

# Simulink<sup>®</sup> Design Optimization<sup>™</sup> Reference



# MATLAB<sup>®</sup>&SIMULINK<sup>®</sup>

R2016a



## How to Contact MathWorks



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



Phone: 508-647-7000



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

*Simulink<sup>®</sup> Design Optimization<sup>™</sup> Reference*

© COPYRIGHT 1998–2016 by The MathWorks, Inc.

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

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

### **Trademarks**

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

### **Patents**

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

**Revision History**

September 2011	Online only
March 2012	Online only
September 2012	Online only
March 2013	Online only
September 2013	Online only
March 2014	Online only
October 2014	Online only
March 2015	Online only
September 2015	Online only
March 2016	Online only

New for Version 2.0 (Release R2011b)
Revised for Version 2.1 (Release R2012a)
Revised for Version 2.2 (Release R2012b)
Revised for Version 2.3 (Release R2013a)
Revised for Version 2.4 (Release R2013b)
Revised for Version 2.5 (Release 2014a)
Revised for Version 2.6 (Release 2014b)
Revised for Version 2.7 (Release 2015a)
Revised for Version 2.8 (Release 2015b)
Revised for Version 3.0 (Release 2016a)



<b>1</b>	<b><u>Blocks — Alphabetical List</u></b>
<b>2</b>	<b><u>Class Reference</u></b>
<b>3</b>	<b><u>Alphabetical List</u></b>



# Blocks — Alphabetical List

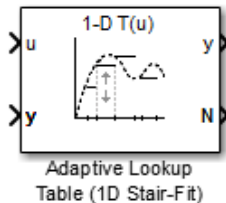
---

# Adaptive Lookup Table (1D Stair-Fit)

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.

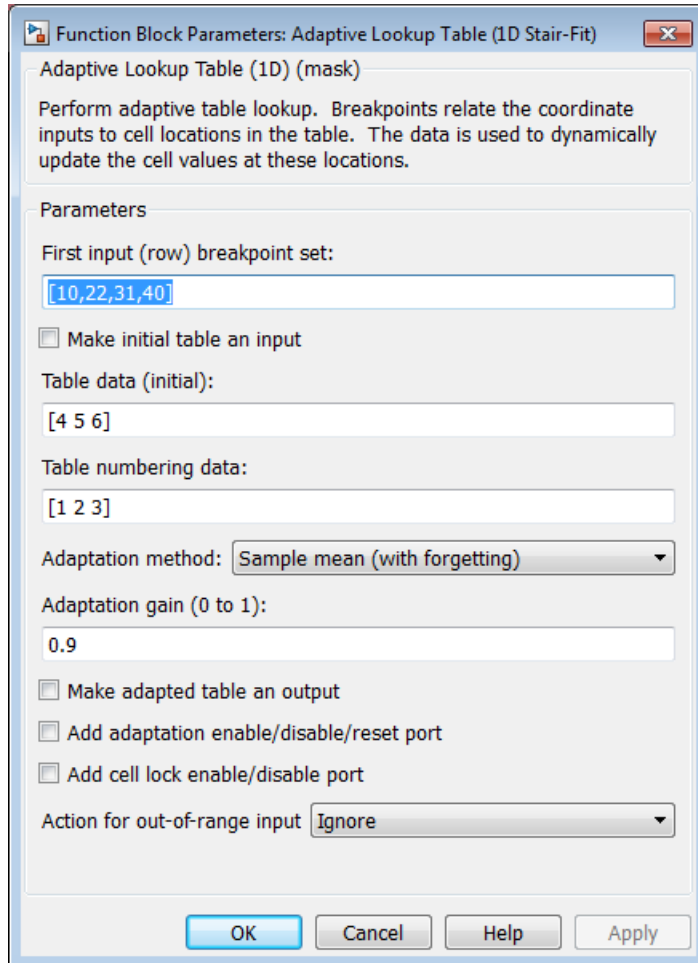
You can use the Adaptive Lookup Table (1D Stair Fit) block to model time-varying systems with one input.

## Data Type Support

Doubles only



## Dialog Box



### First input (row) breakpoint set

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

### 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 **Tin**. 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 **Tout** 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.

**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)

## **More About**

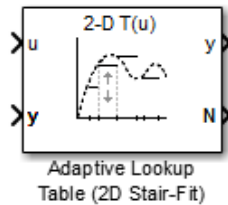
- “What are Adaptive Lookup Tables?”
- “Selecting an Adaptation Method”

# Adaptive Lookup Table (2D Stair-Fit)

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.

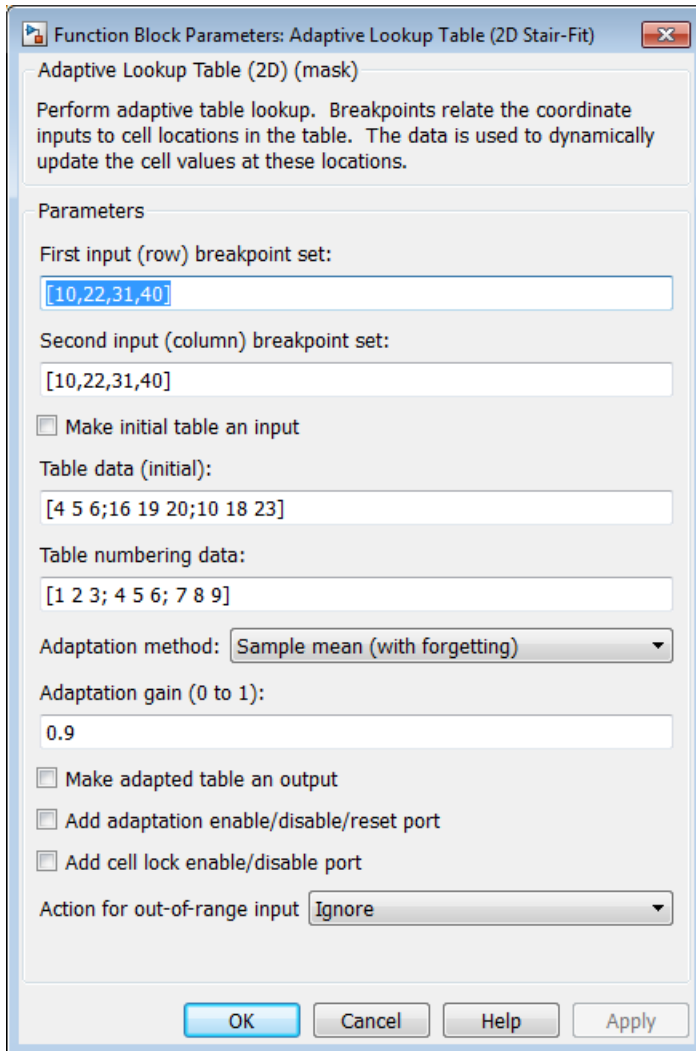
Each indexing parameter  $u$  may take a value within a set of adapting data points, which are called *breakpoints*. Two breakpoints in each dimension define a *cell*. The set of all breakpoints in one of the dimensions defines a *grid*. In the two-dimensional case, each cell has four breakpoints and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

## 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.

**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 **Tin**. 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 **Tout** 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.

**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)

### **More About**

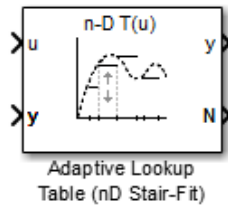
- “What are Adaptive Lookup Tables?”
- “Selecting an Adaptation Method”

# Adaptive Lookup Table (nD Stair-Fit)

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.

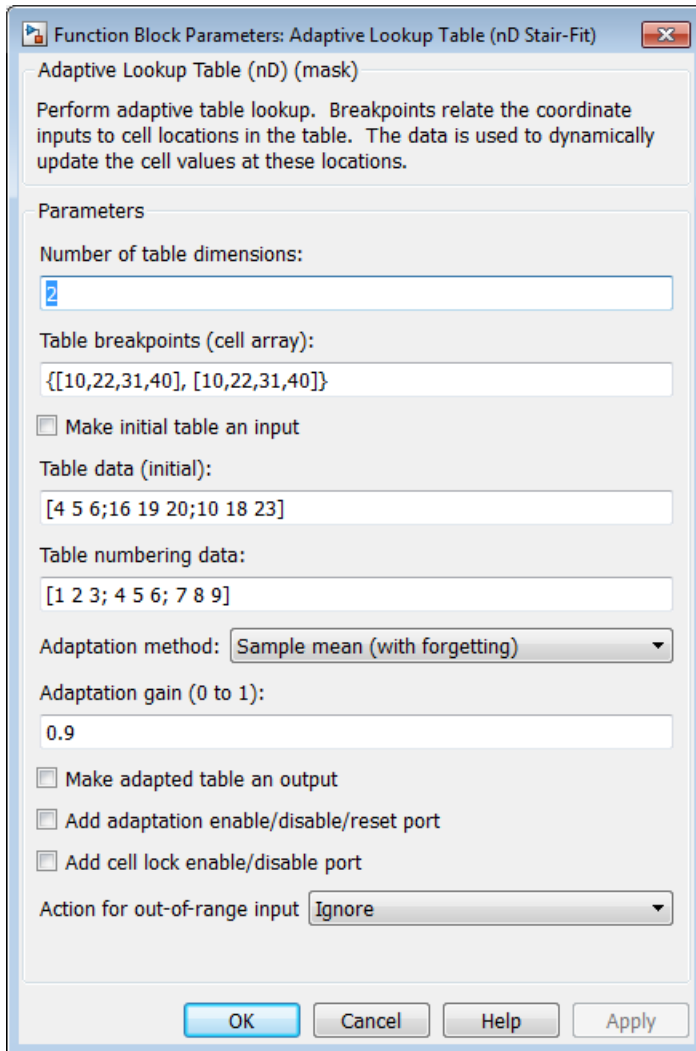
You can use the Adaptive Lookup Table (nD Stair-Fit) block to model time-varying systems with 2 or more inputs.

## Data Type Support

Doubles only



## Dialog Box



### Number of table dimensions

The number of dimensions for the adaptive lookup table.

### 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 **Tin**. 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 **Tout** for the adapted table.

---

**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”

**More About**

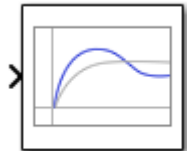
- “What are Adaptive Lookup Tables?”
- “Selecting an Adaptation Method”

# Check Against Reference

Check that model signal tracks reference signal during simulation

## Library

Simulink Design Optimization



## Description Check Against Reference

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<sup>®</sup> 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 Response 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>

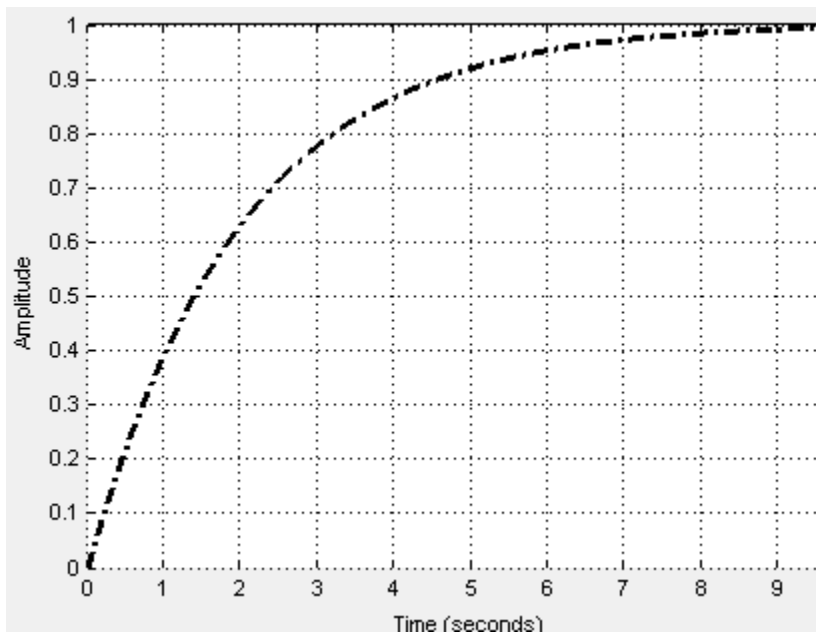
## Include reference signal tracking in assertion

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'

## 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:**  $1 - \exp(-\text{linspace}(0,10)/2)$

### Command-Line Information

**Parameter:** ReferenceAmplitudes

**Type:** string

**Value:**  $1 - \exp(-\text{linspace}(0,10)/2)$  | vector of integers of the same dimension as the time vector

**Default:**  $1 - \exp(-\text{linspace}(0,10)/2)$

## 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

## Enable assertion

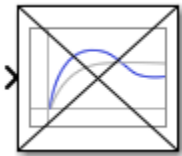
Enable the block to check that bounds specified and included for assertion in the **Bounds** tab are satisfied during simulation. Assertion fails if a bound is not satisfied. A warning, reporting the assertion failure, appears at the MATLAB prompt.

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 you do not specify any bounds.

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** parameter on the **All Parameters** tab lets you to enable or disable all model verification blocks in a model, regardless of the setting of this option in the block.

### 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'

## 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'

## 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 on the **All Parameters** tab of the Configuration Parameters dialog box of the Simulink model is checked. 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

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 Response 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

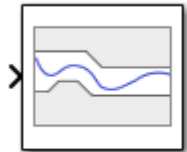
“Design Optimization to Track Reference Signal (GUI)”

# Check Custom Bounds

Check that signal satisfies upper and lower bounds during simulation

## Library

Simulink Design Optimization



## Description Check Custom Bounds

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 Response 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>

## 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.

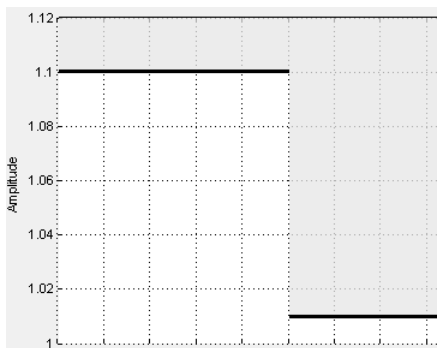


Off

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'

## 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]



## 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]

## 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

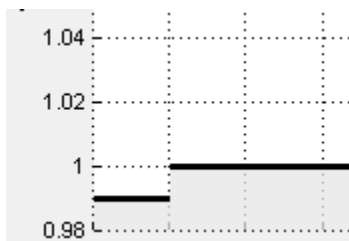
Check that the signal satisfies the specified lower bounds during simulation.

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'

## 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:** [ ]

## 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

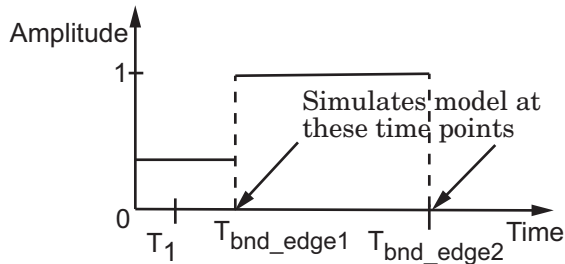
**Type:** string

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

**Default:** [ ]

## 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

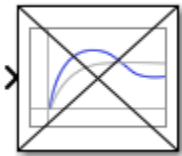
Enable the block to check that bounds specified and included for assertion in the **Bounds** tab are satisfied during simulation. Assertion fails if a bound is not satisfied. A warning, reporting the assertion failure, appears at the MATLAB prompt.

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 you do not specify any bounds.

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** parameter on the **All Parameters** tab lets you to enable or disable all model verification blocks in a model, regardless of the setting of this option in the block.

### 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'



## 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'

## 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 on the **All Parameters** tab of the Configuration Parameters dialog box of the Simulink model is checked. 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

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 Response 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

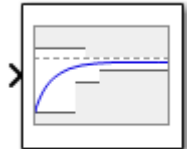
- “Design Optimization to Meet Step Response Requirements (GUI)”
- “Design Optimization to Meet Time- and Frequency-Domain Requirements (GUI)”

# Check Step Response Characteristics

Check that model signal satisfies step response bounds during simulation

## Library

Simulink Design Optimization



Check Step Response  
Characteristics

## 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 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 Response Optimization tool to optimize model response	Click <b>Response Optimization</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>

## Include step response bound in assertion

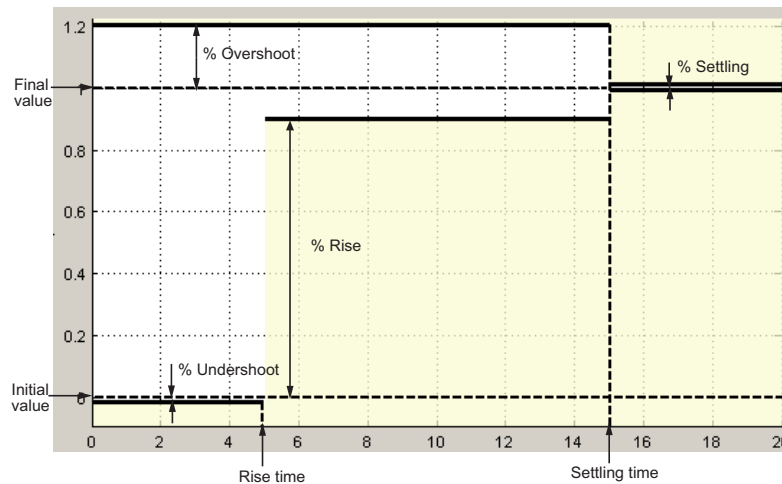
Check that the step response satisfies *all* the characteristics specified in:

- **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.





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.

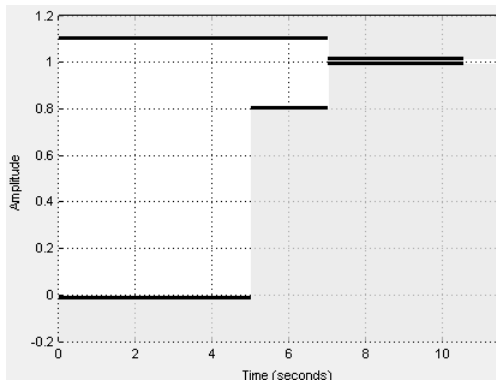


Off

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

## 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

## 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

## % 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  $(100 - \% \text{ settling})$ .

### 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  $(100 - \% \text{ settling})$ . Must be specified inside single quotes (' ').

**Default:** 80

## 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  $(100 - \% \text{ rise})$  and less than  $\% \text{ overshoot}$ .

### 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  $(100 - \% \text{ rise})$  and less than  $\% \text{ overshoot}$ . Must be specified inside single quotes (' ').

**Default:** 1

## % 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

## % 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

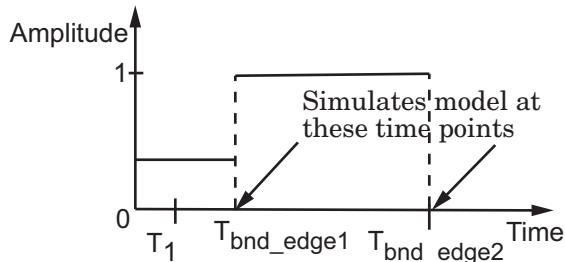
**Type:** string

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

**Default:** 1

## 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

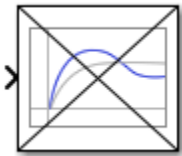
Enable the block to check that bounds specified and included for assertion in the **Bounds** tab are satisfied during simulation. Assertion fails if a bound is not satisfied. A warning, reporting the assertion failure, appears at the MATLAB prompt.

