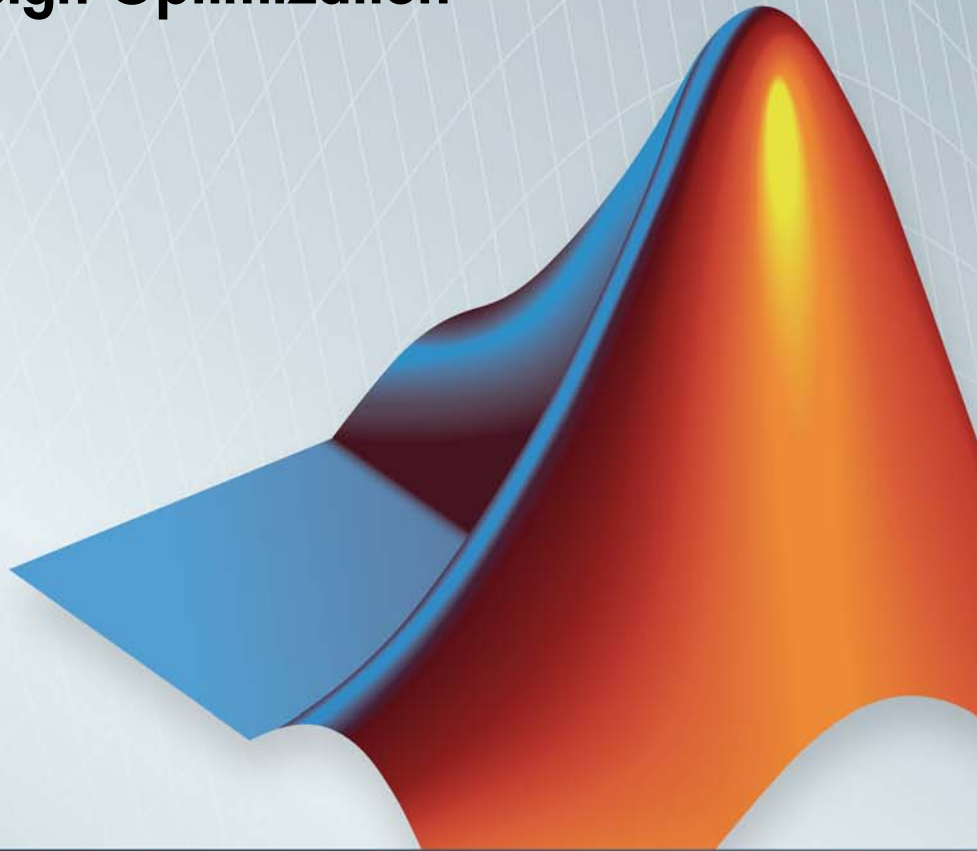


# Simulink® Design Optimization™ Reference

**R2013a**



**MATLAB® & SIMULINK®**



## How to Contact MathWorks



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



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



508-647-7000 (Phone)



508-647-7001 (Fax)



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

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

*Simulink® Design Optimization™ Reference*

© COPYRIGHT 1998–2013 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

September 2011	Online only	New for version 2 (Release R2011b)
March 2012	Online only	New for version 2.1 (Release R2012a)
September 2012	Online only	New for version 2.2 (Release R2012b)
March 2013	Online only	New for version 2.3 (Release R2013a)

## Blocks — Alphabetical List

**1**

## Class Reference

**2**

## Alphabetical List

**3**

## Index



# Blocks — Alphabetical List

---

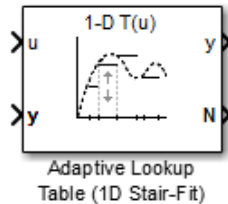
# Adaptive Lookup Table (1D Stair-Fit)

---

**Purpose** One-dimensional adaptive table lookup

**Library** Simulink Design Optimization

## Description



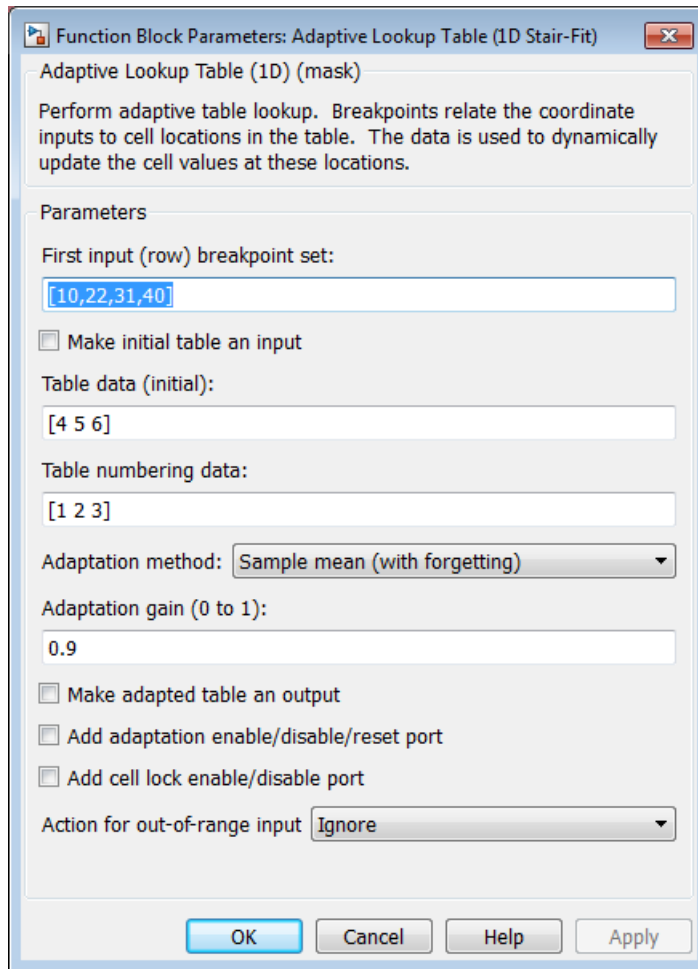
The Adaptive Lookup Table (1D Stair-Fit) block creates a one-dimensional adaptive lookup table by dynamically updating the underlying lookup table. The block uses the outputs,  $y$ , of your system to do the adaptations.

Each indexing parameter  $u$  may take a value within a set of adapting data points, which are called *breakpoints*. Two breakpoints in each dimension define a *cell*. The set of all breakpoints in one of the dimensions defines a *grid*. In the one-dimensional case, each cell has two breakpoints, and the cell is a line segment.

You can use the Adaptive Lookup Table (1D Stair Fit) block to model time-varying systems with one input.

**Data Type Support** Doubles only

# Adaptive Lookup Table (1D Stair-Fit)



## Dialog Box

### First input (row) breakpoint set

The vector of values containing possible block input values. The input vector must be monotonically increasing.

# Adaptive Lookup Table (1D Stair-Fit)

---

## **Make initial table an input**

Selecting this check box forces the Adaptive Lookup Table (1D Stair-Fit) block to ignore the **Table data (initial)** parameter, and creates a new input port  $T_{in}$ . Use this port to input the table data.

## **Table data (initial)**

The initial table output values. This vector must be of size  $N-1$ , where  $N$  is the number of breakpoints.

## **Table numbering data**

Number values assigned to cells. This vector must be the same size as the table data vector, and each value must be unique.

## **Adaptation method**

Choose `Sample mean` or `Sample mean (with forgetting)`. `Sample mean` averages all the values received within a cell. `Sample mean with forgetting` gives more weight to the new data. How much weight is determined by the **Adaptation gain** parameter. For more information, see “Selecting an Adaptation Method”.

## **Adaptation gain (0 to 1)**

A number between 0 and 1 that regulates the weight given to new data during the adaptation. A 0 means short memory (last data becomes the table value), and 1 means long memory (average all data received in a cell).

## **Make adapted table an output**

Selecting this check box creates an additional output port  $T_{out}$  for the adapted table.

## **Add adaptation enable/disable/reset port**

Selecting this check box creates an additional input port `Enable` that enables, disables, or resets the adaptive lookup table. A signal value of 0 applied to the port disables the adaptation, and signal value of 1 enables the adaptation. Setting the signal value to 2 resets the table values to the initial table data.



# Adaptive Lookup Table (1D Stair-Fit)

---

## **Add cell lock enable/disable port**

Selecting this check box creates an additional input port Lock that provides the means for updating only specified cells during a simulation run. A signal value of 0 unlocks the specified cells and signal value of 1 locks the specified cells.

## **Action for out-of-range input**

Ignore or Adapt by extrapolating beyond the extreme breakpoints.

## **See Also**

[Adaptive Lookup Table \(2D Stair-Fit\)](#) | [Adaptive Lookup Table \(nD Stair-Fit\)](#) |

## **Concepts**

- “What are Adaptive Lookup Tables?”
- “Selecting an Adaptation Method”

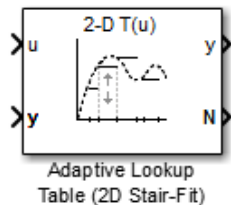
# Adaptive Lookup Table (2D Stair-Fit)

---

**Purpose** Two-dimensional adaptive table lookup

**Library** Simulink Design Optimization

## Description



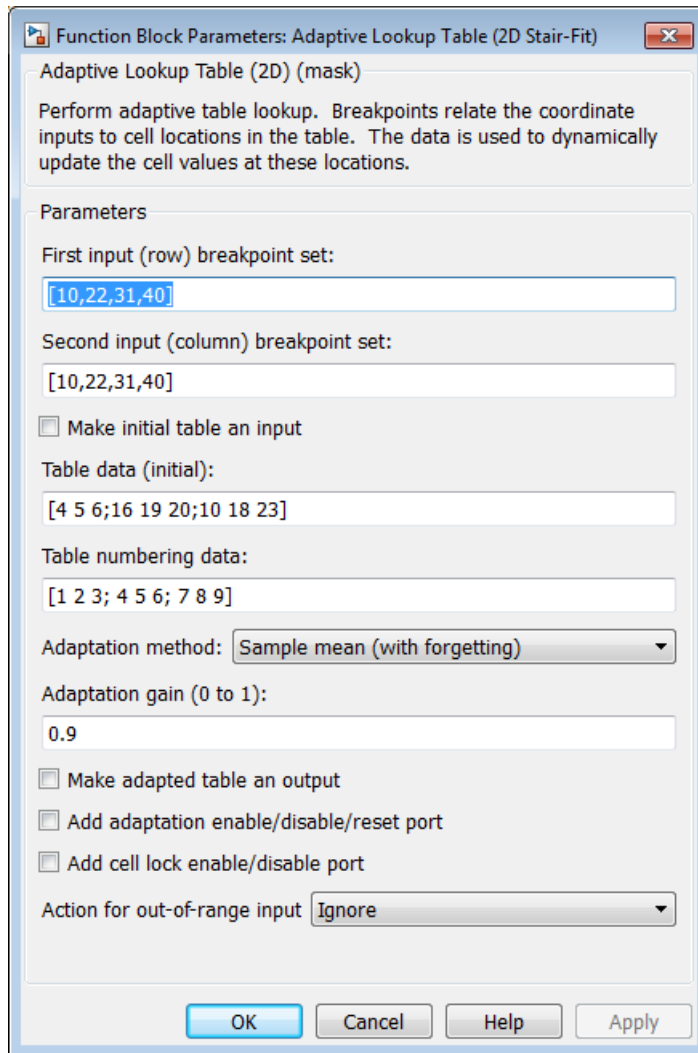
The Adaptive Lookup Table (2D Stair-Fit) block creates a two-dimensional adaptive lookup table by dynamically updating the underlying lookup table. The block uses the outputs,  $y$ , of your system to do the adaptations.

Each indexing parameter  $u$  may take a value within a set of adapting data points, which are called *breakpoints*. Two breakpoints in each dimension define a *cell*. The set of all breakpoints in one of the dimensions defines a *grid*. In the two-dimensional case, each cell has four breakpoints and is a flat surface.

You can use the Adaptive Lookup Table (2D Stair-Fit) block to model time-varying systems with two inputs.

**Data Type Support** Doubles only

# Adaptive Lookup Table (2D Stair-Fit)



## Dialog Box

### First input (row) breakpoint set

The vector of values containing possible block input values for the first input variable. The first input vector must be monotonically increasing.

# Adaptive Lookup Table (2D Stair-Fit)

---

## **Second input (column) breakpoint set**

The vector of values containing possible block input values for the second input variable. The second input vector must be monotonically increasing.

## **Make initial table an input**

Selecting this check box forces the Adaptive Lookup Table (2D Stair-Fit) block to ignore the **Table data (initial)** parameter, and creates a new input port  $T_{in}$ . Use this port to input the table data.

## **Table data (initial)**

The initial table output values. This 2-by-2 matrix must be of size  $(n-1)$ -by- $(m-1)$ , where  $n$  is the number of first input breakpoints and  $m$  is the number of second input breakpoints.

## **Table numbering data**

Number values assigned to cells. This matrix must be the same size as the table data matrix, and each value must be unique.

## **Adaptation method**

Choose **Sample mean** or **Sample mean with forgetting**. **Sample mean** averages all the values received within a cell. **Sample mean with forgetting** gives more weight to the new data. How much weight is determined by the **Adaptation gain** parameter. For more information, see “Selecting an Adaptation Method”.

## **Adaptation gain (0 to 1)**

A number from 0 to 1 that regulates the weight given to new data during the adaptation. A 0 means short memory (last data becomes the table value), and 1 means long memory (average all data received in a cell).

## **Make adapted table an output**

Selecting this check box creates an additional output port  $T_{out}$  for the adapted table.

## **Add adaptation enable/disable/reset port**

Selecting this check box creates an additional input port **Enable** that enables, disables, or resets the adaptive lookup table. A signal value of 0 applied to the port disables the adaptation, and

# Adaptive Lookup Table (2D Stair-Fit)

---

signal value of 1 enables the adaptation. Setting the signal value to 2 resets the table values to the initial table data.

## **Add cell lock enable/disable port**

Selecting this check box creates an additional input port **Lock** that provides the means for updating only specified cells during a simulation run. A signal value of 0 unlocks the specified cells and signal value of 1 locks the specified cells.

## **Action for out-of-range input**

Ignore or Adapt by extrapolating beyond the extreme breakpoints.

## **See Also**

Adaptive Lookup Table (1D Stair-Fit) | Adaptive Lookup Table (nD Stair-Fit) |

## **Concepts**

- “What are Adaptive Lookup Tables?”
- “Selecting an Adaptation Method”

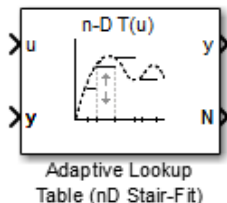
# Adaptive Lookup Table (nD Stair-Fit)

---

**Purpose** Adaptive lookup table of arbitrary dimension

**Library** Simulink Design Optimization

## Description



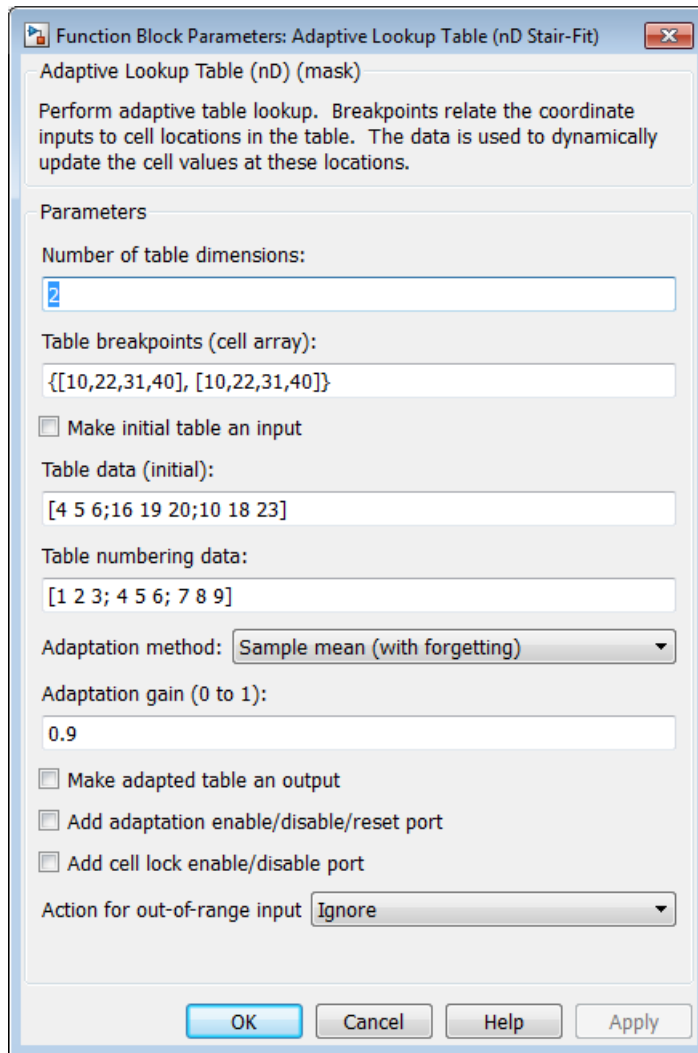
The Adaptive Lookup Table (nD Stair-Fit) block creates an adaptive lookup table of arbitrary dimension by dynamically updating the underlying lookup table. The block uses the outputs of your system to do the adaptations.

Each indexing parameter may take a value within a set of adapting data points, which are called *breakpoints*. Breakpoints in each dimension define a *cell*. The set of all breakpoints in one of the dimensions defines a *grid*. In the n-dimensional case, each cell has two n breakpoints and is an (n-1) hypersurface.

You can use the Adaptive Lookup Table (nD Stair-Fit) block to model time-varying systems with 2 or more inputs.

**Data Type Support** Doubles only

# Adaptive Lookup Table (nD Stair-Fit)



## Dialog Box

### Number of table dimensions

The number of dimensions for the adaptive lookup table.

# Adaptive Lookup Table (nD Stair-Fit)

---

## Table breakpoints (cell array)

A set of one-dimensional vectors that contains possible block input values for the input variables. Each input row must be monotonically increasing, but the rows do not have to be the same length. For example, if the **Number of table dimensions** is 3, you can set the table breakpoints as follows:

```
{[1 2 3], [5 7], [1 3 5 7]}
```

## Make initial table an input

Selecting this check box forces the Adaptive Lookup Table (nD Stair-Fit) block to ignore the **Table data (initial)** parameter, and creates a new input port  $T_{in}$ . Use this port to input the table data.

## Table data (initial)

The initial table output values. This (n-D) array must be of size (n-1)-by-(n-1) ... -by- (n-1), (D times), where D is the number of dimensions and n is the number of input breakpoints.

## Table numbering data

Number values assigned to cells. This vector must be the same size as the table data array, and each value must be unique.

## Adaptation method

Choose **Sample mean** or **Sample mean with forgetting**. **Sample mean** averages all the values received within a cell. **Sample mean with forgetting** gives more weight to the new data. How much weight is determined by the **Adaptation gain** parameter. For more information, see “Selecting an Adaptation Method”.

## Adaptation gain (0 to 1)

A number from 0 to 1 that regulates the weight given to new data during the adaptation. A 0 means short memory (last data becomes the table value), and 1 means long memory (average all data received in a cell).

## Make adapted table an output

Selecting this check box creates an additional output port  $T_{out}$  for the adapted table.



# Adaptive Lookup Table (nD Stair-Fit)

---

---

**Note** The Adaptive Lookup Table (n-D Stair Fit) block cannot output a table of 3 or more dimensions.

---

## **Add adaptation enable/disable/reset port**

Selecting this check box creates an additional input port **Enable** that enables, disables, or resets the adaptive lookup table. A signal value of 0 applied to the port disables the adaptation, and signal value of 1 enables the adaptation. Setting the signal value to 2 resets the table values to the initial table data.

## **Add cell lock enable/disable port**

Selecting this check box creates an additional input port **Lock** that provides the means for updating only specified cells during a simulation run. A signal value of 0 unlocks the specified cells and signal value of 1 locks the specified cells.

## **Action for out-of-range input**

Ignore or Adapt by extrapolating beyond the extreme breakpoints.

## **See Also**

[Adaptive Lookup Table \(1D Stair-Fit\)](#) | [Adaptive Lookup Table \(2D Stair-Fit\)](#) |

## **Related Examples**

- “Model Engine Using n-D Adaptive Lookup Table”

## **Concepts**

- “What are Adaptive Lookup Tables?”
- “Selecting an Adaptation Method”

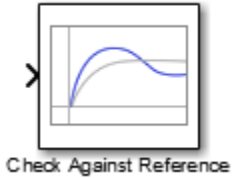
# Check Against Reference

---

**Purpose** Check that model signal tracks reference signal during simulation

**Library** Simulink Design Optimization

**Description** Check that a signal remains within tolerance bounds of a reference signal during simulation:



- If all bounds are satisfied, the block does nothing.
- If a bound is not satisfied, the block asserts, and a warning message appears at the MATLAB® prompt. You can also specify that the block:
  - Evaluate a MATLAB expression.
  - Stop the simulation and bring that block into focus.

During simulation, the block can also output a logical assertion signal:

- If all bounds are satisfied, the signal is true (1).
- If a bound is not satisfied, the signal is false (0).

You can add Check Against Reference blocks on multiple signals to check that they track reference signals.

You can also plot the reference signal on a time plot to graphically verify that the signal tracks that signal.

This block and the other blocks in the Model Verification library test that a signal remains within specified time-domain characteristic bounds. When a model does not violate any bound, you can disable the block by clearing the assertion option. If you modify the model, you can re-enable assertion to ensure that your changes do not cause the model to violate a bound.

If the signal does not satisfy the bounds, you can optimize the model parameters to satisfy the bounds. If you have Simulink® Control Design™ software, you can add frequency-domain bounds such as Bode magnitude and optimize the model response to satisfy both time- and frequency-domain requirements.

The block can be used in all simulation modes for signal monitoring but only in Normal or Accelerator simulation mode for response optimization.

## Parameters

Task	Parameters
Specify a reference signal to: <ul style="list-style-type: none"><li>• Assert that a signal tracks the reference</li><li>• Optimize model response so that a signal tracks the reference</li></ul>	<b>Include reference signal tracking in assertion in Bounds tab.</b>
Specify assertion options (only when you specify reference to track).	In the <b>Assertion</b> tab: <ul style="list-style-type: none"><li>• <b>Enable assertion</b></li><li>• <b>Simulation callback when assertion fails (optional)</b></li><li>• <b>Stop simulation when assertion fails</b></li><li>• <b>Output assertion signal</b></li></ul>
Open Design Optimization tool to optimize model response	Click <b>Response Optimization</b>
Plot reference signal	Click <b>Show Plot</b> .
Display plot window instead of Block Parameters dialog box on double-clicking the block.	<b>Show plot on block open</b>

# Check Against Reference

---

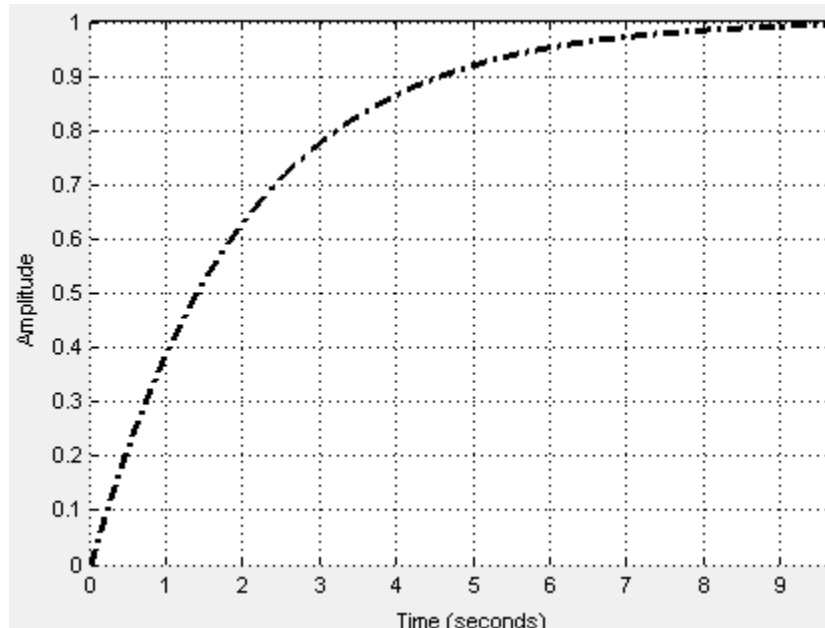
## Include reference signal tracking in assertion

Check that the signal does not track the reference signal specified in “Times (seconds)” on page 1-18 and “Amplitudes” on page 1-19 during simulation.

The software displays a warning if the signal does not track the reference signal.

This parameter is used only if **Enable assertion** in the **Assertion** tab is selected.

The reference signal also appears on a time plot if you click **Show Plot**, as shown in the next figure.



If you clear **Enable assertion**, the bounds are not used for assertion but continue to appear on the plot.

## Settings

Default: On



On

Check that the signal tracks the specified reference signal during simulation.



Off

Do not check that the signal tracks the specified reference signal during simulation.

## Tips

- Clearing this parameter disables the reference signal and the software stops checking that the signal tracks the reference during simulation.
- To only view the bounds on the plot, clear **Enable assertion**.

## Command-Line Information

**Parameter:** EnableReferenceBound

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'on'

# Check Against Reference

---

## **Times (seconds)**

Time vector for the reference signal. Specify the corresponding amplitudes in “Amplitudes” on page 1-19.

## **Settings**

**Default:** linspace(0,10)

## **Command-Line Information**

**Parameter:** ReferenceTimes

**Type:** string

**Value:** linspace(0,10) | vector of positive values of the same dimension as the amplitude vector |

**Default:** linspace(0,10)

## **Amplitudes**

Amplitude of the reference signal corresponding to the time vector specified in “Times (seconds)” on page 1-18.

## **Settings**

**Default:** `1-exp(-linspace(0,10)/2)`

## **Command-Line Information**

**Parameter:** `ReferenceAmplitudes`

**Type:** `string`

**Value:** `1-exp(-linspace(0,10)/2)` | vector of integers of the same dimension as the time vector

**Default:** `1-exp(-linspace(0,10)/2)`

# Check Against Reference

---

## **Absolute tolerance**

Absolute tolerance used to determine bounds as the signal approaches the reference signal.

During simulation, the signal must remain within upper and lower limits respective to the reference signal given by:

$$y_u = (1 + RelTol)y_r + AbsTol$$

$$y_l = (1 - RelTol)y_r - AbsTol$$

where  $y_r$  is the value of the reference at a certain time,  $y_u$  and  $y_l$  are the upper and lower tolerance bounds corresponding to that time point.

The block asserts if the signal violates these limits.

## **Settings**

**Default:**  $\text{eps}^{(1/3)}$

**Minimum:** 0

## **Command-Line Information**

**Parameter:** AbsTolerance

**Type:** string

**Value:**  $\text{eps}^{(1/3)}$  | positive real scalar

**Default:**  $\text{eps}^{(1/3)}$



## Relative tolerance

Relative tolerance used to determine bounds as the signal approaches the reference signal.

During simulation, the signal must remain within upper and lower limits respective to the reference signal given by:

$$y_u = (1 + RelTol)y_r + AbsTol$$

$$y_l = (1 - RelTol)y_r - AbsTol$$

where  $y_r$  is the value of the reference at a certain time,  $y_u$  and  $y_l$  are the upper and lower tolerance bounds corresponding to that time point.

The block asserts if the signal violates these limits.

## Settings

**Default:** 0.01

**Minimum:** 0

## Command-Line Information

**Parameter:** RelTolerance

**Type:** string

**Value:** 0.01 | positive real scalar

**Default:** 0.01

# Check Against Reference

---

## Enable assertion

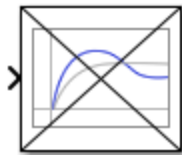
Enable the block to check that bounds specified and included for assertion in the **Bounds** tab are satisfied during simulation. Assertion fails if a bound is not satisfied. A warning, reporting the assertion failure, appears at the MATLAB prompt.

If the assertion fails, you can optionally specify that the block:

- Execute a MATLAB expression, specified in **Simulation callback when assertion fails (optional)**.
- Stop the simulation and bring that block into focus, by selecting **Stop simulation when assertion fails**.

This parameter has no effect if no bounds are included. For example, if you want to use the Linear Analysis Plots blocks for assertion, specify and include bounds in the **Bounds** tab.

Clearing this parameter disables assertion, i.e., the block no longer checks that specified bounds are satisfied. The block icon also updates to indicate that assertion is disabled.



In the Configuration Parameters dialog box of the Simulink model, the **Model Verification block enabling** option in the **Debugging** area of **Data Validity** node, lets you to enable or disable all model verification blocks in a model, regardless of the setting of this option.

## Settings

**Default:** On



On

Check that bounds included for assertion in the **Bounds** tab are satisfied during simulation. A warning, reporting assertion failure, is displayed at the MATLAB prompt if bounds are violated.



Off

Do not check that bounds included for assertion are satisfied during simulation.

## Dependencies

This parameter enables:

- **Simulation callback when assertion fails (optional)**
- **Stop simulation when assertion fails**

## Command-Line Information

**Parameter:** enabled

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'on'

# Check Against Reference

---

## **Simulation callback when assertion fails (optional)**

MATLAB expression to execute when assertion fails.

Because the expression is evaluated in the MATLAB workspace, define all variables used in the expression in that workspace.

### **Settings**

**Default:** []

A MATLAB expression.

### **Dependencies**

**Enable assertion** enables this parameter.

### **Command-Line Information**

**Parameter:** callback

**Type:** string

**Value:** '' | MATLAB expression

**Default:** ''

## Stop simulation when assertion fails

Stop the simulation when a bound specified in the **Bounds** tab is violated during simulation, i.e., assertion fails.

If you run the simulation from a Simulink model window, the Simulation Diagnostics window opens to display an error message. The block where the bound violation occurs is highlighted in the model.

### Settings

**Default:** Off

On

Stop simulation if a bound specified in the **Bounds** tab is violated.

Off

Continue simulation if a bound is violated and produce a warning message at the MATLAB prompt.

### Tips

- Because selecting this option stops the simulation as soon as the assertion fails, assertion failures that might occur later during the simulation are not reported. If you want *all* assertion failures to be reported, do not select this option.

