

Simulink<sup>®</sup> Desktop Real-Time<sup>™</sup>  
Reference



MATLAB<sup>®</sup>

R2016a



## How to Contact MathWorks



Latest news: [www.mathworks.com](http://www.mathworks.com)  
Sales and services: [www.mathworks.com/sales\\_and\\_services](http://www.mathworks.com/sales_and_services)  
User community: [www.mathworks.com/matlabcentral](http://www.mathworks.com/matlabcentral)  
Technical support: [www.mathworks.com/support/contact\\_us](http://www.mathworks.com/support/contact_us)



Phone: 508-647-7000



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

*Simulink<sup>®</sup> Desktop Real-Time<sup>™</sup> Reference*

© COPYRIGHT 1999–2016 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**

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**

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)
March 2013	Online only	Revised for Version 4.2 (Release 2013a)
September 2013	Online only	Revised for Version 4.3 (Release 2013b)
March 2014	Online only	Revised for Version 4.4 (Release 2014a)
October 2014	Online only	Revised for Version 4.5 (Release 2014b)
March 2015	Online only	Revised for Version 5.0 (Release 2015a)
September 2015	Online only	Revised for Version 5.1 (Release 2015b)
March 2016	Online only	Revised for Version 5.2 (Release 2016a)



## Function Reference

1

## Blocks — Alphabetical List

2

## Configuration Parameters

3

<b>Code Generation Configuration Parameters</b> . . . . .	<b>3-2</b>
Simulink Desktop Real-Time Pane . . . . .	3-3
Code replacement library . . . . .	3-5
Shared code placement . . . . .	3-6
Generate assembly listings . . . . .	3-7
Rebuild all . . . . .	3-8
External mode . . . . .	3-9
Transport layer . . . . .	3-10
MEX-file arguments . . . . .	3-11
<b>Code Generation Parameters on All Parameters Tab</b> . . . . .	<b>3-12</b>
Standard math library . . . . .	3-12



# Function Reference

---

# rtwho

Display information about host computer status

## Syntax

rtwho

## Description

rtwho displays information about the Simulink<sup>®</sup> Desktop Real-Time<sup>™</sup> host computer status. For example:

- Simulink Desktop Real-Time version
- Computer type
- Timers (ID, period, and status)
- Hardware drivers (ID, device type, and address)

## Examples

### Get host computer information

Get information about a host computer with a HUMUSOFT<sup>®</sup> MF624 board installed.

```
rtwho
```

```
Simulink Desktop Real-Time version 5.0.0 (C) The MathWorks, Inc.  
1994-2014 Running on 64-bit computer.
```

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

```
DRIVERS: Address     Name
```



0

Humusoft MF624

## **See Also**

sldrtnkernel

**Introduced before R2006a**

## sldrtconfigset

Attach and activate default Simulink Desktop Real-Time configuration set

### Syntax

```
sldrtconfigset model
sldrtconfigset model 'ERT'

configset = sldrtconfigset
configset = sldrtconfigset('','ERT')
```

### Description

`sldrtconfigset model` attaches the default Simulink Desktop Real-Time configuration set `SimulinkDesktopRealTime` to the model, and then activates the configuration set. This set specifies various simulation and code generation parameter values that are useful when working with a Simulink Desktop Real-Time model.

`sldrtconfigset model 'ERT'` attaches the default Simulink Desktop Real-Time configuration set `SimulinkDesktopRealTime_ERT` for Embedded Coder<sup>®</sup> software to the model, and then activates the configuration set. This set specifies simulation and code generation parameter values that are useful when working with a Simulink Desktop Real-Time model in the Embedded Coder environment.

`configset = sldrtconfigset` returns the default Simulink Desktop Real-Time configuration set object. Later, you can attach this configuration set to a model.

`configset = sldrtconfigset('','ERT')` returns the default Simulink Desktop Real-Time configuration set object for the Embedded Coder software. Later, you can attach this configuration set to a model.

