

Simulink® Design Optimization™

User's Guide



MATLAB® & SIMULINK®

R2017b



How to Contact MathWorks



Latest news: www.mathworks.com
Sales and services: www.mathworks.com/sales_and_services
User community: www.mathworks.com/matlabcentral
Technical support: www.mathworks.com/support/contact_us



Phone: 508-647-7000



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

Simulink® Design Optimization™ User's Guide

© COPYRIGHT 1993–2017 by The MathWorks, Inc.

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

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

Trademarks

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

Patents

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

Revision History

March 2009	Online only	New for Version 1 (Release 2009a)
September 2009	Online only	Revised for Version 1.1 (Release 2009b)
March 2010	Online only	Revised for Version 1.1.1 (Release 2010a)
September 2010	Online only	Revised for Version 1.2 (Release 2010b)
April 2011	Online only	Revised for Version 1.2.1 (Release 2011a)
September 2011	Online only	Revised for Version 2.0 (Release 2011b)
March 2012	Online only	Revised for Version 2.1 (Release 2012a)
September 2012	Online only	Revised for Version 2.2 (Release 2012b)
March 2013	Online only	Revised for Version 2.3 (Release 2013a)
September 2013	Online only	Revised for Version 2.4 (Release 2013b)
March 2014	Online only	Revised for Version 2.5 (Release 2014a)
October 2014	Online only	Revised for Version 2.6 (Release 2014b)
March 2015	Online only	Revised for Version 2.7 (Release 2015a)
September 2015	Online only	Revised for Version 2.8 (Release 2015b)
March 2016	Online only	Revised for Version 3.0 (Release 2016a)
September 2016	Online only	Revised for Version 3.1 (Release 2016b)
March 2017	Online only	Revised for Version 3.2 (Release 2017a)
September 2017	Online only	Revised for Version 3.3 (Release 2017b)

	Data Analysis and Processing	
1		
	Model Requirements for Importing Data	1-2
	Select Input Signals	1-3
	Select Output Signals	1-4
	Import Data for Parameter Estimation	1-6
	Create Experiment	1-6
	Time-Domain Data	1-8
	Time-Series Data	1-11
	Complex Data	1-12
	Plot and Analyze Data	1-13
	Why Plot the Data Before Parameter Estimation	1-13
	Plot Data	1-13
	Preprocess Data	1-16
	Ways to Preprocess Data	1-16
	Remove Offset	1-17
	Scale Data	1-18
	Extract Data	1-18
	Filter Data	1-19
	Resample Data	1-20
	Replace Data	1-21
	Parameter Estimation	
2		
	What Is an Experiment?	2-3

Specify Estimation Data	2-4
Create Experiment	2-4
Edit Experiment Data	2-6
Specify Parameters for Estimation	2-9
Choosing Which Parameters to Estimate First	2-9
Add Model Parameters as Variables for Estimation	2-9
Specify Parameters for Estimation	2-12
Specify Known Initial States	2-19
When to Specify Initial States Versus Estimate Initial States	2-19
Specify Model Initial States	2-19
Specify Estimation Options	2-23
Estimate Parameters and States	2-26
Validate Estimation Results	2-34
Configure and Perform Validation	2-34
Compare Measured and Simulated Responses	2-37
Speed Up Parameter Estimation Using Parallel Computing	2-39
When to Use Parallel Computing for Parameter Estimation	2-39
How Parallel Computing Speeds Up Estimation	2-39
Use Parallel Computing for Parameter Estimation	2-43
Configure Your System for Parallel Computing	2-43
Model Dependencies	2-43
Estimate Parameters Using Parallel Computing in the Parameter Estimation Tool	2-44
Estimate Parameters Using Parallel Computing (Code)	2-47
Troubleshooting	2-48
Estimating Initial Conditions for Blocks with External Initial Conditions	2-50
Save and Load Estimation Sessions	2-51
Structure of an Estimation Session	2-51
Save Parameter Estimation Tool Sessions	2-51
Load Parameter Estimation Tool Sessions	2-52

Load Legacy Projects	2-52
How the Software Formulates Parameter Estimation as an Optimization Problem	2-53
Overview of Parameter Estimation as an Optimization Problem	2-53
Cost Function	2-53
Bounds and Constraints	2-55
Optimization Methods and Problem Formulations	2-56
Write a Cost Function	2-62
Anatomy of a Cost Function	2-62
Specify Inputs of the Cost Function	2-63
Compute Requirements	2-64
Specify Outputs of the Cost Function	2-67
Convenience Objects as Additional Inputs	2-69
Gradient Computations	2-74
Estimate Model Parameter Values (Code)	2-76
Estimate Model Parameters and Initial States (Code)	2-88
Estimate Model Parameters Using Multiple Experiments (Code)	2-100
Estimate Model Parameters Per Experiment (Code)	2-112
Estimate Model Parameters with Parameter Constraints (Code)	2-126
Estimate Model Parameter Values (GUI)	2-137
Estimate Model Parameters Per Experiment (GUI)	2-150
Estimate Model Parameters and Initial States (GUI)	2-166
Generate MATLAB Code for Parameter Estimation Problems (GUI)	2-177
Improving Optimization Performance Using Fast Restart (GUI)	2-182

Improving Optimization Performance Using Fast Restart (Code)	2-190
---	--------------

Response Optimization

3

How the Optimization Algorithm Formulates Minimization	
Problems	3-3
Feasibility Problem and Constraint Formulation	3-3
Tracking Problem	3-5
Gradient Descent Method Problem Formulations	3-6
Simplex Search Method Problem Formulations	3-8
Pattern Search Method Problem Formulations	3-9
Gradient Computations	3-10
Specify Signals to Log	3-12
Specify Custom Requirements in the Tool	3-13
Move Constraints	3-16
Move Constraints Graphically	3-16
Position Constraints Exactly	3-17
Specify Time-Domain Design Requirements in the Tool ...	3-19
Specify Piecewise-Linear Lower and Upper Bounds	3-19
Specify Signal Property Requirements	3-21
Specify Step Response Characteristics	3-23
Track Reference Signals	3-26
Impose Elliptic Bound on Phase Plane Trajectory of Two Signals	3-28
Specify Custom Requirements	3-31
Edit Design Requirements	3-34
Edit Design Requirements	3-37
Edit Design Requirement Dialog Box Parameters	3-37
Specify Variable Requirements in the Tool	3-39
Impose Monotonic Constraint Requirement on Variable ...	3-39
Impose Upper Bound on Gradient Magnitude of Variable ...	3-42
Specify Linear or Quadratic Function Matching Constraint .	3-45

Specify Requirement on a Vector Property	3-50
Impose Relational Constraint Between Two Variables	3-53
Specify Frequency-Domain Design Requirements in the	
Tool	3-56
Specify Lower Bounds on Gain and Phase Margin	3-56
Specify Piecewise-Linear Lower and Upper Bounds on	
Frequency Response	3-58
Specify Bound on Closed-Loop Peak Gain	3-60
Specify Lower Bound on Damping Ratio	3-62
Specify Upper and Lower Bounds on Natural Frequency	3-64
Specify Upper Bound on Approximate Settling Time	3-66
Specify Piecewise-Linear Upper and Lower Bounds on Singular	
Values	3-68
Specify Step Response Characteristics	3-70
Specify Custom Requirements	3-72
Specify Design Variables	3-76
Add Model Parameters as Variables for Optimization	3-76
Specify Design Variables	3-79
Update Model with Design Variables Set	3-82
Specify Optimization Options	3-84
Create Linearization I/O Sets	3-86
Interact with Plots	3-89
Response Plots	3-89
Spider Plots	3-90
Iteration Plots	3-92
Compare Requirements and Design Variables Using Spider	
Plot	3-94
Save Design Variable Values for Specific Iteration	3-97
Design Optimization to Meet Time- and Frequency-Domain	
Requirements (GUI)	3-99
Design Optimization to Meet a Custom Objective (GUI)	3-115
Design Optimization to Meet a Custom Objective (Code)	3-133

Design Optimization to Meet Custom Signal Requirements (GUI)	3-142
Design Optimization to Meet Frequency-Domain Requirements (GUI)	3-147
Specify Custom Signal Objective with Uncertain Variable (GUI)	3-164
Design Optimization with Uncertain Variables (Code) ...	3-175
Generate MATLAB Code for Design Optimization Problems (GUI)	3-185
Skip Model Simulation Based on Parameter Constraint Violation (GUI)	3-190
Optimizing Parameters for Robustness	3-201
What Is Robustness?	3-201
Sampling Methods for Uncertain Parameters	3-202
Optimize Parameters for Robustness (GUI)	3-204
Use Accelerator Mode During Simulations	3-212
About Accelerating Optimization	3-212
Limitations	3-212
Setting Accelerator Mode	3-212
Speed Up Response Optimization Using Parallel Computing	3-214
When to Use Parallel Computing for Response Optimization	3-214
How Parallel Computing Speeds Up Optimization	3-214
Use Parallel Computing for Response Optimization	3-218
Configure Your System for Parallel Computing	3-218
Model Dependencies	3-218
Optimize Design Using Parallel Computing (GUI)	3-219
Optimize Design Using Parallel Computing (Code)	3-223
Troubleshooting	3-224
Use Fast Restart Mode During Response Optimization ...	3-226
Response Optimization Tool Workflow for Fast Restart ...	3-226
Command-Line Workflow for Fast Restart	3-227

Troubleshooting	3-228
Optimization Does Not Make Progress	3-229
Should I worry about the scale of my responses and how constraints and design requirements are discretized? . . .	3-229
Why don't the responses and parameter values change at all?	3-229
Why does the optimization stall?	3-229
Optimization Convergence	3-231
What to do if the optimization does not get close to an acceptable solution?	3-231
Why does the optimization terminate before exceeding the maximum number of iterations, with a solution that does not satisfy all the constraints or design requirements?	3-232
What to do if the optimization takes a long time to converge even though it is close to a solution?	3-232
What to do if the response becomes unstable and does not recover?	3-233
Optimization Speed and Parallel Computing	3-234
How can I speed up the optimization?	3-234
Why are the optimization results with and without using parallel computing different?	3-235
Why do I not see the optimization speedup I expected using parallel computing?	3-235
Why does the optimization using parallel computing not make any progress?	3-235
Why does the optimization using parallel computing not stop when I click the Stop optimization button?	3-236
Undesirable Parameter Values	3-237
What to do if the optimization drives the tuned compensator elements and parameters to undesirable values?	3-237
What to do if the optimization violates bounds on parameter values?	3-237
Reverting to Initial Parameter Values	3-239
How do I quit an optimization and revert to my initial parameter values?	3-239
Save and Load Optimization Sessions	3-240
Structure of an Optimization Session	3-240

Save a Session	3-240
Load a Session	3-241
Optimizing Time-Domain Response of Simulink® Models Using Parallel Computing	3-242
Design Optimization to Meet Frequency-Domain Requirements (Code)	3-251

Sensitivity Analysis

4

What is Sensitivity Analysis?	4-2
Specify Parameters for Design Exploration	4-4
Add Model Parameters as Variables	4-4
Select Parameters for Design Exploration	4-7
Generate Parameter Samples for Sensitivity Analysis	4-10
Generate Random Parameter Values	4-10
Generate Gridded Parameter Values	4-20
Specify Time-Domain Requirements	4-28
Match Model Outputs to Measured Signals	4-28
Specify Piecewise-Linear Lower and Upper Bounds	4-33
Specify Signal Property Requirements	4-35
Specify Step Response Characteristics	4-38
Track Reference Signals	4-40
Impose Elliptic Bound on Phase Plane Trajectory of Two Signals	4-42
Specify Custom Requirements	4-46
Specify Parameters Requirements	4-50
Impose Monotonic Constraint Requirement on Variable	4-50
Impose Upper Bound on Gradient Magnitude of Variable	4-53
Specify Linear or Quadratic Function Matching Constraint	4-56
Specify Requirement on a Vector Property	4-60
Impose Relational Constraint Between Two Variables	4-63

Specify Frequency-Domain Requirements	4-66
Specify Lower Bounds on Gain and Phase Margin	4-66
Specify Piecewise-Linear Lower and Upper Bounds on Frequency Response	4-69
Specify Bound on Closed-Loop Peak Gain	4-71
Specify Lower Bound on Damping Ratio	4-72
Specify Upper and Lower Bounds on Natural Frequency . . .	4-73
Specify Upper Bound on Approximate Settling Time	4-74
Specify Piecewise-Linear Upper and Lower Bounds on Singular Values	4-75
Specify Step Response Characteristics	4-77
Specify Custom Requirements	4-79
 Evaluate Design Requirements	 4-83
 Analyze Relation Between Parameters and Design Requirements	 4-91
Visual Analysis	4-91
Statistical Analysis	4-93
 Use Sensitivity Analysis to Configure Estimation and Optimization	 4-99
Export Sensitivity Analysis Results	4-103
 Interact with Plots in the Sensitivity Analysis Tool	 4-106
Parameter Set Plots	4-106
Requirement Plots	4-114
Evaluated Result Scatter Plots	4-116
Evaluated Result Contour Plots	4-123
Statistical Analysis Tornado Plots	4-125
 Validate Sensitivity Analysis	 4-129
Inspect the Generated Parameter Set	4-129
Check Evaluation Results	4-132
Perform Sensitivity Analysis with Different Parameter Set	4-133
 Store Intermediate Data in the Tool	 4-135
 Use Parallel Computing for Sensitivity Analysis	 4-137
Configure Your System for Parallel Computing	4-137
Model Dependencies	4-137
Perform Sensitivity Analysis Using Parallel Computing (GUI)	4-138

Perform Sensitivity Analysis Using Parallel Computing (Code)	4-142
Troubleshooting	4-143
Use Fast Restart Mode During Sensitivity Analysis	4-144
Sensitivity Analysis Tool Workflow for Fast Restart	4-144
Command-Line Workflow for Fast Restart	4-145
Troubleshooting	4-146
Design Exploration Using Parameter Sampling (GUI)	4-147
Identify Key Parameters for Estimation (GUI)	4-168
Explore Design Reliability Using Parameter Sampling (GUI)	4-186
Design Exploration Using Parameter Sampling (Code)	4-200
Identify Key Parameters for Estimation (Code)	4-217

Optimization-Based Control Design

5

Overview of Optimization-Based Compensator Design	5-2
Time-Domain Design Requirements in Simulink	5-4
Specify Piecewise-Linear Lower and Upper Bounds	5-4
Specify Step Response Characteristics	5-5
Track Reference Signals	5-8
Specify Custom Requirements	5-10
Edit Design Requirements	5-12
Frequency-Domain Design Requirements in Simulink	5-15
Specify Lower Bounds on Gain and Phase Margin	5-15
Specify Piecewise-Linear Lower and Upper Bounds on Frequency Response	5-17
Specify Bound on Closed-Loop Peak Gain	5-19
Specify Lower Bound on Damping Ratio	5-21
Specify Upper and Lower Bounds on Natural Frequency	5-23
Specify Upper Bound on Approximate Settling Time	5-25

Specify Piecewise-Linear Upper and Lower Bounds on Singular Values	5-27
Specify Step Response Characteristics	5-29
Specify Custom Requirements	5-31
Time- and Frequency-Domain Requirements in Control System Designer App	5-35
Root Locus Diagrams	5-35
Open-Loop and Prefilter Bode Diagrams	5-37
Open-Loop Nichols Plots	5-37
Step/Impulse Response Plots	5-38
Time-Domain Simulations in Control System Designer App	5-40
Design Optimization-Based Controllers for LTI Systems ...	5-41
Optimize LTI System to Meet Frequency-Domain Requirements	5-42
Design Requirements	5-42
Create an LTI Plant Model	5-43
Open the Control System Designer App	5-44
Open Optimization Based Tuning Method	5-47
Select Tunable Compensator Elements	5-50
Add Design Requirements	5-51
Optimize the System Response	5-60
Create and Display the Closed-Loop System	5-63
Design Linear Controllers for Simulink Models	5-65

Lookup Tables

6

What are Adaptive Lookup Tables?	6-2
Lookup Tables	6-2
Adaptive Lookup Tables	6-2
How to Estimate Lookup Table Values	6-5

Estimate Constrained Values of a Lookup Table	6-6
Objectives	6-6
About the Data	6-6
Lookup Table Output	6-6
Estimate the Monotonically Increasing Table Values Using Default Settings	6-8
Validate the Estimation Results	6-16
Estimate Lookup Table Values from Data	6-21
Objectives	6-21
About the Data	6-21
Open a Parameter Estimation Session	6-21
Estimate the Table Values Using Default Settings	6-23
Validate the Estimation Results	6-30
Building Models Using Adaptive Lookup Table Blocks	6-35
Selecting an Adaptation Method	6-39
Sample Mean	6-39
Sample Mean with Forgetting	6-39
Model Engine Using n-D Adaptive Lookup Table	6-41
Objectives	6-41
About the Data	6-41
Building a Model Using Adaptive Lookup Table Blocks	6-42
Adapting the Lookup Table Values Using Time-Varying I/O Data	6-51
Using Adaptive Lookup Tables in Real-Time Environment	6-55
Design Optimization Using Lookup Table Requirements for Gain Scheduling (Code)	6-56
Design Optimization Using Lookup Table Requirements for Gain Scheduling (GUI)	6-73

Data Analysis and Processing

- “Model Requirements for Importing Data” on page 1-2
- “Import Data for Parameter Estimation” on page 1-6
- “Plot and Analyze Data” on page 1-13
- “Preprocess Data” on page 1-16

Model Requirements for Importing Data


In this section...
“Select Input Signals” on page 1-3
“Select Output Signals” on page 1-4

Before you can analyze and preprocess the estimation data, you must assign the data to the model ports or signals. In order to assign the data, the Simulink model must contain one of the following elements:

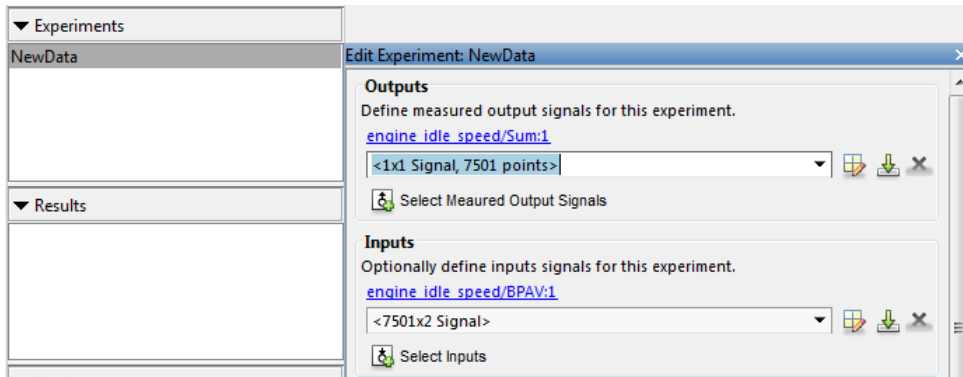
- Root-level model Inport block

Note You do not need an Inport block if your model already contains a fixed input block, such as a Step block.

- Root-level model Outport block
- Logged signal, which can be a root-level signal in the model or a signal in a model subsystem

To enable signal logging for a signal, in the Simulink Editor, select the signal, click the **Simulation Data Inspector** button arrow  and click **Log Selected Signals**. For more information, see “Export Signal Data Using Signal Logging” (Simulink).

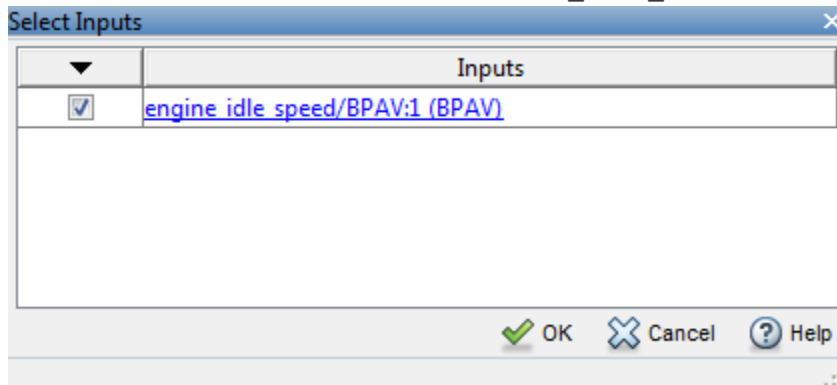
When you create an experiment, as described in “Create Experiment” on page 2-4, the top level input and output ports as well as logged signals are selected by default. You can add or remove the input and output signals using the experiment editor on page 2-6. In the experiment editor, the rows in the **Inputs** panel correspond to the model's root-level Inport blocks.



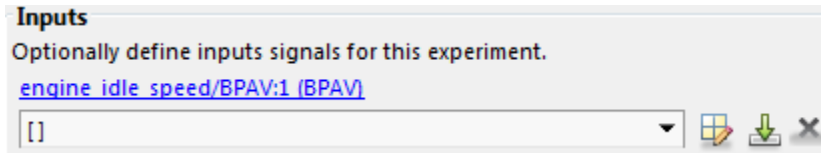
Similarly, the rows in the **Outputs** panel correspond to either the root-level Output blocks or logged signals in the model.

Select Input Signals

You can add the Inport block in the experiment editor by clicking the **Select Inputs** button in the **Inputs** panel to launch the **Select Inputs** dialog box. You can select the Inport block you want by selecting the check box corresponding to it and clicking **OK**. There is only one Inport block for the `engine_idle_speed` model.

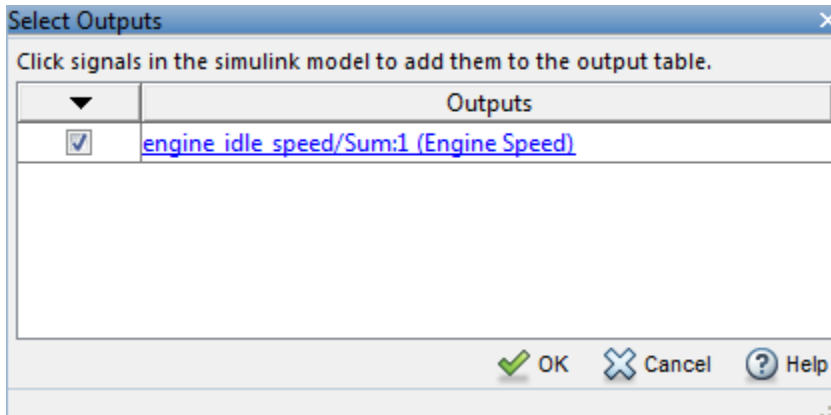


Using the dialog box, you can import the input data by typing, for example, `[time,iodata(:,1)]` in the **Inputs** panel. To learn more about importing data, see Import Data on page 1-6.

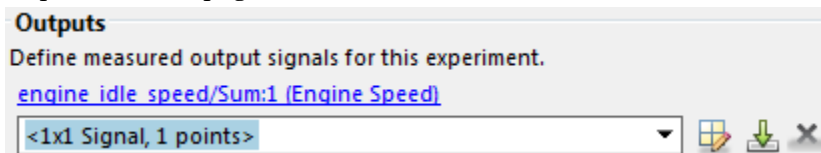


Select Output Signals

You can add the Output block in the experiment editor by clicking **Select Measured Output Signals** in the **Outputs** panel to launch the **Select Outputs** dialog box. You can select the Output block you want by clicking the check box corresponding to it, and clicking **OK**. There is only one Output block for the `engine_idle_speed` model.



Using the dialog box, you can import the output data by typing, for example, `[time,iodata(:,2)]` in the **Inputs** panel. To learn more about importing data, see **Import Data** on page 1-6.



See Also

Related Examples

- “Import Data for Parameter Estimation” on page 1-6

More About

- “What Is an Experiment?” on page 2-3

Import Data for Parameter Estimation

In this section...

“Create Experiment” on page 1-6

“Time-Domain Data” on page 1-8

“Time-Series Data” on page 1-11

“Complex Data” on page 1-12

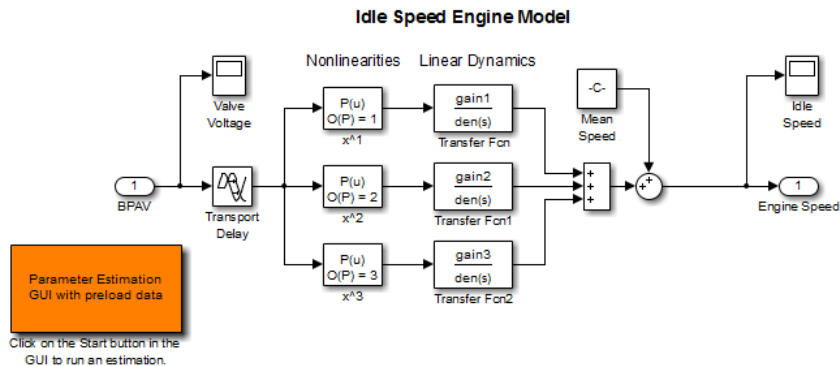
Create Experiment

Before you begin data import, create an experiment. Simulink Design Optimization software provides a tool for setting up the estimation session.

To create an estimation session:

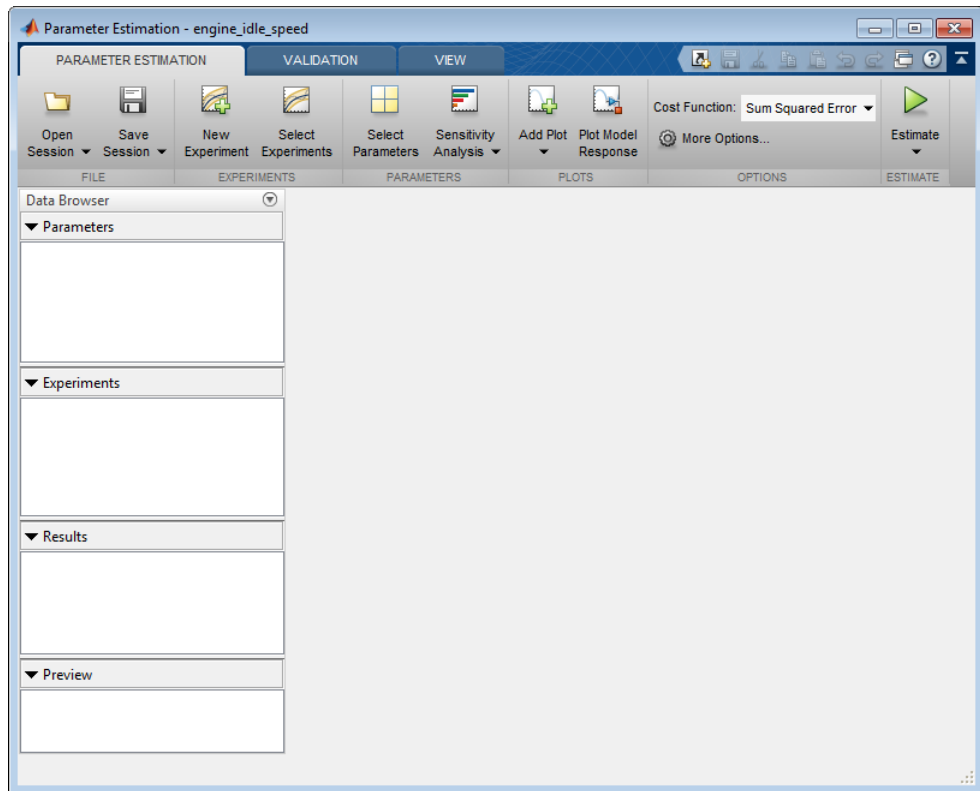
- 1 At the MATLAB® prompt, open the nonlinear idle speed model of an automotive engine by typing :

```
engine_idle_speed
```



The model contains the Inport block BPAV and Outport block Engine Speed for importing input and output data, respectively. To learn more, see “Model Requirements for Importing Data” on page 1-2.

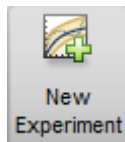
- 2 In the Simulink model window, open the Parameter Estimation tool by selecting **Analysis > Parameter Estimation**.



Parameter Estimation Tool

You can organize the estimation and validation tasks inside **Experiments** under **Data Browser** panel on the left. You can assign each experiment to an estimation task or validation task.

To create an experiment, click the **New Experiment** button.



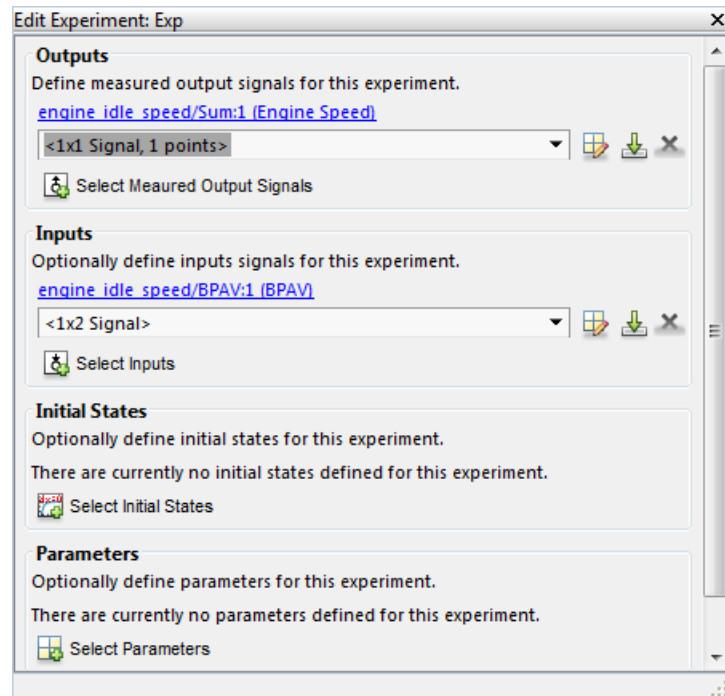
This creates an experiment called `Exp` under **Experiments**. To change the name of the experiment, right-click and select **Rename**. Call it `NewData`.

Note The Simulink model must remain open to perform parameter estimation tasks.

Time-Domain Data

Experiments are collections of signal data, specifically input and output signal data. After you create an experiment, as described in “Create Experiment” on page 1-6, you can import data into your experiment from various sources including MATLAB® variables, MAT-files, Excel® files, or comma-separated-value files.

To import data into your experiment right-click and select **Edit...** This will launch the experiment editor. In the experiment editor, you can define the signals contained in the experiment.

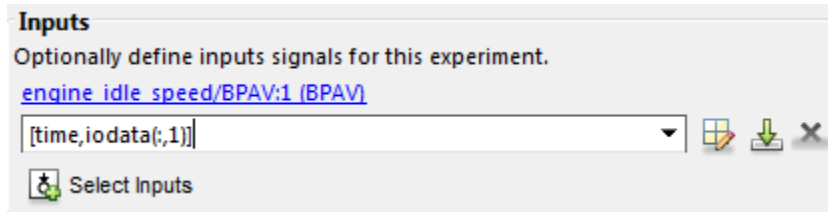


For example, the rows in the **Inputs** panel of the editor correspond to Inport block BPAV in the engine_idle_speed model.

The rows in the **Outputs** panel correspond to Outport block Engine Speed. You can import signal data from files or MATLAB workspace.

Note The Simulink model must contain an Inport or Outport block or logged signals to enable importing data. For more information, see “Model Requirements for Importing Data” on page 1-2. To select more output signals to specify data for, click **Select Measured Output Signals** in the **Outputs** panel.

The idle-speed model of an automotive engine contains the measured data stored in the `iodata` array in the workspace. The array contains two columns: the first for input data, and the second for output data. The time data is in the `time` array in the workspace. You can import the input data by typing `[time,iodata(:,1)]` in the **Inputs** panel.

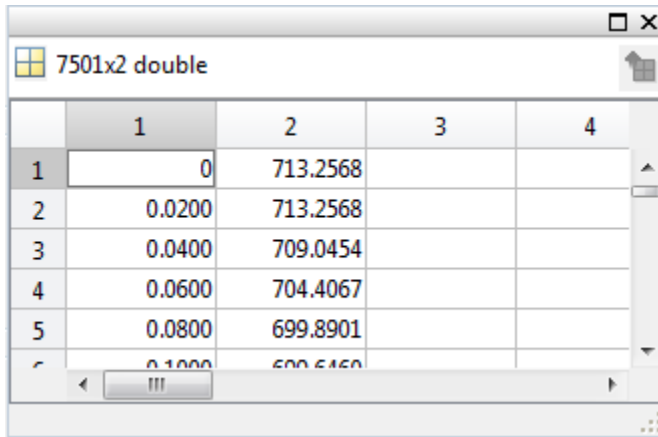


You can import the output data by typing `[time,iodata(:,2)]` in the **Outputs** panel.

You can view the data by clicking . The input data should look like this:

	1	2	3	4
1	0	0		
2	0.0200	0		
3	0.0400	0		
4	0.0600	0		
5	0.0800	0		
...	0.1000	0		

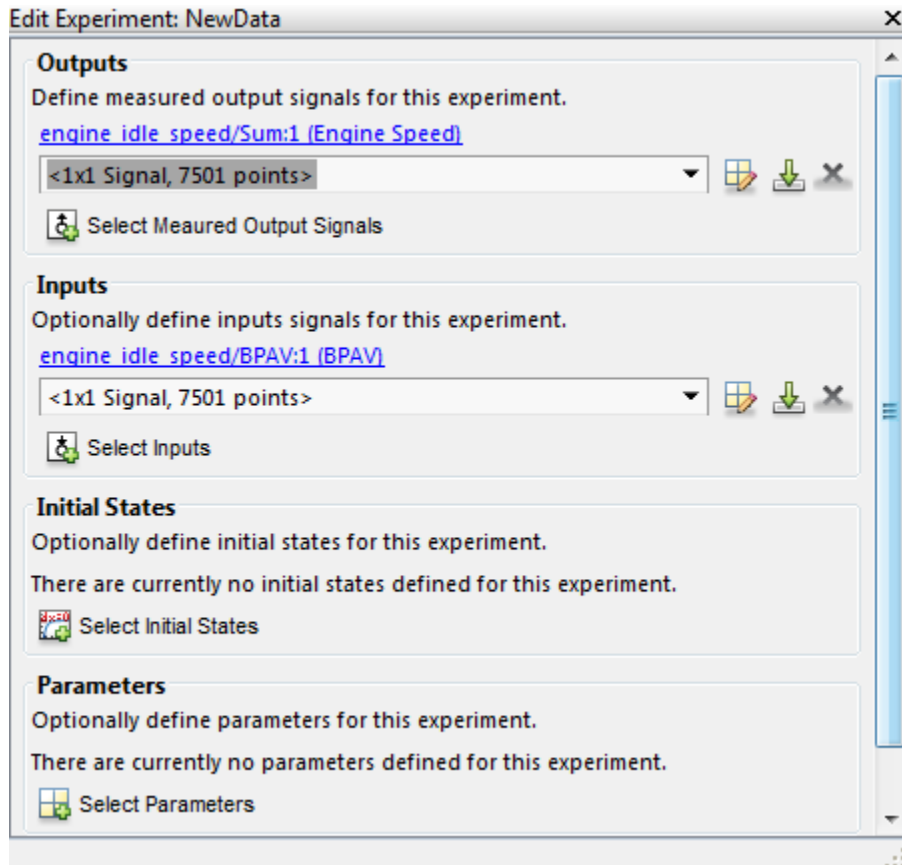
Output data should look like this:




The screenshot shows a window titled "7501x2 double" containing a table with 5 rows and 4 columns. The columns are labeled 1, 2, 3, and 4. The rows are labeled 1 through 5. The data values are as follows:

	1	2	3	4
1	0	713.2568		
2	0.0200	713.2568		
3	0.0400	709.0454		
4	0.0600	704.4067		
5	0.0800	699.8901		

After importing the data for `NewData` experiment, the experiment editor looks like this:



To import data from a file, click the  button.

To learn more about the **Edit Experiment**: dialog box, see “Edit Experiment Data” on page 2-6.

Time-Series Data

Time-series data is stored in time-series objects. For more information, see “Time Series Objects” (MATLAB).

When you import input data from a time-series object, t , for parameter estimation, you must specify the time vector and data as `[t.time, t.inputdata]` in the Inport signal

dialog box. Similarly, to import output data, you must specify the time vector and data as `[t.time, t.outputdata]` in the Output signal dialog box. For more information on how to import data into the experiment, see “Time-Domain Data” on page 1-8.

Complex Data

Complex-valued data is data whose value is a complex number. For example, a signal with the value $1+2j$ is complex. You can use complex data to estimate parameters of electrical systems, such as the magnitude and phase.

Note You must sample the real and imaginary parts of the data as a function of the same time vector.

To use complex data for parameter estimation:

- 1 Split the data into two data sets that contain the real and imaginary parts. To split the data, use the MATLAB functions `real`, and `imag`.
- 2 Create two signals, one for the real part and one for the imaginary part for the Import or Output block.
- 3 Select both signals in the experiment editor.
- 4 Import the data to the corresponding signal as described in “Time-Domain Data” on page 1-8.

See Also

Related Examples

- “Plot and Analyze Data” on page 1-13
- “Preprocess Data” on page 1-16

More About

- “Model Requirements for Importing Data” on page 1-2

Plot and Analyze Data

In this section...

“Why Plot the Data Before Parameter Estimation” on page 1-13

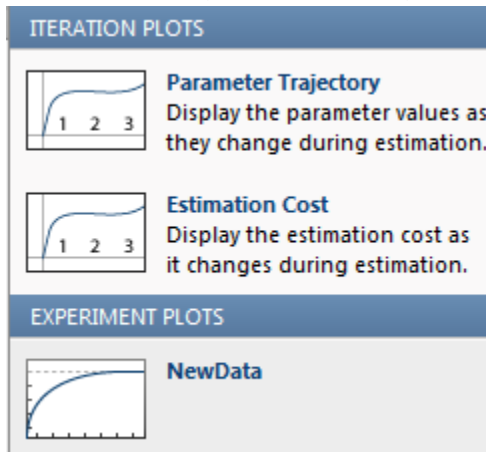
“Plot Data” on page 1-13

Why Plot the Data Before Parameter Estimation

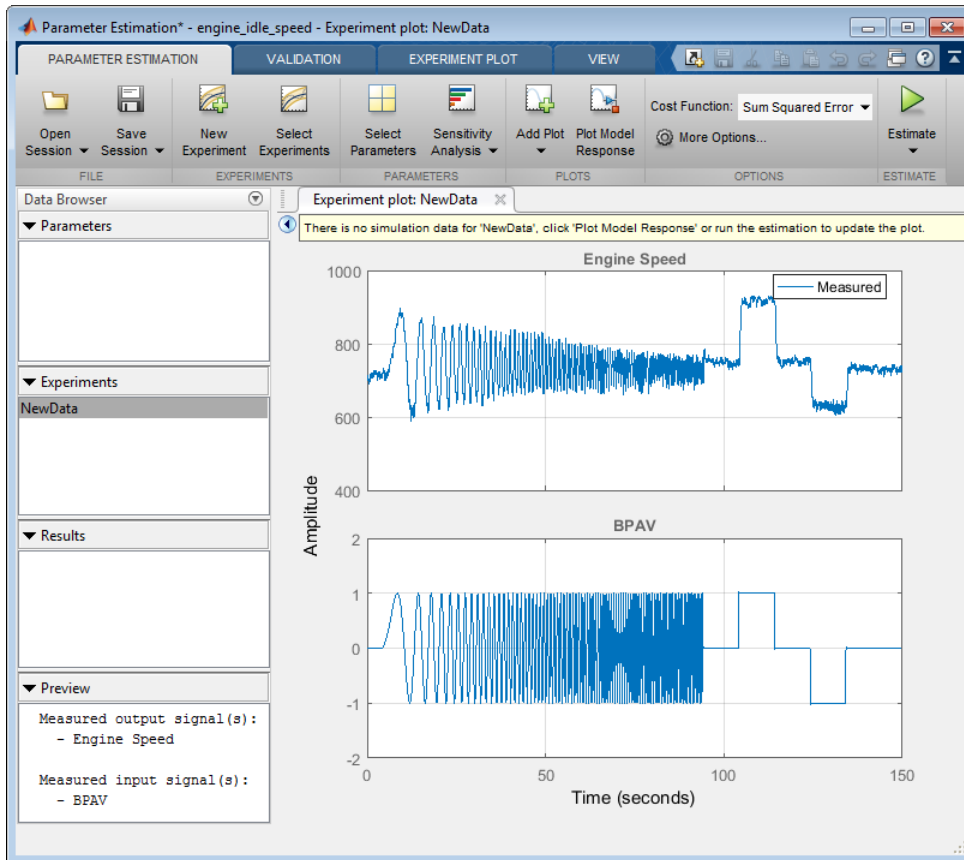
After you import the estimation data as described in “Import Data for Parameter Estimation” on page 1-6, you can remove outliers, smooth, detrend, or otherwise treat the data to prepare for analysis and estimation. To view and analyze data characteristics, plot the data on a time plot.

Plot Data

Use an experiment plot to visualize experiment data. First, create an experiment and import data as described in “Import Data for Parameter Estimation” on page 1-6. To create an experiment plot, in the Parameter Estimation tool, on the **Parameter Estimation** tab, click **Add Plot**, and select **NewData** under **Experiment Plots**.



This creates plots of the input signal for the Inport block BPAV and output signal for the Outport block Engine Speed for the engine_idle_speed model (see “Create Experiment” on page 1-6).



You can also plot the experiment data by right-clicking **NewData** and selecting **Plot measured experiment data** from the list.

Using the time plot, you can examine the data characteristics such as noise, outliers and portions of the data to use for estimating parameters. After you analyze the data, you can preprocess it as described in “Preprocess Data” on page 1-16.

See Also

Related Examples

- “Import Data for Parameter Estimation” on page 1-6
- “Preprocess Data” on page 1-16

More About

- “Model Requirements for Importing Data” on page 1-2

Preprocess Data

In this section...

“Ways to Preprocess Data” on page 1-16

“Remove Offset” on page 1-17

“Scale Data” on page 1-18

“Extract Data” on page 1-18

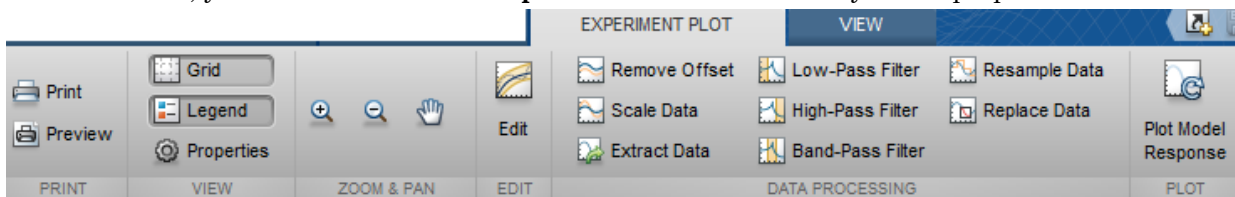
“Filter Data” on page 1-19

“Resample Data” on page 1-20

“Replace Data” on page 1-21

Ways to Preprocess Data

In the Parameter Estimation tool and Sensitivity Analysis tool, you can preprocess imported data before you use it for estimation or evaluation. After plotting the measured data, you have access to the **Experiment Plot** tab where you can preprocess the data.



For information about how to plot the imported data in the Parameter Estimation tool, see “Plot and Analyze Data” on page 1-13. For information about how to import and plot the data in the Sensitivity Analysis tool, see “Match Model Outputs to Measured Signals” on page 4-28.

You can perform the following preprocessing operations:

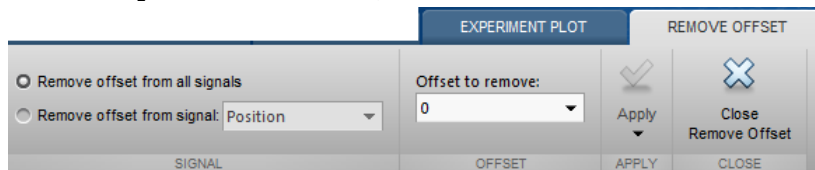
- “Remove Offset” on page 1-17 — Remove mean values, a constant value, or an initial value from the data.
- “Scale Data” on page 1-18 — Scale data by a constant value, signal maximum value, or signal initial value.
- “Extract Data” on page 1-18 — Select a subset of the data to use in the estimation or evaluation. You can graphically select the data to extract, or enter start and end times in the text boxes.

- “Filter Data” on page 1-19 — Process data using a low-pass, high-pass, or band-pass filter.
- “Resample Data” on page 1-20 — Resample data using zero-order hold or linear interpolation.
- “Replace Data” on page 1-21 — Replace data with a constant value, region initial value, region final value, or a line. You can use this functionality to replace outliers.

You can perform as many preprocessing operations on your data as are required for your application. For instance, you can both filter the data and remove an offset.

Remove Offset

On the **Experiment Plot** tab, click **Remove Offset**.





It is important for good estimation results to remove data offsets. In the **Remove Offset** tab, you can remove offset from all signals at once or select a particular signal using the **Remove offset from signal** drop down list. Specify the value to remove using the **Offset to remove** drop down list. The options are:

- A constant value. Enter the value in the box. (Default: 0)
- Mean of the data, to create zero-mean data.
- Signal initial value.

As you change the offset value, the modified data is shown in preview in the plot.

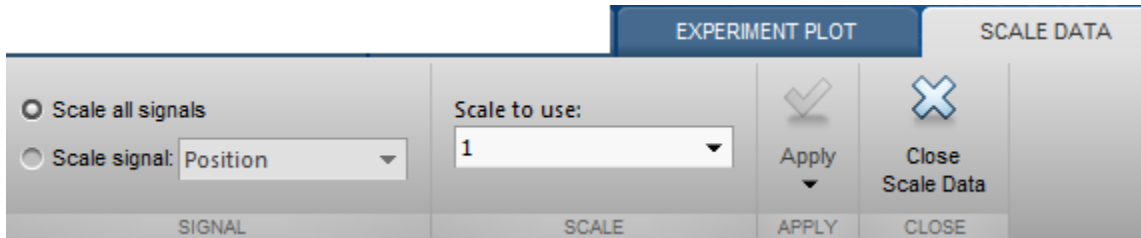
After making choices, update the existing data with the preprocessed data by clicking



Or, to save the modified data values in a new experiment (in the Parameter estimation tool) or requirement (in the Sensitivity Analysis tool), click  and select  **Save As: Create a new experiment from the modified data.**

Scale Data

On the **Experiment Plot** tab, click **Scale Data**.



In the **Scale Data** tab, you can choose to scale all signals or specify a signal to scale. Select the scaling value from the **Scale to use** drop-down list. The options are:

- A constant value. Enter the value in the box. (Default: 1)
- Signal maximum value.
- Signal initial value.

As you change the scaling, the modified data is shown in preview in the plot.

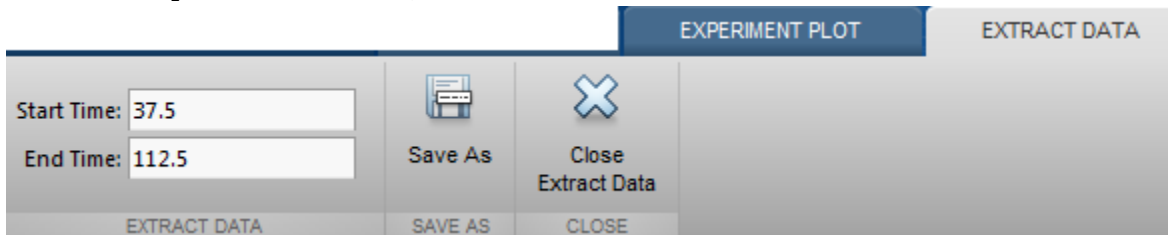
After making choices, update the existing data with the preprocessed data by clicking



Or, to save the modified data values in a new experiment (in the Parameter estimation tool) or requirement (in the Sensitivity Analysis tool), click **Apply** and select **Save As: Create a new experiment from the modified data**.

Extract Data

To extract a portion of your data to use in the estimation or evaluation process, on the **Experiment Plot** tab, click **Extract Data**.

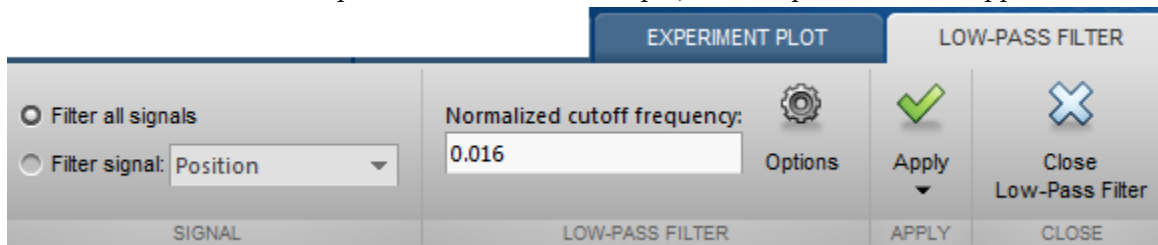


Select a subset of data to use in **Extract Data** tab. You can extract data graphically or by specifying start time and end time. To extract data graphically, click and drag the vertical bars to select a region of the data to use.

After you choose the data to extract, you can save it in a new experiment (in the Parameter estimation tool) or requirement (in the Sensitivity Analysis tool) by clicking **Save As**.

Filter Data

You can filter your data using a low-pass, high-pass, or band-pass filter. A low-pass filter blocks high frequency signals, a high-pass filter blocks low frequency signals, and a band-pass filter combines the properties of both low- and high-pass filters. On the **Experiment Plot** tab click one of the **Low-Pass Filter**, **High-Pass Filter**, or **Band-Pass Filter** to open a new tab. For example, the low-pass filter tab appears as shown:





On the **Low-Pass Filter**, **High-Pass Filter**, or **Band-Pass Filter** tab, you can choose to filter all signals or specify a particular signal. For the low-pass and high-pass filtering, you can specify the normalized cutoff frequency of the signal. Where, a normalized frequency of 1 corresponds to half the sampling rate. For the band-pass filter, you can specify the normalized start and end frequencies. Specify the frequencies by either entering the value in the associated field on the tab. Alternatively, you can specify filter frequencies graphically, by dragging the vertical bars in the frequency-domain plot of your data.

Click **Options** to specify the filter order, and select zero-phase shift filter.

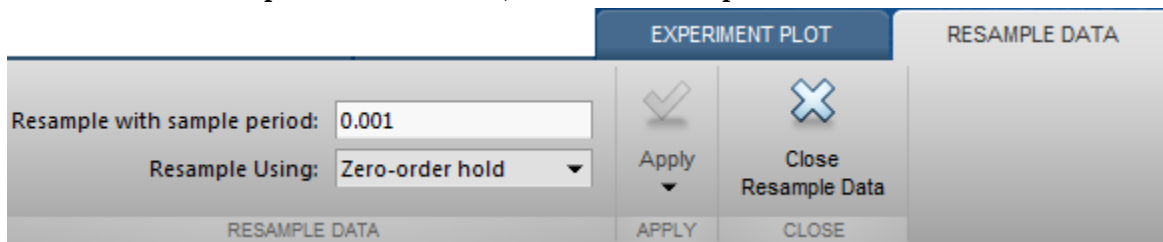
After making choices, update the existing data with the preprocessed data by clicking



Or, to save the modified data values in a new experiment (in the Parameter estimation tool) or requirement (in the Sensitivity Analysis tool), click  and select  **Save As: Create a new experiment from the modified data.**

Resample Data

On the **Experiment Plot** tab, click the **Resample Data** button.



In the **Resample Data** tab, specify the sampling period using the **Resample with sample period:** field. You can resample your data using one of the following interpolation methods:



- **Zero-order hold** — Fill the missing data sample with the data value immediately preceding it.
- **Linear interpolation** — Fill the missing data using a line that connects the two data points.

By default, the resampling method is set to **zero-order hold**. You can select the **linear interpolation** method from the **Resample Using** drop-down list.

The modified data is shown in preview in the plot.

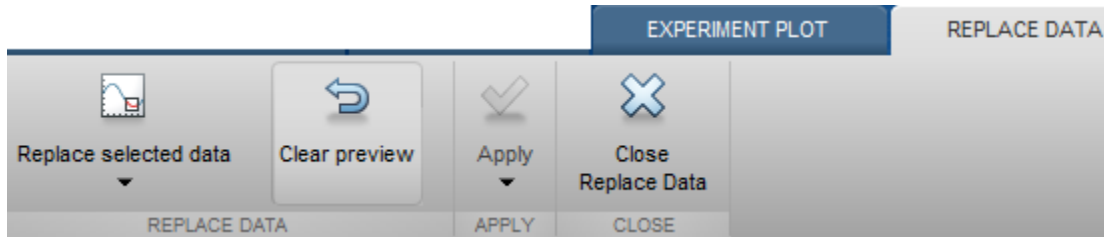
After making choices, update the existing data with the preprocessed data by clicking



Or, to save the modified data values in a new experiment (in the Parameter estimation tool) or requirement (in the Sensitivity Analysis tool), click  and select  **Save As: Create a new experiment from the modified data.**

Replace Data

On the **Experiment Plot** tab click the **Replace Data** button.





In the **Replace Data** tab, select data to replace by dragging across a region in the plot. Once you select data, choose how to replace it using the **Replace selected data** drop-down list. You can replace the data you select with one of these options:

- A constant value
- Region initial value
- Region final value
- A line

The replaced preview data changes color and the replacement data appears on the plot. At any time before updating, click **Clear preview** to clear the data you replaced and start over.

After making choices, update the existing data with the preprocessed data by clicking



Or, to save the modified data values in a new experiment (in the Parameter estimation tool) or requirement (in the Sensitivity Analysis tool), click  and select  **Save As: Create a new experiment from the modified data.**

Replace Data can be useful, for example, to replace outliers. Outliers can be defined as data values that deviate from the mean by more than three standard deviations. When estimating parameters from data containing outliers, the results may not be accurate. Hence, you might choose to replace the outliers in the data before you estimate the parameters.

See Also

More About

- “Import Data for Parameter Estimation” on page 1-6
- “Match Model Outputs to Measured Signals” on page 4-28

Parameter Estimation

- “What Is an Experiment?” on page 2-3
- “Specify Estimation Data” on page 2-4
- “Specify Parameters for Estimation” on page 2-9
- “Specify Known Initial States” on page 2-19
- “Specify Estimation Options” on page 2-23
- “Estimate Parameters and States” on page 2-26
- “Validate Estimation Results” on page 2-34
- “Speed Up Parameter Estimation Using Parallel Computing” on page 2-39
- “Use Parallel Computing for Parameter Estimation” on page 2-43
- “Estimating Initial Conditions for Blocks with External Initial Conditions” on page 2-50
- “Save and Load Estimation Sessions” on page 2-51
- “How the Software Formulates Parameter Estimation as an Optimization Problem” on page 2-53
- “Write a Cost Function” on page 2-62
- “Gradient Computations” on page 2-74
- “Estimate Model Parameter Values (Code)” on page 2-76
- “Estimate Model Parameters and Initial States (Code)” on page 2-88
- “Estimate Model Parameters Using Multiple Experiments (Code)” on page 2-100
- “Estimate Model Parameters Per Experiment (Code)” on page 2-112
- “Estimate Model Parameters with Parameter Constraints (Code)” on page 2-126
- “Estimate Model Parameter Values (GUI)” on page 2-137
- “Estimate Model Parameters Per Experiment (GUI)” on page 2-150
- “Estimate Model Parameters and Initial States (GUI)” on page 2-166
- “Generate MATLAB Code for Parameter Estimation Problems (GUI)” on page 2-177
- “Improving Optimization Performance Using Fast Restart (GUI)” on page 2-182

- “Improving Optimization Performance Using Fast Restart (Code)” on page 2-190

What Is an Experiment?

To estimate unknown parameter values of a Simulink model, first create an experiment. An experiment specifies measured input and output data. During estimation, the experiment input data is used to simulate the model and the model output is compared with the measured experiment output data. For more information about creating experiments and importing data, see “Specify Estimation Data” on page 2-4.

In an experiment, you can specify initial-state values. 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 experiment. For more information, see “Specify Known Initial States” on page 2-19.

To estimate a model parameter on a per-experiment basis, specify the model parameter for each experiment. You can specify the initial values and limits for the parameter value for any of the experiments. Alternatively, you can specify a parameter value as a known quantity, not to be estimated. For more information, see “Specify Parameters for Estimation” on page 2-9. You can choose to update experiments with estimated model initial states and parameter values, or save the results in a new experiment. For more information, see “Specify Estimation Options” on page 2-23.

To use experiments for validating the estimated parameter values, see “Validate Estimation Results” on page 2-34.

See Also

Related Examples

- “Specify Estimation Data” on page 2-4
- “Specify Parameters for Estimation” on page 2-9
- “Specify Known Initial States” on page 2-19
- “Estimate Parameters and States” on page 2-26

More About

- “Edit Experiment Data” on page 2-6

Specify Estimation Data

This topic shows how to specify estimation data for parameter estimation.

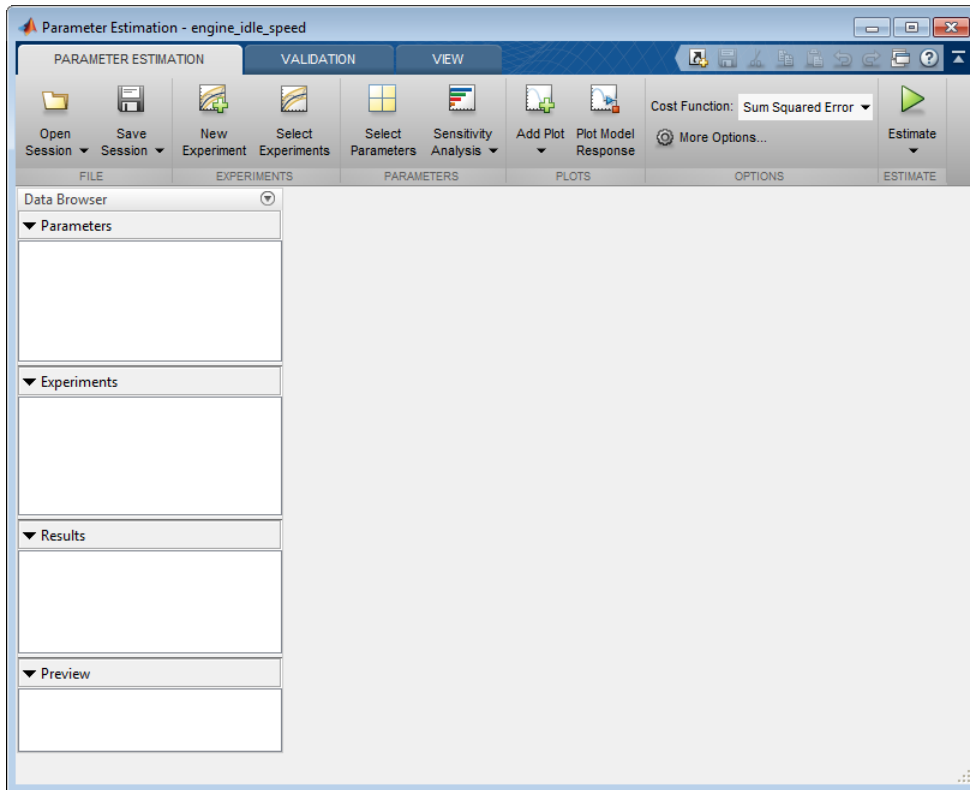
In this section...
“Create Experiment” on page 2-4
“Edit Experiment Data” on page 2-6

Create Experiment

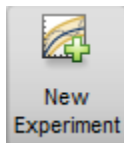
Before you specify estimation data, create an experiment. At the MATLAB prompt, open the nonlinear idle speed model of an automotive engine by typing

```
engine_idle_speed
```

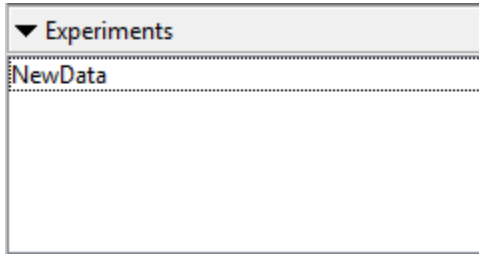
In the Simulink model window, open the Parameter Estimation tool by selecting **Analysis > Parameter Estimation**.



In the Parameter Estimation tool, on the **Parameter Estimation** tab, click **New Experiment**.

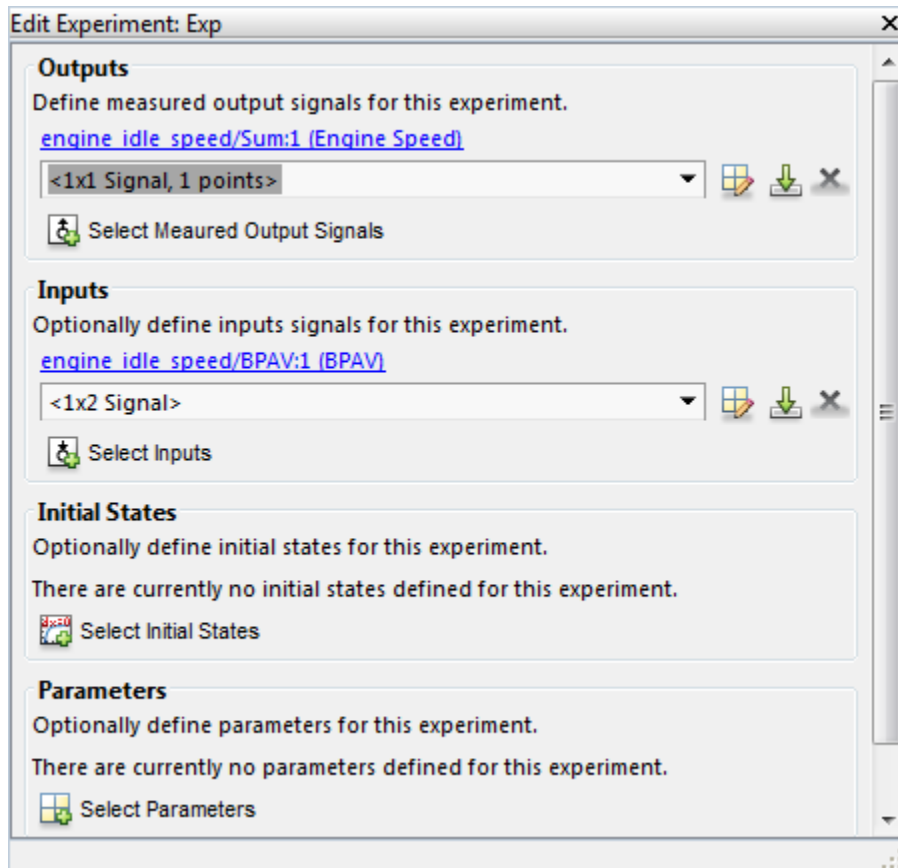


This action creates an experiment called `Exp` in the **Experiments** list in the **Data Browser** panel and opens the experiment editor. To change the name of the experiment, right-click `Exp` and select **Rename**. If you rename it `NewData`, the **Experiments** list now looks like this:



Edit Experiment Data

After creating an experiment, launch the experiment editor by right-clicking on the experiment name and selecting **Edit...** from the list. The experiment editor resembles the following figure.

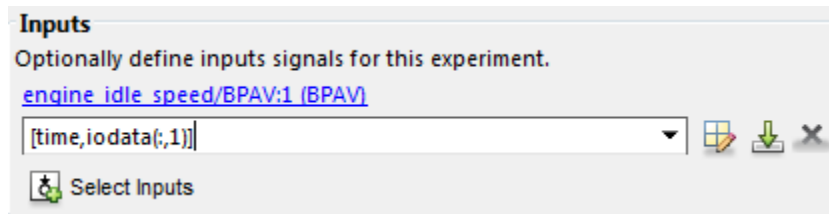


The experiment editor has four panels. You select output signals on page 1-4 and import output data on page 1-6 in the **Outputs** panel. You select input signals on page 1-3 and import input data on page 1-6 in the **Inputs** panel. You can specify model initial states on page 2-19 in the **Initial States** panel. You can specify parameters to estimate on page 2-15 in the **Parameters** panel.

The rows in the **Inputs** panel of the editor correspond to Inport block BPAV in the engine_idle_speed model. See “Import Data for Parameter Estimation” on page 1-6.

The rows in the **Outputs** panel correspond to Outport block Engine Speed. You can import signal data from files or MATLAB workspace.

The idle-speed model of an automotive engine contains the measured data stored in the `iodata` array in the workspace. The array contains two columns: the first for input data, and the second for output data. The time data is in the `time` array in the workspace. Import the input data by typing `[time,iodata(:,1)]` in the dialog box in the **Inputs** panel.



Import the output data by typing `[time,iodata(:,2)]` in the dialog box in the **Outputs** panel.

Note You can have more than one input or output signal, but you can have only one data set for a signal. If you have multiple data sets, create multiple experiments.

See Also

Related Examples

- “Model Requirements for Importing Data” on page 1-2
- “Specify Parameters for Estimation” on page 2-9
- “Specify Known Initial States” on page 2-19
- “Estimate Parameters and States” on page 2-26

More About

- “What Is an Experiment?” on page 2-3

Specify Parameters for Estimation

In this section...

“Choosing Which Parameters to Estimate First” on page 2-9


“Add Model Parameters as Variables for Estimation” on page 2-9

“Specify Parameters for Estimation” on page 2-12

Choosing Which Parameters to Estimate First

Simulink Design Optimization software lets you estimate scalar, vector, and matrix parameters. You can take an iterative approach to estimating model parameters. For example, if you have a large number of parameters to estimate, start by estimating those that most influence the output. After you estimate a subset of parameters and validate the estimated parameters, you can select the remaining parameters for estimation.

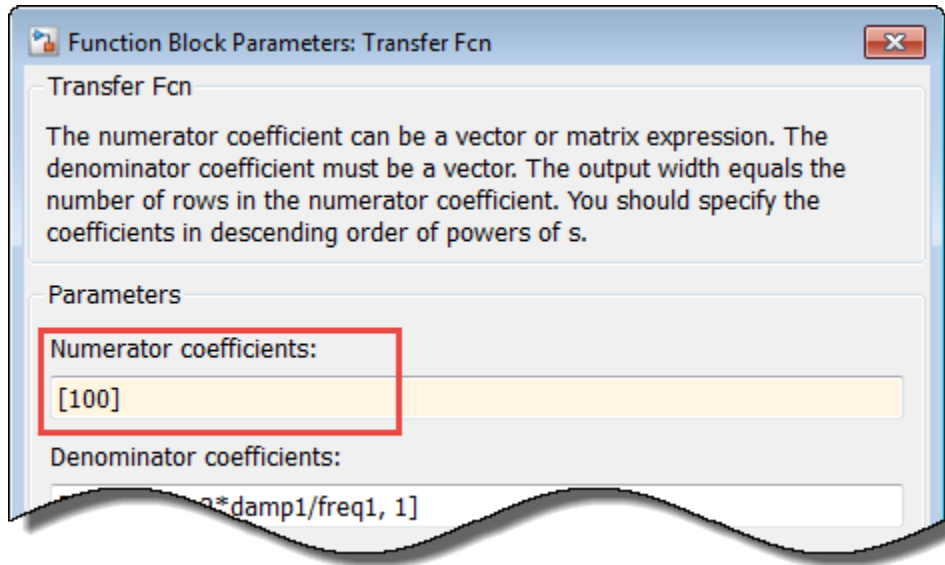
You can also first use sensitivity analysis to identify the parameters that most influence the estimation, and then specify these parameters for estimation. To open the

Sensitivity Analysis tool, in the **Parameter Estimation** tab, click  **Sensitivity Analysis**. In the Sensitivity Analysis tool, you can identify the model parameters that most influence the estimation problem and compute initial values for the estimation parameters.

Add Model Parameters as Variables for Estimation

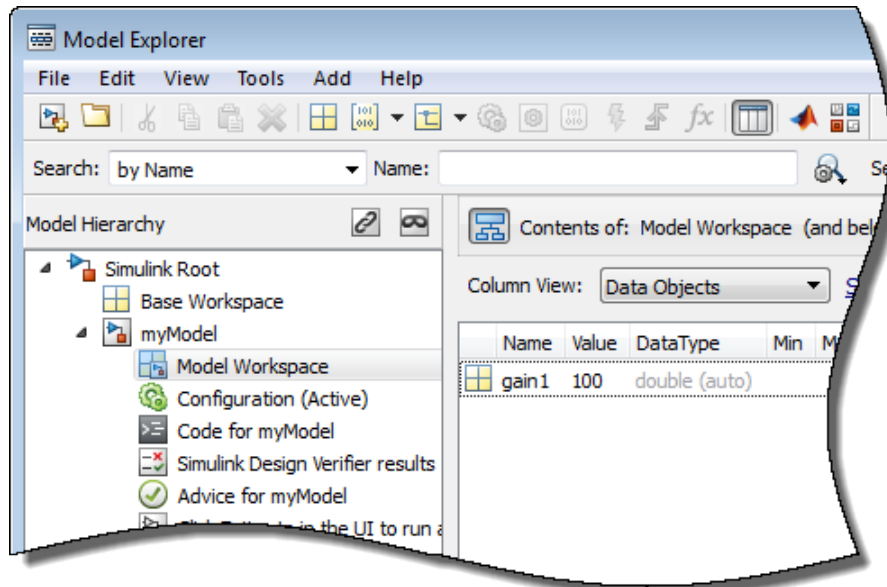
The software can only estimate variables that are in use by the model. Create variables for estimation in the MATLAB or model workspace, and specify your Simulink model or block parameters using these variables.

In this figure, the **Numerator coefficients** parameter of a Transfer Fcn block is specified as a numerical value.



To estimate the **Numerator coefficients** parameter, specify it as variable `gain1`:

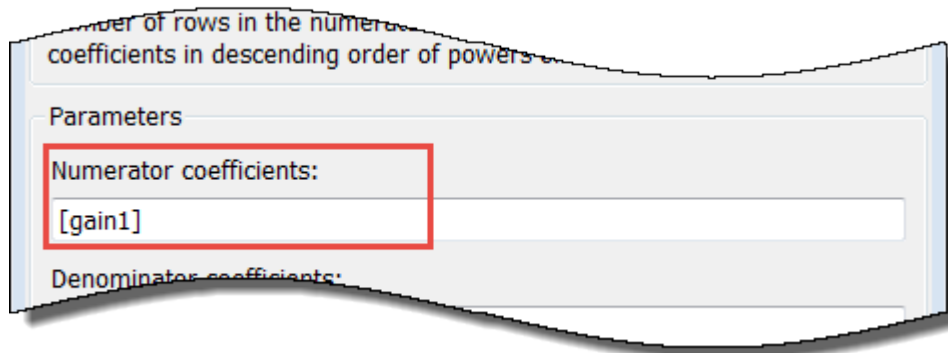
- 1 Create the variable `gain1` in one of the following ways:
 - Add the variables to the model workspace (Simulink), and specify initial values.



- Write initialization code in the **PreloadFcn** callback of the model. For more information, see “Model Callbacks” (Simulink).

```
gain1 = 100
```

- 2 Specify the block parameter as variable `gain1` in the Transfer Fcn block dialog box.



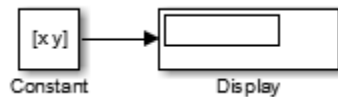
You can now select `gain1` for estimation. See, “Specify Parameters for Estimation” on page 2-12.

Specify Independent Parameters for Estimation

You can also specify independent parameters that do not appear explicitly in the model as variables for estimation. However, you cannot use this workflow with Simulink fast restart.

Suppose that a model parameter K_{int} is related to independent parameters x and y such that $K_{int} = x+y$. To estimate x and y instead of K_{int} :

- Create the independent variables x and y by adding them to the model workspace and specifying initial values.
- The software only allows tuning of variables that are used by model blocks. To ensure that the software detects x and y for tuning, add a Constant block to your model, and specify the **Constant value** of the block as $[x \ y]$. Connect the block to a Display block.



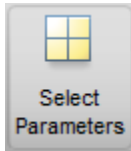
- Write code in the **InitFcn** callback of the model that defines the relationship between K_{int} , x , and y . You must first use the `get_param` function to get the variables x and y from the model workspace before you can use them to define K_{int} .

```
wks = get_param(gcs, 'ModelWorkspace')
x = evalin(wks, 'x')
y = evalin(wks, 'y')
Kint = x+y;
```

You can now select x and y for estimation. Do not estimate the independent and dependent parameters simultaneously. Doing so can lead to incorrect results. For example, do not estimate K_{int} , x and y together.

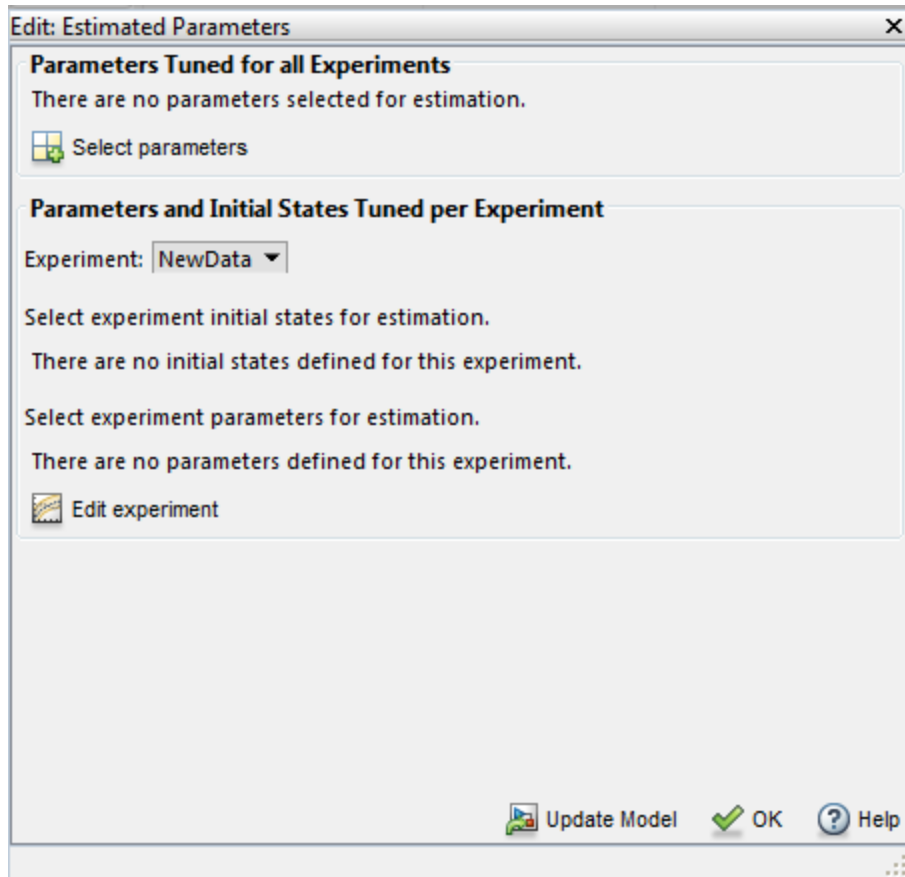
Specify Parameters for Estimation

You can specify the parameters for estimation experiments using the **Estimated Parameters** editor. In the Parameter Estimation tool, on the **Parameter Estimation** tab, click **Select Parameters**.

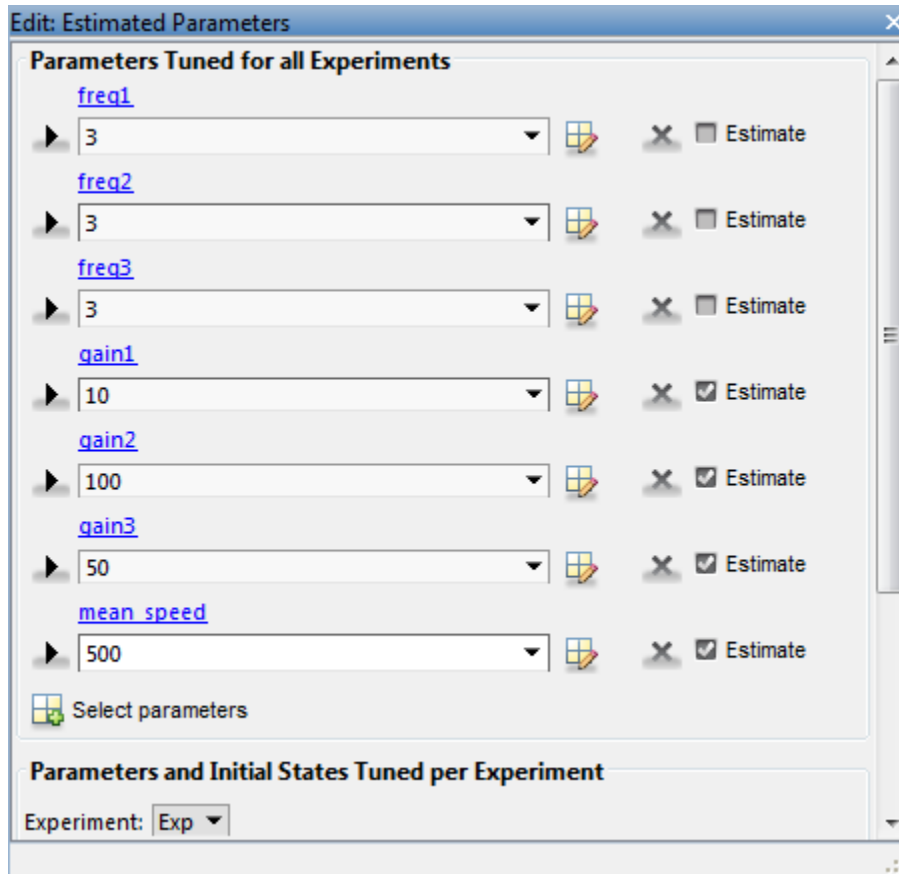


To select parameters for all experiments, click **Select Parameters** in the **Parameters Tuned for all Experiments** panel. This opens the **Select model variables** dialog. Here you can select the parameters you want to estimate by clicking the check box next to it or specifying an expression. For more information see “Select Parameters Using Select Model Variables Dialog Box” on page 2-15.

The editor looks like



For example, in the `engine_idle_speed` model, select `freq1`, `freq2`, `freq3`, `gain1`, `gain2`, `gain3` and `mean_speed` for estimation. You do not need to estimate the parameters all at once. You can first select all the parameters you are interested in, and then later select a subset to estimate. By default, all the parameters are selected for estimation. To deselect the ones you do not want to estimate, clear the **Estimate** check box for a parameter. For this example, only estimate `gain1`, `gain2`, `gain3` and `mean_speed`. Set their initial values 10, 100, 50, and 500, respectively, and then click **OK**. The **Edit: Estimated Parameters** dialog box looks like



To learn how to specify initial values and upper and lower bounds of the parameters, see “Specifying Initial Guesses and Upper/Lower Bounds” on page 2-17.

Select Parameters to Estimate for a Specific Experiment

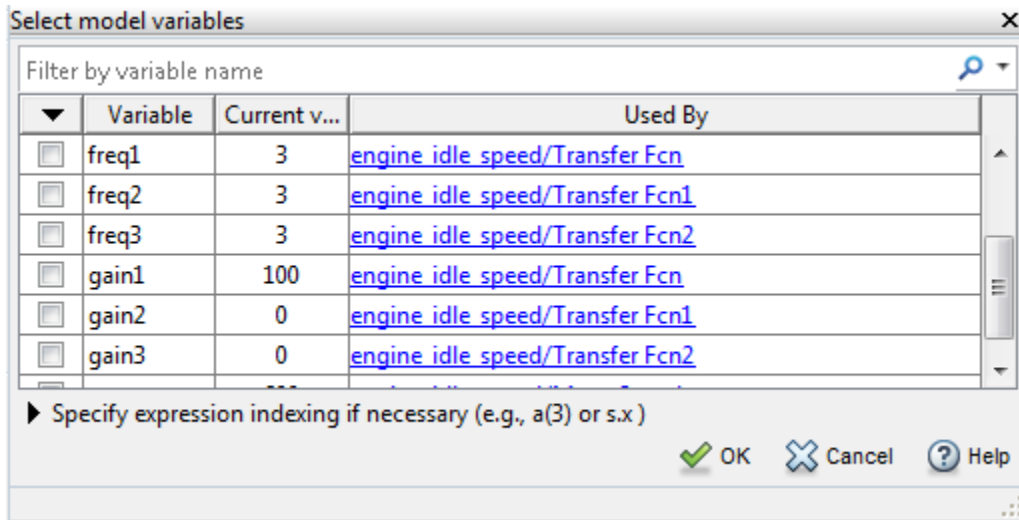
To select the parameters to estimate in a specific experiment, first, select the experiment for estimation as described in “Estimate Parameters and States” on page 2-26. Then, you can use the **Edit:Estimated Parameters** dialog to select parameters to estimate for that experiment. Select the experiment name from the **Experiment:** combo box in the **Parameters and Initial States Tuned per Experiment** panel. Then click **Edit experiment** to launch the experiment editor for the experiment you select.

Alternatively, you can right click the experiment name in the **Experiments** list and select **Edit...** In the experiment editor, click the **Select parameters** button in the **Parameters** panel. In the Select model variables dialog box, you can select the parameters you want to estimate in this experiment by checking the box next to it or specifying an expression. For more information see “Select Parameters Using Select Model Variables Dialog Box” on page 2-15.

Select Parameters Using Select Model Variables Dialog Box

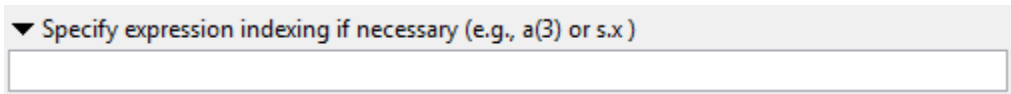
Use this dialog box to specify parameters to estimate. The table lists the variables that the model uses to set block parameter values. The variables can reside in the model workspace, the base workspace, or a data dictionary.

Select variables by clicking the check box next to each variable. If your model contains many variables, filter the list by typing in the **Filter by variable name** field. The **Used By** column lists all blocks in the model that use the variable. When a variable is used in more than one block, all blocks are listed. To highlight blocks in the model that use the variable, click the block name.



The variables that you select must have a numeric value that uses the data type double. If the value of a variable is not a double number, use these techniques:

- To select a single element or a subset of a matrix or array variable, click **Specify expression indexing if necessary**.



Enter an expression such as `myArray(2)`, which selects the second element of an array variable `myArray`.

After you type the expression, press the **Enter** key to add the variable to the list of model variables.

- To use a variable of a numeric data type other than double, convert the variable to a `Simulink.Parameter` object, which separates a parameter value from its data type. Set the `Value` property to a default double number, and use the `DataType` property to control the data type.
- To use the value of a `Simulink.Parameter` object, specify the `Value` property. Enter the expression `myParamObj.Value`.

- To use a numeric field of a structure, enter `myStruct.PID.P1`. If you store the structure in a `Simulink.Parameter` object, enter `myStruct.Value.PID.P1`.
- To use one cell of a cell array, enter `myCells{3}`.

You cannot use mathematical expressions such as $a + b$. Sometimes, models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as $k = a + b$, where a and b are not defined in the model but k is used. To add these independent parameters, see “Add Model Parameters as Variables for Estimation” on page 2-9.

Specifying Initial Guesses and Upper/Lower Bounds

After you select parameters, you can specify

- **Initial guess** — The value the estimation uses to start the process.
- **Minimum** — The smallest allowable parameter value. The default is `-Inf`.
- **Maximum** — The largest allowable parameter value. The default is `+Inf`.

You can enter the initial value in the dialog box below the parameter name. You can specify the minimum and maximum value fields by clicking the arrow. The default minimum and maximum values are `-Inf` and `+Inf`, respectively, but you can select any range you want.

The screenshot shows a dialog box for a parameter named 'gain1'. It contains four input fields: 'Value' (10), 'Minimum' (-Inf), 'Maximum' (Inf), and 'Scale' (10). Each field has a dropdown arrow on the right. To the right of the 'Value' field are three icons: a help icon, a delete icon (X), and a checked 'Estimate' checkbox.

If you believe a parameter lies within a finite range, it is best not to use the default minimum and maximum values. Often, there are computational advantages in specifying finite bounds. It can be very important to specify lower and upper bounds. For example, if a parameter specifies the weight of a part, be sure to specify 0 as the absolute lower bound if better information is unavailable.

Note When you specify the minimum and maximum values for the parameters, it does not affect your settings in the **Parameters** list under **Data Browser** pane. You make these choices for each experiment.

- **Scale** — The scale value to use for normalization. The parameters are scaled, or normalized, by dividing their current value by the scale value. **Scale** is useful, in situations, for example, when parameters have different orders of magnitude.

The default scale value is the next power of 2 greater than the current value of the parameter. For example, if the current parameter value is 15, **Scale** is 16 ($=2^4$). You can edit this field to provide an alternate scaling factor.

See Also

Related Examples

- “Specify Known Initial States” on page 2-19

More About

- “What Is an Experiment?” on page 2-3

Specify Known Initial States

In this section...

“When to Specify Initial States Versus Estimate Initial States” on page 2-19

“Specify Model Initial States” on page 2-19

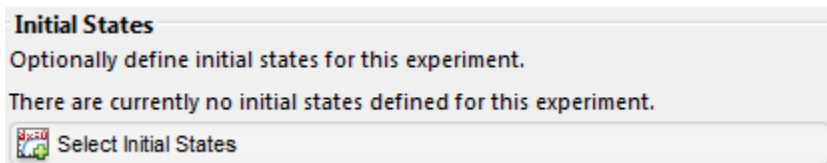
When to Specify Initial States Versus Estimate Initial States

Sets of measured data are often collected at various times and under different initial conditions. When you estimate model parameters using one data set and subsequently run another estimation with a second data set, your parameter values may not match.

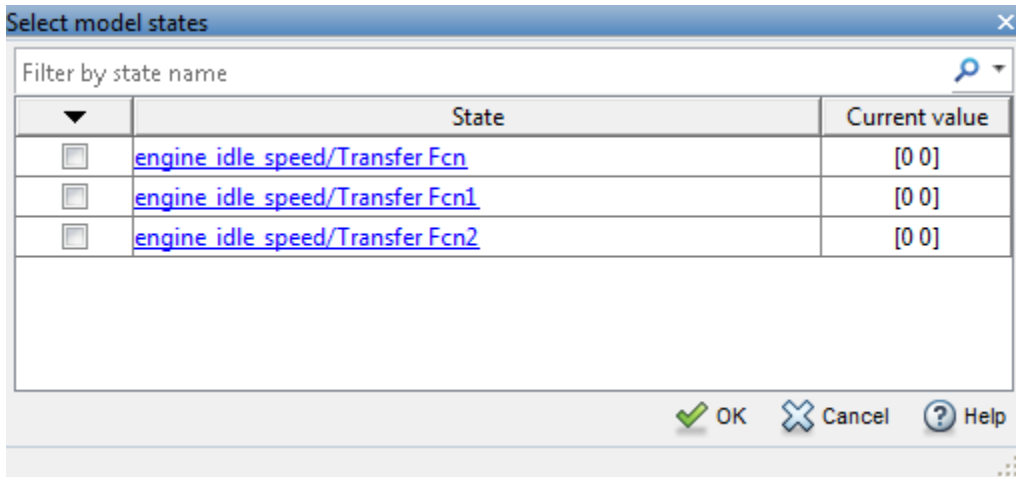
You can use the Parameter Estimation tool to estimate the initial conditions using procedures that are similar to those you use to estimate parameters. You can then use these initial condition estimates as a basis for estimating parameters for your Simulink model.

Specify Model Initial States

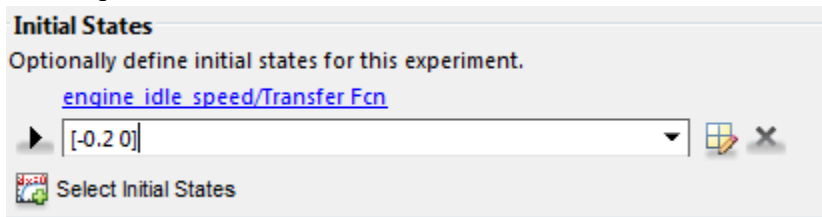
After you select parameters for estimation, as described in “Specify Parameters for Estimation” on page 2-9, you can specify initial conditions of states in your model. By default, the estimation uses initial conditions specified in the Simulink model. If you want to specify initial conditions other than the defaults, use the **Initial States** panel in the experiment editor dialog. For this example, right click `NewData` and select **Edit...** from the list to open the experiment editor on page 2-6. Then, click **Select Initial States** button.



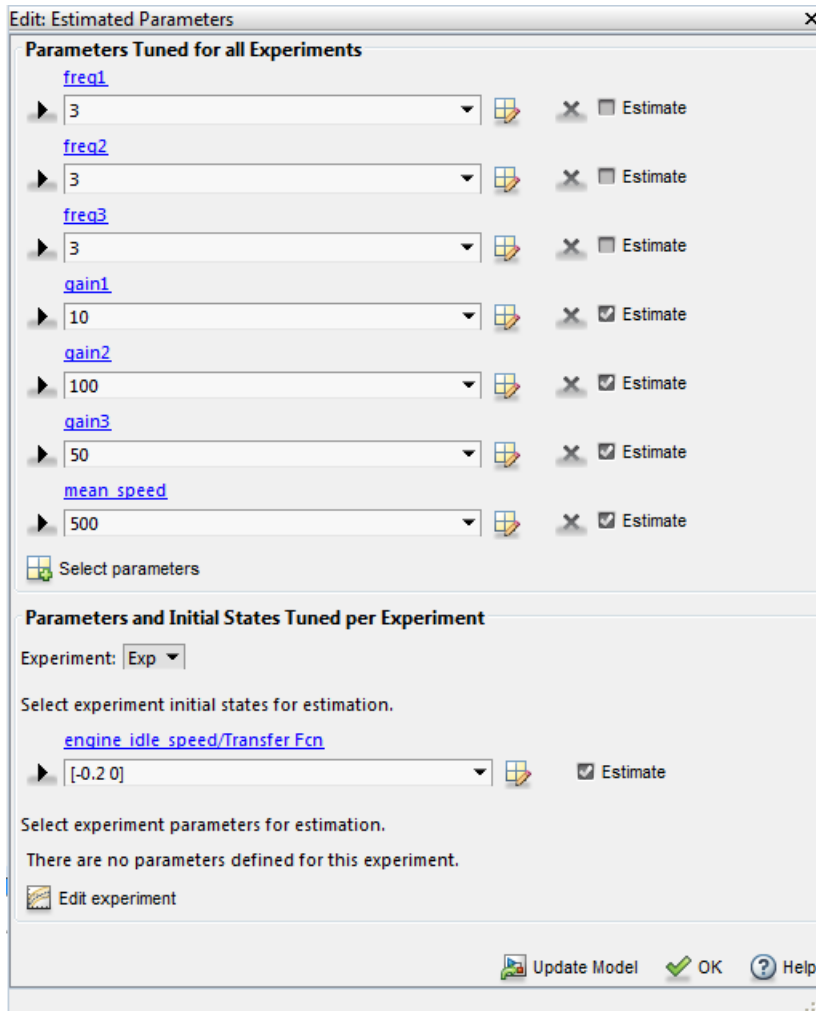
The Select Model States dialog for the `engine_idle_speed` model looks like



Click the check box next to the state you would like to modify. For example, if you select **engine_idle_speed/Transfer Fcn** and enter the initial values -0.2 and 0 , the **Initial States** panel now looks like



Click **Select Parameters** in the **Parameter Estimation** tab. After you also select the parameters as described in “Specify Parameters for Estimation” on page 2-9, the Edit: Estimated Parameters dialog looks like the following figure.



See Also

Related Examples

- “Specify Estimation Data” on page 2-4

- “Specify Parameters for Estimation” on page 2-9

More About

- “Edit Experiment Data” on page 2-6

Specify Estimation Options

This topic shows how to specify estimation options in the Parameter Estimation tool.

After you have specified estimation data and parameters, specify the following estimation options:

1 Goodness of fit criteria (cost function)

Cost Function: Sum Squared Error ▼

The cost function is a function that estimation methods minimize. To specify the method for calculating the cost function, in the **Parameter Estimation** tab of the tool, select one of the following from the **Cost Function** drop-down list:

- **Sum Squared Error** — Uses a least-squares approach (default).
- **Sum-Absolute Error** — Uses the sum of absolute errors.

If the experimental data has many outliers, you can select the robust cost option. 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 having to manually modify your data. To select this option,

in the **Parameter Estimation** tab, click  **More Options** to open the Estimation Options dialog box. In the **Optimization Options** tab of the dialog box, select **Use robust cost**.

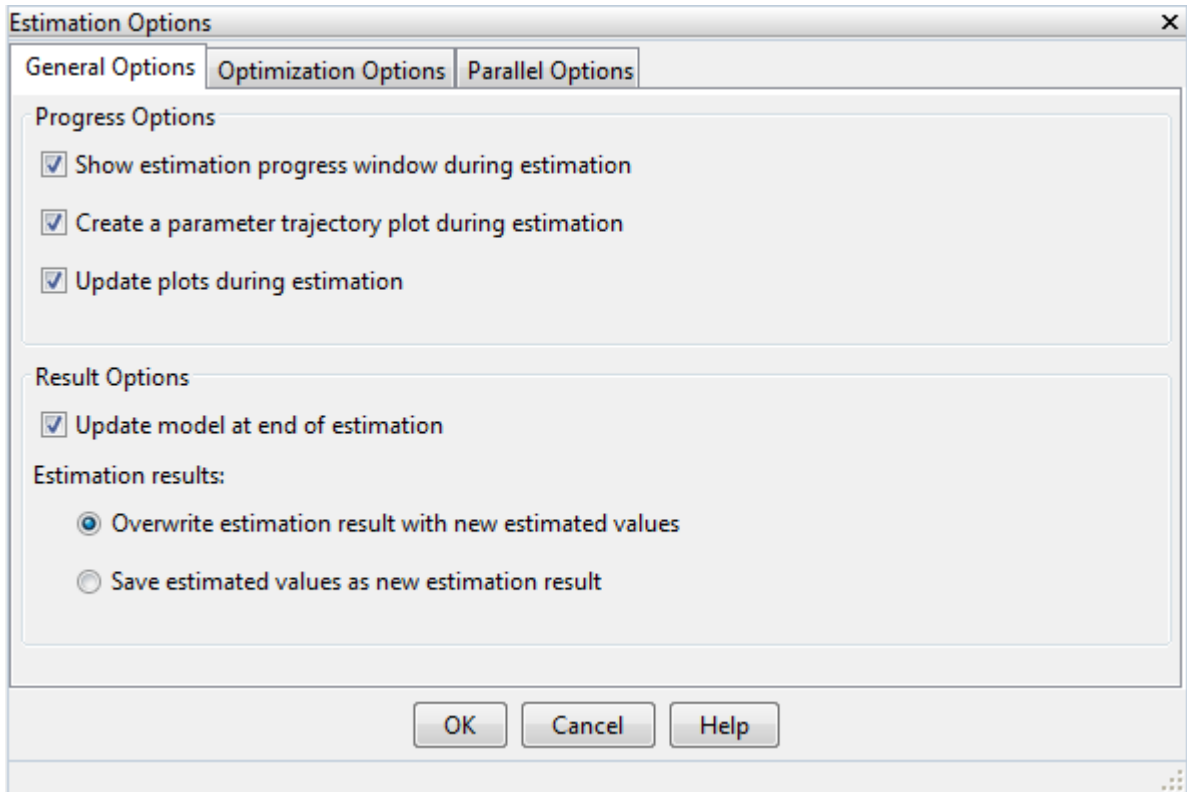
The software uses the error statistics to identify the outliers. The error, e , is calculated as the difference between measured and simulated output. The cost function used, $F(x)$, depends on the method used.

Method Name	Cost Function for Nonoutliers	Cost Function for Outliers
Sum Squared Error	$F(x) = \sum_{t \in NOL} e(t) \times e(t)$ <p>NOL is the set of nonoutlier samples.</p>	$F(x) = \sum_{t \in OL} w \times e(t) $ <p>w is a linear weight. OL is the set of outlier samples.</p>

Method Name	Cost Function for Nonoutliers	Cost Function for Outliers
Sum-Absolute Error	$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>

2 Estimation progress and result options for estimation task

To specify these options, in the **Parameter Estimation** tab, click **More Options** to open the Estimation Options dialog box. In the **General Options** tab, specify the estimation progress and result options. For details about the options, click the **Help** button.



- 3 Optimization options, such as optimization method and optimization termination options.

Specify these options in the **Optimization Options** tab of the Estimation Options dialog box. For details about the options, click the **Help** button.

- 4 Parallel computing options

Specify these options in the **Parallel Options** tab of the Estimation Options dialog box. For details about the options, see “Estimate Parameters Using Parallel Computing in the Parameter Estimation Tool” on page 2-44.

See Also

Related Examples

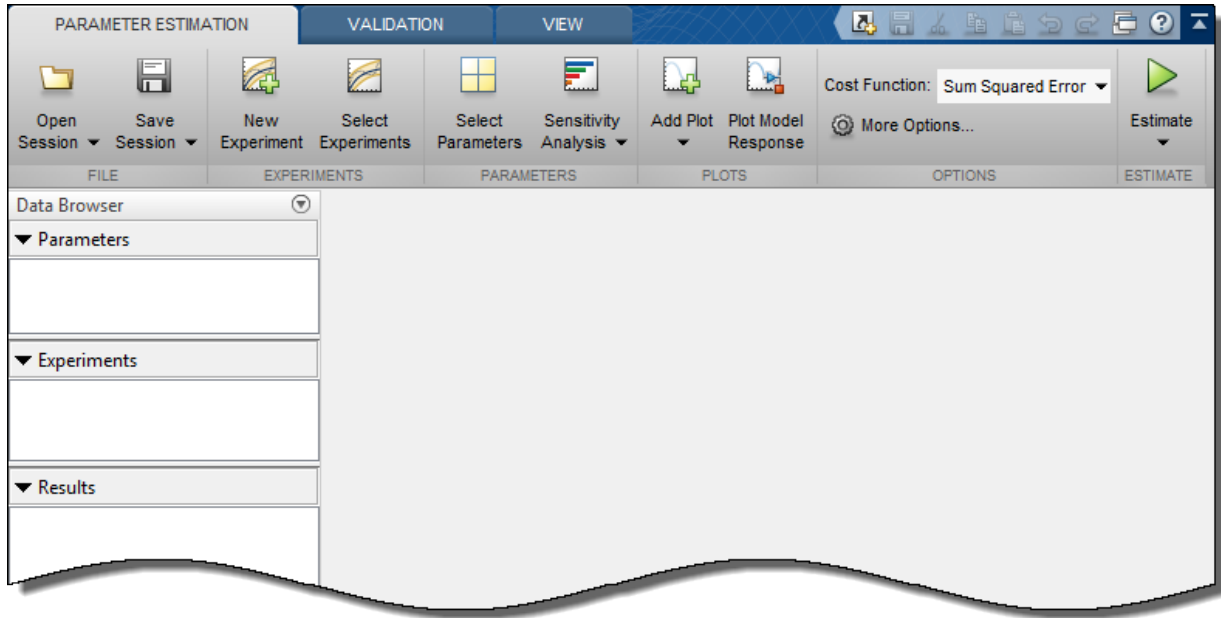
- “Specify Estimation Data” on page 2-4
- “Specify Parameters for Estimation” on page 2-9

More About

- “How the Software Formulates Parameter Estimation as an Optimization Problem” on page 2-53

Estimate Parameters and States

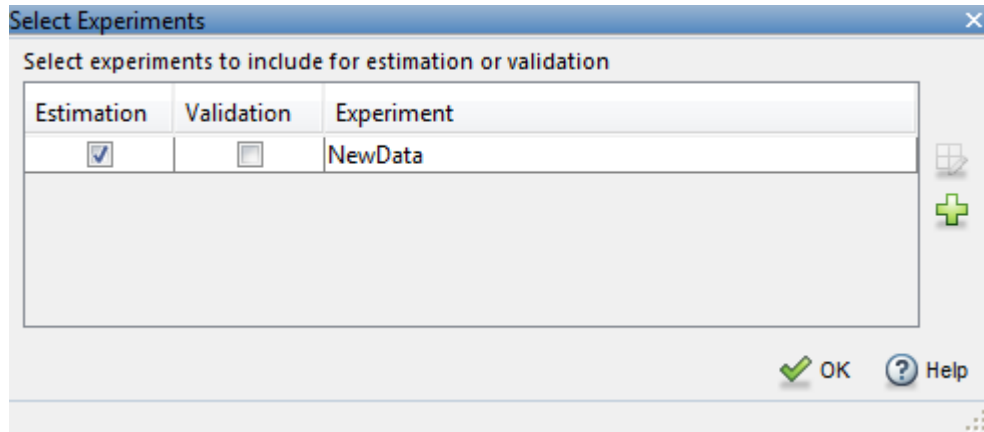
This topic shows how to estimate parameters and states in the Parameter Estimation tool.



To estimate parameters and states after you have specified estimation data on page 2-4, parameters on page 2-9, and estimation options on page 2-23:

- 1 Specify experiments for estimation.

In the Parameter Estimation tool, in the **Parameter Estimation** tab, click **Select Experiments**. In the Select Experiments dialog box, in the **Estimation** column, select the experiment to use.



For information about the **Validation** column, see “Validate Estimation Results” on page 2-34.

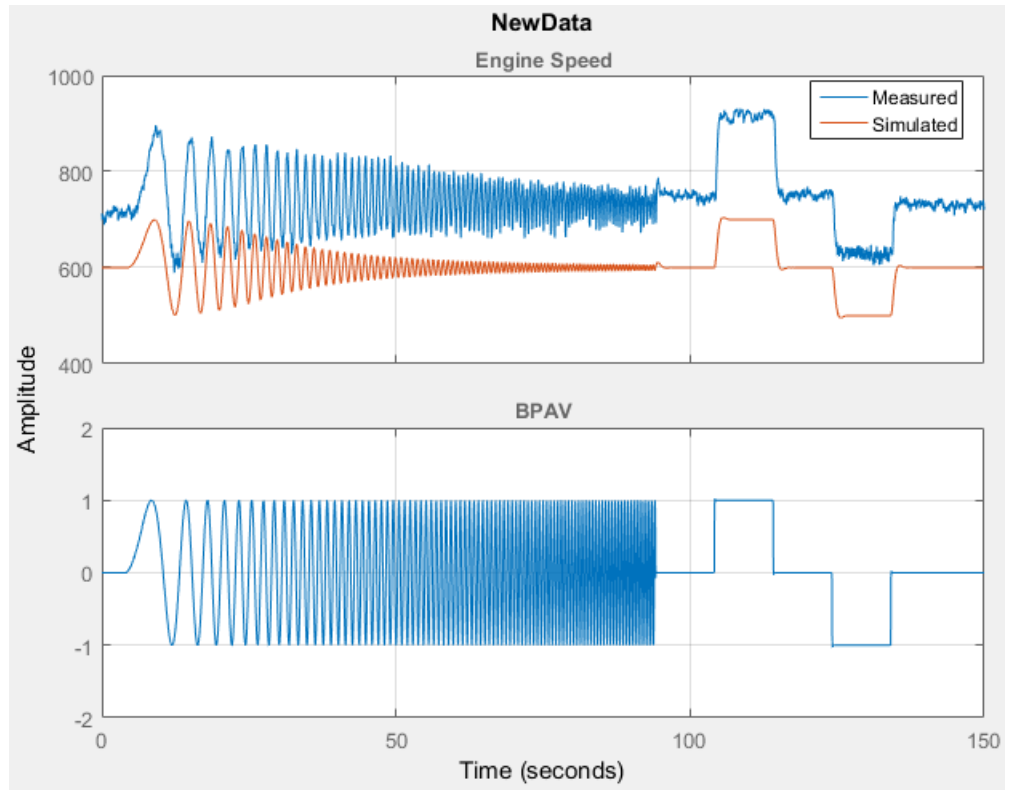
2 (Optional) Create progress plots to view estimation progress.

When you start the estimation, the tool automatically displays a parameter trajectory plot that shows the change in the parameter values during estimation. You can create other plots for viewing the progress of the estimation before you begin estimating the parameters. During estimation, all these plots update with each iteration.

- To plot the measured data, on the **Parameter Estimation** tab, click **Add Plot**. Select the experiment to use for estimation under **Experiment Plots** of the drop-down list.

To add the simulated response to the measured data plot, on the **Parameter Estimation** tab, click **Plot Model Response**.

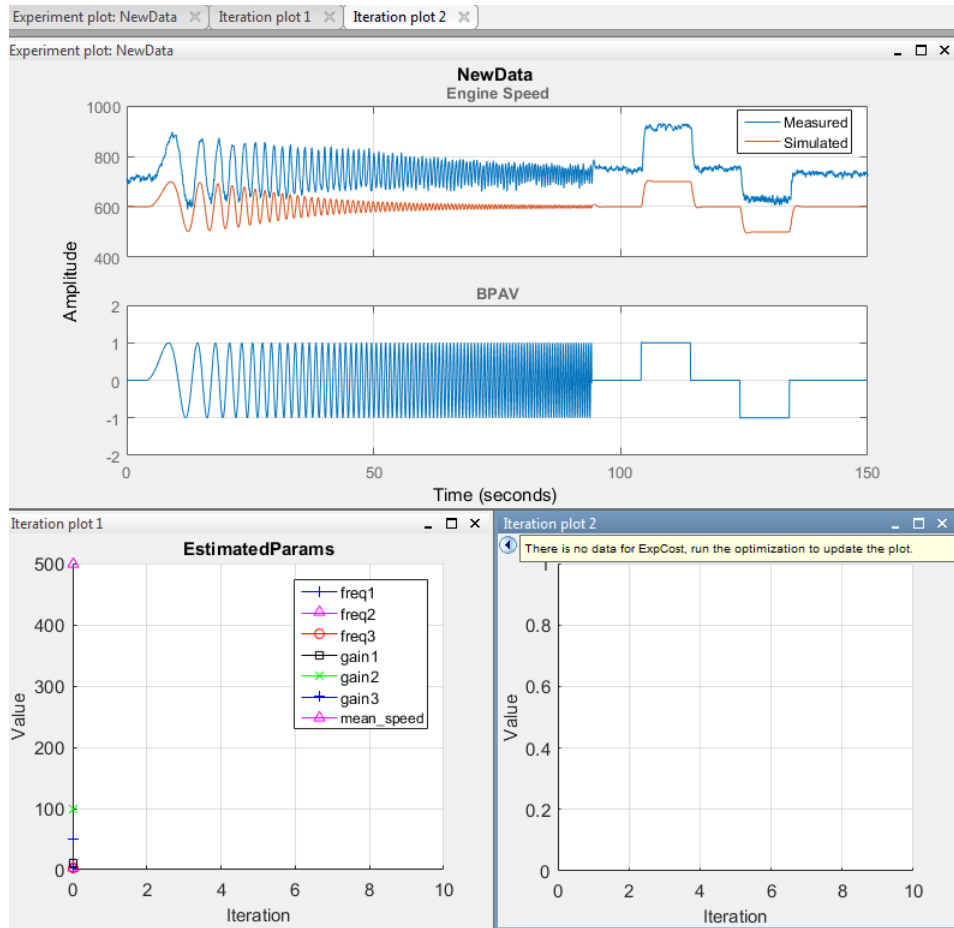
Alternatively, right-click the experiment name in the **Experiments** area of the tool, and select **Plot measured & simulated data** from the menu.