### Dependencies

**Enable assertion** enables this parameter.

### Command-Line Information

**Parameter:** stopWhenAssertionFail

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'off'

# Check Against Reference

---

## Output assertion signal

Output a Boolean signal that, at each time step, is:

- True (1) if assertion succeeds, i.e., all bounds are satisfied
- False (0) if assertion fails, i.e., a bound is violated.

The output signal data type is Boolean only if the **Implement logic signals as Boolean data** option in the **Optimization** pane of the Configuration Parameters dialog box of the Simulink model is selected. Otherwise, the data type of the output signal is double.

Selecting this parameter adds an output port to the block that you can connect to any block in the model.

## Settings

**Default:**Off



On

Output a Boolean signal to indicate assertion status. Adds a port to the block.



Off

Do not output a Boolean signal to indicate assertion status.

## Tips

- Use this parameter to design complex assertion logic. For an example, see “Model Verification Using Simulink Control Design and Simulink Verification Blocks”.

## Command-Line Information

**Parameter:** export


**Type:** string

**Value:** 'on' | 'off'

**Default:** 'off'

## Show plot on block open

Open the plot window instead of the Block Parameters dialog box when you double-click the block in the Simulink model.

Use this parameter if you prefer to open and perform tasks, such as adding or modifying bounds, in the plot window instead of the Block Parameters dialog box. If you want to access the block parameters from the plot window, select **Edit** or click .

For more information on the plot, see **Show Plot**.

### Settings

**Default:** Off

On

Open the plot window when you double-click the block.

Off

Open the Block Parameters dialog box when double-clicking the block.

### Command-Line Information

**Parameter:** LaunchViewOnOpen

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'off'

## Show Plot

Open the plot window.

Use the plot to view:

- System characteristics and signals computed during simulation

You must click this button before you simulate the model to view the system characteristics or signal.

You can display additional characteristics, such as the peak response time, by right-clicking the plot and selecting **Characteristics**.

- Bounds





# Check Against Reference

---

You can specify bounds in the **Bounds** tab of the Block Parameters dialog box or right-click the plot and select **Bounds > New Bound**. For more information on the types of bounds, see the individual reference pages.

You can modify bounds by dragging the bound segment or by right-clicking the plot and selecting **Bounds > Edit Bound**. Before you simulate the model, click **Update Block** to update the bound value in the block parameters.

Typical tasks that you perform in the plot window include:

- Opening the Block Parameters dialog box by clicking  or selecting **Edit**.
- Finding the block that the plot window corresponds to by clicking  or selecting **View > Highlight Simulink Block**. This action makes the model window active and highlights the block.
- Simulating the model by clicking  or selecting **Simulation > Start**. This action also linearizes the portion of the model between the specified linearization input and output.
- Adding legend on the linear system characteristic plot by clicking .

A new option **Response Optimization** appears under **Tools** of Simulink Control Design Model Verification blocks if Simulink Design Optimization™ is installed.

## Response Optimization

Open the Design Optimization tool to optimize the model response to meet design requirements specified in the **Bounds** tab.

This button appears in Simulink Control Design “Model Verification” Block Parameters dialog box only if you have Simulink Design Optimization software.



## See Also

- “Design Optimization to Meet Step Response Requirements (GUI)”
- “Design Optimization to Meet Time- and Frequency-Domain Requirements (GUI)”

## See Also

- Check Custom Bounds
- Check Step Response Characteristics

## Tutorials

“Design Optimization to Track Reference Signal (GUI)”

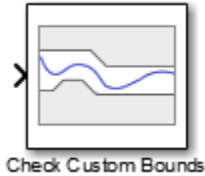
# Check Custom Bounds

---

**Purpose** Check that signal satisfies upper and lower bounds during simulation

**Library** Simulink Design Optimization

**Description** Check that a signal satisfies upper and lower bounds during simulation:



- If all bounds are satisfied, the block does nothing.
- If a bound is not satisfied, the block asserts, and a warning message appears at the MATLAB prompt. You can also specify that the block:
  - Evaluate a MATLAB expression.
  - Stop the simulation and bring that block into focus.

During simulation, the block can also output a logical assertion signal:

- If all bounds are satisfied, the signal is true (1).
- If a bound is not satisfied, the signal is false (0).

You can add Check Custom Bounds blocks on multiple signals to check that they satisfy the bounds.

You can also plot the bounds on a time plot to graphically verify that the signal satisfies the bounds.

This block and the other blocks in the Model Verification library test that a signal remains within specified time-domain characteristic bounds. When a model does not violate any bound, you can disable the block by clearing the assertion option. If you modify the model, you can re-enable assertion to ensure that your changes do not cause the model to violate a bound.

If the signal does not satisfy the bounds, you can optimize the model parameters to satisfy the bounds. If you have Simulink Control Design software, you can add frequency-domain bounds such as Bode magnitude and optimize the model response to satisfy both time- and frequency-domain requirements.

The block can be used in all simulation modes for signal monitoring but only in Normal or Accelerator simulation mode for response optimization.

## Parameters

Task	Parameters
Specify upper and lower bounds to: <ul style="list-style-type: none"> <li>• Assert that a signal satisfies the bounds</li> <li>• Optimize model response so that a signal satisfies the bounds</li> </ul>	In the <b>Bounds</b> tab: <ul style="list-style-type: none"> <li>• <b>Include upper bound in assertion</b></li> <li>• <b>Include lower bound in assertion</b></li> </ul>
Specify assertion options (only when you specify upper and lower bounds).	In the <b>Assertion</b> tab: <ul style="list-style-type: none"> <li>• <b>Enable assertion</b></li> <li>• <b>Simulation callback when assertion fails (optional)</b></li> <li>• <b>Stop simulation when assertion fails</b></li> <li>• <b>Output assertion signal</b></li> </ul>
Open Design Optimization tool to optimize model response	Click <b>Response Optimization</b>
Plot upper and lower bounds	Click <b>Show Plot</b> .
Display plot window instead of Block Parameters dialog box on double-clicking the block.	<b>Show plot on block open</b>

# Check Custom Bounds

---

## Include upper bound in assertion

Check that a signal is less than or equal to upper bounds, specified in **Times (seconds)** and **Amplitudes**, during simulation. The software displays a warning if the signal violates the upper bounds.

This parameter is used for assertion only if **Enable assertion** in the **Assertion** tab is selected.

You can specify multiple upper bounds on various model signals. The bounds also appear on the time plot. If you clear **Enable assertion**, the bounds are not used for assertion but continue to appear on the plot.

### Settings

**Default:** On



On

Check that the signal satisfies the specified upper bounds during simulation.

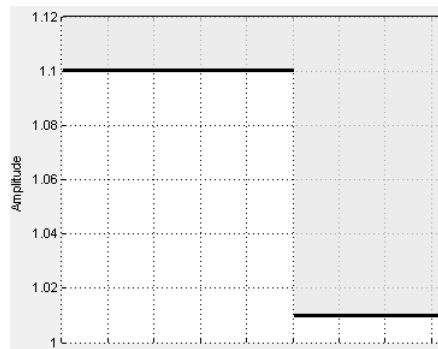


Off

Do not check that the signal satisfies the specified upper bounds during simulation.

### Tips

- Clearing this parameter disables the upper bounds and the software stops checking that the bounds are satisfied during simulation. The bound segments are also greyed out on the plot.



- To only view the bounds on the plot, clear **Enable assertion**.

### Command-Line Information

**Parameter:** EnableUpperBound

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'on'

# Check Custom Bounds

---

## Times (seconds)

Time vector for one or more upper bound segments, specified in seconds.

Specify the corresponding amplitude values in **Amplitudes**.

## Settings

**Default:** [0 5; 5 10]

Must be specified as start and end times:

- Positive finite numbers for a single bound with one edge.
- Matrix of positive finite numbers for a single bound with multiple edges.

For example, type [0.1 1; 1 10] for two edges at times [0.1 1] and [1 10].

- Cell array of matrices with positive finite numbers for multiple bounds.

## Tips

- To assert that amplitudes that correspond to the time vectors are satisfied, select both **Include upper bound in assertion** and **Enable assertion**.
- You can add or modify start and end times from the plot window:
  - To add new time vectors, right-click the yellow area on the plot, and select **Edit**. Click **Insert** to add a new row to the Edit Bound dialog box. Specify the start and end times of the new bound segment in the **Time** column. Specify the corresponding amplitudes in the **Amplitude** column.
  - To modify the start and end times, drag the bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new times in the **Time** column.

You must click **Update Block** before simulating the model.

## Command-Line Information

**Parameter:** UpperBoundTimes

**Type:** string

**Value:** [0 5; 5 10] | positive finite numbers | matrix of positive finite numbers | matrix of positive finite numbers cell array of matrices with positive finite numbers. Must be specified inside single quotes ('').

**Default:** [0 5; 5 10]

# Check Custom Bounds

---

## Amplitudes

Amplitude values for one or more upper bound segments.

Specify the corresponding start and end times in **Times (seconds)**.

## Settings

**Default:** [1.1 1.1; 1.01 1.01]

Must be specified as start and end amplitudes:

- Finite numbers for a single bound with one edge
- Matrix of finite numbers for a single bound with multiple edges.  
For example, type [0 1; 2 3] for two edges at amplitudes [0 1] and [2 3].
- Cell array of matrices with finite numbers for multiple bounds

## Tips

- To assert that amplitude bounds are satisfied, select both **Include upper bound in assertion** and **Enable assertion**.
- You can add or modify amplitudes from the plot window:
  - To add new amplitudes, right-click the plot, and select **Edit**. Click **Insert** to add a new row to the Edit Bound dialog box. Specify the start and end amplitudes of the new bound segment in the **Amplitude** column. Specify the corresponding start and end times in the **Time** column.
  - To modify the start and end amplitudes, drag the bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new amplitudes in the **Amplitude** column.

You must click **Update Block** before simulating the model.

## Command-Line Information

**Parameter:** UpperBoundAmplitudes

**Type:** string



**Value:** [1.1 1.1; 1.01 1.01] | finite numbers | matrix of finite numbers | cell array of matrices with finite numbers. Must be specified inside single quotes ('').

**Default:** [1.1 1.1; 1.01 1.01]

# Check Custom Bounds

---

## Include lower bound in assertion

Check that a signal is greater than or equal to lower bounds, specified in **Times (seconds)** and **Amplitudes**, during simulation.

This parameter is used for assertion only if **Enable assertion** in the **Assertion** tab is selected.

You can specify multiple lower bounds on various model signals. The bounds also appear on the time plot. If you clear **Enable assertion**, the bounds are not used for assertion but continue to appear on the plot.

## Settings

**Default:** Off



On

Check that the signal satisfies the specified lower bounds during simulation.

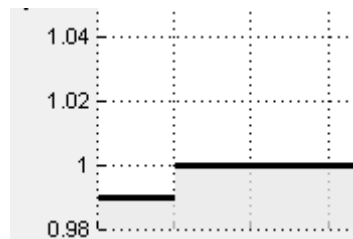


Off

Do not check that the signal satisfies the specified lower bounds during simulation.

## Tips

- Clearing this parameter disables the lower bounds and the software stops checking that the bounds are satisfied during simulation. The bound segments are also greyed out on the plot.



- To only view the bounds on the plot, clear **Enable assertion**.

## Command-Line Information

**Parameter:** EnableLowerBound

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

# Check Custom Bounds

---

## **Times (seconds)**

Time vector for one or more lower bound segments, specified in seconds.

Specify the corresponding amplitude values in **Amplitudes**

## **Settings**

**Default:** []

Must be specified as start and end times:

- Positive finite numbers for a single bound with one edge.
- Matrix of positive finite numbers for a single bound with multiple edges.

For example, type [0.1 1;1 10] for two edges at times [0.1 1] and [1 10].

- Cell array of matrices with positive finite numbers for multiple bounds.

## **Tips**

- To assert that amplitudes that correspond to the time vectors are satisfied, select both **Include lower bound in assertion** and **Enable assertion**.
- You can add or modify start and end times from the plot window:
  - To add new time vectors, right-click the yellow area on the plot, and select **Edit**. Click **Insert** to add a new row to the Edit Bound dialog box. Specify the start and end times of the new bound segment in the **Time** column. Specify the corresponding amplitudes in the **Amplitude** column.
  - To modify the start and end times, drag the bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new times in the **Time** column.

You must click **Update Block** before simulating the model.

## **Command-Line Information**

**Parameter:** LowerBoundTimes

**Type:** string

**Value:** [] | positive finite numbers | matrix of positive finite numbers | cell array of matrices with positive finite numbers. Must be specified inside single quotes (' ').

**Default:** []

# Check Custom Bounds

---

## Amplitudes

Amplitude values for one or more lower bound segments.

Specify the corresponding start and end times in **Times (seconds)**.

## Settings

**Default:** [ ]

Must be specified as start and end amplitudes:

- Finite numbers for a single bound with one edge
- Matrix of finite numbers for a single bound with multiple edges.

For example, type [0 1; 2 3] for two edges at amplitudes [0 1] and [2 3].

- Cell array of matrices with finite numbers for multiple bounds

## Tips

- To assert that amplitude bounds are satisfied, select both **Include lower bound in assertion** and **Enable assertion**.
- You can add or modify amplitudes from the plot window:
  - To add new amplitudes, right-click the plot, and select **Edit**. Click **Insert** to add a new row to the Edit Bound dialog box. Specify the start and end amplitudes of the new bound segment in the **Amplitude** column. Specify the corresponding start and end times in the **Time** column.
  - To modify the start and end amplitudes, drag the bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new amplitudes in the **Amplitude** column.

You must click **Update Block** before simulating the model.

## Command-Line Information

**Parameter:** LowerBoundAmplitudes

**Type:** string

**Value:** [] | finite numbers | matrix of finite numbers | cell array of matrices with finite numbers. Must be specified inside single quotes (' ').

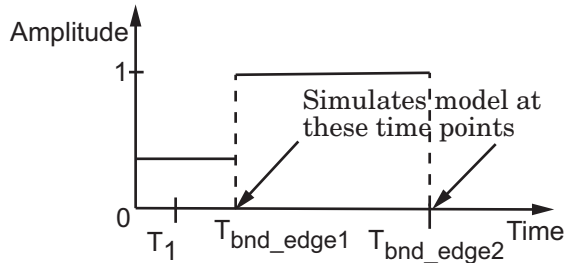
**Default:** []

# Check Custom Bounds

---

## Enable zero-crossing detection

Ensure that the software simulates the model to produce output at the bound edges. Simulating the model at the bound edges prevents the simulation solver from missing a bound edge without asserting that the signal satisfies that bound.



For more information on zero-crossing detection, see “Zero-Crossing Detection” in the *Simulink User Guide*.

## Settings

**Default:** On



On

Simulate model at the bound edges

This setting is ignored if the Simulink solver is fixed step.



Off

Do not simulate model at the bound edges. The software may not compute the output at the bound edges.

## Command-Line Information

**Parameter:** ZeroCross

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'on'



## Enable assertion

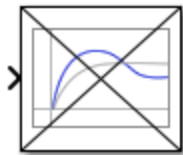
Enable the block to check that bounds specified and included for assertion in the **Bounds** tab are satisfied during simulation. Assertion fails if a bound is not satisfied. A warning, reporting the assertion failure, appears at the MATLAB prompt.

If the assertion fails, you can optionally specify that the block:

- Execute a MATLAB expression, specified in **Simulation callback when assertion fails (optional)**.
- Stop the simulation and bring that block into focus, by selecting **Stop simulation when assertion fails**.

This parameter has no effect if no bounds are included. For example, if you want to use the Linear Analysis Plots blocks for assertion, specify and include bounds in the **Bounds** tab.

Clearing this parameter disables assertion, i.e., the block no longer checks that specified bounds are satisfied. The block icon also updates to indicate that assertion is disabled.



In the Configuration Parameters dialog box of the Simulink model, the **Model Verification block enabling** option in the **Debugging** area of **Data Validity** node, lets you to enable or disable all model verification blocks in a model, regardless of the setting of this option.

## Settings

**Default:** On

# Check Custom Bounds

---

- On  
Check that bounds included for assertion in the **Bounds** tab are satisfied during simulation. A warning, reporting assertion failure, is displayed at the MATLAB prompt if bounds are violated.
- Off  
Do not check that bounds included for assertion are satisfied during simulation.

## Dependencies

This parameter enables:

- **Simulation callback when assertion fails (optional)**
- **Stop simulation when assertion fails**

## Command-Line Information

**Parameter:** enabled

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'on'

## **Simulation callback when assertion fails (optional)**

MATLAB expression to execute when assertion fails.

Because the expression is evaluated in the MATLAB workspace, define all variables used in the expression in that workspace.

### **Settings**

**Default:** []

A MATLAB expression.

### **Dependencies**

**Enable assertion** enables this parameter.

### **Command-Line Information**

**Parameter:** callback

**Type:** string

**Value:** ' ' | MATLAB expression

**Default:** ' '

# Check Custom Bounds

---

## Stop simulation when assertion fails

Stop the simulation when a bound specified in the **Bounds** tab is violated during simulation, i.e., assertion fails.

If you run the simulation from a Simulink model window, the Simulation Diagnostics window opens to display an error message. The block where the bound violation occurs is highlighted in the model.

### Settings

**Default:** Off



On

Stop simulation if a bound specified in the **Bounds** tab is violated.



Off

Continue simulation if a bound is violated and produce a warning message at the MATLAB prompt.

### Tips

- Because selecting this option stops the simulation as soon as the assertion fails, assertion failures that might occur later during the simulation are not reported. If you want *all* assertion failures to be reported, do not select this option.

### Dependencies

**Enable assertion** enables this parameter.

### Command-Line Information

**Parameter:** stopWhenAssertionFail

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'off'

## Output assertion signal

Output a Boolean signal that, at each time step, is:

- True (1) if assertion succeeds, i.e., all bounds are satisfied
- False (0) if assertion fails, i.e., a bound is violated.

The output signal data type is Boolean only if the **Implement logic signals as Boolean data** option in the **Optimization** pane of the Configuration Parameters dialog box of the Simulink model is selected. Otherwise, the data type of the output signal is double.

Selecting this parameter adds an output port to the block that you can connect to any block in the model.

## Settings

**Default:** Off

On

Output a Boolean signal to indicate assertion status. Adds a port to the block.

Off

Do not output a Boolean signal to indicate assertion status.

## Tips

- Use this parameter to design complex assertion logic. For an example, see “Model Verification Using Simulink Control Design and Simulink Verification Blocks”.

## Command-Line Information

**Parameter:** export

**Type:** string

**Value:** 'on' | 'off'


**Default:** 'off'

# Check Custom Bounds

---

## Show plot on block open

Open the plot window instead of the Block Parameters dialog box when you double-click the block in the Simulink model.

Use this parameter if you prefer to open and perform tasks, such as adding or modifying bounds, in the plot window instead of the Block Parameters dialog box. If you want to access the block parameters from the plot window, select **Edit** or click .

For more information on the plot, see **Show Plot**.

### Settings

**Default:** Off

On

Open the plot window when you double-click the block.

Off

Open the Block Parameters dialog box when double-clicking the block.

### Command-Line Information

**Parameter:** LaunchViewOnOpen

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'off'

## Show Plot

Open the plot window.

Use the plot to view:

- System characteristics and signals computed during simulation

You must click this button before you simulate the model to view the system characteristics or signal.





You can display additional characteristics, such as the peak response time, by right-clicking the plot and selecting **Characteristics**.

- Bounds

You can specify bounds in the **Bounds** tab of the Block Parameters dialog box or right-click the plot and select **Bounds > New Bound**. For more information on the types of bounds, see the individual reference pages.

You can modify bounds by dragging the bound segment or by right-clicking the plot and selecting **Bounds > Edit Bound**. Before you simulate the model, click **Update Block** to update the bound value in the block parameters.

Typical tasks that you perform in the plot window include:

- Opening the Block Parameters dialog box by clicking  or selecting **Edit**.
- Finding the block that the plot window corresponds to by clicking  or selecting **View > Highlight Simulink Block**. This action makes the model window active and highlights the block.
- Simulating the model by clicking  or selecting **Simulation > Start**. This action also linearizes the portion of the model between the specified linearization input and output.
- Adding legend on the linear system characteristic plot by clicking .

A new option **Response Optimization** appears under **Tools** of Simulink Control Design Model Verification blocks if Simulink Design Optimization is installed.

## Response Optimization

Open the Design Optimization tool to optimize the model response to meet design requirements specified in the **Bounds** tab.

This button appears in Simulink Control Design “Model Verification” Block Parameters dialog box only if you have Simulink Design Optimization software.

# Check Custom Bounds

---

## See Also

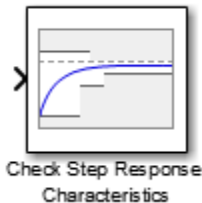
- “Design Optimization to Meet Step Response Requirements (GUI)”
- “Design Optimization to Meet Time- and Frequency-Domain Requirements (GUI)”



**Purpose** Check that model signal satisfies step response bounds during simulation

**Library** Simulink Design Optimization

## Description



Check that a signal satisfies step response bounds during simulation:

- If all bounds are satisfied, the block does nothing.
- If a bound is not satisfied, the block asserts, and a warning message appears at the MATLAB prompt. You can also specify that the block:
  - Evaluate a MATLAB expression.
  - Stop the simulation and bring that block into focus.

During simulation, the block can also output a logical assertion signal:

- If all bounds are satisfied, the signal is true (1).
- If a bound is not satisfied, the signal is false (0).

You can add Check Step Response Characteristics blocks on multiple signals to check that they satisfy the bounds.

You can also plot the bounds on a time plot to graphically verify that the signal satisfies the bounds.

This block and the other blocks in the Model Verification library test that a signal remains within specified time-domain characteristic bounds. When a model does not violate any bound, you can disable the

# Check Step Response Characteristics

block by clearing the assertion option. If you modify the model, you can re-enable assertion to ensure that your changes do not cause the model to violate a bound.

If the signal does not satisfy the bounds, you can optimize the model parameters to satisfy the bounds. If you have Simulink Control Design software, you can add frequency-domain bounds such as Bode magnitude and optimize the model response to satisfy both time- and frequency-domain requirements.

The block can be used in all simulation modes for signal monitoring but only in Normal or Accelerator simulation mode for response optimization.

## Parameters

Task	Parameters
Specify step response bounds to: <ul style="list-style-type: none"><li>• Assert that a signal satisfies the bounds</li><li>• Optimize model response so that a signal satisfies the bounds</li></ul>	<b>Include step response bound in assertion in Bounds tab.</b>
Specify assertion options (only when you specify step response bounds).	In the <b>Assertion</b> tab: <ul style="list-style-type: none"><li>• <b>Enable assertion</b></li><li>• <b>Simulation callback when assertion fails (optional)</b></li><li>• <b>Stop simulation when assertion fails</b></li><li>• <b>Output assertion signal</b></li></ul>
Open Design Optimization tool to optimize model response	Click <b>Response Optimization</b>

# Check Step Response Characteristics

---

<b>Task</b>	<b>Parameters</b>
Plot step response	Click <b>Show Plot</b> .
Display plot window instead of Block Parameters dialog box on double-clicking the block.	<b>Show plot on block open</b>

# Check Step Response Characteristics

---

## Include step response bound in assertion

Check that the step response satisfies *all* the characteristics specified in:

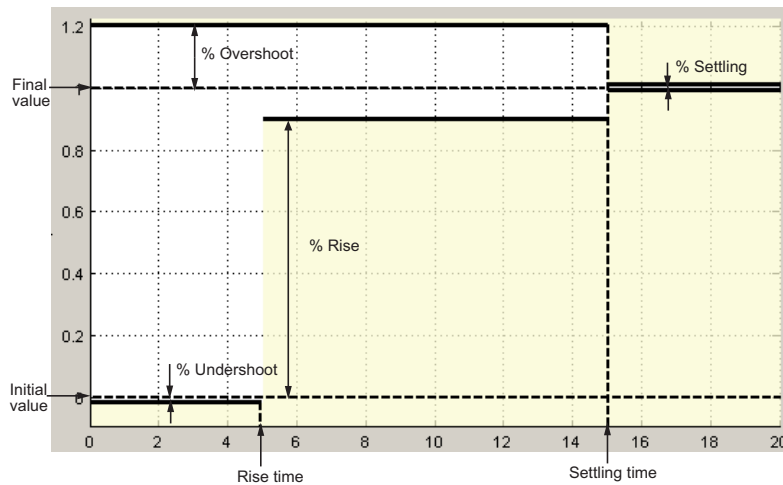
- **Step time (seconds)**
- **Initial value**
- **Final Value**
- **Rise time (seconds)**
- **% Rise**
- **Settling time (seconds)**
- **% Settling**
- **% Overshoot**
- **% Undershoot**

The software displays a warning if the signal violates the specified step response characteristics.

This parameter is used for assertion only if **Enable assertion** in the **Assertion** tab is selected.

The bounds also appear on the step response plot if you click **Show Plot**, as shown in the next figure.

# Check Step Response Characteristics



By default, the line segments represent the following step response requirements:

- Amplitude less than or equal to  $-0.01$  up to the rise time of 5 seconds for 1% undershoot
- Amplitude between 0.9 and 1.2 up to the settling time of 15 seconds
- Amplitude equal to 1.2 for 20% overshoot up to the settling time of 15 seconds
- Amplitude between 0.99 and 1.01 beyond the settling time for 2% settling

If you clear **Enable assertion**, the bounds are not used for assertion but continue to appear on the plot.

## Settings

Default: On

On

Check that the step response satisfies the specified bounds during simulation.

# Check Step Response Characteristics

---

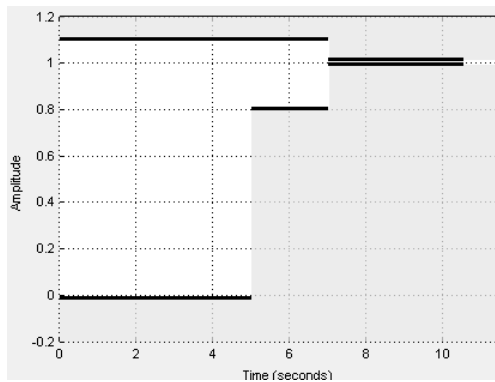


Off

Do not check that the step response satisfies the specified bounds during simulation.

## Tips

- Clearing this parameter disables the step response bounds and the software stops checking that the bounds are satisfied during simulation. The bound segments are also greyed out on the plot.



- To only view the bounds on the plot, clear **Enable assertion**.

## Command-Line Information

**Parameter:** EnableStepResponseBound

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'on'

## **Step time (seconds)**

Time, in seconds, when the step response starts.

### **Settings**

**Default:** 0

**Minimum:** 0

Finite real nonnegative scalar.

### **Tips**

- To assert that step time value is satisfied, select both **Include step response bound in assertion** and **Enable assertion**.
- To modify the step time value from the plot window, drag the corresponding bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new value in **Step time**. You must click **Update Block** before simulating the model.

### **Command-Line Information**

**Parameter:** StepTime

**Type:** string

**Value:** 0 | finite real nonnegative scalar. Must be specified inside single quotes (' ').

**Default:** 0

# Check Step Response Characteristics

---

## Initial value

Value of the signal level before the step response starts.

## Settings

**Default:** 0

Finite real scalar not equal to the final value.

## Tips

- To assert that initial value is satisfied, select both **Include step response bound in assertion** and **Enable assertion**.
- To modify the initial value from the plot window, drag the corresponding bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new value in **Initial value**. You must click **Update Block** before simulating the model.

## Command-Line Information

**Parameter:** InitialValue

**Type:** string

**Value:** 0 | finite real scalar not equal to final value.

Must be specified inside single quotes (' ').

**Default:** 0



## Final value

Final value of the step response.

## Settings

**Default:** 1

Finite real scalar not equal to the initial value.

## Tips

- To assert that final value is satisfied, select both **Include step response bound in assertion** and **Enable assertion**.
- To modify the final value from the plot window, drag the corresponding bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new value in **Final value**. You must click **Update Block** before simulating the model.

## Command-Line Information

**Parameter:** FinalValue

**Type:** string

**Value:** 1 | finite real scalar not equal to the initial value. Must be specified inside single quotes (' ').

**Default:** 1

# Check Step Response Characteristics

---

## **Rise time (seconds)**

Time taken, in seconds, for the signal to reach a percentage of the final value specified in % **Rise**.

### **Settings**

**Default:** 5

**Minimum:** 0

Finite positive real scalar, less than the settling time.

### **Tips**

- To assert that rise time value is satisfied, select both **Include step response bound in assertion** and **Enable assertion**.
- To modify the rise time from the plot window, drag the corresponding bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new value in **Rise time**. You must click **Update Block** before simulating the model.

### **Command-Line Information**

**Parameter:** RiseTime

**Type:** string

**Value:** 5 | finite positive real scalar. Must be specified inside single quotes ('').

**Default:** 5

# Check Step Response Characteristics

---

## **% Rise**

The percentage of final value used with the **Rise time** to define the overall rise time characteristics.

### **Settings**

**Default:** 80

**Minimum:** 0

**Maximum:** 100

Positive real scalar, less than (100 – % settling).

### **Tips**

- To assert that percent rise value is satisfied, select both **Include step response bound in assertion** and **Enable assertion**.
- To modify the percent rise from the plot window, drag the corresponding bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new value in **% Rise**. You must click **Update Block** before simulating the model.

### **Command-Line Information**

**Parameter:** PercentRise

**Type:** string

**Value:** 80 | positive scalar less than (100 – % settling).  
Must be specified inside single quotes (' ').

**Default:** 80

# Check Step Response Characteristics

---

## Settling time (seconds)

The time, in seconds, taken for the signal to settle within a specified range around the final value. This settling range is defined as the final value plus or minus the percentage of the final value, specified in % **Settling**.

## Settings

**Default:** 7

Finite positive real scalar, greater than rise time.

## Tips

- To assert that final value is satisfied, select both **Include step response bound in assertion** and **Enable assertion**.
- To modify the settling time from the plot window, drag the corresponding bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new value in **Settling time**. You must click **Update Block** before simulating the model.