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 you do not specify any bounds.

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** parameter on the **All Parameters** tab lets you to enable or disable all model verification blocks in a model, regardless of the setting of this option in the block.

### 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'

### 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'

## 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 on the **All Parameters** tab of the Configuration Parameters dialog box of the Simulink model is checked. 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

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 Response 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)”

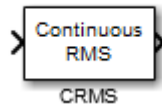
## CRMS

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

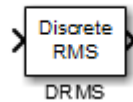
# DRMS

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

## Signal Constraint

Specify desired signal response

---

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

---

## Library

Simulink Design Optimization

# Class Reference

---

## param.Continuous class

**Package:** param

Continuous parameter

### Syntax

```
p = param.Continuous(paramname)
p = param.Continuous(paramname,paramvalue)
```

### Description

A continuous parameter is a numeric parameter that can take any value in a specified interval. The parameter can be scalar- or matrix-valued.

Typically, you use continuous parameters to create parametric models and to estimate or optimize tunable parameters in such models.

### Construction

`p = param.Continuous(paramname)` constructs a `param.Continuous` object and assigns the specified parameter name to the `Name` property and default values to the remaining properties.

`p = param.Continuous(paramname,paramvalue)` assigns the specified parameter value to the `Value` property.

`sdo.getParameterFromModel` also constructs a `param.Continuous` object or an array of `param.Continuous` objects for Simulink model parameters.

### Input Arguments

**paramname**

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



**paramvalue**

Scalar or matrix numeric double

## Properties

**Free**

Flag specifying whether the parameter is tunable or not.

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:** `''`

### **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**

### **Copy Semantics**

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

## **Examples**

### **Construct Continuous Parameter**

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

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

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

## **Alternatives**

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

### **See Also**

`sdo.optimize` | `sdo.getParameterFromModel`

### **How To**

- Class Attributes
- Property Attributes

## param.State class

**Package:** param

**Superclasses:** param.Continuous

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:

- 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

## 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';  
load_system(modelName);  
blockpath = {'sdoAircraft/Actuator Model', ...
```



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

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

## Alternatives

“Specify Known Initial States”

## See Also

sdo.Experiment | sdo.getStateFromModel

## More About

- Class Attributes
- Property Attributes

## **sdo.AnalyzeOptions class**

**Package:** sdo

Analysis options for `sdo.analyze`

### **Syntax**

```
opt = sdo.AnalyzeOptions  
opt = sdo.AnalyzeOptions('Method',method_name)
```

### **Description**

Specify analysis method and method options for sensitivity analysis using `sdo.analyze`.

### **Construction**

`opt = sdo.AnalyzeOptions` creates an `sdo.AnalyzeOptions` object and assigns default values to the properties.

To change a property value, use dot notation. For example:

```
opt = sdo.AnalyzeOptions;  
opt.Method = 'StandardizedRegression';  
opt.MethodOptions = 'Ranked';
```

`opt = sdo.AnalyzeOptions('Method',method_name)` sets the value of the `Method` property to `method_name`.

### **Input Arguments**

#### **method\_name**

Method name, specified as one of the following strings: 'Correlation', 'PartialCorrelation', 'StandardizedRegression', or 'All',

To use multiple methods, specify `method_name` as a cell array of strings.

For example, `method_name = 'PartialCorrelation'`.

For information about each method, see the `Method` property description.

## Properties

### Method

Analysis method used by `sdo.analyze`, specified as one of the following strings or a cell array containing a subset of the following strings:

- `'Correlation'` — Calculates the correlation coefficients,  $R$ . Use to analyze how a model parameter and the cost function outputs are correlated.

$R$  is calculated as follows:

$$R(i, j) = \frac{C(i, j)}{\sqrt{C(i, i)C(j, j)}}$$

$$C = \text{cov}(x, y)$$

$$= E[(x - \mu_x)(y - \mu_y)]$$

$$\mu_x = E[x]$$

$$\mu_y = E[y]$$

$x$  and  $y$  are the input arguments of `sdo.analyze`.

$R$  values are in the  $[-1 \ 1]$  range. The  $(i, j)$  entry of  $R$  indicates the correlation between  $x(i)$  and  $y(j)$ .

- $R(i, j) > 0$  — Variables have positive correlation. The variables increase together.
- $R(i, j) = 0$  — Variables have no correlation.
- $R(i, j) < 0$  — Variables have negative correlation. As one variable increases, the other decreases.
- `'PartialCorrelation'` (Requires a Statistics and Machine Learning Toolbox™ license) — Calculates the partial correlation coefficients,  $R$ . Use to analyze how a model parameter and the cost function are correlated, adjusting to remove the effect of the other parameters.

$R$  is calculated using `partialcorri` in the Statistics and Machine Learning Toolbox software.

- 'StandardizedRegression' — Calculates the standardized regression coefficients,  $R$ . Use when you expect that the model parameters linearly influence the cost function.

$R$  is calculated as follows:

$$R = b_x \frac{\sigma_x}{\sigma_y}$$

Consider a single sample ( $x_1, \dots, x_{Np}$ ) and the corresponding single output,  $y$ .  $b_x$  is the regression coefficient vector calculated using least squares assuming a linear model

$$\hat{y} = b_0 + \sum_{i=1}^{Np} \hat{b}_{x_i} x_i. R \text{ standardizes each element of } b_x \text{ by multiplying it with the ratio of}$$

the standard deviation of the corresponding  $x$  sample ( $\sigma_x$ ) to the standard deviation of  $y$  ( $\sigma_y$ ).

- 'All' — The software calculates results for all applicable combinations of `Method` and `MethodOptions`. This option may be time consuming if you have a large sample set with many parameters and many different cost/constraint outputs.

For  $x$  ( $Ns$ -by- $Np$ ) and  $y$  ( $Ns$ -by- $Nc$ ), all the methods calculate  $R$  as an  $Np$ -by- $Nc$  table. Here  $Ns$  is the number of samples,  $Np$  is the number of model parameters, and  $Nc$  is the number of cost/constraint function evaluations.

**Default:** 'Correlation'

### **MethodOptions**

String specifying the analysis method option that `sdo.analyze` uses, specified as one of the following:

- 'Linear' — Pearson analysis.

Applicable for all methods.

- 'Ranked' — Ranked transformation or Spearman analysis.

Applicable for all methods.

- 'Kendall' — Kendall's tau.

Applicable when `Method` is specified as 'Correlation'.

- 'AllApplicable' — Computes each applicable combination of `Method` and `MethodOptions`.

Applicable when `Method` is specified as 'All'.

For more information about these options, see “Analyze Relation Between Parameters and Design Requirements”.

**Default:** 'Linear'

## Copy Semantics

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

## Examples

### Specify Analysis Options

```
opt = sdo.AnalyzeOptions;  
opt.Method = 'PartialCorrelation';  
opt.MethodOptions = 'Ranked';
```

### See Also

`sdo.analyze`

### More About

- Class Attributes
- Property Attributes
- “Analyze Relation Between Parameters and Design Requirements”

## **sdo.EvaluateOptions class**

**Package:** sdo

Cost function evaluation options for `sdo.evaluate`

### **Syntax**

```
opt = sdo.EvaluateOptions  
opt = sdo.EvaluateOptions(Name,Value)
```

### **Description**

Specify options such as evaluation error handling, display settings, and the use of parallel computing for cost function evaluations using `sdo.evaluate`.

### **Construction**

`opt = sdo.EvaluateOptions` creates an `sdo.EvaluateOptions` object and assigns default values to the properties.

`opt = sdo.EvaluateOptions(Name,Value)` creates an `sdo.EvaluateOptions` object with additional options specified by one or more `Name,Value` pair arguments. `Name` is a property name and `Value` is the corresponding value.

### **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`.

Use `Name,Value` arguments to specify properties of `sdo.EvaluateOptions` object during object creation. For example, `opt =`

`sdo.EvaluateOptions('Display','off')` creates a `sdo.EvaluateOptions` object specifying the `Display` property as `off`.

## Properties

### **UseParallel** — Parallel computing option

`false` or `0` (default) | `true` or `1`

Parallel computing option for `sdo.evaluate`, specified as one of the following:

- `false` or `0` — Do not use parallel computing during cost function evaluation.
- `true` or `1` — Use parallel computing during cost function evaluation.

It is recommended that you also specify values for the `EvaluatedModel`, and `ParallelFileDependencies`, or `ParallelPathDependencies` properties, if needed.

Parallel Computing Toolbox™ software must be installed to enable parallel computing for the cost function evaluation.

### **StopOnEvaluateError** — Handling of cost function evaluation error

`'off'` (default) | `'on'`

Handling of cost function evaluation error, specified as one of the following strings:

- `'on'` — `sdo.evaluate` stops when a cost function evaluation results in an error.
- `'off'` — `sdo.evaluate` continues when a cost function evaluation results in an error. `sdo.evaluate` returns the error using the `info` output argument.

### **Display** — Viewing of display messages for cost function evaluations

`'final'` (default) | `'off'` | `'iter'`

Viewing of display messages for cost function evaluations, specified as one of the following strings:

- `'final'` — Display only the final output.
- `'off'` — Display no output.
- `'iter'` — Display the output for each evaluation.

### **ParallelFileDependencies** — File dependencies to use during parallel evaluation

`{}` (default) | cell array of strings

File dependencies to use during parallel evaluation, specified as a cell array of strings. Each string can specify either an absolute or relative path to a file. These files are copied to the workers during parallel evaluation. Use `sdo.getModelDependencies` to find the dependencies of a Simulink model.

### **ParallelPathDependencies** — Paths to dependencies to use during parallel evaluation

`{}` (default) | cell array of strings

Paths to dependencies to use during parallel evaluation, specified as a cell array of strings. These path dependencies are temporarily added to the workers during parallel evaluation. Use `sdo.getModelDependencies` to find the dependencies of a Simulink model.

### **EvaluatedModel** — Name of Simulink model to be evaluated

`''` (default) | string

Name of Simulink model to be evaluated, specified as a string.

This property is used to configure the model for parallel evaluation (`UseParallel = true`).

## Copy Semantics

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

## Examples

### **Create a Default Evaluation Options Object**

Create an `sdo.EvaluateOptions` object.

```
opt = sdo.EvaluateOptions;
```

Specify handling of cost function evaluation error.



```
opt.StopOnEvaluateError = 'on';
```

### **Specify Cost Function Evaluation Options**

Create an `sdo.EvaluateOptions` object to view output for each evaluation.

```
opt = sdo.EvaluateOptions('Display','iter');
```

### **See Also**

`sdo.evaluate` | `sdo.getModelDependencies`

### **More About**

- Class Attributes
- Property Attributes
- “Use Parallel Computing for Sensitivity Analysis”

## **sdo.Experiment class**

**Package:** sdo

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`.

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)”.

- 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:** []

### **Name**

Experiment name, specified as a string.

**Default:** ''

### **Description**

Experiment description, specified as a string.

**Default:** ''

## **Methods**

## **Copy Semantics**

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

## **Examples**

### **Specify Input-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, respectively.

Specify an experiment for a model.

```
modelName = 'sdoBattery';  
exp = sdo.Experiment(modelName);  
exp.Name = 'Charging';  
exp.Description = 'Battery charging data collected on March 15, 2013.';
```

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  = 'output';  
VoltageSig.PortIndex = 1;  
VoltageSig.Values    = timeseries(Charge_Data(:,3),Charge_Data(:,1));  
  
exp.OutputData = VoltageSig;
```

- “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)”

## Alternatives

“Estimate Parameters and States”

## See Also

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

## More About

- Class Attributes
- Property Attributes

# sdo.OptimizeOptions class

**Package:** sdo

Optimization options

## Syntax

```
opt = sdo.OptimizeOptions  
opt = sdo.OptimizeOptions(Name,Value)
```

## Description

Specify options such as a solver, solver options, and the use of parallel computing during optimization.

## Construction

`opt = sdo.OptimizeOptions` creates an `sdo.OptimizeOptions` object and assigns default values to the properties.

`opt = sdo.OptimizeOptions(Name,Value)` creates an `sdo.OptimizeOptions` object with additional options specified by one or more `Name,Value` pair arguments. `Name` is a property name and `Value` is the corresponding value.

## 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`.

Use `Name,Value` arguments to specify properties of `sdo.OptimizeOptions` object during object creation. For example, `opt = sdo.OptimizeOptions('Method','lsqnonlin')` creates a `sdo.OptimizeOptions` object specifying the `Method` property as `lsqnonlin`.

## Properties

### **GradFcn** — Specify if cost or constraint function returns gradient information

'off' (default) | 'on'

Specify if the cost or constraint function you provide to `sdo.optimize` returns gradient information, specified as one of the following strings:

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

### **Method** — Optimization solver

'fmincon' (default) | 'fminsearch' | 'lsqnonlin' | 'patternsearch'

Optimization solver that `sdo.optimize` uses to solve the optimization problem, specified as one of the following strings:

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

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

### **MethodOptions** — Optimization solver options

[1x1 `optim.options.Fmincon`] (default) | optimization options

Optimization solver options, specified as optimization options. The options are configured based on the `Method` property. For information about the available 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'

To change solver options, use dot notation. For example, `opt.MethodOptions.StepTolerance = 1.5e-3`.



**OptimizedModel** — Name of Simulink model to be optimized

' ' (default) | string

Name of Simulink model to be optimized, specified as a string.

This property is used to configure the model for parallel optimization (`UseParallel = true`).

**ParallelFileDependencies** — File dependencies to use during parallel optimization

{ } (default) | cell array of strings

File dependencies to use during parallel optimization, specified as a cell array of strings. Each string can specify either an absolute or relative path to a file. These files are copied to the workers during parallel optimization. Use `sdo.getModelDependencies` to find the dependencies of a Simulink model.

**ParallelPathDependencies** — Paths to dependencies to use during parallel optimization

{ } (default) | cell array of strings

Paths to dependencies to use during parallel optimization, specified as a cell array of strings. These path dependencies are temporarily added to the workers during parallel optimization. Use `sdo.getModelDependencies` to find the dependencies of a Simulink model.

**Restarts** — Number of times to restart optimization

0 (default) | nonnegative integer

Number of times to restart optimization, if convergence criteria are not satisfied, specified as a nonnegative integer. At each restart, the initial values of the tunable parameters are set to the final value of the previous optimization run.

**StopIfFeasible** — Handling of optimization termination once a feasible solution is found

'on' (default) | 'off'

Handling of optimization termination once a feasible solution satisfying constraints is found, specified as one of the following strings:

- '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.

### **UseParallel** — Parallel computing option

false or 0 (default) | true or 1

Parallel computing option for `fmincon`, `lsqnonlin`, and `patternsearch` optimization solvers, specified as one of the following:

- `false` or `0` — Do not use parallel computing during optimization.
- `true` or `1` — Use parallel computing during optimization.

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

When set to `true`, the methods compute the following in parallel:

- `fmincon` — Finite difference gradients
- `lsqnonlin` — Finite difference gradients
- `patternsearch` — Poll and search set evaluation

---

**Note:** Parallel computing is not supported for `fminsearch`.

---

It is recommended that you also specify values for the `OptimizedModel`, and `ParallelFileDependencies`, or `ParallelPathDependencies` properties, if needed.

## Copy Semantics

Value. To learn how value classes affect copy operations, see [Copying Objects in the MATLAB documentation](#).

## Examples

### **Create Default Optimization Options Object**

```
opt = sdo.OptimizeOptions;
```

### **Specify an Optimization Solver**

```
opts = sdo.OptimizeOptions('Method','lsqnonlin');
```

```
opt.MethodOptions.TolX = 1.5e-3;
```

## See Also

`sdo.optimize` | `sdo.getModelDependencies`

## How To

- “Specify Optimization Options”
- “Speed Up Response Optimization Using Parallel Computing”
- “Speed Up Parameter Estimation Using Parallel Computing”

## sdo.ParameterSpace class

**Package:** sdo

Specify probability distributions for model parameters

### Description

Specify the probability distributions for model parameters, which define the *parameter space*. You use the `sdo.ParameterSpace` object as an input to the `sdo.sample` command and generate samples of the model parameters. The software generates these samples as per the distributions specified for each parameter. You evaluate the cost function for each of these samples using the `sdo.evaluate` command and analyze how the model parameters influence the cost function.

### Construction

`ps = sdo.ParameterSpace(p)` creates an `sdo.ParameterSpace` object for the specified model parameters. The software assigns the parameter names to the `ParameterNames` property and default values to the remaining properties, including `ParameterDistributions`. The software specifies the uniform distribution for each parameter in `p` and sets the values of the two parameters of the uniform distribution as follows:

- **Lower** — Set to `p.Minimum`. If `p.Minimum` is equal to `-Inf`, then the software sets `Lower` to `0.9*p.Value`. Unless `p.Value` is equal to 0, in which case the software sets `Lower` to `-1`.
- **Upper** — Set to `p.Maximum`. If `p.Maximum` is equal to `Inf`, then the software sets `Upper` to `1.1*p.Value`. Unless `p.Value` is equal to 0, in which case the software sets `Upper` to 1.

`ps = sdo.ParameterSpace(p,pdist)` specifies the distribution of each parameter.

### Input Arguments

**p**

Model parameters and states, specified as a vector of `param.Continuous` objects.

For example, `sdo.getParameterFromModel('sdoHydraulicCylinder', {'Ac', 'K'})`.

### **pdist**

Probability distribution of model parameters, specified as a vector of univariate probability distribution objects.

- If `pdist` is the same size as `p`, the software specifies each entry of `pdist` as the probability distribution of the corresponding parameter in `p`.
- If `pdist` contains only one object, the software specifies this object as the probability distribution for all the parameters in `p`.

Use the `makedist` command to create a univariate probability distribution object. For example, `makedist('Normal', 'mu', 100, 'sigma', 10)`.

To check if `pdist` is a univariate distribution object, run `isa(pdist, 'prob.UnivariateDistribution')`.

## **Properties**

### **ParameterNames**

Model parameter names, specified as cell arrays of strings.

This property is read only.

**Default:** ''

### **ParameterDistributions**

Model parameter distributions, specified as a vector of `prob.UnivariateDistribution` objects.

By default, the software specifies a uniform distribution for the model parameters specified by `p`. For each parameter, the software sets the values of the two parameters of the uniform distribution:

- **Lower** — Set to `p.Minimum`. If `p.Minimum` is equal to `-Inf`, then the software sets **Lower** to `0.9*p.Value`. Unless `p.Value` is equal to 0, in which case the software sets **Lower** to `-1`.

- **Upper** — Set to `p.Maximum`. If `p.Maximum` is equal to `Inf`, then the software sets **Upper** to `1.1*p.Value`. Unless `p.Value` is equal to 0, in which case the software sets **Upper** to 1.

Use the `pdist` input argument when constructing `ps` to set the value of this property. Alternatively, use the `sdo.ParameterSpace.setDistribution` method after you have constructed `ps`.