You can edit the labels, adjust the limits, change the units, and the font style of the plot in the **Property Editor**. To open the editor, right-click the experiment plot and select **Properties** from the list.

- To plot the parameter values as they change, click **Add Plot**, and select **Parameter Trajectory**. To add the scaled values or save iteration data, right-click the plot.
- To add a plot for the estimation cost, click **Add Plot**, and select **Estimation Cost**. To add the scaled values or save iteration data, right-click the plot.

Use the **View** tab of the tool to arrange the layout of the plots, so that all the progress plots you created are visible.



- 3 Estimate the parameters and states.

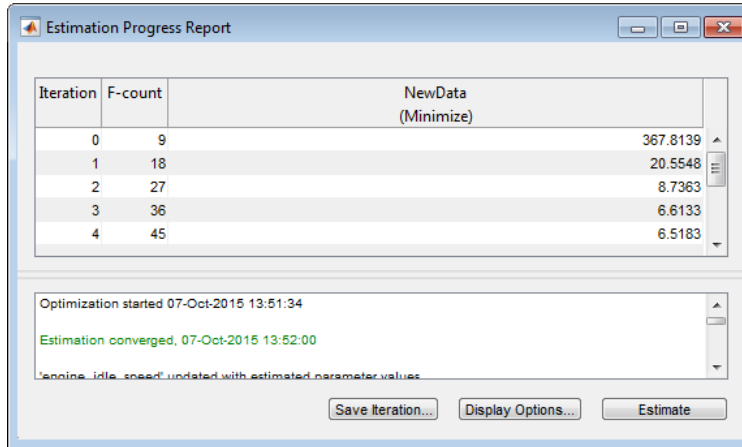
In the **Parameter Estimation** tab, click  **Estimate**.

An Estimation Progress Report window opens at the start of estimation. The report and progress plots update with each iteration.

- 4 View the Estimation Progress Report after the estimation completes.

By default, the report displays the iteration number (**Iteration**), the number of times the objective function is calculated (**F-count**), and cost function value (**f(x)**), for

example, **NewData(Minimize)**). You can change the display table by clicking **Display Options**. To learn more about the display table, see “Iterative Display” (Optimization Toolbox).



The screenshot shows a window titled "Estimation Progress Report". It contains a table with the following data:

Iteration	F-count	NewData (Minimize)
0	9	367.8139
1	18	20.5548
2	27	8.7363
3	36	6.6133
4	45	6.5183

Below the table, the log shows:

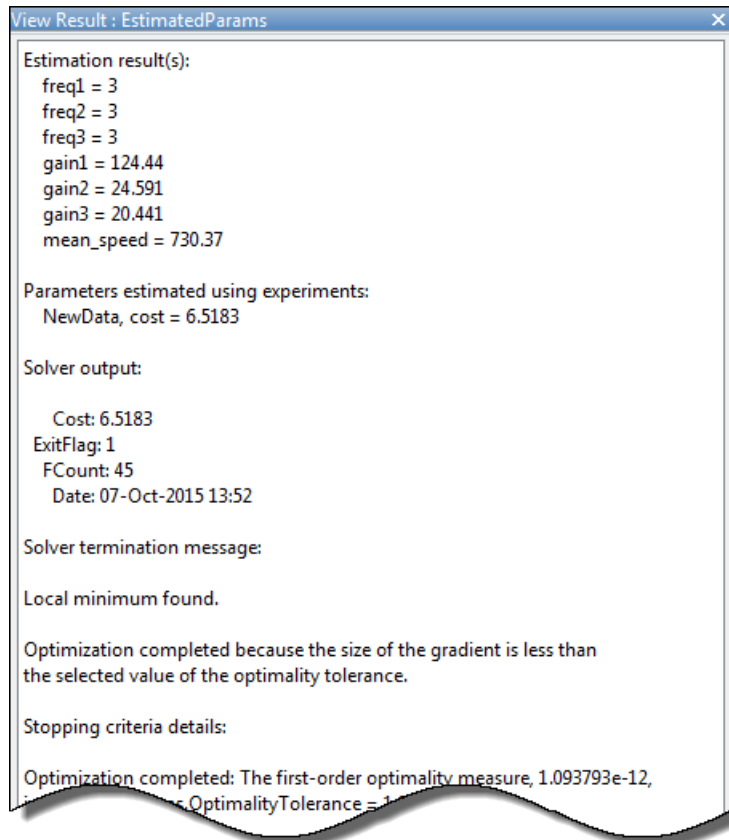
```
Optimization started 07-Oct-2015 13:51:34  
Estimation converged, 07-Oct-2015 13:52:00  
'engine_idle_speed' updated with estimated parameter values
```

At the bottom of the window are three buttons: "Save Iteration...", "Display Options...", and "Estimate".

5 View the estimated results.

The estimation results are saved in a new variable, `EstimatedParams`, in the **Results** area of the tool.

To view the contents of the variable, right-click `EstimatedParams` and select **Open...** from the menu. `EstimatedParams` includes the values of the parameters, the cost function value, and information about the stopping criteria for the estimation.



```
View Result : EstimatedParams
Estimation result(s):
  freq1 = 3
  freq2 = 3
  freq3 = 3
  gain1 = 124.44
  gain2 = 24.591
  gain3 = 20.441
  mean_speed = 730.37

Parameters estimated using experiments:
  NewData, cost = 6.5183

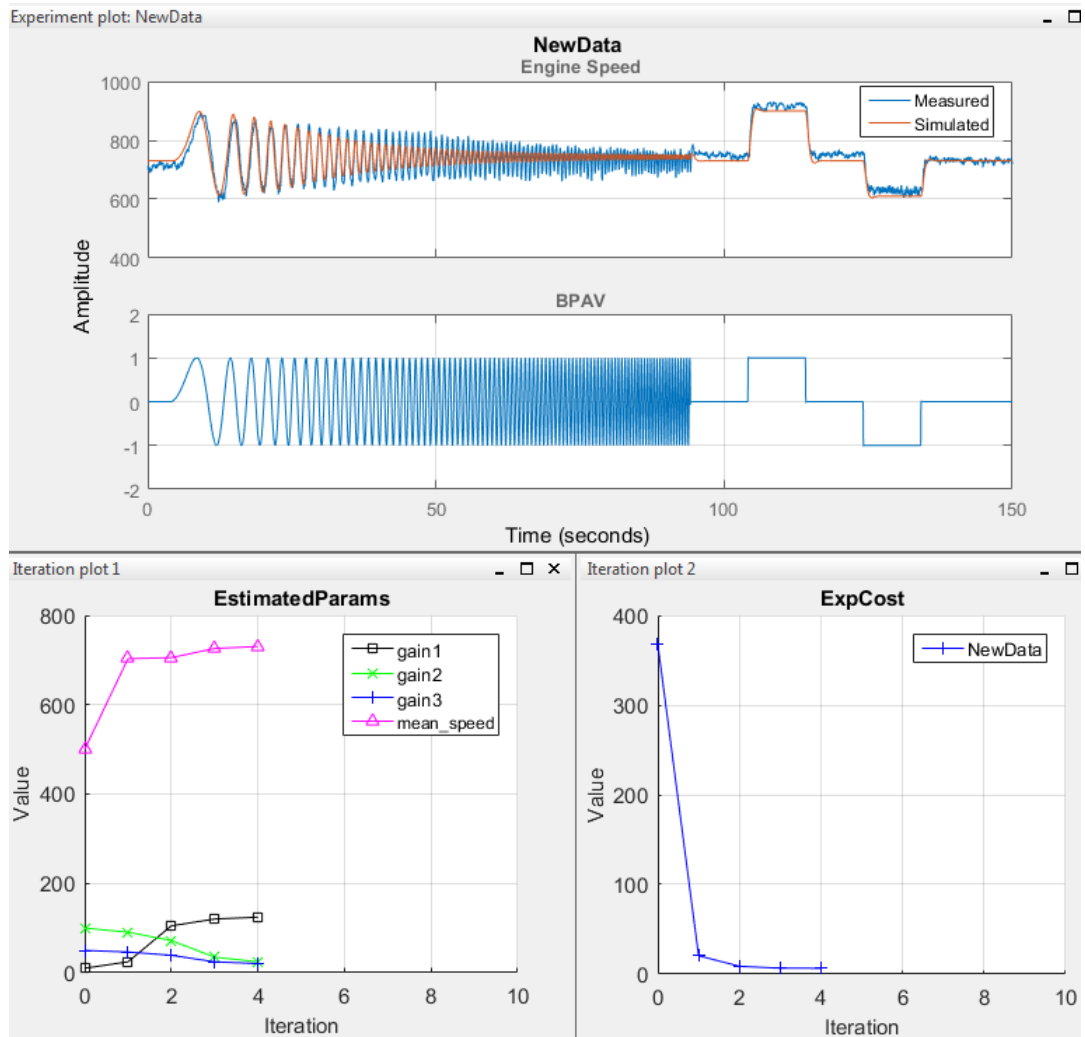
Solver output:
  Cost: 6.5183
  ExitFlag: 1
  FCount: 45
  Date: 07-Oct-2015 13:52

Solver termination message:
  Local minimum found.

  Optimization completed because the size of the gradient is less than
  the selected value of the optimality tolerance.

Stopping criteria details:
  Optimization completed: The first-order optimality measure, 1.093793e-12,
  is less than the OptimalityTolerance = 1e-06.
```

- 6 View the progress plots.



The measured versus simulated data plot shows how closely the simulated data matches the measured estimation data. The estimated parameters plot and cost function plots show the changes in the estimated value of the parameters and estimated cost function for each iteration.

Typically, a lower cost function value indicates the model simulation with the estimated parameters closely matches experimental data. If the optimization went

well, you should see your cost function converge to a minimum value. The lower the cost, the more successful the estimation.

For information on types of problems you may encounter using optimization solvers, see “When the Solver Fails” (Optimization Toolbox), “When the Solver Might Have Succeeded” (Optimization Toolbox), and “When the Solver Succeeds” (Optimization Toolbox).

See Also

Related Examples

- “Specify Estimation Data” on page 2-4
- “Specify Parameters for Estimation” on page 2-9
- “Specify Known Initial States” on page 2-19
- “Specify Estimation Options” on page 2-23
- “Validate Estimation Results” on page 2-34
- “Save and Load Estimation Sessions” on page 2-51

More About

- “What Is an Experiment?” on page 2-3

Validate Estimation Results

This topic shows how to validate estimation results in the Parameter Estimation tool. After you estimate parameters as described in “Estimate Parameters and States” on page 2-26, validate the estimation results using another data set.

Configure and Perform Validation

To validate a model using the Parameter Estimation tool:

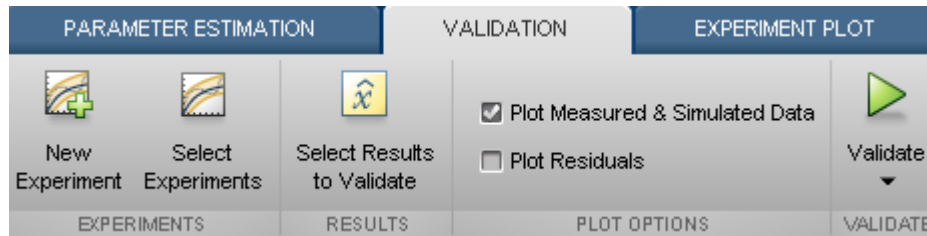
- 1 Load the validation data set.

At the MATLAB prompt, load the validation data into the MATLAB workspace.

```
load iodataval;
```

- 2 Add a new experiment for validation.

In the Parameter Estimation tool, in the **Validation** tab, click **New Experiment**. A new experiment with default name appears in the **Experiments** area of the tool. To rename the experiment `ValidationData`, right-click the experiment and select **Rename** from the drop-down menu.



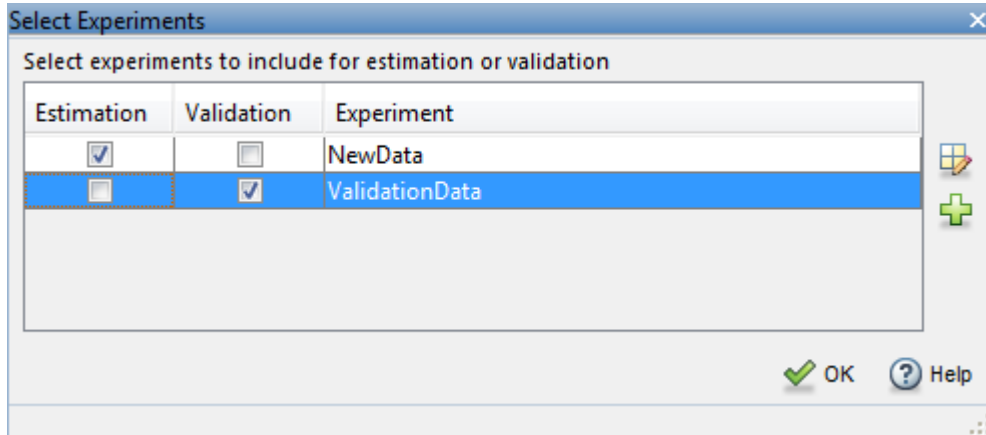
- 3 Import the validation data set into the validation experiment.

Right click the experiment name, and select **Edit**. Specify `[time, iodataval(:, 1)]` in **Inputs**, and `[time, iodataval(:, 2)]` in **Outputs**.

- 4 Specify the experiment for validation.

When you create an experiment, it is by default selected for estimation. To select another experiment for validation, in the **Validation** tab, click **Select Experiments**.

In the Select Experiments dialog box, clear **Estimation** and select **Validation** for the validation experiment.

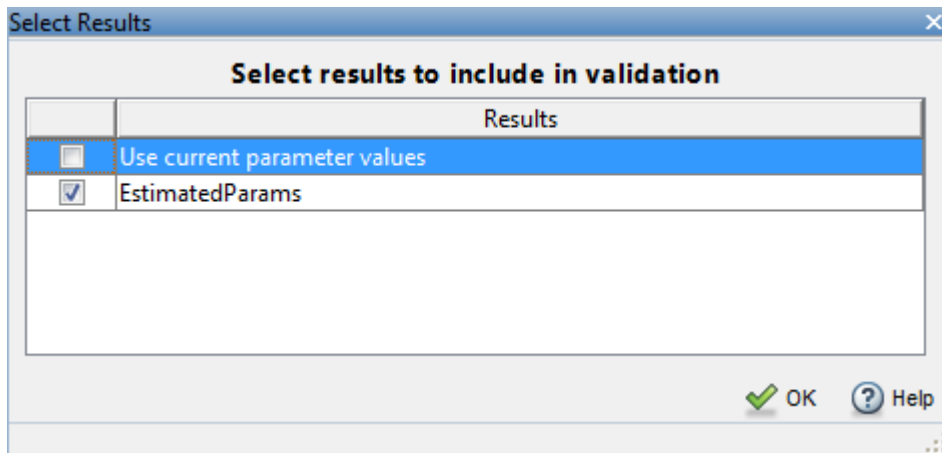


- 5 Specify the estimation results to use in the validation.

After you import validation data, select the estimated parameter values to validate.

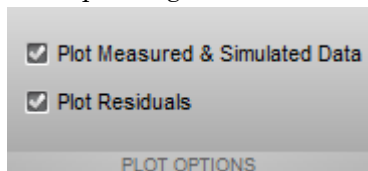
The estimation results are saved in the `EstimatedParams` variable in the **Results** area of the tool. In the **Validation** tab, click **Select Results to Validate**.

To validate the estimated parameters, in the Select Results dialog box, select `EstimatedParams` and clear `Use current parameter values`.



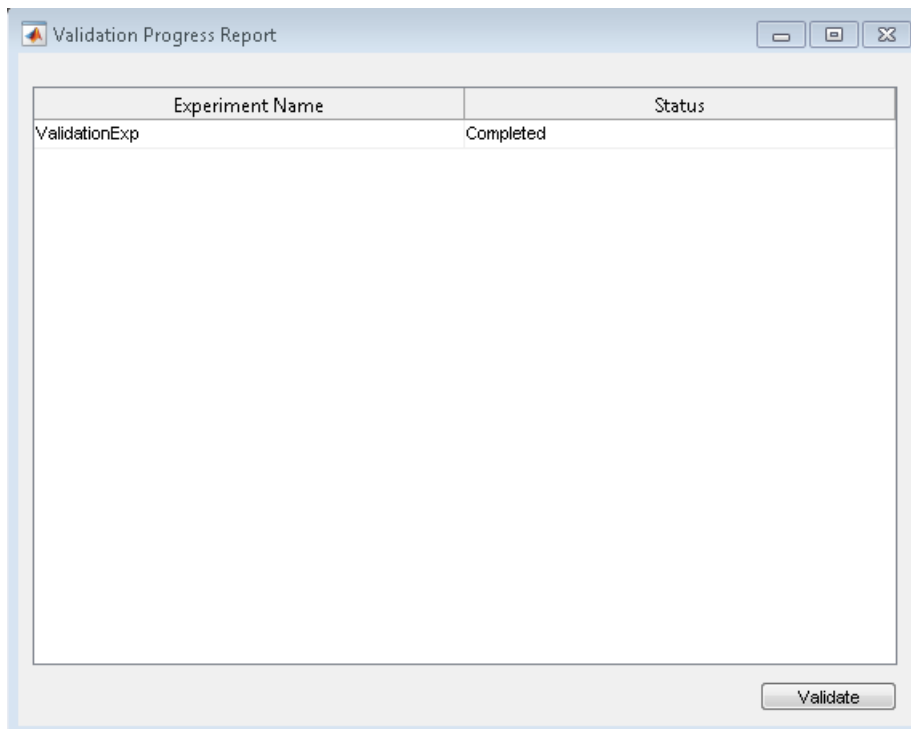
- 6 Select the plots to display at the end of validation.

Parameter Estimation tool can display the measured and simulated responses and the residuals plot at the end of validation. Select the plots to display by checking the corresponding box on the **Validation Tab**.



- 7 Validate the estimation results.

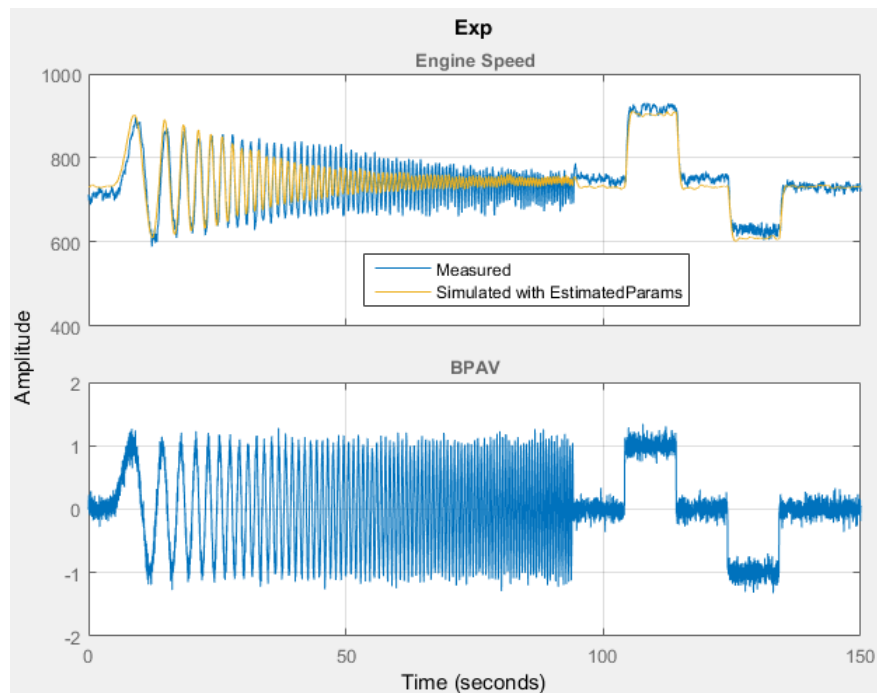
On the **Validation** tab, click **Validate**. The Validation Progress Report shows the status of the validation.



Compare Measured and Simulated Responses

- 1 Compare measured validation data against the model output simulated with the estimated parameters.

The tool displays the experiment plot for each experiment selected for validation. Each experiment plot shows the measured data, and data from simulation using each set of results selected. For example, the following figure shows the experiment plot for the `ValidationExp` data.

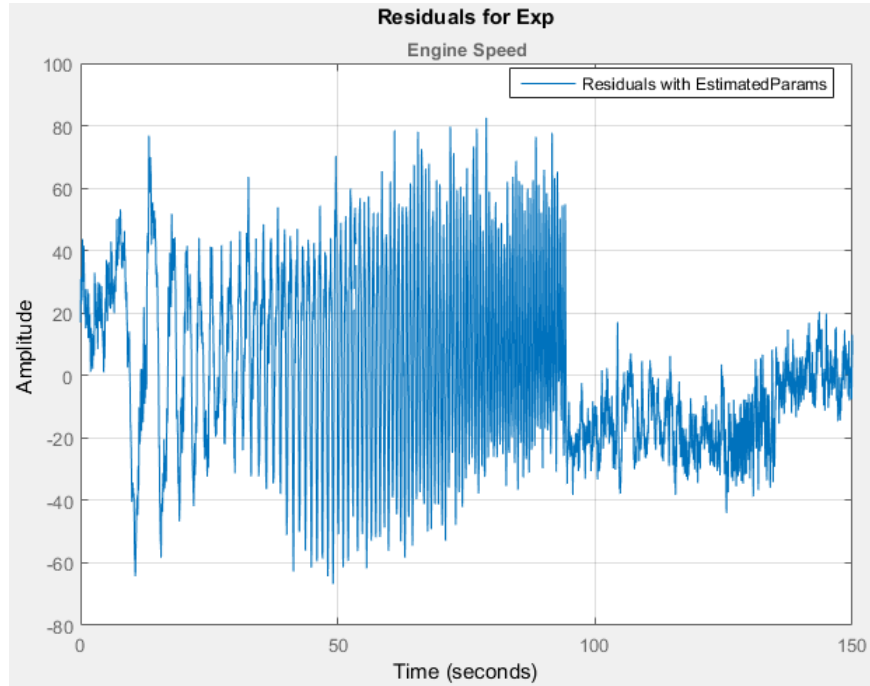


- 2 Examine the Residuals Plot.

The residuals plot shows the difference between simulated response and measured data. When there is a good fit between the simulated output and measured data, the residuals show the following behavior:

- Lie within a small percent of the maximum output variation.
- Do not display any systematic patterns.

For example, you can see from the following figure that the residuals for ValidationExp data satisfy both criteria.



See Also

Related Examples

- “Estimate Parameters and States” on page 2-26

More About

- “What Is an Experiment?” on page 2-3

Speed Up Parameter Estimation Using Parallel Computing

In this section...

“When to Use Parallel Computing for Parameter Estimation” on page 2-39

“How Parallel Computing Speeds Up Estimation” on page 2-39

When to Use Parallel Computing for Parameter Estimation

You can use Simulink Design Optimization software with Parallel Computing Toolbox™ software to speed up parameter estimation of Simulink models. Using parallel computing may reduce the estimation time in the following cases:

- The model contains a large number parameters to estimate, and the estimation method is specified as either `Nonlinear least squares` or `Gradient descent`.
- The `Pattern search` method is selected as the estimation method.
- The model is complex and takes a long time to simulate.

When you use parallel computing, the software distributes independent simulations to run them in parallel on multiple MATLAB sessions, also known as *workers*. The time required to simulate the model dominates the total estimation time. Therefore, distributing the simulations significantly reduces the estimation time.

For information on how the software distributes the simulations and the expected speedup, see “How Parallel Computing Speeds Up Estimation” on page 2-39.

For information on configuring your system and using parallel computing, see “Use Parallel Computing for Parameter Estimation” on page 2-43.

How Parallel Computing Speeds Up Estimation

You can enable parallel computing with the `Nonlinear least squares`, `Gradient descent` and `Pattern search` estimation methods. The following sections describes the potential speedup using parallel computing for estimation:

- “Parallel Computing with Nonlinear least squares and Gradient descent Methods” on page 2-40
- “Parallel Computing with the Pattern search Method” on page 2-41

Parallel Computing with Nonlinear least squares and Gradient descent Methods

When you select `Gradient descent` as the estimation method, the model is simulated during the following computations:

- Objective value computation — One simulation per iteration
- Objective gradient computations — Two simulations for every tuned parameter per iteration
- Line search computations — Multiple simulations per iteration

The total time, T_{total} , taken per iteration to perform these simulations is given by the following equation:

$$T_{total} = T + (N_p \times 2 \times T) + (N_{ls} \times T) = T \times (1 + (2 \times N_p) + N_{ls})$$

where T is the time taken to simulate the model and is assumed to be equal for all simulations, N_p is the number of parameters to estimate, and N_{ls} is the number of line searches. N_{ls} is difficult to estimate and you generally assume it to be equal to one, two, or three.

When you use parallel computing, the software distributes the simulations required for objective gradient computations. The simulation time taken per iteration when the gradient computations are performed in parallel, T_{totalP} , is approximately given by the following equation:

$$T_{totalP} = T + \left(\text{ceil} \left(\frac{N_p}{N_w} \right) \times 2 \times T \right) + (N_{ls} \times T) = T \times \left(1 + 2 \times \text{ceil} \left(\frac{N_p}{N_w} \right) + N_{ls} \right)$$

where N_w is the number of MATLAB workers.

Note The equation does not include the time overheads associated with configuring the system for parallel computing and loading Simulink software on the remote MATLAB workers.

The expected reduction of the total estimation time is given by the following equation:

$$\frac{T_{totalP}}{T_{total}} = \frac{1 + 2 \times \text{ceil} \left(\frac{N_p}{N_w} \right) + N_{ls}}{1 + (2 \times N_p) + N_{ls}}$$

For example, for a model with $N_p=3$, $N_w=4$, and $N_{1s}=3$, the expected reduction of the total

$$\text{estimation time equals } \frac{1 + 2 \times \text{ceil}\left(\frac{3}{4}\right) + 3}{1 + (2 \times 3) + 3} = 0.6 .$$

Parallel Computing with the Pattern search Method

The `Pattern search` method uses search and poll sets to create and compute a set of candidate solutions at each estimation iteration.

The total time, T_{total} , taken per iteration to perform these simulations, is given by the following equation:

$$T_{total} = (T \times N_p \times N_{ss}) + (T \times N_p \times N_{ps}) = T \times N_p \times (N_{ss} + N_{ps})$$

where T is the time taken to simulate the model and is assumed to be equal for all simulations, N_p is the number of parameters to estimate, N_{ss} is a factor for the search set size, and N_{ps} is a factor for the poll set size. N_{ss} and N_{ps} are typically proportional to N_p .

When you use parallel computing, Simulink Design Optimization software distributes the simulations required for the search and poll set computations, which are evaluated in separate `parfor` loops. The simulation time taken per iteration when the search and poll sets are computed in parallel, T_{totalP} , is given by the following equation:

$$\begin{aligned} T_{totalP} &= (T \times \text{ceil}(N_p \times \frac{N_{ss}}{N_w})) + (T \times \text{ceil}(N_p \times \frac{N_{ps}}{N_w})) \\ &= T \times (\text{ceil}(N_p \times \frac{N_{ss}}{N_w}) + \text{ceil}(N_p \times \frac{N_{ps}}{N_w})) \end{aligned}$$

where N_w is the number of MATLAB workers.

Note The equation does not include the time overheads associated with configuring the system for parallel computing and loading Simulink software on the remote MATLAB workers.

The expected speed up for the total estimation time is given by the following equation:

$$\frac{T_{totalIP}}{T_{total}} = \frac{\text{ceil}(N_p \times \frac{N_{ss}}{N_w}) + \text{ceil}(N_p \times \frac{N_{ps}}{N_w})}{N_p \times (N_{ss} + N_{ps})}$$

For example, for a model with $N_p=3$, $N_w=4$, $N_{ss}=15$, and $N_{ps}=2$, the expected speedup

$$\text{equals } \frac{\text{ceil}(3 \times \frac{15}{4}) + \text{ceil}(3 \times \frac{2}{4})}{3 \times (15 + 2)} = 0.27 .$$

Using the `Pattern search` method with parallel computing may not speed up the estimation time. When you do not use parallel computing, the method stops searching for a candidate solution at each iteration as soon as it finds a solution better than the current solution. When you use parallel computing, the candidate solution search is more comprehensive. Although the number of iterations may be larger, the estimation without using parallel computing may be faster.

See Also

Related Examples

- “Use Parallel Computing for Parameter Estimation” on page 2-43

Use Parallel Computing for Parameter Estimation

In this section...

“Configure Your System for Parallel Computing” on page 2-43

“Model Dependencies” on page 2-43

“Estimate Parameters Using Parallel Computing in the Parameter Estimation Tool” on page 2-44

“Estimate Parameters Using Parallel Computing (Code)” on page 2-47

“Troubleshooting” on page 2-48

Configure Your System for Parallel Computing

You can speed up parameter estimation using parallel computing on multicore processors or multiprocessor networks. Use parallel computing with the Parameter Estimation tool and `sdo.optimize` to estimate parameters using the `fmincon`, `lsqonlin`, and `patternsearch` methods. Parallel computing is not supported for the `fminsearch` (**Simplex search**) method.

When you estimate model parameters using parallel computing, the software uses the available parallel pool. If none is available, and you select **Automatically create a parallel pool** in your Parallel Computing Toolbox preferences, the software starts a parallel pool using the settings in those preferences. To open a parallel pool that uses a specific cluster profile, use:

```
parpool(MyProfile);
```

`MyProfile` is the name of a cluster profile.

For information regarding creating a cluster profile, see “Create and Modify Cluster Profiles” (Parallel Computing Toolbox).

Model Dependencies

Model dependencies are any referenced models, data such as model variables, S-functions, and additional files necessary to run the model. Before starting the optimization, verify that the model dependencies are complete. Otherwise, you may get unexpected results.

Making Model Dependencies Accessible to Remote Workers

When you use parallel computing, the Simulink Design Optimization software helps you identify model dependencies. To do so, the software uses the Simulink Manifest Tools. The dependency analysis may not find all the files required by your model. To learn more, see “Scope of Dependency Analysis” (Simulink). If your model has dependencies that are undetected or inaccessible by the parallel pool workers, then add them to the list of model dependencies.

The dependencies 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.
- Path dependencies: the paths to the model dependencies are added to the paths of the parallel pool workers. If you are working in a multi-platform scenario, ensure that the paths are compatible across platforms.

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 dependent files on the local computer.

For more information, see:

- “Estimate Parameters Using Parallel Computing in the Parameter Estimation Tool” on page 2-44
- “Estimate Parameters Using Parallel Computing (Code)” on page 2-47

Estimate Parameters Using Parallel Computing in the Parameter Estimation Tool

To estimate model parameters using parallel computing in the Parameter Estimation tool:


- 1 Ensure that the software can access parallel pool workers that use the appropriate cluster profile.

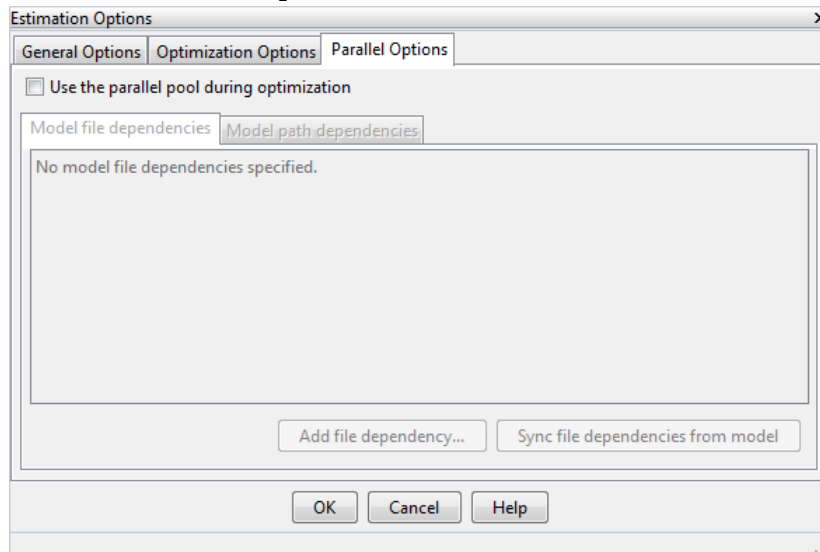
For more information, see “Configure Your System for Parallel Computing” on page 2-43.

- 2 Open the Parameter Estimation tool for the Simulink model.

- 3 Configure the estimation data, estimation parameters and states, and, optionally, estimation settings.

For more information, see “Specify Estimation Data” on page 2-4, “Specify Parameters for Estimation” on page 2-9, and “Specify Estimation Options” on page 2-23.

- 4 On the **Parameter Estimation** tab, click  **More Options** to open the **Estimation Options** dialog box.
- 5 Select the **Parallel Options** tab.



- 6 Select the **Use the parallel pool during optimization** check box.

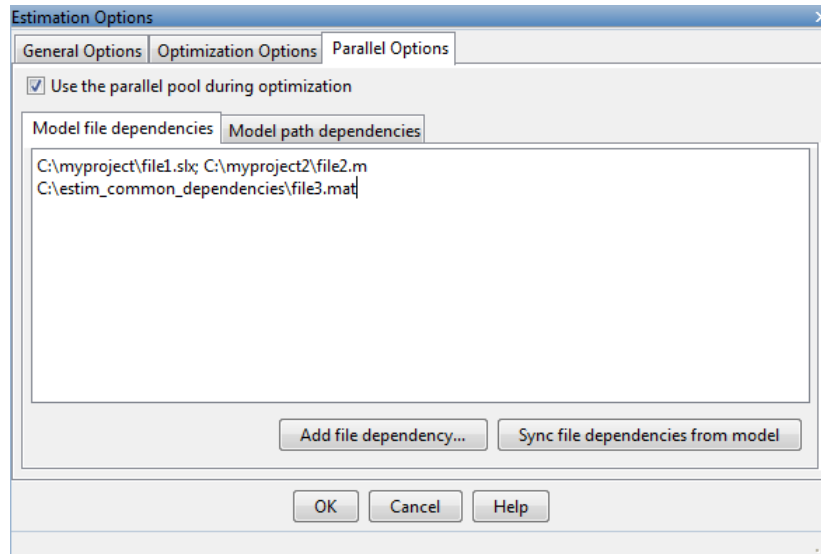
This option checks for dependencies in your Simulink model. The file dependencies are displayed in the **Model file dependencies** list box, and corresponding path to the files in **Model path dependencies**. The files listed in **Model file dependencies** are copied to the remote workers.

Note The automatic dependencies check may not detect all the dependencies in your model.

For more information, see “Model Dependencies” on page 2-43. In this case, add the undetected dependencies manually.

- 7 Add any file dependencies that the automatic check does not detect.

Specify the files in the **Model file dependencies** list box separated by semicolons or on separate lines.



Alternatively, click **Add file dependency** to open a dialog box, and select the file to add.

Note If you do not want to copy the files to the remote workers, delete all entries in the **Model file dependencies** list box. Populate the **Model path dependencies** list box by clicking the **Sync path dependencies from model**, and add any undetected path dependencies. In addition, in the list box, update the paths on local drives to make them accessible to remote workers. For example, change `C:\` to `\\hostname\C$`.

- 8 If you modify the Simulink model, resync the dependencies to ensure that any new dependencies are detected. Click **Sync file dependencies from model** in the **Parallel Options** tab to rerun the automatic dependency check for your model.

This action updates the **Model file dependencies** list box with any new file dependency found in the model.

- 9 Click **OK**.

- 10 In the **Parameter Estimation** tab, click **Estimate** to estimate the model parameters using parallel computing.

For information on troubleshooting problems related to estimation using parallel computing, see “Troubleshooting” on page 2-48.

Estimate Parameters Using Parallel Computing (Code)

To use parallel computing for parameter estimation at the command line:

- 1 Ensure that the software can access parallel pool workers that use the appropriate cluster profile.

For more information, see “Configure Your System for Parallel Computing” on page 2-43.

- 2 Open the model.
- 3 Configure an estimation experiment. For example, see “Estimate Model Parameter Values (Code)” on page 2-76.
- 4 Enable parallel computing using an optimization option set, `opt`.

```
opt = sdo.OptimizeOptions;  
opt.UseParallel = true;
```

- 5 Find the model dependencies.

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

Note `sdo.getModelDependencies` may not detect all the dependencies in your model. For more information, see “Model Dependencies” on page 2-43. In this case, add the undetected dependencies manually.

- 6 Modify files to include any file dependencies that `sdo.getModelDependencies` does not detect.

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

Note If you do not want to copy the files to the remote workers, use the path dependencies. Add any undetected path dependencies to `dirs` and update the paths on local drives to make them accessible to remote workers. See `sdo.getModelDependencies` for more details.

- 7 Add the file dependencies for optimization.

```
opt.ParallelFileDependencies = files;
```

- 8 Run the optimization.

```
[pOpt,opt_info] = sdo.optimize(opt_fcn,param,opt);
```

For information on troubleshooting problems related to estimation using parallel computing, see “Troubleshooting” on page 2-48.

Troubleshooting

- “Why Are the Estimation Results With and Without Parallel Computing Different?” on page 2-48
- “Why Didn’t the Estimation Speed up Using Parallel Computing?” on page 2-48
- “Why Doesn’t the Estimation Using Parallel Computing Make Any Progress?” on page 2-49
- “Why Does the Estimation Using Parallel Computing Continue When I Click Stop?” on page 2-49

Why Are the Estimation Results With and Without Parallel Computing Different?

- Different numerical precision on the client and worker machines can produce marginally different simulation results. Thus, the optimization method can take a different solution path and produce a different result.
- When you use parallel computing with the `Pattern` search method, the search is more comprehensive and can result in a different solution. To learn more, see “Parallel Computing with the Pattern search Method” on page 2-41.

Why Didn’t the Estimation Speed up Using Parallel Computing?

- When you estimate a few parameters or when the model does not take long to simulate, you do not see a speedup in the estimation time. In such cases, the overhead associated with creating and distributing the parallel tasks outweighs the benefits of running the estimation in parallel.
- Using the `Pattern` search method with parallel computing might not speed up the optimization time. Without parallel computing, the method stops the search at each iteration as soon as it finds a solution better than the current solution. The candidate solution search is more comprehensive when you use parallel computing. Although

the number of iterations might be larger, the optimization without using parallel computing might be faster.

To learn more about the expected speedup, see “Parallel Computing with the Pattern search Method” on page 2-41.

Why Doesn't the Estimation Using Parallel Computing Make Any Progress?

To troubleshoot the problem:

- 1 Run the optimization for a few iterations without parallel computing to see if the optimization progresses.
- 2 Check whether the remote workers have access to all model dependencies. Model dependencies include data variables and files required by the model to run.

To learn more, see “Model Dependencies” on page 2-43.

Why Does the Estimation Using Parallel Computing Continue When I Click Stop?

When you use parallel computing with the `Pattern search` method, the software must wait until the current optimization iteration completes before it notifies the workers to stop. The optimization does not terminate immediately when you click **Stop**, and, instead, appears to continue running.

See Also

`parpool` | `sdo.OptimizeOptions` | `sdo.getModelDependencies` | `sdo.optimize`

More About

- “Speed Up Parameter Estimation Using Parallel Computing” on page 2-39
- “Ways to Speed Up Design Optimization Tasks”

Estimating Initial Conditions for Blocks with External Initial Conditions

When an integrator block uses an initial-condition port, which you specify by an IC block, you cannot estimate the initial conditions (ICs) of the integrator using Simulink Design Optimization software. Estimation is not possible because external ICs have priority over the ICs of a specific block to maintain the integrity of the model.

To tune the ICs of an integrator block with external ICs, you must modify the model to make the external signal into a tunable parameter. For example, you can set the IC block that feeds into the integrator to be a tunable variable and estimate it.

See Also

Save and Load Estimation Sessions

In this section...

- “Structure of an Estimation Session” on page 2-51
- “Save Parameter Estimation Tool Sessions” on page 2-51
- “Load Parameter Estimation Tool Sessions” on page 2-52
- “Load Legacy Projects” on page 2-52

This topic shows how to save and load estimation sessions in the Parameter Estimation tool.

Structure of an Estimation Session

The Parameter Estimation tool stores and organizes data from a given Simulink model inside a *session*. An estimation session includes the following information:

- One or more estimation or validation experiments along with their configurations
- Parameter information
- Estimation results
- Estimation settings
- Plots — Changes to plots layout and plot characteristics, such as axis limits, line colors, are not included.

The default session name is the same as the Simulink model name. The session name is shown on the title pane of the Parameter Estimation tool.

Save Parameter Estimation Tool Sessions

Saving a session lets you reuse your estimation settings and results later. You can save the session as a MAT-file or workspace variable:

- To save the session as a MAT-file, in the **Parameter Estimation** tab, in the **Save Session** drop-down list, click **Save to file**. A window opens where you specify the MAT-file name.
- To save the session as a model or MATLAB workspace variable, select **Save to model workspace** or **Save to MATLAB workspace** in the **Save Session** drop-down list.

Load Parameter Estimation Tool Sessions

To load a previously saved MAT-file or workspace sessions:

- 1 Open a Parameter Estimation tool for the model.
- 2 To load a MAT-file, in the **Parameter Estimation** tab click the **Open Session** drop-down list, and select **Open from file**. A window opens where you select the MAT-file to load.

To load a workspace variable, select **Open from model workspace** or **Open from MATLAB workspace** in the **Open Session** drop-down list.

Load Legacy Projects

Open legacy projects that are in MAT-files by selecting `Open from file` from the **Open Session** drop-down list. The Parameter Estimation tool recognizes and converts them into the new session format.

See Also

More About

- “What Is an Experiment?” on page 2-3

How the Software Formulates Parameter Estimation as an Optimization Problem

In this section...

“Overview of Parameter Estimation as an Optimization Problem” on page 2-53

“Cost Function” on page 2-53

“Bounds and Constraints” on page 2-55

“Optimization Methods and Problem Formulations” on page 2-56

Overview of Parameter Estimation as an Optimization Problem

When you perform parameter estimation, the software formulates an optimization problem. The optimization problem solution is the estimated parameter values set. This optimization problem consists of:

- x — Design variables. The model parameters and initial states to be estimated.
- $F(x)$ — Objective function. A function that calculates a measure of the difference between the simulated and measured responses. Also called cost function or estimation error.
- (Optional) $\underline{x} \leq x \leq \bar{x}$ — Bounds. Limits on the estimated parameter values.
- (Optional) $C(x)$ — Constraint function. A function that specifies restrictions on the design variables.

The optimization solver tunes the values of the design variables to satisfy the specified objectives and constraints. The exact formulation of the optimization depends on the optimization method that you use.

Cost Function

- “Types” on page 2-54
- “Time Base” on page 2-54

The software tunes the model parameters to obtain a simulated response (y_{sim}) that tracks the measured response or reference signal (y_{ref}). To do so, the solver minimizes the *cost function* or *estimation error*, a measure of the difference between the simulated and

measured responses. The cost function, $F(x)$, is the objective function of the optimization problem.

Types

The raw estimation error, $e(t)$, is defined as:

$$e(t) = y_{ref}(t) - y_{sim}(t)$$

$e(t)$ is also referred to as the error residuals or, simply, residuals.

Simulink Design Optimization software provides you the following cost functions to process $e(t)$:

Cost Function	Formulation	Option Name in GUI or Command Line
Sum squared error (default)	$F(x) = \sum_{t=0}^{t_N} e(t) \times e(t)$ <p>N is the number of samples.</p>	'SSE'
Sum absolute error	$F(x) = \sum_{t=0}^{t_N} e(t) $ <p>N is the number of samples.</p>	'SAE'
Raw error	$F(x) = \begin{bmatrix} e(0) \\ \vdots \\ e(N) \end{bmatrix}$ <p>N is the number of samples.</p>	'Residuals' This option is available only at the command line.
Custom function	N/A	This option is available only at the command line.

Time Base

The software evaluates the cost function for a specific time interval. This interval is dependent on the measured signal time base and the simulated signal time base.

- The measured signal time base consists of all the time points for which the measured signal is specified. In case of multiple measured signals, this time base is the union of the time points of all the measured signals.
- The simulated signal time base consists of all the time points for which the model is simulated.

If the model uses a variable-step solver, then the simulated signal time base can change from one optimization iteration to another. The simulated and measured signal time bases can be different. The software evaluates the cost function for only the time interval that is common to both. By default, the software uses only the time points specified by the measured signal in the common time interval.

- In the GUI, you can specify the simulation start and stop times in the **Simulation time** area of the **Simulation Options** dialog box.
- At the command line, the software specifies the simulation stop time as the last point of the measured signal time base. For example, the following code simulates the model until the end time of the longest running output signal of `exp`, an `sdo.Experiment` object:

```
sim_obj = createSimulator(exp);
sim_obj = sim(sim_obj);
```

`sim_obj` contains the simulated response for the model associated with `exp`.

Bounds and Constraints

You can specify bounds for the design variables (estimated model parameters), based on your knowledge of the system. Bounds are expressed as:

$$\underline{x} \leq x \leq \bar{x}$$

\underline{x} and \bar{x} are the lower and upper bounds for the design variables.

For example, in a battery discharging experiment, the estimated battery initial charge must be greater than zero and less than `Inf`. These bounds are expressed as:

$$0 < x < \infty$$

For an example of how to specify these types of bounds, see “Estimate Model Parameters and Initial States (Code)” on page 2-88.

You can also specify other constraints, $C(x)$, on the design variables at the command line. $C(x)$ can be linear or nonlinear and can describe equalities or inequalities. $C(x)$ can also specify multiparameter constraints. For example, for a simple friction model, $C(x)$ can specify that the static friction coefficient must be greater than or equal to the dynamic friction coefficient. One way of expressing this constraint is:

$$C(x) : x_1 - x_2 \\ C(x) \leq 0$$

x_1 and x_2 are the dynamic and static friction coefficients, respectively.

For an example of how to specify a constraint, see “Estimate Model Parameters with Parameter Constraints (Code)” on page 2-126.

Optimization Methods and Problem Formulations

An optimization problem can be one of the following types:

- **Minimization problem** — Minimizes an objective function, $F(x)$. You specify the measured signal that you want the model output to track. You can optionally specify bounds for the estimated parameters.
- **Mixed minimization and feasibility problem** — Minimizes an objective function, $F(x)$, subject to specified bounds and constraints, $C(x)$. You specify the measured signal that you want the model to track and bounds and constraints for the estimated parameters.
- **Feasibility problem** — Finds a solution that satisfies the specified constraints, $C(x)$. You specify only bounds and constraints for the estimated parameters. This type of problem is not common in parameter estimation.

The optimization method that you specify determines the formulation of the estimation problem. The software provides the following optimization methods:

Optimization Method Name	Description	Optimization Problem Formulation
<ul style="list-style-type: none"> • User interface: Nonlinear Least Squares • Command line: 'lsqnonlin' 	<p>Minimizes the squares of the residuals, recommended method for parameter estimation.</p> <p>This method requires a vector of error residuals, computed using a fixed time base. Do not use this approach if you have a scalar cost function or if the number of error residuals can change from one iteration to another.</p> <p>This method uses the Optimization Toolbox™ function, lsqnonlin.</p>	<p>Minimization Problem</p> $\min_x \ F(x)\ _2^2 = \min_x (f_1(x)^2 + f_2(x)^2 + \dots + f_n(x)^2)$ <p><i>s.t.</i> $\underline{x} \leq x \leq \bar{x}$</p> <p>$f_1(x), f_2(x), \dots, f_n(x)$ represent residuals. n is the number of samples.</p> <p>Mixed Minimization and Feasibility Problem</p> <p>Not supported.</p> <p>Feasibility Problem</p> <p>Not supported.</p>

Optimization Method Name	Description	Optimization Problem Formulation
<ul style="list-style-type: none"> • User interface: Gradient Descent • Command line: 'fmincon' 	<p>General nonlinear solver, uses the cost function gradient.</p> <p>Use this approach if you want to specify one or any combination of the following:</p> <ul style="list-style-type: none"> • Custom cost functions • Parameter-based constraints • Signal-based constraints <p>This method uses the Optimization Toolbox function, <code>fmincon</code>.</p> <p>For information on how the gradient is computed, see “Gradient Computations” on page 2-74.</p>	<p>Minimization Problem</p> $\min_x F(x)$ $s.t. \underline{x} \leq x \leq \bar{x}$ <p>Mixed Minimization and Feasibility Problem</p> $\min_x F(x)$ $s.t. \quad C(x) \leq 0$ $\underline{x} \leq x \leq \bar{x}$ <hr/> <p>Note When tracking a reference signal, the software ignores the maximally feasible solution option.</p> <hr/> <p>Feasibility Problem</p> <ul style="list-style-type: none"> • If you select the maximally feasible solution option (i.e., the optimization continues after an initial feasible solution is found), the software uses the following problem formulation: $\min_{[x,\gamma]} \gamma$ $s.t. \quad C(x) \leq \gamma$ $\underline{x} \leq x \leq \bar{x}$ $\gamma \leq 0$ <p>γ is a slack variable that permits a feasible solution with $C(x) \leq \gamma$ rather than $C(x) \leq 0$.</p> <ul style="list-style-type: none"> • If you do not select the maximally feasible solution option (i.e., the optimization terminates as soon as a feasible solution is

Optimization Method Name	Description	Optimization Problem Formulation
		<p>found), the software uses the following problem formulation:</p> $\begin{aligned} \min_x & 0 \\ \text{s.t. } & C(x) \leq 0 \\ & \underline{x} \leq x \leq \bar{x} \end{aligned}$
<ul style="list-style-type: none"> • User interface: Simplex Search • Command line: 'fminsearch' 	<p>Based on the Nelder-Mead algorithm, this approach does not use the cost function gradient.</p> <p>Use this approach if your cost function or constraints are not continuous or differentiable.</p> <p>This method uses the Optimization Toolbox functions, <code>fminsearch</code> and <code>fminbnd</code>. <code>fminbnd</code> is used if one scalar parameter is being optimized. Otherwise, <code>fminsearch</code> is used. You cannot specify parameter bounds, $\underline{x} \leq x \leq \bar{x}$, with <code>fminsearch</code>.</p>	<p>Minimization Problem</p> $\min_x F(x)$ <p>Mixed Minimization and Feasibility Problem</p> <p>The software formulates the problem in two steps:</p> <ol style="list-style-type: none"> 1 Finds a feasible solution. $\min_x \max(C(x))$ 2 Minimizes the objective. The software uses the results from step 1 as initial guesses. It maintains feasibility by introducing a discontinuous barrier in the optimization objective. $\min_x \Gamma(x)$ where $\Gamma(x) = \begin{cases} \infty & \text{if } \max(C(x)) > 0 \\ F(x) & \text{otherwise.} \end{cases}$ <p>Feasibility Problem</p> $\min_x \max(C(x))$

Optimization Method Name	Description	Optimization Problem Formulation
<ul style="list-style-type: none"> • User interface: Pattern Search • Command line: 'patternsearch' 	<p>Direct search method, based on the generalized pattern search algorithm, this method does not use the cost function gradient.</p> <p>Use this approach if your cost function or constraints are not continuous or differentiable.</p> <p>This method uses the Global Optimization Toolbox function, <code>patternsearch</code>.</p>	<p>Minimization Problem</p> $\min_x F(x)$ $s.t. \quad \underline{x} \leq x \leq \bar{x}$ <p>Mixed Minimization and Feasibility Problem</p> <p>The software formulates the problem in two steps:</p> <ol style="list-style-type: none"> 1 Finds a feasible solution. $\min_x \max(C(x))$ $s.t. \quad \underline{x} \leq x \leq \bar{x}$ 2 Minimizes the objective. The software uses the results from step 1 as initial guesses. It maintains feasibility by introducing a discontinuous barrier in the optimization objective. $\min_x \Gamma(x)$ $s.t. \quad \underline{x} \leq x \leq \bar{x}$ where $\Gamma(x) = \begin{cases} \infty & \text{if } \max(C(x)) > 0 \\ F(x) & \text{otherwise.} \end{cases}$ <p>Feasibility Problem</p> $\min_x \max(C(x))$ $s.t. \quad \underline{x} \leq x \leq \bar{x}$

See Also

`fminbnd` | `fmincon` | `fminsearch` | `lsqnonlin` | `patternsearch` |
`sdo.Experiment` | `sdo.SimulationTest` | `sdo.requirements.SignalTracking` |
`sdo.requirements.SignalTracking`

Related Examples

- “Estimate Model Parameter Values (Code)” on page 2-76
- “Estimate Model Parameters with Parameter Constraints (Code)” on page 2-126
- “Estimate Parameters from Measured Data”

More About

- “Write a Cost Function” on page 2-62

Write a Cost Function

A cost function is a MATLAB function that evaluates your design requirements using design variable values. After writing and saving the cost function, you can use it for estimation, optimization, or sensitivity analysis at the command line.

When you optimize or estimate model parameters, you provide the saved cost function as an input to `sdo.optimize`. At every optimization iteration, `sdo.optimize` calls this function and uses the function output to decide the optimization direction. When you perform sensitivity analysis using `sdo.evaluate`, you generate sample values of the design variables and evaluate the cost function for each sample value using `sdo.evaluate`.

Anatomy of a Cost Function

To understand the parts of a cost function, consider the following sample function `myCostFunc`. For a design variable x , `myCostFunc` evaluates the objective x^2 and the nonlinearity constraint $x^2-4x+1 \leq 0$.

```
function [vals,derivs] = myCostFunc(params)
% Extract the current design variable values from the parameter object, params.
x = params.Value;
% Compute the requirements (objective and constraint violations) and assign them to vals
vals.F = x.^2;
vals.Cleq = x.^2-4*x+1;
% Compute the cost and constraint derivatives.
derivs.F = 2*x;
derivs.Cleq = 2*x-4;
end
```

This cost function performs the following tasks:

- 1 Specifies the inputs of the cost function.

A cost function must have as input, `params`, a vector of the design variables to be estimated, optimized, or used for sensitivity analysis. Design variables are model parameter objects (`param.Continuous` objects) or model initial states (`param.State` objects).

Since the cost function is called repeatedly during estimation, optimization, or evaluation, you can specify additional inputs to the cost function to help reduce code

redundancy and computation cost. For more information, see “Specify Inputs of the Cost Function” on page 2-63.

2 Computes the requirements.

Requirements can be objectives and constraints based on model parameters, model signals, or linearized models. In this sample cost function, the requirements are based on the design variable x , a model parameter. The cost function first extracts the current values of the design variables and then computes the requirements.

For information about computing requirements based on model parameters, model signals, or linearized models, see “Compute Requirements” on page 2-64.

3 Specifies the requirement values as outputs, `vals` and `derivs`, of the cost function.

A cost function must return `vals`, a structure with one or more fields that specify the values of the objective and constraint violations.

The output can optionally include `derivs`, a structure with one or more fields that specify the values of the gradients of the objective and constraint violations. For more information, see “Specify Outputs of the Cost Function” on page 2-67.

After saving the cost function as a MATLAB file `myCostFunc.m`, to perform the optimization, use the cost function as an input to `sdo.optimize`.

```
[param_opt,opt_info] = sdo.optimize(@myCostFunc,params)
```

When performing sensitivity analysis, to compute the requirements in the cost function for a range of design variable sample values `paramsamples`, use the cost function as an input to `sdo.evaluate`.

```
[y,info] = sdo.evaluate(@myCostFunc,paramsamples)
```

Specify Inputs of the Cost Function

The sample cost function `myCostFunc` takes one input, `params`.

```
function [vals,derivs] = myCostFunc(params)
```

A cost function must have as input, `params`, a vector of the design variables to be estimated, optimized, or used for sensitivity analysis. Design variables are model parameter objects (`param.Continuous` objects) or model initial states (`param.State`

objects). You obtain params by using the `sdo.getParameterFromModel` and `sdo.getStateFromModel` commands.

Specify Multiple Inputs

Because the cost function is called repeatedly during estimation, optimization, or evaluation, you can specify additional inputs to the cost function to help reduce code redundancy and computation cost. However, `sdo.optimize` and `sdo.evaluate` accept a cost function with only one input argument. To use a cost function that accepts more than one input argument, you use an anonymous function. Suppose that the `myCostFunc_multi_inputs.m` file specifies a cost function that takes `params` and `arg1` as inputs. For example, you can make the model name an input argument, `arg1`, and configure the cost function to be used for multiple models. Then, assuming that all input arguments are variables in the workspace, specify an anonymous function `myCostFunc2`, and use it as an input to `sdo.optimize` or `sdo.evaluate`.

```
myCostFunc2 = @(params) myCostFunc_multi_inputs(params, arg1);  
[param_opt, opt_info] = sdo.optimize(@myCostFunc2, params);
```

You can also specify additional inputs using convenience objects provided by Simulink Design Optimization software. You create convenience objects once and pass them as an input to the cost function to reduce code redundancy and computation cost.

For example, you can create a simulator (`sdo.SimulationTest` object) to simulate your model using alternative model parameters without modifying the model, and pass the simulator to your cost function.

```
simulator = sdo.SimulationTest(model)  
myCostFunc2 = @(params) myCostFunc_mult_inputs(params, arg1, arg2, simulator);  
[param_opt, opt_info] = sdo.optimize(@myCostFunc2, params);
```

For more information about the available convenience objects, see “Convenience Objects as Additional Inputs” on page 2-69. For an example, see “Design Optimization to Meet a Custom Objective (Code)” on page 3-133.

Compute Requirements

The sample cost function `myCostFunc` computes the requirements based on a model parameter `x`. In general, requirements can be objectives or constraints based on model parameters, model signals, or linearized models. As seen in `myCostFunc`, you can use MATLAB functions to compute the requirements. You can also use the requirements

objects that Simulink Design Optimization software provides. These objects enable you to specify requirements such as step-response characteristics, gain and phase margin bounds, and Bode magnitude bounds. You can use the `evalRequirement` method of these objects to evaluate the objective and constraint violations. For a list of available requirement objects, see “Convenience Objects as Additional Inputs” on page 2-69.

Parameter-Based Requirements

If you have requirements on model parameters, in the cost function you first extract the current parameter values, and then compute the requirements.

- 1 Extract the current parameter value from `params`.

```
x = params.Value;
```

- 2 Compute the requirement, and specify it as `vals`, the output of the cost function.

Suppose that the objective to be computed is x^2 and the constraint is the nonlinearity constraint x^2-4x+1 .

```
vals.F = x.^2;
vals.Cleq = x.^2-4*x+1;
```

In the context of optimization, x^2 is minimized subject to satisfying the constraints. For sensitivity analysis, the cost and constraints are evaluated for all values of the parameter `params`.

For more information about the output of a cost function, see “Specify Outputs of the Cost Function” on page 2-67.

For an example of a cost function with a parameter-based requirement, see “Design Optimization to Meet a Custom Objective (Code)” on page 3-133. In this example, you minimize the cylinder cross-sectional area, a design variable in a hydraulic cylinder.

Model Signal Requirements

If you have requirements on model signals, in the cost function you simulate the model using current design variable values, extract the signal of interest, and compute the requirement on the signal.

- 1 Simulate the model using the current design variable values in `param`. There are multiple ways to simulate your model:

- **Using `sdo.SimulationTest` object** — If an `sdo.SimulationTest` object, `simulator`, is a cost function input, you update the model parameter values using the `Parameters` property of the simulator. Then use `sim` to simulate the model.

```
simulator.Parameters = params;  
simulator = sim(simulator);
```

For an example, see “Design Optimization to Meet a Custom Objective (Code)” on page 3-133.

- **Using `sdo.Experiment` object** — If you are performing parameter estimation based on input-output data defined in an `sdo.Experiment` object, `exp`, update the design variable values associated with the experiment using the `setEstimatedValues` method. Create a simulator using the `createSimulator` method, and simulate the model using the updated model configuration.

```
exp = setEstimatedValues(exp,params);  
simulator = createSimulator(exp,simulator);  
simulator = sim(simulator);
```

For an example, see “Estimate Model Parameters Per Experiment (Code)” on page 2-112.

- **Using `sim` command** — If you are not using `sdo.SimulationTest` or `sdo.Experiment` objects, use `sdo.setValueInModel` to update the model parameter values, and then call `sim` to simulate the model.

```
sdo.setValueInModel('model_name',param);  
LoggedData = sim('model_name');
```

- 2 Extract the logged signal of interest, `SignalOfInterest`.

Use the `SignalLoggingName` model parameter to get the simulation log name.

```
logName = get_param(simulator.ModelName, 'SignalLoggingName');  
simLog = get(simulator.LoggedData, logName);  
Sig = get(simLog, 'SignalOfInterest');
```

- 3 Evaluate the requirement, and specify it as the output of the cost function.

For example, if you specified a step-response bound on a signal using a `sdo.requirements.StepResponseEnvelope` object, `StepResp`, you can use the `evalRequirement` method of the object to evaluate the objective and constraint violations.

```
vals.Cleq = evalRequirement(StepResp, SignalOfInterest.Values);
```

For an example, see “Design Optimization to Meet Step Response Requirements (Code)”. For more information about the output of a cost function, see “Specify Outputs of the Cost Function” on page 2-67.

Linearization-Based Requirements

If you are optimizing or evaluating frequency-domain requirements, in the cost function you linearize the model, and compute the requirement values. Linearizing the model requires Simulink Control Design™ software.

Use the `SystemLoggingInfo` property of `sdo.SimulationTest` to specify linear systems to log when simulating the model. For an example, see “Design Optimization to Meet Frequency-Domain Requirements (Code)” on page 3-251. Alternatively, use `linearize` to linearize the model.

Note For models in Simulink fast restart mode, you cannot use the `linearize` command.

Specify Outputs of the Cost Function

The sample cost function `myCostFunc` outputs `vals`, a structure with fields that specify the values of the objective and constraint violations. The second output is `derivs`, a structure with fields that specify the derivatives of the objective and constraint.

```
function [vals,derivs] = myCostFunc(params)
```

A cost function must output `vals`, a structure with one or more of the following fields that specify the values of the objective and constraint violations:

- `F` — Value of the cost or objective evaluated at `param`.
- `Cleq` — Value of the nonlinear inequality constraint violations evaluated at `param`. For optimization, the solver ensures $Cleq \leq 0$.
- `Ceq` — Value of the nonlinear equality constraint violations evaluated at `param`. For optimization, the solver ensures $Ceq = 0$.
- `leq` — Value of the linear inequality constraint violations evaluated at `param`. For optimization, the solver ensures $leq \leq 0$.

- `eq` — Value of the linear equality constraint violations evaluated at `param`. For optimization, the solver ensures `eq = 0`.
- `Log` — Additional optional information from evaluation.

If you have multiple constraints of one type, concatenate the values into a vector, and specify this vector as the corresponding field value. For instance, if you have a hydraulic cylinder, you can specify nonlinear inequality constraints on the piston position (`Cleq1`) and cylinder pressure (`Cleq2`). In this case, specify the `Cleq` field of the output structure `vals` as:

```
vals.Cleq = [Cleq1; Cleq2];
```

For an example, see “Design Optimization to Meet a Custom Objective (Code)” on page 3-133.

By default, the `sdo.optimize` command computes the objective and constraint gradients using numeric perturbation. You can also optionally return the gradients as an additional cost function output, `derivs`. Where `derivs` must contain the derivatives of all applicable objective and constraint violations and is specified as a structure with one or more of the following fields:

- `F` — Derivatives of the cost or objective.
- `Cleq` — Derivatives of the nonlinear inequality constraints.
- `Ceq` — Derivatives of the nonlinear equality constraints.

The derivatives are not required for sensitivity analysis. For estimation or optimization, specify the `GradFcn` property of `sdo.OptimizeOptions` as `'on'`.

Multiple Objectives

Simulink Design Optimization software does not support multi-objective optimization. However, you can return the objective value (`vals.F`) as a vector that represents the multiple objective values. The software sums the elements of the vector and minimizes this sum. The exception to this behavior is in the use of the nonlinear least squares (`lsqnonlin`) optimization method. The nonlinear least squares method, used for parameter estimation, requires that you return the error residuals as a vector. In this case, the software minimizes the sum square of this vector. If you are tracking multiple signals and using `lsqnonlin`, then concatenate the error residuals for the different signals into one vector. Specify this vector as the `F` field value.

For an example of single-objective optimization using the gradient descent method, see “Design Optimization to Meet a Custom Objective (Code)” on page 3-133.

For an example of multiple-objective optimization using the nonlinear least squares method, see “Estimate Model Parameters Per Experiment (Code)” on page 2-112.

Convenience Objects as Additional Inputs

A cost function must have as input, `params`, a vector of the design variables to be estimated, optimized, or used for sensitivity analysis. You can specify additional inputs to the cost function using convenience objects provided by the Simulink Design Optimization software. You create convenience objects once and pass them as an input to the cost function to reduce code redundancy and computation cost. For information about specifying additional inputs to the cost function, see “Specify Multiple Inputs” on page 2-64.

Convenience Object	Class Name	Description
<p>Simulator objects</p>	<p><code>sdo.SimulationTest</code></p>	<p>Use the simulator object to simulate the model using alternative inputs, model parameters, and initial-state values without modifying the model. Use the <code>SystemLoggingInfo</code> property of <code>sdo.SimulationTest</code> to specify linear systems to log when you have frequency-domain requirements.</p> <p>In the cost function, use the <code>sim</code> method to simulate the model. Then extract the model response from the object, and evaluate the requirements.</p> <p>For an example, see “Design Optimization to Meet a Custom Objective (Code)” on page 3-133.</p> <hr/> <p>Note To perform estimation, optimization, or evaluation using Simulink fast restart, it is necessary to create the simulator before the cost function, and then pass the simulator to the cost function.</p>

Convenience Object	Class Name	Description
Requirements objects	<p>Available Requirements Objects</p> <p>Time-domain requirements:</p> <ul style="list-style-type: none"> • <code>sdo.requirements.SignalBound</code> • <code>sdo.requirements.StepResponseEnvelope</code> • <code>sdo.requirements.SignalTracking</code> • <code>sdo.requirements.PhasePlaneEllipse</code> • <code>sdo.requirements.PhasePlaneRegion</code> <p>Parameter requirements:</p> <ul style="list-style-type: none"> • <code>sdo.requirements.FunctionMatching</code> • <code>sdo.requirements.MonotonicVariable</code> • <code>sdo.requirements.RelationalConstraint</code> • <code>sdo.requirements.SmoothnessConstraint</code> <p>Frequency-domain requirements:</p> <ul style="list-style-type: none"> • <code>sdo.requirements.GainPhaseMargin</code> • <code>sdo.requirements.BodeMagnitude</code> • <code>sdo.requirements.ClosedLoopPeakGain</code> 	<p>Use these objects to specify time-domain and frequency-domain costs or constraints that depend on the design variable values.</p> <p>In the cost function, use the <code>evalRequirement</code> method of the object to evaluate how closely the current design variables satisfy your design requirement.</p> <p>For an example, see “Design Optimization to Meet Step Response Requirements (Code)”.</p>

Convenience Object	Class Name	Description
	<ul style="list-style-type: none"> • <code>sdo.requirements.PZDampingRatio</code> • <code>sdo.requirements.PZNaturalFrequency</code> • <code>sdo.requirements.PZSettlingTime</code> • <code>sdo.requirements.SignalTracking</code> • <code>sdo.requirements.StepResponseEnvelope</code> • <code>sdo.requirements.OpenLoopGainPhase</code> 	
Experiment objects	<code>sdo.Experiment</code>	<p>Use an experiment object to specify the input-output data, model parameters, and initial-state values for parameter estimation.</p> <p>In the cost function, update the design variable values associated with the experiment using the <code>setEstimatedValues</code> method. Then, to simulate the model using the updated model configuration, create a simulator using the <code>createSimulator</code> method.</p> <p>For an example, see “Estimate Model Parameters Per Experiment (Code)” on page 2-112.</p>

See Also

`param.Continuous` | `sdo.Experiment` | `sdo.OptimizeOptions` |
`sdo.SimulationTest` | `sdo.evaluate` | `sdo.optimize` | `sdo.setValueInModel`

Related Examples

- “How the Optimization Algorithm Formulates Minimization Problems” on page 3-3
- “Design Optimization to Meet a Custom Objective (Code)” on page 3-133
- “How the Software Formulates Parameter Estimation as an Optimization Problem” on page 2-53
- “Estimate Model Parameter Values (Code)” on page 2-76
- “What is Sensitivity Analysis?” on page 4-2
- “Identify Key Parameters for Estimation (Code)” on page 4-217

Gradient Computations

For the Gradient descent (`fmincon`) optimization solver, the gradients are computed using numerical perturbation:

$$dx = \sqrt[3]{eps} \times \max\left(|x|, \frac{1}{10} x_{typical}\right)$$

$$dL = \max(x - dx, x_{min})$$

$$dR = \min(x + dx, x_{max})$$

$$F_L = opt_fcn(dL)$$

$$F_R = opt_fcn(dR)$$

$$\frac{dF}{dx} = \frac{(F_L - F_R)}{(dL - dR)}$$

- x is a scalar design variable.
- x_{min} is the lower bound of x .
- x_{max} is the upper bound of x .
- $x_{typical}$ is the scaled value of x .
- opt_fcn is the objective function.

dx is relatively large to accommodate simulation solver tolerances.

If you want to compute the gradients in any other way, you can do so in the cost function you write for performing design optimization programmatically. See `sdo.optimize` and `GradFcn` of `sdo.OptimizeOptions` for more information.

See Also

`fmincon`

More About

- “How the Software Formulates Parameter Estimation as an Optimization Problem” on page 2-53

- “How the Optimization Algorithm Formulates Minimization Problems” on page 3-3

Estimate Model Parameter Values (Code)

This example shows how to use experimental data to estimate model parameter values.

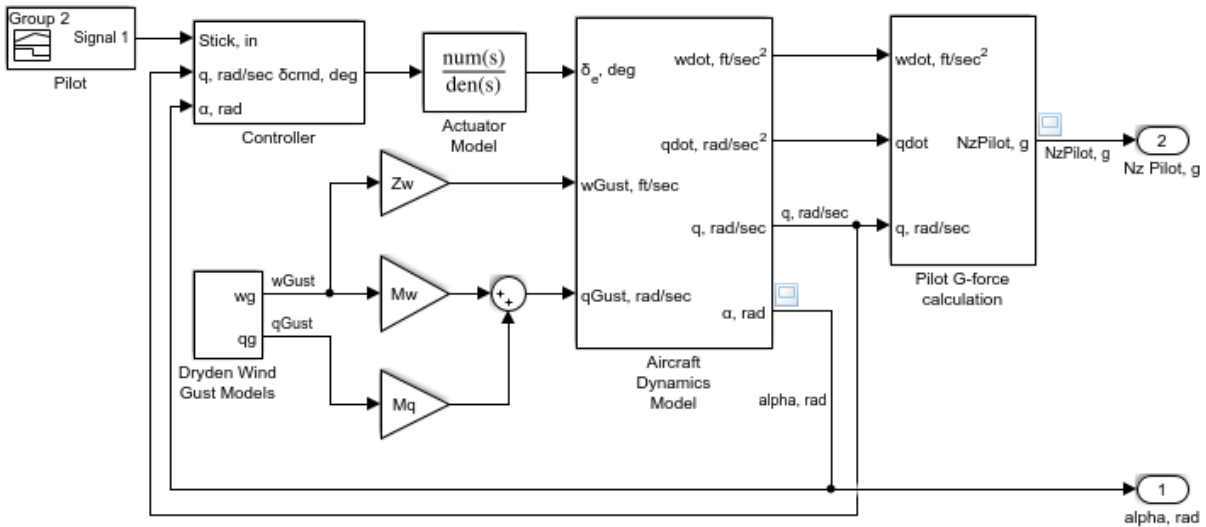
Aircraft Model

The Simulink model, `sdoAircraftEstimation`, models the longitudinal flight control system of an aircraft.

```
open_system('sdoAircraftEstimation')
```

Aircraft Longitudinal Flight Control

This demonstration models a flight algorithm of an aircraft.



Copyright 1990-2012 The MathWorks, Inc.

Estimation Problem

You use measured data to estimate the aircraft model parameters and states.

Measured output data:

- Pilot G force, output of the Pilot G-force calculation block
- Angle of attack, fourth output of the Aircraft Dynamics Model block

Parameters:

- Actuator time constant, T_a , used by the Actuator Model block
- Vertical velocity, Z_d , used by the Aircraft Dynamics Model block
- Pitch rate gains, M_d , used by the Aircraft Dynamics Model block

State:

- Initial state of the first-order actuator model, `sdoAircraftEstimation/Actuator Model`

Define the Estimation Experiment

Get the measured data.

```
[time,iodata] = sdoAircraftEstimation_Experiment;
```

The `sdoAircraftEstimation_Experiment` function returns the measured output data, `iodata`, and the corresponding time vector. The first column of `iodata` is the pilot G force and the second column is the angle of attack.

To see the code for this function, type `edit sdoAircraftEstimation_Experiment`.

Create an experiment object to store the measured input/output data.

```
Exp = sdo.Experiment('sdoAircraftEstimation');
```

Create an object to store the measured pilot G-Force output.

```
PilotG = Simulink.SimulationData.Signal;
PilotG.Name = 'PilotG';
PilotG.BlockPath = 'sdoAircraftEstimation/Pilot G-force calculation';
PilotG.PortType = 'outport';
PilotG.PortIndex = 1;
PilotG.Values = timeseries(iodata(:,2),time);
```

Create an object to store the measured angle of attack (alpha) output.

```
AoA = Simulink.SimulationData.Signal;
AoA.Name = 'AngleOfAttack';
AoA.BlockPath = 'sdoAircraftEstimation/Aircraft Dynamics Model';
AoA.PortType = 'outport';
AoA.PortIndex = 4;
AoA.Values = timeseries(iodata(:,1),time);
```

Add the measured pilot G-Force and angle of attack data to the experiment as the expected output data.

```
Exp.OutputData = [...
    PilotG; ...
    AoA];
```

Add the initial state for the `Actuator Model` block to the experiment. Set its `Free` field to true so that it is estimated.

```
Exp.InitialStates = sdo.getStateFromModel('sdoAircraftEstimation','Actuator Model');
Exp.InitialStates.Minimum = 0;
Exp.InitialStates.Free = true;
```

Compare the Measured Output and the Initial Simulated Output

Create a simulation scenario using the experiment and obtain the simulated output.

```
Simulator = createSimulator(Exp);
Simulator = sim(Simulator);
```

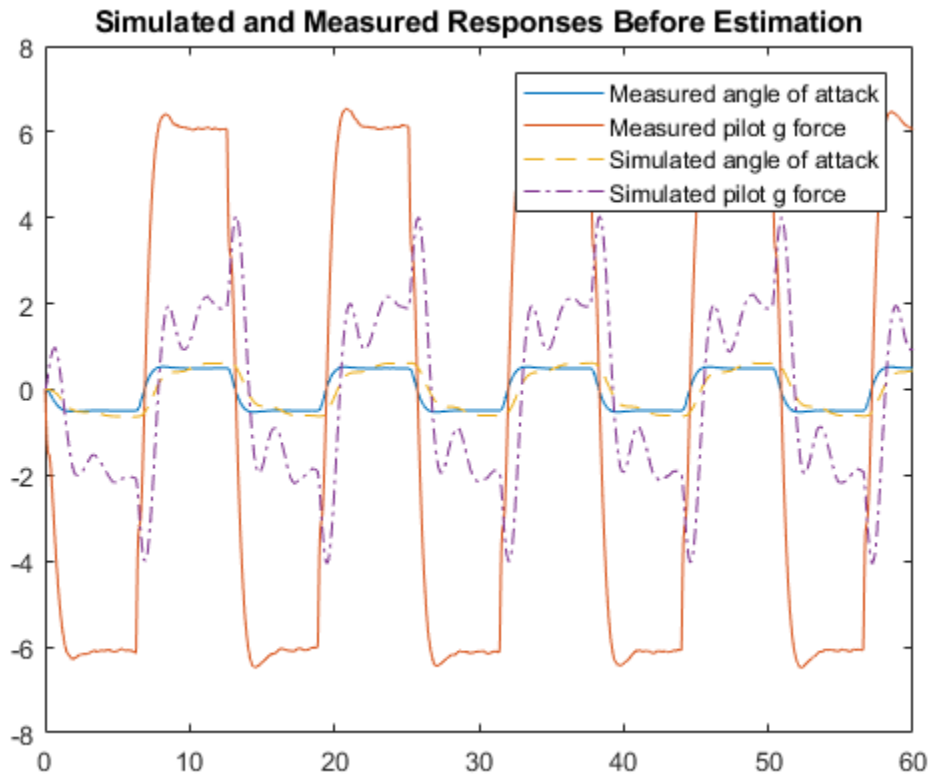
Search for the pilot G-Force and angle of attack signals in the logged simulation data.

```
SimLog = find(Simulator.LoggedData,get_param('sdoAircraftEstimation','SignalLoggi
PilotGSignal = find(SimLog,'PilotG');
AoASignal = find(SimLog,'AngleOfAttack');
```

Plot the measured and simulated data.

As expected, the model response does not match the experimental output data.

```
plot(time, iodata, ...
    AoASignal.Values.Time,AoASignal.Values.Data,'--', ...
    PilotGSignal.Values.Time,PilotGSignal.Values.Data,'-.');
title('Simulated and Measured Responses Before Estimation')
legend('Measured angle of attack', 'Measured pilot g force', ...
    'Simulated angle of attack', 'Simulated pilot g force');
```

Specify the Parameters to Estimate

Select the model parameters that describe the flight control actuation system. Specify bounds for the estimated parameter values based on our understanding of the actuation system.

```
p = sdo.getParameterFromModel('sdoAircraftEstimation', {'Ta', 'Md', 'Zd'});
p(1).Minimum = 0.01; %Ta
p(1).Maximum = 1;
p(2).Minimum = -10; %Md
p(2).Maximum = 0;
p(3).Minimum = -100; %Zd
p(3).Maximum = 0;
```

Get the actuator initial state value that is to be estimated from the experiment.

```
s = getValuesToEstimate(Exp);
```

Group the model parameters and initial states to be estimated together.

```
v = [p;s]
```

```
v(1,1) =
```

```
    Name: 'Ta'  
    Value: 0.5000  
  Minimum: 0.0100  
  Maximum: 1  
    Free: 1  
    Scale: 0.5000  
    Info: [1x1 struct]
```

```
v(2,1) =
```

```
    Name: 'Md'  
    Value: -1  
  Minimum: -10  
  Maximum: 0  
    Free: 1  
    Scale: 1  
    Info: [1x1 struct]
```

```
v(3,1) =
```

```
    Name: 'Zd'  
    Value: -80  
  Minimum: -100  
  Maximum: 0  
    Free: 1  
    Scale: 128  
    Info: [1x1 struct]
```

```
v(4,1) =
```

```
    Name: 'sdoAircraftEstimation/Actuator...'
```

```

    Value: 0
  Minimum: 0
  Maximum: Inf
    Free: 1
    Scale: 1
  dxValue: 0
  dxFree: 1
    Info: [1x1 struct]

```

```
4x1 param.Continuous
```

Define the Estimation Objective Function

Create an estimation objective function to evaluate how closely the simulation output, generated using the estimated parameter values, matches the measured data.

Use an anonymous function with one input argument that calls the `sdoAircraftEstimation_Objective` function. We pass the anonymous function to `sdo.optimize`, which evaluates the function at each optimization iteration.

```
estFcn = @(v) sdoAircraftEstimation_Objective(v, Simulator, Exp);
```

The `sdoAircraftEstimation_Objective` function:

- Has one input argument that specifies the actuator parameter values and the actuator initial state.
- Has one input argument that specifies the experiment object containing the measured data.
- Returns a vector of errors between simulated and experimental outputs.

The `sdoAircraftEstimation_Objective` function requires two inputs, but `sdo.optimize` requires a function with one input argument. To work around this, `estFcn` is an anonymous function with one input argument, `v`, but it calls `sdoAircraftEstimation_Objective` using two input arguments, `v` and `Exp`.

For more information regarding anonymous functions, see "Anonymous Functions".

The `sdo.optimize` command minimizes the return argument of the anonymous function `estFcn`, that is, the residual errors returned by

`sdoAircraftEstimation_Objective`. For more details on how to write an objective/constraint function to use with the `sdo.optimize` command, type `help sdoExampleCostFunction` at the MATLAB command prompt.

To examine the estimation objective function in more detail, type `edit sdoAircraftEstimation_Objective` at the MATLAB command prompt.

```
type sdoAircraftEstimation_Objective
```

```
function vals = sdoAircraftEstimation_Objective(v, Simulator, Exp)
%SDOAIKRAFTESTIMATION_OBJECTIVE
%
%   The sdoAircraftEstimation_Objective function is used to compare model
%   outputs against experimental data.
%
%   vals = sdoAircraftEstimation_Objective(v, Exp)
%
%   The |v| input argument is a vector of estimated model parameter values
%   and initial states.
%
%   The |Simulator| input argument is a simulation object used
%   simulate the model with the estimated parameter values.
%
%   The |Exp| input argument contains the estimation experiment data.
%
%   The |vals| return argument contains information about how well the
%   model simulation results match the experimental data and is used by
%   the |sdo.optimize| function to estimate the model parameters.
%
%   See also sdo.optimize, sdoExampleCostFunction,
%   sdoAircraftEstimation_cmddemo
%
% Copyright 2012-2015 The MathWorks, Inc.

%%
% Define a signal tracking requirement to compute how well the model output
% matches the experiment data. Configure the tracking requirement so that
% it returns the tracking error residuals (rather than the
% sum-squared-error) and does not normalize the errors.
%
r = sdo.requirements.SignalTracking;
r.Type      = '==';
```

```

r.Method      = 'Residuals';
r.Normalize   = 'off';

%%
% Update the experiments with the estimated parameter values.
%
Exp = setEstimatedValues(Exp,v);

%%
% Simulate the model and compare model outputs with measured experiment
% data.
%
Simulator = createSimulator(Exp,Simulator);
Simulator = sim(Simulator);

SimLog      = find(Simulator.LoggedData,get_param('sdoAircraftEstimation','SignalLogging'));
PilotGSignal = find(SimLog,'PilotG');
AoASignal   = find(SimLog,'AngleOfAttack');

PilotGError = evalRequirement(r,PilotGSignal.Values,Exp.OutputData(1).Values);
AoAError    = evalRequirement(r,AoASignal.Values,Exp.OutputData(2).Values);

%%
% Return the residual errors to the optimization solver.
%
vals.F = [PilotGError(:); AoAError(:)];
end

```

Estimate the Parameters

Use the `sdo.optimize` function to estimate the actuator parameter values and initial state.

Specify the optimization options. The estimation function `sdoAircraftEstimation_Objective` returns the error residuals between simulated and experimental data and does not include any constraints, making this problem ideal for the 'lsqnonlin' solver.

```

opt = sdo.OptimizeOptions;
opt.Method = 'lsqnonlin';

```

Estimate the parameters.

```
vOpt = sdo.optimize(estFcn,v,opt)
```

2 Parameter Estimation

Optimization started 14-Aug-2017 23:59:35

Iter	F-count	f(x)	Step-size	First-order optimality
0	8	27972.2	1	
1	17	10124.8	0.4744	5.69e+04
2	26	3128.2	0.3854	1.24e+04
3	35	872.811	0.4289	2.81e+03
4	44	238.66	0.515	617
5	53	71.7155	0.4939	148
6	62	16.8531	0.4257	44
7	71	1.77655	0.301	11.3
8	80	0.0415982	0.1344	1.36
9	89	0.00184978	0.02513	0.203
10	98	0.00115139	0.007944	0.128

Local minimum possible.

lsqnonlin stopped because the final change in the sum of squares relative to its initial value is less than the selected value of the function tolerance.

vOpt(1,1) =

```
Name: 'Ta'  
Value: 0.0500  
Minimum: 0.0100  
Maximum: 1  
Free: 1  
Scale: 0.5000  
Info: [1x1 struct]
```

vOpt(2,1) =

```
Name: 'Md'  
Value: -6.8849  
Minimum: -10  
Maximum: 0  
Free: 1  
Scale: 1  
Info: [1x1 struct]
```

vOpt(3,1) =

```

        Name: 'Zd'
        Value: -63.9881
        Minimum: -100
        Maximum: 0
        Free: 1
        Scale: 128
        Info: [1x1 struct]

vOpt(4,1) =

        Name: 'sdoAircraftEstimation/Actuator...'
        Value: 1.1100e-04
        Minimum: 0
        Maximum: Inf
        Free: 1
        Scale: 1
        dxValue: 0
        dxFree: 1
        Info: [1x1 struct]

4x1 param.Continuous

```

Compare the Measured Output and the Final Simulated Output

Update the experiments with the estimated parameter values.

```
Exp = setEstimatedValues(Exp,vOpt);
```

Simulate the model using the updated experiment and compare the simulated output with the experimental data.

The model response using the estimated parameter values closely matches the experiment output data.

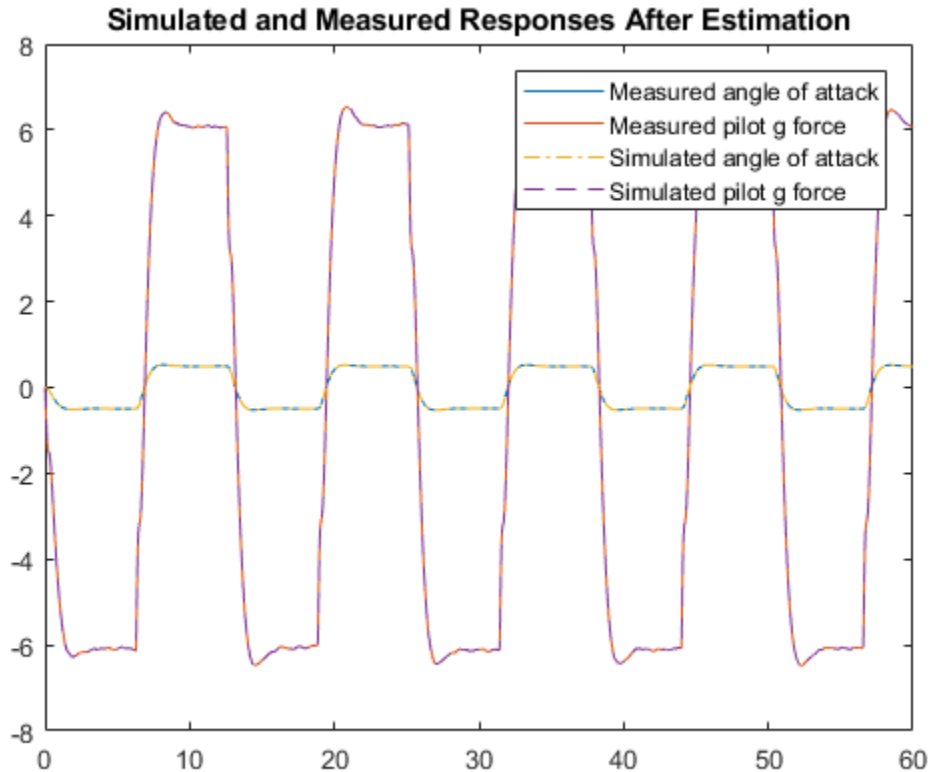
```

Simulator = createSimulator(Exp, Simulator);
Simulator = sim(Simulator);
SimLog = find(Simulator.LoggedData, get_param('sdoAircraftEstimation', 'SignalLoggi
PilotGSignal = find(SimLog, 'PilotG');
AoASignal = find(SimLog, 'AngleOfAttack');

plot(time, iodata, ...

```

```
AoASignal.Values.Time,AoASignal.Values.Data,'-.', ...  
PilotGSignal.Values.Time,PilotGSignal.Values.Data,'--')  
title('Simulated and Measured Responses After Estimation')  
legend('Measured angle of attack', 'Measured pilot g force', ...  
       'Simulated angle of attack', 'Simulated pilot g force');
```



Update the Model Parameter Values

Update the model with the estimated actuator parameter values. Do not update the model actuator initial state (fourth element of `vOpt`) as it is dependent on the experiment.

```
sdo.setValueInModel('sdoAircraftEstimation',vOpt(1:3));
```


Related Examples

To learn how to estimate model parameters using the Parameter Estimation Tool, see "Estimate Model Parameter Values (GUI)".

Close the model.

```
bdclose('sdoAircraftEstimation')
```

Estimate Model Parameters and Initial States (Code)

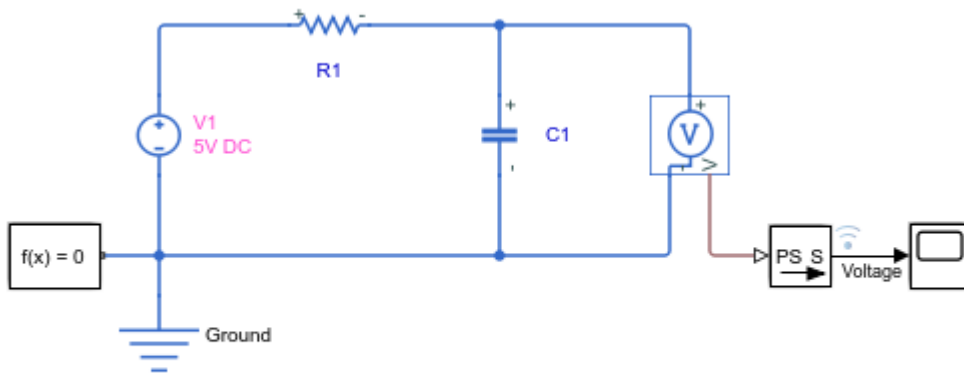
This example shows how to estimate the initial state and parameters of a model.

This example requires Simscape®.

RC Circuit Model

The Simulink model, `sdoRCCircuit`, models a simple resistor-capacitor (RC) circuit.

```
open_system('sdoRCCircuit');
```



Copyright 2011 The MathWorks, Inc.

Estimation Problem

You use the measured data to estimate the RC model parameter and state values.

Measured output data:

- Capacitor voltage, output of the PS-Simulink Converter block

Parameter:

- Capacitance, $C1$, used by the $C1$ block

State:

- Initial voltage of the capacitor, C1

Define the Estimation Experiment

Get the measured data.

```
load sdoRCCircuit_ExperimentData
```

The variables `time` and `data` are loaded into the workspace, where `data` is the measured capacitor voltage for times `time`.

Create an experiment object to store the experimental voltage data.

```
Exp = sdo.Experiment('sdoRCCircuit');
```

Create an object to store the measured capacitor voltage output.

```
Voltage = Simulink.SimulationData.Signal;
Voltage.Name      = 'Voltage';
Voltage.BlockPath = 'sdoRCCircuit/PS-Simulink Converter';
Voltage.PortType  = 'outport';
Voltage.PortIndex = 1;
Voltage.Values    = timeseries(data,time);
```

Add the measured capacitor data to the experiment as the expected output data.

```
Exp.OutputData = Voltage;
```

Compare the Measured Output and the Initial Simulated Output

Create a simulation scenario using the experiment and obtain the simulated output.

```
Simulator = createSimulator(Exp);
Simulator = sim(Simulator);
```

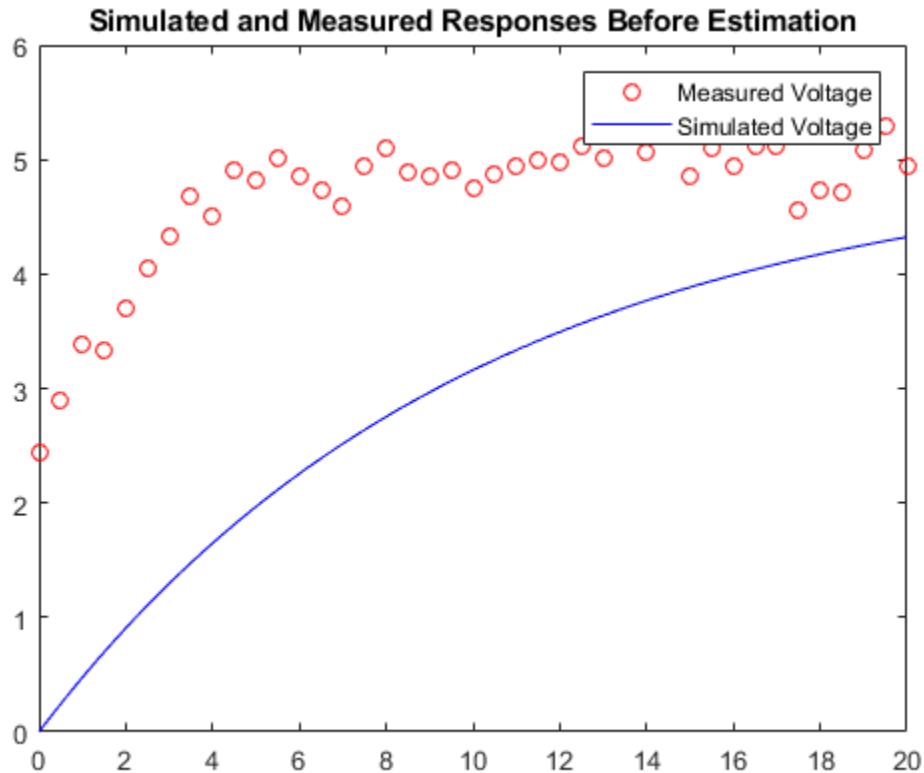
Search for the voltage signal in the logged simulation data.

```
SimLog      = find(Simulator.LoggedData, get_param('sdoRCCircuit', 'SignalLoggingName'));
Voltage     = find(SimLog, 'Voltage');
```

Plot the measured and simulated data.

The model response does not match the experimental output data.

```
plot(time,data,'ro',Voltage.Values.Time,Voltage.Values.Data,'b')
title('Simulated and Measured Responses Before Estimation')
legend('Measured Voltage','Simulated Voltage')
```



Specify the Parameters to Estimate

Select the capacitance parameter from the model. Specify an initial guess for the capacitance value (460 uF) and a minimum bound (0 F).

```
p = sdo.getParameterFromModel('sdoRCCircuit','C1');
p.Value = 460e-6;
p.Minimum = 0;
```

Define the Estimation Objective Function

Create an estimation objective function to evaluate how closely the simulation output, generated using the estimated parameter value, matches the measured data.

Use an anonymous function with one input argument that calls the `sdoRCCircuit_Objective` function. We pass the anonymous function to `sdo.optimize`, which evaluates the function at each optimization iteration.

```
estFcn = @(v) sdoRCCircuit_Objective(v, Simulator, Exp);
```

The `sdoRCCircuit_Objective` function:

- Has one input argument that specifies the estimated circuit capacitance value.
- Has one input argument that specifies the experiment object containing the measured data.
- Returns a vector of errors between simulated and experimental outputs.

The `sdoRCCircuit_Objective` function requires two inputs, but `sdo.optimize` requires a function with one input argument. To work around this, `estFcn` is an anonymous function with one input argument, `v`, but it calls `sdoRCCircuit_Objective` using two input arguments, `v` and `Exp`.

For more information regarding anonymous functions, see "Anonymous Functions".

The optimization solver minimizes the residual errors. For more details on how to write an objective/constraint function to use with the `sdo.optimize` command, type `help sdoExampleCostFunction` at the MATLAB command prompt.

To examine the estimation object function in more detail, type `edit sdoRCCircuit_Objective` at the MATLAB command prompt.

```
type sdoRCCircuit_Objective
```

```
function vals = sdoRCCircuit_Objective(v, Simulator, Exp)
%SDORCCIRCUIT_OBJECTIVE
%
%   The sdoRCCircuit_Objective function is used to compare model
%   outputs against experimental data.
%
```

```
% vals = sdoRCCircuit_Objective(v,Exp)
%
% The |v| input argument is a vector of estimated model parameter values
% and initial states.
%
% The |Simulator| input argument is a simulation object used
% simulate the model with the estimated parameter values.
%
% The |Exp| input argument contains the estimation experiment data.
%
% The |vals| return argument contains information about how well the
% model simulation results match the experimental data and is used by
% the |sdo.optimize| function to estimate the model parameters.
%
% See also sdo.optimize, sdoExampleCostFunction, sdoRCCircuit_cmddemo
%

% Copyright 2012-2015 The MathWorks, Inc.

%%
% Define a signal tracking requirement to compute how well the model output
% matches the experiment data. Configure the tracking requirement so that
% it returns the tracking error residuals (rather than the
% sum-squared-error) and does not normalize the errors.
%
r = sdo.requirements.SignalTracking;
r.Type      = '==';
r.Method    = 'Residuals';
r.Normalize = 'off';

%%
% Update the experiments with the estimated parameter values.
%
Exp = setEstimatedValues(Exp,v);

%%
% Simulate the model and compare model outputs with measured experiment
% data.
%
Simulator = createSimulator(Exp,Simulator);
Simulator = sim(Simulator);

SimLog = find(Simulator.LoggedData,get_param('sdoRCCircuit','SignalLoggingName'));
Voltage = find(SimLog,'Voltage');
```

```

VoltageError = evalRequirement(r,Voltage.Values,Exp.OutputData(1).Values);

%%
% Return the residual errors to the optimization solver.
%
vals.F = VoltageError(:);
end

```

Estimate the Parameters

Use the `sdo.optimize` function to estimate the capacitance value.

Specify the optimization options. The estimation function `sdoRCCircuit_Objective` returns the error residuals between simulated and experimental data and does not include any constraints, making this problem ideal for the 'lsqnonlin' solver.

```

opt = sdo.OptimizeOptions;
opt.Method = 'lsqnonlin';

```

Estimate the parameters.

```
pOpt = sdo.optimize(estFcn,p,opt)
```

```
Optimization started 15-Aug-2017 00:09:53
```

Iter	F-count	f(x)	Step-size	First-order optimality
0	3	55.0041	1	
1	6	21.0161	0.2124	17.2
2	9	11.5085	0.1272	6.08
3	12	9.56468	0.06553	1.99
4	15	9.27666	0.02744	0.442
5	18	9.27666	0.00717	0.442
6	21	9.27131	0.001793	0.356

Local minimum possible.

lsqnonlin stopped because the final change in the sum of squares relative to its initial value is less than the selected value of the function tolerance.

```
pOpt =
```

```

    Name: 'C1'
    Value: 1.1600e-04

```

```
Minimum: 0
Maximum: Inf
Free: 1
Scale: 0.0020
Info: [1x1 struct]
```

```
1x1 param.Continuous
```

Compare the Measured Output and the Simulated Output

Update the experiment with the estimated capacitance value.

```
Exp = setEstimatedValues(Exp,pOpt);
```

Create a simulation scenario using the experiment and obtain the simulated output.

```
Simulator = createSimulator(Exp, Simulator);
Simulator = sim(Simulator);
```

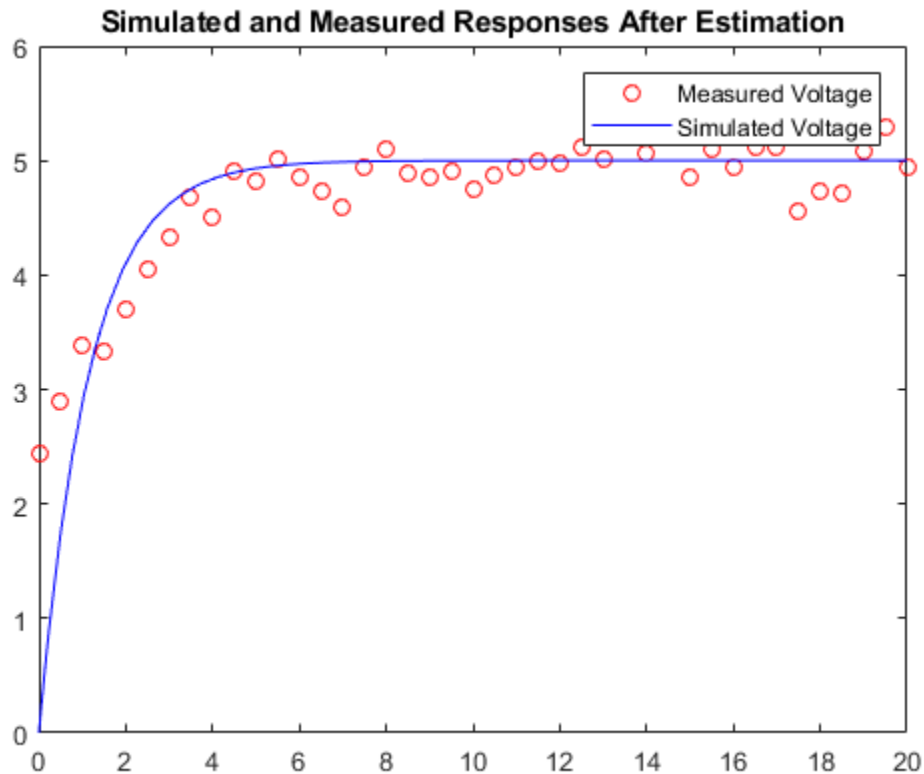
Search for the voltage signal in the logged simulation data.

```
SimLog = find(Simulator.LoggedData, get_param('sdoRCCircuit', 'SignalLoggingName'));
Voltage = find(SimLog, 'Voltage');
```

Plot the measured and simulated data.

The simulated and measured signals match well, except for near time zero. This mismatch is because the capacitor initial voltage defined in the model does not match the initial voltage from the experiment.

```
plot(time, data, 'ro', Voltage.Values.Time, Voltage.Values.Data, 'b')
title('Simulated and Measured Responses After Estimation')
legend('Measured Voltage', 'Simulated Voltage')
```

Estimate the Initial State

Add the capacitor initial voltage for the C1 block to the experiment. Set its initial guess value to 1 V.

```
Exp.InitialStates = sdo.getStateFromModel('sdoRCCircuit','C1');
Exp.InitialStates.Value = 1;
```

Recreate the estimation function to use the experiment with initial state estimation

```
estFcn = @(v) sdoRCCircuit_Objective(v, Simulator, Exp);
```

Get the initial state and capacitance value that is to be estimated from the experiment.

```
v = getValuesToEstimate(Exp);
```

Estimate the parameters.

```
vOpt = sdo.optimize(estFcn,v,opt)
```

```
Optimization started 15-Aug-2017 00:10:22
```

Iter	F-count	f(x)	Step-size	First-order optimality
0	5	4.66337	1	
1	10	2.01883	1.533	21
2	15	1.34889	0.1257	0.0803
3	20	1.34365	0.0525	0.12
4	25	1.34363	0.001294	0.000711

```
Local minimum found.
```

```
Optimization completed because the size of the gradient is less than  
the selected value of the optimality tolerance.
```

```
vOpt(1,1) =
```

```
    Name: 'sdoRCCircuit/C1:sdoRCCircuit.C1.vc'  
    Value: 2.3596  
    Minimum: -Inf  
    Maximum: Inf  
    Free: 1  
    Scale: 1  
    dxValue: 0  
    dxFree: 1  
    Info: [1x1 struct]
```

```
vOpt(2,1) =
```

```
    Name: 'C1'  
    Value: 2.2638e-04  
    Minimum: 0  
    Maximum: Inf  
    Free: 1  
    Scale: 0.0020  
    Info: [1x1 struct]
```

```
2x1 param.Continuous
```

Compare the Measured Output and the Final Simulated Output

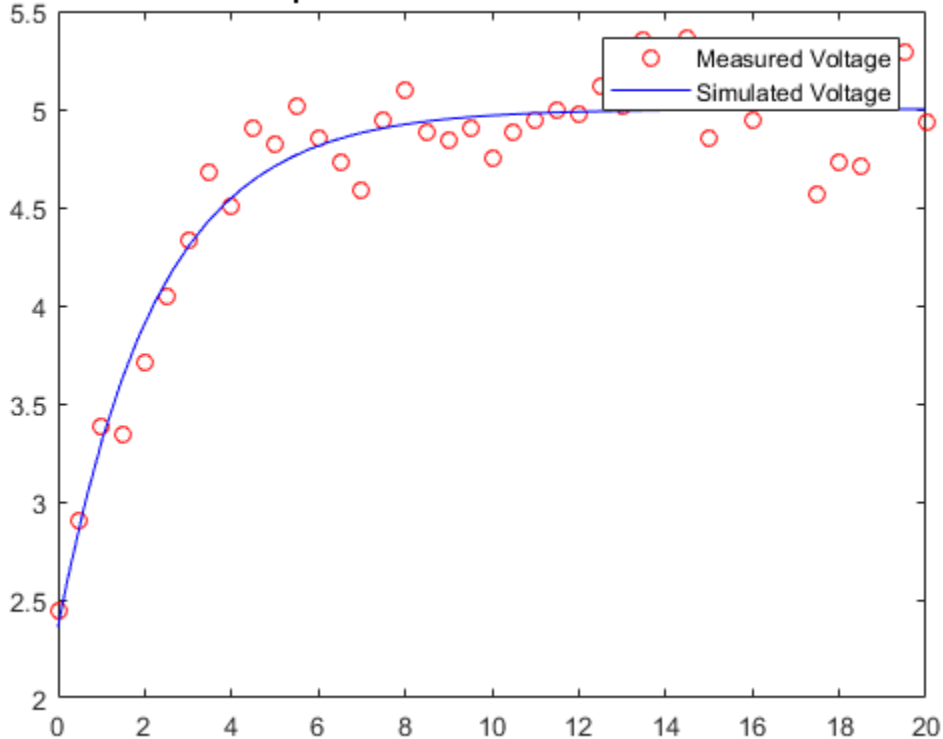
Update the experiment with the estimated capacitance and capacitor initial voltage values.

```
Exp = setEstimatedValues(Exp, vOpt);
```

Simulate the model with the estimated initial-state and parameter values and compare the simulated output with the experiment data.

```
Simulator = createSimulator(Exp, Simulator);  
Simulator = sim(Simulator);  
SimLog     = find(Simulator.LoggedData, get_param('sdoRCCircuit', 'SignalLoggingName'));  
Voltage    = find(SimLog, 'Voltage');  
  
plot(time, data, 'ro', Voltage.Values.Time, Voltage.Values.Data, 'b')  
title('Simulated and Measured Responses After Initial State and Model Parameter Estimat  
legend('Measured Voltage', 'Simulated Voltage')
```

Simulated and Measured Responses After Initial State and Model Parameter Estimation



Update the Model Parameter Values

Update the model with the estimated capacitance value. Do not update the model capacitor initial voltage (first element of `vOpt`) as it is dependent on the experiment.

```
sdo.setValueInModel('sdoRCCircuit',vOpt(2));
```

Related Examples

To learn how to estimate model parameters using the `sdo.optimize` command, see "Estimate Model Parameters and Initial States (GUI)".