## Command-Line Information

**Parameter:** SettlingTime

**Type:** string

**Value:** 7 | positive finite real scalar greater than rise time. Must be specified inside single quotes (' ').

**Default:** 7

## **% Settling**

The percentage of the final value that defines the settling range of the **Settling time** characteristic.

### **Settings**

**Default:** 1

**Minimum:** 0

**Maximum:** 100

Real positive finite scalar, less than (100 – % rise) and less than % overshoot.

### **Tips**

- To assert that percent settling value is satisfied, select both **Include step response bound in assertion** and **Enable assertion**.
- To modify the percent settling from the plot window, drag the corresponding bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new value in **% Settling**. You must click **Update Block** before simulating the model.

### **Command-Line Information**

**Parameter:** PercentSettling

**Type:** string

**Value:** 1 | Real positive finite scalar less than (100 % rise) and less than % overshoot. Must be specified inside single quotes (' ').

**Default:** 1

# Check Step Response Characteristics

---

## **% Overshoot**

The amount by which the signal can exceed the final value before settling, specified as a percentage.

### **Settings**

**Default:** 10

**Minimum:** 0

**Maximum:** 100

Positive real scalar, greater than % settling.

### **Tips**

- To assert that percent overshoot value is satisfied, select both **Include step response bound in assertion** and **Enable assertion**.
- To modify the percent overshoot from the plot window, drag the corresponding bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new value in **% Overshoot**. You must click **Update Block** before simulating the model.

### **Command-Line Information**

**Parameter:** PercentOvershoot

**Type:** string

**Value:** 10 | Positive real scalar greater than % settling. Must be specified inside single quotes (' ').

**Default:** 10

# Check Step Response Characteristics

---

## **% Undershoot:**

The amount by which the signal can undershoot the initial value, specified as a percentage.

### **Settings**

**Default:** 1

**Minimum:** 0

**Maximum:** 100

Positive finite real scalar.

### **Tips**

- To assert that percent undershoot value is satisfied, select both **Include step response bound in assertion** and **Enable assertion**.
- To modify the percent undershoot from the plot window, drag the corresponding bound segment. Alternatively, right-click the segment, and select **Edit**. Specify the new value in **% Undershoot**. You must click **Update Block** before simulating the model.

### **Command-Line Information**

**Parameter:** PercentUndershoot

**Type:** string

**Value:** 1 | positive finite real scalar between 0 and 100. Must be specified inside single quotes (' ').

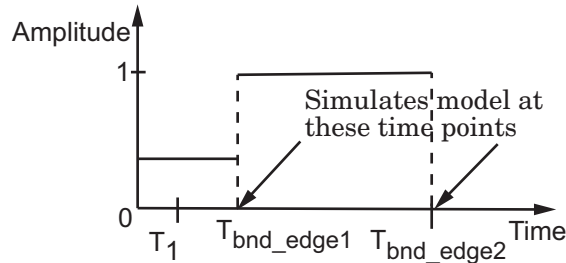
**Default:** 1

# Check Step Response Characteristics

---

## Enable zero-crossing detection

Ensure that the software simulates the model to produce output at the bound edges. Simulating the model at the bound edges prevents the simulation solver from missing a bound edge without asserting that the signal satisfies that bound.



For more information on zero-crossing detection, see “Zero-Crossing Detection” in the *Simulink User Guide*.

## Settings

**Default:** On



On

Simulate model at the bound edges

This setting is ignored if the Simulink solver is fixed step.



Off

Do not simulate model at the bound edges. The software may not compute the output at the bound edges.

## Command-Line Information

**Parameter:** ZeroCross

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'on'



# Check Step Response Characteristics

## Enable assertion

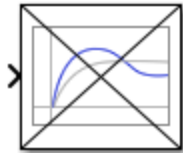
Enable the block to check that bounds specified and included for assertion in the **Bounds** tab are satisfied during simulation. Assertion fails if a bound is not satisfied. A warning, reporting the assertion failure, appears at the MATLAB prompt.

If the assertion fails, you can optionally specify that the block:

- Execute a MATLAB expression, specified in **Simulation callback when assertion fails (optional)**.
- Stop the simulation and bring that block into focus, by selecting **Stop simulation when assertion fails**.

This parameter has no effect if no bounds are included. For example, if you want to use the Linear Analysis Plots blocks for assertion, specify and include bounds in the **Bounds** tab.

Clearing this parameter disables assertion, i.e., the block no longer checks that specified bounds are satisfied. The block icon also updates to indicate that assertion is disabled.



In the Configuration Parameters dialog box of the Simulink model, the **Model Verification block enabling** option in the **Debugging** area of **Data Validity** node, lets you to enable or disable all model verification blocks in a model, regardless of the setting of this option.

## Settings

**Default:** On

# Check Step Response Characteristics

---

- On  
Check that bounds included for assertion in the **Bounds** tab are satisfied during simulation. A warning, reporting assertion failure, is displayed at the MATLAB prompt if bounds are violated.
- Off  
Do not check that bounds included for assertion are satisfied during simulation.

## Dependencies

This parameter enables:

- **Simulation callback when assertion fails (optional)**
- **Stop simulation when assertion fails**

## Command-Line Information

**Parameter:** enabled

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'on'

## Simulation callback when assertion fails (optional)

MATLAB expression to execute when assertion fails.

Because the expression is evaluated in the MATLAB workspace, define all variables used in the expression in that workspace.

## Settings

**Default:** []

A MATLAB expression.

## Dependencies

**Enable assertion** enables this parameter.

## Command-Line Information

**Parameter:** callback

**Type:** string

**Value:** '' | MATLAB expression

# Check Step Response Characteristics

---

Default: ''

# Check Step Response Characteristics

---

## Stop simulation when assertion fails

Stop the simulation when a bound specified in the **Bounds** tab is violated during simulation, i.e., assertion fails.

If you run the simulation from a Simulink model window, the Simulation Diagnostics window opens to display an error message. The block where the bound violation occurs is highlighted in the model.

### Settings

**Default:** Off



On

Stop simulation if a bound specified in the **Bounds** tab is violated.



Off

Continue simulation if a bound is violated and produce a warning message at the MATLAB prompt.

### Tips

- Because selecting this option stops the simulation as soon as the assertion fails, assertion failures that might occur later during the simulation are not reported. If you want *all* assertion failures to be reported, do not select this option.

### Dependencies

**Enable assertion** enables this parameter.

### Command-Line Information

**Parameter:** stopWhenAssertionFail

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'off'

## Output assertion signal

Output a Boolean signal that, at each time step, is:

- True (1) if assertion succeeds, i.e., all bounds are satisfied
- False (0) if assertion fails, i.e., a bound is violated.

The output signal data type is Boolean only if the **Implement logic signals as Boolean data** option in the **Optimization** pane of the Configuration Parameters dialog box of the Simulink model is selected. Otherwise, the data type of the output signal is double.

Selecting this parameter adds an output port to the block that you can connect to any block in the model.

## Settings

**Default:** Off

On

Output a Boolean signal to indicate assertion status. Adds a port to the block.

Off

Do not output a Boolean signal to indicate assertion status.

## Tips

- Use this parameter to design complex assertion logic. For an example, see “Model Verification Using Simulink Control Design and Simulink Verification Blocks”.

## Command-Line Information

**Parameter:** export

**Type:** string

**Value:** 'on' | 'off'


**Default:** 'off'

# Check Step Response Characteristics

---

## Show plot on block open

Open the plot window instead of the Block Parameters dialog box when you double-click the block in the Simulink model.

Use this parameter if you prefer to open and perform tasks, such as adding or modifying bounds, in the plot window instead of the Block Parameters dialog box. If you want to access the block parameters from the plot window, select **Edit** or click .

For more information on the plot, see **Show Plot**.

### Settings

**Default:** Off



On

Open the plot window when you double-click the block.



Off

Open the Block Parameters dialog box when double-clicking the block.

### Command-Line Information

**Parameter:** LaunchViewOnOpen

**Type:** string

**Value:** 'on' | 'off'

**Default:** 'off'

## Show Plot

Open the plot window.

Use the plot to view:

- System characteristics and signals computed during simulation

You must click this button before you simulate the model to view the system characteristics or signal.





You can display additional characteristics, such as the peak response time, by right-clicking the plot and selecting **Characteristics**.

- Bounds

You can specify bounds in the **Bounds** tab of the Block Parameters dialog box or right-click the plot and select **Bounds > New Bound**. For more information on the types of bounds, see the individual reference pages.

You can modify bounds by dragging the bound segment or by right-clicking the plot and selecting **Bounds > Edit Bound**. Before you simulate the model, click **Update Block** to update the bound value in the block parameters.

Typical tasks that you perform in the plot window include:

- Opening the Block Parameters dialog box by clicking  or selecting **Edit**.
- Finding the block that the plot window corresponds to by clicking  or selecting **View > Highlight Simulink Block**. This action makes the model window active and highlights the block.
- Simulating the model by clicking  or selecting **Simulation > Start**. This action also linearizes the portion of the model between the specified linearization input and output.
- Adding legend on the linear system characteristic plot by clicking .

# Check Step Response Characteristics

---

A new option **Response Optimization** appears under **Tools** of Simulink Control Design Model Verification blocks if Simulink Design Optimization is installed.

## **Response Optimization**

Open the Design Optimization tool to optimize the model response to meet design requirements specified in the **Bounds** tab.

This button appears in Simulink Control Design “Model Verification” Block Parameters dialog box only if you have Simulink Design Optimization software.

### **See Also**

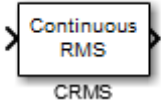
- “Design Optimization to Meet Step Response Requirements (GUI)”
- “Design Optimization to Meet Time- and Frequency-Domain Requirements (GUI)”



**Purpose** Compute continuous-time, cumulative root mean square (CRMS) of signal

**Library** Simulink Design Optimization

**Description** Attach the CRMS block to a signal to compute its continuous-time, cumulative root mean square value. Use in conjunction with the Signal Constraint block to optimize the signal energy.



The continuous-time, cumulative root mean square value of a signal  $u(t)$  is defined as

$$R.M.S = \sqrt{\frac{1}{T} \int_0^T \|u(t)\|^2 dt}$$

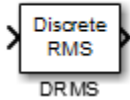
The R.M.S value gives a measure of the average energy in the signal.

**See Also** DRMS, Signal Constraint

**Purpose** Compute discrete-time, cumulative root mean square (DRMS) of signal

**Library** Simulink Design Optimization

**Description** Attach the DRMS block to a signal to compute its discrete-time, cumulative root mean square value. Use in conjunction with the Signal Constraint block to optimize the signal energy.



The discrete-time, cumulative root mean square value of a signal  $u(t_i)$  is defined as

$$R.M.S = \sqrt{\frac{1}{N} \sum_{i=1}^N \|u(t_i)\|^2}$$

The R.M.S value gives a measure of the average energy in the signal.

**See Also** CRMS, Signal Constraint

**Purpose** Specify desired signal response

---

**Note** Signal Constraint has been removed. Use `sdouupdate` to replace it with the equivalent block from the **Signal Constraints** block library.

---

**Library** Simulink Design Optimization

# Signal Constraint

---

# Class Reference

---

# param.Continuous

---

<b>Purpose</b>	Continuous parameter
<b>Syntax</b>	<code>p = param.Continuous(paramname)</code> <code>p = param.Continuous(paramname,paramvalue)</code>
<b>Description</b>	<p>A continuous parameter is a numeric parameter that can take any value in a specified interval. The parameter can be scalar- or matrix-valued.</p> <p>Typically, you use continuous parameters to create parametric models and to estimate or optimize tunable parameters in such models.</p>
<b>Construction</b>	<p><code>p = param.Continuous(paramname)</code> constructs a <code>param.Continuous</code> object and assigns the specified parameter name to the <code>Name</code> property and default values to the remaining properties.</p> <p><code>p = param.Continuous(paramname,paramvalue)</code> assigns the specified parameter value to the <code>Value</code> property.</p> <p><code>sdo.getParameterFromModel</code> also constructs a <code>param.Continuous</code> object or an array of <code>param.Continuous</code> objects for Simulink model parameters.</p>
<b>Input Arguments</b>	<p><b>paramname</b> Parameter name, specified as a string inside single quotes ( ' ').</p> <p><b>paramvalue</b> Scalar or matrix numeric double</p>
<b>Properties</b>	<p><b>Free</b> Flag specifying whether the parameter is tunable or not.</p> <p>Set the <code>Free</code> property to <code>true</code> (1) for tunable parameters and <code>false</code> (0) for parameters you do not want to tune (fixed).</p> <p>The dimension of this property must match the dimension of the <code>Value</code> property.</p> <p>For matrix-valued parameters, you can:</p>

- Fix individual matrix elements. For example `p.Free = [true false; false true]` or `p.Free([2 3]) = false`.
- Use scalar expansion to fix all matrix elements. For example `p.Free = false`.

**Default:** `true (1)`

## Info

Structure array specifying parameter units and labels.

The structure has `Label` and `Unit` fields.

The array dimension must match the dimension of the `Value` property.

Use this property to store parameter units and labels that describe the parameter. For example `p.Info(1,1).Unit = 'N/m'`; or `p.Info(1,1).Label = 'spring constant'`.

**Default:** `' '` for both `Label` and `Unit` fields

## Maximum

Upper bound for the parameter value.

The dimension of this property must match the dimension of the `Value` property.

For matrix-valued parameters, you can:

- Specify upper bounds on individual matrix elements. For example `p.Maximum([1 4]) = 5`.
- Use scalar expansion to set the upper bound for all matrix elements. For example `p.Maximum = 5`.

**Default:** `Inf`

## Minimum

# param.Continuous

---

Lower bound for the parameter value.

The dimension of this property must match the dimension of the Value property.

For matrix-valued parameters, you can:

- Specify lower bounds on individual matrix elements. For example `p.Minimum([1 4]) = -5`.
- Use scalar expansion to set the lower bound for all matrix elements. For example `p.Minimum = -5`.

**Default:** `Inf`

## Name

Parameter name.

This property is read-only and is set at object construction.

**Default:** `''`

## Scale

Scaling factor used to normalize the parameter value.

The dimension of this property must match the dimension of the Value property.

For matrix-valued parameters, you can:

- Specify scaling for individual matrix elements. For example `p.Scale([1 4]) = 1`.
- Use scalar expansion to set the scaling for all matrix elements. For example `p.Scale = 1`.

**Default:** `1`

## Value



Scalar or matrix value of a parameter.

The dimension of this property is set at object construction.

**Default:** 0

## Methods

`isreal` Determine if parameter value, minimum and maximum are real

## Copy Semantics

Value. To learn how value classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Construct a `param.Continuous` object and specify the maximum value.

```
p = param.Continuous('K',eye(2));  
p.Maximum = 5;
```

## Alternatives

“Design Optimization to Meet Step Response Requirements (GUI)”

## See Also

`sdo.optimize` | `sdo.getParameterFromModel`

## Tutorials

- “Design Optimization to Meet Step Response Requirements (Code)”
- “Design Optimization to Meet a Custom Objective (Code)”

## How To

- Class Attributes
- Property Attributes

# param.State

---

**Superclasses** Continuous

**Purpose** Specify tuning parameters for model states

**Description** A *state parameter* is a numeric parameter, representing a state associated with a model, that can take any value in a specified interval. The parameter can take scalar or matrix values.

You use state parameters to estimate or specify the initial state values of a model.

**Construction** You obtain a state parameter using the `sdo.getStateFromModel` function.

For example, use

```
s = sdo.getStateFromModel('sdoMassSpringDamper','Position');
```

to obtain the state parameter of the `Position` block of the `sdoMassSpringDamper` Simulink model.

**Properties** **Free**

Flag specifying whether the state parameter is tunable or not.

Set the `Free` property to `true` (1) for tunable state parameters and `false` (0) for state parameters you do not want to tune, to designate them as fixed.

The dimension of this property must match the dimension of the `Value` property.

For matrix-valued state parameters, you can:

- Fix individual matrix elements. For example, `p.Free = [true false; false true]` or `p.Free([2 3]) = false`.
- Use scalar expansion to fix all matrix elements. For example, `p.Free = false`.

**Default:** true (1)

## Info

Structure array specifying state parameter units and labels.

The structure has `Label` and `Unit` fields.

The array dimension must match the dimension of the `Value` property.

Use this property to store state parameter units and labels. For example, `p.Info(1,1).Unit = 'N/m'`; or `p.Info(1,1).Label = 'spring constant'`.

**Default:** '' for both `Label` and `Unit` fields

## Maximum

Upper bound for the state parameter value.

The dimension of this property must match the dimension of the `Value` property.

For matrix-valued state parameters, you can:

- Specify upper bounds on individual matrix elements. For example, `p.Maximum([1 4]) = 5`.
- Use scalar expansion to set the upper bound for all matrix elements. For example `p.Maximum = 5`.

**Default:** Inf

## Minimum

Lower bound for the state parameter value.

The dimension of this property must match the dimension of the `Value` property.

For matrix-valued state parameters, you can:

# param.State

---

- Specify lower bounds on individual matrix elements. For example `p.Minimum([1 4]) = -5`.
- Use scalar expansion to set the lower bound for all matrix elements. For example `p.Minimum = -5`.

**Default:** `Inf`

## Name

State parameter name.

This read-only property is set at object construction.

**Default:** `''`

## Scale

Scaling factor used to normalize the state parameter value.

The dimension of this property must match the dimension of the Value property.

For matrix-valued state parameters, you can:

- Specify scaling for individual matrix elements. For example `p.Scale([1 4]) = 1`.
- Use scalar expansion to set the scaling for all matrix elements. For example `p.Scale = 1`.

**Default:** `1`

## Value

State parameter value.

You can specify the value as either a scalar or a matrix.

The dimension of this property is set at object construction.

**Default:** `0`

## **dxFree**

Flag specifying whether the state parameter derivative (with respect to time) is tunable or not.

Set the `dxFree` property to `true` (1) for tunable state parameter derivatives and `false` (0) for state parameter derivatives you do not want to tune (fixed).

The dimension of this property must match the dimension of the `Value` property.

For matrix-valued state parameter derivatives, you can:

- Fix individual matrix elements. For example `p.dxFree = [true false; false true]` or `p.dxFree([2 3]) = false`.
- Use scalar expansion to fix all matrix elements. For example `p.dxFree = false`.

**Default:** `true` (1)

## **dxValue**

State parameter derivative (with respect to time) value.

The dimension of this property must match the dimension of the `Value` property.

**Default:** 0

## **Methods**

### **Inherited Methods**

`isreal`

Determine if parameter value, minimum and maximum are real

## **Copy Semantics**

`Value`. To learn how value classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

### Get State Parameters from Model

```
modelname = 'sdoAircraft';  
open_system(modelname);  
  
blockpath = {'sdoAircraft/Actuator Model', ...  
            'sdoAircraft/Controller/Proportional plus integral compensator'};  
  
s = sdo.getStateFromModel(modelname,blockpath);
```

## Alternatives

“Specify Known Initial States”

## See Also

`sdo.getStateFromModel` `sdo.Experiment` |

## Related Examples

- “Estimate Model Parameter Values (Code)”
- “Estimate Model Parameters and Initial States (Code)”

## Concepts

- Class Attributes
- Property Attributes

**Purpose** Specify experiment I/O data, model parameters, and initial-state values

**Description** An *experiment* specifies input and output data for a Simulink model. You can also specify model parameters and initial-state values.

Typically, you use experiments to estimate unknown model parameter values. You can also use the `createSimulator` method of an experiment to create a simulation object. Use the simulation object to simulate the model and compare measured and simulated data.

**Construction** `exp = sdo.Experiment(modelname)`

Constructs an `sdo.Experiment` object. It assigns the specified model name to the `ModelName` property and default values to the remaining properties.

### Input Arguments

#### **modelname**

Simulink model name, specified as a string inside single quotes (' ').

The model must either be open or appear on the MATLAB path.

### Properties

#### **InitialStates**

Model initial-state for the experiment, specified as a `param.State` object.

To specify multiple initial-states, use a vector of `param.State` objects.

To obtain model initial states, use `sdo.getStateFromModel`.

Use this property only for specifying initial-states that differ from the initial state values defined in the model.

- To estimate the value of an initial state, set the `Free` property of the initial state to `true`.

# sdo.Experiment

---

When you have multiple experiments for a given model, you can estimate model initial states on a per-experiment basis. To do so, specify the model initial states for each experiment. You can optionally specify an initial guess for the initial state values for any of the experiments using the `Value` property of the state parameters.

- To specify an initial state value as a known quantity, not to be estimated, set its `Free` property to `false`.

After specifying the initial states that you are estimating for an experiment, use `sdo.Experiment.getValuesToEstimate`. `sdo.Experiment.getValuesToEstimate` returns a vector of all the model parameters and initial states that you want to estimate. You use this vector as an input to `sdo.optimize` to specify the parameters that you want to estimate.

**Default:** []

## **InputData**

Experiment input data.

Specify signals to apply to root-level input ports. For information on supported forms of input data, see “Forms of Input Data”.

**Default:** []

## **ModelName**

Simulink model name associated with the experiment, specified as a string.

The model must appear on the MATLAB path.

**Default:** ''

## **OutputData**



Experiment output data, specified as a `Simulink.SimulationData.Signal` object.

To specify multiple output signals, use a vector of `Simulink.SimulationData.Signal` objects.

**Default:** []

## Parameters

Model parameter value for the experiment, specified as a `param.Continuous` object.

To specify values for multiple parameters, use a vector of `param.Continuous` objects.

To obtain model parameters, use `sdo.getParameterFromModel`.

Use this property only for specifying parameters values that differ from the parameters values defined in the model.

- To estimate the value of a parameter, set the `Free` property of the parameter to `true`.

When you have multiple experiments for a given model, you can:

- Estimate a model parameter on a per-experiment basis. To do so, specify the model parameter for each experiment. You can optionally specify the initial guess for the parameter value for any of the experiments using the `Value` property.
- Estimate one value for a model parameter using all the experimental data. To do so, do not specify the model parameter for the experiments. Instead, call `sdo.optimize` with the model parameter directly.

For an example of estimating model parameters on a per-experiment basis and using data from multiple experiments, see “Estimate Model Parameters Per Experiment (Code)”.

# sdo.Experiment

---

- To specify a parameter value as a known quantity, not to be estimated, set its `Free` property to `false`.

After specifying the parameters that you are estimating for an experiment, use `sdo.Experiment.getValuesToEstimate`. `sdo.Experiment.getValuesToEstimate` returns a vector of all the model parameters and initial states that you want to estimate. You use this vector as an input to `sdo.optimize` to specify the parameters that you want to estimate.

**Default:** []

## Methods

<code>createSimulator</code>	Create simulation object from experiment to compare measured and simulated data
<code>getValuesToEstimate</code>	Get model initial states and parameters for estimation from experiment
<code>setEstimatedValues</code>	Update experiments with estimated model initial states and parameter values

## Copy Semantics

Value. To learn how value classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

### Specify Input and Output Data for Parameter Estimation

Load the measured experiment data.

```
load sdoBattery_ExperimentData
```

The variable `Charge_Data`, which contains the data measured during a battery charging experiment, is loaded into the MATLAB workspace. The first column contains time data. The second and third columns contain the current and voltage data.

Specify an experiment for a model.

```
modelName = 'sdoBattery';  
exp = sdo.Experiment(modelName);
```

Specify input data for the experiment.

```
exp.InputData = timeseries(Charge_Data(:,2),Charge_Data(:,1));
```

Specify output data for the experiment.

```
VoltageSig = Simulink.SimulationData.Signal;  
VoltageSig.Name = 'Voltage';  
VoltageSig.BlockPath = 'sdoBattery/SOC -> Voltage';  
VoltageSig.PortType = 'outport';  
VoltageSig.PortIndex = 1;  
VoltageSig.Values = timeseries(Charge_Data(:,3),Charge_Data(:,1));  
  
exp.OutputData = VoltageSig;
```

## Alternatives

“Run Estimation”

## See Also

[sdo.optimize](#) | [sdo.getStateFromModelparam.Continuous](#) | [param.State](#) |

## Related Examples

- “Estimate Model Parameter Values (Code)”
- “Estimate Model Parameters and Initial States (Code)”
- “Estimate Model Parameters using Multiple Experiments (Code)”
- “Estimate Model Parameters Per Experiment (Code)”
- “Estimate Model Parameters with Parameter Constraints (Code)”

## Concepts

- Class Attributes
- Property Attributes

# sdo.OptimizeOptions

---

<b>Purpose</b>	Optimization options
<b>Syntax</b>	<code>opt = sdo.OptimizeOptions</code> <code>opt = sdo.OptimizeOptions(Name,Value)</code>
<b>Description</b>	Specify options such as solver, solver options and whether to use parallel computing during optimization.
<b>Construction</b>	<code>opt = sdo.OptimizeOptions</code> creates an <code>sdo.OptimizeOptions</code> object and assigns default values to the properties.  <code>opt = sdo.OptimizeOptions(Name,Value)</code> creates an <code>sdo.OptimizeOptions</code> object with additional options specified by one or more <code>Name,Value</code> pair arguments. <code>Name</code> is a property name and <code>Value</code> is the corresponding value. <code>Name</code> must appear inside single quotes ( ' '). You can specify several name-value pair arguments in any order as <code>Name1,Value1,...,NameN,ValueN</code> .

## Input Arguments

### Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside single quotes ( ' '). You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

### GradFcn

String that indicates whether the cost/constraint function you provide to `sdo.optimize` returns gradient information:

- 'on' — The cost/constraint function returns gradient information
- 'off' — The cost/constraint function does not return gradient information. The software uses central differences to compute the gradients.

**Default:** 'off'

## Method

String specifying the optimization solver that `sdo.optimize` uses to solve the optimization problem:

- 'fmincon'
- 'fminsearch'
- 'lsqnonlin'
- 'patternsearch' (requires Global Optimization Toolbox software)

See the Optimization Toolbox and Global Optimization Toolbox documentation for more information on these solvers.

**Default:** 'fmincon'

## MethodOptions

Structure with fields specifying optimization solver options. The structure fields are configured based on the `Method` property.

You can change solver options. For example  
`opt.MethodOptions.TolX = 1.5e-3.`

For information on the optimization solver options, see:

- “Optimization Options” when `Method` is specified as 'fmincon', 'fminsearch', or 'lsqnonlin'
- `psoptimset` and “Pattern Search Options” when `Method` is specified as 'patternsearch'

**Default:** [1x1 struct]

## OptimizedModel

# sdo.OptimizeOptions

---

String displaying a Simulink model name to be optimized.

**Default:** ''

## **ParallelPathDependencies**

Cell array of strings specifying path dependencies to use during parallel optimization. These path dependencies are temporarily added to the workers during parallel optimization.

**Default:** {}

## **Restarts**

Nonnegative integer specifying the number of times the optimization solver restarts the optimization if convergence criteria are not satisfied. At each restart, the initial value of the tunable parameters are set to the final value of the previous optimization run.

**Default:** 0

## **StopIfFeasible**

Terminate optimization once a feasible solution satisfying the constraints is found:

- 'on' — Terminate as soon a feasible solution is found
- 'off' — Continue to search for solutions that are typically located further inside the constraint region

The software ignores this option when you track a reference signal or your problem has a cost.

**Default:** 'on'

## **UseParallel**

Parallel computing option for `fmincon`, `lsqnonlin` and `patternsearch` optimization solvers:

- 'never' — Do not use parallel computing during optimization
- 'always' — Use parallel computing during optimization

Parallel Computing Toolbox™ software must be installed to enable parallel computing for the optimization methods.

When set to 'always', the methods compute the following in parallel:

- `fmincon` — Finite difference gradients
- `lsqnonlin` — Finite difference gradients
- `patternsearch` — Population evaluation

**Default:** 'never'

## Properties

### GradFcn

String that indicates whether the cost/constraint function you provide to `sdo.optimize` returns gradient information:

- 'on' — The cost/constraint function returns gradient information
- 'off' — The cost/constraint function does not return gradient information. The software uses central differences to compute the gradients.

**Default:** 'off'

### Method

String specifying the optimization solver that `sdo.optimize` uses to solve the optimization problem:

# sdo.OptimizeOptions

---

- 'fmincon'
- 'fminsearch'
- 'lsqnonlin'
- 'patternsearch' (requires Global Optimization Toolbox software)

See the Optimization Toolbox and Global Optimization Toolbox documentation for more information on these solvers.

**Default:** 'fmincon'

## MethodOptions

Structure with fields specifying optimization solver options. The structure fields are configured based on the Method property.

You can change solver options. For example  
`opt.MethodOptions.TolX = 1.5e-3.`

For information on the optimization solver options, see:

- “Optimization Options” when Method is specified as 'fmincon', 'fminsearch', or 'lsqnonlin'
- `psoptimset` and “Pattern Search Options” when Method is specified as 'patternsearch'

**Default:** [1x1 struct]

## OptimizedModel

String displaying a Simulink model name to be optimized.

**Default:** ''

## ParallelPathDependencies



Cell array of strings specifying path dependencies to use during parallel optimization. These path dependencies are temporarily added to the workers during parallel optimization.

**Default:** {}

## **Restarts**

Nonnegative integer specifying the number of times the optimization solver restarts the optimization if convergence criteria are not satisfied. At each restart, the initial value of the tunable parameters are set to the final value of the previous optimization run.

**Default:** 0

## **StopIfFeasible**

Terminate optimization once a feasible solution satisfying the constraints is found:

- 'on' — Terminate as soon a feasible solution is found
- 'off' — Continue to search for solutions that are typically located further inside the constraint region

The software ignores this option when you track a reference signal or your problem has a cost.

**Default:** 'on'

## **UseParallel**

Parallel computing option for `fmincon`, `lsqnonlin` and `patternsearch` optimization solvers:

- 'never' — Do not use parallel computing during optimization
- 'always' — Use parallel computing during optimization

# sdo.OptimizeOptions

---

Parallel Computing Toolbox software must be installed to enable parallel computing for the optimization methods.