**Default:** []

### RankCorrelation

Correlation between parameters, specified as a matrix.

When you call `sdo.sample`, the software generates samples that are correlated as specified by this matrix (where the correlation refers to ranked correlation). You can specify the sampling method using the `Method` property of an `sdo.SampleOptions`.

- If you specify `Method` as `'random'` or `'lhs'`, the software uses the Iman-Conover algorithm to impose the correlation specified by `RankCorrelation`.
- If you specify `Method` as `'copula'`, the software uses a copula to impose the correlation specified by `RankCorrelation`. Use the `MethodOptions` property of the `sdo.SampleOptions` object to specify the copula family.

Specify [] when the parameters are uncorrelated.

**Default:** []

### Options

Sampling method options, specified as an `sdo.SampleOptions` object.

**Default:** `sdo.SampleOptions`

### Notes

Text notes associated with `ps`, specified as a string or cell array of strings.

**Default:** ''

**Default:**

## Methods

## Copy Semantics

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

## Examples

### Specify Parameter Distributions for Sampling

Obtain the model parameters of interest.

```
load_system('sdoHydraulicCylinder');  
p = sdo.getParameterFromModel('sdoHydraulicCylinder',{'Ac','K'});
```

Construct an `sdo.ParameterSpace` object for Ac and K.

```
ps = sdo.ParameterSpace(p);
```

You can use `ps` as an input to `sdo.sample` and generate samples. By default, the software specifies a uniform distribution for both parameters.

Suppose you want to specify the normal distribution for Ac and the uniform distribution for K, with K in the [30000 70000] range.

```
pdistAc = makedist('Normal','mu',p(1).Value,'sigma',2);  
pdistK = makedist('Uniform','lower',30000,'upper',70000);  
ps1 = sdo.ParameterSpace(p,[pdistAc;pdistK]);
```

### See Also

`sdo.ParameterSpace.addParameter` | `makedist` | `sdo.getParameterFromModel` | `sdo.sample`

### More About

- Class Attributes
- Property Attributes

## sdo.requirements.BodeMagnitude class

**Package:** sdo.requirements

Bode magnitude bound

### Syntax

```
bode_req = sdo.requirements.BodeMagnitude  
bode_req = sdo.requirements.BodeMagnitude(Name,Value)
```

### Description

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 `sdo.optimize`.

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

You must have Simulink Control Design software to specify bode magnitude requirements.

### Construction

`bode_req = sdo.requirements.BodeMagnitude` creates an `sdo.requirements.BodeMagnitude` object and assigns default values to its properties.

`bode_req = sdo.requirements.BodeMagnitude(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 `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]

#### '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'
- '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  $-\text{inf}$  and  $+\text{inf}$  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  $n \times 2$  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'

- '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  $-\text{inf}$  and  $+\text{inf}$  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:

- ' $\leq$ ' — Upper bound
- ' $\geq$ ' — 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

## 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 class

**Package:** sdo.requirements

Closed loop peak gain bound

## Description

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 `sdo.optimize`.

You must have Simulink Control Design software to specify closed-loop peak gain bounds.

## Construction

`pkgain_req = sdo.requirements.ClosedLoopPeakGain` creates a `sdo.requirements.ClosedLoopPeakGain` object and assigns default values to its properties.

`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`.

## 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 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:** '<='

## 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

## 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 class

**Package:** sdo.requirements

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:

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

**Default:** '>='



## Methods

## 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 Margins and Check Nichols Characteristics block.

## See Also

`copy` | `get` | `set`

## How To

- Class Attributes
- Property Attributes

## sdo.requirements.OpenLoopGainPhase class

**Package:** sdo.requirements

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  $n \times 2$  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:** '>='

## Properties

### **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  $n \times 2$  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:** '>='

## **Methods**

### **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](#)

## **sdo.requirements.PZDampingRatio class**

**Package:** sdo.requirements

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:** ''

**'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

### 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

## **How To**

- Class Attributes
- Property Attributes

## **sdo.requirements.PZNaturalFrequency class**

**Package:** sdo.requirements

Natural frequency bound

### **Description**

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 `sdo.optimize`.

You must have Simulink Control Design software to specify natural frequency requirements.

### **Construction**

`pznatfreq_req = sdo.requirements.PZNaturalFrequency` creates a `sdo.requirements.PZNaturalFrequency` object and assigns default values to its properties.

`pznatfreq_req = sdo.requirements.pznatfreq_req(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:** ''

**'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'

**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'
- '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:** '>='

## Methods

## 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.

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

## Alternatives

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

## See Also

`copy` | `get` | `set`

## How To

- Class Attributes



- Property Attributes

## sdo.requirements.PZSettlingTime class

**Package:** sdo.requirements

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.

**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'
- '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

## 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 a Check Pole-Zero Characteristics block.

### **See Also**

copy | get | set

### **How To**

- Class Attributes
- Property Attributes

# sdo.requirements.SignalBound class

**Package:** sdo.requirements

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  $-\infty$ .

## 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.

**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:** ''

### **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.

**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**

## **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 class

**Package:** sdo.requirements

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'

- '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

**'RobustCost'**

Enable or disable robust treatment of outliers when evaluating the requirement. The software uses a Huber loss function to handle the outliers in the cost function and improves the fit quality. This option reduces the influence of outliers on the estimation without you manually modifying your data.

Must be one of the following:

- 'on' — When you call the `evalRequirement` method, the software uses a Huber loss function to evaluate the cost for the tracking error outliers. The tracking error is calculated as  $e(t)=y_{ref}(t)-y_{test}(t)$ . The software uses the error statistics to identify the outliers.

The exact cost function used,  $F(x)$ , depends on the requirement evaluation Method.

Method Name	Cost Function for Nonoutliers	Cost Function for Outliers
'SSE'	$F(x) = \sum_{t \in NOL} e(t) \times e(t)$ <p><i>NOL</i> is the set of nonoutlier samples.</p>	$F(x) = \sum_{t \in OL} w \times  e(t) $ <p><i>w</i> is a linear weight. <i>OL</i> is the set of outlier samples.</p>
'SAE'	$F(x) = \sum_{t \in NOL}  e(t) $ <p><i>NOL</i> is the set of nonoutlier samples.</p>	$F(x) = \sum_{t \in OL} w$ <p><i>w</i> is a constant value. <i>OL</i> is the set of outlier samples.</p>
'Residuals'	<p>The software does not remove the outliers.</p> $F(x) = \begin{bmatrix} e(0) \\ \vdots \\ e(N) \end{bmatrix}$ <p><i>N</i> is the number of samples.</p>	

- 'off'

**Default:** 'off'

**'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

**RobustCost**

Enable or disable robust treatment of outliers when evaluating the requirement. The software uses a Huber loss function to handle the outliers in the cost function and improves the fit quality. This option reduces the influence of outliers on the estimation without you manually modifying your data.

Must be one of the following:

- 'on' — When you call the `evalRequirement` method, the software uses a Huber loss function to evaluate the cost for the tracking error outliers. The tracking error is calculated as  $e(t)=y_{ref}(t)-y_{test}(t)$ . The software uses the error statistics to identify the outliers.

The exact cost function used,  $F(x)$ , depends on the requirement evaluation Method.

Method Name	Cost Function for Nonoutliers	Cost Function for Outliers
'SSE'	$F(x) = \sum_{t \in NOL} e(t) \times e(t)$ <p><i>NOL</i> is the set of nonoutlier samples.</p>	$F(x) = \sum_{t \in OL} w \times  e(t) $ <p><i>w</i> is a linear weight. <i>OL</i> is the set of outlier samples.</p>
'SAE'	$F(x) = \sum_{t \in NOL}  e(t) $ <p><i>NOL</i> is the set of nonoutlier samples.</p>	$F(x) = \sum_{t \in OL} w$ <p><i>w</i> is a constant value. <i>OL</i> is the set of outlier samples.</p>

Method Name	Cost Function for Nonoutliers	Cost Function for Outliers
'Residuals'	The software does not remove the outliers. $F(x) = \begin{bmatrix} e(0) \\ \vdots \\ e(N) \end{bmatrix}$ $N$ is the number of samples.	

- 'off'

**Default:** 'off'

### 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.

## Methods

### 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

## sdo.requirements.SingularValue class

**Package:** sdo.requirements

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 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'
- '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  $-\text{inf}$  and  $+\text{inf}$  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  $n \times 2$  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'
- '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  $-\text{inf}$  and  $+\text{inf}$  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

## 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

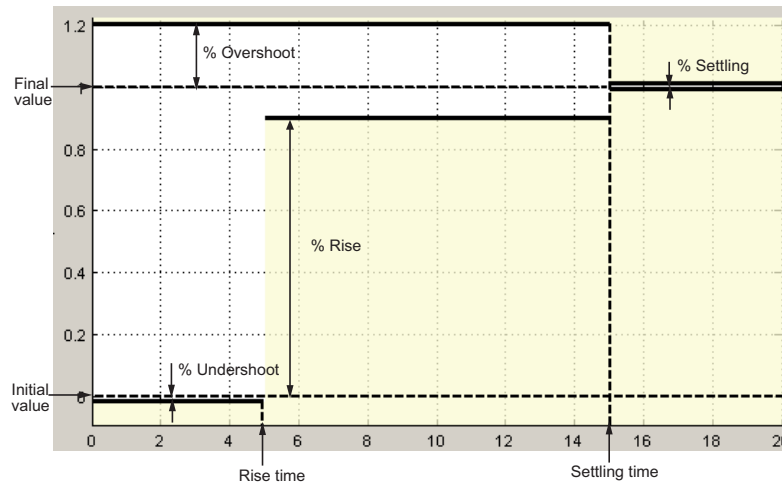
# sdo.requirements.StepResponseEnvelope class

**Package:** sdo.requirements

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.

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'**

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-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**

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 '<='.

## Methods

## 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

## sdo.SampledParameter class

**Package:** sdo

**Superclasses:** param.Continuous

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.

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

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

### 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.SampleOptions class

**Package:** sdo

Parameter sampling options for `sdo.sample`

## Description

Specify method options to generate parameter samples, using `sdo.sample`, for sensitivity analysis.

## Construction

`opt = sdo.SampleOptions` creates an `sdo.SampleOptions` object and assigns default values to its properties.

Use dot notation to modify the property values. For example:

```
opt = sdo.SampleOptions;  
opt.Method = 'lhs';
```

## Properties

### Method

Sampling method, specified as one of the following strings:

- `'random'` — Random samples drawn from the probability distributions specified for the parameters.

Suppose you specified a value for the `RankCorrelation` property of the `sdo.ParameterSpace` object that you use for sampling. The software uses the Iman-Conover method to impose the parameter correlations.

- `'lhs'` — Latin hypercube samples drawn from the probability distributions specified for the parameters. Use this option for a more systematic space-filling approach than random sampling.

Suppose you specified a value for the `RankCorrelation` property of the `sdo.ParameterSpace` object that you use for sampling. The software uses the Iman-Conover method to impose the parameter correlations.

Requires a Statistics and Machine Learning Toolbox license.

- `'copula'` — Random samples drawn from a copula. Use this option to impose correlations between the parameters. When you use this option, you must specify the value of the `RankCorrelation` property of the `sdo.ParameterSpace` object that you use for sampling.

Requires a Statistics and Machine Learning Toolbox license.

For more information about the sampling methods, see “Generate Parameter Samples for Sensitivity Analysis”.

**Default:** `'random'`

### **MethodOptions**

Sample method options, applicable only when `Method` is `'copula'`, specified as a structure with the following fields:

- `Family` — Copula family, specified as one of the following strings:
  - `'Gaussian'` — Gaussian copula
  - `'t'` — t copula

**Default:** `'Gaussian'`

- `Type` — Rank correlation type, specified as one of the following strings
  - `'Spearman'` — Spearman’s rank correlation
  - `'Kendall'` — Kendall’s rank correlation

**Default:** `'Spearman'`

- `DOF` — Degrees of freedom of t copula, specified as a positive integer.

For a Gaussian copula, specify `DOF` as `[]`. Specification of `DOF` is required for a t copula.

**Default:** `[]`

## Copy Semantics

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

## Examples

### Specify Random Sampling Method

```
opt = sdo.SampleOptions

opt =

SampleOptions with properties:

    Method: 'random'
MethodOptions: [0x0 struct]
```

### Specify Latin Hypercube Sampling Method

```
opt = sdo.SampleOptions;
opt.Method = 'lhs';
```

### Specify Copula-Based Sampling Method

```
opt = sdo.SampleOptions;
opt.Method = 'copula';
opt.MethodOptions.Family = 't';
opt.MethodOptions.DOF = 2;
```

## See Also

sdo.sample

## More About

- Class Attributes
- Property Attributes
- “Generate Parameter Samples for Sensitivity Analysis”

## **sdo.SimulationTest class**

**Package:** sdo

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. You can also specify linear systems to compute if you have Simulink Control Design toolbox. 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

### SystemLoggingInfo

Linear system logging settings.

This property is a vector of `sdo.SystemLoggingInfo` objects.

If you specify the `SystemLoggingInfo` property, the `sim` method linearizes the model during simulation.

---

**Note:** You can also use `linearize` command from Simulink Control Design to compute linear systems. However, to use fast restart, you must use `SystemLoggingInfo` property and `sim` instead.

---

**Default:** []

### **modelName**

Simulink model name associated with the simulation scenario. The model must be on the MATLAB path.

### **Name**

Name of the scenario

**Default:** ''

### **Parameters**

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**

## **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.

```
Pressures = Simulink.SimulationData.SignalLoggingInfo;  
Pressures.BlockPath = 'sdoHydraulicCylinder/Cylinder Assembly';  
Pressures.OutputPortIndex = 1;  
simulator = sdo.SimulationTest('sdoHydraulicCylinder');
```

Specify model signals to log.

```
simulator.LoggingInfo.Signals = [Pressures];
```

### 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);
```

- “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)”
- “Design Optimization to Meet Frequency-Domain Requirements (Code)”

## Alternatives

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

### See Also

sdo.SystemLoggingInfo | sdo.Experiment | sdo.optimize | sdo.evaluate |  
sdo.Experiment.createSimulator

## **How To**

- Class Attributes
- Property Attributes

# sdo.SystemLoggingInfo class

**Package:** sdo

Specify linear system logging information

## Syntax

```
sys = sdo.SystemLoggingInfo  
sys = sdo.SystemLoggingInfo(Name, Value)
```

## Description

Specify linear system logging information. Use `sdo.SystemLoggingInfo` object to set the `SystemLoggingInfo` property of `sdo.SimulationTest`, to specify linear systems to log when simulating the model. You can configure `sdo.SystemLoggingInfo` to compute the linear system with or without using any frequency-domain check blocks defined in the model.

## Construction

`sys = sdo.SystemLoggingInfo` constructs an `sdo.SystemLoggingInfo` object, `sys`, with default linear system logging settings. To modify settings for your specific application, use dot notation.

`sys = sdo.SystemLoggingInfo(Name, Value)` specifies additional linear system logging settings, using 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`.

Use `Name`, `Value` arguments to specify properties of `sdo.SystemLoggingInfo` object during object creation. For example, `sys = sdo.SystemLoggingInfo('LoggingName', 'linear_system1')` creates a `sdo.SystemLoggingInfo` object specifying the `LoggingName` property as `linear_system1`.

## Properties

### **Source** — Specify how to compute the linear system

' ' (default) | string

Specify how to compute the linear system, with or without using frequency-domain check blocks in the model, specified as a string. To use a frequency-domain check block in the model, specify `Source` as the full path of the check block. To not use a model check block, set `Source` to a string name or use the default value ' '.

### **LoggingName** — Name used for the computed linear system

'sys' (default) | string

Name used for the computed linear system, specified as a string. `LoggingName` appears in the `sdo.SimulationTest.LoggedData` when the simulation is run.

### **LinearizationIOs** — Linearization input/output points

[] (default) | vector of linearization IOs

Linearization input/output (IO) points, specified as a vector of linearization IOs. Create `LinearizationIOs` using the `linio` command from Simulink Control Design.

If `Source` is specified as path to a model frequency-domain check block, and `LinearizationIOs` is non-empty, the linearization IO points of the check block are overwritten when the model is simulated.

### **SnapshotTimes** — Linearization snapshot times

[] (default) | scalar | vector of scalars

Linearization snapshot times, specified as a scalar or vector of scalars.

If `Source` is specified as path to a model frequency-domain check block, and `SnapshotTimes` is non-empty, the linearization snapshot times of the check block are overwritten when the model is simulated.

### LinearizationOptions — Linearization options

[] (default) | linearization option set

Linearization options to use when computing the linear system, specified as a linearization option set. To set these options, use the `linearizeOptions` command from Simulink Control Design.

If `Source` is specified as path to a model frequency-domain check block, and `LinearizationOptions` is non-empty, the linearization options of the check block are overwritten when the model is simulated.

## Copy Semantics

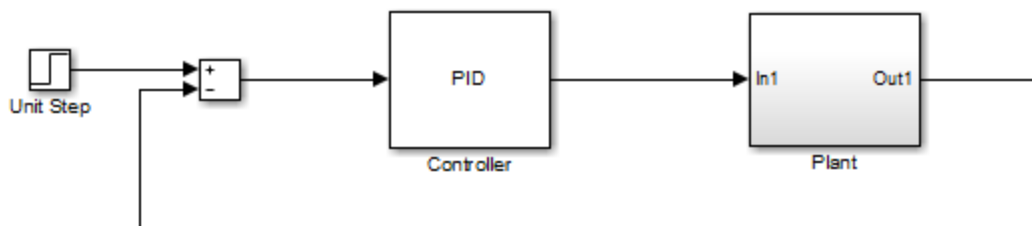
Value. To learn how value classes affect copy operations, see [Copying Objects in the MATLAB documentation](#).

## Examples

### Specify Linear System Logging Settings

Open the model.

```
open_system('sldo_model1')
```



Specify the input and output points that define the linear system to be computed.

```
I0s(1) = linio('sldo_model1/Sum',1,'input');  
I0s(2) = linio('sldo_model1/Plant',1,'output');
```

Create an `sdo.SystemLoggingInfo` object to specify the linear system logging settings.

```
sys1 = sdo.SystemLoggingInfo;  
sys1.Source = 'Specified I0s';  
sys1.LoggingName = 'Linear_System';  
sys1.LinearizationI0s = I0s;  
sys1.SnapshotTimes = 0;
```

- “Design Optimization to Meet Frequency-Domain Requirements (Code)”

### See Also

`fastRestart` | `linearizeOptions` | `linio` | `sdo.SimulationTest`

### More About

- “Use Fast Restart Mode During Response Optimization”
- “Use Fast Restart Mode During Sensitivity Analysis”

**Introduced in R2015b**



# Alphabetical List

---

### **copy**

Copy requirement

### **Syntax**

```
copy_req = copy(req)
```

### **Description**

`copy_req = copy(req)` copies a requirement object (`sdo.requirements.StepResponseEnvelope, ...`) to a new object of the same type.

For more information, see `copy` in the MATLAB documentation.

### **Input Arguments**

**req**

requirement object (`sdo.requirements.StepResponseEnvelope, ...`)

### **Output Arguments**

**copy\_req**

requirement object (`sdo.requirements.StepResponseEnvelope, ...`), which is a copy of `req`.

### **See Also**

`get` | `handle`

# evalRequirement

**Class:** sdo.requirements.BodeMagnitude

**Package:** sdo.requirements

Evaluate Bode magnitude bound for linear system

## Syntax

```
c = evalRequirement(req,lin_sys)
```

## Description

`c = evalRequirement(req,lin_sys)` evaluate whether a linear system satisfies the specified piecewise-linear Bode magnitude bound.

## Input Arguments

### **req**

sdo.requirements.BodeMagnitude object.

For MIMO systems, the bound applies to each input/output (I/O) channel.

### **lin\_sys**

Linear system (tf, ss, zpk, frd, genss, or genfrd).

## Output Arguments

### **c**

Column vector indicating the maximum signed distance of the system gain to each edge specified in `req`. Negative values indicate that the bound edge is satisfied and positive values 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.

### Examples

Evaluate Bode magnitude requirement.

