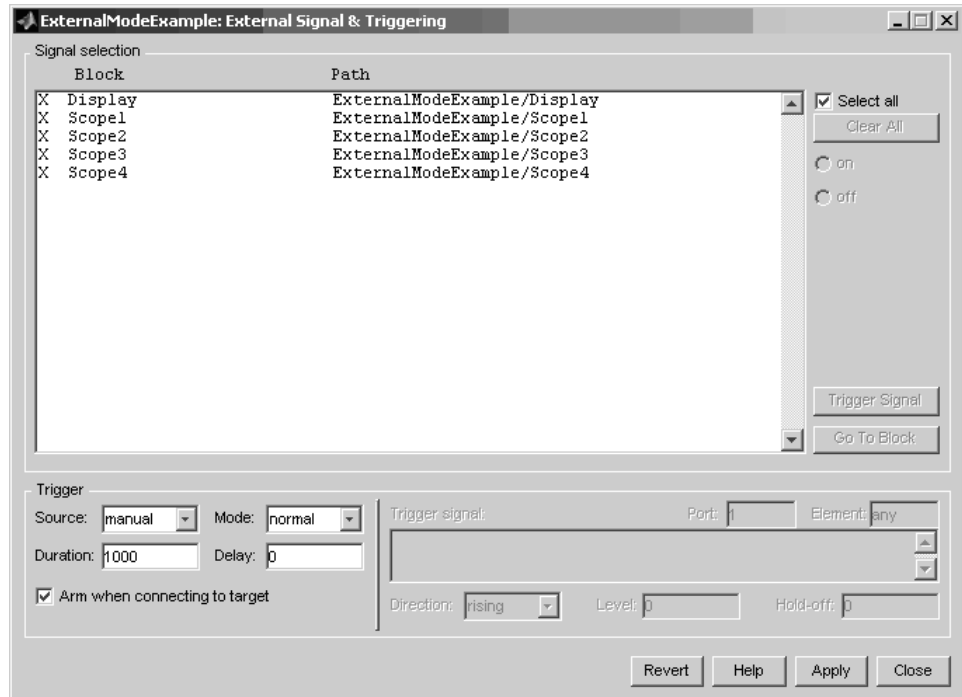
- 2 Return to your model, and update the model (press **Ctrl+D**) to apply the changed parameter.

### Viewing and Storing Signal Data

To view the logged signals attach a supported scope type to the signal (see “Limitations” on page 2-26 for supported scope types).

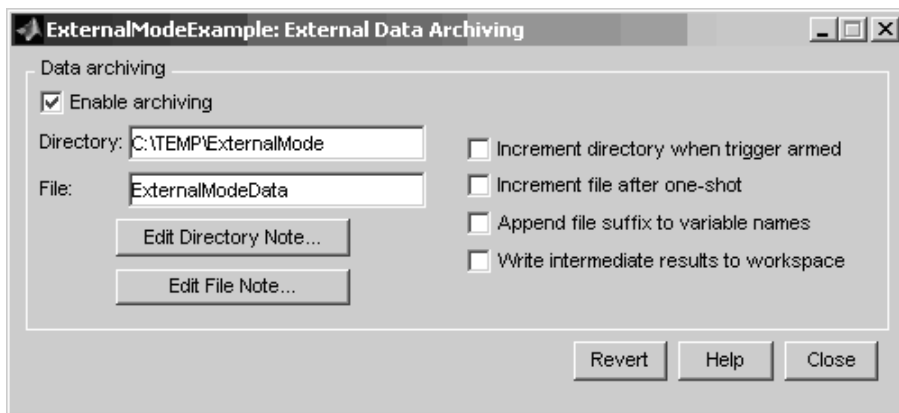
Select which signals you want to log by using the External Signal & Triggering dialog box. Access the External Mode Control Panel from the Tools menu, and click the **Signal & Triggering** button. By default, all displays appear as selected to be logged, as shown in the following example. Edit these settings if you do not want to log all displays. Individual displays can be selected manually.



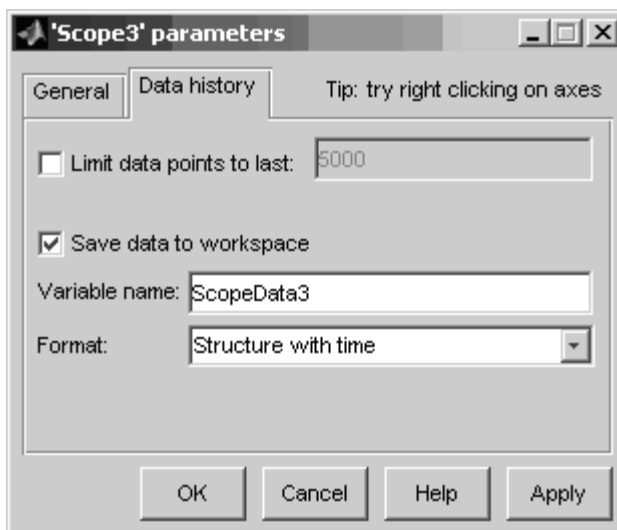


**Storing signal data for further analysis.** It is possible to store the logged data for further analysis in MATLAB.

- 1 To use the Data Archiving feature of external mode, click **Data Archiving** in the External Mode Control Panel. The External Data Archiving dialog box appears.



- a Select the check box **Enable archiving**
  - b Edit the **Directory** and **Filename** and any other desired settings.
  - c Close the dialog box.
- 2 Open the Scope parameters, and select the check box **Save data to workspace**.



- 3** You may want to edit the **Variable name** in the edit box. The data that is displayed on the scope at the end of the external mode session is available in the workspace with this variable name.

The data that was previously displayed in the scope is stored in `.mat` files as previously setup using Data Archiving.

For example, at the end of an external mode session, the following variable and files could be available in the workspace and current directory:

- A variable `ScopeData5` with the data currently displayed on the scope:

```
ScopeData5

ScopeData5 =

        time: [56x1 double]
        signals: [1x1 struct]
        blockName: 'mpc555rt_ccp/Scope1'
```

- In the current directory, `.mat` files for the three previous **Durations** of scope data:

```
ExternalMode_0.mat
ExternalMode_2.mat
ExternalMode_1.mat
```

## Manual Configuration For External Mode

As an alternative to using the Switch External Mode Configuration block, you can configure models manually for build and execution with external mode.

To configure a model to be built for external mode:

- 1** Select **Inline parameters** (under Optimization in the Configuration Parameters dialog box). The **Inline parameters** option is required for ASAP2 generation.
- 2** Select **Normal** simulation mode (in either the Simulation menu, or the drop-down list in the toolbar).

- 3 Select ASAP2 as the **Interface** (under **Real-Time Workshop, Interface**, in the **Data Exchange** pane, in the Configuration Parameters dialog box).

After you build the model, you can configure it for external mode execution:

- 1 Make sure **Inline parameters** are selected (under **Optimization** in the Configuration Parameters dialog box). The **Inline parameters** option is required for external mode.
- 2 Select **External** simulation mode (in either the **Simulation** menu, or the drop-down list in the toolbar).
- 3 Select **External** mode as the **Interface** (under **Real-Time Workshop, Interface**, in the **Data Exchange** pane, in the Configuration Parameters dialog box).

### Limitations

Logging of multiple signals feeding the same scope block is not supported. Instead, log each signal with its own scope block. These multiple signals can be on the same Simulink line, or can be multiple lines feeding the same scope (i.e. the scope can have multiple axes).

Only the following kinds of scopes are supported with External Mode Logging:

- Simulink Scope block
- Simulink Display block
- Viewer type: scope — To use this option, right-click a signal in the model, and select **Create & Connect Viewer > Simulink > Scope**. The other scope types listed there are not supported (e.g., floating scope).

Before connecting to external mode, you also need to right-click the signal, and select **Signal Properties**. In the dialog box, select the **Test point** check box, and click **OK**.

GRT is supported but only for parameter tuning.

If a signal comes directly from a Rate Transition block, external mode may fail to detect the correct sample time. To work around this, place a non-virtual

block (e.g., Contiguous Copy) in between the Rate Transition block and the signal to log.

It is not possible to log signals with very fast sample times (e.g., 0.0001) without losing data.

Subsystem builds are not supported for external mode, only top-level builds are supported.

Logging and tuning of nonscalars is not supported. It is possible to log nonscalar signals by breaking the signal down into its scalar components. For an example of how to do this signal deconstruction, see the CCP demo models, which use a Demux and Signal Conversion block with contiguous copy.

Logging and tuning of complex numbers is not supported. It is possible to work with complex numbers by breaking the complex number down into its real and imaginary components. This breakdown can be performed using the following blocks in the Simulink Math Operations library: Complex to Real-Imag, Real-Imag to Complex, Magnitude-Angle to Complex, Complex to Magnitude-Angle.

## Using a Third Party Calibration Tool

The Target Support Package IC1 product allows an ASAP2 data definition file to be generated during the code generation process. This file can be used by a third party tool to access data from the real-time application while it is executing.

ASAP2 is a data definition standard by the Association for Standardization of Automation and Measuring Systems (ASAM). ASAP2 is a standard description for data measurement, calibration, and diagnostic systems. The Target Support Package IC1 product lets you export an ASAP2 file containing information about your model during the code generation process. See also “Compatibility with Calibration Packages” on page 7-30.

Before you begin generating ASAP2 files with the Target Support Package IC1 product, you should read the “Generating ASAP2 Files” section of the Real-Time Workshop documentation. That section describes how to define the signal and parameter information required by the ASAP2 file generation process.

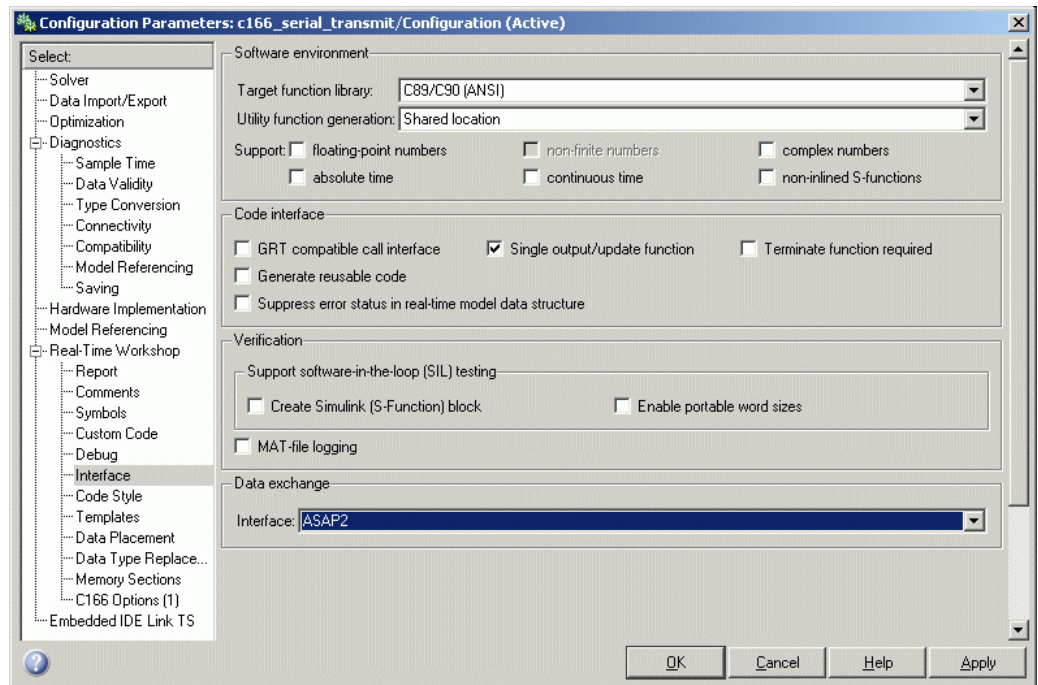
Select the ASAP2 option before the build process as follows:

**1 Select Simulation > Configuration Parameters.**

The Configuration Parameters dialog box appears.

**2 Select Interface (under Real-Time Workshop) in the tree.**

**3 Select the ASAP2 option from the Interface drop-down menu, in the Data exchange frame, as shown following.**



**4 Click Apply.**

The build process creates an ASAM-compliant ASAP2 data definition file for the generated C code.

- The standard Real-Time Workshop ASAP2 file generation does not include the memory address attributes in the generated file. Instead,

it leaves a placeholder that must be replaced with the actual address by postprocessing the generated file.

- The map file options in the template project need to be set up a certain way for this procedure to work. If you have created your own template projects, and you do not have the correct settings, you see the following instructions:

```
Warning: It was not possible to do ASAP2 processing on your
.map file.This is because your EDE project template is not
configured to generate a .map file in the correct format.
To generate a .map file in the correct format you need to
setup the following options in your EDE project template:
Generate section map should be checked on
Generate register map should be checked off
Generate symbol table should be checked on
Format list file into pages should be checked off
Generate summary should be checked off
Page width should be equal to 132 characters
Symbol colums should be 1
You can change these options via Project -> Project Options
-> Linker/Locator -> Map File -> Map File Format.
```

The Target Support Package IC1 product performs this postprocessing for you. To do this, it first extracts the memory address information from the map file generated during the link process. Secondly, it replaces the placeholders in the ASAP2 file with the actual memory addresses. This postprocessing is performed automatically and requires no additional input from you.

For an example of a model that is configured to generate an ASAP2 file, see `c166_ccp`.





# Integrating Your Own Device Drivers

---

This section includes the following topics:

- “Integrating Hand-Coded Device Drivers with a Simulink Model” on page 3-2
- “Preparing Input and Output Signals to the Device Driver Functions ” on page 3-4
- “Calling the Device Driver Functions from c166\_main.c” on page 3-7
- “Adding the I/O Driver Source to the List of Files to Build” on page 3-10
- “Tutorial: Using the Example Driver Functions” on page 3-12

## Integrating Hand-Coded Device Drivers with a Simulink Model

The Target Support Package IC1 product has a limited set of I/O device driver blocks. This means that, for most applications, it is necessary to write some device driver code by hand.

This approach requires the following steps:

- 1** Identify the model inputs/outputs that must be read from/written to device driver functions.
- 2** Set the data type and storage class for each input or output signal so that it is compatible with your device driver code.
- 3** Use the hooks provided in the automatically generated `c166_main.c` to call your device driver initialization, input, and output functions.
- 4** Add your device driver source code to the list of files that must be included in the build process.

Each of these steps is described in the following sections. An example model is provided: `c166_user_io`.

An alternative approach is to create Simulink I/O blocks that automatically generate the device driver code. This approach may be worth considering if you need to reconfigure the I/O behavior frequently. If you want to take this alternative approach, you should consult the documentation on S-functions and TLC. See the section *Developing Device Drivers for Embedded Targets* in the document *Developing Embedded Targets in the Real-Time Workshop Embedded Coder* documentation.

A useful tool for creating C166 device drivers is the freeware Digital Application Engineer DAvE from Infineon. You can find this at the following URL:

<http://www.infineon.com/dave>

Using this package along with the hardware User's Manual greatly eases the task of developing your own device driver code.

For a guide to creating device drivers, see “Developing Device Drivers for Embedded Targets” in the Developing Embedded Targets for Real-Time Workshop Embedded Coder documentation.

## Preparing Input and Output Signals to the Device Driver Functions

Structure your model similarly to `c166_user_io`. Place the control algorithm that will be targeted onto the C166 microcontroller hardware in a separate subsystem. Before generating code, you can run this model in closed-loop simulation; this allows you to validate the correct behavior of your control algorithm before running it in real time.

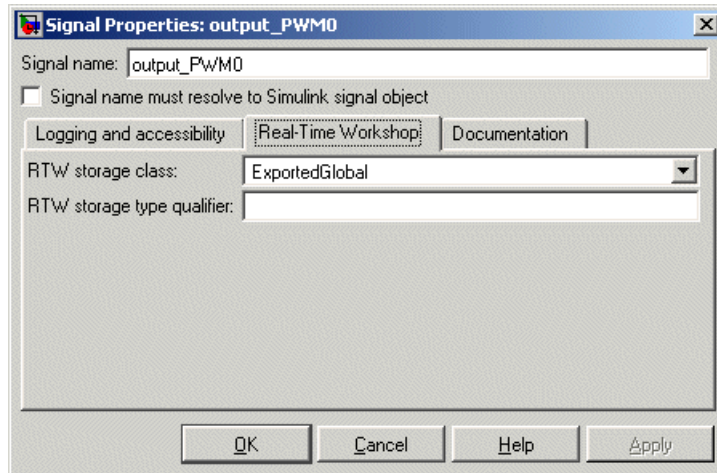
When structuring your model in this way, you should make sure that all the input and output signals to the control algorithm are channeled through top-level input or output ports in the control algorithm subsystem.

By default, when you generate code for the control algorithm subsystem, the Real-Time Workshop build process chooses variable names and data structures for each of the top-level input and output signals. However, in this case, you must ensure that the variables are global, and that their names and data structures match those that are required by the hand-written device driver functions.

The example model `c166_user_io` illustrates some alternative ways to achieve this. The simplest method is to

- 1 Select one of the signals in your model connected to a top-level output port in the control algorithm subsystem. As an example, open the demo `c166_user_io`.
- 2 Open the controller subsystem.
- 3 Click the `output_PWM0` signal.
- 4 Select the menu item **Edit > Signal Properties**.

The **Signal Properties** dialog box appears, as in the example following.



- 5 Enter the required variable name for your signal in the **Signal name** edit box. This must match the variable name required by your hand written device driver functions.
- 6 Click the **Real-Time Workshop** tab and select `ExportedGlobal` from the **RTW storage class** drop-down menu.

When you generate code for this model, the Real-Time Workshop build process uses the variable name that you have specified and creates an `extern` declaration in the model header file. By using a `#include` directive to include this model header file in your device driver source code, it is possible for the device driver functions to read or write this variable that is defined in the Real-Time Workshop generated code.

A more sophisticated approach is to use custom storage classes. By using custom storage classes, you can collect a number of input or output variables together into a C struct, resulting in more readable code. The LED output signal in the `c166_user_io` uses a custom storage class, which uses a single bit in a bitfield variable. See “Tutorial: Using the Example Driver Functions” on page 3-12 for details about the different ways the model variables are defined and referenced to interface the hand-coded driver functions and the automatically generated code.

By defining your own custom storage classes, you have complete control over the data structures that are used for any signal in the model. See the custom storage class documentation in the Real-Time Workshop Embedded Coder documentation for more details.

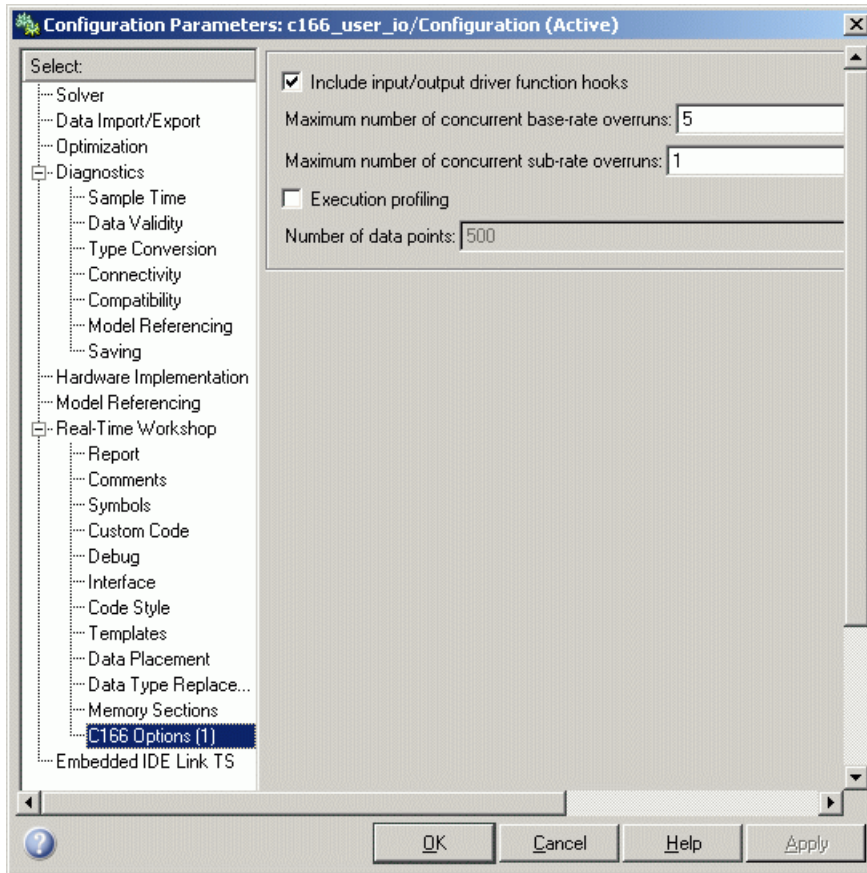
## Calling the Device Driver Functions from c166\_main.c

You should check the option to include I/O driver function hooks. When you use Real-Time Workshop code generation for this model, it includes some extra calls to user-supplied I/O device driver functions:

- 1** Select **Simulation > Configuration Parameters**.

The Configuration Parameters dialog box appears.

- 2** Select **C166 Options (1)**, under **Real-Time Workshop** in the tree, as shown in the example below.



**3** Select the check box option for including I/O driver function hooks.

These functions are

`user_io_initialize` — called following model initialization

`base_rate_model_inputs` — read model inputs, called at the base sample rate

`base_rate_model_outputs` — write model outputs, called at the base sample rate



`sub_rate_i_model_inputs` — read model inputs, called at the start of sub-rate *i*, where *i*=1, 2, ...

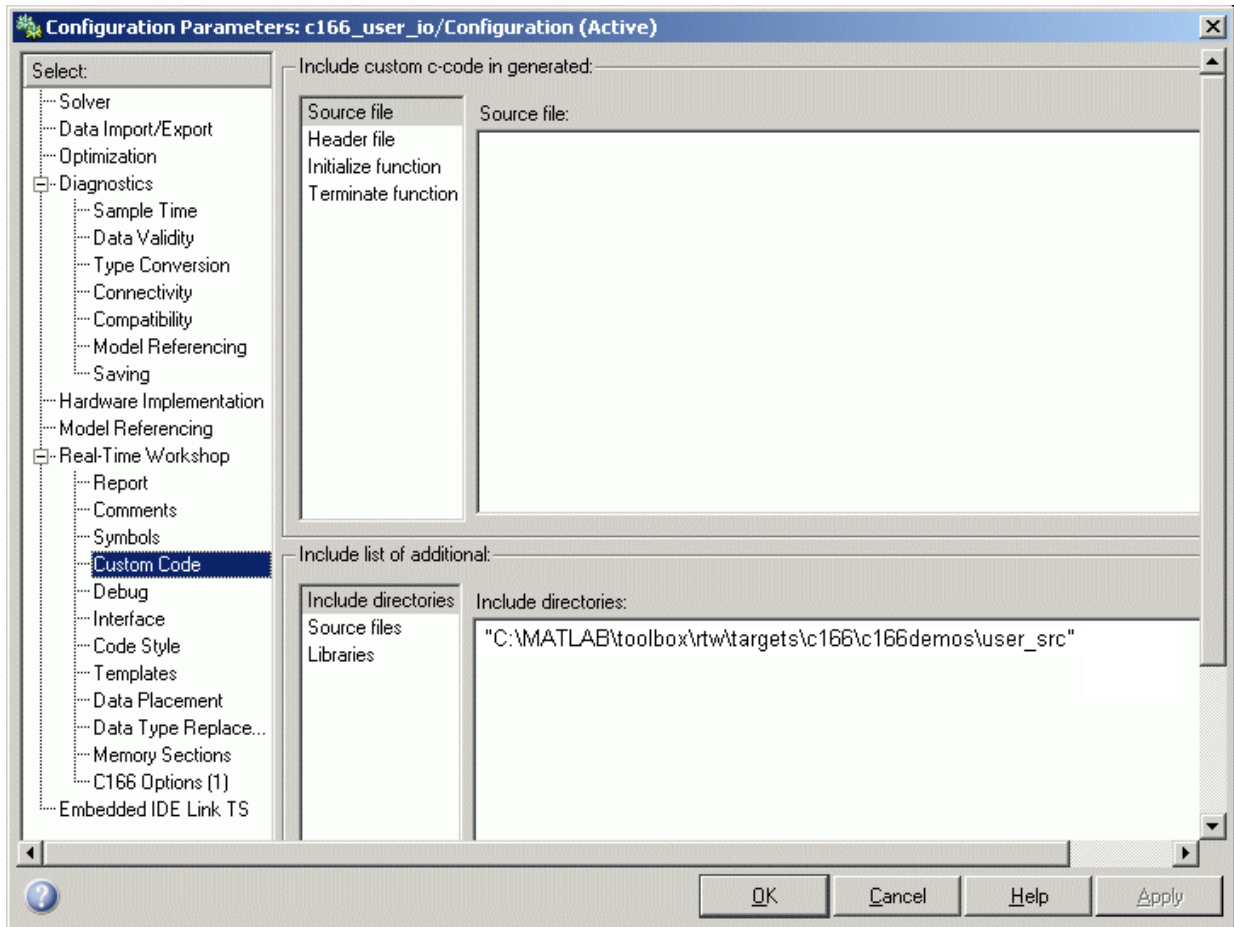
`sub_rate_i_model_outputs` — write model outputs, called at the start of sub-rate *i*, where *i*=1, 2, ...

If you are using the automatically generated `c166_main.c`, then these function names are fixed.

For an example implementation of these functions, open the model `c166_user_io` and follow the link to open the I/O driver source files. These are described in “Tutorial: Using the Example Driver Functions” on page 3-12.

## Adding the I/O Driver Source to the List of Files to Build

You must tell the Real-Time Workshop build process to compile and link the I/O driver source files that you have written. You do so by adding the files to the custom code dialog box. Access the Configuration Parameters dialog box, look under **Real-Time Workshop > Custom Code**, and add the necessary **Include Directories** and **Source Files**, as shown in the following figure.



You are now ready to build your model and run it in real time.

You can examine an example of this in the example model `c166_user_io`. See the instructions in “Tutorial: Using the Example Driver Functions” on page 3-12. Step 8 shows you how to specify the location of your own hand-coded drivers.

# Tutorial: Using the Example Driver Functions

The example model `c166_user_io` demonstrates how to integrate user-defined device driver code. In this tutorial, you generate code from the controller subsystem, which automatically downloads and runs on the target.

The model `c166_user_io` illustrates three alternative methods for using global variables to interface the hand-written driver functions with the Real-Time Workshop automatically generated code. The three different methods are illustrated by these signals:

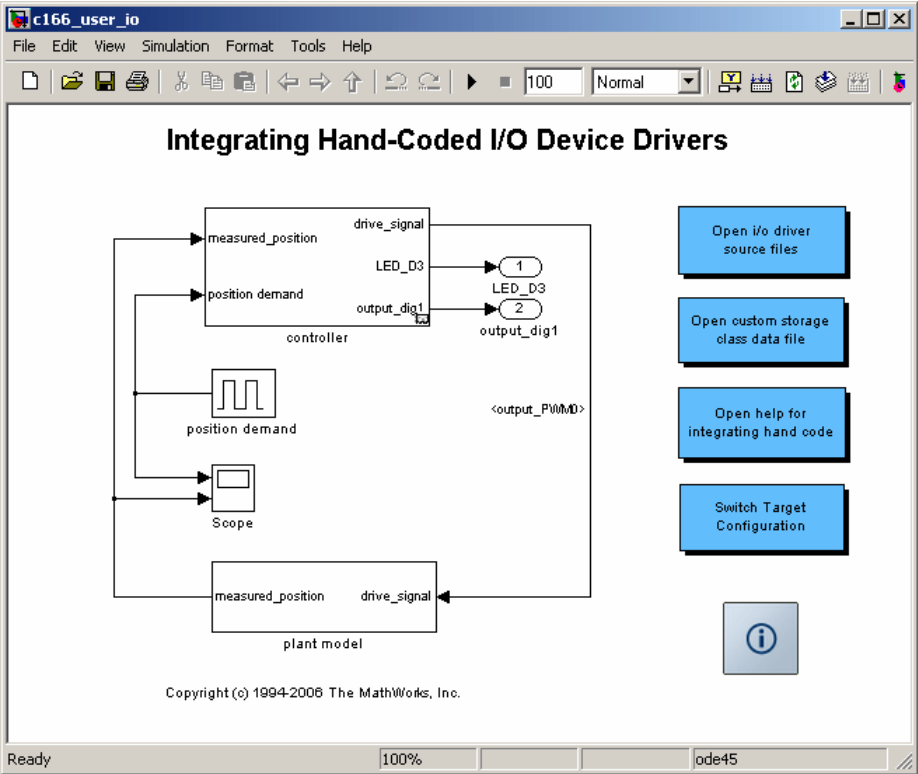
- `input_adc0`
- `output_PWM0`
- `output_led_D3`

For `input_adc0`, the variable is defined in the hand code and referenced in the Real-Time Workshop code.