When set to 'always', the methods compute the following in parallel:

- `fmincon` — Finite difference gradients
- `lsqnonlin` — Finite difference gradients
- `patternsearch` — Population evaluation

**Default:** 'never'

## Copy Semantics

Value. To learn how value classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Specify an optimization solver.

```
opts = sdo.OptimizeOptions;  
opts.Method = 'fminsearch';
```

## See Also

`sdo.optimize` | `sdo.getModelDependencies`

## How To

- “Optimization Options”
- “Speedup Using Parallel Computing”

<b>Purpose</b>	Bode magnitude bound
<b>Syntax</b>	<pre>bode_req = sdo.requirements.BodeMagnitude bode_req = sdo.requirements.BodeMagnitude(Name, Value)</pre>
<b>Description</b>	<p>Specify frequency-dependent piecewise-linear upper and lower magnitude bounds on a linear system. You can then optimize your model to meet the requirements using <code>sdo.optimize</code>.</p> <p>You can specify upper or lower bounds, include multiple linear edges, and extend them to + or -infinity..</p> <p>You must have Simulink Control Design software to specify bode magnitude requirements.</p>
<b>Construction</b>	<p><code>bode_req = sdo.requirements.BodeMagnitude</code> creates an <code>sdo.requirements.BodeMagnitude</code> object and assigns default values to its properties.</p> <p><code>bode_req = sdo.requirements.BodeMagnitude(Name,Value)</code> uses additional options specified by one or more <code>Name,Value</code> pair arguments. <code>Name</code> is a property name and <code>Value</code> is the corresponding value. <code>Name</code> must appear inside single quotes (' '). You can specify several name-value pair arguments in any order as <code>Name1,Value1,...,NameN,ValueN</code>.</p> <p><b>Input Arguments</b></p> <p><b>Name-Value Pair Arguments</b></p> <p>Specify optional comma-separated pairs of <code>Name,Value</code> arguments. <code>Name</code> is the argument name and <code>Value</code> is the corresponding value. <code>Name</code> must appear inside single quotes (' '). You can specify several name and value pair arguments in any order as <code>Name1,Value1,...,NameN,ValueN</code>.</p> <p><b>BoundFrequencies</b></p> <p>Frequency values for the gain bound.</p>

# sdo.requirements.BodeMagnitude

---

Specify the start and end frequencies for all the edges in the piecewise-linear bound. The property must be a  $n \times 2$  array of finite doubles, where each row specifies the start and end frequencies of an edge in the piecewise-linear bound. The start and end frequencies must define a positive length. The number of rows must match the number of rows of the `BoundMagnitudes` property.

Use `set` to set this and the `BoundMagnitudes` properties simultaneously.

Use the `FrequencyUnits` property to specify the frequency units.

**Default:** [1 10]

## **BoundMagnitudes**

Magnitude values for the gain bound.

Specify the start and end gain values for all the edges in the piecewise-linear bound. The property must be a  $n \times 2$  array of finite doubles where each row specifies the start and end gains of an edge in the piecewise-linear bound. The number of rows must match the number of rows of the `BoundFrequencies` property.

Use `set` to set this and the `BoundFrequencies` properties simultaneously.

Use the `MagnitudeUnits` property to specify the magnitude units.

**Default:** [0 0]

## **Description**

Requirement description. Must be a string.

**Default:** ''

## **FrequencyScale**

Frequency-axis scaling.

Use this property to determine the value of the bound between edge start and end points. Must be one of the following strings:

- 'linear'
- 'log'

For example, if bound edges are at frequencies  $f_1$  and  $f_2$ , and the bound is to be evaluated at  $f_3$ , the edges are interpolated as a straight lines. The x-axis is either linear or logarithmic.

**Default:** 'log'

## **FrequencyUnits**

Frequency units of the requirement. Must be one of the following strings:

- 'Hz'
- 'rad/s'
- 'rpm'
- 'kHz'
- 'MHz'
- 'GHz'
- 'rad/nanosecond'
- 'rad/microsecond'
- 'rad/millisecond'
- 'rad/minute'
- 'rad/hour'
- 'rad/day'

# sdo.requirements.BodeMagnitude

---

- 'rad/week'
- 'rad/month'
- 'rad/year'
- 'cycles/nanosecond'
- 'cycles/microsecond'
- 'cycles/millisecond'
- 'cycles/hour'
- 'cycles/day'
- 'cycles/week'
- 'cycles/month'
- 'cycles/year'

**Default:** 'rad/s'

## **MagnitudeUnits**

Magnitude units of the requirement. Must be:

- 'db' (decibels)
- 'abs' (absolute units)

**Default:** 'db'

## **Name**

Requirement name. Must be a string.

**Default:** ''

## **OpenEnd**

Extend bound in a negative or positive frequency direction.

Specify whether the first and last edge of the bound extends to `inf` and `+inf` respectively. Use to bound signals that extend beyond the frequency values specified by the `BoundFrequencies` property.

Must be a 1x2 logical array of `true` or `false`. If `true`, the first or last edge of the piecewise linear bound is extended in the negative or positive direction.

**Default:** [0 0]

## Type

Magnitude bound type. Must be:

- '`<=`' — Upper bound
- '`>=`' — Lower bound

Use to specify whether the piecewise-linear bound is an upper or lower bound. Use for upper bound and for lower bound.

## Properties

### BoundFrequencies

Frequency values for the gain bound.

Specify the start and end frequencies for all the edges in the piecewise-linear bound. The property must be a `nx2` array of finite doubles, where each row specifies the start and end frequencies of an edge in the piecewise-linear bound. The start and end frequencies must define a positive length. The number of rows must match the number of rows of the `BoundMagnitudes` property.

Use `set` to set this and the `BoundMagnitudes` properties simultaneously.

Use the `FrequencyUnits` property to specify the frequency units.

**Default:** [1 10]

# sdo.requirements.BodeMagnitude

---

## BoundMagnitudes

Magnitude values for the gain bound.

Specify the start and end gain values for all the edges in the piecewise-linear bound. The property must be a  $n \times 2$  array of finite doubles where each row specifies the start and end gains of an edge in the piecewise-linear bound. The number of rows must match the number of rows of the `BoundFrequencies` property.

Use `set` to set this and the `BoundFrequencies` properties simultaneously.

Use the `MagnitudeUnits` property to specify the magnitude units.

**Default:** [0 0]

## Description

Requirement description. Must be a string.

**Default:** ''

## FrequencyScale

Frequency-axis scaling.

Use this property to determine the value of the bound between edge start and end points. Must be one of the following strings:

- 'linear'
- 'log'

For example, if bound edges are at frequencies  $f_1$  and  $f_2$ , and the bound is to be evaluated at  $f_3$ , the edges are interpolated as a straight lines. The x-axis is either linear or logarithmic.

**Default:** 'log'



## FrequencyUnits

Frequency units of the requirement. Must be one of the following strings:

- 'Hz'
- 'rad/s'
- 'rpm'
- 'kHz'
- 'MHz'
- 'GHz'
- 'rad/nanosecond'
- 'rad/microsecond'
- 'rad/millisecond'
- 'rad/minute'
- 'rad/hour'
- 'rad/day'
- 'rad/week'
- 'rad/month'
- 'rad/year'
- 'cycles/nanosecond'
- 'cycles/microsecond'
- 'cycles/millisecond'
- 'cycles/hour'
- 'cycles/day'
- 'cycles/week'
- 'cycles/month'

# sdo.requirements.BodeMagnitude

---

- 'cycles/year'

**Default:** 'rad/s'

## **MagnitudeUnits**

Magnitude units of the requirement. Must be:

- 'db' (decibels)
- 'abs' (absolute units)

**Default:** 'db'

## **Name**

Requirement name. Must be a string.

**Default:** ''

## **OpenEnd**

Extend bound in a negative or positive frequency direction.

Specify whether the first and last edge of the bound extends to `inf` and `+inf` respectively. Use to bound signals that extend beyond the frequency values specified by the `BoundFrequencies` property.

Must be a 1x2 logical array of `true` or `false`. If `true`, the first or last edge of the piecewise linear bound is extended in the negative or positive direction.

**Default:** [0 0]

## **Type**

Magnitude bound type. Must be:

- '<=' — Upper bound

- '>=' — Lower bound

Use to specify whether the piecewise-linear bound is an upper or lower bound. Use for upper bound and for lower bound.

## Methods

`evalRequirement` Evaluate Bode magnitude bound for linear system

## Copy Semantics

Handle. To learn how handle classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Construct a Bode magnitude requirements object and specify bound frequencies and magnitudes.

```
r = sdo.requirements.BodeMagnitude;  
set(r,'BoundFrequencies', [0.1 10; 10 100],...  
'BoundMagnitudes',[1 1; 0.1 0.1])
```

Alternatively, you can specify the frequency and magnitude during construction.

```
r = sdo.requirements.BodeMagnitude(...  
'BoundFrequencies', [1 10; 10 100], ...  
'BoundMagnitudes', [1 1; 1 0]);
```

## Alternatives

Use `getbounds` to get the bounds specified in a Check Bode Characteristics block.

## See Also

`copy` | `get` | `set`

## How To

- Class Attributes
- Property Attributes

# sdo.requirements.ClosedLoopPeakGain

---

<b>Purpose</b>	Closed loop peak gain bound
<b>Description</b>	<p>Specify lower or equality bounds on the closed loop peak gain of a linear system. The closed loop can be formed using negative, positive or no feedback. You can then optimize the model response to meet these bounds using <code>sdo.optimize</code>.</p> <p>You must have Simulink Control Design software to specify closed-loop peak gain bounds.</p>
<b>Construction</b>	<p><code>pkgain_req = sdo.requirements.ClosedLoopPeakGain</code> creates a <code>sdo.requirements.ClosedLoopPeakGain</code> object and assigns default values to its properties.</p> <pre>pkgain_req = sdo.requirements.ClosedLoopPeakGain(Name,Value) uses additional options specified by one or more Name,Value pair arguments. Name is a property name and Value is the corresponding value. Name must appear inside single quotes (' '). You can specify several name-value pair arguments in any order as Name1,Value1,...,NameN,ValueN.</pre> <p><b>Input Arguments</b></p> <p><b>Name-Value Pair Arguments</b></p> <p>Specify optional comma-separated pairs of <code>Name,Value</code> arguments. <code>Name</code> is the argument name and <code>Value</code> is the corresponding value. <code>Name</code> must appear inside single quotes (' '). You can specify several name and value pair arguments in any order as <code>Name1,Value1,...,NameN,ValueN</code>.</p> <p><b>Description</b></p> <p>Requirement description. Must be a string.</p> <p><b>Default:</b> ''</p> <p><b>FeedbackSign</b></p>

Feedback loop sign to determine the peak gain of the linear system.

Must be -1 or 1. Use -1 if the loop has negative feedback and 1 if the loop has positive feedback.

**Default:** 1

## **MagnitudeUnits**

Magnitude units of the requirement.

Must be 'db' (decibels) or 'abs' (absolute units).

**Default:** 'abs'

## **Name**

Requirement name. Must be a string.

**Default:** ''

## **PeakGain**

Peak gain bound.

**Default:** 2

## **Type**

Peak gain requirement type. Must be one of the following strings:

- '<=' — Upper bound
- '==' — Equality bound
- 'min' — Minimization objective

**Default:** '<='

# sdo.requirements.ClosedLoopPeakGain

---

## Properties

### Description

Requirement description. Must be a string.

**Default:** ''

### FeedbackSign

Feedback loop sign to determine the peak gain of the linear system.

Must be -1 or 1. Use -1 if the loop has negative feedback and 1 if the loop has positive feedback.

**Default:** 1

### MagnitudeUnits

Magnitude units of the requirement.

Must be 'db' (decibels) or 'abs' (absolute units).

**Default:** 'abs'

### Name

Requirement name. Must be a string.

**Default:** ''

### PeakGain

Peak gain bound.

**Default:** 2

### Type

Peak gain requirement type. Must be one of the following strings:

- '<=' — Upper bound

- '==' — Equality bound
- 'min' — Minimization objective

**Default:** '<='

## Methods

`evalRequirement` Evaluate peak gain bound for linear system

## Copy Semantics

Handle. To learn how handle classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Construct a closed loop peak gain object and specify peak gain requirement.

```
r = sdo.requirements.ClosedLoopPeakGain;  
r.PeakGain = 2;
```

Alternatively, you can specify the peak gain during construction:

```
r = sdo.requirements.ClosedLoopPeakGain('PeakGain',2);
```

## Alternatives

Use `getbounds` to get the bounds specified in Check Nichols Characteristics block.

## See Also

`copy` | `get` | `set`

## How To

- Class Attributes
- Property Attributes

# sdo.requirements.GainPhaseMargin

---

**Purpose** Gain and phase margin bounds

**Description** Specify lower or equality bounds on the gain and phase margin of a linear system. You can then optimize the model response to meet the bounds using `sdo.optimize`.

You must have Simulink Control Design software to specify gain and phase margin requirements.

**Construction** `gainphase_req = sdo.requirements.GainPhaseMargin` creates a `sdo.requirements.GainPhaseMargin` object and assigns default values to its properties.

```
gainphase_req =  
sdo.requirements.GainPhaseMargin(Name,Value) uses additional  
options specified by one or more Name,Value pair arguments. Name is a  
property name and Value is the corresponding value. Name must appear  
inside single quotes ( ' '). You can specify several name-value pair  
arguments in any order as Name1,Value1,...,NameN,ValueN.
```

## Input Arguments

### Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside single quotes ( ' '). You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

### Description

Requirement description. Must be a string.

**Default:** ' '

### FeedbackSign

Feedback loop sign to determine the gain and phase margins of the linear system.



Must be -1 or 1. Use -1 if the loop has negative feedback and 1 if the loop has positive feedback.

**Default:** 1

## **GainMargin**

Gain margin bound. Use `MagnitudeUnits` to specify the gain units. Set to `[]` to specify a bound on the phase margin only.

**Default:** 10

## **MagnitudeUnits**

Magnitude units of the requirement. Must be:

- 'db' (decibels)
- 'abs' (absolute units)

**Default:** 'db'

## **Name**

Requirement name. Must be a string.

**Default:** ''

## **PhaseMargin**

Phase margin bound. Must be in degrees and a positive finite scalar. Set to `[]` to specify a bound on the gain margin only.

**Default:** 60

## **PhaseUnits**

Phase units of the requirement. Must be one of the following strings:

# sdo.requirements.GainPhaseMargin

---

- 'deg' (degrees)
- 'rad' (radians)

**Default:** 'deg'

## Type

Gain and phase margin requirement type. Must be one of the following strings:

- '>=' — Lower bound
- '==' — Equality bound
- 'max' — Maximization objective

**Default:** '>='

## Properties

### Description

Requirement description. Must be a string.

**Default:** ''

### FeedbackSign

Feedback loop sign to determine the gain and phase margins of the linear system.

Must be -1 or 1. Use -1 if the loop has negative feedback and 1 if the loop has positive feedback.

**Default:** -1

### GainMargin

Gain margin bound. Use `MagnitudeUnits` to specify the gain units. Set to `[]` to specify a bound on the phase margin only.

**Default:** 10

## **MagnitudeUnits**

Magnitude units of the requirement. Must be:

- 'db' (decibels)
- 'abs' (absolute units)

**Default:** 'db'

## **Name**

Requirement name. Must be a string.

**Default:** ''

## **PhaseMargin**

Phase margin bound. Must be in degrees and a positive finite scalar. Set to [] to specify a bound on the gain margin only.

**Default:** 60

## **PhaseUnits**

Phase units of the requirement. Must be one of the following strings:

- 'deg' (degrees)
- 'rad' (radians)

**Default:** 'deg'

## **Type**

Gain and phase margin requirement type. Must be one of the following strings:

- '>=' — Lower bound

# sdo.requirements.GainPhaseMargin

---

- '==' — Equality bound
- 'max' — Maximization objective

**Default:** '>='

## Methods

`evalRequirement` Evaluate gain and phase margin bounds for linear system

## Copy Semantics

Handle. To learn how handle classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Construct a gain and phase margin object and specify gain and phase margin requirement.

```
r = sdo.requirements.GainPhaseMargin;  
r.GainMargin = 5;  
r.PhaseMargin = 55;
```

Alternatively, you can specify the gain and phase margins during construction.

```
r = sdo.requirements.GainPhaseMargin(...  
    'GainMargin',5, ...  
    'PhaseMargin', 55);
```

## Alternatives

Use `getbounds` to get the bounds specified in a Check Gain and Phase Margin Characteristics and Check Nichols Characteristics block.

## See Also

`copy` | `get` | `set`

## How To

- Class Attributes
- Property Attributes

## Purpose

Nichols response bound

## Description

Specify piecewise-linear bounds on the Nichols (gain-phase) response of a linear system. You can then optimize the model response to meet these bounds using `sdo.optimize`.

You can specify an upper or lower bound, include multiple linear edges, and extend the bounds to `+` or `inf`.

You must have Simulink Control Design software to specify open-loop gain and phase requirements.

## Construction

`olgainphase_req = sdo.requirements.OpenLoopGainPhase` creates a `sdo.requirements.OpenLoopGainPhase` object and assigns default values to its properties.

```
gainphase_req =  
sdo.requirements.OpenLoopGainPhase(Name,Value) uses additional  
options specified by one or more Name,Value pair arguments. Name is a  
property name and Value is the corresponding value. Name must appear  
inside single quotes ( ' '). You can specify several name-value pair  
arguments in any order as Name1,Value1,...,NameN,ValueN.
```

## Input Arguments

### Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside single quotes ( ' '). You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

### BoundGains

Gain values for a piecewise linear bound.

Specify the start and end values in decibels for all the edges in the piecewise-linear bound. The property must be a `nx2` array of finite doubles, where each row specifies the start and end gain values

## sdo.requirements.OpenLoopGainPhase

---

of an edge. The number of rows must match the number of rows of the `BoundPhases` property.

Use `set` to set this and the `BoundPhases` properties simultaneously.

**Default:** [ -10 -10]

### **BoundPhases**

Phase values for a piecewise-linear bound.

Specify the start and end values in degrees for all the edges in the piecewise-linear bound. The property must be a  $n \times 2$  array of finite doubles, where each row specifies the start and end phase values of an edge. The number of rows must match the number of rows of the `BoundGains` property.

Use `set` to set this and the `BoundGains` properties simultaneously.

**Default:** [ -180 -90]

### **Description**

Requirement description. Must be a string.

**Default:** ''

### **MagnitudeUnits**

Magnitude units of the requirement. Must be:

- 'db' (decibels)
- 'abs' (absolute units)

**Default:** 'db'

### **Name**

Requirement name. Must be a string.

**Default:** ''

## **OpenEnd**

Extend bound in a negative or positive time direction.

Use to bound signals that extend beyond the coordinates specified by the **BoundPhases** and **BoundGains** properties.

Must be a 1x2 logical array. If **true**, the first or last edge of the bound is extended to infinity.

**Default:** [0 0]

## **PhaseUnits**

Phase units of the requirement. Must be one of the following strings:

- 'deg' (degrees)
- 'rad' (radians)

**Default:** 'deg'

## **Type**

Gain and phase requirement type. Must be one of the following strings:

- '>=' — Lower bound
- '<=' — Upper bound

**Default:** '>='

## **Properties**

### **BoundGains**

Gain values for a piecewise linear bound.

# sdo.requirements.OpenLoopGainPhase

---

Specify the start and end values in decibels for all the edges in the piecewise-linear bound. The property must be a  $n \times 2$  array of finite doubles, where each row specifies the start and end gain values of an edge. The number of rows must match the number of rows of the `BoundPhases` property.

Use `set` to set this and the `BoundPhases` properties simultaneously.

**Default:** [ -10 -10]

## **BoundPhases**

Phase values for a piecewise-linear bound.

Specify the start and end values in degrees for all the edges in the piecewise-linear bound. The property must be a  $n \times 2$  array of finite doubles, where each row specifies the start and end phase values of an edge. The number of rows must match the number of rows of the `BoundGains` property.

Use `set` to set this and the `BoundGains` properties simultaneously.

**Default:** [ -180 -90]

## **Description**

Requirement description. Must be a string.

**Default:** ''

## **MagnitudeUnits**

Magnitude units of the requirement. Must be:

- 'db' (decibels)
- 'abs' (absolute units)



**Default:** 'db'

## **Name**

Requirement name. Must be a string.

**Default:** ''

## **OpenEnd**

Extend bound in a negative or positive time direction.

Use to bound signals that extend beyond the coordinates specified by the **BoundPhases** and **BoundGains** properties.

Must be a 1x2 logical array. If **true**, the first or last edge of the bound is extended to infinity.

**Default:** [0 0]

## **PhaseUnits**

Phase units of the requirement. Must be one of the following strings:

- 'deg' (degrees)
- 'rad' (radians)

**Default:** 'deg'

## **Type**

Gain and phase requirement type. Must be one of the following strings:

- '>=' — Lower bound
- '<=' — Upper bound

**Default:** '>='

# sdo.requirements.OpenLoopGainPhase

---

## Methods

`evalRequirement` Evaluate gain and phase bounds on Nichols response of linear system

## Copy Semantics

Handle. To learn how handle classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Construct an open-loop gain and phase object, and specify gain and phase requirements.

```
r = sdo.requirements.OpenLoopGainPhase;  
set(r,'BoundPhases',[-120 -120; -120 -150; -150 -180],...  
    'BoundGains',[20 0; 0 -20; -20 -20]);
```

Alternatively, you can specify the gain and phase requirements during construction:

```
r = sdo.requirements.OpenLoopGainPhase('BoundPhases',...  
    [-120 -120; -120 -150; -150 -180],'BoundGains',...  
    [20 0; 0 -20; -20 -20]);
```

## Alternatives

Use `getbounds` to get the bounds specified in a Check Nichols Characteristics block.

## See Also

`copy` | `get` | `set`

## How To

- Class Attributes
- Property Attributes

**Purpose** Damping ratio bound

**Description** Specify bounds on the damping ratio of the poles of a linear system. You can then optimize the model response to meet these bounds using `sdo.optimize`. You can also use this object to specify overshoot bound. You must have Simulink Control Design software to specify damping ratio requirements.

**Construction** `damp_req = sdo.requirements.PZDampingRatio` creates a `sdo.requirements.PZDampingRatio` object and assigns default values to its properties.

`gainphase_req = sdo.requirements.PZDampingRatio(Name,Value)` uses additional options specified by one or more `Name,Value` pair arguments. `Name` is a property name and `Value` is the corresponding value. `Name` must appear inside single quotes (' '). You can specify several name-value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

## Input Arguments

### Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside single quotes (' '). You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

### DampingRatio

Damping ratio bound. Must be a finite scalar between 0 and 1.

**Default:** 0.7071

### Description

Requirement description. Must be a string.

**Default:** ''

# sdo.requirements.PZDampingRatio

---

**Name**

Requirement name. Must be a string.

**Default:** ''

**Type**

Damping ratio bound type. Must be one of the following strings:

- '<=' — Upper bound
- '>=' — Lower bound
- '==' — Equality bound
- 'max' — Maximization objective

**Default:** '>='

**Properties****DampingRatio**

Damping ratio bound. Must be a finite scalar between 0 and 1.

**Default:** 0.7071

**Description**

Requirement description. Must be a string.

**Default:** ''

**Name**

Requirement name. Must be a string.

**Default:** ''

**Type**

Damping ratio bound type. Must be one of the following strings:

- '<=' — Upper bound
- '>=' — Lower bound
- '==' — Equality bound
- 'max' — Maximization objective

**Default:** '>='

## Methods

evalRequirement	Evaluate damping ratio bound on linear system
getOvershoot	Convert damping ratio to equivalent overshoot value
setOvershoot	Set overshoot to an equivalent damping ratio

## Copy Semantics

Handle. To learn how handle classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Construct a damping ratio object and specify the damping ratio.

```
r = sdo.requirements.PZDampingRatio;  
r.DampingRatio = 0.1;
```

Alternatively, you can specify the damping ratio during construction.

```
r = sdo.requirements.PZDampingRatio('DampingRatio',0.1);
```

## Alternatives

Use `getbounds` to get the bounds specified in a Check Pole-Zero Characteristics block.

## See Also

`copy` | `get` | `set`

# sdo.requirements.PZDampingRatio

---

## How To

- Class Attributes
- Property Attributes

<b>Purpose</b>	Natural frequency bound
<b>Description</b>	<p>Specify bounds on the natural frequency of the poles of a linear system. You can then optimize the model response to meet these bounds using <code>sdo.optimize</code>.</p> <p>You must have Simulink Control Design software to specify natural frequency requirements.</p>
<b>Construction</b>	<p><code>pznatfreq_req = sdo.requirements.PZNaturalFrequency</code> creates a <code>sdo.requirements.PZNaturalFrequency</code> object and assigns default values to its properties.</p> <p><code>pznatfreq_req = sdo.requirements.pznatfreq_req(Name,Value)</code> uses additional options specified by one or more <code>Name,Value</code> pair arguments. <code>Name</code> is a property name and <code>Value</code> is the corresponding value. <code>Name</code> must appear inside single quotes ( <code>'</code> ). You can specify several name-value pair arguments in any order as <code>Name1,Value1,...,NameN,ValueN</code>.</p>
	<b>Input Arguments</b>
	<b>Name-Value Pair Arguments</b>
	<p>Specify optional comma-separated pairs of <code>Name,Value</code> arguments. <code>Name</code> is the argument name and <code>Value</code> is the corresponding value. <code>Name</code> must appear inside single quotes ( <code>'</code> ). You can specify several name and value pair arguments in any order as <code>Name1,Value1,...,NameN,ValueN</code>.</p>
	<b>Description</b>
	<p>Requirement description. Must be a string.</p>
	<b>Default:</b> <code>'</code>
	<b>FrequencyUnits</b>
	<p>Frequency units of the requirement. Must be one of the following strings:</p>

## sdo.requirements.PZNaturalFrequency

---

- 'Hz'
- 'rad/s'
- 'rpm'
- 'kHz'
- 'MHz'
- 'GHz'
- 'rad/nanosecond'
- 'rad/microsecond'
- 'rad/millisecond'
- 'rad/minute'
- 'rad/hour'
- 'rad/day'
- 'rad/week'
- 'rad/month'
- 'rad/year'
- 'cycles/nanosecond'
- 'cycles/microsecond'
- 'cycles/millisecond'
- 'cycles/hour'
- 'cycles/day'
- 'cycles/week'
- 'cycles/month'
- 'cycles/year'

**Default:** 'rad/s'



**Name**

Requirement name. Must be a string.

**Default:** ''

**NaturalFrequency**

Natural frequency bound. Must be in radians/second and a positive finite scalar.

**Default:** 2

**Type**

Natural frequency bound type. Must be one of the following strings:

- '<=' — Upper bound
- '>=' — Lower bound
- '==' — Equality bound
- 'max' — Maximization objective

**Default:** '>='

**Properties****Description**

Requirement description. Must be a string.

**Default:** ''

**FrequencyUnits**

Frequency units of the requirement. Must be one of the following strings:

- 'Hz'

# sdo.requirements.PZNaturalFrequency

---

- 'rad/s'
- 'rpm'
- 'kHz'
- 'MHz'
- 'GHz'
- 'rad/nanosecond'
- 'rad/microsecond'
- 'rad/millisecond'
- 'rad/minute'
- 'rad/hour'
- 'rad/day'
- 'rad/week'
- 'rad/month'
- 'rad/year'
- 'cycles/nanosecond'
- 'cycles/microsecond'
- 'cycles/millisecond'
- 'cycles/hour'
- 'cycles/day'
- 'cycles/week'
- 'cycles/month'
- 'cycles/year'

**Default:** 'rad/s'

**Name**

# sdo.requirements.PZNaturalFrequency

---

Requirement name. Must be a string.

**Default:** ''

## NaturalFrequency

Natural frequency bound. Must be in radians/second and a positive finite scalar.

**Default:** 2

## Type

Natural frequency bound type. Must be one of the following strings:

- '<=' — Upper bound
- '>=' — Lower bound
- '==' — Equality bound
- 'max' — Maximization objective

**Default:** '>='

## Methods

evalRequirement	Evaluate natural frequency bound on linear system
-----------------	---

## Copy Semantics

Handle. To learn how handle classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Construct a natural frequency object and specify the natural frequency.

```
r = sdo.requirements.PZNaturalFrequency;  
r.NaturalFrequency = 1;
```

Alternatively, you can specify the natural frequency during construction.

# sdo.requirements.PZNaturalFrequency

---

```
r = sdo.requirements.PZNaturalFrequency(...  
    'NaturalFrequency',1);
```

**Alternatives** Use `getbounds` to get the bounds specified in Check Pole-Zero Characteristics block.

**See Also** `copy` | `get` | `set`

**How To**

- Class Attributes
- Property Attributes

## Purpose

Settling time bound

## Description

Specify bounds on the real component of the poles of a linear system. The real component of poles are used to approximate the settling time. You can then optimize the model response to meet these bounds using `sdo.optimize`.

You must have Simulink Control Design software to specify settling time requirements.

## Construction

`settime_req = sdo.requirements.PZSettlingTime` creates a `sdo.requirements.PZSettlingTime` object and assigns default values to its properties.

`settime_req = sdo.requirements.PZSettlingTime(Name,Value)` uses additional options specified by one or more `Name,Value` pair arguments. `Name` is a property name and `Value` is the corresponding value. `Name` must appear inside single quotes (' '). You can specify several name-value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

## Input Arguments

### Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside single quotes (' '). You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

### Description

Requirement description. Must be a string.

**Default:** ''

### Name

Requirement name. Must be a string.

# sdo.requirements.PZSettlingTime

---

**Default:** ''

## **SettlingTime**

Settling time bound. Must be in seconds and a positive finite scalar.

**Default:** 2

## **TimeUnits**

Time units of the requirement. Must be one of the following strings:

- 'nanoseconds'
- 'microseconds'
- 'milliseconds'
- 'seconds'
- 'minutes'
- 'hours'
- 'days'
- 'weeks'
- 'months'
- 'years'

**Default:** 'second'

## **Type**

Settling time bound type. Must be one of the following strings:

- '<=' — Upper bound
- '>=' — Lower bound

- '=' — Equality bound
- 'min' — Minimization objective

**Default:** '<='

## Properties

### Description

Requirement description. Must be a string.

**Default:** ''

### Name

Requirement name. Must be a string.

**Default:** ''

### SettlingTime

Settling time bound. Must be in seconds and a positive finite scalar.