```
req = sdo.requirements.BodeMagnitude;  
sys = tf(1,[1 2 2 1])  
c = evalRequirement(req,sys);
```

c is negative, which indicates that the system satisfies the gain requirement.

### See Also

`sdo.requirements.BodeMagnitude` | `get` | `set` | `copy`

# evalRequirement

Evaluate peak gain bound for linear system

## Syntax

```
c = evalRequirement(req,lin_sys)
```

## Description

`c = evalRequirement(req,lin_sys)` 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`.

## Input Arguments

**req**

`sdo.requirements.ClosedLoopPeakGain` object.

**lin\_sys**

Linear system (`tf`, `ss`, `zpk`, `frd`, `genss`, or `genfrd`).

## Output Arguments

**c**

- Signed distance of the closed-loop peak gain to the bound if the `Type` property of `req` is `<=` or `==`. When `<=`, 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.
- Peak gain if the `Type` property of `req` is `min`.

### Examples

Evaluate peak gain requirement.

```
req = sdo.requirements.ClosedLoopPeakGain;  
sys = tf(0.5,[1 3 3 1]);  
c = evalRequirement(req,sys);
```

`c` is negative, which indicates that the system satisfies the gain requirement.

### See Also

`sdo.requirements.ClosedLoopPeakGain` | `get` | `set` | `copy`

# evalRequirement

**Class:** sdo.requirements.GainPhaseMargin

**Package:** sdo.requirements

Evaluate gain and phase margin bounds for linear system

## Syntax

```
c = evalRequirement(req,lin_sys)
```

## Description

`c = evalRequirement(req,lin_sys)` 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 `FeedbackSign` property of `req`.

## Input Arguments

**req**

sdo.requirements.GainPhaseMargin object.

**lin\_sys**

Linear system (tf, ss, zpk, frd, genss, or genfrd).

## Output Arguments

**c**

- Signed distance of the computed gain and phase margins to the bound if the `Type` property of `req` is `>=` or `==`.

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 ==, any number other than 0 indicates that the bound is not satisfied.

- Negative of the gain and phase margins such that minimizing the values maximizes the margins if the `Type` property of `req` is 'max'. Unstable loops return positive values.

## Examples

Evaluate gain and phase margin requirements.

```
req = sdo.requirements.GainPhaseMargin;  
sys = tf(0.5,[1 3 3 1]);  
c = evalRequirement(req,sys);
```

`c` is negative, which indicates that the system satisfies the gain and phase margin requirement.

## See Also

| `get` | `set` | `copy`



# evalRequirement

**Class:** sdo.requirements.OpenLoopGainPhase

**Package:** sdo.requirements

Evaluate gain and phase bounds on Nichols response of linear system

## Syntax

```
c = evalRequirement(req,lin_sys)
```

## Description

`c = evalRequirement(req,lin_sys)` evaluates whether a linear system satisfies the specified open-loop gain and phase bounds on the Nichols response.

## Input Arguments

**req**

sdo.requirements.OpenLoopGainPhase object.

**lin\_sys**

Linear system (tf, ss, zpk, frd, genss, or genfrd).

## Output Arguments

**c**

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.

### Examples

Evaluate open-loop gain and phase requirements.

```
req = sdo.requirements.OpenLoopGainPhase;  
sys = tf(0.5,[1 3 3 1]);  
c = evalRequirement(req,sys);
```

### See Also

`sdo.requirements.OpenLoopGainPhase` | `get` | `set` | `copy`

# evalRequirement

**Class:** sdo.requirements.PZDampingRatio

**Package:** sdo.requirements

Evaluate damping ratio bound on linear system

## Syntax

```
c = evalRequirement(req,lin_sys)
```

## Description

`c = evalRequirement(req,lin_sys)` evaluates whether the poles of a linear system satisfies the specified damping ratio bound.

## Input Arguments

**req**

sdo.requirements.PZDampingRatio object.

**lin\_sys**

Linear system (tf, ss, zpk, frd, genss, or genfrd).

## Output Arguments

**c**

- Signed distance of the damping ratio of each pole of the linear system to the bound, if the **Type** property of **req** is `>=`, `<=` or `==`. Negative values indicate that the bound is satisfied while positive values indicate that the bound is violated. When `==`, any value other than 0 indicates that the bound is violated.
- Negative of the damping ratio such that minimizing the values maximizes the damping ratio, if the **Type** property of **req** is `'max'`.

### 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.

### See Also

| `get` | `set` | `copy`

# evalRequirement

**Class:** sdo.requirements.PZNaturalFrequency

**Package:** sdo.requirements

Evaluate natural frequency bound on linear system

## Syntax

```
c = evalRequirement(req,lin_sys)
```

## Description

`c = evalRequirement(req,lin_sys)` evaluates whether the poles of a linear system satisfies the specified natural frequency bound.

## Input Arguments

### **req**

Requirement object (sdo.requirements.StepResponseEnvelope, ...).

For MIMO systems, the bound applies to each input/output (I/O) channel.

### **lin\_sys**

Linear system (tf, ss, zpk, frd, genss, or genfrd).

## Output Arguments

### **c**

- Signed distance of the natural frequency of each system pole to the bound. If the `Type` property of `req` is `>=`, `<=`, negative values indicate that the bound is satisfied while positive values indicate that the bound is violated. If `==`, any value other than 0 indicates that the bound is violated.

- Negative of the natural frequency of the linear system poles such that minimizing the values maximizes the natural frequency, if the `Type` property of `req` is 'max'.

### Examples

Evaluate natural frequency requirement.

```
req = sdo.requirements.PZNaturalFrequency;  
sys = tf(0.5,[1 3 3 1]);  
c = evalRequirement(req,sys);
```

`c` is positive, which indicates that the system does not satisfy the natural frequency requirement.

### See Also

`sdo.requirements.PZNaturalFrequency` | `get` | `set` | `copy`

# evalRequirement

**Class:** sdo.requirements.PZSettlingTime

**Package:** sdo.requirements

Evaluate settling time bound on linear system

## Syntax

```
c = evalRequirement(req,lin_sys)
```

## Description

`c = evalRequirement(req,lin_sys)` evaluates whether the poles of a linear system satisfies the specified settling time bound.

## Input Arguments

**req**

sdo.requirements.PZSettlingTime object.

**lin\_sys**

Linear system (tf, ss, zpk, frd, genss, or genfrd).

## Output Arguments

**c**

- Signed distance of the real component of each system pole to the bound, if the **Type** property of **req** is `<=` or `==`. Negative values indicate that the bound is satisfied while positive values indicate that the bound is violated. If `==`, values other than 0 indicate that the bound is violated.
- Pole locations such that minimizing the values minimizes the settling time, if the **Type** property of **req** is `'min'`.

### Examples

Evaluate settling time requirement.

```
req = sdo.requirements.PZSettlingTime;  
sys = tf(0.5,[1 3 3 1]);  
c = evalRequirement(req,sys);
```

c is positive, which indicates that the system does not satisfy the settling time requirement.

### See Also

| get | set | copy



# evalRequirement

**Class:** sdo.requirements.SignalBound

**Package:** sdo.requirements

Evaluate piecewise-linear bound

## Syntax

```
c = evalRequirement(req,sig)
```

## Description

`c = evalRequirement(req,sig)` evaluate whether a signal satisfies the specified piecewise-linear bounds.

## Input Arguments

**req**

sdo.requirements.SignalBound object.

**sig**

MATLAB `timeseries` object or `nxm` array, where the 1st column is time and subsequent columns are signal values.

## Output Arguments

**c**

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.

### Examples

Evaluate piecewise-linear bound on signal.

```
req = sdo.requirements.SignalBound;  
sig = timeseries(1-exp(-(0:10)'));  
c = evalRequirement(req,sig);
```

c is negative, which indicates that the signal satisfies the bounds.

### See Also

get | set | copy

# evalRequirement

**Class:** sdo.requirements.SignalTracking

**Package:** sdo.requirements

Evaluate tracking requirement

## Syntax

```
c = evalRequirement(req,sig)
```

```
c = evalRequirement(req,sig,ref)
```

## Description

`c = evalRequirement(req,sig)` evaluates whether a test point signal, `sig`, tracks the reference signal specified by a requirement object, `req`.

`c = evalRequirement(req,sig,ref)` evaluates whether `sig` tracks the reference signal specified by `ref`. `req` 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 `sig` and `ref`, and re-use the requirement object, `req`.

## Input Arguments

### **req**

sdo.requirements.SignalTracking object.

### **sig**

MATLAB `timeseries` object or `nxm` array, where the 1st column is time and subsequent columns are signal values.

### **ref**

Reference signal, specified as a MATLAB `timeseries` object.

## Output Arguments

**c**

- Measure of how well the test point signal matches the reference signal, if the **Type** property of **req** is `'=='`. Specify the algorithm used to compute the tracking measure through the **Method** property.
- Signed distance of the test point signal to the reference signal, if the **Type** property of **req** is `'>='` or `'<='`. Negative values indicate the bound is satisfied while positive values indicate that the bound is violated.

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.

Create a reference and test point signal. Then, use a requirement object to evaluate the requirement.

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.

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](#)

# evalRequirement

**Class:** sdo.requirements.SingularValue

**Package:** sdo.requirements

Evaluate singular value bound on linear system

## Syntax

```
c = evalRequirement(req,lin_sys)
```

## Description

`c = evalRequirement(req,lin_sys)` evaluates whether a linear system satisfies the specified singular values bound.

## Input Arguments

### **req**

sdo.requirements.SingularValue object.

For MIMO systems, the bound applies to each input/output (I/O) channel.

### **lin\_sys**

Linear system (tf, ss, zpk, frd, genss, or genfrd).

## Output Arguments

### **c**

Column vector indicating the maximum signed distance of the system gain to each edge specified in `req`. 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.

## Examples

Evaluate singular value requirement.

```
req = sdo.requirements.SingularValue;  
sys = tf(1,[1 2 2 1]);  
c = evalRequirement(req,sys);
```

c is negative, which indicates that the system satisfies the gain requirement.

## See Also

sdo.requirements.SingularValue | [get](#) | [set](#) | [copy](#)

# evalRequirement

**Class:** sdo.requirements.StepResponseEnvelope

**Package:** sdo.requirements

Evaluate satisfaction of step response requirement

## Syntax

```
evaluation = evalRequirement(requirement,signal)
```

## Description

`evaluation = evalRequirement(requirement,signal)` evaluates whether `signal` satisfies `requirement`, the step response requirement.

## Input Arguments

### **requirement** — Step response requirement

sdo.requirements.StepResponseEnvelope object

Step response requirement, specified as an sdo.requirements.StepResponseEnvelope object.

### **signal** — Signal to be evaluated

timeseries object | matrix | numeric or generalized LTI model

Signal to be evaluated, specified as one of the following:

- timeseries object
- Matrix of size  $n \times m$  — Where  $n$  is the number of time points in the signal and  $m$  is the number of channels in the signal. The first column is time and subsequent columns are signal values.
- Numeric or generalized linear time invariant (LTI) model — Available with Control System Toolbox™ software.



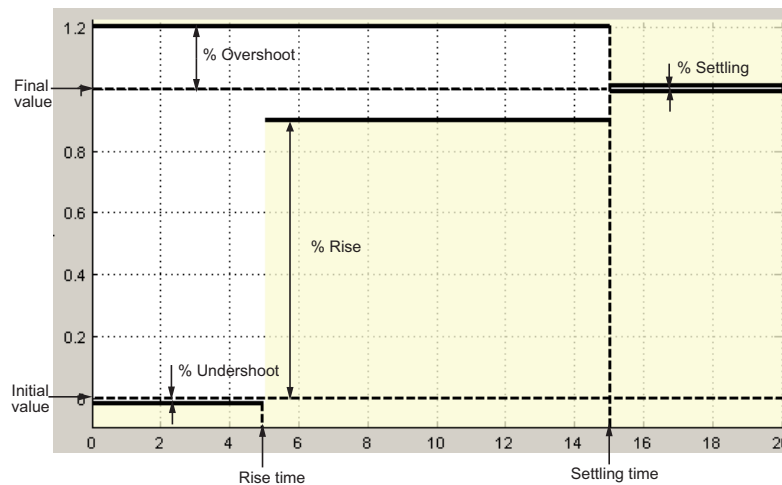
## Output Arguments

### evaluation — Evaluation of the step response requirement

column vector

Evaluation of the step response requirement, returned as a column vector. Negative values indicate that the requirement is satisfied, and positive values indicate that the requirement is violated. `evaluation` consists of the maximum distances between signal and step response bounds. The last entry in `evaluation` is a stability measure.

The maximum signal distances from upper bounds are returned before lower bounds. The step bounds are considered in the following order:



- Upper bound for the overshoot — Signal between `StepTime` (time,  $t = 0$ ) and `SettlingTime` of the requirement is used.
- Upper bound for the settling time — Signal between `SettlingTime` and  $1.5 * \text{SettlingTime}$  is used.
- Additional upper bound for the settling time — If signal extends beyond  $1.5 * \text{SettlingTime}$ , there is an additional upper bound for the settling time. Maximum signal distance from this upper bound is also returned. The signal between  $1.5 * \text{SettlingTime}$  and end of signal is used.
- Lower bound for the undershoot — Signal between `StepTime` and `RiseTime` is used.
- Lower bound for the % rise — Signal between `RiseTime` and `SettlingTime` is used.

- Lower bound for the settling time — Signal between `SettlingTime` and `1.5*SettlingTime` is used.
- Additional lower bound for the settling time — If `signal` extends beyond `1.5*SettlingTime`, there is an additional lower bound for the settling time. Maximum signal distance from this lower bound is also returned. The signal between `1.5*SettlingTime` and end of signal is used.

The last value in `evaluation` is a measure of stability from the end of `signal` to infinity. Negative values indicate that the projected signal is not deviating from the step response after end of `signal`. A positive value indicates that the projected signal is deviating.

## Examples

### Evaluate Step Response Requirement

Create a step response requirement.

```
requirement = sdo.requirements.StepResponseEnvelope;
```

A requirement is created with the default settling time of 7 seconds.

Specify the signal to be evaluated.

```
signal = timeseries(1-exp(-(0:10)'));;
```

The signal data extends to 10 seconds.

Evaluate the step response requirement.

```
evaluation = evalRequirement(requirement,signal)
```

```
evaluation =
```

```
-0.0917  
-0.0099  
-1.0000  
-0.2416  
-0.0092  
-0.0299
```

The maximum distance of the signal from the step response bounds is returned in `evaluation(1:5)`, followed by the stability value. Negative values indicate that the requirement is satisfied.

The maximum distance from the bounds are returned in the following order:

- 1 Distance from upper bound for the overshoot
- 2 Distance from upper bound for the settling time
- 3 Distance from lower bound for the undershoot
- 4 Distance from lower bound for the % rise
- 5 Distance from lower bound for the settling time

`evaluation` does not include the distance of the signal from bounds beyond  $1.5 \times \text{settling time}$  because the signal data does not extend beyond  $1.5 \times \text{settling time}$ .

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

## See Also

`get` | `set` | `copy` | `sdo.requirements.StepResponseEnvelope`

# get

Get property values

## Syntax

```
get(req)  
get(req,PropertyName)
```

## Description

`get(req)` returns the value of all properties of the requirement object (`sdo.requirements.StepResponseEnvelope, ...`).

`get(req,PropertyName)` returns value of a specific property. Use a cell array of property names to return a cell array with multiple property values.

## Input Arguments

### **req**

Requirement object (`sdo.requirements.StepResponseEnvelope, ...`).

### **PropertyName**

Name of the requirement object (`sdo.requirements.StepResponseEnvelope, ...`) property.

## Alternatives

You can access data in properties using dot notation:

```
object.PropertyName
```

## set

Set property values

### Syntax

```
set (req, Name, Value, )
```

### Description

`set (req, Name, Value, )` sets the property value of a requirement object (`sdo.requirements.StepResponseEnvelope, ...`). Specify the property name and value using one or more `Name, Value` pair arguments.

### Input Arguments

#### **req**

Requirement object (`sdo.requirements.StepResponseEnvelope, ...`)

#### **Name, Value**

Property name of a requirement object (`sdo.requirements.StepResponseEnvelope, ...`), and the corresponding value to set.

### Examples

Specify property values.

```
r = sdo.requirements.SignalBound;  
set(r, 'BoundTimes', [0 5; 5 10], ...  
    'BoundMagnitudes', [1.1 1.1; 1.01 1.01]);
```

### Alternatives

You can set data in properties using dot notation: `object.PropertyName`. However, you can not simultaneously set the value of more than one property using dot notation, use `set` instead.

### More About

#### Tips

- Use `set` to simultaneously change properties that you cannot change independently.

# getbounds

Get bounds specified in Check block

## Syntax

```
bnds = getbounds(blockpath)
```

## Description

`bnds = getbounds(blockpath)` returns the bounds specified in the Check block specified by `blockpath`.

## Input Arguments

### **blockpath**

Check block to get bounds from, specified as a full block path inside single quotes (' '). A block path is of the form *model/subsystem/block* that uniquely identifies a block in the model. The Simulink model must be open.

## Output Arguments

### **bnds**

Cell array. The number of elements and objects they contain depends on the Check block type.

- **Check Step Response Characteristics:** Cell array of one element that contains a `sdo.requirements.StepResponseEnvelope` object.
- **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 `sdo.requirements.SignalBound` objects.
- **Check Against Reference:** Cell array of one element that contains a `sdo.requirements.SignalTracking` object.

---

**Note:** Programmatically changing the bound values in the object returned does not update them in the Block Parameters dialog box.

---

## Examples

### Get Bounds from Check Block

Retrieve bounds from a Check Step Response Characteristics block.

```
load_system('sldo_model1_stepblk');  
allBlkReq = getbounds('sldo_model1_stepblk/Step Response');
```

Type `allBlkReq{1}` to view the cell array element.