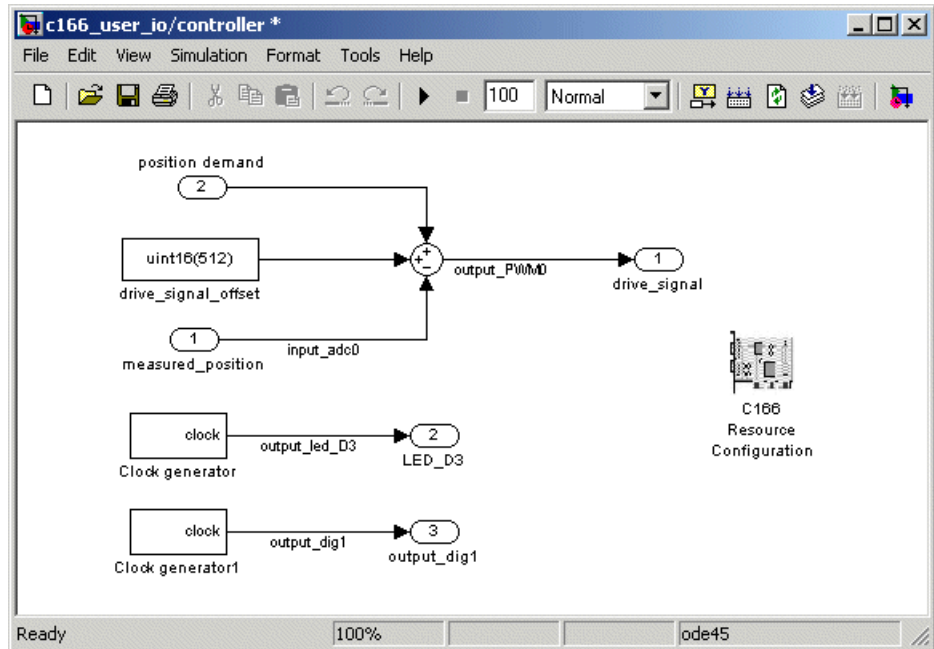
For `output_PWM0`, the variable is defined in the Real-Time Workshop code and referenced in the hand code.

For `output_led_D3`, a more sophisticated approach is used, involving custom storage classes. In this case, the variable is again defined in the Real-Time Workshop code and referenced by the hand code; the difference is that the variable is defined and referenced as a bitfield using C166 microcontroller bit-addressable memory:

- 1 Open the model `c166_user_io`.

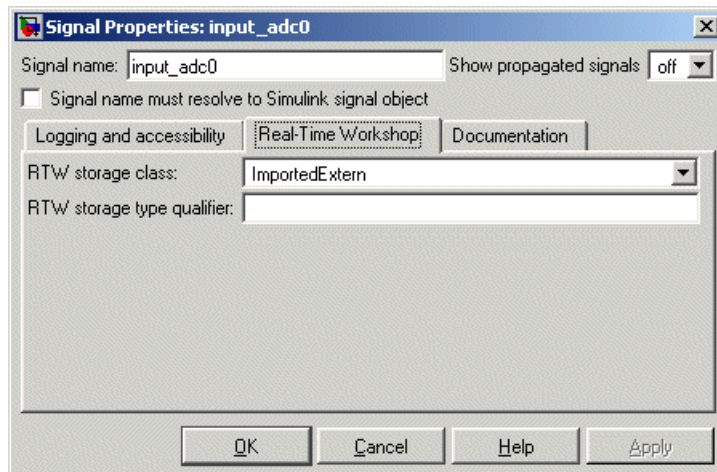


2 Open the controller subsystem by double-clicking and select the signal `input_adc0`.



3 Select the menu item **Edit > Signal Properties**.

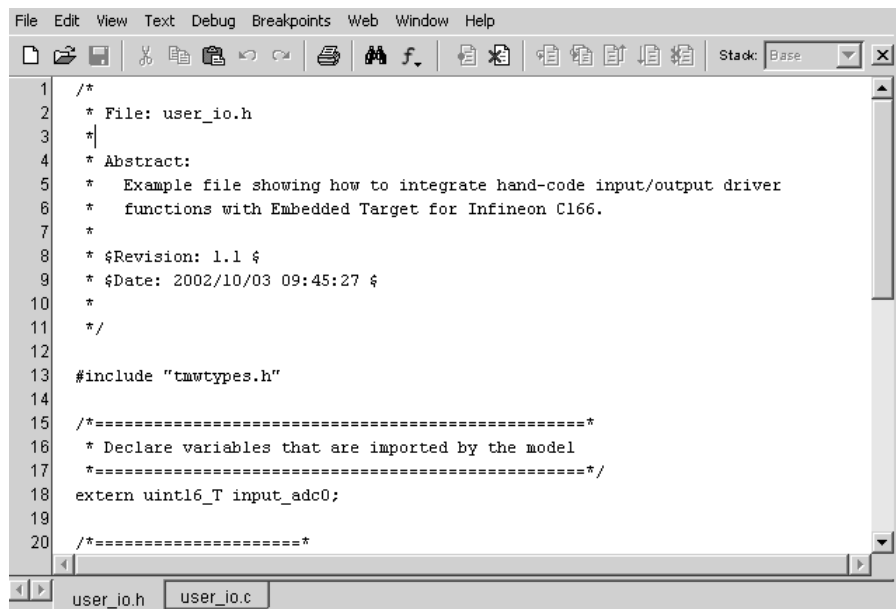
The Signal Properties dialog box appears.



Click the **Real-Time Workshop** tab and observe that the **RTW storage class** is `ImportedExtern`. When you generate code for this model, the specified variable name `input_adc0` is used, and an extern declaration is created in the model header file. Since the Real-Time Workshop storage class is `ImportedExtern`, this variable must be defined in the hand-written driver code. When you open the file `user_io.c` in the next step, you will find the line `uint16_T input_adc0` that provides this definition.

- 4 In the top level model, double-click the link **Open the i/o driver source files**.

Two source files open in the MATLAB editor, `user_io.h` and `user_io.c`.



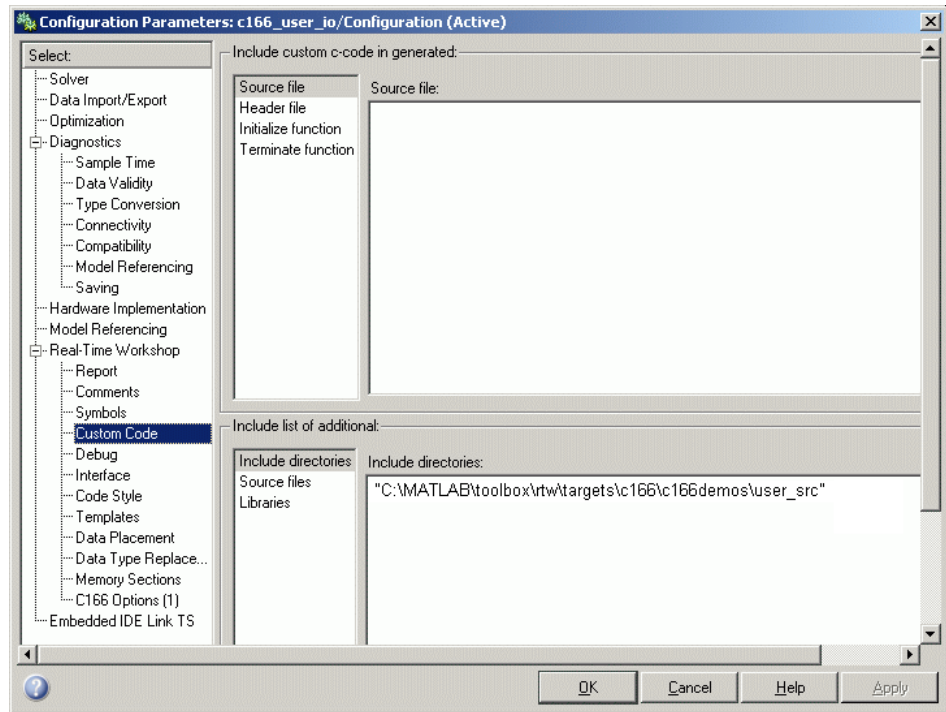
```

File Edit View Text Debug Breakpoints Web Window Help
[Icons] Stack: Base X
1  /*
2  * File: user_io.h
3  *
4  * Abstract:
5  * Example file showing how to integrate hand-code input/output driver
6  * functions with Embedded Target for Infineon C166.
7  *
8  * $Revision: 1.1 $
9  * $Date: 2002/10/03 09:45:27 $
10 *
11 */
12
13 #include "tmwtypes.h"
14
15 /*=====
16 * Declare variables that are imported by the model
17 *=====*/
18 extern uint16_T input_adc0;
19
20 /*=====
[Navigation] user_io.h user_io.c

```

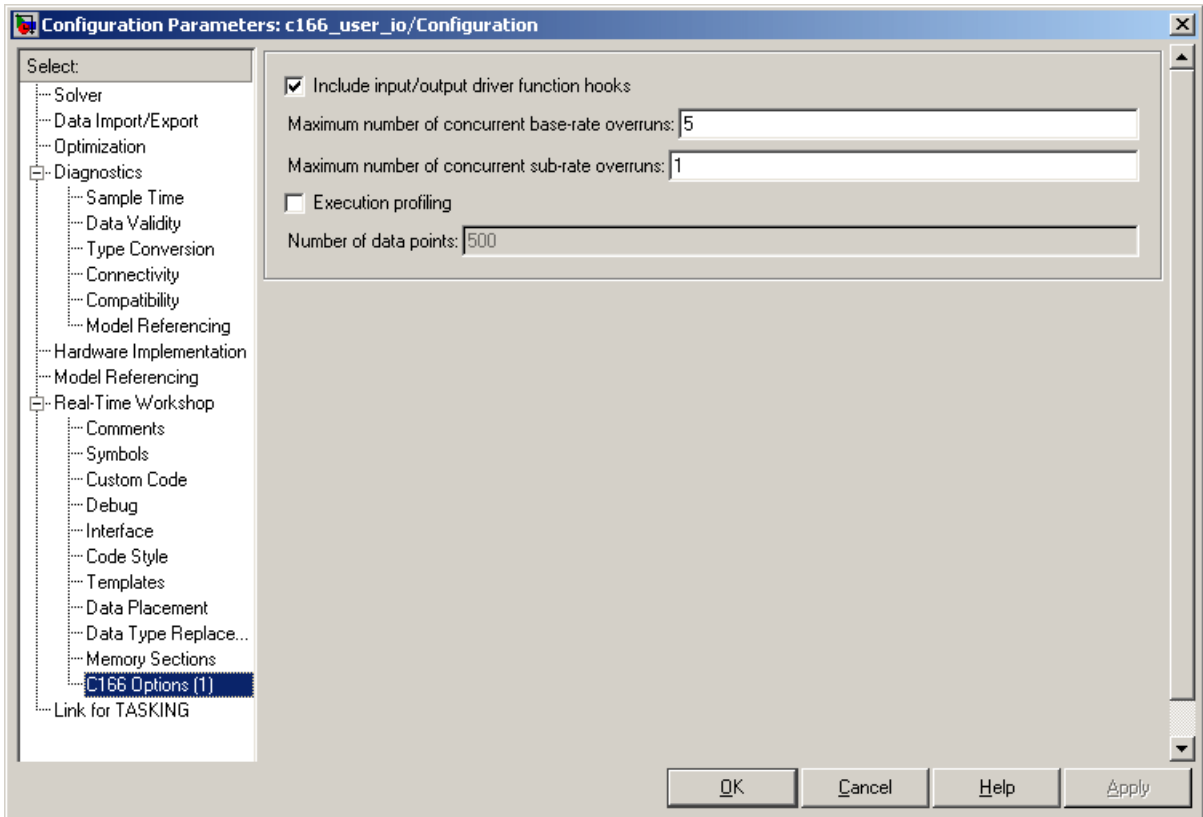
- 5 Click the `user_io.h` tab, as shown above. Here you can see `extern uint16_T input_adc0` under the heading `Declare variables that are imported by the model`. Also look at the `#include` directive in `user_io.c`. The extern declaration and incorporating the header file into the build makes it possible for the device driver functions to read or write this variable that is defined in the Real-Time Workshop generated code.

- 6 You need to instruct the Real-Time Workshop product to compile and link the hand-coded I/O driver source files in the build process. You do so by adding the files to the custom code dialog box. Access the Configuration Parameters dialog box, select **Real-Time Workshop > Custom Code** in the tree, and review the Include Directories and Source Files, as shown in the following figure.



- 7 Select **C166 Options (1)** (under **Real-Time Workshop** in the tree). Observe the selected option **Include input/output driver function hooks**.





This instructs the Real-Time Workshop build process to include extra calls to the user-supplied I/O device driver functions when code is generated for this model.

- 8 Select **Interface** in the tree. Observe the option **Floating-point numbers** is *not* selected.


If your model does not use floating point, you should make sure this option is not checked to use integer code only. Using only integer code results in smaller code size and faster real-time execution. It also speeds up the build process because libraries that are used only by floating-point applications are not included.

Explore the `user_io.c` file. This example file is intended to show you some hand-coded input/output driver functions and how they can be integrated with the Target Support Package IC1 product.

You can see sections for initializing these input/output drivers: ADC, digital I/O, and Pulse Width Modulation (PWM).

- 9 Close the Signal Properties dialog box and Configuration Parameters dialog box if they are still open.

Prior to generating code, you can run the model in closed-loop simulation;

just click Start Simulation (  ) in the toolbar. You can open the Scope block to see the model output. If you use this model as a basis for integrating your own device driver code, this closed-loop simulation allows you to validate the correct behavior of your control algorithm before running it in real time.

- 10 Generate code by right-clicking the controller subsystem and selecting **Real-Time Workshop > Build Subsystem**.
- 11 Click **Build** in the Build code for Subsystem: Controller dialog box that appears. Watch the messages as the process proceeds and code is generated.

If you are using a Phytex phyCORE module with HD200 development board, the digital output is connected to the LED D3. You can see successful execution of the code when the LED blinks.

# Custom Storage Class for C166 Microcontroller Bit-Addressable Memory

---

This section contains the following topics:

- “Specifying C166 Microcontroller Bit-Addressable Memory” on page 4-2
- “Using the Bitfield Example Model” on page 4-3

# Specifying C166 Microcontroller Bit-Addressable Memory

Target Support Package IC1 allows you to take advantage of Infineon C166 microcontroller bit-addressable memory. The example model `c166_bitfields` demonstrates this. By using bit-addressable memory, the compiler is able to use special assembler instructions that significantly reduce code size and increase execution speed.

---

**Note** This feature requires the Real-Time Workshop Embedded Coder product.

---

This is done by using the custom storage class `SimulinkC166.Signal`. To specify that a signal in the model should use bit-addressable memory, you must perform the following steps:

- 1 Ensure that the signal has the Simulink data type 'boolean'.
- 2 Attach a label to the signal, either by using **Edit > Signal Properties** or by double-clicking the signal and typing in the name directly; this label will be used as the bitfield variable name in the generated code.
- 3 Create a new Simulink data object of type `SimulinkC166.Signal` with the same name as the signal label. See the file `c166bitfielddata.m` for an example.
- 4 Select **View > Model Explorer** and click the base workspace to inspect all the Simulink data objects that are available to the model.
- 5 Build the model.

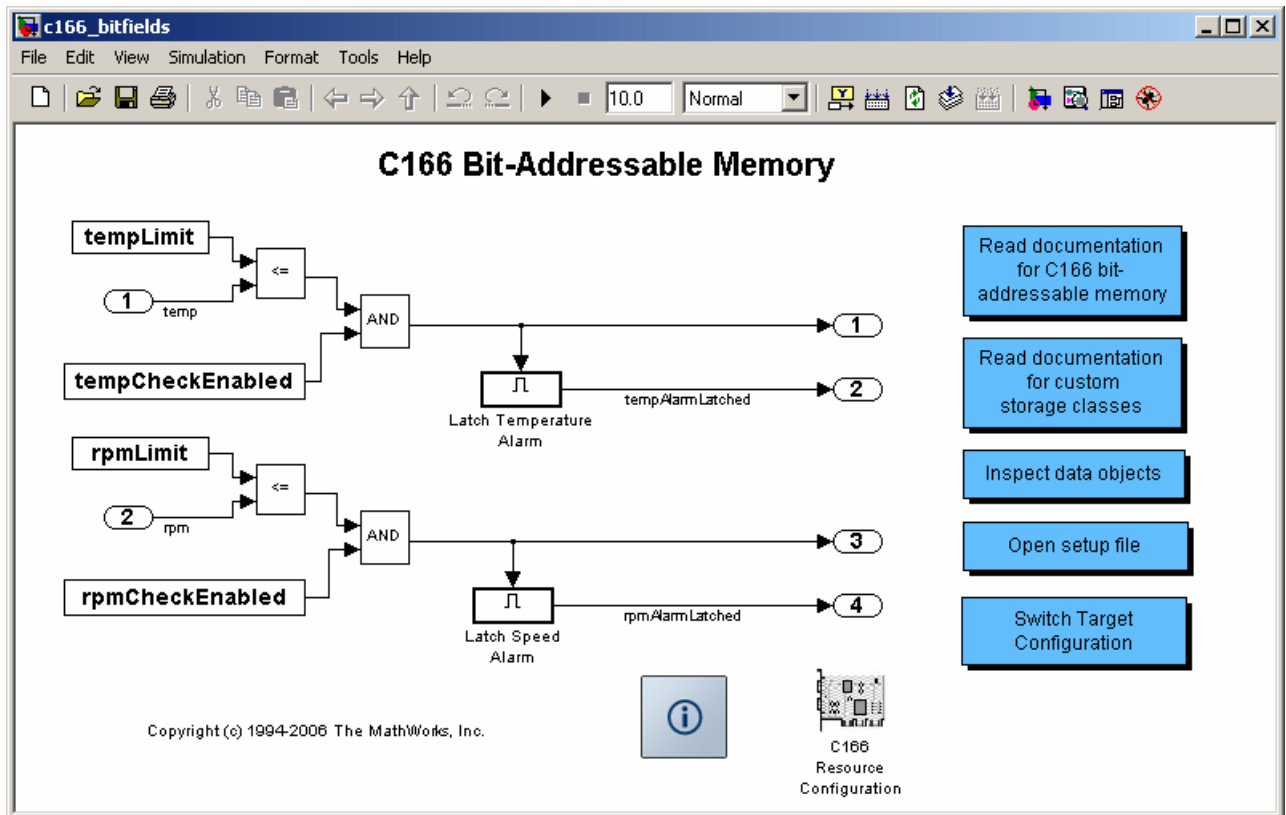
One of the signals in the demo model `c166_user_io` also uses the custom storage class `SimulinkC166.Signal` to specify that the signal uses bit-addressable memory. You can compare this with the `c166_bitfields` example; it is included in the steps in “Using the Bitfield Example Model” on page 4-3.

## Using the Bitfield Example Model

You can use the example model `c166_bitfields` to see the automatic debugger start at the end of the build.

Follow these steps:

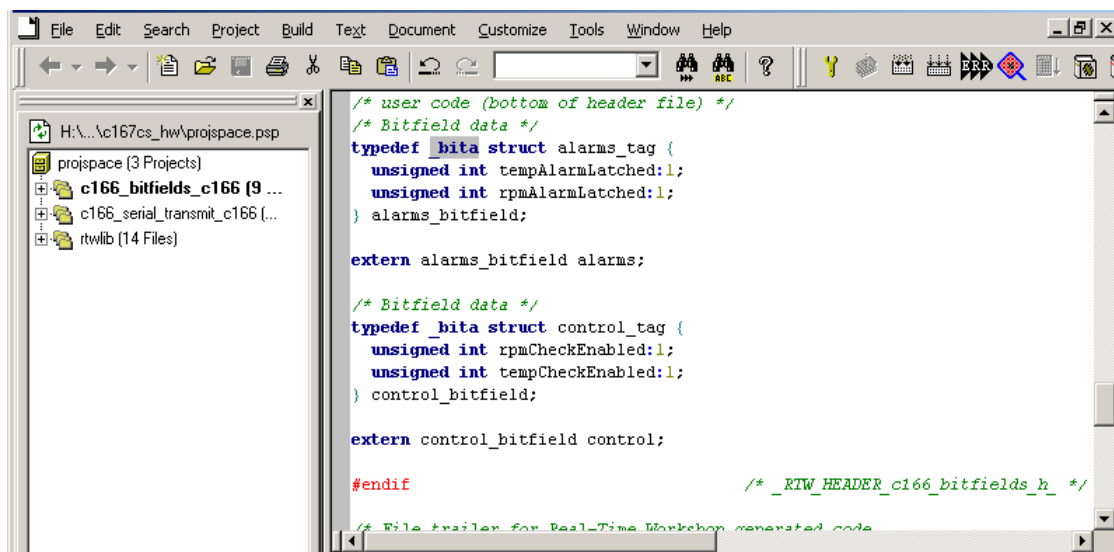
- 1 Open `c166_bitfields`.



- 2 Press **Ctrl+B** to build the model.

- 3 Examine the project generated code in the TASKING EDE:
  - a Select **Search > Multiple Sources**.

- b In the dialog box, select Project Space under Multiple Sources, and enter `_bita` for the search string.



The screenshot shows a MATLAB editor window with a menu bar (File, Edit, Search, Project, Build, Text, Document, Customize, Tools, Window, Help) and a toolbar. The left pane shows a project tree with the following structure:

- H:\...c167cs\_hw\projSPACE.psp
  - projSPACE (3 Projects)
    - c166\_bitfields\_c166 (9 ...)
    - c166\_serial\_transmit\_c166 (...)
    - rtwlib (14 Files)

The main editor area contains the following C code:

```
/* user code (bottom of header file) */  
/* Bitfield data */  
typedef _bita struct alarms_tag {  
    unsigned int tempAlarmLatched:1;  
    unsigned int rpmAlarmLatched:1;  
} alarms_bitfield;  
  
extern alarms_bitfield alarms;  
  
/* Bitfield data */  
typedef _bita struct control_tag {  
    unsigned int rpmCheckEnabled:1;  
    unsigned int tempCheckEnabled:1;  
} control_bitfield;  
  
extern control_bitfield control;  
  
#endif                                     /* _RTW_HEADER_c166_bitfields_h_ */  
  
/* File trailer for Real-Time Workshop generated code
```

- 4 You can double-click **Open setup file** in the model to open the file `c166bitfielddata.m` in the MATLAB editor.

```

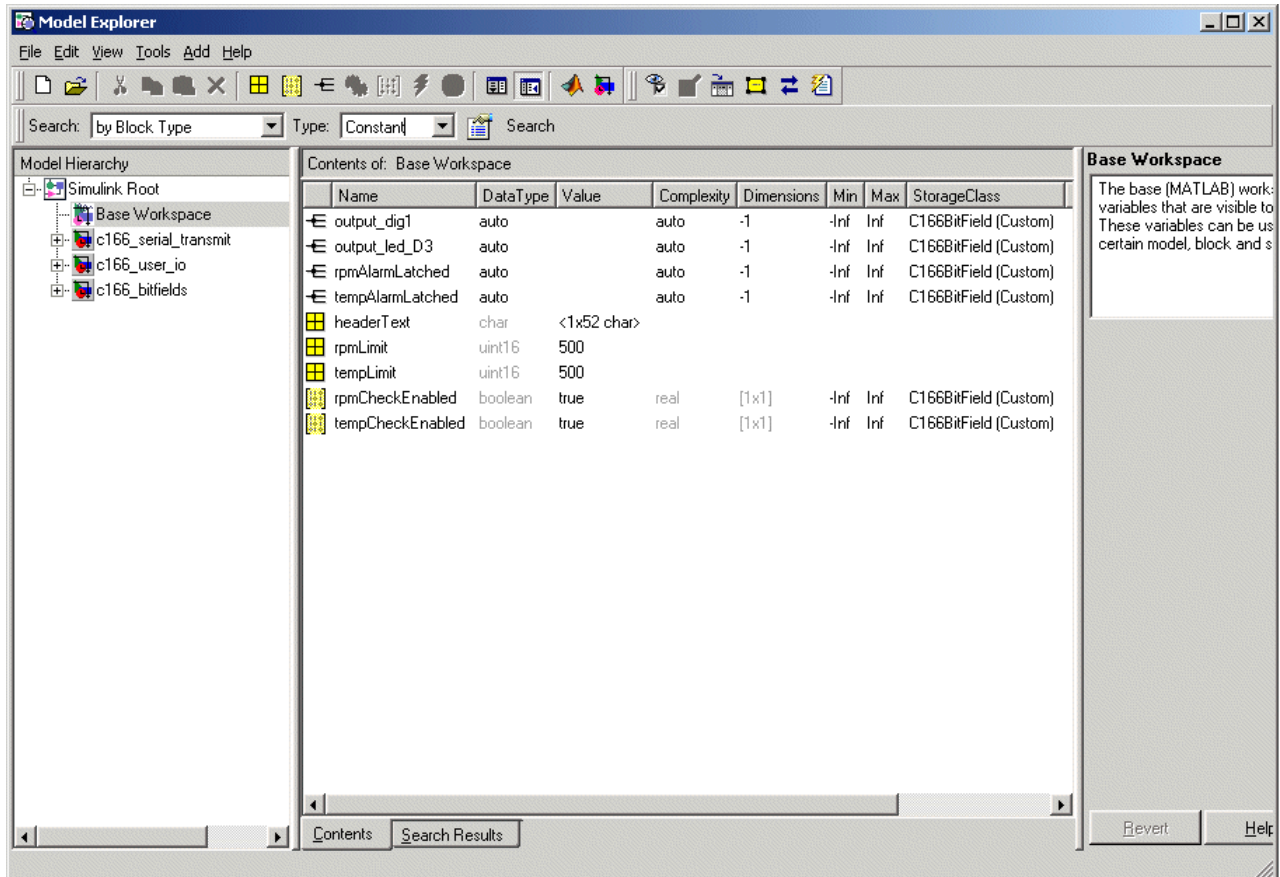
File Edit View Text Debug Breakpoints Web Window Help
Stack Base
1 | % C166BITFIELDDDATA create data for C166 bitfield demo model
2 |
3 | % Copyright 2002 The MathWorks, Inc.
4 | % $Revision: 1.1 $
5 | % $Date: 2002/10/03 09:45:24 $
6 |
7 | cscdemoclearws
8 |
9 | tempAlarm = SimulinkC166.Signal;
10 | tempAlarm.RTWInfo.CustomAttributes.BitFieldName = 'alarms';
11 |
12 | tempAlarmLatched = SimulinkC166.Signal;
13 | tempAlarmLatched.RTWInfo.CustomAttributes.BitFieldName = 'alarms';
14 |
15 | rpmAlarm = SimulinkC166.Signal;
16 | rpmAlarm.RTWInfo.CustomAttributes.BitFieldName = 'alarms';
17 |
18 | rpmAlarmLatched = SimulinkC166.Signal;
19 | rpmAlarmLatched.RTWInfo.CustomAttributes.BitFieldName = 'alarms';
20 |
21 | templimit = uint16(500);

```

This file creates a new Simulink data object using the custom storage class `SimulinkC166.Signal`. By using custom storage classes, you can collect a number of input or output variables together into a C struct, resulting in more readable code. By defining your own custom storage classes, you have complete control over the data structures that are used for any signal in the model. See the custom storage class documentation in the *Real-Time Workshop Embedded Coder User's Guide* for more details. You can double-click **Read documentation for custom storage classes** in the model to go directly to the relevant Real-Time Workshop Embedded Coder help section.

