

Real-Time Windows Target™

Reference

R2012b

MATLAB®

How to Contact MathWorks



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



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



508-647-7000 (Phone)



508-647-7001 (Fax)



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

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

Real-Time Windows Target™ Reference

© COPYRIGHT 1999–2012 by The MathWorks, Inc.

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

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

Trademarks

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

Patents

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

Revision History

March 2008	Online only	New for Version 3.1 (Release 2008a)
October 2008	Online only	Revised for Version 3.2 (Release 2008b)
March 2009	Online only	Revised for Version 3.3 (Release 2009a)
September 2009	Online only	Revised for Version 3.4 (Release 2009b)
March 2010	Online only	Revised for Version 3.5 (Release 2010a)
September 2010	Online only	Revised for Version 3.6 (Release 2010b)
April 2011	Online only	Revised for Version 3.7 (Release 2011a)
September 2011	Online only	Revised for Version 3.8 (Release 2011b)
March 2012	Online only	Revised for Version 4.0 (Release 2012a)
September 2012	Online only	Revised for Version 4.1(Release 2012b)

Function Reference

1

Block Reference

2

Input	2-2
Output	2-3
Synchronization	2-4

Blocks — Alphabetical List

3

Configuration Parameters

4

Code Generation Pane: Real-Time Windows Target ...	4-2
Real-Time Windows Target Tab Overview	4-3
Code replacement library	4-4
Shared code placement	4-6
Generate assembly listings	4-7
Rebuild all	4-8
External mode	4-9
Transport layer	4-11
MEX-file arguments	4-13

Function Reference

Purpose Display information about Real-Time Windows Target status

Syntax rtwho

Description rtwho display information about the Real-Time Windows Target™ PC status. For example, it lists:

- Real-Time Windows Target version
- Computer type
- Timers (ID, period, and status)
- Hardware drivers (ID, device type, and address)
- Kernel performance

Examples This command displays PC information.

```
>> rtwho
```

```
Real-Time Windows Target version 4.0.0 (C) The MathWorks, Inc. 1994-2011  
Running on 64-bit computer.
```

```
TIMERS: Period      Running  
         0.001      Yes  
         0.001      Yes
```

```
DRIVERS: Address    Name  
         0          Humusoft MF624
```


Purpose	Attach and activate default Real-Time Windows Target configuration set
Syntax	<pre>rtwinconfigset('model') rtwinconfigset('model', 'ERT') configset=rtwinconfigset() configset=rtwinconfigset('','ERT')</pre>
Input Arguments	<p><i>model</i> The model to which to attach the default Real-Time Windows Target configuration set.</p> <p>'ERT' The default Real-Time Windows Target configuration set for Embedded Coder™ software to be attached to <i>model</i>.</p>
Output Arguments	<p><i>configset</i> The default Real-Time Windows Target configuration set, which later can be attached to a model.</p>
Description	<p>The <code>rtwinconfigset</code> function, when called with argument <i>model</i>, attaches the default Real-Time Windows Target configuration set RTWin to the model, then activates the configuration set. This default configuration set specifies various simulation and code-generation parameter values that are useful when working with a Real-Time Windows Target model. In most cases, using <code>rtwinconfigset</code> provides all the configuration parameter values that the model needs.</p> <p>When called with no argument, <code>rtwinconfigset</code> returns the default Real-Time Windows Target configuration set object. Later, you can attach this configuration set to a model. See “Configure a Model”.</p> <p>The <code>rtwinconfigset('model', 'ERT')</code> function attaches the default Real-Time Windows Target configuration set for Embedded Coder software to the model, then activates the configuration set. This default configuration set specifies various simulation and code-generation parameter values that are useful when working with a Real-Time Windows Target model in the Embedded Coder environment.</p>

rtwinconfigset

The `rtwinconfigset(' ', 'ERT')` function returns the default Real-Time Windows Target configuration set object for the Embedded Coder software. Later, you can attach this configuration set to a model.

How To

- Configuration Sets
- Referencing Configuration Sets
- Specifying the Default Configuration Set

Purpose Install and remove Real-Time Windows Target kernel

Syntax

```
rtwintgt -setup  
rtwintgt -install  
rtwintgt -uninstall  
rtwintgt -version
```

Description

`rtwintgt -setup` installs the Real-Time Windows Target kernel on your system. It performs the same operation as `rtwintgt -install`.

`rtwintgt -install` installs the Real-Time Windows Target kernel on your system. It performs the same operation as `rtwintgt -setup`. On Windows Vista™ and Windows® 7 systems, `rtwintgt -install` requires User Account Control (UAC) to be enabled.

`rtwintgt -uninstall` removes the Real-Time Windows Target kernel from your system. On Windows Vista and Windows 7 systems, `rtwintgt -uninstall` requires User Account Control (UAC) to be enabled.

`rtwintgt -version` displays the Real-Time Windows Target currently installed on your system.

Block Reference

Input (p. 2-2)

Input data to a Real-Time Windows
Target application

Output (p. 2-3)

Output data from a Real-Time
Windows Target application

Synchronization (p. 2-4)

Synchronization of Real-Time
Windows Target application with
Simulink[®]

Input

Analog Input	Select and connect analog input channels
Counter Input	Select and connect counter input channels
Digital Input	Select and connect digital input lines or channels
Encoder Input	Select and connect encoder input channels
Other Input	Connect with input sources that other input blocks cannot
Packet Input	Receive unformatted binary data
Stream Input	Receive formatted ASCII data

Output

Analog Output	Select and connect analog output channels
Digital Output	Select and connect digital output lines or channels
Frequency Output	Generate and output a pulse-width-modulated square wave
Other Output	Connect with output sources that other output blocks cannot
Packet Output	Transmit unformatted binary data
Stream Output	Transmit formatted ASCII data

Synchronization

Real-Time Sync

Synchronize Simulink to real time

Blocks — Alphabetical List

Analog Input

Purpose Select and connect analog input channels

Library Real-Time Windows Target

Description The Analog Input block allows you to select and connect specific analog input channels to your Simulink model. After you add an Analog Input block to your model, you can enter the parameters for its I/O driver. The following procedure uses the Humusoft® AD512 I/O board as an example:

Parameters **Sample time**
Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block will also synchronize your model with the real-time clock at this sample rate.