**Default:** 2

### TimeUnits

Time units of the requirement. Must be one of the following strings:

- 'nanoseconds'
- 'microseconds'
- 'milliseconds'
- 'seconds'
- 'minutes'
- 'hours'

# sdo.requirements.PZSettlingTime

---

- 'days'
- 'weeks'
- 'months'
- 'years'

**Default:** 'second'

## Type

Settling time bound type. Must be one of the following strings:

- '<=' — Upper bound
- '>=' — Lower bound
- '==' — Equality bound
- 'min' — Minimization objective

**Default:** '<='

## Methods

evalRequirement	Evaluate settling time bound on linear system
-----------------	---

## Copy Semantics

Handle. To learn how handle classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Construct a settling time object and specify the settling time requirement.

```
r = sdo.requirements.PZSettlingTime;  
r.SettlingTime = 2.5;
```

Alternatively, you can specify the setting time during construction.

```
r = sdo.requirements.PZSettlingTime('SettlingTime',2.5);
```



**Alternatives** Use getbounds to get the bounds specified in Check Pole-Zero Characteristics block.

**See Also** copy | get | set

**How To**

- Class Attributes
- Property Attributes

# sdo.requirements.SignalBound

---

**Purpose** Piecewise-linear amplitude bound

**Description** Specify piecewise-linear upper or lower amplitude bounds on a time-domain signal. You can then optimize the model response to meet these bounds using `sdo.optimize`.

You can include multiple linear edges, and extend to `+` or `inf`.

**Construction** `sig_req = sdo.requirements.SignalBound` creates an `sdo.requirements.SignalBound` object and assigns default values to its properties.

`sig_req = sdo.requirements.SignalBound(Name,Value)` uses additional options specified by one or more `Name,Value` pair arguments. `Name` is a property name and `Value` is the corresponding value. `Name` must appear inside single quotes (`' '`). You can specify several name-value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

## Input Arguments

### Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside single quotes (`' '`). You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

### BoundMagnitudes

Magnitude values for the piecewise-linear bound.

Specify the start and end magnitude values for all edges in the bound. The property must be a `nx2` array of finite doubles, where each row specifies the start and end magnitude values of an edge. The number of rows must match the number of rows of the `BoundTimes` property.

Use `set` to set this and `BoundTimes` properties simultaneously.

**Default:** [1 1]

## **BoundTimes**

Time values of the piecewise-linear bound.

Specify the start and end times for all the edges in the piecewise-linear bound. The property must be a  $n \times 2$  array of finite doubles where each row specifies the start and end times of an edge. The start and end times must define a positive length. The number of rows must match the number of rows of the `BoundMagnitudes` property.

Use `set` to set this and `BoundMagnitudes` properties simultaneously.

**Default:** [0 10]

## **Description**

Requirement description. Must be a string.

**Default:** ''

## **Name**

Requirement name. Must be a string.

**Default:** ''

## **OpenEnd**

Extend bound in a negative or positive time direction.

Specify whether the first and last edge of the bound extends to `inf` and `+inf` respectively. Use to bound signals that extend beyond the time values specified by the `BoundTimes` property.

Must be a  $1 \times 2$  logical array. If `true`, the first or last edge of the bound is extended in a negative or positive direction, respectively.

# sdo.requirements.SignalBound

---

**Default:** [0 0]

## **TimeUnits**

Time units of the requirement. Must be one of the following strings:

- 'nanoseconds'
- 'microseconds'
- 'milliseconds'
- 'seconds'
- 'minutes'
- 'hours'
- 'days'
- 'weeks'
- 'months'
- 'years'

**Default:** 'second'

## **Type**

Bound type.

Specify whether the piecewise-linear requirement is an upper or lower bound. Must be one of the following strings:

- '<=' — Upper bound
- '>=' — Lower bound

**Default:** '<='

## Properties

### BoundMagnitudes

Magnitude values for the piecewise-linear bound.

Specify the start and end magnitude values for all edges in the bound. The property must be a  $n \times 2$  array of finite doubles, where each row specifies the start and end magnitude values of an edge. The number of rows must match the number of rows of the `BoundTimes` property.

Use `set` to set this and `BoundTimes` properties simultaneously.

**Default:** [1 1]

### BoundTimes

Time values of the piecewise-linear bound.

Specify the start and end times for all the edges in the piecewise-linear bound. The property must be a  $n \times 2$  array of finite doubles where each row specifies the start and end times of an edge. The start and end times must define a positive length. The number of rows must match the number of rows of the `BoundMagnitudes` property.

Use `set` to set this and `BoundMagnitudes` properties simultaneously.

**Default:** [0 10]

### Description

Requirement description. Must be a string.

**Default:** ''

### Name

Requirement name. Must be a string.

**Default:** ''

# sdo.requirements.SignalBound

---

## OpenEnd

Extend bound in a negative or positive time direction.

Specify whether the first and last edge of the bound extends to `-inf` and `+inf` respectively. Use to bound signals that extend beyond the time values specified by the `BoundTimes` property.

Must be a 1x2 logical array. If `true`, the first or last edge of the bound is extended in a negative or positive direction, respectively.

**Default:** [0 0]

## TimeUnits

Time units of the requirement. Must be one of the following strings:

- 'nanoseconds'
- 'microseconds'
- 'milliseconds'
- 'seconds'
- 'minutes'
- 'hours'
- 'days'
- 'weeks'
- 'months'
- 'years'

**Default:** 'second'

## Type

Bound type.

Specify whether the piecewise-linear requirement is an upper or lower bound. Must be one of the following strings:

- '<=' — Upper bound
- '>=' — Lower bound

**Default:** '<='

## Methods

`evalRequirement` Evaluate piecewise-linear bound

## Examples

Construct a signal bound object and specify piecewise-linear bounds.

```
r = sdo.requirements.SignalBound;  
set(r, 'BoundTimes', [0 10; 10 20], ...  
     'BoundMagnitudes', [1.1 1.1; 1.01 1.01])
```

Alternatively, you can specify the bounds during construction:

```
r = sdo.requirements.SignalBound(...  
     'BoundTimes', [0 10; 10 20], ...  
     'BoundMagnitudes', [1.1 1.1; 1.01 1.01]);
```

## Alternatives

Use `getbounds` to get the bounds specified in a Check Custom Bounds block.

## See Also

`copy` | `get` | `set`

## How To

- Class Attributes
- Property Attributes

# sdo.requirements.SignalTracking

---

**Purpose** Reference signal to track

**Description** Specify a tracking requirement on a time-domain signal. You can then optimize the model response to track the reference using `sdo.optimize`. You can specify an equality, upper or lower bound requirement.

**Construction** `track_req = sdo.requirements.SignalTracking` creates an `sdo.requirements.SignalTracking` object and assigns default values to its properties.

`track_req = sdo.requirements.SignalTracking(Name,Value)` uses additional options specified by one or more `Name,Value` pair arguments. `Name` is a property name and `Value` is the corresponding value. `Name` must appear inside single quotes (' '). You can specify several name-value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

## Input Arguments

### Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside single quotes (' '). You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

### AbsTol

Absolute tolerance used to determine bounds as the signal approaches the reference signal. The bounds on the reference signal are given by:

$$y_u = (1 + RelTol)y_r + AbsTol$$

$$y_l = (1 - RelTol)y_r - AbsTol$$

where  $y_r$  is the value of the reference at a certain time,  $y_u$  and  $y_l$  are the upper and lower tolerance bounds corresponding to that time point.



**Default:** 0

## **Description**

Requirement description. Must be a string.

**Default:** ''

## **InterpolationTimes**

Time points to use when comparing reference and testpoint signals. Linear interpolation is used to compare the signals at the same timepoints.

Must be one of the following strings:

- 'Reference only' — Compare the signals at the time points of the reference signal only
- 'Testpoint only' — Compare the signals at the time points of the testpoint signal only
- 'Reference and Testpoint' — Compare the signals at the time points of both the reference and testpoint signals

**Default:** 'Reference only'

## **Method**

Algorithm for evaluating the requirement when the **Type** property is '=='.

When the requirement is evaluated using `evalRequirement`, the software computes the error between the reference and testpoint signals. This property specifies how the error signal  $e(t) = y_s(t) - y_r(t)$  should be processed.

Must be one of the following strings:

- 'SSE'

# sdo.requirements.SignalTracking

---

- 'SAE'
- 'Residuals'

**Default:** 'SSE'

## **Name**

Requirement name. Must be a string.

**Default:** ''

## **Normalize**

Enable or disable normalization when evaluating the requirement. The maximum absolute value of the reference signal is used for normalization. Must be 'on' or 'off'.

**Default:** 'on'

## **ReferenceSignal**

Reference signal to track. Must be a MATLAB `timeseries` object with real finite data points.

**Default:** [1x1 `timeseries`]

## **RelTol**

Relative tolerance used to determine bounds as the signal approaches the reference signal. The bounds on the reference signal are given by:

$$y_u = (1 + RelTol)y_r + AbsTol$$

$$y_l = (1 - RelTol)y_r - AbsTol$$

**Default:** 0

## **Type**

Tracking requirement type. Must be one of the following strings:

- '==' — Tracking objective.
- '<=' — Upper bound
- '>=' — Lower bound

**Default:** '=='

## Weights

Weights to use when evaluating the tracking error between the reference and testpoint signals. Use weights to increase or decrease the significance of different time points.

Must be real finite positive vector with the same number of elements as the `Time` property of the MATLAB timeseries object in the `ReferenceSignal` property.

## Properties

### AbsTol

Absolute tolerance used to determine bounds as the signal approaches the reference signal. The bounds on the reference signal are given by:

$$y_u = (1 + RelTol)y_r + AbsTol$$

$$y_l = (1 - RelTol)y_r - AbsTol$$

where  $y_r$  is the value of the reference at a certain time,  $y_u$  and  $y_l$  are the upper and lower tolerance bounds corresponding to that time point.

**Default:** 0

### Description

Requirement description. Must be a string.

**Default:** ''

## InterpolationTimes

Time points to use when comparing reference and testpoint signals. Linear interpolation is used to compare the signals at the same timepoints.

Must be one of the following strings:

- 'Reference only' — Compare the signals at the time points of the reference signal only
- 'Testpoint only' — Compare the signals at the time points of the testpoint signal only
- 'Reference and Testpoint' — Compare the signals at the time points of both the reference and testpoint signals

**Default:** 'Reference only'

## Method

Algorithm for evaluating the requirement when the `Type` property is '=='.

When the requirement is evaluated using `evalRequirement`, the software computes the error between the reference and testpoint signals. This property specifies how the error signal  $e(t) = y_s(t) - y_r(t)$  should be processed.

Must be one of the following strings:

- 'SSE'
- 'SAE'
- 'Residuals'

**Default:** 'SSE'

## Name

Requirement name. Must be a string.

**Default:** ''

## Normalize

Enable or disable normalization when evaluating the requirement. The maximum absolute value of the reference signal is used for normalization. Must be 'on' or 'off'.

**Default:** 'on'

## ReferenceSignal

Reference signal to track. Must be a MATLAB `timeseries` object with real finite data points.

**Default:** [1x1 timeseries]

## RelTol

Relative tolerance used to determine bounds as the signal approaches the reference signal. The bounds on the reference signal are given by:

$$y_u = (1 + RelTol)y_r + AbsTol$$

$$y_l = (1 - RelTol)y_r - AbsTol$$

**Default:** 0

## Type

Tracking requirement type. Must be one of the following strings:

- '==' — Tracking objective.
- '<=' — Upper bound
- '>=' — Lower bound

**Default:** '=='

# sdo.requirements.SignalTracking

---

## Weights

Weights to use when evaluating the tracking error between the reference and testpoint signals. Use weights to increase or decrease the significance of different time points.

Must be real finite positive vector with the same number of elements as the `Time` property of the MATLAB timeseries object in the `ReferenceSignal` property.

## Methods

`evalRequirement` Evaluate tracking requirement

## Copy Semantics

Handle. To learn how handle classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Construct a signal tracking object and specify a reference signal.

```
r = sdo.requirements.SignalTracking;  
r.ReferenceSignal = timeseries(1-exp(-(0:10)'));
```

Alternatively, you can specify the reference signal during construction.

```
r = sdo.requirements.SignalTracking(...  
    'ReferenceSignal',timeseries(1-exp(-(0:10)')));
```

## Alternatives

Use `getbounds` to get the bounds specified in a Check Against Reference block.

## See Also

`copy` | `get` | `set`

## How To

- Class Attributes
- Property Attributes

## Purpose

Singular value bound

## Description

Specify frequency-dependent piecewise-linear upper and lower bounds on the singular values of a linear system. You can then optimize the model response to meet these bounds using `sdo.optimize` to .

You can specify upper or lower bounds, include multiple edges, and extend them to + or -infinity.

You must have Simulink Control Design software to specify singular value requirements.

## Construction

`singval_req = sdo.requirements.SingularValue` creates a `sdo.requirements.SingularValue` object and assigns default values to its properties.

`singval_req = sdo.requirements.SingularValue(Name,Value)` uses additional options specified by one or more `Name,Value` pair arguments. `Name` is a property name and `Value` is the corresponding value. `Name` must appear inside single quotes (' '). You can specify several name-value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

## Input Arguments

### Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name,Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside single quotes (' '). You can specify several name and value pair arguments in any order as `Name1,Value1,...,NameN,ValueN`.

### BoundFrequencies

Frequency values for the gain bound.

Specify the start and end frequencies for all the edges in the piecewise-linear bound. The property must be a  $n \times 2$  array of finite doubles, where each row specifies the start and end frequencies of an edge in the piecewise-linear bound. The start

# sdo.requirements.SingularValue

---

and end frequencies must define a positive length. The number of rows must match the number of rows of the `BoundMagnitudes` property.

Use `set` to set this and the `BoundMagnitudes` properties simultaneously.

Use the `FrequencyUnits` property to specify the frequency units.

**Default:** [1 10]

## **BoundMagnitudes**

Magnitude values for the gain bound.

Specify the start and end gain values for all the edges in the piecewise-linear bound. The property must be a `nx2` array of finite doubles where each row specifies the start and end gains of an edge in the piecewise-linear bound. The number of rows must match the number of rows of the `BoundFrequencies` property.

Use `set` to set this and the `BoundFrequencies` properties simultaneously.

Use the `MagnitudeUnits` property to specify the magnitude units.

**Default:** [0 0]

## **Description**

Requirement description. Must be a string.

**Default:** ''

## **FrequencyScale**

Frequency-axis scaling.

Use this property to determine the value of the bound between edge start and end points. Must be one of the following strings:



- 'linear'
- 'log'

For example, if bound edges are at frequencies  $f_1$  and  $f_2$ , and the bound is to be evaluated at  $f_3$ , the edges are interpolated as a straight lines. The x-axis is either linear or logarithmic.

**Default:** 'log'

## FrequencyUnits

Frequency units of the requirement. Must be one of the following strings:

- 'Hz'
- 'rad/s'
- 'rpm'
- 'kHz'
- 'MHz'
- 'GHz'
- 'rad/nanosecond'
- 'rad/microsecond'
- 'rad/millisecond'
- 'rad/minute'
- 'rad/hour'
- 'rad/day'
- 'rad/week'
- 'rad/month'
- 'rad/year'

# sdo.requirements.SingularValue

---

- 'cycles/nanosecond'
- 'cycles/microsecond'
- 'cycles/millisecond'
- 'cycles/hour'
- 'cycles/day'
- 'cycles/week'
- 'cycles/month'
- 'cycles/year'

**Default:** 'rad/s'

## **MagnitudeUnits**

Magnitude units of the requirement. Must be:

- 'db' (decibels)
- 'abs' (absolute units)

**Default:** 'db'

## **Name**

Requirement name. Must be a string.

**Default:** ''

## **OpenEnd**

Extend bound in a negative or positive frequency direction.

Specify whether the first and last edge of the bound extends to  $-\infty$  and  $+\infty$  respectively. Use to bound signals that extend beyond the frequency values specified by the `BoundFrequencies` property.

Must be a 1x2 logical array of `true` or `false`. If `true`, the first or last edge of the piecewise linear bound is extended in the negative or positive direction.

**Default:** [0 0]

## Type

Magnitude bound type. Must be:

- '`<=`' — Upper bound
- '`>=`' — Lower bound

Use to specify whether the piecewise-linear bound is an upper or lower bound. Use for upper bound and for lower bound.

## Properties

### BoundFrequencies

Frequency values for the gain bound.

Specify the start and end frequencies for all the edges in the piecewise-linear bound. The property must be a  $n \times 2$  array of finite doubles, where each row specifies the start and end frequencies of an edge in the piecewise-linear bound. The start and end frequencies must define a positive length. The number of rows must match the number of rows of the `BoundMagnitudes` property.

Use `set` to set this and the `BoundMagnitudes` properties simultaneously.

Use the `FrequencyUnits` property to specify the frequency units.

**Default:** [1 10]

### BoundMagnitudes

Magnitude values for the gain bound.

# sdo.requirements.SingularValue

---

Specify the start and end gain values for all the edges in the piecewise-linear bound. The property must be a nx2 array of finite doubles where each row specifies the start and end gains of an edge in the piecewise-linear bound. The number of rows must match the number of rows of the `BoundFrequencies` property.

Use `set` to set this and the `BoundFrequencies` properties simultaneously.

Use the `MagnitudeUnits` property to specify the magnitude units.

**Default:** [0 0]

## Description

Requirement description. Must be a string.

**Default:** ''

## FrequencyScale

Frequency-axis scaling.

Use this property to determine the value of the bound between edge start and end points. Must be one of the following strings:

- 'linear'
- 'log'

For example, if bound edges are at frequencies `f1` and `f2`, and the bound is to be evaluated at `f3`, the edges are interpolated as a straight lines. The x-axis is either linear or logarithmic.

**Default:** 'log'

## FrequencyUnits

Frequency units of the requirement. Must be one of the following strings:

- 'Hz'
- 'rad/s'
- 'rpm'
- 'kHz'
- 'MHz'
- 'GHz'
- 'rad/nanosecond'
- 'rad/microsecond'
- 'rad/millisecond'
- 'rad/minute'
- 'rad/hour'
- 'rad/day'
- 'rad/week'
- 'rad/month'
- 'rad/year'
- 'cycles/nanosecond'
- 'cycles/microsecond'
- 'cycles/millisecond'
- 'cycles/hour'
- 'cycles/day'
- 'cycles/week'
- 'cycles/month'
- 'cycles/year'

# sdo.requirements.SingularValue

---

**Default:** 'rad/s'

## **MagnitudeUnits**

Magnitude units of the requirement. Must be:

- 'db' (decibels)
- 'abs' (absolute units)

**Default:** 'db'

## **Name**

Requirement name. Must be a string.

**Default:** ''

## **OpenEnd**

Extend bound in a negative or positive frequency direction.

Specify whether the first and last edge of the bound extends to `-inf` and `+inf` respectively. Use to bound signals that extend beyond the frequency values specified by the `BoundFrequencies` property.

Must be a 1x2 logical array of `true` or `false`. If `true`, the first or last edge of the piecewise linear bound is extended in the negative or positive direction.

**Default:** [0 0]

## **Type**

Magnitude bound type. Must be:

- '<=' — Upper bound
- '>=' — Lower bound

Use to specify whether the piecewise-linear bound is an upper or lower bound. Use for upper bound and for lower bound.

## Methods

`evalRequirement` Evaluate singular value bound on linear system

## Copy Semantics

Handle. To learn how handle classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Construct a singular value object and specify bound frequencies and magnitudes.

```
r = sdo.requirements.SingularValue;  
set(r, 'BoundFrequencies', [1 10; 10 100], ...  
      'BoundMagnitudes', [1 1; 1 0]);
```

Alternatively, you can specify the frequency and magnitude during construction.

```
r = sdo.requirements.SingularValue(...  
      'BoundFrequencies', [1 10; 10 100], ...  
      'BoundMagnitudes', [1 1; 1 0]);
```

## Alternatives

Use `getbounds` to get the bounds specified in a Check Singular Value Characteristics block.

## See Also

`copy` | `get` | `set`

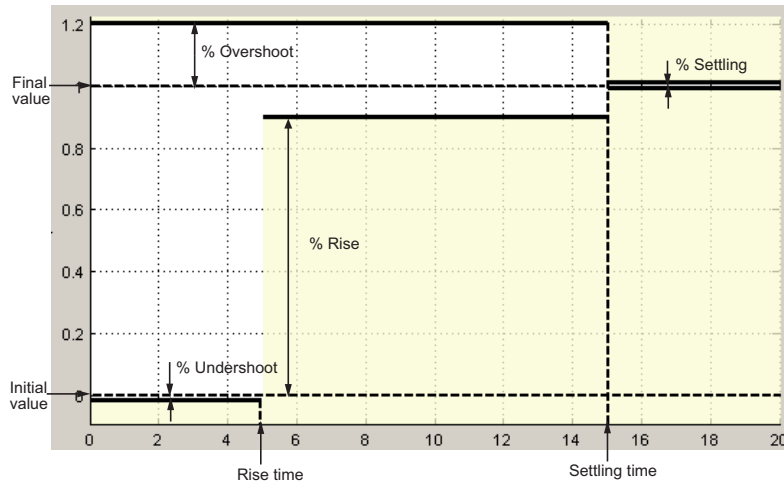
## How To

- Class Attributes
- Property Attributes

# sdo.requirements.StepResponseEnvelope

**Purpose** Step response bound on signal

**Description** Specify a step response envelope requirement on a time-domain signal. Step response characteristics such as rise-time and percentage overshoot define the step response envelope.



## Construction

`step_req = sdo.requirements.StepResponseEnvelope` creates an `sdo.requirements.StepResponseEnvelope` object and assigns default values to its properties.

`step_req = sdo.requirements.StepResponseEnvelope(Name, Value)` uses additional options specified by one or more `Name, Value` pair arguments. `Name` is a property name and `Value` is the corresponding value. `Name` must appear inside single quotes ( ' '). You can specify several name-value pair arguments in any order as `Name1, Value1, ..., NameN, ValueN`.



## Input Arguments

### Name-Value Pair Arguments

Specify optional comma-separated pairs of `Name`, `Value` arguments. `Name` is the argument name and `Value` is the corresponding value. `Name` must appear inside single quotes ( ' '). You can specify several name and value pair arguments in any order as `Name1, Value1, ..., NameN, ValueN`.

### Description

Requirement description. Must be a string.

**Default:** ''

### FinalValue

Final value of the step response. Must be a finite real scalar not equal to the `InitialValue` property.

**Default:** 1

### InitialValue

Value of the signal level before the step response starts. Must be a finite real scalar not equal to the `FinalValue` property.

**Default:** 0

### Name

Requirement name. Must be a string.

**Default:** ''

### PercentOvershoot

The percentage amount by which the signal can exceed the final value before settling.

# sdo.requirements.StepResponseEnvelope

---

Must be a real finite scalar between [0 100] and greater than PercentSettling.

Use set to set this and the PercentSettling properties simultaneously.

**Default:** 10

## **PercentRise**

The percentage of final value used with the RiseTime property to define the overall rise time characteristics.

Must be a real finite scalar between [0 100] and less than (100–PercentSettling).

Use set to set this and the PercentSettling properties simultaneously.

**Default:** 80

## **PercentSettling**

The percentage of the final value that defines the settling range of settling time characteristic specified in the SettlingTime property.

Must be a real positive finite scalar between [0 100] and less than (100 – PercentRise) and less than PercentOvershoot.

Use set to set this and the PercentOvershoot and PercentRise properties simultaneously.

**Default:** 1

## **PercentUndershoot**

The percentage amount by which the signal can undershoot the initial value.

Must be a positive finite scalar between [0 100].

**Default:** 1

## **RiseTime**

Time taken, in seconds, for the signal to reach a percentage of the final value specified in **PercentRise**.

Must be a finite positive real scalar and less than the **SettlingTime**. Time is relative to the **StepTime**.

Use **set** to set this and the **StepTime** and **SettlingTime** properties simultaneously.

**Default:** 5

## **SettlingTime**

Time taken, in seconds, for the signal to settle within a specified range around the final value. This settling range is defined as the final value plus or minus the percentage of the final value, specified in **PercentSettling**.

Must be a finite positive real scalar, greater than **RiseTime**. Time is relative to the **StepTime**.

Use **set** to set this and the **RiseTime** properties simultaneously.

**Default:** 7

## **StepTime**

Time, in seconds, when the step response starts.

Must be a finite real nonnegative scalar, less than the **RiseTime** property.

Use **set** to set this and the **RiseTime** properties simultaneously.

**Default:** 0

## **TimeUnits**

# sdo.requirements.StepResponseEnvelope

---

Time units of the requirement. Must be one of the following strings:

- 'nanoseconds'
- 'microseconds'
- 'milliseconds'
- 'seconds'
- 'minutes'
- 'hours'
- 'days'
- 'weeks'
- 'months'
- 'years'

**Default:** 'second'

## **Type**

Step response bound type.

This property is read-only and set to '<='.

## **Properties**

### **Description**

Requirement description. Must be a string.

**Default:** ''

### **FinalValue**

Final value of the step response. Must be a finite real scalar not equal to the `InitialValue` property.

**Default:** 1

## **InitialValue**

Value of the signal level before the step response starts. Must be a finite real scalar not equal to the `FinalValue` property.

**Default:** 0

## **Name**

Requirement name. Must be a string.

**Default:** ''

## **PercentOvershoot**

The percentage amount by which the signal can exceed the final value before settling.

Must be a real finite scalar between [0 100] and greater than `PercentSettling`.

Use `set` to set this and the `PercentSettling` properties simultaneously.

**Default:** 10

## **PercentRise**

The percentage of final value used with the `RiseTime` property to define the overall rise time characteristics.

Must be a real finite scalar between [0 100] and less than  $(100 - \text{PercentSettling})$ .

Use `set` to set this and the `PercentSettling` properties simultaneously.

**Default:** 80

## **PercentSettling**

# sdo.requirements.StepResponseEnvelope

---

The percentage of the final value that defines the settling range of settling time characteristic specified in the `SettlingTime` property.

Must be a real positive finite scalar between [0 100] and less than  $(100 - \text{PercentRise})$  and less than `PercentOvershoot`.

Use `set` to set this and the `PercentOvershoot` and `PercentRise` properties simultaneously.

**Default:** 1

## **PercentUndershoot**

The percentage amount by which the signal can undershoot the initial value.

Must be a positive finite scalar between [0 100].

**Default:** 1

## **RiseTime**

Time taken, in seconds, for the signal to reach a percentage of the final value specified in `PercentRise`.

Must be a finite positive real scalar and less than the `SettlingTime`. Time is relative to the `StepTime`.

Use `set` to set this and the `StepTime` and `SettlingTime` properties simultaneously.

**Default:** 5

## **SettlingTime**

Time taken, in seconds, for the signal to settle within a specified range around the final value. This settling range is defined as the final value plus or minus the percentage of the final value, specified in `PercentSettling`.

Must be a finite positive real scalar, greater than RiseTime.  
Time is relative to the StepTime.

Use `set` to set this and the RiseTime properties simultaneously.

**Default:** 7

## StepTime

Time, in seconds, when the step response starts.

Must be a finite real nonnegative scalar, less than the RiseTime property.

Use `set` to set this and the RiseTime properties simultaneously.

**Default:** 0

## TimeUnits

Time units of the requirement. Must be one of the following strings:

- 'nanoseconds'
- 'microseconds'
- 'milliseconds'
- 'seconds'
- 'minutes'
- 'hours'
- 'days'
- 'weeks'
- 'months'
- 'years'

**Default:** 'second'

# sdo.requirements.StepResponseEnvelope

---

## Type

Step response bound type.

This property is read-only and set to '<='.

## Methods

evalRequirement                      Evaluate step response bound

## Copy Semantics

Handle. To learn how handle classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Construct a step response bound object and specify percent overshoot.

```
r = sdo.requirements.StepResponseEnvelope;  
r.PercentOvershoot = 20;
```

Alternatively, you can specify the percent overshoot during construction:

```
r = sdo.requirements.StepResponseEnvelope('PercentOvershoot',20);
```

## Alternatives

Use `getbounds` to get the bounds specified in a Check Step Response Characteristics block.

## See Also

`copy` | `get` | `set`

## How To

- Class Attributes
- Property Attributes



**Superclasses** Continuous

**Purpose** Sampled parameter

**Syntax**

```
p = sdo.SampledParameter(paramname)
p = sdo.SampledParameter(paramname,paramvalue)
p = sdo.SampledParameter(paramname,paramvalue,samplevalues)
```

**Description** A sampled parameter is a numeric parameter with a nominal value and set of sample values. The parameter can be scalar- or matrix-valued.

Typically, you use sampled parameters to create parametric models and evaluate model variations for robustness testing.

**Construction** `p = sdo.SampledParameter(paramname)` constructs a `sdo.SampledParameter` object for a parameter and assigns the specified name to the `Name` property and default values to the remaining properties.

`p = sdo.SampledParameter(paramname,paramvalue)` assigns the specified parameter value to the `Value` property.

`p = sdo.SampledParameter(paramname,paramvalue,samplevalues)` assigns the specified sample values to the `SampleValues` property.

## Input Arguments

### **paramname**

Parameter name, specified as a string inside single quotes ( ' ').

### **paramvalue**

Scalar or matrix parameter value.

### **samplevalues**

Scalar, matrix or cell array of parameter sample values.

## Properties

### **Free**

Flag specifying whether the parameter is tunable or not.

# sdo.SampledParameter

---

Set the `Free` property to `true` (1) for tunable parameters and `false` (0) for parameters you do not want to tune (fixed).

The dimension of this property must match the dimension of the `Value` property.

For matrix-valued parameters, you can:

- Fix individual matrix elements. For example `p.Free = [true false; false true]` or `p.Free([2 3]) = false`.
- Use scalar expansion to fix all matrix elements. For example `p.Free = false`.

**Default:** `true` (1)

## Info

Structure array specifying parameter units and labels.

The structure has `Label` and `Unit` fields.

The array dimension must match the dimension of the `Value` property.

Use this property to store parameter units and labels that describe the parameter. For example `p.Info(1,1).Unit = 'N/m'`; or `p.Info(1,1).Label = 'spring constant'`.

**Default:** `' '` for both `Label` and `Unit` fields

## Maximum

Upper bound for the parameter value.

The dimension of this property must match the dimension of the `Value` property.

For matrix-valued parameters, you can:

- Specify upper bounds on individual matrix elements. For example `p.Maximum([1 4]) = 5`.