- 5 You can double-click **Inspect data objects** to inspect all the Simulink data objects that are available to the model.

## 4 Custom Storage Class for C166® Microcontroller Bit-Addressable Memory



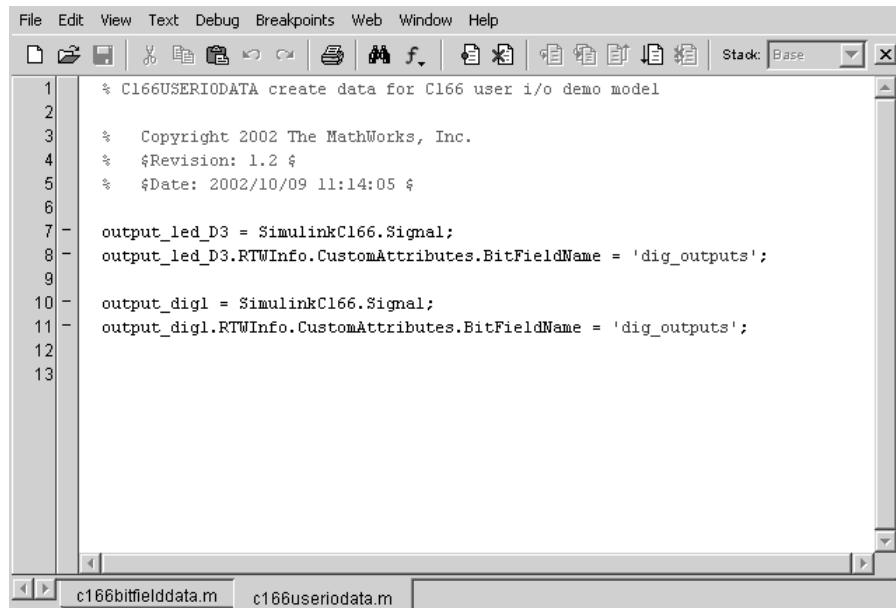
Here you can see the SimulinkC166.Signal data object and you can click on each object to inspect the properties.

**6** One of the signals in the demo model `c166_user_io` also uses the custom storage class `SimulinkC166.Signal` to specify that the signal uses bit-addressable memory. Open `c166_user_io`.

**7** Double-click **Open custom storage class data file**.

The file `c166useriodata.m` opens in the MATLAB editor.





```
File Edit View Text Debug Breakpoints Web Window Help
Stack Base
1 % C166USERIODATA create data for C166 user i/o demo model
2
3 % Copyright 2002 The MathWorks, Inc.
4 % $Revision: 1.2 $
5 % $Date: 2002/10/09 11:14:05 $
6
7 output_led_D3 = SimulinkC166.Signal;
8 output_led_D3.RTWInfo.CustomAttributes.BitFieldName = 'dig_outputs';
9
10 output_dig1 = SimulinkC166.Signal;
11 output_dig1.RTWInfo.CustomAttributes.BitFieldName = 'dig_outputs';
12
13
c166bitfielddata.m c166useriodata.m
```

Compare with `c166bitfielddata.m`.

For more details on the variables in this model, see “Tutorial: Using the Example Driver Functions” on page 3-12.



# Execution Profiling

---

This section contains the following topics:

- “Overview of Execution Profiling” on page 5-2
- “Real-Time Workshop Options for Execution Profiling” on page 5-6
- “Multitasking Demo Model” on page 5-10

## Overview of Execution Profiling

### In this section...

“Introducing Execution Profiling” on page 5-2

“The Profiling Command” on page 5-3

“Definitions” on page 5-5

“Execution Profiling Blocks” on page 5-5

### Introducing Execution Profiling

The Target Support Package IC1 product provides a set of utilities for recording, uploading, and analyzing execution profile data for timer-based tasks and asynchronous Interrupt Service Routines (ISRs). With these utilities, you can

- Generate a graphical display that shows when timer-based tasks and interrupt service routines are activated, preempted, resumed, and completed.
- Generate a report with information on
  - Maximum number of overruns for each timer-based task since model execution began
  - Maximum turnaround time for each timer-based task since model execution began
  - Analysis of profiling data for timer-based tasks and asynchronous interrupts over a period of time

To perform execution-profiling analysis on a model, you must perform the following steps:

- 1 Place a copy of the appropriate execution profiling block in your model:
  - Execution Profiling via ASC0 if using a serial connection
  - Execution Profiling via CAN A if using CAN with a C166 processor
  - Execution Profiling via TwinCAN A if using CAN with an XC16x processor variant

- 2** Select the **Execution profiling** option under **Real-Time Workshop** options in the Configuration Parameters dialog box. See “Real-Time Workshop Options for Execution Profiling” on page 5-6.
- 3** Connect the target processor to your host PC (with a serial or CAN cable).
- 4** Build, download, and run the model.
- 5** Initiate execution profiling by running the command `profile_c166`. See below for more information on the profiling command.

Two forms of execution profiling are provided:

- 1** The worst-case values for task turnaround times and number of concurrent task overruns since model execution began are updated whenever a previous worst-case value is exceeded.
- 2** A snapshot of task and ISR activity may be recorded over a period of time; the length of this period depends on how much memory is reserved to log the data.

## The Profiling Command

Use the profiling command as follows:

```
profile_c166(connection)
```

Specify your connection as 'can' or 'serial', to collect data via a CAN or serial connection between the target and the host computer. Make sure the model includes the appropriate C166 execution profiling block (CAN or ASC0), to provide an interface between the target-side profiling engine and the host-side computer from which this command is run.

`PROFDATA = profile_c166(connection)` collects and displays execution profiling data from a C166 target microcontroller that is running a suitably configured application generated by the Target Support Package IC1 product. `PROFDATA` contains the execution profiling data in the format documented by `exprofile_unpack`.

The data collected is unpacked then displayed in a summary HTML report and as a MATLAB graphic.

To use the serial connection, the C166 board must be connected via a serial cable to one of the host computer's serial ports. This function defaults to port ASC0 on the C166 and port COM1 on the host computer. If the 'BitRate' argument is not provided, the default of 57600 baud is used.

```
PROFDATA = PROFILE_C166('serial', 'SerialPort', serialport)
```

sets the serial port to the specified `serialport`, which should be one of COM1, COM2, etc.

Optionally, you can specify the bit rate as follows:

```
PROFDATA = PROFILE_C166('serial', 'BitRate', bitrate)
```

This specification sets the `bitrate` for serial connection to the target. `bitrate` must be the same as the bit rate specified for the application that is running on the target.

Alternatively, you can set the bitrate for the serial connection to the target automatically as follows:

```
profdata = profile_c166('serial', 'ModelName', modelName)
```

This specification automatically sets the bit rate by analyzing *modelName* and extracting the correct serial connection bit rate setting from the model. *modelName* should be set to the name of a model which is currently open and running on the target.

To use the CAN connection, you must have suitable CAN hardware installed. If no Application Channel is specified, this function will use the channel 'MATLAB 1'. The bit rate is a property of the Application Channel; to change the bit rate, you must use a different Application Channel, or change the bit rate by running the Vector Informatik configuration utility. To run this utility, make sure that `vcanconf.exe` is on your System Path, then type `vcanconf` from a Windows command prompt.

You can specify the Application Channel as follows:

```
profdata = profile_c166('can', 'CANChannel', canchannel)
```

canchannel specifies the Vector Informatik CAN Application Channel, and must be of the form 'MATLAB 1', 'MATLAB 2' etc.

## Definitions

**Task turnaround time** is the elapsed time between start and finish of a task. If the task is not preempted, then the task turnaround time is equal to the task execution time.

**Task execution time** is that part of the time between task start and finish when the task is actually running and not preempted by another task. Note that the task execution time cannot be measured directly, but is inferred from the task start and finish time and the intervening periods during which it was preempted by another task. Note that, in performing these calculations, no account is taken of processor time consumed by the scheduler while switching tasks: this means that, in cases where preemption has occurred, the reported task execution times will overestimate the true values.

**Concurrent task overruns** occur when a timer task does not complete before that same task is next scheduled to run. Depending on how the real-time scheduler is configured, a task overrun may be handled as a real-time failure. Alternatively, a small number of concurrent task overruns may be allowed to accommodate cases where a task occasionally takes longer than normal to complete.

## Execution Profiling Blocks

See the block reference sections:

- C166 Execution Profiling via ASC0
- C166 Execution Profiling via CAN A
- C166 Execution Profiling via TwinCAN A

## Real-Time Workshop Options for Execution Profiling

### In this section...

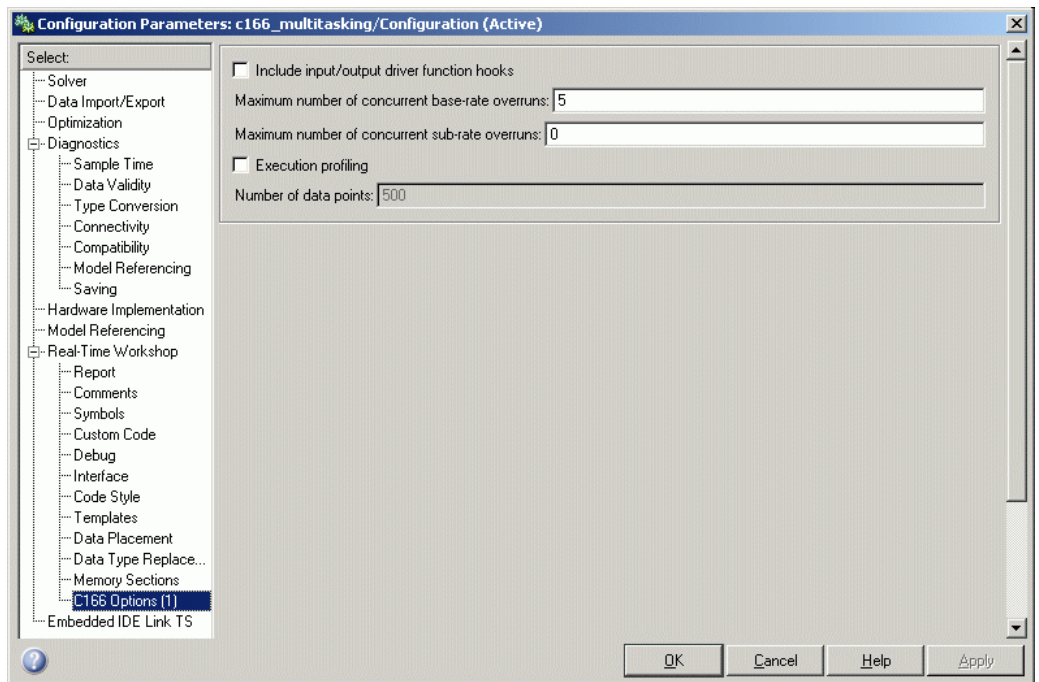
“Execution Profiling” on page 5-6

“Number of Data Points” on page 5-7

“Task Scheduler Overrun Options” on page 5-7

## Execution Profiling

You can see the options for execution profiling by selecting **C166 Options (1)** (under **Real-Time Workshop** in the tree) in the Configuration Parameters dialog box.



If the **Execution Profiling** option is selected, then the generated code for the model will be “instrumented” with function calls at the beginning and



end of each task or ISR to be profiled. These function calls read a timer (on C166 a free running timer is selected from the options in the C166 Resource Configuration block) and log this reading along with a task identifier.

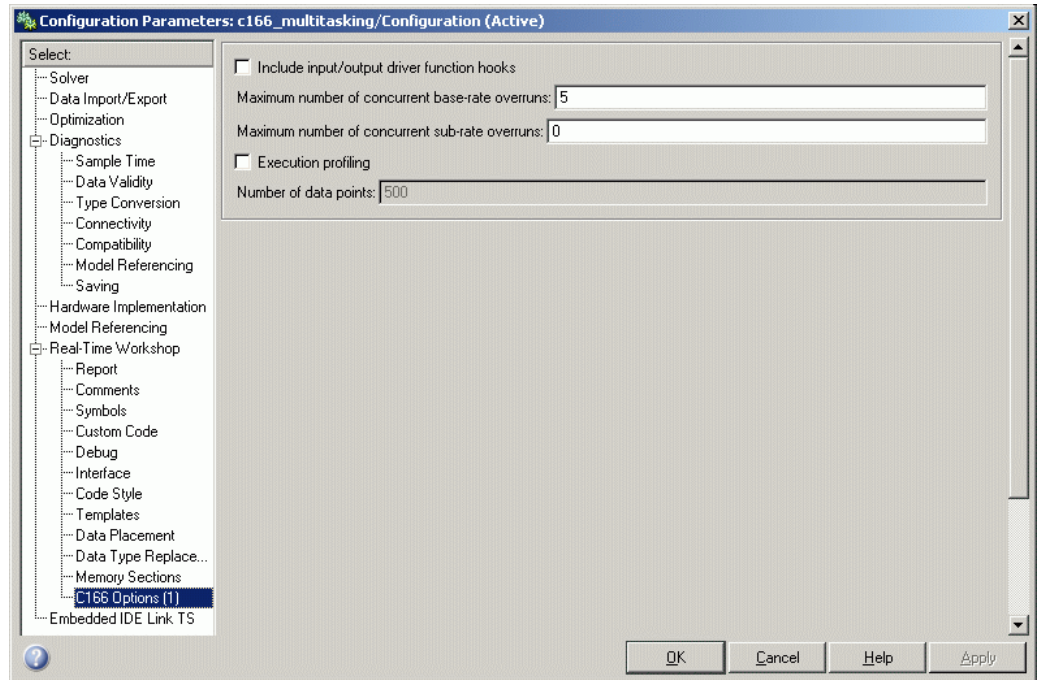
When code for the model is generated, these functions will update data on the worst-case turnaround time for each timer-based task as well as the worst-case number of concurrent task overruns, whenever a previous worst-case value is exceeded. Additionally, when a trigger is provided, data will be logged over a period of time to record all task start and finish times. The trigger signal can be supplied, for example, by the block C166 Execution Profiling via CAN A.

## **Number of Data Points**

When a snapshot of task and ISR activity is logged, this data is stored in memory that is statically allocated at build time. Each data point requires 4 bytes on C166. The larger the number of data points to be stored, the more RAM that must be reserved for this purpose. At the end of a logging run, the data must be uploaded to the host computer for analysis; this is typically achieved by using one of the C166 execution profiling blocks — via ASCO, CAN A, or TwinCAN A. See the reference pages for C166 Execution Profiling via ASCO, C166 Execution Profiling via CAN A, and C166 Execution Profiling via TwinCAN A.

## **Task Scheduler Overrun Options**

These scheduler options configure the allowable number of concurrent task overruns. You can see these options on the **C166 Options (1)** section in the Configuration Parameters dialog box.



You can use the options **Maximum number of concurrent base-rate overruns** and **Maximum number of concurrent sub-rate overruns** to configure the behavior of the scheduler when any of the timer based tasks do not complete within their allowed sample time. It is useful to allow task overruns in the case where a task may occasionally take longer than usual to complete (e.g., if extra processing is required when a special event occurs); if the task overrun is only occasional, then it is possible for the scheduler to catch up after the extra processing has been completed.

If the maximum number of concurrent overruns for any task is exceeded, this is deemed to be a failure and the real-time application is stopped.

As an example, if the base rate is 1 ms and the maximum number of concurrent base-rate overruns is set to 5 then it is possible for the base rate task to run for almost 6 ms before failure occurs. Once the overrun has occurred, it is necessary for subsequent executions of the base rate to complete in less than 1 ms in order that the lost time is recovered.

The occurrence of base-rate overruns does not affect the numerical behavior of the algorithm (although reading/writing external devices will of course be delayed).

If sub-rate overruns are allowed, then the transfer of data between different rates (via rate-transition blocks) in the model may be affected; this causes the numerical behavior in real time to differ from the behavior in simulation. To see an illustration of this effect, try running the demo model `c166_multitasking`, described in the next section. To disallow sub-rate overruns and ensure that this effect does not occur, you should set **Maximum number of concurrent sub-rate overruns** to zero.

---

**Note** Allowing sub-rate overruns may cause non-determinism and loss of integrity for data transferred between different rates in the model. Set this value to zero if you require sub-rate overruns to be handled as a failure (recommended).

---

If you allow sub-rate overruns, then the behavior of any Rate-Transition blocks may be affected. Specifically, if the model contains a Rate Transition block where the option "Ensure deterministic data transfer (maximum delay)" is selected, then this setting may not be honored.

## Multitasking Demo Model

### In this section...

“Introducing the Multitasking Demo” on page 5-10

“Running the Multitasking Demo” on page 5-11

“Interpreting the MATLAB Graphic” on page 5-13

“The Generated HTML Report” on page 5-14

### Introducing the Multitasking Demo

The demo model `c166_multitasking` illustrates both execution profiling and the preemptive multitasking scheduler with configurable overrun handling.

The model is multirate, having tasks running at 1 ms, 4 ms, and 16 ms. It is configured to use the preemptive multitasking scheduler.

A special feature of this model is that each task is designed to perform an increasing number of calculations to increase the processor loading until that task reaches a target turnaround time. This behavior ensures that task overruns occur to demonstrate the behavior of the model in this situation.

Each block in the model, labeled `Load base rate`, `Load sub-rate 1`, `Load sub-rate 2` performs calculations, the result of which should always be 1 both in simulation and in real time. Any other result is a failure and should never occur.

The `Test Rate Interaction` blocks are designed to test whether data is transferred between tasks in a deterministic manner. In simulation, the output of each of these blocks is always zero, indicating that there is no drift between tasks running at different rates. When running in real time, under normal circumstances, the output is also zero; in this case the real-time behavior is deterministic and exactly matches the results in simulation. Even if task preemption and base-rate overruns occur, the output of these blocks will be zero so that the real-time behavior faithfully reproduces the results in simulation. The circumstance under which drift occurs is if sub-rate overruns occur during execution in real time; if this behavior is not desired, you should disallow sub-rate overruns by setting the maximum allowed number of

sub-rate overruns to zero in the **C166 Options (1)** section in the Configuration Parameters dialog box (see “Task Scheduler Overrun Options” on page 5-7).

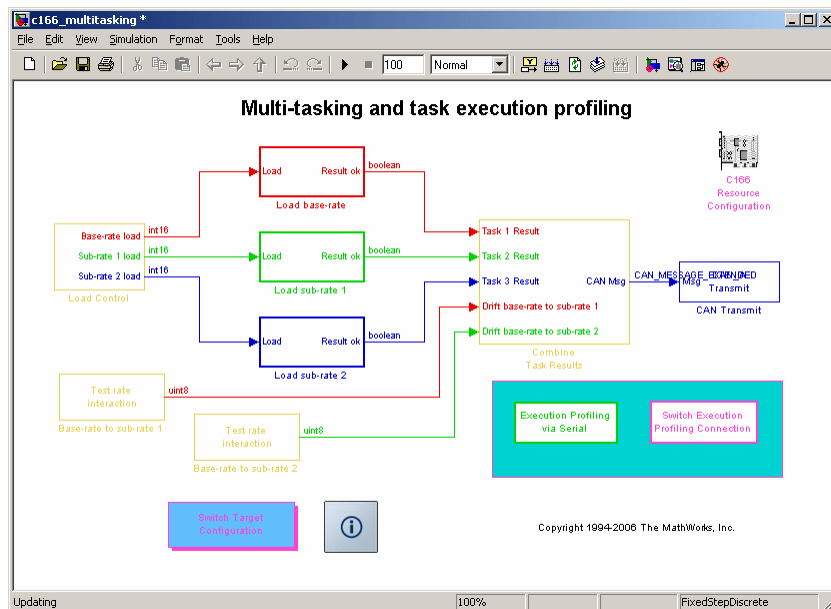
You can double-click the block provided in the model to switch between profiling over serial or CAN connections.

## Running the Multitasking Demo

- 1 Open the model by typing at the command line

```
c166_multitasking
```

If viewing in the Help browser, you can click the link to open the model. If you update the diagram you can see the sample-time colors.



- 2 Select **Simulation > Configuration Parameters**.

The Configuration Parameters dialog box appears.

- 3** Select **Embedded IDE Link TS** in the tree and change the **Build action** to **Create, Build and Execute Application Project**. Click **OK** to dismiss the dialog box.
- 4** Make sure the target is connected to the host PC via serial or CAN cable. The default setting in this demo model is serial. You can double-click the **Switch Execution Profiling Connection** block to toggle between blocks for serial and CAN. See below for instructions if using CAN.
- 5** To build and run the model, select the model window, and then press **Ctrl+B**.

Watch the messages in the command window as code is generated and loaded into the TASKING EDE, then the CrossView Pro Debugger starts, connects to the target, and downloads the code.

- 6** In the CrossView window, click **Run** in the toolbar to start the application running on the target.
- 7** At the command line, type

```
profile_c166 ('serial')
```

You will see messages in the command window as `profile_c166` runs.

When the data has been obtained the Help browser and a figure window appear, displaying the HTML report and the task execution profile.

- 8** Scroll to view the HTML report on task timings and use the controls to zoom in on the MATLAB graphic to examine the details of the task overruns.

If using CAN, be sure to use CAN channel 0 (not 1) on the PC. You can double-click the **Switch Execution Profiling Connection** block in the model to switch to CAN, and follow the same instructions as for a serial connection, except step 7 when the application is running. At the command line, type

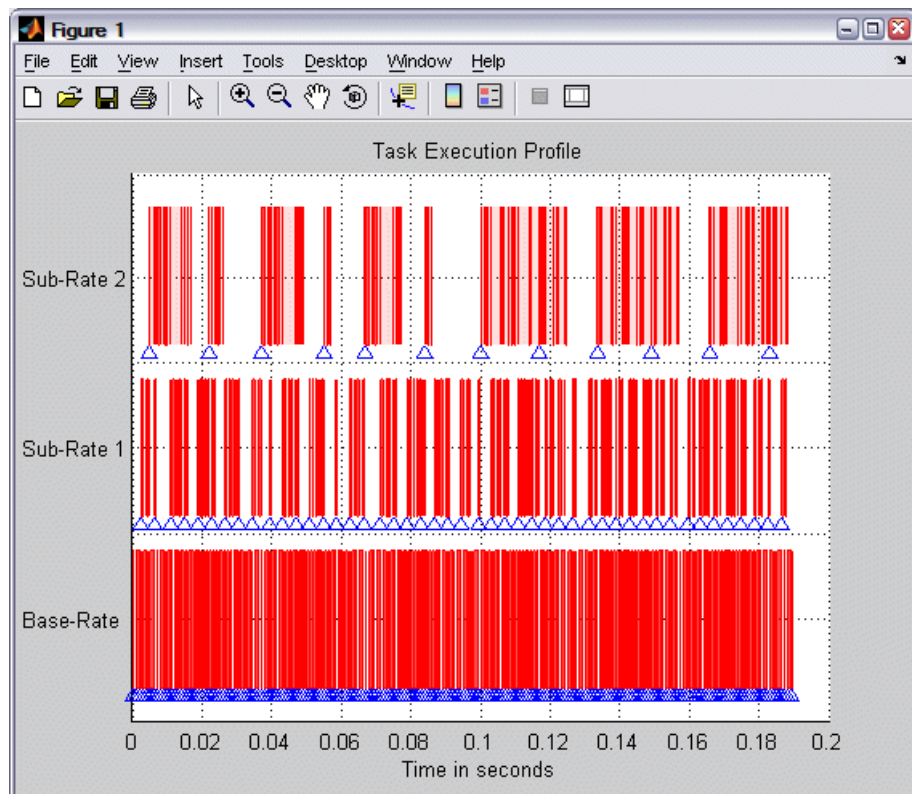
```
profile_c166 ('CAN')
```

You will see command line messages as the function tests the CAN channel, and requests and collects profiling data. When using CAN, it can be useful to run a monitor program such as `bttest32` to verify that the model is running —

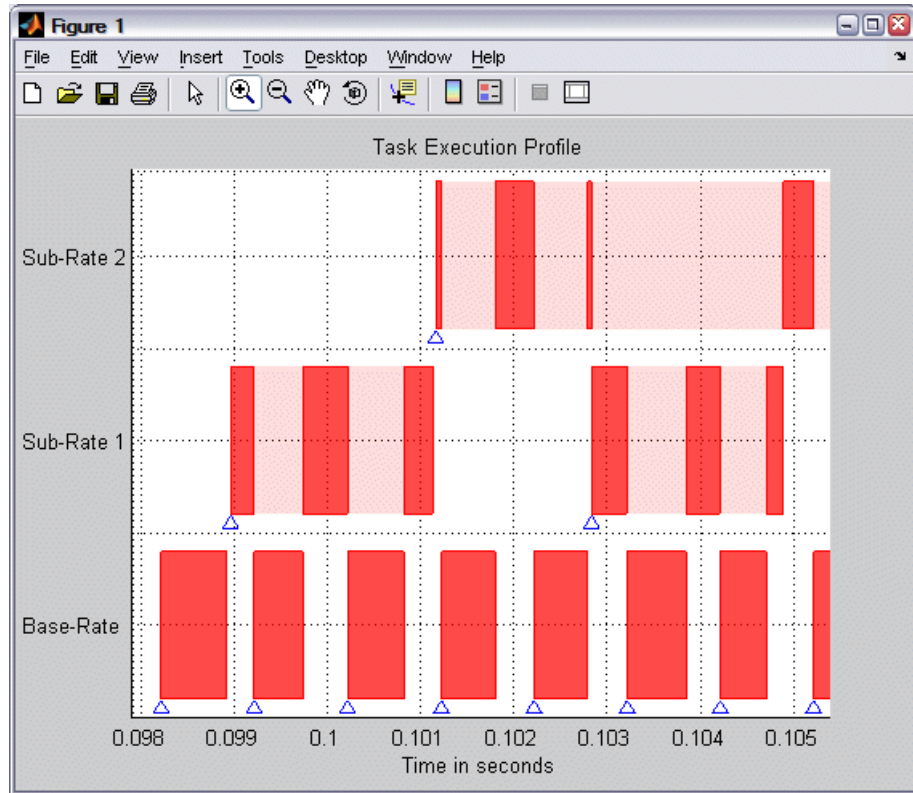
for example you will see messages appearing on the CAN bus and you can see that you have connected the correct CAN channel.

## Interpreting the MATLAB Graphic

Dark shaded areas show the region where a task is executing. Light shaded areas show the region where a task is preempted by a higher priority task or ISR. Triangles indicate the beginning of a task. An example is shown following.



Zoom in to see the details of times that tasks are executing and being preempted, as shown in the following example.



## The Generated HTML Report

See “Definitions” on page 5-5 for the terms task turnaround time, task execution time, and concurrent task overruns.

All times are in seconds. The timer resolution is 4e-007 seconds and the measurement range is 0.026214 seconds.