```
allBlkReq{1}
```

```
ans =
```

```
StepResponseEnvelope with 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'
```

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

### See Also

[sdo.optimize](#) | [Check Step Response Characteristics](#) | [Check Against Reference](#) | [Check Custom Bounds](#)



## getOvershoot

**Class:** sdo.requirements.PZDampingRatio

**Package:** sdo.requirements

Convert damping ratio to equivalent overshoot value

### Syntax

```
overshoot = getOvershoot(req)
```

### Description

`overshoot = getOvershoot(req)` converts the damping ratio value specified in the `DampingRatio` property of an `sdo.requirements.PZDampingRatio` object to an equivalent approximate second-order overshoot value.

### Input Arguments

**req**

`sdo.requirements.PZDampingRatio` object.

### Output Arguments

**overshoot**

Approximate second-order percent overshoot value, equivalent to the damping ratio value in `DampingRatio` property of `sdo.requirements.PZDampingRatio`.

### Examples

Convert damping ratio to approximate second-order overshoot value.

```
r = sdo.requirements.PZDampingRatio;  
r.DampingRatio = 0.1;  
overshoot = getOvershoot(r);
```

### **See Also**

sdo.requirements.PZDampingRatio | setOvershoot | evalRequirement

## setOvershoot

**Class:** sdo.requirements.PZDampingRatio

**Package:** sdo.requirements

Set overshoot to an equivalent damping ratio

### Syntax

```
req1 = setOvershoot(req,percent_overshoot)
```

### Description

`req1 = setOvershoot(req,percent_overshoot)` sets the damping ratio value to a value equivalent to percent overshoot.

### Input Arguments

**req**

sdo.requirements.PZDampingRatio object.

**percent\_overshoot**

Percent overshoot value to compute damping ratio.

### Output Arguments

**req1**

sdo.requirements.PZDampingRatio object whose `DampingRatio` property is the damping ratio value equivalent to `percent_overshoot`.

### Examples

Specify overshoot bound.

```
req = sdo.requirements.PZDampingRatio  
setOvershoot(req,20)
```

#### **See Also**

sdo.requirements.PZDampingRatio | getOvershoot | evalRequirement

# makedist

Create probability distribution object

## Syntax

```
pd = makedist(distname)
pd = makedist(distname,Name,Value)
```

## Description

`pd = makedist(distname)` creates a probability distribution object for the distribution `distname`, using the default parameter values.

Use `makedist` to specify normal or uniform distribution objects. If you have a Statistics and Machine Learning Toolbox license, you can use `makedist` to create objects for other distributions, such as the Gamma or Weibull distributions. For more information, see `makedist` in the Statistics and Machine Learning Toolbox documentation.

`pd = makedist(distname,Name,Value)` creates a probability distribution object with one or more distribution parameter values specified by name-value pair arguments.

## Examples

### Create Normal Distribution Object

Create a normal distribution object using default parameter values.

```
pd = makedist('Normal')
```

```
pd =
    NormalDistribution
    Normal distribution
    mu = 0
```

```
sigma = 1
```

#### Specify Parameters for a Normal Distribution Object

Create a normal distribution object with a mean value of  $\mu = 75$ , and a standard deviation of  $\sigma = 10$ .

```
pd = makedist('Normal', 'mu', 75, 'sigma', 10);
```

## Input Arguments

#### **distname** — Distribution name

string

Distribution name, specified as one of the following strings. The distribution specified by **distname** determines the class type of the returned probability distribution object.

Distribution Name	Description	Distribution Class
'Normal'	Normal distribution	prob.NormalDistribution
'Uniform'	Uniform distribution	prob.UniformDistribution

## 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**.

Example: `makedist('Normal', 'mu', 10)` specifies a normal distribution with parameter  $\mu$  equal to 10, and parameter  $\sigma$  equal to the default value of 1.

#### Normal Distribution

##### 'mu' — Mean

0 (default) | scalar value

Example: 'mu', 2

Data Types: single | double

**'sigma' — Standard deviation**

1 (default) | nonnegative scalar value

Example: 'sigma', 2

Data Types: single | double

**Uniform Distribution****'lower' — Lower parameter**

0 (default) | scalar value

Example: 'lower', -4

Data Types: single | double

**'upper' — Upper parameter**

1 (default) | scalar value greater than lower

Example: 'upper', 2

Data Types: single | double

## Output Arguments

**pd — Probability distribution**

probability distribution object

Probability distribution, returned as a probability distribution object of the type specified by `distname`.

**See Also**

`sdo.ParameterSpace`

**Introduced in R2014a**

## sdo.analyze

Analyze how model parameters influence cost function

### Syntax

```
r = sdo.analyze(x,y)
r = sdo.analyze(x,y,opts)
```

### Description

`r = sdo.analyze(x,y)` returns an  $N_p$ -by- $N_c$  table containing the pairwise linear correlation coefficients between each pair of columns in the `x` and `y` tables. `x` contains  $N_s$  samples of  $N_p$  model parameters. `y` contains  $N_s$  rows, each row corresponds to the cost function evaluation for a sample in `x`. Each column in `y` corresponds to a cost or constraint.

`r = sdo.analyze(x,y,opts)` specifies the analysis method(s) and method options using `opts`, an `sdo.AnalyzeOptions` object. If you specify multiple analysis methods, `r` is returned as a structure with fields for the results of each specified analysis method and method option combination.

### Examples

#### Analyze Influence of Model Parameters on Cost Function

Create arbitrary `param.Continuous` objects.

```
p1 = param.Continuous('x1',1);
p2 = param.Continuous('x2',1500);
p = [p1;p2];
```

Specify the parameter space definition for the parameters.

```
ps = sdo.ParameterSpace(p);
```

Sample the parameters.

```
x = sdo.sample(ps,50);
```



Evaluate the cost function for the samples.

```
y = sdo.evaluate(@(p) sdoExampleCostFunction(p),ps,x);
```

Model evaluated at 50 samples.

Obtain the pairwise linear correlation coefficients for the parameters and the cost function.

```
r = sdo.analyze(x,y);
```

### Specify Analysis Method

Create arbitrary `param.Continuous` objects.

```
p1 = param.Continuous('x1',1);
p2 = param.Continuous('x2',1500);
p = [p1;p2];
```

Specify the parameter space definition for the parameters.

```
ps = sdo.ParameterSpace(p);
```

Sample the parameters.

```
x = sdo.sample(ps,50);
```

Evaluate the cost function for the samples.

```
y = sdo.evaluate(@(p) sdoExampleCostFunction(p),ps,x);
```

Model evaluated at 50 samples.

Create an options object to use `Correlation Method`.

```
opt = sdo.AnalyzeOptions;
opt.Method = 'Correlation';
```

Obtain the pairwise linear correlation coefficients for the parameters and the cost function.

```
r = sdo.analyze(x,y,opt)
```

```
r =
```

```
          F          Cleq          leq
```

x1	0.9997	-0.9997	1
x2	-0.35144	0.35324	-0.35244

`r` is a structure with table fields, with one table for each type of analysis specified by `opt`.

- “Identify Key Parameters for Estimation (Code)”

## Input Arguments

### **x** — Model parameter samples

table

Model parameter samples, specified as an  $N_s$ -by- $N_p$  table.

$N_s$  is the number of samples, and  $N_p$  is the number of model parameters.

Generally, you use `sdo.sample` to generate `x`.

### **y** — Cost function evaluations

table

Cost function and constraint evaluations for each sample in `x`, specified as an  $N_s$ -by- $N_c$  table.

$N_s$  is the number of samples, and  $N_c$  is the number of cost and constraints returned by the cost function.

Generally, you use `sdo.evaluate` to generate `y`.

### **opts** — Analysis options

`sdo.AnalyzeOptions` object

Analysis options, specified as an `sdo.AnalyzeOptions` object.

## Output Arguments

### **r** — Analysis results

table | structure

Analysis results, returned as a table (when you specify a single analysis method) or a structure with table fields (when you specify multiple analysis methods).

Each table returned by `r` is an  $N_p$ -by- $N_c$  table.  $N_p$  is the number of parameters, and  $N_c$  is the number of cost and constraints returned by the cost function.

## More About

- “Analyze Relation Between Parameters and Design Requirements”

## See Also

`sdo.AnalyzeOptions` | `sdo.evaluate`

**Introduced in R2014a**

## sdo.evaluate

Evaluate cost function for samples

### Syntax

```
[y,info] = sdo.evaluate(fcn,params)
[y,info] = sdo.evaluate(fcn,params,param_samples)
[y,info] = sdo.evaluate( ____,opts)
```

### Description

`[y,info] = sdo.evaluate(fcn,params)` evaluates the cost function, `fcn`, for samples of the parameter space specified by `params` (`sdo.ParameterSpace` object). The software generates a table of samples with  $2Np+1$  rows and  $Np$  columns. These samples are generated based on the parameter space specifications in `params`, per its `ParameterDistributions`, `RankCorrelation`, and `Options` properties.  $Np$  is the number of parameters specified in `params`. `fcn` takes sample values and computes model goal values. A model goal could be a cost, constraint, or assessment of difference between experimental data and model simulation. `sdo.evaluate` applies `fcn` to each row of the table of samples. `y` is a table with one column for each model goal output returned by `fcn` and  $2Np+1$  rows. Additional evaluation information is returned in `info`.

`[y,info] = sdo.evaluate(fcn,params,param_samples)` evaluates the cost function for the specified parameter samples table, `param_samples`. For this syntax, you can specify `params` as an `sdo.ParameterSpace` object or a vector of `param.Continuous` objects. `y` is a table with one column for each model goal (cost or constraint) output returned by `fcn`. `y` contains as many rows as `param_samples`.

`[y,info] = sdo.evaluate( ____,opts)` specifies evaluation options that configure the evaluation error handling, display, and parallel computing options. This syntax can include any of the input argument combinations in the previous syntaxes.

## Examples

### Evaluate Cost Function Value for Parameter Samples

Create an arbitrary `param.Continuous` object.

```
p = param.Continuous('x',1);
```

Specify the parameter space definition for the model parameter.

```
ps = sdo.ParameterSpace(p);
```

Evaluate the cost function.

```
[y,info] = sdo.evaluate(@(p) sdoExampleCostFunction(p),ps);
```

```
Model evaluated at 3 samples.
```

The software generates three samples ( $2Np + 1$ ), and evaluates the `sdoExampleCostFunction` cost function for each sample. `Np` is the number of parameters (= 1).

- “Design Exploration using Parameter Sampling (Code)”
- “Identify Key Parameters for Estimation (Code)”

## Input Arguments

### **fcn** — Function to be evaluated by `sdo.evaluate`

function handle

Function to be evaluated by `sdo.evaluate`, specified as a function handle.

The function requires:

- One input argument, which is a vector of `param.Continuous` object.

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`. `F` is a 1x1 double.
  - **C1eq** — Value of the nonlinear inequality constraint violations evaluated at `p`.

**Cleq** is a double  $m \times 1$  vector, where  $m$  is the number of nonlinear inequality constraints.

- **Ceq** — Value of the nonlinear equality constraint violations evaluated at  $p$ .

The value is a double  $r \times 1$  vector, where  $r$  is the number of nonlinear equality constraints.

- **leq** — Value of the linear inequality constraint violations evaluated at  $p$ .

**leq** is a double  $n \times 1$  vector, where  $n$  is the number of linear inequality constraints.

- **eq** — Value of the linear equality constraint violations evaluated at  $p$ .

**eq** is a double  $s \times 1$  vector or `[]`, where  $s$  is the number of linear equality constraints.

- **Log** — Additional optional information from function evaluation. If specified, this is returned in the **Log** field of output **info**.

---

**Note:** You can use the same function handle **fcn** for sensitivity analysis, response optimization, or parameter estimation. For optimization and estimation, the solver seeks values of  $p$  that minimize **F** while satisfying constraints **Cleq**, **Ceq**, **leq** and **eq**. For more information, see `sdo.optimize` and “Write a Cost Function”.

---

#### **params** — Model parameters and states

`sdo.ParameterSpace` object | vector of `param.Continuous` objects

Model parameters and states, specified as an `sdo.ParameterSpace` object or a vector of `param.Continuous` objects. If you specify **params** as a vector of `param.Continuous` objects, you must also specify **param\_samples**.

#### **param\_samples** — Parameter samples

table

Parameter samples, specified as a table. **param\_samples** contains columns that correspond to free scalar parameters and rows that are samples of these parameters. *Free scalar parameters* refers to all the parameters specified by **params** whose **Free** property is set to 1. Specifying this property value as 1 indicates that the software can vary the value of this parameter for each evaluation.

Each column name must be equal to the name of the corresponding scalar parameter.

**opts — Evaluation options**

sdo.EvaluateOptions object

Evaluation options, specified as an sdo.EvaluateOptions object.

## Output Arguments

**y — Cost function evaluation**

table

Cost function and constraint evaluations, returned as a table.

y is a table with one column for each cost or constraint output returned by fcn, and  $N_s$  rows.

If you specify param\_samples,  $N_s$  is equal to the number of rows of param\_samples. Otherwise,  $N_s$  is equal to  $2N_p+1$ .  $N_p$  is the number of parameters specified in params.

**info — Evaluation information**

structure

Evaluation information, returned as a structure with the following fields:

- **Status** — Evaluation status for each sample, returned as a cell array of strings.

Each entry of the cell array is one of the following strings:

- 'success' — Model evaluation was successful
- 'failure' — Model evaluation resulted in all NaN results
- 'error' — Model evaluation resulted in an error
- **Log** — Additional evaluation information retrieved from the Log field of the cost function, fcn.
- **Stats** — Time to evaluate all samples, returned as a structure with the following fields:
  - **StartTime** — Evaluation start time, returned as a six-element date vector containing the current date and time in decimal form: [year month day hour minute seconds]

- **EndTime** — Evaluation end time, returned as a six-element date vector containing the current date and time in decimal form: [year month day hour minute seconds]

To determine the total evaluation time, use  
`etime(info.Stats.EndTime,info.Stats.StartTime)`.

### See Also

`sdo.ParameterSpace` | `sdo.analyze` | `sdo.EvaluateOptions` | `sdo.optimize` |  
`sdo.sample`

**Introduced in R2014a**



# createSimulator

**Class:** sdo.Experiment

**Package:** sdo

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 from Experiment**

Specify an experiment.

```
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.

```
load_system('sdoRCCircuit');  
p = sdo.getParameterFromModel('sdoRCCircuit','C1');  
p.Value = 1e-6;
```

```
p.Free = false;
experiment = sdo.Experiment('sdoRCCircuit');
experiment.Parameters = p;
```

Create a simulation scenario for the experiment.

```
sim_obj = createSimulator(experiment);
sim_obj.Parameters.Value
```

```
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.

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

## See Also

`sdo.SimulationTest`

# getValuesToEstimate

**Class:** sdo.Experiment

**Package:** sdo

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`.

## 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.

```
load_system('sdoRCCircuit');
experiment = sdo.Experiment('sdoRCCircuit');
experiment.InitialStates = sdo.getStateFromModel('sdoRCCircuit','C1');
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'
    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

`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(1,2)`, the capacitance of the C1 block, is the model parameter specified by `experiment` for estimation.

- “Estimate Model Parameters using Multiple Experiments (Code)”
- “Estimate Model Parameters Per Experiment (Code)”

### See Also

`sdo.Experiment` | `setEstimatedValues`

# setEstimatedValues

**Class:** sdo.Experiment

**Package:** sdo

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.

```
load_system('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  $\mu$  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.

- “Estimate Model Parameters using Multiple Experiments (Code)”
- “Estimate Model Parameters Per Experiment (Code)”

### **See Also**

`sdo.Experiment.getValuesToEstimate`

## **sdo.getModelDependencies**

**Package:** sdo

List of model file and path dependencies

### **Syntax**

```
[dirs,files] = sdo.getModelDependencies(modelname)
```

### **Description**

`[dirs,files] = sdo.getModelDependencies(modelname)` returns dependencies of a Simulink model. The dependencies are required for parallel computing of parameter estimation, response optimization, or sensitivity analysis tasks. The model must be open for the dependency analysis.

`sdo.getModelDependencies` may not return a complete list of model dependencies; some dependencies are undetectable. To learn more, see “Scope of Dependency Analysis” in the Simulink documentation. If your model has dependencies that are undetected or inaccessible by the parallel pool workers, then add them to the list of model dependencies.

### **Input Arguments**

**modelname**

Simulink model name, 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.

### **files**

Cell array of files that are 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 Dependencies Required for Parallel Computing**

Copy Simulink model and boiler library to temporary folder.

```
pathToLib = boilerpressure_setup;
```

Add folder to search path and open model.

```
origPath = addpath(pathToLib);  
boilerpressure_demo
```

Get model dependencies.

```
[dirs, files] = sdo.getModelDependencies('boilerpressure_demo');
```

The paths listed in `dirs` are the paths to all the file dependencies listed in `files`.

Enable parallel computing and add model dependencies.

```
opts = sdo.OptimizeOptions;  
opts.UseParallel = true;  
opts.ParallelFileDependencies = files;
```

### **Add Additional Files to Model File Dependency List**

Copy Simulink model and boiler library to temporary folder.

```
pathToLib = boilerpressure_setup;
```

Add folder to search path and open model.

```
origPath = addpath(pathToLib);  
boilerpressure_demo
```

Get model dependencies.

```
[dirs, files] = sdo.getModelDependencies('boilerpressure_demo');
```

Append an additional file, `filename.m` located in `'C:\matlab\work\'`

```
files = vertcat(files, 'C:\matlab\work\filename.m');
```

Enable parallel computing and add model dependencies.

```
opts = sdo.OptimizeOptions;  
opts.UseParallel = true;  
opts.ParallelFileDependencies = files;
```

#### **Make Local Paths Accessible to Remote Workers**

Using file dependencies is recommended, however, in some cases it can be better to choose path dependencies. For example, if parallel computing is set up on a local multi-core computer, using path dependencies is preferred as using file dependencies creates multiple copies of the dependency files on the local computer. This example shows how to use path dependencies for setting up parallel computing.

Copy Simulink model and boiler library to temporary folder.

```
pathToLib = boilerpressure_setup;
```

Add folder to search path and open model.

```
origPath = addpath(pathToLib);  
boilerpressure_demo
```

Get model dependencies.

```
[dirs, files] = sdo.getModelDependencies('boilerpressure_demo');
```

Add undetected path dependencies.

```
dirs = vertcat(dirs, '//hostname/C$/matlab/work');
```

Replace `C:/` with valid network path accessible to remote workers.

```
dirs = regexprep(dirs, 'C://', '////hostname//C//');
```

Enable parallel computing and add model dependencies.