- Use scalar expansion to set the upper bound for all matrix elements. For example `p.Maximum = 5`.

**Default:** `Inf`

## Minimum

Lower bound for the parameter value.

The dimension of this property must match the dimension of the `Value` property.

For matrix-valued parameters, you can:

- Specify lower bounds on individual matrix elements. For example `p.Minimum([1 4]) = -5`.
- Use scalar expansion to set the lower bound for all matrix elements. For example `p.Minimum = -5`.

**Default:** `Inf`

## Name

Parameter name.

This property is read-only and is set at object construction.

**Default:** `''`

## SampleValues

Set of sample values for the parameter.

Must be a cell array of values. The elements of the cell array must have the same dimension as the `Value` property. If the `Value` property is a scalar, this property can be a vector.

**Default:** `[-1 1]`

## Scale

# sdo.SampledParameter

---

Scaling factor used to normalize the parameter value.

The dimension of this property must match the dimension of the Value property.

For matrix-valued parameters, you can:

- Specify scaling for individual matrix elements. For example `p.Scale([1 4]) = 1`.
- Use scalar expansion to set the scaling for all matrix elements. For example `p.Scale = 1`.

**Default:** 1

## Value

Scalar or matrix value of a parameter.

The dimension of this property is set at object construction.

**Default:** 0

## Methods

### Inherited Methods

isreal

Determine if parameter value, minimum and maximum are real

## Copy Semantics

Value. To learn how value classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

Specify sample values during construction.

```
p = sdo.SampledParameter('K', eye(2), {0.9*eye(2) 1.1*eye(2)});
```

---

Construct a sampled parameter object and set its sample values.

```
p = sdo.SampledParameter('K',eye(2));  
p.SampleValues = {0.9*eye(2) 1.1*eye(2)};
```

**Alternatives**    “Optimize Parameters for Robustness (GUI)”

**See Also**        param.Continuous | sdo.optimize

**How To**

- “Optimizing Parameters for Robustness”
- Class Attributes
- Property Attributes

# sdo.SimulationTest

---

**Purpose** Simulation scenario description

**Syntax** `sim_obj = sdo.SimulationTest(modelname)`

**Description** Create a scenario to simulate a Simulink model. A simulation scenario specifies input signals, model parameter and initial state values, and signals to log for a model. Use a simulation scenario to simulate a model with alternative inputs and model parameter and initial state values, without modifying the model.

**Construction** `sim_obj = sdo.SimulationTest(modelname)` constructs an `sdo.SimulationTest` object and assigns the specified model name to the `ModelName` property and default values to the remaining properties.

You can also construct an `sdo.SimulationTest` object using the `sdo.Experiment.createSimulator` method of an `sdo.Experiment` object. The `createSimulator` method configures the properties of the `sdo.SimulationTest` object to simulate the model associated with the experiment.

## Input Arguments

### **modelname**

Simulink model name, specified as a string inside single quotes (' ').

The model must be on the MATLAB path.

## Properties

### **InitialState**

Model initial state for simulation.

This property can be any initial state format that `sim` command supports.

### **Inputs**

Input signals.

Specify signals to apply to root level input ports when simulating the model. The signal can be any input signal format that the `sim` command supports.

**Default:** []

## LoggedData

Data logged during simulation.

You must also specify the signals to log in the `LoggingInfo` property. The logged data is stored in a `Simulink.SimulationOutput` object and is populated by the `sim` method.

This property is read-only.

**Default:** []

## LoggingInfo

Signals to log when simulating a model.

This property is a `Simulink.SimulationData.ModelLoggingInfo` object. Specify the signals to log in its `Signals` property.

**Default:** 1x1 `Simulink.SimulationData.ModelLoggingInfo` object

## ModelName

Simulink model name associated with the simulation scenario. The model must be on the MATLAB path.

## Name

Name of the scenario

**Default:** ''

## Parameters

# sdo.SimulationTest

---

Parameter values.

The software changes the model parameters to the specified values before simulating the model and restores them to their original value after the simulation completes.

This property must be a `param.Continuous` object.

**Default:** []

## Methods

<code>find</code>	Find logged data set
<code>sim</code>	Simulate Simulink model using simulation scenario
<code>who</code>	List logged data names

## Copy Semantics

Value. To learn how value classes affect copy operations, see Copying Objects in the MATLAB documentation.

## Examples

### Create Simulation Scenario for Model

Create a simulation scenario for a model and specify model signals to log.

```
Pressures = Simulink.SimulationData.SignalLoggingInfo;  
Pressures.BlockPath = 'sdoHydraulicCylinder/Cylinder Assembly';  
Pressures.OutputPortIndex = 1;  
simulator = sdo.SimulationTest('sdoHydraulicCylinder');  
simulator.LoggingInfo.Signals = [Pressures];
```

### Create Simulation Scenario from Experiment

Specify an experiment.

```
experiment = sdo.Experiment('sdoRCCircuit');
```

Create a simulation scenario for the experiment.



```
sim_obj = createSimulator(experiment);
```

## Alternatives

“Design Optimization to Meet Step Response Requirements (GUI)”

## See Also

`sdo.optimize` | `sdo.Experiment.createSimulator` | `sdo.Experiment`

## Tutorials

- “Design Optimization to Meet Step Response Requirements (Code)”
- “Design Optimization to Meet a Custom Objective (Code)”
- “Estimate Model Parameter Values (Code)”
- “Estimate Model Parameters and Initial States (Code)”

## How To

- Class Attributes
- Property Attributes



# Alphabetical List

---

# copy

---

<b>Purpose</b>	Copy requirement
<b>Syntax</b>	<code>copy_req = copy(req)</code>
<b>Description</b>	<code>copy_req = copy(req)</code> copies a requirement object ( <code>sdo.requirements.StepResponseEnvelope, ...</code> ) to a new object of the same type.  For more information, see <code>copy</code> in the MATLAB documentation.
<b>Input Arguments</b>	<b>req</b> requirement object ( <code>sdo.requirements.StepResponseEnvelope, ...</code> )
<b>Output Arguments</b>	<b>copy_req</b> requirement object ( <code>sdo.requirements.StepResponseEnvelope, ...</code> ), which is a copy of <code>req</code> .
<b>See Also</b>	<code>get</code>   <code>handle</code>

# sdo.requirements.BodeMagnitude.evalRequirement

---

<b>Purpose</b>	Evaluate Bode magnitude bound for linear system
<b>Syntax</b>	<code>c = evalRequirement(req,lin_sys)</code>
<b>Description</b>	<code>c = evalRequirement(req,lin_sys)</code> evaluate whether a linear system satisfies the specified piecewise-linear Bode magnitude bound.
<b>Input Arguments</b>	<p><b>req</b></p> <p>sdo.requirements.BodeMagnitude object.</p> <p>For MIMO systems, the bound applies to each input/output (I/O) channel.</p> <p><b>lin_sys</b></p> <p>Linear system (tf, ss, zpk, frd, genss, or genfrd).</p>
<b>Output Arguments</b>	<p><b>c</b></p> <p>Column vector indicating the maximum signed distance of the system gain to each edge specified in <code>req</code>. Negative values indicate that the bound edge is satisfied and positive values that the bound edge is violated.</p> <p>For MIMO systems, a matrix of signed distances where each column represents an I/O pair and gives the distance of that IO pair gain to each edge in the bounds.</p>
<b>Examples</b>	<p>Evaluate Bode magnitude requirement.</p> <pre>req = sdo.requirements.BodeMagnitude; sys = tf(1,[1 2 2 1]); c = evalRequirement(req,sys);</pre> <p><code>c</code> is negative, which indicates that the system satisfies the gain requirement.</p>
<b>See Also</b>	<code>sdo.requirements.BodeMagnitude</code>   <code>get</code>   <code>set</code>   <code>copy</code>

# sdo.requirements.ClosedLoopPeakGain.evalRequirement

<b>Purpose</b>	Evaluate peak gain bound for linear system
<b>Syntax</b>	<code>c = evalRequirement(req,lin_sys)</code>
<b>Description</b>	<code>c = evalRequirement(req,lin_sys)</code> evaluates whether a linear system satisfies the specified peak gain (infinity norm of the system) bound. The closed loop is computed using the feedback sign specified in the FeedbackSign property of req.
<b>Input Arguments</b>	<b>req</b> sdo.requirements.ClosedLoopPeakGain object. <b>lin_sys</b> Linear system (tf, ss, zpk, frd, genss, or genfrd).
<b>Output Arguments</b>	<b>c</b> <ul style="list-style-type: none"><li>• Signed distance of the closed-loop peak gain to the bound if the Type property of req is &lt;= or ==. When &lt;=, negative values indicate that the bound is satisfied while positive values indicate the bound is violated. When ==, any value other than 0 indicate that the bound is violated.</li><li>• Peak gain if the Type property of req is min.</li></ul>
<b>Examples</b>	Evaluate peak gain requirement. <pre>req = sdo.requirements.ClosedLoopPeakGain; sys = tf(0.5,[1 3 3 1]); c = evalRequirement(req,sys);</pre> <p>c is negative, which indicates that the system satisfies the gain requirement.</p>
<b>See Also</b>	sdo.requirements.ClosedLoopPeakGain   get   set   copy

# sdo.requirements.GainPhaseMargin.evalRequirement

<b>Purpose</b>	Evaluate gain and phase margin bounds for linear system
<b>Syntax</b>	<code>c = evalRequirement(req,lin_sys)</code>
<b>Description</b>	<code>c = evalRequirement(req,lin_sys)</code> evaluates whether a linear system satisfies the specified gain and phase margin bounds. The gain and phase margins are computed using the feedback sign specified in the <code>FeedbackSign</code> property of <code>req</code> .
<b>Input Arguments</b>	<b>req</b> sdo.requirements.GainPhaseMargin object. <b>lin_sys</b> Linear system (tf, ss, zpk, frd, genss, or genfrd).
<b>Output Arguments</b>	<b>c</b> <ul style="list-style-type: none"><li>• Signed distance of the computed gain and phase margins to the bound if the <code>Type</code> property of <code>req</code> is <code>&gt;=</code> or <code>==</code>. Signed distance to the gain margin bound appear before the signed distance to the phase margin bound. Negative values indicate that the bound is satisfied while positive values indicate the bound is violated. Unstable loops return positive values. When <code>==</code>, any number other than 0 indicates that the bound is not satisfied.</li><li>• Negative of the gain and phase margins such that minimizing the values maximizes the margins if the <code>Type</code> property of <code>req</code> is <code>'max'</code>. Unstable loops return positive values.</li></ul>
<b>Examples</b>	Evaluate gain and phase margin requirements. <pre>req = sdo.requirements.GainPhaseMargin; sys = tf(0.5,[1 3 3 1]); c = evalRequirement(req,sys);</pre>

## **sdo.requirements.GainPhaseMargin.evalRequirement**

---

`c` is negative, which indicates that the system satisfies the gain and phase margin requirement.

get | set | copy



# sdo.requirements.OpenLoopGainPhase.evalRequirement

---

<b>Purpose</b>	Evaluate gain and phase bounds on Nichols response of linear system
<b>Syntax</b>	<code>c = evalRequirement(req,lin_sys)</code>
<b>Description</b>	<code>c = evalRequirement(req,lin_sys)</code> evaluates whether a linear system satisfies the specified open-loop gain and phase bounds on the Nichols response.
<b>Input Arguments</b>	<b>req</b> sdo.requirements.OpenLoopGainPhase object. <b>lin_sys</b> Linear system (tf, ss, zpk, frd, genss, or genfrd).
<b>Output Arguments</b>	<b>c</b> Vector of maximum signed distances of the response to each piecewise linear edge. Negative values indicate that the bound edge is satisfied and positive values indicate the bound is violated.
<b>Examples</b>	Evaluate open-loop gain and phase requirements.  <pre>req = sdo.requirements.OpenLoopGainPhase; sys = tf(0.5,[1 3 3 1]); c = evalRequirement(req,sys);</pre>
<b>See Also</b>	sdo.requirements.OpenLoopGainPhase   get   set   copy

# sdo.requirements.PZDampingRatio.evalRequirement

---

<b>Purpose</b>	Evaluate damping ratio bound on linear system
<b>Syntax</b>	<code>c = evalRequirement(req,lin_sys)</code>
<b>Description</b>	<code>c = evalRequirement(req,lin_sys)</code> evaluates whether the poles of a linear system satisfies the specified damping ratio bound.
<b>Input Arguments</b>	<b>req</b> sdo.requirements.PZDampingRatio object. <b>lin_sys</b> Linear system (tf, ss, zpk, frd, genss, or genfrd).
<b>Output Arguments</b>	<b>c</b> <ul style="list-style-type: none"><li>• Signed distance of the damping ratio of each pole of the linear system to the bound, if the <b>Type</b> property of <b>req</b> is <code>&gt;=</code>, <code>&lt;=</code> or <code>==</code>. Negative values indicate that the bound is satisfied while positive values indicate that the bound is violated. When <code>==</code>, any value other than 0 indicates that the bound is violated.</li><li>• Negative of the damping ratio such that minimizing the values maximizes the damping ratio, if the <b>Type</b> property of <b>req</b> is <code>'max'</code>.</li></ul>

**Examples** Evaluate damping ratio requirement.

```
req = sdo.requirements.PZDampingRatio;  
sys = tf(0.5,[1 3 3 1]);  
c = evalRequirement(req,sys);
```

`c` is negative, which indicates that the system satisfies the damping ratio requirement.

get | set | copy

# sdo.requirements.PZNaturalFrequency.evalRequirement

<b>Purpose</b>	Evaluate natural frequency bound on linear system
<b>Syntax</b>	<code>c = evalRequirement(req, lin_sys)</code>
<b>Description</b>	<code>c = evalRequirement(req, lin_sys)</code> evaluates whether the poles of a linear system satisfies the specified natural frequency bound.
<b>Input Arguments</b>	<p><b>req</b></p> <p>Requirement object (<code>sdo.requirements.StepResponseEnvelope, ...</code>).</p> <p>For MIMO systems, the bound applies to each input/output (I/O) channel.</p> <p><b>lin_sys</b></p> <p>Linear system (<code>tf, ss, zpk, frd, genss, or genfrd</code>).</p>
<b>Output Arguments</b>	<p><b>c</b></p> <ul style="list-style-type: none"><li>• Signed distance of the natural frequency of each system pole to the bound. If the <code>Type</code> property of <code>req</code> is <code>&gt;=</code>, <code>&lt;=</code>, negative values indicate that the bound is satisfied while positive values indicate that the bound is violated. If <code>==</code>, any value other than 0 indicates that the bound is violated.</li><li>• Negative of the natural frequency of the linear system poles such that minimizing the values maximizes the natural frequency, if the <code>Type</code> property of <code>req</code> is <code>'max'</code>.</li></ul>
<b>Examples</b>	<p>Evaluate natural frequency requirement.</p> <pre>req = sdo.requirements.PZNaturalFrequency; sys = tf(0.5,[1 3 3 1]); c = evalRequirement(req,sys);</pre> <p><code>c</code> is positive, which indicates that the system does not satisfy the natural frequency requirement.</p>

# sdo.requirements.PZNaturalFrequency.evalRequirement

---

## See Also

`sdo.requirements.PZNaturalFrequency` | `get` | `set` | `copy`

# sdo.requirements.PZSettlingTime.evalRequirement

<b>Purpose</b>	Evaluate settling time bound on linear system
<b>Syntax</b>	<code>c = evalRequirement(req, lin_sys)</code>
<b>Description</b>	<code>c = evalRequirement(req, lin_sys)</code> evaluates whether the poles of a linear system satisfies the specified settling time bound.
<b>Input Arguments</b>	<b>req</b> sdo.requirements.PZSettlingTime object. <b>lin_sys</b> Linear system (tf, ss, zpk, frd, genss, or genfrd).
<b>Output Arguments</b>	<b>c</b> <ul style="list-style-type: none"><li>• Signed distance of the real component of each system pole to the bound, if the <code>Type</code> property of <code>req</code> is <code>&lt;=</code> or <code>==</code>. Negative values indicate that the bound is satisfied while positive values indicate that the bound is violated. If <code>==</code>, values other than 0 indicate that the bound is violated.</li><li>• Pole locations such that minimizing the values minimizes the settling time, if the <code>Type</code> property of <code>req</code> is <code>'min'</code>.</li></ul>
<b>Examples</b>	Evaluate settling time requirement.  <pre>req = sdo.requirements.PZSettlingTime; sys = tf(0.5,[1 3 3 1]); c = evalRequirement(req,sys);</pre> <code>c</code> is positive, which indicates that the system does not satisfy the settling time requirement. <code>get   set   copy</code>

# sdo.requirements.SignalBound.evalRequirement

---

<b>Purpose</b>	Evaluate piecewise-linear bound
<b>Syntax</b>	<code>c = evalRequirement(req,sig)</code>
<b>Description</b>	<code>c = evalRequirement(req,sig)</code> evaluate whether a signal satisfies the specified piecewise-linear bounds.
<b>Input Arguments</b>	<b>req</b> sdo.requirements.SignalBound object. <b>sig</b> MATLAB timeseries object or nxm array, where the 1st column is time and subsequent columns are signal values.
<b>Output Arguments</b>	<b>c</b> Column vector indicating the maximum signed distance of the signal to each edge. Negative values indicate that the bound edge is satisfied and positive values indicate that the bound edge is violated. Matrix if multi-channeled signal.
<b>Examples</b>	Evaluate piecewise-linear bound on signal. <pre>req = sdo.requirements.SignalBound; sig = timeseries(1-exp(-(0:10)')); c = evalRequirement(req,sig);</pre> <p>c is negative, which indicates that the signal satisfies the bounds.</p>
<b>See Also</b>	<a href="#">get</a>   <a href="#">set</a>   <a href="#">copy</a>

# sdo.requirements.SignalTracking.evalRequirement

<b>Purpose</b>	Evaluate tracking requirement
<b>Syntax</b>	<pre>c = evalRequirement(req,sig) c = evalRequirement(req,sig,ref)</pre>
<b>Description</b>	<p><code>c = evalRequirement(req,sig)</code> evaluates whether a test point signal, <code>sig</code>, tracks the reference signal specified by a requirement object, <code>req</code>.</p> <p><code>c = evalRequirement(req,sig,ref)</code> evaluates whether <code>sig</code> tracks the reference signal specified by <code>ref</code>. <code>req</code> specifies the error computation options. Estimating parameters for multiple experiments requires you to repeatedly compare test point and reference signal sets. Use this syntax if you use the same evaluation criteria for all comparisons. You vary <code>sig</code> and <code>ref</code>, and re-use the requirement object, <code>req</code>.</p>
<b>Input Arguments</b>	<p><b>req</b> sdo.requirements.SignalTracking object.</p> <p><b>sig</b> MATLAB timeseries object or nxm array, where the 1st column is time and subsequent columns are signal values.</p> <p><b>ref</b> Reference signal, specified as a MATLAB timeseries object.</p>
<b>Output Arguments</b>	<p><b>c</b></p> <ul style="list-style-type: none"><li>• Measure of how well the test point signal matches the reference signal, if the <code>Type</code> property of <code>req</code> is <code>'=='</code>. Specify the algorithm used to compute the tracking measure through the <code>Method</code> property.</li><li>• Signed distance of the test point signal to the reference signal, if the <code>Type</code> property of <code>req</code> is <code>'&gt;='</code> or <code>'&lt;='</code>. Negative values indicate the bound is satisfied while positive values indicate that the bound is violated.</li></ul>

# sdo.requirements.SignalTracking.evalRequirement

---

The command compares the reference and test point signals only at time points that are in the range of both signals. Time points outside this range are ignored. The software uses the interpolation method specified by `ref.InterpolationTimes` to compare the data in the valid time range.

## Examples

### Evaluate Signal Tracking Requirement

Create the reference data.

```
time = (0:0.1:10)';  
data = 1-exp(-time);
```

Create the signal tracking requirement object. Specify the reference signal.

```
req = sdo.requirements.SignalTracking;  
req.ReferenceSignal = timeseries(data,time);
```

Obtain the test point signal.

```
sig = timeseries(1-exp(-time/2),time);
```

Evaluate the signal tracking requirement.

```
c = evalRequirement(req,sig);
```

### Evaluate Tracking Using Requirement Object to Specify Error Computation Method

When you estimate parameters for multiple experiments, you repeatedly compare test point and reference signal sets. If you use the same evaluation criteria for all comparisons, you can use the `c = evalrequirement(req,sig,ref)` syntax. You vary `sig` and `ref`, and re-use the requirement object, `req`. `req` specifies the estimation error computation options.

For this example, create a reference and test point signal. Then, use a requirement object to evaluate the requirement.



# sdo.requirements.SignalTracking.evalRequirement

---

Create the reference signal.

```
time = (0:0.1:10)';  
data = 1-exp(-time);  
ref = timeseries(data,time);
```

Create the signal tracking requirement object. Specify the error computation method.

For this example, specify 'Residuals' as the algorithm for error computation.

```
req = sdo.requirements.SignalTracking;  
req.Method = 'Residuals';
```

Obtain the test point signal.

```
sig = timeseries(1-exp(-time/2),time);
```

Evaluate the signal tracking requirement.

```
c = evalRequirement(req,sig,ref);
```

## See Also

[get](#) | [set](#) | [copy](#)

# sdo.requirements.SingularValue.evalRequirement

---

<b>Purpose</b>	Evaluate singular value bound on linear system
<b>Syntax</b>	<code>c = evalRequirement(req,lin_sys)</code>
<b>Description</b>	<code>c = evalRequirement(req,lin_sys)</code> evaluates whether a linear system satisfies the specified singular values bound.
<b>Input Arguments</b>	<p><b>req</b> sdo.requirements.SingularValue object. For MIMO systems, the bound applies to each input/output (I/O) channel.</p> <p><b>lin_sys</b> Linear system (tf, ss, zpk, frd, genss, or genfrd).</p>
<b>Output Arguments</b>	<p><b>c</b> Column vector indicating the maximum signed distance of the system gain to each edge specified in <code>req</code>. Negative values indicate that the bound edge is satisfied and positive values indicate that the bound edge is violated. For MIMO systems, a matrix of signed distances where each column represents an I/O pair and gives the distance of that IO pair gain to each edge in the bounds.</p>
<b>Examples</b>	<p>Evaluate singular value requirement.</p> <pre>req = sdo.requirements.SingularValue; sys = tf(1,[1 2 2 1]); c = evalRequirement(req,sys);</pre> <p><code>c</code> is negative, which indicates that the system satisfies the gain requirement.</p>
<b>See Also</b>	sdo.requirements.SingularValue   get   set   copy

# sdo.requirements.StepResponseEnvelope.evalRequirement

**Purpose** Evaluate step response bound

**Syntax** `c = evalRequirement(req,sig)`

**Description** `c = evalRequirement(req,sig)` evaluate whether a signal satisfies specified step response bounds.

**Input Arguments**

**req**

sdo.requirements.StepResponseEnvelope object.

**sig**

MATLAB `timeseries` object or `nxm` array, where the 1st column is time and subsequent columns are signal values.

Numeric or generalized linear time invariant (LTI) model, if you have Simulink Control Design software.

**Output Arguments**

**c**

Column vector indicating the maximum signed distance of the signal to each edge in the step response envelope.

Signed distances to upper bound edges appear before signed distances to lower bounds edges. Negative values indicate that the bound edge is satisfied and positive values indicate that the bound edge is violated.

**Examples**

Evaluate step response bounds on signal.

```
req = sdo.requirements.StepResponseEnvelope;  
sig = timeseries(1-exp(-(0:10)'));  
c = evalRequirement(req,sig);
```

**See Also**

`get` | `set` | `copy`

# get

---

<b>Purpose</b>	Get property values
<b>Syntax</b>	<code>get(req)</code> <code>get(req,PropertyName)</code>
<b>Description</b>	<code>get(req)</code> returns the value of all properties of the requirement object ( <code>sdo.requirements.StepResponseEnvelope, ...</code> ). <code>get(req,PropertyName)</code> returns value of a specific property. Use a cell array of property names to return a cell array with multiple property values.
<b>Input Arguments</b>	<b>req</b> Requirement object ( <code>sdo.requirements.StepResponseEnvelope, ...</code> ).  <b>PropertyName</b> Name of the requirement object ( <code>sdo.requirements.StepResponseEnvelope, ...</code> ) property.
<b>Alternatives</b>	“Getting Property Values”

---

<b>Purpose</b>	Set property values
<b>Syntax</b>	<code>set(req,Name,Value,)</code>
<b>Description</b>	<code>set(req,Name,Value,)</code> sets the property value of a requirement object ( <code>sdo.requirements.StepResponseEnvelope, ...</code> ). Specify the property name and value using one or more <code>Name,Value</code> pair arguments.
<b>Tips</b>	<ul style="list-style-type: none"><li>• Use <code>set</code> to simultaneously change properties that you cannot change independently.</li></ul>
<b>Input Arguments</b>	<p><b>req</b> Requirement object (<code>sdo.requirements.StepResponseEnvelope, ...</code>)</p> <p><b>Name,Value</b> Property name of a requirement object (<code>sdo.requirements.StepResponseEnvelope, ...</code>), and the corresponding value to set.</p>
<b>Examples</b>	<p>Specify property values.</p> <pre>r = sdo.requirements.SignalBound; set(r,'BoundTimes',[0 5;5 10], ...     'BoundMagnitudes',[1.1 1.1; 1.01 1.01]);</pre>
<b>Alternatives</b>	“Setting Property Values”

# findconstr

---

**Purpose** Extract design requirements specified in Signal Constraint block

---

**Note** `findconstr` will be removed in a future version. Use `getbounds` instead.

---

**Syntax** `constraints=findconstr(proj, 'blockname')`

**Description** `constraints=findconstr(proj, 'blockname')` extracts design requirements specified in the Signal Constraint block `blockname` of the Simulink model associated with the response optimization project `proj`. `constraints` contains data that defines the design requirements. Modify the `UpperBoundX`, `UpperBoundY`, `LowerBoundX` and `LowerBoundY` properties of `constraints` to specify new design requirements, and the `ReferenceX` and `ReferenceY` properties to add a reference signal. The Simulink model associated with `proj` must be open.

---

**Note** The Signal Constraint block does not update to display the modified constraints and reference signal. However, the updated constraint bounds and reference signal specified in the constraint object are used when you optimize the parameters from the command line.

---

**See Also** `getsro` | `newsro` | `optimize`

**Tutorials** • “Design Optimization to Meet Step Response Requirements (Code)”

**How To** • “How the Optimization Algorithm Formulates Minimization Problems”

**Purpose** List of model path dependencies

---

**Note** finddepend will be removed in a future version. Use `sdo.getModelDependencies` instead.

---

**Syntax** `dirs=finddepend(proj)`

**Description** `dirs=finddepend(proj)` returns paths containing Simulink model dependencies required for parameter optimization using parallel computing. `proj` is a response optimization project for the model created using `newsro` or `getsro`. `dirs` is a cell array, where each element is a path string. `dirs` is empty when `finddepend` does not detect any model dependencies. Append paths to `dirs` when the list of paths is empty or incomplete.

`finddepend` does not return a complete list of model dependency paths when the dependencies are undetectable.

**Examples** List model path dependencies required for parallel computing:

```
% Copy Simulink boiler library to temporary folder.
pathToLib = boilerpressure_setup;
% Add folder to search path.
addpath(pathToLib);
% Open Simulink model.
boilerpressure_demo
% Extract response optimization project.
proj=getsro('boilerpressure_demo');
% Enable parallel computing.
optimset(proj,'UseParallel','always');
% Get model dependency paths.
dirs=finddepend(proj)
% Add paths to optimization project.
optimset(proj,'ParallelPathDependencies',dirs)
```

---

Make local paths accessible to remote workers:

```
% Copy Simulink boiler library to temporary folder.
pathToLib = boilerpressure_setup;
% Add folder to search path.
addpath(pathToLib);
% Open Simulink model.
boilerpressure_demo
% Extract response optimization project.
proj=getstro('boilerpressure_demo');
% Enable parallel computing.
optimset(proj,'UseParallel','always');
% Get model dependency paths.
dirs=finddepend(proj)
% The resulting path is on a local drive, C:/.
% Replace C:/ with valid network path accessible to remote workers.
dirs = regexprep(dirs,'C:/','\\\\hostname\C$')
% Add paths to optimization project.
optimset(proj,'ParallelPathDependencies',dirs)
```

---

Append path to model path dependency list:

```
% Copy Simulink boiler library to temporary folder.
pathToLib = boilerpressure_setup;
% Add folder to search path.
addpath(pathToLib);
% Open Simulink model.
boilerpressure_demo
% Extract response optimization project.
proj=getstro('boilerpressure_demo');
% Enable parallel computing.
optimset(proj,'UseParallel','always');
% Get model dependency paths.
dirs=finddepend(proj)
```



```
% Append an additional path.  
dirs=vertcat(dirs,'\\hostname\C$\matlab\work')  
% Add paths to optimization project.  
optimset(proj,'ParallelPathDependencies',dirs)
```

## Alternatives

Identify model path dependencies using the GUI:

- 1** In the Simulink model, double-click the Signal Constraint block to open the Block Parameters: Signal Constraint window.
- 2** In the Block Parameters window, select **Optimization > Parallel Options** to open the **Parallel Options** tab.
- 3** Select the **Use the matlabpool during optimization** option to identify the model dependencies automatically.

## See Also

newsro | getsro | optimget | optimset

## Tutorials

- Improving Optimization Performance Using Parallel Computing

## How To

- “Speedup Using Parallel Computing”
- “Analyze Model Dependencies”

# findpar

---

## Purpose

Find specifications for given tuned parameter

---

**Note** findpar will be removed in a future version. Use `sdo.getParameterFromModel` instead.

---

## Syntax

```
p=findpar(proj, 'param')
```

## Description

`p=findpar(proj, 'param')` returns a tuned parameters object for the parameter with the name `param` within the response optimization project, `proj`. The tuned parameters object defines specifications for each tuned parameter that the optimization method uses, such as initial guesses, lower bounds, etc.