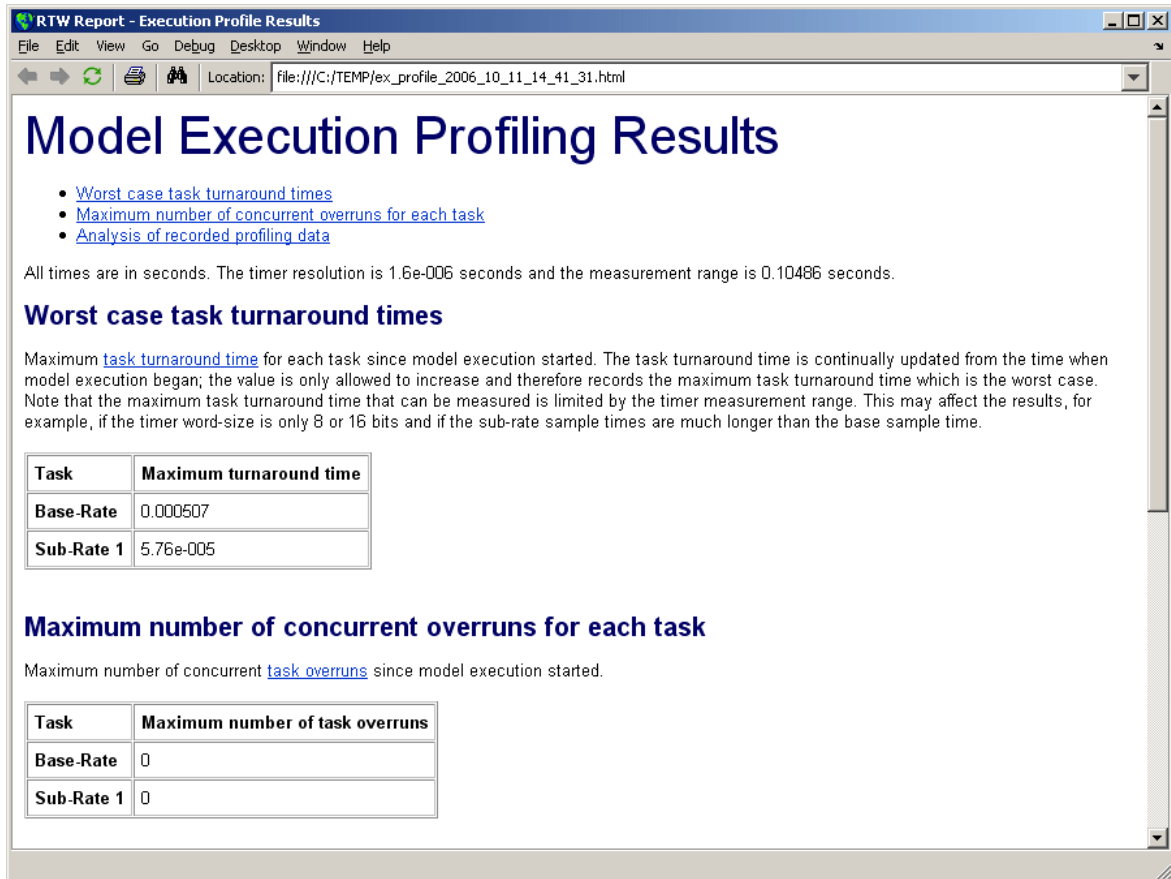
The report contains the following information:

- Worst-case task turnaround times



- Maximum task turnaround time for each task since model execution started. Note that the maximum task turnaround time that can be measured is limited by the timer measurement range.
- Maximum number of overruns for each task
  - Maximum number of concurrent task overruns since model execution started
- Analysis of recorded profiling data
  - Analysis of task turnaround times and task execution times based on recorded data over a period of 0.18139 second

Examples are shown following.



The screenshot shows a web browser window titled "RTW Report - Execution Profile Results". The address bar shows the file path: file:///C:/TEMP/ex\_profile\_2006\_10\_11\_14\_41\_31.html. The main content area displays the title "Model Execution Profiling Results" in a large blue font. Below the title are three bullet points with links: "Worst case task turnaround times", "Maximum number of concurrent overruns for each task", and "Analysis of recorded profiling data". A paragraph of text explains that all times are in seconds, with a timer resolution of 1.6e-006 seconds and a measurement range of 0.10486 seconds. The section "Worst case task turnaround times" is followed by a paragraph explaining that the task turnaround time is continually updated from the start of model execution and records the maximum value. Below this is a table with two columns: "Task" and "Maximum turnaround time". The table has three rows: "Base-Rate" with a value of 0.000507, and "Sub-Rate 1" with a value of 5.76e-005. The section "Maximum number of concurrent overruns for each task" is followed by a paragraph explaining that it shows the maximum number of concurrent task overruns. Below this is another table with two columns: "Task" and "Maximum number of task overruns". This table has three rows: "Base-Rate" with a value of 0, and "Sub-Rate 1" with a value of 0.

### Model Execution Profiling Results

- [Worst case task turnaround times](#)
- [Maximum number of concurrent overruns for each task](#)
- [Analysis of recorded profiling data](#)

All times are in seconds. The timer resolution is 1.6e-006 seconds and the measurement range is 0.10486 seconds.

#### Worst case task turnaround times

Maximum [task turnaround time](#) for each task since model execution started. The task turnaround time is continually updated from the time when model execution began; the value is only allowed to increase and therefore records the maximum task turnaround time which is the worst case. Note that the maximum task turnaround time that can be measured is limited by the timer measurement range. This may affect the results, for example, if the timer word-size is only 8 or 16 bits and if the sub-rate sample times are much longer than the base sample time.

Task	Maximum turnaround time
Base-Rate	0.000507
Sub-Rate 1	5.76e-005

#### Maximum number of concurrent overruns for each task

Maximum number of concurrent [task overruns](#) since model execution started.

Task	Maximum number of task overruns
Base-Rate	0
Sub-Rate 1	0

**RTW Report - Execution Profile Results**

File Edit View Go Debug Desktop Window Help

Location: file:///C:/TEMP/ex\_profile\_2006\_10\_11\_14\_41\_31.html

### Analysis of profiling data recorded over 1.6695 seconds.

Profiling data was recorded over 1.6695 seconds. The recorded data for [task turnaround times](#) and [task execution times](#) is presented in the table below.

Task	Maximum turnaround time	Average turnaround time	Maximum execution time	Average execution time	Average sample time
Base-Rate	0.000462 at 1.03	0.000462	0.000462 at 1.03	0.000462	0.01
Sub-Rate 1	2.24e-005 at 1.14	2.24e-005	2.24e-005 at 1.14	2.24e-005	0.02

**Task turnaround time** is the elapsed time between start and finish of the task. If the task is not pre-empted then the task turnaround time is equal to the task execution time.

**Task execution time** is that part of the time between task start and finish when the task is actually running and not pre-empted by another task. Note that the task execution time cannot be measured directly, but is inferred from the task start and finish time and the intervening periods during which it was preempted by another task. Note that, in performing these calculations, no account is taken of processor time consumed by the scheduler while switching tasks: this means that, in cases where preemption has occurred, the reported task execution times will overestimate the true values.

**Task overruns** occur when a timer task does not complete before that same task is next scheduled to run. Depending on how the real-time scheduler is configured, a task overrun may be handled as a real-time failure. Alternatively, a small number of task overruns may be allowed in order to accommodate cases where a task occasionally takes longer than normal to complete. If a task overrun has occurred and the same task is again scheduled to run *before* the first overrun has been cleared then two *concurrent* task overruns are said to have occurred.



# Block Reference

---

C166 Drivers (p. 6-2)

CAN Message Blocks and CAN  
Drivers (p. 6-6)

Target Support Package IC1  
device driver blocks

Blocks that provide CAN  
functionality

## C166 Drivers

Top-Level Blocks (p. 6-2)	Resource configuration for C166 microcontrollers
Asynchronous/Synchronous Serial Interface (p. 6-2)	Serial transmit and receive
CAN Interface (p. 6-3)	Controller Area Network (CAN) utilities
C-CAN Interface (p. 6-3)	Controller Area Network (CAN) utilities for C-CAN
Execution Profiling (p. 6-4)	Configure execution profiling over CAN, TwinCAN, or serial connection
TwinCAN Interface (p. 6-4)	Controller Area Network (CAN) utilities for XC16x
Interrupts (p. 6-5)	Generate function-call triggers on interrupt
Utilities (p. 6-5)	Configure for predefined hardware configurations
Digital Input/Output (p. 6-5)	Configure digital input/output

## Top-Level Blocks

C166 Resource Configuration	Support device configuration for Infineon C166 microcontrollers
-----------------------------	---

## Asynchronous/Synchronous Serial Interface

Serial Receive	Configure C166 microcontroller for serial receive
Serial Transmit	Configure Infineon C166 microcontroller for serial transmit

## CAN Interface

CAN Bus Status	Output Bus Off or Error Warning state of CAN module
CAN Calibration Protocol (C166)	Implement CAN Calibration Protocol (CCP) standard
CAN Receive	Receive CAN messages from CAN module on Infineon C166 microcontrollers
CAN Reset	Reset CAN module
CAN Transmit	Transmit CAN messages via CAN module on Infineon C166 microcontrollers

For information about CAN message blocks and CAN drivers, see “CAN Message Blocks and CAN Drivers” on page 6-6.

## C-CAN Interface

C-CAN Receive	Receive CAN messages from C-CAN module on ST10 microcontrollers
C-CAN Transmit	Transmit CAN messages via C-CAN module on ST10 microcontrollers
CAN Calibration Protocol (C166, C-CAN)	Implement CAN Calibration Protocol (CCP) standard with C-CAN

For information about CAN message blocks and CAN drivers, see “CAN Message Blocks and CAN Drivers” on page 6-6.

## Execution Profiling

C166 Execution Profiling via ASC0	Provide serial interface to execution profiling engine
C166 Execution Profiling via C-CAN 1	Provide CAN interface to execution profiling engine via C-CAN channel 1 on ST10 microcontrollers
C166 Execution Profiling via CAN A	Provide CAN interface to execution profiling engine via CAN channel A
C166 Execution Profiling via TwinCAN A	Provide CAN interface to execution profiling engine via TwinCAN channel A for XC16x variants of Infineon C166 microcontrollers

## TwinCAN Interface

CAN Calibration Protocol (C166, TwinCAN)	Implement CAN Calibration Protocol (CCP) standard for XC16x variants of Infineon C166 microcontrollers
TwinCAN Bus Status	Output Bus Off or Error Warning state of a CAN node on XC16x variants of Infineon C166 microcontrollers
TwinCAN Receive	Receive CAN messages via TwinCAN module on XC16x variants of Infineon C166 microcontrollers
TwinCAN Reset	Reset CAN node on XC16x variants of Infineon C166 microcontrollers
TwinCAN Transmit	Transmit CAN messages from TwinCAN module on XC16x variants of Infineon C166 microcontrollers



## Interrupts

Fast External Interrupt

Generate asynchronous function-call trigger when interrupt occurs

## Utilities

Switch External Mode Configuration

Configure model for external mode or executable building

Switch Target Configuration

Configure model and Target Preferences to one of a set of predefined hardware configurations

## Digital Input/Output

Digital In

Digital input driver that reads value of specified port or pin number

Digital Out

Digital output driver that sets logical state of specified pin

## **CAN Message Blocks and CAN Drivers**

CAN Pack

Pack individual signals into CAN message

CAN Unpack

Unpack individual signals from CAN messages

For information about obsolete CAN message blocks and CAN drivers, see CAN Blocks Transition.

# Blocks — Alphabetical List

---

# C166 Execution Profiling via ASC0

---

## Purpose

Provide serial interface to execution profiling engine

## Library

Target Support Package IC1/ C166 Driver Library/ Execution Profiling

## Description



The C166 Execution Profiling via ASC0 block provides a serial interface to the execution profiling engine. On receipt of a start command message, logging of execution profile data begins. On completion of a logging run, the recorded data is automatically returned via the serial interface (ASC0). See also the MATLAB command `profile_c166`.

`profile_c166('serial')` collects and displays execution profiling data from an InfineonC166 target microcontroller that is running a suitably configured application generated by the Target Support Package IC1 product.

The data collected is unpacked and then displayed in a summary HTML report and as a MATLAB graphic.

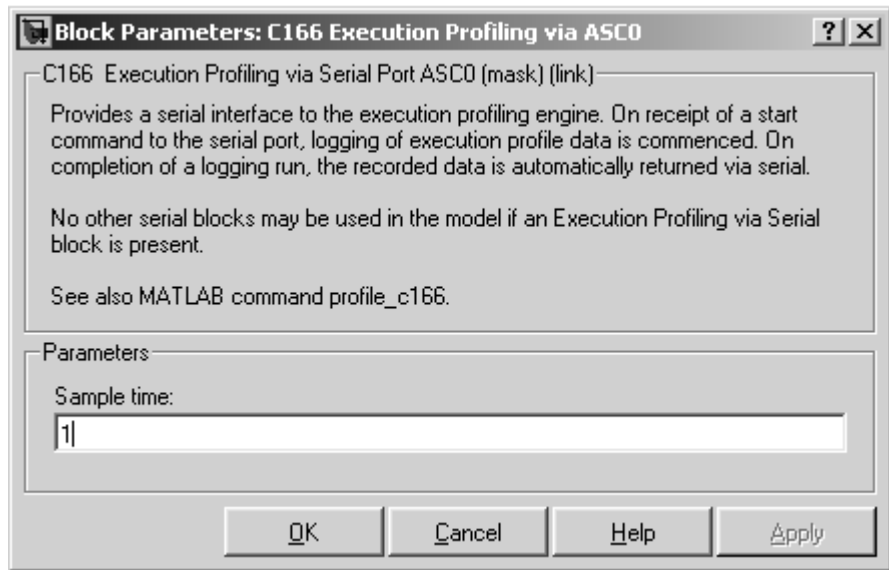
See “The Profiling Command” on page 5-3 for instructions for setting the bit rate automatically or manually, and setting the serial port.

To configure a model for use with execution profiling, you must perform the following steps:

- 1 Check the appropriate option in the **Target Specific Options** tab of the Real-Time Workshop Options dialog box.
- 2 Make sure the model includes a C166 Execution Profiling block that provides an interface between the target-side profiling engine, and the host-side computer from which this command is run.

For more information, see Chapter 5, “Execution Profiling” which includes instructions for the example demo `c166_multitasking`.

## Dialog Box



### Sample time

The sample time of the block. The faster the sample time of the block, the faster data will be uploaded at the end of the execution profiling run. You may want to run this block slower than the fastest rate in the system because the execution profiling itself imposes some loading on the processor. You can minimize this extra loading by not running it at the fastest rate.

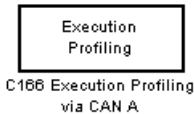
# C166 Execution Profiling via CAN A

---

**Purpose** Provide CAN interface to execution profiling engine via CAN channel A

**Library** Target Support Package IC1/ C166 Driver Library/ Execution Profiling

**Description** The C166 Execution Profiling via CAN A block provides a CAN interface to the execution profiling engine. On receipt of a start command message, logging of execution profile data begins. On completion of a logging run, the recorded data is automatically returned via CAN. You must specify the message identifiers for the start command and the returned data. These identifiers must be compatible with the values used by the host-side part of the execution profiling utility. See also the MATLAB command `profile_c166`.



`profile_c166(CAN)` collects and displays execution profiling data from an Infineon C166 target microcontroller that is running a suitably configured application generated by the Target Support Package IC1 product. The data collected is unpacked then displayed in a summary HTML report and as a MATLAB graphic.

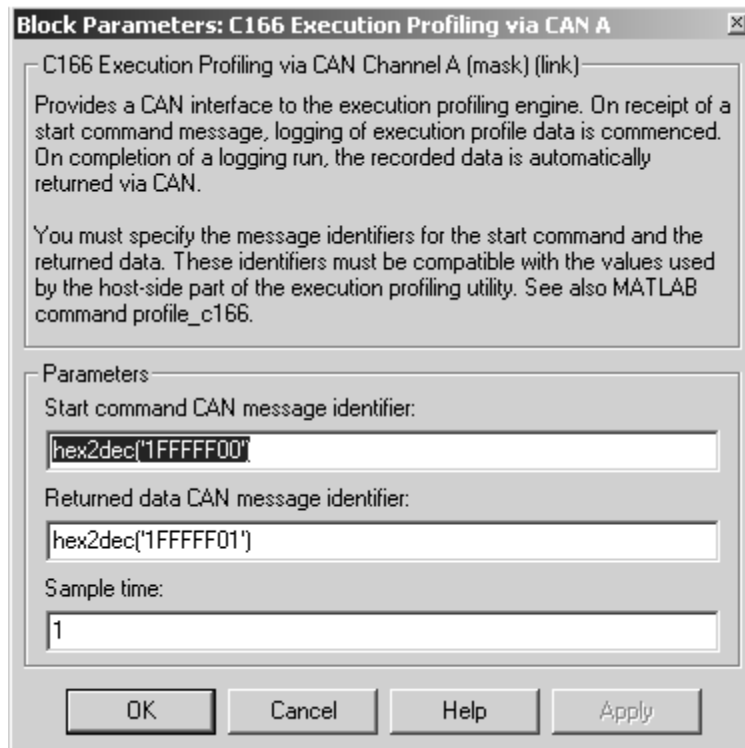
To use the CAN connection, you must have suitable CAN hardware installed on the host computer. See “The Profiling Command” on page 5-3 for instructions for setting the CAN Application Channel and bit rate.

To configure a model for use with execution profiling, you must perform the following steps:

- 1 Check the appropriate option in the **Target Specific Options** tab of the Real-Time Workshop Options dialog box.
- 2 Make sure the model includes a C166 Execution Profiling block that provides an interface between the target-side profiling engine, and the host-side computer from which this command is run.

For more information, see Chapter 5, “Execution Profiling” which includes instructions for the example demo `c166_multitasking`.

## Dialog Box



### Start command CAN message identifier

Set the identifier of the message to start logging execution profiling data. You should use the default unless you have modified `profile_c166`. This identifier must be compatible with the values used by the host-side part of the execution profiling utility (`profile_c166`).

The utility `profile_c166` provides a mechanism for initiating an execution profiling run and for uploading the recorded data to the host machine. To perform this procedure using a CAN connection between host and target, `profile_c166` first sends a CAN message that is a command to start an execution profiling

# C166 Execution Profiling via CAN A

---

run. The CAN identifier for this message must be specified as the same value on the target as on the host. The host-side values are hard-coded in `profile_c166`. If you are using an unmodified version of the host-side utility, you should use the default value for this CAN message identifier. These are visible to help you avoid using the same identifier for other tasks.

## **Returned data CAN message identifier**

Set the message identifier for the returned data. As with the message identifier for the start command, the value specified here must be the same as the hard-coded value in `profile_c166`.

## **Sample time**

The sample time of the block. The faster the sample time of the block, the faster data will be uploaded at the end of the execution profiling run. You may want to run this block slower than the fastest rate in the system because the execution profiling itself imposes some loading on the processor. You can minimize this extra loading by not running it at the fastest rate.



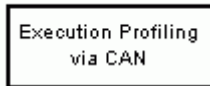
# C166 Execution Profiling via C-CAN 1

---

**Purpose** Provide CAN interface to execution profiling engine via C-CAN channel 1 on ST10 microcontrollers

**Library** Target Support Package IC1/ C166 Driver Library/ Execution Profiling

## Description



C166 Execution Profiling  
via C-CAN 1

The C166 Execution Profiling via C-CAN 1 block is for the C-CAN interface and performs the same functions as the C166 Execution Profiling via CAN A block. For block parameter descriptions, see the C166 Execution Profiling via CAN A reference page.

# C166 Execution Profiling via TwinCAN A

---

<b>Purpose</b>	Provide CAN interface to execution profiling engine via TwinCAN channel A for XC16x variants of Infineon C166 microcontrollers
<b>Library</b>	Target Support Package IC1/ C166 Driver Library/ Execution Profiling
<b>Description</b>	The C166 Execution Profiling via TwinCAN A block is for the TwinCAN interface and performs the same functions as the C166 Execution Profiling via CAN A block. For block parameter descriptions, see the C166 Execution Profiling via CAN A reference page.

<b>Purpose</b>	Support device configuration for Infineon C166 microcontrollers
<b>Library</b>	Target Support Package IC1/ C166 Driver Library
<b>Description</b>	<p>The C166 Resource Configuration block differs in function and behavior from conventional blocks. Therefore, we refer to this block as the C166 Resource Configuration <i>object</i>.</p> <p>The C166 Resource Configuration object is required to provide information that is used to configure driver blocks and timer interrupts.</p> <ul style="list-style-type: none"><li>• You must include this block in your model if<ul style="list-style-type: none"><li>▪ You are using any of the driver blocks supplied with the Target Support Package IC1 product</li><li>▪ You are taking advantage of the automatically generated scheduler that is driven by timer interrupts.</li></ul></li><li>• You do not need to include the C166 Resource Configuration object in your model if you are not using any of the C166 driver library blocks, and if you do not require the automatically generated scheduler (for example, if you are supplying your own <code>main.c</code>).</li></ul> <p>The C166 Resource Configuration object maintains configuration settings that apply to the Infineon C166 microcontroller. Although the C166 Resource Configuration object resembles a conventional block in appearance, it is not connected to other blocks via input or output ports. This is because the purpose of the C166 Resource Configuration object is to provide information to other blocks in the model. C166 device driver blocks register their presence with the C166 Resource Configuration object when they are added to a model or subsystem; they can then query the C166 Resource Configuration object for required information.</p> <p>To install a C166 Resource Configuration object in a model or subsystem, open the C166 Drivers library and select the C166 Resource Configuration icon. Then drag and drop it into your model or subsystem, like a conventional block.</p>

# C166 Resource Configuration

---

Having installed a C166 Resource Configuration object into your model or subsystem, you can then select and edit configuration settings in the C166 Resource Configuration window. See “Using the C166 Resource Configuration Window” on page 7-12 for further information.

---

**Note** If your model or subsystem requires a C166 Resource Configuration object (see above), you should place it at the top-level system for which you are going to generate code. If your whole model is going to run on the target processor, put the C166 Resource Configuration object at the root level of the model. If you are going to generate code from separate subsystems (to run specific subsystems on the target), place a C166 Resource Configuration object at the top level of each subsystem. You should not have more than one C166 Resource Configuration object in the same branch of the model hierarchy. Errors will result if these conditions are not met.

---

When the C166 Resource Configuration block is placed into a model, it modifies the `preloadfcn` callback of the model. If you wish to add a command to the `preloadfcn` callback of a model that already has an C166 Resource Configuration block, do not remove the commands that are already installed. Instead, copy the installed `preloadfcn` callback and append your commands. Then set the `preloadfcn` to the merged command. If you corrupt the `preloadfcn`, you can retrieve the command from any model that has a C166 Resource Configuration block, as the `preloadfcn` will be the same for all models. You can retrieve the `preloadfcn` with the following command: `plf = get_param(bdroot, 'preloadfcn')`

## Types of Configurations

A *configuration* is a collection of parameter values affecting the operation of one or more device driver blocks in the Target Support Package IC1 library. The C166 Resource Configuration object currently supports the following types of configurations:

- “C166 System Configuration Parameters” on page 7-14 (c166drivers): C166 microcontroller clocks and other CPU-related parameters
- “Asynchronous/Synchronous Serial Interface Configuration Parameters” on page 7-16Asynchronous/Synchronous Serial Interface Configuration: parameters related to the serial driver blocks and Simulink external mode
- “CAN Configuration Parameters” on page 7-18: parameters for CAN interrupt levels
- “TwinCAN Configuration Parameters” on page 7-21: parameters for TwinCAN interrupt levels
- “C-CAN Configuration Parameters” on page 7-22: parameters for C-CAN interrupt levels

## Dialog Box

The C166 drivers configuration always appears in the active configuration pane. If there are also blocks in your model from the Asynchronous/Synchronous Serial Interface (ASCO) sublibrary, you will also see the configuration for these, as seen in the next example. If you add an ASCO block to a model without any ASCO blocks, the appropriate configuration is created and activated in the C166 Resource Configuration block. Similarly, if you add CAN blocks to a model, a CAN configuration is created.

You can see an example like this by opening the demo model `c166_serial_transmit` and double-clicking on the C166 Resource Configuration block.

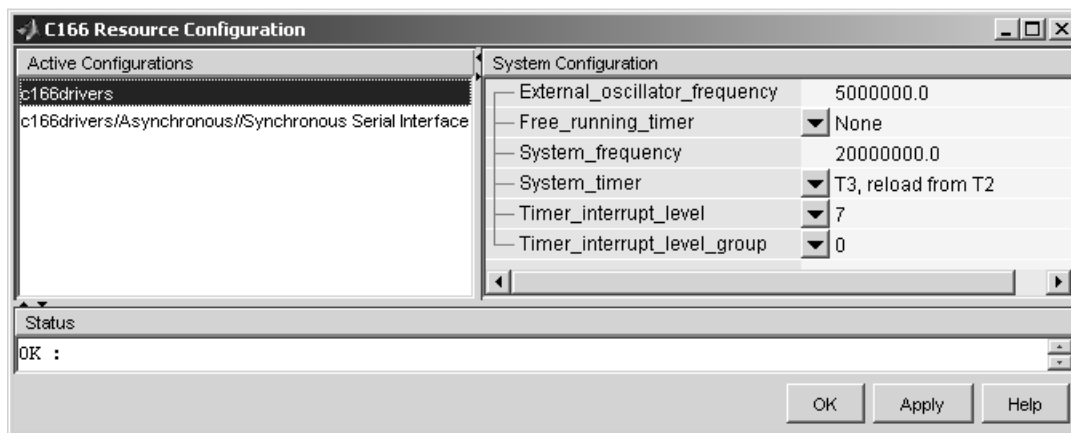
A configuration remains active until all blocks associated with it are removed from the model or subsystem. At that point, the configuration is in an *inactive* state. Inactive configurations are lost from the C166 Resource Configuration window when the model is saved and reopened. You can reactivate a configuration by simply adding an appropriate block into the model.

# C166 Resource Configuration

## Using the C166 Resource Configuration Window

To open the C166 Resource Configuration window, install a C166 Resource Configuration object in your model or subsystem and double-click on the C166 Resource Configuration icon. The C166 Resource Configuration window then opens.

This example shows the C166 Resource Configuration window for a model that has active configurations for the C166 microcontroller (c166drivers) and for the Asynchronous/Synchronous Serial Interface (ASCO) blocks, as found in the demo c166\_serial\_transmit.



The C166 Resource Configuration window consists of the following elements:

- **Active Configurations** panel: This panel displays a list of currently active configurations. To edit a configuration, click its entry in the list. The parameters for the selected configuration then appear in the **System Configuration** panel.

To link back to the library associated with an active configuration, right-click its entry in the list. From the menu that appears, select **Go to library**.

To see documentation associated with an active configuration, right-click its entry in the list. From the menu that appears, select **Help**.

- **System Configuration** panel: This panel lets you edit the parameters of the selected configuration. The parameters of each configuration type are detailed in “C166 Resource Configuration Window Parameters” on page 7-13.

---

**Note** Click **Apply** to make your changes take effect.

---

- **Status** panel: The **Status** panel displays error messages that may arise if resource allocation conflicts are detected in the configuration.
- **OK** button: Dismisses the window.

## **C166 Resource Configuration Window Parameters**

The following sections describe the parameters for each type of configuration in the C166 Resource Configuration window. The default parameter settings are optimal for most purposes. If you want to change the settings, read the relevant sections of the C166 User’s Manual. You can find this document at the Infineon Web site at the following URL:

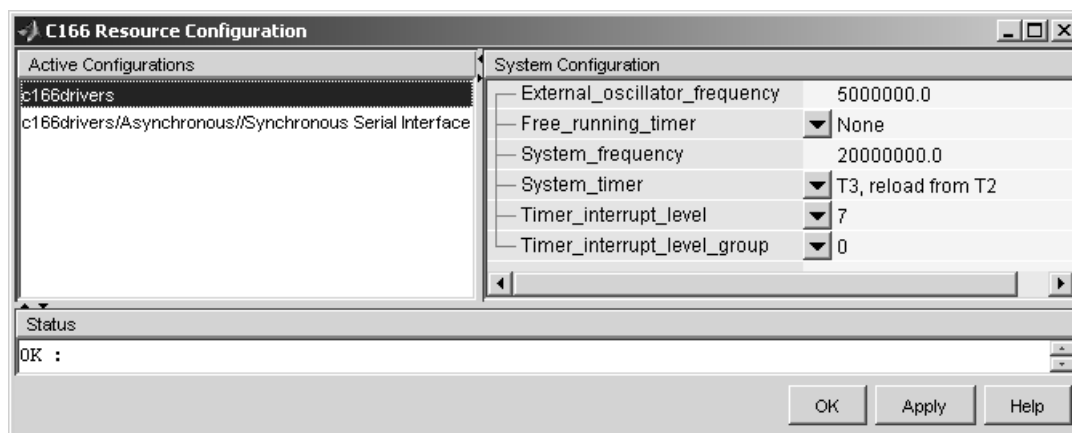
<http://www.infineon.com/>

For the ST10 User’s Manual, see the ST Microelectronics Web site at following URL:

<http://www.st.com/>

# C166 Resource Configuration

## C166 System Configuration Parameters



### External\_oscillator\_frequency

Depending on your hardware variant, the Real-Time Clock (RTC) may be driven directly by the external oscillator input and it is, therefore, important that the external oscillator frequency is set correctly. Otherwise, if the RTC is used to provide any timing services, the behavior will be incorrect. The default value for external oscillator frequency is 5 MHz. You should check your hardware manual to establish the correct value for your setup. Note you can choose the RTC as a `System_timer`, see below.