```
opts = sdo.OptimizeOptions;  
opts.UseParallel = true;  
opts.ParallelPathDependencies = dirs;
```

- Improving Optimization Performance Using Parallel Computing

## Alternatives

- “Use Parallel Computing for Parameter Estimation”
- “Use Parallel Computing for Response Optimization”
- “Use Parallel Computing for Sensitivity Analysis”

## More About

### Tips

- `files` lists the model dependencies, and `dirs` lists the corresponding paths to these dependencies.

The model dependencies are required during parallel computing and are made accessible to the parallel pool workers by specifying one of the following:

- File dependencies: the model dependency files are copied to the parallel pool workers.

Use `files` to set the `ParallelFileDependencies` property of `sdo.OptimizeOptions` to use for parallel computing.

- Path dependencies: the paths to the model dependencies are specified to the parallel pool workers.

Use `dirs` to set the `ParallelPathDependencies` property of `sdo.OptimizeOptions` to use for parallel computing.

- Modify `files` and `dirs` to include dependencies that `sdo.getModelDependencies` cannot detect.
- Using file dependencies is recommended, however, in some cases it can be better to choose path dependencies. For example, if parallel computing is set up on a local

multi-core computer, using path dependencies is preferred as using file dependencies creates multiple copies of the dependency files on the local computer.

- “Speed Up Response Optimization Using Parallel Computing”
- “Speed Up Parameter Estimation Using Parallel Computing”
- “Analyze Model Dependencies”

### **See Also**

`sdo.OptimizeOptions` | `sdo.optimize` | `sdo.evaluate` | `sdo.EvaluateOptions`

# sdo.getParameterFromModel

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, 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

```
load_system('sldo_model1_stepblk');  
p_des = sdo.getParameterFromModel('sldo_model1_stepblk','Kp');
```

### Get Multiple Model Parameters as Optimization Design Variables

```
paramname = {'Kp','Ki','Kd'};  
load_system('sldo_model1_stepblk');  
p_des = sdo.getParameterFromModel('sldo_model1_stepblk',paramname);
```

### Get All Model Parameters as Optimization Design Variables

```
load_system('sldo_model1_stepblk');  
p_des = sdo.getParameterFromModel('sldo_model1_stepblk');
```

- “Design Optimization to Meet Step Response Requirements (Code)”
- “Estimate Model Parameter Values (Code)”

## Alternatives

“Specify Design Variables”

### See Also

`sdo.optimize` | `sdo.setValueInModel`



# sdo.getStateFromModel

**Package:** sdo

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

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

### Get Model States

```
modelName = 'sdoAircraft';  
load_system(modelname);  
s = sdo.getStateFromModel(modelname);
```

`s` is a vector containing nine `param.State` objects, which represent all the states of the `sdoAircraft` model.

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

## See Also

`sdo.Experiment` | `param.State`

# sdo.getValueFromModel

**Package:** sdo

Get design variable value from model

## Syntax

```
param_value = sdo.getValueFromModel(modelname,param_des)
```

## Description

`param_value = sdo.getValueFromModel(modelname,param_des)` gets the value of a design variable in a Simulink model. The model must be open.

## Input Arguments

### **modelname**

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

### **param\_des**

Design variables, specified as:

- A string inside single quotes (' ') for one variable or a cell array of strings for multiple variables
- A `param.Continuous` object for one variable or a vector of objects for multiple variables, created using `sdo.getParameterFromModel`

## Output Arguments

### **param\_value**

Design variable value in the model.

A cell array for multiple variable values.

## Examples

### Get Current Design Variable Value From Model

```
load_system('sldo_model1_stepblk');  
p_value = sdo.getValueFromModel('sldo_model1_stepblk', 'Kp');
```

Alternatively, type:

```
p_des = sdo.getParameterFromModel('sldo_model1_stepblk', 'Kp');  
p_value = sdo.getValueFromModel('sldo_model1_stepblk', p_des);
```

### See Also

sdo.optimize

## sdo.scatterPlot

Scatter plot of samples

### Syntax

```
sdo.scatterPlot(X,Y)
sdo.scatterPlot(X)
[H,AX,BigAX,P,PAx] = sdo.scatterPlot( ___ )
```

### Description

`sdo.scatterPlot(X,Y)` creates a matrix of subaxes containing scatter plots of the columns of  $X$  against the columns of  $Y$ . If  $X$  is  $p$ -by- $n$  and  $Y$  is  $p$ -by- $m$ , then `sdo.scatterPlot` creates a matrix of  $n$ -by- $m$  subaxes.  $X$  and  $Y$  must have the same number of rows.

`sdo.scatterPlot(X)` is the same as `sdo.scatterPlot(X,X)`, except that the subaxes along the diagonal are replaced with histogram plots of the data in the corresponding column of  $X$ . For example, the subaxes along the diagonal in the  $i$ th column is replaced by `hist(X(:,i))`.

`[H,AX,BigAX,P,PAx] = sdo.scatterPlot( ___ )` returns the handles to the graphic objects. Use these handles to customize the scatter plot. For example, you can specify titles for the subaxes.

### Examples

#### Scatter Plot of Parameter Samples and Cost Function Evaluations

Generally, you use the `sdo.scatterPlot(X,Y)` syntax with  $X$  specifying the samples and  $Y$  specifying the cost function value for each sample. Use the `sdo.evaluate` command to perform the cost function evaluation to generate  $Y$ . For this example, obtain 100 samples of the  $AC$  and  $K$  parameters of the `sdoHydraulicCylinder` model. Calculate the cost function as a function of  $AC$  and  $K$ . Create a scatter plot to see the sample and cost function values.

Load the `sdoHydraulicCylinder` model.

```
load_system('sdoHydraulicCylinder');
```

Generate 100 samples of the `Ac` and `K` parameters.

```
p = sdo.getParameterFromModel('sdoHydraulicCylinder',{'Ac','K'});
```

```
ps = sdo.ParameterSpace(p);
```

```
X = sdo.sample(ps,100);
```

The first operation obtains the `Ac` and `K` parameters as a vector, `p`. The second operation creates an `sdo.ParameterSpace` object, `ps`, that specifies the probability distributions of the parameter samples. The third operation generates 100 samples of each parameter, returned as a `Table`, `X`.

Calculate the cost function value table.

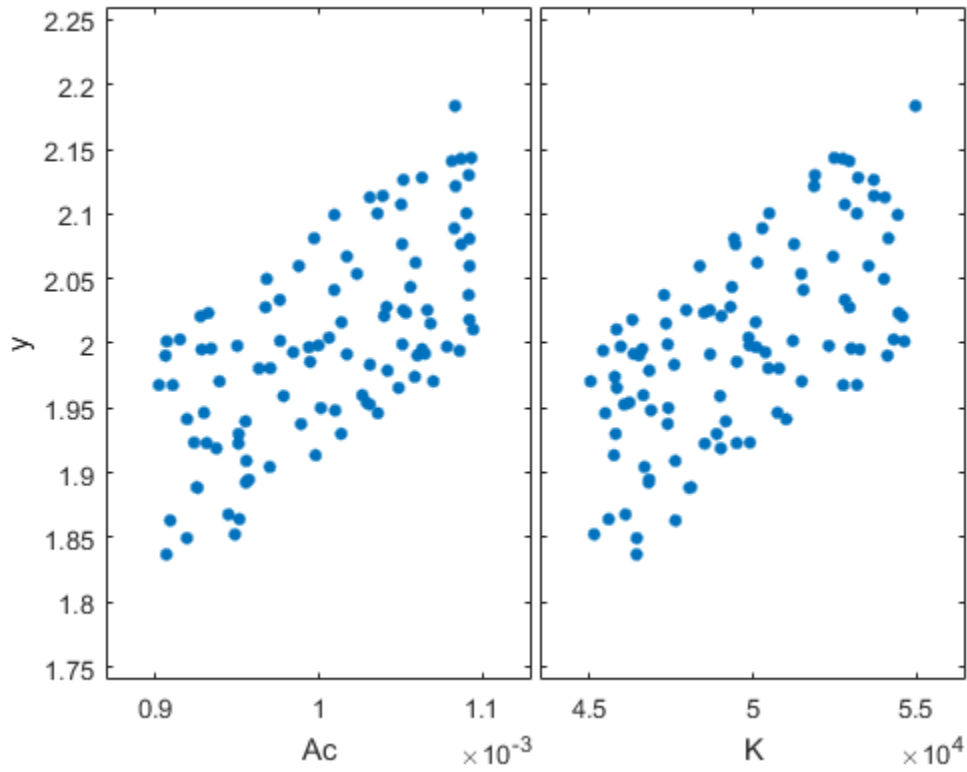
```
Ac_mean = mean(X(:,1));
```

```
K_mean = mean(X(:,2));
```

```
Y = table(X(:,1)/Ac_mean+X(:,2)/K_mean,'VariableNames',{'y'});
```

Create a scatter plot of `X` and `Y`.

```
sdo.scatterPlot(X,Y);
```



### Scatter Plot of Parameter Samples

Sample the AC and K parameters of the `sdoHydraulicCylinder` model. Use a scatter plot to analyze the samples.

Load the `sdoHydraulicCylinder` model.

```
load_system('sdoHydraulicCylinder');
```

Generate 100 samples of the AC and K parameters.

```
p = sdo.getParameterFromModel('sdoHydraulicCylinder',{'Ac','K'});
```

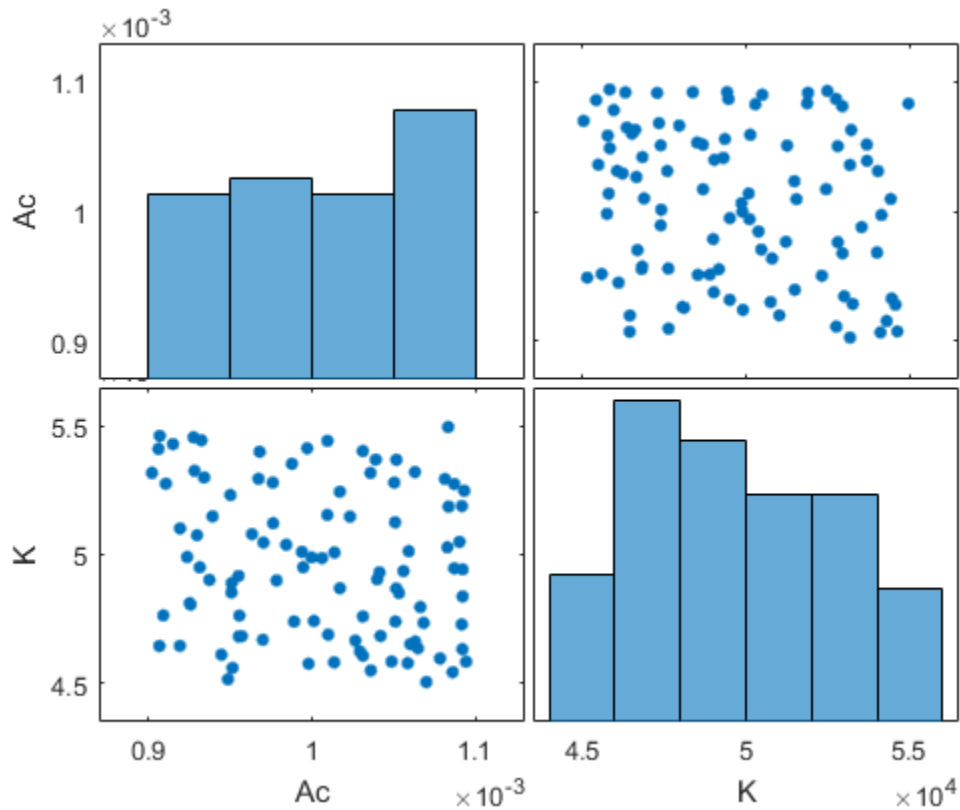
```
ps = sdo.ParameterSpace(p);
```

```
X = sdo.sample(ps, 100);
```

The first operation obtains the AC and K parameters as a vector, **p**. The second operation creates an `sdo.ParameterSpace` object, **ps**, that specifies the probability distributions of the parameter samples. The third operation generates 100 samples of each parameter, returned as a `Table`, **X**.

Create a scatter plot of **X**.

```
sdo.scatterPlot(X);
```



#### Set Scatter Plot Properties Using Handles

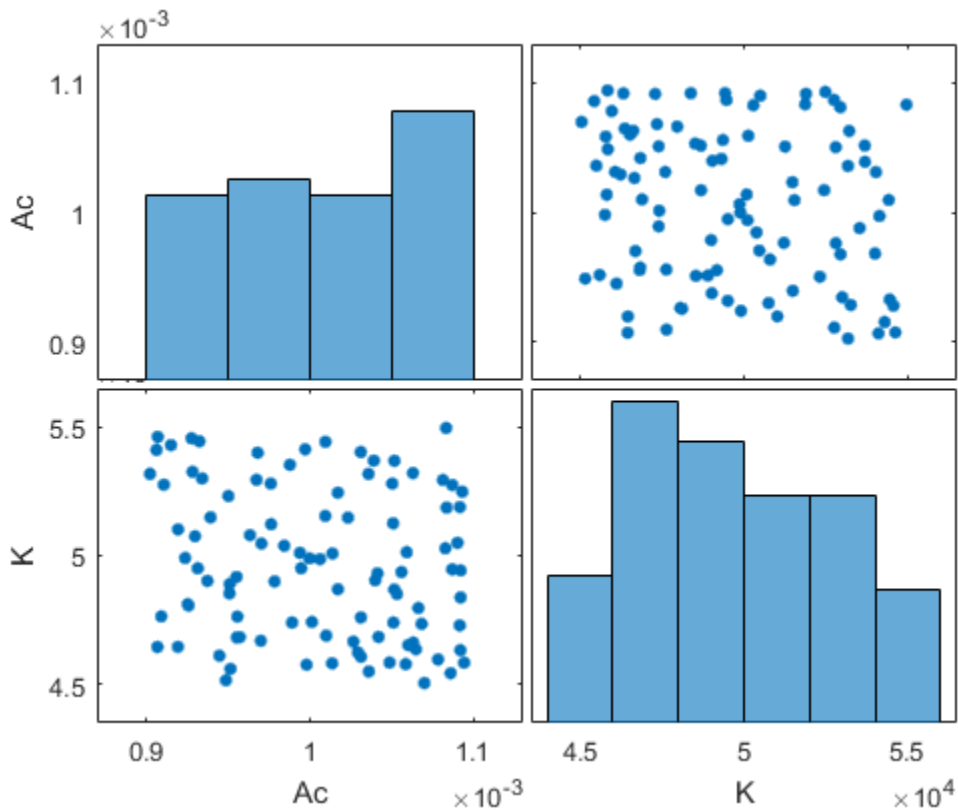
Generate samples of the AC and K parameters of the `sdoHydraulicCylinder` model.



```
load_system('sdoHydraulicCylinder');
p = sdo.getParameterFromModel('sdoHydraulicCylinder',{'Ac','K'});
ps = sdo.ParameterSpace(p);
X = sdo.sample(ps,100);
```

Create a scatter plot matrix and return the object handles and the axes handles.

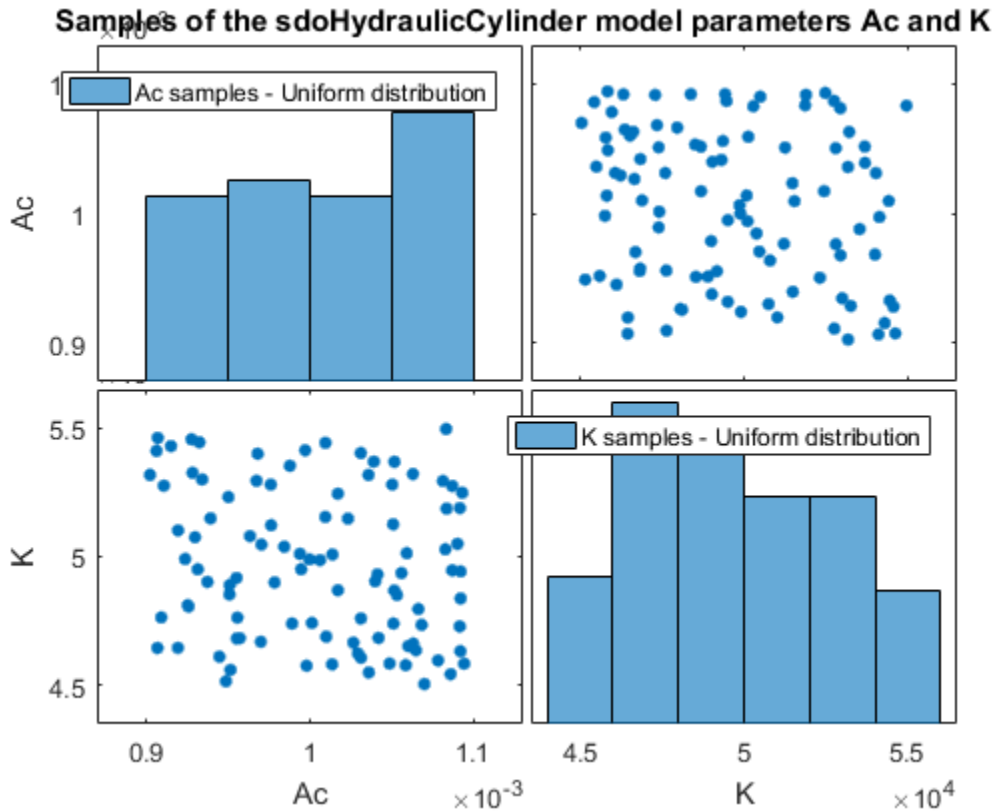
```
figure
[H,AX,BigAX,P,PAx] = sdo.scatterPlot(X);
```



To set properties for the scatter plots, use the handles in H. To set properties for the histograms, use the patch handles in P. To set axes properties, use the axes handles, AX, BigAX, and PAX.

Specify a title for the plot matrix and add legends specifying the sample distribution for each parameter.