Close the model

```
bdclose('sdoRCCircuit')
```

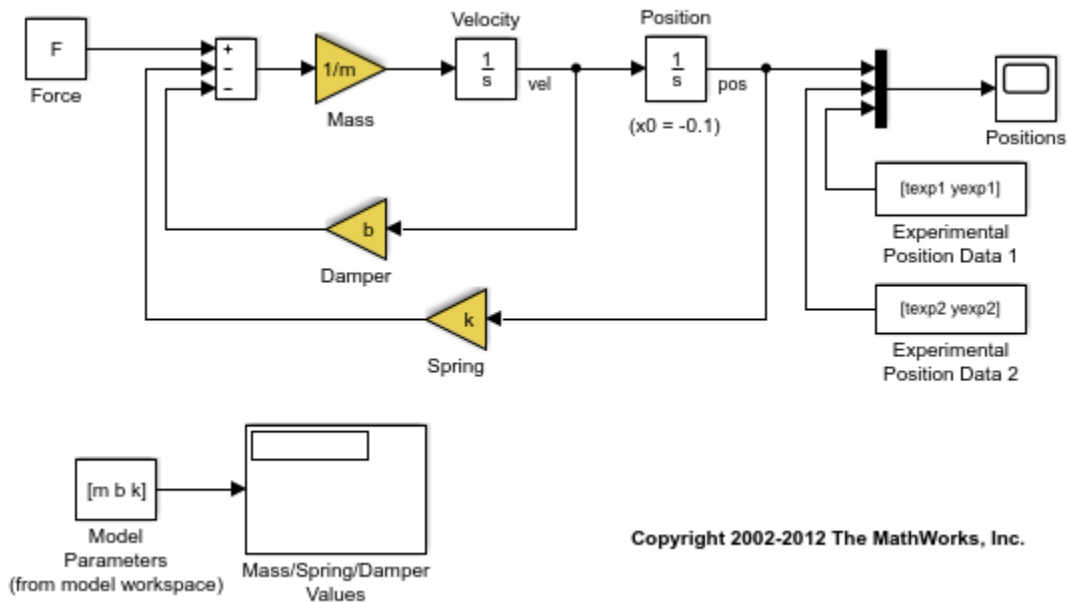
Estimate Model Parameters Using Multiple Experiments (Code)

This example shows how to estimate model parameters from multiple sets of experimental data. You estimate the parameters of a mass-spring-damper system.

Open the Model and Get Experimental Data

This example uses the `sdoMassSpringDamper` model. The model includes two integrators to model the velocity and position of a mass in a mass-spring-damper system.

```
open_system('sdoMassSpringDamper');
```



Load the experiment data.

```
load sdoMassSpringDamper_ExperimentData
```

The variables `texp1`, `yexp1`, `texp2`, and `yexp2` are loaded into the workspace. `yexp1` and `yexp2` describe the mass position for times `texp1` and `texp2` respectively.

Define the Estimation Experiments

Create a 2-element array of experiment objects to store the measured data for the two experiments.

Create an experiment object for the first experiment.

```
Exp = sdo.Experiment('sdoMassSpringDamper');
```

Create an object to store the measured mass position output.

```
MeasuredPos          = Simulink.SimulationData.Signal;  
MeasuredPos.Values   = timeseries(yexpl,texpl);  
MeasuredPos.BlockPath = 'sdoMassSpringDamper/Position';  
MeasuredPos.PortType = 'outport';  
MeasuredPos.PortIndex = 1;  
MeasuredPos.Name     = 'Position';
```

Add the measured mass position data to the experiment as the expected output data.

```
Exp.OutputData = MeasuredPos;
```

Create an object to specify the initial state for the Velocity block. The initial velocity of the mass is 0 m/s.

```
sVel          = sdo.getStateFromModel('sdoMassSpringDamper','Velocity');  
sVel.Value = 0;  
sVel.Free  = false;
```

`sVel.Free` is set to `false` because the initial velocity is known and does not need to be estimated.

Create an object to specify the initial state for the Position block. Specify a guess for the initial mass position. Set the `Free` field of the initial position object to `true` so that it is estimated.

```
sPos          = sdo.getStateFromModel('sdoMassSpringDamper','Position');  
sPos.Free    = true;  
sPos.Value   = -0.1;
```

Add the initial states to the experiment.

```
Exp.InitialStates = [sVel;sPos];
```

Create a 2-element array of experiments. As the two experiments are identical except for the expected output data, copy the first experiment twice.

```
Exp = [Exp; Exp];
```

Modify the expected output data of the second experiment object in Exp.

```
Exp(2).OutputData.Values = timeseries(yexp2,texp2);
```

Compare the Measured Output and the Initial Simulated Output

Create a simulation scenario using the first experiment and obtain the simulated output.

```
Simulator = createSimulator(Exp(1));  
Simulator = sim(Simulator);
```

Search for the position signal in the logged simulation data.

```
SimLog = find(Simulator.LoggedData,get_param('sdoMassSpringDamper','SignalLoggingName')  
Position = find(SimLog,'Position');
```

Obtain the simulated position signal for the second experiment.

```
Simulator = createSimulator(Exp(2),Simulator);  
Simulator = sim(Simulator);  
SimLog = find(Simulator.LoggedData,get_param('sdoMassSpringDamper','SignalLoggingName')  
Position(2) = find(SimLog,'Position');
```

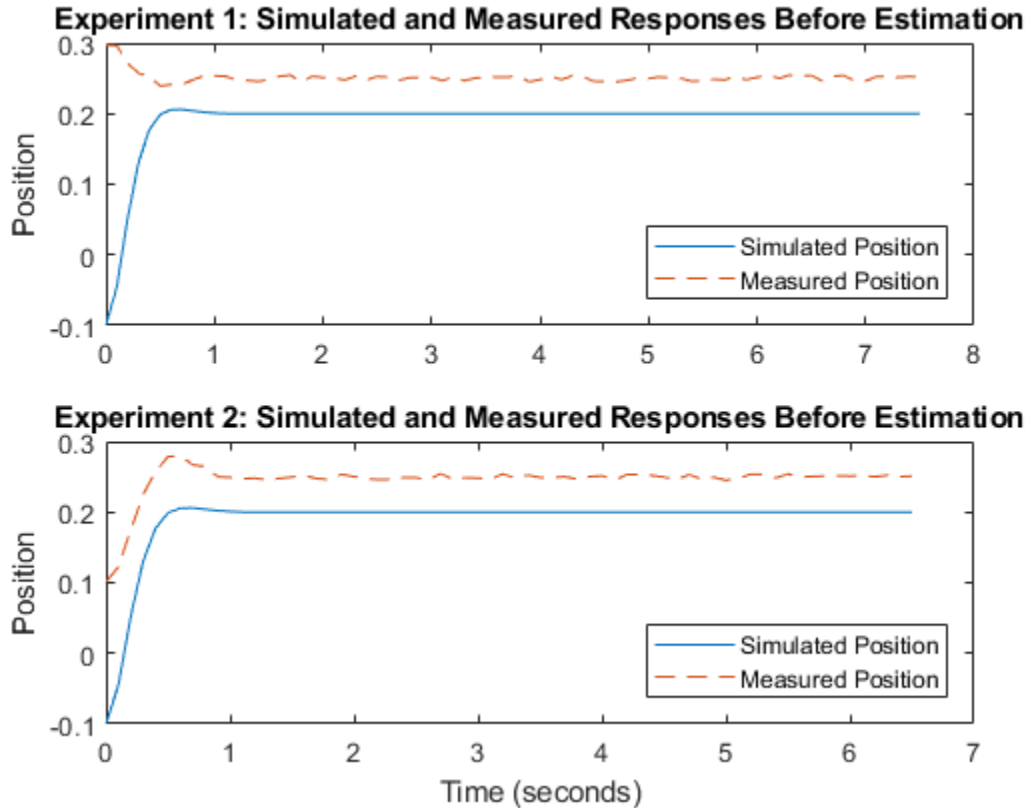
Plot the measured and simulated data.

The model response does not match the experimental output data.

```
subplot(211)  
plot(...  
    Position(1).Values.Time,Position(1).Values.Data, ...  
    Exp(1).OutputData.Values.Time, Exp(1).OutputData.Values.Data,'--')  
title('Experiment 1: Simulated and Measured Responses Before Estimation')  
ylabel('Position')  
legend('Simulated Position','Measured Position','Location','SouthEast')  
subplot(212)  
plot(...  
    Position(2).Values.Time,Position(2).Values.Data, ...  
    Exp(2).OutputData.Values.Time, Exp(2).OutputData.Values.Data,'--')  
title('Experiment 2: Simulated and Measured Responses Before Estimation')  
xlabel('Time (seconds)')
```



```
ylabel('Position')
legend('Simulated Position','Measured Position','Location','SouthEast')
```



Specify Parameters to Estimate

Select the mass m , spring constant k , and damping coefficient b parameters from the model. Specify that the estimated values for these parameters must be positive.

```
p = sdo.getParameterFromModel('sdoMassSpringDamper', {'b', 'k', 'm'});
p(1).Minimum = 0;
p(2).Minimum = 0;
p(3).Minimum = 0;
```

Get the position initial state values to be estimated from the experiment.

```
s = getValuesToEstimate(Exp);
```

s contains two initial state objects, both for the `Position` block. Each object corresponds to an experiment in `Exp`.

Group the model parameters and initial states to be estimated together.

```
v = [p;s]
```

```
v(1,1) =
```

```
    Name: 'b'  
    Value: 100  
  Minimum: 0  
  Maximum: Inf  
    Free: 1  
    Scale: 128  
    Info: [1x1 struct]
```

```
v(2,1) =
```

```
    Name: 'k'  
    Value: 500  
  Minimum: 0  
  Maximum: Inf  
    Free: 1  
    Scale: 512  
    Info: [1x1 struct]
```

```
v(3,1) =
```

```
    Name: 'm'  
    Value: 8  
  Minimum: 0  
  Maximum: Inf  
    Free: 1  
    Scale: 8  
    Info: [1x1 struct]
```

```
v(4,1) =
```

```

        Name: 'sdoMassSpringDamper/Position'
        Value: -0.1000
        Minimum: -Inf
        Maximum: Inf
        Free: 1
        Scale: 0.1250
        dxValue: 0
        dxFree: 1
        Info: [1x1 struct]

v(5,1) =

        Name: 'sdoMassSpringDamper/Position'
        Value: -0.1000
        Minimum: -Inf
        Maximum: Inf
        Free: 1
        Scale: 0.1250
        dxValue: 0
        dxFree: 1
        Info: [1x1 struct]

5x1 param.Continuous

```

Define the Estimation Objective

Create an estimation objective function to evaluate how closely the simulation output, generated using the estimated parameter values, matches the measured data.

Use an anonymous function with one input argument that calls the `sdoMassSpringDamper_Objective` function. We pass the anonymous function to `sdo.optimize`, which evaluates the function at each optimization iteration.

```
estFcn = @(v) sdoMassSpringDamper_Objective(v, Simulator, Exp);
```

The `sdoMassSpringDamper_Objective` function:

- Has one input argument that specifies the mass, spring constant and damper values as well as the initial mass position.
- Has one input argument that specifies the experiment object containing the measured data.

- Returns a vector of errors between simulated and experimental outputs.

The `sdoMassSpringDamper_Objective` function requires two inputs, but `sdo.optimize` requires a function with one input argument. To work around this, `estFcn` is an anonymous function with one input argument, `v`, but it calls `sdoMassSpringDamper_Objective` using two input arguments, `v` and `Exp`.

For more information regarding anonymous functions, see "Anonymous Functions".

The `sdo.optimize` command minimizes the return argument of the anonymous function `estFcn`, that is, the residual errors returned by `sdoMassSpringDamper_Objective`. For more details on how to write an objective/constraint function to use with the `sdo.optimize` command, type `help sdoExampleCostFunction` at the MATLAB command prompt.

To examine the estimation objective function in more detail, type `edit sdoMassSpringDamper_Objective` at the MATLAB command prompt.

```
type sdoMassSpringDamper_Objective
```

```
function vals = sdoMassSpringDamper_Objective(v, Simulator, Exp)
%SDOMASSSPRINGDAMPER_OBJECTIVE
%
%   The sdoMassSpringDamper_Objective function is used to compare model
%   outputs against experimental data.
%
%   vals = sdoMassSpringDamper_Objective(v, Exp)
%
%   The |v| input argument is a vector of estimated model parameter values
%   and initial states.
%
%   The |Simulator| input argument is a simulation object used
%   simulate the model with the estimated parameter values.
%
%   The |Exp| input argument contains the estimation experiment data.
%
%   The |vals| return argument contains information about how well the
%   model simulation results match the experimental data and is used by
%   the |sdo.optimize| function to estimate the model parameters.
%
%   see also sdo.optimize, sdoExampleCostFunction
%
```

```
% Copyright 2012-2015 The MathWorks, Inc.

%%
% Define a signal tracking requirement to compute how well the model output
% matches the experiment data. Configure the tracking requirement so that
% it returns the tracking error residuals (rather than the
% sum-squared-error) and does not normalize the errors.
%
r = sdo.requirements.SignalTracking;
r.Type      = '==';
r.Method    = 'Residuals';
r.Normalize = 'off';

%%
% Update the experiments with the estimated parameter values.
%
Exp = setEstimatedValues(Exp,v);

%%
% Simulate the model and compare model outputs with measured experiment
% data.
%
Error = [];
for ct=1:numel(Exp)

    Simulator = createSimulator(Exp(ct),Simulator);
    Simulator = sim(Simulator);

    SimLog = find(Simulator.LoggedData,get_param('sdoMassSpringDamper','SignalLoggingName'));
    Position = find(SimLog,'Position');

    PositionError = evalRequirement(r,Position.Values,Exp(ct).OutputData.Values);

    Error = [Error; PositionError(:)];
end

%%
% Return the residual errors to the optimization solver.
%
vals.F = Error(:);
end
```

Estimate the Parameters

Use the `sdo.optimize` function to estimate the actuator parameter values and initial state.

Specify the optimization options. The estimation function `sdoMassSpringDamper_Objective` returns the error residuals between simulated and experimental data and does not include any constraints, making this problem ideal for the 'lsqnonlin' solver.

```
opt = sdo.OptimizeOptions;  
opt.Method = 'lsqnonlin';
```

Estimate the parameters. Notice that the initial mass position is estimated twice, once for each experiment.

```
vOpt = sdo.optimize(estFcn,v,opt)
```

```
Optimization started 15-Aug-2017 00:08:27
```

Iter	F-count	f(x)	Step-size	First-order optimality
0	11	0.777696	1	
1	22	0.00413099	3.696	0.00648
2	33	0.00118327	0.3194	0.00243
3	44	0.0011106	0.06718	5.09e-05

Local minimum found.

```
Optimization completed because the size of the gradient is less than  
the selected value of the optimality tolerance.
```

```
vOpt(1,1) =
```

```
    Name: 'b'  
    Value: 58.1959  
  Minimum: 0  
  Maximum: Inf  
    Free: 1  
   Scale: 128  
   Info: [1x1 struct]
```

```
vOpt(2,1) =
```

```
Name: 'k'  
Value: 399.9452  
Minimum: 0  
Maximum: Inf  
Free: 1  
Scale: 512  
Info: [1x1 struct]
```

```
vOpt(3,1) =
```

```
Name: 'm'  
Value: 9.7225  
Minimum: 0  
Maximum: Inf  
Free: 1  
Scale: 8  
Info: [1x1 struct]
```

```
vOpt(4,1) =
```

```
Name: 'sdoMassSpringDamper/Position'  
Value: 0.2995  
Minimum: -Inf  
Maximum: Inf  
Free: 1  
Scale: 0.1250  
dxValue: 0  
dxFree: 1  
Info: [1x1 struct]
```

```
vOpt(5,1) =
```

```
Name: 'sdoMassSpringDamper/Position'  
Value: 0.0994  
Minimum: -Inf  
Maximum: Inf  
Free: 1  
Scale: 0.1250  
dxValue: 0  
dxFree: 1
```

```
Info: [1x1 struct]
```

```
5x1 param.Continuous
```

Compare the Measured Output and the Final Simulated Output

Update the experiments with the estimated parameter values.

```
Exp = setEstimatedValues(Exp,vOpt);
```

Obtain the simulated output for the first experiment.

```
Simulator = createSimulator(Exp(1), Simulator);  
Simulator = sim(Simulator);  
SimLog = find(Simulator.LoggedData, get_param('sdoMassSpringDamper', 'SignalLoggingName'));  
Position(1) = find(SimLog, 'Position');
```

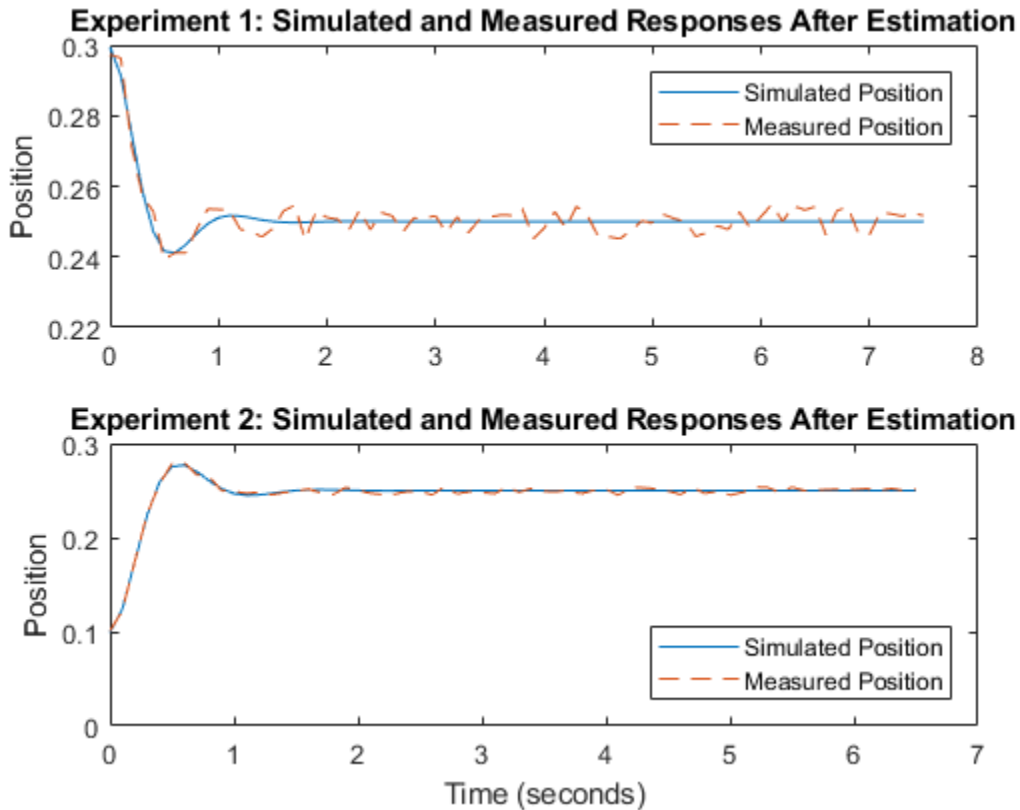
Obtain the simulated output for the second experiment.

```
Simulator = createSimulator(Exp(2), Simulator);  
Simulator = sim(Simulator);  
SimLog = find(Simulator.LoggedData, get_param('sdoMassSpringDamper', 'SignalLoggingName'));  
Position(2) = find(SimLog, 'Position');
```

Plot the measured and simulated data.

The model response using the estimated parameter values nicely matches the output data for the experiments.

```
subplot(211)  
plot(...  
    Position(1).Values.Time, Position(1).Values.Data, ...  
    Exp(1).OutputData.Values.Time, Exp(1).OutputData.Values.Data, '--')  
title('Experiment 1: Simulated and Measured Responses After Estimation')  
ylabel('Position')  
legend('Simulated Position', 'Measured Position', 'Location', 'NorthEast')  
subplot(212)  
plot(...  
    Position(2).Values.Time, Position(2).Values.Data, ...  
    Exp(2).OutputData.Values.Time, Exp(2).OutputData.Values.Data, '--')  
title('Experiment 2: Simulated and Measured Responses After Estimation')  
xlabel('Time (seconds)')  
ylabel('Position')  
legend('Simulated Position', 'Measured Position', 'Location', 'SouthEast')
```

Update the Model Parameter Values

Update the model m , k , and b parameter values. Do not update the model initial position value as this is dependent on the experiment.

```
sdo.setValueInModel('sdoMassSpringDamper',vOpt(1:3));
```

Close the model

```
bdclose('sdoMassSpringDamper')
```

Estimate Model Parameters Per Experiment (Code)

This example shows how to use multiple experiments to estimate a mix of model parameter values; some that are estimated using all the experiments and others that are estimated using individual experiments. The example also shows how to configure estimation experiments with experiment dependent parameter values.

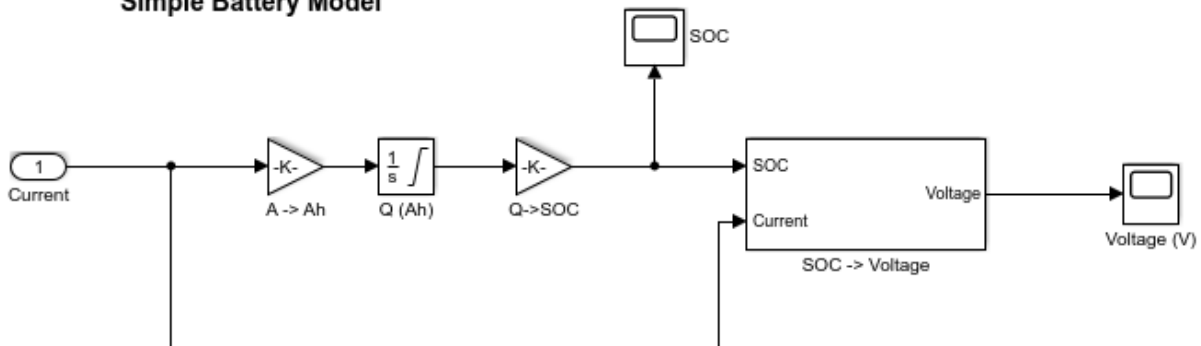
You estimate the parameters of a rechargeable battery based on data collected in experiments that discharge and charge the battery.

Open the Model and Get Experimental Data

This example estimates parameters of a simple, rechargeable battery model, `sdoBattery`. The model input is the battery current and the model output, the battery terminal voltage, is computed from the battery state-of-charge.

```
open_system('sdoBattery');
```

Simple Battery Model



Copyright 2012-2014 The MathWorks, Inc.

The model is based on the equation

$$E = (1 - Loss) * V - K * Q_{max} * \frac{1 - s}{s}$$

Where:

- E is the battery terminal voltage in Volts.
- V is the battery constant voltage in Volts.
- K is the battery polarization resistance in Ohms.
- Q_{max} is the maximum battery capacity in Ampere-Hour.
- s is the battery charge state, with 1 being fully charged and 0 zero charge. The battery state-of-charge is computed from the integral of the battery current with a +ve current indicating discharge and a -ve current indicating charging. The battery initial state-of-charge is specified by Q_0 in Ampere-Hour.
- $Loss$ is the voltage drop when charging, expressed as a fraction of the battery constant voltage. When the battery is discharging this value is zero.

V , K , Q_{max} , Q_0 , and $Loss$ are variables defined in the model workspace.

Load the experiment data. A 1.2V (6500mAh) battery was subjected to a discharge experiment and a charging experiment.

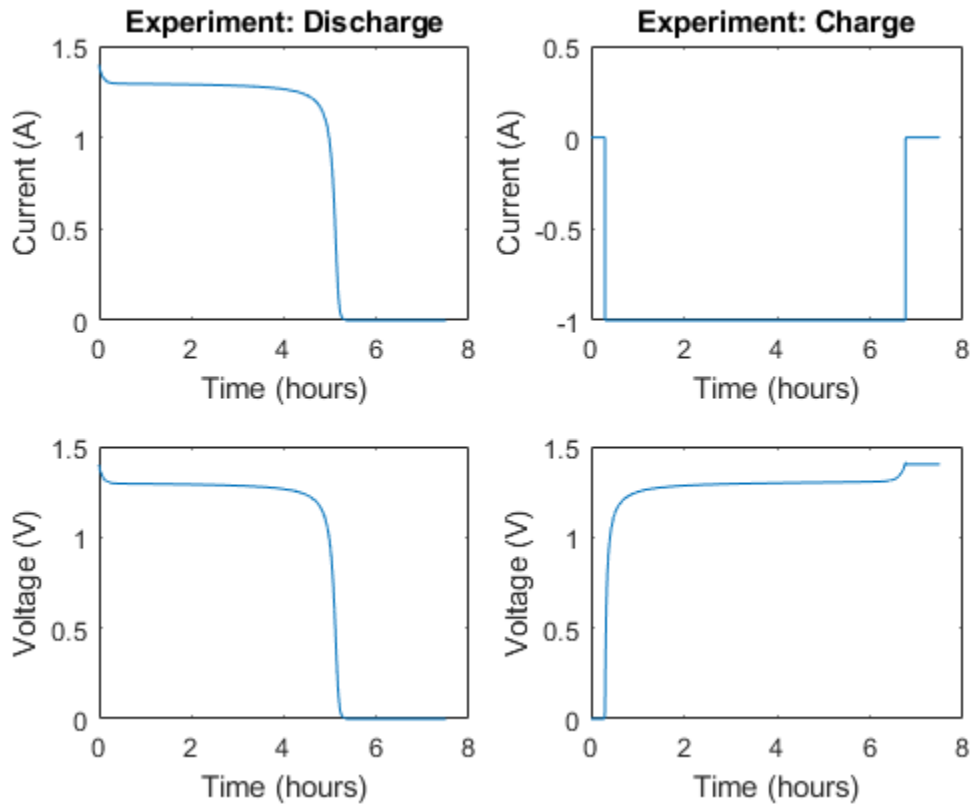
```
load sdoBattery_ExperimentData
```

The variables `Charge_Data` and `DCharge_Data` are loaded into the workspace. The first column of `Charge_Data` contains time data. The second and third columns of `Charge_Data` describe the current and voltage during a battery charging experiment. `DCharge_Data` is similarly structured and contains data for a battery discharging experiment.

Plot the Experiment Data

```
subplot(221),
plot(DCharge_Data(:,1)/3600,DCharge_Data(:,2))
title('Experiment: Discharge')
xlabel('Time (hours)')
ylabel('Current (A)')
subplot(223)
plot(DCharge_Data(:,1)/3600,DCharge_Data(:,3))
xlabel('Time (hours)')
ylabel('Voltage (V)')
subplot(222),
plot(Charge_Data(:,1)/3600,Charge_Data(:,2))
title('Experiment: Charge')
```

```
xlabel('Time (hours)')
ylabel('Current (A)')
subplot(224)
plot(Charge_Data(:,1)/3600,Charge_Data(:,3))
xlabel('Time (hours)')
ylabel('Voltage (V)')
```



Define the Estimation Experiments

Create a 2-element array of experiment objects to specify the measured data for the two experiments.

Create an experiment object for the battery discharge experiment. The measured current data is specified as a timeseries in the experiment object.

```
DCharge_Exp = sdo.Experiment('sdoBattery');
```

Specify the input data (current) as a timeseries object.

```
DCharge_Exp.InputData = timeseries(DCharge_Data(:,2),DCharge_Data(:,1));
```

Create an object to specify the measured voltage output data.

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

Add the voltage signal to the discharge experiment as the expected output data.

```
DCharge_Exp.OutputData = VoltageSig;
```

Specify the battery initial charge state for the experiment. The battery charge state is modeled by the Q (Ah) block and it's initial value is specified by the variable Q0. Create a parameter for the Q0 variable and add the parameter to the experiment. Q0 is experiment dependent and assumes different values in the discharging and charging experiments.

```
Q0 = sdo.getParameterFromModel('sdoBattery','Q0');
Q0.Value = 6.5;
Q0.Free = false;
```

Q0.Free is set to false because the initial battery charge is known and does not need to be estimated.

Add the Q0 parameter to the experiment.

```
DCharge_Exp.Parameters = Q0;
```

Create an experiment object to store the charging experiment data. Add the measured current input and measured voltage output data to the object.

```
Charge_Exp = sdo.Experiment('sdoBattery');
Charge_Exp.InputData = timeseries(Charge_Data(:,2),Charge_Data(:,1));
VoltageSig.Values    = timeseries(Charge_Data(:,3),Charge_Data(:,1));
Charge_Exp.OutputData = VoltageSig;
```

Add the battery initial charge and charging loss fraction parameters to the experiment. For this experiment, the initial charge (Q_0) is known (0 Ah), but the value of the charging loss fraction ($Loss$) is not known.

```
Q0.Value = 0;

Loss = sdo.getParameterFromModel('sdoBattery','Loss');
Loss.Free = true;
Loss.Minimum = 0;
Loss.Maximum = 0.5;

Charge_Exp.Parameters = [Q0;Loss];
```

`Loss.Free` is set to true so that the value of `Loss` is estimated.

Collect both experiments into one vector.

```
Exp = [DCharge_Exp; Charge_Exp];
```

Compare the Measured Output and the Initial Simulated Output

Create a simulation scenario using the first (discharging) experiment and obtain the simulated output.

```
Simulator = createSimulator(Exp(1));
Simulator = sim(Simulator);
```

Search for the voltage signal in the logged simulation data.

```
SimLog = find(Simulator.LoggedData,get_param('sdoBattery','SignalLoggingName'));
Voltage(1) = find(SimLog,'Voltage');
```

Obtain the simulated voltage signal for the second (charging) experiment.

```
Simulator = createSimulator(Exp(2),Simulator);
Simulator = sim(Simulator);
SimLog = find(Simulator.LoggedData,get_param('sdoBattery','SignalLoggingName'));
Voltage(2) = find(SimLog,'Voltage');
```

Plot the measured and simulated data.

The model response does not match the experimental output data.

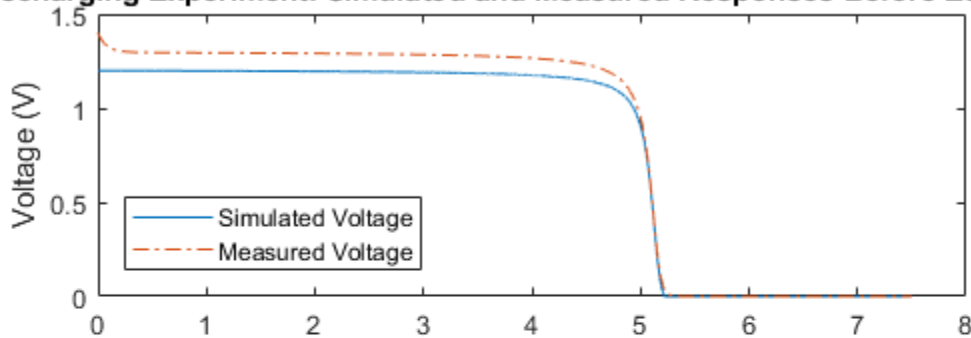
```
subplot(211)
plot(...
```

```

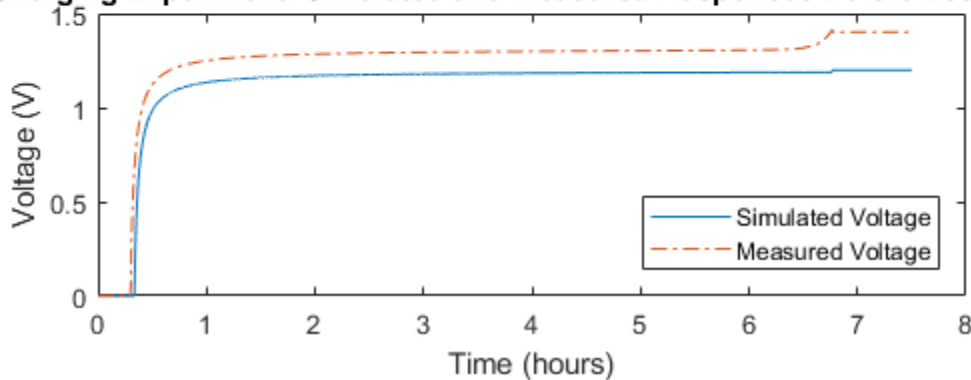
    Voltage(1).Values.Time/3600,Voltage(1).Values.Data, ...
    Exp(1).OutputData.Values.Time/3600, Exp(1).OutputData.Values.Data,'-.')
title('Discharging Experiment: Simulated and Measured Responses Before Estimation')
ylabel('Voltage (V)')
legend('Simulated Voltage','Measured Voltage','Location','SouthWest')
subplot(212)
plot(...
    Voltage(2).Values.Time/3600,Voltage(2).Values.Data, ...
    Exp(2).OutputData.Values.Time/3600, Exp(2).OutputData.Values.Data,'-.')
title('Charging Experiment: Simulated and Measured Responses Before Estimation')
xlabel('Time (hours)')
ylabel('Voltage (V)')
legend('Simulated Voltage','Measured Voltage','Location','SouthEast')

```

Discharging Experiment: Simulated and Measured Responses Before Estimation



Charging Experiment: Simulated and Measured Responses Before Estimation



Specify Parameters to Estimate

Estimate the values of the battery voltage V , the battery polarization resistance K , and the charging loss fraction $Loss$. The V and K parameters are estimated using all the experiment data while the $Loss$ parameter is estimated using only the charging data.

Select the battery voltage V and the battery polarization resistance K parameters from the model. Specify minimum and maximum bounds for these parameters.

```
p = sdo.getParameterFromModel('sdoBattery',{ 'V','K' });  
  
p(1).Minimum = 0;  
p(1).Maximum = 2;  
  
p(2).Minimum = 1e-6;  
p(2).Maximum = 1e-1;
```

Get the experiment-specific $Loss$ parameter from the experiment.

```
s = getValuesToEstimate(Exp);
```

Group all the parameters to be estimated.

```
v = [p;s]
```

```
v(1,1) =
```

```
    Name: 'V'  
    Value: 1.2000  
    Minimum: 0  
    Maximum: 2  
    Free: 1  
    Scale: 2  
    Info: [1x1 struct]
```

```
v(2,1) =
```

```
    Name: 'K'  
    Value: 1.0000e-03  
    Minimum: 1.0000e-06  
    Maximum: 0.1000  
    Free: 1  
    Scale: 0.0020
```



```

Info: [1x1 struct]

v(3,1) =

    Name: 'Loss'
    Value: 0.0100
    Minimum: 0
    Maximum: 0.5000
    Free: 1
    Scale: 0.0156
    Info: [1x1 struct]

3x1 param.Continuous

```

Define the Estimation Objective

Create an estimation objective function to evaluate how closely the simulation output, generated using the estimated parameter values, matches the measured data.

Use an anonymous function with one input argument that calls the `sdoBattery_Objective` function. We pass the anonymous function to `sdo.optimize`, which evaluates the function at each optimization iteration.

```
estFcn = @(v) sdoBattery_Objective(v, Simulator, Exp);
```

The `sdoBattery_Objective` function:

- Has one input argument that specifies the estimated battery parameter values.
- Has one input argument that specifies the experiment object containing the measured data.
- Returns a vector of errors between simulated and experimental outputs.

The `sdoBattery_Objective` function requires two inputs, but `sdo.optimize` requires a function with one input argument. To work around this, `estFcn` is an anonymous function with one input argument, `v`, but it calls `sdoBattery_Objective` using two input arguments, `v` and `Exp`.

For more information regarding anonymous functions, see "Anonymous Functions".

The `sdo.optimize` command minimizes the return argument of the anonymous function `estFcn`, that is, the residual errors returned by `sdoBattery_Objective`. For more details on how to write an objective/constraint function to use with the `sdo.optimize` command, type `help sdoExampleCostFunction` at the MATLAB command prompt.

To examine the estimation objective function in more detail, type `edit sdoBattery_Objective` at the MATLAB command prompt.

```
type sdoBattery_Objective
```

```
function vals = sdoBattery_Objective(v, Simulator, Exp)
%SDOBATTERY_OBJECTIVE
%
% The sdoBattery_Objective function is used to compare model
% outputs against experimental data.
%
% vals = sdoBattery_Objective(v, Exp)
%
% The |v| input argument is a vector of estimated model parameter values
% and initial states.
%
% The |Simulator| input argument is a simulation object used
% simulate the model with the estimated parameter values.
%
% The |Exp| input argument contains the estimation experiment data.
%
% The |vals| return argument contains information about how well the
% model simulation results match the experimental data and is used by
% the |sdo.optimize| function to estimate the model parameters.
%
% See also sdo.optimize, sdoExampleCostFunction, sdoBattery_cmddemo
%

% Copyright 2012-2015 The MathWorks, Inc.

%%
% Define a signal tracking requirement to compute how well the model output
% matches the experiment data. Configure the tracking requirement so that
% it returns the tracking error residuals (rather than the
% sum-squared-error) and does not normalize the errors.
%
r = sdo.requirements.SignalTracking;
```

```

r.Type      = '==';
r.Method    = 'Residuals';
r.Normalize = 'off';

%%
% Update the experiments with the estimated parameter values.
%
Exp = setEstimatedValues(Exp,v);

%%
% Simulate the model and compare model outputs with measured experiment
% data.
%
Error = [];
for ct=1:numel(Exp)

    Simulator = createSimulator(Exp(ct),Simulator);
    Simulator = sim(Simulator);

    SimLog = find(Simulator.LoggedData,get_param('sdoBattery','SignalLoggingName'));
    Voltage = find(SimLog,'Voltage');

    VoltageError = evalRequirement(r,Voltage.Values,Exp(ct).OutputData(1).Values);

    Error = [Error; VoltageError(:)];
end

%%
% Return the residual errors to the optimization solver.
%
vals.F = Error(:);
end

```

Estimate the Parameters

Use the `sdo.optimize` function to estimate the battery parameter values.

Specify the optimization options. The estimation function `sdoBattery_Objective` returns the error residuals between simulated and experimental data and does not include any constraints, making this problem ideal for the 'lsqnonlin' solver.

```

opt = sdo.OptimizeOptions;
opt.Method = 'lsqnonlin';

```

Estimate the parameters.

```
vOpt = sdo.optimize(estFcn,v,opt)
```

```
Optimization started 14-Aug-2017 23:58:01
```

Iter	F-count	f(x)	Step-size	First-order optimality
0	7	3272.22	1	
1	14	619.356	0.1634	3.15e+05
2	21	411.131	0.2175	28.7
3	28	405.529	0.3838	2.16e+03
4	35	403.727	0.2767	15.2
5	42	403.379	0.1645	1.14e+03

```
Local minimum possible.
```

```
lsqnonlin stopped because the final change in the sum of squares relative to its initial value is less than the selected value of the function tolerance.
```

```
vOpt(1,1) =
```

```
    Name: 'V'  
    Value: 1.3083  
  Minimum: 0  
  Maximum: 2  
    Free: 1  
    Scale: 2  
    Info: [1x1 struct]
```

```
vOpt(2,1) =
```

```
    Name: 'K'  
    Value: 0.0010  
  Minimum: 1.0000e-06  
  Maximum: 0.1000  
    Free: 1  
    Scale: 0.0020  
    Info: [1x1 struct]
```

```
vOpt(3,1) =
```

```
    Name: 'Loss'  
    Value: 5.1801e-05  
  Minimum: 0
```

```

Maximum: 0.5000
Free: 1
Scale: 0.0156
Info: [1x1 struct]

```

```
3x1 param.Continuous
```

Compare the Measured Output and the Final Simulated Output

Update the experiments with the estimated parameter values.

```
Exp = setEstimatedValues(Exp,vOpt);
```

Obtain the simulated output for the first (discharging) experiment.

```

Simulator = createSimulator(Exp(1), Simulator);
Simulator = sim(Simulator);
SimLog = find(Simulator.LoggedData, get_param('sdoBattery', 'SignalLoggingName'));
Voltage(1) = find(SimLog, 'Voltage');

```

Obtain the simulated output for the second (charging) experiment.

```

Simulator = createSimulator(Exp(2), Simulator);
Simulator = sim(Simulator);
SimLog = find(Simulator.LoggedData, get_param('sdoBattery', 'SignalLoggingName'));
Voltage(2) = find(SimLog, 'Voltage');

```

Plot the measured and simulated data.

The simulation results match the experimental data well except in the regions when the battery is fully charged. This is not unexpected as the simple battery model does not model the exponential voltage drop when the battery is fully charged.

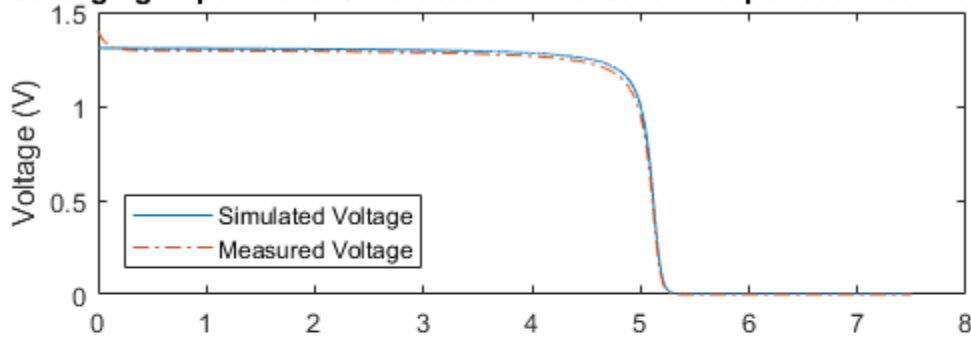
```

subplot(211)
plot(...
    Voltage(1).Values.Time/3600, Voltage(1).Values.Data, ...
    Exp(1).OutputData.Values.Time/3600, Exp(1).OutputData.Values.Data, '-.')
title('Discharging Experiment: Simulated and Measured Responses After Estimation')
ylabel('Voltage (V)')
legend('Simulated Voltage', 'Measured Voltage', 'Location', 'SouthWest')
subplot(212)
plot(...
    Voltage(2).Values.Time/3600, Voltage(2).Values.Data, ...

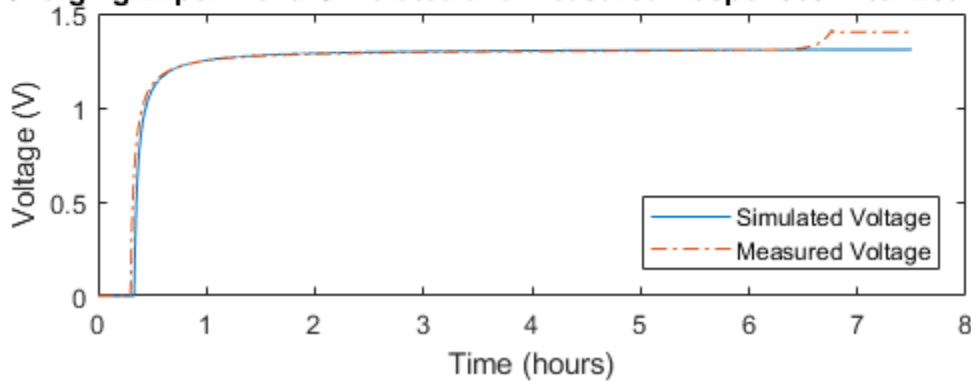
```

```
Exp(2).OutputData.Values.Time/3600, Exp(2).OutputData.Values.Data, '-.')
title('Charging Experiment: Simulated and Measured Responses After Estimation')
xlabel('Time (hours)')
ylabel('Voltage (V)')
legend('Simulated Voltage', 'Measured Voltage', 'Location', 'SouthEast')
```

Discharging Experiment: Simulated and Measured Responses After Estimator



Charging Experiment: Simulated and Measured Responses After Estimator



Update the Model Parameter Values

Update the model v , K , and $Loss$ parameter values.

```
sdo.setValueInModel('sdoBattery', vOpt);
```

Related Examples

To learn how to estimate the battery parameters using the Parameter Estimation Tool, see "Estimate Model Parameters Per Experiment (GUI)".

Close the model

```
bdclose('sdoBattery')
```

Estimate Model Parameters with Parameter Constraints (Code)

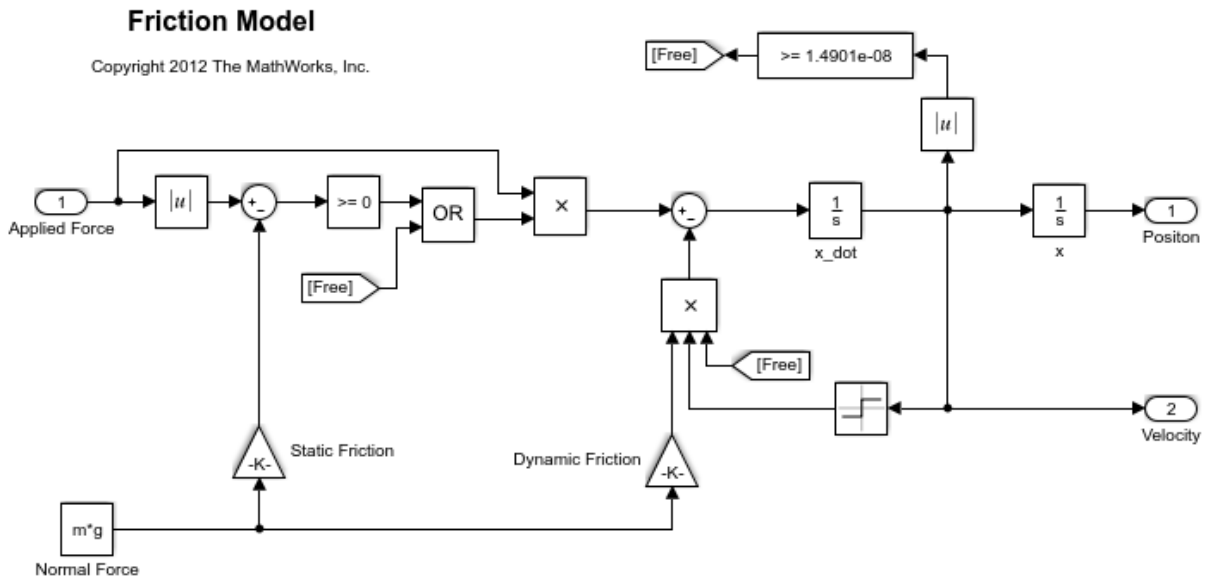
This example shows how to estimate model parameters while imposing constraints on the parameter values.

You estimate dynamic and static friction coefficients of a simple friction system.

Open the Model and Get Experimental Data

This example estimates parameters for a simple friction system, `sdoFriction`. The model input is the force applied to a mass and the model outputs are the mass position and velocity.

```
open_system('sdoFriction');
```



The model is based on a mass sliding on a surface. The mass is subject to a static friction that must be overcome before the mass moves and a dynamic friction once the mass moves. The static friction, u_{static} , is a fraction of the mass normal force; similarly the dynamic friction, u_{dynamic} , is a fraction of the mass normal force.

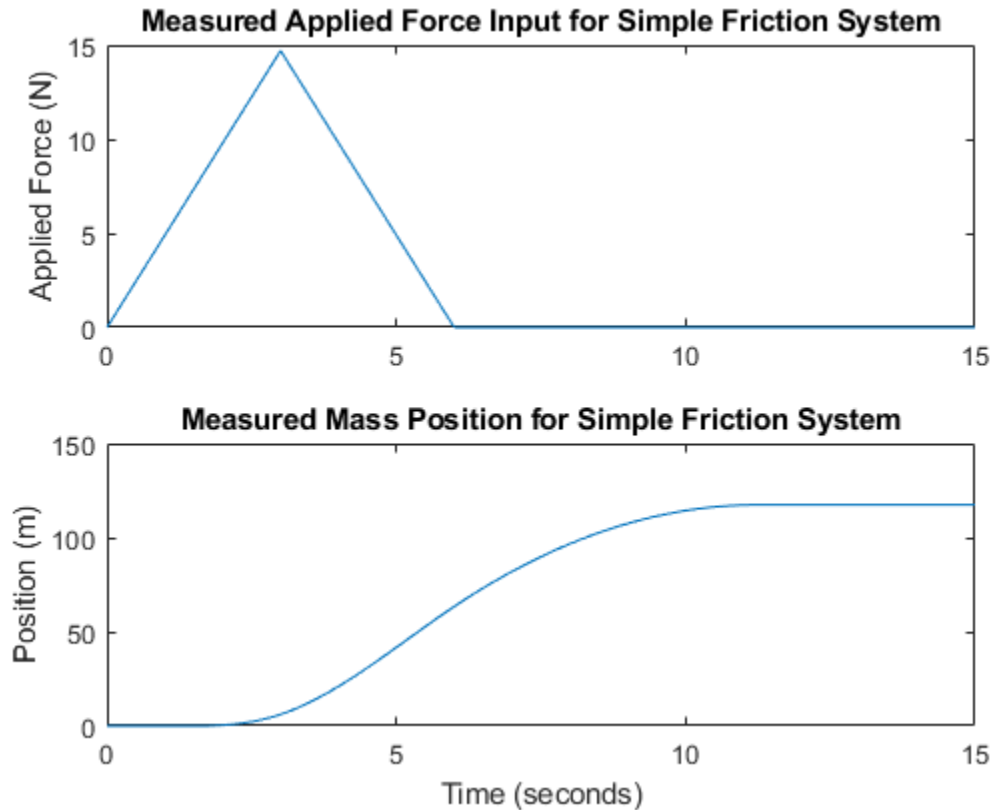
Load the experiment data. The mass was subjected to an applied force and its position recorded.

```
load sdoFriction_ExperimentData
```

The variables `AppliedForce`, `Position`, and `Velocity` are loaded into the workspace. The first column of each of these variables represents time and the second column represents the measured data. Because velocity is the first derivative of position, we only use the position measurements for this example.

Plot the Experiment Data

```
subplot(211),  
plot(AppliedForce(:,1),AppliedForce(:,2))  
title('Measured Applied Force Input for Simple Friction System');  
ylabel('Applied Force (N)')  
subplot(212)  
plot(Position(:,1),Position(:,2))  
title('Measured Mass Position for Simple Friction System');  
xlabel('Time (seconds)')  
ylabel('Position (m)')
```



Define the Estimation Experiment

Create an experiment object to specify the experiment data.

```
Exp = sdo.Experiment('sdoFriction');
```

Specify the input data (applied force) as a timeseries object.

```
Exp.InputData = timeseries(AppliedForce(:,2),AppliedForce(:,1));
```

Create an object to specify the measured mass position output.

```
PositionSig = Simulink.SimulationData.Signal;  
PositionSig.Name = 'Position';  
PositionSig.BlockPath = 'sdoFriction/x';
```

```
PositionSig.PortType = 'outport';
PositionSig.PortIndex = 1;
PositionSig.Values = timeseries(Position(:,2),Position(:,1));
```

Add the measured mass position data to the experiment as the expected output data.

```
Exp.OutputData = PositionSig;
```

Compare the Measured Output and the Initial Simulated Output

Create a simulation scenario using the experiment and obtain the simulated output.

```
Simulator = createSimulator(Exp);
Simulator = sim(Simulator);
```

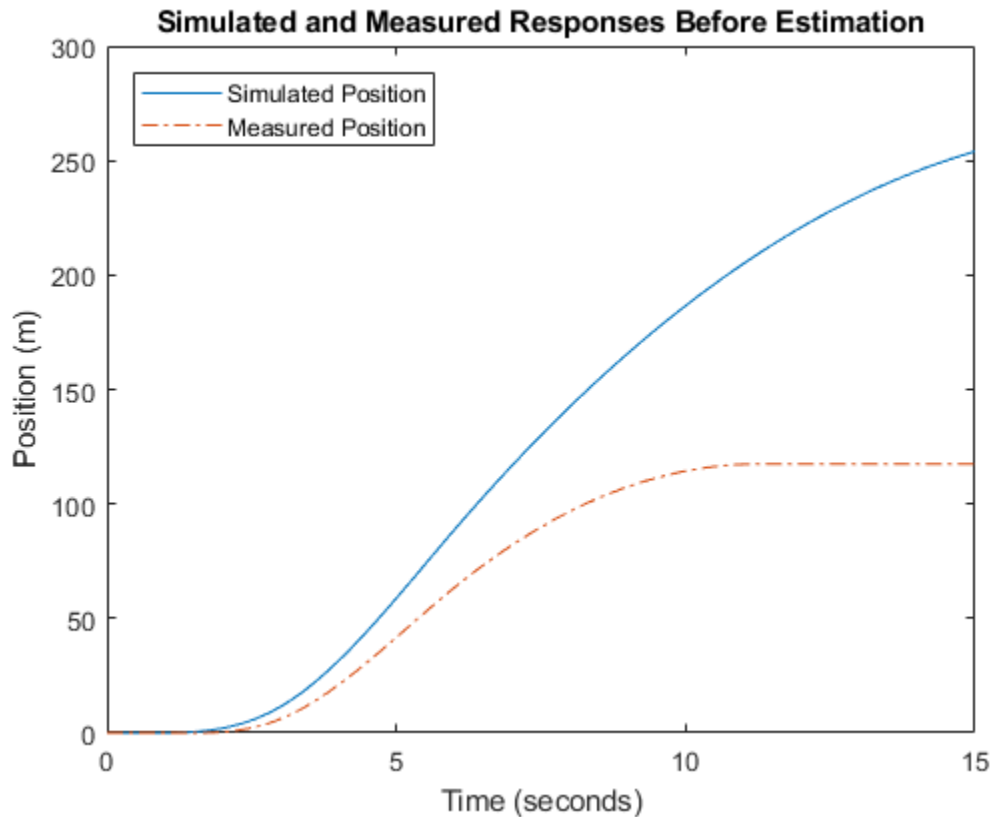
Search for the position signal in the logged simulation data.

```
SimLog = find(Simulator.LoggedData,get_param('sdoFriction','SignalLoggingName'));
Position = find(SimLog,'Position');
```

Plot the measured and simulated data.

As expected, the model response does not match the experimental output data.

```
figure
plot(...
    Position.Values.Time,Position.Values.Data, ...
    Exp.OutputData.Values.Time, Exp.OutputData.Values.Data,'-.')
title('Simulated and Measured Responses Before Estimation')
ylabel('Position (m)')
xlabel('Time (seconds)')
legend('Simulated Position','Measured Position','Location','NorthWest')
```



Specify Parameters to Estimate

Estimate the `u_static` and `u_dynamic` friction coefficients using the experiment data. These coefficients are used as gains in the `Static Friction` and `Dynamic Friction` blocks, respectively. Physics indicates that friction coefficients should be constrained so that $u_{\text{static}} \geq u_{\text{dynamic}}$; this parameter constraint is implemented in the estimation objective function.

Select the `u_static` and `u_dynamic` model parameters. Specify bounds for the estimated parameter values. Both coefficients are limited to the range [0 1].

```
p = sdo.getParameterFromModel('sdoFriction',{'u_static','u_dynamic'});
```

```
p(1).Minimum = 0;  
p(1).Maximum = 1;  
  
p(2).Minimum = 0;  
p(2).Maximum = 1;
```

Define the Estimation Objective

Create an estimation objective function to evaluate how closely the simulation output, generated using the estimated parameter values, matches the measured data.

Use an anonymous function with one input argument that calls the `sdoFriction_Objective` function. We pass the anonymous function to `sdo.optimize`, which evaluates the function at each optimization iteration.

```
estFcn = @(v) sdoFriction_Objective(v, Simulator, Exp);
```

The `sdoFriction_Objective` function:

- Has one input argument that specifies the estimated friction coefficients.
- Has one input argument that specifies the experiment object containing the measured data.
- Returns the sum-squared-error errors between simulated and experimental outputs, and returns the parameter constraint.

The `sdoFriction_Objective` function requires two inputs, but `sdo.optimize` requires a function with one input argument. To work around this, `estFcn` is an anonymous function with one input argument, `v`, but it calls `sdoFriction_Objective` using two input arguments, `v` and `Exp`.

For more information regarding anonymous functions, see "Anonymous Functions".

The `sdo.optimize` command minimizes the return argument of the anonymous function `estFcn`, that is, the residual errors returned by `sdoFriction_Objective`. For more details on how to write an objective/constraint function to use with the `sdo.optimize` command, type `help sdoExampleCostFunction` at the MATLAB command prompt.

To examine the estimation objective function in more detail, type `edit sdoFriction_Objective` at the MATLAB command prompt.

```
type sdoFriction_Objective
```

```
function vals = sdoFriction_Objective(p, Simulator, Exp)
%SDOFRICTION_OBJECTIVE
%
% The sdoFriction_Objective function is used to compare model
% outputs against experimental data and measure how well constraints are
% satisfied.
%
% vals = sdoFriction_Objective(p, Exp)
%
% The |p| input argument is a vector of estimated model parameter values.
%
% The |Simulator| input argument is a simulation object used
% simulate the model with the estimated parameter values.
%
% The |Exp| input argument contains the estimation experiment data.
%
% The |vals| return argument contains information about how well the
% model simulation results match the experimental data and how well
% constraints are satisfied. The |vals| argument is used by the
% |sdo.optimize| function to estimate the model parameters.
%
% See also sdo.optimize, sdoExampleCostFunction, sdoFriction_cmddemo
%

% Copyright 2012-2015 The MathWorks, Inc.

%%
% Define a signal tracking requirement to compute how well the model output
% matches the experiment data. Configure the tracking requirement so that
% it returns the sum-squared-error.
%
r = sdo.requirements.SignalTracking;
r.Type = '==';
r.Method = 'SSE';

%%
% Update the experiments with the estimated parameter values.
%
Exp = setEstimatedValues(Exp, p);

%%
% Simulate the model and compare model outputs with measured experiment
```

```

% data.
%
Simulator = createSimulator(Exp, Simulator);
Simulator = sim(Simulator);

SimLog = find(Simulator.LoggedData, get_param('sdoFriction', 'SignalLoggingName'));
Position = find(SimLog, 'Position');

PositionError = evalRequirement(r, Position.Values, Exp.OutputData(1).Values);

%%
% Measure how well the parameters satisfy the friction coefficient constraint,
% |u_static| >= |u_dynamic|. Note that constraints are returned to the
% optimizer in a c <= 0 format. The friction coefficient constraint is
% rewritten accordingly.
PConstr = p(2).Value - p(1).Value; % u_dynamic - u_static <= 0

%%
% Return the sum-squared-error and constraint violation to the optimization
% solver.
%
vals.F = PositionError(:);
vals.Cleq = PConstr;
end

```

The friction coefficient constraint, $u_{\text{static}} \geq u_{\text{dynamic}}$, is implemented in the `sdoFriction_Objective` function as $u_{\text{dynamic}} - u_{\text{static}} \leq 0$. This is because the optimizer requires constraint values in a $\mathbf{c} \leq \mathbf{0}$ format. For more information, type `help sdo.optimize` at the MATLAB command prompt.

Estimate the Parameters

Use the `sdo.optimize` function to estimate the friction model parameter values.

Specify the optimization options. The estimation function `sdoFriction_Objective` returns the sum-squared-error between simulated and experimental data and includes a parameter constraint. The default 'fmincon' solver is ideal for this type of problem.

Estimate the parameters.

```
pOpt = sdo.optimize(estFcn, p)
```

```
Optimization started 15-Aug-2017 00:02:50
```

2 Parameter Estimation

Iter	F-count	f(x)	max constraint	Step-size	First-order optimality
0	5	27.7267	0		
1	11	22.5643	0	2.21	72.9
2	15	17.4771	0	0.51	16
3	22	0.762174	0	1.33	10.7
4	29	0.40765	0	0.263	3.15
5	34	0.0254254	0	0.0897	1.22
6	39	0.00525359	0	0.0296	0.27
7	44	0.00394384	0	0.0203	0.213
8	49	0.00196687	0	0.11	0.215
9	58	0.00189047	0	0.0324	0.239
10	74	0.00180601	0	0.0135	0.231
11	86	0.00174013	0	0.015	0.217
12	103	0.00174013	0	0.00163	0.217

Local minimum possible. Constraints satisfied.

fmincon stopped because the size of the current step is less than the selected value of the step size tolerance and constraints are satisfied to within the selected value of the constraint tolerance.

pOpt(1,1) =

```
Name: 'u_static'  
Value: 0.7938  
Minimum: 0  
Maximum: 1  
Free: 1  
Scale: 0.5000  
Info: [1x1 struct]
```

pOpt(2,1) =

```
Name: 'u_dynamic'  
Value: 0.4032  
Minimum: 0  
Maximum: 1  
Free: 1  
Scale: 0.2500  
Info: [1x1 struct]
```



```
2x1 param.Continuous
```

Compare the Measured Output and the Final Simulated Output

Update the experiments with the estimated parameter values.

```
Exp = setEstimatedValues(Exp,pOpt);
```

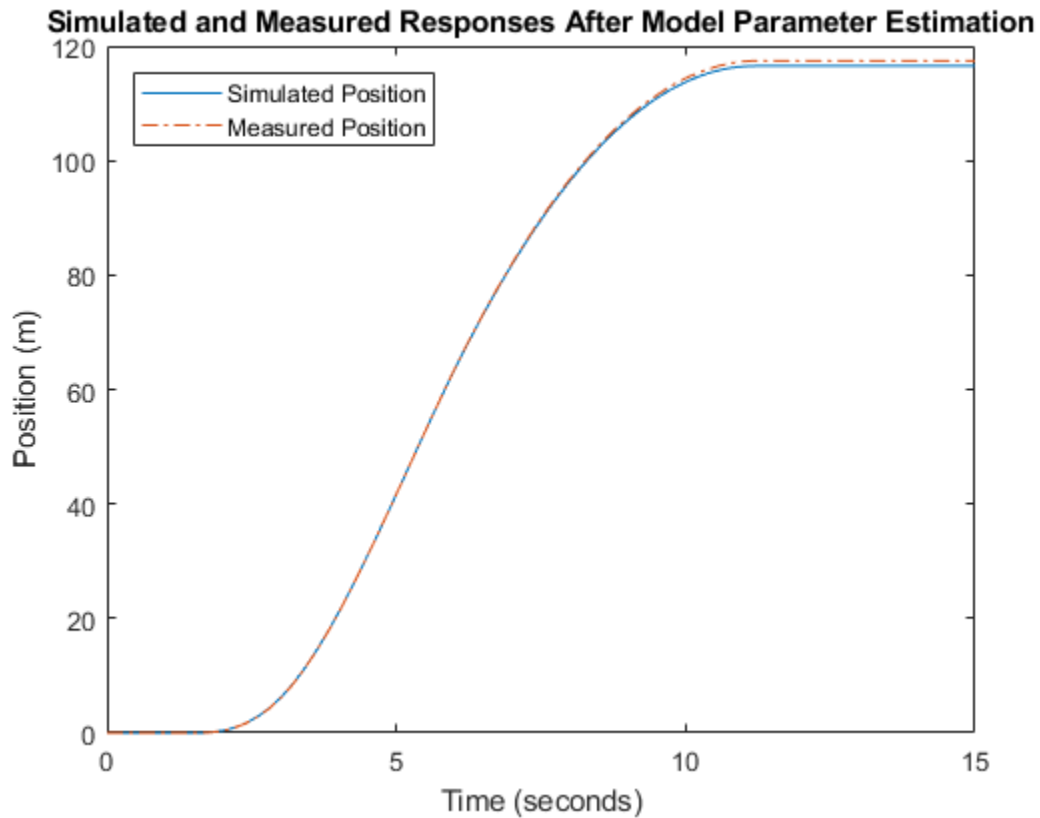
Obtain the simulated output for the experiment.

```
Simulator = createSimulator(Exp,Simulator);  
Simulator = sim(Simulator);  
SimLog = find(Simulator.LoggedData,get_param('sdoFriction','SignalLoggingName'));  
Position = find(SimLog,'Position');
```

Plot the measured and simulated data.

It can be seen that the model response using the estimated parameter values nicely matches the experiment output data.

```
plot(...  
    Position.Values.Time,Position.Values.Data, ...  
    Exp.OutputData.Values.Time, Exp.OutputData.Values.Data,'-.')  
title('Simulated and Measured Responses After Model Parameter Estimation')  
ylabel('Position (m)')  
xlabel('Time (seconds)')  
legend('Simulated Position','Measured Position','Location','NorthWest')
```



Update the Model Parameter Values

Update the model `u_static` and `u_dynamic` parameter values.

```
sdo.setValueInModel('sdoFriction', pOpt);
```

Close the model

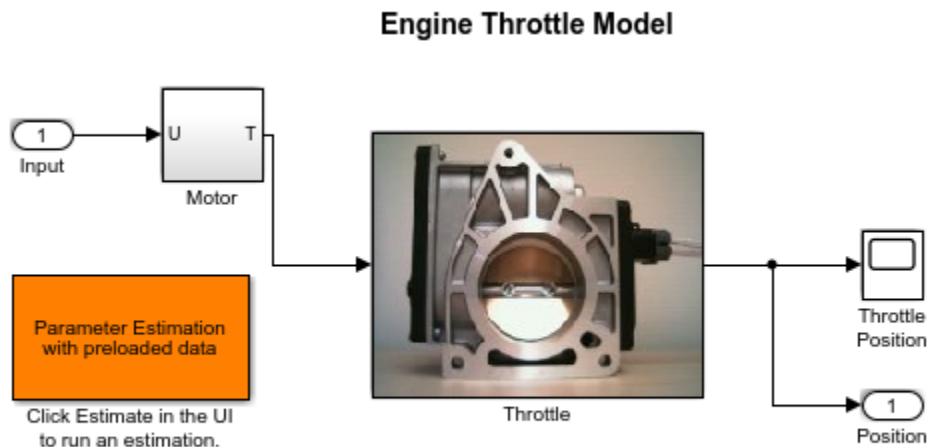
```
bdclose('sdoFriction')
```

Estimate Model Parameter Values (GUI)

This example shows how to use experiment data to estimate model parameters. You estimate the parameters of an engine throttle system.

Simulink® Model of the Engine Throttle System

The Simulink® model for the engine throttle system, `spe_engine_throttle`, is shown below.



Throttle Model Description

The throttle controls the air mass flow into the intake manifold of an engine. The throttle body contains a butterfly valve that opens when the driver presses down on the accelerator pedal. This lets more air enter the cylinders and causes the engine to produce more torque.

A DC motor controls the opening angle of the butterfly valve. There is also a spring attached to the valve to return it to its closed position when the DC motor is de-energized. The amount of rotation of the valve is limited to approximately 90 degrees. Therefore, if a large command input is applied to the motor, the valve hits the hard stops preventing it from rotating further.

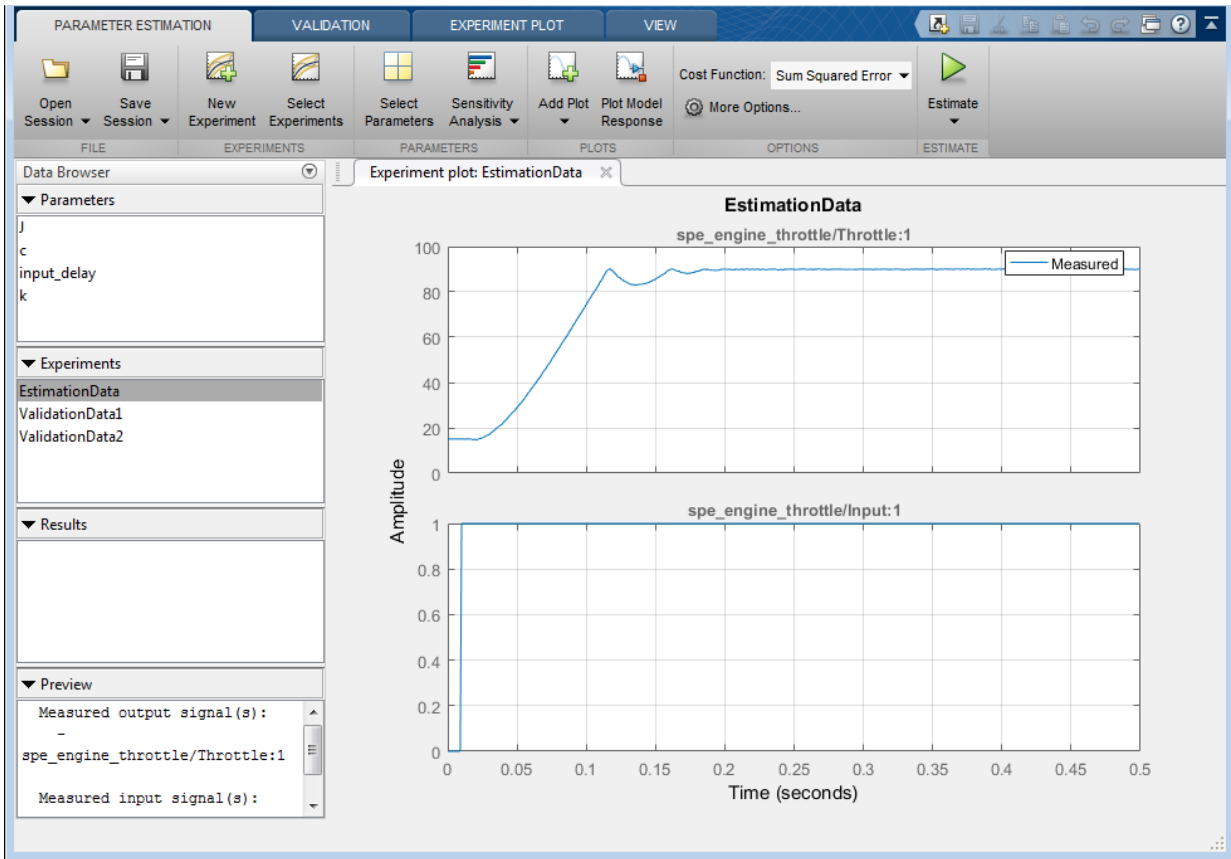
The motor is modeled as a torque gain and a time-delay input with parameters `Kt` and `input_delay`. The butterfly valve is modeled as a mass-spring-damper system with parameters `J`, `c` and `k`. This system is augmented with hard stops to limit the valve opening to 90 degrees. We know the model components, however, the parameter values of the system are not known accurately.

Estimation Experiment Data

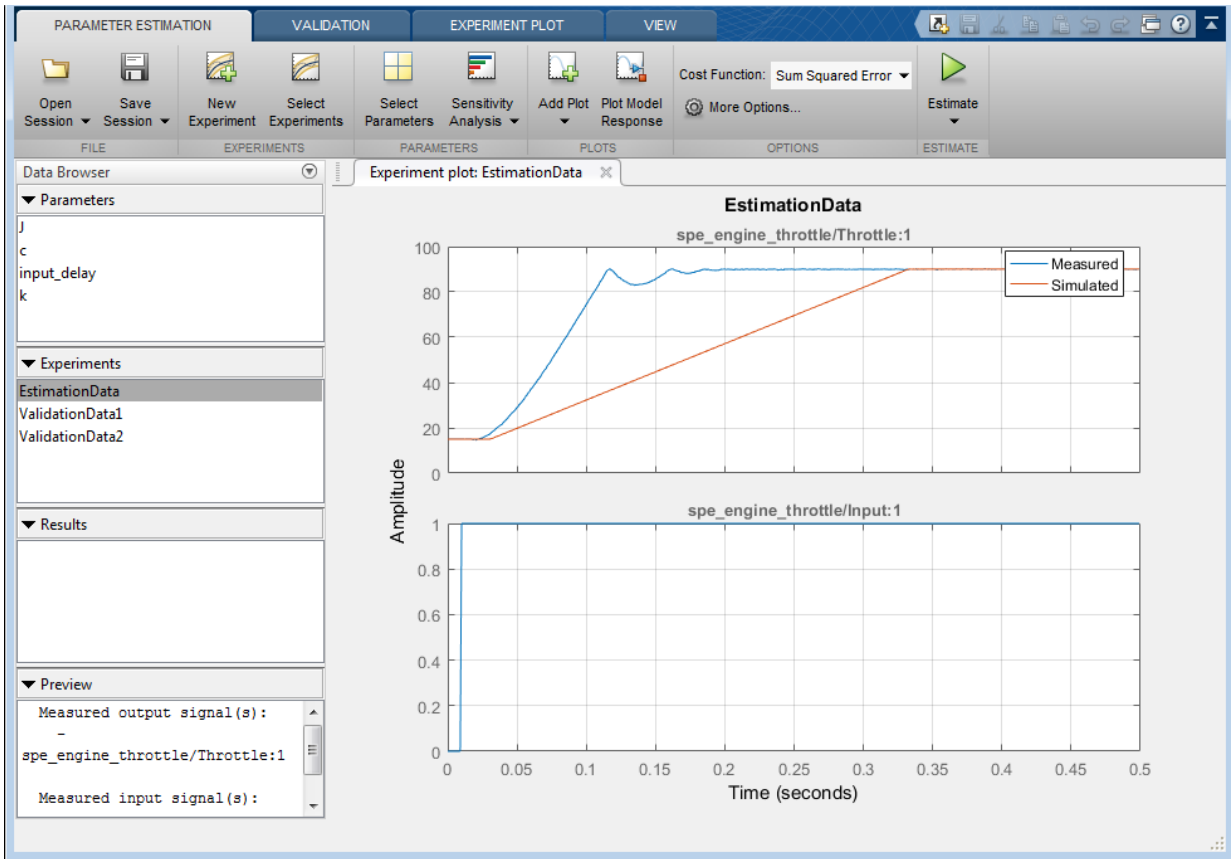
Double-click the `Parameter Estimation GUI` with preloaded data block in the model to open a pre-configured estimation GUI session.

The saved estimation project defines three experiments; the `EstimationData` experiment will be used for parameter estimation, while `ValidationData1`, `ValidationData2` are used for validating the estimated parameters. The `EstimateData` experiment is plotted.

The signal data for the experiments can be imported from various sources including MATLAB® variables, MAT files, Excel® files, or comma-separated-value files. See "Importing and Preprocessing Experiment Data (GUI)" for more information.



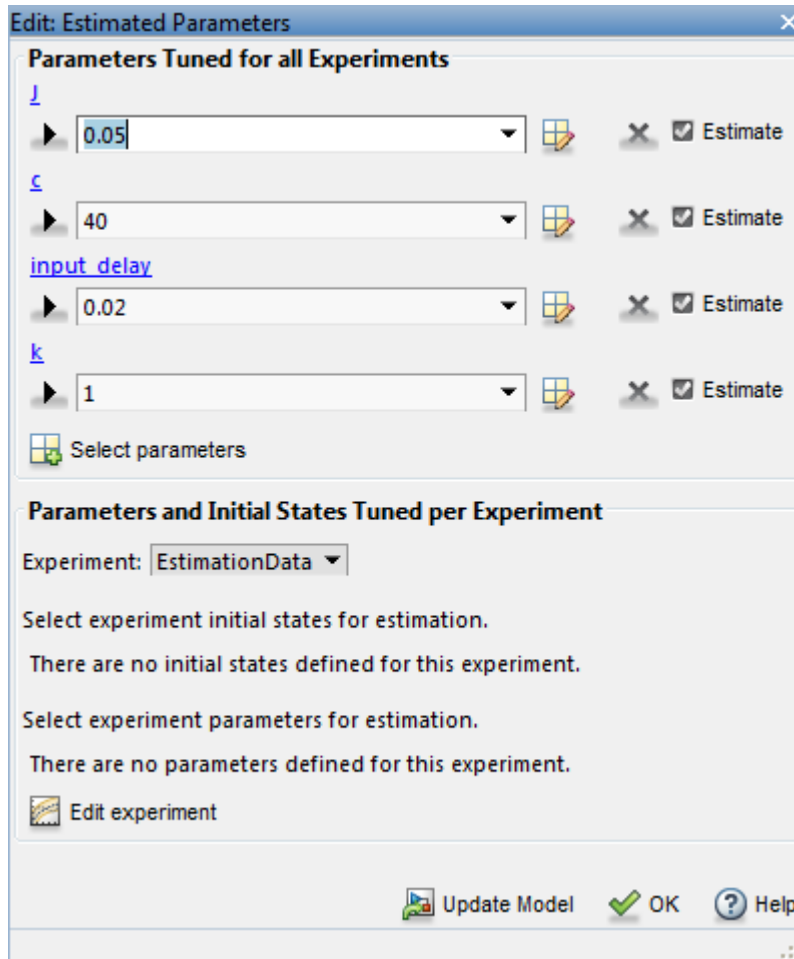
The experiment plot is also used to see how well the measured data matches the current model. Click **Plot Model Response** to display simulated signal data on the experiment plots.



The simulation results show that the model does not match the measured data and that model parameters need to be estimated.

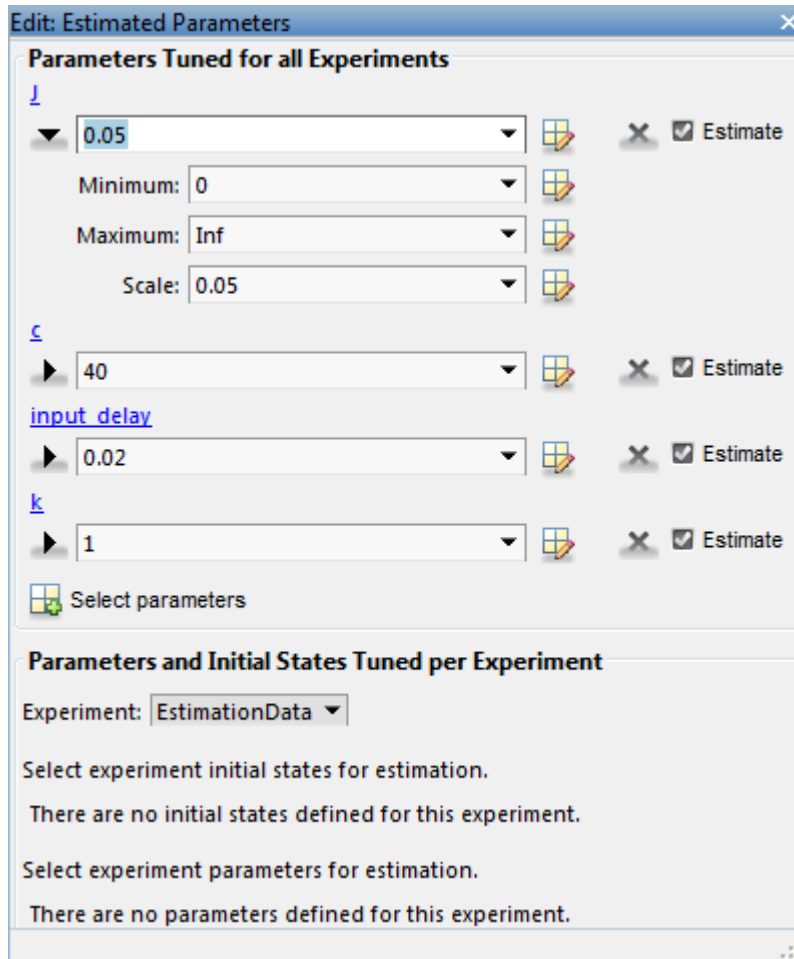
Estimated Parameters

The next step is to define the parameter to estimate. Click **Select Parameters** to open a dialog to select model parameters to estimate. In this example we have preselected the four unknown parameters; the butterfly valve inertia, J ; the damping coefficient, c ; the return spring constant, k ; and the time lag in motor response, `input_delay`.



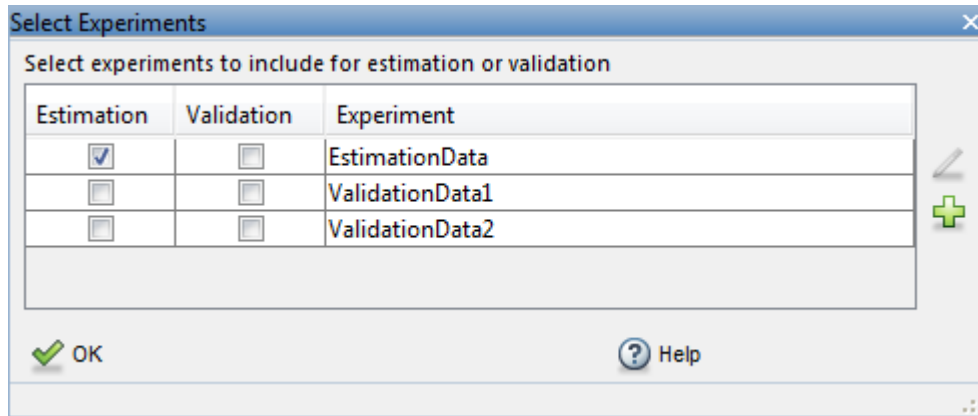
Since we know from physical insight that all of these parameters have positive values, we set their lower limits to zero. We also put an upper bound of 0.1 sec on the `input_delay` parameter. We can also select an initial value for the parameters. These may come from some quick calculations of some formulas that determine the parameters.

Click the right arrow toggle button to modify the parameter minimum and maximum bounds.



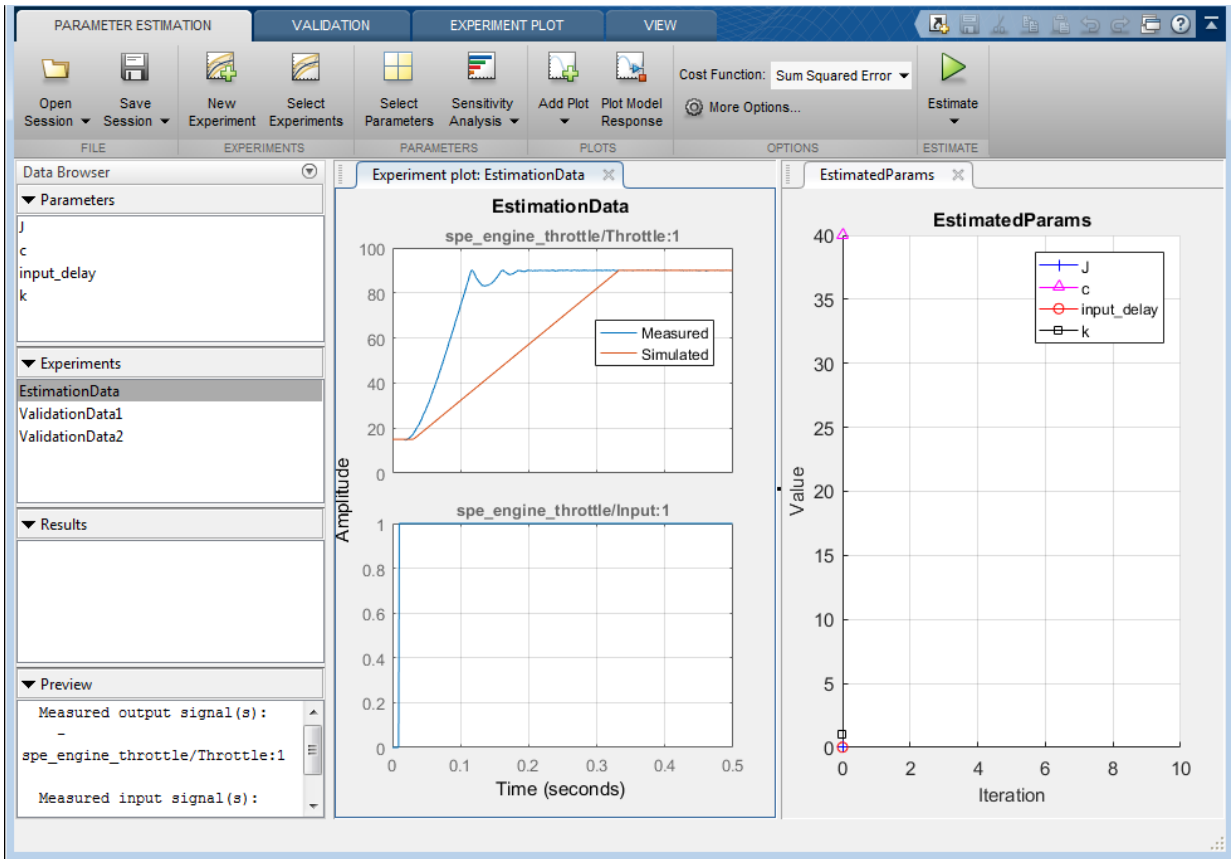
The Estimation Task

With the parameters for estimation selected we select experiments to use for estimation. Click **Select Experiments** and select `EstimationData` for estimation.



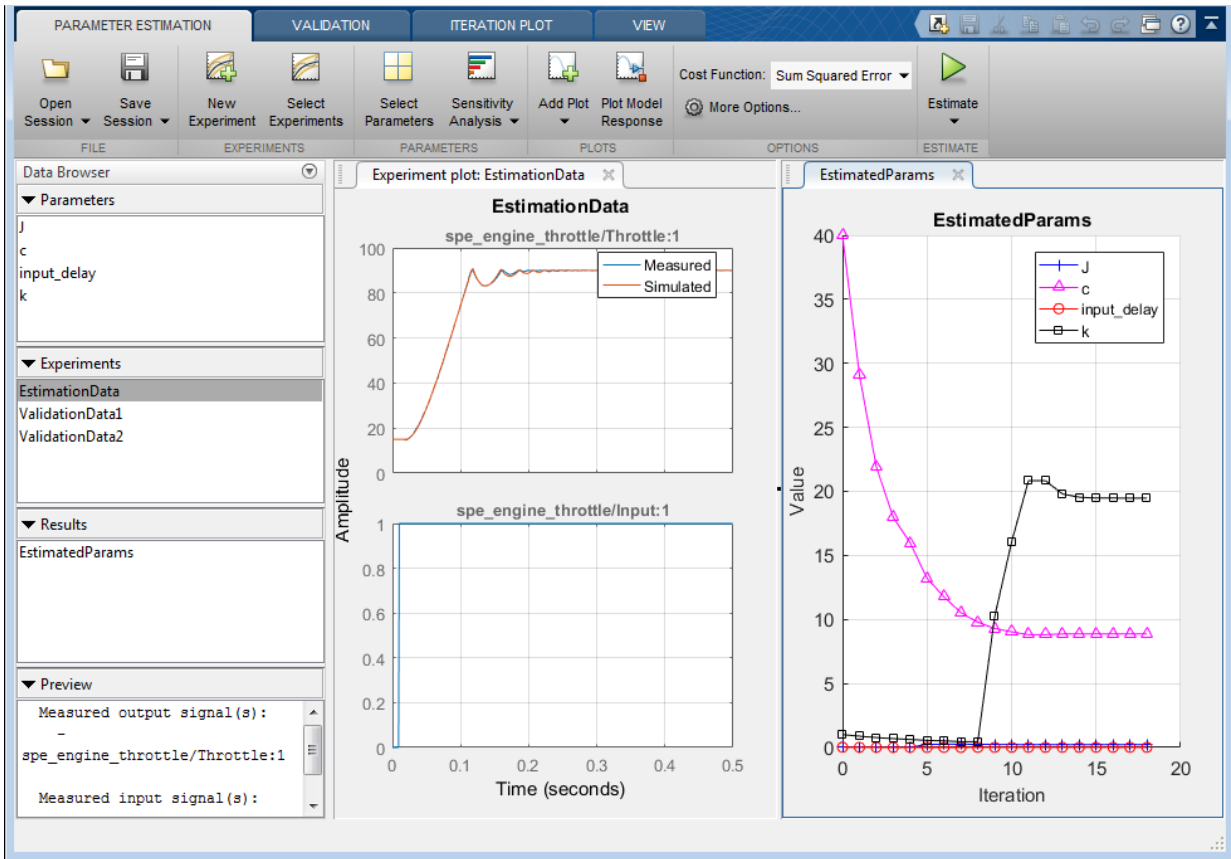
We are now almost ready to start our estimation but first create plots to monitor the estimation progress. Click **Add Plot** and select **Parameter Trajectory**. This creates a plot that shows how the estimated parameter values change during estimation. Click the **View** tab to layout the plots so that the `Experiment plot:EstimationData` and `EstimatedParams` are both visible.

2 Parameter Estimation



Click the **Estimate** button to start the estimation. You can modify estimation options by setting the **Cost Function** combobox and clicking **More Options...**

While the estimation is running the plots update and a dialog showing estimation progress appears. The progress dialog shows the estimation iterations, the number of times the model has been evaluated (F-count), and the estimation cost at each iteration.



Iteration	F-count	EstimationData (Minimize)
0	9	32.04
1	18	12.30
2	27	3.65
3	36	1.13
4	45	0.66
5	54	0.32
6	63	0.11
7	72	0.02
8	81	0.00
9	90	0.00
10	99	0.00
11	108	0.00

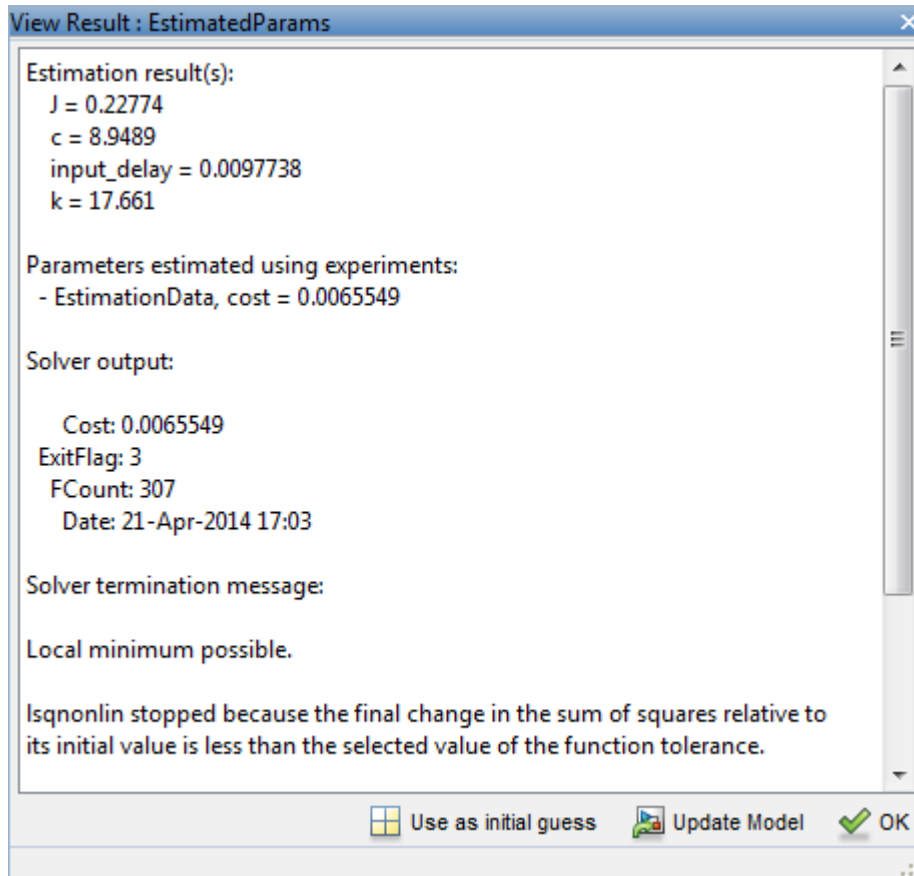
Optimization started 21-Apr-2014 16:50:09

Estimation converged, 21-Apr-2014 16:52:25

Estimated experiment values written to the workspace

Save Iteration... Display Options... Estimate

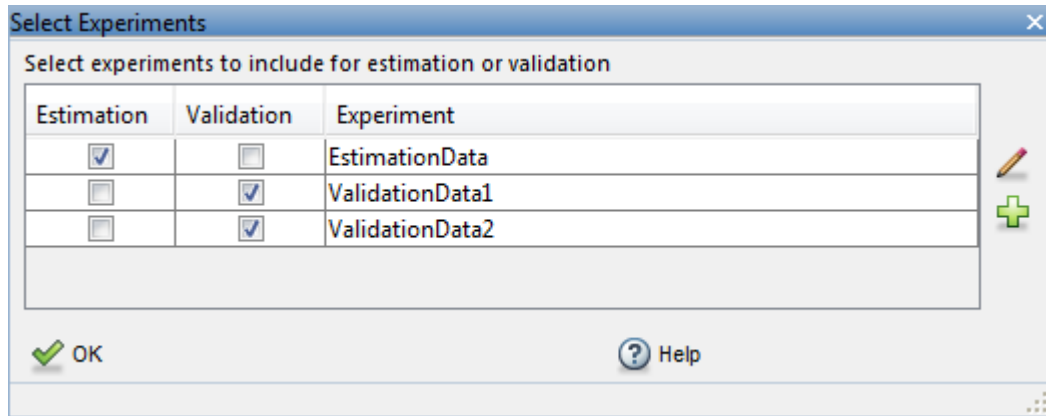
After a number of iterations the estimation converges and terminates. The model is updated with the estimated parameters and the estimation results are saved in the data browser. Right click `EstimatedParams` and select **Open...** to see details of the estimation result.



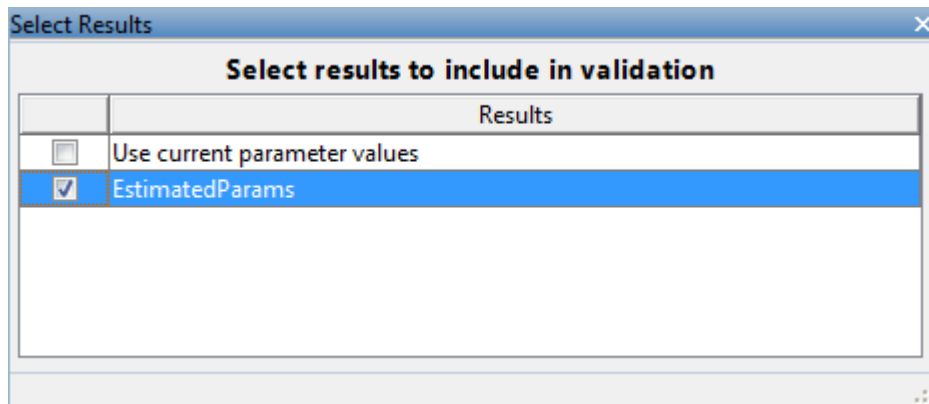
Validation

It is important to validate the estimation results against other experiments. A successful estimation will not only match the experimental data that was used for estimation but also other independent measured data that were collected in experiments.

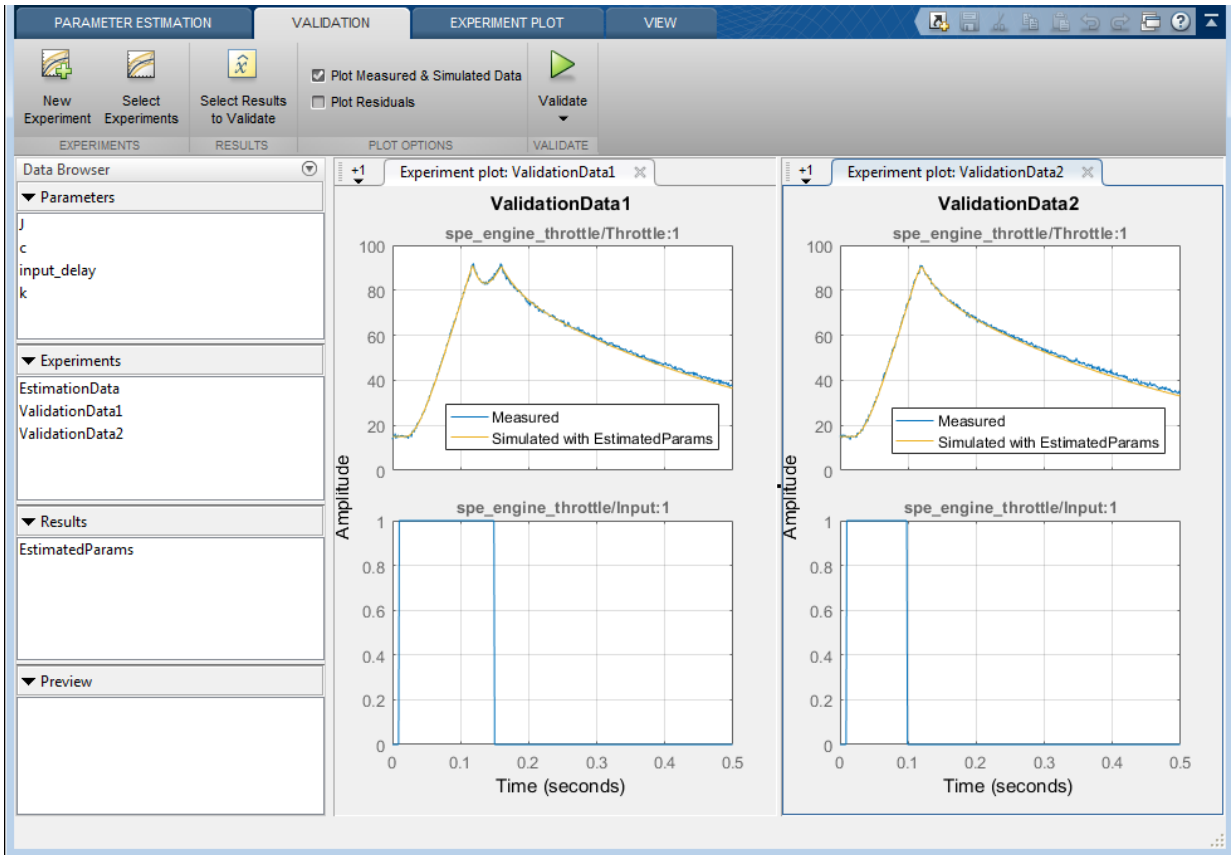
Click the **Validation** tab and click **Select Experiments** to select experiments for validation. Select both `ValidationData1` and `ValidationData2` for validation.



Click **Select Results** to select the estimation result(s) to use for validation. Select EstimatedParams and deselect Use current parameter values.



Click **Validate** to validate the estimation result against the validation experiments. Validation simulates the model using the estimated parameters and selected experiments and creates plots showing the measured and simulation data. Use the **View** tab to layout the plots so that the Experiment plot:ValidationData1 and Experiment plot:ValidationData2 are both visible.



The validation plots confirm that our estimation was successful, showing that the estimated parameters are robust enough to handle a variety of inputs.

Related Examples

To learn how to estimate model parameters using the `sdo.optimize` command, see "Estimate Model Parameter Values (Code)".

Close the model.

Estimate Model Parameters Per Experiment (GUI)

This example shows how to use multiple experiments to estimate a mix of model parameter values; some that are estimated using all the experiments and others that are estimated using individual experiments. The example also shows how to configure estimation experiments with experiment dependent parameter values.

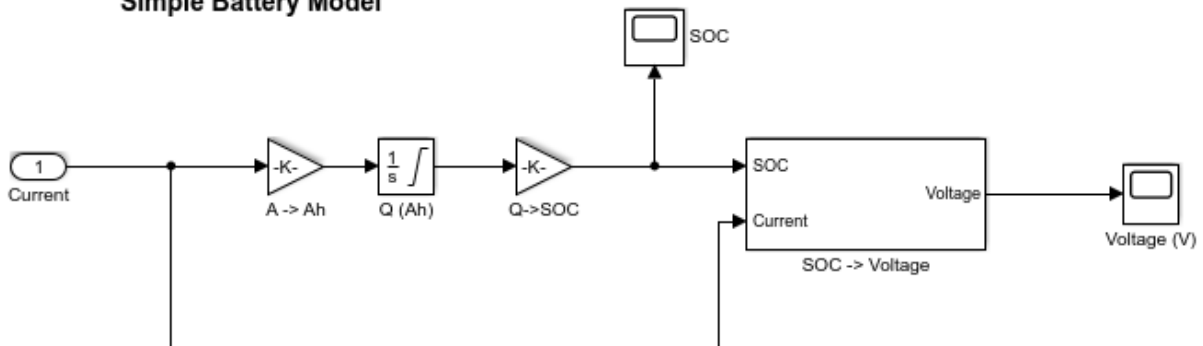
You estimate the parameters of a rechargeable battery based on data collected in experiments that discharge and charge the battery.

Open the Model and Get Experimental Data

This example estimates parameters of a simple, rechargeable battery model, `sdoBattery`. The model input is the battery current and the model output, the battery terminal voltage, is computed from the battery state-of-charge.

```
open_system('sdoBattery');
```

Simple Battery Model



Copyright 2012-2014 The MathWorks, Inc.

The model is based on the equation

$$E = (1 - Loss) * V - K * Q_{max} * \frac{1 - s}{s}$$

Where:

- E is the battery terminal voltage in Volts.
- V is the battery constant voltage in Volts.
- K is the battery polarization resistance in Ohms.
- Q_{max} is the maximum battery capacity in Ampere-Hour.
- s is the battery charge state, with 1 being fully charged and 0 zero charge. The battery state-of-charge is computed from the integral of the battery current with a +ve current indicating discharge and a -ve current indicating charging. The battery initial state-of-charge is specified by Q_0 in Ampere-Hour.
- $Loss$ is the voltage drop when charging, expressed as a fraction of the battery constant voltage. When the battery is discharging this value is zero.

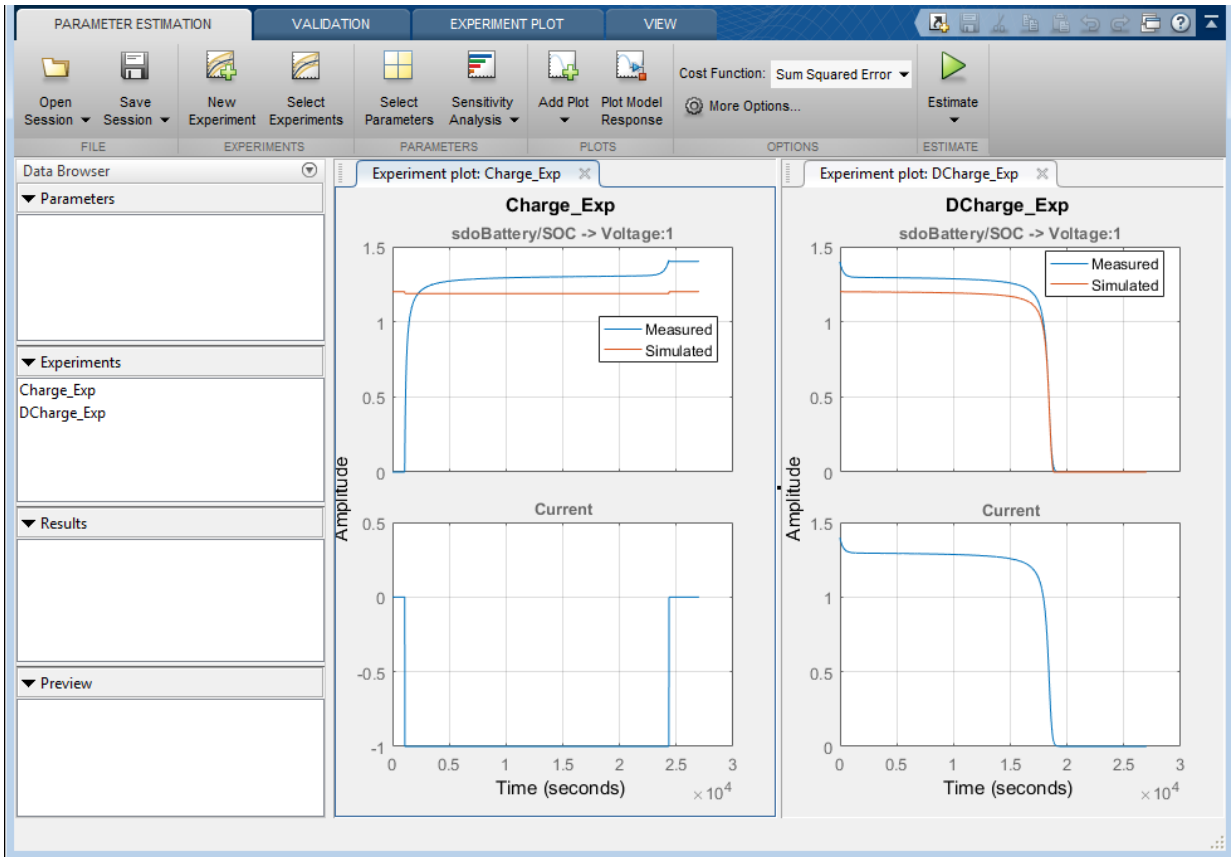
V , K , Q_{max} , Q_0 , and $Loss$ are variables defined in the model workspace.

Estimation Experiment Data

A 1.2V (6500mAh) battery was subjected to a discharge and a charging experiment. This experiment data has been loaded into a preconfigured estimation tool session.

From the `sdoBattery` model click **Analysis** and select the **Parameter Estimation...** menu item to launch the Parameter Estimation tool. From the Parameter Estimation tool click **Open Session** and select **Open from model workspace** and open the `sdoBattery_sdoSession` session. The measured charge and discharge experiment data are loaded and plotted.

Click the **View** tab to layout the plots so that the `Experiment_plot:Charge_Exp` and `Experiment_plot:DCharge_Exp` are both visible. Click **Plot Model Response** to see how well the model simulation matches the measured experiment data.