### Examples

#### Set and get default configuration set

Set model `sldrtex_vdp` to the default configuration set. Read the configuration set into a variable.

```

open_system sldrtex_vdp
sldrtconfigset sldrtex_vdp

configset = sldrtconfigset;
configset.Name

ans =

SimulinkDesktopRealTime

```

### Set and get default ERT configuration set

Open model `sldrtex_vdp` and set `gcs` to the default Embedded Coder configuration set. Read the configuration set into a variable.

```

open_system('sldrtex_vdp')
sldrtconfigset(gcs, 'ERT')

configset = sldrtconfigset('', 'ERT');
configset.Name

ans =

SimulinkDesktopRealTime_ERT

```

- “Configure a Model”
- “Manage a Configuration Set”
- “Manage a Configuration Reference”

## Input Arguments

### **model1** — Model to associate with default configuration set

string

The model to which you attach the default Simulink Desktop Real-Time configuration set.

Example: `sldrtex_vdp`

## Output Arguments

**configset** — Default configuration set  
structure

The default Simulink Desktop Real-Time configuration set, which later you can attach to a model.

**Introduced in R2015a**

# sldrtkernel

Install and remove Simulink Desktop Real-Time kernel

## Syntax

```
sldrtkernel -install  
sldrtkernel -uninstall  
sldrtkernel -version  
sldrtkernel -setup
```

## Description

`sldrtkernel -install` installs the Simulink Desktop Real-Time kernel on your system.

On Windows<sup>®</sup> systems, installing or uninstalling the kernel can require you to authorize requests from User Account Control (UAC). On Mac OS X systems, installing or uninstalling the kernel can require you to enter an `osascript` password.

`sldrtkernel -uninstall` removes the Simulink Desktop Real-Time kernel from your system.

`sldrtkernel -version` displays the Simulink Desktop Real-Time version installed on your system.

`sldrtkernel -setup` installs the Simulink Desktop Real-Time kernel on your system.

## Examples

### Update Kernel Version

Checking the version of the kernel, uninstalling the kernel, and installing a new version

Check the kernel version.

```
sldrkernel -version
```

```
The installed version of the Simulink Desktop Real-Time kernel  
is 5.0.0.
```

Uninstall the kernel. At the command prompt, type **y**.

```
sldrkernel -uninstall
```

```
You are going to uninstall the Simulink Desktop Real-Time kernel.  
Do you want to proceed? [y] : y  
The Simulink Desktop Real-Time kernel has been successfully  
uninstalled.
```

Reinstall the kernel. At the command prompt, type **y**.

```
sldrkernel -install
```

```
You are going to install the Simulink Desktop Real-Time kernel.  
Do you want to proceed? [y] : y  
The Simulink Desktop Real-Time kernel has been successfully  
installed.
```

## **See Also**

rtwho

**Introduced in R2015a**

# Blocks — Alphabetical List

---

# Analog Input

Read from one or multiple analog input channels

## Library

Simulink Desktop Real-Time

## Description

Connect to and read from specific analog input channels into your Simulink model. After you add an **Analog Input** block to your model, you can enter the parameters for its I/O driver.

## Parameters

### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is `< no board selected >`. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

### Delete current board

Click this button to delete the current board. The initial selection of the list of registered boards changes to `< no board selected >`, and the **Delete current board** and **Board setup** buttons become inactive.

### Board setup

Click this button to set up the board.



A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

**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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

**Maximum missed ticks**

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

**Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

**Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

**Input channels**

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

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

To select the first three analog input channels, enter:

[1,2,3]

### **Input range**

From the list, choose the input range for the analog input channels that you entered in the **Input channels** box. For example, if the board has an input range of  $\pm 5$  V, choose **-5 to 5 V**.

If you want the input range to be different for different analog channels, 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, analog input ranges of 0 to +5 volts and of -5 to +5 volts are converted to 0 to +1.
- **Raw** — Returns a value of 0 to  $2^n - 1$ . For example, a 12-bit A/D converter returns values of 0 to  $2^{12} - 1$  (0 to 4095). The advantage of this method is that the returned value is an integer with no round-off errors.