### Free\_running\_timer

This parameter allows one of the on-chip timers to be configured for use with execution profiling. The selected timer is configured to run indefinitely at a known frequency and is used by the execution profiling engine to record the times at which tasks start or finish executing. See Chapter 5, “Execution Profiling” for more details.

To find supported timer configurations, you should check the General Purpose Timer section of the relevant User’s Manual for your C166 microcontroller derivative.



## **System\_frequency**

You must set the system frequency of your C166 microcontroller hardware here. Note that the value depends on your hardware type and configuration. If you choose an incorrect value the model will be correspondingly fast or slow.

## **System\_timer**

You must select which timer to use for generating interrupts to drive the model update rate. You should select a timer, or timer pair, that you do not intend to use for any other purpose within your application. We recommend you choose a pair of timers, e.g., T6, with reload from CAPREL. This will give the best possible sample time accuracy with no long term drift caused by higher priority interrupts. If you choose a single timer, e.g., T2 or RTC, the timer value will be reloaded within the timer interrupt service routine. With this approach, any delay in servicing the timer interrupt will be added to the time until the next timer interrupt is generated.

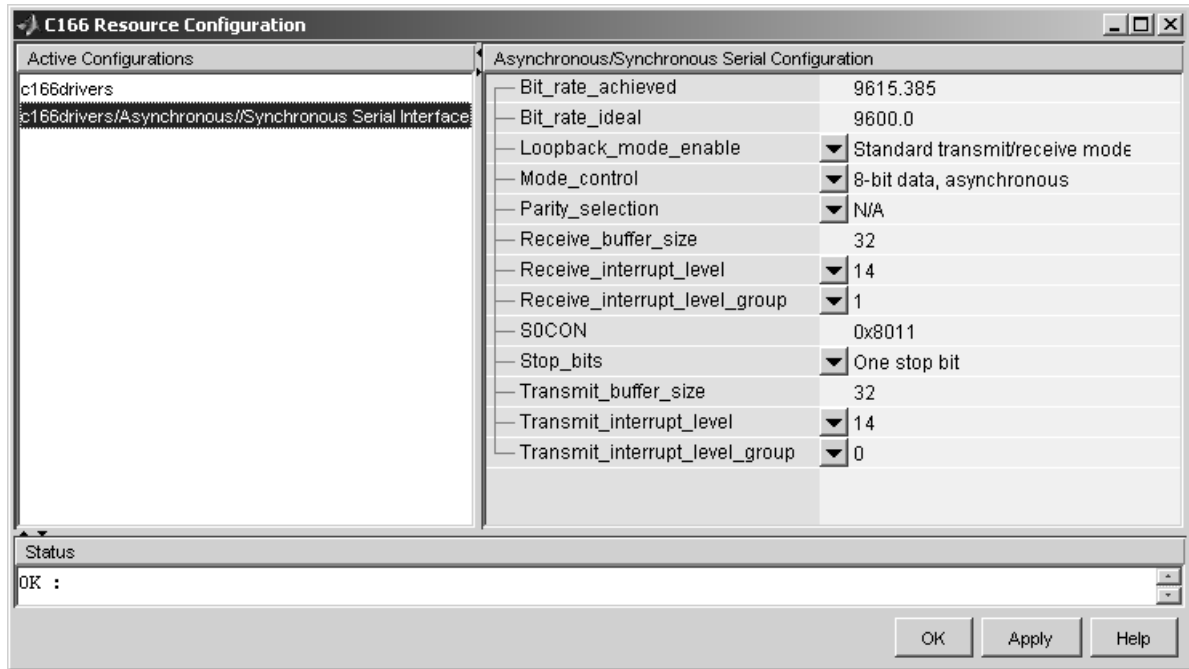
To find supported timer configurations, you should check the General Purpose Timer section of the relevant User's Manual for your C166 microcontroller derivative.

## **Timer\_interrupt\_level and Timer\_interrupt\_level\_group**

These two parameters together set the priority of sample time interrupts. You should choose values such that the sample time interrupts are suitably prioritized relative to other interrupts used by your application.

# C166 Resource Configuration

## Asynchronous/Synchronous Serial Interface Configuration Parameters



### **Bit\_rate\_achieved**

This read-only field shows the achieved serial interface bit rate. In general, this value differs slightly from the requested bit rate, but is the closest value that can be achieved by setting allowed values in C166 register S0BG and bitfield S0BRS of register S0CON.

### **Bit\_rate\_ideal**

Enter the desired bit rate for serial communications in this field. Appropriate register settings are calculated automatically. You can verify the actual bit rate in the `Bit_rate_achieved` field.

**Loopback\_mode\_enable**

Select this entry to operate the serial interface in loopback mode. This may be useful for test purposes where the serial interface is required to receive data that it transmitted itself.

**Mode\_control**

Select the desired combination of word length and parity/no parity. See the C166 User's Manual for more details.

**Parity\_selection**

If parity is enabled, you must select odd or even.

**Receive\_buffer\_size**

You must select the size of the RAM buffer that will be used by the serial receive driver. The maximum allowed value is 254.

**Receive\_interrupt\_level** and **Receive\_interrupt\_level\_group**

Set the receive interrupt priority here. Note that the Target Support Package IC1 drivers allow only interrupt levels 14 and 15 to be used. The reason for this is that the drivers use the peripheral event controller (PEC), which provides very fast interrupt response but is restricted to levels 14 and 15.

**S0CON**

This is a noneditable field that shows the value of the serial interface register S0CON and how it varies as dialog box settings are changed.

**Stop\_bits**

You must select either 1 or 2 stop bits.

**Transmit\_buffer\_size**

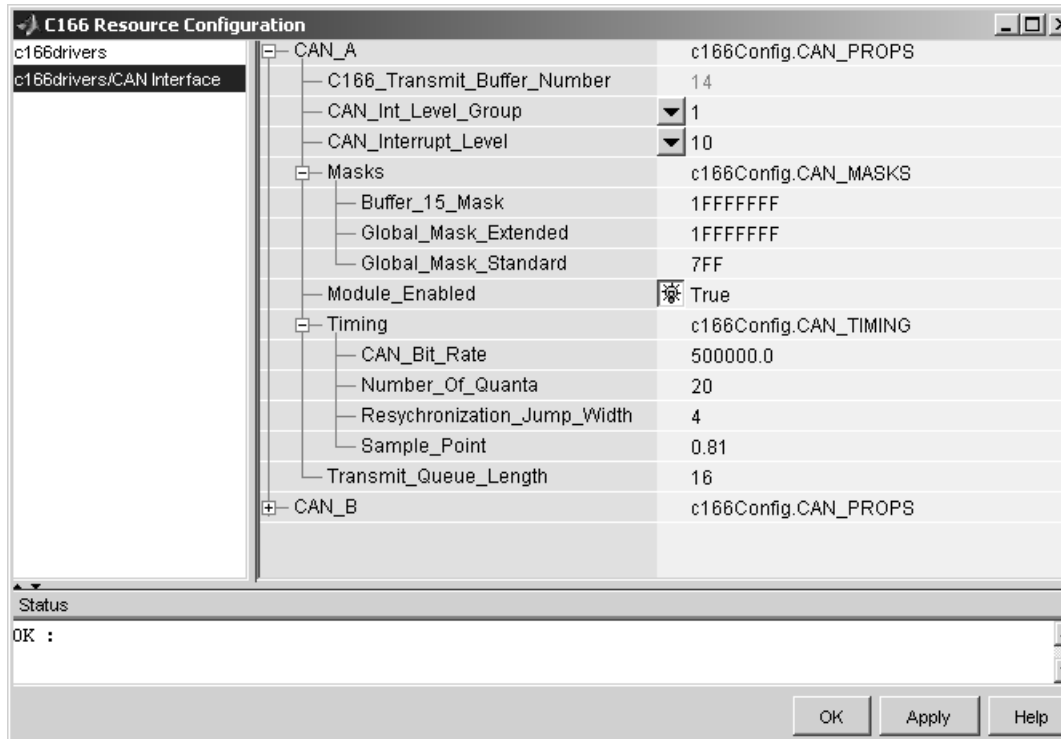
See `Receive_buffer_size`.

**Transmit\_interrupt\_level** and **Transmit\_interrupt\_level\_group**

See Receive parameters above.

# C166 Resource Configuration

## CAN Configuration Parameters



The parameters listed below are the same for CAN modules A and B.

### **C166\_Transmit\_Buffer\_Number**

This parameter is read only; all transmitted messages are sent from buffer 14.

### **CAN\_Int\_Level\_Group and CAN\_Interruption\_Level**

These two parameters together set the priority of sample time interrupts. You should choose values such that the sample time interrupts are suitably prioritized relative to other interrupts

used by your application. Note that CAN module interrupts must be set to a higher priority than timer interrupts. Use the **Validate Configuration** button to make sure you do not select an interrupt level that is already in use.

## **Masks**

You can use these mask configuration parameters to choose to ignore certain bits. In general, a CAN message is received only if its identifier is an exact match with the identifier specified in one of the receive buffers. You can use mask parameters to indicate that some of the bits in the received message identifier are “don’t care.”

### **Buffer\_15\_Mask**

This mask applies to buffer 15 only. Each bit in the mask that is set to zero causes the corresponding bit in the received message identifier to be ignored when comparing it to the message identifier that buffer 15 is configured to receive.

### **Global\_Mask\_Extended**

This mask applies to any of buffers 1 to 14 that are configured to receive messages with an extended identifier. Each bit in the mask that is set to zero causes the corresponding bit in the received message identifier to be ignored when comparing it to the message identifier that this buffer is configured to receive.

### **Global\_Mask\_Standard**

This mask applies to any of buffers 1 to 14 that are configured to receive messages with a standard identifier. Each bit in the mask that is set to zero causes the corresponding bit in the received message identifier to be ignored when comparing it to the message identifier that this buffer is configured to receive.

### **Module\_Enabled**

If the module is enabled, then initialization code for that CAN module is generated. Use this setting to prevent generation of driver code for a CAN module that is not required, or not available on your hardware variant.

# C166 Resource Configuration

---

## **Timing**

### **CAN\_Bit\_Rate**

Enter the desired bit rate. The default bit rate is 500000.

### **Number\_Of\_Quanta**

The number of CAN module clock ticks per message bit.

### **Resynchronization\_Jump\_Width**

The maximum number of clock ticks that the CAN device can resynchronize over when it detects that it is losing message synchronization.

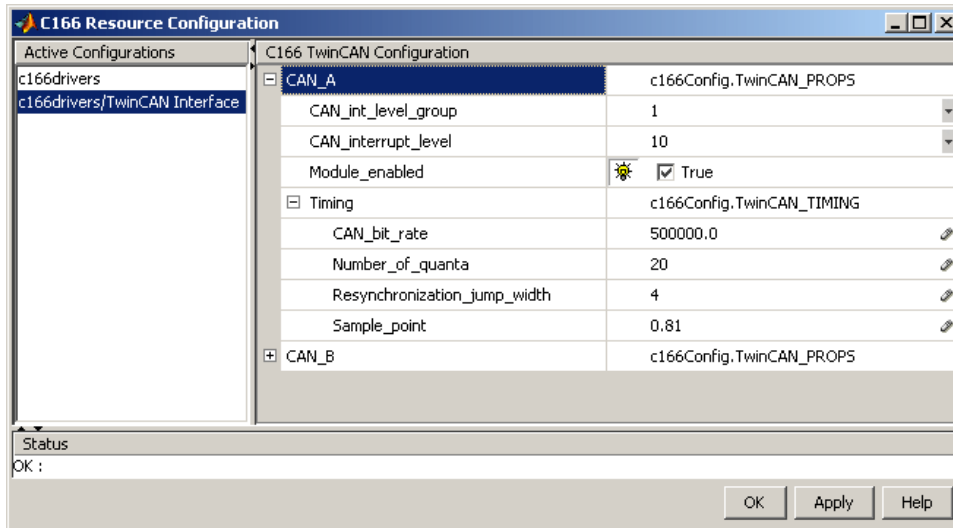
### **Sample\_Point**

The point in the message where the CAN module samples the value of the message bit.

### **Transmit\_Queue\_Length**

Length (number of messages) of the transmit queue. The transmit queue holds messages that are waiting to be transmitted. An increase in performance can be achieved by reducing the queue length. However, if the queue's length is too small, it may become full, causing messages to be lost.

## TwinCAN Configuration Parameters



The TwinCAN Configuration Parameters are a subset of the “CAN Configuration Parameters” on page 7-18, plus these additional parameters:

### **TwinCAN\_Rx\_Pin**

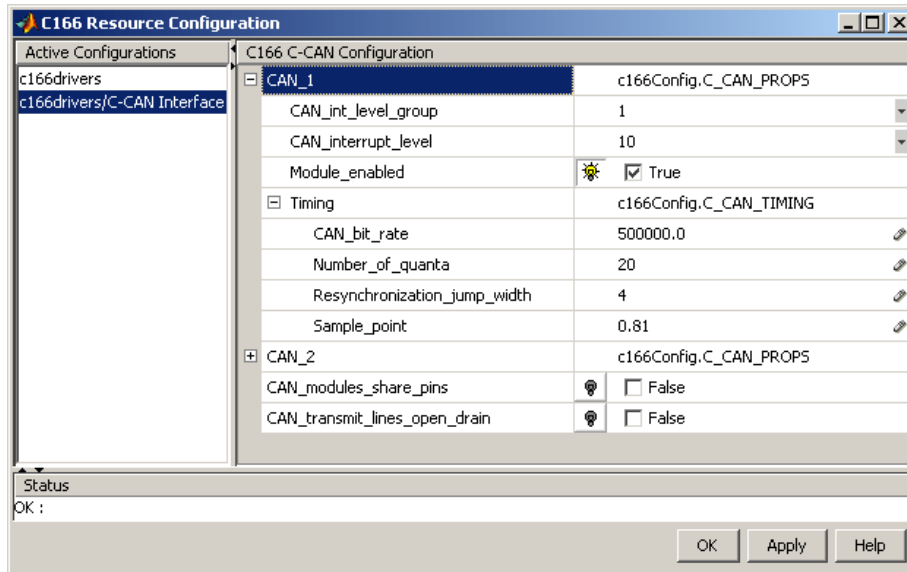
Select the desired pin. The default is P4.5(CAN\_A) or P9.0(CAN\_B).

### **TwinCAN\_Tx\_Pin**

Select the desired pin. The default is P4.6(CAN\_A) or P9.1(CAN\_B).

# C166 Resource Configuration

## C-CAN Configuration Parameters



The C-CAN Configuration Parameters are the same subset of the “CAN Configuration Parameters” on page 7-18 as the TwinCAN Configuration Parameters, plus the following two settings. The parameters are the same for C-CAN modules 1 and 2.

### **CAN\_modules\_share\_pins**

When this option is not selected (the default), C-CAN modules 1 and 2 are connected to separate I/O pins. Use this option if C-CAN modules 1 and 2 both share the same microcontroller I/O pins P4.5 (receive) and P4.6 (transmit). In this mode both CAN modules are connected to the same CAN bus via a shared transceiver. This option takes effect only if both C-CAN modules are enabled. See the microcontroller User Manual for more details.

### **CAN\_transmit\_lines\_open\_drain**

When selected, the transmit lines for both CAN 1 and CAN 2 are configured for open drain. Use this option if both C-CAN modules



are connected (externally from the microcontroller) to the same CAN transceiver. This option takes effect only if both C-CAN modules are enabled and if the option for both modules to use shared pins is not selected. See the microcontroller User Manual for more details.

# CAN Bus Status

## Purpose

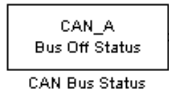
Output Bus Off or Error Warning state of CAN module

## Library

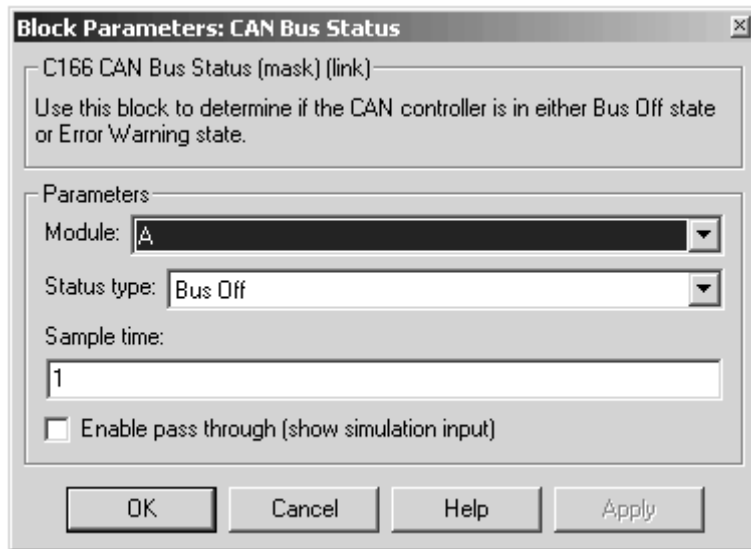
Target Support Package IC1/ C166 Driver Library/ CAN Interface

## Description

The CAN Bus Status block provides an indicator of the state of the selected CAN module. The block has a single output that may be set to indicate either the Bus Off or Error Warning state of the module.



## Dialog Box



### Module

Select CAN module A or B.

### Status type

Choose Bus Off or Error Warning.

### Sample time

The sample time of this block.

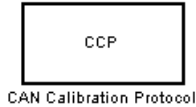
## Purpose

Implement CAN Calibration Protocol (CCP) standard

## Library

Target Support Package IC1/ C166 Driver Library/ CAN Interface

## Description



The CAN Calibration Protocol (C166) block provides an implementation of a subset of the CAN Calibration Protocol (CCP) Version 2.1. CCP is a protocol for communicating between the target processor and the host machine over CAN. In particular, a calibration tool (see “Compatibility with Calibration Packages” on page 7-30) running on the host can communicate with the target, allowing remote signal monitoring and parameter tuning.

This block processes a Command Receive Object (CRO) and outputs the resulting Data Transmission Object (DTO) and Data Acquisition (DAQ) messages.

For more information on CCP, refer to *ASAM Standards: ASAM MCD: MCD 1a* on the Association for Standardization of Automation and Measuring Systems (ASAM) Web site at <http://www.asam.de>.

## Using the DAQ Output

---

**Note** The CCP Data Acquisition (DAQ) List mode of operation is only supported with the Real-Time Workshop Embedded Coder product. If this is not available then custom storage classes `canlib.signal` are ignored during code generation: this means that the CCP DAQ Lists mode of operation cannot be used.

You can use the CCP Polling mode of operation with or without the Real-Time Workshop Embedded Coder product.

---

The DAQ output is the output for any CCP Data Acquisition (DAQ) lists that have been set up. You can use the ASAP2 file generation feature of the Real-Time (RT) target to

- Set up signals to be transmitted using CCP DAQ lists.

# CAN Calibration Protocol (C166)

---

- Assign signals in your model to a CCP event channel automatically (see “Parameter Tuning and Signal Logging” on page 2-18).

Once these signals are set up, event channels then periodically fire events that trigger the transmission of DAQ data to the host. When this occurs, CAN messages with the appropriate CCP/DAQ data appear on the DAQ output, along with an associated function call trigger.

The calibration tool (see “Compatibility with Calibration Packages” on page 7-30) must use CCP commands to assign an event channel and data to the available DAQ lists, and interpret the synchronous response.

Using DAQ lists for signal monitoring has the following advantages over the polling method:

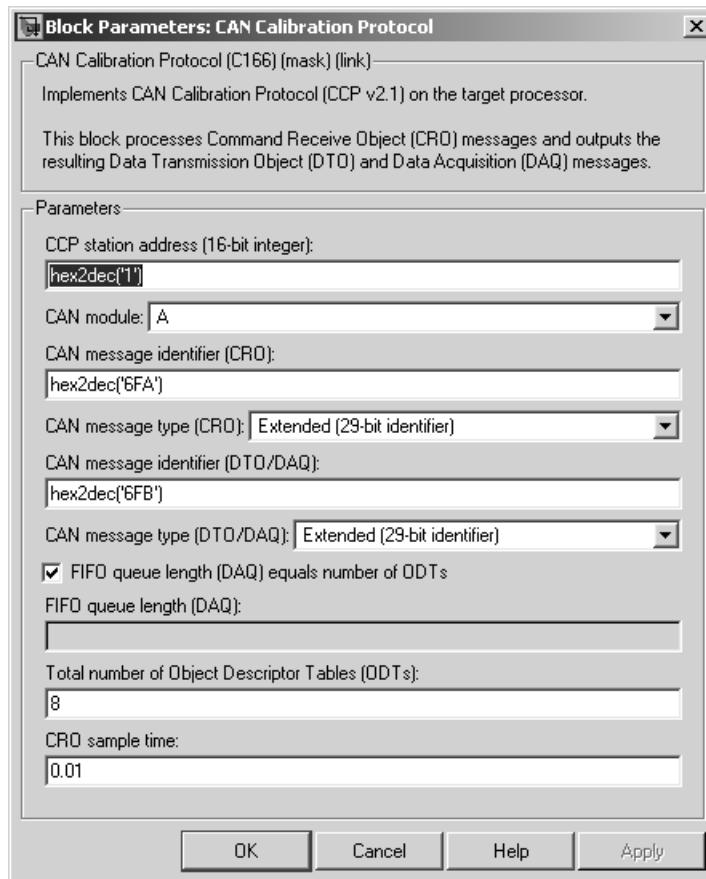
- There is no need for the host to poll for the data. Network traffic is halved.
- The data is transmitted at the correct update rate for the signal. Therefore, there is no unnecessary network traffic generated.
- Data is guaranteed to be consistent. The transmission takes place after the signals have been updated, so there is no risk of interruptions while sampling the signal.

---

**Note** The Target Support Package IC1 product does not currently support event channel prescalers.

---

## Dialog Box



**Block Parameters: CAN Calibration Protocol**

CAN Calibration Protocol (C166) (mask) (link)

Implements CAN Calibration Protocol (CCP v2.1) on the target processor.

This block processes Command Receive Object (CRO) messages and outputs the resulting Data Transmission Object (DTO) and Data Acquisition (DAQ) messages.

Parameters

CCP station address (16-bit integer):  
hex2dec('1')

CAN module: A

CAN message identifier (CRO):  
hex2dec('6FA')

CAN message type (CRO): Extended (29-bit identifier)

CAN message identifier (DTO/DAQ):  
hex2dec('6FB')

CAN message type (DTO/DAQ): Extended (29-bit identifier)

FIFO queue length (DAQ) equals number of ODTs

FIFO queue length (DAQ):

Total number of Object Descriptor Tables (ODTs):  
8

CRO sample time:  
0.01

OK Cancel Help Apply

### CAN station address (16 bit integer)

The station address of the target. The station address is interpreted as a `uint16`. It is used to distinguish between different targets. By assigning unique station addresses to targets sharing the same CAN bus, it is possible for a single host to communicate with multiple targets.

### CAN module

Choose CAN module A or B.

# CAN Calibration Protocol (C166)

---

## **CAN message identifier (CRO)**

Specify the CAN message identifier for the Command Receive Object (CRO) message you want to process.

## **CAN message type (CRO)**

The incoming message type. Select either Standard(11-bit identifier) or Extended(29-bit identifier).

## **CAN message identifier (DTO/DAQ)**

The message identifier is the CAN message ID used for Data Transmission Object (DTO) and Data Acquisition (DAQ) message outputs.

## **CAN message type (DTO/DAQ)**

The message type to be transmitted by the DTO and DAQ outputs. Select either Standard(11-bit identifier) or Extended(29-bit identifier).

## **FIFO queue length (DAQ) equals number of ODTs**

Leave this check box selected to automatically set the FIFO queue length equal to the number of Object Descriptor Tables (ODTs) (recommended). Clear the check box to set the length of the FIFO queue manually.

## **FIFO queue length (DAQ)**

Specify the FIFO queue length manually. This is enabled if you clear the check box to set the queue length automatically.

## **Total number of Object Descriptor Tables (ODTs)**

The default number of Object Descriptor Tables (ODTs) is 8. These ODTs are shared equally between all available DAQ lists. You can choose a value between 0 and 254, depending on how many signals you wish to log simultaneously. You must make sure you allocate at least 1 ODT per DAQ list, or your build will fail. The calibration tool will give an error message if there are too few ODTs for the number of signals you specify for monitoring. Be aware that too many ODTs can make the sample time overrun. If you choose more than the maximum number of ODTs (254), the build will fail.

A single ODT uses 56 bytes of memory. Using all 254 ODTs would require over 14 KB of memory, a large proportion of the available memory on the target. To conserve memory on the target, the default number is low, allowing DAQ list signal monitoring with reduced memory overhead and processing power.

As an example, if you have five different rates in a model, and you are using three rates for DAQ, then this will create three DAQ lists and you must make sure you have at least three ODTs. ODTs are shared equally among DAQ lists and, therefore, you will end up with one ODT per DAQ list. With less than three ODTs, you get zero ODTs per DAQ list and the behavior is undefined.

Taking this example further, say you have three DAQ lists with one ODT each, and start trying to monitor signals in a calibration tool. If you try to assign too many signals to a particular DAQ list (that is, signals requiring more space than seven bytes (one ODT) in this case), then the calibration tool will report this as an error.

## **CRO sample time**

The sample time for CRO messages.

## **Supported CCP Commands**

The following CCP commands are supported by the CAN Calibration Protocol (C166) block:

- CONNECT
- DISCONNECT
- DNLOAD
- DNLOAD\_6
- EXCHANGE\_ID
- GET\_CCP\_VERSION
- GET\_DAQ\_SIZE
- GET\_S\_STATUS

# CAN Calibration Protocol (C166)

---

- SET\_DAQ\_PTR
- SET\_MTA
- SET\_S\_STATUS
- SHORT\_UP
- START\_STOP
- START\_STOP\_ALL
- TEST
- UPLOAD
- WRITE\_DAQ

## **Compatibility with Calibration Packages**

The above commands support

- Synchronous signal monitoring via calibration packages that use DAQ lists
- Asynchronous signal monitoring via calibration packages that poll the target
- Asynchronous parameter tuning via CCP memory programming

This CCP implementation has been tested successfully with the Vector-Informatik CANape calibration package running in both DAQ list and polling mode, and with the Accurate Technologies, Inc., Vision, calibration package running in DAQ list mode. (Note that Accurate Technologies, Inc., Vision does not support the polling mechanism for signal monitoring).



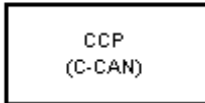
# CAN Calibration Protocol (C166, C-CAN)

---

**Purpose** Implement CAN Calibration Protocol (CCP) standard with C-CAN

**Library** Target Support Package IC1/ C166 Driver Library/ C-CAN Interface

## Description



CAN Calibration Protocol

The CAN Calibration Protocol (C166, C-CAN) block is for the C-CAN interface and performs the same functions as the CAN Calibration Protocol (C166) block. For block parameter descriptions, see the CAN Calibration Protocol (C166) reference page.

# CAN Calibration Protocol (C166, TwinCAN)

---

<b>Purpose</b>	Implement CAN Calibration Protocol (CCP) standard for XC16x variants of Infineon C166 microcontrollers
<b>Library</b>	Target Support Package IC1/ C166 Driver Library/ CAN Interface
<b>Description</b>	The CAN Calibration Protocol (C166, TwinCAN) block is for the TwinCAN interface and performs the same functions as the CAN Calibration Protocol (C166) block. For block parameter descriptions, see the CAN Calibration Protocol (C166) reference page.

**Purpose** Pack individual signals into CAN message

**Library** CAN Communication

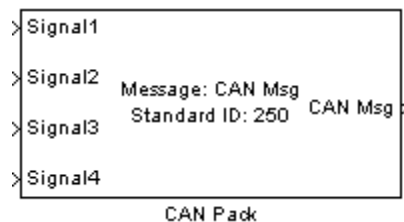
**Description** The CAN Pack block loads signal data into a message at specified intervals during the simulation.

---

**Note** To use this block, you also need a license for Simulink software.

---

CAN Pack block has one input port by default. The number of input ports is dynamic and depends on the number of signals you specify for the block. For example, if your block has four signals, it has four input ports.



This block has one output port, CAN Msg. The CAN Pack block takes the specified input parameters and packs the signals into a message.