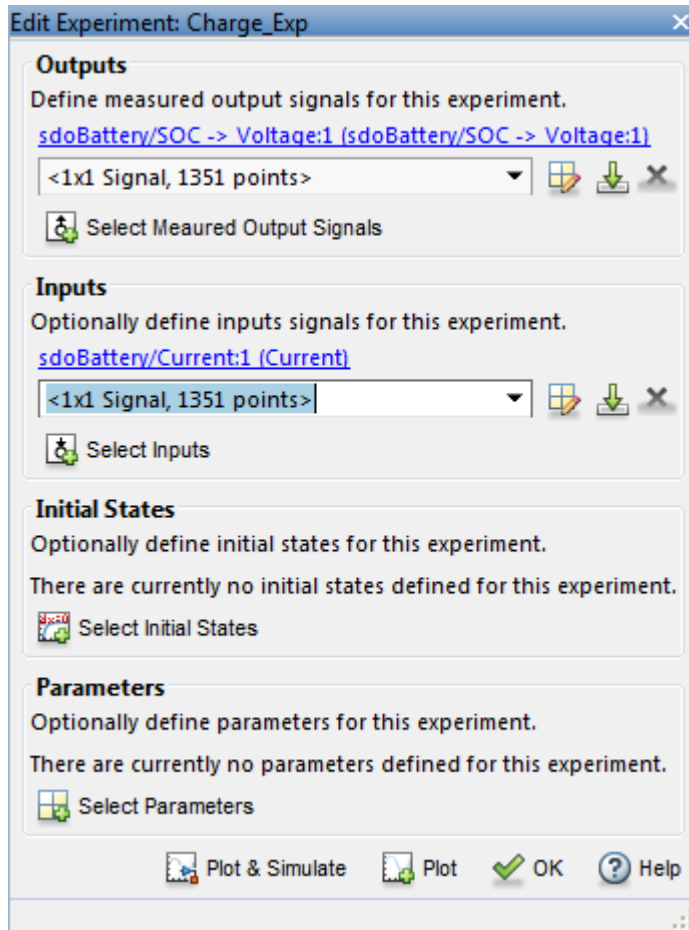


The plots show that the battery initial charge Q_0 is not set correctly for the Charge_Exp experiment and that the model V , K , and $Loss$ parameters need to be estimated.

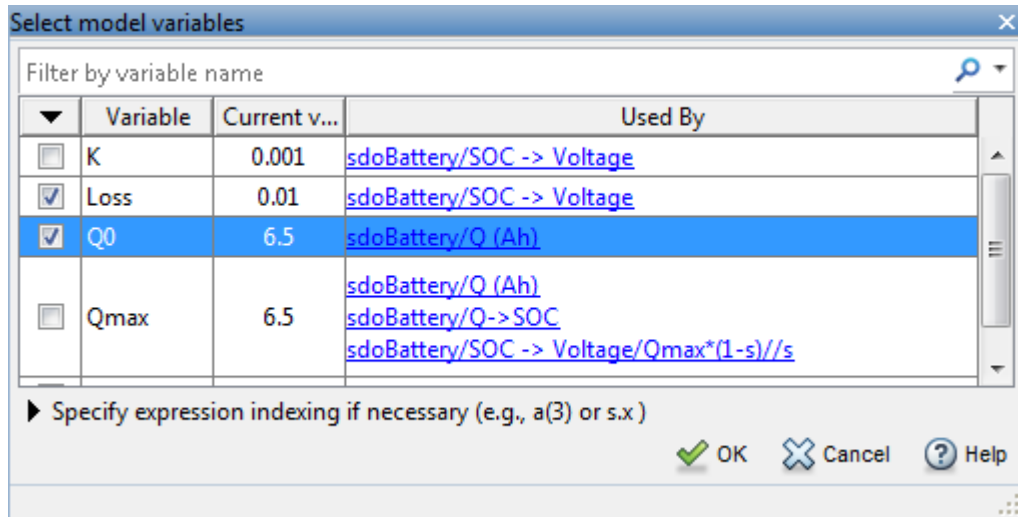
Setting Experiment Parameter Values

The previous plot indicates that the Charge_Exp battery initial charge, Q_0 , is not set correctly. Add the initial charge to both experiments.

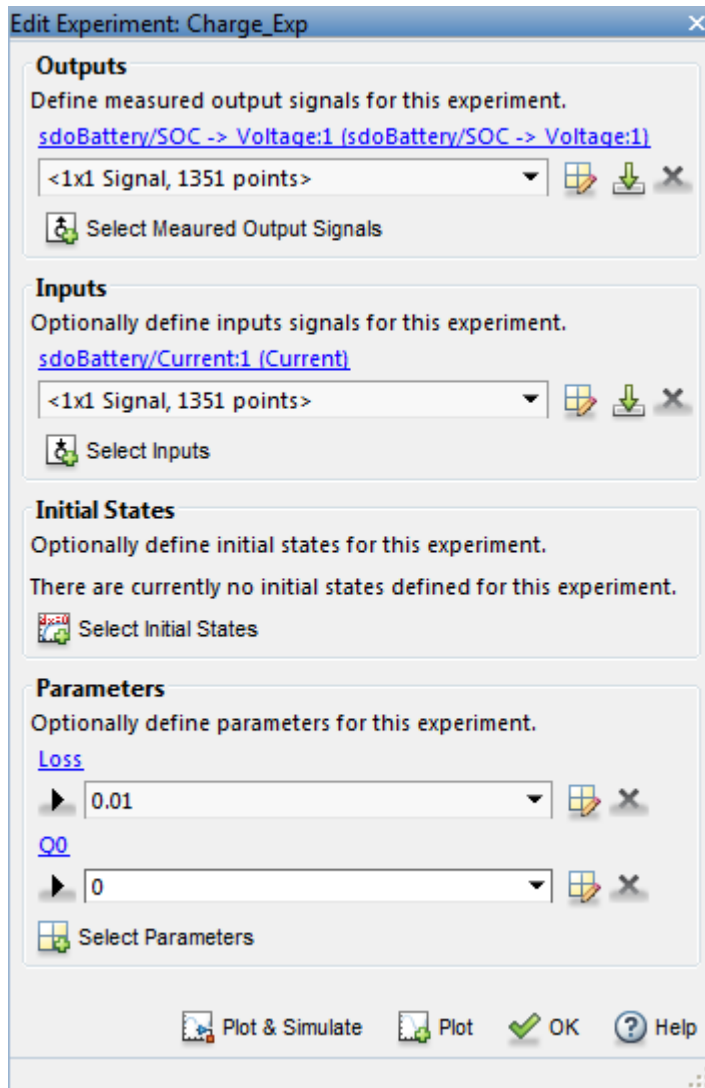
Right click Charge_Exp and select **Edit....** A dialog to edit the experiment opens.



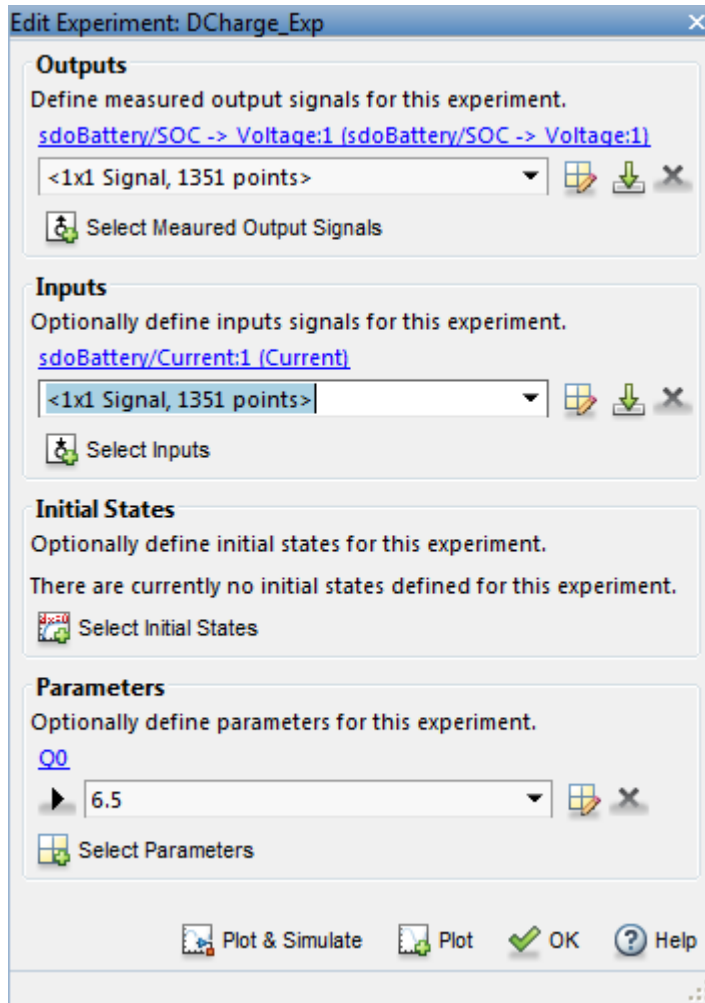
Click **Select Parameters** to open a dialog to add model parameters to the experiment. Select `Loss` and `Q0` to add to the experiment. Select `Loss` as we need to estimate this parameter using only the `Charge_Exp` experiment. Click **Ok** to add the `Q0` and `Loss` parameters to the experiment.



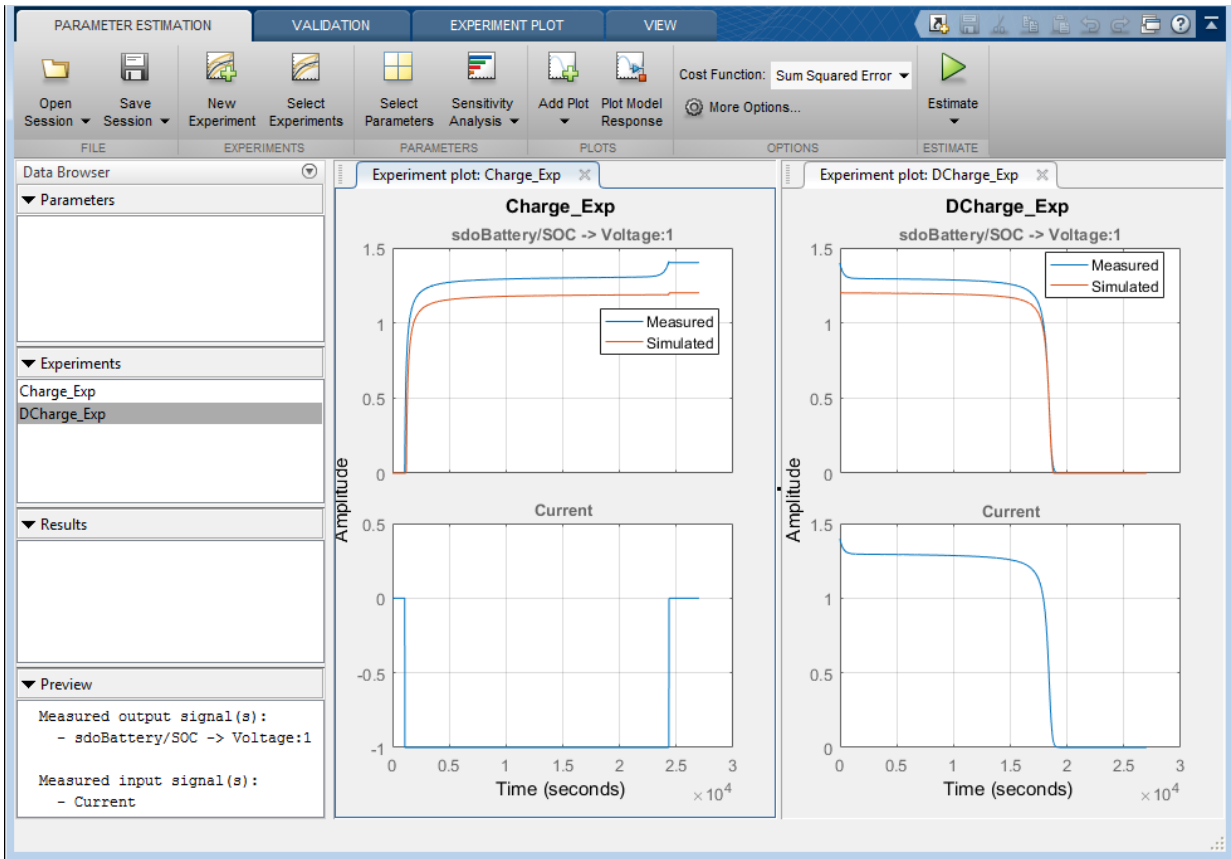
Set the battery initial charge Q_0 in the `Charge_Exp` to 0, i.e. there is no initial charge.



Similarly add the battery initial charge Q_0 to the `DCharge_Exp` experiment and set the initial charge to 6.5., i.e. for this experiment there is an initial charge.



Now that the experiments are updated with the correct initial battery charge click **Plot Model Response** to simulate the model and compare measured and simulated data.

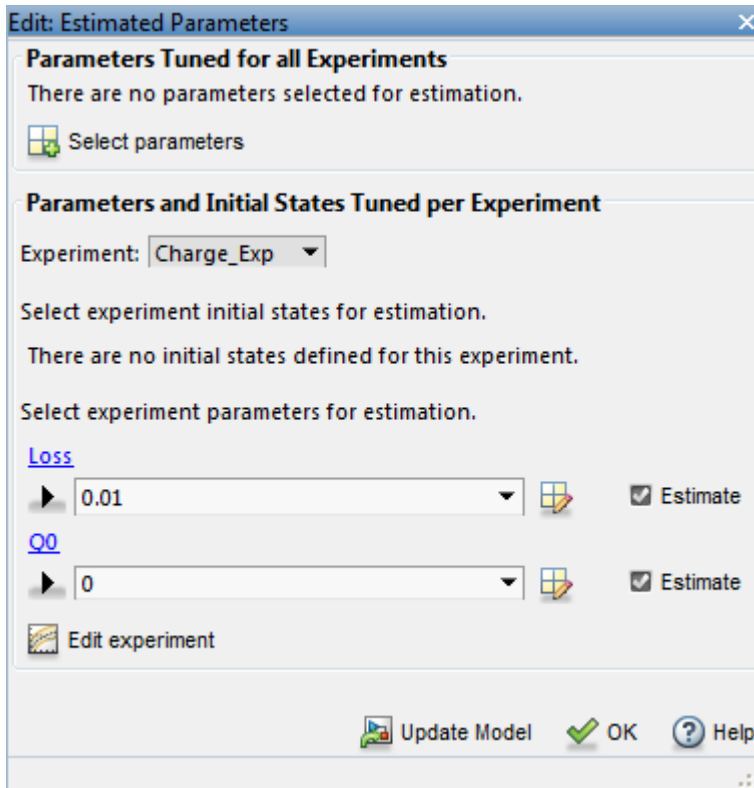


The experiment plots show that the experiment initial conditions match but the battery response does not. The next step is to estimate the κ and V model parameters.

Select Estimation Parameters

The previous plot showed that the model response does not match the measured data and we need to estimate the model V and κ parameters.

Click **Select Parameters** to open a dialog to select model parameters.



The upper portion of the select parameters dialog has a section for parameters that are tuned using all experiments. Click **Select Parameters** and add the v and κ model parameters to the estimated parameters. Set the v minimum to 0 and the maximum to 2, similarly set the κ minimum to $1e-6$ and maximum to 0.1.

Edit: Estimated Parameters [X]

Parameters Tuned for all Experiments

K

▼ 0.001 [X] Estimate

Minimum: 1e-06 [X]

Maximum: 0.1 [X]

Scale: 0.001953125 [X]

V

▼ 1.2 [X] Estimate

Minimum: 0 [X]

Maximum: 2 [X]

Scale: 2 [X]

[X] Select parameters

Parameters and Initial States Tuned per Experiment

Experiment: Charge_Exp ▼

Select experiment initial states for estimation.

There are no initial states defined for this experiment.

Select experiment parameters for estimation.

Loss

▶ 0.01 [X] Estimate

Q0

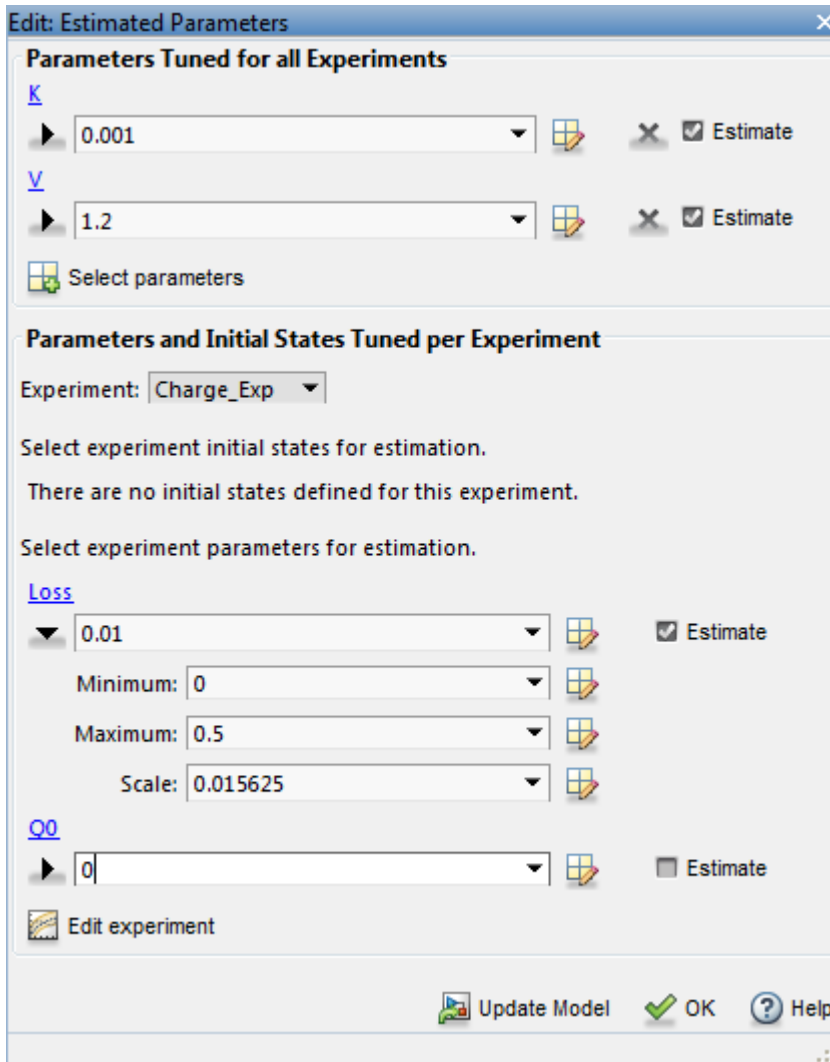
▶ 0 [X] Estimate

[X] Edit experiment

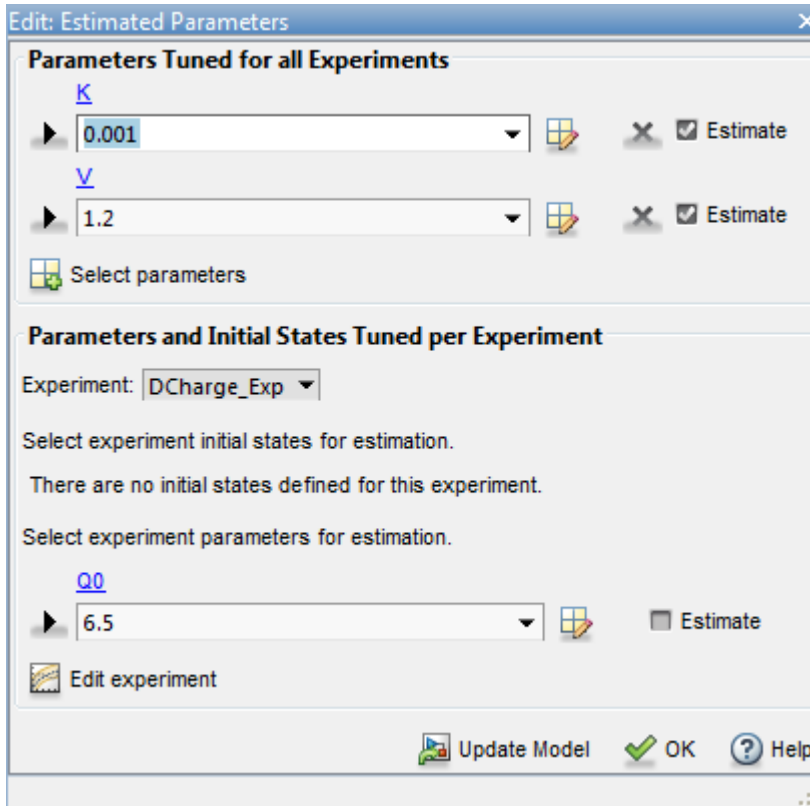
[X] Update Model [X] OK [X] Help

The lower section of the dialog has a section for initial states and parameters that are tuned using individual experiments.

For the `Charge_Exp` we tune the `Loss` parameter and set its minimum to 0 maximum to 0.5. The battery initial charge `Q0` is fixed to 0 and should not be estimated; uncheck **Estimate**.



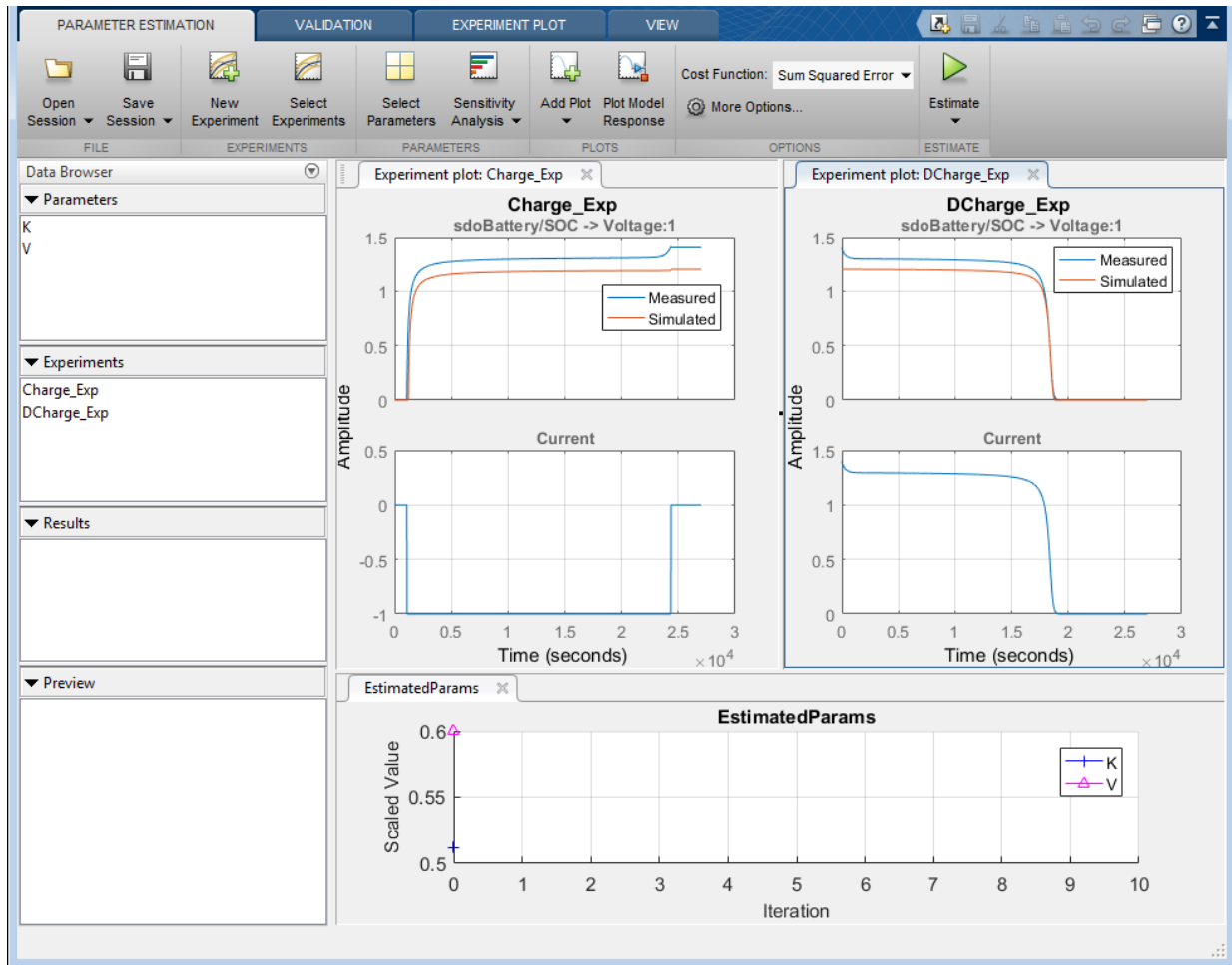
Select DCharge_Exp from the **Experiment** combobox to view the parameter settings for the DCharge_Exp experiment. The battery initial charge Q_0 is fixed to 6.5 and should not be estimated; uncheck **Estimate**



Estimate Parameter Values

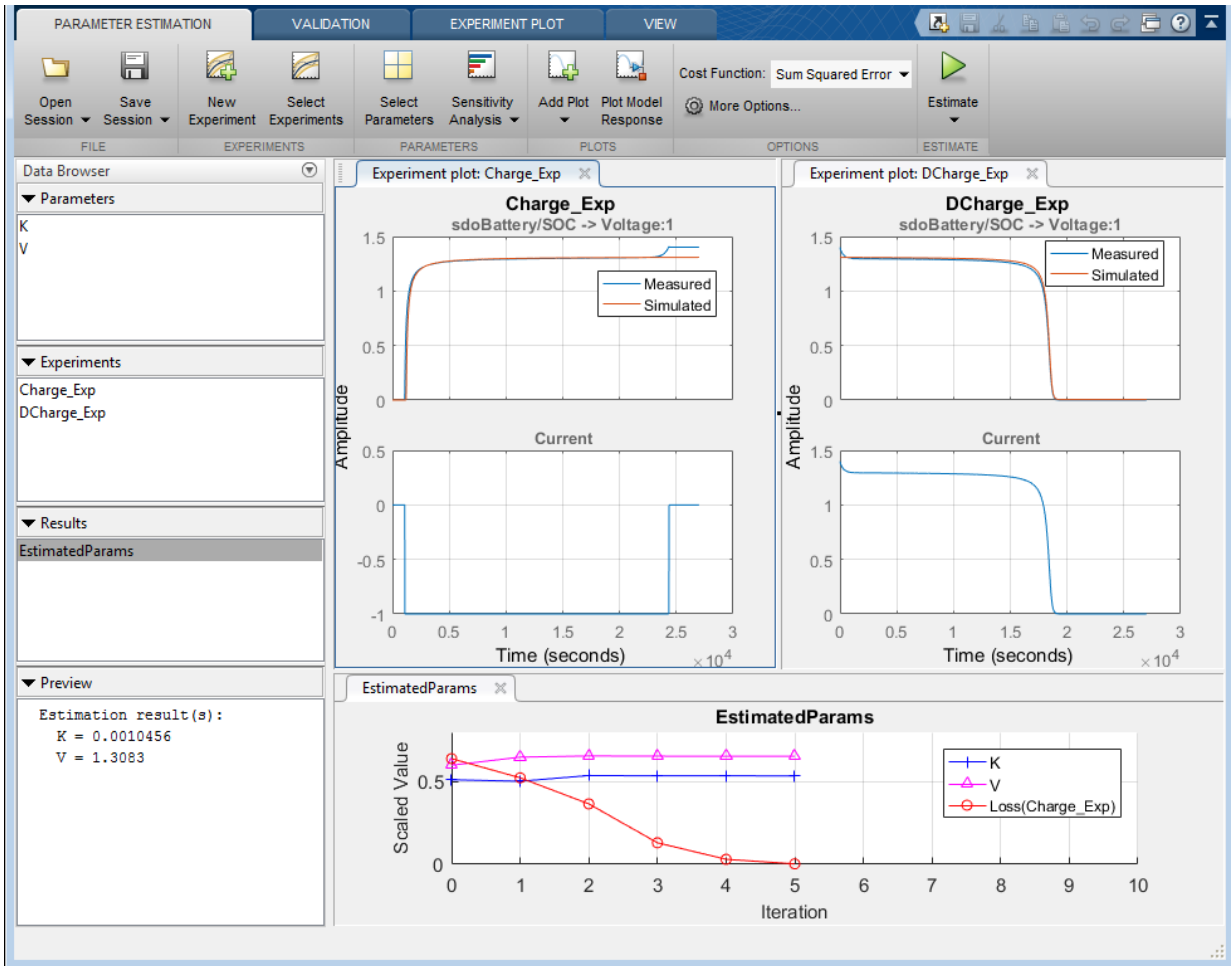
The experiments and estimated parameters are configured and we are ready to run the estimation. First create a plot to monitor the estimation progress. Click **Add Plot** and select **Parameter Trajectory**. This creates a plot that shows how the estimated parameter values change during estimation. Click the **View** tab to layout the plots so that the experiment and parameter trajectory plots are all visible.

2 Parameter Estimation



Click the **Estimate** button to start the estimation. You can modify estimation options by setting the **Cost Function** combobox and clicking **More Options...**

While the estimation is running the plots update and a dialog showing estimation progress appears. The progress dialog shows the estimation iterations, the number of times the model has been evaluated (F-count), and the estimation cost at each iteration.



Iteration	F-count	Charge_Exp (Minimize)	DCharge_Exp (Minimize)
0	7	12.6204	4.1010
1	14	2.2115	1.3692
2	21	1.9613	0.2716
3	28	1.9663	0.2372
4	35	1.9765	0.2169
5	42	1.9756	0.2164

Optimization started 23-Apr-2014 15:10:19

Estimation converged, 23-Apr-2014 15:11:15

Estimated experiment values written to the workspace

After a number of iterations the estimation converges and terminates. The experiment plots show the measured and simulation data matching well. The `EstimatedParams` plot shows the `V`, `K`, and `Loss` parameters changing during the estimation; the scale of `V` and `K`, `Loss` are different, right click on the plot and select **Show scaled values** to see how all the parameters changed from their original values.

Related Examples

To learn how to estimate parameters per experiment using the `sdo.optimize` command, see "Estimate Model Parameters Per Experiment (GUI)".

Close the model

```
bdclose('sdoBattery')
```

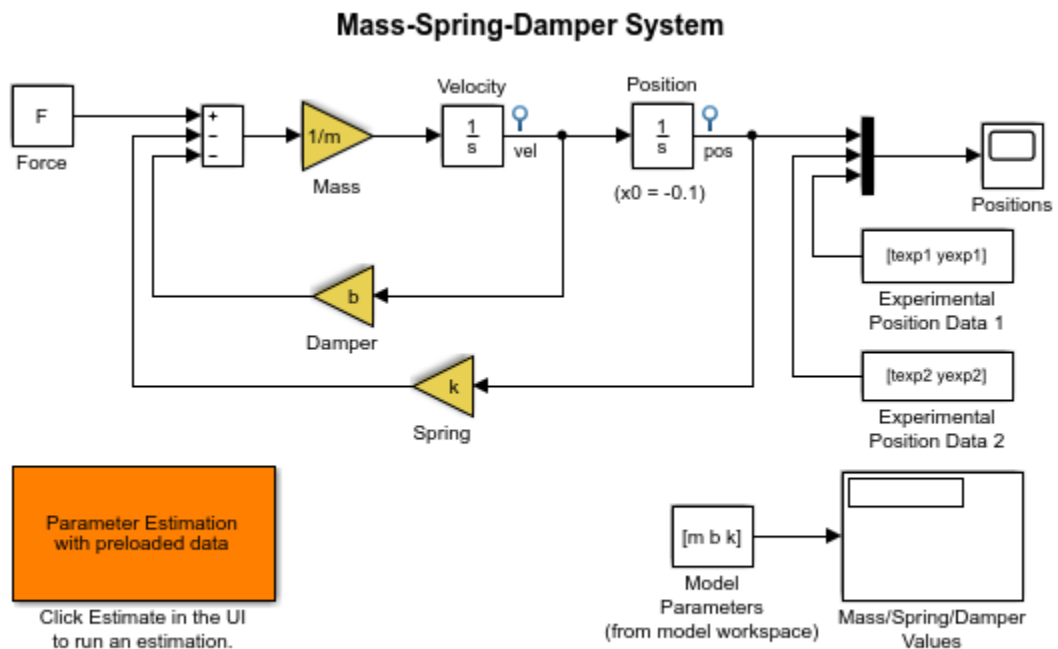
Estimate Model Parameters and Initial States (GUI)

This example shows how to estimate the physical parameters - mass (m), spring constant (k) and damping (b) of a simple mass-spring-damper model. This example illustrates the significance of initial state estimation.

Simulink® Model of the Mass Spring Damper System

The Simulink model for the mass-spring-damper system, `msd_system`, is shown below.

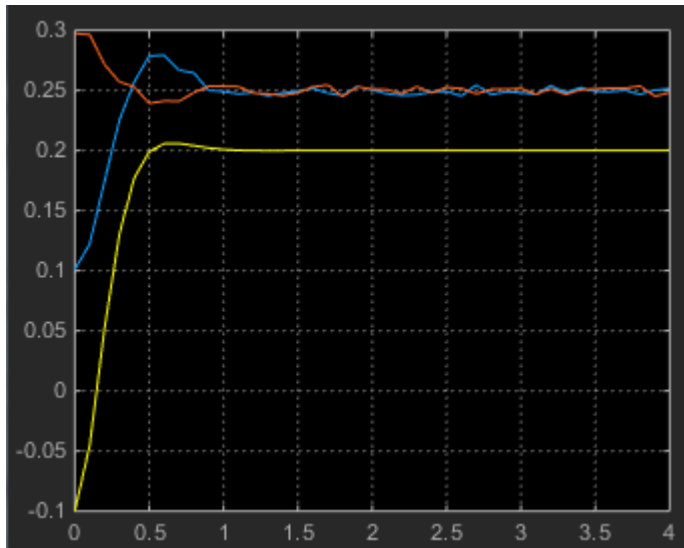
The model's output is the displacement response (position) of the mass in a mass-spring-damper system, subject to a constant force (F), and an initial displacement (x_0). x_0 is the initial condition of the Position integrator block. Run the simulation once to observe the response of the model to a nominal set of parameter values.



Copyright 2002-2014 The MathWorks, Inc.

Experimental Data Sets

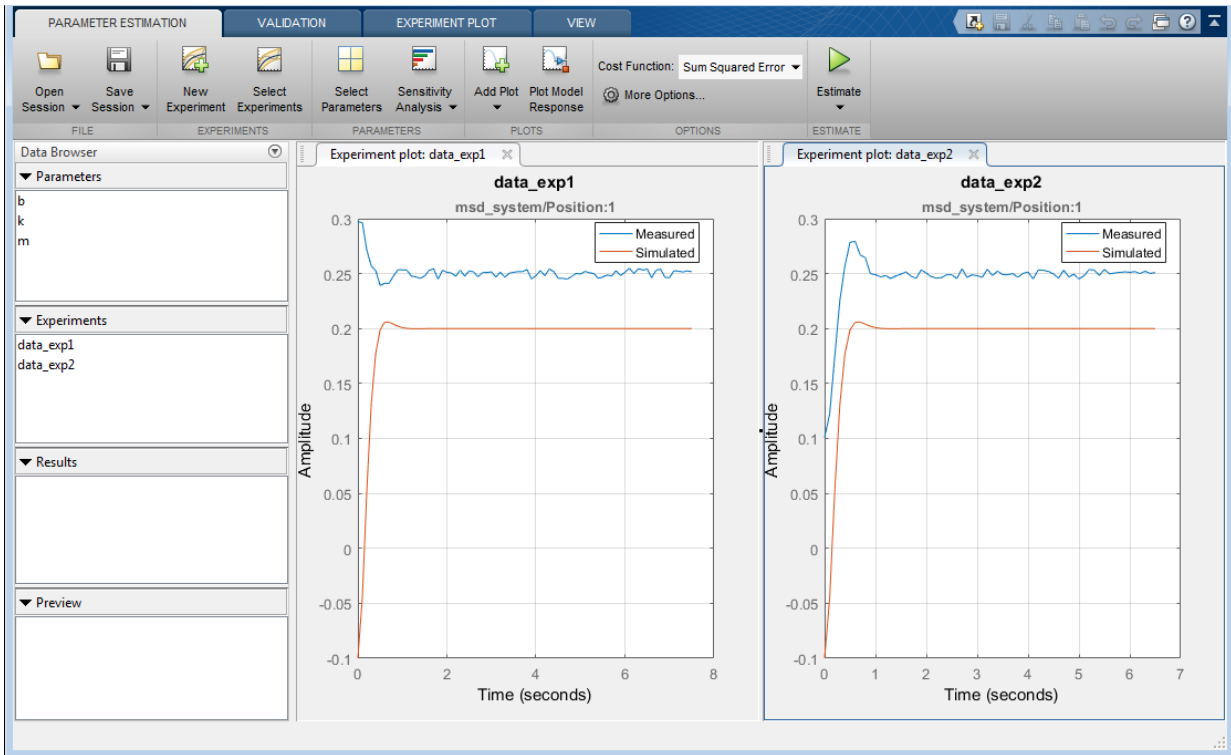
For estimation of the model parameters (m , b and k), two sets of experimental data are used. These data sets were obtained using two different initial positions (0.1 and 0.3), and contain additive noise. A plot of these data sets is shown below (orange and cyan curves), along with the simulated response (yellow curve) of the Simulink model for $x_0 = -0.1$ and a nominal set of parameter values ($m=8$, $k=500$, $b=100$).



Estimation of Model Parameters

The model has three parameters (k , b , m) that appear in the Gain blocks of the Simulink model `msd_system`. We estimate these parameters using Parameter Estimation.

Double-click the Parameter Estimation GUI with preloaded data block in the model to open a pre-configured estimation GUI session. The experimental data sets are already loaded in the project (`data_exp1` and `data_exp2`). Click the **View** tab to layout the plots so that the `Experiment plot:data_exp1` and `Experiment plot:data_exp2` are both visible. Click **Plot Model Response** to simulate the model for the two experiments. The plots show that the model simulation does not match the experiment data.

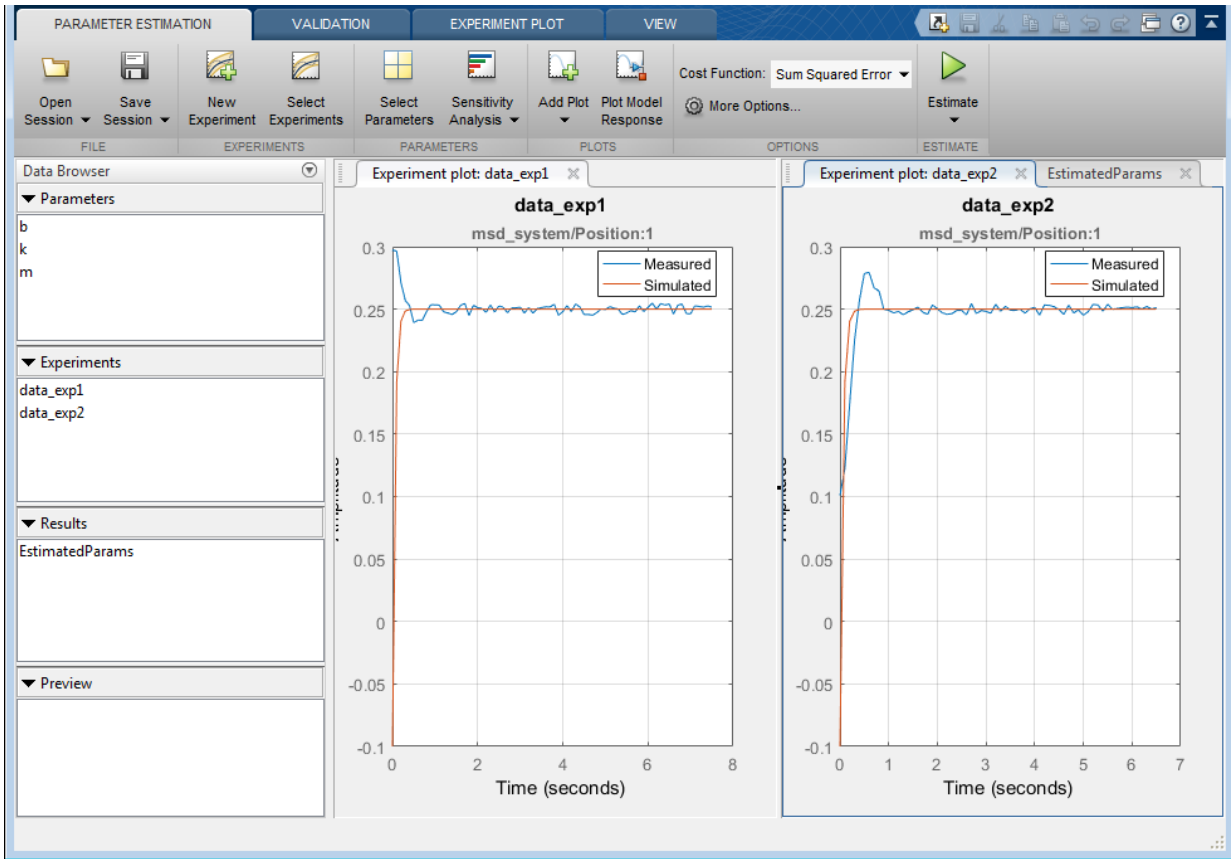


Parameter Estimation with No State Estimation

The tool has been configured to estimate the model parameters using both `data_exp1` and `data_exp2` experiments, click **Select Parameters** to see the selected parameters and **Select Experiments** to see the experiments selected for estimation.

Click **Estimate** to start the estimation. You can modify estimation options by setting the **Cost Function** combobox and clicking **More Options...**

While the estimation is running the plots update and a dialog showing estimation progress appears. The progress dialog shows the estimation iterations, the number of times the model has been evaluated (F-count), and the estimation cost at each iteration.



Iteration	F-count	data_exp1 (Minimize)	data_exp2 (Minimize)
0	7	4.8737	2.49
1	14	4.8737	2.49
2	21	3.7331	1.18
3	28	3.2630	0.87
4	35	3.2630	0.87
5	42	3.0230	0.74
6	49	2.8380	0.70
7	56	2.7386	0.66
8	63	2.3614	0.64
9	70	2.3129	0.56
10	77	2.0777	0.59
11	84	1.9252	0.67

Optimization started 22-Apr-2014 13:19:47
Estimation converged, 22-Apr-2014 13:21:29
 Estimated experiment values written to the workspace

Save Iteration... Display Options... Estimate

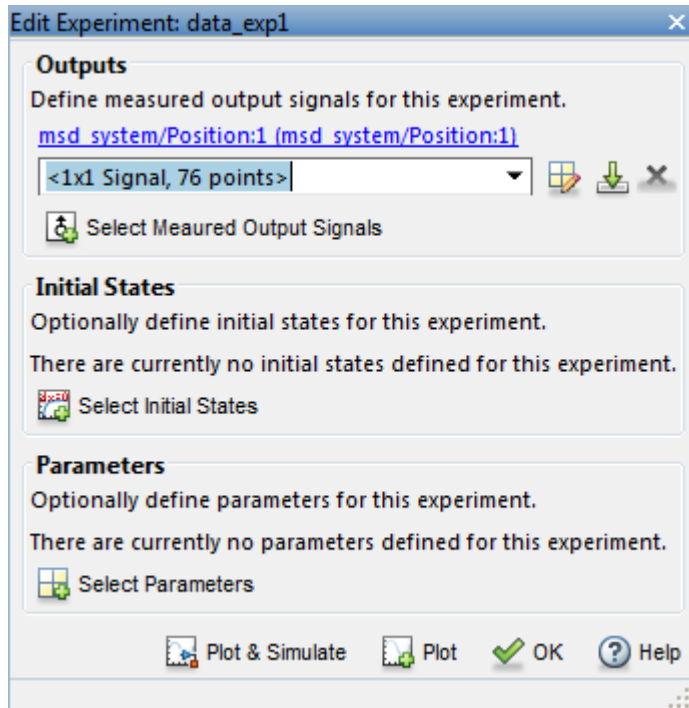
After a number of iterations the estimation converges and terminates. The model is updated with the estimated parameters and the estimation results are saved in the data browser.

The `data_exp1` and `data_exp2` experiment plots show that the model parameters have been tuned to match the measured experiment data as closely as possible. The simulated measured signals match well from the 2 second mark onward but don't match well before 2 seconds. The simulation results for both experiments start at -0.1. This is the initial condition of the model which was not estimated; these plots show that the initial condition should also be estimated.

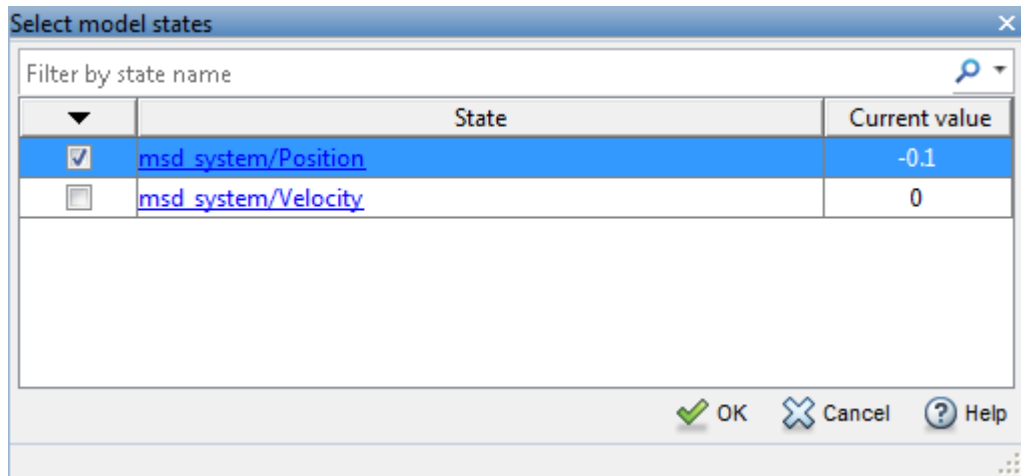
Parameter Estimation with Initial State Estimation

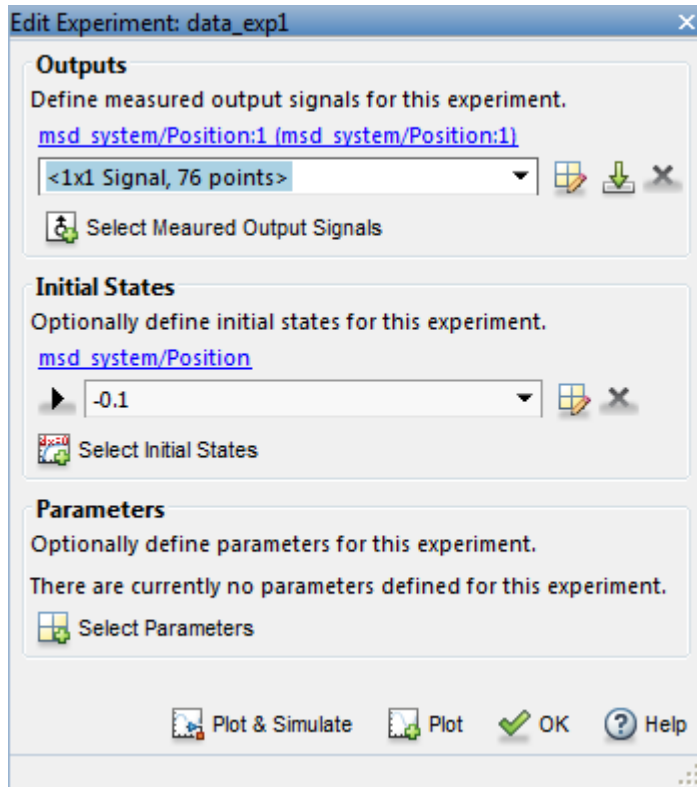
The `data_exp1` and `data_exp2` experiments specify the measured output data but as seen above must also specify the model initial state. We now add the initial states to the experiments and estimate them.

Right click `data_exp1` and select **Edit...** to open a dialog to configure the experiment.



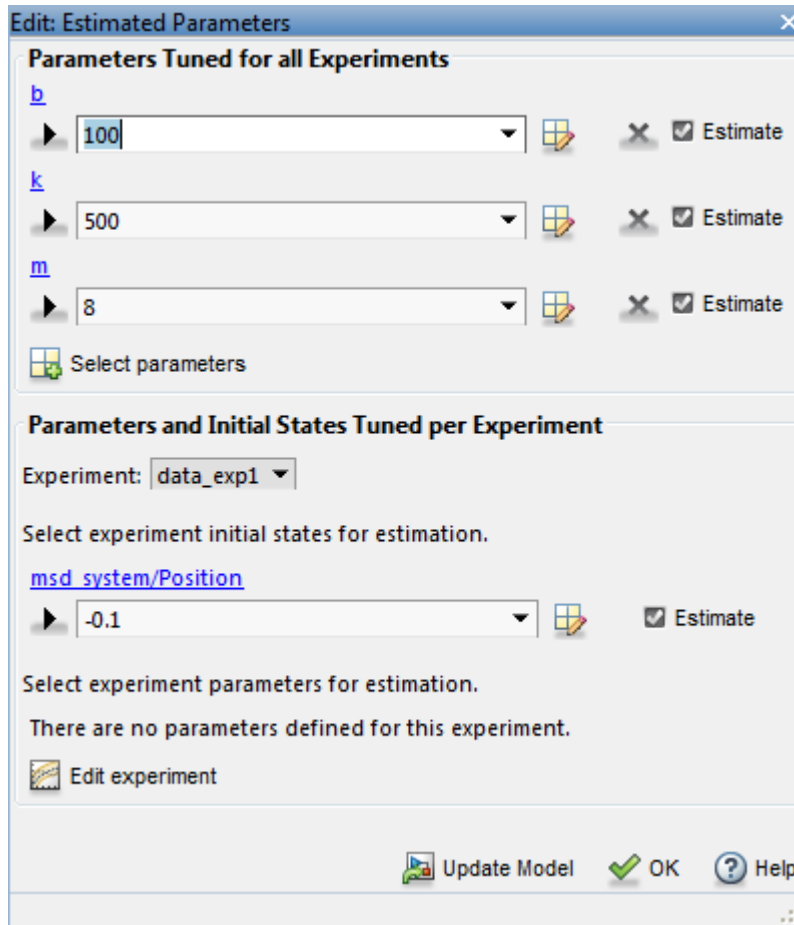
Click **Select Initial States** and select the position state. Click **OK** to close the state selector and add the selected state to the experiment.





Right click `data_exp2` and select **Edit..** and add the position state to the experiment.

The experiments are now configured to include initial states that can be estimated. Click **Select Parameters**.

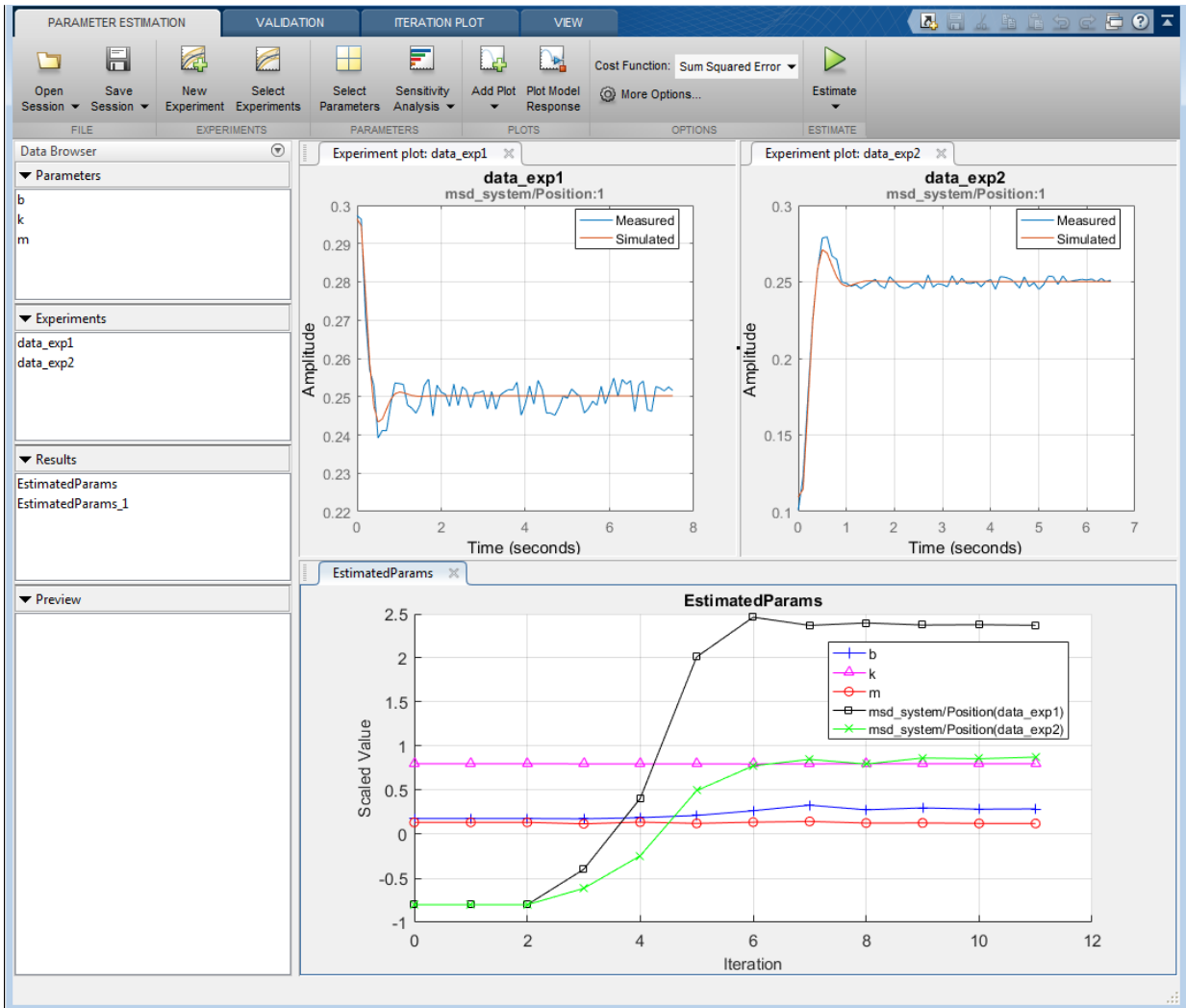


The upper portion of the select parameters dialog has a section for parameters that are tuned using all experiments selected for estimation. The lower section of the dialog has a combo-box to select an experiment and widgets to specify initial states and parameters that are tuned using only the selected experiment. For this problem the `data_exp1` and `data_exp2` experiments estimate the model initial state for each experiment.

Now we are ready to start our estimation but first create plots to monitor the estimation progress. Click **Add Plot** and select **Parameter Trajectory**, right click the plot and select **Show scaled values**. This creates a plot that shows how the estimated parameter values change during estimation. Click the **View** tab to layout the plots so that the

Experiment plot:data_exp1, Experiment plot:data_exp2, and Iteration plot 1 are both visible.

Click the **Estimate** button to start the estimation.



After a number of iterations the estimation converges and terminates. The `data_exp1` and `data_exp2` experiment plots show how estimating the initial value improves the estimation fit. The `EstimatedParams` plot shows the estimated initial state for the two experiments, the plot also shows that the estimated `k` value did not change while `b` and `m` changed slightly. You can confirm this by clicking `EstimatedParams` and examining the preview pane and then clicking `EstimatedParams1` and examining the preview pane. Alternatively right click `EstimatedParams` and select **Open...** to open a dialog to view the results.

<p>▼ Results</p> <p>EstimatedParams</p> <p>EstimatedParams1</p>	<p>▼ Results</p> <p>EstimatedParams</p> <p>EstimatedParams1</p>
<p>▼ Preview</p> <p>Estimation result(s):</p> <p>b = 17.559</p> <p>k = 400.07</p> <p>m = 1.0676</p>	<p>▼ Preview</p> <p>Estimation result(s):</p> <p>b = 28.593</p> <p>k = 399.79</p> <p>m = 0.97266</p>

This example shows that it is important to independently estimate initial states for each experiment in order to obtain the correct estimates of the model parameters.

Related Examples

To learn how to estimate model parameters and initial states using the `sdo.optimize` command, see "Estimate Model Parameters and Initial States (Code)".

Close the model

Generate MATLAB Code for Parameter Estimation Problems (GUI)

This example shows how to automatically generate a MATLAB function to solve a Parameter Estimation problem. You use the Parameter Estimation tool to define an estimation problem for a mass-spring-damper and generate MATLAB code to solve this estimation problem.

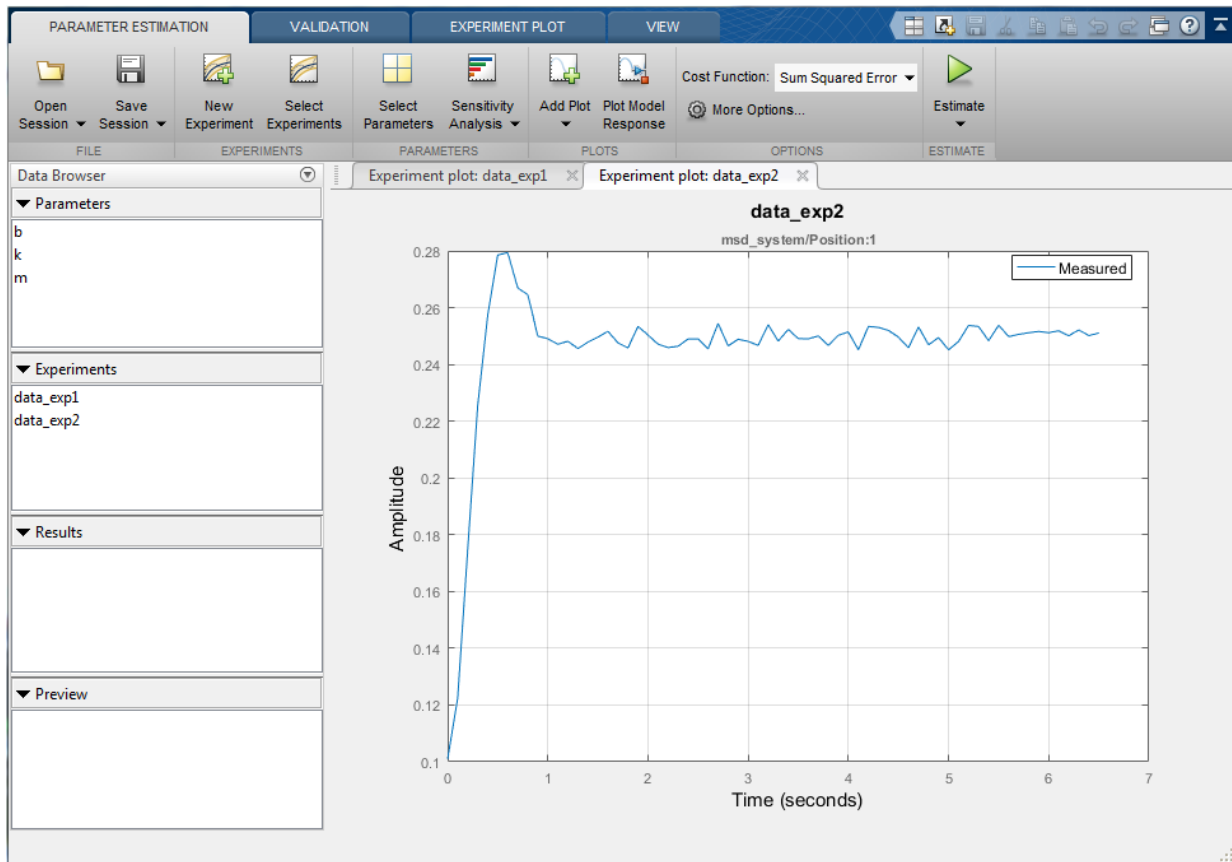
Mass-Spring-Damper Estimation Problem

The "Estimate Model Parameters and Initial States" example shows how to use the Parameter Estimation tool to estimate parameters of a mass-spring-damper model. In this example we load a pre-configured Parameter Estimation tool session based on that example.

Use the following commands to load the pre-configured estimation session

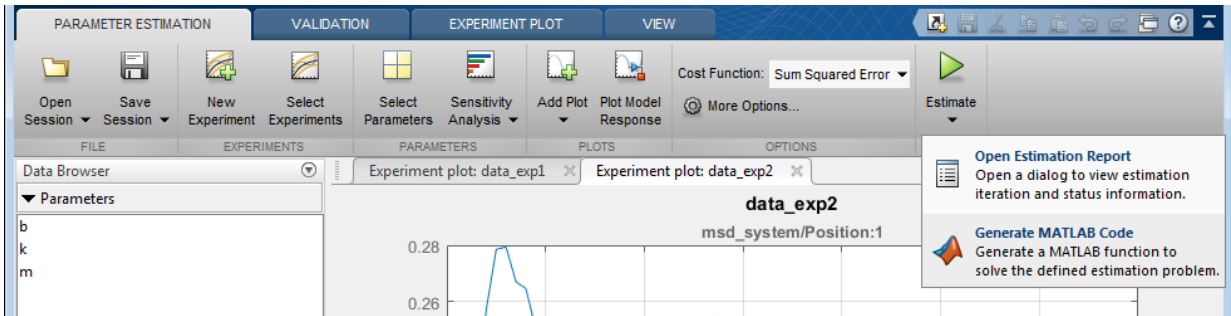
```
load sdoMassSpringDamper_sdoession  
spetool(SDOSessionData)
```

2 Parameter Estimation



Generate MATLAB Code

From the **Estimate** list, select **Generate MATLAB Code**.



The generated code is added to the MATLAB editor as an unsaved MATLAB function.

```

1 function [pOpt, Info] = spe_msd_system(p)
2 %SPE_MSD_SYSTEM
3 %
4 % Solve a parameter estimation problem for the msd_system model.
5 %
6 % The function returns estimated parameter values, pOpt,
7 % and estimation termination information, Info.
8 %
9 % The, p, input argument defines the model parameters to estimate,
10 % if omitted the parameters specified in the function body are estimated.
11 %
12 % Modify the function to include or exclude new experiments or
13 % change the estimation options.
14 %
15 % Auto-generated by SPETOOL on 22-Oct-2014 07:39:23.
16 %
17
18 %% Open the model.
19 open_system('msd_system')
20
21 %% Specify Model Parameters to Estimate
22 %
23 if nargin < 1 || isempty(p)
24     p = sdo.getParameterFromModel('msd_system',{'b','k','m'});
25     p(1).Minimum = 0;
26     p(1).Scale = 100;
27     p(2).Minimum = 0;
28     p(2).Scale = 500;
29     p(3).Minimum = 0;
    
```

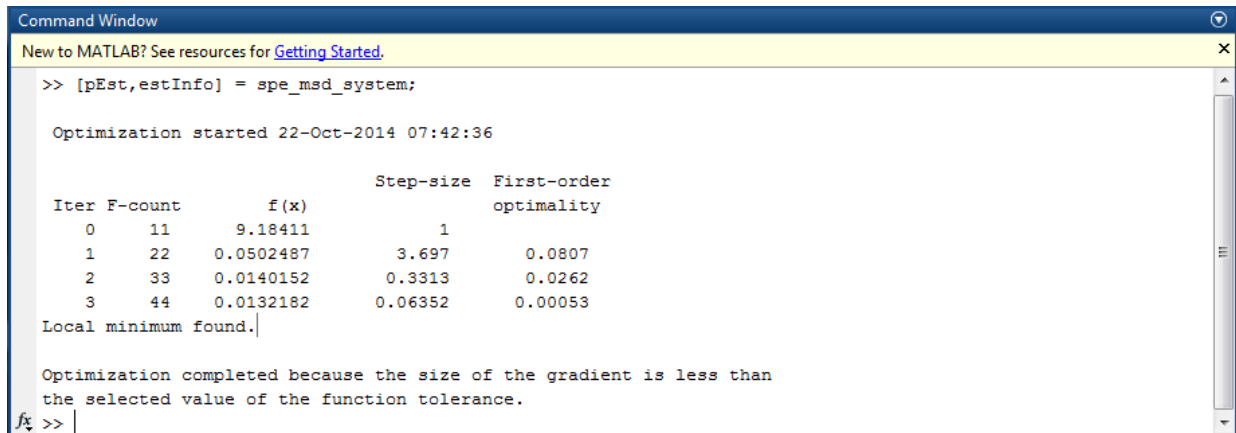
Examine the generated code. Significant code portions are:

- **Specify Model Parameters to Estimate** - Definition of the model parameters being estimated.
- **Define the Estimation Experiments** - Definition of the measured and expected signal data to use for estimation.
- **Create Estimation Objective Function** - Creation of an anonymous function that calls the subfunction `msd_system_optFcn`, which evaluates the model using each experiment and compares simulation and measured experiment outputs. This anonymous function is called by `sdo.optimize` at each iteration of the optimization problem to solve the estimation problem.
- **Estimate the Parameters** - Solve the estimation problem using the `sdo.optimize` command.

Select **Save** from the MATLAB editor to save the generated function.

Run Generated Code

Run the generated function.



```
Command Window
New to MATLAB? See resources for Getting Started.
>> [pEst,estInfo] = spe_msd_system;

Optimization started 22-Oct-2014 07:42:36

Iter F-count      f(x)            Step-size  First-order
      F-count      f(x)            Step-size  optimality
  0    11          9.18411         1
  1    22    0.0502487     3.697      0.0807
  2    33    0.0140152     0.3313     0.0262
  3    44    0.0132182     0.06352    0.00053
Local minimum found.

Optimization completed because the size of the gradient is less than
the selected value of the function tolerance.
fx >>
```

The first output argument, `pOpt`, contains the optimized parameter values and the second output argument, `optInfo`, contains optimization information.

Modify the Generated Code

You can:

- Modify the generated `spe_msd_system` function to include or exclude new experiments or change estimation options.
- Call the generated `spe_msd_system` function with a different set of parameters to estimate.

For details on how to write an objective/constraint function to use with the `sdo.optimize` command, type `help sdoExampleCostFunction` at the MATLAB command prompt.

Close the model

Improving Optimization Performance Using Fast Restart (GUI)

This example shows how to use the Fast Restart feature of Simulink® to speed up optimization of a model. You use Fast Restart to estimate the parameters of an engine throttle model.

How Fast Restart Speeds up the Optimization

Simulation of Simulink models requires that the model be compiled before it is simulated. In this context compilation of a model means analyzing and formatting the model so that it can be simulated. The idea of Fast Restart is to perform the model compilation once and reuse the compiled information for subsequent simulations, see "How Fast Restart Improves Iterative Simulations" in the Simulink documentation for a description of Fast Restart.

During optimization the model is repeatedly simulated (often tens or hundreds of times) Fast Restart means that the model is only compiled once for these simulation in comparison to non-fast restart where the model is recompiled each time.

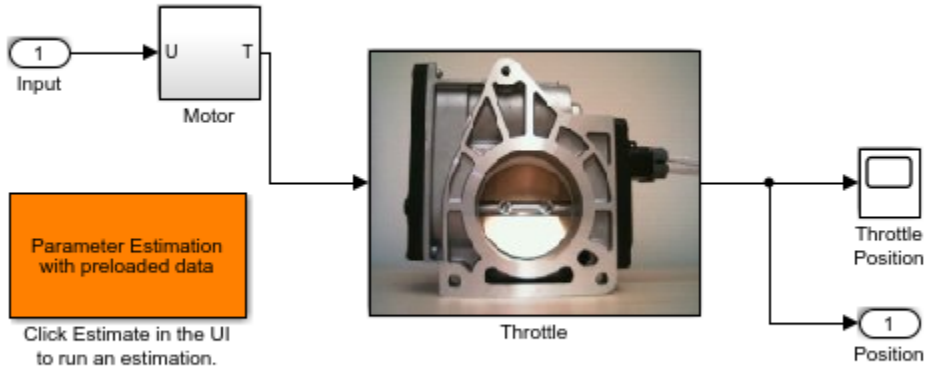
Models where compilation is a significant portion of overall simulation time benefit the most from Fast Restart. Further once a model is compiled not all model parameters can be changed, specifically only tunable parameters can be changed, see "Factors Affecting Fast Restart" in the Simulink documentation for more information.

Open Model and Estimation Tool

Load the model and click the "Parameter Estimation with preloaded data" block to load a preconfigured parameter estimation problem. The goal is to tune the parameters of an engine throttle model to match measured data. For details on the problem setup see the "Estimate Model Parameter Values (GUI)" example.

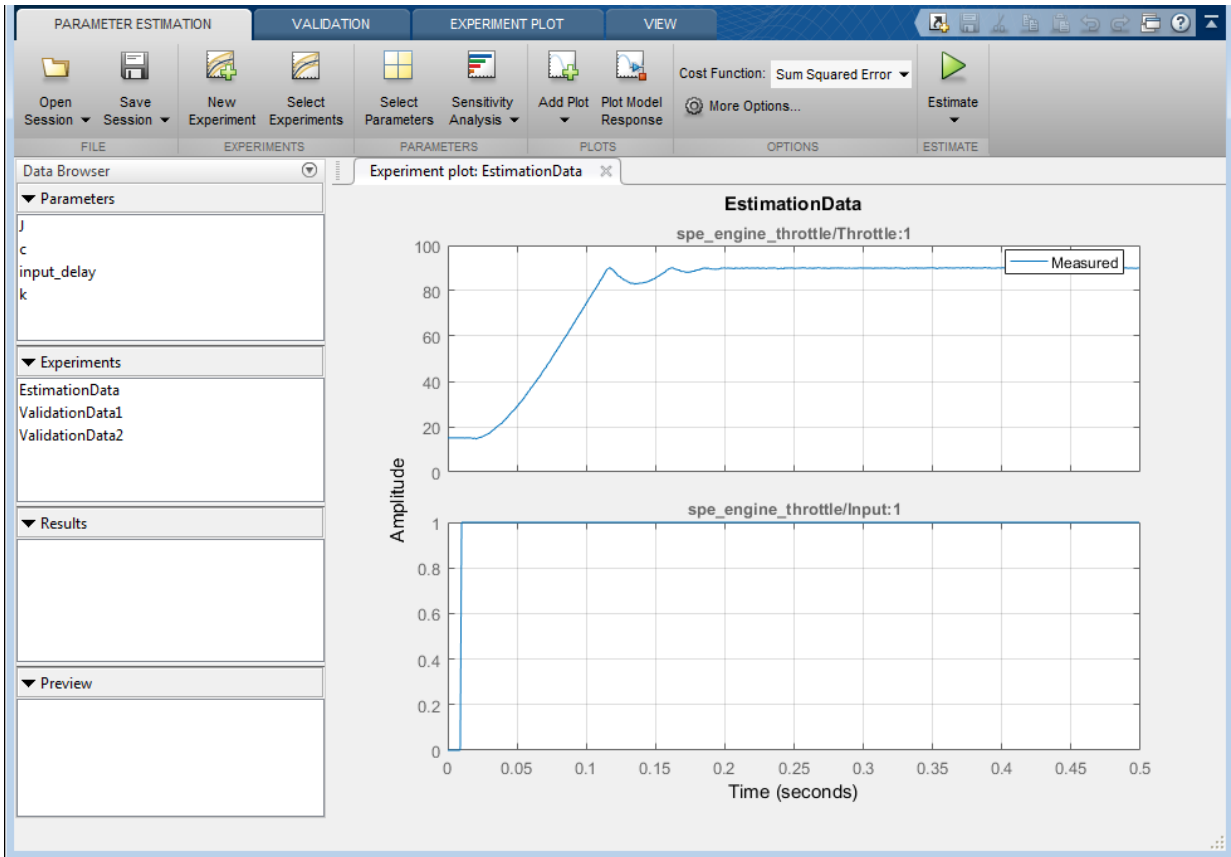
```
open_system('spe_engine_throttle')
```


Engine Throttle Model



Copyright (c) 2002-2014 The MathWorks, Inc.

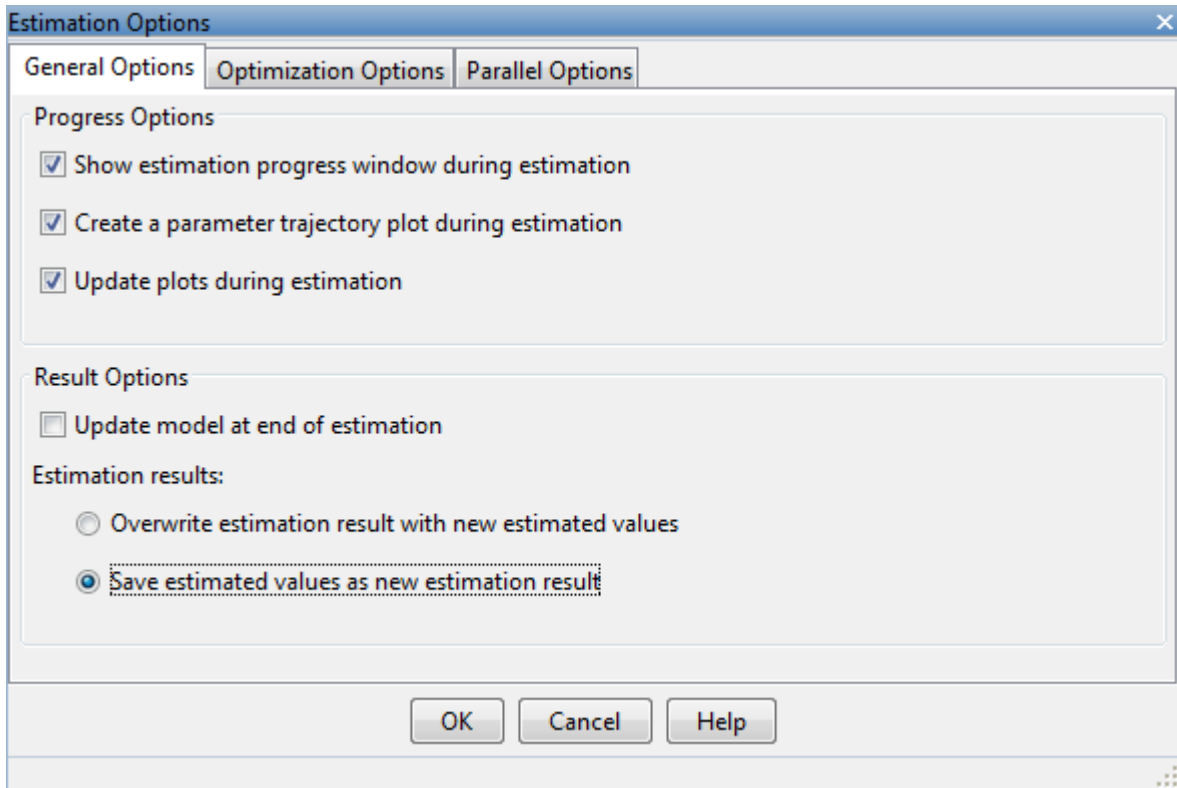
2 Parameter Estimation



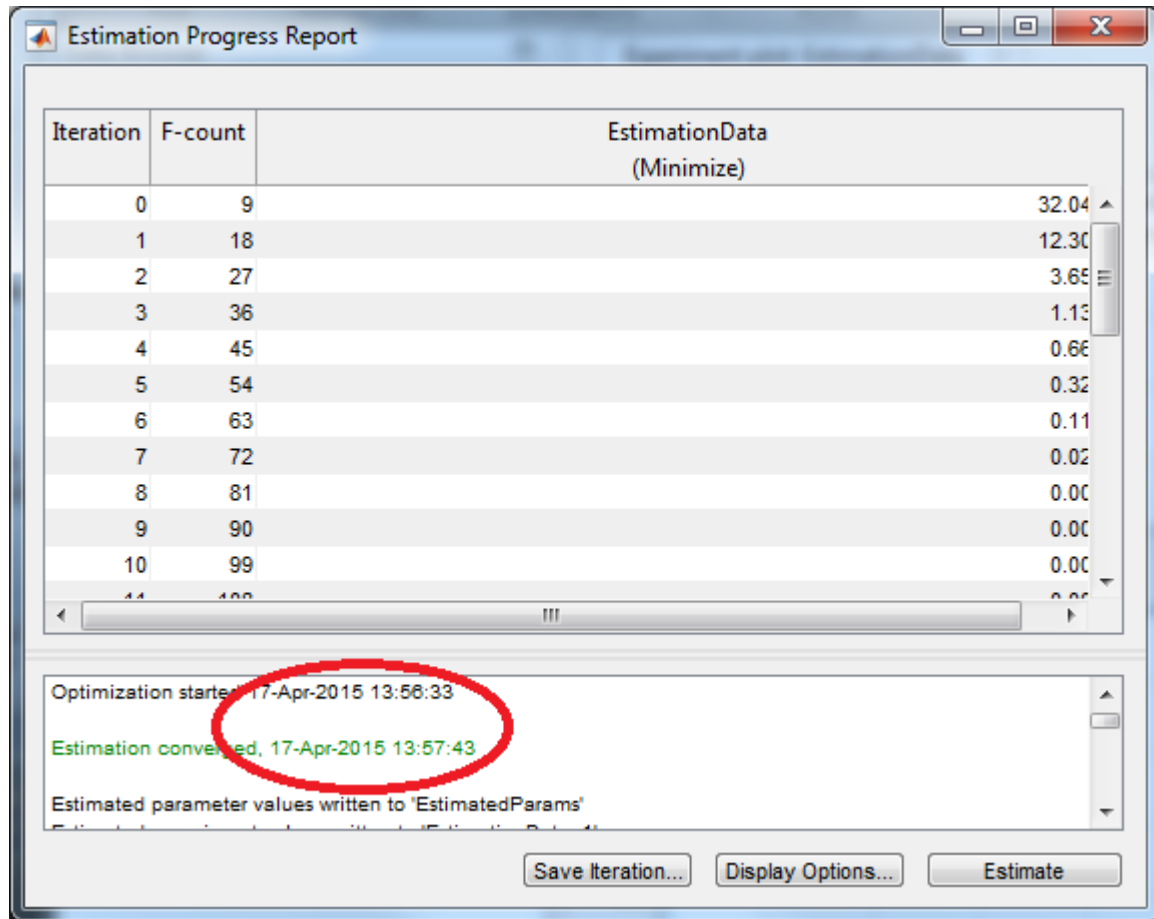
Estimate Without Using Fast Restart

To compare the estimation with and without fast restart, change the estimation options in the tool to not update the model with estimated values.

Click **Model Options...** in the Parameter Estimation tool and clear **Update model at end of estimation**, and select **Save estimated values as new estimation result**.

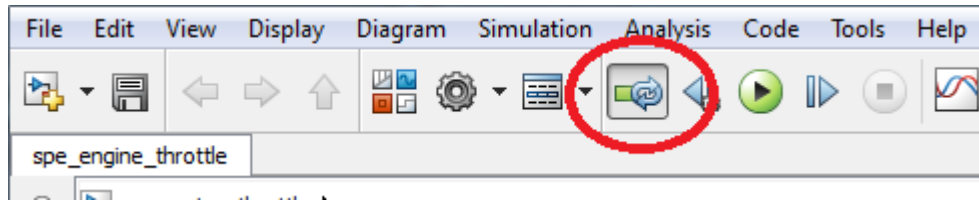


Click **Estimate** in the tool to estimate the model parameter values. The estimation progress report shows the estimation start and end time.

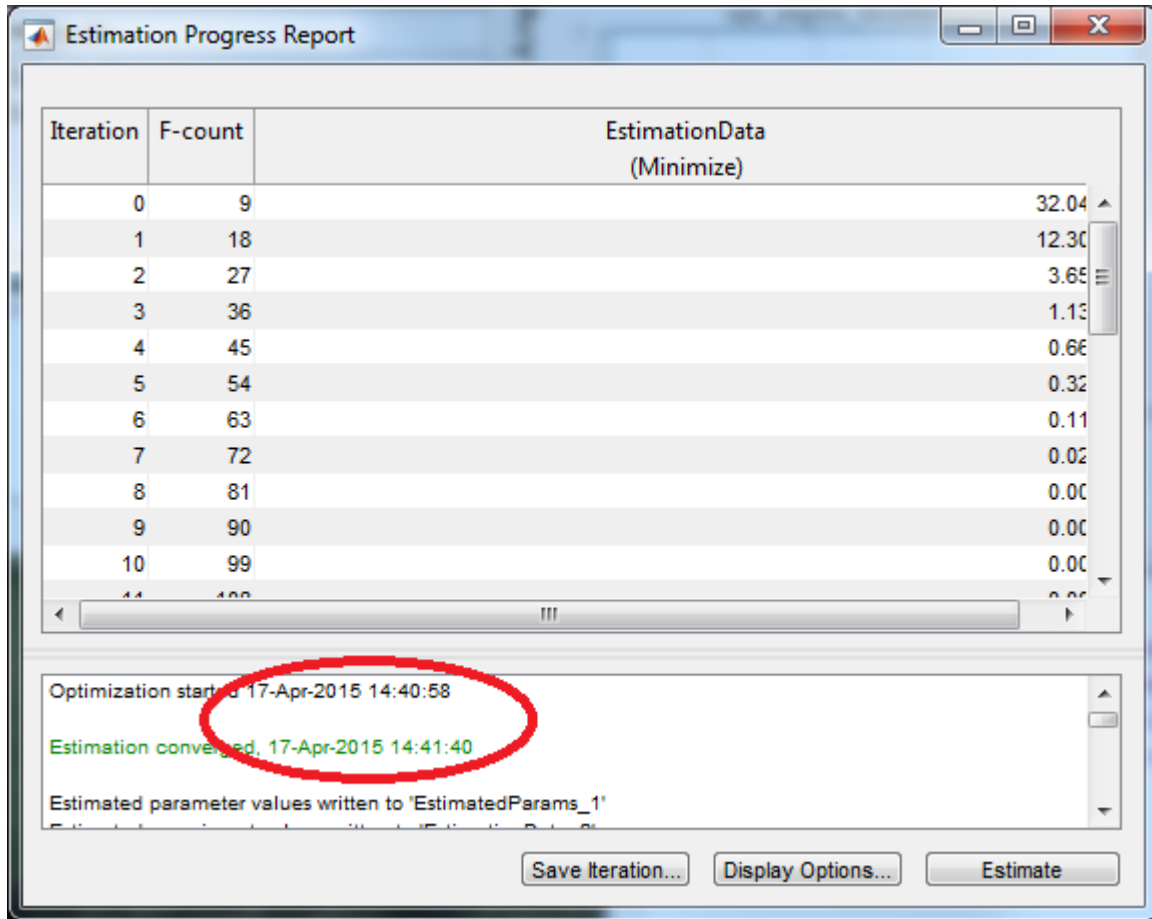


Estimate Using Fast Restart

To configure the model to use Fast Restart during simulation, click **Enable Fast Restart** in the Simulink model.



Click **Estimate** in the Parameter Estimation tool. The estimation progress report shows the estimation start and end time. Note the reduction in total estimation time compared to the estimation without using fast restart, in this case around 28 seconds or 45% of the original estimation time.



Related Examples

The `Generate MATLAB Code` feature of the Parameter Estimation and Response Optimization tool will generate the MATLAB code to configure the model for fast restart if the tool is configured to use fast restart.

To learn how to use Fast Restart at the command line see "Improving Optimization Performance using Fast Restart (Code)".

Close the model.

```
bdclose('spe_engine_throttle')
```

See Also

Related Examples

- “Improving Optimization Performance Using Fast Restart (Code)” on page 2-190
- “Use Fast Restart Mode During Response Optimization” on page 3-226
- “Use Fast Restart Mode During Sensitivity Analysis” on page 4-144

More About

- “Ways to Speed Up Design Optimization Tasks”

Improving Optimization Performance Using Fast Restart (Code)

This example shows how to use the Fast Restart feature of Simulink® to speed up optimization of a model. You use Fast Restart to estimate the parameters of an engine throttle model.

How Fast Restart Speeds up the Optimization

Simulation of Simulink models requires that the model be compiled before it is simulated. In this context compilation of a model means analyzing and formatting the model so that it can be simulated. The idea of Fast Restart is to perform the model compilation once and reuse the compiled information for subsequent simulations, see "How Fast Restart Improves Iterative Simulations" in the Simulink documentation for a description of Fast Restart.

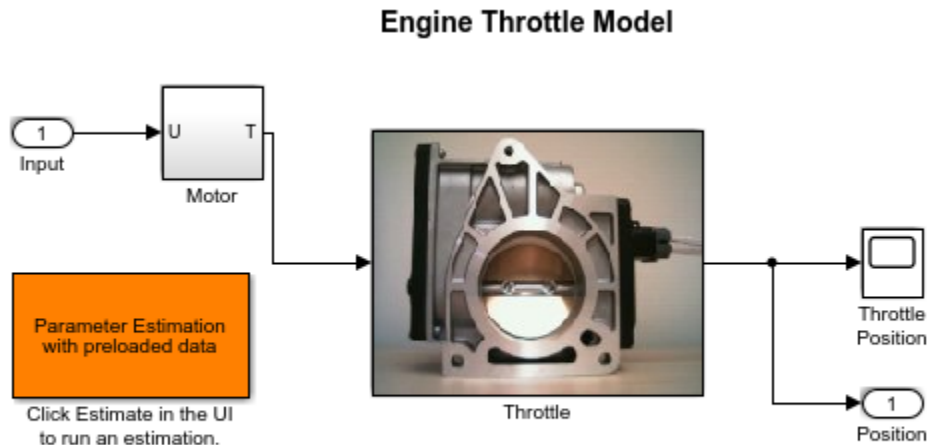
During optimization the model is repeatedly simulated (often tens or hundreds of times) Fast Restart means that the model is only compiled once for these simulation in comparison to non-fast restart where the model is recompiled each time.

Models where compilation is a significant portion of overall simulation time benefit the most from Fast Restart. Further once a model is compiled not all model parameters can be changed, specifically only tunable parameters can be changed, see "Factors Affecting Fast Restart" in the Simulink documentation for more information.

Open Model

Load the engine throttle model. The goal is to tune the parameters of the model to match measured data. For details on the problem setup see the "Estimate Model Parameter Values (GUI)" example.

```
open_system('spe_engine_throttle')
```

Define the Estimation Problem Data

This example focuses on the command line interface for using Fast Restart during estimation. For a detailed description of the estimation command line interface see "Estimate Model Parameter Values (Code)".

Specify the model parameter values to estimate and any parameter bounds.

```
p = sdo.getParameterFromModel('spe_engine_throttle',{'J','c','input_delay','k'});
p(1).Minimum = 0;
p(2).Minimum = 0;
p(3).Minimum = 0;
p(3).Maximum = 0.1;
p(4).Minimum = 0;
```

Define the estimation experiment. The measured experiment data is loaded from the `sdoFastRestart_ExperimentData` MATLAB file. The MATLAB file contains a `Input_SignalData` and `Output_SignalData` variable specifying the experiment input and output signal data.

```
load spe_engine_throttle_ExperimentData
Exp = sdo.Experiment('spe_engine_throttle');
```

```
Input = Simulink.SimulationData.Signal;
Input.Values = Input_SignalData;
Input.BlockPath = 'spe_engine_throttle/Input';
Input.PortType = 'inport';
Input.PortIndex = 1;
Input.Name = 'spe_engine_throttle/Input:1';
Exp.InputData = Input;
Output = Simulink.SimulationData.Signal;
Output.Values = Output_SignalData;
Output.BlockPath = 'spe_engine_throttle/Throttle';
Output.PortType = 'outport';
Output.PortIndex = 1;
Output.Name = 'spe_engine_throttle/Throttle:1';
Exp.OutputData = Output;
```

Create a model simulator from the experiment

```
Simulator = createSimulator(Exp);
```

Configure the Simulator for Fast Restart

The simulator controls whether the model is simulated using fast restart or not. The `fastRestart` command is used to configure the simulator to use Fast Restart.

The `spe_engine_throttle` model uses a variable-step solver, and may not output values at the times in the measured experiment data. To output values at the times of the measured data, use the `set_param` command to specify the model logging output times as a workspace variable. In the estimation objective function, the variable is then used to specify output times to be the same as the measured experiment data. The model `OutputTimes` is set before configuring the simulator for fast restart, as once the model is configured for fast restart, the model logging configuration can not change.

```
set_param('spe_engine_throttle', 'OutputOption', 'SpecifiedOutputTimes', 'OutputTimes', 'OutputTimes');
Simulator = fastRestart(Simulator, 'on');
```

The simulator can now be used during estimation, and the model will be simulated using fast restart.

Run the Estimation

Create an estimation objective function to evaluate how closely the simulation output, generated using the estimated parameter values, matches the measured data. Use an anonymous function with one argument that calls the

`spe_engine_throttle_Objective` function. The `spe_engine_throttle_Objective` function includes the `Simulator` argument that has been configured to use fast restart.

```
optimfcn = @(P) spe_engine_throttle_Objective(P,Simulator,Exp);
```

Set the optimization options, and run the optimization.

```
Options = sdo.OptimizeOptions;
Options.Method = 'lsqnonlin';
[pOpt,Info] = sdo.optimize(optimfcn,p,Options);
```

```
Optimization started 15-Aug-2017 00:02:05
```

Iter	F-count	f(x)	Step-size	First-order optimality
0	9	32.048	1	
1	18	12.24	0.6495	18
2	27	3.5944	0.3919	8.65
3	36	1.11908	0.188	3.11
4	45	0.648846	0.197	1.25
5	54	0.286999	1.174	1.12
6	63	0.145554	0.3484	0.417
7	72	0.0876181	0.5157	0.0797
8	81	0.0675225	0.4867	0.0587
9	90	0.0546981	3.473	0.119
10	99	0.0546981	7.271	0.119
11	108	0.0530811	1.818	0.0693
12	117	0.0530811	1.898	0.0693
13	126	0.0522303	0.4745	0.0322

Local minimum possible.

`lsqnonlin` stopped because the final change in the sum of squares relative to its initial value is less than the selected value of the function tolerance.

Restore the Model

Restore the simulator fast restart settings. This clears the logging and other settings used for the optimization problem.

```
Simulator = fastRestart(Simulator,'off');
set_param('spe_engine_throttle','OutputOption','RefineOutputTimes','OutputTimes','[]');
```

Related Examples

You can also generate code to configure your model for fast restart in the Parameter Estimation and Response Optimization tools. Configure the model for fast restart as described in "Improving Optimization Performance using Fast Restart (GUI)". Then use the Generate MATLAB Code feature of the tool.

Close the model.

```
bdclose('spe_engine_throttle')
```

See Also

`fastRestart` | `sdo.SimulationTest`

Related Examples

- "Improving Optimization Performance Using Fast Restart (GUI)" on page 2-182
- "Use Fast Restart Mode During Response Optimization" on page 3-226
- "Use Fast Restart Mode During Sensitivity Analysis" on page 4-144

More About

- "Ways to Speed Up Design Optimization Tasks"

Response Optimization

- “How the Optimization Algorithm Formulates Minimization Problems” on page 3-3
- “Specify Signals to Log” on page 3-12
- “Specify Custom Requirements in the Tool” on page 3-13
- “Move Constraints” on page 3-16
- “Specify Time-Domain Design Requirements in the Tool” on page 3-19
- “Edit Design Requirements” on page 3-37
- “Specify Variable Requirements in the Tool” on page 3-39
- “Specify Frequency-Domain Design Requirements in the Tool” on page 3-56
- “Specify Design Variables” on page 3-76
- “Update Model with Design Variables Set” on page 3-82
- “Specify Optimization Options” on page 3-84
- “Create Linearization I/O Sets” on page 3-86
- “Interact with Plots” on page 3-89
- “Compare Requirements and Design Variables Using Spider Plot” on page 3-94
- “Save Design Variable Values for Specific Iteration” on page 3-97
- “Design Optimization to Meet Time- and Frequency-Domain Requirements (GUI)” on page 3-99
- “Design Optimization to Meet a Custom Objective (GUI)” on page 3-115
- “Design Optimization to Meet a Custom Objective (Code)” on page 3-133
- “Design Optimization to Meet Custom Signal Requirements (GUI)” on page 3-142
- “Design Optimization to Meet Frequency-Domain Requirements (GUI)” on page 3-147
- “Specify Custom Signal Objective with Uncertain Variable (GUI)” on page 3-164
- “Design Optimization with Uncertain Variables (Code)” on page 3-175
- “Generate MATLAB Code for Design Optimization Problems (GUI)” on page 3-185
- “Skip Model Simulation Based on Parameter Constraint Violation (GUI)” on page 3-190

- “Optimizing Parameters for Robustness” on page 3-201
- “Use Accelerator Mode During Simulations” on page 3-212
- “Speed Up Response Optimization Using Parallel Computing” on page 3-214
- “Use Parallel Computing for Response Optimization” on page 3-218
- “Use Fast Restart Mode During Response Optimization” on page 3-226
- “Optimization Does Not Make Progress” on page 3-229
- “Optimization Convergence” on page 3-231
- “Optimization Speed and Parallel Computing” on page 3-234
- “Undesirable Parameter Values” on page 3-237
- “Reverting to Initial Parameter Values” on page 3-239
- “Save and Load Optimization Sessions” on page 3-240
- “Optimizing Time-Domain Response of Simulink® Models Using Parallel Computing” on page 3-242
- “Design Optimization to Meet Frequency-Domain Requirements (Code)” on page 3-251

How the Optimization Algorithm Formulates Minimization Problems

When you optimize parameters of a Simulink model to meet design requirements, Simulink Design Optimization software automatically converts the requirements into a constrained optimization problem and then solves the problem using optimization techniques. The constrained optimization problem iteratively simulates the Simulink model, compares the results of the simulations with the constraint objectives, and uses optimization methods to adjust tuned parameters to better meet the objectives.

This topic describes how the software formulates the constrained optimization problem used by the optimization algorithms. For each optimization algorithm, the software formulates one of the following types of minimization problems:

- Feasibility on page 3-3
- Tracking on page 3-5
- Mixed feasibility and tracking

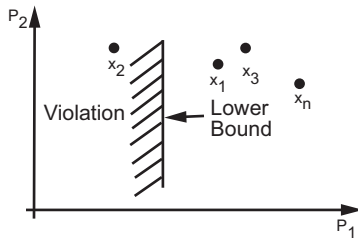
For more information on how each optimization algorithm formulates these problems, see:

- “Gradient Descent Method Problem Formulations” on page 3-6
- “Simplex Search Method Problem Formulations” on page 3-8
- “Pattern Search Method Problem Formulations” on page 3-9
- “Gradient Computations” on page 3-10

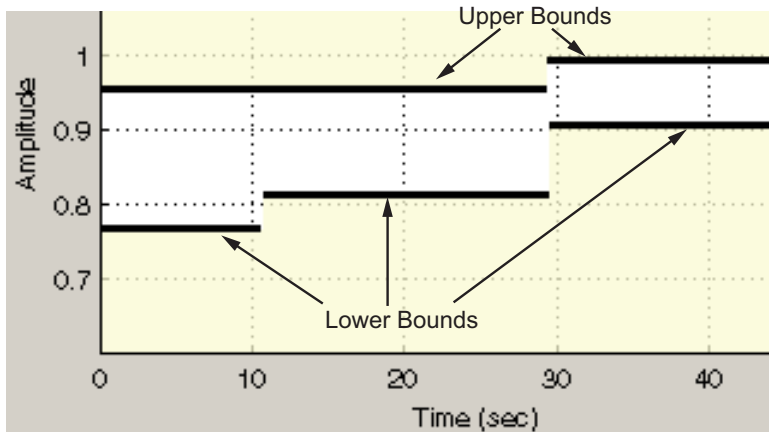
Feasibility Problem and Constraint Formulation

Feasibility means that the optimization algorithm finds parameter values that satisfy all constraints to within specified tolerances but does not minimize any objective or cost function in doing so.

In the following figure, x_1 , x_3 , and x_n represent a combination of parameter values P_1 and P_2 and are feasible solutions because they do not violate the lower bound constraint.



In a Simulink model, you constrain a signal by specifying lower and upper bounds in a Check block (Check Step Response Characteristics, ...) or a requirement object (sdo.requirements.StepResponseEnvelope, ...), as shown in the following figure.



These constraints are piecewise linear bounds. A piecewise linear bound y_{bnd} with n edges can be represented as:

$$y_{bnd}(t) = \begin{cases} y_1(t) & t_1 \leq t \leq t_2 \\ y_2(t) & t_2 \leq t \leq t_3 \\ \vdots & \vdots \\ y_n(t) & t_n \leq t \leq t_{n+1} \end{cases},$$

The software computes the signed distance between the simulated response and the edge. The signed distance for lower bounds is:

$$c = \begin{bmatrix} \max_{t_1 \leq t \leq t_2} y_{bnd} - y_{sim} \\ \max_{t_2 \leq t \leq t_3} y_{bnd} - y_{sim} \\ \max_{t_n \leq t \leq t_{n+1}} y_{bnd} - y_{sim} \end{bmatrix},$$

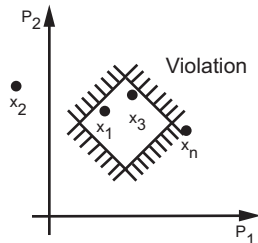
where y_{sim} is the simulated response and is a function of the parameters being optimized.

The signed distance for upper bounds is:

$$c = \begin{bmatrix} \max_{t_1 \leq t \leq t_2} y_{sim} - y_{bnd} \\ \max_{t_2 \leq t \leq t_3} y_{sim} - y_{bnd} \\ \max_{t_n \leq t \leq t_{n+1}} y_{sim} - y_{bnd} \end{bmatrix}.$$

At the command line, `opt_fcn` supplies c directly from the `Cleq` field of `vals`.

If *all* the constraints are met ($c \leq 0$) for some combination of parameter values, then that solution is said to be feasible. In the following figure, x_1 and x_3 are feasible solutions.



When your model has multiple requirements or vector signals feeding a requirement, the constraint vector is extended with the constraint violations for each signal and bound:

$$C = [c_1; c_2; \dots; c_n].$$

Tracking Problem

In addition to lower and upper bounds, you can specify a reference signal in a Check Against Reference block or `sdo.requirements.SignalTracking` object, which the

Simulink model output can track. The tracking objective is a sum-squared-error tracking objective.

You specify the reference signal as a sequence of time-amplitude pairs:

$$y_{ref}(t_{ref}), t_{ref} \in \{T_{ref0}, T_{ref1}, \dots, T_{refN}\}.$$

The software computes the simulated response as a sequence of time-amplitude pairs:

$$y_{sim}(t_{sim}), t_{sim} \in \{T_{sim0}, T_{sim1}, \dots, T_{simN}\},$$

where some values of t_{sim} may match the values of t_{ref} .

A new time base, t_{new} , is formed from the union of the elements of t_{ref} and t_{sim} . Elements that are not within the minimum-maximum range of both t_{ref} and t_{sim} are omitted:

$$t_{new} = \{t : t_{sim} \cup t_{ref}\}$$

Using linear interpolation, the software computes the values of y_{ref} and y_{sim} at the time points in t_{new} and then computes the scaled error:

$$e(t_{new}) = \frac{(y_{sim}(t_{new}) - y_{ref}(t_{new}))}{\max_{t_{new}} |y_{ref}|}.$$

Finally, the software computes the weighted, integral square error:

$$f = \int w(t) e(t)^2 dt.$$

Note The weight $w(t)$ is 1 by default. You can specify a different value of weight only at the command line.

When your model has requirements or vector signals feeding a requirement, the tracking objective equals the sum of the individual tracking integral errors for each signal:

$$F = \sum f_i.$$

Gradient Descent Method Problem Formulations

The Gradient Descent method uses the Optimization Toolbox function `fmincon` to optimize model parameters to meet design requirements.

Problem Type	Problem Formulation
Feasibility Problem	<p>The software formulates the constraint $C(x)$ as described in “Feasibility Problem and Constraint Formulation” on page 3-3.</p> <ul style="list-style-type: none"> If you select the maximally feasible solution option (i.e., the optimization continues after an initial feasible solution is found), the software uses the following problem formulation: $\begin{aligned} \min_{[x,\gamma]} \quad & \gamma \\ \text{s.t.} \quad & C(x) \leq \gamma \\ & \underline{x} \leq x \leq \bar{x} \\ & \gamma \leq 0 \end{aligned}$ <p>γ is a slack variable that permits a feasible solution with $C(x) \leq \gamma$ rather than $C(x) \leq 0$.</p> If you do not select the maximally feasible solution option (i.e., the optimization terminates as soon as a feasible solution is found), the software uses the following problem formulation: $\begin{aligned} \min_x \quad & 0 \\ \text{s.t.} \quad & C(x) \leq 0 \\ & \underline{x} \leq x \leq \bar{x} \end{aligned}$
Tracking Problem	<p>The software formulates the tracking objective $F(x)$ as described in “Tracking Problem” on page 3-5 and minimizes the tracking objective:</p> $\begin{aligned} \min_x \quad & F(x) \\ \text{s.t.} \quad & \underline{x} \leq x \leq \bar{x} \end{aligned}$

Problem Type	Problem Formulation
Mixed Feasibility and Tracking Problem	<p>The software minimizes following problem formulation:</p> $\begin{aligned} \min_x \quad & F(x) \\ \text{s.t.} \quad & C(x) \leq 0 \\ & \underline{x} \leq x \leq \bar{x} \end{aligned}$ <p>Note When tracking a reference signal, the software ignores the maximally feasible solution option.</p>

Simplex Search Method Problem Formulations

The Simplex Search method uses the Optimization Toolbox function `fminsearch` and `fminbnd` to optimize model parameters to meet design requirements. `fminbnd` is used if one scalar parameter is being optimized, otherwise `fminsearch` is used. You cannot use parameter bounds $\underline{x} \leq x \leq \bar{x}$ with `fminsearch`.

Problem Type	Problem Formulation
Feasibility Problem	<p>The software formulates the constraint $C(x)$ as described in “Feasibility Problem and Constraint Formulation” on page 3-3 and then minimizes the maximum constraint violation:</p> $\min_x \max(C(x))$
Tracking Problem	<p>The software formulates the tracking objective $F(x)$ as described in “Tracking Problem” on page 3-5 and then minimizes the tracking objective:</p> $\min_x F(x)$

Problem Type	Problem Formulation
Mixed Feasibility and Tracking Problem	<p>The software formulates the problem in two steps:</p> <ol style="list-style-type: none"> 1 Finds a feasible solution. $\min_x \max(C(x))$ 2 Minimizes the tracking objective. The software uses the results from step 1 as initial guesses and maintains feasibility by introducing a discontinuous barrier in the optimization objective. $\min_x \Gamma(x)$ where $\Gamma(x) = \begin{cases} \infty & \text{if } \max(C(x)) > 0 \\ F(x) & \text{otherwise.} \end{cases}$

Pattern Search Method Problem Formulations

The Pattern Search method uses the Global Optimization Toolbox function `patternsearch` to optimize model parameters to meet design requirements.

Problem Type	Problem Formulation
Feasibility Problem	<p>The software formulates the constraint $C(x)$ as described in “Feasibility Problem and Constraint Formulation” on page 3-3 and then minimizes the maximum constraint violation:</p> $\min_x \max(C(x))$ <p><i>s.t.</i> $\underline{x} \leq x \leq \bar{x}$</p>
Tracking Problem	<p>The software formulates the tracking objective $F(x)$ as described in “Tracking Problem” on page 3-5 and then minimizes the tracking objective:</p> $\min_x F(x)$ <p><i>s.t.</i> $\underline{x} \leq x \leq \bar{x}$</p>

Problem Type	Problem Formulation
Mixed Feasibility and Tracking Problem	<p>The software formulates the problem in two steps:</p> <p>1 Finds a feasible solution.</p> $\min_x \max(C(x))$ $s.t. \quad \underline{x} \leq x \leq \bar{x}$ <p>2 Minimizes the tracking objective. The software uses the results from step 1 as initial guesses and maintains feasibility by introducing a discontinuous barrier in the optimization objective.</p> $\min_x \Gamma(x)$ $s.t. \quad \underline{x} \leq x \leq \bar{x}$ <p>where</p> $\Gamma(x) = \begin{cases} \infty & \text{if } \max(C(x)) > 0 \\ F(x) & \text{otherwise.} \end{cases}$

Gradient Computations

For the Gradient descent (fmincon) optimization solver, the gradients are computed using numerical perturbation:

$$dx = \sqrt[3]{eps} \times \max\left(|x|, \frac{1}{10} x_{typical}\right)$$

$$dL = \max(x - dx, x_{min})$$

$$dR = \min(x + dx, x_{max})$$

$$F_L = opt_fcn(dL)$$

$$F_R = opt_fcn(dR)$$

$$\frac{dF}{dx} = \frac{(F_L - F_R)}{(dL - dR)}$$

- x is a scalar design variable.
- x_{min} is the lower bound of x .

- x_{max} is the upper bound of x .
- $x_{typical}$ is the scaled value of x .
- opt_fcn is the objective function.

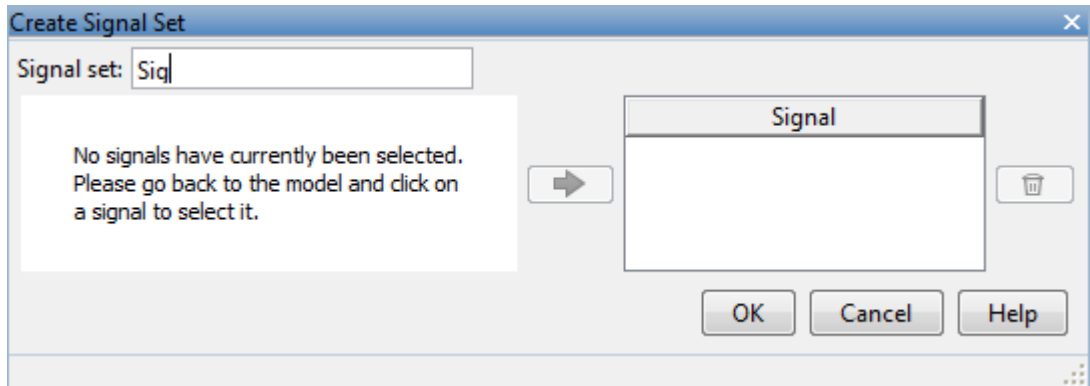
dx is relatively large to accommodate simulation solver tolerances.

If you want to compute the gradients in any other way, you can do so in the cost function you write for performing design optimization programmatically. See `sdo.optimize` and `GradFcn` of `sdo.OptimizeOptions` for more information.


Specify Signals to Log

Design requirements require logged model signals. During optimization, the model is simulated using the current value of the design variables and the logged signal is used to evaluate the design requirements.

- 1 In the **Response Optimization tool**, select **Signal** in the **New** drop-down list. A window opens where you select a signal to log.
- 2 In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- 3 Select the signal and click  to add it to the signal set.
- 4 In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.


See Also


- “Design Optimization to Track Reference Signal (GUI)”
- `sdo.SimulationTest`

Specify Custom Requirements in the Tool

This topic shows how to specify custom requirements in the Response Optimization tool.

You can specify custom requirements, such as minimizing system energy. To specify custom requirements:

- 1 In the Response Optimization tool, in **New** drop-down menu, select **Custom Requirement**. The Create Requirement dialog box opens where you specify the custom requirement.
- 2 Specify a requirement name in **Name**.
- 3 Specify the requirement type in the **Type** drop-down menu.
- 4 Specify the name of the function that contains the custom requirement in **Function**. The field must be specified as a function handle using @. The function must be on the MATLAB path. Click  to review or edit the function.