Tip If you are using a fixed-step solver, you must enter the value you entered in the **Fixed step size** box from the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks
Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port
Select this check box to display the current number of missed ticks as a block output.

Yield CPU when waiting
Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Input channels

Enter a channel vector that selects the analog input channels you are using on this board. The vector can be any valid MATLAB® vector form. For example, to select all eight analog input channels on the AD512 board, enter

```
[1,2,3,4,5,6,7,8] or [1:8]
```

If you want to use the first three analog input channels, enter

```
[1,2,3]
```

Input range

From the list, choose the input range for all of the analog input channels you entered in the Input channels box. For example, with the AD512 board, choose -5 to 5 V.

If you want the input range to be different for different analog channels, you need to add an I/O block for each different input range.

Block output signal

From the list, choose from the following options:

- **Volts** — Returns a value equal to the analog voltage.
- **Normalized bipolar** — Returns a full range value of -1 to +1 regardless of the input voltage range.
- **Normalized unipolar** — Returns a full range value of 0 to +1 regardless of the input voltage range. For example, an analog input range of 0 to +5 volts and -5 to +5 volts would both be converted to 0 to +1.
- **Raw** — Returns a value of 0 to $2^n - 1$. For example, a 12-bit A/D converter would return values of 0 to $2^{12} - 1$ (0 to 4095). The advantage of this method is the returned value is always an integer with no round-off errors.

Analog Input

Output data type

Set to specify the type of data that the block will output to the model.

Purpose Select and connect analog output channels

Library Real-Time Windows Target

Description The Analog Output block allows you to select and connect specific analog output channels to your Simulink model. After you add an Analog Output block to your model, you can enter the parameters for its I/O driver. The block outputs The following procedure uses the Humusoft AD512 I/O board as an example.

Parameters **Sample time**

Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block will also synchronize your model with the real-time clock at this sample rate.

Tip If you are using a fixed-step solver, you must enter the value you entered in the **Fixed step size** box from the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks

Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port

Select this check box to display the current number of missed ticks as a block output.

Yield CPU when waiting

Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Analog Output

Output channels

Enter a channel vector that selects the analog output channels you are using on this board. The vector can be any valid MATLAB vector form. For example, to select both analog output channels on the AD512 board, enter

[1,2] or [1:2]

Output range

From the list, choose the input range for all of the analog input channels you entered in the Input channels box. For example, with the AD512 board, choose -5 to 5 V.

If you want the input range to be different for different analog channels, you need to add an I/O block for each different input range.

Block input signal

From the list, choose from the following options:

- Volts — Expects a value equal to the analog output voltage.
- Normalized bipolar — Expects a value between -1 and +1 that is converted to the full range of the output voltage regardless of the output voltage range.
- Normalized unipolar — Expects a value between 0 and +1 that is converted to the full range of the output voltage regardless of the output voltage range. For example, an analog output range of 0 to +5 volts and -5 to +5 volts would both be converted from values between 0 and +1.
- Raw — Expects a value of 0 to $2^n - 1$. For example, a 12-bit A/D converter would expect a value between 0 and $2^{12} - 1$ (0 to 4095). The advantage of this method is the expected value is always an integer with no round-off errors.

Initial value

Enter the initial value for each analog output channel you entered in the **Output channels** box. For example, if you entered [1,2]

in the **Output channels** box, and you want an initial value of 0 volts, enter [0,0].

Final value

Enter a final value for each analog channel you entered in the **Output channels** box. For example, if you entered [1,2] in the **Output channels** box, and you want final values of 0 volts, enter [0,0].

Counter Input

Purpose Select and connect counter input channels

Library Real-Time Windows Target

Description The Counter Input block allows you to select and connect specific counter input channels to your Simulink model. After you have added a Counter Input block to your model, you can enter the parameters for its I/O driver. The following procedure uses the Humusoft MF604 I/O board as an example.

Parameters **Sample time**
Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block will also synchronize your model with the real-time clock at this sample rate.

Tip If you are using a fixed-step solver, you must enter the value you entered in the **Fixed step size** box from the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks
Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port
Select this check box to display the current number of missed ticks as a block output.

Yield CPU when waiting
Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Input channels

Enter a channel vector that selects the counter input channels you are using on this board. The vector can be any valid MATLAB vector form. For example, to select all four counter input channels on the MF604 board, enter

[1,2,3,4] or [1:4]

Reset after read

Determines if the counter should be reset to zero after its value has been read. From the list, choose one of the following options:

- `never` — Do not reset after reading.
- `always` — Always reset after reading.
- `level` — Reset after reading if block input is nonzero. This will add an input to the Counter Input block.
- `rising edge` — Reset after reading if block input changes from zero to nonzero between the last two successive readings. This will add an input to the Counter Input block.
- `falling edge` — Reset after reading if the block input changes from nonzero to zero between last two successive readings. This will add an input to the Counter Input block.
- `either edge` — Reset after reading if the block input changes either from zero to nonzero or from nonzero to zero between the last two successive readings. This will add an input to the Counter Input block.

Clock input source

Determines the clock input source to increment the counter. From the list, select

- `input pin rising edge` — Clock edge low to high transitions
- `input pin falling edge` — Clock edge high to low transitions
- `internal clock` — Internal time base

Counter Input

If you set the **Gate input functionality** parameter to `enable when high, latch & reset on edge` or `enable when low, latch & reset on edge`, you can measure positive or negative pulse lengths in units of the internal time base. You can use this combination with the National Instruments® PCI/PXI-60xx and National Instruments PCI/PCIe/PXI/PXIe-62xx drivers for pulse width measurement.

Not all counter chips support selecting the input edge. In this case, only supported options appear in the pull-down menu.