### Other Supported Features

The CAN Pack block supports:

- The use of Simulink® Accelerator™ mode. Using this feature, you can speed up the execution of Simulink models.
- The use of model referencing. Using this feature, your model can include other Simulink models as modular components.
- Code generation using Real-Time Workshop to deploy models to targets.

---

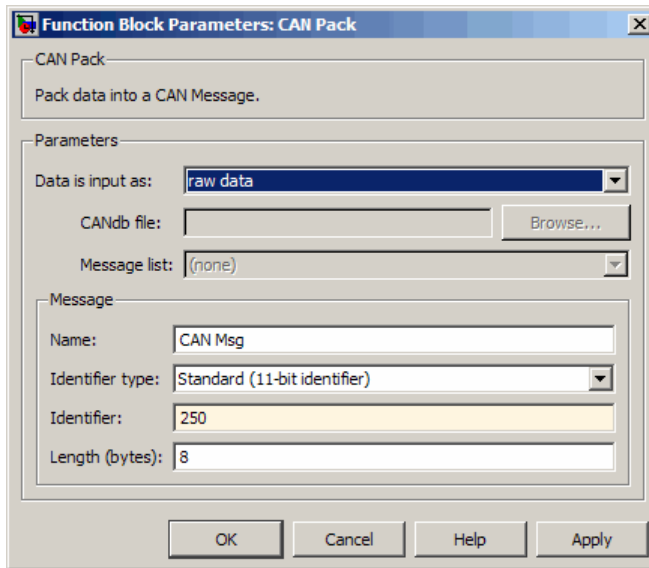
**Note** Code generation is not supported if your signal information consists of signed or unsigned integers greater than 32-bits long.

---

For more information on these features, see the Simulink documentation.

## Dialog Box

Use the Function Block Parameters dialog box to select your CAN Pack block parameters.



## Parameters

### Data is input as

Select your data signal:

- **raw data:** Input data as a uint8 vector array. If you select this option, you only specify the message fields. All other signal

parameter fields are unavailable. This option opens only one input port on your block.

- **manually specified signals:** Allows you to specify data signal definitions. If you select this option, use the **Signals** table to create your signals. The number of input ports on your block depends on the number of signals you specify.

Function Block Parameters: CAN Pack

CAN Pack  
Pack data into a CAN Message.

Parameters

Data is input as: manually specified signals

CANdb file:  Browse...

Message list: (none)

Message

Name: CAN Msg

Identifier type: Standard (11-bit identifier)

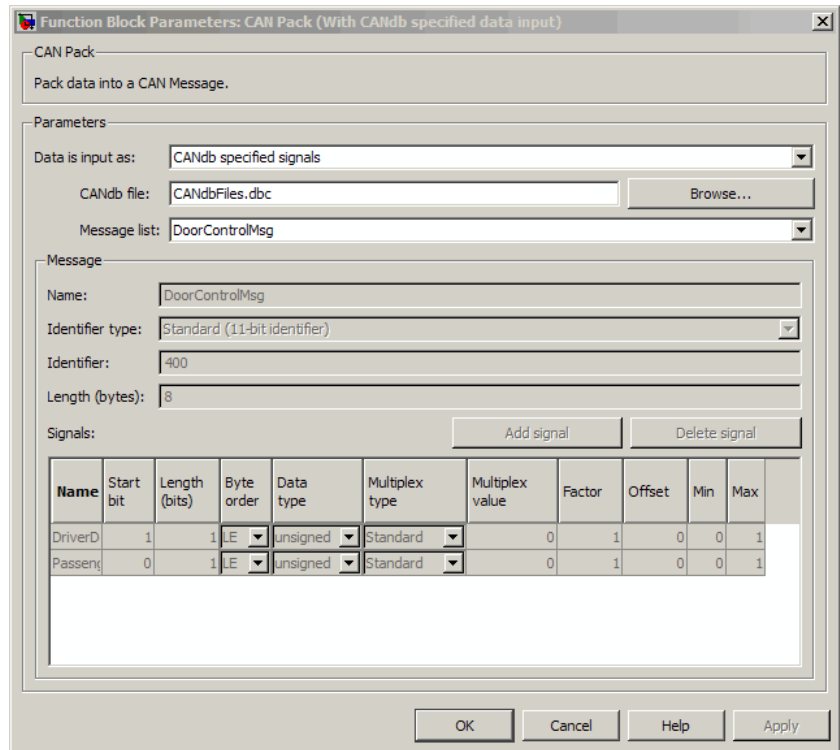
Identifier: 250

Length (bytes): 8

Signals:

Name	Start bit	Length (bits)	Byte order	Data type	Multiplex type	Multiplex value	Factor	Offset	Min	Max
Signal1	0	8	LE	signed	Standard	0	1	0	-Inf	Inf
Signal2	8	8	LE	signed	Standard	0	1	0	-Inf	Inf
Signal3	16	8	LE	signed	Standard	0	1	0	-Inf	Inf
Signal4	24	8	LE	signed	Standard	0	1	0	-Inf	Inf

- **CANdb specified signals:** Allows you to specify a CAN database file that contains message and signal definitions. If you select this option, select a CANdb file. The number of input ports on your block depends on the number of signals specified in the CANdb file for the selected message.



## CANdb file

This option is available if you specify that your data is input via a CANdb file in the **Data is input as** list. Click **Browse** to find the appropriate CANdb file on your system. The message list specified in the CANdb file populates the **Message** section of the dialog box. The CANdb file also populates the **Signals** table for the selected message.

## Message list

This option is available if you specify that your data is input via a CANdb file in the **Data is input as** field and you select a CANdb file in the **CANdb file** field. Select the message to display signal details in the **Signals** table.

## Message

### Name

Specify a name for your CAN message. The default is `CAN Msg`. This option is available if you choose to input raw data or manually specify signals. This option is unavailable if you choose to use signals from a `CANdb` file.

### Identifier type

Specify whether your CAN message identifier is a `Standard` or an `Extended` type. The default is `Standard`. A standard identifier is an 11-bit identifier and an extended identifier is a 29-bit identifier. This option is available if you choose to input raw data or manually specify signals. For `CANdb` specified signals, the **Identifier type** inherits the type from the database.

### Identifier

Specify your CAN message ID. This number must be a positive integer from 0 through 2047 for a standard identifier and from 0 through 536870911 for an extended identifier. You can also specify hexadecimal values using the `hex2dec` function. This option is available if you choose to input raw data or manually specify signals.

### Length (bytes)

Specify the length of your CAN message in bytes. If you are using `CANdb` specified signals for your data input, the `CANdb` file defines the length of your message. If not, this field defaults to 8. This option is available if you choose to input raw data or manually specify signals.

## Signals Table

This table appears if you choose to specify signals manually or define signals using a `CANdb` file.

If you are using a `CANdb` file, the data in the file populates this table automatically and you cannot edit any fields. To edit signal information, switch to manually specified signals.

If you have selected to specify signals manually, create your signals manually in this table. Each signal you create has the following values:

**Name**

Specify a descriptive name for your signal. The Simulink block in your model displays this name. The default is `Signal [row number]`.

**Start bit**

Specify the start bit of the data. The start bit is the least significant bit counted from the start of the message data. The start bit must be an integer from 0 through 63.

**Length (bits)**

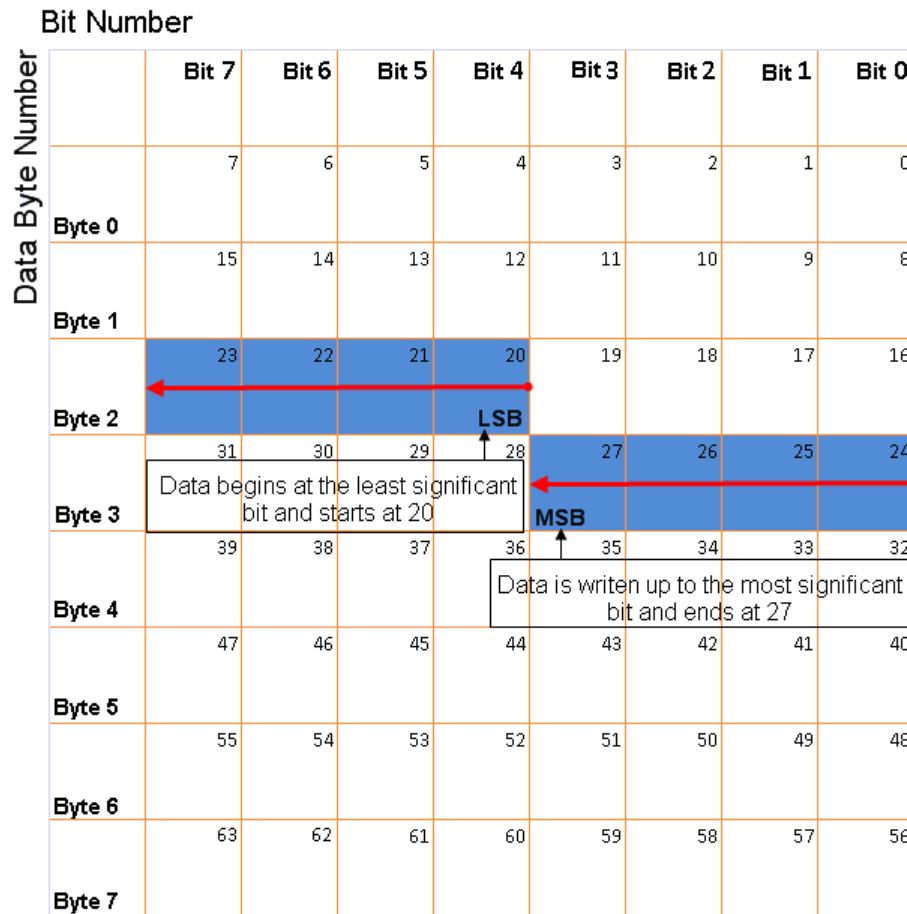
Specify the number of bits the signal occupies in the message. The length must be an integer from 1 through 64.

**Byte order**

Select either of the following options:

- **LE:** Where the byte order is in little-endian format (Intel®). In this format you count bits from the start, which is the least significant bit, to the most significant bit, which has the highest bit index. For example, if you pack one byte of data in little-endian format, with the start bit at 20, the data bit table resembles this figure.





### Little Endian Byte Order Counted from the Least Significant Bit to the Highest Address

- BE: Where byte order is in big-endian format (Motorola®). In this format you count bits from the start, which is the least significant bit, to the most significant bit. For example, if you pack one byte of data in big-endian format, with the start bit at 20, the data bit table resembles this figure.

# CAN Pack

		Bit Number							
		Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
Data Byte Number	Byte 0	7	6	5	4	3	2	1	0
	Byte 1	15	14	13	12	11	10	9	8
	Byte 2	23	22	21	20	19	18	17	16
	Byte 3	31	30	29	28	27	26	25	24
	Byte 4	39	38	37	36	35	34	33	32
	Byte 5	47	46	45	44	43	42	41	40
	Byte 6	55	54	53	52	51	50	49	48
	Byte 7	63	62	61	60	59	58	57	56

## Big Endian Byte Order Counted from the Least Significant Bit to the Lowest Address

### Data type

Specify how the signal interprets the data in the allocated bits.  
Choose from:

- signed (default)

- unsigned
- single
- double

## Multiplex type

Specify how the block packs the signals into the CAN message at each timestep:

- **Standard:** The signal is always packed at each timestep.
- **Multiplexor:** The Multiplexor signal, or the mode signal is always packed. You can specify only one Multiplexor signal per message.
- **Multiplexed:** The signal is packed if the value of the Multiplexor signal (mode signal) at run time matches the configured **Multiplex value** of this signal.

For example, a message has four signals with the following types and values.

Signal Name	Multiplex Type	Multiplex Value
Signal-A	Standard	N/A
Signal-B	Multiplexed	1
Signal-C	Multiplexed	0
Signal-D	Multiplexor	N/A

In this example

- The block packs Signal-A (Standard signal) and Signal-D (Multiplexor signal) in every timestep.
- If the value of Signal-D is 1 at a particular timestep, then the block packs Signal-B along with Signal-A and Signal-D in that timestep.

- If the value of Signal-D is 0 at a particular timestep, then the block packs Signal-C along with Signal-A and Signal-D in that timestep.
- If the value of Signal-D is not 1 or 0, the block does not pack either of the Multiplexed signals in that timestep.

### **Multiplex value**

This option is available only if you have selected the **Multiplex type** to be Multiplexed. The value you provide here must match the Multiplexor signal value at run time for the block to pack the Multiplexed signal. The **Multiplex value** must be a positive integer or zero.

### **Factor**

Specify the **Factor** value to apply to convert the physical value (signal value) to the raw value packed in the message. See “Conversion Formula” on page 7-43 to understand how physical values are converted to raw values packed into a message.

### **Offset**

Specify the **Offset** value to apply to convert the physical value (signal value) to the raw value packed in the message. See “Conversion Formula” on page 7-43 to understand how physical values are converted to raw values packed into a message.

### **Min**

Specify the minimum physical value of the signal. The default value is `-inf` (negative infinity). You can specify any number for the minimum value. See “Conversion Formula” on page 7-43 to understand how physical values are converted to raw values packed into a message.

### **Max**

Specify the maximum physical value of the signal. The default value is `inf`. You can specify any number for the maximum value. See “Conversion Formula” on page 7-43 to understand how physical values are converted to raw values packed into a message.

## Conversion Formula

The conversion formula is

$$\text{raw\_value} = (\text{physical\_value} - \text{Offset}) / \text{Factor}$$

where `physical_value` is the value of the signal after it is saturated using the specified **Min** and **Max** values. `raw_value` is the packed signal value.

## See Also

CAN Unpack

# CAN Receive

---

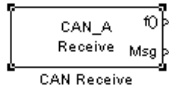
## Purpose

Receive CAN messages from CAN module on Infineon C166 microcontrollers

## Library

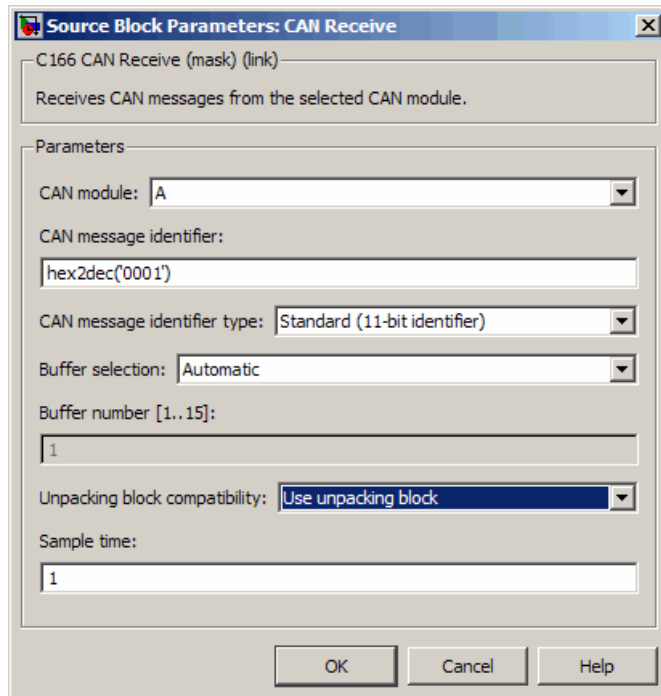
Target Support Package IC1/ C166 Driver Library/ CAN Interface

## Description



The CAN Receive block receives CAN messages from a CAN module. The CAN Receive block can reserve one of the buffers on the CAN module. Alternatively, you can instruct the CAN Receive block to select a hardware buffer automatically from the available buffers. The CAN Receive block has two outputs: a data output and a function-call trigger output. The CAN Receive block polls its message buffer at a rate determined by the block's sample time. When the CAN Receive block detects that a message has arrived, the function-call trigger is activated. You should use a function-call subsystem, activated by the trigger, to decode the message available at the CAN Receive block data output.

## Dialog Box



### **CAN module**

Select CAN module A or B. The CAN modules can receive messages independently.

### **CAN message identifier**

The identifier of the message you want to receive. Note that if you have set the CAN configuration parameters in your model to mask out certain bits (e.g., the message identifier field), you may receive messages with identifiers other than the identifier specified here. See “CAN Configuration Parameters” on page 7-18.

### **CAN message identifier type**

The type of message you want to receive. Select either Standard(11-bit identifier) or Extended(29-bit identifier).

# CAN Receive

---

## Buffer selection

Choose **Automatic** or **Manual**. When the automatic option is selected, the CAN Receive block automatically selects a receive buffer from the available buffers. Use this automatic buffer selection, unless you want to use buffer 15 with its individually programmable mask.

## Buffer number [1..15]

This field is enabled if the **Buffer selection** is **Manual**. The buffer number specifies the identifier of the receive buffer for this block. Select **Automatic** buffer selection instead of manually specifying the buffer, unless you want to use buffer 15 with its individually programmable mask.

## Unpacking block compatibility

Select **Use unpacking block** or **Use message unpacking block (obsolete)**. Choose the latter only if you are using the obsolete Can Message Blocks library (`canblocks.mdl`).

---

**Note** If you have models that use host-side CAN blocks from the obsolete Can Message Blocks library (`canblocks.mdl`), you will see an obsolescence warning message. You should update your models, as the host-side CAN blocks may be removed in a future release

---

## Sample time

Determines the rate at which to sample the buffer to see if a new message has arrived.

---

**Note** The CAN Receive block sample time must be set to a value that is smaller than the minimum time between CAN messages that will be received into the corresponding buffer. If more than one message is received into a buffer during a single sample interval, the older message will be overwritten.

---



## Purpose

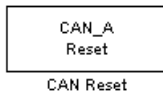
Reset CAN module

## Library

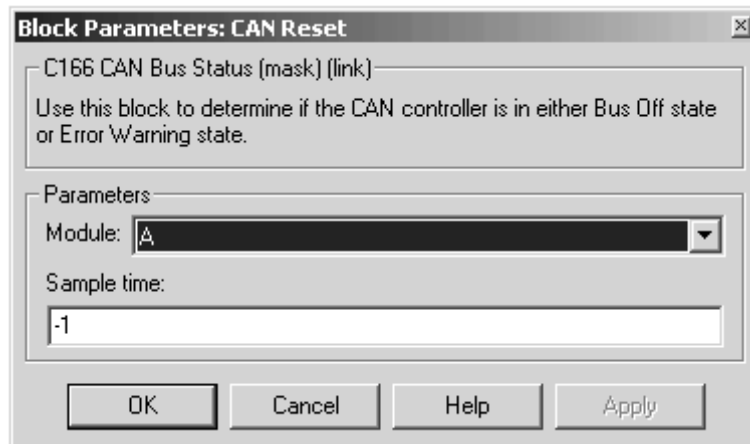
Target Support Package IC1/ C166 Driver Library/CAN Interface

## Description

The CAN Reset block reinitializes the CAN module. We recommend that you place this block in a triggered subsystem, with a sample time of -1 (inherited).



## Dialog Box



### Module

Select CAN module A or B.

### Sample time

The sample time of this block.

# CAN Transmit

---

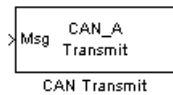
## Purpose

Transmit CAN messages via CAN module on Infineon C166 microcontrollers

## Library

Target Support Package IC1/C166 Driver Library/CAN Interface

## Description



The CAN Transmit block transmits a CAN message onto the CAN bus. Three modes of transmission are available with the CAN Transmit block.

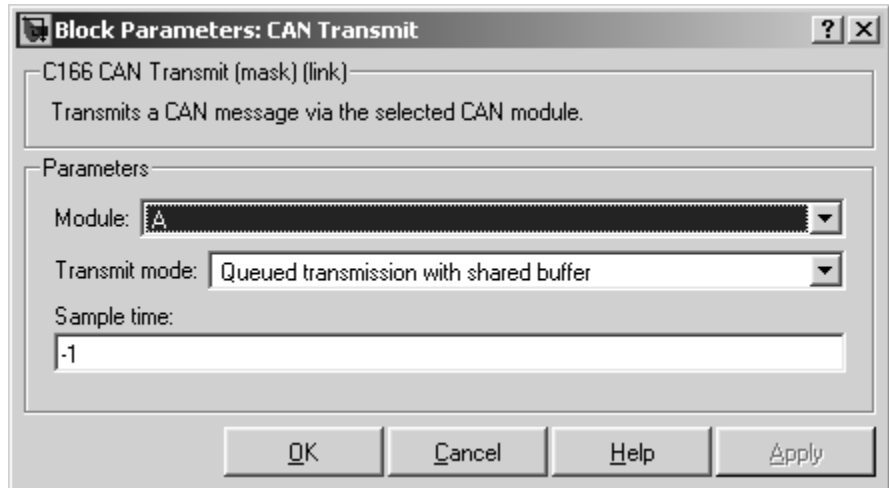
The default mode is to use a priority-based message queue shared by all transmit blocks operating in this mode; the priority-based message queue operates with CAN buffer 14; when a message is successfully transmitted from this buffer, an interrupt is generated and the highest priority message from the queue is loaded into the hardware buffer ready to be transmitted. This mode has the advantage of allowing several messages with different identifiers to be transmitted without each message requiring a dedicated hardware buffer. Note that although messages are taken from the queue in order of priority, it is possible for a low priority message to be present in the hardware buffer and higher priority messages cannot then be transferred from the queue until transmission of the low priority message is complete.

The second transmit mode is to use a dedicated CAN buffer; in this case, messages to be transmitted are loaded directly into a CAN buffer that is used exclusively by the block. No queue is used, which means that in case the previous message has not been transmitted, it will be overwritten by the new one. This transmit mode does not use interrupts. An advantage of using the dedicated buffer mode is that there is reduced delay in transmitting high-priority messages, and reduced processor overhead that is otherwise required for queue management and servicing interrupts.

The third transmit mode is to use a First In First Out (FIFO) queue with dedicated buffer. In this mode, messages are placed in a queue and then transmitted on a first in, first out basis. This mode is useful if several messages, possibly with the same CAN identifier, must be transmitted in sequence; this may be a requirement if CAN is being used for data acquisition.

The CAN Transmit block should be connected to CAN Message Packing/Unpacking blocks. Do not ground the block or leave it unconnected.

## Dialog Box



### Module

Select CAN module A or B.. The CAN modules can receive messages independently.

### Transmit mode

Select one of the three modes described above: queued transmission with shared buffer, direct transmission with dedicated buffer, or FIFO queue with dedicated buffer.

### Buffer selection

Only for selecting dedicated buffers — available only if you select direct transmission or FIFO queue transmit modes. Choose either automatic or manual selection of the hardware buffer number.

### Buffer number

This option is available only if the buffer selection is available and set to manual. You must select a buffer number between 1 and 14. Note if more than one message is ready to be transmitted,

# CAN Transmit

---

then the one in the lower buffer number will be sent first. Select buffer numbers such that the higher the message priority, the lower the buffer number.

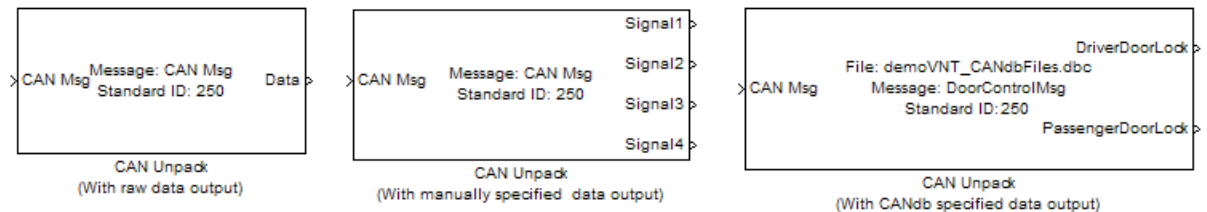
## **Sample time**

Choose -1 to inherit the sample time from the driving blocks. The CAN Transmit block does not inherit constant sample times and runs at the base rate of the model if driven by invariant signals.

**Purpose** Unpack individual signals from CAN messages

**Library** CAN Communication

## Description



The CAN Unpack block unpacks a CAN message into signal data using the specified output parameters at every timestep. Data is output as individual signals.

---

**Note** To use this block, you also need a license for Simulink software.

---

The CAN Unpack block has one output port by default. The number of output ports is dynamic and depends on the number of signals you specify for the block to output. For example, if your block has four signals, it has four output ports.



## Other Supported Features

The CAN Unpack block supports:

- The use of Simulink Accelerator mode. Using this feature, you can speed up the execution of Simulink models.
- The use of model referencing. Using this feature, your model can include other Simulink models as modular components.
- Code generation using Real-Time Workshop to deploy models to targets.

---

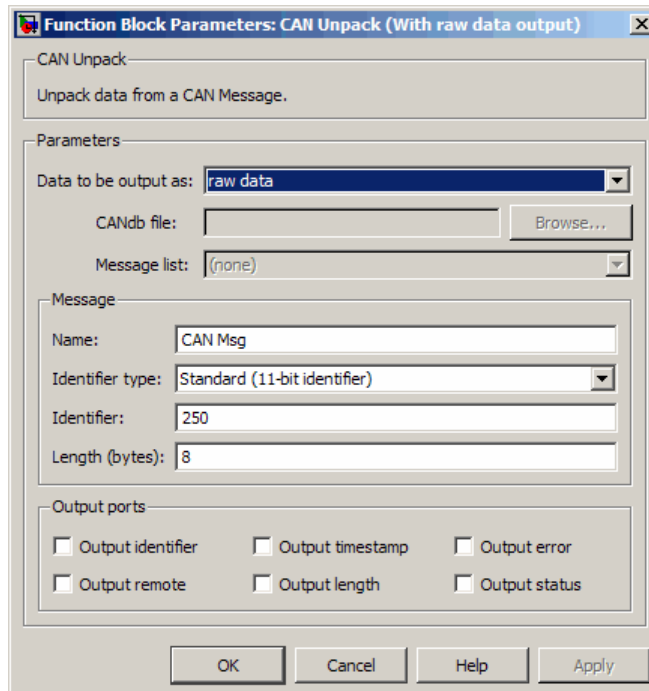
**Note** Code generation is not supported if your signal information consists of signed or unsigned integers greater than 32-bits long.

---

For more information on these features, see the Simulink documentation.

## Dialog Box

Use the Function Block Parameters dialog box to select your CAN message unpacking parameters.



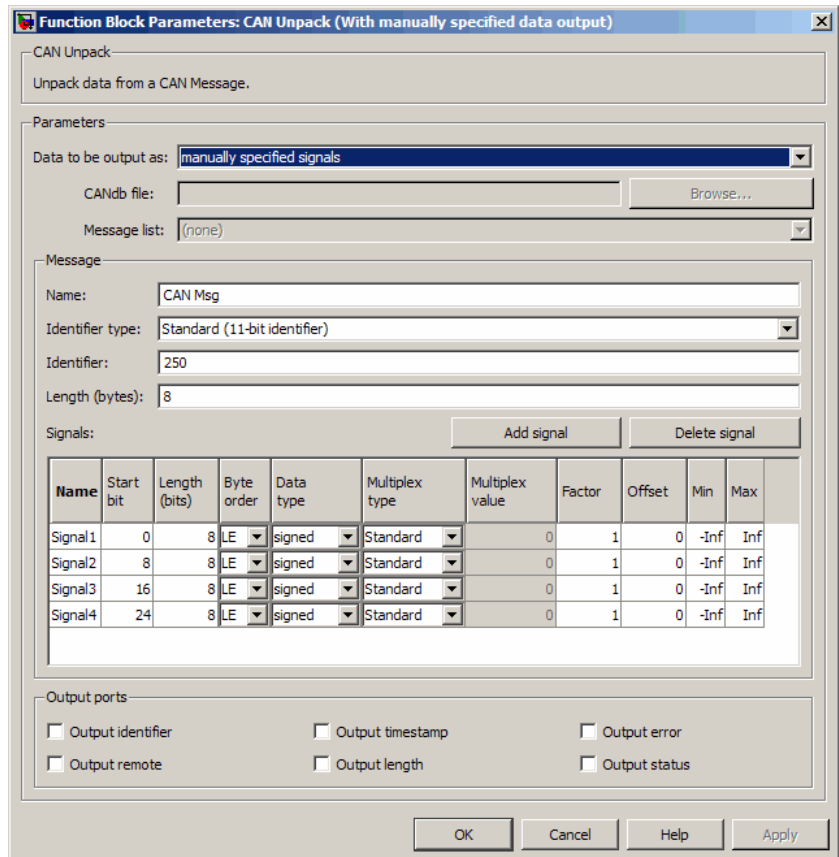
## Parameters

### Data to be output as

Select your data signal:

- **raw data:** Output data as a uint8 vector array. If you select this option, you only specify the message fields. All other signal parameter fields are unavailable. This option opens only one output port on your block.
- **manually specified signals:** Allows you to specify data signals. If you select this option, use the Signals table to create your signals message manually.