If the function does not exist, clicking  opens a template MATLAB file. Use this file to implement the custom requirement. The default function name is myCustomRequirement.

- 5 (Optional) To prevent the solver from considering specific parameter combinations, select **Error if constraint is violated**. Use this option for parameter-only constraints.

During an optimization iteration, the solver first evaluates requirements with this option selected.

- If the constraint is violated, the solver skips evaluating any remaining requirements and proceeds to the next iteration.
- If the constraint is *not* violated, the solver evaluates the remaining requirements for the current iteration. If any of the remaining requirements bound signals or systems, then the solver simulates the model.

For more information, see “Skip Model Simulation Based on Parameter Constraint Violation (GUI)” on page 3-190.

Note If you select this check box, then do not specify signals or systems to bound. If you *do* specify signals or systems, then this check box is ignored.

- 6 (Optional) Specify the signal or system, or both, to be bound.

You can apply this requirement to model signals, or a linearization of your Simulink model (requires Simulink Control Design), or both.


Click **Select Signals and Systems to Bound (Optional)** to view the signal and linearization I/O selection area.

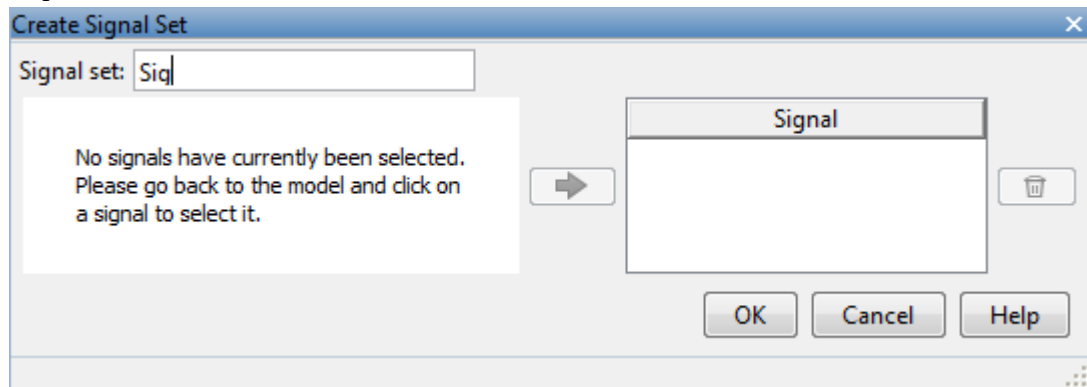
- To apply this requirement to a model signal:

In the **Signal** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, it appears in the list. Select the corresponding check box.

If you have not selected a signal to log:

- Click . A Create Signal Set dialog box opens where you specify the logged signal.
- In the Simulink model window, click the signal to which you want to add a requirement.




The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- Select the signal and click  to add it to the signal set.
- In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

- To apply this requirement to a linear system:
 - a** Specify the simulation time at which the model is linearized in **Snapshot Times**. For multiple simulation snapshot times, specify a vector.
 - b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box. For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

7 Click **OK**.

A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.

See Also

Related Examples

- “Design Optimization to Meet a Custom Objective (GUI)” on page 3-115
- “Design Optimization to Meet Custom Signal Requirements (GUI)” on page 3-142
- “Specify Time-Domain Design Requirements in the Tool” on page 3-19
- “Specify Frequency-Domain Design Requirements in the Tool” on page 3-56

Move Constraints

In this section...

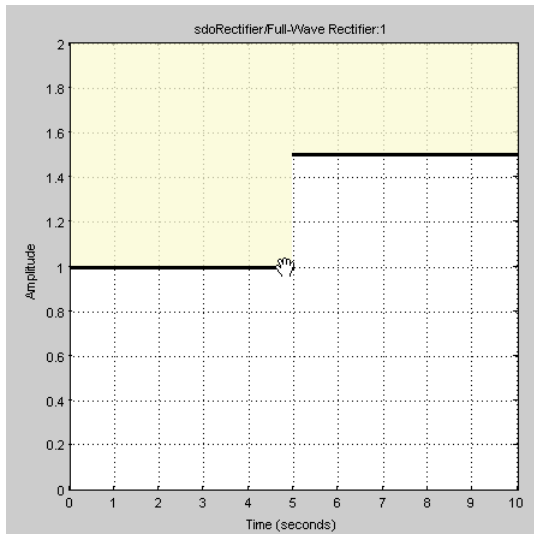
“Move Constraints Graphically” on page 3-16

“Position Constraints Exactly” on page 3-17

Constraint-bound edges define time-domain constraints you would like to place on a particular signal in your model. You can position these edges, which appear as a yellow shaded region bordered by a black line, graphically on page 3-16 or exactly on page 3-17.

Move Constraints Graphically

Use the mouse to click and drag edges in the amplitude versus time plot, as shown in the following figure.



- To move a constraint edge boundary or to change the slope of a constraint edge, position the pointer over a constraint edge endpoint, and press and hold down the left mouse button. The pointer should change to a hand symbol. While still holding the button down, drag the pointer to the target location, and release the mouse button. Note that the edges on either side of the boundary might not maintain their slopes.

- To move an entire constraint edge up, down, left, or right, position the mouse pointer over the edge and press and hold down the left mouse button. The pointer should change to a four-way arrow. While still holding the button down, drag the pointer to the target location, and release the mouse button. Note that the edges on either side of the boundary might not maintain their slopes.

To move a constraint edge to a perfectly horizontal or vertical position, hold down the **Shift** key while clicking and dragging the constraint edge. This causes the constraint edge to *snap* to a horizontal or vertical position.

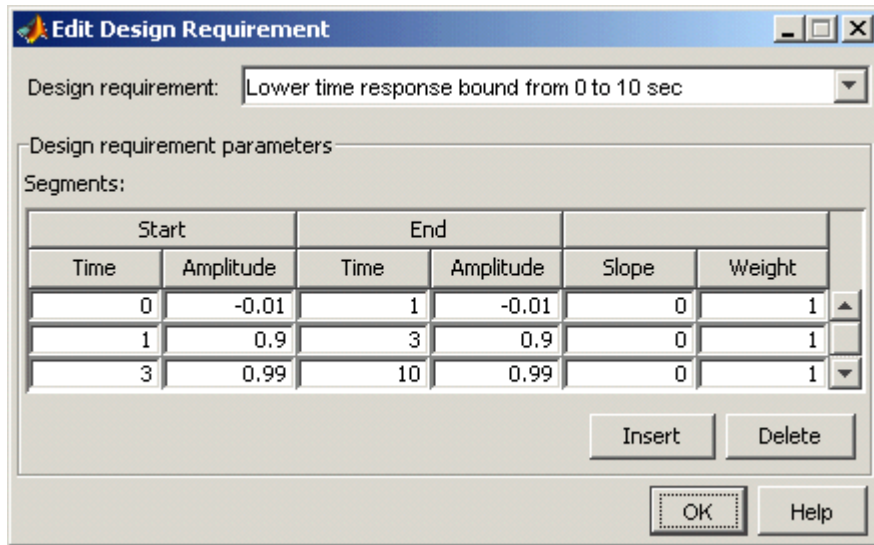
When moving constraint bound edges, it is sometimes helpful to display gridlines on the axes for careful alignment of the constraint bound edges. To turn the gridlines on or off, right-click within the axes and select **Grid**.

Note You can move a lower bound constraint edge above an upper bound constraint edge, or vice versa, but this produces an error when you attempt to run the optimization.

Position Constraints Exactly

To position a constraint edge exactly:

- 1 Position the pointer over the edge you want to move and right-click. Select **Edit** to open the Edit Design Requirement dialog box.



- 2 Specify the position of each constraint edge in the **Time** and **Amplitude** columns.

See Also

More About

- “Specify Time-Domain Design Requirements in the Tool” on page 3-19
- “Specify Frequency-Domain Design Requirements in the Tool” on page 3-56

Specify Time-Domain Design Requirements in the Tool

In this section...

- “Specify Piecewise-Linear Lower and Upper Bounds” on page 3-19
- “Specify Signal Property Requirements” on page 3-21
- “Specify Step Response Characteristics” on page 3-23
- “Track Reference Signals” on page 3-26
- “Impose Elliptic Bound on Phase Plane Trajectory of Two Signals” on page 3-28
- “Specify Custom Requirements” on page 3-31
- “Edit Design Requirements” on page 3-34

In the Response Optimization tool, you can specify the following time-domain requirements:


- **Signal Bound** — “Specify Piecewise-Linear Lower and Upper Bounds” on page 3-19
- **Signal Property** — “Specify Signal Property Requirements” on page 3-21
- **Step Response Envelope** — “Specify Step Response Characteristics” on page 3-23
- **Signal Tracking** — “Track Reference Signals” on page 3-26
- **Ellipse Region Constraint** — “Impose Elliptic Bound on Phase Plane Trajectory of Two Signals” on page 3-28
- **Custom Requirement** — “Specify Custom Requirements” on page 3-31

After you specify the constraints, you can see if the requirements are satisfied by optimizing the design variables. For more information, see “Specify Optimization Options” on page 3-84.

Specify Piecewise-Linear Lower and Upper Bounds

To specify upper and lower bounds on a signal:

- 1 In the Response Optimization tool, select **Signal Bound** in the **New** drop-down list. A window opens where you specify upper or lower bounds on a signal.
- 2 Specify a requirement name in the **Name** box.
- 3 Select the requirement type using the **Type** list.


- 4 Specify the edge start and end times and corresponding amplitude in the **Time (s)** and **Amplitude** columns.
- 5 Click  to specify additional bound edges.

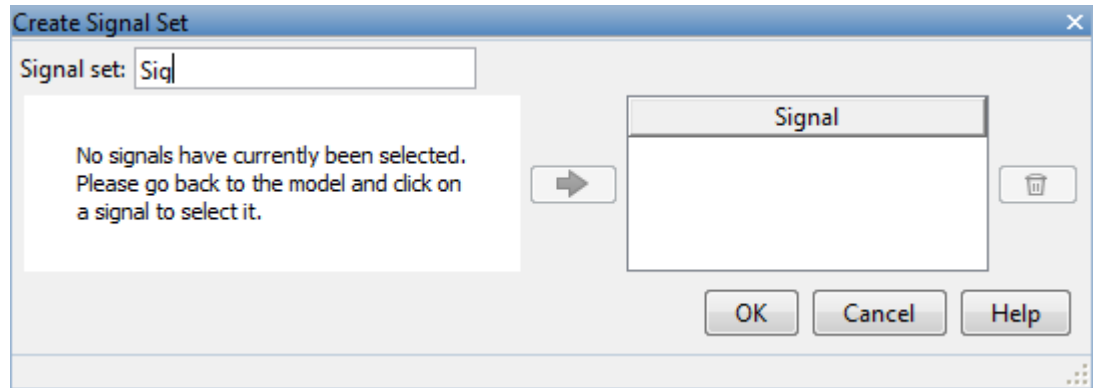
Select a row and click  to delete a bound edge.

- 6 In the **Select Signals to Bound** area, select a logged signal to apply the requirement to.


If you have already selected signals, as described in “Specify Signals to Log” on page 3-12, they appear in the list. Select the corresponding check-box.

If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.
- b In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

7 Click **OK**.

A variable with the specified requirement name appears in the **Data** area of the tool. A graphical display of the requirement also appears in the Response Optimization tool window.

8 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can add a Check Custom Bounds block to your model to specify piecewise-linear bounds.

Specify Signal Property Requirements

To specify signal property requirements:

- 1 In the Response Optimization tool, select **Signal Property** in the **New** drop-down list. A Create Requirement window opens where you specify signal property requirements.
- 2 In the **Name** box, specify a requirement name.
- 3 In the **Specify Property** area, specify a signal property requirement using the **Property** and **Type** lists and the **Bound** box.

Property List

Property	Description	Time weighting available
Signal minimum	Minimum of the signal	No
Signal maximum	Maximum of the signal	No
Signal final value	Last signal value	No
Signal mean	Average of signal value	Yes
Signal median	Middle value of signal	Yes
Signal variance	Variance of signal	Yes
Signal interquartile range	Difference between the 75th and 25th percentiles of signal values	No

Property	Description	Time weighting available
Signal sum	$\sum_{i=t_0}^{t_N} S(i)$, where $S(t_0), \dots, S(t_N)$ is the signal to constrain.	Yes
Signal sum square	$\sum_{i=t_0}^{t_N} S(i)^2$	Yes
Signal sum absolute	$\sum_{i=t_0}^{t_N} S(i) $	Yes

For signal properties where the **Time-Weighted** option is available, you can select it to weight the property computation by the time intervals between samples.

Custom Signal Property

You can add a custom signal property to the **Property** list by editing the function `sdo.requirements.signalPropertyFcns`.

- a** At the MATLAB command prompt, enter `edit sdo.requirements.signalPropertyFcns`.
- b** Add your signal property function to the `FcnData` cell array.


Your signal property function must be on the path.

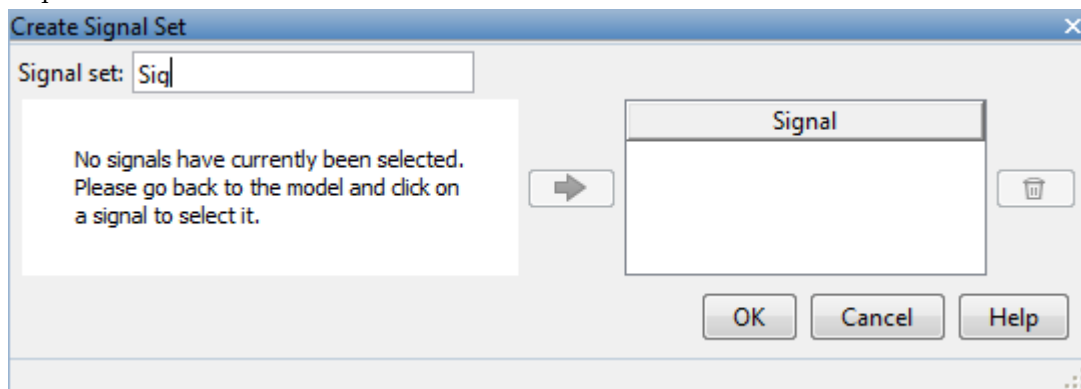
- 4** In the **Select Signals to Bound** area, select the logged signal to which you want to apply the requirement.

The signal selected must have numeric type data (either floating-point or integer). Also, if the property selected is Signal median, Signal variance, or Signal interquartile range, then the signal data must be floating-point (either double or single).


If you have already selected a signal, as described in “Specify Signals to Log” on page 3-12, the signal appears in the list. Select the corresponding check box for that signal.

If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.
- b In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

- 5 Click **OK**.

A variable with the specified requirement name appears in the **Data** area of the tool. An iteration plot depicting the signal property for each iteration also appears in the Response Optimization tool window.

Specify Step Response Characteristics

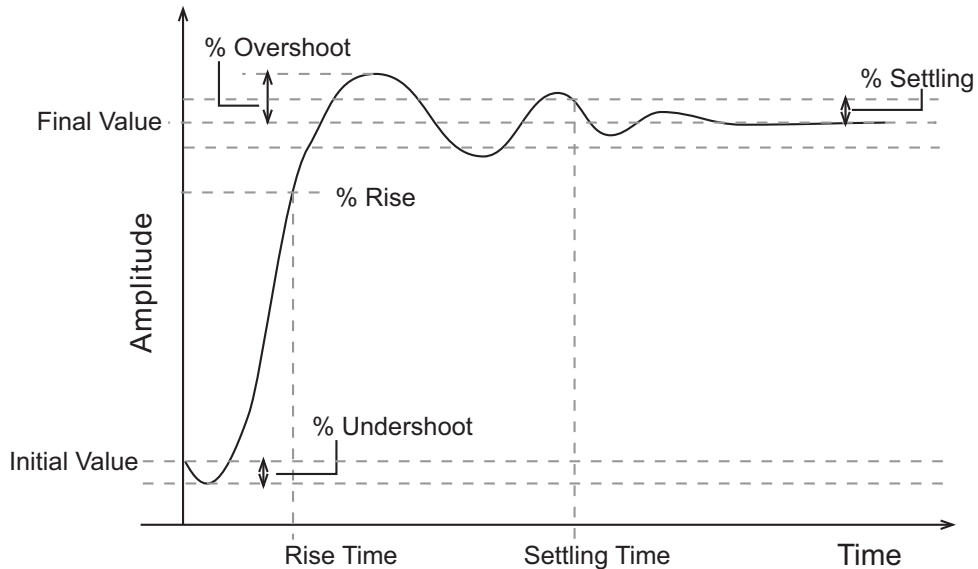
To apply a step response requirement to a signal in your model, specify the step response characteristics as follows:

- 1 Select a step response requirement from the Response Optimization tool.

In the **New** drop-down menu of the tool, in the **New Time Domain Requirement** section, select **Step Response Envelope**.

A Create Requirement dialog box opens where you specify the step response requirements on a signal.

- 2 Specify a requirement name in the **Name** field of the dialog box.
- 3 Specify the step response characteristics:




- **Initial value** — Input level before the step occurs
- **Step time** — Time at which the step takes place
- **Final value** — Input level after the step occurs
- **Rise time** — The time taken for the response signal to reach a specified percentage of the step range. The step range is the difference between the final and initial values.
- **% Rise** — The percentage of the step range used with **Rise time** to define the overall rise time characteristics.
- **Settling time** — Time taken until the response signal settles within a specified region around the final value. This settling region is defined as the final step value plus or minus the specified percentage of the final value.

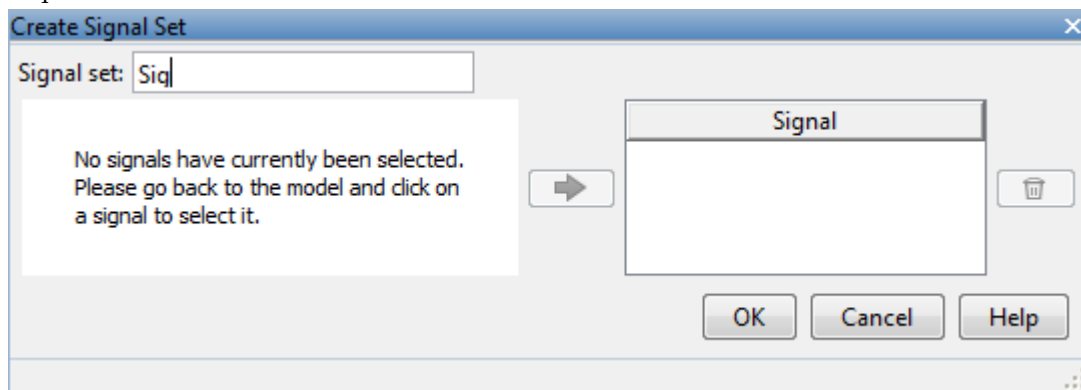
- **% Settling** — The percentage of the final value that defines the settling range of settling time characteristic specified in **Settling time**.
 - **% Overshoot** — The amount by which the response signal can exceed the final value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.
 - **% Undershoot** — The amount by which the response signal can undershoot the initial value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.
- 4 Specify the signal to be bound.

To apply this requirement to a model signal, in the **Select Signals to Bound** area, select a logged signal to which you will apply the requirement.

If you have already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, it appears in the list. Select the corresponding check-box.

If you have not selected a signal to log:

- Click . The Create Signal Set dialog box opens where you specify the logged signal.
- In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- Select the signal and click  to add it to the signal set.

- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

Alternatively, you can use the Check Step Response Characteristics block to specify step response bounds for a signal.

See Also

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

Track Reference Signals

Use reference tracking to force a model signal to match a desired signal. To track a reference signal:


- 1 In the Response Optimization tool, select **Signal Tracking** in the **New** drop-down list. A window opens where you specify the reference signal to track.
- 2 Specify a requirement name in the **Name** box.
- 3 Define the reference signal by entering vectors, or variables from the workspace, in the **Time vector** and **Amplitude** fields.

Click **Update reference signal data** to use the new amplitude and time vector as the reference signal.

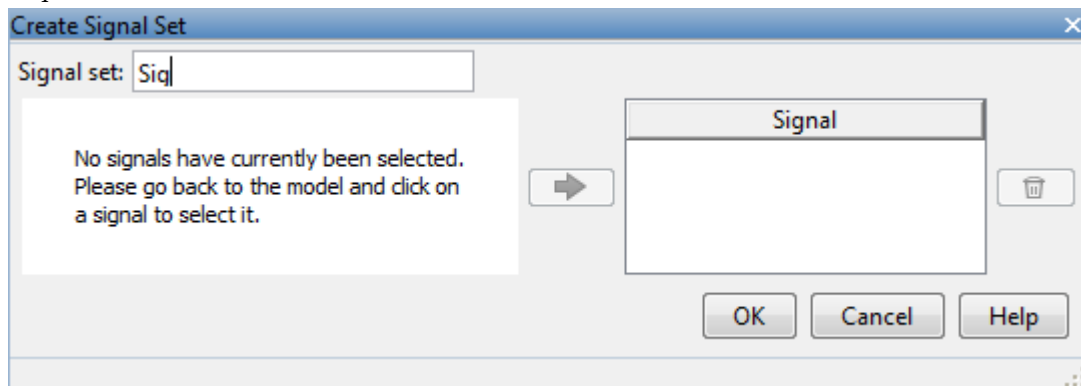
- 4 Specify how the optimization solver minimizes the error between the reference and model signals using the **Tracking Method** list:
 - SSE — Reduces the sum of squared errors
 - SAE — Reduces the sum of absolute errors
- 5 In the **Specify Signal to Track Reference Signal** area, select a logged signal to apply the requirement to.

If you already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, they appear in the list. Select the corresponding check-box.


If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.

- b In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

- e Select the check-box corresponding to the signal and click **OK**.

A variable with the specified requirement name appears in the **Data** area of the tool. A graphical display of the signal bound also appears in the Response Optimization tool window.

Note When tracking a reference signal, the software ignores the maximally feasible solution option. For more information on this option, in the **Response Optimization** tab, click **Options > Optimization Options**, and click **Help**.

Alternatively, you can use the Check Against Reference block to specify a reference signal to track.

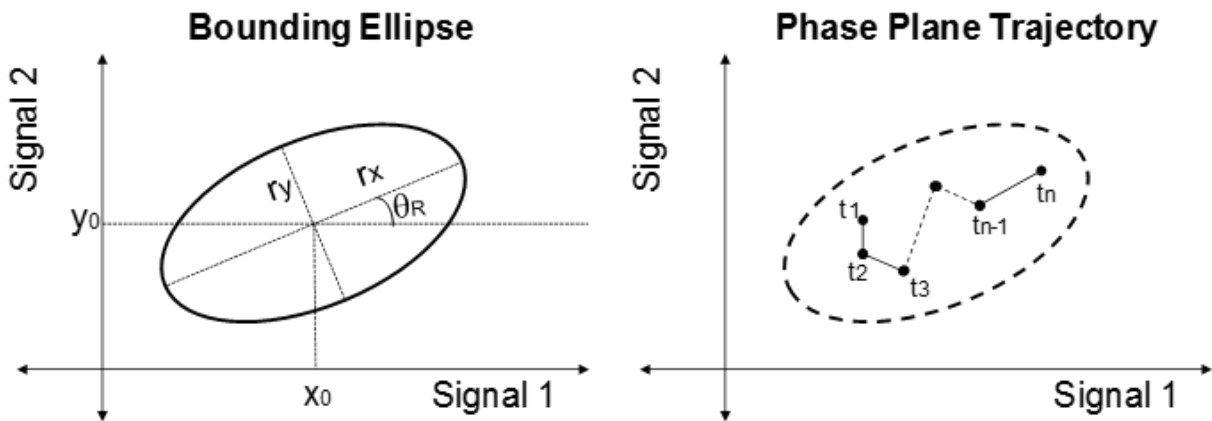
See Also

“Design Optimization to Track Reference Signal (GUI)”

Impose Elliptic Bound on Phase Plane Trajectory of Two Signals

You can impose an elliptic bound on the phase plane trajectory of two signals in your Simulink model. The phase plane trajectory is a plot of the two signals against each other. You specify the radii, center, and rotation of the bounding ellipse. You also specify whether you require the trajectory of the two signals to lie inside or outside the ellipse.

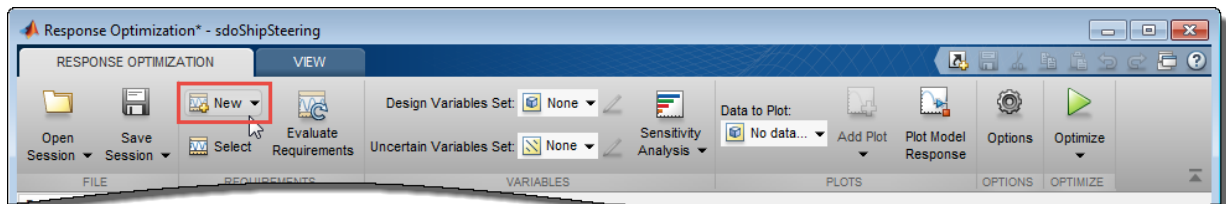
The following image shows the bounding ellipse and an example of the phase plane trajectory of two signals.



The X-Y plane is the phase plane defined by the two signals. r_x and r_y are the radii of the bounding ellipse along the x and y axes, and θ_R is the rotation of the ellipse about the center. The ellipse center is at (x_0, y_0) . In the image, the phase plane trajectory of the signals lies within the bounding ellipse for all time points t_1 to t_n .

To specify the elliptical bound requirement:

- 1 In the Response Optimization tool, in **New** drop-down list, select **Ellipse Region Constraint**.



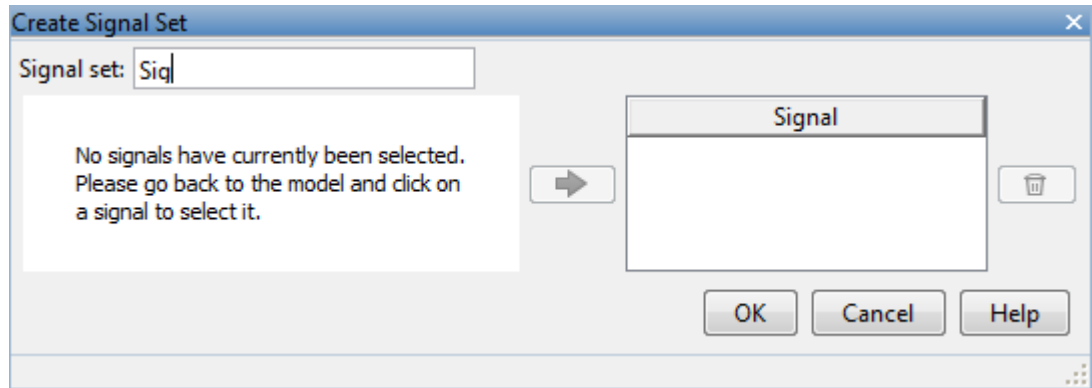
In the Create Requirement dialog box, specify the requirement.


The screenshot shows a dialog box titled "Create Requirement" with a close button (X) in the top right corner. The main title is "Ellipse Region Constraint" and the subtitle is "Constrain a pair of signals to an elliptical region in signal phase plane." Below this, there is a "Name:" field containing the text "PhasePlaneEllipse". A section titled "Specify ellipse bound" is expanded, showing a table with columns for "Signal", "Semi-axis length", and "Center". There are two rows in the table, each with a "Select" button and the text "Click to select signal." The first row has a semi-axis length of "1" and a center of "0". The second row has a semi-axis length of "0.5" and a center of "0". Below the table is an "Angle (rad):" field with the value "0". At the bottom, there is a "Type:" dropdown menu set to "Signal <= bound".

- 2 Specify a requirement name in **Name**.
- 1 Specify the two signals that you want to impose the requirement on. The signals define the X-Y plane of the bounding ellipse. To specify the signals, click the corresponding **Select** buttons.

This screenshot shows the "Specify ellipse bound" section of the dialog box after the signals have been selected. The "Signal" column now contains two entries: ".../Controller/Derivative with LPF" and "sdoShipSteering/Controller/Speed". The "Semi-axis length" column contains "1" and "0.5", and the "Center" column contains "0" and "0". The "Angle (rad):" field remains "0". The signal names are highlighted in red in the original image.

When you click **Select**, the Create Signal Set dialog box opens.



In the Simulink model window, click the signal to which you want to add the requirement. The Create Signal Set dialog box updates with the name of the block and the port number where the selected signal is located. Select the signal, and click  to add it to the signal set.

Once you have specified the logged signal in the Create Signal Set dialog box, the signal appears in the Create Requirement dialog box.

- 2 Specify the radii of the bounding ellipse as real positive finite values in **Semi-axis length**. You specify r_x and r_y that are the x -axis and y -axis radii before any rotation about the ellipse center.
- 3 Specify the location of the center of the bounding ellipse in **Center**. You specify x_0 and y_0 , the x and y coordinates of the center, as real finite values.
- 4 Specify the angle of rotation of the ellipse about its center as a real finite scalar in **Angle (rad)**.
- 5 Specify the bound **Type** as one of the following:
 - ' \leq ' — Ellipse is an upper bound. The phase plane trajectory of the two signals should lie inside or on the ellipse.
 - ' \geq ' — Ellipse is a lower bound. The phase plane trajectory of the two signals should lie outside or on the ellipse.
- 1 (Optional) To create an iteration plot that shows the evaluated requirement value for each optimization iteration, select **Create Plot**. The plot is populated when you perform optimization. During optimization, the software computes the signed minimum distance of each point in the phase plane trajectory to the bounding

ellipse. The maximum of these signed distances is returned and plotted at each iteration. A positive value indicates that the requirement has been violated and at least one of the trajectory points lies outside the bounding region.

- 2 Click **OK**.


A new variable, with the specified requirement name, appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.

Specify Custom Requirements

You can specify custom requirements, such as minimizing system energy. To specify custom requirements:

- 1 In the Response Optimization tool, in **New** drop-down menu, select **Custom Requirement**. The Create Requirement dialog box opens where you specify the custom requirement.
- 2 Specify a requirement name in **Name**.
- 3 Specify the requirement type in the **Type** drop-down menu.
- 4 Specify the name of the function that contains the custom requirement in **Function**. The field must be specified as a function handle using @. The function must be on the

MATLAB path. Click  to review or edit the function.

If the function does not exist, clicking  opens a template MATLAB file. Use this file to implement the custom requirement. The default function name is myCustomRequirement.

- 5 (Optional) To prevent the solver from considering specific parameter combinations, select **Error if constraint is violated**. Use this option for parameter-only constraints.

During an optimization iteration, the solver first evaluates requirements with this option selected.

- If the constraint is violated, the solver skips evaluating any remaining requirements and proceeds to the next iteration.

- If the constraint is *not* violated, the solver evaluates the remaining requirements for the current iteration. If any of the remaining requirements bound signals or systems, then the solver simulates the model.

For more information, see “Skip Model Simulation Based on Parameter Constraint Violation (GUI)” on page 3-190.

Note If you select this check box, then do not specify signals or systems to bound. If you *do* specify signals or systems, then this check box is ignored.

6 (Optional) Specify the signal or system, or both, to be bound.

You can apply this requirement to model signals, or a linearization of your Simulink model (requires Simulink Control Design), or both.


Click **Select Signals and Systems to Bound (Optional)** to view the signal and linearization I/O selection area.

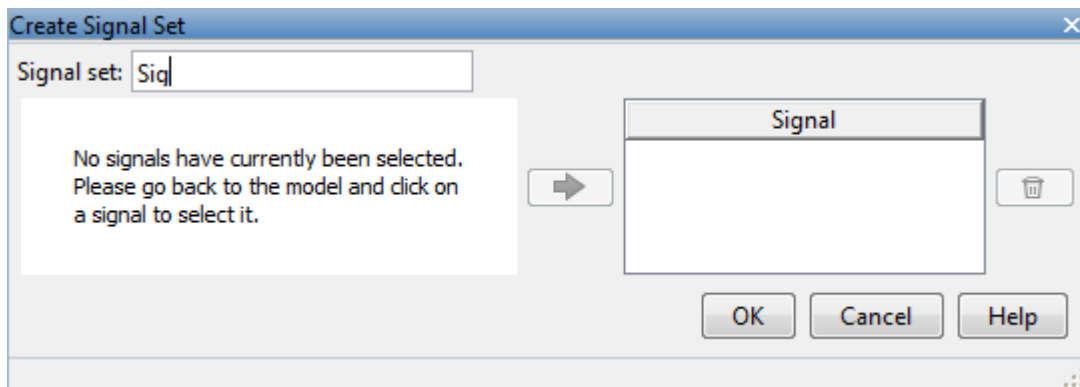
- To apply this requirement to a model signal:

In the **Signal** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, it appears in the list. Select the corresponding check box.

If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.
- b In the Simulink model window, click the signal to which you want to add a requirement.




The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

- To apply this requirement to a linear system:
 - a Specify the simulation time at which the model is linearized in **Snapshot Times**. For multiple simulation snapshot times, specify a vector.
 - b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box. For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 7 Click **OK**.

A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.

See Also

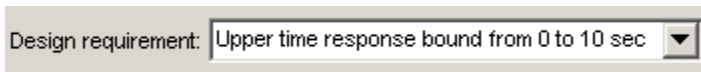
- “Design Optimization to Meet a Custom Objective (GUI)” on page 3-115
- “Design Optimization to Meet a Custom Objective (Code)” on page 3-133

Edit Design Requirements

The Edit Design Requirement dialog box allows you to exactly position constraint segments and to edit other properties of these constraints. The dialog box has two main components:

- An upper panel to specify the constraint you are editing
- A lower panel to edit the constraint parameters

The upper panel of the Edit Design Requirement dialog box resembles the image in the following figure.



In the **Control System Designer** app in Control System Toolbox™, you can edit design requirements from the analysis plots. The **Design requirement** drop-down list will contain all the requirements on that plot.

Edit Design Requirement Dialog Box Parameters

The particular parameters shown within the lower panel of the Edit Design Requirement dialog box depend on the type of constraint/requirement. In some cases, the lower panel contains a grid with one row for each segment and one column for each constraint parameter. The following table summarizes the various constraint parameters.

Edit Design Requirement Dialog Box Parameters

Parameter	Found in	Description
Time	Upper and lower time response bounds on step and impulse response plots	Defines the time range of a segment within a constraint/requirement.
Amplitude	Upper and lower time response bounds on step and impulse response plots	Defines the beginning and ending amplitude of a constraint segment.
Slope (1/s)	Upper and lower time response bounds	Defines the slope, in 1/s, of a constraint segment. It is an alternative method of specifying the magnitude values. Entering a new Slope value changes any previously defined magnitude values.
Final value	Step response bounds	Defines the input level after the step occurs.
Rise time	Step response bounds	Defines a constraint segment for a particular rise time.
% Rise	Step response bounds	The percentage of the step's range used to describe the rise time.
Settling time	Step response bounds	Defines a constraint segment for a particular settling time.
% Settling	Step response bounds	The percentage of the final value that defines the settling region used to describe the settling time.
% Overshoot	Step response bounds	The percentage amount by which the signal can exceed the final value before settling.
% Undershoot	Step response bounds	Defines the constraint segments for a particular percent undershoot.

See Also

Related Examples

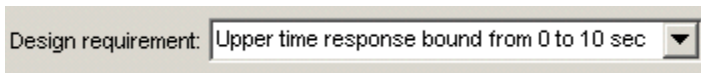
- “Specify Design Variables” on page 3-76
- “Specify Variable Requirements in the Tool” on page 3-39
- “Specify Frequency-Domain Design Requirements in the Tool” on page 3-56
- “Specify Optimization Options” on page 3-84
- “Design Optimization Using Lookup Table Requirements for Gain Scheduling (GUI)” on page 6-73

Edit Design Requirements

The Edit Design Requirement dialog box allows you to exactly position constraint segments and to edit other properties of these constraints. The dialog box has two main components:

- An upper panel to specify the constraint you are editing
- A lower panel to edit the constraint parameters

The upper panel of the Edit Design Requirement dialog box resembles the image in the following figure.



In the context of the **Control System Designer** app in Control System Toolbox, **Design requirement** is associated with both the analysis plot or editor that contains the requirement and the particular requirement itself. To edit other constraints within the app, select another design requirement from the drop-down menu.

Edit Design Requirement Dialog Box Parameters

The particular parameters shown within the lower panel of the Edit Design Requirement dialog box depend on the type of constraint/requirement. In some cases, the lower panel contains a grid with one row for each segment and one column for each constraint parameter. The following table summarizes the various constraint parameters.

Edit Design Requirement Dialog Box Parameters

Parameter	Found in	Description
Time	Upper and lower time response bounds on step and impulse response plots	Defines the time range of a segment within a constraint/requirement.
Amplitude	Upper and lower time response bounds on step and impulse response plots	Defines the beginning and ending amplitude of a constraint segment.
Slope (1/s)	Upper and lower time response bounds	Defines the slope, in 1/s, of a constraint segment. It is an alternative method of specifying the magnitude values. Entering a new Slope value changes any previously defined magnitude values.
Final value	Step response bounds	Defines the input level after the step occurs.
Rise time	Step response bounds	Defines a constraint segment for a particular rise time.
% Rise	Step response bounds	The percentage of the step's range used to describe the rise time.
Settling time	Step response bounds	Defines a constraint segment for a particular settling time.
% Settling	Step response bounds	The percentage of the final value that defines the settling region used to describe the settling time.
% Overshoot	Step response bounds	The percentage amount by which the signal can exceed the final value before settling.
% Undershoot	Step response bounds	Defines the constraint segments for a particular percent undershoot.

Specify Variable Requirements in the Tool

In this section...

“Impose Monotonic Constraint Requirement on Variable” on page 3-39

“Impose Upper Bound on Gradient Magnitude of Variable” on page 3-42

“Specify Linear or Quadratic Function Matching Constraint” on page 3-45

“Specify Requirement on a Vector Property” on page 3-50

“Impose Relational Constraint Between Two Variables” on page 3-53

In the Response Optimization tool, you can specify the following constraints on Simulink model parameters that are specified as variables:

- **Monotonic Variable** — “Impose Monotonic Constraint Requirement on Variable” on page 3-39
- **Smoothness Constraint** — “Impose Upper Bound on Gradient Magnitude of Variable” on page 3-42
- **Function Matching** — “Specify Linear or Quadratic Function Matching Constraint” on page 3-45
- **Vector Property** — “Specify Requirement on a Vector Property” on page 3-50
- **Relational Constraint** — “Impose Relational Constraint Between Two Variables” on page 3-53

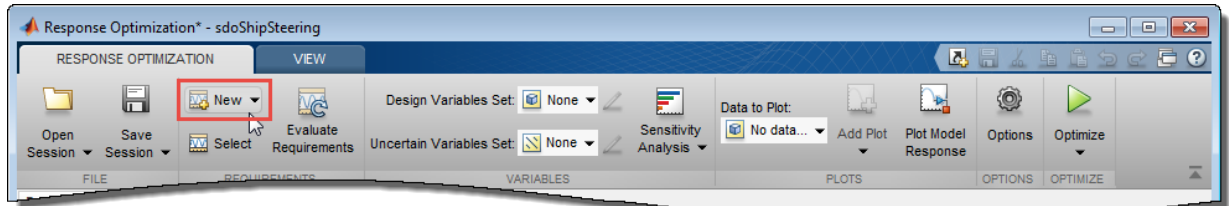
For information about how to specify a model parameter as a variable, see “Add Model Parameters as Variables for Optimization” on page 3-76. After you specify the constraints, you can see if the requirements are satisfied by optimizing the design variables. For more information, see “Specify Optimization Options” on page 3-84.

Impose Monotonic Constraint Requirement on Variable

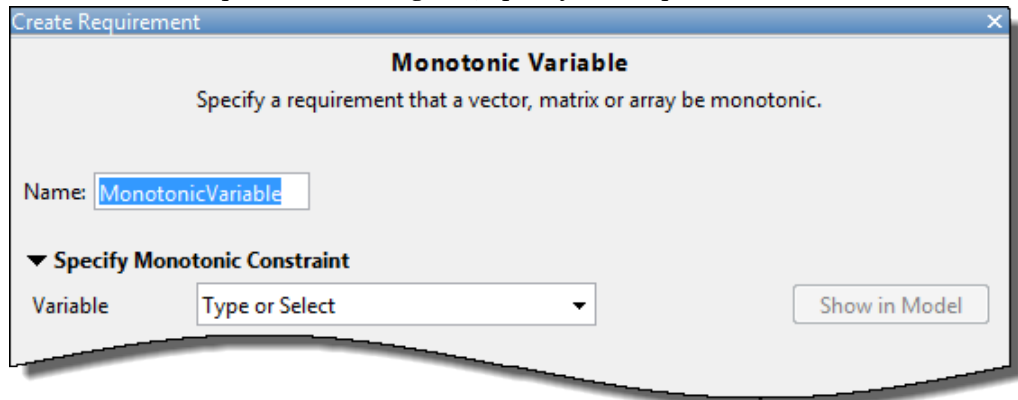
You can impose a monotonic constraint requirement on a design variable in your Simulink model. For example, constrain a variable to be monotonically increasing. The variable can be a vector, matrix, or multidimensional array that is a parameter in your model, such as the breakpoints of a lookup table.

To specify the requirement:

- 1 In the Response Optimization tool, in **New** drop-down menu, select **Monotonic Variable**.



In the Create Requirement dialog box, specify the requirement.



- 2 Specify a requirement name in **Name**.
- 3 Specify the name of the variable in **Variable**. The variable must be a vector, matrix, or multidimensional array of data type `double` or `single`.

You can type the name of a nonscalar variable, or select the variable from the drop-down list. The list is prepopulated with all the nonscalar variables in your model. To choose a subset of an array or matrix variable V , type an expression. For example, specify **Variable** as $V(1, :)$ to use the first row of the variable. To use a numeric nonscalar field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$.

Sometimes, models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as $k = a + b$, where a and b are not defined in the model but k is used. To add these

independent parameters as variables in the Response Optimization tool, see “Add Model Parameters as Variables for Optimization” on page 3-76.

- 1 Specify the monotonicity for each dimension of the variable.

After you select the variable, the dialog updates to show **Dimension 1** to **Dimension n**, corresponding to the n dimensions of the variable. For example, for a 2-dimensional variable κ of size 3-by-5, the dialog updates as shown.

The screenshot shows a dialog box titled "Create Requirement" with a sub-header "Smoothness Constraint". Below the sub-header is the instruction "Specify a smoothness requirement on a vector, matrix or array." The "Name" field contains "SmoothnessReq". Under the "Specify Smoothness Constraint" section, the "Gradient maximum magnitude" is set to "1". The "Dependent Variable" is set to κ . The "Independent Variable" section has two dimensions: "Dimension 1" and "Dimension 2", both set to "Type or Select". To the right of each dimension field is a "Show in Model" button. The dialog also has a "Show in Model" button next to the "Dependent Variable" field.

Specify the monotonicity for the first dimension in **Dimension 1** and for the n^{th} -dimension in **Dimension n** as one of the following options:

- **Strictly increasing** — Each element of the variable is greater than the previous element in that dimension.
- **Increasing** — Each element of the variable is greater than or equal to the previous element in that dimension.
- **Decreasing** — Each element of the variable is less than or equal to the previous element in that dimension.
- **Strictly decreasing** — Each element of the variable is less than the previous element in that dimension.

- **Not constrained** — No constraint exists between the elements of the variable in that dimension.
- 1 (Optional) To create an iteration plot that shows the evaluated requirement value for each optimization iteration, select **Create Plot**. The plot is populated when you perform optimization. The plot shows the evaluated requirement value corresponding to each dimension of the variable. A positive value indicates that the requirement has been violated.
 - 2 Click **OK**.

A new variable, with the specified requirement name, appears in the **Data** area of the Response Optimization tool.

Impose Upper Bound on Gradient Magnitude of Variable

You can impose an upper bound on the gradient magnitude of a variable in your Simulink model. The variable can be a vector, matrix, or multidimensional array that is a parameter in your model, such as the data of a lookup table. For example, consider a car engine controller whose gain changes under different operating conditions determined by the car speed. You can use a gradient bound constraint to limit the rate at which the controller gain changes per unit change in vehicle speed.

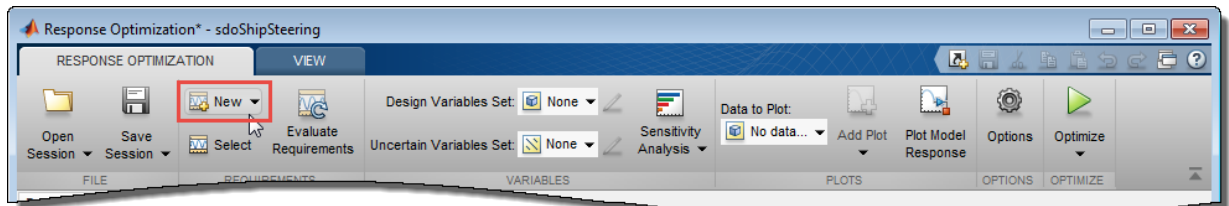
For an N -dimensional variable F that is a function of independent variables x_1, \dots, x_N , the gradient magnitude is defined as:

$$|\nabla F| = \sqrt{\left(\frac{\partial F}{\partial x_1}\right)^2 + \left(\frac{\partial F}{\partial x_2}\right)^2 + \dots + \left(\frac{\partial F}{\partial x_N}\right)^2}$$

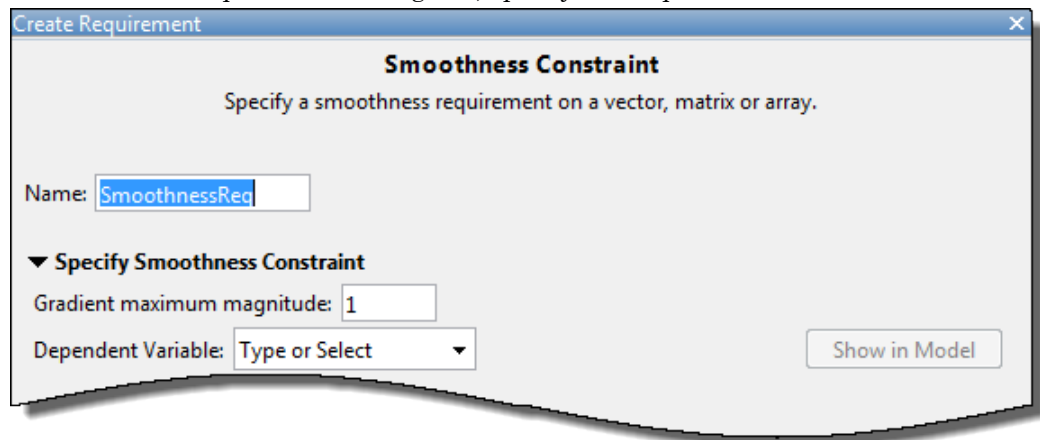
To compute the gradient magnitude, the software computes the partial derivative in each dimension by computing the difference between successive F data in that dimension and dividing by the spacing between the data in that dimension. You specify F and the spacing between the data. The software checks whether the gradient magnitude of the variable data is less than or equal to a specified bound. If the gradient magnitude of the data is greater than the required bound, the variable data is not smooth.

To specify the requirement:

- 1 In the Response Optimization tool, in **New** drop-down list, select **Smoothness Constraint**.



In the Create Requirement dialog box, specify the requirement.



- 2 Specify a requirement name in **Name**.
- 3 Specify the gradient magnitude bound as a nonnegative finite real scalar in **Gradient maximum magnitude**.
- 4 Specify the variable F that you want to impose the requirement on in **Dependent Variable**. The variable must be a vector, matrix, or multidimensional array of data type `double` or `single`. The variable must be a parameter in your model or a constant that you enter.

You can type the name of a nonscalar variable or constant, or select the variable from the drop-down list. The list is prepopulated with all the nonscalar variables in your model. To choose a subset of an array or matrix variable V , type an expression. For example, specify **Variable** as $V(1, :)$ to use the first row of the variable. To use a numeric nonscalar field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$.

Sometimes, models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as $k = a + b$, where a and b are not defined in the model but k is used. To add these independent parameters as variables in the Response Optimization tool, see “Add Model Parameters as Variables for Optimization” on page 3-76.

- 1 Specify the spacing between points of **Dependent Variable** data in each dimension in **Independent Variable**.

After you select the **Dependent Variable**, the dialog updates to show **Dimension 1** to **Dimension n**, corresponding to the n dimensions of the dependent variable. For example, for a 1-dimensional variable k , the dialog updates as shown.

The screenshot shows a dialog box titled "Create Requirement" with a sub-title "Smoothness Constraint". The main instruction is "Specify a smoothness requirement on a vector, matrix or array." The "Name" field contains "SmoothnessReq". Under the "Specify Smoothness Constraint" section, the "Gradient maximum magnitude" is set to "1". The "Dependent Variable" is set to "k", with a "Show in Model" button to its right. The "Independent Variable" section shows "Dimension 1" set to "Type or Select", with a note "<4 element vector, or scalar>" and a "Show in Model" button to its right.

The first dimension specifies the spacing going down the dependent variable data rows, and the second specifies spacing across the columns. The N th dimension specifies the spacing along the N th dimension of dependent variable data. You can specify the independent variables in each dimension as scalars or vectors.

- **Scalars** — Specify the spacing between dependent variable data F in the corresponding dimension as a nonzero scalar. For example, suppose that **Dependent Variable** is two-dimensional, and the spacing between data in the first dimension is 5 and in the second dimension is 2. In the **Independent Variable** section, specify **Dimension 1** as 5 and **Dimension 2** as 2.

- **Vectors** — Specify the coordinates of F data in the corresponding dimension as real, numeric, monotonic vectors. The software uses the coordinates to compute the spacing between the dependent variable data points in the corresponding dimension. The length of the vector must match the length of F in the corresponding dimension. You do not have to specify coordinates with uniform spacing. For example, suppose that F is two-dimensional, and the length of the data in the first and second dimension is 3 and 5, respectively. The coordinates of the data in the first dimension are [1 2 3]. In the second dimension, the spacing is not uniform and the coordinates of the data are [1 2 10 20 30]. In the **Independent Variable** section, specify **Dimension 1** as [1 2 3] and **Dimension 2** as [1 2 10 20 30].

You can also specify the independent variables by typing the name of a variable, or selecting a variable from the drop-down list. The list is prepopulated with all the variables in your model that have the appropriate size. To choose a subset of an array or matrix variable V , type an expression. For example, specify as $V(1, :)$ to use the first row of the variable. To use a numeric field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$.

- 1 (Optional) To create an iteration plot that shows the evaluated requirement value for each optimization iteration, select **Create Plot**. The plot is populated when you perform optimization. A positive value indicates that the requirement has been violated.
- 2 Click **OK**.

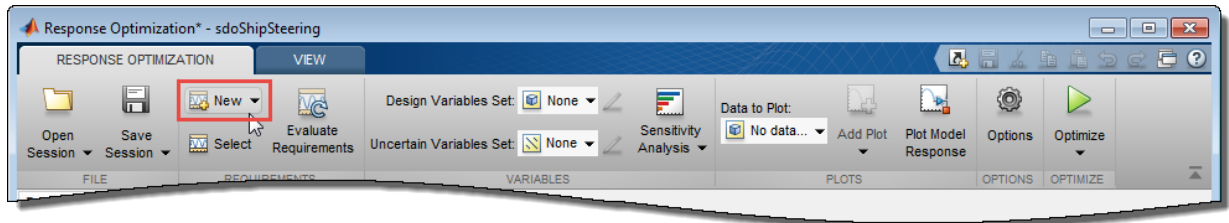
A new variable, with the specified requirement name, appears in the **Data** area of the Response Optimization tool.

Specify Linear or Quadratic Function Matching Constraint

In the tool, you can constrain a variable's values to match a linear or quadratic function. The variable can be a vector, matrix, or a multidimensional array that is a parameter in your model, such as the data of a lookup table in your model. To specify the requirement:

- 1 In Response Optimization tool, from the **New** drop-down list, select **Function Matching**.

3 Response Optimization



In the Create Requirement dialog box, specify the requirement. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.

Create Requirement (X)

Function Matching

Specify that values must match a designated function.

Name:

▼ **Specify Function Matching Constraint**

Functional Relation: ▼

Dependent Variable: ▼

Independent Variable

Dimension 1: ▼ <4 element vector>

Center and scale independent variables

▼ Center and Scale Settings

Use automatic centers and scales

Use custom centers and scales

Independent Variable	Center	Scale
Dimension 1:	<input type="text" value="12"/>	<input type="text" value="2.5820"/>

Create Plot

- 2 Specify the function to be matched. To do so, set **Functional Relation** to one of the following values:

- **Linear** — Data from variable V are fit to a linear function. For example, for a two-dimensional variable with independent variables, X_1 and X_2 , the linear function has the form:

$$V = a_0 + a_1X_1 + a_2X_2$$

The software calculates the fit coefficients a_0 , a_1 , and a_2 and then calculates the sum of squares of the error between the data and the linear function.

- **Quadratic with no cross-terms** — Data are fit to a quadratic function with no cross-terms. For a two-dimensional variable, the pure quadratic function has the form:

$$V = a_0 + a_1X_1 + a_2X_1^2 + a_3X_2 + a_4X_2^2$$

- **Quadratic with all cross-terms** — Variable data are fit to a quadratic function that includes cross-terms. For a two-dimensional variable, the quadratic function has the form:

$$V = a_0 + a_1X_1 + a_2X_1^2 + a_3X_2 + a_4X_2^2 + a_5X_1X_2$$

If the variable is one-dimensional, there are no cross-terms and so the computation is the same as when **Functional relation** is **Quadratic with no cross-terms**.

- 3 Specify the variable V to which you want to apply the requirement in **Dependent Variable**. The variable must be a vector, matrix, or multidimensional array of data type `double` or `single` that is a parameter in your model.

Type the name of a nonscalar variable, or select a variable from the drop-down list. The list is prepopulated with all the nonscalar variables in your model. To see where the selected variable is used on your model, click **Show in Model**. To choose a subset of an array or matrix variable A , type an expression. For example, specify $A(1, :)$ to use the first row of the variable. To use a numeric nonscalar field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$.

Sometimes models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as $k = a + b$, where a and b are not defined in the model but k is used. To add these independent parameters as design variables in the tool, see “Add Model Parameters as Variables for Optimization” on page 3-76.

- 4 Specify the independent variable vectors used for computing the function in **Independent Variable**. The independent variables are specified as real, numeric, monotonic vectors.

The number of independent variables must equal the number of dimensions of the dependent variable V . For example, you specify two independent variables when V is a matrix, and use three independent variables when V is three-dimensional. The first independent variable vector specifies coordinates going down the rows of V , and the second independent variable vector specifies coordinates going across the columns of V . The n^{th} independent variable vector specifies coordinates along the n^{th} dimension of V . The number of elements in each independent variable vector must match the size of V in the corresponding dimension. The independent variable vectors must be monotonically increasing or decreasing.

You can also specify the independent variables by typing the name of a variable, or selecting a variable from the drop-down list. The list is prepopulated with all the variables in your model that have the appropriate size. To choose a subset of an array or matrix variable A , type an expression. For example, specify $A(1, :)$ to use the first row of the variable. To use a numeric field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$. To use an equally spaced vector, select $[1 \ 2 \ \dots N]$ from the drop-down menu.

- 5 Specify whether you want to center and scale the independent variables. When you select the **Center and scale independent variables** option, the independent variable vectors you specify are divided by a scale value after subtracting a center value. Centering can improve numerical conditioning when one or more independent variable vectors have a mean that differs from 0 by several orders of magnitude. Scaling can improve numerical conditioning when independent variable vectors differ from each other by several orders of magnitude.

To specify the center and scale values for each independent variable, expand the **Center and Scale Settings** section, and select one of the following:

- **Use automatic centers and scales** - The center and scale values are the mean and standard deviation for each independent variable. Using the mean and standard deviation values to center and scale the independent variables is the default option.
- **Use custom centers and scales** - Specify the **Center** and **Scale** values for each independent variable. The independent variable vectors are divided by the corresponding **Scale** value after subtracting the value you specify in **Center**.

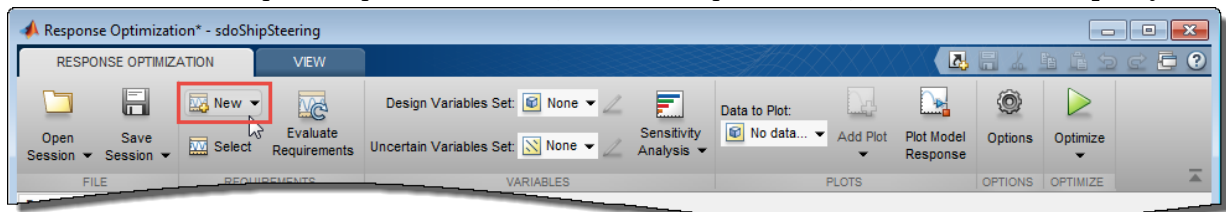
- (Optional) Select the **Create Plot** option to create an iteration plot that shows the evaluated requirement value for each optimization iteration. The software computes an error signal that is the difference between the dependent variable data and the specified function of the independent variables. The sum of squares of this error is plotted when you perform optimization. A positive value indicates that the requirement has been violated, and 0 value indicates that the requirement is satisfied. The closer the value is to 0, the better the match between the function and dependent variable data.
- Close the Create Requirement dialog box.

The requirement created in the **Requirements** area of the tool is updated with the specified characteristics.

Specify Requirement on a Vector Property

You can specify a requirement on a vector property, such as the mean value of the vector. The vector must be a parameter in your model. To specify the requirement:

- In the Response Optimization tool, in **New** drop-down list, select **Vector Property**.



In the Create Requirement dialog box, specify the requirement.

The screenshot shows a dialog box titled "Create Requirement" with a close button (X) in the top right corner. The main title is "Vector Property" and the subtitle is "Specify a requirement on a vector property such as the mean value." Below the subtitle, there is a "Name:" label followed by a text input field containing "VectorProperty". Underneath, there is a section titled "Specify Vector Property" with a downward arrow. This section contains four fields: "Property:" with a dropdown menu showing "Vector mean"; "Type:" with a dropdown menu showing "Constrain property to be <= the bound"; "Bound:" with a text input field containing "0"; and "Variable:" with a dropdown menu showing "Type or Select". To the right of these fields is a button labeled "Show in Model".

- 2 Specify a requirement name in **Name**.
- 1 Specify the vector property in **Property**. For a vector V with N elements, you can specify one of the following properties:

- Vector mean — $mean(V)$
- Vector median — $median(V)$
- Vector variance — $variance(V)$
- Vector inter-quartile range — Difference between the 75th and 25th percentiles of the vector values.

•

• Vector sum — $\sum_{i=1}^N V(i)$

•

• Vector sum of squares — $\sum_{i=1}^N V(i)^2$

•

• Vector sum of absolute values — $\sum_{i=1}^N |V(i)|$

- Vector minimum — $\min(V)$
 - Vector maximum — $\max(V)$
- 2 Specify the type of requirement you want to impose on the vector property in **Type**. You can set an upper or lower bound on the vector property, or require the property to equal a particular value. You can also choose to maximize or minimize the vector property. For example, to maximize the mean value of your vector, specify **Property** as `Vector mean` and **Type** as `Maximize the property`.
 - 3 Specify the value of the bound imposed on the vector property in **Bound**. Specify the bound as a finite real scalar value. For example, if for a vector variable V you require $\text{mean}(V) = 5$, specify **Property** as `Vector mean`, **Type** as `Constrain property to be == the bound`, and **Bound** as `5`.
 - 1 Specify the name of the variable in **Variable**. The variable must be a vector, matrix, or multidimensional array of data type `double` or `single`.

You can type the name of a nonscalar variable, or select the variable from the dropdown list. The list is prepopulated with all the nonscalar variables in your model. To choose a subset of an array or matrix variable V , type an expression. For example, specify **Variable** as `V(1, :)` to use the first row of the variable. To use a numeric nonscalar field x of a structure S , type `S.x`. You cannot use mathematical expressions such as `a + b`.

Sometimes, models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as `k = a + b`, where a and b are not defined in the model but k is used. To add these independent parameters as variables in the Response Optimization tool, see “Add Model Parameters as Variables for Optimization” on page 3-76.

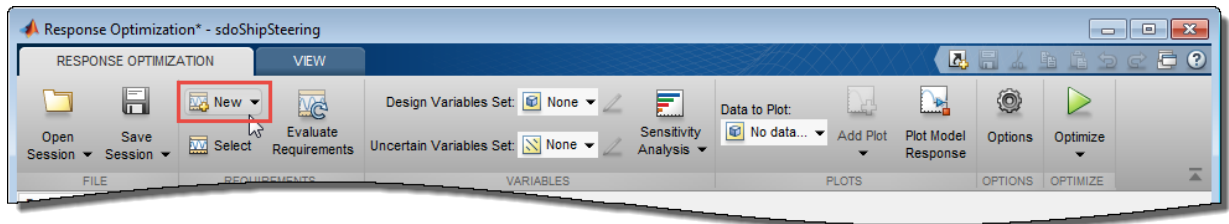
- 2 (Optional) To create an iteration plot that shows the evaluated requirement value for each optimization iteration, select **Create Plot**. The plot is populated when you perform optimization. A positive value indicates that the requirement has been violated.
- 3 Click **OK**.

A new variable, with the specified requirement name, appears in the **Data** area of the Response Optimization tool.

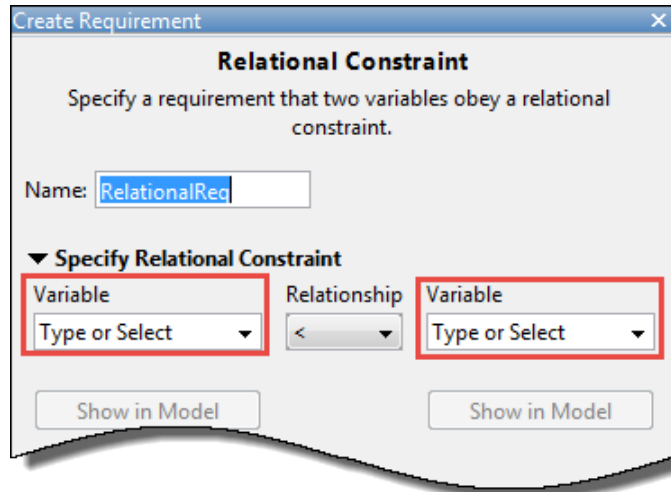
Impose Relational Constraint Between Two Variables

You can impose a relational constraint requirement on a pair of variables in your Simulink model. For example, require that variable a is always greater than variable b . To specify the requirement:

- 1 In the Response Optimization tool, in **New** drop-down list, select **Relational Constraint**.



In the Create Requirement dialog box, specify the requirement.



- 2 Specify a requirement name in **Name**.
- 3 Specify the name of the two variables in **Variable**. The variables can be vectors or arrays but must be the same size.

Type the names of two nonscalar variables, or select the variables from the drop-down lists. The lists are prepopulated with all the nonscalar variables in your model.

To see where a selected variable is used on your model, click **Show in Model**. To choose a subset of an array or matrix variable V , type an expression. For example, specify **Variable** as $V(1, :)$ to use the first row of the variable. To use a numeric nonscalar field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$.

Sometimes, models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as $k = a + b$, where a and b are not defined in the model but k is used. To add these independent parameters as variables in the Response Optimization tool, see “Add Model Parameters as Variables for Optimization” on page 3-76.

1 Specify the relation between the elements of the two variables as one of the following in **Relationship**:

- '<' — Each data element in the first variable is less than the corresponding element in the second variable.
- '<=' — Each data element in the first variable is less than or equal to the corresponding element in the second variable.
- '>' — Each data element in the first variable is greater than the corresponding element in the second variable.
- '>=' — Each data element in the first variable is greater than or equal to the corresponding element in the second variable.
- '==' — Each data element in the first variable is equal to the corresponding element in the second variable.
- '~=' — Each data element in the first variable is not equal to the corresponding element in the second variable.

1 (Optional) To create an iteration plot that shows the evaluated requirement value for each optimization iteration, select **Create Plot**. The plot is populated when you perform optimization. The plot shows the evaluated requirement value corresponding to each element of the variables. The interpretation of the evaluated requirement value depends on the requirement **Type**.

Type	Evaluated Requirement Value	
	Requirement is Satisfied	Requirement is Violated
'>' or '<'	Negative number	Positive number, or 0 if the elements are equal

Type	Evaluated Requirement Value	
	Requirement is Satisfied	Requirement is Violated
'>=' or '<='	Negative number, or 0 if the elements are equal	Positive number
'=='	0	Non-zero number
'~= '	0	1

2 Click **OK**.

A new variable, with the specified requirement name, appears in the **Data** area of the Response Optimization tool.

See Also

Related Examples

- “Specify Design Variables” on page 3-76
- “Specify Time-Domain Design Requirements in the Tool” on page 3-19
- “Specify Frequency-Domain Design Requirements in the Tool” on page 3-56
- “Specify Optimization Options” on page 3-84
- “Design Optimization Using Lookup Table Requirements for Gain Scheduling (GUI)” on page 6-73

Specify Frequency-Domain Design Requirements in the Tool

In this section...

“Specify Lower Bounds on Gain and Phase Margin” on page 3-56

“Specify Piecewise-Linear Lower and Upper Bounds on Frequency Response” on page 3-58

“Specify Bound on Closed-Loop Peak Gain” on page 3-60

“Specify Lower Bound on Damping Ratio” on page 3-62

“Specify Upper and Lower Bounds on Natural Frequency” on page 3-64

“Specify Upper Bound on Approximate Settling Time” on page 3-66

“Specify Piecewise-Linear Upper and Lower Bounds on Singular Values” on page 3-68

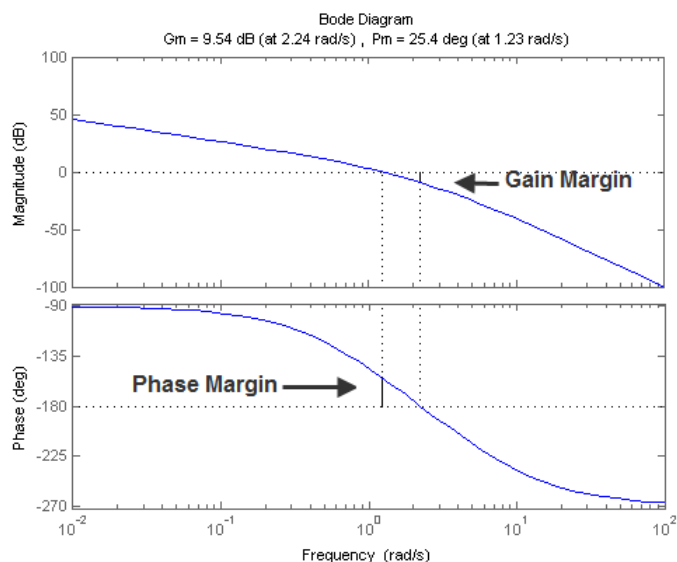
“Specify Step Response Characteristics” on page 3-70

“Specify Custom Requirements” on page 3-72

Specify Lower Bounds on Gain and Phase Margin

To specify lower bounds on the gain and phase margin of a linear system:

- 1 In the Response Optimization tool, select **Gain and Phase Margin** in the **New** list. A window opens where you specify lower bounds on the gain and phase margin of your linear system.
- 2 Specify a requirement name in **Name**.
- 3 Specify bounds on the gain margin or phase margin, or both.



- **Gain margin** — Amount of gain increase or decrease required to make the loop gain unity at the frequency where the phase angle is -180° .
- **Phase margin** — Amount of phase increase or decrease required to make the phase angle -180° when the loop gain is 1.0


To specify a lower bound on the gain margin or phase margin, or both, select the corresponding check box and enter the lower bound value.

- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

5 Click **OK**.

A variable with the specified requirement name appears in the **Data** area of the tool. A graphical display of the requirement also appears in the Response Optimization tool window.

6 (Optional) In the graphical display, you can:


- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Gain and Phase Margins block to specify bounds on the gain and phase margin. (Requires Simulink Control Design.)

Specify Piecewise-Linear Lower and Upper Bounds on Frequency Response

To specify upper or lower bounds on the magnitude of a system response:

- 1** In the Response Optimization tool, select **Bode Magnitude** in the **New** list. A window opens where you specify the lower or upper bounds on the magnitude of the system response.
- 2** Specify a requirement name in the **Name** box.
- 3** Specify the requirement type using the **Type** list.
- 4** Specify the edge start and end frequencies and corresponding magnitude in the **Frequency** and **Magnitude** columns.
- 5** Insert or delete bound edges.

Click  to specify additional bound edges.


Select a row and click  to delete a bound edge.

- 6 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

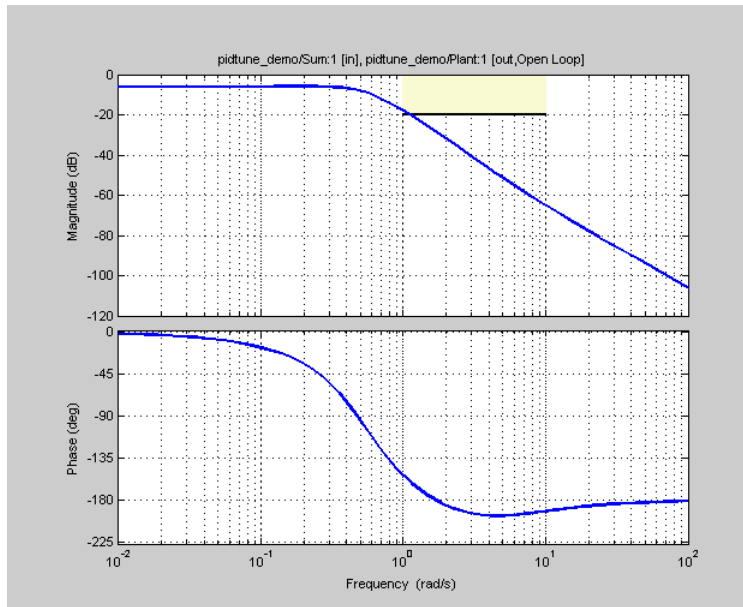
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 7 Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool window. A graphical display of the requirement also appears in the Response Optimization tool window.



8 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Bode Characteristics block to specify bounds on the magnitude of the system response. (Requires Simulink Control Design.)

Specify Bound on Closed-Loop Peak Gain


To specify an upper bound on the closed-loop peak response of a system:

- 1 In the Response Optimization tool, select **Closed-Loop Peak Gain** in the **New** list. A window opens where you specify an upper bound on the closed-loop peak gain of the system.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify the upper bound on the closed-loop peak gain in the **Closed-Loop peak gain** box.
- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a** Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

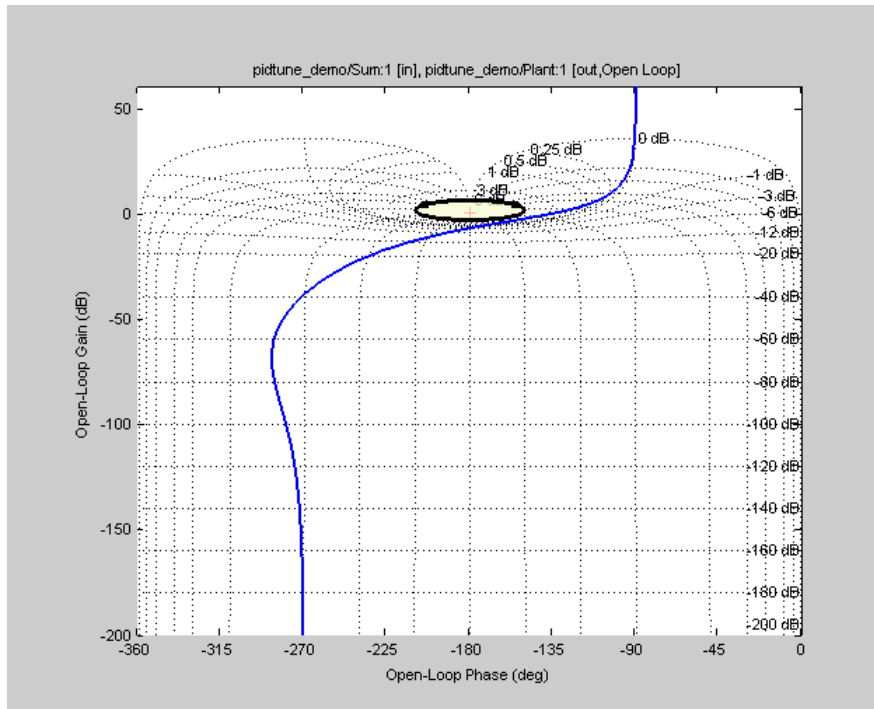
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 5** Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool window. A graphical display of the requirement also appears in the Response Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Nichols Characteristics block to specify bounds on the magnitude of the system response. (Requires Simulink Control Design.)

Specify Lower Bound on Damping Ratio

To specify a lower bound on the damping ratio of the system:


- 1 In the Response Optimization tool, select **Damping Ratio** in the **New** list. A window opens where you specify a lower bound on the damping ratio of the system.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify the lower bound on the damping ratio in the **Damping ratio** box.

- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

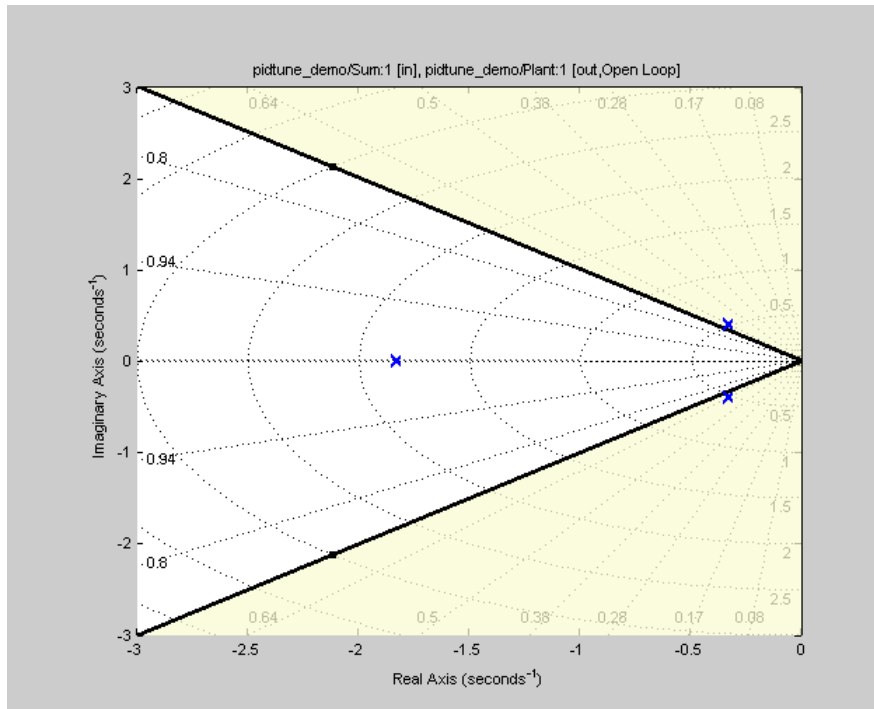
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 5 Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Pole-Zero Characteristics block to specify a bound on the damping ratio. (Requires Simulink Control Design.)

Specify Upper and Lower Bounds on Natural Frequency

To specify a bound on the natural frequency of the system:


- 1 In the Response Optimization tool, select **Natural Frequency** in the **New** list. A window opens where you specify a bound on the natural frequency of the system.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify a lower or upper bound on the natural frequency in the **Natural frequency** box.

- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

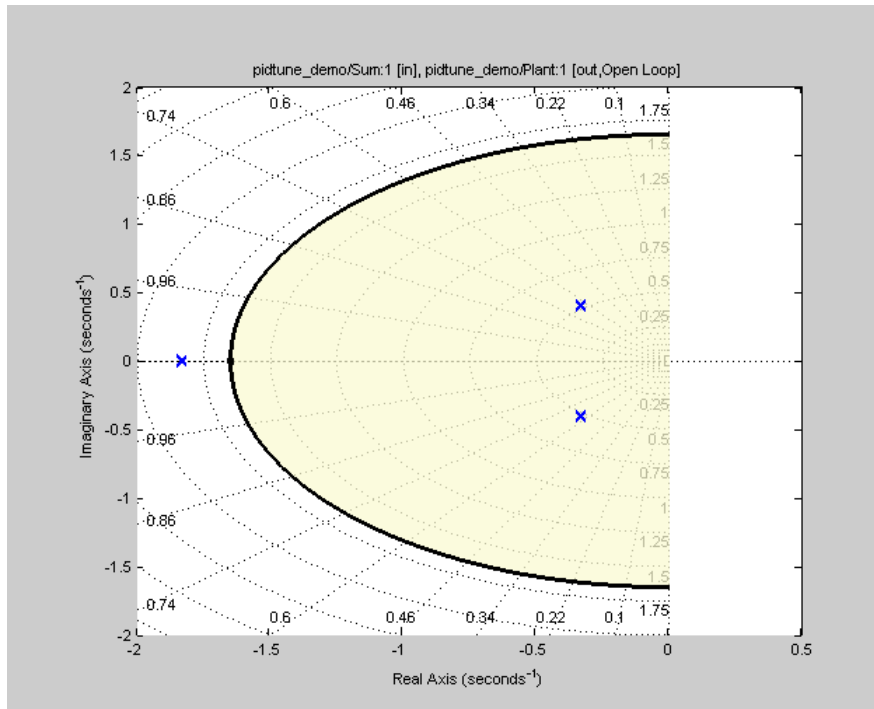
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 5 Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Pole-Zero Characteristics block to specify a bound on the natural frequency. (Requires Simulink Control Design.)

Specify Upper Bound on Approximate Settling Time

To specify an upper bound on the approximate settling time of the system:


- 1 In the Response Optimization tool, select **Settling Time** in the **New** list. A window opens where you specify an upper bound on the approximate settling time of the system.
- 2 Specify a requirement name in the **Name** box.

- 3 Specify the upper bound on the approximate settling time in the **Settling time** box.
- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

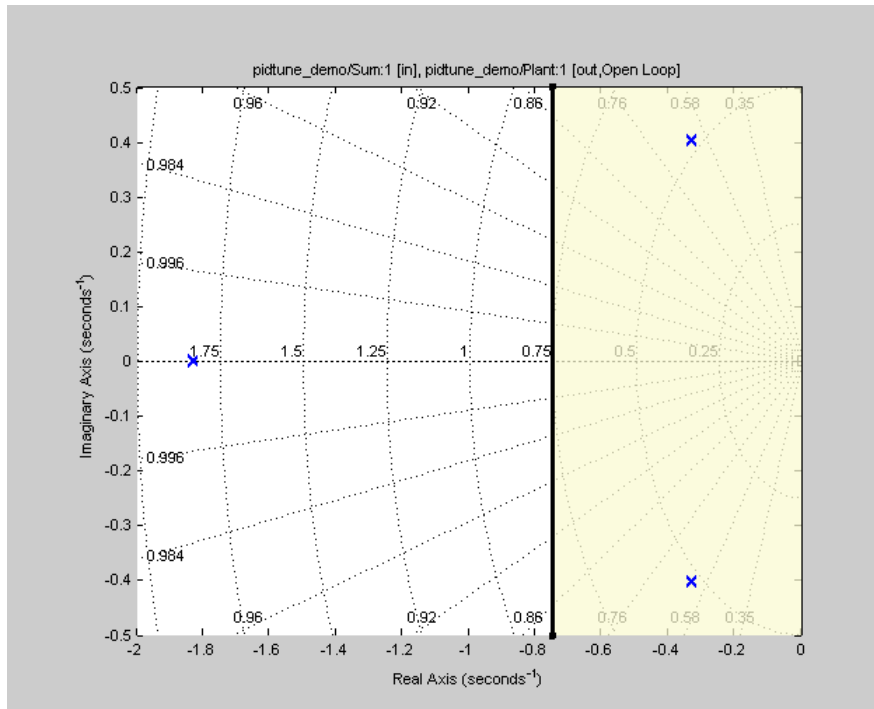
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 5 Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17


Alternatively, you can use the Check Pole-Zero Characteristics block to specify the approximate settling time. (Requires Simulink Control Design.)

Specify Piecewise-Linear Upper and Lower Bounds on Singular Values

To specify piecewise-linear upper and lower bounds on the singular values of a system:

- 1 In the Response Optimization tool, select **Singular Values** in the **New** list. A window opens where you specify the lower or upper bounds on the singular values of the system.
- 2 Specify a requirement name in the **Name** box.

- 3 Specify the requirement type using the **Type** list.
- 4 Specify the edge start and end frequencies and corresponding magnitude in the **Frequency** and **Magnitude** columns, respectively.
- 5 Insert or delete bound edges.

Click  to specify additional bound edges.


Select a row and click  to delete a bound edge.

- 6 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

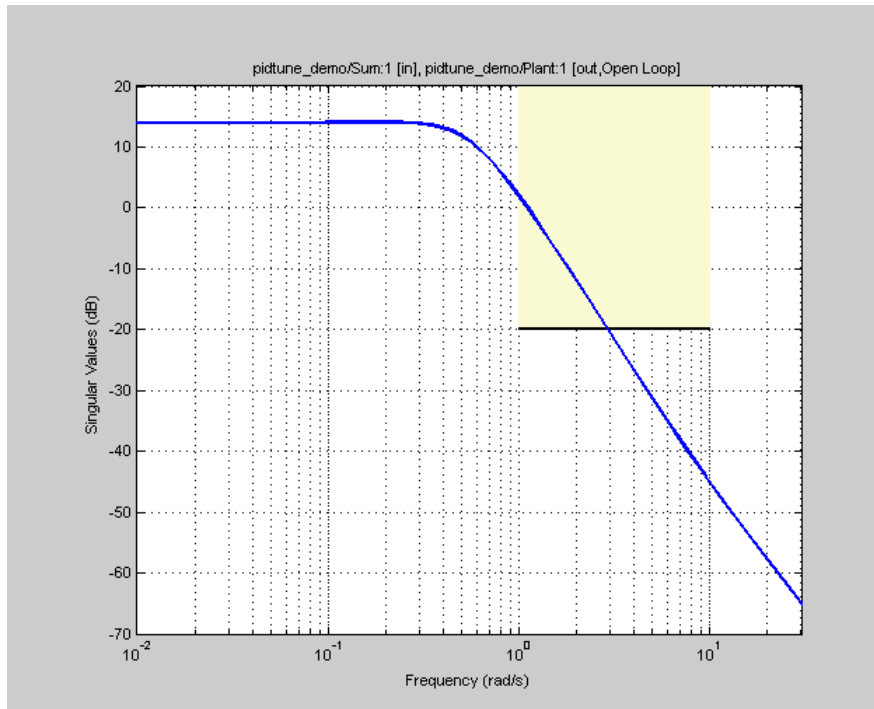
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 7 Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.



8 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Singular Value Characteristics block to specify bounds on the singular value. (Requires Simulink Control Design).

Specify Step Response Characteristics

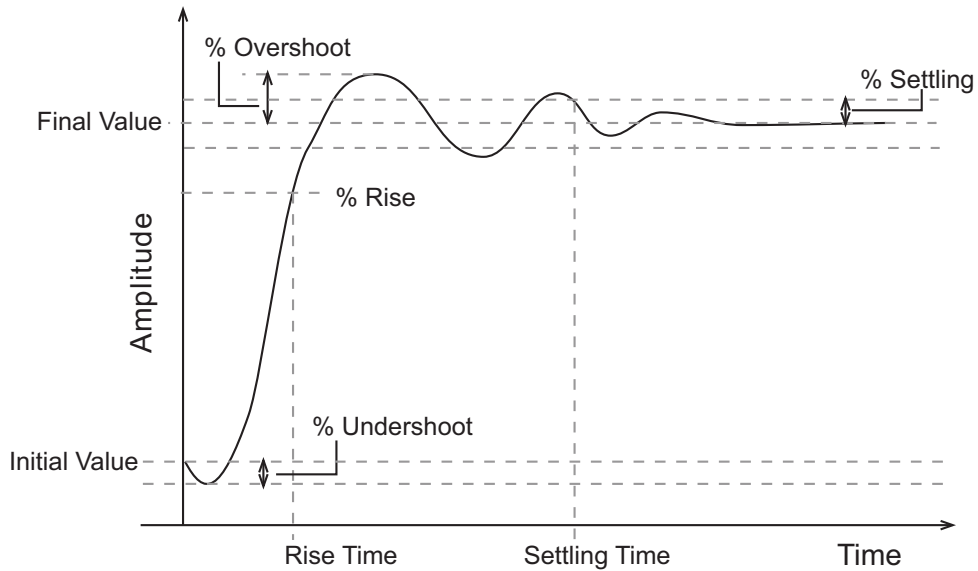
To apply a step response requirement to a linearization of your model (requires Simulink Control Design), specify the step response characteristics as follows:

- 1 Select a step response requirement from the Response Optimization tool.

In the **New** drop-down menu of the tool, in the **New Frequency Domain Requirement** section, select **Step Response Envelope**.

A Create Requirement dialog box opens where you specify the step response requirements.

- 2 Specify a requirement name in the **Name** field of the dialog box.
- 3 Specify the step response characteristics:




- **Initial value** — Input level before the step occurs
- **Step time** — Time at which the step takes place
- **Final value** — Input level after the step occurs
- **Rise time** — The time taken for the response signal to reach a specified percentage of the step range. The step range is the difference between the final and initial values.
- **% Rise** — The percentage of the step range used with **Rise time** to define the overall rise time characteristics.
- **Settling time** — Time taken until the response signal settles within a specified region around the final value. This settling region is defined as the final step value plus or minus the specified percentage of the final value.
- **% Settling** — The percentage of the final value that defines the settling range of settling time characteristic specified in **Settling time**.

- **% Overshoot** — The amount by which the response signal can exceed the final value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.
 - **% Undershoot** — The amount by which the response signal can undershoot the initial value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.
- 4 Specify the systems to be bound.

To apply this requirement to a linearization of your Simulink model:

- a** In the **Select Systems to Bound** area, specify the simulation time at which the model is linearized in **Snapshot Times**. For multiple simulation snapshot times, specify a vector.
- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.


For more information on linearization, see “What Is Linearization?” (Simulink Control Design).


Alternatively, you can use the Check Step Response Characteristics block to specify step response bounds for a signal.

Specify Custom Requirements

You can specify custom requirements, such as minimizing system energy. To specify custom requirements:

- 1 In the Response Optimization tool, in **New** drop-down menu, select **Custom Requirement**. The Create Requirement dialog box opens where you specify the custom requirement.
- 2 Specify a requirement name in **Name**.

- 3 Specify the requirement type in the **Type** drop-down menu.
- 4 Specify the name of the function that contains the custom requirement in **Function**. The field must be specified as a function handle using @. The function must be on the MATLAB path. Click  to review or edit the function.

If the function does not exist, clicking  opens a template MATLAB file. Use this file to implement the custom requirement. The default function name is `myCustomRequirement`.

- 5 (Optional) To prevent the solver from considering specific parameter combinations, select **Error if constraint is violated**. Use this option for parameter-only constraints.

During an optimization iteration, the solver first evaluates requirements with this option selected.

- If the constraint is violated, the solver skips evaluating any remaining requirements and proceeds to the next iteration.
- If the constraint is *not* violated, the solver evaluates the remaining requirements for the current iteration. If any of the remaining requirements bound signals or systems, then the solver simulates the model.

For more information, see “Skip Model Simulation Based on Parameter Constraint Violation (GUI)” on page 3-190.

Note If you select this check box, then do not specify signals or systems to bound. If you *do* specify signals or systems, then this check box is ignored.

- 6 (Optional) Specify the signal or system, or both, to be bound.

You can apply this requirement to model signals, or a linearization of your Simulink model (requires Simulink Control Design), or both.


Click **Select Signals and Systems to Bound (Optional)** to view the signal and linearization I/O selection area.

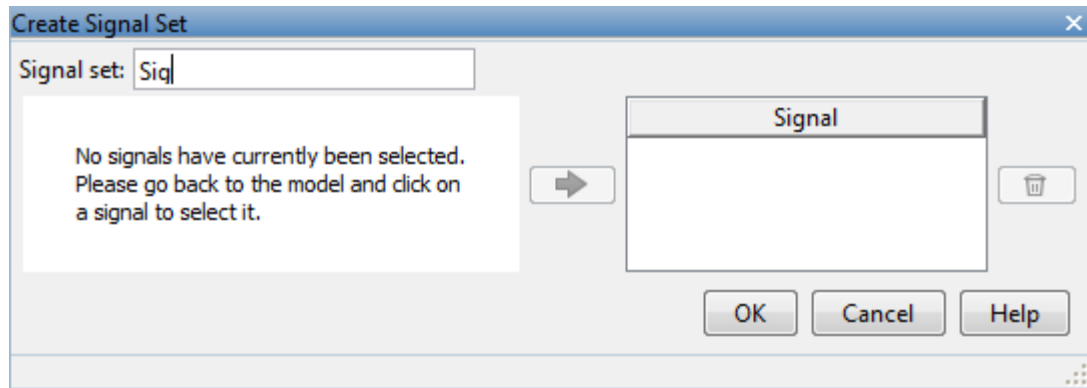
- To apply this requirement to a model signal:

In the **Signal** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, it appears in the list. Select the corresponding check box.

If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.
- b In the Simulink model window, click the signal to which you want to add a requirement.




The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

- To apply this requirement to a linear system:
 - a Specify the simulation time at which the model is linearized in **Snapshot Times**. For multiple simulation snapshot times, specify a vector.
 - b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box. For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

7 Click **OK**.

A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.

See Also

- “Design Optimization to Meet a Custom Objective (GUI)” on page 3-115
- “Design Optimization to Meet a Custom Objective (Code)” on page 3-133

See Also

Related Examples


- “Specify Design Variables” on page 3-76
- “Specify Time-Domain Design Requirements in the Tool” on page 3-19
- “Specify Variable Requirements in the Tool” on page 3-39
- “Specify Optimization Options” on page 3-84

Specify Design Variables

This topic shows how to specify design variables for optimization.

Before running the optimization, you must specify the model parameters to optimize. These parameters form the design variables set for optimization. By tuning these parameters, Simulink Design Optimization software attempts to make the signals meet the requirements. Simulink Design Optimization software optimizes the response signals of the model by varying the tuned parameters so that the response signals lie within the constraint bound segments or closely match a specified reference signal. The design variables can be scalar, vector, matrix, or an expression that evaluates to one of these values.

You can also use sensitivity analysis for finding the parameters that most influence the optimization problem and use these as design variables. To open the **Sensitivity**

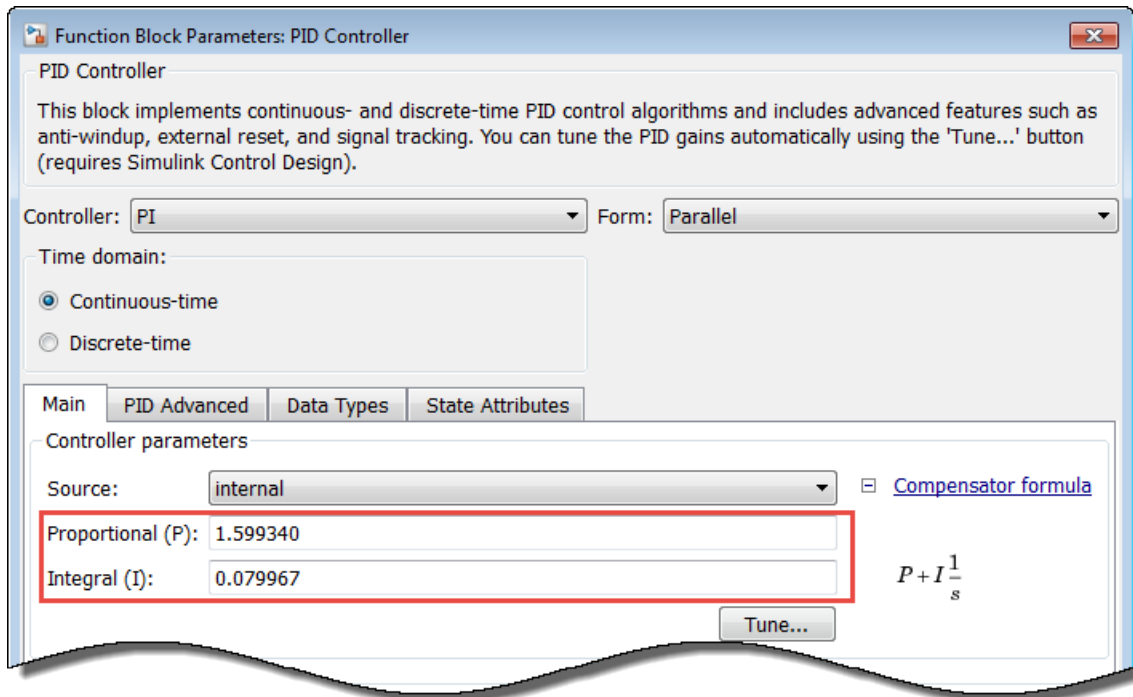
Analysis tool, in the **Response Optimization** tab, click  **Sensitivity Analysis**. In the Sensitivity Analysis tool, you can explore the response optimization design space by altering the design variables, identify the parameters that most influence the optimization problem, and compute initial values.

In this section...
“Add Model Parameters as Variables for Optimization” on page 3-76
“Specify Design Variables” on page 3-79

Add Model Parameters as Variables for Optimization

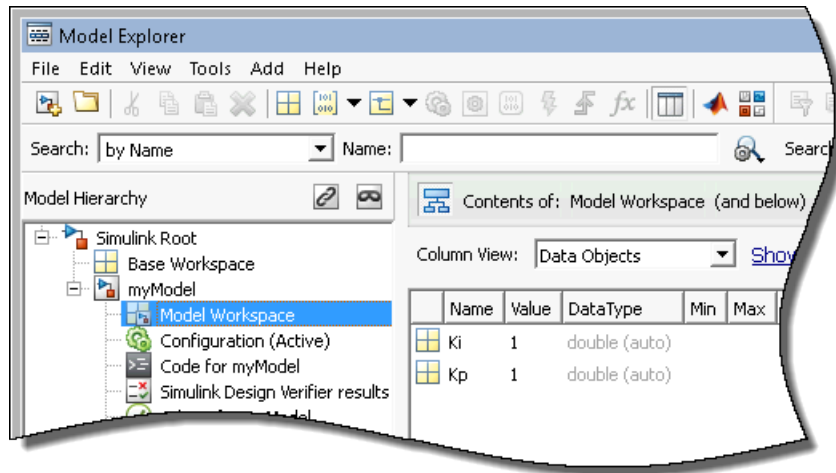
The software can only optimize variables that are in use by the Simulink model. Create variables for optimization in the MATLAB or model workspace, and specify your model or block parameters using these variables.

In this figure, the **Proportional (P)** and **Integral (I)** gain parameters of a PID Controller block are specified as numerical values.



To optimize the gain parameters, specify them as variables K_p and K_i :

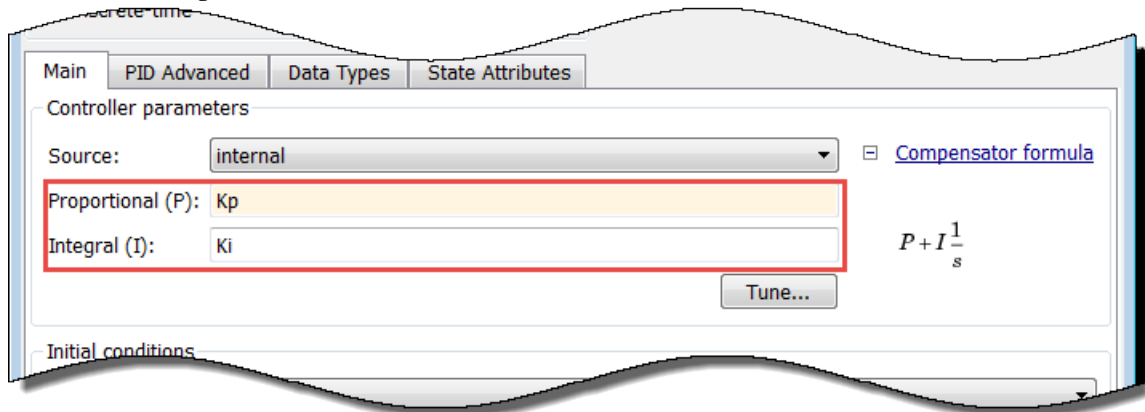
- 1 Create the variables K_p and K_i in one of the following ways:
 - Add the variables to the model workspace (Simulink), and specify initial values.



- Write initialization code in the **PreloadFcn** callback of the model. For more information, see “Model Callbacks” (Simulink).

```
Kp = 1;
Ki = 1;
```

- 2 Specify the gain parameters as the variables K_p and K_i in the PID Controller block dialog box.



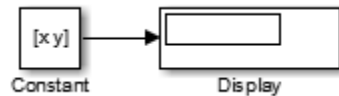
You can now select K_p and K_i for optimization. See, “Specify Design Variables” on page 3-79.

Specify Independent Parameters for Optimization

You can also specify independent parameters that do not appear explicitly in the model as variables for optimization. However, you cannot use this workflow with Simulink fast restart.

Suppose that a model parameter K_{int} is related to independent parameters x and y such that $K_{int} = x+y$. To optimize x and y instead of K_{int} :

- Create the independent variables x and y by adding them to the model workspace and specifying initial values.
- The software only allows tuning of variables that are used by model blocks. To ensure that the software detects x and y for tuning, add a Constant block to your model, and specify the **Constant value** of the block as $[x \ y]$. Connect the block to a Display block.



- Write code in the **InitFcn** callback of the model that defines the relationship between K_{int} , x , and y . You must first use the `get_param` function to get the variables x and y from the model workspace before you can use them to define K_{int} .

```
wks = get_param(gcs, 'ModelWorkspace')
x = evalin(wks, 'x')
y = evalin(wks, 'y')
Kint = x+y;
```

You can now select x and y for optimization. Do not optimize the independent and dependent parameters simultaneously. Doing so can lead to incorrect results. For example, do not optimize K_{int} , x and y together.

Specify Design Variables

To specify the parameters to be tuned using the Response Optimization tool:

- 1 In the **Design Variables Set** list, select **New**.

A window opens where you specify design variables. All parameters in use by the model are displayed in this window.

- 2 Select one or more parameter names and click



to add the selected parameters to a design variables set.

Note You can add the same parameter to multiple design variable sets.

- 3 (Optional) Specify design variable settings.

Setting	Description	Default
Variable	The name of the parameter.	Not an editable field
Value	Value of the model parameter. This value is used by the optimization method as the initial value and is modified during optimization.	Current value of the parameter in the model. If you edit this column, click Update model variable values to update the values in the model.
Minimum	The minimum value or lower bound for the parameter. You can edit this field to provide an alternate minimum value.	-Inf
Maximum	The maximum value or upper bound for the parameter. You can edit this field to provide an alternate maximum value.	Inf
Scale	During optimization, the design variables are scaled, or normalized, by dividing their current value by a scale value. You can edit this field to provide an alternate scaling factor.	Next power of 2 greater than the current value of the parameter

The check-box indicates whether the parameter is selected as an design variable in the set. Select it if you want this parameter to be tuned during the optimization. Deselect if you do not want this parameter to be tuned during the optimization but you would like to keep it on the list of tuned parameters (for a subsequent optimization).

Expand **Variable Detail** to see the block in the model that contains this parameter.

- 4 Click **OK** to create a design variable set.

See Also

Related Examples

- “Optimize Parameters for Robustness (GUI)” on page 3-204
- “Update Model with Design Variables Set” on page 3-82
- “Save Design Variable Values for Specific Iteration” on page 3-97

Update Model with Design Variables Set

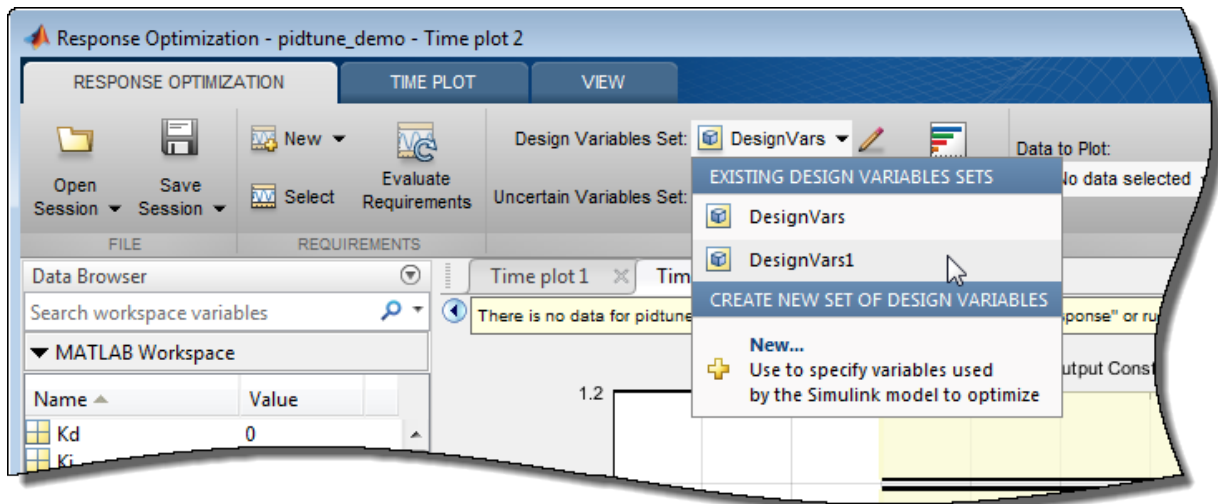
This example shows how to update a model with a set of design variables.

Open the Simulink model and load the pre-configured Response Optimization tool session.

```
load('pidtune_demo_sdoession_update_dv.mat')
sdotool(SDOSessionData)
```

The Response Optimization tool opens and loads the preconfigured session. In the **Data** area, DesignVars1 is a set of tuned design variables.

In the **Design Variables Set** list, select the design variable set, DesignVars1.

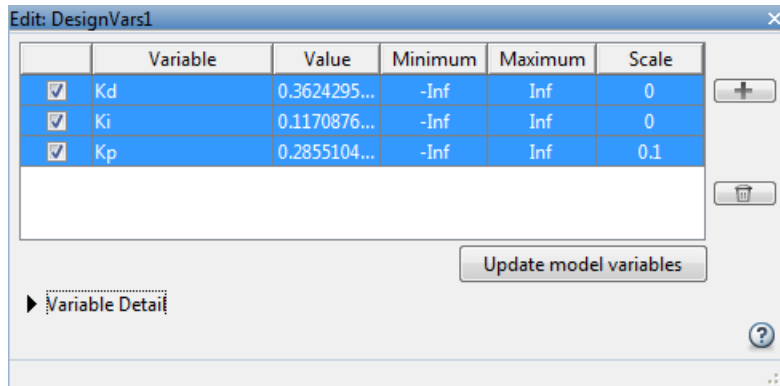


Open the Edit dialog box.

Click  for the **Design Variable Set** list.

Select the variables you want to update in the model.

For this example, select K_d , K_i , and K_p .



Click **Update model variables**.

Plot the model response.

In the **Response Optimization** tab, click **Plot Current Response**.

See Also

Related Examples


- “Save Design Variable Values for Specific Iteration” on page 3-97

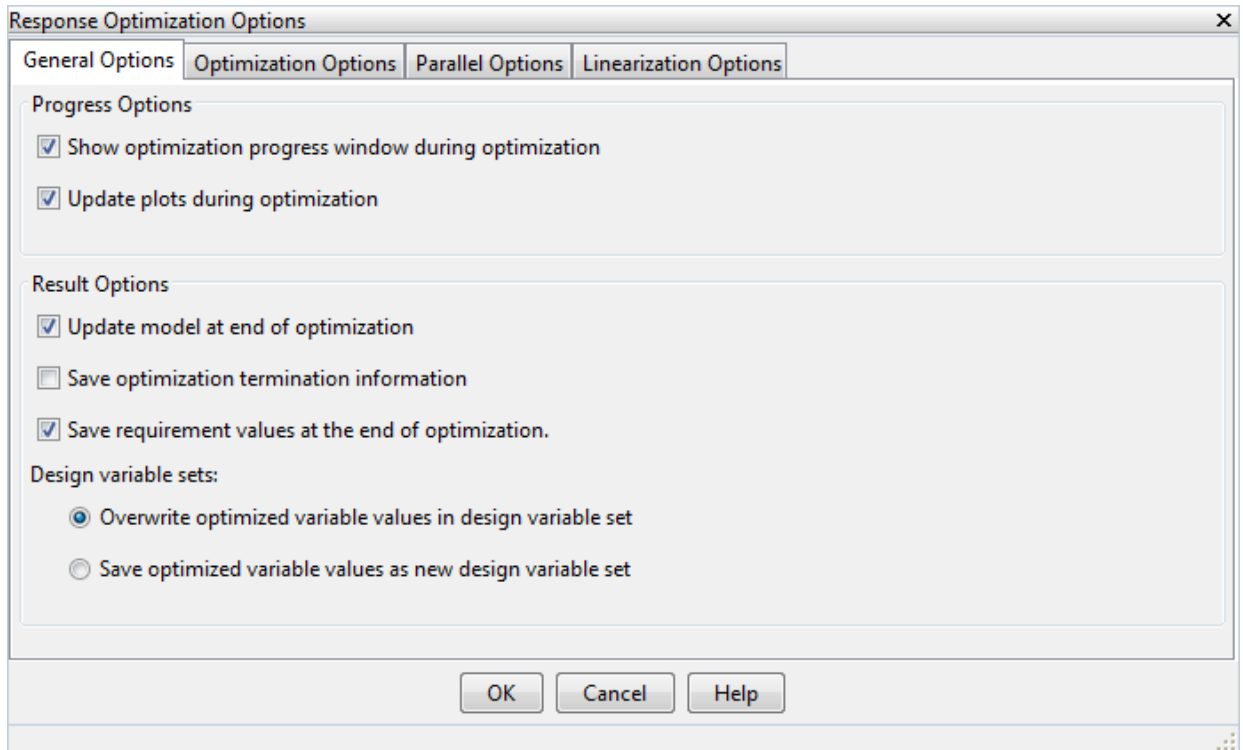
Specify Optimization Options

This topic shows how to specify optimization options in the Response Optimization tool, after you have configured the design variables and design requirements.