Gate input functionality

Defines the action of the counter's gate pin. From the list, select

- `none` — Gate is disabled.
- `enable when high` — Counting is disabled when the gate is low and enabled when the gate is high.
- `enable when low` — Counting is disabled when the gate is high and enabled when the gate is low.
- `start on rising edge` — Counting is disabled until low to high transition of the gate occurs.
- `start on falling edge` — Counting is disabled until high to low transition of the gate occurs.
- `reset on rising edge` — Counter is reset when low to high transition of the gate occurs.
- `reset on falling edge` — Counter is reset when high to low transition of the gate occurs.
- `latch on rising edge` — The count of the counter is remembered when low to high transition of the gate occurs.
- `latch on falling edge` — The count of the counter is remembered when high to low transition of the gate occurs.

- **latch & reset on rising edge** — The count of the counter is remembered and then the counter is reset when low to high transition of the gate occurs.
- **latch & reset on falling edge** — The count of the counter is remembered and then the counter is reset when high to low transition of the gate occurs.
- **enable when high, latch & reset on edge** — Enables pulse counting when the gate input goes high, counts the clock while it is high, and remembers (latches) the pulse count. Then resets the counter to prepare it for the next pulse in the gate input.

If you set the **Clock input source** parameter to **internal clock**, you can measure positive pulse lengths in units of the internal time base. You can use this combination with the National Instruments PCI/PXI-60xx and National Instruments PCI/PXI-62xx drivers for pulse width measurement.

- **enable when low, latch & reset on edge** — Enables pulse counting when the gate input goes high, counts the clock while it is low, and remembers (latches) the pulse count. Then resets the counter to prepare it for the next pulse in the gate input.

If you set the **Clock input source** parameter to **internal clock**, you can measure negative pulse lengths in units of the internal time base. You can use this combination with the National Instruments PCI/PXI-60xx and National Instruments PCI/PXI-62xx drivers for pulse width measurement.

Not all counter chips support all gate modes. Only supported gate modes appear in the pull-down menu.

Output data type

From the list, select the type of data that the block will output to the model.

Counter Input

Counter Input Block Example

The Real-Time Windows Target software includes an example that shows the operation of the Counter Input block. To see this example, type `rtcounter` in the MATLAB Command Window, or start MATLAB Help and choose **Real-Time Windows Target > Demos > Frequency Measurement**.

Tip You can run examples in both Simulink Normal Mode (initial setting) and in Simulink External Mode.

Purpose	Select and connect digital input lines or channels
Library	Real-Time Windows Target
Description	The Digital Input block allows you to select and connect specific digital lines or channels to your Simulink model. After you have added a Digital Input block to your model, you can enter the parameters for its I/O driver. The following procedure uses the Humusoft AD512 I/O board as an example.

Parameters

Sample time

Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block will also synchronize your model with the real-time clock at this sample rate.

Tip If you are using a fixed-step solver, you must enter the value you entered in the **Fixed step size** box from the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks

Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port

Select this check box to display the current number of missed ticks as a block output.

Yield CPU when waiting

Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Digital Input

Input channels

Enter a channel vector that selects the digital input channels you are using on this board. The vector can be any valid MATLAB vector form. For example, to select all eight digital input channels on the AD512 board, enter

```
[1,2,3,4,5,6,7,8] or [1:8]
```

If you want to use the first four digital input lines, enter

```
[1,2,3,4]
```

If you have one 8-bit digital channel, enter [1]. If you have two 8-bit digital channels, enter [1 9], and from the Channel mode list, choose Byte.

Channel mode

From the list, choose one of the following options:

- **Bit** — Returns a value of 0 or 1.
- **Byte** — Groups eight digital lines into one digital channel and returns a value of 0 to 255.

Output data type

Set to specify the type of data that the block will output to the model.

Purpose	Select and connect digital output lines or channels
Library	Real-Time Windows Target
Description	The Digital Output block allows you to select and connect specific digital lines or channels to your Simulink model. After you have added a Digital Output block to your model, you can enter the parameters for its I/O driver. The following procedure uses the Humusoft AD512 I/O board as an example.

Parameters

Sample time

Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block will also synchronize your model with the real-time clock at this sample rate.

Tip If you are using a fixed-step solver, you must enter the value you entered in the **Fixed step size** box from the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks

Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port

Select this check box to display the current number of missed ticks as a block output.

Yield CPU when waiting

Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Digital Output

Output channels

Enter a channel vector that selects the digital output channels you are using on this board. The vector can be any valid MATLAB vector form. For example, to select all eight digital output channels on the AD512 board, enter

```
[1,2,3,4,5,6,7,8] or [1:8]
```

If you want to use the first four digital output lines, enter

```
[1,2,3,4]
```

If you have one 8-bit digital channel, enter [1]. If you have two 8-bit digital channels, enter [1 9], and from the **Channel mode** list, choose **Byte**.

Channel mode

From the list, choose from one of the following:

- **Bit** — Expects a value of 0 or 1.
- **Byte** — Expects a value of 0 to 255 that is converted to one digital channel of eight digital lines.

Initial value

Enter the initial values for each digital output line or channel you entered in the **Output channels** box. For example, if you entered [1,2,3,4] in the **Output channels** box, and you want initial values of 0 and 1, enter

```
[0,0,1,1]
```

If you choose **Byte** from the **Channel mode** list, enter a value between 0 and 255 for each digital output channel. For example, for one byte (8 digital lines) with an initial value of 25, enter [25]. For two bytes (16 digital lines) with initial values of 25 and 50, enter [25 50].

First value

Enter a final value for each digital output channel you entered in the **Output channels** box. For example, if you entered [1,2,3,4] in the **Output channels** box, and you want final values of 0, enter

[0,0,0,0]

If you choose **Byte** from the **Channel mode** list, enter a value between 0 and 255 for each digital output channel.