```
title('Samples of the sdoHydraulicCylinder model parameters Ac and K')
legend(PAx(1),'Ac samples - Uniform distribution')
legend(PAx(2),'K samples - Uniform distribution')
```



- “Design Exploration using Parameter Sampling (Code)”
- “Identify Key Parameters for Estimation (Code)”

## Input Arguments

### **X — Sampled data**

table

Sampled data, specified as a table.

### **Y — Cost function evaluation data**

table

Cost function evaluation data, specified as a table.

## Output Arguments

### **H — Line object handles**

matrix

Line object handles, returned as a matrix. This is a unique identifier, which you can use to query and modify the properties of a specific line object. The line objects are used to create the scatter plots.

### **AX — Subaxes handles**

matrix

Subaxes handles, returned as a matrix. This is a unique identifier, which you can use to query and modify the properties of a specific subaxes.

### **BigAX — Big axes handle**

scalar

Big axes handle, returned as a scalar. This is a unique identifier, which you can use to query and modify properties of the big axes. **BigAX** is left as the current axes (**gca**) so that a subsequent **title**, **xlabel**, or **ylabel** command will center text with respect to the big axes.

### **P — Patch object handles**

vector | []

Patch object handles, returned as a vector or []. If histogram plots are created, then **P** is returned as a vector of patch object handles for the histogram plots. These are unique

identifiers, which you can use to query and modify the properties of a specific patch object. If no histogram plots are created, then **P** is returned as empty brackets.

### **PAX — Handle to invisible histogram axes**

vector | []

Handle to invisible histogram axes, returned as a vector or []. If histogram plots are created, then **PAX** is returned as a vector of histogram axes handles. These are unique identifiers, which you can use to query and modify the properties of a specific axes, such as the axes scale. If no histogram plots are created, then **PAX** is returned as empty brackets.

## **More About**

- “Analyze Relation Between Parameters and Design Requirements”

## **See Also**

`sdo.evaluate` | `sdo.sample`

**Introduced in R2014a**

# isreal

**Class:** param.Continuous

**Package:** param

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.

**Default:**

## Examples

**Determine if Parameter Value and Minimum/Maximum Bounds are Real**

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

```
ans =
```

```
1
```

Because the `Value`, `Minimum`, and `Maximum` properties of all parameters in `p` are real, `isreal` returns 1.

#### **See Also**

`param.Continuous`

# Parameter Estimation Tool

Estimate model parameters and initial states

## Description

The Parameter Estimation tool estimates parameters and initial states of a Simulink model using measured data. The tool increases model accuracy so that the model reflects the measured hardware behavior. For example, you can automatically estimate electric motor resistance, inductance, and inertia from measured voltage and motor speed data.

Using this tool, you can:

- Import and preprocess measured data.
- Find the most influential parameters to estimate (with the Sensitivity Analysis tool).
- Estimate model parameters and initial states, and monitor estimation progress.
- Validate estimation results.

You can generate MATLAB code from the tool, and accelerate parameter estimation using parallel computing and Simulink fast restart.

## Open the Parameter Estimation Tool App

- Simulink model editor: Select **Analysis > Parameter Estimation**.
- MATLAB command prompt: Enter `spetool('modelname')`.

## Examples

- “How the Software Formulates Parameter Estimation as an Optimization Problem”
- “Estimate Parameters from Measured Data”
- “Estimate Model Parameter Values (GUI)”
- “Estimate Model Parameters Per Experiment (GUI)”
- “Estimate Model Parameters and Initial States (GUI)”

### **Programmatic Use**

`spetool('modelName')` opens the Parameter Estimation tool and creates a new session. The model must be open or on the MATLAB path.

`spetool(spesession)` opens a previously saved Parameter Estimation tool session.

### **See Also**

`sdo.optimize`

**Introduced before R2006a**



# Response Optimization Tool

Optimize model response to satisfy design requirements, test model robustness

## Description

The Response Optimization tool automatically optimizes system parameters to improve design characteristics such as response time, bandwidth, and energy consumption. The system parameters can be tuned to meet time-domain and frequency-domain requirements such as overshoot and phase margin, and custom requirements.

Using this tool, you can:

- Specify design requirements.
- Incorporate parameter uncertainty to validate the robustness of your design.
- Find the most influential parameters to optimize (with the Sensitivity Analysis tool).
- Optimize model parameters, and monitor optimization progress.

You can generate MATLAB code from the tool, and you can accelerate response optimization using parallel computing and Simulink fast restart.

## Open the Response Optimization Tool App

- Simulink model editor: Select **Analysis > Response Optimization**.
- MATLAB command prompt: Enter `sdotool('modelName')`.

## Examples

- “How the Optimization Algorithm Formulates Minimization Problems”
- “Design Optimization to Meet Step Response Requirements (GUI)”
- “Design Optimization to Track Reference Signal (GUI)”
- “Design Optimization Using Frequency-Domain Check Blocks (GUI)”
- “Design Optimization to Meet a Custom Objective (GUI)”
- “Specify Custom Signal Objective with Uncertain Variable (GUI)”

## **Programmatic Use**

`sdotool('modelName')` opens the Response Optimization tool and creates a new session. The model must be open or on the MATLAB path.

`sdotool(sdosession)` opens a previously saved Response Optimization tool session.

## **See Also**

`sdo.optimize`

**Introduced in R2011b**

# addParameter

**Class:** sdo.ParameterSpace

**Package:** sdo

Add parameter to `sdo.ParameterSpace` object

## Syntax

```
ps = addParameter(ps0,p)
ps = addParameter(ps0,p,pdist)
```

## Description

`ps = addParameter(ps0,p)` adds a model parameter, `p`, to an `sdo.ParameterSpace` object, `ps0`, and returns the updated object, `ps`. The software updates the `ParameterNames` property to include the parameter name.

The software also updates the `ParameterDistributions` property to specify the uniform distribution for the parameter. The software sets the values of the two parameters of the uniform distribution:

- **Lower** — Set to `p.Minimum`. If `p.Minimum` is equal to `-Inf`, then the software sets `Lower` to `0.9*p.Value`. Unless `p.Value` is equal to 0, in which case the software sets `Lower` to `-1`.
- **Upper** — Set to `p.Maximum`. If `p.Maximum` is equal to `Inf`, then the software sets `Upper` to `1.1*p.Value`. Unless `p.Value` is equal to 0, in which case the software sets `Upper` to 1.

`ps = addParameter(ps0,p,pdist)` specifies the probability distribution of `p`.

## Input Arguments

### **ps0**

Parameter space, specified as an `sdo.ParameterSpace` object.

### **p**

Model parameters and states, specified as a vector of `param.Continuous` objects.

For example, `sdo.getParameterFromModel('sdoHydraulicCylinder', {'Ac', 'K'})`.

### **pdist**

Probability distribution of model parameters, specified as a vector of univariate probability distribution objects.

- If `pdist` is the same size as `p`, the software specifies each entry of `pdist` as the probability distribution of the corresponding parameter in `p`.
- If `pdist` contains only one distribution, the software specifies this object as the probability distribution for all the parameters in `p`.

Use the `makedist` command to create a univariate probability distribution object. For example, `makedist('Normal', 'mu', 10, 'sigma', 3)`.

To check if `pdist` is a univariate distribution object, run `isa('pdist', 'prob.UnivariateDistribution')`.

## Output Arguments

### **ps**

Updated parameter space, returned as an `sdo.ParameterSpace` object.

## Examples

### **Add Parameters to Parameter Space Object**

Create an `sdo.ParameterSpace` object, `ps`, for the AC parameter of `sdoHydraulicCylinder` model.

```
load_system('sdoHydraulicCylinder');  
pAc = sdo.getParameterFromModel('sdoHydraulicCylinder', 'Ac');  
ps = sdo.ParameterSpace(pAc);
```

Add the K parameter to ps.

```
pK = sdo.getParameterFromModel('sdoHydraulicCylinder','K');  
ps = addParameter(ps,pK);
```

### Add Parameter with Specified Distribution to Parameter Space Object

Create an sdo.ParameterSpace object for the Ac and C1 parameters of the sdoHydraulicCylinder model.

```
load_system('sdoHydraulicCylinder');  
p = sdo.getParameterFromModel('sdoHydraulicCylinder',{'Ac','C1'});  
ps = sdo.ParameterSpace(p);
```

Add the K parameter to ps. Specify a normal distribution for K.

```
pK = sdo.getParameterFromModel('sdoHydraulicCylinder','K');  
pKdist = makedist('Normal','mu',pK.Value,'sigma',2);  
ps = addParameter(ps,pK,pKdist);
```

### See Also

sdo.ParameterSpace.removeParameter | makedist | sdo.getParameterFromModel | sdo.sample

### More About

- “Generate Parameter Samples for Sensitivity Analysis”

# removeParameter

**Class:** sdo.ParameterSpace

**Package:** sdo

Remove parameter from sdo.ParameterSpace object

## Syntax

```
ps = removeParameter(ps0,p)
```

## Description

ps = removeParameter(ps0,p) removes the parameter, p, from the sdo.ParameterSpace object, ps0, and returns the updated object, ps.

## Input Arguments

### ps0

Parameter space, specified as an sdo.ParameterSpace object.

### p

Parameters to be removed, specified as:

- Vector of param.Continuos objects — Parameter objects. For example, p = sdo.getParameterFromModel('sdoHydraulicCylinder','Ac').
- String — Parameter name. For example, 'Ac'.

## Output Arguments

### ps

Updated parameter space, returned as an sdo.ParameterSpace object.

## Examples

### Remove Parameter From sdo.ParameterSpace Object

Create an `sdo.ParameterSpace` object, `ps`, for the `Ac` and `K` parameters of the `sdoHydraulicCylinder` model.

```
load_system('sdoHydraulicCylinder');
p = sdo.getParameterFromModel('sdoHydraulicCylinder',{'Ac','K'});
ps = sdo.ParameterSpace(p)
```

```
ps =
```

```
ParameterSpace with properties:
```

```
ParameterNames: {'Ac' 'K'}
ParameterDistributions: [1x2 prob.UniformDistribution]
RankCorrelation: []
Options: [1x1 sdo.SampleOptions]
Notes: []
```

Remove `K` from `ps`.

```
ps = removeParameter(ps,p(2))
```

```
ps =
```

```
ParameterSpace with properties:
```

```
ParameterNames: {'Ac'}
ParameterDistributions: [1x1 prob.UniformDistribution]
RankCorrelation: []
Options: [1x1 sdo.SampleOptions]
Notes: []
```

Remove `AC` from `ps` using the parameter name.

```
ps = removeParameter(ps, 'Ac');
```

### See Also

sdo.ParameterSpace | sdo.ParameterSpace.AddParameter |  
sdo.getParameterFromModel



# setDistribution

**Class:** sdo.ParameterSpace

**Package:** sdo

Set distribution of parameter in sdo.ParameterSpace object

## Syntax

```
ps = setDistribution(ps0,p,pdist)
```

## Description

`ps = setDistribution(ps0,p,pdist)` updates the `ParameterDistributions` property of the `sdo.ParameterSpace` object, `ps0`, for the specified parameters, `p`, and returns the updated object, `ps`.

## Input Arguments

### ps0

Parameter space, specified as an `sdo.ParameterSpace` object.

### p

Parameters whose distributions are to be updated, specified as:

- Vector of `param.Continuos` objects — Parameter objects. For example, `p = sdo.getParameterFromModel('sdoHydraulicCylinder','Ac')`.
- String — Parameter name. For example, `'Ac'`.

### pdist

Probability distribution for model parameters, specified as a vector of univariate probability distribution objects.

- If `pdist` is the same size as `p`, the software specifies each entry of `pdist` as the probability distribution of the corresponding parameter in `p`.
- If `pdist` contains only one distribution, the software specifies this object as the probability distribution for all the parameters in `p`.

Use the `makedist` command to create a univariate probability distribution object. For example, `makedist('Normal','mu',10,'sigma',3)`.

To check if `pdist` is a univariate distribution object, run `isa('pdist','prob.UnivariateDistribution')`.

## Output Arguments

**ps**

Updated parameter space, returned as an `sdo.ParameterSpace` object.

## Examples

### Set Distribution of Parameters in Parameter Space

Create an `sdo.ParameterSpace` object for the `Ac` and `K` parameters of `sdoHydraulicCylinder` model.

```
load_system('sdoHydraulicCylinder');  
p = sdo.getParameterFromModel('sdoHydraulicCylinder',{'Ac','K'});  
ps = sdo.ParameterSpace(p);
```

By default, a uniform distribution is specified for all parameters in `p`.

```
ps.ParameterDistributions
```

```
ans =
```

```
1x2 UniformDistribution array
```

Specify a normal distribution for `Ac` and `K`.

```
pAcdist = makedist('Normal','mu',p(1).Value,'sigma',1);
```

```
pKdist = makedist('Normal','mu',p(2).Value,'sigma',3);  
ps = setDistribution(ps,p,[pAcdist;pKdist]);
```

### **See Also**

makedist | sdo.getParameterFromModel | sdo.sample

## **sdo.setCheckBlockEnabled**

**Package:** sdo

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

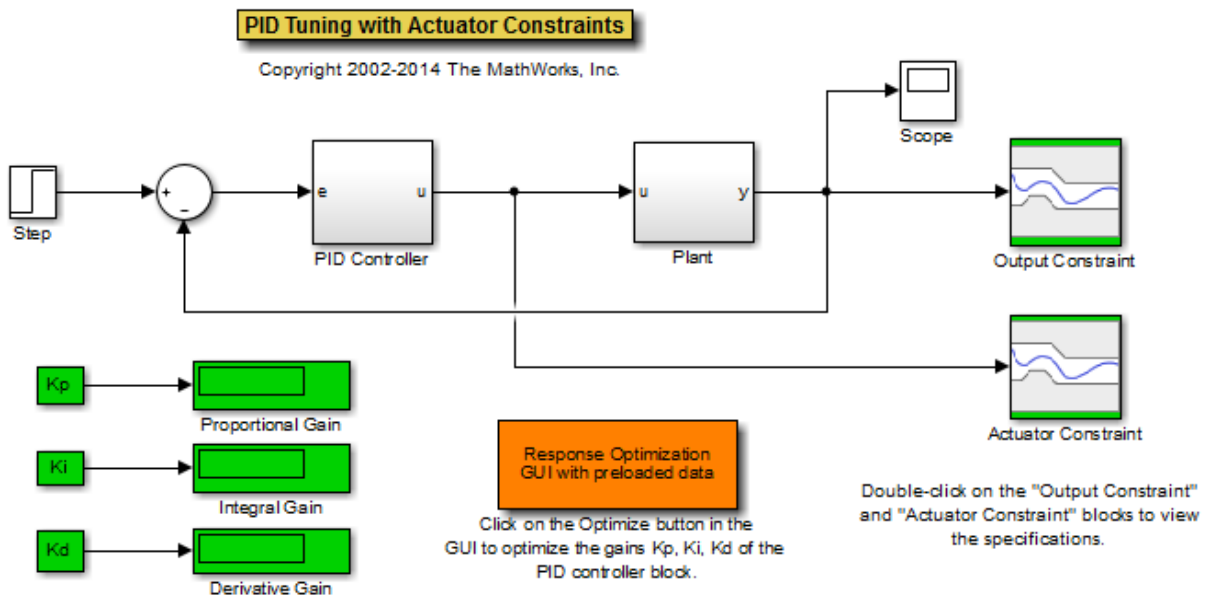
Original values of the Enabled block parameter of the model check blocks, returned as a cell array of strings.

## Examples

### Disable Model Check Blocks

Open the model.

```
modelname = 'pidtune_demo';
open_system(modelname);
```



Disable the model check blocks in a model.

```
state = 'off';
```

```
chkBlkState = sdo.setCheckBlockEnabled(modelname, state);
```

Restore the enabled state of the model check blocks.

```
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

**Package:** sdo

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, 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.

**Default:**

## **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`



# sdo.optimize

**Package:** sdo

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` (the default optimization method) to solve a design optimization problem of the form:

$$\min_p F(p) \text{ subject to } \begin{cases} C_{leq}(p) \leq 0 \\ C_{eq}(p) = 0 \\ A \times p \leq B \\ A_{eq} \times p = B_{eq} \\ lb \leq p \leq ub \end{cases}$$

where

- $p$  — Design variable
- $C_{leq}, C_{eq}$  — Nonlinear inequality and equality constraints
- $A, B$  — Linear inequality constraints
- $A_{eq}, B_{eq}$  — Linear equality constraints
- $lb, ub$  — Upper and lower bounds on  $p$

[param\_opt,opt\_info] = sdo.optimize(opt\_fcn,param,options) specifies the optimization options. For parameter estimation, you typically use the Nonlinear Least Squares method:

```
opts = sdo.OptimizeOptions('Method','lsqnonlin');
```

[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  $Cleq(p) \leq 0$ .  
**Cleq** is a double  $m \times 1$  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  $r \times 1$  vector, where **r** is the number of nonlinear equality constraints.
  - **leq** — Value of the linear inequality constraint violations evaluated at **p**. The solver satisfies  $leq(p) \leq 0$ .  
**leq** is a double  $n \times 1$  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  $s \times 1$  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 `Cleq`, `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  $n \times 1$  vector that contains  $dF(p)/dp$ , where  $n$  is the number of scalar parameters.
- `Cleq` — Double  $n \times m$  matrix that contains  $dCleq(p)/dp$ , where  $m$  is the number of nonlinear inequality constraints.
- `Ceq` — Double  $n \times r$  matrix that contains  $dCeq(p)/dp$ , where  $r$  is the number of nonlinear equality constraints.

For an example, type `edit sdoExampleCostFunction`.

**Default:**

**param**

A `param`. Continuous object or a vector of objects.

**Default:**

**options**

Optimization options.

`options` is an options set, created using `sdo.OptimizeOptions`. Use this options set to specify:

- Optimization method
- Maximum number of iterations
- Tolerances

**Default:**

### **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
- **Options** — Optimization options, specified using `sdo.OptimizeOptions`

**Default:**

## **Output Arguments**

### **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.
- **Cleq** — 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.

- **leq** — Optimized linear equality constraint violations.

The field appears if you specify a linear equality constraint in `opt_fcn`.

The value is a double `nx1` 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);
```

```
Optimization started 15-Feb-2016 15:16:25
```

Iter	F-count	f(x)	max constraint	Step-size	First-order optimality
0	3	1	0		
1	5	0.09	0	0.7	0.59
2	6	0.0716349	0.001047	0.0324	0.0129
3	7	0.0717968	9.127e-08	0.000302	2.37e-06

Local minimum found that satisfies the constraints.

Optimization completed because the objective function is non-decreasing in feasible directions, to within the selected value of the optimality tolerance, and constraints are satisfied to within the selected value of the constraint tolerance

- “Design Optimization to Meet Step Response Requirements (Code)”
- “Estimate Model Parameter Values (Code)”
- “Design Optimization to Meet a Custom Objective (Code)”
- “Write a Cost Function”

## Alternatives

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

## More About

### 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.

- “Create Function Handle”

## See Also

`param.Continuous` | `sdo.OptimizeOptions`

## sdo.sample

Generate parameter samples

### Syntax

```
x = sdo.sample(ps)
x = sdo.sample(ps,N)
x = sdo.sample( ____,opt)
```

### Description

`x = sdo.sample(ps)` generates samples using the specified parameter space definition, `ps`. The output sample table, `x`, has  $2Np+1$  rows and  $Np$  columns. Each column corresponds to a parameter and each row corresponds to a sample of the parameters.  $Np$  is the number of parameters in `ps`. The samples are generated as per the `ParameterDistributions`, `RankCorrelation`, and `Options` property of `ps`.

`x = sdo.sample(ps,N)` specifies the number of samples to be generated. `x` is a table with  $N$  rows and  $Np$  columns.

`x = sdo.sample( ____,opt)` specifies sampling options such as the sampling method. This syntax can include any of the input argument combinations in the previous syntaxes.

### Examples

#### Generate Parameter Samples

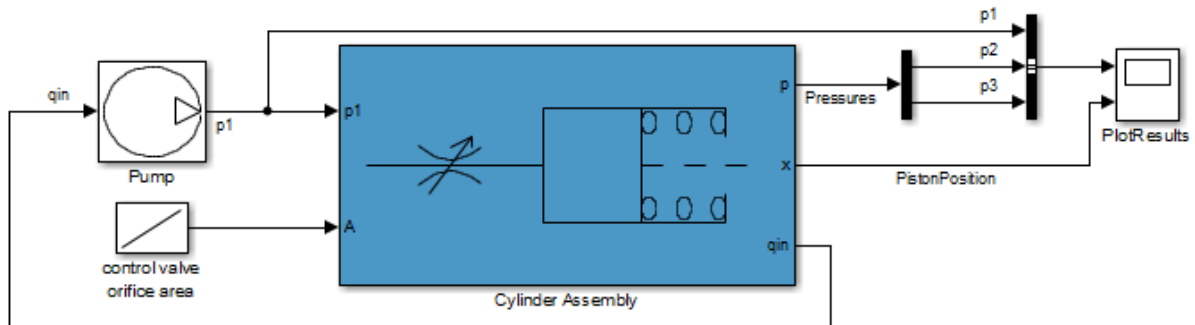
Open the model.

```
open_system('sdoHydraulicCylinder');
```





## Single Hydraulic Cylinder Simulation



Copyright 1990-2011 The MathWorks, Inc.

Obtain the parameters from the model.