After you have configured the design variables and design requirements, specify the following optimization options:

1 Optimization progress and result options for optimization task

To specify these options, in the **Response Optimization** tab, click  **Options** to open the Response Optimization Options dialog box. In the **General Options** tab, specify the optimization progress and result options. For details about the options, click the **Help** button.



2 Optimization method and termination options

Specify these options in the **Optimization Options** tab of the Response Optimization Options dialog box.

3 Parallel computing options

Specify these options in the **Parallel Options** tab of the Response Optimization Options dialog box. For details about the options, see “Optimize Design Using Parallel Computing (GUI)” on page 3-219.

4 Linearization options

Specify these options in the **Linearization Options** tab of the Response Optimization Options dialog box.

See Also

Related Examples

- “Specify Design Variables” on page 3-76
- “Specify Time-Domain Design Requirements in the Tool” on page 3-19
- “Specify Frequency-Domain Design Requirements in the Tool” on page 3-56
- “Specify Custom Requirements in the Tool” on page 3-13


More About

- “How the Optimization Algorithm Formulates Minimization Problems” on page 3-3

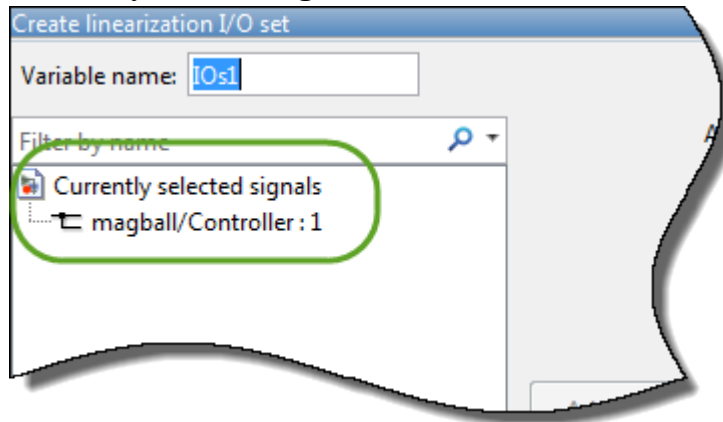
Create Linearization I/O Sets

This example shows how to create a linearization input/output set in the Response Optimization tool or Sensitivity Analysis tool.

To create a linearization I/O set:

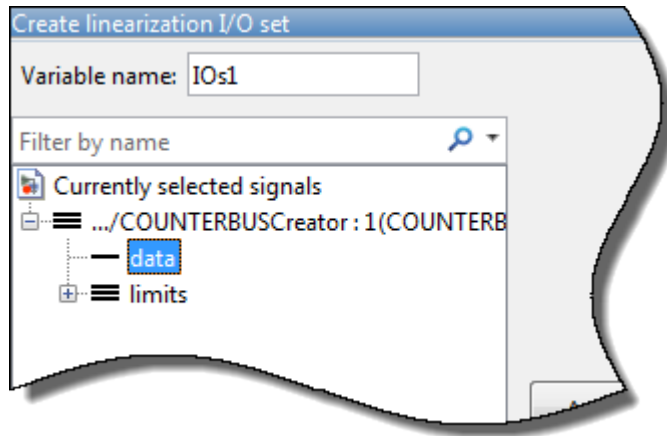
- 1 Open the Create Linearization I/O Set dialog box using one of the following methods:
 - In a requirement dialog box, in the **Select Systems to Bound** section, click .
 - In the Response Optimization tool, in the **New** drop-down list, select **Linearization I/Os**.
- 1 In your Simulink model, select one or more signals that you want to define as analysis points.

The selected signals appear in the Create linearization I/O set dialog box under **Currently selected signals**.

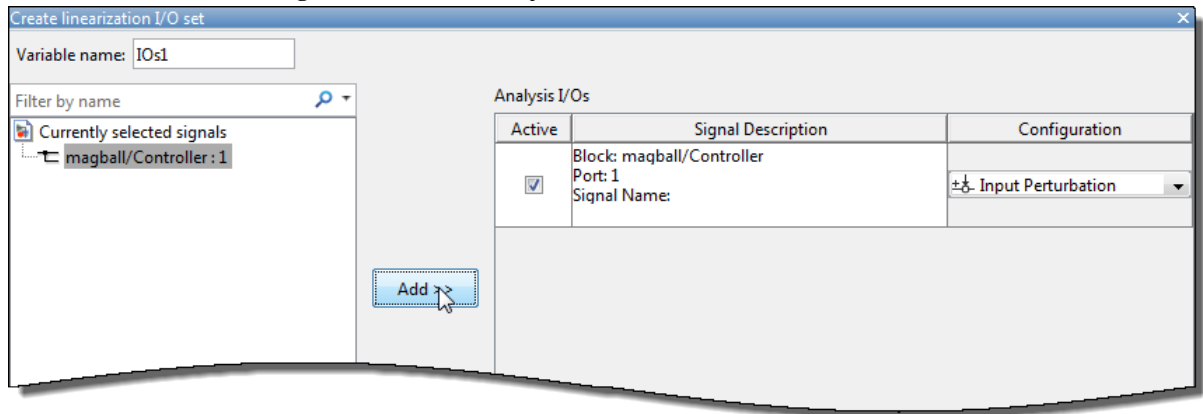


- 2 Under **Currently selected signals**, click the signal you want to add. To select multiple signals, hold **Ctrl** and click each signal you want to add.




To add a signal from within a bus signal, expand the bus and select the signal. For example, select the data signal within the COUNTERBUS signal.








3 To add the signal to list of **Analysis I/Os**, click **Add**.



4 In the **Configuration** drop-down list for the signal, select the type of analysis point you want to define:

-  **Input Perturbation** — Specifies an additive input to a signal.
-  **Output Measurement** — Takes a measurement at a signal.
-  **Loop Break** — Specifies a loop opening.

-  **Open-Loop Input** — Specifies a loop break followed by an input perturbation.
-  **Open-Loop Output** — Specifies an output measurement followed by a loop break.
-  **Loop Transfer** — Specifies an output measurement before a loop break followed by an input perturbation.
-  **Sensitivity** — Specifies an input perturbation followed by an output measurement.
-  **Complementary Sensitivity** — Specifies an output measurement followed by an input perturbation.

For more information on the different types of analysis points, see “Specify Portion of Model to Linearize” (Simulink Control Design).

- 5 Repeat steps 1–4 for any other signals you want to define as analysis points.

Tip To highlight the source block of an analysis point in the Simulink model, in the **Analysis I/Os** list, select the analysis point, and click **Highlight**.

- 6 In the **Variable name** box, enter a name for the I/O set.
- 7 Click **OK**.

See Also

Related Examples

- “Design Optimization to Meet Frequency-Domain Requirements (GUI)” on page 3-147

More About

- “What Is Linearization?” (Simulink Control Design)

Interact with Plots

In this section...

“Response Plots” on page 3-89

“Spider Plots” on page 3-90

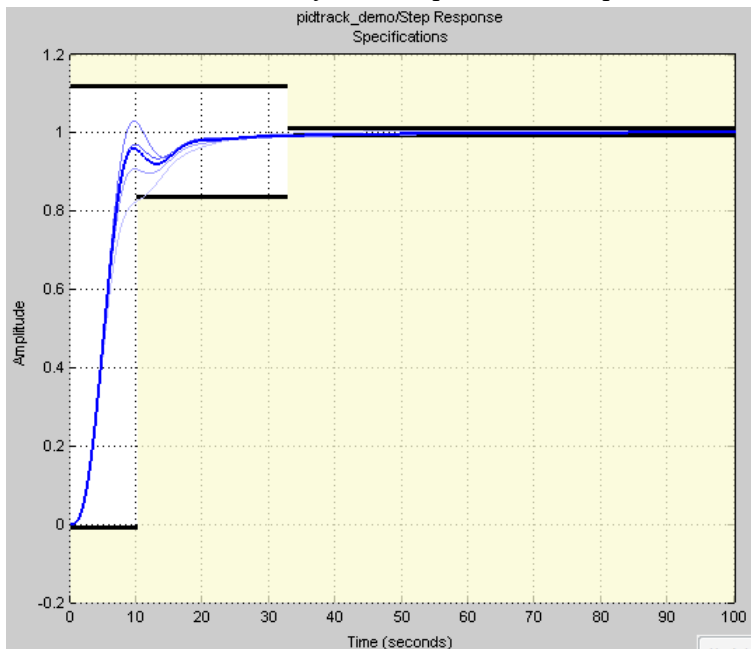
“Iteration Plots” on page 3-92

This topic shows how to interact with plots in the Response Optimization tool.

Response Plots

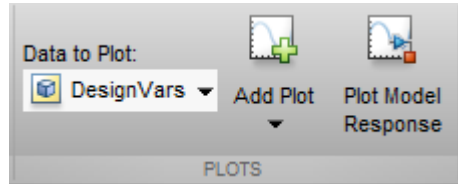
View model signals and the requirements applied to the signal using a response plot. You can also plot the frequency response of a system (requires Simulink Control Design software).

The response plot shows the system response as it varies during optimization. You can also view the uncertain system responses in the plot.



To plot a response plot:

- 1 In the **Response Optimization** tab of the tool, in the **Data to Plot** drop-down list, choose a signal, linear system, or requirement.



- 2 Select a response plot in the **Add Plot** drop-down list.

You can add to an existing plot, or create a plot. The drop-down list has entries for the supported plot types for the given plot variable.

- 3 Display the current model response.

Click **Plot Model Response**. The current response appears as a thick line on the response plot.

- 4 Configure the plot display.

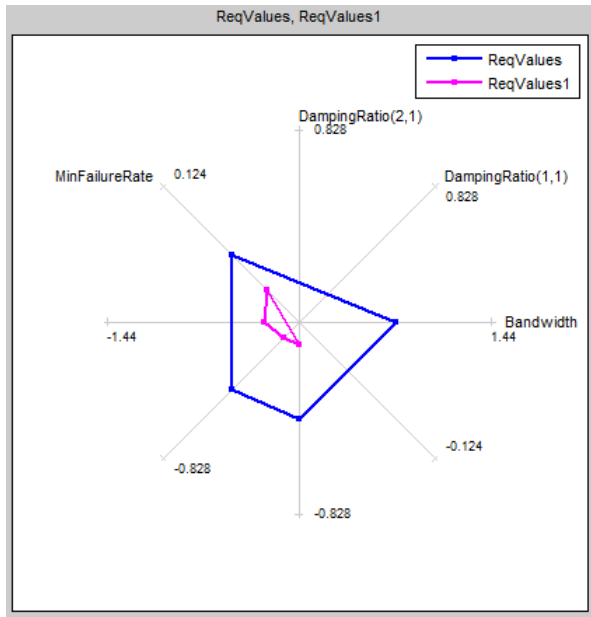
- To select the signal or systems to display, right-click within the white space in the plot, and choose from **Signals** and **Systems** in the menu.
- To control whether the response signal displays at intermediate steps during optimization, right-click within the white space in the plot and select **Responses** > **Show Iteration Responses**. The response at an intermediate step is based on parameter values at that intermediate step in the optimization.
- Modify plot properties.

You can change plot properties such as plot title, axis labels, axes limits, and units. Right-click the white space in a plot and select **Axes Properties** to open the Property Editor dialog box.

Spider Plots

Compare the values of design variable sets or evaluated requirements using a spider plot.

Spider plots depict multivariate data using a separate axis for each variable. The various axes are arranged clockwise and have a common intersecting point.



To plot a spider plot for a design variable set or evaluated requirements:

- 1 In the **Response Optimization** tab of the tool, in **Data to Plot**, choose design variables, or evaluated requirements.
- 2 Select a Spider plot in the **Add Plot** drop-down list.

You can add to an existing plot, or create a new one. The drop-down list has entries for the supported plot types for the given plot variable.

- 3 Configure the plot display.

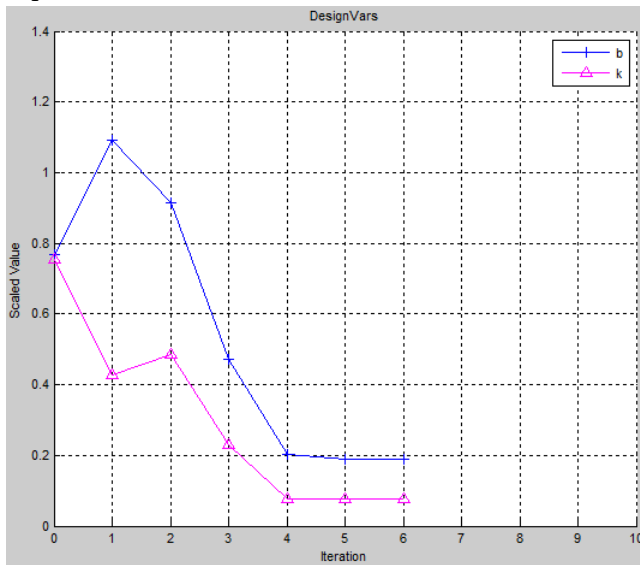
To view only some of the variables or requirement values in a given plot, right-click the plot, and select the variables or requirements in the **Show** list.

For information on using a spider plot to compare design variables sets or evaluated requirements, see “Compare Requirements and Design Variables Using Spider Plot” on page 3-94

Iteration Plots

Plot the values of design variables and requirements as they vary during optimization using an iteration plot.

Iteration plots depict the value of the plot variables for each iteration. The x-axis represents the iteration number.



To plot an iteration plot for a design variable set or requirements:

- 1 In the **Response Optimization** tab of the tool, in **Data to Plot**, choose design variables or requirements.
- 2 Select an iteration plot in the **Add Plot** drop-down list.

You can add to an existing plot, or create a plot. The drop-down list has entries for the supported plot types for the given plot variable.

- 3 Configure the plot display.

To view scaled values of the plotted variables, right-click the plot, and select **Show scaled values**.

- 4 Save design variables and requirements values for a given iteration using the iteration plots.

For more information, see “Save Design Variable Values for Specific Iteration” on page 3-97.

See Also

Related Examples

- “Save Design Variable Values for Specific Iteration” on page 3-97
- “Compare Requirements and Design Variables Using Spider Plot” on page 3-94

Compare Requirements and Design Variables Using Spider Plot

This example shows how to use a spider plot to compare requirement evaluations before and after optimizing the response. You can use a similar procedure to compare the values of sets of design variables.

Open the Simulink model and load the pre-configured Response Optimization Tool session.

For this example, which uses a distillation column model, the step response requirements are preconfigured and loaded in the model workspace.

- 1 Open the distillation model.

```
sys = 'distillation_demo';  
open_system(sys)
```

- 2 Open the Response Optimization Tool.

In the Simulink model window, select **Analysis > Response Optimization**.

Alternatively, click the Response Optimization GUI with preloaded data block in the model and skip the next step.

- 3 Load the preconfigured Response Optimization Tool session.

Click the **Response Optimization** tab. In the **Open Session** drop-down list, select **Open from model workspace**. A window opens where you select the Response Optimization Tool session to load. Select `distillation_optim` and click **OK**.

The preconfigured step response requirements are loaded in the Response Optimization Tool.

Evaluate the requirement before optimization.

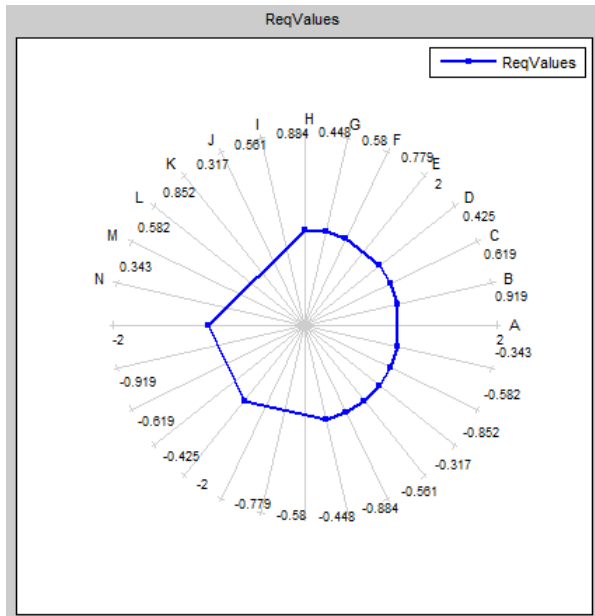
In the **Response Optimization** tab, click **Evaluate Requirements**.

A new variable, `ReqValues`, containing the evaluation of the requirements appears in the **Data** area.

When optimizing the model response, you create a set of requirements that it must satisfy. If the requirements are violated, meaning that they evaluate to non-negative values, the design variables must be optimized. After the optimization, you can compare the original requirement value with the requirement evaluated using the optimized design variable values.

Plot the requirement value before optimization.

- 1 In the **Data to Plot** list, select ReqValues.
- 2 In the **Add Plot** list, select Spider plot.



The plot has an axis for each edge-and-signal combination defined in the `distillation_demo/Desired Step Response` check block. Points on each axis represent the violation for that signal-edge combination and the plot connects these points to form a closed polygon representing the initial design. Note that some points are negative, representing satisfied constraints, and some positive, representing violated constraints.

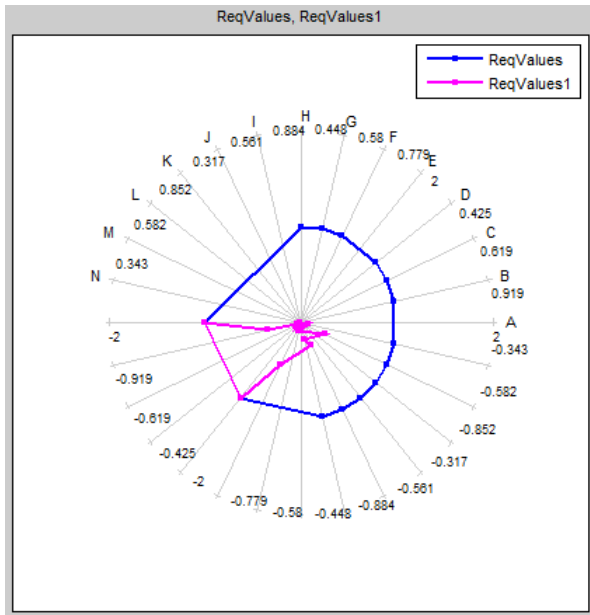
Optimize the model.

Click **Optimize**.

A new variable, `ReqValues1`, containing the evaluation of the requirements using the optimized design variables appears in the **Data** area.

Compare the requirement values before and after optimization.

- 1 In the **Data to Plot** list, select ReqValues1.
- 2 In the **Add Plot** list, select Spider plot 1.



The optimized requirement values, ReqValues1, are all negative or zero, indicating that all the constraints are satisfied.

See Also

More About

- “Save Design Variable Values for Specific Iteration” on page 3-97

Save Design Variable Values for Specific Iteration

This example shows how to save the design variable values for specific optimization iterations.

During optimization, the optimization solver simulates the model using a different set of design variables at each iteration. After the optimization completes, you can export the values for an iteration from the iteration plot of the design variable set.

For this example, load a preconfigured Response Optimization tool session. Optimize the model, and export the design variable set values for the third iteration.

Open the Simulink model and load the preconfigured Response Optimization tool session.

```
load('distillation_demo_sdosession_export_iter_dv.mat')  
sdotool(SDOSessionData)
```

The Response Optimization tool opens and loads the preconfigured session. **Iteration Plot 1** is configured to plot the values of `DesignVars` for each optimization iteration.

Click **Optimize**.

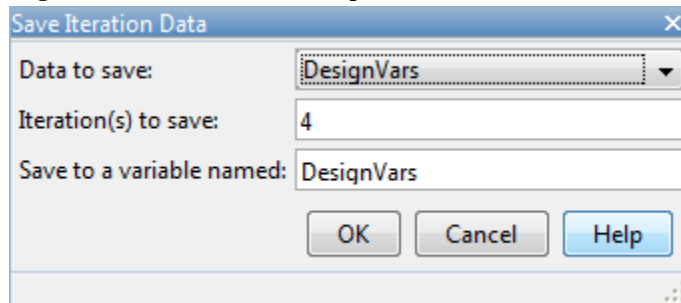
The optimization completes after four iterations.

Select the iteration plot of the design variable set.

Click **Iteration plot 1**.

Open the Save Iteration Data dialog box.

Right-click on the iteration plot, and select **Save Iteration Data**.



You can also access the Save Iteration Data dialog box from the Optimization Progress Report. To do so, in the progress report, click **Save Iteration**.

Specify details about the design variable data to save:

- In the **Data to save** list, select `DesignVars`.
- In **Iteration(s) to save**, enter 3.

To specify multiple iterations, use a vector of integers. For example, `[0 2 5]`.

- In **Save to a variable named**, enter `DesignVars_iter`.

Export the design variable values set.

Click **OK**. The exported data variable, `DesignVars_iter_3`, appears in the **Data** area of the tool.

Note The iteration number is added as a suffix to the saved data variable name.

See Also

Related Examples

- “Update Model with Design Variables Set” on page 3-82
- “Compare Requirements and Design Variables Using Spider Plot” on page 3-94

More About

- “Interact with Plots” on page 3-89

Design Optimization to Meet Time- and Frequency-Domain Requirements (GUI)

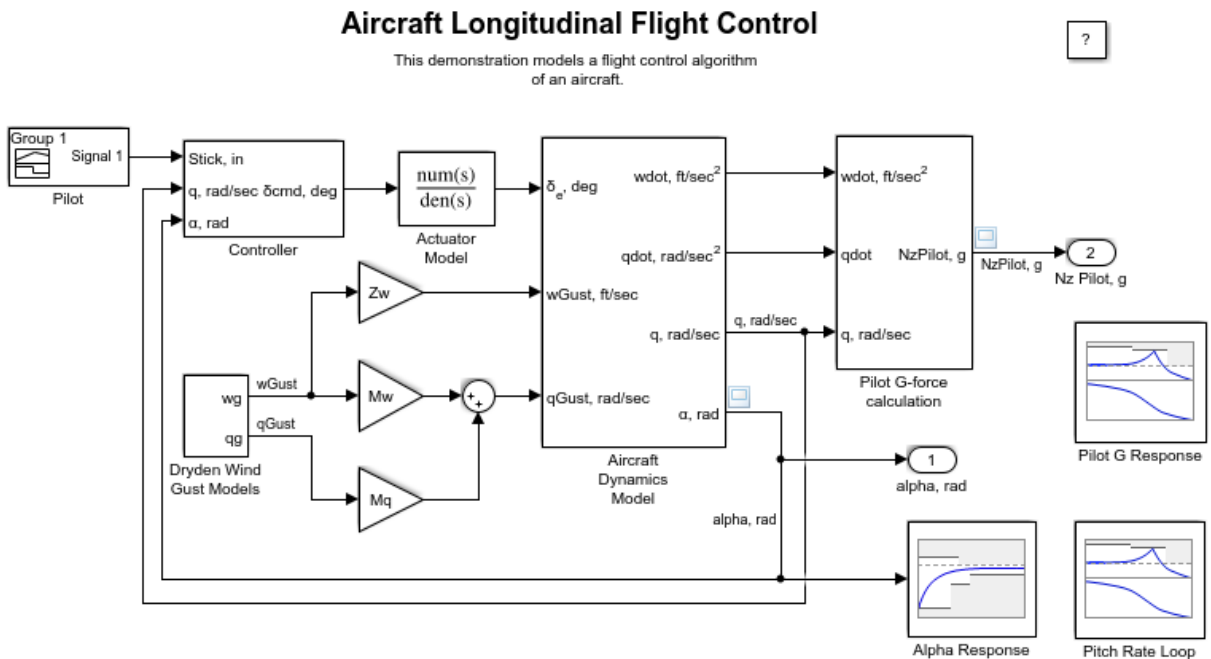
This example shows how to tune a controller to satisfy time- and frequency-domain design requirements using the Response Optimization tool.

The example requires Simulink® Control Design™.

Aircraft Longitudinal Flight Control Model

Open the Simulink model.

```
sys = 'sdoAircraft';
open_system(sys);
```



The aircraft model is based on the Simulink `slexAircraftExample` model. The model includes:

- Subsystems to model aircraft dynamics (`Aircraft Dynamics Model`), wind gusts (`Dryden Wind Gust Models`), and pilot G-forces (`Pilot G-force calculation`).
- A step change applied to the aircraft joystick at 1 second into the simulation that causes the aircraft to pitch upward.

Controller Design Problem

You tune the controller gains to meet the following time- and frequency-domain design requirements:

- Angle-of-attack `alpha` response to a step change in the joystick has a rise time of less than 1 second, less than 1% overshoot, and settles to within 1% of steady state within less than 5 seconds
- Pitch-rate control loop has good tracking below 1 rad/s and 20 dB noise rejection above 100 rad/s
- Closed loop response from joystick to pilot G-Force is below 0 dB above 5 rad/s.

These requirements reduce the high frequency G-forces experienced by the pilot in response to joystick changes while still maintaining flight performance.

The model includes the following blocks (from Simulink® Design Optimization™ and Simulink Control Design Model Verification libraries):

- `Alpha Response` specifies the alpha step response requirement.

Check Step Response Characteristics

Assert that the input signal satisfies bounds specified by step response characteristics.

Bounds **Assertion**

Include step response bound in assertion

Step time (seconds):

Initial value: Final value:

Rise time (seconds): % Rise:

Settling time (seconds): % Settling:

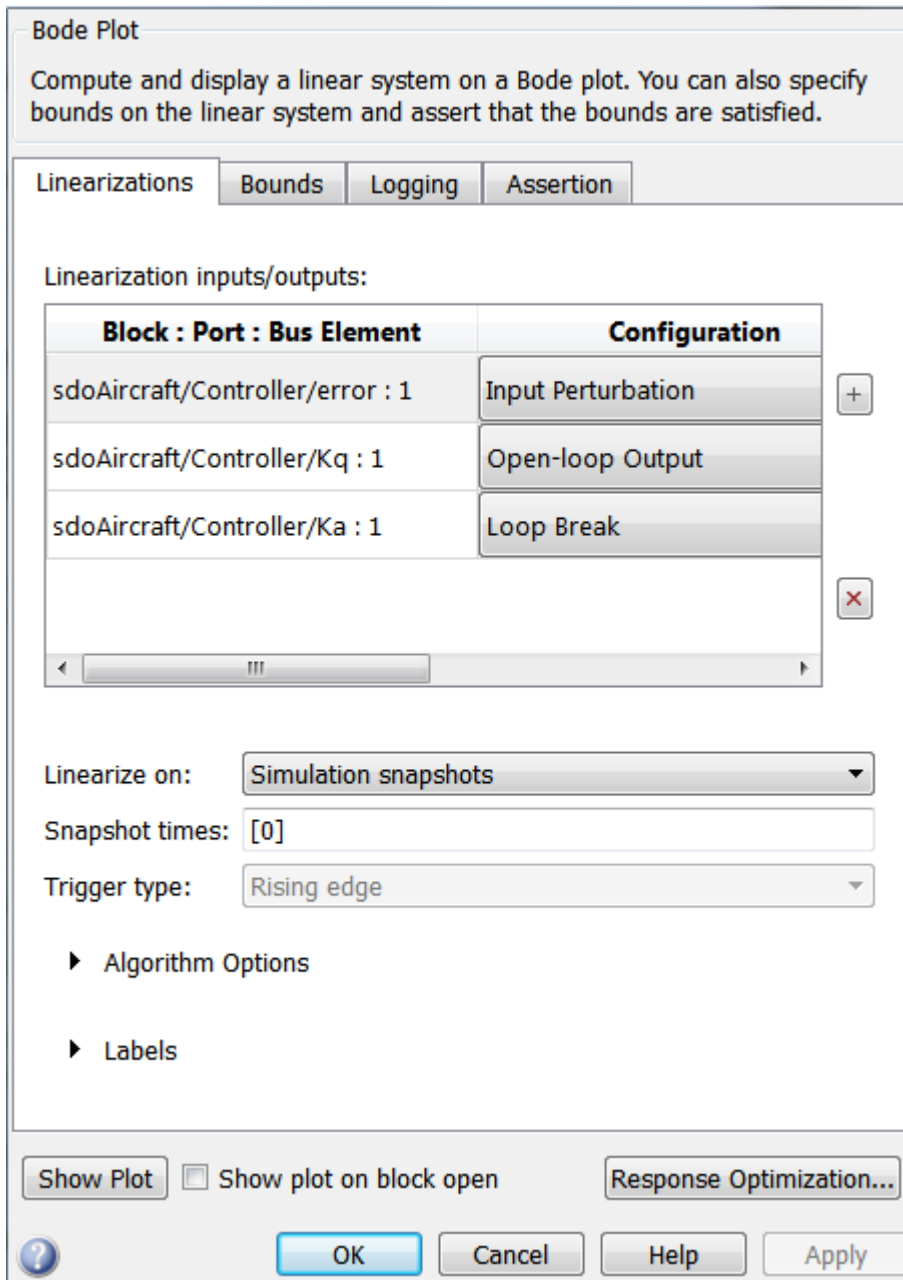
% Overshoot: % Undershoot:

Enable zero-crossing detection

Show Plot Show plot on block open Response Optimization...

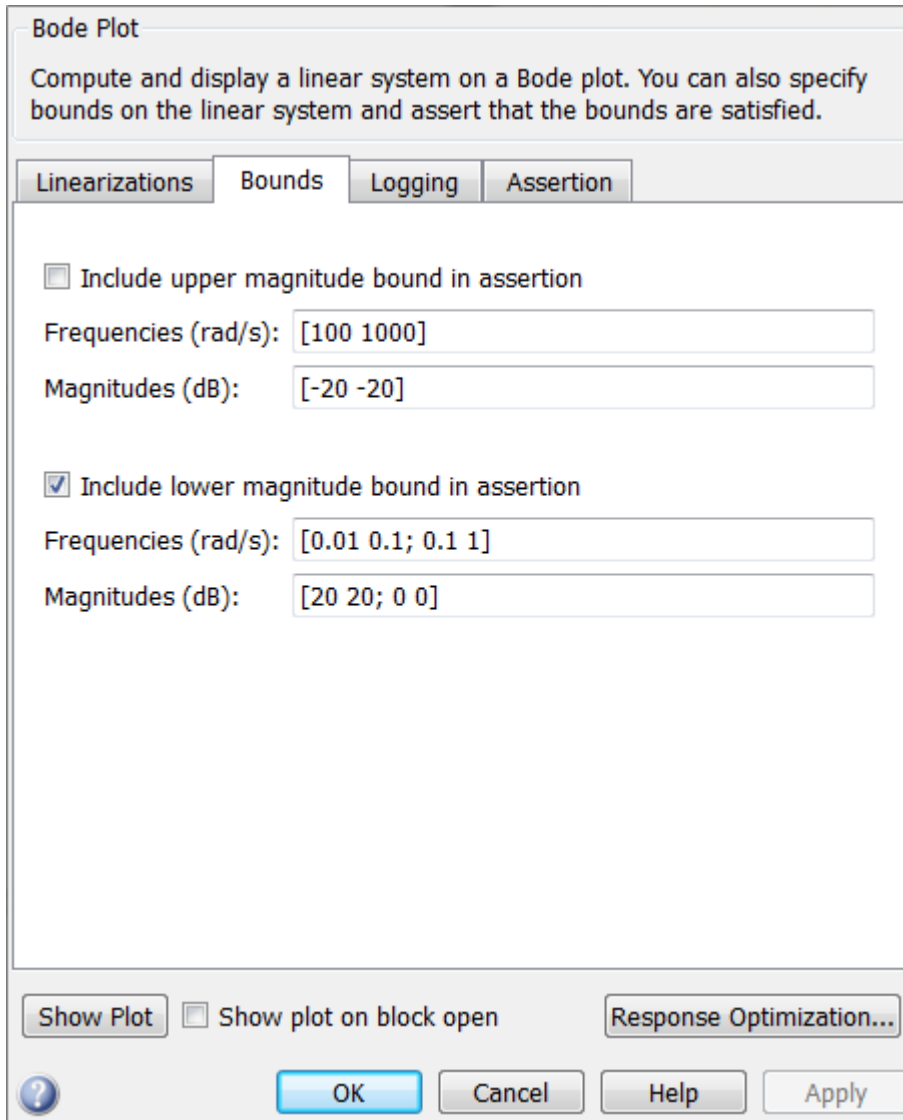
- Pitch Rate Loop specifies the pitch-rate performance requirement.

The linearization inputs/outputs are already selected in the **Linearizations** tab. The pitch-rate loop starts from the input of the controller (the controller error signal) and ends at the output of the pitch-rate sensor. The angle-of-attack loop is opened signal so that the block only computes the pitch-rate loop response. The linear system is computed at a simulation time of 0.



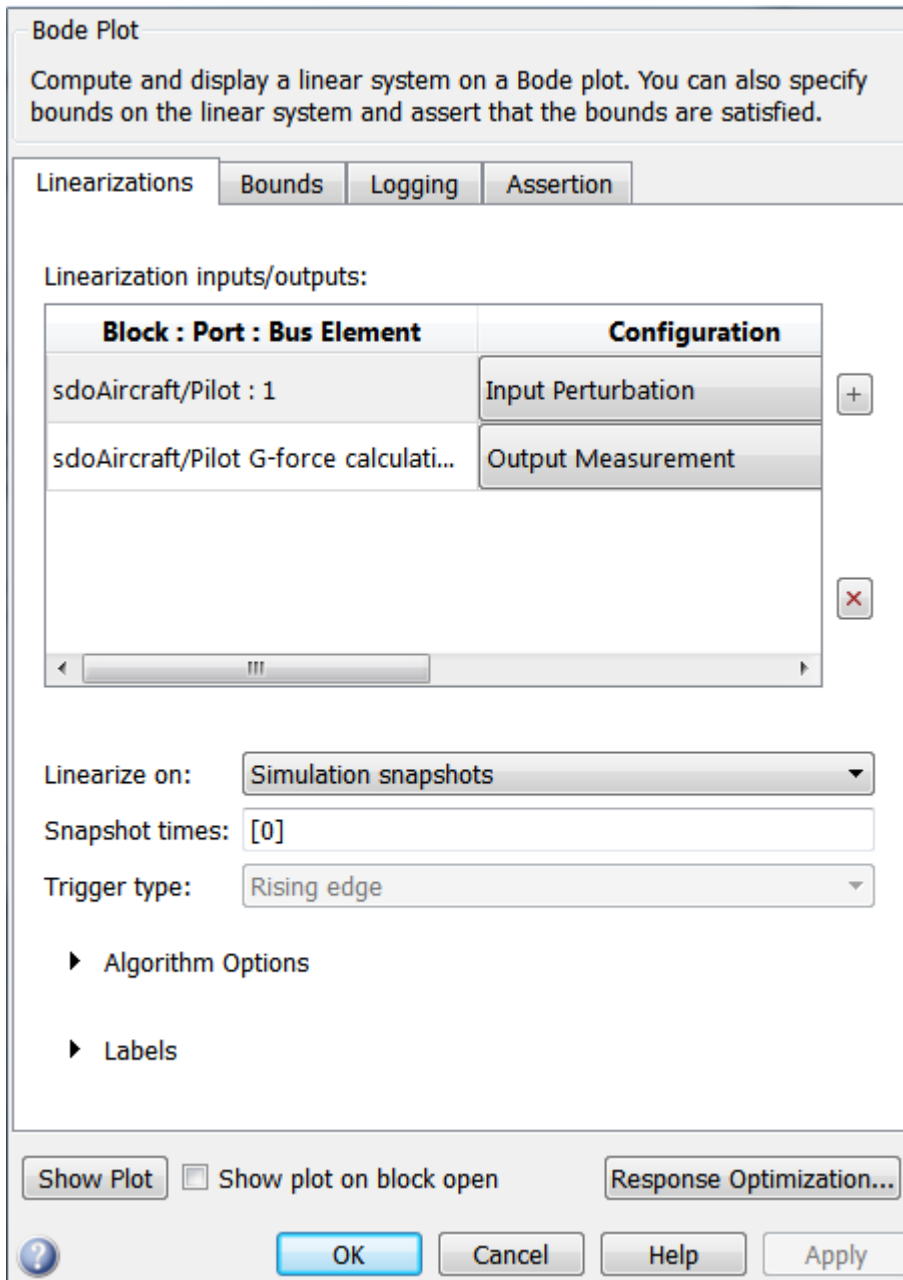
The **Bounds** tab specifies the following pitch-rate loop shape requirements:

- Greater than 20 dB over the range 0.01 rad/s to 0.1 rad/s
- Greater than 0 dB over the range 0.1 rad/s to 1 rad/s
- Less than -20 dB over the range 100 rad/s to 1000 rad/s



- Pilot G Response specifies the G-force requirement.

The linearization inputs/outputs are already selected in the **Linearizations** tab. The linear system is computed at a simulation time of 0.



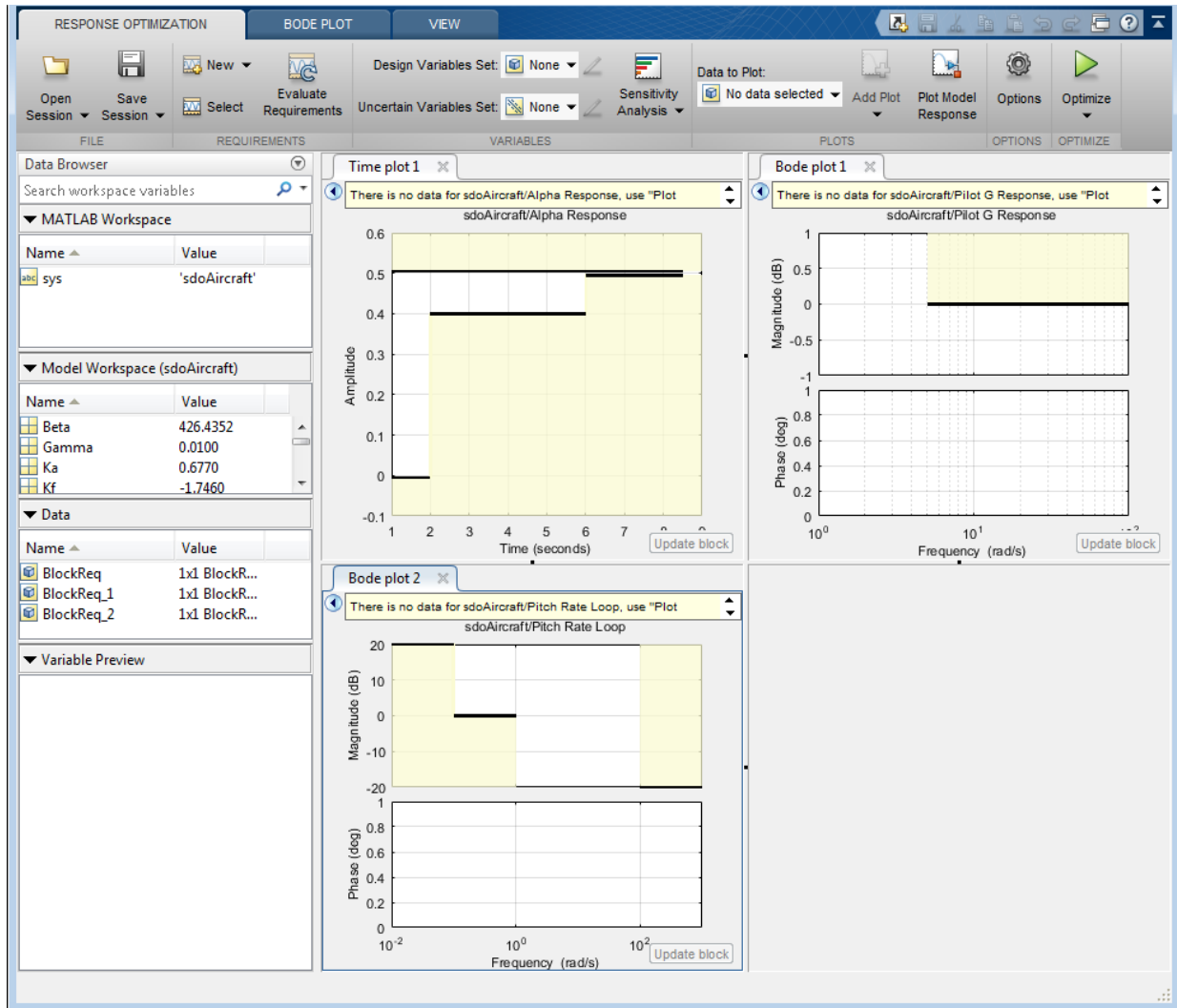
The **Bounds** tab specifies the G-force requirements of less than 0 dB over the range 5 rad/s 100 rad/s.

The image shows a software dialog box titled "Bode Plot". At the top, there is a text box with the instruction: "Compute and display a linear system on a Bode plot. You can also specify bounds on the linear system and assert that the bounds are satisfied." Below this is a tabbed interface with four tabs: "Linearizations", "Bounds", "Logging", and "Assertion". The "Bounds" tab is currently selected. It contains two sections. The first section is titled "Include upper magnitude bound in assertion" and has a checked checkbox. Below it are two input fields: "Frequencies (rad/s):" with the value "[5 100]" and "Magnitudes (dB):" with the value "[0 0]". The second section is titled "Include lower magnitude bound in assertion" and has an unchecked checkbox. Below it are two empty input fields: "Frequencies (rad/s):" and "Magnitudes (dB):". At the bottom of the dialog, there are several buttons: "Show Plot", "Show plot on block open" (with an unchecked checkbox), "Response Optimization...", "OK", "Cancel", "Help", and "Apply". A help icon (question mark) is located in the bottom-left corner.

Open the Response Optimization Tool

Open the Response Optimization tool to configure and run design optimization problems interactively. Click **Response Optimization** on the Block Parameters dialog of Alpha Response, Pitch Rate Loop or Pilot G Response block. Alternatively, type `sdotool('sdoAircraft')`. To show multiple requirement plots at the same time, use the **View** tab in the tool.

3 Response Optimization



The tool detects the requirements specified in the Model Verification blocks and automatically includes them as requirements to satisfy.

Specify Design Variables

Specify the following model parameters as design variables for optimization:

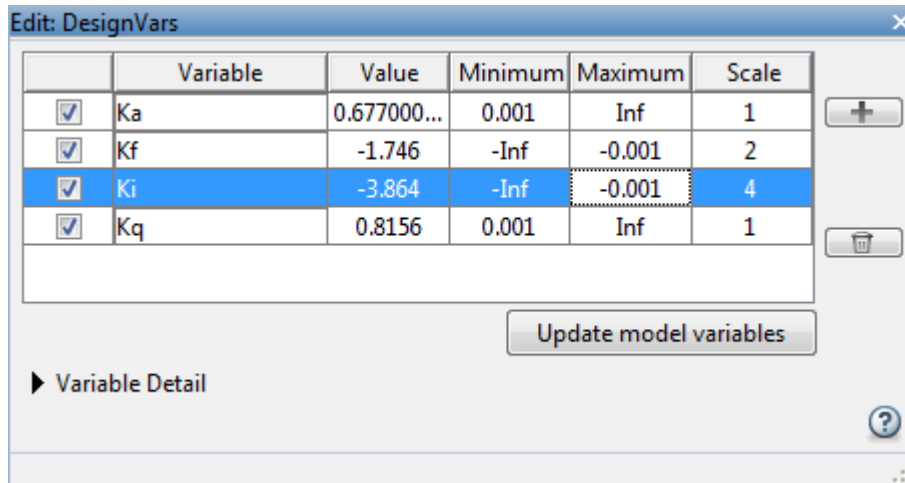
- Controller gains K_i and K_f
- Pitch-rate sensor gain K_q
- Alpha sensor gain K_a

In the **Design Variables Set** drop-down list, select **New**. A dialog to select model parameters for optimization opens.

Select K_i , K_f , K_q and K_a . Click << to add the selected parameters to the design variables set.

Specify minimum and maximum gain values, the K_i and K_f values must remain negative while K_a and K_q must remain positive.

Press **Enter** after you enter the values.

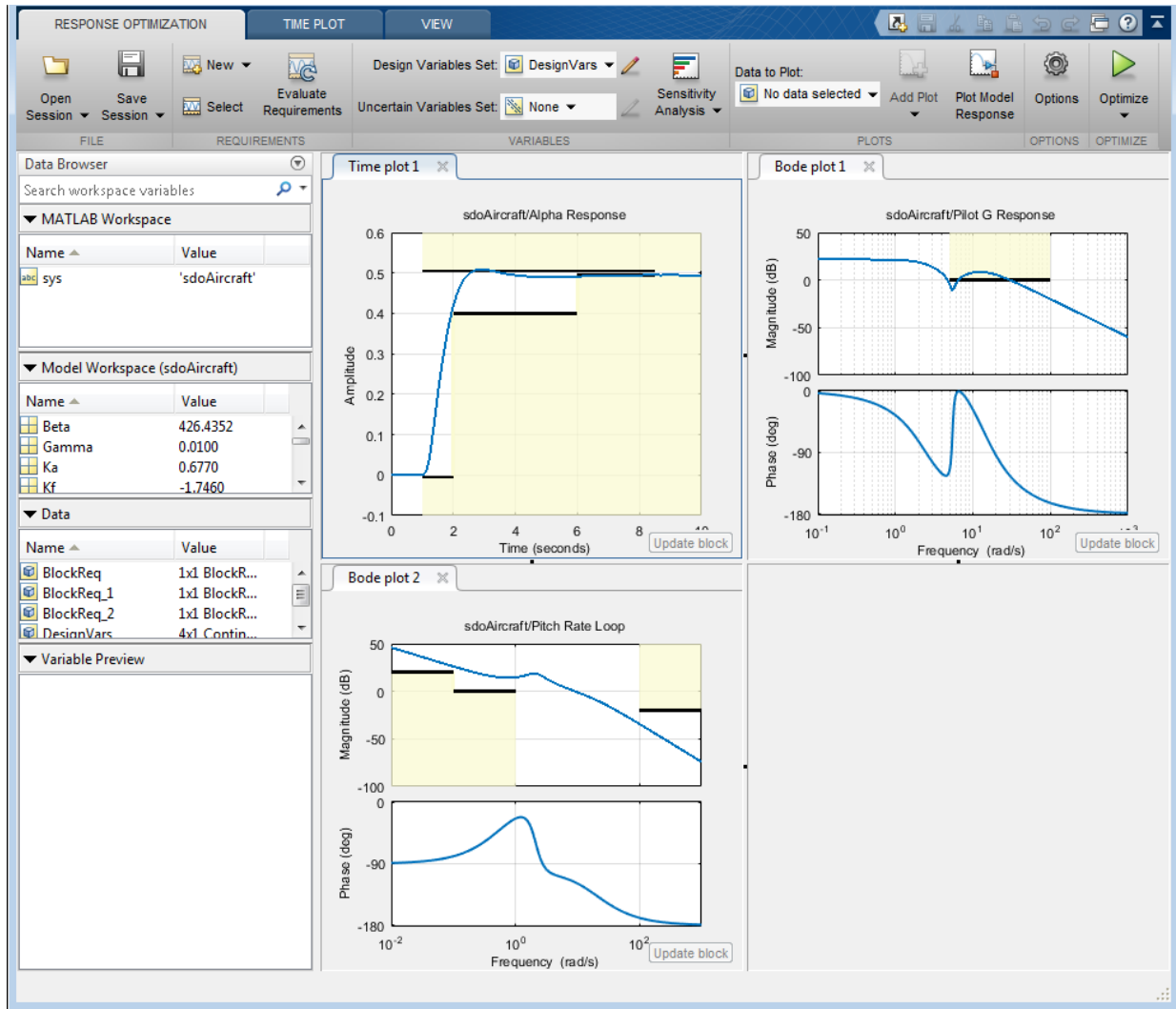


Click **OK**. A new variable `DesignVars` appears in the **Response Optimization Tool Workspace**.

Evaluate the Initial Design

Click **Plot Model Response** to simulate the model and check how well the initial design satisfies the design requirements.

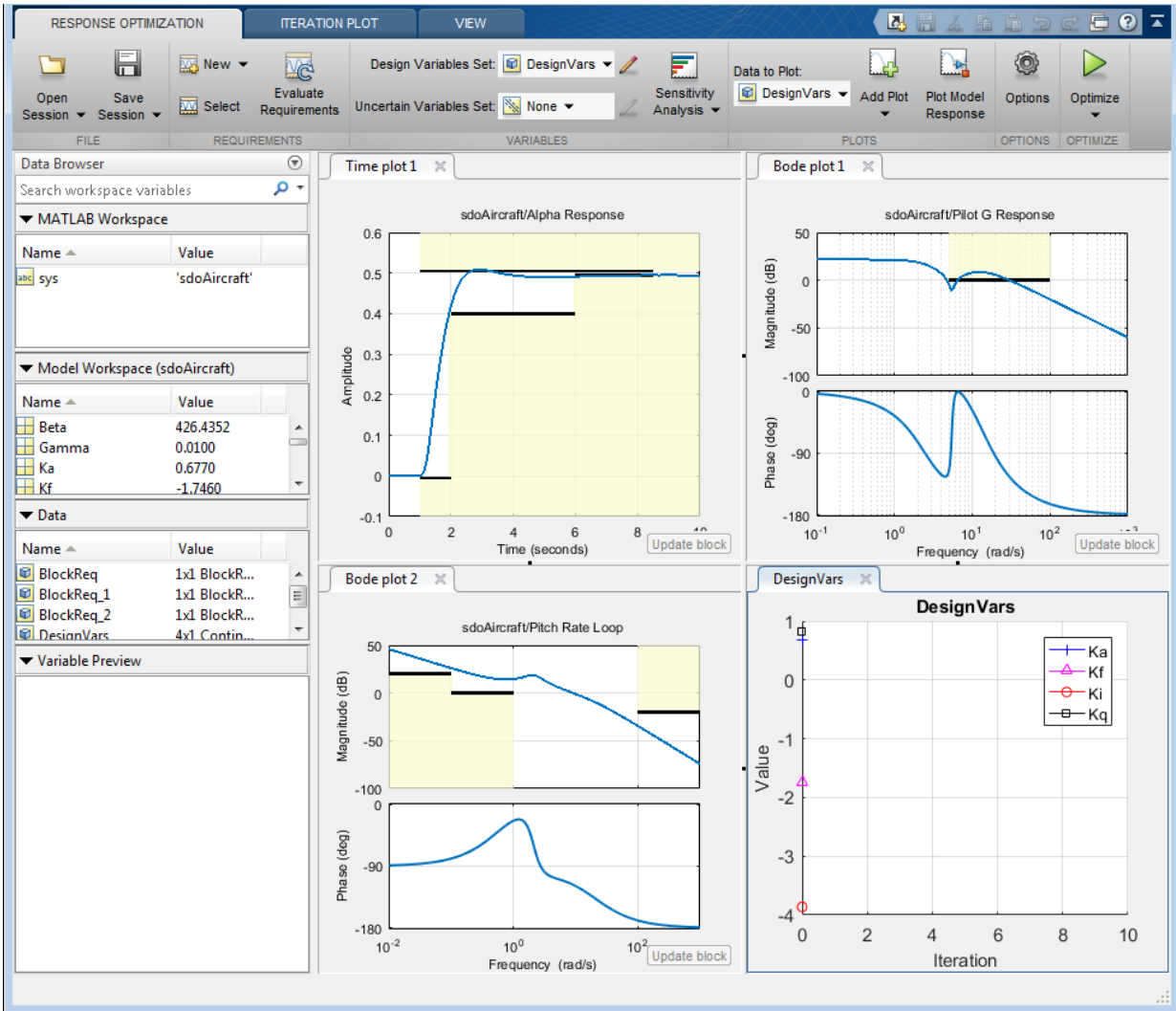
3 Response Optimization



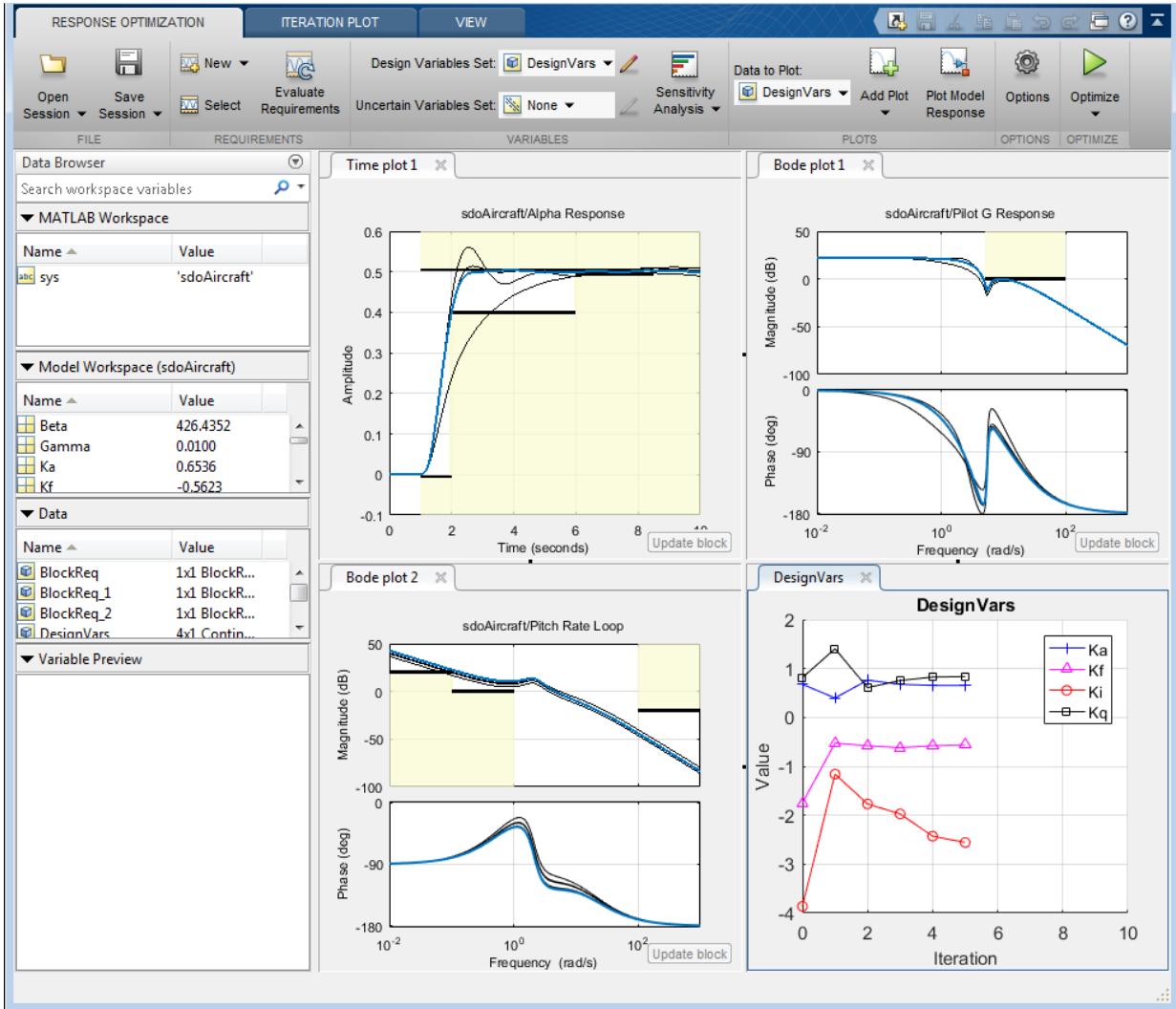
The plots indicate that the current design does not satisfy the pilot G-force requirement and the alpha step response overshoot requirement is violated.

Optimize the Design

Create a plot to display how the controller variables are modified during the optimization. In the **Data To Plot** drop-down list, select **DesignVars**, which contains the optimization design variables K_i , K_f , K_q and K_a . In the **Add Plot** drop-down list, select **Iteration plot**.



Click **Optimize**.



Iteration	F-count	Alpha Response (... (<=0)	Pilot G Response ... (<=0)	Pitch Rate Loop (... (<=0)	Pitch Rate Loop (... (>=0)
0	9	0.1450	8.5841	-0.7160	0.3122
1	20	0.4060	-1.7578	-1.0067	0.0215
2	31	0.1081	-1.0047	-1.3244	-0.1505
3	41	0.2407	-0.0478	-1.2014	-0.0129
4	51	0.0067	0.0584	-1.1881	0.1180
5	61	2.2579e-04	0.0034	-1.1965	0.1432

Optimization started 01-Oct-2012 11:15:57

Optimization converged, 01-Oct-2012 11:16:39

Optimized variable values written to 'DesignVars' in the Design Optimization workspace

Save Iteration... Display Options... Optimize

To load a pre-configured file and run the optimization, click **Open** in the **Response Optimization** tab and select `sdoAircraft_sdoession.mat`. Alternatively load the project by typing:

```
>> load sdoAircraft_sdoession
```

```
>> sdotool(SDOSessionData)
```

The optimization progress window updates at each iteration and shows that the optimization converged after 5 iterations.

The Alpha Response and Pilot G Response plots indicate that the design requirements are satisfied. The DesignVars plot shows that the controller gains converged to new values.

To view the optimized design variable values, click DesignVars in the **Response Optimization Tool Workspace**. The optimized values of the design variables are automatically updated in the Simulink model.

```
% Close the model  
bdclose('sdoAircraft')
```

Design Optimization to Meet a Custom Objective (GUI)

This example shows how to optimize a design to meet a custom objective using the Response Optimization tool. You optimize the cylinder parameters to minimize the cylinder geometry and satisfy design requirements.

Hydraulic Cylinder Model

The hydraulic cylinder model is based on the Simulink model `sldemo_hydcyl`. The model includes:

- `Pump` and `Cylinder Assembly` subsystems. For more information on the subsystems, see "Single Hydraulic Cylinder Simulation" (Simulink).
- A step change applied to the cylinder control valve orifice area that causes the cylinder piston position to change.

Hydraulic Cylinder Design Problem

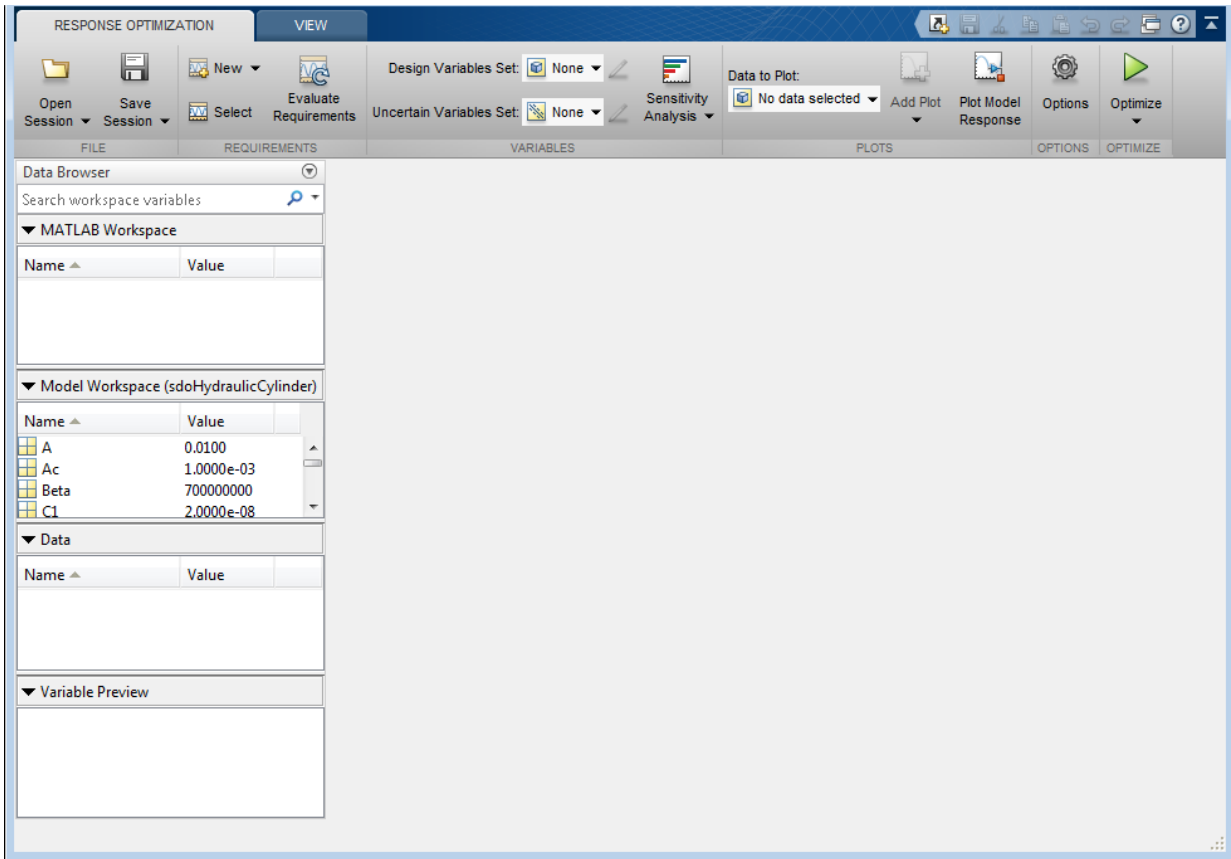
You tune the cylinder cross-sectional area and piston spring constant to meet the following design requirements:

- Ensure that the piston position has a step response rise time of less than 0.04 seconds and setting time of less than 0.05 seconds.
- Limit the maximum cylinder pressures to $1.75e6$ N/m.
- Minimize the cylinder cross-sectional area.

Open the Response Optimization Tool

Open the Response Optimization tool to configure and run design optimization problems interactively using the command

```
sdotool('sdoHydraulicCylinder')
```

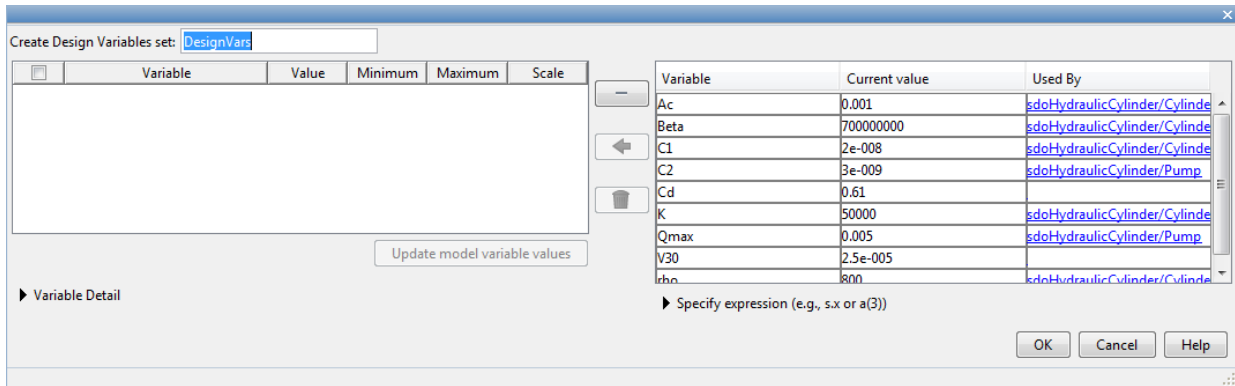


Specify Design Variables

Specify the following model parameters as design variables for optimization:

- Cylinder cross-sectional area A_c
- Piston spring constant K

In the **Design Variables Set** drop-down list, select **New**. A dialog to select model parameters for optimization opens.

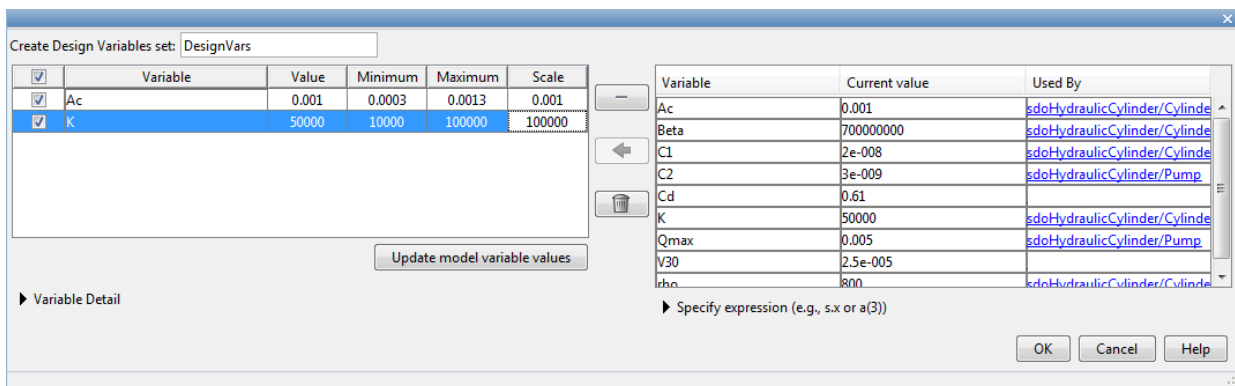


Select A_c and K . Click \leftarrow to add the selected parameters to the design variables set.

Limit the cylinder cross-sectional area to circular area with radius between 1 and 2 centimeters and the piston spring constant to a range of $1e4$ to $10e4$ N/m. To do so, specify the maximum and minimum for the corresponding variable in the **Maximum** and **Minimum** columns.

Because the variable values are different orders of magnitude, scale A_c by $1e-3$ and K by $1e5$.

Press **Enter** after you specify the values.



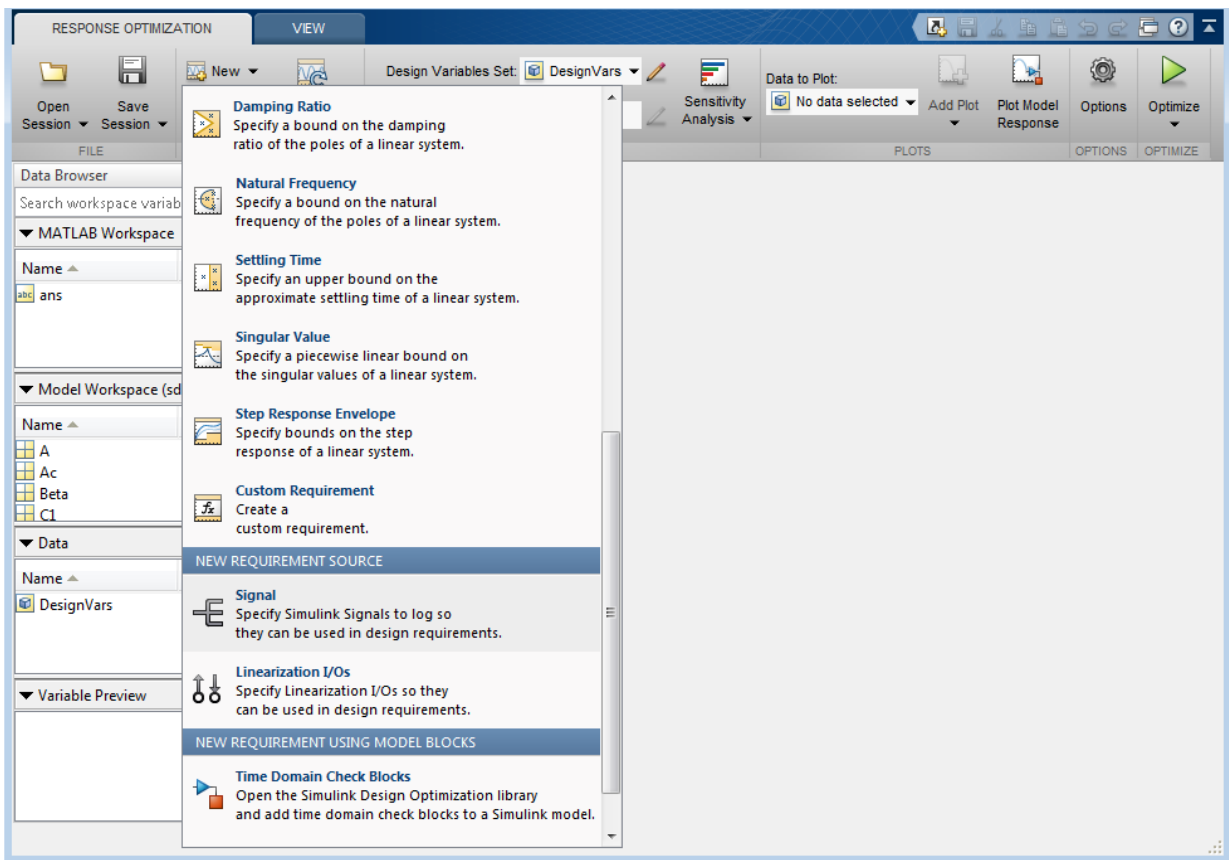
Click **OK**. A new variable `DesignVars` appears in the **Response Optimization Tool Workspace**.

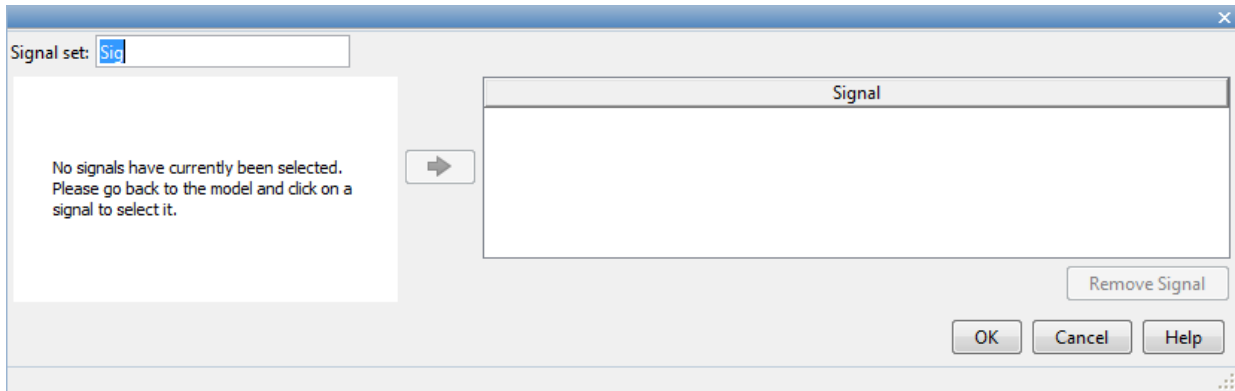
Specify Design Requirements

The design requirements require logged model signals. During optimization, the model is simulated using the current value of the design variables and the logged signal is used to evaluate the design requirements.

Log the cylinder pressures, which is the first output port of the `Cylinder Assembly` block.

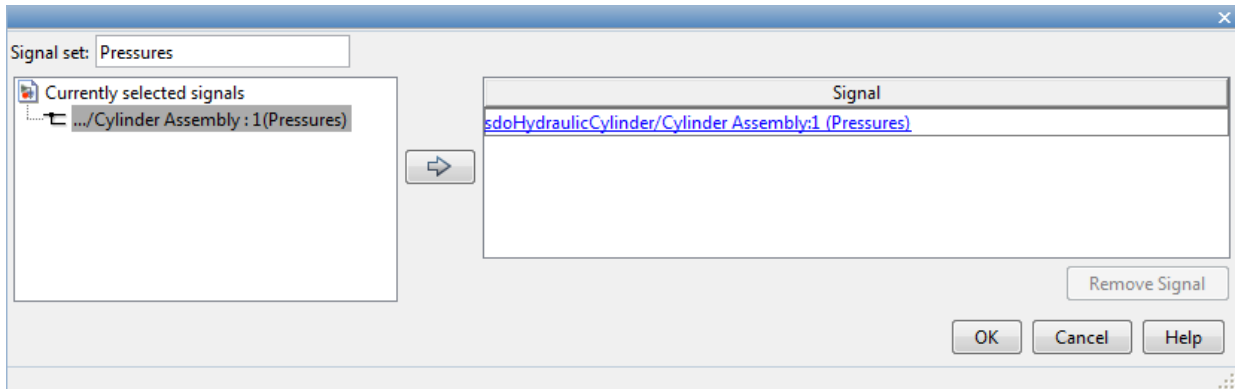
In the **New** drop-down list, select **Signal**. A dialog to select model signals to log opens.





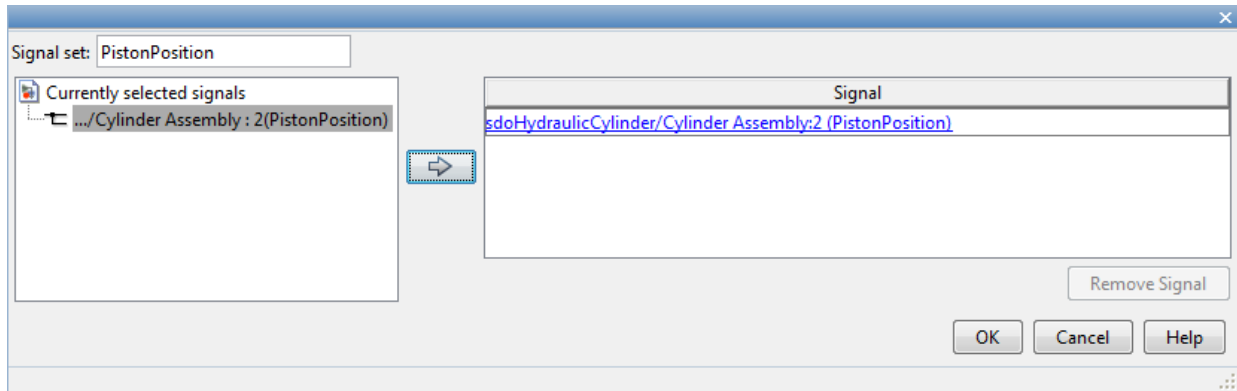
Enter `Pressures` as the signal name in the **Signal set** field. Then, in the Simulink model, click the first output port of the `Cylinder Assembly` block named `Pressure`. The dialog updates to display the selected signal.

Select the signal in the dialog and click `->` to add it to the signal set.



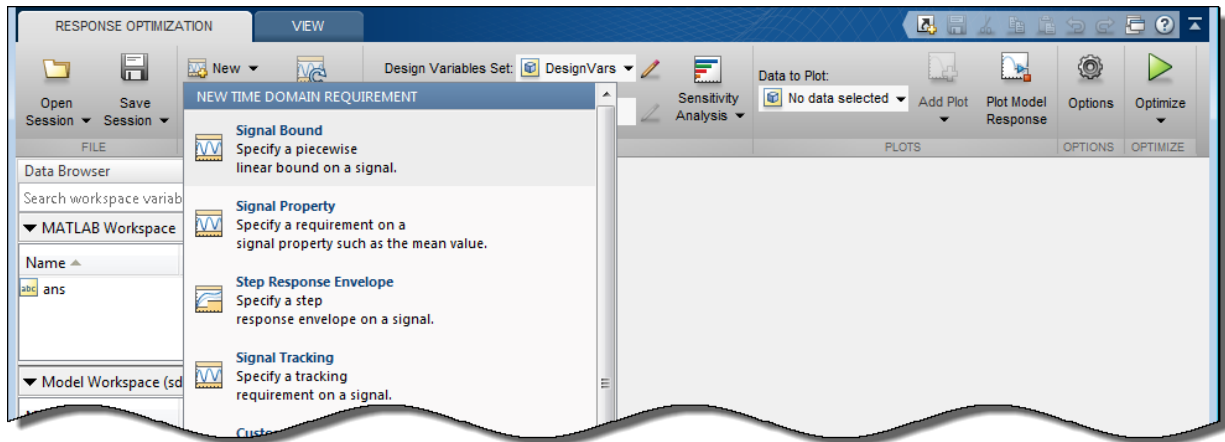
Click **OK**. A new variable **Pressures** appears in the **Response Optimization Tool Workspace**.

Similarly, log the piston position, which is the second output of the `Cylinder Assembly` block, in a variable named `PistonPosition`.



Specify the maximum cylinder pressure requirement of less than $1.75e6$ N/m.

In the **New** drop-down list, select **Signal Bound**. A dialog to create a signal bound requirement opens.



Signal Bound
Use to specify a piecewise linear bound on a signal.

Name:

▼ **Specify Signal Bound**

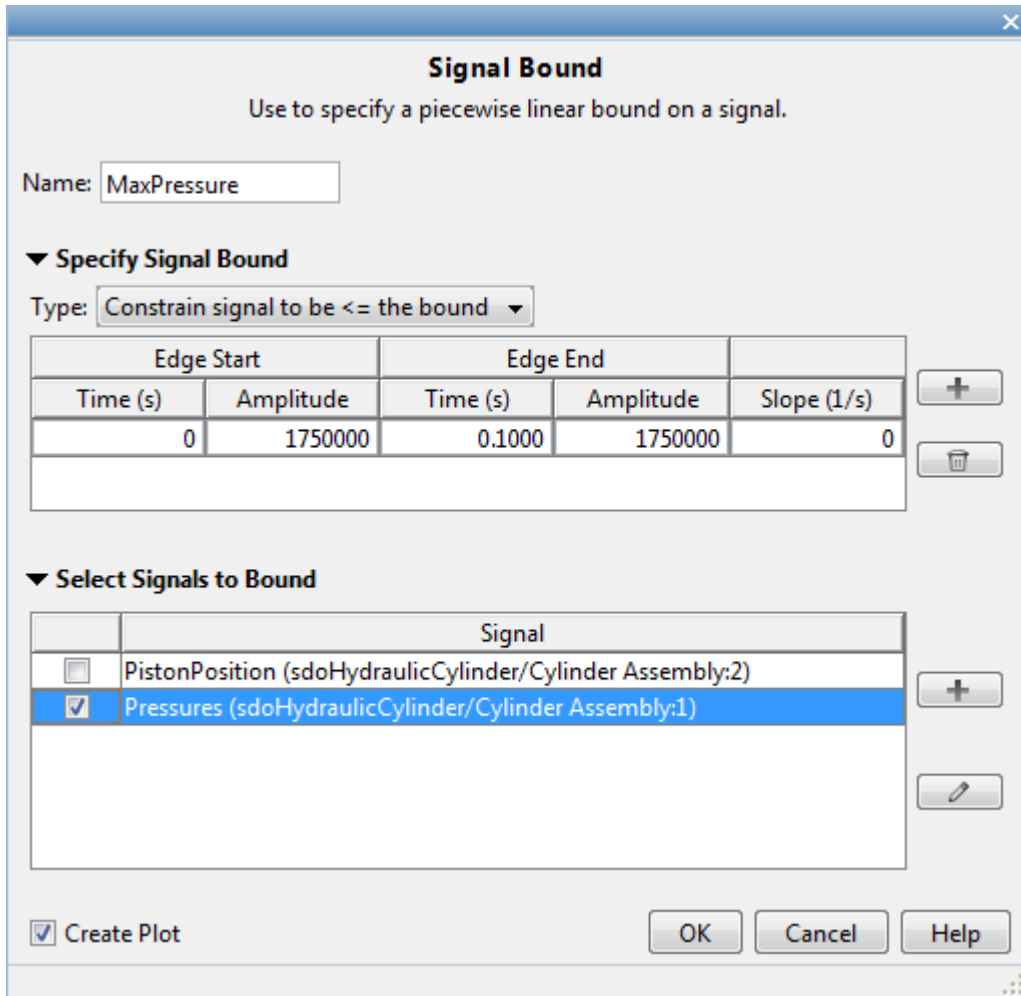
Type:

Edge Start		Edge End		Slope (1/s)
Time (s)	Amplitude	Time (s)	Amplitude	
0	1	10	1	0

► **Select Signals to Bound**

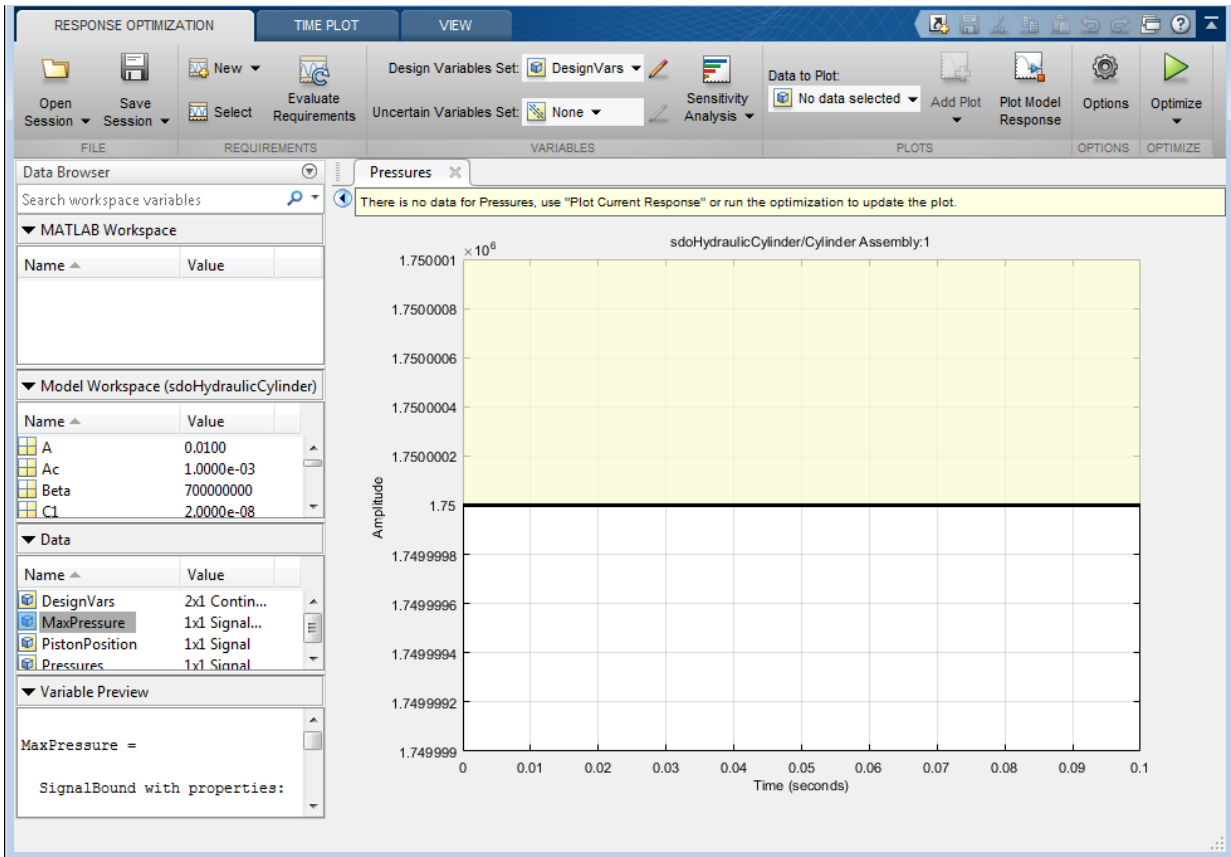
Create Plot

Designate the **Requirement Name** as `MaxPressure`. In both the start and end **Amplitude** columns, enter the maximum pressure requirement of $1.75e6$ N/m, and set the **Edge End Time** to 0.1 s. In the **Select Signals to Bound** area, select `Pressures`, the signal on which this requirement applies.



Click **OK**.

- A new `MaxPressure` variable appears in the **Response Optimization Tool Workspace**.
- A graphical view of the maximum pressure requirement is automatically created.



Specify the piston position step response requirement of rise time of less than 0.04 seconds and a settling time of less than 0.05 seconds.

In the **New** drop-down list of the **Response Optimization** tab, select **Step Response Envelope**. A dialog to create a step response requirement opens.

Specify a requirement named `PistonResponse`, and the required rise and settling time bounds. Select `PistonPosition` as the signal to apply the step response requirement to.

Step Response Envelope
Use to specify a step response envelope on a signal.

Name:

▼ **Specify Step Response Characteristics**

Initial value: Final value:

Step time : seconds

Rise time : seconds % Rise:

Settling time : seconds % Settling:

% Overshoot: % Undershoot:

▼ **Select Signals to Bound**

	Signal	
<input checked="" type="checkbox"/>	PistonPosition (sdoHydraulicCylinder/Cylinder Assembly:2)	+
<input type="checkbox"/>	Pressures (sdoHydraulicCylinder/Cylinder Assembly:1)	

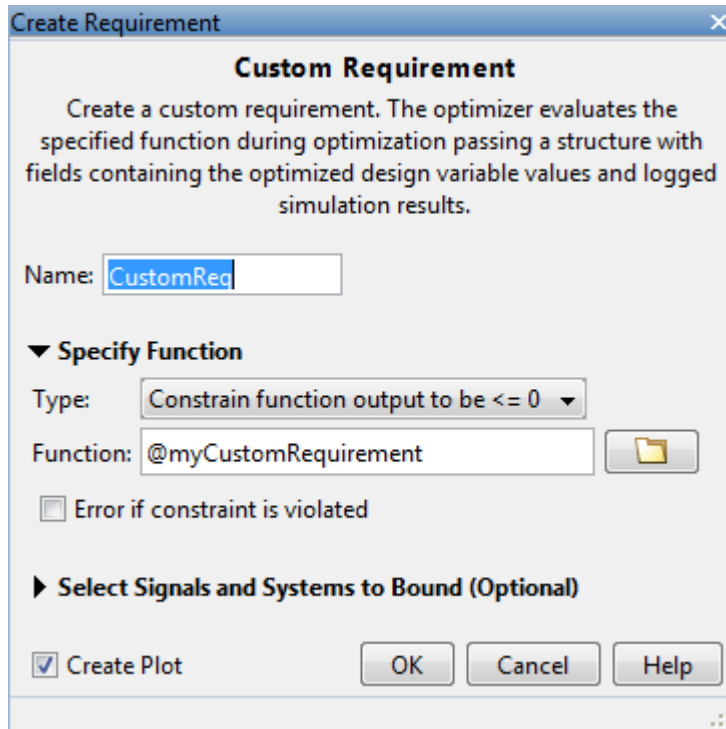
Create Plot

Click **OK**.

Specify Custom Objective

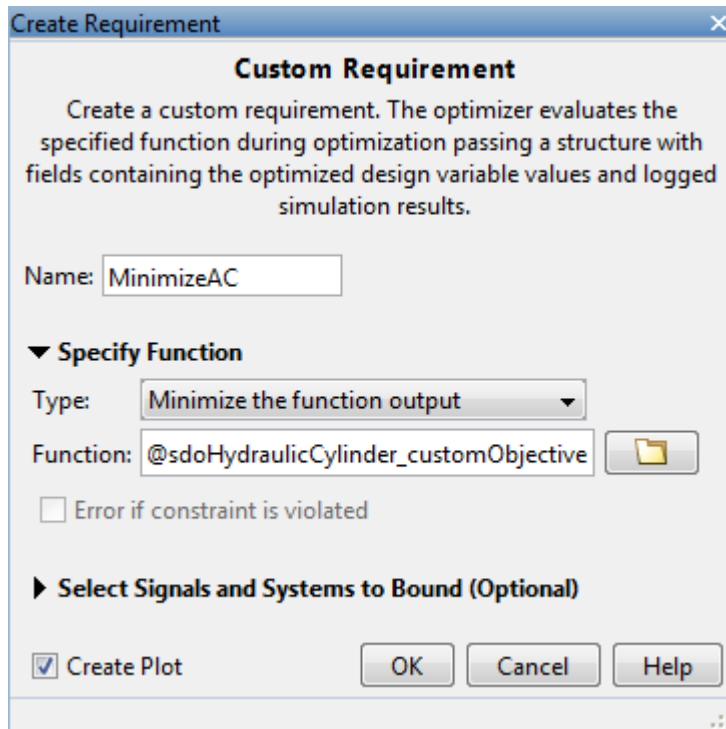
The custom objective is to minimize the cylinder cross-sectional area.

In the **New** drop-down list, select **Custom Requirement**. A dialog to create custom requirement opens.



Specify a function to call during optimization in the **Requirement Function** field. At each optimization iteration, the software calls the function and passes the current design variable values. You can also optionally pass logged signals to the custom requirement. Here, you use `sdoHydraulicCylinder_customObjective` as the custom requirement function, which returns the value of the cylinder cross-sectional area.

In the **Requirement Type** drop-down list, specify whether the requirement is an objective to minimize (`min`), an inequality constraint (`<=`), or an equality constraint (`==`).



```
type sdoHydraulicCylinder_customObjective
```

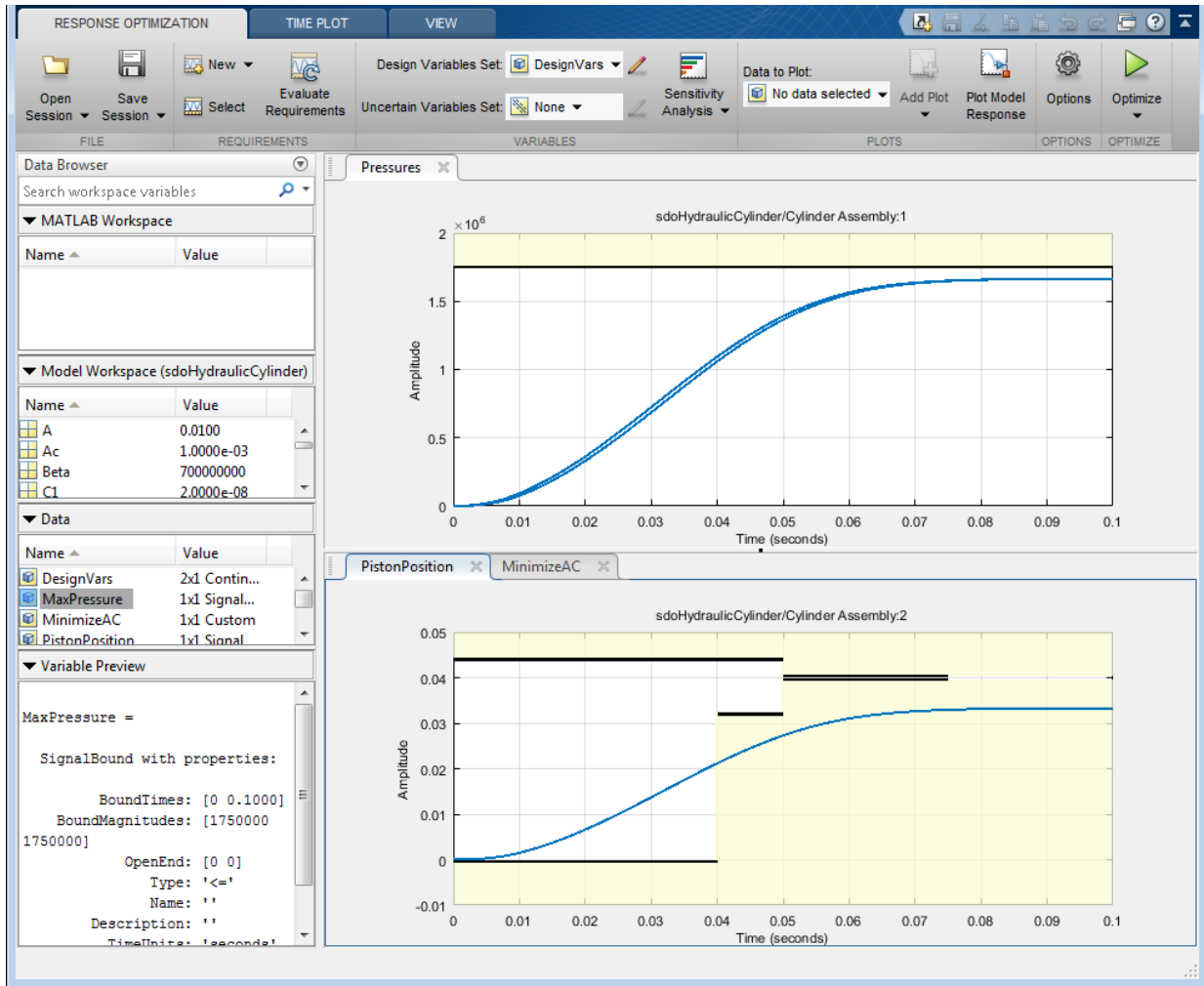
```
function objective = sdoHydraulicCylinder_customObjective(data)
%SDOHYDRAULICCYLINDER_CUSTOMOBJECTIVE
%
% The sdoHydraulicCylinder_customObjective function is used to define a
% custom requirement that can be used in the graphical SDTOOL environment.
%
% The |data| input argument is a structure with fields containing the
% design variable values chosen by the optimizer.
%
% The |objective| return argument is the objective value to be minimized by
% the SDTOOL optimization solver.
%
% Copyright 2011 The MathWorks, Inc.
```

```
%For the cylinder design problem we want to minimize the cylinder  
%cross-sectional area so return the cylinder cross-sectional area as an  
%objective value.  
Ac = data.DesignVars(1);  
objective = Ac.Value;  
end
```

Evaluate the Initial Design

Click **Plot Model Response** to simulate the model and check how well the initial design satisfies the design requirements. To show both requirement plots at the same time, use the plot layout widgets in the **View** tab.

3 Response Optimization

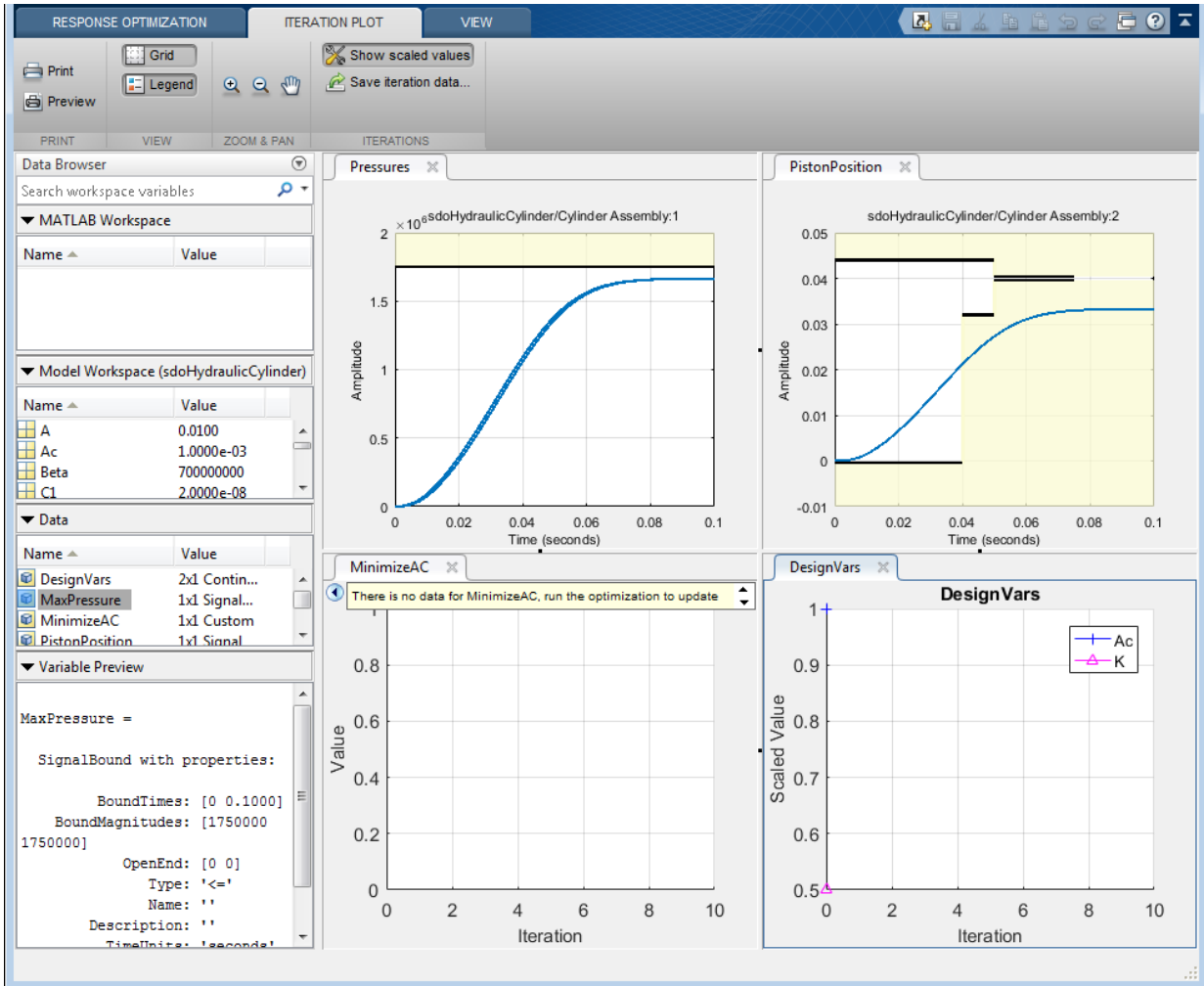


From the plots, see that the maximum pressure requirement is satisfied but the piston position step response requirement is not satisfied.

Optimize the Design

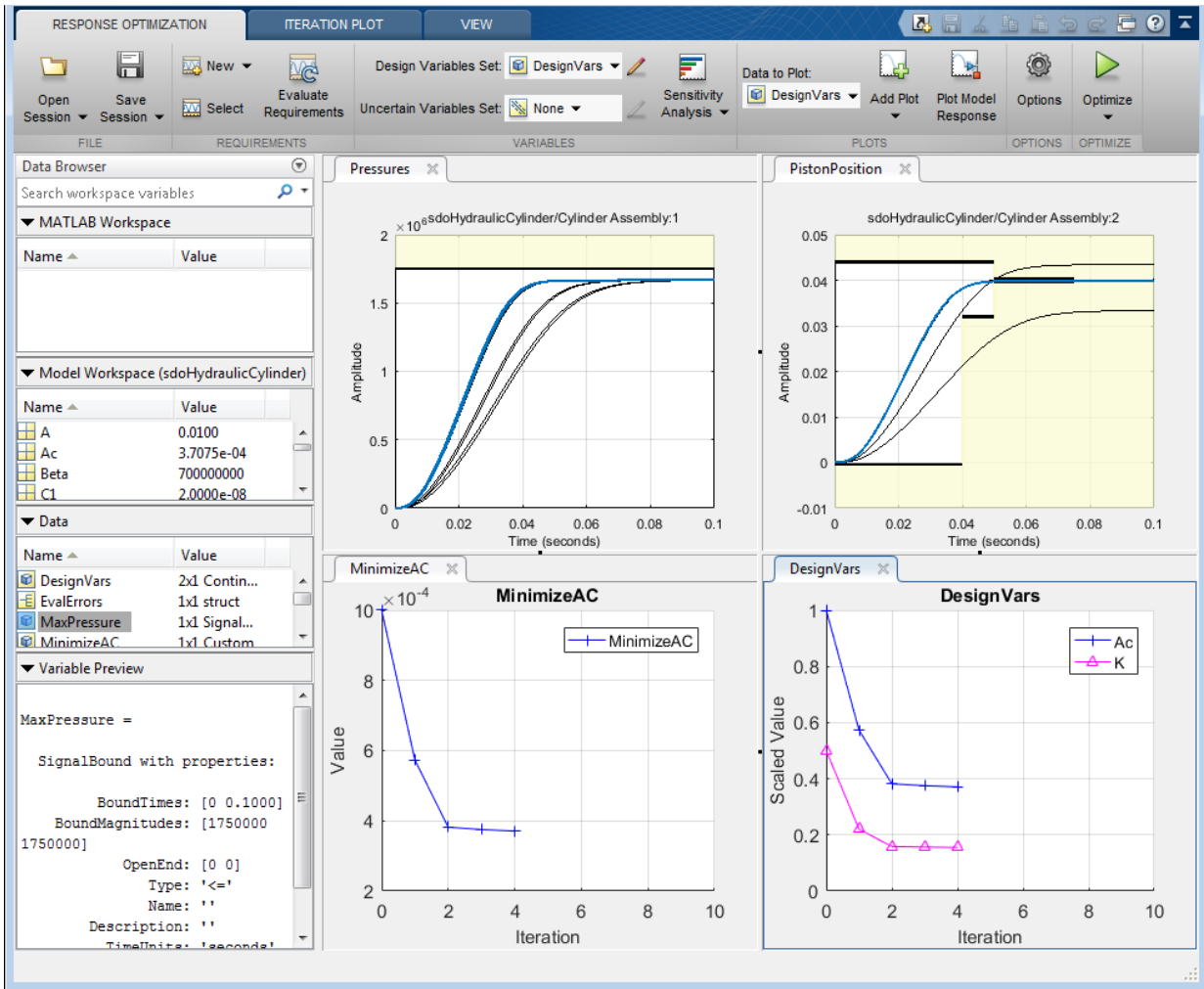
Create a plot to display how the cylinder cross-sectional area and piston spring constant are modified during optimization.

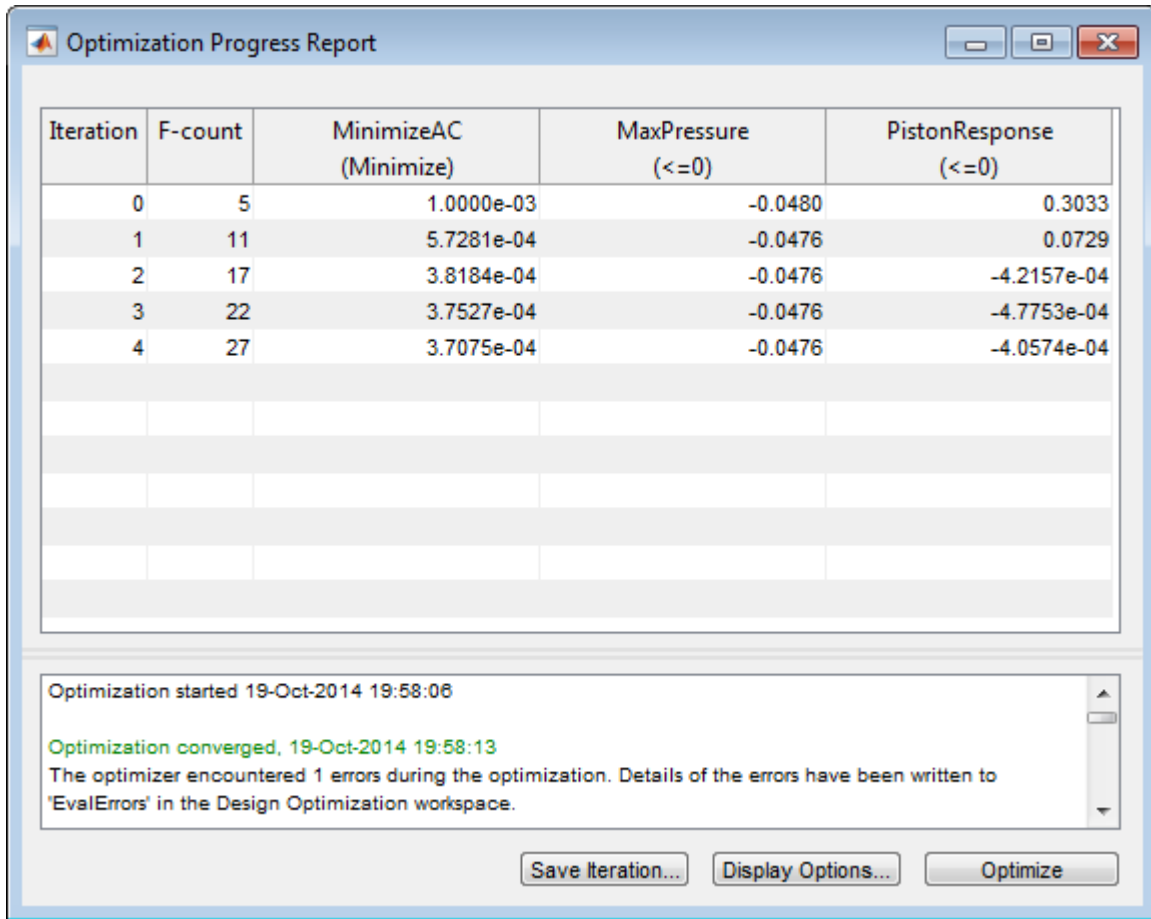
In the **Data to Plot** drop-down list, select **DesignVars**, which contains the optimization design variables A_c and K . In the **Add Plot** drop-down, create a new iteration plot to show the design variable trajectories. For this new plot, click **Show scaled values** in the **Iteration Plot** tab, to facilitate viewing the two trajectories on the same axes.



Click **Optimize** in the **Response Optimization** tab.

3 Response Optimization





The optimization progress window updates at each iteration and shows that the optimization converged after 4 iterations.

The `Pressures` and `PistonPosition` plots indicate that the design requirements are satisfied. The `MinimizeAC` plot shows that the cylinder cross-sectional area A_c is minimized.

To view the optimized design variable values, click the variable name in the **Response Optimization Tool Workspace**. The optimized values of the design variables are automatically updated in the Simulink model.

Related Examples

To learn how to optimize the cylinder design using the `sdo.optimize` command, see "Design Optimization to Meet Custom Objective (Code)".

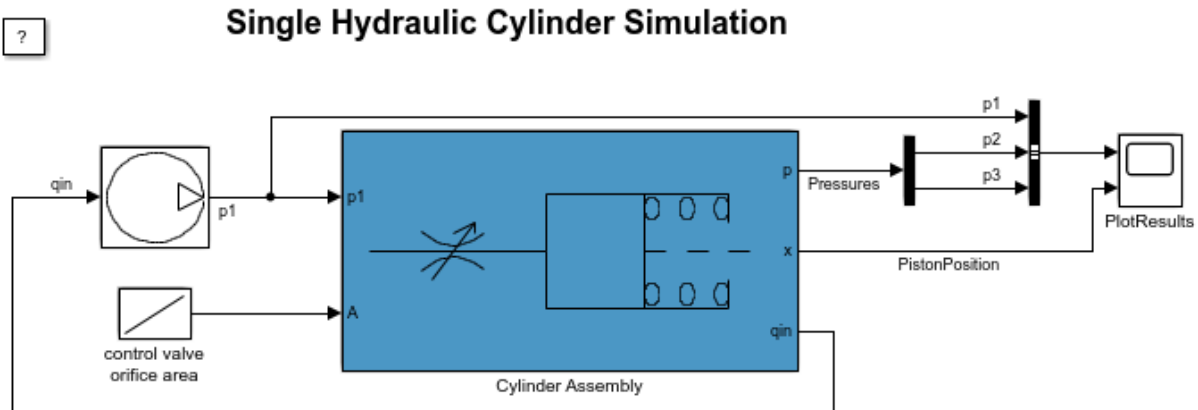
Design Optimization to Meet a Custom Objective (Code)

This example shows how to optimize a design to meet custom objective using `sdo.optimize`. You optimize the cylinder parameters to minimize the cylinder geometry and satisfy design requirements.

Hydraulic Cylinder Model

Open the Simulink model.

```
sys = 'sdoHydraulicCylinder';
open_system(sys);
```



Copyright 1990-2011 The MathWorks, Inc.