Encoder Input

Purpose Select and connect encoder input channels

Library Real-Time Windows Target

Description The Encoder Input block allows you to select and connect specific encoder input channels to your Simulink model. After you have added an Encoder Input block to your model, you can enter the parameters for its I/O driver. The following procedure uses the Humusoft MF604 I/O board as an example.

Parameters **Sample time**
Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block will also synchronize your model with the real-time clock at this sample rate.

Tip If you are using a fixed-step solver, you must enter the value you entered in the **Fixed step size** box from the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks
Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port
Select this check box to display the current number of missed ticks as a block output.

Yield CPU when waiting
Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Input channels

Enter a channel vector that selects the encoder input channels you are using on this board. The vector can be any valid MATLAB vector form. For example, to select all four encoder input channels on the MF604 board, enter

[1,2,3,4] or [1:4]

Quadrature mode

Encoders typically use two sets of stripes, shifted in phase, to optically detect the amplitude and direction of movement. This parameter specifies which encoder stripe edges should be counted. From the list, select

- `double` — Counts the rising edges from both stripe sets
- `single` — Counts the rising edges from one stripe set
- `quadruple` — Counts rising and falling edges from both stripe sets

Quadruple mode yields four times more pulses per revolution than the single mode. Therefore, quadruple is more accurate and should be used unless other parameters dictate otherwise.

Reset input function

The encoder interface chip has a reset pin in addition to encoder inputs. This pin is usually connected to the index output of the encoder. However, it can be connected to any signal or not be used at all. This parameter specifies the function of this pin. From the list, select

- `gate` — Enables encoder counting
- `reset` — Level reset of the encoder count
- `rising edge index` — Resets the encoder count on the rising edge
- `falling edge index` — Resets the encoder count on the falling edge

Encoder Input

Input filter clock frequency

The encoder interface chip has a built-in low-pass filter that attempts to filter out any high frequencies, which are interpreted as noise. This parameter is the cutoff frequency (Hz) of this filter. The cutoff frequency you specify is rounded to the nearest frequency supported by the chip.

If the encoder is moving slowly and high-frequency noise is present, employ the filter to eliminate the noise. This keeps the noise from being counted as encoder pulses. If the encoder is moving quickly, the filter can filter out all of the high-frequency pulses, including those you want to count. In this case, consider leaving the filter disabled by setting the cutoff frequency to Inf.

Output data type

From the list, select the type of data that the block will output to the model.

Purpose	Generate and output a pulse-width-modulated square wave
Library	Real-Time Windows Target
Description	<p>The Frequency Output block generates a pulse-width-modulated square wave that alternates between low (0) and high (1) with a specified frequency and duty cycle. The frequency is specified in Hertz. The duty cycle is specified as a decimal fraction between 0 and 1 inclusive. This fraction specifies the amount of time that the output signal value is high (1). For example, a duty cycle of .7 specifies that the output is high 70% of the time, and low 30% of the time.</p> <p>After you have added a Frequency Output block to your model, double-click the Frequency Output block to open the Block Parameters: Frequency Output dialog box.</p> <p>If you specify a non-default value for Output signal frequency source or Output signal duty source, the appearance of the dialog box changes as shown below.</p>
Parameters	<p>Sample time Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block will also synchronize your model with the real-time clock at this sample rate.</p> <hr/> <p>Tip If you are using a fixed-step solver, you must enter the value you entered in the Fixed step size box from the Configuration Parameters dialog box, or an integer multiple of that value.</p> <hr/> <p>Maximum missed ticks Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.</p>

Frequency Output

Show “Missed Ticks” port

Select this check box to display the current number of missed ticks as a block output.

Yield CPU when waiting

Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Output channels

Enter a vector that selects the output channels you are using on this board. The vector can be any valid MATLAB vector form.

Output signal frequency source

From the list, select the source that specifies the frequency to output. The frequency is specified in Hertz. The **Output signal frequency source** can be either of the following:

external (Default)

The frequency is specified by an input signal. An input port appears on the block to accept the signal. A **Frequency initial value** and a **Frequency final value** may be required.

internal

The frequency is specified by a tunable parameter named **Frequency**. If you specify **internal**, **Frequency** replaces **Frequency initial value**, and **Frequency final value** is disabled.

Frequency

This tunable parameter appears when **Output signal frequency source** is **internal**. Specify the desired frequency in Hertz.

Frequency initial value

This parameter appears when **Output signal frequency source** is **external**. Optionally specify a frequency in Hertz. The specified frequency takes effect when you connect to the target, and persists until simulation starts, at which time the value of the frequency signal takes effect. You can use this parameter

to specify initial conditions and give them time to stabilize. If no value appears, connecting to the target has no effect on the frequency.

Frequency final value

This parameter appears when **Output signal frequency source** is **external**. Optionally specify a frequency in Hertz. The specified frequency takes effect when simulation completes and persists indefinitely. Disconnecting from the target does not change the frequency. You can use this parameter to put a connected device into a neutral state after simulation. If no value appears, the frequency in effect at the end of simulation persists afterwards.

Output signal duty source

Specify the source that specifies the duty cycle. The specification is a decimal fraction between 0 and 1 inclusive that determines the amount of time that the output signal value is high (1). For example, a duty cycle of .7 specifies that the output is high 70% of the time, and low 30% of the time. Two duty cycle values have special significance:

0

The output signal is continuously low (0) regardless of any frequency specification currently in effect.

1

The output signal is continuously high (1) regardless of any frequency specification currently in effect.

From the list, select one of the following:

internal (Default)

The duty cycle is specified by a tunable parameter named **Duty**.

external

The duty cycle is specified by an input signal. An input port appears on the block to accept the signal. If you specify

Frequency Output

external, **Duty initial value** replaces **Duty**, and **Duty final value** is enabled.

Duty

This tunable parameter appears when **Output signal duty source** is **internal**. Specify the desired duty cycle as a decimal fraction between 0 and 1. Specifying 0 or 1 overrides the currently specified frequency and enforces a continuously low or high output signal for the duration of the specification.