### **Output data type**

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

### **Related Examples**

- “Real-Time Controller”
- “Real-Time Filter”

# Analog Output

Write to one or multiple analog output channels

## Library

Simulink Desktop Real-Time

## Description

Connect to and write to specific analog output channels from your Simulink model. After you add an Analog Output block to your model, you can enter the parameters for its I/O driver.

## Parameters

### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is `< no board selected >`. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

### Delete current board

Click this button to delete the current board. The initial selection of the list of registered boards changes to `< no board selected >`, and the **Delete current board** and **Board setup** buttons become inactive.

### Board setup

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

### **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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

### **Maximum missed ticks**

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

### **Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

### **Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

### **Output channels**

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

[1,2] or [1:2]

### **Output range**

From the list, choose the input range for the analog input channels that you entered in the **Input channels** box. For example, if the board has an output range of  $\pm 5$  V, choose **-5 to 5 V**.

If you want the input range to be different for different analog channels, 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, analog output ranges of 0 to +5 volts and of -5 to +5 volts are 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 that the expected value is an integer with no round-off errors.

### Initial value

Enter the initial value for each analog output channel that 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 that 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].

## Related Examples

- “Real-Time Signal Generator”
- “Real-Time Controller”
- “Real-Time Filter”

# Counter Input

Read from one or multiple counter input channels

## Library

Simulink Desktop Real-Time

## Description

Connect to and read from specific counter input channels into your Simulink model. After you have added a **Counter Input** block to your model, you can enter the parameters for its I/O driver.

## Parameters

### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is `< no board selected >`. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

### Delete current board

Click this button to delete the current board. The initial selection of the list of registered boards changes to `< no board selected >`, and the **Delete current board** and **Board setup** buttons become inactive.

### Board setup

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

**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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

**Maximum missed ticks**

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

**Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

**Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

**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 the first four counter input channels on the board, enter:

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

**Reset after read**

Determines if the software should reset the counter to zero after reading its value. 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 the block input is nonzero. Adds an input to the Counter Input block.
- **rising edge** — Reset after reading if the block input changes from zero to nonzero between the last two successive readings. Adds 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. Adds 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. Adds 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

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 National Instruments® drivers for pulse width measurement.

Not all counter chips support selecting the input edge. In this case, only supported options appear in the list.

### **Gate input functionality**

Defines the action of the counter gate input pin. From the list, select:

- **none** — Enable counting unconditionally, ignoring gate input.
- **enable when high** — Disable counting when gate input is low and enable counting when gate input is high.



- **enable when low** — Disable counting when gate input is high and enable counting when gate input is low.
- **start on rising edge** — Disable counting until low-to-high gate transition occurs.
- **start on falling edge** — Disable counting until high-to-low gate transition occurs.
- **reset on rising edge** — Reset counter to zero when low-to-high gate transition occurs.
- **reset on falling edge** — Reset counter to zero when high-to-low gate transition occurs.
- **latch on rising edge** — Store count in register when low-to-high gate transition occurs. Return contents of register.
- **latch on falling edge** — Store count in register when high-to-low gate transition occurs. Return contents of register.
- **latch & reset on rising edge** — Store count in register and then reset counter to zero when low-to-high gate transition occurs. Return contents of register.
- **latch & reset on falling edge** — Store count in register and then reset counter to zero when high-to-low gate transition occurs. Return contents of register.
- **enable when high, latch & reset on edge** — Disable counting while gate input is low. Start counting when low-to-high gate transition occurs. Count while gate input is high. Store count in register and then reset counter to zero when high-to-low gate transition occurs. Return contents of register.

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 most National Instruments drivers for pulse width measurement.

- **enable when low, latch & reset on edge** — Disable counting while gate input is high. Start counting when high-to-low gate transition occurs. Count while gate input is low. Store count in register and then reset counter to zero when low-to-high gate transition occurs. Return contents of register.

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 National Instruments drivers for pulse width measurement.

Not all counter chips support all gate input modes. Only supported modes appear in the list.