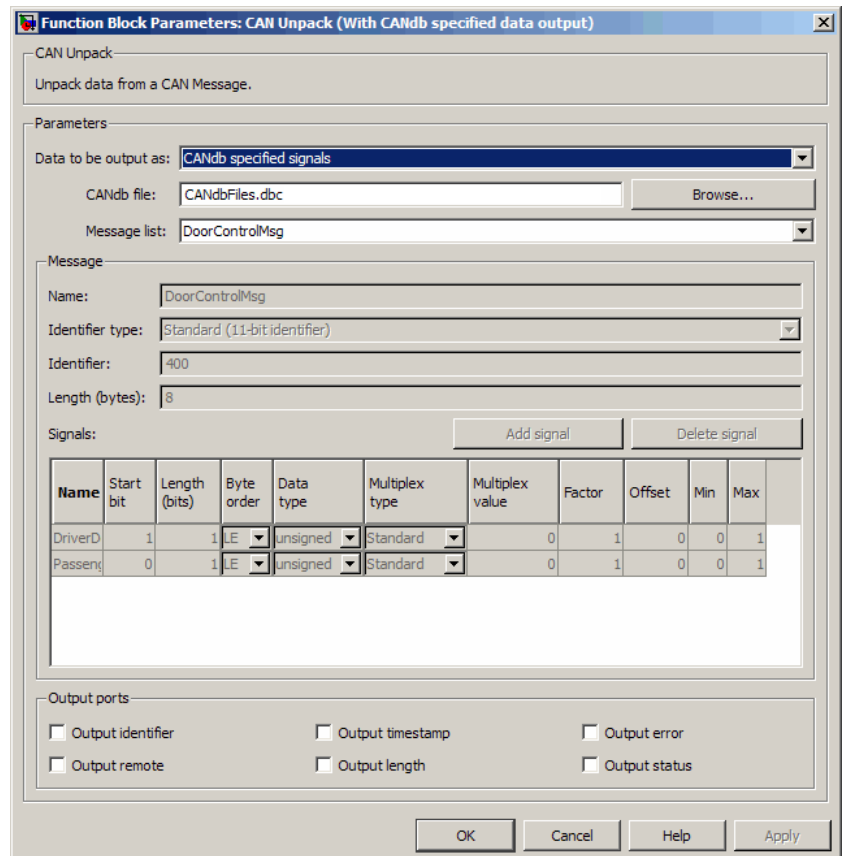
# CAN Unpack



The number of output ports on your block depends on the number of signals you specify. For example, if you specify four signals, your block has four output ports.

- **CANdb specified signals:** Allows you to specify a CAN database file that contains data signals. If you select this option, select a CANdb file.





The number of output ports on your block depends on the number of signals specified in the CANdb file. For example, if the selected message in the CANdb file has four signals, your block has four output ports.

### CANdb file

This option is available if you specify that your data is input via a CANdb file in the **Data to be output as** list. Click **Browse** to find the appropriate CANdb file on your system. The messages and signal definitions specified in the CANdb file populate the

**Message** section of the dialog box. The signals specified in the CANdb file populate **Signals** table.

## **Message list**

This option is available if you specify that your data is to be output as a CANdb file in the **Data to be output as** list and you select a CANdb file in the **CANdb file** field. You can select the message that you want to view. The **Signals** table then displays the details of the selected message.

## **Message**

### **Name**

Specify a name for your CAN message. The default is CAN Msg. This option is available if you choose to output raw data or manually specify signals.

### **Identifier type**

Specify whether your CAN message identifier is a **Standard** or an **Extended** type. The default is **Standard**. A standard identifier is an 11-bit identifier and an extended identifier is a 29-bit identifier. This option is available if you choose to output raw data or manually specify signals. For CANdb-specified signals, the **Identifier type** inherits the type from the database.

### **Identifier**

Specify your CAN message ID. This number must be a integer from 0 through 2047 for a standard identifier and from 0 through 536870911 for an extended identifier. If you specify 1, the block unpacks all messages that match the length specified for the message. You can also specify hexadecimal values using the hex2dec function. This option is available if you choose to output raw data or manually specify signals.

### **Length (bytes)**

Specify the length of your CAN message in bytes. If you are using **CANdb specified signals** for your output data, the CANdb file defines the length of your message. If not, this field defaults to

8. This option is available if you choose to output raw data or manually specify signals.

## Signals Table

This table appears if you choose to specify signals manually or define signals using a CANdb file.

If you are using a CANdb file, the data in the file populates this table automatically and you cannot edit any fields. To edit signal information, switch to manually specified signals.

If you have selected to specify signals manually, create your signals manually in this table. Each signal you create has the following values:

### Name

Specify a descriptive name for your signal. The Simulink block in your model displays this name. The default is Signal [row number].

### Start bit

Specify the start bit of the data. The start bit is the least significant bit counted from the start of the message. The start bit must be an integer from 0 through 63.

### Length (bits)

Specify the number of bits the signal occupies in the message. The length must be an integer from 1 through 64.

### Byte order

Select either of the following options:

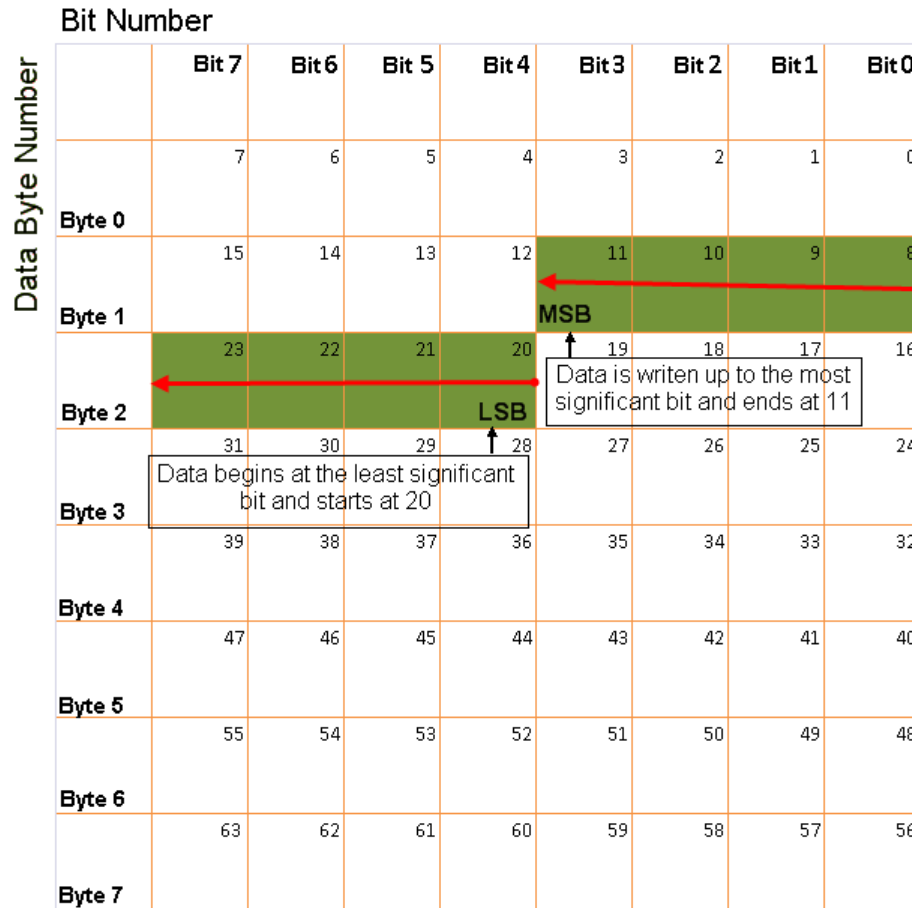
- LE: Where the byte order is in little-endian format (Intel). In this format you count bits from the start, which is the least significant bit, to the most significant bit, which has the highest bit index. For example, if you pack one byte of data in little-endian format, with the start bit at 20, the data bit table resembles this figure.

# CAN Unpack

		Bit Number							
Data Byte Number		Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
	Byte 0		7	6	5	4	3	2	1
Byte 1		15	14	13	12	11	10	9	8
Byte 2		23	22	21	20	19	18	17	16
Byte 3		31	30	29	28	27	26	25	24
Byte 4		39	38	37	36	35	34	33	32
Byte 5		47	46	45	44	43	42	41	40
Byte 6		55	54	53	52	51	50	49	48
Byte 7		63	62	61	60	59	58	57	56

## Little Endian Byte Order Counted from the Least Significant Bit to the Highest Address

- BE: Where the byte order is in big-endian format (Motorola). In this format you count bits from the start, which is the least significant bit, to the most significant bit. For example, if you pack one byte of data in big-endian format, with the start bit at 20, the data bit table resembles this figure.



## Big Endian Byte Order Counted from the Least Significant Bit to the Lowest Address

### Data type

Specify how the signal interprets the data in the allocated bits.  
Choose from:

- signed (default)

# CAN Unpack

---

- unsigned
- single
- double

## Multiplex type

Specify how the block unpacks the signals from the CAN message at each timestep:

- **Standard:** The signal is always unpacked at each timestep.
- **Multiplexor:** The Multiplexor signal, or the mode signal is always unpacked. You can specify only one Multiplexor signal per message.
- **Multiplexed:** The signal is unpacked if the value of the Multiplexor signal (mode signal) at run time matches the configured **Multiplex value** of this signal.

For example, if a message has four signals with the following values.

Signal Name	Multiplex Type	Multiplex Value
Signal-A	Standard	N/A
Signal-B	Multiplexed	1
Signal-C	Multiplexed	0
Signal-D	Multiplexor	N/A

In this example

- The block unpacks Signal-A (Standard signal) and Signal-D (Multiplexor signal) in every timestep.
- If the value of Signal-D is 1 at a particular timestep, then the block unpacks Signal-B along with Signal-A and Signal-D in that timestep.

- If the value of Signal-D is 0 at a particular timestep, then the block unpacks Signal-C along with Signal-A and Signal-D in that timestep.
- If the value of Signal-D is not 1 or 0, the block does not unpack either of the Multiplexed signals in that timestep.

### **Multiplex value**

This option is available only if you have selected the **Multiplex type** to be Multiplexed. The value you provide here must match the Multiplexor signal value at run time for the block to unpack the Multiplexed signal. The **Multiplex value** must be a positive integer or zero.

### **Factor**

Specify the **Factor** value applied to convert the unpacked raw value to the physical value (signal value). See “Conversion Formula” on page 7-62 to understand how unpacked raw values are converted to physical values.

### **Offset**

Specify the **Offset** value applied to convert the physical value (signal value) to the unpacked raw value. See “Conversion Formula” on page 7-62 to understand how unpacked raw values are converted to physical values.

### **Min**

Specify the minimum raw value of the signal. The default value is `-inf` (negative infinity). You can specify any number for the minimum value. See “Conversion Formula” on page 7-62 to understand how unpacked raw values are converted to physical values.

### **Max**

Specify the maximum raw value of the signal. The default value is `inf`. You can specify any number for the maximum value. See “Conversion Formula” on page 7-62 to understand how unpacked raw values are converted to physical values.

## Output Ports

Selecting an **Output ports** option adds an output port to your block.

### Output identifier

Select this option to output a CAN message identifier. The data type of this port is **uint32**.

### Output remote

Select this option to output the message remote frame status. This option adds a new output port to the block. The data type of this port is **uint8**.

### Output timestamp

Select this option to output the message time stamp. This option adds a new output port to the block. The data type of this port is **double**.

### Output length

Select this option to output the length of the message in bytes. This option adds a new output port to the block. The data type of this port is **uint8**.

### Output error

Select this option to output the message error status. This option adds a new output port to the block. The data type of this port is **uint8**.

### Output status

Select this option to output the message received status. The status is 1 if the block receives new message and 0 if it does not. This option adds a new output port to the block. The data type of this port is **uint8**.

If you do not select any **Output ports** option, the number of output ports on your block depends on the number of signals you specify.

## Conversion Formula

The conversion formula is



$$\text{physical\_value} = \text{raw\_value} * \text{Factor} + \text{Offset}$$

where `raw_value` is the unpacked signal value. `physical_value` is the scaled signal value which is saturated using the specified **Min** and **Max** values.

## See Also

CAN Pack

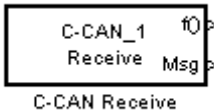
# C-CAN Receive

---

**Purpose** Receive CAN messages from C-CAN module on ST10 microcontrollers

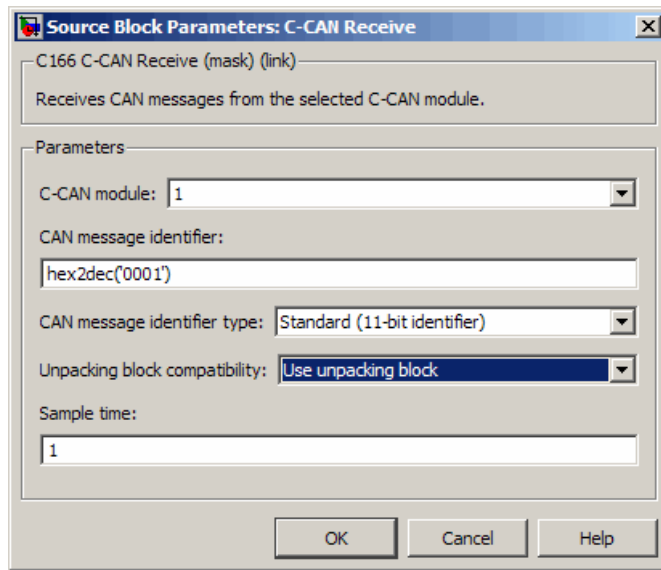
**Library** Target Support Package IC1/ C166 Driver Library/ C-CAN Interface

## Description



The C-CAN Receive block receives CAN messages from a C-CAN module. The C-CAN Receive block automatically reserves one of the buffers on the C-CAN module. The C-CAN Receive block has two outputs: a data output and a function-call trigger output. The C-CAN Receive block polls its message buffer at a rate determined by the block's sample time. When the CAN Receive block detects that a message has arrived, the function-call trigger is activated. You should use a function-call subsystem, activated by the trigger, to decode the message available at the CAN Receive block data output.

## Dialog Box



### **C-CAN module**

Select C-CAN module 1 or 2. The C-CAN modules can receive messages independently.

### **CAN message identifier**

The identifier of the message you want to receive.

### **CAN message identifier type**

The type of message you want to receive. Select either Standard(11-bit identifier) or Extended(29-bit identifier).

### **Unpacking block compatibility**

Select Use unpacking block or Use message unpacking block (obsolete). Choose the latter only if you are using the obsolete Can Message Blocks library (canblocks.mdl).

# C-CAN Receive

---

---

**Note** If you have models that use host-side CAN blocks from the obsolete Can Message Blocks library (`canblocks.mdl`), you will see an obsolescence warning message. You should update your models, as the host-side CAN blocks may be removed in a future release

---

## Sample time

Determines the rate at which to sample the buffer to see if a new message has arrived.

---

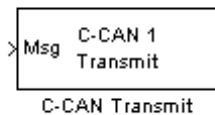
**Note** The C-CAN Receive block sample time must be set to a value that is smaller than the minimum time between CAN messages that will be received into the corresponding buffer. If more than one message is received into a buffer during a single sample interval, the older message will be overwritten.

---

**Purpose** Transmit CAN messages via C-CAN module on ST10 microcontrollers

**Library** Target Support Package IC1/C166 Driver Library/C-CAN Interface

## Description



The C-CAN Transmit block transmits a CAN message onto the CAN bus. Two modes of transmission are available with the C-CAN Transmit block.

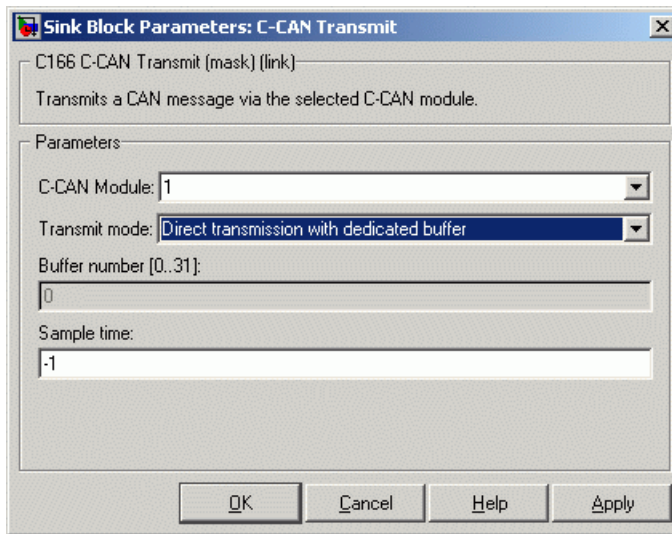
The default transmit mode is to use a dedicated CAN buffer; in this case, messages to be transmitted are loaded directly into a CAN buffer that is used exclusively by the block. No queue is used, which means that in case the previous message has not been transmitted, it will be overwritten by the new one. This transmit mode does not use interrupts. An advantage of using the dedicated buffer mode is that there is reduced delay in transmitting high-priority messages, and reduced processor overhead that is otherwise required for queue management and servicing interrupts.

The other transmit mode is to use a First In First Out (FIFO) queue with dedicated buffer. In this mode, messages are placed in a queue and then transmitted on a first in, first out basis. This mode is useful if several messages, possibly with the same CAN identifier, must be transmitted in sequence; this may be a requirement if CAN is being used for data acquisition.

The C-CAN Transmit block should be connected to CAN Message Packing/Unpacking blocks. Do not ground the block or leave it unconnected.

# C-CAN Transmit

## Dialog Box



### C-CAN Module

Select C-CAN module 1 or 2.. The C-CAN modules can receive messages independently.

### Transmit mode

Select one of the two modes described above: direct transmission with dedicated buffer, or FIFO queue with dedicated buffer.

### Length (number of messages) of FIFO queue

This option is available only if you select the transmit mode FIFO queue with dedicated buffer.

### Buffer number

This parameter is for information only. It may be useful for reviewing code to know which hardware buffer is used for which block.

### Sample time

Choose -1 to inherit the sample time from the driving blocks. The CAN Transmit block does not inherit constant sample times and runs at the base rate of the model if driven by invariant signals.

## Purpose

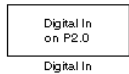
Digital input driver that reads value of specified port or pin number

## Library

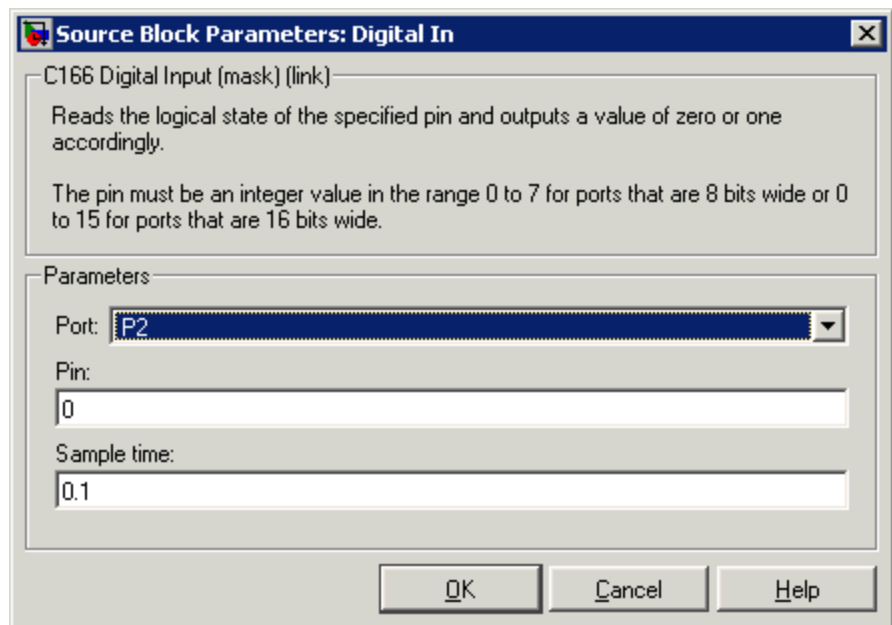
Target Support Package IC1/C166 Driver Library/Digital Input/Output

## Description

The Digital In block reads the logical state of the specified pin and outputs a value of zero or one accordingly.



## Dialog Box



### Port

Select a port. Options are P0L, P0H, P1L, P1H, P2–P8.

### Pin

The pin must be an integer value in the range 0 to 7 for ports that are 8 bits wide, or 0 to 15 for ports that are 16 bits wide.

## **Sample time**

The time interval between samples. The default is 0.1. See “Specifying Sample Time” in the Simulink documentation for more information.



## Purpose

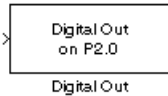
Digital output driver that sets logical state of specified pin

## Library

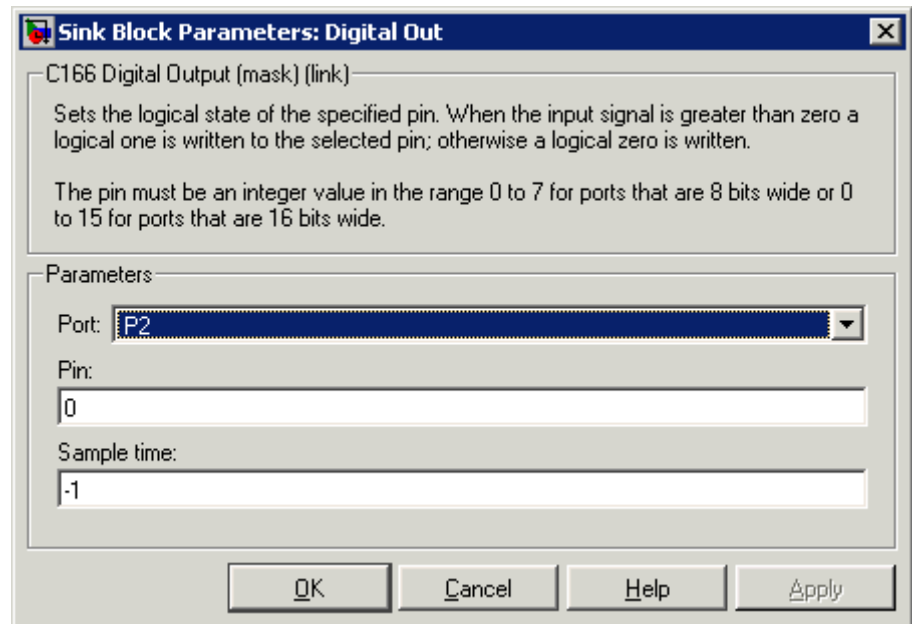
Target Support Package IC1/ C166 Driver Library/ Digital Input/Output

## Description

The Digital Out block sets the logical state of the specified pin according to the input signal. When the input signal is greater than zero, a logical one is written to the selected pin; otherwise a logical zero is written.



## Dialog Box



## Port

Select a port. Options are P0L, P0H, P1L, P1H, P2–P8 (not P5).

# Digital Out

---

## **Pin**

The pin must be an integer value in the range 0 to 7 for ports that are 8 bits wide, or 0 to 15 for ports that are 16 bits wide.

## **Sample time**

The time interval between samples. The default is -1, inherited. See “Specifying Sample Time” in the Simulink documentation for more information.

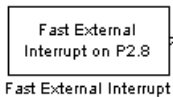
## Purpose

Generate asynchronous function-call trigger when interrupt occurs

## Library

Target Support Package IC1/ C166 Driver Library/ Interrupts

## Description



The Fast External Interrupt block executes a function-call triggered subsystem in the context of the service routine for a fast external interrupt. To generate the interrupt, you must select one of the upper eight pins of Port 2 (P2.8 to P2.15)

The function-call subsystem will be executed as an asynchronous task. Use this block to assign the task a Simulink priority and a CPU interrupt level. The settings that you assign must be consistent with priorities and interrupt levels of other tasks defined in the model.

### Limitations on XC16x Hardware

On XC16x hardware, this block is unable to generate code to enable fast external interrupts. Fast external interrupts must be enabled by setting bits in the register EXICON. On XC16x devices this register is write protected after execution of the special EINIT instruction by the processor's register security mechanism. It is not possible for the driver block to generate code that is executed before the EINIT instruction.

If you want to use this block on XC16x hardware, you must set the required bits in register EXICON in the Project Options within the Tasking EDE. For example, to enable fast external interrupts on rising or falling edges for both of pins P2.8 and P2.9 (as required by the demo model `c166_async`), follow these steps:

- 1** Build the model `c166_async`.
- 2** Open the project `c166_async_c166` within the Tasking EDE.
- 3** Select **Project > Project Options**.
- 4** In the Project Options dialog box, select in the tree **Application > Startup > EXICON**.
  - a** Set the value to `0x000F`.

# Fast External Interrupt

---

- b** Select the check box to **Include in startup code**.
- 5** From within the Tasking EDE, re-build the project `c166_async_c166`.
- 6** Download to the XC16x by launching Crossview from the CrossView button in the Tasking EDE.

Alternatively, you can create a new template project with the required setting for EXICON. You can easily create a new template project from the MATLAB **Start** menu by selecting **Start > Links and Targets > Embedded IDE Link TS > Create New Template Projects**.

## Dialog Box

### Port 2 pin number

Select a port. Options are 8 to 15.

### Trigger mode

Select from Rising or falling edge (the default), Rising edge, Falling edge, or Disabled.

### Priority

Set a Simulink priority. The default is 30.

### Interrupt level

Select an interrupt level from 1 to 15. The default is 5.

### Interrupt level group

Select an interrupt level group from 0 to 3. The default is 1.

### Show simulation input

Select this check box (and click **Apply**) to get an input port for simulation.

**Purpose** Configure C166 microcontroller for serial receive

**Library** Target Support Package IC1/ C166 Driver Library/  
Asynchronous/Synchronous Serial Interface

**Description** The Serial Receive block receives bytes over the Infineon C166 microcontroller Synchronous/Asynchronous Serial Interface ASC0. It requests either a fixed number of bytes to be received, or by enabling the first input, a variable number of bytes can be requested each time this block is called.

When the block is called, the requested number of bytes are retrieved from a FIFO buffer that is internal to the device driver. If this buffer contains fewer bytes than the number requested, these bytes are pulled from the buffer and made available at the block output. The number of bytes actually retrieved from the buffer is made available at the second output. This block retrieves only those bytes that have already been received and placed in the internal buffer; it never waits for additional data to be received.

Whenever bytes are received at the serial interface, a Peripheral Event Controller (PEC) interrupt is generated to move the byte into the internal buffer. If there is no more space available in the internal buffer, any additional data is lost. The PEC interrupts are extremely fast and have minimal effect on the rest of the application.

To configure the serial interface bit rate, buffer size, PEC interrupt priority, and other parameters, see “Asynchronous/Synchronous Serial Interface Configuration Parameters” on page 7-16.

# Serial Receive

---

---

**Note** If your model contains a serial transmit or receive block, it is not possible to perform on-chip debugging over the same serial interface. Attempting to use the debugger in this case causes an error. If you need to debug an application that includes the serial transmit and receive blocks, you must run the debugger using a hardware simulator; alternatively, it may be possible to run your debugger on-chip without using the serial interface, for example, if debugging over CAN is available. See “Debugging and Using The Code Profile Report” on page 2-12.