Duty initial value

This parameter appears when **Output signal duty source** is **external**. Optionally specify a duty cycle. The specified duty cycle takes effect when you connect to the target, and persists until simulation starts. If no value appears, connecting to the target has no effect on the duty cycle. You can specify any duty cycle. Often the **Duty initial value** is 0 or 1, enforcing a continuously low or high initial output signal. Specifying 0 or 1 overrides any **Frequency initial value**.

Duty final value

This parameter appears when **Output signal duty source** is **external**. Optionally specify a duty cycle. The specified duty cycle takes effect when simulation completes and persists indefinitely. Disconnecting from the target does not change the duty cycle. If no value appears, the duty cycle in effect at the end of simulation persists indefinitely afterwards. You can specify any duty cycle. Often the **Duty final value** is 0 or 1, enforcing a continuously low or high initial output signal. Specifying 0 or 1 overrides any **Frequency final value**.

Purpose	Connect with input sources that other input blocks cannot
Library	Real-Time Windows Target
Description	<p>The Real-Time Windows Target Other Input block can be used for interfacing input signals that other Real-Time Windows Target input blocks do not accommodate. The Other Input block is rarely used, and for only a few drivers. For details, see the documentation for the driver that you intend to use.</p> <p>If none of the available blocks, including Other Input, provide what you need, you can create your own I/O blocks to work with Real-Time Windows Target applications. See “Custom I/O Driver Blocks” for details.</p>
Parameters	<p>Sample time Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block will also synchronize your model with the real-time clock at this sample rate.</p> <hr/> <p>Tip If you are using a fixed-step solver, you must enter the value you entered in the Fixed step size box from the Configuration Parameters dialog box, or an integer multiple of that value.</p> <hr/> <p>Maximum missed ticks Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.</p> <p>Show “Missed Ticks” port Select this check box to display the current number of missed ticks as a block output.</p>

Other Input

Yield CPU when waiting

Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Input channels

Enter a channel vector that selects the other input channels you are using on this board. The vector can be any valid MATLAB vector form.

Optional parameters

Use this field to enter any optional parameters required by the driver.

Output data type

Set to specify the type of data that the block will output to the model.

Purpose Connect with output sources that other output blocks cannot

Library Real-Time Windows Target

Description The Real-Time Windows Target Other Output block can be used for interfacing input signals that other Real-Time Windows Target output blocks do not accommodate. The Other Output block is rarely used, and for only a few drivers. For details, see the documentation for the driver that you intend to use.

If none of the available blocks, including Other Output, provide what you need, you can create your own I/O blocks to work with Real-Time Windows Target applications. See “Custom I/O Driver Blocks” for details.

Parameters **Sample time**

Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block will also synchronize your model with the real-time clock at this sample rate.

Tip If you are using a fixed-step solver, you must enter the value you entered in the **Fixed step size** box from the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks

Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port

Select this check box to display the current number of missed ticks as a block output.

Other Output

Yield CPU when waiting

Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Output channels

Enter a channel vector that selects the analog output channels you are using on this board. The vector can be any valid MATLAB vector form.

Optional parameters

Use this field to enter any optional parameters required by the driver.

Initial value

Enter the initial value for each analog output channel you entered in the **Output channels** box.

Final value

Enter a final value for each analog channel you entered in the **Output channels** box.

Purpose Receive unformatted binary data

Library Real-Time Windows Target

Description The Packet Input block receives unformatted binary data. After you add a Packet Input block to your model, double-click the block to set its parameters.

The driver must be **Standard Devices > Serial Port**, **Standard Devices > UDP Protocol**, **Standard Devices > File**, or **Vector > CAN Device**. When you install a UDP device, enter port addresses in decimal format in the Standard Devices UDP Protocol dialog box.

Parameters **Sample time**

Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block also synchronizes your model with the real-time clock at this sample rate.

Tip If you are using a fixed-step solver, you must enter the same value as you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks

Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port

Select this check box to display the current number of missed ticks as a block output.

Packet Input

Yield CPU when waiting

Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Packet identifier

Enter the ID of the packet to receive. If your protocol does not have packet IDs, this parameter is disabled.

Extended identifier

Check this check box if **Packet identifier** is an extended identifier.

Input packet size

Enter the number of bytes expected in each input packet. This number must be the same as the number of bytes required by the type specifications in **Block output data types**.

Block output data types

Enter a string, or a cell array of strings, that specifies how the data in each packet obtained from the device is to be typed and grouped for input to the application. The Packet Input block has an output port corresponding to each string in **Block output data types**. Changing the number of strings automatically changes the number of output ports.

Each string has the format $[n^*]datatype$. The data described by the string has the type specified by *datatype* and the width specified by *n*; or 1 if *n* is not specified. For example, 'double' means one double value, and '4*int8' means a vector of four int8 values.

By providing a cell array of such strings, you can convert an input packet into the types required, packaged into vectors as required. For example, specifying {'int16', 'int16', 'double'} creates three ports. The first and second each output an int16 value, and the third outputs a double. Specifying {'2*int16', 'double'} creates two ports: a vector of two int16 values, and a scalar double.

Note An out-of-range value is forced to the maximum value for that datatype.

Input packet field byte order

From the list, select:

- Little Endian

Default. Send multibyte values in little-endian format.

- Big Endian

Send multibyte values in big-endian format.

Show “Data Ready” port

Select this check box to indicate that the block has an output port that signals 1 if the block has new data available, and 0 otherwise.

Show “Data Error” port

Select this check box to indicate that the block has an output port that signals 1 if a data error has occurred, and 0 otherwise.

Show packet timestamp port

Select this check box to show the time stamp for the CAN message packets. If your protocol does not have packet time stamps, the block ignores this value.

Packet Output

Purpose Transmit unformatted binary data

Library Real-Time Windows Target

Description The Packet Output block sends unformatted binary data. After you add a Packet Output block to your model, double-click the block to set its parameters.