### **Output data type**

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

### **Related Examples**

- “Frequency Measurement”
- “PWM Frequency and Duty Measurement”

---

# Digital Input

Read from one or multiple digital input channels

## Library

Simulink Desktop Real-Time

## Description

Connect to and read from specific digital lines or channels into your Simulink model. After you have added a **Digital Input** block to your model, you can enter the parameters for its I/O driver.

## Parameters

### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is `< no board selected >`. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

### Delete current board

Click this button to delete the current board. The initial selection of the list of registered boards changes to `< no board selected >`, and the **Delete current board** and **Board setup** buttons become inactive.

### Board setup

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

### **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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

### **Maximum missed ticks**

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

### **Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

### **Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

### **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 the first eight digital input channels, 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. Returns a value of 0 to 255.

**Output data type**

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

## Digital Output

Write to one or multiple digital output channels

### Library

Simulink Desktop Real-Time

### Description

Connect to and write to specific digital lines or channels from your Simulink model. After you have added a **Digital Output** block to your model, you can enter the parameters for its I/O driver.

### Parameters

#### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is `< no board selected >`. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

#### Delete current board

Click this button to delete the current board. The initial selection of the list of registered boards changes to `< no board selected >`, and the **Delete current board** and **Board setup** buttons become inactive.

#### Board setup

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

### **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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

### **Maximum missed ticks**

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

### **Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

### **Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

### **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 the first eight digital output channels, 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 that 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 (eight digital lines) with an initial value of 25, enter [25]. For two bytes (sixteen digital lines) with initial values of 25 and 50, enter [25 50].

### **First value**

Enter a final value for each digital output channel that 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

Read from one or multiple incremental encoder input channels

## Library

Simulink Desktop Real-Time

## Description

Connect to and read from specific encoder input channels into your Simulink model. After you have added an **Encoder Input** block to your model, you can enter the parameters for its I/O driver.

## Parameters

### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is **< no board selected >**. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

### Delete current board

Click this button to delete the current board. The initial selection of the list of registered boards changes to **< no board selected >**, and the **Delete current board** and **Board setup** buttons become inactive.

### Board setup

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

### **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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

### **Maximum missed ticks**

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

### **Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

### **Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

### **Input channels**

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

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

### **Quadrature mode**

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

- **single** — Counts the rising edges from one stripe set.
- **double** — Counts the rising edges from both stripe sets.
- **quadruple** (default) — 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. Use quadruple mode unless other parameters dictate otherwise.

### **Reset input function**

The encoder interface chip has a reset pin in addition to encoder inputs. In most cases, you connect this pin to the index output of the encoder. However, you can connect it to any signal or not 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.

### **Input filter clock frequency**

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

If the encoder is moving slowly and high-frequency noise is present, use the filter to eliminate the noise. With the filter in operation, the chip does not count the noise as encoder pulses. If the encoder is moving quickly, the filter can filter out the high-frequency pulses, including those that you want to count. In this case, consider disabling the filter by setting the cutoff frequency to **Inf**.

### **Output data type**

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

# Frequency Output

Write to one or multiple frequency or PWM output channels

## Library

Simulink Desktop Real-Time

## Description

Generate and output a pulse-width-modulated square wave to an analog output.

The square wave alternates between low (0) and high (1) with a specified frequency and duty cycle.

You specify the frequency in hertz (Hz).

You specify the duty cycle as a decimal fraction from 0 through 1 inclusive. It determines the amount of time that the output signal value is high (1).

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.

## Parameters

### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is **< no board selected >**. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

### Delete current board

Click this button to delete the current board. The initial selection of the list of registered boards changes to < no board selected >, and the **Delete current board** and **Board setup** buttons become inactive.

### **Board setup**

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

### **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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

### **Maximum missed ticks**

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

### **Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

### **Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

### **Output channels**

Enter a vector that selects the output channels that 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, in hertz. The **Output signal frequency source** can be either of the following:

- **internal** — A tunable parameter named **Frequency** specifies the frequency.

If you specify **internal**, **Frequency** replaces **Frequency initial value**, and **Frequency final value** is disabled.

- **external** (default) — An input signal specifies the frequency. An input port appears on the block to accept the signal.