---

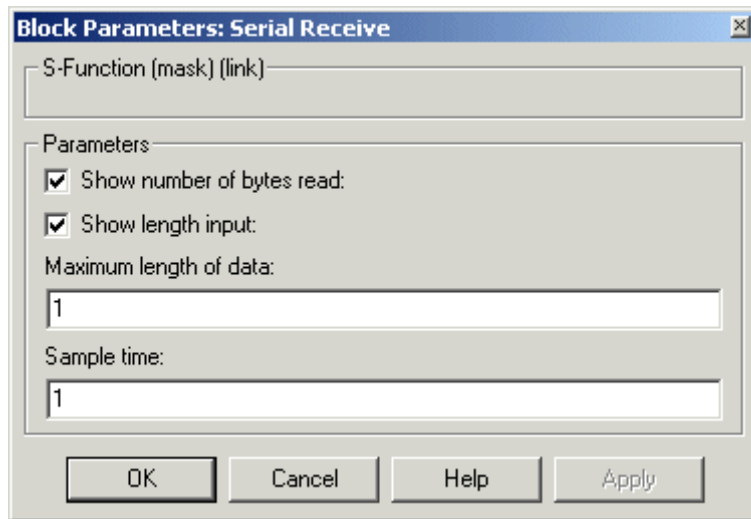
## Block Inputs and Outputs

The input can be enabled so a variable number of bytes can be requested each time.

The first output pulls bytes from the buffer — either the number requested or the number available, whichever is the lower. Note that the number requested is the value of input signal if supplied, or the width of output signal otherwise.

The second output is the number of bytes actually retrieved from the buffer.

## Dialog Box



### **Show number of bytes read**

Enables second output to show actual number of bytes retrieved from the buffer.

### **Show length input**

Enables inport so you can vary the number of bytes requested per call.

### **Maximum length of data**

Set this as required up to the maximum buffer size. You can set receive and transmit buffer size (up to a maximum of 256 bytes) within the C166 Resource Configuration object. See “Asynchronous/Synchronous Serial Interface Configuration Parameters” on page 7-16.

### **Sample time**

The time interval between samples. The default is 1. To inherit the sample time, set this parameter to -1. See “Specifying Sample Time” in the Simulink documentation for more information.

# Serial Transmit

---

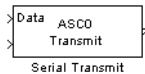
## Purpose

Configure Infineon C166 microcontroller for serial transmit

## Library

Target Support Package IC1/ C166 Driver Library/  
Asynchronous/Synchronous Serial Interface

## Description



The Serial Transmit block transmits bytes over the Infineon C166 microcontroller Synchronous/Asynchronous Serial Interface ASC0. You can use it either to transmit a fixed number of bytes, or by enabling the second input, transmit a variable number of bytes each time this block is called.

When the block is called, the specified number of bytes are placed in a FIFO buffer that is internal to the device driver. If this buffer is already full, or if the number of spaces available is too few, then not all of the bytes requested will actually be queued for transmit; in this case, the number of bytes actually transmitted can be determined from block output.

Once bytes are queued for transmit, they will be sent as fast as possible by the serial interface hardware with no further intervention required by the main application. Note that after each byte is sent, a Peripheral Event Controller (PEC) interrupt is generated to fetch the next byte from the internal buffer. The PEC interrupts are extremely fast and have minimal effect on the rest of the application.

To configure the serial interface bit rate, buffer size, PEC interrupt priority, and other parameters, see “Asynchronous/Synchronous Serial Interface Configuration Parameters” on page 7-16.



---

**Note** If your model contains a serial transmit or receive block, it is not possible to perform on-chip debugging over the same serial interface. Attempting to use the debugger in this case causes an error. If you need to debug an application that includes the serial transmit and receive blocks, you must run the debugger using a hardware simulator; alternatively, it may be possible to run your debugger on-chip without using the serial interface, for example if debugging over CAN is available. See “Debugging and Using The Code Profile Report” on page 2-12.

---

## Block Inputs and Outputs

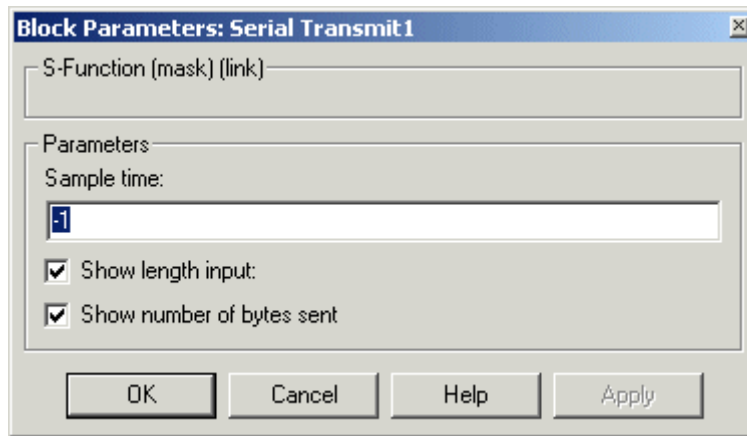
The first input contains the data to be transmitted; this input signal may be either a vector or scalar with data type `uint8`.

The optional second input must be a scalar and may be used to control the number of bytes transmitted. The number of bytes to transmit should not be greater than the width of the first input signal.

The block output port `actual number of bytes output` gives the number of bytes queued for transmit. If there was sufficient space in the buffer, this number will be equal to the requested number of bytes to transmit.

# Serial Transmit

## Dialog Box



### Sample time

The time interval between samples. To inherit the sample time, leave this parameter at the default -1. See “Specifying Sample Time” in the Simulink documentation for more information.

### Show length input

Enable/disable the number of bytes to send. If not selected, the number of bytes sent is just the width of the first inport; if selected, the second input is enabled, which controls the number of bytes to send.

### Show number of bytes sent

Enable/disable the number of bytes actually sent. If selected, this value is available from the first output.

# Switch External Mode Configuration

---

<b>Purpose</b>	Configure model for external mode or executable building
<b>Library</b>	Target Support Package IC1/ C166 Driver Library/ Utilities
<b>Description</b>	<p>Place the Switch External Mode Configuration block in your model and double-click it to run a convenience function to configure your model for building an executable, or executing your model in external mode. When you double-click the block, a dialog box appears. Choose either <b>Building an executable</b> or <b>External mode</b>, and click <b>OK</b>.</p> <p>When you choose building an executable, messages at the command line inform you the following steps are taken to configure your model:</p> <ol style="list-style-type: none"><li><b>1 Inline parameters</b> are selected (under Optimization in the Configuration Parameters dialog box). This is required for ASAP2 generation</li><li><b>2 Normal</b> simulation mode is selected (in the Simulation menu, and drop-down list in the toolbar).</li><li><b>3</b> ASAP2 is selected as the <b>Interface</b> (under <b>Real-Time Workshop</b>, Interface, in the Data Exchange pane, in the Configuration Parameters dialog box).</li></ol> <p>When you choose external mode, messages at the command line inform you the following steps are taken to configure your model:</p> <ol style="list-style-type: none"><li><b>1 Inline parameters</b> are selected (under Optimization in the Configuration Parameters dialog box). This is required for external mode.</li><li><b>2 External</b> simulation mode is selected (in the Simulation menu, and drop-down list in the toolbar).</li><li><b>3</b> External mode is selected as the <b>Interface</b> (under <b>Real-Time Workshop</b>, Interface, in the Data Exchange pane, in the Configuration Parameters dialog box).</li></ol>

# Switch External Mode Configuration

---

See “Using External Mode” on page 2-18 for instructions for converting a model to use external mode for signal logging and parameter tuning.

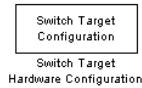
## Purpose

Configure model and Target Preferences to one of a set of predefined hardware configurations

## Library

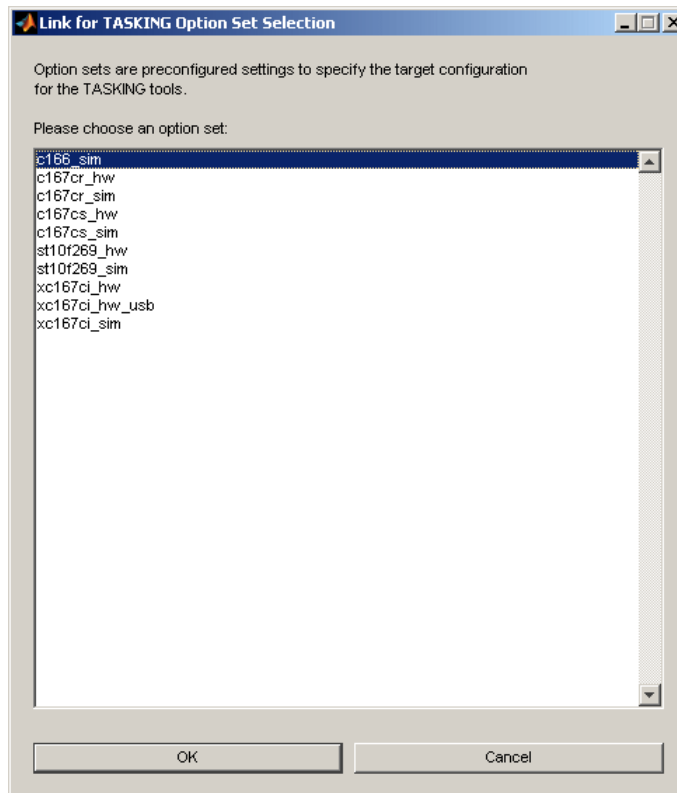
Target Support Package IC1/ C166 Driver Library/ Utilities

## Description



Place the Switch Target Configuration block in your model and double-click it to run a convenience function that configures your model and Target Preferences to one of a set of predefined configurations. The Embedded IDE Link TS Option Set Selection dialog box appears, and you must choose a configuration for your processor type from the list. The suffixes '\_hw' and '\_sim' mean hardware or instruction set simulator. See "Option Sets" in the Embedded IDE Link TS documentation for more information.

# Switch Target Configuration



The predefined configurations include settings for

- Phytex phyCORE-167 ST10F269
- Phytex phyCORE-167 C167CS
- Phytex kitCON-167 C167CR
- Infineon XC167CI Starter Kit

The option set `xc167ci_hw_usb` is for USB wiggler connection to the XC167CI.

# Switch Target Configuration

---

---

**Note** You must change jumper 501 when switching between USB wiggler and on-board parallel port wiggler for this target.

---

# TwinCAN Bus Status

---

## Purpose

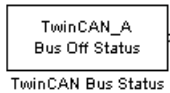
Output Bus Off or Error Warning state of a CAN node on XC16x variants of Infineon C166 microcontrollers

## Library

Target Support Package IC1/ C166 Driver Library/ TwinCAN Interface

## Description

The TwinCAN Bus Status block is for the TwinCAN interface and performs the same functions as the CAN Bus Status block. For block parameter descriptions, see the CAN Bus Status reference page.





<b>Purpose</b>	Receive CAN messages via TwinCAN module on XC16x variants of Infineon C166 microcontrollers
<b>Library</b>	Target Support Package IC1/ C166 Driver Library/ TwinCAN Interface
<b>Description</b>	<p>The TwinCAN Receive block receives CAN messages from a TwinCAN module. The TwinCAN Receive automatically reserves one of the buffers on the TwinCAN module. The TwinCAN Receive block has two outputs: a data output and a function call trigger output. The TwinCAN Receive block polls its message buffer at a rate determined by the block's sample time. When the TwinCAN Receive block detects that a message has arrived, the function call trigger is activated. You should use a function call subsystem, activated by the trigger, to decode the message available at the TwinCAN Receive block data output.</p> <p>This block has the same parameters as the CAN Receive block, except there is no option to <code>Automatically select buffer</code> or <code>Buffer number</code>. For block parameter descriptions, see the CAN Receive reference page.</p>

# TwinCAN Reset

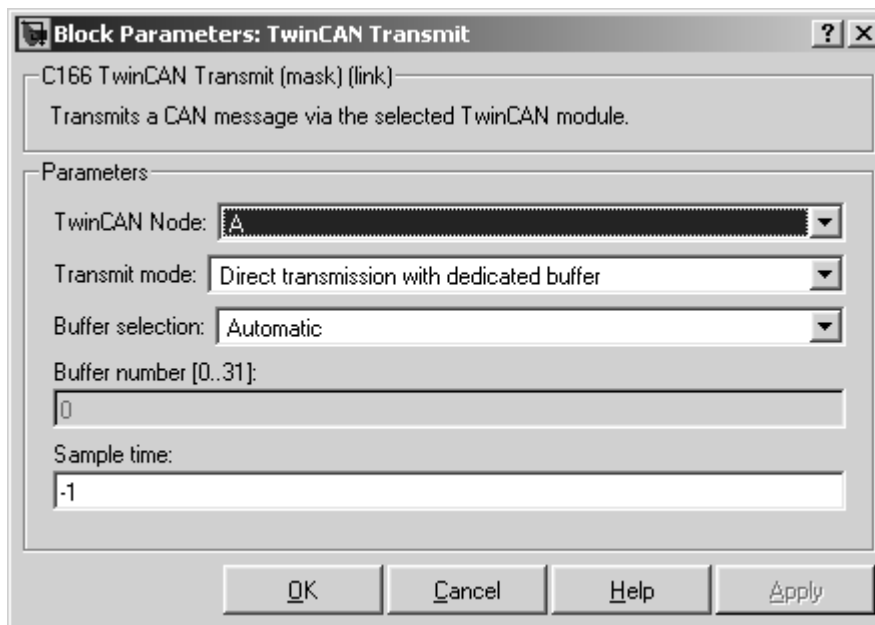
---

<b>Purpose</b>	Reset CAN node on XC16x variants of Infineon C166 microcontrollers
<b>Library</b>	Target Support Package IC1/C166 Driver Library/TwinCAN Interface
<b>Description</b>	The TwinCAN Reset block is for the TwinCAN interface and performs the same functions as the CAN Reset block. For block parameter descriptions, see the CAN Reset reference page.

<b>Purpose</b>	Transmit CAN messages from TwinCAN module on XC16x variants of Infineon C166 microcontrollers
<b>Library</b>	Target Support Package IC1/C166 Driver Library/ TwinCAN Interface
<b>Description</b>	<p>The TwinCAN Transmit block transmits a CAN message onto the CAN bus. Two modes of transmission are available with the CAN Transmit block, as described below.</p> <p>The first transmit mode is to use a dedicated CAN buffer; in this case, messages to be transmitted are loaded directly into a CAN buffer that is used exclusively by the block. No queue is used, which means that in case the previous message has not been transmitted, it will be overwritten by the new one. This transmit mode does not use interrupts. An advantage of using the dedicated buffer mode is that there is minimal delay in transmitting high-priority messages.</p> <p>The second transmit mode is to use a First In First Out (FIFO) queue with dedicated buffer. In this mode, messages are placed in a queue and then transmitted on a first in, first out basis. This mode is useful if several messages, possibly with the same CAN identifier, must be transmitted in sequence; this may be a requirement if CAN is being used for data acquisition.</p> <p>The TwinCAN Transmit block should be connected to CAN Message Packing/Unpacking blocks. Do not ground the block or leave it unconnected.</p>

# TwinCAN Transmit

## Dialog Box



### **TwinCAN Node**

Select node A or node B.

### **Transmit mode**

Select one of the modes described above: direct transmission with dedicated buffer, or FIFO queue with dedicated buffer.

### **Buffer selection**

Choose either automatic or manual selection of the hardware buffer number.

### **Buffer number [0..31]**

This option is available only if the buffer selection is available and set to manual. You must select a buffer number between 0 and 31. Note if more than one message is ready to be transmitted, then the one in the lower buffer number will be sent first. Select buffer numbers such that the higher the message priority, the lower

the buffer number. Note that the hardware buffers are shared between node A and node B of the TwinCAN module.

## **Sample time**

Choose -1 to inherit the sample time from the driving blocks. The TwinCAN Transmit block does not inherit constant sample times and runs at the base rate of the model if driven by invariant signals.

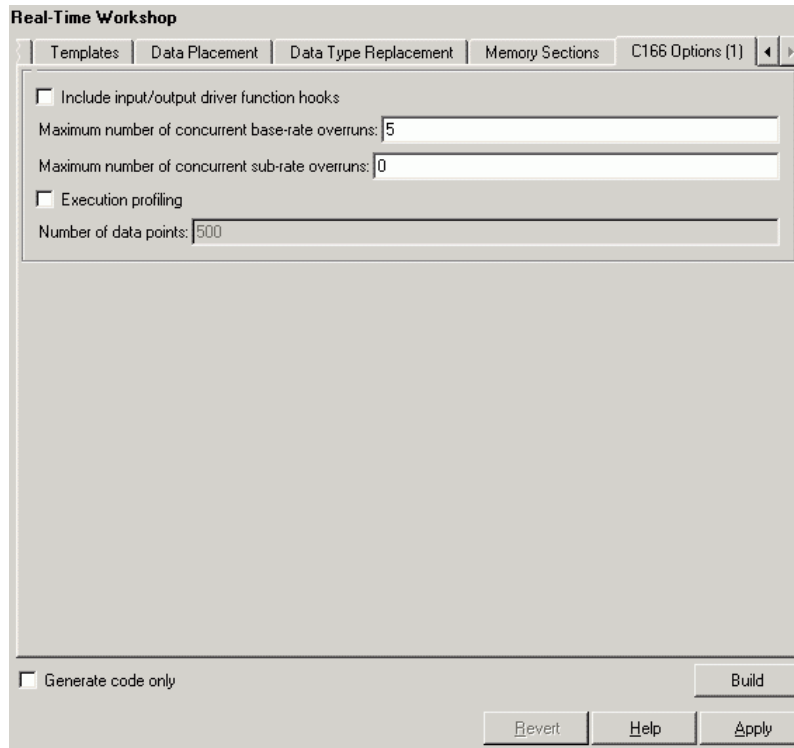
# TwinCAN Transmit

---

# Configuration Parameters

---

## Real-Time Workshop Pane: C166 Options



### In this section...

“C166 Options Tab Overview” on page 8-3

“Include input/output driver function hooks” on page 8-4

“Maximum number of concurrent base-rate overruns:” on page 8-5

“Maximum number of concurrent sub-rate overruns:” on page 8-7

“Execution profiling” on page 8-9

“Number of data points:” on page 8-10



## **C166 Options Tab Overview**

Parameters for integrating your own device driver code and controlling execution profiling with the Target Support Package IC1 product.

### **Configuration**

This pane appears only if you specify the `C166.tlc` or `C166_grt.tlc` system target file.

### **See Also**

- Overview of C166 Configuration Parameters
- Getting Started

### Include input/output driver function hooks

Specify whether to integrate your own device driver code.

#### Settings

**Default:** Off

On

Include input/output driver function hooks. When you generate code for this model, it includes some extra calls to user-supplied input/output device driver functions, to read and write model inputs and outputs. See [Calling the Device Driver Functions from c166\\_main.c](#) for function names and instructions.

Off

Do not include input/output driver function hooks.

#### Command-Line Information

**Parameter:** InputOutputDriverHooks

**Type:** logical

**Value:** 0 | 1

**Default:** 0

#### See Also

[Integrating Your Own Device Drivers](#)

## Maximum number of concurrent base-rate overruns:

Configure allowable base-rate overruns.

### Settings

**Default:** 5

**Minimum:** 0

**Maximum:** No maximum value — it depends on available memory.

### Tips

- Use this option to configure the behavior of the scheduler when timer based tasks do not complete within their allowed sample time.
- It is useful to allow task overruns in the case where a task may occasionally take longer than usual to complete (e.g. if extra processing is required when a special event occurs); if the task overrun is only occasional then it is possible for the scheduler to 'catch up' after the extra processing has been completed.
- If the maximum number of concurrent overruns for any task is exceeded, this is deemed to be a failure and the real-time application is stopped. This in turn will result in a watchdog timer timeout and the processor will be reset.
- The occurrence of base-rate overruns does not affect the numerical behavior of the algorithm (although reading/writing external devices will of course be delayed).

### Command-Line Information

**Parameter:** BaseRateMaxOverrunsValue

**Type:** int

**Value:** 0 | 1 | 2...

**Default:** 5

### **See Also**

- Task Scheduler Overrun Options
- Execution Profiling

## Maximum number of concurrent sub-rate overruns:

Configure allowable sub-rate overruns.

### Settings

**Default:** 0

**Minimum:** 0

**Maximum:** No maximum value — it depends on available memory.

### Tips

---

**Note** Allowing sub-rate overruns may cause non-determinism and loss of integrity for data transferred between different rates in the model. Set this value to zero if you require sub-rate overruns to be handled as a failure (recommended).

---

- If this option is set to a value greater than zero, then the behavior of any Rate-Transition blocks may be affected. Specifically, if the model contains a Rate Transition block where the option "Ensure deterministic data transfer (maximum delay)" is selected, then this setting may not be honored.
- If sub-rate overruns are allowed then the transfer of data between different rates (via rate-transition blocks) in the model may be affected; this causes the numerical behavior in real-time to differ from the behavior in simulation. To see an illustration of this effect try running the demo model `c166_multitasking`. To disallow sub-rate overruns and ensure that this effect does not occur, you should set **Maximum number of concurrent sub-rate overruns** to zero.

### Command-Line Information

**Parameter:** SubRateMaxOverrunsValue

**Type:** int

**Value:** 0 | 1 | 2...

**Default:** 0

### **See Also**

- Task Scheduler Overrun Options
- Execution Profiling

## Execution profiling

Specify whether to configure code for execution profiling.

### Settings

**Default:** Off



On

Include function calls in the generated code for the model at the beginning and end of each task or asynchronous Interrupt Service Routine (ISR) to be profiled. When you perform an execution profiling run, these function calls read a timer and log this reading, along with a task identifier, for uploading and analyzing.



Off

Do not add function calls for execution profiling.

### Tip

When code for the model is generated, these function calls update data on the worst-case turnaround time for each timer-based task as well as the worst-case number of concurrent task overruns, whenever a previous worst case value is exceeded. Additionally, when a trigger is provided, data can be logged over a period of time to record all task start and task finish times. The trigger signal can be supplied by the execution profiling blocks.

### Dependency

This parameter enables **Number of data points**.

### See Also

- Real-Time Workshop Options for Execution Profiling
- Execution Profiling

### Number of data points:

Specify number of data points to log for execution profiling runs.

#### Settings

**Default:** 500

**Minimum:** This depends on the number of tasks. Three is a sensible minimum to get useful information back.

**Maximum:** No maximum value - it depends on available memory.

#### Tip

When a snapshot of task and ISR activity is logged, this data is stored in memory that is statically allocated at build time. Each data point requires 4 bytes on C166 microcontrollers. The larger the number of data points to be stored, the more RAM that must be reserved for this purpose. At the end of a logging run, the data must be uploaded to the host computer for analysis; this is typically achieved by using one of the C166 execution profiling blocks.

#### Dependency

This parameter is enabled by **Execution Profiling**.

#### Command-Line Information

**Parameter:** ExecutionProfilingNumSamples

**Type:** int

**Value:** 3 | 4 | 5...

**Default:** 500

#### See Also

- Number of Data Points
- Execution Profiling



# Examples

---

Use this list to find examples in the documentation.

## **Simple Example Applications**

“Example Model 1: c166\_serial\_transmit” on page 2-4

“Example 2: c166\_serial\_io” on page 2-9

“Debugging and Using The Code Profile Report” on page 2-12

“RAM / ROM Code Profile Report” on page 2-14

“Parameter Tuning and Signal Logging” on page 2-18

## **Real-Time Target**

“Using External Mode” on page 2-18

## **Integrating Hand-Coded Device Drivers**

“Tutorial: Using the Example Driver Functions” on page 3-12

## **Bit-Addressable Memory**

“Using the Bitfield Example Model” on page 4-3

## **Execution Profiling**

“Multitasking Demo Model” on page 5-10

## A

- ASAP2 files
  - generating for C166 2-14
- ASAP2 files, generating 2-27

## B

- bit-addressable memory 4-1
- blocks
  - C-CAN Receive 7-64
  - C-CAN Transmit 7-67
  - C166 Execution Profiling via ASC0 7-2
  - C166 Execution Profiling via C-CAN 1 7-7
  - C166 Execution Profiling via CAN A 7-4
  - C166 Execution Profiling via TwinCAN A 7-8
  - C166 Resource Configuration 7-9
  - CAN Bus Status 7-24
  - CAN Calibration Protocol (C166) 7-25
  - CAN Calibration Protocol (C166,  
C-CAN) 7-31
  - CAN Calibration Protocol (C166,  
TwinCAN) 7-32
  - CAN Pack 7-33
  - CAN Receive 7-44
  - CAN Reset 7-47
  - CAN Transmit 7-48
  - CAN Unpack 7-51
  - Digital In 7-69
  - Digital Out 7-71
  - Fast External Interrupt 7-73
  - Serial Receive 7-75
  - Serial Transmit 7-78
  - Switch External Mode Configuration 7-81
  - Switch Target Configuration 7-83
  - TwinCAN Bus Status 7-86
  - TwinCAN Receive 7-87
  - TwinCAN Reset 7-88
  - TwinCAN Transmit 7-89

## C

- C-CAN Receive block 7-64
- C-CAN Transmit block 7-67
- C166 Execution Profiling via ASC0 block 7-2
- C166 Execution Profiling via C-CAN 1 block 7-7
- C166 Execution Profiling via CAN A block 7-4
- C166 Execution Profiling via TwinCAN A  
block 7-8
- C166 Resource Configuration block 7-9
- CAN Bus Status block 7-24
- CAN Calibration Protocol (C166) block 7-25
- CAN Calibration Protocol (C166, C-CAN)  
block 7-31
- CAN Calibration Protocol (C166, TwinCAN)  
block 7-32
- CAN Pack block 7-33
- CAN Receive block 7-44
- CAN Reset block 7-47
- CAN Transmit block 7-48
- CAN Unpack block 7-51
- Configuration Class blocks 1-28
- configuration parameters
  - pane
    - Execution profiling 8-9
    - Include input/output driver function  
hooks 8-4
    - Maximum number of concurrent  
base-rate overruns: 8-5
    - Maximum number of concurrent sub-rate  
overruns: 8-7
    - Number of data points: 8-10
  - Real-Time Workshop Pane: C166 Options  
Tab 8-3
- custom storage class 4-1

## D

- device driver blocks
  - C-CAN Receive 7-64
  - C-CAN Transmit 7-67

- C166 Digital In 7-69
  - C166 Digital Out 7-71
  - C166 Execution Profiling via ASC0 7-2
  - C166 Execution Profiling via C-CAN 1 7-7
  - C166 Execution Profiling via CAN A 7-4
  - C166 Execution Profiling via TwinCAN A 7-8
  - C166 Resource Configuration 7-9
  - C166 Serial Receive 7-75
  - C166 Serial Transmit 7-78
  - CAN Bus Status 7-24
  - CAN Calibration Protocol (C166) 7-25
  - CAN Calibration Protocol (C166,  
C-CAN) 7-31
  - CAN Calibration Protocol (C166,  
TwinCAN) 7-32
  - CAN Receive 7-44
  - CAN Reset 7-47
  - CAN Transmit 7-48
  - Digital In 7-69
  - Digital Out 7-71
  - Fast External Interrupt 7-73
  - Serial Receive 7-75
  - Serial Transmit 7-78
  - Switch Target Configuration 7-83
  - TwinCAN Bus Status 7-86
  - TwinCAN Receive 7-87
  - TwinCAN Reset 7-88
  - TwinCAN Transmit 7-89
  - Digital In block 7-69
  - Digital Out block 7-71
  - downloading code 2-7
- E**
- example model
    - c166\_bitfields 4-1
    - c166\_fuelsys 2-14
    - c166\_multitasking 5-1
    - c166\_serial\_io 2-9
    - c166\_serial\_transmit 2-4
    - c166\_user\_io 3-1
  - execution profiling 5-1
- F**
- Fast External Interrupt block 7-73
  - fixed-point example 2-14
- G**
- generating code 2-7
- I**
- installation of Target Support Package™ IC1 1-6
  - integrating hand-coded device drivers 3-1
- M**
- multitasking 5-1
- R**
- real-time target
    - C166 tutorial 2-2
- S**
- Serial Receive block 7-75
  - Serial Transmit block 7-78
  - Switch External Mode Configuration block 7-81
  - Switch Target Configuration block 7-83
- T**
- Target Support Package™ IC1
    - feature summary 1-2
  - Target Support Package IC1 product 1-1
  - TwinCAN Bus Status block 7-86
  - TwinCAN Receive block 7-87
  - TwinCAN Reset block 7-88
  - TwinCAN Transmit block 7-89