The driver must be **Standard Devices > Serial Port**, **Standard Devices > UDP Protocol**, **Standard Devices > File**, or **Vector > CAN Device**. When you install a UDP device, enter port addresses in decimal format in the Standard Devices UDP Protocol dialog box.

Any serial output port can send data with sample rates up to 500 Hz. To enable a faster sample rate of up to 10 kHz for some output serial ports, see the following. The software does not support this option with all types of serial output hardware.

To enable a faster rate click **Install new board**, select **Standard Devices > Serial Port**, and select **Direct port access**.

Parameters **Sample time**

Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block also synchronizes your model with the real-time clock at this sample rate.

Tip If you are using a fixed-step solver, you must enter the same value as you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks

Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this

number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port

Select this check box to display the current number of missed ticks as a block output.

Yield CPU when waiting

Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Packet identifier

Enter the ID of the packet to receive. If your protocol does not have packet IDs, this parameter is disabled.

Extended identifier

Check this check box if **Packet identifier** is an extended identifier.

Output packet size

Enter the number of bytes to be transmitted in the output packet. This number must be the same as the number of bytes required by the type specifications in **Output packet field data types**.

Output packet field data types

Enter a string, or a cell array of strings, that specifies how data provided by the application will be formatted into a packet for output to the device. The Packet Output block has an input port corresponding to each string in **Output packet field data types**. Changing the number of strings automatically changes the number of ports.

Each string has the format $[n^*]datatype$. The data described by the string has the type specified by *datatype* and the width specified by *n*; or 1 if *n* is not specified. For example, 'double' means one double value, and '4*int8' means a vector of four int8 values.

The signal input to each port of the Packet Output block can be a scalar or vector of any Simulink data type. The string for each

Packet Output

port specifies the type to be used when its signal is output to the device. If the format string for a port matches the type of the signal input to that port, the signal value appears verbatim in the output packet.

You can also perform type conversion on output. For example, if an input signal is a four-element `int16` vector, but the corresponding string is `'4*int8'`, each of the four integers is converted to an `int8` before being written to the packet. The resulting data occupies four bytes in the output packet.

Note An out-of-range value is forced to the maximum value for that data type.

Output packet field byte order

From the list, select:

- Little Endian
Default. Receive multibyte values in little-endian format.
- Big Endian
Receive multibyte values in big-endian format.

Show “Data Ready” port

Select this check box to indicate that the block has an output port that signals 1 if the block is ready to accept new data, and 0 otherwise.

Show “Data Error” port

Select this check box to indicate that the block has an output port that signals 1 if a data error has occurred, and 0 otherwise.

Initial value

Optional. Enter a vector that has the same number of elements as the sum of the widths of the input signals across all ports. The

specified data is sent when simulation begins, before any other data that is output during simulation.

Final value

Optional. Enter a vector that has the same number of elements as the sum of the widths of the input signals across all ports. The specified data is sent when simulation ends, after any other data that is output during simulation.

Real-Time Sync

Purpose Synchronize Simulink to real time

Library Real-Time Windows Target

Description The Real-Time Sync block allows you to synchronize your Simulink model with the real-time clock at Real-Time Sync block sample hits. The only parameters are those dealing with sample time and missed ticks. There are no input ports.

Parameters **Sample time**
Enter a value representing how frequently you want to synchronize your model with the real-time clock.

Tip If you are using a fixed-step solver, you must enter the value you entered in the **Fixed step size** box from the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks
Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port
Select this check box to display the current number of missed ticks as a block output.

Yield CPU when waiting
Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Purpose Receive formatted ASCII data

Library Real-Time Windows Target

Description The Stream Input block receives formatted ASCII data. After you have added a Stream Input block to your model, double-click the Stream Input block to open the Block Parameters: Stream Input dialog box.

The driver must be **Standard Devices > Serial Port**, **Standard Devices > File**, or **Standard Devices > UDP Protocol**. Specify parameter values as required, then click **OK** or **Apply**. When you install a UDP device, enter port addresses in decimal format in the Standard Devices UDP Protocol dialog box.

The parameters are:

Parameters

Sample time

Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block will also synchronize your model with the real-time clock at this sample rate.

Tip If you are using a fixed-step solver, you must enter the value you entered in the **Fixed step size** box from the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks

Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port

Select this check box to display the current number of missed ticks as a block output.

Yield CPU when waiting

Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Block output data types

A string or a cell array of strings. The block has as many output ports as the number of strings. Each string specifies a data type by name, and optionally a number of elements. For example, 'double' means one double value, and '4*int8' means four int8 values.

The values made available on output ports are grouped and typed according to the **Block output data types** strings. For example, if **Block output data types** is {'2*int8', '3*double'}, the block outputs an int8 vector of width 2 on the first output port, and a double vector of width 3 on the second output port.

Format string

A specification in the same format used by C library I/O routines like scanf. The format string describes the data to be received. The number of elements in the string must equal the number of data items specified in **Block output data types**.

For example, if **Block output data types** is {'2*int8', '3*double'}, and **Format string** is '%d %d %f %f %f', the block reads an ASCII representation of two integers and three doubles. The block makes the resulting values available to the application in an int8 vector of width 2 on the first output port, and a double vector of width 3 on the second output port.

If the data type specified for a value in **Block output data types** differs from the type of the corresponding element in **Format string**, type conversion occurs automatically. The block reads data as specified by **Format string**, converts the data to match the **Block output data types**, and provides the data to the application on the block output ports.

Message termination

A string, cell array of strings, or a number. If the value is a string, receiving this sequence of characters terminates data input. If the value is a cell array, any of the strings in the cell array terminates data input. If the value is a number, data input terminates after reading the specified number of characters.

Show “Data Ready” port

If enabled, the block has an output port that signals 1 if the block has new data available, and 0 otherwise.

Show “Data Error” port

If enabled, the block has an output port that signals 1 if a data error has occurred, and 0 otherwise.

Stream Output

Purpose Transmit formatted ASCII data

Library Real-Time Windows Target

Description The Stream Output block sends formatted ASCII data. After you have added a Stream Output block to your model, double-click the Stream Output block to open the Block Parameters: Stream Output dialog box.