```
p = sdo.getParameterFromModel('sdoHydraulicCylinder',{'Ac','K'});
```

Create an `sdo.ParameterSpace` object to specify the sample distributions.

```
ps = sdo.ParameterSpace(p);
```

Generate samples for the parameters.

```
x = sdo.sample(ps);
```

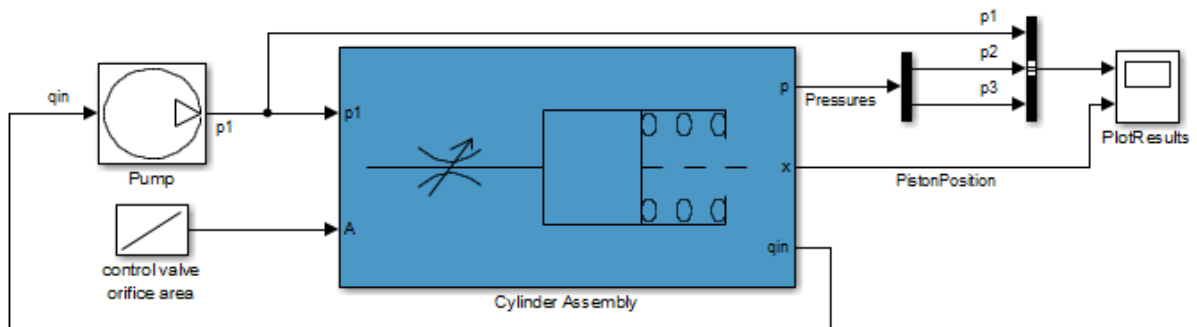
### Specify Number of Samples

Open the model.

```
open_system('sdoHydraulicCylinder');
```



## Single Hydraulic Cylinder Simulation



Copyright 1990-2011 The MathWorks, Inc.

Obtain the parameters from the model.

```
p = sdo.getParameterFromModel('sdoHydraulicCylinder',{'Ac','K'});
```

Create an `sdo.ParameterSpace` object to specify the sample distributions.

```
ps = sdo.ParameterSpace(p);
```

Generate 50 samples for the parameters.

```
x = sdo.sample(ps,50);
```

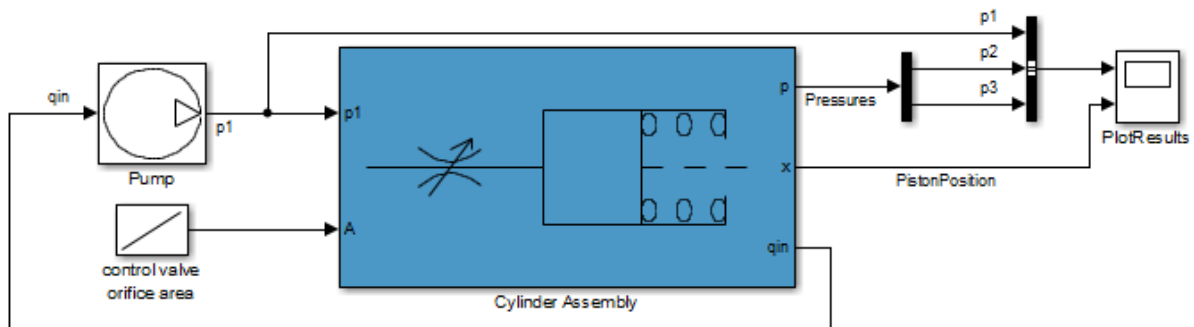
### Specify Sampling Options

Open the model.

```
open_system('sdoHydraulicCylinder');
```



## Single Hydraulic Cylinder Simulation



Copyright 1990-2011 The MathWorks, Inc.

Obtain the parameters from the model.

```
p = sdo.getParameterFromModel('sdoHydraulicCylinder',{'Ac','K'});
```

Create an `sdo.ParameterSpace` object to specify the sample distributions.

```
ps = sdo.ParameterSpace(p);
```

Specify the sampling method used by the software.

```
opt = sdo.SampleOptions;  
opt.Method = 'lhs';
```

The 'lhs' (Latin hypercube) sampling method requires a Statistics and Machine Learning Toolbox™ license.

Generate 50 samples for the parameters using Latin hypercube sampling.

```
x = sdo.sample(ps,50,opt);
```

- “Design Exploration using Parameter Sampling (Code)”
- “Identify Key Parameters for Estimation (Code)”

### Input Arguments

#### **ps** — Parameter space distribution

`sdo.ParameterSpace` object

Parameter space distribution definition, specified as an `sdo.ParameterSpace` object.

#### **N** — Number of samples

positive integer

Number of samples to be generated for the parameters, specified as a positive integer.

Ideally, you want to use the smallest number of samples that yield useful results, because each sample requires a model evaluation.

As the number of parameters increases, the number of samples needed to explore the design space generally increases. For correlation or regression analysis, consider using  $10Np$  samples, where  $Np$  is the number of parameters.

Example: 10

#### **opt** — Sampling options

`sdo.SampleOptions` object

Sampling options, specified as an `sdo.SampleOptions` object.

### Output Arguments

#### **x** — Parameter samples

table

Parameter samples, returned as a `table`.

`x` has  $Ns$  rows and  $Np$  columns. Each column corresponds to a parameter and each row corresponds to a sample of the parameters.  $Np$  is the number of parameters in `ps`. If you specify `N`,  $Ns$  is equal to `N`. Otherwise,  $Ns$  is equal to  $2Np+1$ .

### More About

- “Generate Parameter Samples for Sensitivity Analysis”

## **See Also**

sdo.SampleOptions | sdo.evaluate

**Introduced in R2014a**

# fastRestart

**Class:** sdo.SimulationTest

**Package:** sdo

Simulate Simulink model in fast restart mode using simulation scenario

## Syntax

```
Simulator_out = fastRestart(Simulator,EnablefastRestart)
```

## Description

`Simulator_out = fastRestart(Simulator,EnablefastRestart)` configures a Simulink model and simulation scenario specified in `sdo.SimulationTest` object, `Simulator`, for simulation in fast restart mode. Fast restart configures the model to compile once when first simulated. Subsequent model simulations reuse the compiled data, speeding up subsequent simulation runs.

## Tips

- To optimize, evaluate, or estimate a Simulink model, first create a simulator configured for fast restart (`Simulator_out`). Then use `Simulator_out` as an input to your cost function. If you create a simulator in the cost function, you cannot use fast restart mode.
- When you enable fast restart, you can change only tunable properties of the model.
- To linearize the model, specify the linearization logging information in the `SystemLoggingfInfo` property of `Simulator` before configuring the model for fast restart.

## Input Arguments

**Simulator** — Simulation scenario for Simulink model

`sdo.SimulationTest` object (default)

Simulation scenario for Simulink model, specified as an `sdo.SimulationTest` object. A simulation scenario specifies input signals, model parameter and initial state values, and signals to log for a model.

If you want to linearize the model, for example if you have frequency-domain design requirements, specify the linearization logging information in the `SystemLoggingInfo` property of `Simulator`.

### **EnablefastRestart** — Configuration of model and simulation scenario for fast restart

'on' (default) | 'off'

Configuration of model and simulation scenario for fast restart, specified as a string with one of the following values:

- 'on' — Initializes a Simulink model to simulate in the fast restart mode using simulation scenario specified in `Simulator` object. Fast restart configures the model to compile once when first simulated. Subsequent model simulations reuse the compiled data, speeding up subsequent simulation runs.

Once you have initialized a model in fast restart, you can change only tunable properties of the model.

- 'off' — Turns off the fast restart mode. Use this option to change nontunable properties of your model.

## Output Arguments

### **Simulator\_out** — Simulation scenario configured for fast restart

`sdo.SimulationTest` object

Simulation scenario configured for fast restart, returned as a `sdo.SimulationTest` object.

## Examples

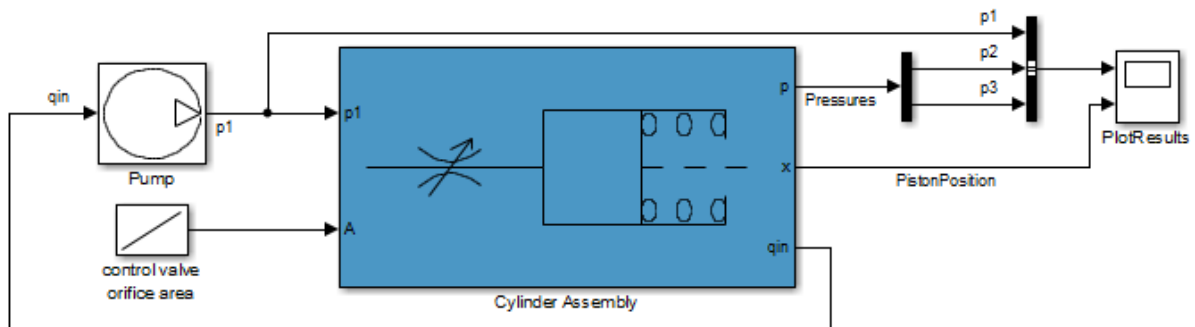
### **Simulate a Model in Fast Restart Mode**

Open the model.

```
open_system('sdoHydraulicCylinder')
```



## Single Hydraulic Cylinder Simulation



Copyright 1990-2011 The MathWorks, Inc.

Create a simulation scenario for the model.

```
simulator = sdo.SimulationTest('sdoHydraulicCylinder');
```

Configure the model and `simulator` for fast restart.

```
simulator = fastRestart(simulator,'on');
```

Simulate the model.

```
simulator = sim(simulator);
```

The first simulation in fast restart mode requires the model to compile. Subsequent simulations execute in fast restart mode and reuse the compiled data.

Adjust tunable model parameters.

```
Ac = sdo.getParameterFromModel('sdoHydraulicCylinder','Ac');
Ac.Value = 0.5;
simulator.Parameters = Ac;
```

Simulate the model again.

```
simulator = sim(simulator);
```

The model simulates in fast restart mode.



Turn off fast restart mode.

```
simulator = fastRestart(simulator, 'off');
```

- “Improving Optimization Performance using Fast Restart (Code)”
- “Use Fast Restart Mode During Response Optimization”
- “Use Fast Restart Mode During Sensitivity Analysis”

## See Also

[find](#) | [sdo.evaluate](#) | [sdo.optimize](#) | [sdo.SimulationTest](#) | [sim](#) | [who](#)

**Introduced in R2015b**

# find

**Class:** sdo.SimulationTest

**Package:** sdo

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');
```

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

## See Also

[sim](#) | [sdo.optimize](#) | [fastRestart](#) | [who](#)

# sim

**Class:** sdo.SimulationTest

**Package:** sdo

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

`fastRestart` | `find` | `sdo.optimize` | `who`

# who

**Class:** sdo.SimulationTest

**Package:** sdo

List logged data names

## Syntax

```
names = who(sim_obj)
```

## Description

`names = who(sim_obj)` returns a list of logged data names.

## Input Arguments

**sim\_obj**

sdo.SimulationTest object

## Output Arguments

**names**

Cell array of logged data set names.

## Examples

List logged data set names.

Log model signals.

```
Pressures = Simulink.SimulationData.SignalLoggingInfo;
```

```
Pressures.BlockPath      = 'sdoHydraulicCylinder/Cylinder Assembly';  
Pressures.OutputPortIndex = 1;
```

Store logged signal data.

```
simulator = sdo.SimulationTest('sdoHydraulicCylinder');  
simulator.LoggingInfo.Signals = Pressures;  
simulator = sim(simulator);
```

Find logged data sets.

```
names = who(simulator);
```

## See Also

[find](#) | [fastRestart](#) | [sdo.optimize](#) | [sim](#)

# sdotool

Open Response Optimization tool

## Syntax

```
sdotool(modelname)  
sdotool(sdosession)
```

## Description

`sdotool(modelname)` opens the Response Optimization tool and creates a new session. The model must be open or on the MATLAB path.

`sdotool(sdosession)` opens a previously saved Response Optimization tool session.

## Input Arguments

### **modelname**

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

### **sdosession**

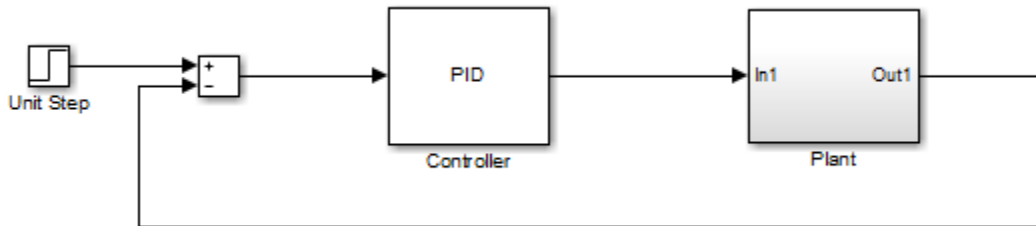
Response Optimization tool session variable, saved in a MAT-file, model or MATLAB workspace.

## Examples

### **Create a New Response Optimization Tool Session**

```
sdotool('sldo_model1');
```





### Open Response Optimization Tool Using a Saved Session

```
load sdoAircraft_sdoession;
sdotool(SDOSessionData);
```

SDOSessionData is the Response Optimization tool session variable saved in the sdoAircraft\_sdoession.mat file.

- “Design Optimization to Meet a Custom Objective (GUI)”

## More About

### Tips

- sdotool also updates **Signal Constraint** blocks in the model to the equivalent blocks from the **Signal Constraints** block library.

# sdouupdate

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 Response 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 Response Optimization tool session.

## Input Arguments

### **modelname**

Simulink model name, 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**

Response Optimization tool session name.

### **See Also**

Response Optimization tool

# Sensitivity Analysis Tool

Explore design space and determine most influential model parameters

## Description

The Sensitivity Analysis tool lets you explore the design space and determine the most influential Simulink model parameters using design of experiments, Monte Carlo simulations, and correlation analysis.

Using this tool, you can:

- Select and sample parameters using design of experiments.
- Specify design requirements.
- Perform Monte Carlo simulations to evaluate the design requirement at selected parameter values.
- Analyze and visualize model sensitivity to parameters.

You can accelerate evaluation of design requirements using parallel computing and Simulink fast restart.

## Open the Sensitivity Analysis Tool App

- Simulink model editor: Select **Analysis > Sensitivity Analysis**.
- MATLAB command prompt: Enter `ssatool('modelName')`.

## Examples

- “What is Sensitivity Analysis?”
- “Identify Key Parameters for Estimation (GUI)”
- “Design Exploration using Parameter Sampling (GUI)”
- “Generate Parameter Samples for Sensitivity Analysis”
- “Evaluate Design Requirements”
- “Analyze Relation Between Parameters and Design Requirements”

- “Interact with Plots in the Sensitivity Analysis Tool”

## **Programmatic Use**

`ssatool('modelName')` opens the Sensitivity Analysis tool and creates a new session. The model must be open or on the MATLAB path.

`ssatool(ssasession)` opens a previously saved Sensitivity Analysis tool session.

## **See Also**

`sdo.evaluate`

**Introduced in R2016a**

## spetool

Create Estimation Task in Parameter Estimation Tool

## Syntax

```
spetool('modelName')
```

## Description

`spetool('modelName')` opens the Simulink model with the name `modelName` and creates an estimation task in the Parameter Estimation tool.

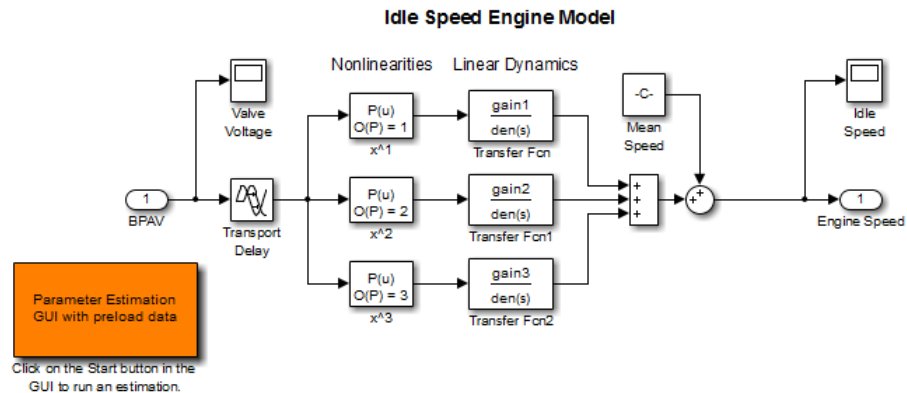
## Examples

Create an estimation task by typing the following command at the MATLAB prompt:

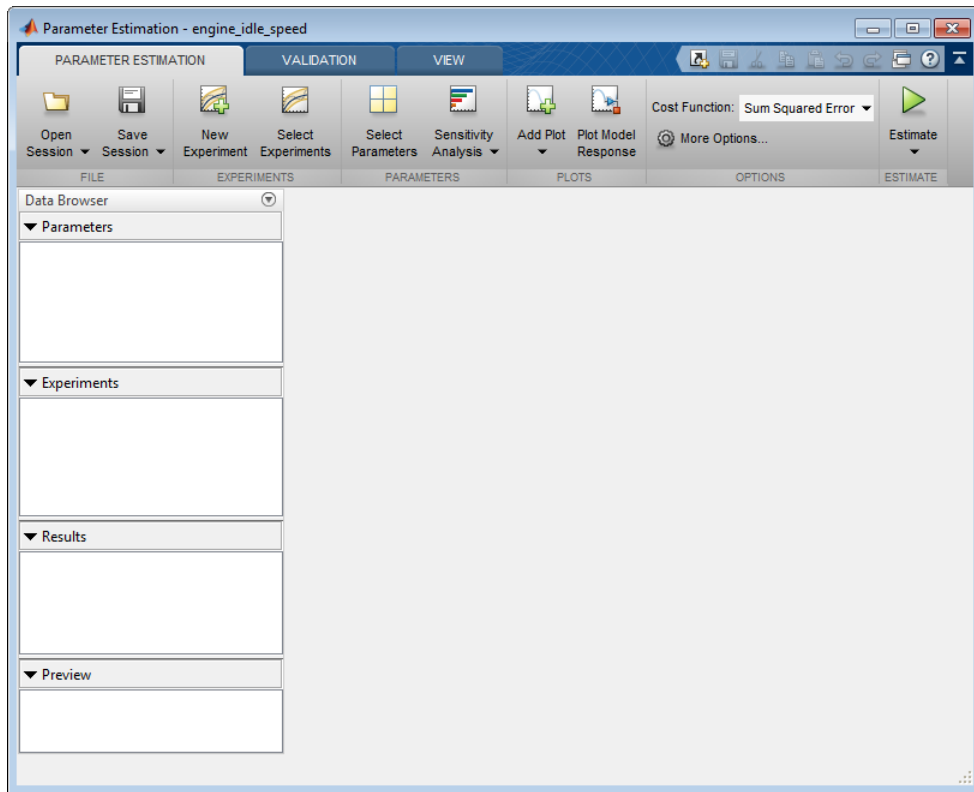
```
spetool('engine_idle_speed')
```

This command opens the following:

- Simulink model



- Parameter Estimation tool containing a session with an estimation experiment



## More About

- “Import Data for Parameter Estimation”