The properties of each tuned parameter object are

Name	A string giving the parameter's name.
Value	The current value of the parameter. This changes during the optimization.
InitialGuess	The initial guess for the parameter value for the optimization.
Minimum	The minimum value this parameter can take. By default, it is set to <code>-Inf</code> .
Maximum	The maximum value this parameter can take. By default, it is set to <code>Inf</code> .
TypicalValue	A value that the tuned parameter is scaled by during the optimization.
ReferencedBy	The block, or blocks, in which the parameter appears.

Description	An optional string giving a description of the parameter.
Tuned	Set to 1 or 0 to indicate if this parameter is to be tuned or not.

Edit these properties to specify additional information about your parameters.

**See Also**

`getsro` | `newsro` | `optimize`

**Concepts**

- “Estimate Model Parameter Values (GUI)”

# getbounds

---

<b>Purpose</b>	Get bounds specified in Check block
<b>Syntax</b>	<code>bnds = getbounds(blockpath)</code>
<b>Description</b>	<code>bnds = getbounds(blockpath)</code> returns the bounds specified in the Check block specified by <code>blockpath</code> .
<b>Input Arguments</b>	<b>blockpath</b> Check block to get bounds from, specified as a full block path inside single quotes ( ' '). A block path is of the form <i>model/subsystem/block</i> that uniquely identifies a block in the model. The Simulink model must be open.
<b>Output Arguments</b>	<b>bnds</b> Cell array. The number of elements and objects they contain depends on the Check block type. <ul style="list-style-type: none"><li>• Check Step Response Characteristics: Cell array of one element that contains a <code>sdo.requirements.StepResponseEnvelope</code> object.</li><li>• Check Custom Bounds: Cell array of two elements — the first and second elements contain the following upper and lower bound values, respectively. Both elements are <code>sdo.requirements.SignalBound</code> objects.</li><li>• Check Against Reference: Cell array of one element that contains a <code>sdo.requirements.SignalTracking</code> object.</li></ul>

---

**Note** Programmatically changing the bound values in the object returned does not update them in the Block Parameters dialog box.

---

**Examples** Retrieve bounds from a Check Step Response Characteristics block.

```
sldo_model1_stepblk;  
allBlkReq = getbounds('sldo_model1_stepblk/Step Response');
```

Type `allBlkReq{1}` to view the cell array element.

```
ans =
```

```
sdo.requirements.StepResponseEnvelope handle  
Package: sdo.requirements
```

```
Properties:
```

```
    InitialValue: 0  
    FinalValue: 1  
    StepTime: 0  
    RiseTime: 5  
    PercentRise: 80  
    SettlingTime: 7  
    PercentSettling: 1.0000  
    PercentOvershoot: 10.0000  
    PercentUndershoot: 1  
    Type: '<='  
    Name: ''  
    Description: ''  
    TimeUnits: 'seconds'
```

## See Also

[sdo.optimize](#) | [Check Against Reference](#) | [Check Custom Bounds](#) | [Check Step Response Characteristics](#)

## Tutorials

- “Design Optimization to Meet Step Response Requirements (Code)”

# sdo.requirements.PZDampingRatio.getOvershoot

---

<b>Purpose</b>	Convert damping ratio to equivalent overshoot value
<b>Syntax</b>	<code>overshoot = getOvershoot(req)</code>
<b>Description</b>	<code>overshoot = getOvershoot(req)</code> converts the damping ratio value specified in the <code>DampingRatio</code> property of <code>pzdampratio_req</code> object to an equivalent approximate second-order overshoot value.
<b>Input Arguments</b>	<b>req</b> sdo.requirements.PZDampingRatio object.
<b>Output Arguments</b>	<b>overshoot</b> Approximate second-order percent overshoot value, equivalent to the damping ratio value in <code>DampingRatio</code> property of <code>pzdampratio_req</code> .
<b>Examples</b>	Convert damping ratio to approximate second-order overshoot value.  <pre>r = sdo.requirements.PZDampingRatio; r.DampingRatio = 0.1; overshoot = getOvershoot(r);</pre>
<b>See Also</b>	<code>sdo.requirements.PZDampingRatio</code>   <code>evalRequirement</code>   <code>setOvershoot</code>

# sdo.requirements.PZDampingRatio.setOvershoot

---

<b>Purpose</b>	Set overshoot to an equivalent damping ratio
<b>Syntax</b>	<code>req1 = setOvershoot(req,percent_overshoot)</code>
<b>Description</b>	<code>req1 = setOvershoot(req,percent_overshoot)</code> sets the damping ratio value to a value equivalent to percent overshoot.
<b>Input Arguments</b>	<b>req</b> sdo.requirements.PZDampingRatio object. <b>percent_overshoot</b> Percent overshoot value to compute damping ratio.
<b>Output Arguments</b>	<b>req1</b> sdo.requirements.PZDampingRatio object whose DampingRatio property is the damping ratio value equivalent to percent_overshoot.
<b>Examples</b>	Specify overshoot bound.  <code>req = sdo.requirements.PZDampingRatio setOvershoot(req,20)</code>
<b>See Also</b>	sdo.requirements.PZDampingRatio   evalRequirement   getOvershoot

# getsro

---

**Purpose** Extract response optimization project for given Simulink model

---

**Note** getsro will be removed in a future version. For examples of the new optimization workflow, see:

- “Design Optimization to Meet Step Response Requirements (GUI)”
  - “Design Optimization to Meet Step Response Requirements (Code)”
- 

**Syntax** `proj=getsro('modelName')`

**Description** `proj=getsro('modelName')` returns the response optimization project, `proj`, currently associated with the Simulink model with name, `modelName`. The model should be open and contain Simulink Design Optimization blocks. Use the project with the `optimize` function to optimize response signals in the model by tuning specified parameters.

**See Also** `findconstr` | `findpar` | `newsro` | `optimize`



**Purpose** Multi-dimensional grid of uncertain parameter values

---

**Note** gridunc will be removed in a future version.

---

**Syntax** `uncpar=gridunc('ParameterName',Values,...)`

**Description** `uncpar=gridunc('ParameterName',Values,...)` constructs a multi-dimensional grid that contains all value combinations of uncertain parameters *ParameterName*. *uncpar* contains the uncertain parameter values.

### Input Arguments

#### **ParameterName\_n**

Name of the uncertain parameter in a response optimization project.

#### **Values\_n**

Values of corresponding uncertain parameter *ParameterName\_n*:

- $\{first\_value, \dots, last\_value\}$  for vector- or matrix-valued parameters. Dimensions of each cell element of  $\{first\_value, \dots, last\_value\}$  must match the dimensions of the parameter.
- $\{first\_value, \dots, last\_value\}$  or  $[first\_value, \dots, last\_value]$  for scalar-valued parameters.

### Algorithms

For parameters with values specified as  $\{first\_value, \dots, last\_value\}$  or  $[first\_value, \dots, last\_value]$ , gridunc creates a hypercube which consists of:

- $2^S$  vertices, where  $S$  is the number of parameters. *first\_value* and *last\_value* of each parameter form the vertices of the hypercube.
- Number of samples equal to the product of number of values specified for each parameter.

## Alternatives

To create a grid of uncertain parameter values using the GUI:

- 1** In the Simulink model, double-click the Signal Constraint block to open the Block Parameters: Signal Constraint window.
- 2** In the Block Parameters window, select **Optimization > Uncertain Parameters** to open the Uncertain Parameters dialog box.
- 3** Select Grid as the **Sampling method**.
- 4** Specify the uncertain parameters and their values:
  - a** Click **Add** to open the Add Parameters dialog box.
  - b** Select the uncertain parameters and click **OK** to add them to the Uncertain Parameters dialog box.
  - c** Specify the values for the corresponding parameter in the **Sample Values** column.

## See Also

setunc | randunc

## How To

- “Sampling Methods for Uncertain Parameters”
- “Optimizing Parameters for Robustness”

**Purpose** Specify initial parameter values in response optimization project

---

**Note** `initpar` will be removed in a future version. For examples of the new optimization workflow, see:

- “Design Optimization to Meet Step Response Requirements (GUI)”
  - “Design Optimization to Meet Step Response Requirements (Code)”
- 

**Syntax** `initpar(proj)`

**Description** `initpar(proj)` updates the parameters’ initial guess in `proj` with the current parameter values in the model or base workspace. `proj` is a response optimization project, created using `getsro` or `newsro`.

Before using `initpar`, open the Simulink model associated with `proj`.

**Examples** Initialize a response optimization project with updated parameter values:

**1** Run an optimization to meet step response requirements.

```
% Open Simulink model.
sldo_model1_constrblk
% Create response optimization project to optimize
% parameters Kd, Ki and Kp.
proj=newsro('sldo_model1_constrblk',{'Kd','Ki','Kp'})
% Extract design requirements from Signal Constraint block.
constr=findconstr(proj,...
'sldo_model1_constrblk/Signal Constraint')
% Specify step response requirements.
constr.LowerBoundX=[0 10;10 15;15 50];
constr.LowerBoundY=[0 10;10 30;30 50];
constr.UpperBoundX=[0 30;30 50];
constr.UpperBoundY=[1.1 1.1;1.01 1.01];
% View initial value of Kd.
```

```
proj.parameter(1).InitialGuess
% Optimize parameters to meet step response requirements.
optimize(proj);
```

- 2 Specify a reference signal, and initialize proj with updated parameter values before running a new optimization.

```
% Specify reference signal.
constr.CostEnable='on';
constr.ReferenceX=linspace(0,50,200)';
constr.ReferenceY=1-exp(-0.3*linspace(0,50,200))';
% The values of Kd, Ki and Kp are not updated
% in proj. For example:
proj.parameter(1).InitialGuess
% Update proj with updated parameter values.
initpar(proj);
% View the updated initial value of Kd, which is now updated.
% The same is true for Ki and Kp.
proj.parameter(1).InitialGuess
% Rerun optimization to meet step requirements and
% track the reference signal simultaneously.
optimize(proj);
```

## See Also

[newsro](#) | [findpar](#) | [findconstr](#) | [optimize](#)

## Tutorials

- “Design Optimization to Meet Step Response Requirements (Code)”

**Purpose**

New response optimization project with default settings

---

**Note** newsro will be removed in a future version. For examples of the new optimization workflow, see:

- “Design Optimization to Meet Step Response Requirements (GUI)”
  - “Design Optimization to Meet Step Response Requirements (Code)”
- 

**Syntax**

```
proj=newsro('modelName',params)
```

**Description**

`proj=newsro('modelName',params)` creates a response optimization project *proj* for the Simulink model *modelName*. *params* is a cell array of strings that specifies model parameters for optimization. *proj* contains default design requirements, parameter settings, and optimization options in the Signal Constraint block. The model must contain at least one Signal Constraint block.

**Alternatives**

Create a response optimization project with default settings using the GUI:

- 1** Drag and drop a Signal Constraint block from the Simulink Design Optimization library into the Simulink model.
- 2** Connect the Signal Constraint block to the signal that must meet specific design requirements.
- 3** Double-click the Signal Constraint block to create a response optimization project with default settings.

**See Also**

`findconstr` | `findpar` | `getsro` | `optimize`

**Tutorials**

- “Design Optimization to Meet Step Response Requirements (Code)”

# optimget

---

**Purpose** Current optimizer settings

---

**Note** `optimget` will be removed in a future version. Use `sdo.OptimizeOptions` instead.

---

**Syntax** `opt_settings=optimget(proj)`

**Description** `opt_settings=optimget(proj)` returns the current optimization settings object, `opt_settings`, for the response optimization project `proj`.

Use `optimset` to modify the optimization options.

For more information on the settings and their possible values, see the `optimset` reference page.

**See Also** `optimset` | `simget` | `simset`

**Purpose**

Run response optimization project

---

**Note** `optimize` will be removed in a future version. Use `sdo.optimize` instead.

---

**Syntax**

```
result=optimize(proj)
```

**Description**

`result=optimize(proj)` optimizes the responses specified in the response optimization project, `proj`, with the constraints, parameters, and settings. The response optimization results are displayed after each iteration. The tuned parameters are changed in the workspace. Enter the parameter name at the MATLAB prompt to see its new value.

A results object, `result`, is also returned. The properties of this object are

- **Cost:** The final value of the cost function.
- **ExitFlag:** 1 if the optimization terminated successfully, 0 if it did not.
- **Iteration:** The number of iterations.

For more information on the results properties, see the reference pages for the Optimization Toolbox™ functions `fmincon` and `fminsearch` and the Global Optimization Toolbox function `patternsearch`.

**See Also**

`findconstr` | `findpar` | `getsro` | `newsro` | `optimget` | `optimset`

**Tutorials**

- “Design Optimization to Meet Step Response Requirements (Code)”

**How To**

- “How the Optimization Algorithm Formulates Minimization Problems”

# optimset

**Purpose** Modify optimization settings

---

**Note** optimset will be removed in a future version. Use sdo.OptimizeOptions instead.

---

**Syntax** optimset(proj, 'Property1', Value1, 'Property2', Value2, ...)

**Description** optimset(proj, 'Property1', Value1, 'Property2', Value2, ...) modifies the optimization settings within the response optimization project, proj. The value of the optimization setting, Property1, is set to Value1, Property2 is set to Value2, etc.

Property	Description	Possible Settings
Method	The optimization method used. The following methods are available: <ul style="list-style-type: none"><li>• fmincon — Optimization Toolbox function fmincon</li><li>• patternsearch — Global Optimization Toolbox function patternsearch</li><li>• fminsearch — Optimization Toolbox function fminsearch</li></ul>	{'fmincon'}   'patternsearch'   'fminsearch'
Algorithm	The algorithm used by the optimization method.	For fmincon optimization method: <ul style="list-style-type: none"><li>• {'active-set'}</li><li>• 'trust-region-reflective'</li><li>• 'interior-point'</li></ul>



Property	Description	Possible Settings
Display	<p>The level of information that the optimization displays:</p> <ul style="list-style-type: none"> <li>• <code>off</code> — No output</li> <li>• <code>iter</code> — Output at each iteration</li> <li>• <code>final</code> — Final output only</li> <li>• <code>notify</code> — Output only if the function does not converge</li> </ul>	'off'   {'iter'}   'final'   'notify'
GradientType	<p>Method used to calculate gradients when using 'fmincon' as the Method. Use one of the following finite difference methods for gradient calculation:</p> <ul style="list-style-type: none"> <li>• <code>basic</code> — Default method for computing the gradients</li> <li>• <code>refined</code> — Offers a more robust and less noisy gradient calculation method than 'basic'</li> </ul> <p>The <code>refined</code> method is sometimes more expensive, and does not work with certain models such as SimPowerSystems™ models.</p>	{'basic'}   'refined'

# optimset

Property	Description	Possible Settings
MaximallyFeasible	<p>Option to specify that the optimization continue after an initial, feasible solution has been found:</p> <ul style="list-style-type: none"><li>• 0 — Terminate the optimization as soon as an initial solution that satisfies the constraints is found. The resulting response signal may lie very close to the constraint segment.</li><li>• 1 — Continue the optimization after an initial solution is found. The optimization can continue to search for a maximally feasible solution that is typically located further inside the constraint region.</li></ul> <hr/> <p><b>Note</b> The software ignores this option when you track a reference signal.</p> <hr/>	{0}   1
MaxIter	Maximum number of iterations allowed	Positive integer value
TolCon	Termination tolerance on the constraints	Positive scalar value
TolFun	Termination tolerance on the function value	Positive scalar value
TolX	Termination tolerance on the parameter values	Positive scalar value

Property	Description	Possible Settings
Restarts	<p>In some optimizations, the Hessian may become ill-conditioned, and the optimization does not converge. In these cases, it is sometimes useful to restart the optimization after it stops, using the endpoint of the previous optimization as the starting point for the next one. To automatically restart the optimization, use this option to indicate the number of times you want to restart.</p>	Nonnegative integer value
UseParallel	<p>Parallel computing option for the following optimization methods:</p> <ul style="list-style-type: none"> <li>• fmincon</li> <li>• patternsearch</li> </ul> <hr/> <p><b>Note</b> Parallel Computing Toolbox software must be installed to enable parallel computing for the optimization methods.</p> <hr/> <p>When set to 'always', the methods compute the following in parallel:</p> <ul style="list-style-type: none"> <li>• fmincon — Computes finite difference gradients</li> </ul>	'always'   {'never'}

# optimset

---

Property	Description	Possible Settings
	<ul style="list-style-type: none"><li>• <code>patternsearch</code> — Performs population evaluation</li></ul> Disable the option by setting to 'never'.	
<code>ParallelPathDependencies</code>	Option to store model path dependencies when using parallel computing	Cell array of strings
<code>SearchMethod</code>	Search options for use with the <code>patternsearch</code> method	See “Search Options” in the Global Optimization Toolbox documentation.

For more information on the possible settings and the values they can take, see the reference page for `optimset` in the MATLAB documentation.

## See Also

`optimget` | `simget` | `simset`

**Purpose** Random values of uncertain parameters

---

**Note** randunc will be removed in a future version. For examples of specifying uncertain parameters, see:

- “Design Optimization with Uncertain Variables (Code)”
  - “Specify Custom Signal Objective with Uncertain Variable (GUI)”
- 

**Syntax** `uncpar=randunc(N,'ParameterName',Range,...)`

**Description** `uncpar=randunc(N,'ParameterName',Range,...)` generates random values of uncertain parameters, specified as comma separated 'ParameterName' and Range value pairs. Range specifies the lower and upper bounds for the uncertain parameter. Enter Range as a cell array {[Min],[Max]} for vector- and scalar-valued parameters or vector [Min,Max] for scalar-valued parameters. Dimensions of each cell element must match the corresponding parameter dimension. N is the number of samples inside the hypercube formed by Min and Max of each parameter. uncpair contains the uncertain parameter values.

**Algorithms** For parameters  $p$  with range specified as [Min,Max] or {[Min],[Max]}, randunc interprets the range of the uncertain parameters as:

$$\text{Min}(i,j) \leq p(i,j) \leq \text{Max}(i,j)$$

randunc generates a set of uncertain parameter values consists of the following:

- All vertices of the hypercube specified by Min and Max values of the parameters. The total number of vertices of the hypercube is  $2^S$ , where  $S$  is the number of uncertain parameters.
- $N$  random samples inside the hypercube.

**Alternatives** To generate random values of uncertain parameters using the GUI:

- 1** In the Simulink model, double-click the Signal Constraint block to open the Block Parameters: Signal Constraint window.
- 2** In the Block Parameters window, select **Optimization > Uncertain Parameters** to open the Uncertain Parameters dialog box.
- 3** Select Random (Monte Carlo) as the **Sampling method**.
- 4** Specify the number of samples in the **Number of samples** field.
- 5** Specify the uncertain parameters and their range:
  - a** Click **Add** to open the Add Parameters dialog box.
  - b** Select the uncertain parameters and click **OK** to add them to the Uncertain Parameters dialog box.
  - c** Specify the range for the corresponding parameter in the **Min** and **Max** columns.

## See Also

setunc | gridunc

## How To

- “Sampling Methods for Uncertain Parameters”
- “Optimizing Parameters for Robustness”

**Purpose**

Specify parameter uncertainty in response optimization project

---

**Note** setunc will be removed in a future version. For examples of specifying uncertain parameters, see:

- “Design Optimization with Uncertain Variables (Code)”
  - “Specify Custom Signal Objective with Uncertain Variable (GUI)”
- 

**Syntax**

setunc(proj,unc\_settings)

**Description**

setunc(proj,unc\_settings) sets the parameter uncertainty specifications for the response optimization project, proj. Use the function gridunc or randunc to specify the uncertainty settings, unc\_settings.

**See Also**

gridunc | randunc

# simget

---

**Purpose** Current simulation settings

---

**Note** `simget` will be removed in a future version.

---

**Syntax** `simoptions=simget('proj')`

**Description** `simoptions=simget('proj')` returns a object containing the current simulation settings, `simoptions`, used by the response optimization project, `proj`. To modify the project's simulation settings, use the `simset` function.

The project's simulation settings are a subset of the parameters that you can set for Simulink models and blocks. For a detailed list of these settings and the possible values they can take, see “About Model Parameters” in the Simulink documentation. The default values of the simulation settings are the same as those used by the Simulink model the project is associated with. Changes that are made to the project's simulation settings are only used during simulations that are run as part of the optimization, and they do not affect the simulation settings for the model.

**See Also** `optimget` | `optimset` | `simset`



**Purpose** Modify simulation settings

---

**Note** `simset` will be removed in a future version.

---

**Syntax** `simset(proj, 'setting1', value1, 'setting2', value2, ...)`

**Description** `simset(proj, 'setting1', value1, 'setting2', value2, ...)` modifies the simulation settings within the response optimization project, `proj`. The value of the simulation setting, `setting1`, is set to `value1`, `setting2` is set to `value2`, etc.

The project's simulation settings are a subset of the parameters that you can set for Simulink models and blocks. For a detailed list of these settings and the possible values they can take, see “About Model Parameters” in the Simulink documentation. The default values of the simulation options for the project are the same as those used by the Simulink model the project is associated with. Changes that are made to the project's simulation settings are only used during simulations that are run as part of the optimization, and they do not affect the simulation settings for the model.

**See Also** `optimget` | `optimset` | `simget`

# sdo.Experiment.createSimulator

---

**Purpose** Create simulation object from experiment to compare measured and simulated data

**Syntax**  
`sim_obj = createSimulator(experiment)`  
`sim_obj = createSimulator(experiment,sim_obj0)`

**Description** `sim_obj = createSimulator(experiment)` creates a `sdo.SimulationTest` object to simulate a model using the parameters and inputs specified in an experiment. You compare the simulated and measured outputs. `sim_obj` specifies the model stop time as the end time of the longest running experiment output signal.

`sim_obj = createSimulator(experiment,sim_obj0)` updates the values of the `Parameters`, `InitialStates`, `Input` and `LoggingInfo` properties of the `sdo.SimulationTest` object, `sim_obj0`. It does so using the corresponding properties specified by `experiment`. `sim_obj0.ModelName` must be the same as `experiment.ModelName`. You use this syntax to avoid creating a simulation scenario object (`sdo.SimulationTest` object) repeatedly and, instead, modify an existing simulation scenario object.

## Input Arguments

### **experiment**

Experiment, specified as an `sdo.Experiment` object.

### **sim\_obj0**

Simulation scenario, specified as an `sdo.SimulationTest` object.

Typically, you use the `createSimulator` method of an experiment to create `sim_obj0`, which returns an appropriately configured simulation scenario. You can construct `sim_obj0` using the syntax `sim_obj0 = sdo.SimulationTest(modelname)`. However, if you do so, then `sim_obj0.ModelName` must be the same as `experiment.ModelName`.

## Output Arguments

### **sim\_obj**

Simulation scenario, returned as an `sdo.SimulationTest` object.

The properties of `sim_obj` are configured to simulate the model associated with `experiment` using the parameters, initial states and inputs defined by `experiment`.

When you use the syntax `sim_obj = createSimulator(experiment,sim_obj0)`, `sim_obj` is the same object as `sim_obj0`. However, it contains the `Parameters`, `InitialStates`, and `Input` property values of `experiment`. The `LoggingInfo` property of `sim_obj` is extended to include any additional signals from `experiment.OutputData`.

## Examples

### Create Simulation Scenario for Experiment

Specify an experiment for a model.

```
experiment = sdo.Experiment('sdoRCCircuit');
```

Create a simulation scenario for the experiment.

```
sim_obj = createSimulator(experiment);
```

### Update Simulation Scenario for Experiment

Specify an experiment and a model parameter value for the experiment.

```
experiment = sdo.Experiment('sdoRCCircuit');
```

```
open_system('sdoRCCircuit');  
p = sdo.getParameterFromModel('sdoRCCircuit','C1');  
p.Value = 1e-6;  
p.Free = false;
```

```
experiment.Parameters = p;
```

Create a simulation scenario for the experiment.

```
sim_obj = createSimulator(experiment);  
sim_obj.Parameters.Value
```

# sdo.Experiment.createSimulator

---

```
ans =  
  
1.0000e-06
```

Modify the model parameter value for the experiment.

```
experiment.Parameters.Value = 2e-6;
```

Update the simulation scenario.

```
sim_obj = createSimulator(experiment,sim_obj);  
sim_obj.Parameters.Value
```

```
ans =  
  
2.0000e-06
```

The value of the model parameter associated with `sim_obj` is updated.

**See Also** [sdo.SimulationTest](#) |

## Related Examples

- “Estimate Model Parameter Values (Code)”
- “Estimate Model Parameters and Initial States (Code)”

# sdo.Experiment.getValuesToEstimate

---

**Purpose** Get model initial states and parameters for estimation from experiment

**Syntax** `parameters = getValuesToEstimate(experiment)`

**Description** `parameters = getValuesToEstimate(experiment)` returns the model initial states and parameters of an experiment that you want to estimate.

When you estimate parameters for multiple experiments, `getValuesToEstimate` tags each parameter to track its corresponding experiment. To update the experiments with their corresponding estimated parameter values, use `setEstimatedValues`.

## Input Arguments

### **experiment**

Experiment, specified as an `sdo.Experiment` object.

To get the model initial states and parameters for multiple experiments, use a vector of `sdo.Experiment` objects.

To specify that you want to estimate the value of a model initial state or parameter for an experiment, set its `Free` property to `true`. For example, `experiment.InitialStates(1).Free = true`.

## Output Arguments

### **parameters**

Model initial states and parameters of an experiment that you want to estimate, returned as a vector of `param.Continuous` objects.

When `experiment` specifies multiple experiments, `getValuesToEstimate` tags each entry of `parameters` to track its corresponding experiment. To update the experiments with their corresponding estimated parameter values, use `setEstimatedValues`.

# sdo.Experiment.getValuesToEstimate

---

## Examples

### Get Model Initial States and Parameters to Estimate from Experiment

Specify an experiment with a model initial state and parameter that you want to estimate.

```
experiment = sdo.Experiment('sdoRCCircuit');  
  
experiment.InitialStates = sdo.getStateFromModel('sdoRCCircuit','C1');  
  
open_system('sdoRCCircuit');  
experiment.Parameters = sdo.getParameterFromModel('sdoRCCircuit','C1');
```

Get the model initial states and parameters that you want to estimate from the experiment.

```
val = getValuesToEstimate(experiment)  
  
val(1,1) =  
  
    Name: 'sdoRCCircuit/C1:sdoRCCircuit.C1.vc_0'  
    Value: 0  
    Minimum: -Inf  
    Maximum: Inf  
    Free: 1  
    Scale: 1  
    dxValue: 0  
    dxFree: 1  
    Info: [1x1 struct]  
  
val(2,1) =  
  
    Name: 'C1'  
    Value: 1.0000e-03  
    Minimum: -Inf  
    Maximum: Inf  
    Free: 1
```

```
Scale: 0.0020  
Info: [1x1 struct]
```

```
2x1 param.Continuous
```

```
lists of methods, superclasses
```

`val(1,1)`, the initial voltage of the model capacitor block, C1, is the initial state specified by experiment for estimation. Execute `class(val(1,1))` to see that `val(1,1)` is a `param.State` object, representing a model initial state.

`val(2,1)`, the capacitance of the C1 block, is the model parameter specified by experiment for estimation.

## See Also

`sdo.Experiment` | `setEstimatedValues` |

## Related Examples

- “Estimate Model Parameters using Multiple Experiments (Code)”
- “Estimate Model Parameters Per Experiment (Code)”

# sdo.Experiment.setEstimatedValues

---

**Purpose** Update experiments with estimated model initial states and parameter values

**Syntax** `experiment = setEstimatedValue(experiment0,parameters)`

**Description** `experiment = setEstimatedValue(experiment0,parameters)` updates the experiment with the estimated model initial states and parameter values.

`setEstimatedValues` is used with the `getValuesToEstimate` method. You use `getValuesToEstimate` to obtain the parameters that you want to estimate from an experiment. When you estimate parameters for multiple experiments, `getValuesToEstimate` tags each parameter to track its corresponding experiment. You use `setEstimatedValues` to update the experiments with their corresponding estimated parameter values.

## Input Arguments

### **experiment0**

Experiment, specified as an `sdo.Experiment` object.

To specify multiple experiments, use a vector of `sdo.Experiment` objects.

### **parameters**

Estimated model initial states and parameters for experiments, specified as a vector of `param.Continuous` objects.

You obtain estimated parameters using `sdo.optimize`.

## Output Arguments

### **experiment**

Updated experiment, returned as an `sdo.Experiment` object.

If `experiment0` is a vector of experiments, then `experiment` is a corresponding vector of updated `sdo.Experiment` objects.

`setEstimatedValues` updates the values of the parameters and initial states specified in each of the experiments in `experiment0` using the corresponding entry in `parameters`.



## Examples

### Update Experiment with Estimated Parameter Value

Specify an experiment.

```
experiment = sdo.Experiment('sdoRCCircuit');
```

Typically, you also specify measured input/output data for the experiment.

Specify a model parameter for estimation.

```
sdoRCCircuit;  
C1_parameter = sdo.getParameterFromModel('sdoRCCircuit','C1');  
C1_parameter.Value = 460e-6;
```

```
experiment.Parameters = C1_parameter;
```

C1\_parameter is the capacitance parameter of the C1 block. The initial guess for its value is 460  $\mu\text{F}$ .

Estimate the parameter value.

Typically, you use `sdo.optimize` to get the estimated parameter values for an experiment. For this example, directly change the value of the capacitance parameter.

```
C1_parameter.Value = 1e-6;
```

Update the experiment with the estimated parameter.

```
experiment = setEstimatedValues(experiment,C1_parameter);
```

Use `experiment.Parameters.Value` to verify that the capacitance parameter's value is updated.

**See Also** `sdo.Experiment.getValuesToEstimate` |

## Related Examples

- “Estimate Model Parameters using Multiple Experiments (Code)”
- “Estimate Model Parameters Per Experiment (Code)”

# sdo.getModelDependencies

---

**Purpose** List of model path dependencies

**Syntax** `dirs = sdo.getModelDependencies(modelname)`

**Description** `dirs = sdo.getModelDependencies(modelname)` returns paths containing model dependencies required for response optimization using parallel computing. The model must be open.

`sdo.getModelDependencies` does not return a complete list of model dependency paths when the dependencies are undetectable.

**Tips**

- Modify `dirs` to include:
  - Path dependencies that `sdo.getModelDependencies` cannot detect such as paths that contain code for your model callbacks
  - Paths that the workers cannot access directly such as path dependencies on local drive
- Use `dirs` to set the `ParallelPathDependencies` property of `sdo.OptimizeOptions` to use for parallel computing.