The hydraulic cylinder model is based on the Simulink model `sldemo_hydcyl`. The model includes:

- Pump and Cylinder Assembly subsystems. For more information on the subsystems, see "Single Hydraulic Cylinder Simulation" (Simulink).
- A step change applied to the cylinder control valve orifice area that causes the cylinder piston position to change.

Hydraulic Cylinder Design Problem

You tune the cylinder cross-sectional area and piston spring constant to meet the following design requirements:

- Ensure that the piston position has a step response rise time of less than 0.04 seconds and settling time of less than 0.05 seconds.
- Limit the maximum cylinder pressures to 1.75e6 N/m.
- Minimize the cylinder cross-sectional area.

Specify Design Variables

Select the following model parameters as design variables for optimization:

- Cylinder cross-sectional area A_c
- Piston spring constant K

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

Limit the cylinder cross-sectional area to a circular area with radius between 1 and 2 centimeters.

```
Ac.Minimum = pi*1e-2^2; % m^2  
Ac.Maximum = pi*2e-2^2; % m^2
```

Limit the piston spring constant to a range of 1e4 to 10e4 N/m.

```
K.Minimum = 1e4; % N/m  
K.Maximum = 10e4; % N/m
```

Specify Design Requirements

The design requirements require logged model signals. During optimization, the model is simulated using the current value of the design variables and the logged signal is used to evaluate the design requirements.

Log the following signals:

- Cylinder pressures, available at the first output port of the `Cylinder Assembly` block

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

- Piston position, available at the second output port of the Cylinder Assembly block

```
PistonPosition = Simulink.SimulationData.SignalLoggingInfo;
PistonPosition.BlockPath = 'sdoHydraulicCylinder/Cylinder Assembly';
PistonPosition.OutputPortIndex = 2;
```

Create an object to store the logging information and use later to simulate the model

```
simulator = sdo.SimulationTest('sdoHydraulicCylinder');
simulator.LoggingInfo.Signals = [PistonPosition, Pressures];
```

Specify the piston position step response requirement of rise time of less than 0.04 seconds and settling time less than of 0.05 seconds.

```
PistonResponse = sdo.requirements.StepResponseEnvelope;
set(PistonResponse, ...
    'RiseTime', 0.04, ...
    'FinalValue', 0.04, ...
    'SettlingTime', 0.05, ...
    'PercentSettling', 1);
```

Specify the maximum cylinder pressure requirement of less than 1.75e6 N/m.

```
MaxPressure = sdo.requirements.SignalBound;
set(MaxPressure, ...
    'BoundTimes', [0 0.1], ...
    'BoundMagnitudes', [1.75e6 1.75e6], ...
    'Type', '<=');
```

For convenience, collect the performance requirements into a single structure to use later.

```
requirements = struct(...
    'PistonResponse', PistonResponse, ...
    'MaxPressure', MaxPressure);
```

Create Objective/Constraint Function

To optimize the cylinder cross-sectional area and piston spring constant, create a function to evaluate the cylinder design. This function is called at each optimization iteration.

Here, use an anonymous function with one argument that calls the `sdoHydraulicCylinder_design` function.

```
evalDesign = @(p) sdoHydraulicCylinder_design(p,simulator,requirements);
```

The function:

- Has one input argument that specifies the cylinder cross-sectional area and piston spring constant values.
- Returns the optimization objective value and optimization constraint violation values.

The optimization solver minimizes the objective value and attempts to keep the optimization constraint violation values negative. Type `help sdoExampleCostFunction` for more details on how to write the objective/constraint function.

The `sdoHydraulicCylinder_design` function uses the `simulator` and `requirements` objects to evaluate the design. Type `edit sdoHydraulicCylinder_design` to examine the function in more detail.

```
type sdoHydraulicCylinder_design
```

```
function design = sdoHydraulicCylinder_design(p,simulator,requirements)
%SDOHYDRAULICCYLINDER_DESIGN
%
% The sdoHydraulicCylinder_design function is used to evaluate a cylinder
% design.
%
% The |p| input argument is the vector of cylinder design parameters.
%
% The |simulator| input argument is a sdo.SimulinkTest object used to
% simulate the |sdoHydraulicCylinder| model and log simulation signals
%
% The |requirements| input argument contains the design requirements used
% to evaluate the cylinder design
%
% The |design| return argument contains information about the design
% evaluation that can be used by the |sdo.optimize| function to optimize
% the design.
%
% see also sdo.optimize, sdoExampleCostFunction
```

```

% Copyright 2011 The MathWorks, Inc.

%% Simulate the model
%
% Use the simulator input argument to simulate the model and log model
% signals.
%
% First ensure that we simulate the model with the parameter values chosen
% by the optimizer.
%
simulator.Parameters = p;
% Simulate the model and log signals.
%
simulator = sim(simulator);
% Get the simulation signal log, the simulation log name is defined by the
% model |SignalLoggingName| property
%
logName = get_param('sdoHydraulicCylinder','SignalLoggingName');
simLog = get(simulator.LoggedData,logName);

%% Evaluate the design requirements
%
% Use the requirements input argument to evaluate the design requirements
%
% Check the PistonPosition signal against the stepresponse requirement
%
PistonPosition = get(simLog,'PistonPosition');
cPiston = evalRequirement(requirements.PistonResponse,PistonPosition.Values);
% Check the Pressure signals against the maximum requirement
%
Pressures = find(simLog,'Pressures');
cPressure = evalRequirement(requirements.MaxPressure,Pressures.Values);
% Use the PistonResponse and MaxPressure requirements as non-linear
% constraints for optimization.
design.Cleq = [cPiston(:);cPressure(:)];
% Add design objective to minimize the Cylinder cross-sectional area
Ac = p(1);           %Since we called sdo.optimize(evalDesign,[Ac;K])
design.F = Ac.Value;
end

```

Evaluate the Initial Design

Call the objective function with the initial cylinder cross-sectional area and initial piston spring constant.

```
initDesign = evalDesign([Ac;K]);
```

The function simulates the model and evaluates the design requirements. The scope shows that the maximum pressure requirement is satisfied but the piston position step response requirement is not satisfied.

`initDesign` is a structure with the following fields:

- `Cleq` shows that some of the inequality constraints are positive indicating they are not satisfied by the initial design.

```
initDesign.Cleq
```

```
ans =
```

```
-0.3839  
-0.1861  
-0.1836  
-1.0000  
0.3033  
0.2909  
0.1671  
0.2326  
-0.0480  
-0.0480
```

- `F` shows the optimization objective value (in this case the cylinder cross-sectional area). The initial design cross-sectional area, as expected, has the same value as the initial cross-sectional area parameter `Ac`.

```
initDesign.F
```

```
ans =
```

```
1.0000e-03
```

Optimize the Design

Pass the objective function, initial cross-sectional area and piston spring constant values to `sdo.optimize`.

```
[pOpt,optInfo] = sdo.optimize(evalDesign,[Ac;K]);
```

```
Optimization started 15-Aug-2017 01:28:27
```

Iter	F-count	f(x)	max constraint	Step-size	First-order optimality
0	5	0.001	0.3033		
1	11	0.00057281	0.07293	0.48	85.4
2	17	0.000391755	0	0.128	28
3	22	0.000388463	0	0.00232	0.00414
4	27	0.000384272	0	0.00291	0.00222
5	32	0.000374608	0	0.00677	0.00134
6	37	0.000373106	0	0.00109	0.000544

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.

The optimization repeatedly evaluates the cylinder design by adjusting the cross-sectional area and piston spring constant to meet the design requirements. From the scope, see that the maximum pressure and piston response requirements are met.

The `sdo.optimize` function returns:

- `pOpt` shows the optimized cross-sectional area and piston spring constant values.

```
pOpt
```

```
pOpt(1,1) =
```

```

    Name: 'Ac'
    Value: 3.7311e-04
  Minimum: 3.1416e-04
  Maximum: 0.0013
    Free: 1
   Scale: 0.0020
    Info: [1x1 struct]
```

```
pOpt(2,1) =
```

```
    Name: 'K'
```

```
Value: 1.5650e+04
Minimum: 10000
Maximum: 100000
Free: 1
Scale: 65536
Info: [1x1 struct]
```

```
2x1 param.Continuous
```

- `optInfo` is a structure that contains optimization termination information such as number of optimization iterations and the optimized design.

```
optInfo
```

```
optInfo =
```

```
struct with fields:
```

```
    Cleq: [10x1 double]
         F: 3.7311e-04
    Gradients: [1x1 struct]
    exitflag: 1
    iterations: 6
    SolverOutput: [1x1 struct]
    Stats: [1x1 struct]
```

For example, the `Cleq` field shows the optimized non-linear inequality constraints are all non-positive to within optimization tolerances, indicating that the maximum pressure and piston response requirements are satisfied.

```
optInfo.Cleq
```

```
ans =
```

```
-0.0999
-0.0165
-0.0165
-1.0000
-0.2059
-0.0015
```



```
-0.0034  
-0.0002  
-0.0476  
-0.0476
```

The `F` field contains the optimized cross-sectional area. The optimized cross-sectional area value is nearly 50% less than the initial value.

```
optInfo.F
```

```
ans =
```

```
3.7311e-04
```

Update the Model Variable Values

By default, the model variables `Ac` and `K` are not updated at the end of optimization. Use the `setValueInModel` command to update the model variable values.

```
sdo.setValueInModel('sdoHydraulicCylinder',pOpt)
```

Related Examples

To learn how to optimize the cylinder design using the Response Optimization tool, see "Design Optimization to Meet Custom Objective (GUI)".

```
% Close the model  
bdclose('sdoHydraulicCylinder')
```

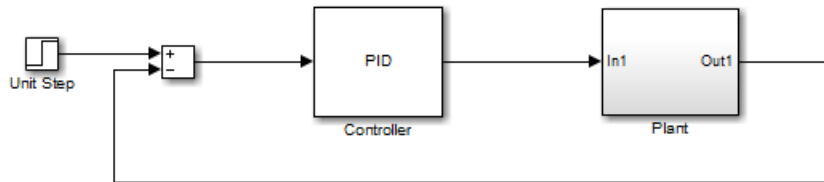
Design Optimization to Meet Custom Signal Requirements (GUI)

This example shows how to optimize a design to meet a custom signal requirement. You optimize the controller parameters to minimize the plant actuation signal energy while satisfying step response requirements.

- 1 Load a saved Response Optimization tool session.

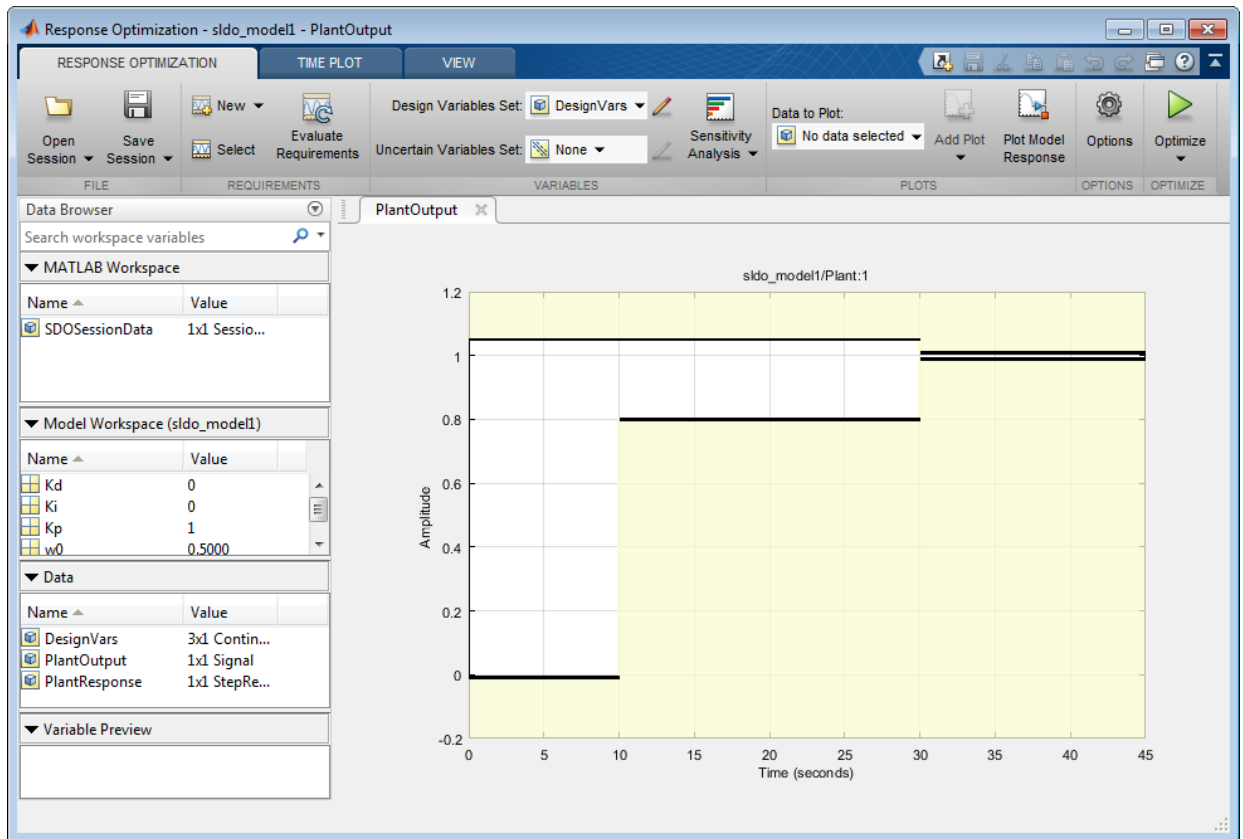
```
load sldo_model1_custom_signal_session
sdotool(SDOSessionData);
```

The following Simulink model opens.



The Response Optimization tool, configured with the following settings, also opens:

- Step response characteristics, specified on the output of the Plant block, that the model output must satisfy:
 - Maximum overshoot of 5%
 - Maximum rise time of 10 seconds
 - Maximum settling time of 30 seconds
- Design variable set with the controller parameters K_p , K_i and K_d . These parameters have a minimum value of 0.
- The variables for step requirements (PlantResponse), logged signal (PlantOutput) and design variables (DesignVars) which appear in the **Data** area.



2 Specify a signal to log. You apply the custom requirement on this logged signal.

a Select **New > Signal**.

A window opens where you select a signal to log.

b In the Simulink model window, click the output of the Controller block.

The window updates to display the selected signal.

c Select the signal and click  to add it to the signal set.

d In **Signal set**, enter `PlantActuator`.

Click **OK**. A new variable `PlantActuator` appears in the **Data** area.

3 Specify the custom requirement to apply to the signal.

The custom requirement calls the objective function `sldo_model1_minimize_energy` which returns the energy in the `PlantActuator` signal. The signal energy is minimized. This function accepts:

- An input argument `data` which is a structure with fields for the design variables in the **Data** area. Signals are logged for the nominal and uncertain parameter values if there are any.
- Returns the objective value to be minimized.

Tip To see the contents of this function, type `edit sldo_model1_minimize_energy`.

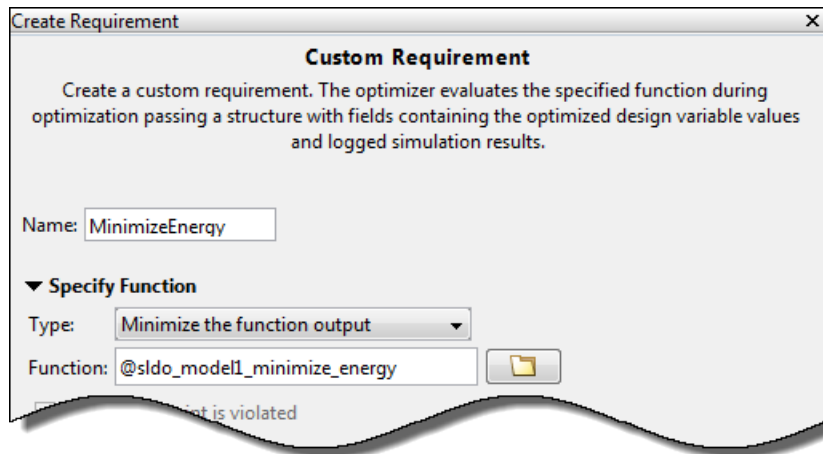
a Select **New > Custom Requirement**.

A window opens where you specify the custom requirement.

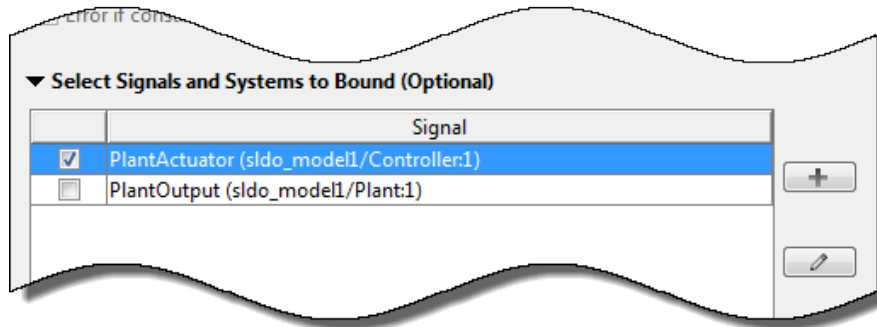
b Specify `MinimizeEnergy` as the **Name**.

c Specify `@sldo_model1_minimize_energy` as the **Function**.

d Select `Minimize` the function output as the **Type**.

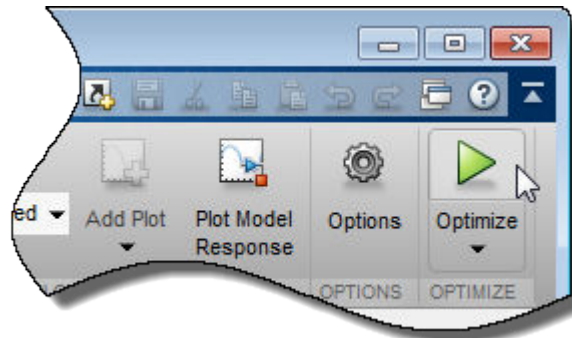


4 In the **Select Signals and Systems to Bound** area, select the `PlantActuator` check box to associate the custom requirement with that signal.

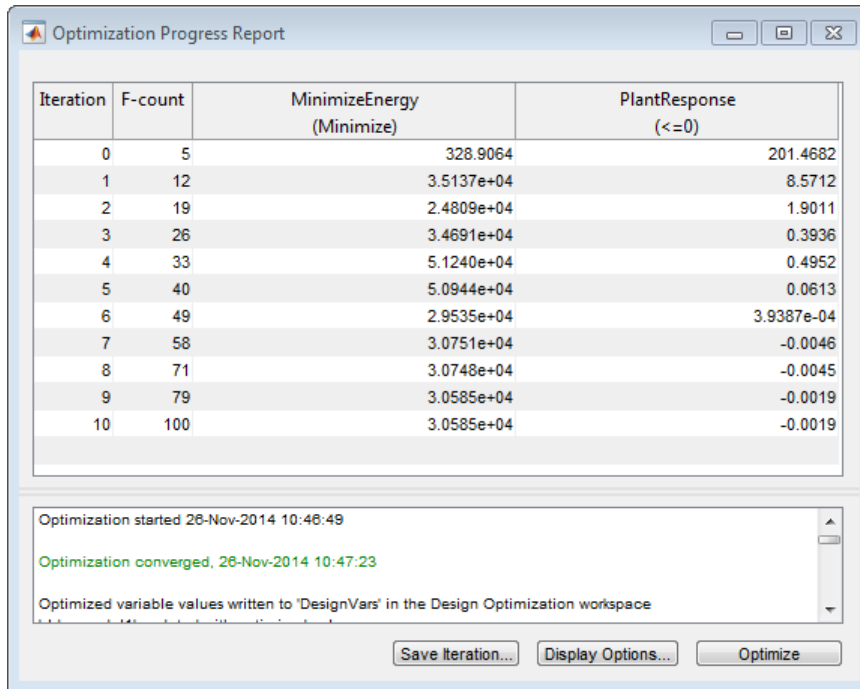


Click **OK**. A new variable appears in the **Data** area of the tool. The window also updates to graphically display the custom signal requirement.

- 5 Click **Optimize**.



After a few iterations, the optimization converges to meet both the custom signal and step response requirements.



The screenshot shows a window titled "Optimization Progress Report" with a table of results and a log of progress. The table has four columns: Iteration, F-count, MinimizeEnergy (Minimize), and PlantResponse (<=0). The log shows that optimization started on 26-Nov-2014 at 10:46:49 and converged at 10:47:23. The optimized variable values were written to 'DesignVars' in the Design Optimization workspace.

Iteration	F-count	MinimizeEnergy (Minimize)	PlantResponse (<=0)
0	5	328.9064	201.4682
1	12	3.5137e+04	8.5712
2	19	2.4809e+04	1.9011
3	26	3.4691e+04	0.3936
4	33	5.1240e+04	0.4952
5	40	5.0944e+04	0.0613
6	49	2.9535e+04	3.9387e-04
7	58	3.0751e+04	-0.0046
8	71	3.0748e+04	-0.0045
9	79	3.0585e+04	-0.0019
10	100	3.0585e+04	-0.0019

Optimization started 26-Nov-2014 10:46:49
Optimization converged, 26-Nov-2014 10:47:23
Optimized variable values written to 'DesignVars' in the Design Optimization workspace

Save Iteration... Display Options... Optimize

6 Close the model.

```
setOption(sdotool('sldo_model1'),'NoPromptClose',true)  
bdclose('sldo_model1')
```

See Also

Related Examples

- “Design Optimization Using Frequency-Domain Check Blocks (GUI)”

Design Optimization to Meet Frequency-Domain Requirements (GUI)

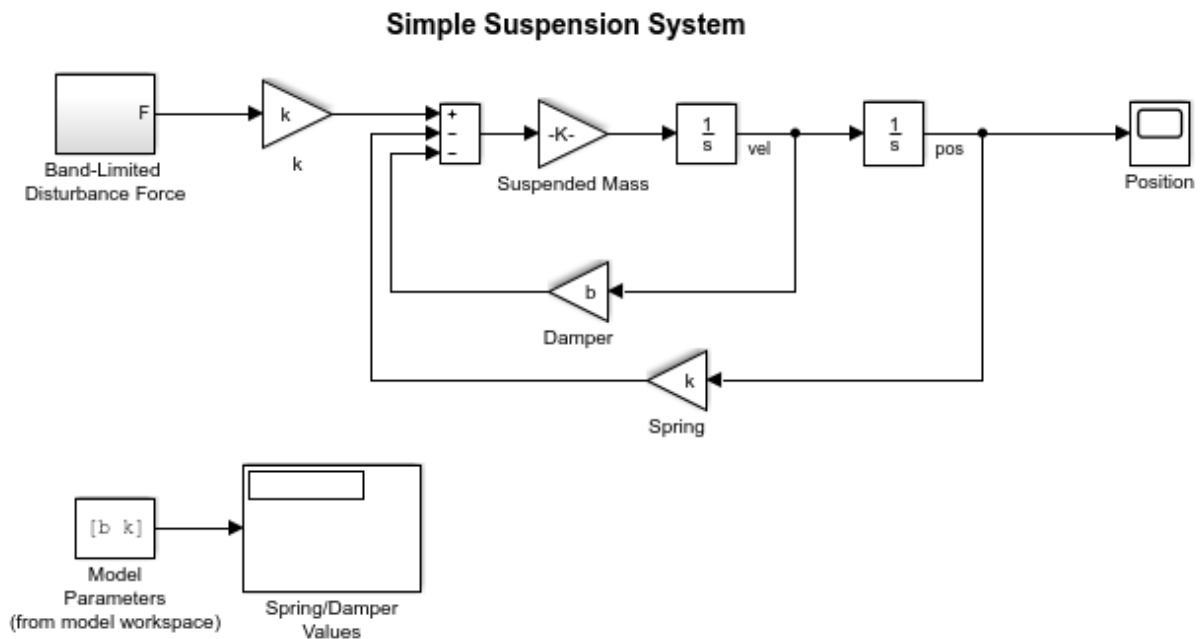
This example shows how to tune model parameters to meet frequency-domain requirements using the Response Optimization tool.

This example requires Simulink® Control Design™.

Suspension Model

Open the Simulink Model.

```
open_system('sdoSimpleSuspension')
```



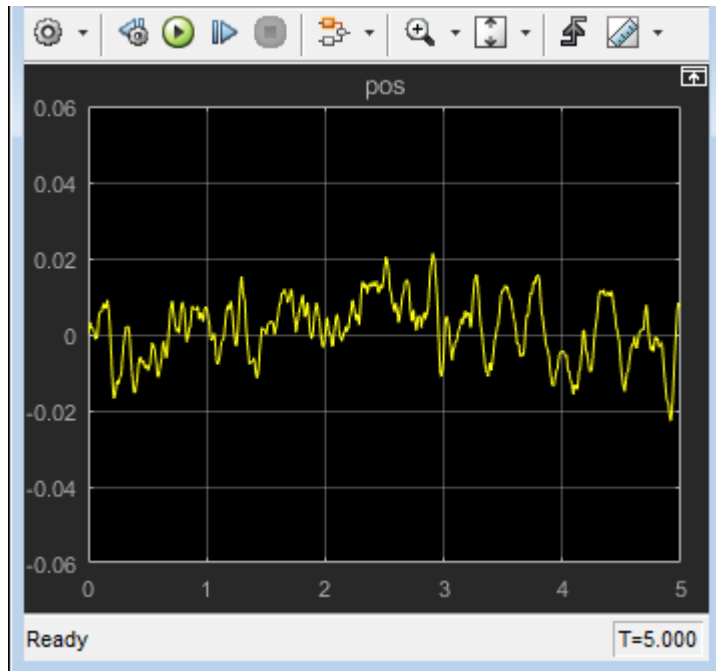
Copyright 2002-2015 The MathWorks, Inc.

Mass-spring-damper models represent simple suspension systems and for this example we tune the system to meet typical suspension requirements. The model implements the

second order system representing a mass-spring-damper using Simulink blocks and includes:

- a Mass gain block parameterized by the total suspended mass, m_0+m_{Load} . The total mass is the sum of a nominal mass, $|m_0|$, and a variable load mass, m_{Load} .
- a Damper gain block parameterized by the damping coefficient, b .
- a Spring gain block parameterized by the spring constant, k .
- two integrator blocks to compute the mass velocity and position.
- a Band-Limited Disturbance Force block applying a disturbance force to the mass. The disturbance force is assumed to be band-limited white noise.

Simulate the model to view the system response to the applied disturbance force.



Design Problem

The initial system has a bandwidth that is too high. This can be seen from the spiky position signal. You tune the spring and damper values to meet the following requirements:

- The -3dB system bandwidth must not exceed 10 rad/s.
- The damping ratio of the system must be less than $1/\sqrt{2}$. This ensures that no frequencies in pass band are amplified by the system.
- Minimize the expected failure rate of the system. The expected failure rate is described by a Weibull distribution dependent on the mass, spring, and damper values.
- These requirements must all be satisfied as the load mass ranges from 0 to 20.

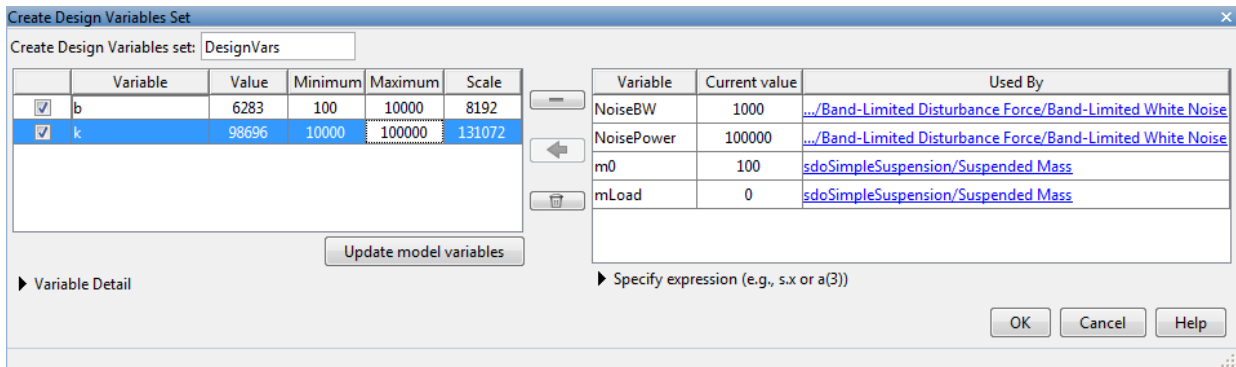
Open the Response Optimization Tool

In the Simulink model **Analysis** menu, select **Response Optimization**.

Specify Design Variables

In the **Design Variables Set** list, select **New**. Add the **b** and **k** model variables to the design variable set.

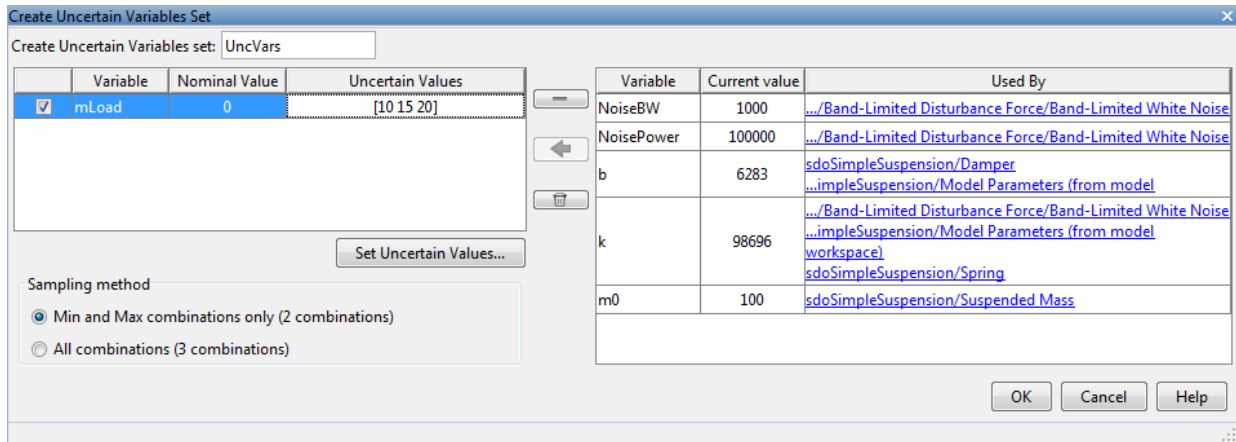
- Specify the Minimum and Maximum values for the **b** variable as 100 and 10000 respectively.
- Specify the Minimum and Maximum values for the **k** variable as 10000 and 100000 respectively.



Click **OK**. A new variable, `DesignVars`, appears in the **Response Optimization Tool Workspace**.

In the **Uncertain Variables Set** list, select **New**. Add the `mLoad` variable to the uncertain variables set.

- Specify the **Uncertain Values** value for the `mLoad` variable as [10 15 20]



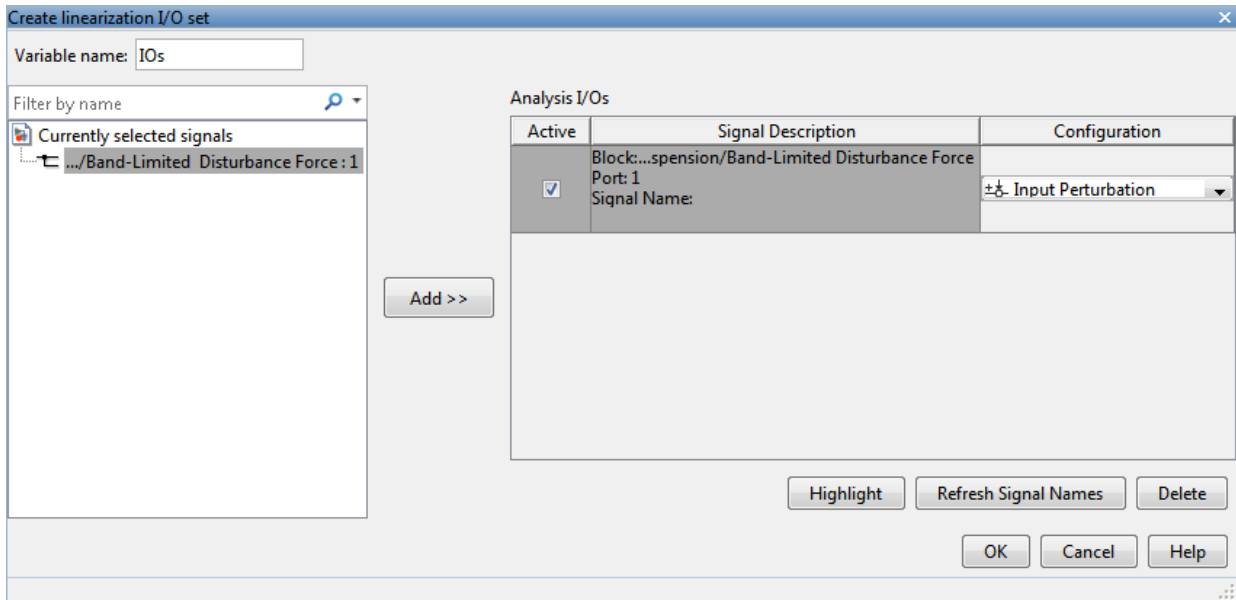
Click **OK**. A new variable, `UncVars`, appears in the **Response Optimization Tool Workspace**.

Specify Linear Analysis Input/Output Points

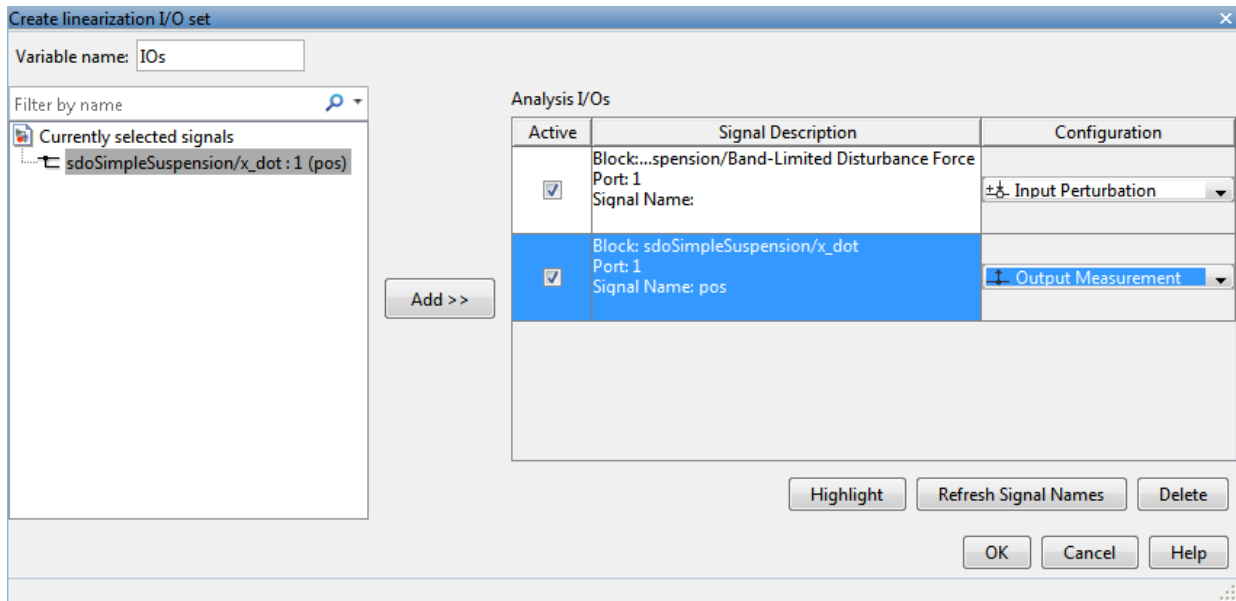
Specify the input/output points defining the linear system used to compute the bandwidth and damping ratio.

To specify the input/output points:

- In the **New** list, select **Linearization I/Os**.
- In the Simulink model, click the signal at the output of the `Band-Limited Disturbance Force` block. The **Create linearization I/O set** dialog box is updated and the chosen signal appears in it.
- In the **Create linearization I/O** dialog box, select the signal and click **Add**.
- In the **Configuration** list for the selected signal, choose **Input Perturbation** to specify it as an input signal.



- Similarly, add the **pos** signal from the Simulink model. Specify this signal as an output. In the **Configuration** list, select **Output Measurement**.



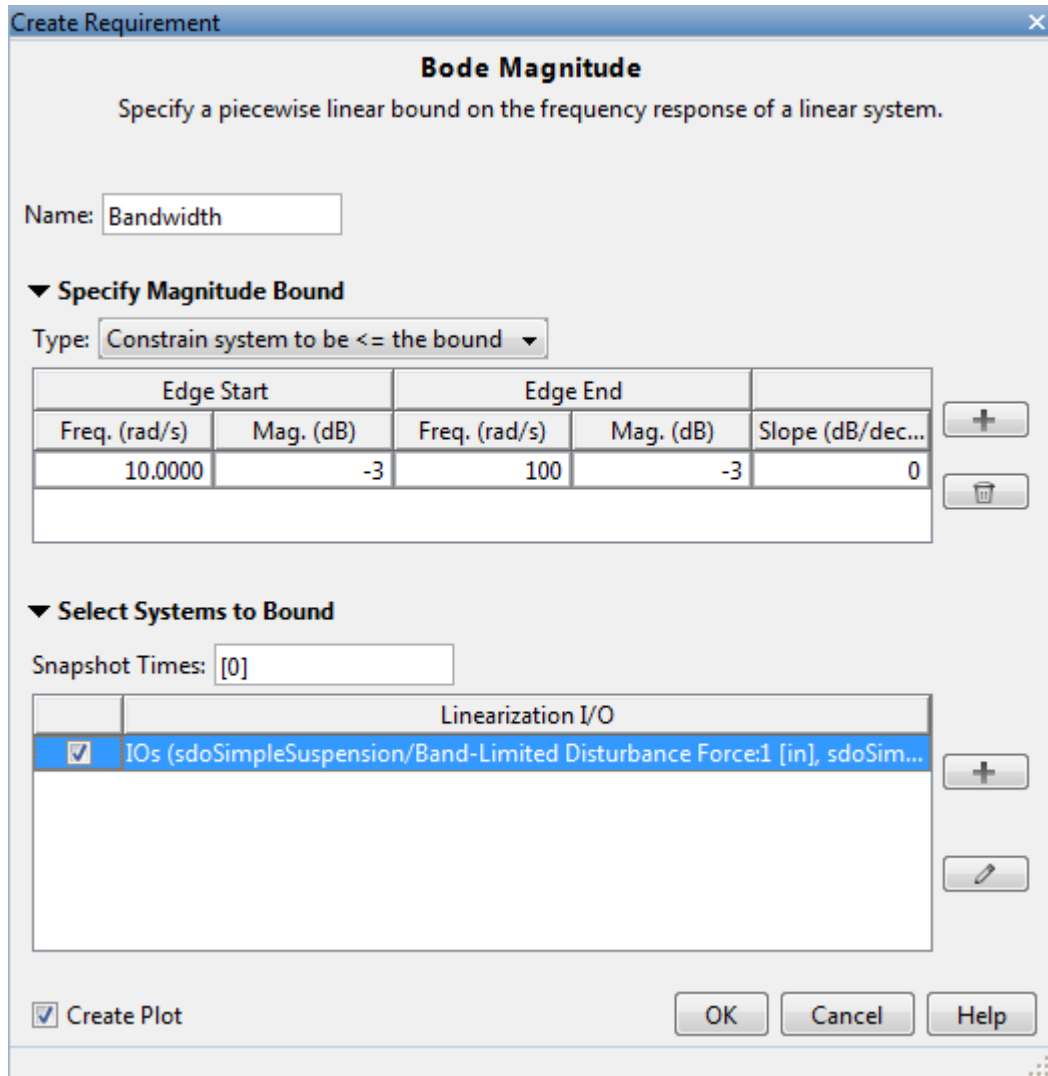
- Click **OK**. A new variable, `IOs`, appears in the **Response Optimization Tool Workspace**.

Add Bandwidth and Damping-Ratio Requirements

Tune the spring and damper values to satisfy bandwidth and damping ratio requirements.

To specify the bandwidth requirement:

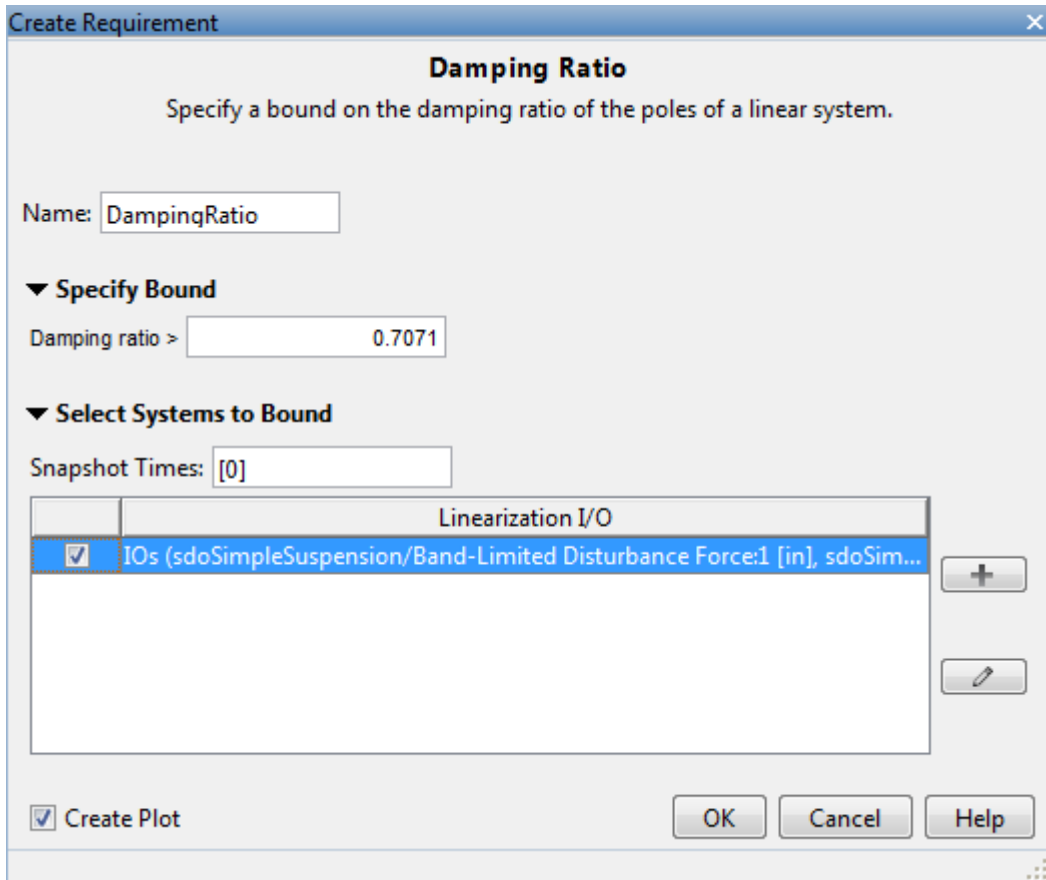
- Open a dialog to specify bounds on the Bode magnitude. In the **New** list, select **Bode Magnitude**.
- Specify the requirement name as `Bandwidth`.
- Specify the edge start frequency and magnitude as 10 rad/s and -3db, respectively.
- Specify the edge end frequency and magnitude as 100 rad/s and -3db, respectively.
- Specify the input/output set to which the requirement applies by clicking **Select Systems to Bound**. Select the `IOs` check box .



- Click **OK**. A new requirement, **Bandwidth**, appears in the **Response Optimization Tool Workspace** and a graphical view of the bandwidth requirement is automatically created.

To specify the damping ratio requirement:

- Open a dialog to specify bounds on the damping ratio. In the **New** list, select **Damping Ratio**.
- Specify the damping ratio bound value as 0.7071.
- Specify the input/output set to which the requirement applies by clicking **Select Systems to Bound**. Select the **IOs** check box .



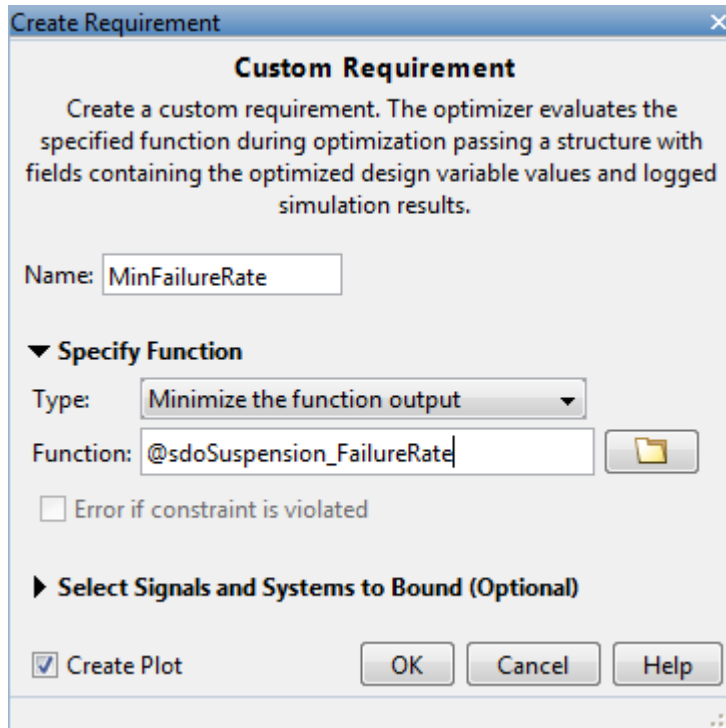
- Click **OK**. A new requirement, DampingRatio, appears in the **Response Optimization Tool Workspace** and a graphical view of the damping ratio requirement is automatically created.

Add a Reliability Requirement

Tune the spring and damper values to minimize the expected failure rate over a lifetime of 100e3 miles. The failure rate is computed using a Weibull distribution on the damping ratio of the system. As the damping ratio increases the failure rate is expected to increase.

Specify the reliability requirement as a custom requirement:

- Open a dialog box to specify the custom requirements. In the **New** list, select **Custom Requirement**.
- Specify the custom requirement name as `MinFailureRate`.
- In the **Specify Function** area, select **Minimize the function output** from the **Type** list.
- Specify the function as `@sdoSuspension_FailureRate`.



- Click **OK**. A new requirement, `MinFailureRate`, appears in the **Response Optimization Tool Workspace** and a graphical view of the custom requirement is automatically created.

The `@sdoSuspension_FailureRate` function returns expected failure rate for a lifetime of 100e3 miles.

```
type sdoSuspension_FailureRate

function pFailure = sdoSuspension_FailureRate(data)
%SDOSUSPENSION_FAILURERATE
%
% The sdoSuspension_FailureRate function is used to define a custom
% requirement that can be used in the graphical SDTOOL environment.
%
% The |data| input argument is a structure with fields containing the
% design variable values chosen by the optimizer.
%
% The |pFailure| return argument is the failure rate to be minimized by the
% SDTOOL optimization solver. The failure rate is given by a Weibull
% distribution that is a function of the mass, spring and damper values.
% The design minimizes the failure rate for a 100e3 mile lifetime.
%

% Copyright 2012 The MathWorks, Inc.

%Get the spring and damper design values
allVarNames = {data.DesignVars.Name};
idx          = strcmp(allVarNames,'k');
k            = data.DesignVars(idx).Value;
idx          = strcmp(allVarNames,'b');
b            = data.DesignVars(idx).Value;

%Get the nominal mass from the model workspace
wksp = get_param('sdoSimpleSuspension','ModelWorkspace');
m     = evalin(wksp,'m0');

%The expected failure rate is defined by the Weibull cumulative
%distribution function, 1-exp(-(x/l)^k), where k=3, l is a function of the
%mass, spring and damper values, and x the lifetime.
d     = b/2/sqrt(m*k);
pFailure = 1-exp(-(100e3*d/250e3)^3);
end
```


Optimize the Design

Before running the optimization be sure to have completed the earlier steps. Alternatively, you can load the `sdoSimpleSuspension_sdoSession` from the model workspace into the Response Optimization tool.

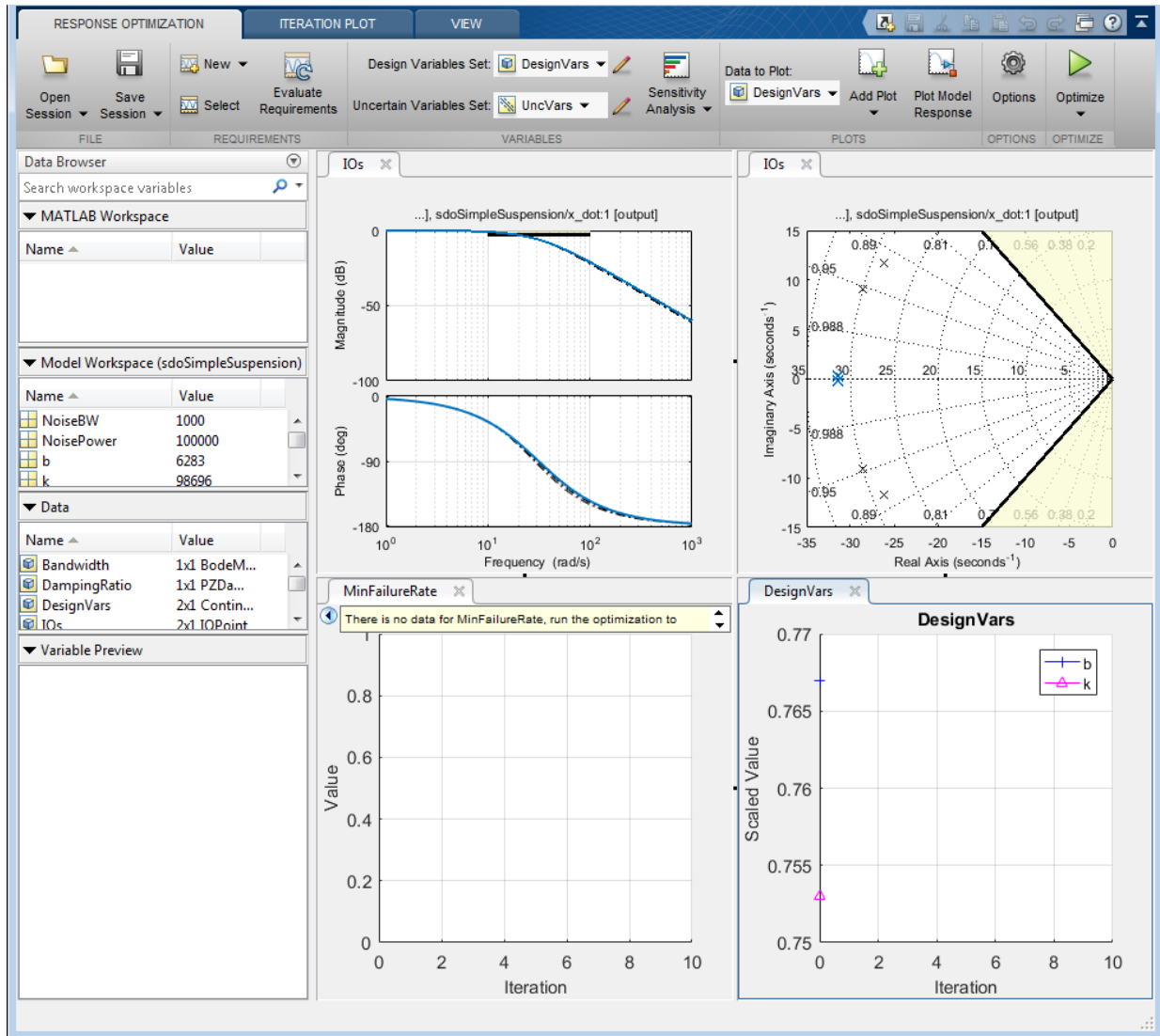
To save the initial design variable values and later compare them with the optimized values configure the optimization.

- Click **Options**.
- Select the `Save optimized variable values as new design variable set option`.

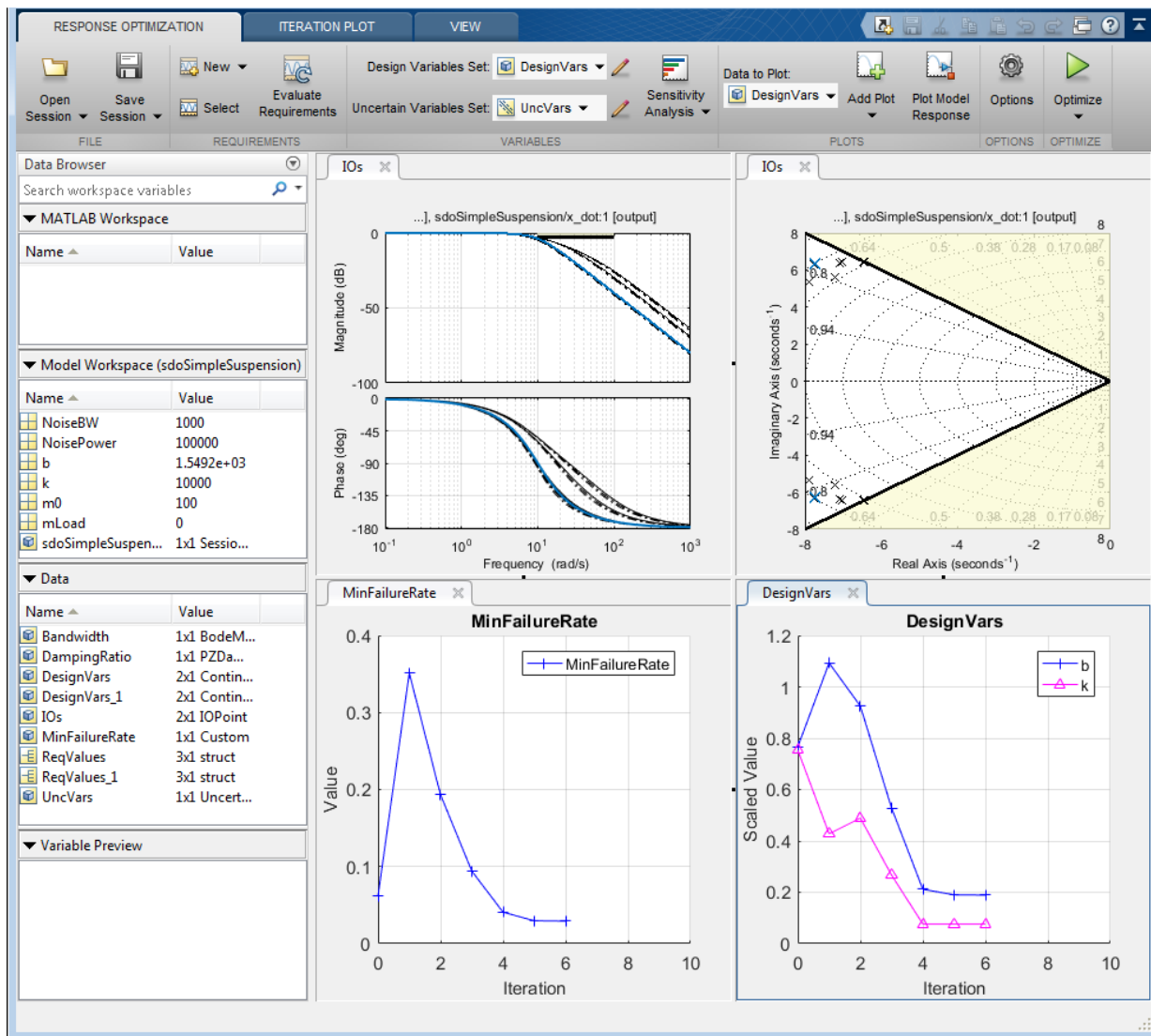
To study how the design variable values change during optimization:

- In the **Data to Plot** list, select **DesignVars**.
- In the **Add Plot** list, and select **Iteration Plot**.
- View the design variables in an appropriately scaled manner. Right-click on the `DesignVars` plot and select **Show scaled values**.

To evaluate the requirements at the initial design point, click **Evaluate Requirements** . The requirement plots are updated and a `ReqValues` variable is added to the **Response Optimization Tool Workspace**.



To optimize the design, click **Optimize**. The plots are updated during optimization. At the end of optimization, the optimal design values are written to the `DesignVars1` variable. The requirement values for the optimized design are written to the `ReqValues1` variable.



Iteration	F-count	MinFailureRate (Minimize)	Bandwidth (≤ 0)	DampingRatio (≥ 0)
0	5	0.0620	0.7642	0.2910
1	10	0.3513	-0.6699	0.4142
2	15	0.1933	-0.0462	0.4142
3	20	0.0942	0.0253	0.4142
4	24	0.0408	-0.5935	0.1192
5	28	0.0297	-0.2810	0.0047
6	32	0.0293	-0.2674	-1.5701e-16

Optimization started 29-Dec-2015 18:37:37

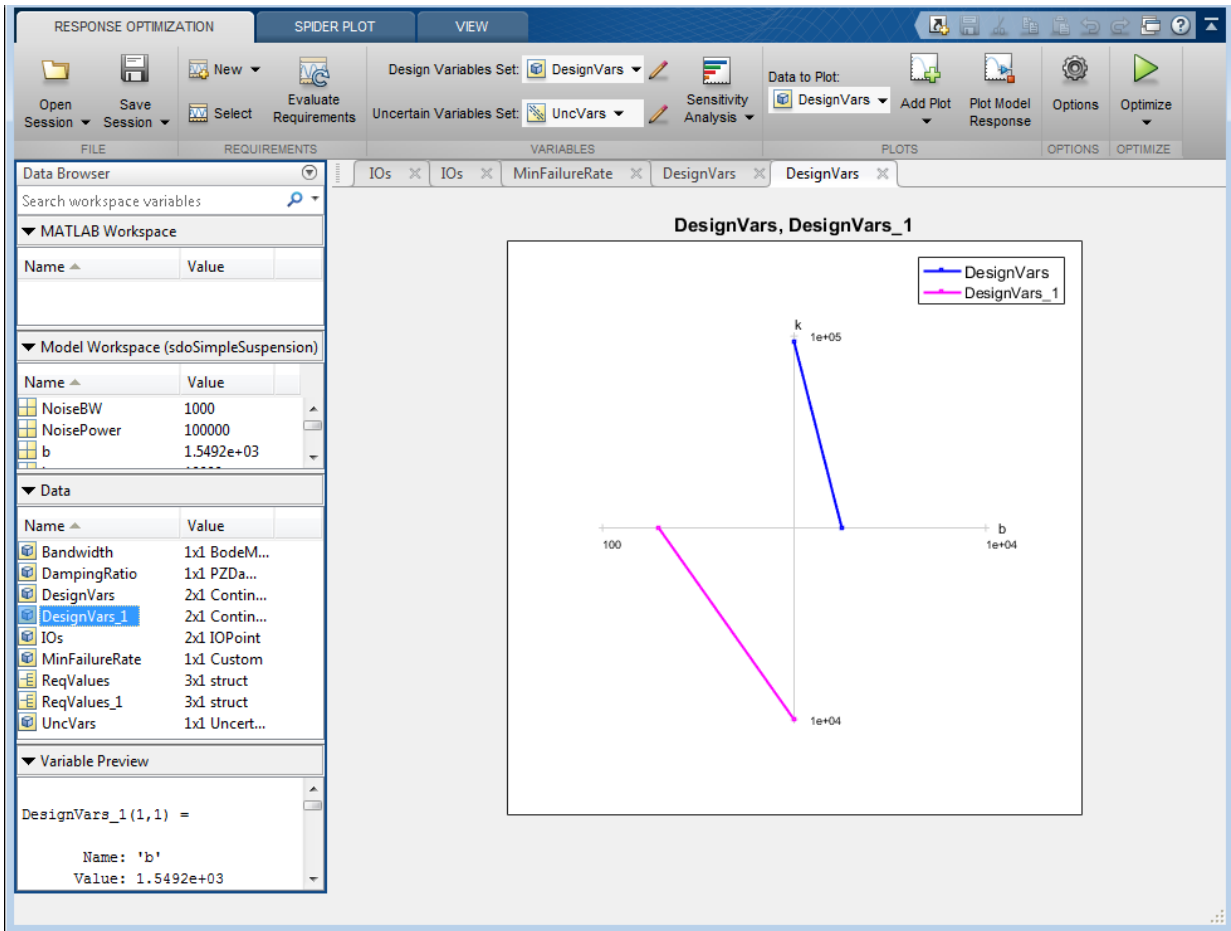
Optimization converged, 29-Dec-2015 18:39:53

Optimized variable values written to 'DesignVars_1' in the Design Optimization workspace

Analyze the Design

To compare design variables before and after optimization:

- In the **Data to Plot** list, select **DesignVars**.
- In the **Add Plot** list, select **Spider Plot**.
- To add the optimized design variables to the same plot, select **DesignVars1** in the **Response Optimization Tool Workspace** and drag it onto the Spider plot. Alternatively, in the **Data to Plot** list, select **DesignVars1**. Then, in the **Add Plot** list, select **Spider plot 1** from the **Add to Existing Plot** section.

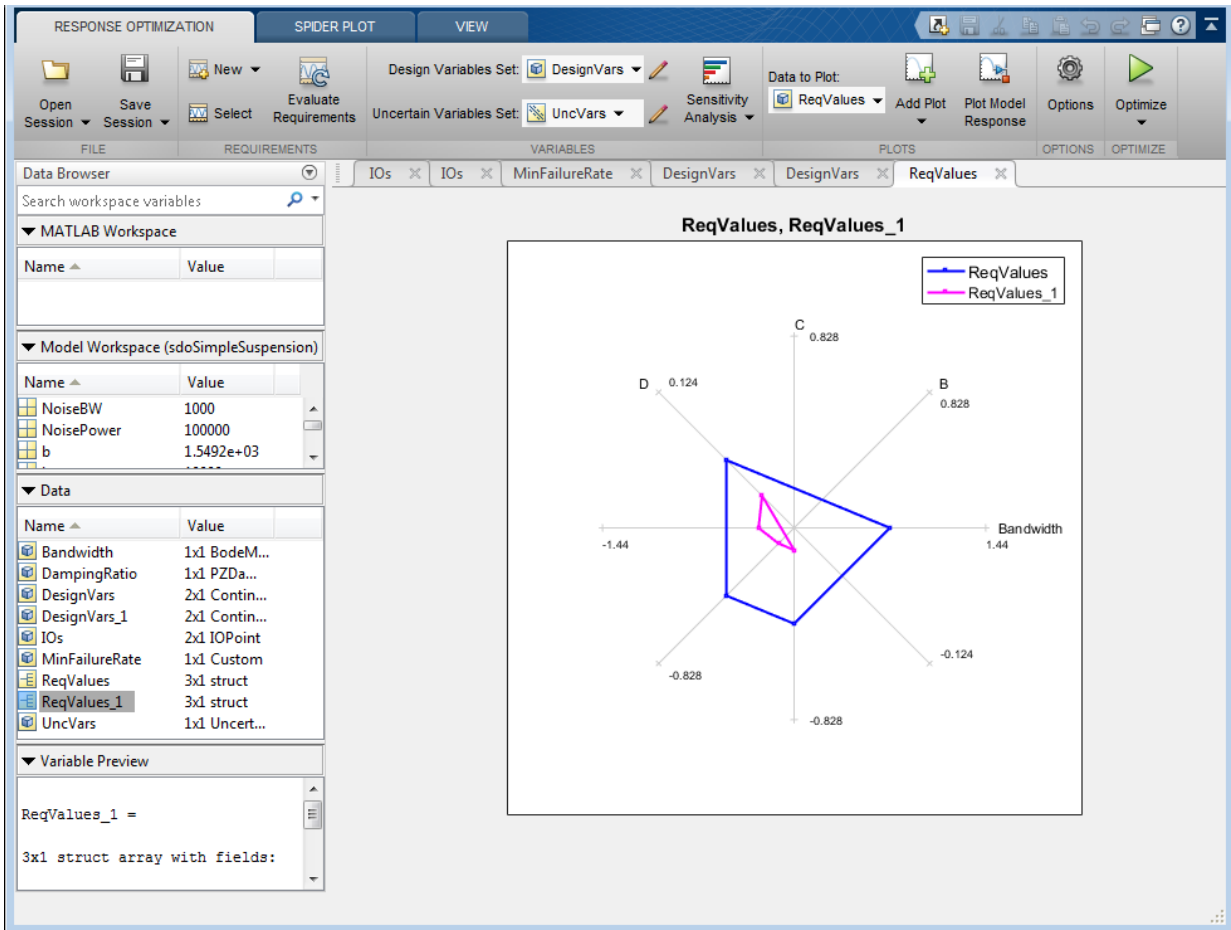


The plot shows that the optimizer reduced both the k and b values for the optimal design.

To compare requirements before and after optimization:

- In the **Data to Plot** list, select **ReqValues**.
- In the **Add Plot** list, select **Spider Plot**.
- To add the optimized requirement values to the same plot, select ReqValues1 in the **Response Optimization Tool Workspace** and drag it onto the Spider plot.

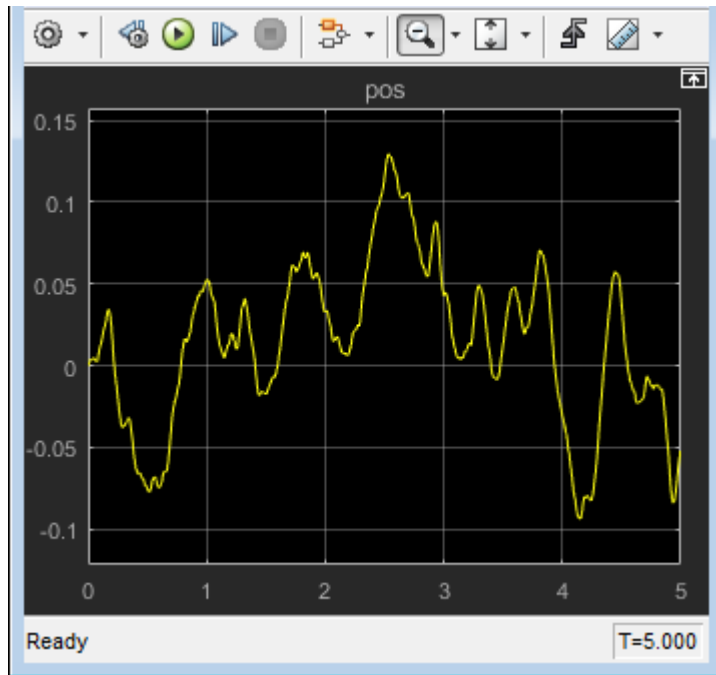
Alternatively, in the **Data to Plot** list, select ReqValues1. Then, in the **Add Plot** list, select **Spider plot 2** from the **Add to Existing Plot** section.



The plot shows that the optimal design has a lower failure rate (the MinFailureRate axis) and better satisfies the bandwidth requirement. The value plotted on the bandwidth axis is the difference between the bandwidth bound and the bandwidth value. The optimization satisfies the bound by keeping this value negative; a more negative value indicates better satisfaction of the bound.

The improved reliability and bandwidth are achieved by pushing the damping ratio closer to the damping ratio bound. The plot has two axes for the damping ratio requirement, one for each system pole, and the plotted values are the difference between the damping ratio bound and the damping ratio value. The optimization satisfies the bound by keeping this value negative.

Finally the simulated mass position is smoother than the initial position response (indication of a lower bandwidth as required) at the expense of larger position deflection.



Related Examples

To learn how to optimize the suspension design using the `sdo.optimize` command, see "Design Optimization to Meet Frequency-Domain Requirements (Code)".

```
% Close the model  
bdclose('sdoSimpleSuspension')
```

Specify Custom Signal Objective with Uncertain Variable (GUI)

This example shows how to specify a custom objective function for a model signal. You calculate the objective function value using a variable that models parameter uncertainty.

Competitive Population Dynamics Model

The Simulink model `sdoPopulation` models a simple two-organism ecology using the competitive Lotka-Volterra equations:

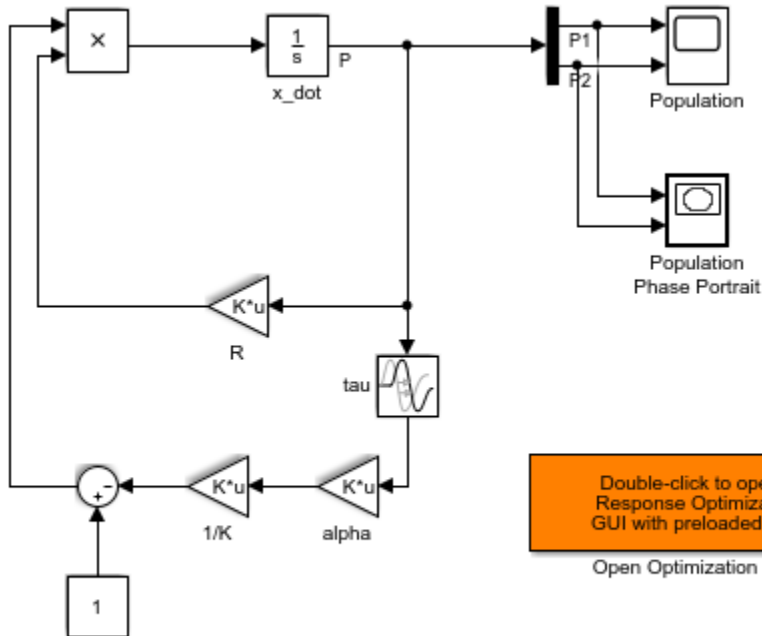
$$\frac{dP_1}{dt} = R_1 P_1 \left(1 - \frac{P_1(t - \tau_1) + \alpha P_2(t - \tau_2)}{K} \right)$$
$$\frac{dP_2}{dt} = R_2 P_2 \left(1 - \frac{P_2(t - \tau_2) + \alpha P_1(t - \tau_1)}{K} \right)$$

- P_n is the population size of the n-th organism.
- R_n is the inherent per capita growth rate of each organism.
- τ_n is the competitive delay for each organism.
- K is the carrying capacity of the organism environment.
- α is the proximity of the two populations and how strongly they affect each other.

The model uses normalized units.

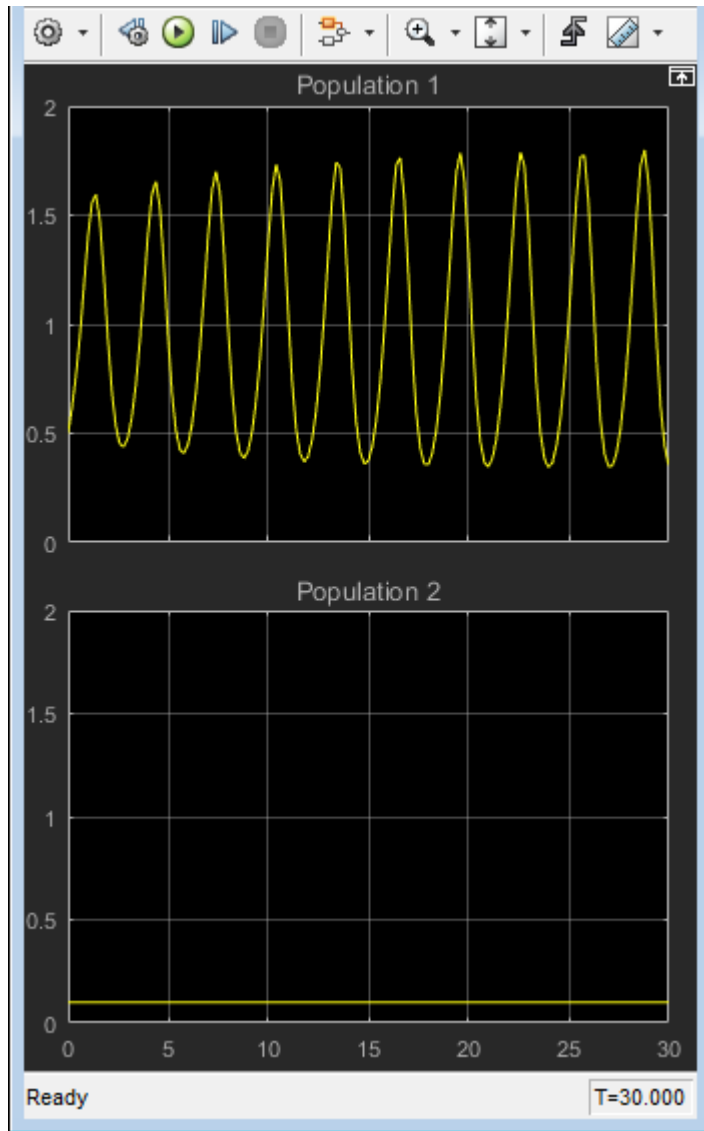
Open the model.

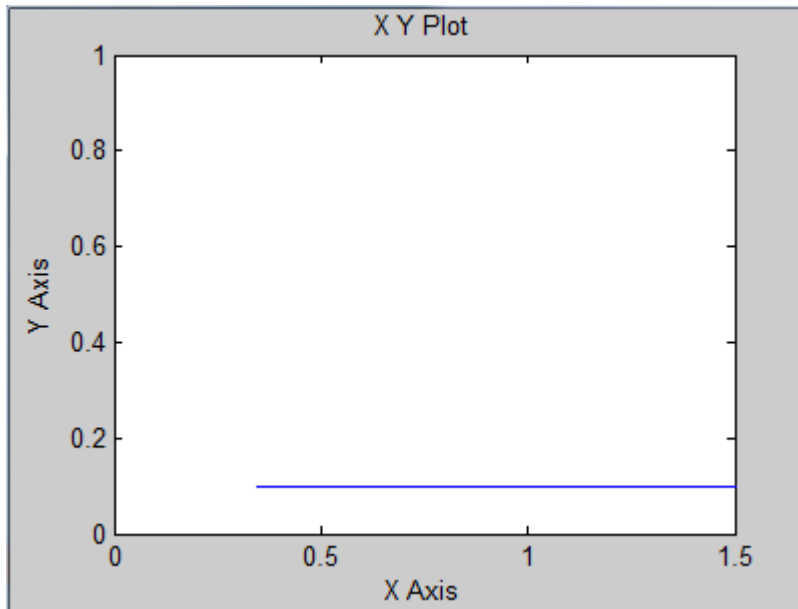
```
open_system('sdoPopulation')
```

Double-click to open a Response Optimization GUI with preloaded data

Open Optimization Tool.





The two-dimensional signal, P , models the population sizes for P_1 (first element) and P_2 (second element). The model is initially configured with one organism, P_1 , dominating the ecology. The Population scope shows the P_1 population oscillating between high and low values, while P_2 is constant at 0.1. The Population Phase Portrait block shows the population sizes of the two organisms in relation to each other.

Population Stabilization Design Problem

Tune the R_2 , τ_2 , and α values to meet the following design requirements:

- Minimize the population range, that is, the maximum difference between P_1 and P_2 .
- Stabilize P_1 and P_2 , that is, ensure that neither organism population dies off or grows extremely large.

You must tune the parameters for different values of the carrying capacity, K . This ensures robustness to environment carrying-capacity uncertainty.

Open Response Optimization Tool

Double-click the Open Optimization Tool block in the model to open a pre configured Response Optimization tool session. The session specifies the following variables:

- DesignVars - Design variables set for the R_2 , τ_2 , and α model parameters.
- K_unc - Uncertain parameter modeling the carrying capacity of the organism environment (K). K_unc specifies the nominal value and two sample values.
- P1 and P2 - Logged signals representing the populations of the two organisms.

Specify Custom Signal Objective Function

Specify a custom requirement to minimize the maximum difference between the two population sizes. Apply this requirement to the P1 and P2 model signals.

- 1 Open the Create Requirement dialog box. In the **New** list, select **Custom Requirement**.
- 2 Specify the following in the Create Requirement dialog box:
 - **Name** - Enter PopulationRange.
 - **Type** - Select **Minimize the function output** from the list.
 - **Function** - Enter @sdoPopulation_PopRange. For more information about this function, see **Custom Signal Objective Function Details**.
 - **Select Signals and Systems to Bound (Optional)** - Select the P1 and P2 check boxes.

Create Requirement


Custom Requirement

Create a custom requirement. The optimizer evaluates the specified function during optimization passing a structure with fields containing the optimized design variable values and logged simulation results.

Name:



▼ **Specify Function**

Type:



Function: 

Error if constraint is violated

▼ **Select Signals and Systems to Bound (Optional)**

	Signal	
<input checked="" type="checkbox"/>	P1 (sdoPopulation/Demux:1)	
<input checked="" type="checkbox"/>	P2 (sdoPopulation/Demux:2)	

Snapshot Times:

	Linearization I/O	
	Create Linearization I/Os defining a system so that it can be used in requirements.	
		

Create Plot

3. Click **OK**.

A new variable, `PopulationRange`, appears in the **Response Optimization Tool Workspace**.

Custom Signal Objective Function Details

`PopulationRange` uses the `sdoPopulation_PopRange` function. This function computes the maximum difference between the populations, across different environment carrying capacity values. By minimizing this value, you can achieve both design goals. The function is called by the optimizer at each iteration step.

To view the function, type `edit sdoPopulation_PopRange`. The following discusses details of this function.

Input/Output

The function accepts `data`, a structure with the following fields:

- `DesignVars` - Current iteration values of R_2 , τ_2 , and α .
- `Nominal` - Logged signal data, obtained by simulating the model using parameter values specified by `data.DesignVars` and nominal values for all other parameters. The `Nominal` field is itself a structure with fields for each logged signal. The field names are the logged signal names. The custom requirement uses the logged signals, `P1` and `P2`. Therefore, `data.Nominal.P1` and `data.Nominal.P2` are timeseries objects corresponding to `P1` and `P2`.
- `Uncertain` - Logged signal data, obtained by simulating the model using the sample values of the uncertain variable `K_unc`. The `Uncertain` field is a vector of `N` structures, where `N` is the number of sample values specified for `K_unc`. Each element of this vector is similar to `data.Nominal` and contains simulation results obtained from a corresponding sample value specified for `K_unc`.

The function returns the maximum difference between the population sizes across different carrying capacities. The following code snippet in the function performs this action:

```
val = max(maxP(1)-minP(2),maxP(2)-minP(1));
```

Data Time Range

When computing the design goals, discard the initial population growth data to eliminate biases from the initial-condition. The following code snippet in the function performs this action:

```
%Get the population data
tMin = 5; %Ignore signal values prior to this time
iTime = data.Nominal.P1.Time > tMin;
sigData = [data.Nominal.P1.Data(iTime), data.Nominal.P2.Data(iTime)];
```

iTime represents the time interval of interest, and the columns of sigData contain P1 and P2 data for this interval.

Optimization for Different Values of Carrying Capacity

The function includes the effects of varying the carrying capacity by iterating through the elements of data.Uncertain. The following code snippet in the function performs this action:

```
...
for ct=1:numel(data.Uncertain)
    iTime = data.Uncertain(ct).P1.Time > tMin;
    sigData = [data.Uncertain(ct).P1.Data(iTime), data.Uncertain(ct).P2.Data(iTime)];

    maxP = max([maxP; max(sigData)]); %Update maximum if new signals are bigger
    minP = min([minP; min(sigData)]); %Update minimum if new signals are smaller
end
...
```

The maximum and minimal populations are obtained across all the simulations contained in data.Uncertain.

Optimize Design

Click **Optimize**.

The optimization converges after a number of iterations.

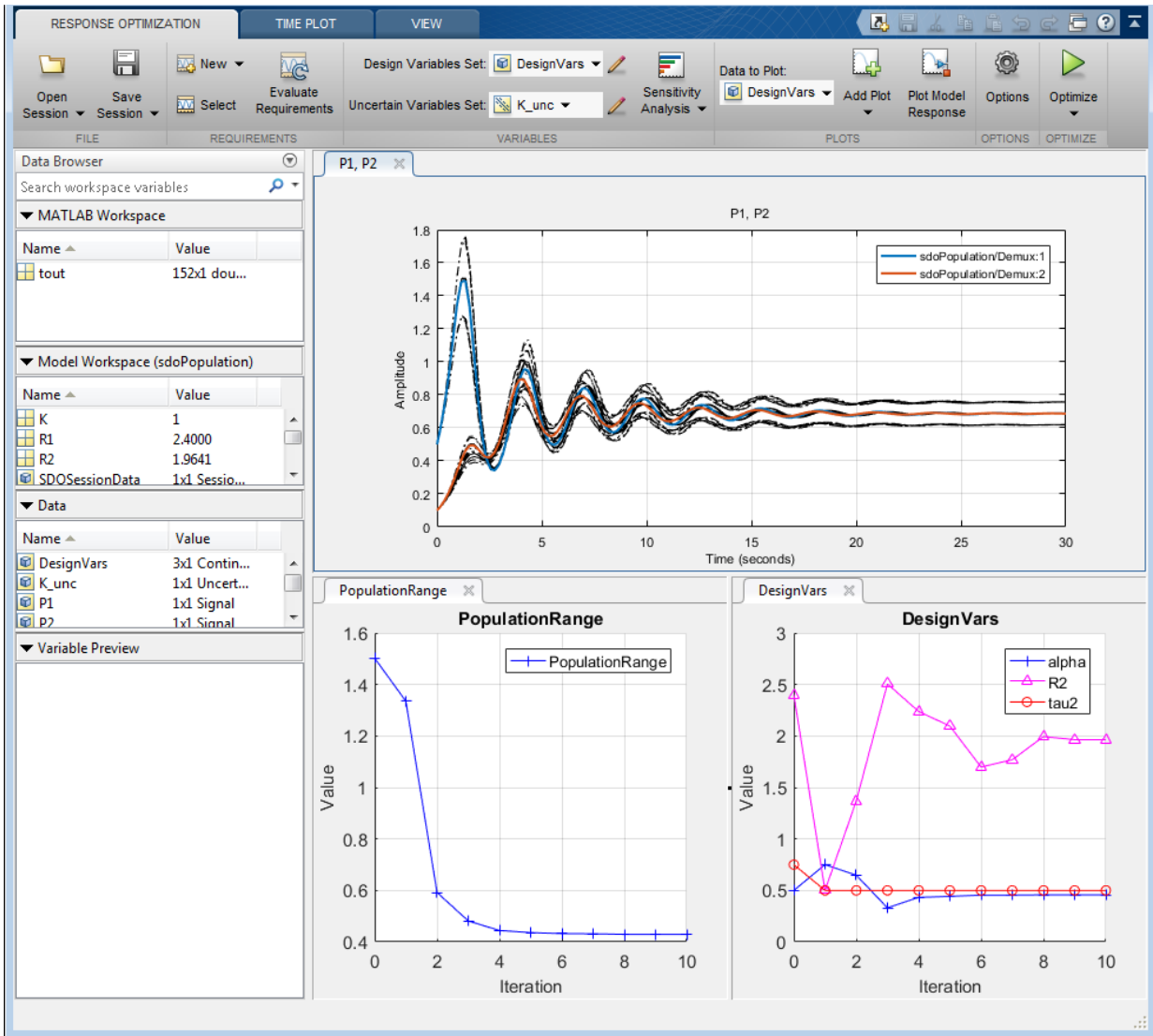
Iteration	F-count	PopulationRange (min)
0	8	1.5024
1	13	1.3350
2	20	0.5918
3	29	0.4814
4	36	0.4443
5	43	0.4361
6	51	0.4322
7	58	0.4316
8	65	0.4294
9	77	0.4294
10	88	0.4294

Optimization started 09-Oct-2012 12:01:35

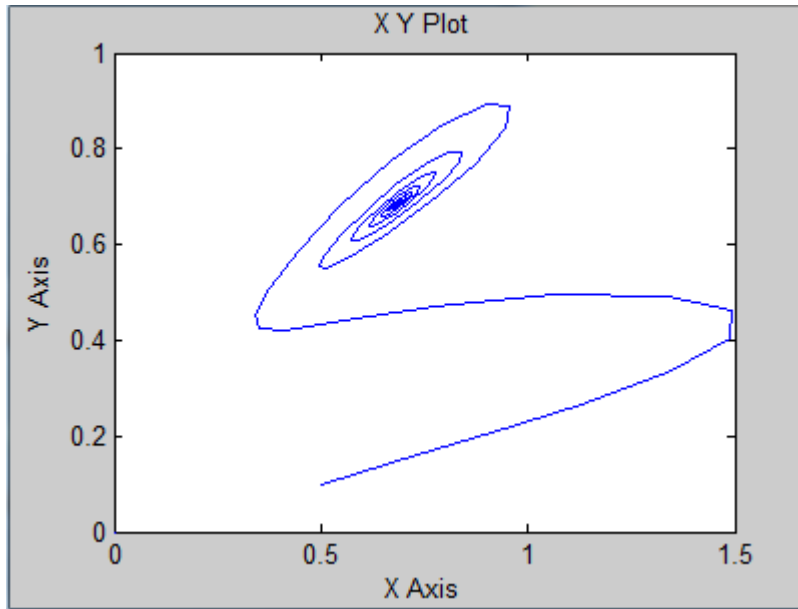
Optimization converged, 09-Oct-2012 12:03:32

Optimized variable values written to 'DesignVars' in the Design Optimization workspace

The P_1, P_2 plot shows the population dynamics, with the first organism population in blue and the second organism population in red. The dotted lines indicate the population dynamics for different environment capacity values. The $PopulationRange$ plot shows that the maximum difference between the two organism populations reduces over time.



The Population Phase Portrait block shows the populations initially varying, but they eventually converge to stable population sizes.



```
% Close the model  
bdclose('sdoPopulation')
```

Design Optimization with Uncertain Variables (Code)

This example shows how to optimize a design when there are uncertain variables. You optimize the dimensions of a Continuously Stirred Tank Reactor (CSTR) to minimize product concentration variation and production cost in case of varying, or uncertain, feed stock.

Continuously Stirred Tank Reactor (CSTR) Model

Continuously Stirred Tank Reactors (CSTRs) are common in the process industry. The Simulink model, `sdCSTR`, models a jacketed diabatic (i.e., non-adiabatic) tank reactor described in [1]. The CSTR is assumed to be perfectly mixed, with a single first-order exothermic and irreversible reaction, $A \rightarrow B$. A , the reactant, is converted to B , the product.

In this example, you use the following two-state CSTR model, which uses basic accounting and energy conservation principles:

$$\frac{dC_A}{dt} = \frac{F}{A * h} (C_{feed} - C_A) - r * C_A$$

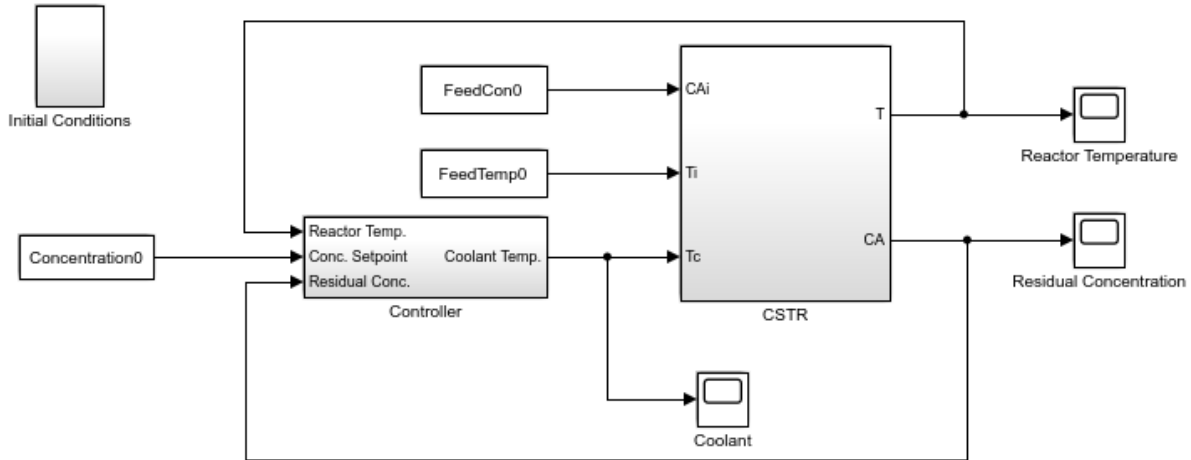
$$\frac{dT}{dt} = \frac{F}{A * h} (T_{feed} - T) - \frac{H}{c_p \rho} r - \frac{U}{c_p * \rho * h} (T - T_{cool})$$

$$r = k_0 * e^{-\frac{E}{RT}}$$

- C_A , and C_{feed} - Concentrations of A in the CSTR and in the feed [kgmol/m³]
- T , T_{feed} , and T_{cool} - CSTR, feed, and coolant temperatures [K]
- F and ρ - Volumetric flow rate [m³/h] and the density of the material in the CSTR [1/m³]
- h and A - Height [m] and heated cross-sectional area [m²] of the CSTR.
- k_0 - Pre-exponential non-thermal factor for reaction $A \rightarrow B$ [1/h]
- E and H - Activation energy and heat of reaction for $A \rightarrow B$ [kcal/kgmol]
- R - Boltzmann's gas constant [kcal/(kgmol * K)]
- c_p and U - Heat capacity [kcal/K] and heat transfer coefficients [kcal/(m² * K * h)]

Open the Simulink model.

```
open_system('sdoCSTR');
```



Copyright 2012-2015 The MathWorks, Inc.

The model includes a cascaded PID controller in the Controller subsystem. The controller regulates the reactor temperature, T , and reactor residual concentration, C_A .

CSTR Design Problem

Assume that the CSTR is cylindrical, with the coolant applied to the base of the cylinder. Tune the CSTR cross-sectional area, A , and CSTR height, h , to meet the following design goals:

- Minimize the variation in residual concentration, C_A . Variations in the residual concentration negatively affect the quality of the CSTR product. Minimizing the variations also improves CSTR profit.
- Minimize the mean coolant temperature T_{cool} . Heating or cooling the jacket coolant temperature is expensive. Minimizing the mean coolant temperature improves CSTR profit.

The design must allow for variations in the quality of supply feed concentration, C_{feed} , and feed temperature, T_{feed} . The CSTR is fed with feed from different suppliers. The

quality of the feed differs from supplier to supplier and also varies within each supply batch.

Specify Design Variables

Select the following model parameters as design variables for optimization:

- Cylinder cross-sectional area A
- Cylinder height h

```
p = sdo.getParameterFromModel('sdoCSTR', {'A', 'h'});
```

Limit the cross-sectional area to a range of [1 2] m².

```
p(1).Minimum = 1;
p(1).Maximum = 2;
```

Limit the height to a range of [1 3] m.

```
p(2).Minimum = 1;
p(2).Maximum = 3;
```

Specify Uncertain Variables

Select the feed concentration and feed temperature as uncertain variables. You evaluate the design using different values of feed temperature and concentration.

```
pUnc = sdo.getParameterFromModel('sdoCSTR', {'FeedCon0', 'FeedTemp0'});
```

Create a parameter space for the uncertain variables. Use normal distributions for both variables. Specify the mean as the current parameter value. Specify a variance of 5% of the mean for the feed concentration and 1% of the mean for the temperature.

```
uSpace = sdo.ParameterSpace(pUnc);
uSpace = setDistribution(uSpace, 'FeedCon0', makedist('normal', pUnc(1).Value, 0.05*pUnc(1).Value));
uSpace = setDistribution(uSpace, 'FeedTemp0', makedist('normal', pUnc(2).Value, 0.01*pUnc(2).Value));
```

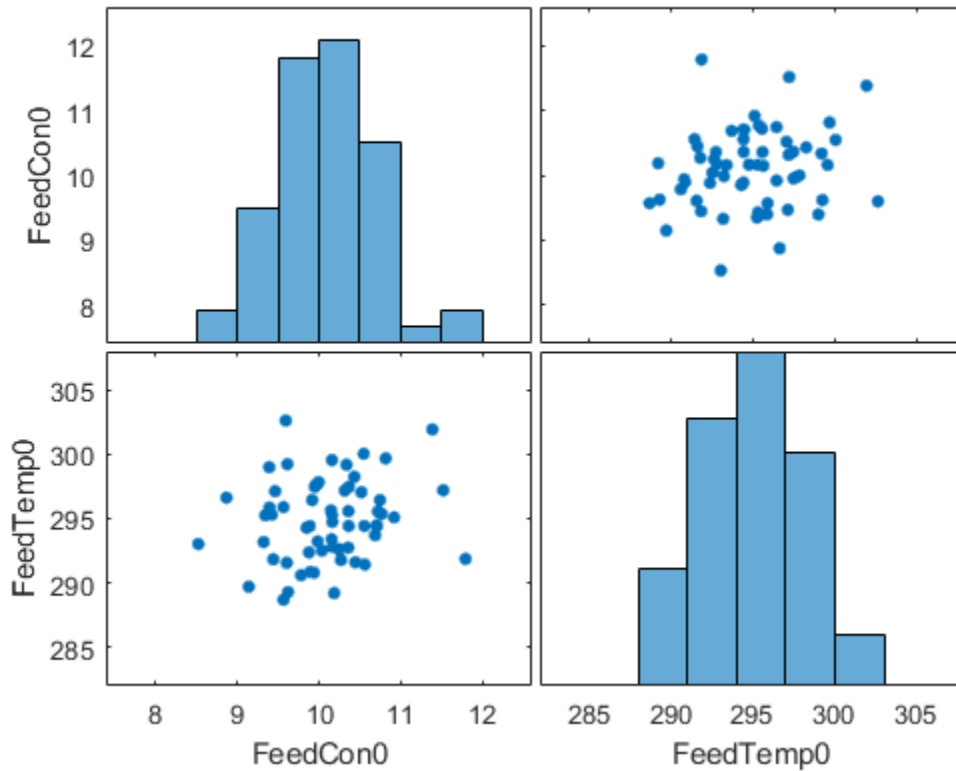
The feed concentration is inversely correlated with the feed temperature. Add this information to the parameter space.

```
%uSpace.RankCorrelation = [1 -0.6; -0.6 1];
```

The rank correlation matrix has a row and column for each parameter with the (i,j) entry specifying the correlation between the i and j parameters.

Sample the parameter space. The scatter plot shows the correlation between concentration and temperature.

```
rng('default'); %For reproducibility
uSmpl = sdo.sample(uSpace, 60);
sdo.scatterPlot(uSmpl)
```



Ideally you want to evaluate the design for every combination of points in the design and uncertain spaces, which implies $30 \times 60 = 1800$ simulations. Each simulation takes around 0.5 sec. You can use parallel computing to speed up the evaluation. For this example you instead only use the samples that have maximum & minimum concentration and temperature values, reducing the evaluation time to around 1 min.

```
[~,iminC] = min(uSmpl.FeedCon0);
[~,imaxC] = max(uSmpl.FeedCon0);
[~,iminT] = min(uSmpl.FeedTemp0);
[~,imaxT] = max(uSmpl.FeedTemp0);
uSmpl = uSmpl(unique([iminC,imaxC,iminT,imaxT]),:);
```

Specify Design Requirements

The design requirements require logging model signals. During optimization, the model is simulated using the current value of the design variables. Logged signals are used to evaluate the design requirements.

Log the following signals:

- CSTR concentration, available at the second output port of the `sdoCSTR/CSTR` block

```
Conc = Simulink.SimulationData.SignalLoggingInfo;
Conc.BlockPath      = 'sdoCSTR/CSTR';
Conc.OutputPortIndex = 2;
Conc.LoggingInfo.NameMode = 1;
Conc.LoggingInfo.LoggingName = 'Concentration';
```

- Coolant temperature, available at the first output of the `sdoCSTR/Controller` block

```
Coolant = Simulink.SimulationData.SignalLoggingInfo;
Coolant.BlockPath      = 'sdoCSTR/Controller';
Coolant.OutputPortIndex = 1;
Coolant.LoggingInfo.NameMode = 1;
Coolant.LoggingInfo.LoggingName = 'Coolant';
```

Create and configure a simulation test object to log the required signals.

```
simulator = sdo.SimulationTest('sdoCSTR');
simulator.LoggingInfo.Signals = [Conc,Coolant];
```

Create Objective/Constraint Function

Create a function to evaluate the CSTR design. This function is called at each optimization iteration.

Use an anonymous function with one argument that calls the `sdoCSTR_design` function.

```
evalDesign = @(p) sdoCSTR_design(p,simulator,pUnc,uSmpl);
```

The `evalDesign` function:

- Has one input argument that specifies the CSTR dimensions
- Returns the optimization objective value

The `sdoCSTR_design` function uses a `for` loop that iterates through the sample values specified for the feed concentration. Within the loop, the function:

- Simulates the model using the current iterate, feed concentration, and feed temperature values
- Calculates the residual concentration variation and coolant temperature costs

To view the objective function, type `edit sdoCSTR_design`.

Evaluate Initial Design

Call the `evalDesign` function with the initial CSTR dimensions.

```
dInit = evalDesign(p)
```

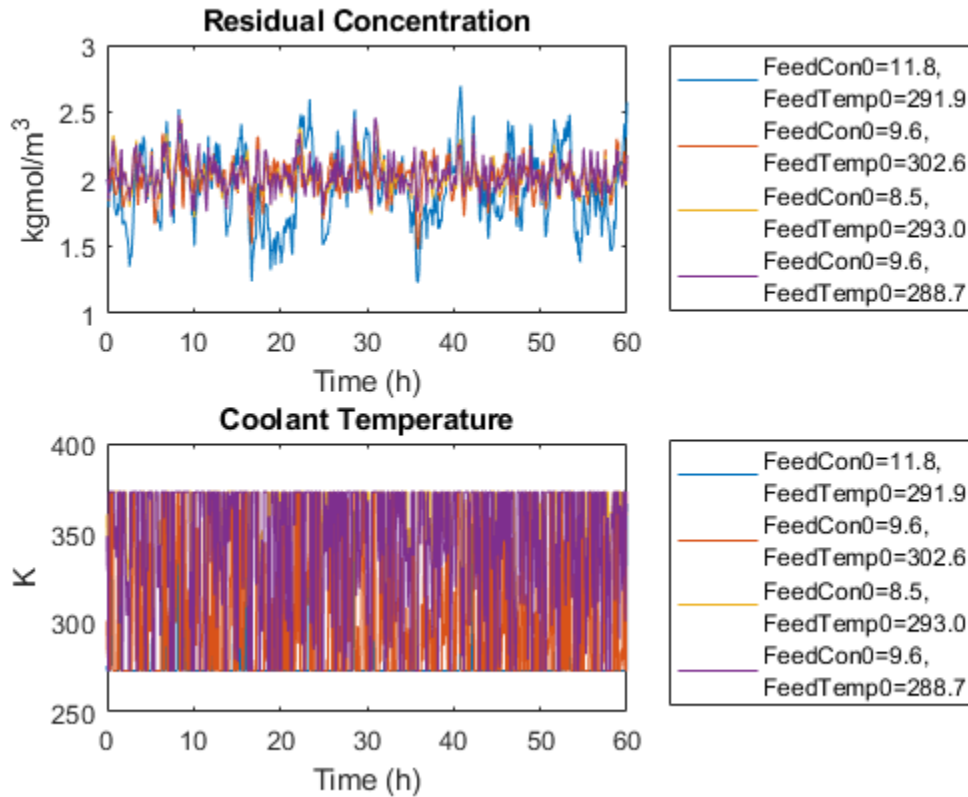
```
dInit =
```

```
    struct with fields:
```

```
        F: 11.3358  
    costConc: 6.4387  
    costCoolant: 4.8971
```

Plot the model response for the initial design. Simulate the model using the sample feed concentration values. The plot shows the variation in the residual concentration and coolant temperature.

```
sdoCSTR_plotModelResponse(p, simulator, pUnc, uSmpl);
```

The `sdoCSTR_plotModelResponse` function plots the model response. To view this function, type `edit sdoCSTR_plotModelResponse`.

Optimize Design

Pass the objective function and initial CSTR dimensions to `sdo.optimize`.

```
pOpt = sdo.optimize(evalDesign,p)
```

```
Optimization started 15-Aug-2017 01:29:58
```

Iter	F-count	f(x)	max constraint	Step-size	First-order optimality
------	---------	------	-------------------	-----------	---------------------------

3 Response Optimization

0	4	5.17935	0		
1	8	3.81245	0	2.01	7.81
2	12	2.65827	0	0.574	3.06
3	16	2.53697	0	0.162	0.267
4	20	2.52022	0	0.0154	0.249
5	24	2.48886	0	0.072	0.195
6	28	2.48857	0	0.0316	0.148
7	32	2.47868	0	0.0136	0.204
8	38	2.4757	0	0.0462	0.801
9	44	2.47295	0	0.018	0.378
10	52	2.46467	0	0.00387	0.275
11	59	2.46467	0	0.000856	0.275

Local minimum possible. Constraints satisfied.

fmincon stopped because the size of the current step is less than the selected value of the step size tolerance and constraints are satisfied to within the selected value of the constraint tolerance.

pOpt(1,1) =

```
Name: 'A'  
Value: 2  
Minimum: 1  
Maximum: 2  
Free: 1  
Scale: 0.5000  
Info: [1x1 struct]
```

pOpt(2,1) =

```
Name: 'h'  
Value: 2.2185  
Minimum: 1  
Maximum: 3  
Free: 1  
Scale: 2  
Info: [1x1 struct]
```

2x1 param.Continuous

Evaluate Optimized Design

Call the `evalDesign` function with the optimized CSTR dimensions.

```
dFinal = evalDesign(pOpt)
```

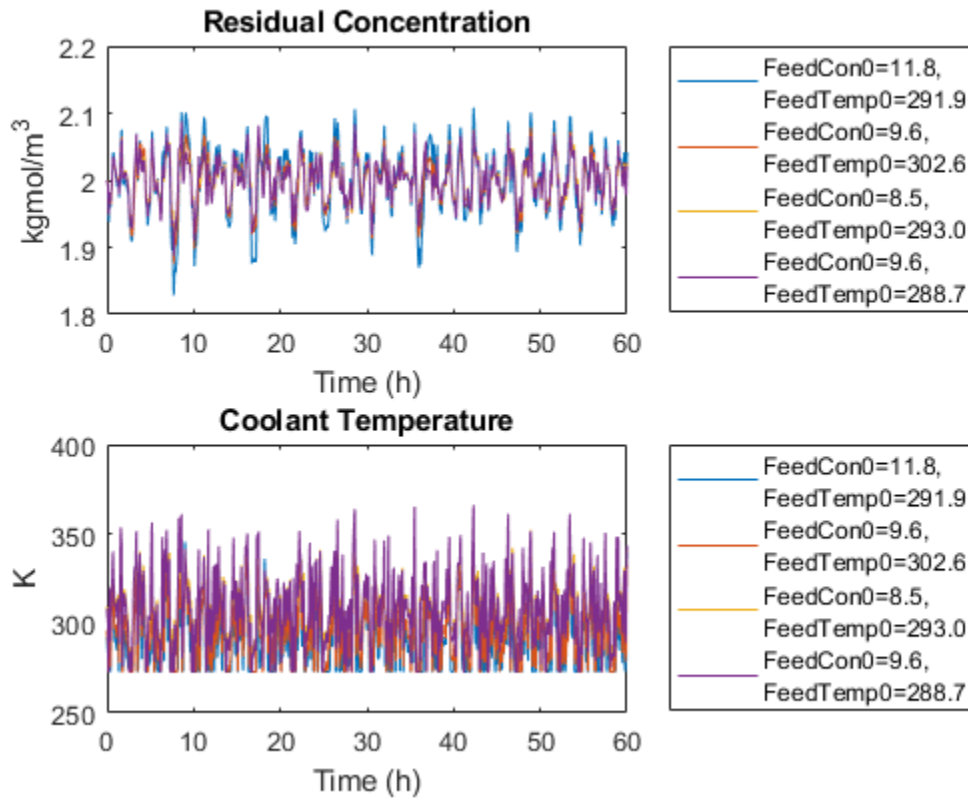
```
dFinal =
```

```
    struct with fields:
```

```
        F: 2.4647  
    costConc: 1.4416  
    costCoolant: 1.0231
```

Plot the model response for the optimized design. Simulate the model using the sample feed concentration values. The optimized design reduces the residual concentration variation and average coolant temperature for different feed stocks.

```
sdoCSTR_plotModelResponse(pOpt, simulator, pUnc, uSmpl);
```



Related Examples

To learn how to use sensitivity analysis to explore the CSTR design space and select an initial design for optimization, see "Design Optimization with Uncertain Variables (Code)".

References

[1] Bequette, B.W. *Process Dynamics: Modeling, Analysis and Simulation*. 1st ed. Upper Saddle River, NJ: Prentice Hall, 1998.

```
% Close the model
bdclose('sdoCSTR')
```

Generate MATLAB Code for Design Optimization Problems (GUI)

This example shows how to automatically generate a MATLAB function to solve a Design Optimization problem. You use the Response Optimization tool to define an optimization problem for a hydraulic cylinder design and generate MATLAB code to solve this optimization problem.