The software can require additional settings for **Frequency initial value** and **Frequency final value**.

### **Frequency**

Specify the desired frequency in hertz.

This tunable parameter appears when **Output signal frequency source** is **internal**.

### **Frequency initial value**

Optionally specify an initial frequency in hertz. The specified frequency takes effect when you connect to the target. It 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.

This parameter appears when **Output signal frequency source** is **external**.

### **Frequency final value**

Optionally specify a final frequency in hertz. The specified frequency takes effect when simulation is complete 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.

This parameter appears when **Output signal frequency source** is **external**.

### **Output signal duty source**

From the list, specify the source that determines the duty cycle. The **Output signal duty source** can be either of the following:

- **internal** (default) — A tunable parameter named **Duty** specifies the duty cycle.
- **external** — An inport signal specifies the duty cycle. An input port appears on the block to accept the signal.

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

### Duty

Specify the duty cycle as a decimal fraction from 0 through 1 inclusive. It determines the amount of time that the output signal value is high (1). For example, a duty cycle of 0.7 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 the frequency specification currently in effect.
- 1 — The output signal is continuously high (1) regardless of the frequency specification currently in effect.

This tunable parameter appears when **Output signal duty source** is **internal**.

### Duty initial value

Optionally specify an initial duty cycle. The specified duty cycle takes effect when you connect to the target. It 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** setting.

This parameter appears when **Output signal duty source** is **external**.

### Duty final value

Optionally specify a final duty cycle. The specified duty cycle takes effect when simulation is complete 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** setting.

This parameter appears when **Output signal duty source** is external.



# Other Input

Read from one or more input channels not handled by specific blocks

## Library

Simulink Desktop Real-Time

## Description

Connect to and read from hardware sources requiring specialized driver support that other Simulink Desktop Real-Time input blocks do not provide. 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.

If none of the available blocks, including **Other Input**, provide what you need, you can create your own I/O blocks to work with Simulink Desktop Real-Time applications. See “Custom I/O Driver Basics” for details.

## Parameters

### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is **< no board selected >**. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

### Delete current board

Click this button to delete the current board. The initial selection of the list of registered boards changes to **< no board selected >**, and the **Delete current board** and **Board setup** buttons become inactive.

### Board setup

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

### 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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

### Maximum missed ticks

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

### Show “Missed Ticks” port

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

### Yield CPU when waiting

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

### 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 optional parameters required by the driver.

**Output data type**

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

# Other Output

Write to one or more output channels not handled by specific blocks

## Library

Simulink Desktop Real-Time

## Description

Connect to and write to hardware sinks requiring specialized driver support that other Simulink Desktop Real-Time output blocks do not provide. 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 Simulink Desktop Real-Time applications. See “Custom I/O Driver Basics” for details.

## Parameters

### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is **< no board selected >**. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

### Delete current board

Click this button to delete the current board. The initial selection of the list of registered boards changes to **< no board selected >**, and the **Delete current board** and **Board setup** buttons become inactive.

**Board setup**

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

**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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

**Maximum missed ticks**

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

**Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

**Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

**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 optional parameters that the driver requires.

**Initial value**

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

**Final value**

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

# Packet Input

Read binary data or a CAN message from a communication channel

## Library

Simulink Desktop Real-Time

## Description

Read unformatted binary data from a port or file. 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

### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is **< no board selected >**. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

### Delete current board

Click this button to delete the current board. The initial selection of the list of registered boards changes to **< no board selected >**, and the **Delete current board** and **Board setup** buttons become inactive.

### **Board setup**

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

### **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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

### **Maximum missed ticks**

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

### **Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

### **Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

### **Packet identifier**

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

### **Extended identifier**



Select 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 changes the number of output ports.

Each string has the format  $[n^*]datatype$ . The data that the string describes 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. You can also type 'CAN\_MESSAGE' to specify a single **CAN\_MESSAGE** value.

By providing a cell array of such strings, you can convert an input packet into the types required, packaged into vectors. 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:** The software forces an out-of-range value to the maximum value for that data type.