**Input Arguments**

**modelname**  
Simulink model name that uses the parameter, specified as a string inside single quotes (' ').

**Output Arguments**

**dirs**  
Cell array of paths that contain model dependencies.

The cell array is empty when the model does not have any dependencies or `sdo.getModelDependencies` does not detect any dependencies.

**Examples**

List model path dependencies required for parallel computing.

Open Simulink model.

```
% Copy Simulink boiler library to temporary folder.  
pathToLib = boilerpressure_setup;
```

```
% Add folder to search path.
addpath(pathToLib);
% Open Simulink model.
boilerpressure_demo

Enable parallel computing.

opts = sdo.OptimizeOptions;
opts.UseParallel = 'always';

Get model dependency paths.

dirs = sdo.getModelDependencies('boilerpressure_demo');

Add model dependencies to use during parallel computing.

opts.ParallelPathDependencies = dirs;
```

---

```
Make local paths accessible to remote workers.

Open Simulink model.

% Copy Simulink boiler library to temporary folder.
pathToLib = boilerpressure_setup;
% Add folder to search path.
addpath(pathToLib);
% Open Simulink model.
boilerpressure_demo

Enable parallel computing.

opts = sdo.OptimizeOptions;
opts.UseParallel = 'always';

Get model dependency paths.

dirs = sdo.getModelDependencies('boilerpressure_demo');
```

## sdo.getModelDependencies

---

The resulting path is on a local drive, C:/.

Replace C:/ with valid network path accessible to remote workers.

```
dirs = regexprep(dirs, 'C:/', '\\\hostname\C$\')
```

Add model dependencies to use during parallel computing.

```
opts.ParallelPathDependencies = dirs;
```

---

Add path to model path dependency list.

Open Simulink model.

```
% Copy Simulink boiler library to temporary folder.  
pathToLib = boilerpressure_setup;  
% Add folder to search path.  
addpath(pathToLib);  
% Open Simulink model.  
boilerpressure_demo
```

Enable parallel computing.

```
opts = sdo.OptimizeOptions;  
opts.UseParallel = 'always';
```

Get model dependency paths.

```
dirs = sdo.getModelDependencies('boilerpressure_demo');
```

Append an additional path.

```
dirs=vertcat(dirs, '\\hostname\C$\matlab\work')
```

Add model dependencies to use during parallel computing.

```
opts.ParallelPathDependencies = dirs;
```

**Alternatives**    “How to Use Parallel Computing”

**See Also**        `sdo.optimize`

**Tutorials**        • Improving Optimization Performance Using Parallel Computing

**How To**            • “Speedup Using Parallel Computing”  
                          • “Analyze Model Dependencies”

# sdo.getParameterFromModel

---

**Purpose** Design variable for optimization

**Syntax**  
`p_des = sdo.getParameterFromModel(modelname,paramname)`  
`p_des = sdo.getParameterFromModel(modelname)`

**Description** `p_des = sdo.getParameterFromModel(modelname,paramname)` creates an object from a Simulink model parameter that you can tune to satisfy design requirements during optimization. The model must be open.

`p_des = sdo.getParameterFromModel(modelname)` creates model parameter objects for all the parameters in the model.

## Input Arguments

### **modelname**

Simulink model name that uses the parameter, specified as a string inside single quotes ( ' ').

### **paramname**

Model parameter name, specified as a string inside single quotes ( ' ') for one parameter or a cell array of strings for multiple parameters.

## Output Arguments

### **p\_des**

A param.Continuous object for one parameter or an array of objects for multiple parameters.

If paramname is not specified, then p\_des contains all the parameters of the model.

The Value property of the object is set to the current value of the model parameter.

## Examples

### **Get Model Parameter as Optimization Design Variable**

```
modelname = 'sldo_model1_stepblk';  
paramname = 'Kp';
```

```
sldo_model1_stepblk;  
  
p_des = sdo.getParameterFromModel(modelname,paramname);
```

## **Get Multiple Model Parameters as Optimization Design Variables**

```
modelname = 'sldo_model1_stepblk';  
paramname = {'Kp','Ki','Kd'};
```

```
sldo_model1_stepblk;  
  
p_des = sdo.getParameterFromModel(modelname,paramname);
```

## **Get All Model Parameters as Optimization Design Variables**

```
modelname = 'sldo_model1_stepblk';  
  
sldo_model1_stepblk;  
  
p_des = sdo.getParameterFromModel(modelname);
```

### **Alternatives**

“Specify Design Variables”

### **See Also**

[sdo.optimize](#) | [sdo.setValueInModel](#)

### **Tutorials**

- “Design Optimization to Meet Step Response Requirements (Code)”
- “Estimate Model Parameter Values (Code)”

# sdo.getStateFromModel

---

## Purpose

Initial state for estimation from Simulink model

## Syntax

```
s = sdo.getStateFromModel(modelname,blockpath)
s = sdo.getStateFromModel(modelname)
```

## Description

`s = sdo.getStateFromModel(modelname,blockpath)` creates a state parameter object for the state of a specified block in a Simulink model. Use the state object to either specify the initial-state value in an experiment or estimate it.

`s = sdo.getStateFromModel(modelname)` creates state parameter objects for all the states in the model.

## Input Arguments

### **modelname**

Simulink model name, specified as a string inside single quotes ( ' '). The model must be open.

### **blockpath**

Block path of the block containing the required state, specified as a string inside single quotes ( ' ').

To specify multiple blocks, use a cell array of block path strings.

## Output Arguments

### **s**

Model state, returned as a `param.State` object.

`s.Value` is the initial value of the state in the model.

When you use the syntax `s =`

`sdo.getStateFromModel(modelname,blockpath)`, `s` contains the state of the corresponding block.

If `blockpath` specifies multiple blocks, then `sdo.getStateFromModel` returns a vector of `param.State` objects.



## Examples

### Get States from Model

```
modelname = 'sdoAircraft';  
open_system(modelname);  
  
blockpath = {'sdoAircraft/Actuator Model', ...  
            'sdoAircraft/Controller/Proportional plus integral compensator'};  
  
s = sdo.getStateFromModel(modelname,blockpath);
```

### Get All Model States

```
modelname = 'sdoAircraft';  
open_system(modelname);  
  
s = sdo.getStateFromModel(modelname);
```

s is a vector containing nine `param.State` objects, which represent all the states of the `sdoAircraft` model.

## See Also

`sdo.Experiment` | `param.State` |

## Related Examples

- “Estimate Model Parameter Values (Code)”
- “Estimate Model Parameters and Initial States (Code)”

# sdo.getValueFromModel

---

<b>Purpose</b>	Get design variable value from model
<b>Syntax</b>	<code>param_value = sdo.getValueFromModel(modelname,param_des)</code>
<b>Description</b>	<code>param_value = sdo.getValueFromModel(modelname,param_des)</code> gets the value of a design variable in a Simulink model. The model must be open.
<b>Input Arguments</b>	<p><b>modelname</b> Simulink model name that uses the parameter, specified as a string inside single quotes (' ').</p> <p><b>param_des</b> Design variables, specified as:</p> <ul style="list-style-type: none"><li>• A string inside single quotes (' ') for one variable or a cell array of strings for multiple variables</li><li>• A <code>param.Continuous</code> object for one variable or a vector of objects for multiple variables, created using <code>sdo.getParameterFromModel</code></li></ul>
<b>Output Arguments</b>	<p><b>param_value</b> Design variable value in the model. A cell array for multiple variable values.</p>
<b>Examples</b>	<p>Get current design variable value from a model.</p> <pre>sldo_model1_stepblk; p_value = sdo.getValueFromModel('sldo_model1_stepblk','Kp');</pre> <p>Alternatively, type</p> <pre>p_des = sdo.getParameterFromModel('sldo_model1_stepblk','Kp'); p_value = sdo.getValueFromModel('sldo_model1_stepblk',p_des);</pre>
<b>See Also</b>	<code>sdo.optimize</code>

**Purpose** Determine if parameter value, minimum and maximum are real

**Syntax** `isreal(param_obj)`

**Description** `isreal(param_obj)` returns true (1) if the Value, Minimum and Maximum properties of `param_obj` are all real.

**Input Arguments** **param\_obj**  
A `param.Continuous` object.

**Examples** Determine if the value, maximum and minimum of a parameter are all real.

```
p= param.Continuous('K',eye(2));  
isreal(p)
```

Because the Value, Minimum and Maximum properties of all parameters in `p` are all real, `isreal` returns 1.

**See Also** `param.Continuous`

# sdo.setCheckBlockEnabled

---

**Purpose** Enable or disable all check blocks in model

**Syntax** `chk_blk_state = sdo.setCheckBlockEnabled(modelname, state)`

**Description** `chk_blk_state = sdo.setCheckBlockEnabled(modelname, state)` sets the Enabled parameter of all the check blocks in an open Simulink model to the specified value. The function returns the original value of the Enabled parameter of all the model check blocks.

Use this function to disable the check blocks (model verification blocks) in a model before running an optimization for the model. After optimization completes, you can restore the enabled state of the model check blocks by calling this function again. Use the output from the previous call as the second input for the function.

## Input Arguments

### **modelName**

Simulink model name, specified as a string inside single quotes (' '). The model must be open.

### **state**

Switch enabling or disabling model check blocks, specified as either 'on' or 'off'.

To restore the enabled state of the model check blocks, specify **state** as the output from the previous call to `sdo.setCheckBlockEnabled`.

## Output Arguments

### **chk\_blk\_state**

Original values of the Enabled block parameter of the model check blocks, returned as a cell array of strings.

## Examples

### **Disable Model Check Blocks**

Disable the model check blocks in a model.

```
modelName = 'pidtune_demo';
```

```
open_system(modelname);  
  
state = 'off';  
  
chkBlkState = sdo.setCheckBlockEnabled(modelname, state);
```

To restore the enabled state of the model check blocks, use:

```
sdo.setCheckBlockEnabled(modelname, chkBlkState)
```

### **Alternatives**

You can open each model verification block in a model and select or clear the **Enable assertion** check box.

# sdo.setValueInModel

---

## Purpose

Set design variable value in model

## Syntax

```
sdo.setValueInModel(modelname,param_des)
sdo.setValueInModel(modelname,param_des,value)
```

## Description

`sdo.setValueInModel(modelname,param_des)` sets the value of a parameter in an open Simulink model to the Value property of the design variable `param_des`.

You generally use this command to update the Simulink model with optimized parameter values.

`sdo.setValueInModel(modelname,param_des,value)` sets the parameter to the value you specify.

## Input Arguments

### **modelname**

Simulink model name that uses the parameter, specified as a string inside single quotes ( ' ').

### **param\_des**

Design variable, specified as

- A `param.Continuous` object for one variable or a vector of objects for multiple variables, created using `sdo.getParameterFromModel`.
- A string inside single quotes ( ' ') for one variable or a cell array of strings for multiple variables.

You must also specify the `value` argument.

### **value**

Value to set for the design variable.

Use a cell array with the same number of elements as the number of variables in `param_des` for setting values of multiple design variables.

## Examples

Change the design variable value in a model.

```
sldo_model1_stepblk;  
p_des = sdo.getParameterFromModel('sldo_model1_stepblk','Kp');  
p_des.Value = 1.1*p_des.Value;  
sdo.setValueInModel('sldo_model1_stepblk',p_des);
```

The value of Kp is set to the Value property of p\_des.

## Alternatives

“Update Model with Design Variables Set”

## See Also

sdo.optimize

## Tutorials

- “Design Optimization to Meet Step Response Requirements (Code)”
- “Design Optimization to Track Reference Signal (GUI)”
- “Design Optimization Using Frequency-Domain Check Blocks (GUI)”

# sdo.optimize

---

**Purpose** Design optimization problem solution

**Syntax**  
[param\_opt,opt\_info] = sdo.optimize(opt\_fcn,param)  
[param\_opt,opt\_info] = sdo.optimize(opt\_fcn,param,options)  
[param\_opt,opt\_info] = sdo.optimize(prob)

**Description** [param\_opt,opt\_info] = sdo.optimize(opt\_fcn,param) uses fmincon to solve a design optimization problem of the form:

$$\min_p F(p)$$

subject to

$$\begin{aligned} C_{leq}(p) &\leq 0, \\ C_{eq}(p) &= 0, \\ A \times p &\leq B, \\ A_{eq} \times p &= B_{eq}, \\ lb &\leq p \leq ub \end{aligned}$$

where

- $p$  is the design variable
- $C_{leq}$  is the nonlinear inequality constraint
- $C_{eq}$  is the nonlinear equality constraint
- $A, B$  are the linear inequality constraint
- $A_{eq}, B_{eq}$  are the linear equality constraints
- $lb$  is the upper bound on  $p$
- $ub$  is the lower bound on  $p$

[param\_opt,opt\_info] = sdo.optimize(opt\_fcn,param,options) uses optimization options specified in options.



[param\_opt,opt\_info] = sdo.optimize(prob) uses a structure that contains the function to be minimized, design variables and optimization options.

## Input Arguments

### opt\_fcn

Function to be minimized. The optimization solver calls this function during optimization.

The function requires:

- One input argument, which is a vector of param. Continuous objects to be tuned.

To pass additional input arguments, use an anonymous function. For example, new\_fcn = @(p) fcn(p,arg1,arg2, ...).

- One output argument, which is a structure with one or more of the following fields:
  - F — Value of the cost function evaluated at p. The solver minimizes F.  
F is a 1x1 double.
  - Cleq — Value of the nonlinear inequality constraint violations evaluated at p. The solver satisfies Cleq(p) <= 0.  
Cleq is a double mx1 vector, where m is the number of nonlinear inequality constraints.
  - Ceq — Value of the nonlinear equality constraint violations evaluated at p. The solver satisfies Ceq(p) == 0.  
The value is a double rx1 vector, where r is the number of nonlinear equality constraints.
  - leq — Value of the linear inequality constraint violations evaluated at p. The solver satisfies leq(p) <= 0.  
leq is a double nx1 vector, where n is the number of linear inequality constraints.

- `eq` — Value of the linear equality constraint violations evaluated at `p`. The solver satisfies `eq(p) == 0`.

`eq` is a double `sx1` vector or `[]`, where `s` is the number of linear equality constraints.

To specify a pure feasibility problem, omit `F` or set `F = []`. To specify a minimization problem, omit `C1eq`, `Ceq`, `leq` and `eq` or set their values to `[]`.

The software computes gradients of the cost and constraint violations using numeric perturbation. If you want to specify how the gradients are computed, include a second output argument and set the `GradFcn` property of `sdo.OptimizeOptions` to `'on'`. This argument must be a structure with one or more of the following fields:

- `F` — Double `nx1` vector that contains  $dF(p)/dp$ , where `n` is the number of scalar parameters.
- `C1eq` — Double `nxm` matrix that contains  $dC1eq(p)/dp$ , where `m` is the number of nonlinear inequality constraints.
- `Ceq` — Double `nxr` matrix that contains  $dCeq(p)/dp$ , where `r` is the number of nonlinear equality constraints.

For an example, type `edit sdoExampleCostFunction`.

## **param**

A `param.Continuous` object or a vector of objects

## **options**

Optimization options, specified using `sdo.OptimizeOptions`

## **prob**

Structure with the following fields:

- `OptFcn` — Name of the function to be minimized. See `opt_fcn` for the input and output argument requirements of this function.
- `Parameters` — A `param.Continuous` object or a vector of objects

## Output Arguments

- **Options** — Optimization options, specified using `sdo.OptimizeOptions`

### **param\_opt**

A `param.Continuous` object or vector of objects, containing the optimized parameter values in the `Value` property.

### **opt\_info**

Optimization information. Structure with one or more of the following fields:

- **F** — Optimized cost (objective) value.
- **C1eq** — Optimized nonlinear inequality constraint violations.

The field appears if you specify a nonlinear inequality constraint in `opt_fcn`.

The value is a  $m \times 1$  vector, where the order of the elements correspond to the order specified in `opt_fcn`. Positive values indicate that the constraint has not been satisfied. Check `exitflag` to confirm that the optimization succeeded.

- **Ceq** — Optimized nonlinear equality constraint violations.

The field appears if you specify a nonlinear equality constraint in `opt_fcn`.

The value is a double  $r \times 1$  vector, where the order of the elements correspond to the order specified in `opt_fcn`. Any nonzero values indicate that the constraint has not been satisfied. Check `exitflag` to confirm that the optimization succeeded.

- **1eq** — Optimized linear equality constraint violations.

The field appears if you specify a linear equality constraint in `opt_fcn`.

The value is a double  $n \times 1$  vector, where the order of the elements correspond to the order specified in `opt_fcn`. Nonzero values

indicate that the constraint has not been satisfied. Check `exitflag` to confirm that the optimization succeeded.

- `eq` — Optimized linear equality constraint violations.

The field appears if you specify linear equality constraints in `opt_fcn`.

The value is a double `sx1` vector, where the order of the elements correspond to the order specified in `opt_fcn`. Nonzero values indicate that the constraint has not been satisfied. Check `exitflag` to confirm that the optimization succeeded.

- `Gradients` — Cost and constraint gradients at the optimized parameter values. See “How the Optimization Algorithm Formulates Minimization Problems” on how the solver computes gradients.

This field appears if the solver specified in the `Method` property of `sdo.OptimizeOptions` computes gradients.

The value is a structure whose fields are dependent on `opt_fcn`.

- `exitflag` — Integer identifying the reason the algorithm terminated. See `fmincon`, `patternsearch` and `fminsearch` for a list of the values and the corresponding termination reasons.
- `iterations` — Number of optimization iterations
- `SolverOutput` — A structure with solver-specific output information. The fields of this structure depends on the optimization solver specified in the `Method` property of `sdo.OptimizeOptions`. See `fmincon`, `patternsearch` and `fminsearch` for a list of solver outputs and their description.
- `Stats` — A structure that contains statistics collected during optimization, such as start and end times, number of function evaluations and restarts.

## Examples

### Optimize Model Response

Create design variables.

```
p = param.Continuous('x',1);
```

Specify optimization options.

```
opts = sdo.OptimizeOptions;
opts.GradFcn = 'on';
```

Optimize the parameter.

```
[pOpt,opt_info] = sdo.optimize(@(p) sdoExampleCostFunction(p),p,opts);
```

## Tips

- By default, the software displays the optimization information for each iteration in the MATLAB command window. To learn more about the information displayed, see:
  - “Iterative Display” when the optimization method is specified as 'fmincon' (default), 'fminsearch', or 'lsqnonlin'
  - “Display to Command Window Options” when the optimization method is specified as 'patternsearch'

You can configure the level of this display using the `MethodOptions.Display` property of an optimization option set.

## Alternatives

“Design Optimization to Meet Step Response Requirements (GUI)”  
 |  
 •  
 •

## See Also

`sdo.OptimizeOptions` | `function_handle` (@) `param.Continuous` |

## Related Examples

- “Design Optimization to Meet Step Response Requirements (Code)”
- “Design Optimization to Meet a Custom Objective (Code)”
- “Estimate Model Parameter Values (Code)”

## Concepts

- “Optimization Options”
- “Writing a Cost Function”

# sdo.SimulationTest.find

---

**Purpose** Find logged data set

**Syntax** `data = find(sim_obj,data_name)`

**Description** `data = find(sim_obj,data_name)` searches for an element with a specific name in the `LoggedData` property of `sim_obj`. Use `who` to find possible names.

**Input Arguments**

**sim\_obj**  
sdo.SimulationTest object

**data\_name**  
Data set name to search for, specified as a string inside single quotes (' ').

**Output Arguments**

**data**  
Logged simulation data for the data set name specified in `data_name`.

**Examples** Find logged data set.

Log model signals.

```
Pressures = Simulink.SimulationData.SignalLoggingInfo;  
Pressures.BlockPath = 'sdoHydraulicCylinder/Cylinder Assembly';  
Pressures.OutputPortIndex = 1;  
simulator = sdo.SimulationTest('sdoHydraulicCylinder');  
simulator.LoggingInfo.Signals = Pressures;
```

Run a simulation

```
sim = sim(simulator)
```

Search for logged data.

```
sim_log = find(simulator,'sdoHydraulicCylinder');
```

## See Also

`sdo.optimize` | `sim` | `who`

## Tutorials

- “Design Optimization to Meet Step Response Requirements (Code)”
- “Design Optimization to Meet a Custom Objective (Code)”

# sdo.SimulationTest.sim

---

**Purpose** Simulate Simulink model using simulation scenario

**Syntax** `sim_out = sim(sim_obj)`

**Description** `sim_out = sim(sim_obj)` simulates a Simulink model using a simulation scenario.

**Tips**

- Before simulating the model, specify the parameter values and signals to log in the `Parameters` and `LoggingInfo` properties of the `sim_obj`. The software restores the parameter values and logging settings to their original values after simulation.

**Input Arguments**

**sim\_obj**  
sdo.SimulationTest object

**Output Arguments**

**sim\_out**  
sdo.SimulationTest object which contains the logged data in the `LoggedData` property.

**Examples** Simulate a model and log model signal during simulation.

Log model signals.

```
Pressures = Simulink.SimulationData.SignalLoggingInfo;  
Pressures.BlockPath = 'sdoHydraulicCylinder/Cylinder Assembly';  
Pressures.OutputPortIndex = 1;  
simulator = sdo.SimulationTest('sdoHydraulicCylinder');  
simulator.LoggingInfo.Signals = [Pressures];
```

Specify parameter values.

```
Ac = sdo.getParameterFromModel('sdoHydraulicCylinder','Ac');  
Ac.Value = 0.5;  
simulator.Parameters = Ac;
```

Simulate the model.



```
sim_obj = sim(simulator);
```

The specified signal Pressure is logged during simulation.

### See Also

`sdo.optimize` | `find` | `who`

### Tutorials

- “Design Optimization to Meet Step Response Requirements (Code)”
- “Design Optimization to Meet a Custom Objective (Code)”

# sdo.SimulationTest.who

---

<b>Purpose</b>	List logged data names
<b>Syntax</b>	<code>names = who(sim_obj)</code>
<b>Description</b>	<code>names = who(sim_obj)</code> returns a list of logged data names.
<b>Input Arguments</b>	<b>sim_obj</b> sdo.SimulationTest object
<b>Output Arguments</b>	<b>names</b> Cell array of logged data set names.
<b>Examples</b>	<p>List logged data set names.</p> <p>Log model signals.</p> <pre>Pressures = Simulink.SimulationData.SignalLoggingInfo; Pressures.BlockPath = 'sdoHydraulicCylinder/Cylinder Assembly'; Pressures.OutputPortIndex = 1;</pre> <p>Store logged signal data.</p> <pre>simulator = sdo.SimulationTest('sdoHydraulicCylinder'); simulator.LoggingInfo.Signals = Pressures; simulator = sim(simulator);</pre> <p>Find logged data sets.</p> <pre>names = who(simulator);</pre>
<b>See Also</b>	<code>sdo.optimize</code>   <code>find</code>   <code>sim</code>
<b>Tutorials</b>	<ul style="list-style-type: none"><li>• “Design Optimization to Meet Step Response Requirements (Code)”</li><li>• “Design Optimization to Meet a Custom Objective (Code)”</li></ul>

---

<b>Purpose</b>	Open Design Optimization tool
<b>Syntax</b>	<code>sdotool(modelname)</code> <code>sdotool(sdosession)</code>
<b>Description</b>	<code>sdotool(modelname)</code> opens the Design Optimization tool and creates a new session. The model must be open or on the MATLAB path. <code>sdotool(sdosession)</code> opens a previously saved Design Optimization tool session.
<b>Tips</b>	<ul style="list-style-type: none"><li>• <code>sdotool</code> also updates Signal Constraint blocks in the model to the equivalent blocks from the <b>Signal Constraints</b> block library.</li></ul>
<b>Input Arguments</b>	<b>modelname</b> Simulink model name, specified as a string inside single quotes (' ').  <b>sdosession</b> Design Optimization tool session variable, saved in a MAT-file, model or MATLAB workspace.
<b>Examples</b>	<b>Create a New Design Optimization Tool Session</b> <pre>sdotool('pidtune_demo');</pre> <b>Open Design Optimization Tool Using a Saved Session</b> <pre>load sdoAircraft_sdosession; sdotool(SDOSessionData);</pre> SDOSessionData is the Design Optimization tool session variable saved in the sdoAircraft_sdosession.mat file.
<b>Tutorials</b>	<ul style="list-style-type: none"><li>• “Design Optimization to Meet a Custom Objective (GUI)”</li></ul>

# sdouupdate

---

**Purpose** Update model containing Signal Constraint block

**Syntax**  
`sdouupdate(modelname)`  
`sdouupdate(modelname,noprompt)`  
`session = sdouupdate(modelname)`

**Description** `sdouupdate(modelname)` replaces Signal Constraint blocks in a Simulink model with equivalent blocks from the **Signal Constraints** library. If the model has an associated response optimization project, this command replaces it with a session that you can use with the Design Optimization tool, after prompting you to update. The model must be open.

`sdouupdate(modelname,noprompt)` updates the response optimization project without prompting you.

`session = sdouupdate(modelname)` returns the Design Optimization tool session.

**Input Arguments**

**modelname**  
Simulink model name that uses the parameter, specified as a string inside single quotes (' ').

**noprompt**  
Whether to prompt you about updating the response optimization project (`false`) or not (`true`).

**Default:** `false`

**Output Arguments**

**session**  
Design Optimization tool session name.

**See Also** `sdotool`

**Purpose** Create Estimation Task in Control and Estimation Tools Manager GUI

**Syntax** `spetool('modelname')`

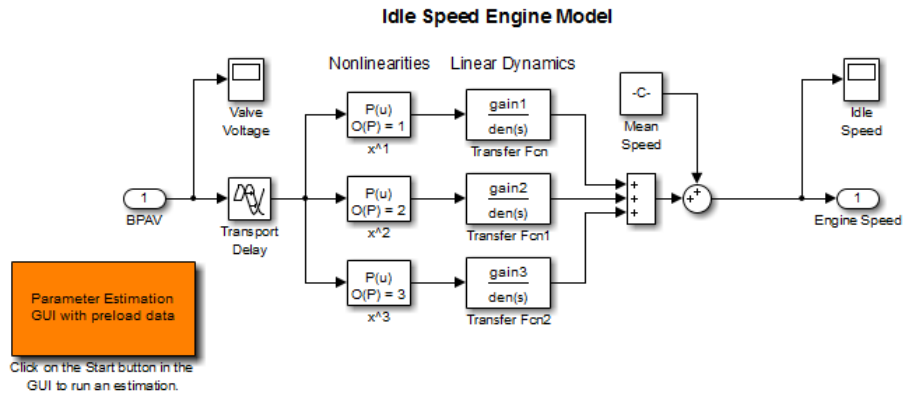
**Description** `spetool('modelname')` opens the Simulink model with the name `modelname` and creates an estimation task in the Control and Estimation Tools Manager GUI.

**Examples** Create an estimation task by typing the following command at the MATLAB prompt:

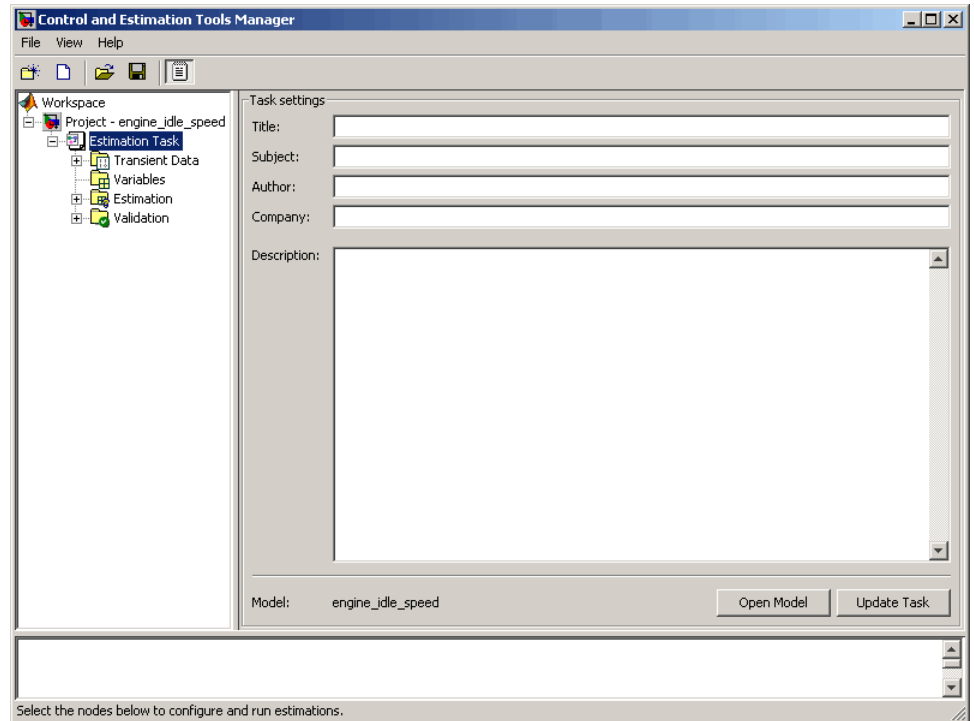
```
spetool('engine_idle_speed')
```

This command opens the following:

- Simulink model



- Control and Estimation Tools Manager containing a project with an estimation task



## How To

- “Import Data (GUI)”

## Symbols and Numerics

1D adaptive lookup table 1-2

## C

CRMS block 1-77

## D

DRMS block 1-78

## E

enable assertion 1-22 1-45 1-69

Estimation Task 3-83

## F

findconstr function 3-20

finddepend function 3-21

findpar function 3-24

## G

getsro function 3-30

gridunc function 3-31

## I

initpar function 3-33

## N

newsro function 3-35

## O

optimget function 3-36

optimize function 3-37

optimset function 3-38

output assertion signal 1-26 1-49 1-73

## R

randunc function 3-43

## S

setunc function 3-45

show plot on block open 1-27 1-50 1-74

Signal Constraint block 1-79

simget function 3-46

simset function 3-47

simulation callback 1-24 1-47 1-70

Stop simulation when assertion fails 1-25 1-48  
1-72