The driver must be **Standard Devices > Serial Port**, **Standard Devices > File**, or **Standard Devices > UDP Protocol**. Specify parameter values as required, then click **OK** or **Apply**. When you install a UDP device, enter port addresses in decimal format in the Standard Devices UDP Protocol dialog box.

Any serial output port can send data with sample rates up to 500 Hz. To enable a faster sample rate of up to 10 kHz for some output serial ports, see the following. The software does not support this option with all types of serial output hardware.

- 1** Click **Install new board**.
- 2** From the drop-down list, select **Standard Devices > Serial Port**.
The Standard Devices Serial Port dialog box is displayed.
- 3** In the Standard Devices Serial Port dialog box, select the **Direct port access** check box.
- 4** Continue as usual, click **OK** when done.

Parameters **Sample time**
Enter a value representing how frequently you want the block to execute and interact with the I/O hardware. The block will also synchronize your model with the real-time clock at this sample rate.

Tip If you are using a fixed-step solver, you must enter the value you entered in the **Fixed step size** box from the Configuration Parameters dialog box, or an integer multiple of that value.

Maximum missed ticks

Enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by more than this number, the block reports an error and simulation stops. Otherwise, simulation continues even if some ticks are missed.

Show “Missed Ticks” port

Select this check box to display the current number of missed ticks as a block output.

Yield CPU when waiting

Select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

Input port sizes

Enter a port width, or vector of port widths. The number of elements determines the number of input ports. Each port has the width specified by the corresponding element. For example, specifying 3, 1, 2 indicates that the block has three input ports. The first is a vector of width 3, the second is a scalar, and the third is a vector of width 2, for a total of six elements. Only the widths need to be specified; any Simulink data type can be input.

Format string

Enter a specification in the same format used by C library I/O routines like `printf`. The format string describes the data to be sent. The format string must have the same number of specifiers as the sum of the elements of **Input port sizes**.

For example, assume that **Input port sizes** is 3, 1, 2. Then a **Format string** of `'%d %d %d %f %d %d'` would output ASCII representing: three integers from the vector on the first input

Stream Output

port; a double from the scalar on the input second port; and two integers from the vector on the third input port.

If the data type specified in the **Format string** for a value differs from the type of the actual value, type conversion occurs automatically, and converted data that conforms to the **Format string** is output as ASCII to the device.

Show “Data Ready” port

Select this check box to indicate that the block has an output port that signals 1 if the block is ready to accept new data, and 0 otherwise.

Show “Data Error” port

Select this check box to indicate that the block has an output port that signals 1 if a data error has occurred, and 0 otherwise.

Initial string

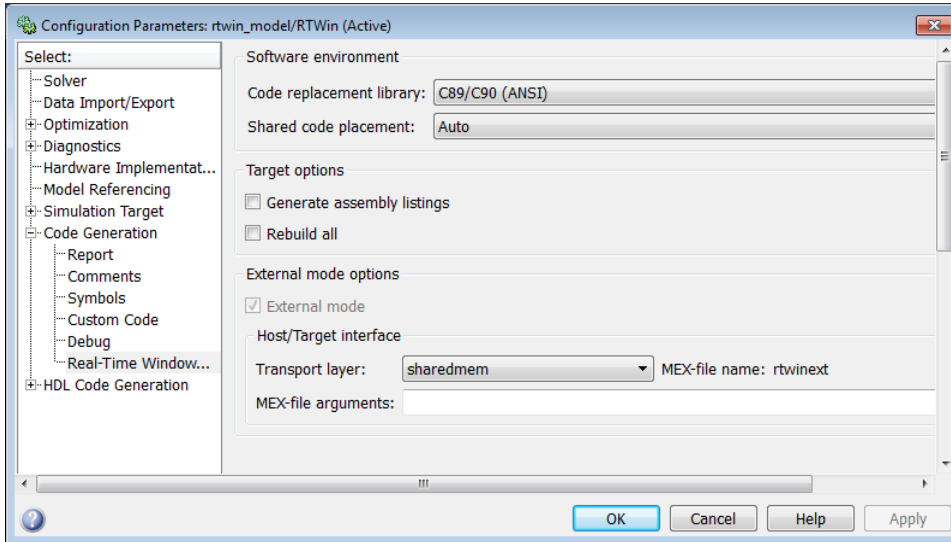
Optional. Enter a string that is sent when simulation begins, before any other data that is output during simulation. The string can contain anything; it could be used to initialize a device. It is sent literally as specified, without translation using the **Format string**.

Final string

Optional. Enter a string that is sent when simulation ends, after any other data that is output during simulation. The string can contain anything; it could be used to shut down a device. It is sent literally as specified, without translation using the **Format string**.

Configuration Parameters

Code Generation Pane: Real-Time Windows Target



In this section...

“Real-Time Windows Target Tab Overview” on page 4-3

“Code replacement library” on page 4-4

“Shared code placement” on page 4-6

“Generate assembly listings” on page 4-7

“Rebuild all” on page 4-8

“External mode” on page 4-9

“Transport layer” on page 4-11

“MEX-file arguments” on page 4-13

Real-Time Windows Target Tab Overview

Control the code created by Simulink Coder™ code generation software for a Real-Time Windows Target application.

Configuration

To enable the xPC Target™ options pane, select `rtwin.tlc` for the **System target file** parameter on the code generation pane.

Code replacement library

Specify a floating-point math library extension.

Settings

Default: C89/C90 (ANSI)

C89/C90 (ANSI)

Generates calls to the ISO[®]/IEC 9899:1990 C standard math library for floating-point functions.

C99 (ISO)

Generates calls to the ISO/IEC 9899:1999 C standard math library.

GNU99 (GNU)

Generates calls to the GNU[®] gcc math library, which provides C99 extensions as defined by compiler option `-std=gnu99`.