---

### 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.

### **Related Examples**

- “Packet Input/Output”
- “CAN Input/Output”
- “CAN Input/Output with Vehicle Network Toolbox”

# Packet Output

Write binary data or a CAN message to a communication channel

## Library

Simulink Desktop Real-Time

## Description

Connect to and write unformatted binary data to a port, file, or CAN device. 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, click **Install new board**, select **Standard Devices > Serial Port**, and select **Direct port access**.

The software does not support this option with all types of serial output hardware.

## Parameters

### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is **< no board selected >**. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

### **Delete current board**

Click this button to delete the current board. The initial selection of the list of registered boards changes to < **no board selected** >, and the **Delete current board** and **Board setup** buttons become inactive.

### **Board setup**

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

### **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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

### **Maximum missed ticks**

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

### **Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

### **Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

**Packet identifier**

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

**Extended identifier**

Select 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 changes the number of ports.

Each string has the format  $[n^*]datatype$ . The data that the string describes 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. You can also type 'CAN\_MESSAGE' to specify a single CAN\_MESSAGE value.

The signal input to each port of the **Packet Output** block can be a scalar or vector of the data type. The string for each 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:** The software forces an out-of-range value 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. Before other data that is output during simulation is sent, the specified data is sent when simulation begins.

### **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. After other data that is output during simulation, the specified data is sent when simulation ends.

## **Related Examples**

- “Packet Input/Output”
- “CAN Input/Output”
- “CAN Input/Output with Vehicle Network Toolbox”

# Real-Time Sync

Synchronize model execution to real time in normal or accelerator mode

## Library

Simulink Desktop Real-Time

## Description

Synchronize your Simulink model with the real-time kernel clock at **Real-Time Sync** block sample hits. The parameters deal only with sample time and missed ticks. There are no input ports.

The **Real-Time Sync** block operates only in the Simulink normal and accelerator modes. It has no function in Simulink external mode. If the normal or accelerator mode model contains another Simulink Desktop Real-Time block, the other block also synchronizes the Simulink model with the real-time clock. Therefore, do not include the **Real-Time Sync** block in such a model.

## 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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

### Maximum missed ticks

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the

software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

### **Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

### **Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

### **Related Examples**

- “Real-Time Van der Pol Simulation”



# Stream Input

Read formatted ASCII data from a communication channel

## Library

Simulink Desktop Real-Time

## Description

Read formatted ASCII data from a port or file. 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.

## Parameters

### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is **< no board selected >**. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

### Delete current board

Click this button to delete the current board. The initial selection of the list of registered boards changes to **< no board selected >**, and the **Delete current board** and **Board setup** buttons become inactive.

### Board setup

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

### 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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

### Maximum missed ticks

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

### Show “Missed Ticks” port

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

### Yield CPU when waiting

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

### 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. 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.

## Related Examples

- “Stream Input/Output”

## Stream Output

Write formatted ASCII data to a communication channel

### Library

Simulink Desktop Real-Time

### Description

Write formatted ASCII data to a port or file. 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**. 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, click **Install new board**, select **Standard Devices > Serial Port**, and select **Direct port access**.

The software does not support this option with all types of serial output hardware.

### Parameters

#### Install new board

Click this button to register a board with Simulink Desktop Real-Time.

When you click **Install new board**, the software displays a list of manufacturers of supported boards. When you select a manufacturer, the software displays a list of boards available from that manufacturer. When you select a board, the software adds the board to the list of registered boards and makes that board the current board.

By default, the initial selection in the list of registered boards is **< no board selected >**. When you select a board in the list, the **Delete current board** and **Board setup** buttons become active.

**Delete current board**

Click this button to delete the current board. The initial selection of the list of registered boards changes to < **no board selected** >, and the **Delete current board** and **Board setup** buttons become inactive.

**Board setup**

Click this button to set up the board.

A board-specific dialog box opens for you to use to set up the board. For more information to set up the board, see the board manufacturer documentation.

**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 value that you entered in the **Fixed step size** box in the Configuration Parameters dialog box, or an integer multiple of that value.