C++ (ISO)

Generates calls to the ISO/IEC 14882:2003 C++ standard math library. This setting is visible only if you selected C++ for the **Language** parameter on the Code Generation pane of the Configuration Parameters dialog box.

Tips

- Before setting this parameter, verify that your compiler supports the library you want to use. If you select a parameter value that your compiler does not support, compiler errors can occur.
- Restriction — Stateflow[®] software supports only C89/C90 (ANSI). Selecting a different parameter has no effect on code generated for Stateflow components.

Command-Line Information

Parameter: CodeReplacementLibrary

Type: string

Value: 'ANSI_C' | 'C99 (ISO)' | 'GNU99 (GNU)' | 'C++ (ISO)'

Default: 'ANSI_C'

Recommended Settings

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

Shared code placement

Specify the location for generating shared utility code

Settings

Default: Auto

Auto

Operates as follows:

- When the model contains Model blocks, place utility code within the `slprj/target/_sharedutils` folder.
- When the model does not contain Model blocks, place utility code in the build folder (generally, in `model.c` or `model.cpp`).

Shared location

Directs code for utilities to be placed within the `slprj` folder in your working folder.

Command-Line Information

Parameter: UtilityFuncGeneration

Type: string

Value: 'Auto' | 'Shared location'

Default: 'Auto'

Recommended Settings

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

Generate assembly listings

Instruct the C compiler to generate assembly listings for the generated code.

Settings

Default: off



On

Generates assembly listings.



Off

Suppresses assembly listings.

Command-Line Information

Parameter: CCListing

Type: Boolean

Value: on | off

Default: off

Recommended Settings

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

Rebuild all

Force all the object files to always be rebuilt regardless of their time stamps.

Settings

Default: off



On

Rebuilds all object files at every build.



Off

Rebuilds only object files whose time stamps show they are outdated.

Command-Line Information

Parameter: RebuildAll

Type: Boolean

Value: on | off

Default: off

Recommended Settings

Application	Setting
Debugging	On
Traceability	No impact
Efficiency	No run-time impact. (Build may take longer.)
Safety precaution	On

External mode

Enable client/server communication between Simulink software and an application.

Settings

Default: on



On

Enables external mode.



Off

Disables external mode

Tip

- This check box is selected by default and grayed out.
- Code generation is only supported in external mode.

Dependencies

Selecting **External mode** enables:

- **Transport layer**
- **MEX-file arguments**
- **Static memory allocation**

Command-Line Information

Parameter: ExtMode

Type: string

Value: 'on' | 'off'

Default: 'on'

Recommended Settings

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

See Also

“Create a Transport Layer for External Communication”

Transport layer

Specify the transport protocol for external mode communication.

Settings

Default: sharedmem

sharedmem

Uses a shared memory transport mechanism. The MEX-file name is `rtwinext`.

Tips

- The MEX-file name displayed next to **Transport layer** cannot be edited in the Configuration Parameters dialog box.
- For targets provided by The MathWorks, like Real-Time Windows Target, the MEX-file name is specified in:

```
matlabroot/toolbox/simulink/simulink/extmode_transports.m
```

Dependency

This parameter is enabled by checking **External mode**.

Command-Line Information

Parameter: ExtModeTransport

Type: integer

Value: 0

Default: 0

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact

See Also

- “Create a Transport Layer for External Communication”
- “Target Interfacing”

MEX-file arguments

Specify external mode MEX arguments.

Settings

Default: ''

For a shared memory transport, `rtwintext` allows two optional arguments. The arguments are positional, so the first argument must appear if the second appears.

- An argument that is currently unused. Specify '' for this argument.
- An argument that specifies verbosity. Specify 0 for Nonverbose, or 1 for Verbose.

The default is Nonverbose mode (0). To specify Verbose mode, set **MEX-file arguments** to:

```
'' 1
```

where '' is the empty string and 1 specifies Verbose mode.

Dependency

This parameter is enabled by checking **External mode**.

Command-Line Information

Parameter: ExtModeMexArgs
Type: string followed by integer
Value: '' | '' 0 | '' 1
Default: ''

Recommended Settings

Application	Setting
Debugging	No impact
Traceability	No impact

Application	Setting
Efficiency	No impact
Safety precaution	No impact

See Also

- “Target Interfacing”
- “Choose Communication Protocol for Client and Server”

A

Analog Input block
 configuring 3-2 3-36
Analog Output block
 configuring 3-5

C

configuration parameters
 pane
 External mode 4-9
 Generate assembly listings 4-7
 Rebuild all 4-8
 Real-Time Windows Target tab 4-3
 Simulink Coder (interface)
 mex-file arguments 4-13
 target floating-point math
 environment 4-4
 transport layer 4-11
 utility function 4-6
configuration set
 specifying default 1-3
configuring
 Analog Input block 3-2 3-36
 Analog Output block 3-5
 Counter Input block 3-8
 Digital Input block 3-13
 Digital Output block 3-15
 Encoder Input block 3-18
 Frequency Output block 3-21
 Packet Input block 3-29
 Packet Output block 3-32
 Stream Input block 3-37
 Stream Output block 3-40
Counter Input block
 configuring 3-8

D

Digital Input block

 configuring 3-13

Digital Output block
 configuring 3-15

E

Encoder Input block
 configuring 3-18

F

Frequency Output block
 configuring 3-21

I

I/O blocks
 Analog Input block 3-2 3-36
 Analog Output block 3-5
 Counter Input block 3-8
 Digital Input block 3-13
 Digital Output block 3-15
 Encoder Input block 3-18
 Frequency Output block 3-21
 Packet Input block 3-29
 Packet Output block 3-32
 Stream Input block 3-37
 Stream Output block 3-40

P

Packet Input block
 configuring 3-29
Packet Output block
 configuring 3-32

R

Real-Time Windows Target kernel
 installing 1-5
 removing 1-5

rtwho function 1-2

rtwinconfigset function 1-3

rtwintgt function 1-5

configuring 3-37

Stream Output block

configuring 3-40

S

Stream Input block