---

**Maximum missed ticks**

In normal or accelerator mode, enter the number of timer ticks that your model can lag behind the real-time kernel. When the model lags by this number or less, the software assumes that the lag is temporary and allows the model to catch up, even if the model misses some ticks. When the model lags by more than this number, the software reports an error and simulation stops.

In external mode, the software ignores this value.

**Show “Missed Ticks” port**

In normal or accelerator mode, select this check box to display the number of missed ticks as a block output.

In external mode, The "Missed Ticks" port displays zero.

**Yield CPU when waiting**

In normal or accelerator mode, select this check box to grant other programs more CPU time while the kernel waits for a response from the hardware.

In external mode, the software ignores this value.

### 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. You specify only the widths.

### 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' outputs ASCII representing: three integers from the vector on the first input 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, 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. Before any other data that is output during simulation, **Initial string** is sent when simulation begins.

The string can contain anything. You can use it to initialize a device. It is sent literally as specified, without translation using **Format string**.

### Final string

Optional. Enter a string. After any other data that is output during simulation, **Final string** is sent when simulation ends.

The string can contain anything. You can use it to shut down a device. It is sent literally as specified, without translation using the **Format string**

## **Related Examples**

- “Stream Input/Output”





# Configuration Parameters

---

- “Code Generation Configuration Parameters” on page 3-2
- “Code Generation Parameters on All Parameters Tab” on page 3-12

## Code Generation Configuration Parameters

In this section...
“Simulink Desktop Real-Time Pane” on page 3-3
“Code replacement library” on page 3-5
“Shared code placement” on page 3-6
“Generate assembly listings” on page 3-7
“Rebuild all” on page 3-8
“External mode” on page 3-9
“Transport layer” on page 3-10
“MEX-file arguments” on page 3-11

## Simulink Desktop Real-Time Pane

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

### Configuration

The **Simulink Desktop Real-Time** node in the Configuration Parameters dialog box allows you to specify how the software generates the real-time application. To reveal the **Simulink Desktop Real-Time** node, in the **Code Generation** pane, in the **System target file** list, select one of these settings:

- `sldrt.tlc`

Generate system target code for Simulink Desktop Real-Time.

- `sldrtert.tlc`

Generate system target code for a Simulink Desktop Real-Time using the required Embedded Coder software.

---

**Note:** If you open a model that was originally saved with **System target file** set to `rtwin.tlc`, the software updates the setting to `sldrt.tlc`, and likewise with `rtwinert.tlc` and `sldrtert.tlc`. To retain the updated setting, save the updated model.

---



## Code replacement library

Specify a target-specific floating-point math library extension

### Settings

**Default:** None

None

Does not use a code replacement library.

GNU C99 extensions

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

### 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.

### Command-Line Information

**Parameter:** CodeReplacementLibrary

**Type:** string

**Value:** 'None' | 'GNU99 (GNU)'

**Default:** 'None'

### 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'

## 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

### Rebuild all

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

#### Settings

**Default:** off

On

Rebuilds the object files at every build.

Off

Rebuilds object files whose time stamps show they are outdated.

#### Command-Line Information

**Parameter:** RebuildAll

**Type:** Boolean

**Value:** on | off

**Default:** off



## 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'

### 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 `sldrtext`.

#### Tips

- The MEX-file name displayed next to **Transport layer** cannot be edited in the Configuration Parameters dialog box.
- For targets provided by MathWorks®, like Simulink Desktop Real-Time, 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

#### 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, `sldrttext` allows two optional positional arguments. If the second argument appears, the first argument must appear.

- An argument that is 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:** ''

### See Also

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

# Code Generation Parameters on All Parameters Tab

## Standard math library

Specify a standard floating-point math library

### Settings

**Default:** C89/C90 (ANSI)

**C89/C90 (ANSI)**

Generates calls to the ISO<sup>®</sup>/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.

### 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.

### Command-Line Information

**Parameter:** TargetLangStandard

**Type:** string

**Value:** 'C89/C90 (ANSI)' | 'C99 (ISO)'

**Default:** 'C89/C90 (ANSI)'