Hydraulic Cylinder Design Problem

The "Design Optimization to Meet a Custom Objective Using the Response Optimization Tool" example shows how to use the Response Optimization tool to optimize a cylinder design. In this example we load a pre-configured Response Optimization tool session based on that example.

```
load sdoHydraulicCylinder_sdoSession
```

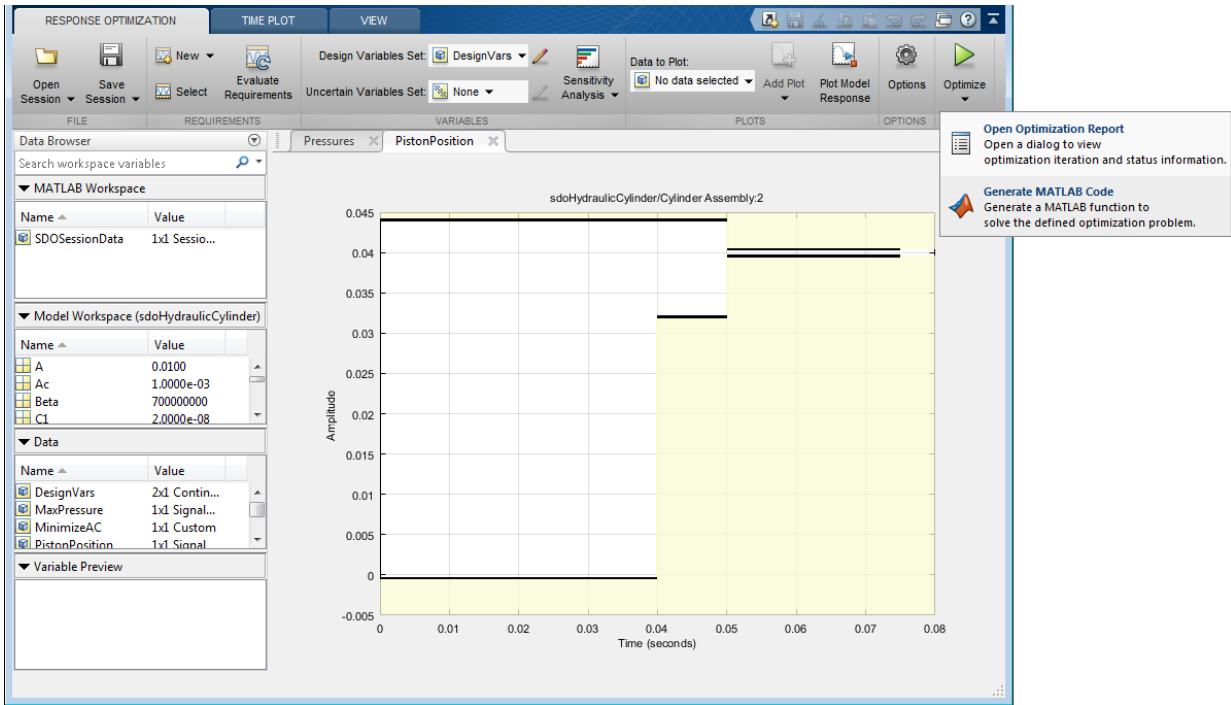
Use the following command to open the Response Optimization tool

```
sdoTool(SDOSessionData)
```

Generate MATLAB Code

From the **Optimize** list, select **Generate MATLAB Code**.

3 Response Optimization



The generated code is added to the MATLAB editor as an unsaved MATLAB function.

```

1  function [Optimized_DesignVars, Info] = sdo_sdoHydraulicCylinder(DesignVars)
2  %SDO_SDOHYDRAULICCYLINDER
3  %
4  % Solve a design optimization problem for the sdoHydraulicCylinder model.
5  %
6  % The function returns optimized parameter values, Optimized_DesignVars,
7  % and optimization termination information, Info.
8  %
9  % The, DesignVars, input argument defines the model parameters to optimize,
10 % if omitted the parameters specified in the function body are optimized.
11 %
12 % Modify the function to include or exclude new design requirements or
13 % change the optimization options.
14 %
15 % Auto-generated by SDOTOOL on 15-Mar-2012 13:14:45.
16 %
17
18 %% Open the model.
19 open_system('sdoHydraulicCylinder')
20
21 %% Specify Design Variables
22 %
23 % Specify model parameters as design variables to optimize.
24 if nargin < 1 || isempty(DesignVars)
25     DesignVars = sdo.getParameterFromModel('sdoHydraulicCylinder',{'Ac','K'});
26     DesignVars(1).Minimum = 0.0003;
27     DesignVars(1).Maximum = 0.0013;
28     DesignVars(1).Scale = 0.001;

```

Examine the generated code. Significant code portions are:

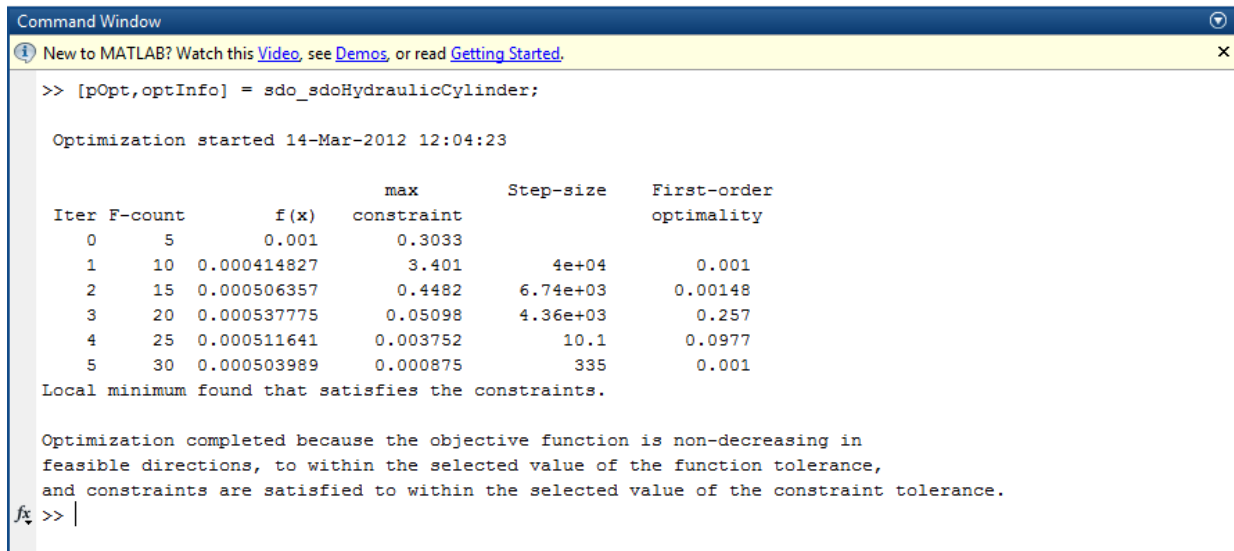
- **Specify Design Variables** - Definition of the model parameters being optimized.
- **Specify Design Requirements** - Definition of the design requirements.
- **Create Optimization Objective Function** - Creation of an anonymous function that calls the subfunction `sdoHydraulicCylinder_optFcn`, which evaluates the cylinder design. `sdo.optimize` calls the anonymous function at each iteration.
- **Evaluate custom parameter requirement functions** - Evaluates the custom requirement, `MinimizeAC`, that uses the `sdoHydraulicCylinder_customObjective` function.

- **Optimize the Design** - Optimization using the `sdo.optimize` command.

Select **Save** from the MATLAB editor to save the generated function.

Run Generated Code

Run the generated function.



```
Command Window
New to MATLAB? Watch this Video, see Demos, or read Getting Started.
>> [pOpt,optInfo] = sdo_sdoHydraulicCylinder;

Optimization started 14-Mar-2012 12:04:23

Iter F-count      f(x)      max      Step-size      First-order
      constraint
0     5          0.001      0.3033
1    10  0.000414827      3.401      4e+04          0.001
2    15  0.000506357      0.4482      6.74e+03      0.00148
3    20  0.000537775      0.05098      4.36e+03      0.257
4    25  0.000511641      0.003752      10.1          0.0977
5    30  0.000503989      0.000875      335          0.001

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 function tolerance,
and constraints are satisfied to within the selected value of the constraint tolerance.

fx >> |
```

The first output argument, `pOpt`, contains the optimized parameter values and the second output argument, `optInfo`, contains optimization information.

Modify the Generated Code

You can:

- Modify the generated `sdo_sdoHydraulicCylinder` function to include or exclude new design requirements or change the optimization options.
- Call the generated `sdo_sdoHydraulicCylinder` function with a different set of parameters to optimize.

For details on how to write an objective/constraint function to use with the `sdo.optimize` command, type `help sdoExampleCostFunction` at the MATLAB command prompt.

Close the model

Skip Model Simulation Based on Parameter Constraint Violation (GUI)

This example shows how to optimize a design and specify parameter-only constraints that prevent the model from being evaluated in an invalid solution space.

During optimization, the solver may try a design variable set that results in a model simulation error, which can be computationally expensive. If you can define a parameter-only constraint that identifies such a design variable set, then the solver can use the constraint to skip such sets. In other words, you can configure the optimization to be more efficient by disallowing design variable sets that lead to simulation errors.

In this example, you optimize thermostat settings to minimize temperature set-point deviations while satisfying some constraints. One of the constraints applies to the model parameters that define the thermostat switch on/switch off points. If the switch-off point is greater than the switch-on point, evaluating the model leads to a simulation error.

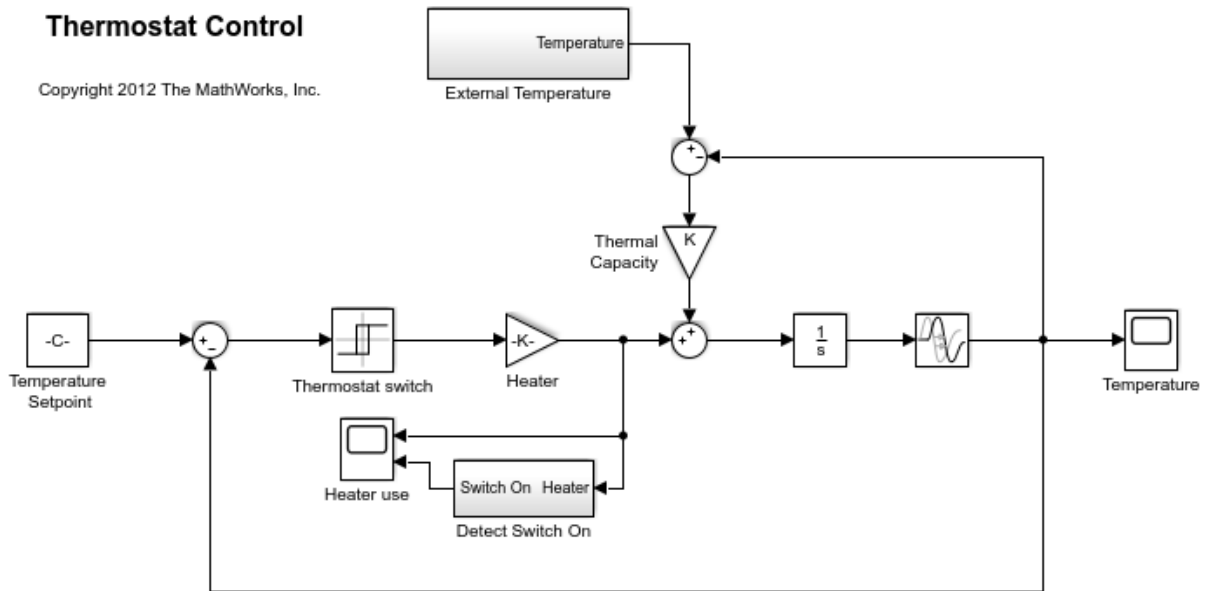
Thermostat Model

Open the model.

```
open_system('sdoThermostat');
```

Thermostat Control

Copyright 2012 The MathWorks, Inc.



The model describes a simple heater & thermostat that regulate the temperature of a room. The room is subject to external temperature fluctuations. The room temperature is computed using a first-order heat-flow equation:

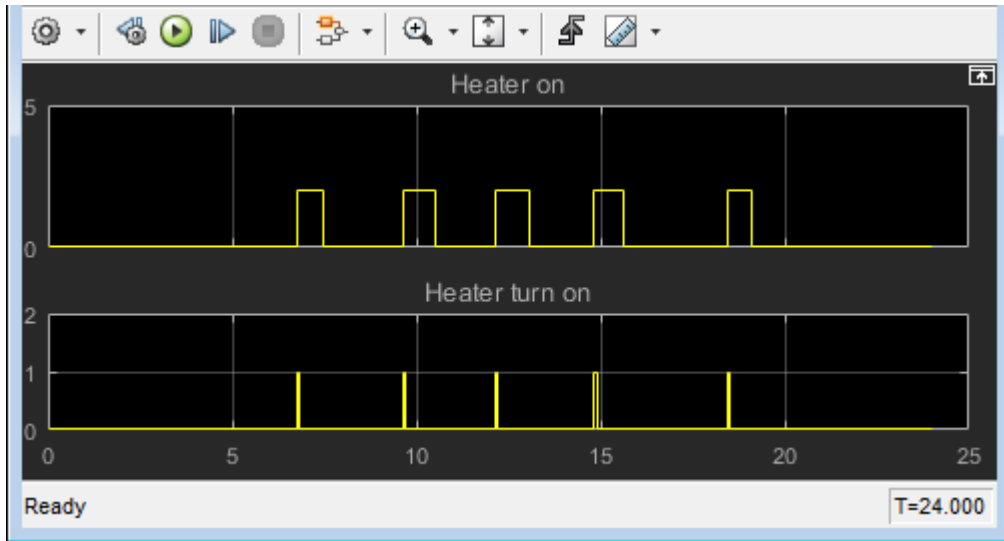
$$\frac{dT}{dt} = K(T_e - T) + Q$$

Where:

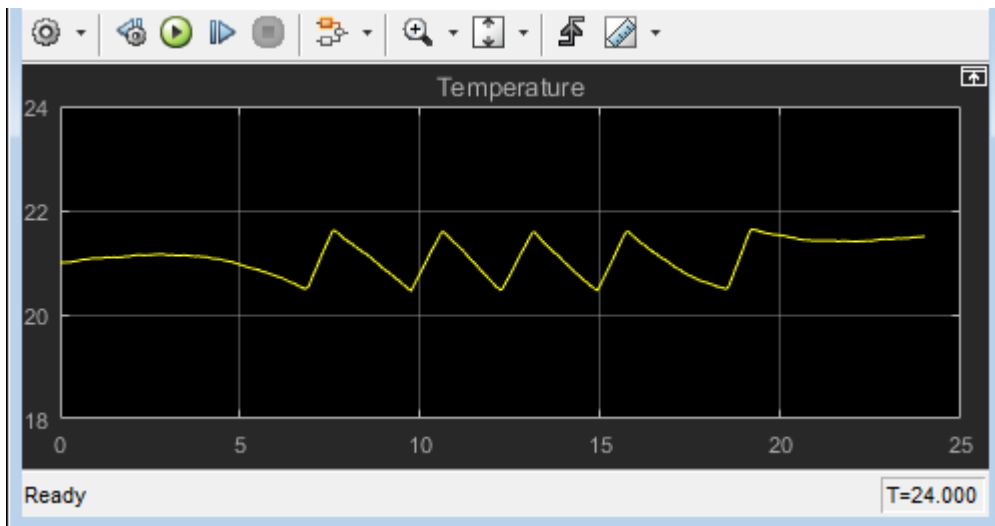
- T is the room temperature (C).
- T_e the external temperature (C).
- Q the heat supplied by the heater (W).
- K the room thermal capacity (J/C).

The heater is controlled by a thermostat that turns on when the difference between the room temperature and temperature set-point exceeds a threshold. The heater turns off when the error drops below a threshold.

The heater operation is displayed in the Heater use scope. The upper axis is the delivered heat and the lower axis shows the times when the heater is switched on.



The room temperature is displayed in the Temperature scope.



Thermostat Design Problem

You tune the thermostat turn-on and turn-off temperature thresholds, and also the heater power. The `Thermostat_switch` block specifies the turn-on and turn-off thresholds using the variables `H_on` and `H_off`. The `Heater` block specifies the heater power using the variable `Hgain`.

The design requirements are:

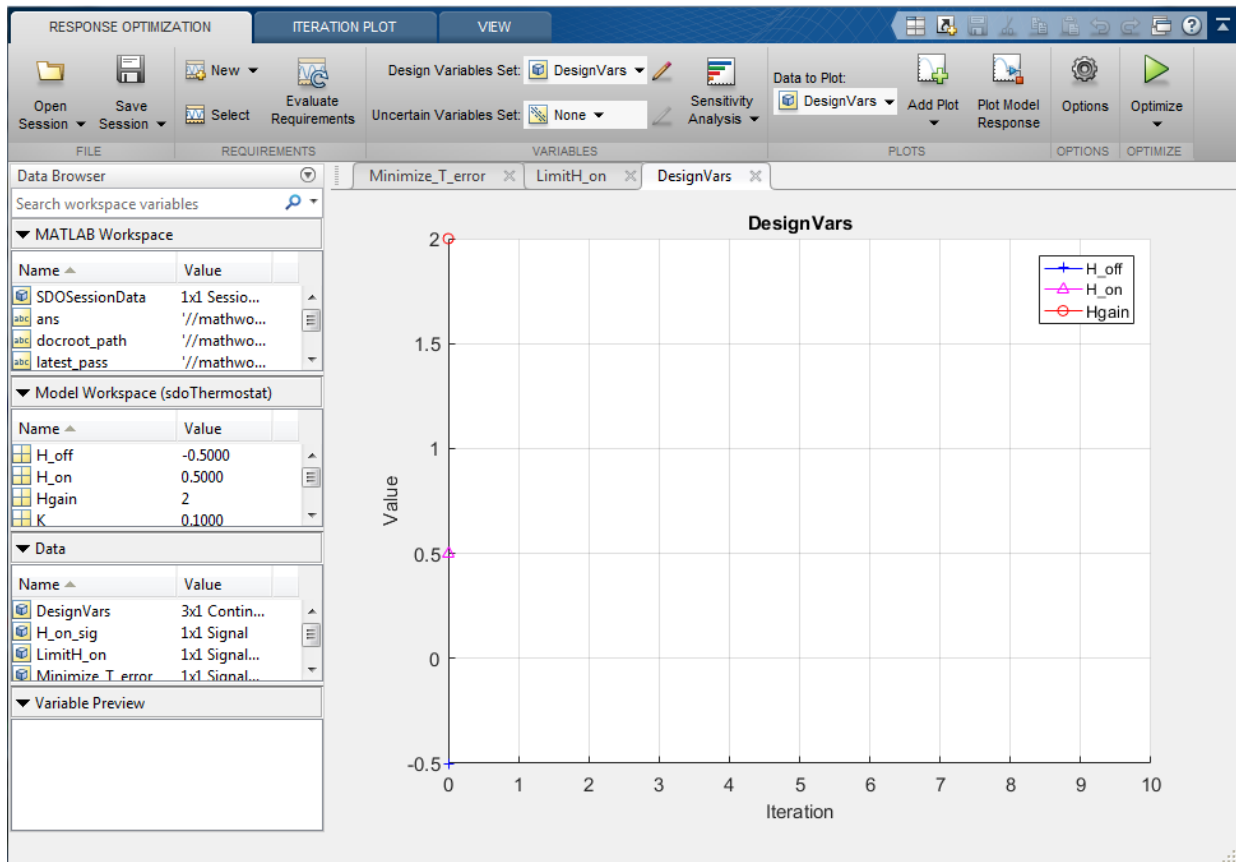
- Minimize the difference between the room temperature and temperature set-point over a 24 hour period.
- The heater must not turn on more than 12 times during the 24 hour period.
- The thermostat turn-on temperature must be greater than the thermostat turn-off temperature. If this constraint is violated, the model is invalid and cannot be simulated or evaluated.

Open the Response Optimization Tool

Open a pre-configured Response Optimization tool session using the commands

```
load sdoThermostat_sdoessionion
sdotool(SDOSessionData)
```

3 Response Optimization



The pre-configured session specifies the following variables:

- DesignVars - Design variables set for the H_on, H_off, and Hgain model parameters.
- Minimize_T_error - Requirement to minimize the temperature deviation from the set-point.
- LimitH_on - Requirement to limit the number of times the thermostat is turned on.
- H_on_sig and T_error - Logged signals. H_on_sig represents when the heater is on. T_error is the difference between the room temperature and the set-point.

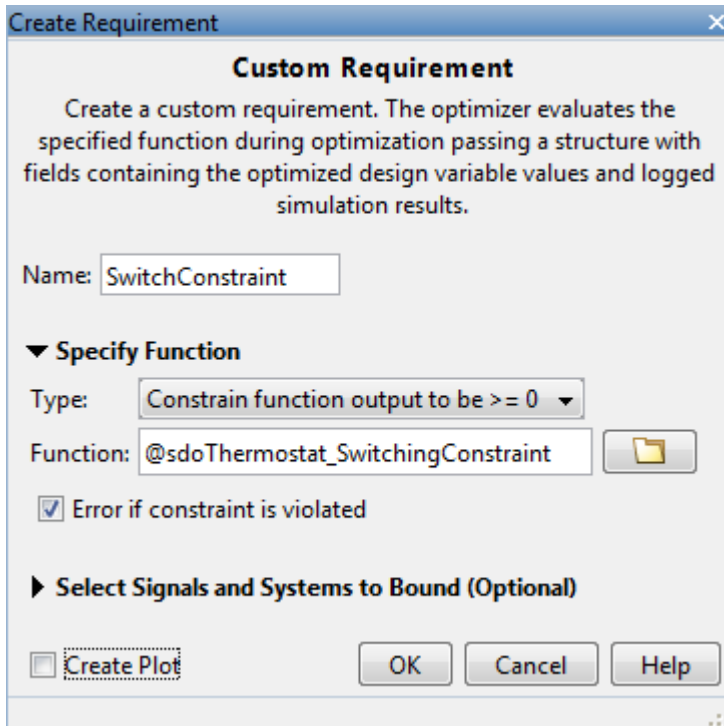
Specify Parameter Constraint

The $H_{on} > H_{off}$ requirement is not yet defined. Use a custom requirement to specify this constraint and configure the requirement to error if it is not satisfied.

In the **New** drop-down list, select **Custom Requirement**. The Create Requirement dialog opens.

In this dialog, specify the following:

- **Name** - SwitchConstraint.
- **Type** - Select Constrain the function output to be ≥ 0 from the **Type** list.
- **Function** - @sdoThermostat_SwitchingConstraint.
- **Error if constraint is violated** - Select this check box.



The software calls the `sdoThermostat_SwitchingConstraint` function at each optimization iteration with a structure containing all the design variables. The output of the `sdoThermostat_SwitchingConstraint` function is the difference between the `H_on` and `H_off` values. This difference must be positive for the requirement to be satisfied.

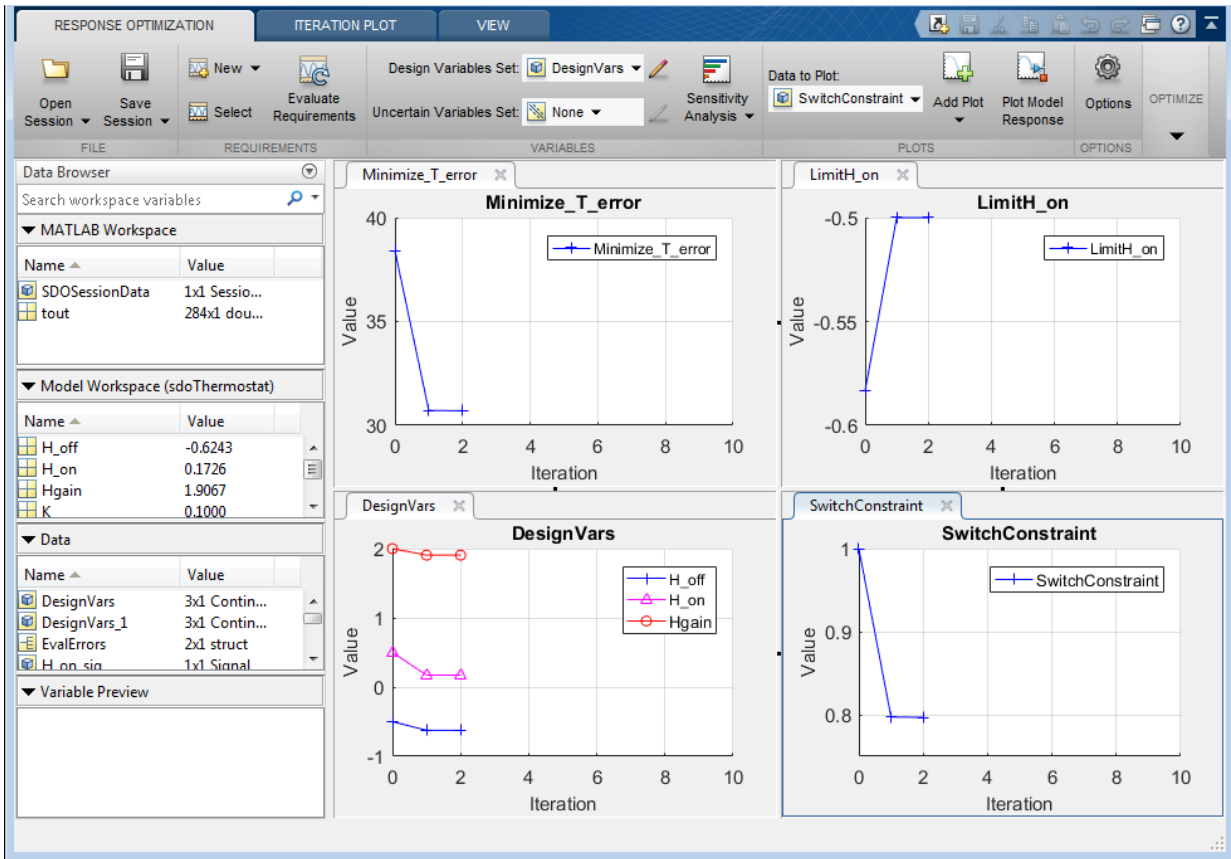
The software evaluates custom requirements that test parameter-only constraints, such as `SwitchConstraint`, before simulating the model and evaluating the remaining requirements.

- If the constraint is violated while the **Error if constraint is violated** check box is *selected*, the software does not simulate the model to evaluate the remaining requirements. Instead, the solver assigns the cost function a NaN value for this iteration, evaluates the terminating conditions, and continues.
- If the constraint is violated while the **Error if constraint is violated** check box is *cleared*, the solver will attempt to simulate the model to evaluate the remaining requirements. Simulating the model may lead to a hard error; for example, simulating the thermostat model when `SwitchConstraint` is violated will lead to an error. In this case, the solver assigns the cost function a NaN value for this iteration, evaluates the terminating conditions, and continues.

To examine the constraint function, type `edit sdoThermostat_SwitchingConstraint`. The requirement that `H_on > H_off` is implemented as `H_on - H_off > 0`

Optimize the Design

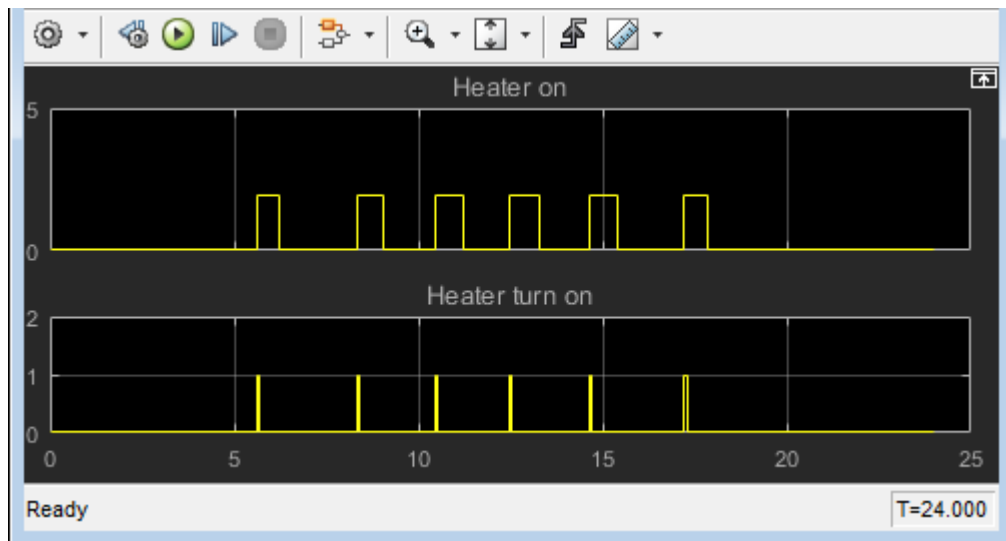
Click **Optimize**.



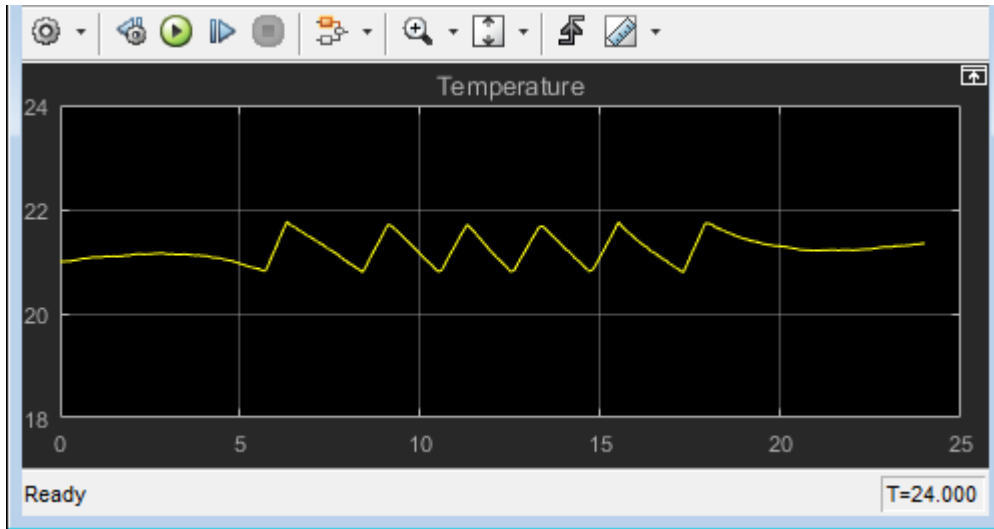
triggers the `SwitchConstraint` requirement and prevents model simulation for the relevant iterations.

View Optimized Model Response

Simulate the model with the optimized thermostat settings. The optimized heater operation is displayed in the `Heater use` scope where the upper axis is the delivered heat and the lower axis the heater switch on times.



The optimized room temperature is displayed in the `Temperature` scope.



Close the model

```
bdclose('sdoThermostat')
```

Optimizing Parameters for Robustness

In this section...

“What Is Robustness?” on page 3-201

“Sampling Methods for Uncertain Parameters” on page 3-202

“Optimize Parameters for Robustness (GUI)” on page 3-204

What Is Robustness?

A design is robust when its response does not violate design requirements under model parameter variations. Your model may contain parameters whose values are not precisely known. Such parameters vary over a given range of values and are defined as uncertain parameters. You may know the nominal value and the range of values in which these uncertain parameters vary.

You can use Simulink Design Optimization software to incorporate the parameter uncertainty to test the robustness of your design. When you optimize parameters for robustness, the optimization solver uses the responses computed using all the uncertain parameter values to adjust the design variable values.

You can specify the same parameter both as a design *and* uncertain variable. However, you cannot use a parameter both as a design and uncertain variable in the same optimization run. Also, you cannot add uncertainty to controller or plant parameters during optimization-based control design in the SISO Design Tool.

The uncertain variables can be scalar, vector, matrix or an expression.

You can test and optimize parameters for model robustness in the following ways:

- **Before Optimization.** Specify the parameter uncertainty *before* you optimize the parameters to meet the design requirements. In this case, the optimization method optimizes the signals based on both nominal parameter values as well as the uncertain values. This mode requires more computational time.
- **After Optimization.** Specify the parameter uncertainty *after* you have optimized the model parameters to meet design requirements. You can then test the effect of the uncertain parameters by plotting the model's response. If the response violates the design requirements, you can optimize the parameters again by including the parameter uncertainty during the optimization.

Related Examples

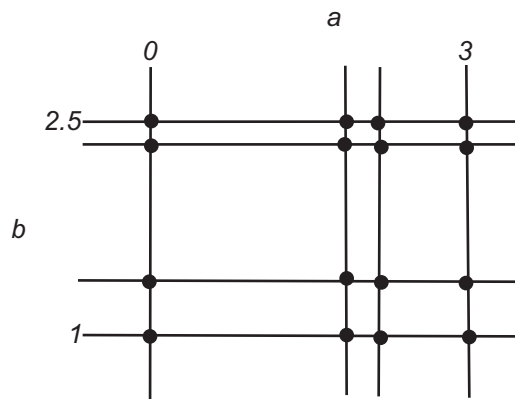
“Optimize Parameters for Robustness (GUI)” on page 3-204

More About

“Sampling Methods for Uncertain Parameters” on page 3-202

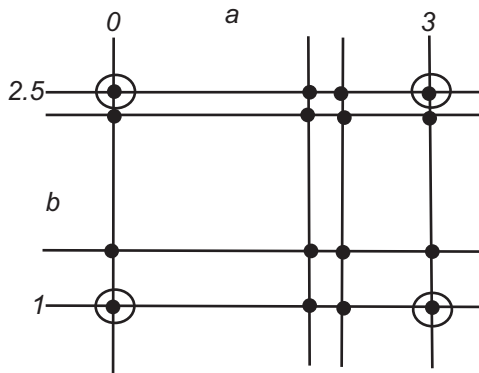
Sampling Methods for Uncertain Parameters

Sample values for uncertain parameters are a vector of numerical values. You can specify the vector yourself or generate a vector of random numbers using the software. The sample values you specify can be uniformly distributed or random. For example, four sample values for two uncertain parameters a and b in the range $[0 \ 3]$ and $[1 \ 2.5]$ may look like the following figure.



There are two methods to determine the number of sample values to use during optimization:

- Only the combination of minimum and maximum values (circled)

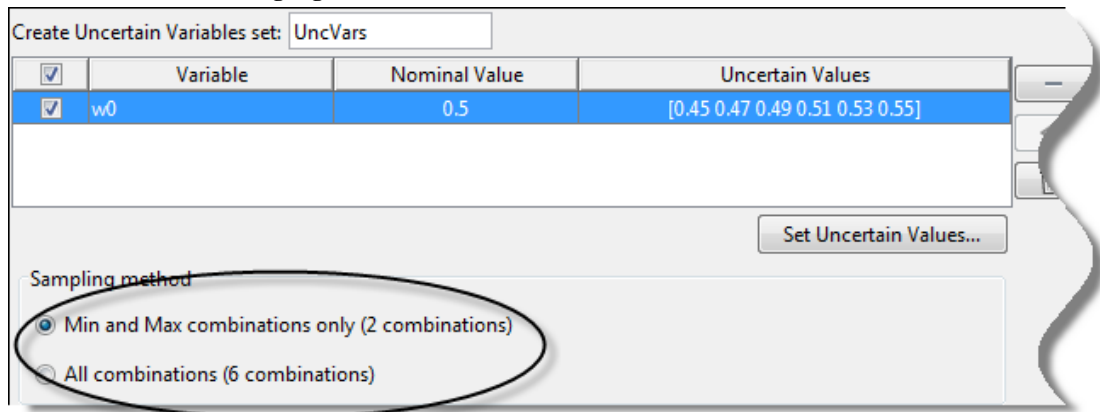


- Combination of the entire set of values (all solid dots in the previous figure)

Tip Using only the minimum and maximum values during optimization increases the computation speed when compared to using the entire set of values.

For the previous example, there are 4 combinations using the minimum and maximum values and 16 combinations if you use all sample values.

In the Response Optimization tool, you specify the sampling method using the options as shown in the following figure.



Related Examples

“Optimize Parameters for Robustness (GUI)” on page 3-204

More About

- “What Is Robustness?” on page 3-201

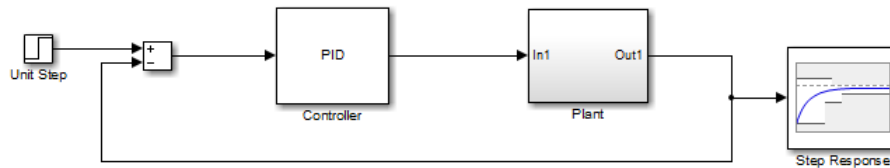
Optimize Parameters for Robustness (GUI)

This example shows how to optimize parameters for model robustness.

- 1 Load a saved Response Optimization tool session.

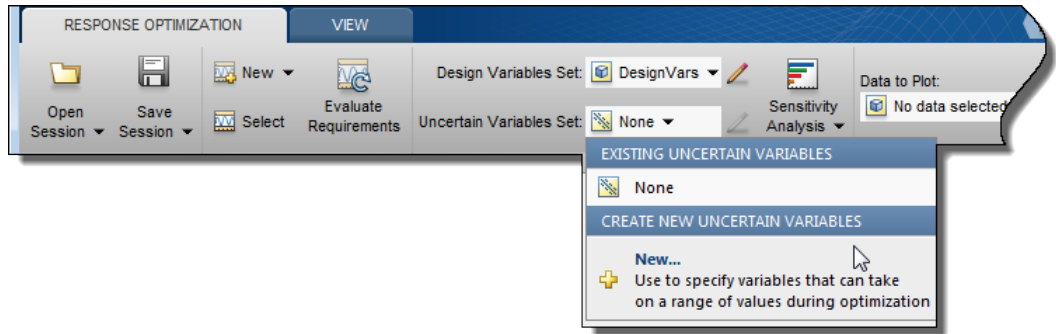
```
load sldo_model1_desreq_optim_sdoession;  
sdotool(SDOSessionData);
```

The `sdotool` command opens the following Simulink model and a saved Response Optimization tool session.

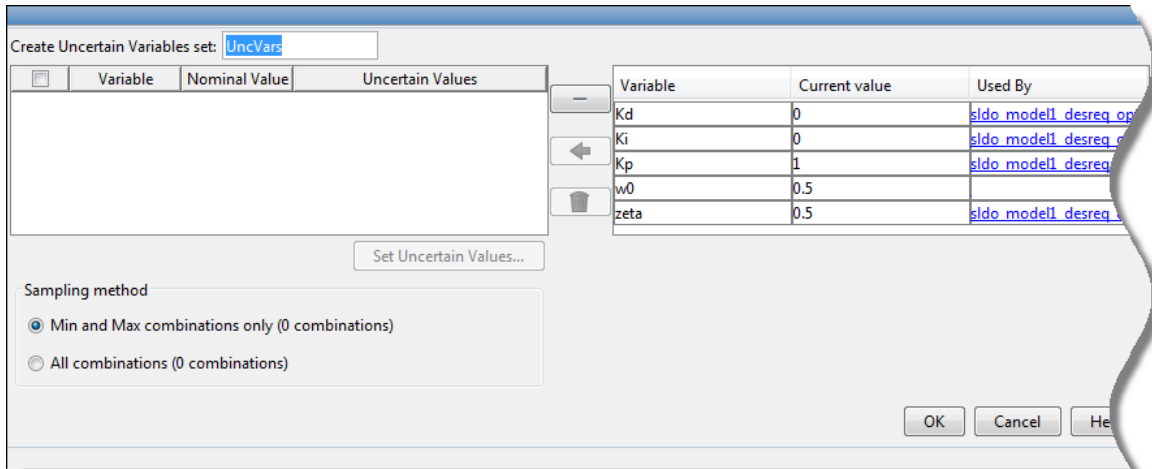


The parameters of this model, K_P , K_i and K_d , have already been optimized to meet the following step response requirements:


- Maximum overshoot of 5%
 - Maximum rise time of 10 seconds
 - Maximum settling time of 30 seconds
- 2 Specify parameter uncertainty.
 - a In the **Uncertain Variables Set** drop-down list, select **New**.

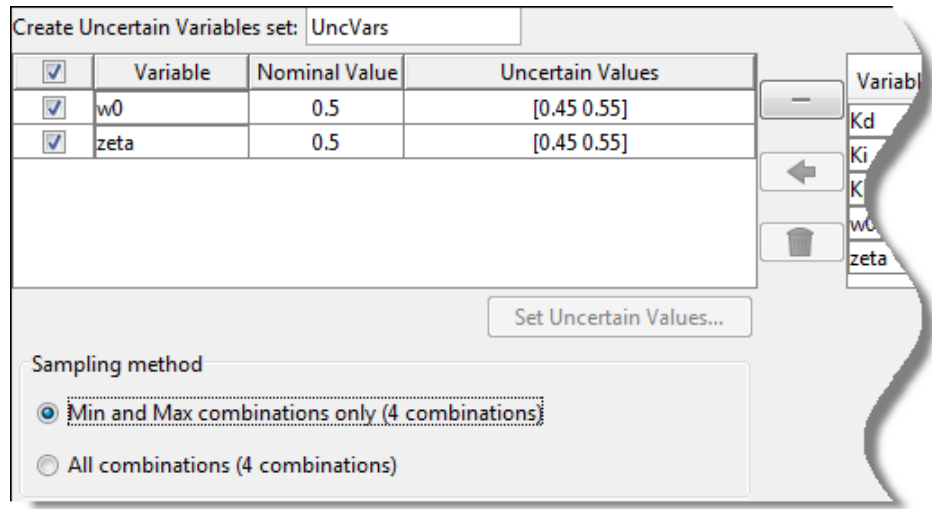


A window opens where you specify uncertain variables.



b Click `w0` and `zeta` to select them.

c Click  to add the selected parameters to an uncertain variables set.



The software displays the following parameter settings:

- **Variable** — Parameter name
- **Nominal Value** — Nominal value of the parameters as specified in the Simulink model
- **Uncertain Values** — Values that the uncertain parameter can take. By default, the maximum and minimum values vary by 10% of the nominal value.

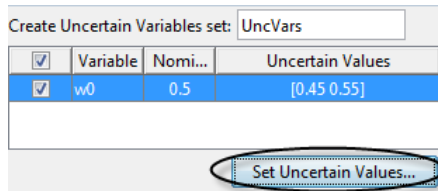
The total number of sample values to use during optimization is a combination of the maximum and minimum values of the uncertain parameters.

The check-box indicates that the parameter is included in the uncertain variable set. The default uncertain variable set name is `UncVars`.

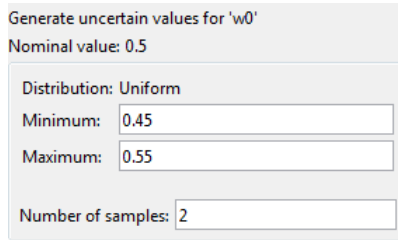
Click **OK**. A new variable `UncVars` appears in **Data** area of the Response Optimization tool.

Specify Random Values

Instead of specifying sample values, you can auto-generate random values in a specific range. Select a parameter and click **Set Uncertain Values**.

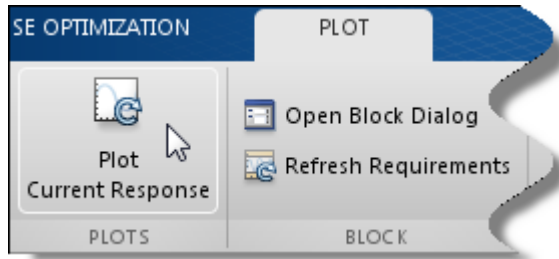


A window opens where you specify the range and the number of samples.

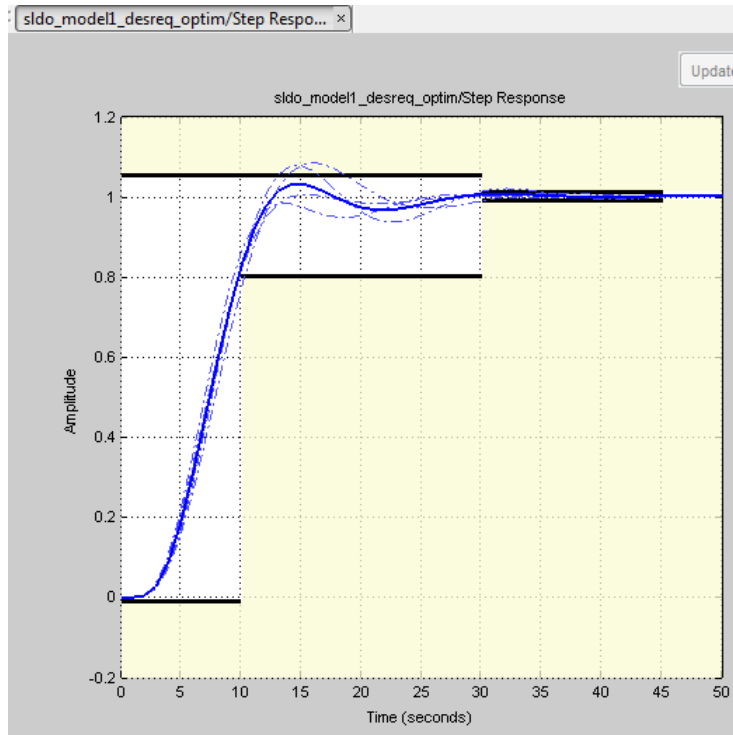


3 Test the model robustness to the uncertain parameters.

a Click **Plot Model Response**.



The step response plot, displaying the requirements, updates.

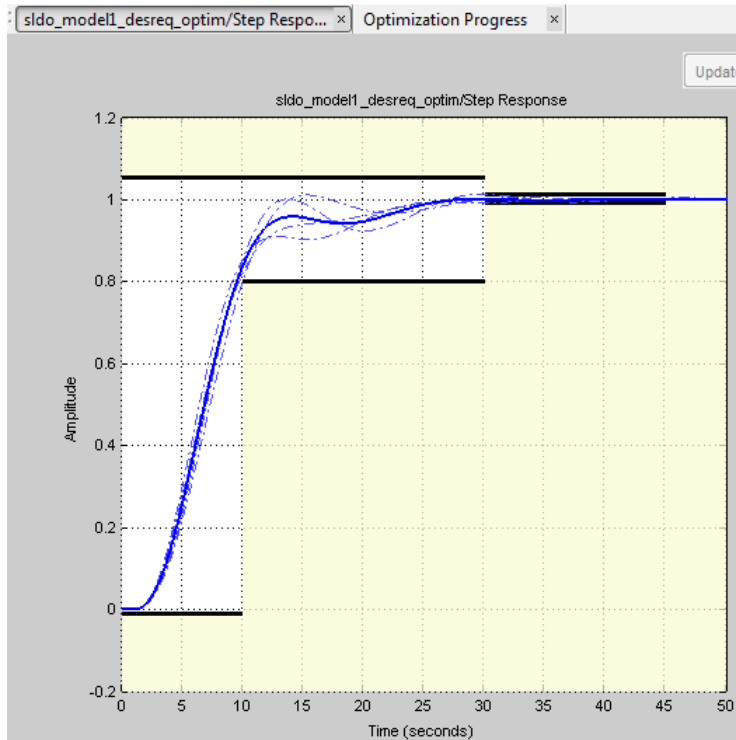


- The solid curve corresponds to the model response computed using the optimized parameters and nominal values of the uncertain parameter.
- The four dashed curves correspond to the model response with the minimum and maximum values of the uncertain parameters.

The dashed plot lines show that the response during the period of 10 to 20 seconds violates the design requirements.

- 4 Optimize the parameters for model robustness. Click **Optimize**.

Tip To view only the final responses of the model, right-click the white area in the plot and uncheck **Responses > Show Iteration Responses**.



The final responses appear as the thick solid and dashed curves. The nominal and uncertain responses with parameter variations now meet the design requirements.

More About

- “What Is Robustness?” on page 3-201
- “Sampling Methods for Uncertain Parameters” on page 3-202

See Also

Related Examples

- “Design Optimization with Uncertain Variables (Code)” on page 3-175

Use Accelerator Mode During Simulations

In this section...
“About Accelerating Optimization” on page 3-212
“Limitations” on page 3-212
“Setting Accelerator Mode” on page 3-212

About Accelerating Optimization

Simulink Design Optimization software supports `Normal` and `Accelerator` simulation modes. You can accelerate the design optimization computations by changing the simulation mode of your Simulink model to `Accelerator`. For information about these modes, see “How Acceleration Modes Work” (Simulink).

The default simulation mode is `Normal`. In this mode, Simulink uses interpreted code, rather than compiled C code during simulations.

In the `Accelerator` mode, Simulink Design Optimization software runs simulations during optimization with compiled C code. Using compiled C code speeds up the simulations and reduces the time to optimize the model response signals.

Limitations

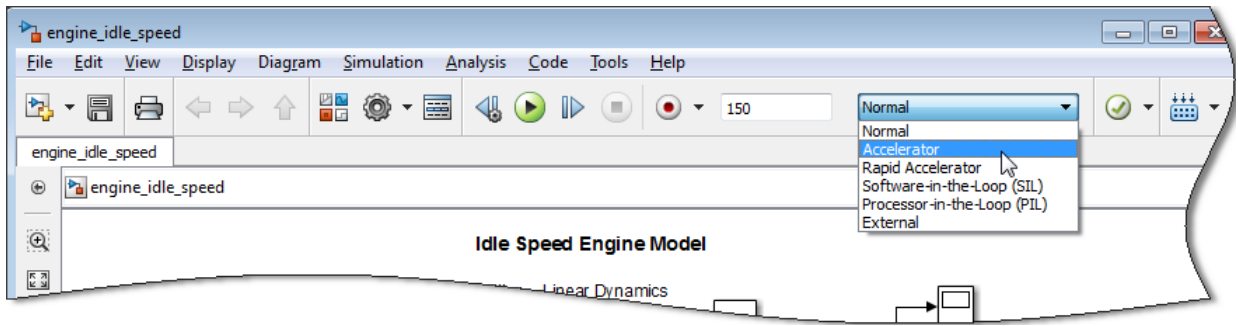
You cannot use the `Accelerator` mode if your model contains algebraic loops. If the model contains MATLAB function blocks, you must either remove them or replace them with `Fcn` blocks.

If the model structure changes during optimization, the model is compiled to regenerate the C code for each iteration. In this case, using the `Accelerator` mode increases the computation time. To learn more about code regeneration, see “Code Regeneration in Accelerated Models” (Simulink).

Setting Accelerator Mode

To set the simulation mode to `Accelerator`, open the Simulink model window and perform one of the following actions:

- Select under **Simulation > Mode > Accelerator**.
- Choose Accelerator from the drop-down list as shown in the next figure.



Tip To obtain the maximum performance from the Accelerator mode, close all Scope blocks in your model.

See Also

More About

- “Ways to Speed Up Design Optimization Tasks”

Speed Up Response Optimization Using Parallel Computing

In this section...
“When to Use Parallel Computing for Response Optimization” on page 3-214
“How Parallel Computing Speeds Up Optimization” on page 3-214

When to Use Parallel Computing for Response Optimization

You can use Simulink Design Optimization software with Parallel Computing Toolbox software to speed up the response optimization of a Simulink model. Using parallel computing may reduce model optimization time in the following cases:

- The model contains a large number of tuned parameters, and the `Gradient descent` method is selected for optimization.
- The `Pattern search` method is selected for optimization.
- The model contains a large number of uncertain parameters and uncertain parameter values.
- The model is complex and takes a long time to simulate.

When you use parallel computing, the software distributes independent simulations to run them in parallel on multiple MATLAB sessions, also known as *workers*. Distributing the simulations significantly reduces the optimization time because the time required to simulate the model dominates the total optimization time.

For information on how the software distributes the simulations and the expected speedup, see “How Parallel Computing Speeds Up Optimization” on page 3-214.

For information on configuring your system and using parallel computing, see “Use Parallel Computing for Response Optimization” on page 3-218.

How Parallel Computing Speeds Up Optimization

You can enable parallel computing with the `Gradient descent` and `Pattern search` optimization methods. When you enable parallel computing, the software distributes independent simulations during optimization on multiple MATLAB sessions. The following topics describe which simulations are distributed and the potential speedup using parallel computing:

- “Parallel Computing with the Gradient Descent Method” on page 3-215
- “Parallel Computing with the Pattern Search Method” on page 3-216

Parallel Computing with the Gradient Descent Method

When you select `Gradient descent` as the optimization method, the model is simulated during the following computations:

- Constraint and objective value computation — One simulation per iteration
- Constraint and objective gradient computations — Two simulations for every tuned parameter per iteration
- Line search computations — Multiple simulations per iteration

The total time, T_{total} , taken per iteration to perform these simulations is given by the following equation:

$$T_{total} = T + (N_p \times 2 \times T) + (N_{ls} \times T) = T \times (1 + (2 \times N_p) + N_{ls})$$

where T is the time taken to simulate the model and is assumed to be equal for all simulations, N_p is the number of tuned parameters, and N_{ls} is the number of line searches. N_{ls} is difficult to estimate and you generally assume it to be equal to one, two, or three.

When you use parallel computing, the software distributes the simulations required for constraint and objective gradient computations. The simulation time taken per iteration when the gradient computations are performed in parallel, T_{totalP} , is approximately given by the following equation:

$$T_{totalP} = T + \left(\text{ceil} \left(\frac{N_p}{N_w} \right) \times 2 \times T \right) + (N_{ls} \times T) = T \times \left(1 + 2 \times \text{ceil} \left(\frac{N_p}{N_w} \right) + N_{ls} \right)$$

where N_w is the number of MATLAB workers.

Note The equation does not include the time overheads associated with configuring the system for parallel computing and loading Simulink software on the remote MATLAB workers.

The expected speedup for the total optimization time is given by the following equation:

$$\frac{T_{totalP}}{T_{total}} = \frac{1 + 2 \times \text{ceil}\left(\frac{N_p}{N_w}\right) + N_{ls}}{1 + (2 \times N_p) + N_{ls}}$$

For example, for a model with $N_p=3$, $N_w=4$, and $N_{ls}=3$, the expected speedup equals

$$\frac{1 + 2 \times \text{ceil}\left(\frac{3}{4}\right) + 3}{1 + (2 \times 3) + 3} = 0.6 .$$

For an example of the performance improvement achieved with the Gradient descent method, see Improving Optimization Performance Using Parallel Computing.

Parallel Computing with the Pattern Search Method

The Pattern search optimization method uses search and poll sets to create and compute a set of candidate solutions at each optimization iteration.

The total time, T_{total} , taken per iteration to perform these simulations, is given by the following equation:

$$T_{total} = (T \times N_p \times N_{ss}) + (T \times N_p \times N_{ps}) = T \times N_p \times (N_{ss} + N_{ps})$$

where T is the time taken to simulate the model and is assumed to be equal for all simulations, N_p is the number of tuned parameters, N_{ss} is a factor for the search set size, and N_{ps} is a factor for the poll set size. N_{ss} and N_{ps} are typically proportional to N_p .

When you use parallel computing, Simulink Design Optimization software distributes the simulations required for the search and poll set computations, which are evaluated in separate `parfor` loops. The simulation time taken per iteration when the search and poll

sets are computed in parallel, T_{totalP} , is given by the following equation:

$$\begin{aligned} T_{totalP} &= (T \times \text{ceil}(N_p \times \frac{N_{ss}}{N_w})) + (T \times \text{ceil}(N_p \times \frac{N_{ps}}{N_w})) \\ &= T \times (\text{ceil}(N_p \times \frac{N_{ss}}{N_w}) + \text{ceil}(N_p \times \frac{N_{ps}}{N_w})) \end{aligned}$$

where N_w is the number of MATLAB workers.

Note The equation does not include the time overheads associated with configuring the system for parallel computing and loading Simulink software on the remote MATLAB workers.

The expected speed up for the total optimization time is given by the following equation:

$$\frac{T_{totalP}}{T_{total}} = \frac{\text{ceil}(N_p \times \frac{N_{ss}}{N_w}) + \text{ceil}(N_p \times \frac{N_{ps}}{N_w})}{N_p \times (N_{ss} + N_{ps})}$$

For example, for a model with $N_p=3$, $N_w=4$, $N_{ss}=15$, and $N_{ps}=2$, the expected speedup

$$\text{equals } \frac{\text{ceil}(3 \times \frac{15}{4}) + \text{ceil}(3 \times \frac{2}{4})}{3 \times (15 + 2)} = 0.27 .$$

Note Using the `Pattern` search method with parallel computing may not speed up the optimization time. To learn more, see “Why do I not see the optimization speedup I expected using parallel computing?” on page 3-235

For an example of the performance improvement achieved with the `Pattern` search method, see [Improving Optimization Performance Using Parallel Computing](#).

See Also

Related Examples

- “Use Parallel Computing for Response Optimization” on page 3-218

Use Parallel Computing for Response Optimization

In this section...
“Configure Your System for Parallel Computing” on page 3-218
“Model Dependencies” on page 3-218
“Optimize Design Using Parallel Computing (GUI)” on page 3-219
“Optimize Design Using Parallel Computing (Code)” on page 3-223
“Troubleshooting” on page 3-224

Configure Your System for Parallel Computing

You can speed up model optimization using parallel computing on multicore processors or multiprocessor networks. Use parallel computing with the Response Optimization tool and `sdo.optimize` to optimize using the `fmincon`, `lsqonlin`, and `patternsearch` methods. Parallel computing is not supported for the `fminsearch` (**Simplex search**) method.

When you optimize model parameters using parallel computing, the software uses the available parallel pool. If none is available, and you select **Automatically create a parallel pool** in your Parallel Computing Toolbox preferences, the software starts a parallel pool using the settings in those preferences. To open a parallel pool that uses a specific cluster profile, use:

```
parpool(MyProfile);
```

`MyProfile` is the name of a cluster profile.

For information regarding creating a cluster profile, see “Create and Modify Cluster Profiles” (Parallel Computing Toolbox).

Model Dependencies

Model dependencies are any referenced models, data such as model variables, S-functions, and additional files necessary to run the model. Before starting the optimization, verify that the model dependencies are complete. Otherwise, you may get unexpected results.

Making Model Dependencies Accessible to Remote Workers

When you use parallel computing, the Simulink Design Optimization software helps you identify model dependencies. To do so, the software uses the Simulink Manifest Tools. The dependency analysis may not find all the files required by your model. To learn more, see “Scope of Dependency Analysis” (Simulink). If your model has dependencies that are undetected or inaccessible by the parallel pool workers, then add them to the list of model dependencies.

The dependencies 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.
- Path dependencies: the paths to the model dependencies are added to the paths of the parallel pool workers. If you are working in a multi-platform scenario, ensure that the paths are compatible across platforms.

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 dependent files on the local computer.

For more information, see:

- “Optimize Design Using Parallel Computing (GUI)” on page 3-219
- “Optimize Design Using Parallel Computing (Code)” on page 3-223

Optimize Design Using Parallel Computing (GUI)


To optimize a model response using parallel computing in the Response Optimization tool:

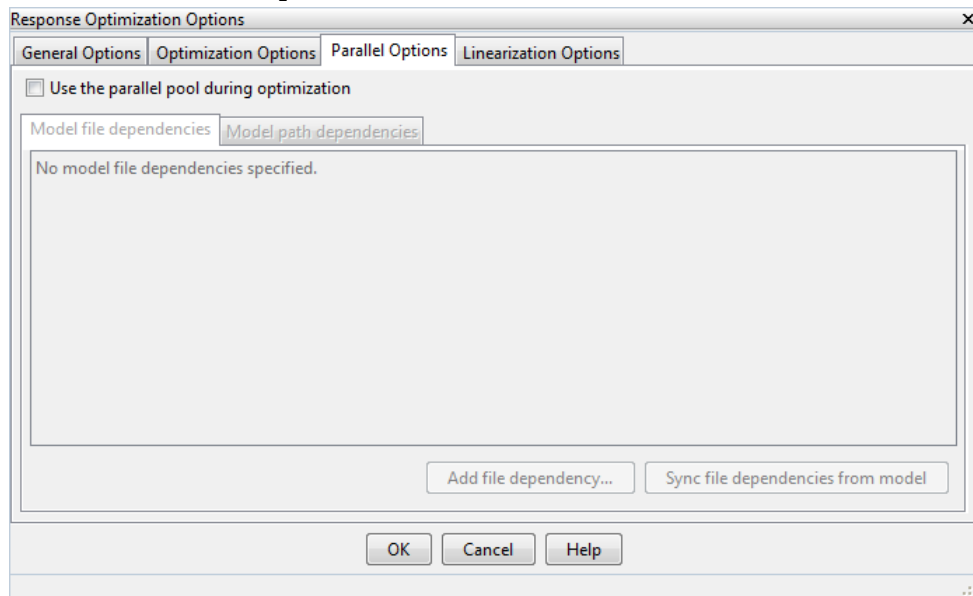
- 1 Ensure that the software can access parallel pool workers that use the appropriate cluster profile.

For more information, see “Configure Your System for Parallel Computing” on page 3-218.

- 2 Open the Response Optimization tool for the Simulink model.
- 3 Configure the design variables, design requirements, and, optionally, optimization settings.

For more information, see “Specify Design Variables” on page 3-76, “Specify Time-Domain Design Requirements in the Tool” on page 3-19, “Specify Frequency-Domain Design Requirements in the Tool” on page 3-56, and “Specify Optimization Options” on page 3-84.

- 4 On the **Response Optimization** tab, click  **Options** to open the **Response Optimization Options** dialog box.
- 5 Select the **Parallel Options** tab.

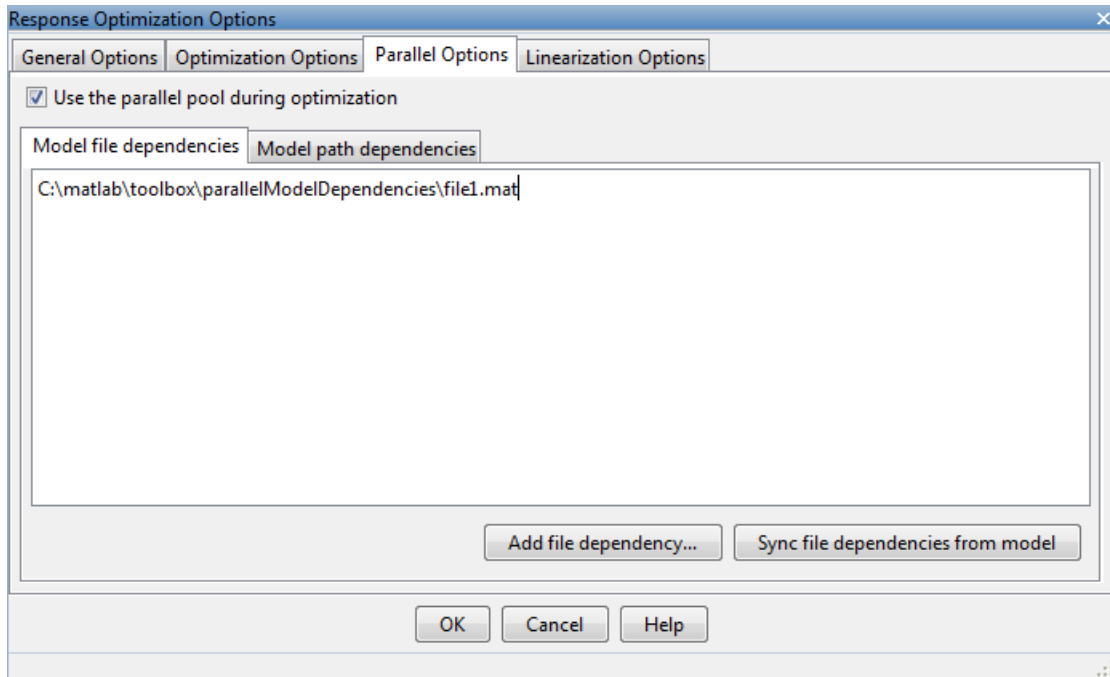


- 6 Select the **Use the parallel pool during optimization** check box.

This option checks for dependencies in your Simulink model. The file dependencies are displayed in the **Model file dependencies** list box, and corresponding path to the files in **Model path dependencies**. The files listed in **Model file dependencies** are copied to the remote workers.

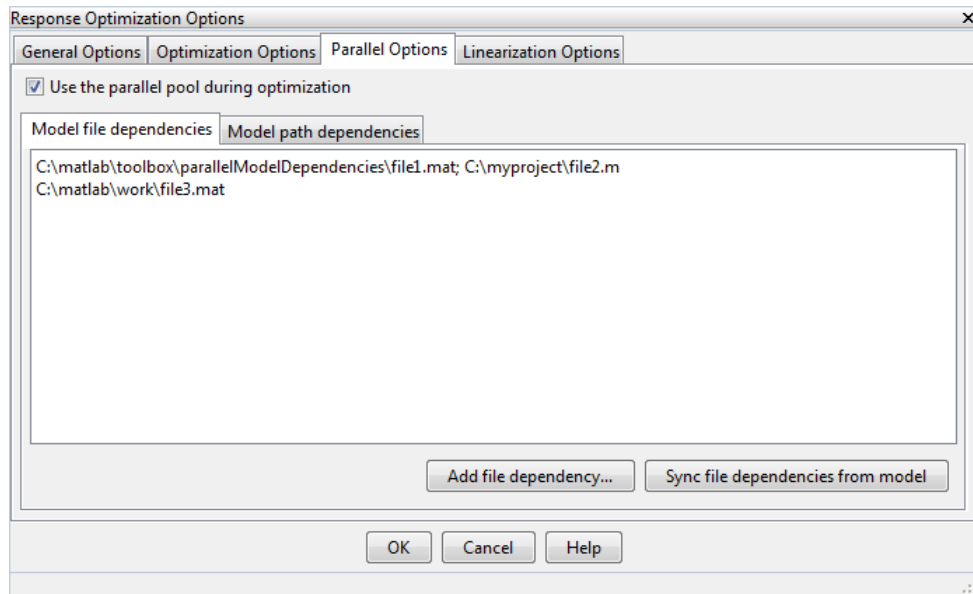
Note The automatic dependencies check may not detect all the dependencies in your model.

For more information, see “Model Dependencies” on page 3-218. In this case, add the undetected dependencies manually.



- 7 Add any file dependencies that the automatic check does not detect.

Specify the files in the **Model file dependencies** list box separated by semicolons or on separate lines.



Alternatively, click **Add file dependency** to open a dialog box, and select the file to add.

Note If you do not want to copy the files to the remote workers, delete all entries in the **Model file dependencies** list box. Populate the **Model path dependencies** list box by clicking the **Sync path dependencies from model**, and add any undetected path dependencies. In addition, in the list box, update the paths on local drives to make them accessible to remote workers. For example, change `C:\` to `\\hostname\C$`.

- 8 If you modify the Simulink model, resync the dependencies to ensure that any new dependencies are detected. Click **Sync file dependencies from model** in the **Parallel Options** tab to rerun the automatic dependency check for your model.

This action updates the **Model file dependencies** list box with any new file dependency found in the model.

- 9 Click **OK**.
- 10 In the Response Optimization tool, click **Optimize** to optimize the model response using parallel computing.

For information on troubleshooting problems related to optimization using parallel computing, see “Troubleshooting” on page 3-224.

Optimize Design Using Parallel Computing (Code)

To optimize a model response using parallel computing at the command line:

- 1 Ensure that the software can access parallel pool workers that use the appropriate cluster profile.

For more information, see “Configure Your System for Parallel Computing” on page 3-218.

- 2 Open the model.
- 3 Specify design requirements and design variables. For example see, “Design Optimization to Meet Step Response Requirements (Code)”.
- 4 Enable parallel computing using an optimization option set, `opt`.

```
opt = sdo.OptimizeOptions;
opt.UseParallel = true;
```

- 5 Find the model dependencies.

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

Note `sdo.getModelDependencies` may not detect all the dependencies in your model. For more information, see “Model Dependencies” on page 3-218. In this case, add the undetected dependencies manually.

- 6 Modify files to include any file dependencies that `sdo.getModelDependencies` does not detect.

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

Note If you do not want to copy the files to the remote workers, use the path dependencies. Add any undetected path dependencies to `dirs` and update the paths on local drives to make them accessible to remote workers. See `sdo.getModelDependencies` for more details.

- 7 Add the file dependencies for optimization.

```
opt.ParallelFileDependencies = files;
```

8 Run the optimization.

```
[pOpt, opt_info] = sdo.optimize(opt_fcn, param, opt);
```

For information on troubleshooting problems related to optimization using parallel computing, see “Troubleshooting” on page 3-224.

Troubleshooting

- “Why Are the Optimization Results With and Without Using Parallel Computing Different?” on page 3-224
- “Why Don’t I See the Optimization Speed up I Expected Using Parallel Computing?” on page 3-224
- “Why Doesn’t the Optimization Using Parallel Computing Make Any Progress?” on page 3-225
- “Why Doesn’t the Optimization Using Parallel Computing Stop When I Click the Stop Optimization Button?” on page 3-225

Why Are the Optimization Results With and Without Using Parallel Computing Different?

- Different numerical precision on the client and worker machines can produce marginally different simulation results. Thus, the optimization method can take a different solution path and produce a different result.
- When you use parallel computing with the `Pattern search` method, the search is more comprehensive and can result in a different solution.

To learn more, see “Parallel Computing with the Pattern Search Method” on page 3-216.

Why Don’t I See the Optimization Speed up I Expected Using Parallel Computing?

- When you optimize a model that does not have a large number of parameters or does not take long to simulate, you might not see a speedup in the optimization time. In such cases, the overhead associated with creating and distributing the parallel tasks outweighs the benefits of running the optimization in parallel.
- Using the `Pattern search` method with parallel computing might not speed up the optimization time. Without parallel computing, the method stops the search at each iteration when it finds a solution better than the current solution. The candidate solution search is more comprehensive when you use parallel computing. Although

the number of iterations might be larger, the optimization without using parallel computing might be faster.

To learn more about the expected speedup, see “Parallel Computing with the Pattern Search Method” on page 3-216.

Why Doesn't the Optimization Using Parallel Computing Make Any Progress?

To troubleshoot the problem:

- 1 Run the optimization for a few iterations without parallel computing to see if the optimization progresses.
- 2 Check whether the remote workers have access to all model dependencies. Model dependencies include data variables and files required by the model to run.

To learn more, see “Model Dependencies” on page 3-218.

Why Doesn't the Optimization Using Parallel Computing Stop When I Click the Stop Optimization Button?

When you use parallel computing with the `Pattern search` method, the software must wait until the current optimization iteration completes before it notifies the workers to stop. The optimization does not terminate immediately when you click **Stop**, and, instead, appears to continue running.

See Also

`parpool` | `sdo.OptimizeOptions` | `sdo.getModelDependencies` | `sdo.optimize`

Related Examples

- “Optimizing Time-Domain Response of Simulink® Models Using Parallel Computing” on page 3-242

More About

- “Speed Up Response Optimization Using Parallel Computing” on page 3-214
- “Ways to Speed Up Design Optimization Tasks”

Use Fast Restart Mode During Response Optimization

In this section...
“Response Optimization Tool Workflow for Fast Restart” on page 3-226
“Command-Line Workflow for Fast Restart” on page 3-227
“Troubleshooting” on page 3-228

This topic shows how to speed up response optimization using Simulink fast restart. You can use the fast restart feature to speed up response optimization of tunable parameters (Simulink) of a model.

Fast restart enables you to perform iterative simulations without compiling a model or terminating the simulation each time. Using fast restart, you compile a model only once. You can then tune parameters and simulate the model again without spending time on compiling. Fast restart associates multiple simulation phases with a single compile phase to make iterative simulations more efficient. You see a speedup of design optimization tasks using fast restart in models that have a long compilation phase. See “How Fast Restart Improves Iterative Simulations” (Simulink).

When you enable fast restart, you can only change tunable properties of the model during simulation. For more information about the limitations, see “Factors Affecting Fast Restart” (Simulink).

You can optimize using fast restart in the Response Optimization tool on page 3-226 or at the command line on page 3-227.

Response Optimization Tool Workflow for Fast Restart

To optimize a model response using fast restart in the Response Optimization tool:

- 1 Open the Simulink model.
- 2 Enable fast restart in the model.

Click **Fast Restart**  in the model window.

- 3 Open the Response Optimization tool for the model.
- 4 Configure the design variables, design requirements, and, optionally, optimization settings.

For more information, see “Specify Design Variables” on page 3-76, “Specify Time-Domain Design Requirements in the Tool” on page 3-19, “Specify Frequency-Domain Design Requirements in the Tool” on page 3-56, and “Specify Optimization Options” on page 3-84.

- 5 Click **Optimize** to optimize the model response in fast restart mode.
- 6 Disable fast restart.

In the model window, click **Fast Restart**



Command-Line Workflow for Fast Restart

To optimize a model response using fast restart at the command line:

- 1 Open the Simulink model.
- 2 Create a model simulation scenario. You must create a simulation scenario with logging information before configuring the model for fast restart. You cannot modify logging information once the model has been compiled for fast restart.

```
Simulator = sdo.SimulationTest('model');
```

Specify model signals to log during model simulation.

For response optimization problems that include frequency-domain requirements, the model is linearized using Simulink Control Design. Use the `SystemLoggingInfo` property of the `sdo.SimulationTest` object, `Simulator`, to specify linear systems to log when simulating the model. For an example, see “Design Optimization to Meet Frequency-Domain Requirements (Code)” on page 3-251.

Note In fast restart mode, you cannot use the `linearize` command from Simulink Control Design to specify and compute linear systems. Using `linearize` generates an error.

- 3 Specify design requirements, `Requirements`, and design variables, `param`. For and example, see “Design Optimization to Meet Step Response Requirements (Code)”.
- 4 Configure the model and simulation scenario for fast restart.

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

- 5 Create an optimization cost function, `myCostfcn`, and pass `Simulator` to the cost function as an input. For more information, see “Write a Cost Function” on page 2-62. In the cost function, the simulator configured for fast restart is used to update the model parameters, simulate the model, and log signals.

Use an anonymous function with one argument that calls `myCostfcn`.

```
optimfcn = @(param) myCostfcn(param, Simulator, Requirements);
```

Here, `myCostfcn` is a cost function that takes design variables, `param`, simulation scenario, `Simulator`, and design requirements, `Requirements`, as inputs.

- 6 Run the optimization.

```
[param_opt, opt_info] = sdo.optimize(optimfcn, param);
```

- 7 Restore the simulator fast restart settings.

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

Troubleshooting

Why Don't I See the Optimization Speedup I Expected Using Fast Restart?

You see a speedup of design optimization tasks using fast restart in models that have a long compilation phase. If the compilation phase of your model is not long, you do not see a significant change in optimization speed.

See Also

`fastRestart` | `sdo.SimulationTest` | `sdo.optimize`

More About

- “Ways to Speed Up Design Optimization Tasks”

Optimization Does Not Make Progress

In this section...

“Should I worry about the scale of my responses and how constraints and design requirements are discretized?” on page 3-229

“Why don't the responses and parameter values change at all?” on page 3-229

“Why does the optimization stall?” on page 3-229

Should I worry about the scale of my responses and how constraints and design requirements are discretized?

No, Simulink Design Optimization software automatically normalizes constraints, design requirement and response data.

Why don't the responses and parameter values change at all?

The optimization problem you formulated might be nonsmooth. This means that small parameter changes have no effect on the amount by which response signals satisfy or violate the constraints and only large changes will make a difference. Try switching to a search-based method such as simplex search or pattern search. Alternatively, look for initial guesses outside of the dead zone where parameter changes have no effect. If you are optimizing the response of a Simulink model, you could also try removing nonlinear blocks such as Quantizer or Dead Zone.

Why does the optimization stall?

When optimizing a Simulink model, certain parameter combinations can make the simulation stall for models with strong nonlinearities or frequent mode switching. In these cases, the ODE solvers take smaller and smaller step sizes. Stalling can also occur when the model's ODEs become too stiff for some parameter combinations. A symptom of this behavior is when the Simulink model status is `Running` and clicking the **Stop** button fails to interrupt the optimization. When this happens, you can try one of the following solutions:

- Switch to a different ODE solver, especially one of the stiff solvers.
- Specify a minimum step size.

- Disable zero crossing detection if chattering is occurring.
- Tighten the lower and upper bounds on parameters that cause simulation difficulties. In particular, eliminate regions of the parameter space where some model assumptions are invalid and the model behavior can become erratic.

Optimization Convergence

In this section...

“What to do if the optimization does not get close to an acceptable solution?” on page 3-231

“Why does the optimization terminate before exceeding the maximum number of iterations, with a solution that does not satisfy all the constraints or design requirements?” on page 3-232

“What to do if the optimization takes a long time to converge even though it is close to a solution?” on page 3-232

“What to do if the response becomes unstable and does not recover?” on page 3-233

What to do if the optimization does not get close to an acceptable solution?

- If you are using pattern search, check that you have specified appropriate maximum and minimum values for all your tuned parameters or compensator elements. The pattern search method looks inside these bounds for a solution. When they are set to their default values of `Inf` and `-Inf`, the method searches within $\pm 100\%$ of the initial values of the parameters. In some cases this region is not large enough and changing the maximum and minimum values can expand the search region.
- Your optimization problem might have local minima. Consider running one of the search-based methods first to get closer to an acceptable solution.
- Reduce the number of tuned parameters and compensator elements by removing from the design variables or from the **Compensators** pane (when using a SISO Design Task) those parameters that you know only mildly influence the optimized responses. After you identify reasonable values for the key parameters, add the fixed parameters back to the tunable list and restart the optimization using these reasonable values as initial guesses.
- The software may have encountered errors during the optimization. Review the errors to determine if you can make changes to improve the optimization results. Changes may require modifications to the model, requirements, or optimization settings.
 - In the Response Optimization tool, the software creates a structure named `EvalErrors` in the **Data** area when the optimization completes with errors. Export this structure to the MATLAB workspace and examine its contents at the

command line. `EvalErrors` has two fields, `Errors` and `DesignVars`, containing the errors encountered during optimization and the corresponding design variable values. To reproduce a specific error, use `sdo.setValueInModel` to run the model using the design variables that correspond to the error.

- At the command line, the second output of `sdo.optimize`, `opt_info`, is a structure that provides information regarding the optimization. `opt_info.exitflag` identifies the reason the optimization terminated. For more information regarding exit flags, see “Exit Flags and Exit Messages” (Optimization Toolbox).

Why does the optimization terminate before exceeding the maximum number of iterations, with a solution that does not satisfy all the constraints or design requirements?

- It might not be possible to achieve your specifications. Try relaxing the constraints or design requirements that the response signals violate the most. After you find an acceptable solution to the relaxed problem, tighten some constraints again and restart the optimization.
- The optimization might have converged to a local minimum that is not a feasible solution. Restart the optimization from a different initial guess and/or use one of the search-based methods to identify another local minimum that satisfies the constraints.

What to do if the optimization takes a long time to converge even though it is close to a solution?

- In a Response Optimization tool, click **Stop** to interrupt the optimization when you think the current optimized response signals are acceptable.

When you use a SISO Design Task, click **Stop Optimization** in the **Optimization** panel of the **Response Optimization** node in the Control and Estimation Tools Manager, when you think the current optimized response signals are acceptable.

- If you use the gradient descent method, try restarting the optimization. Restarting resets the Hessian estimate and might speed up convergence.
- Increase the convergence tolerances in the Optimization Options dialog to force earlier termination.

- Relax some of the constraints or design requirements to increase the size of the feasibility region.

What to do if the response becomes unstable and does not recover?

While the optimization formulation has explicit safeguards against unstable or divergent response signals, the optimization can sometimes venture into an unstable region where simulation results become erratic and gradient methods fail to find a way back to the stable region. In these cases, you can try one of the following solutions:

- Add or tighten the lower and upper bounds on compensator element and parameter values. Instability often occurs when you allow some parameter values to become too large.
- Use a search-based method to find parameter values that stabilize the response signals and then start the gradient-based method using these initial values.
- When optimizing responses in a SISO Design Task, you can try adding additional design requirements that achieve the same or similar goal. For example, in addition to a settling time design requirement on a step response plot, you could add a settling time design requirement on a root-locus plot that restricts the location of the real parts of the poles. By adding overlapping design requirements in this way, you can force the optimization to meet the requirements.

Optimization Speed and Parallel Computing

In this section...

“How can I speed up the optimization?” on page 3-234

“Why are the optimization results with and without using parallel computing different?” on page 3-235

“Why do I not see the optimization speedup I expected using parallel computing?” on page 3-235

“Why does the optimization using parallel computing not make any progress?” on page 3-235

“Why does the optimization using parallel computing not stop when I click the Stop optimization button?” on page 3-236

How can I speed up the optimization?

- The optimization time is dominated by the time it takes to simulate the model. When optimizing a Simulink model, you can enable the Accelerator mode using **Simulation > Mode > Accelerator** in the Simulink Editor, to dramatically reduce the optimization time.

Note The Rapid Accelerator mode in Simulink software is not supported for speeding up the optimization. For more information, see “Use Accelerator Mode During Simulations” on page 3-212.

- The choice of ODE solver can also significantly affect the overall optimization time. Use a stiff solver when the simulation takes many small steps, and use a fixed-step solver when such solvers yield accurate enough simulations for your model. (These solvers must be accurate in the entire parameter search space.)
- Reduce the number of tuned compensator elements or parameters and constrain their range to narrow the search space.
- When specifying parameter uncertainty (not available when optimizing responses in a SISO Design Task), keep the number of sample values small since the number of simulations grows exponentially with the number of samples. For example, a grid of 3 parameters with 10 sample values for each parameter requires $10^3=1000$ simulations per iteration.

Why are the optimization results with and without using parallel computing different?

- Different numerical precision on the client and worker machines can produce marginally different simulation results. Thus, the optimization method can take a different solution path and produce a different result.
- When you use parallel computing with the `Pattern` search method, the search is more comprehensive and can result in a different solution.

To learn more, see “Parallel Computing with the Pattern Search Method” on page 3-216.

Why do I not see the optimization speedup I expected using parallel computing?

- When you optimize a model that does not have a large number of parameters or does not take long to simulate, you might not see a speedup in the optimization time. In such cases, the overhead associated with creating and distributing the parallel tasks outweighs the benefits of running the optimization in parallel.
- Using the `Pattern` search method with parallel computing might not speed up the optimization time. Without parallel computing, the method stops the search at each iteration when it finds a solution better than the current solution. The candidate solution search is more comprehensive when you use parallel computing. Although the number of iterations might be larger, the optimization without using parallel computing might be faster.

To learn more about the expected speedup, see “Parallel Computing with the Pattern Search Method” on page 3-216.

Why does the optimization using parallel computing not make any progress?

To troubleshoot the problem:

- 1 Run the optimization for a few iterations without parallel computing to see if the optimization progresses.
- 2 Check whether the remote workers have access to all model dependencies. Model dependencies include data variables and files required by the model to run.

To learn more, see “Model Dependencies” on page 3-218.

Why does the optimization using parallel computing not stop when I click the Stop optimization button?

When you use parallel computing with the `Pattern search` method, the software must wait until the current optimization iteration completes before it notifies the workers to stop. The optimization does not terminate immediately when you click **Stop**, and, instead, appears to continue running.

Undesirable Parameter Values

In this section...

“What to do if the optimization drives the tuned compensator elements and parameters to undesirable values?” on page 3-237

“What to do if the optimization violates bounds on parameter values?” on page 3-237

What to do if the optimization drives the tuned compensator elements and parameters to undesirable values?

- When a tuned compensator element or parameter is positive, or when its value is physically constrained to a given range, enter the lower and upper bounds (**Minimum** and **Maximum**) in one of the following:
 - Dialog box to select design variables (in Response optimization tool)
 - **Compensators** pane (in a SISO Design Task)

This information helps guide the optimization method towards a reasonable solution.

- Specify initial guesses that are within the range of desirable values.
- In the **Compensators** pane in a SISO Design Task, verify that no integrators/differentiators are selected for optimization. Optimizing the pole/zero location of integrators/differentiators can result in drastic changes in the system gain and lead to undesirable values.

What to do if the optimization violates bounds on parameter values?

The `Gradient descent` optimization method `fmincon` violates the parameter bounds when it cannot simultaneously satisfy the signal constraints and the bounds. When this happens, try one of the following:

- Specify a different value for the parameter bound and restart the optimization. A guideline is to adjust the bound by 1% of the typical value.

For example, for a parameter with a typical value of 1 and lower bound of 0, change the lower bound to 0.01.

- Relax the signal constraints and restart the optimization. This approach results in a different solution path for the `Gradient descent` method.

- Restart the optimization immediately after it terminates by clicking **Optimize** in the Response Optimization tool. This approach uses the previous optimization results as the starting point for the next optimization cycle to refine the results.
- Use the following two-step approach to perform the optimization:

- 1 Run an initial optimization to satisfy the signal constraints.

For example, run the optimization using the `Simplex` search method. This method satisfies the signal constraints but does not support the bounds on parameter values. The results obtained using this method provide the starting point for the optimization performed in the next step. To learn more about this method, see the `fminsearch` function reference page in the Optimization Toolbox documentation.

- 2 Reconfigure the optimization by selecting a different optimization method to satisfy both the signal constraints and the parameter bounds.

For example, change the optimization method to `Gradient descent` and run the optimization again.


Tip If Global Optimization Toolbox software is installed, you can select the `Pattern search` optimization method to optimize the model response.

Reverting to Initial Parameter Values

How do I quit an optimization and revert to my initial parameter values?

- Before running an optimization, do one of the following:
 - In the Response Optimization tool, click Options. Uncheck **Update model at end of optimization** in the **General Options** tab.
 - In the Response Optimization tool, click Options. Select **Save optimized variable values as new design variable set** in the **General Options** tab.
 - Make a copy of the design variable set in the **Data** area.

If you want to revert to the initial parameter values after the optimization terminates or you stop the optimization by clicking **Stop**, select the design variable that contains

the initial values in the **Design Variable Set** drop-down list and click  adjacent to **Design variables Set**. Select the design variables in the dialog box and click **Update model variable values** to revert the model parameters to their original values.

- When using a SISO Design Task, the **Start Optimization** button becomes a **Stop Optimization** button after the optimization has begun. To quit the optimization, click the **Stop Optimization** button. To revert to the initial parameter values, select **Edit > Undo Optimize compensators** from the menu in the SISO Design Tool window.

Save and Load Optimization Sessions

In this section...
“Structure of an Optimization Session” on page 3-240
“Save a Session” on page 3-240
“Load a Session” on page 3-241

Structure of an Optimization Session

The Response Optimization tool stores and organizes data from a given Simulink model inside a *session*. An optimization session includes the following information:

- Design variables and uncertain variables
- Design requirements
- Optimization results
- Optimization settings
- Plots — Changes to plots layout and plot characteristics, such as axis limits, line colors, are not included.

The default session name is the same as the Simulink model name. The session name displays on the title pane of Response optimization tool.

Save a Session

Saving a session lets you reuse your settings and optimization results later. Each Response Optimization tool session is associated with a Simulink model.

You can save the session as either a MAT-file or workspace variable.

- To save the session as a MAT-file, in the **Response Optimization** tab, in the **Save Session** drop-down list, click **Save to file**. A window opens where you specify the MAT-file name.
- To save the session as a model or MATLAB workspace variable, in the **Save Session** drop-down list, select **Save to model workspace** or **Save to MATLAB workspace**.

Load a Session

To load a previously saved MAT-file or workspace session:

- 1 Open the Response Optimization tool for the model.
- 2 To load a MAT-file, in the **Response Optimization** tab, click the **Open Session** drop-down list, and select **Open from file**. A window opens where you select the MAT-file to load.

To load a workspace variable, select **Open from model workspace** or **Open from MATLAB workspace** in the **Open Session** drop-down list.

See Also

Related Examples

- “Specify Design Variables” on page 3-76
- “Specify Time-Domain Design Requirements in the Tool” on page 3-19
- “Specify Frequency-Domain Design Requirements in the Tool” on page 3-56
- “Specify Optimization Options” on page 3-84

Optimizing Time-Domain Response of Simulink® Models Using Parallel Computing

This example shows how to use parallel computing to optimize the time-domain response of a Simulink® model. You use Simulink® Design Optimization™ and Parallel Computing Toolbox™ to tune the gains of a discrete PI controller of a boiler to meet the design requirements. The example also shows how the software automatically handles model file dependencies.

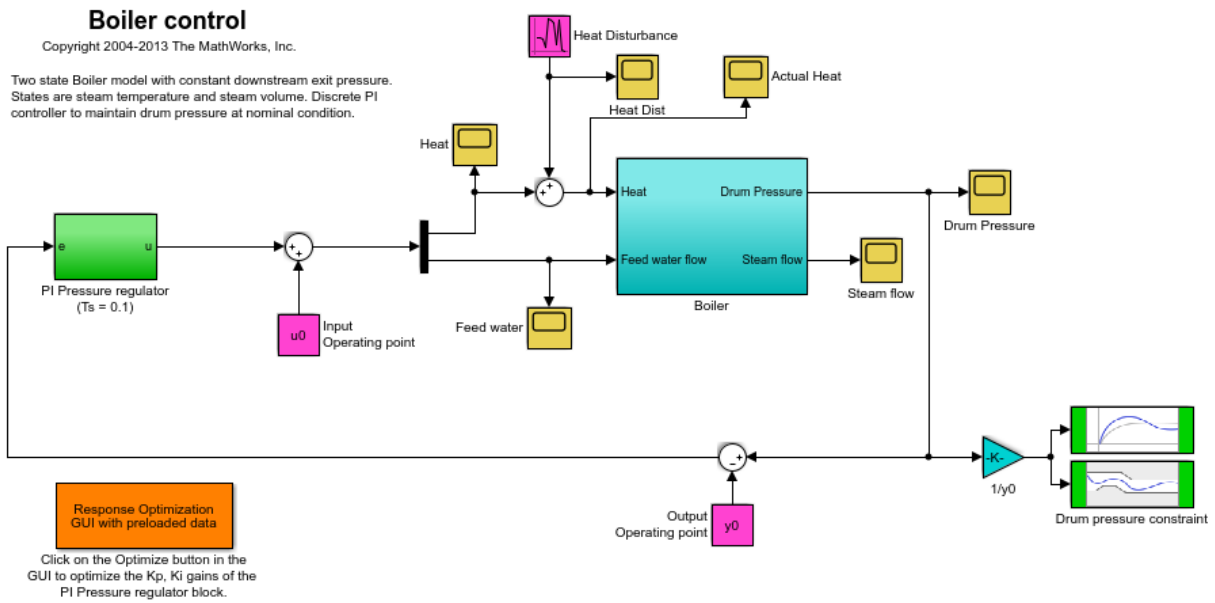
This example requires Parallel Computing Toolbox™.

Opening the Model

The Simulink model consists of a boiler model and a discrete PI controller. When using parallel computing, Simulink Design Optimization performs a model dependency check, which recognizes the boiler model library as an installed library.

In order to illustrate how model dependencies are handled when using parallel computing, we copy the boiler model and library block to a temporary folder before opening the model.

```
pathToLib = boilerpressure_setup;    %Copies boiler model and library to a temporary fol
addpath(pathToLib);
open_system('boilerpressure_demo')
```



Design Requirements

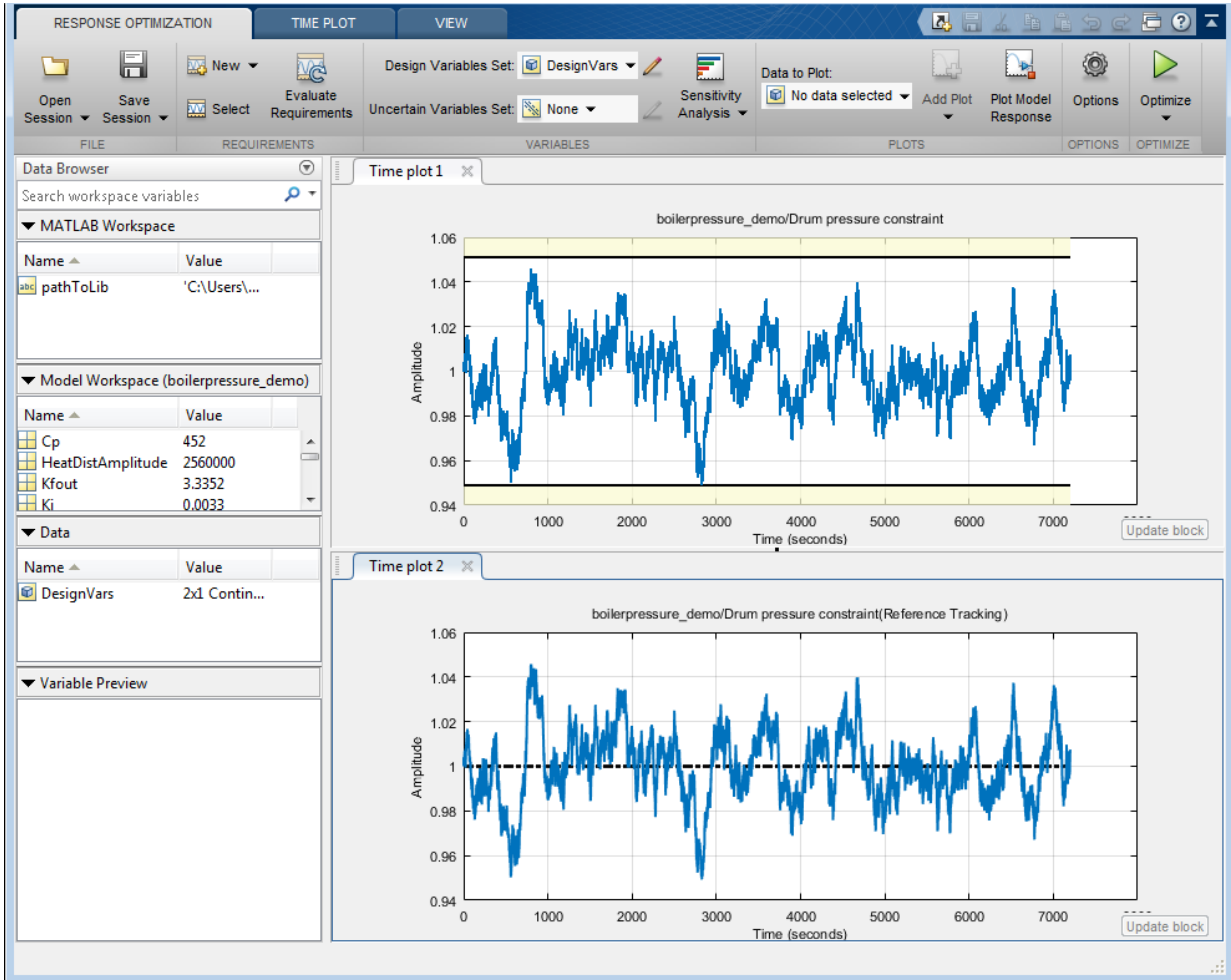
The boiler pressure is regulated by a discrete PI controller. The design requirement for the controller is to limit the pressure variation of the boiler within $\pm 5\%$ of the nominal pressure.

The initial controller has fairly good regulation characteristics but in the presence of additional heat disturbances, modeled by the Heat Disturbance block, we want to tune the controller performance to provide tighter pressure regulation.

Double-click the 'Response Optimization GUI with preloaded data' block in the Simulink model to open a pre-configured Response Optimization Tool. The Response Optimization Tool is configured with:

1. Upper and lower bounds representing a $\pm 5\%$ allowable range on the drum pressure
2. A reference tracking objective to minimize the deviation of the drum pressure from nominal
3. The PI controller gains, K_p and K_i , are selected for tuning

Click **Plot Model Response** to display the drum pressure variations with the initial controller.

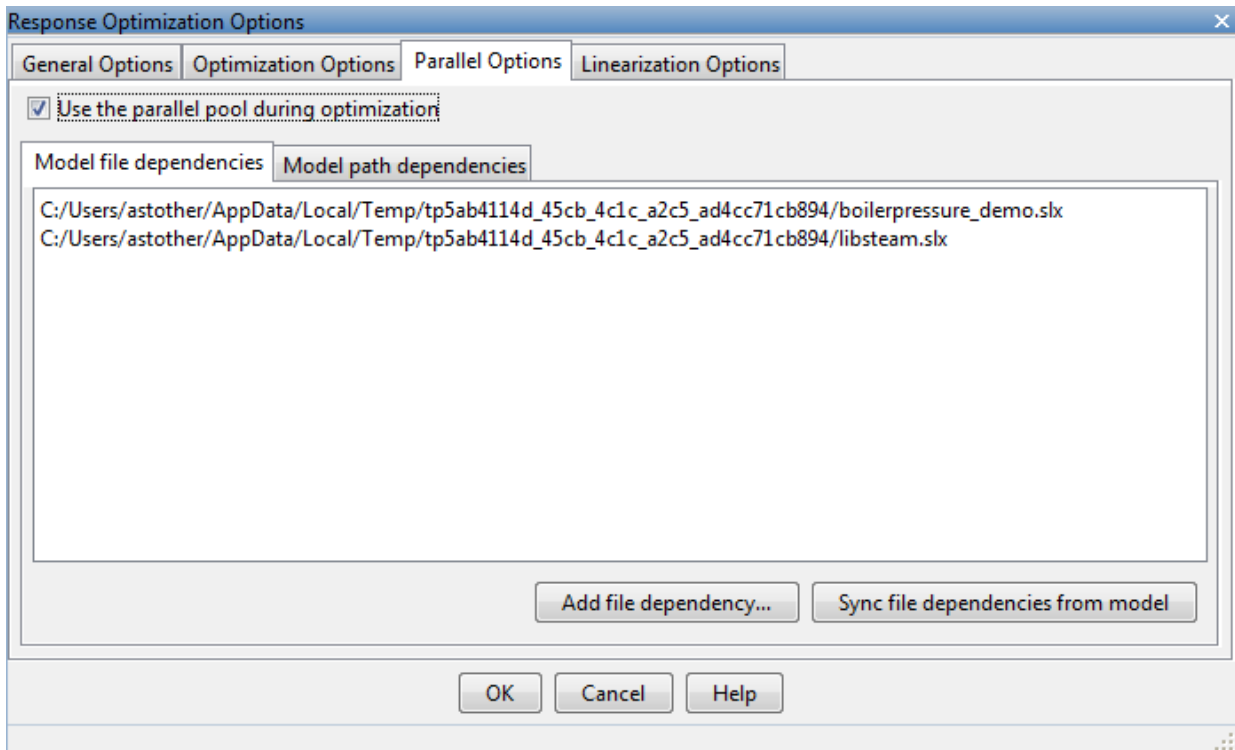


Configuring and Running the Optimization in the GUI Using Parallel Computing

When computing the model response with the initial controller, this complex model took a long time to simulate. Using parallel computing can reduce the optimization time by simulating the model in parallel. For more information on parallel computing and

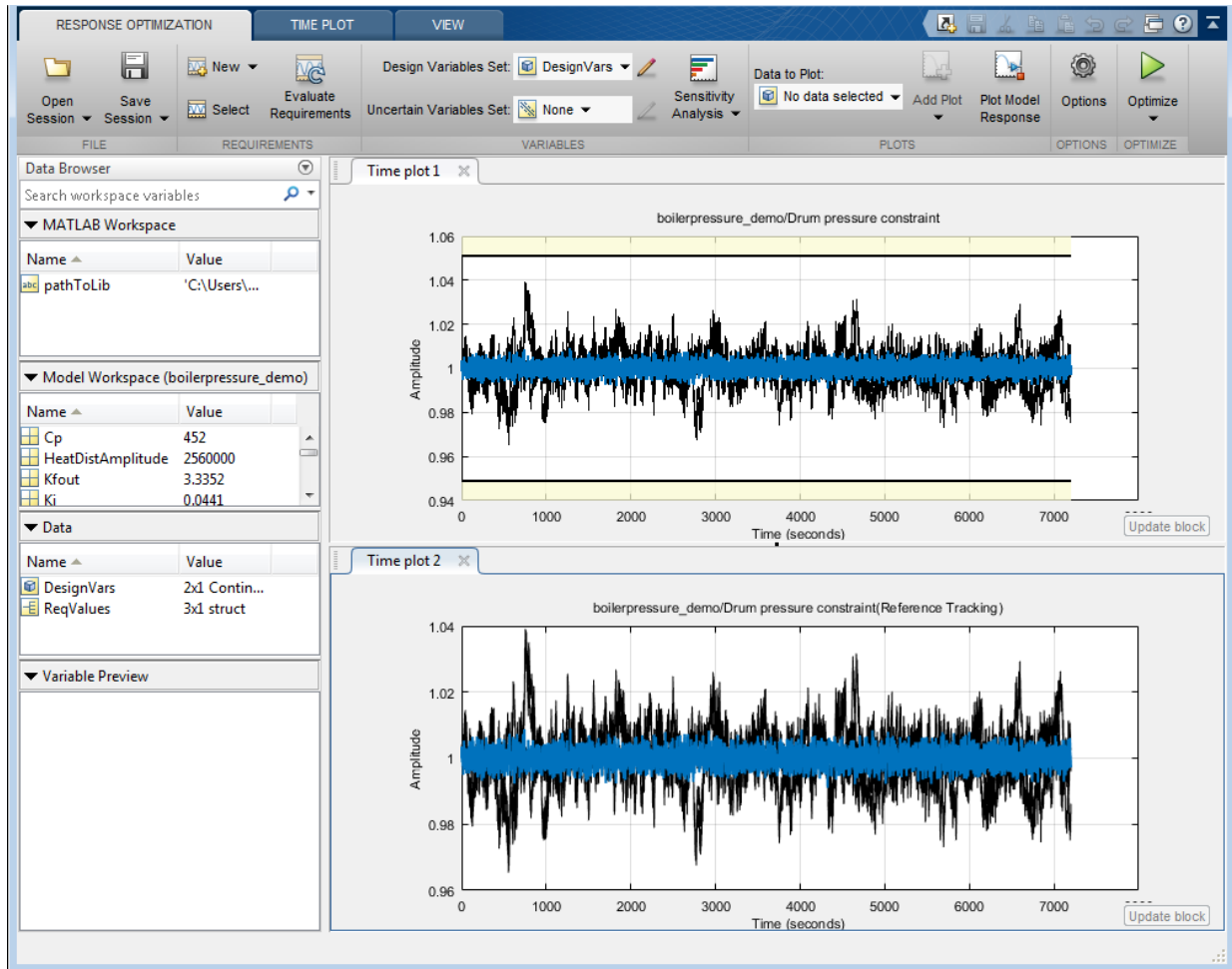
optimization performance see the tutorial "Improving Optimization Performance Using Parallel Computing".

To configure the optimization problem to use parallel computing click **Options** in the Response Optimization Tool and select the **Parallel Options** tab. Select the "Use the parallel pool during optimization" option. This triggers an automated search for any model dependencies. In this example, the Steam table library (`libsteam.slx`) is found as a model dependency (in addition to the `boilerpressure_demo` model itself), and is displayed in the Model file dependencies list box.



Clicking OK configures the optimization to use parallel computing.

To run the optimization click the **Optimize** button. A progress window opens displaying optimization progress and the plots update to show the optimized response.



The final response shows that the optimized regulator tracks the reference pressure much more closely and the drum pressure constraints are satisfied.

Configuring and Running the Optimization at the Command Line

You can also use the command line functions to configure the optimization to use parallel computing and run the optimization.

Select the model variables for optimization and set lower limits

```
p = sdo.getParameterFromModel('boilerpressure_demo',{'Kp','Ki'});
p(1).Minimum = 0.001;
p(2).Minimum = 0.001;
```

Select the model signal to bound and create a simulator to simulate the model.

```
nPressure = Simulink.SimulationData.SignalLoggingInfo;
nPressure.BlockPath = 'boilerpressure_demo/1//y0';
nPressure.OutputPortIndex = 1;
nPressure.LoggingInfo.NameMode = 1;
nPressure.LoggingInfo.LoggingName = 'nPressure';

simulator = sdo.SimulationTest('boilerpressure_demo');
simulator.LoggingInfo.Signals = nPressure;
```

Get the optimization requirements defined by the check blocks in the model so that we can use them in the optimization problem.

```
bnds = getbounds('boilerpressure_demo/Drum pressure constraint');
PressureLimits = [bnds{:}];
bnds = getbounds('boilerpressure_demo/Drum pressure constraint(Reference Tracking)');
PressureRegulation = [bnds{:}];
requirements = struct(...
    'PressureLimits', PressureLimits, ...
    'PressureRegulation', PressureRegulation);
```

Define the function called during optimization. Notice that the function uses the simulator and requirements defined earlier to evaluate the design.

```
evalDesign = @(p) boilerpressure_design(p,simulator,requirements);
type boilerpressure_design

function design = boilerpressure_design(p,simulator,requirements)
%BOILERPRESSURE_DESIGN
%
% The boilerpressure_design function is used to evaluate a boiler
% controller design design.
%
% The |p| input argument is the vector of controller parameters.
%
% The |simulator| input argument is a sdo.SimulinkTest object used to
% simulate the |boilerpressure_demo| model and log simulation signals.
%
% The |requirements| input argument contains the design requirements used
```

```
% to evaluate the boiler controller design.
%
% The |design| return argument contains information about the design
% evaluation that can be used by the |sdo.optimize| function to optimize
% the design.
%
% see also sdo.optimize, sdoExampleCostFunction

% Copyright 2011 The MathWorks, Inc.

%% Simulate the model
%
% Use the simulator input argument to simulate the model and log model
% signals.
%
% First ensure that we simulate the model with the parameter values chosen
% by the optimizer.
%
simulator.Parameters = p;
%%
% Simulate the model and log signals.
%
simulator = sim(simulator);
%%
% Get the simulation signal log, the simulation log name is defined by the
% model |SignalLoggingName| property
%
logName = get_param('boilerpressure_demo','SignalLoggingName');
simLog = get(simulator.LoggedData,logName);

%% Evaluate the design requirements
%
% Use the requirements input argument to evaluate the design requirements
%
% Check the Pressure signal against the |PressureLimits| requirements.
%
nPressure = get(simLog,'nPressure');
c = [...
    evalRequirement(requirements.PressureLimits(1),nPressure.Values); ...
    evalRequirement(requirements.PressureLimits(2),nPressure.Values)];
%%
% Use the PressureLimits requirements as non-linear constraints for
% optimization.
design.Cleq = c(:);
```

```
%%
% Check the pressure signal against the |PressureRegulation| requirement.
%
f = evalRequirement(requirements.PressureRegulation,nPressure.Values);
%%
% Use the PressureRegulation requirement as an objective for optimization.
design.F = f;
end
```

Setup optimization options to use the parallel pool and specify the model and model files dependencies.

```
opt = sdo.OptimizeOptions;
opt.UseParallel = true;
opt.OptimizedModel = 'boilerpressure_demo';
[dirs,files] = sdo.getModelDependencies('boilerpressure_demo');
opt.ParallelFileDependencies = files;
```

To run the optimization using the parallel pool pass the optimization options, `opt`, to the `sdo.optimize` command.

```
>> [pOpt,info] = sdo.optimize(evalDesign,p,opt);
Starting parallel pool (parpool) using the 'local' profile ... connected to 4 workers.
Configuring parallel workers for optimization...
Parallel workers configured for optimization.
```

```
Optimization started 18-Nov-2014 10:31:21
```

Iter	F-count	f(x)	max constraint	Step-size	First-order optimality
0	5	17.5068	0		
1	10	11.6563	0	1.25	32.2
2	15	8.32632	0	1.27	17.4
3	20	0.694611	0	70.1	0.0391
4	25	0.562489	0	4.3	0.0355
5	31	0.543708	0	2.79	0.0316
6	36	0.543708	0	0.0515	0.0316

```
Local minimum possible. Constraints satisfied.
```

```
fmincon stopped because the size of the current step is less than
the selected value of the step size tolerance and constraints are
satisfied to within the selected value of the constraint tolerance.
Removing data from parallel workers...
Data removed from parallel workers.
```

Closing the Model

After the model is optimized, we remove the boiler model and library file from the temporary folder.

```
bdclose('boilerpressure_demo')
rmpath(pathToLib)
boilerpressure_cleanup(pathToLib)
```

Design Optimization to Meet Frequency-Domain Requirements (Code)

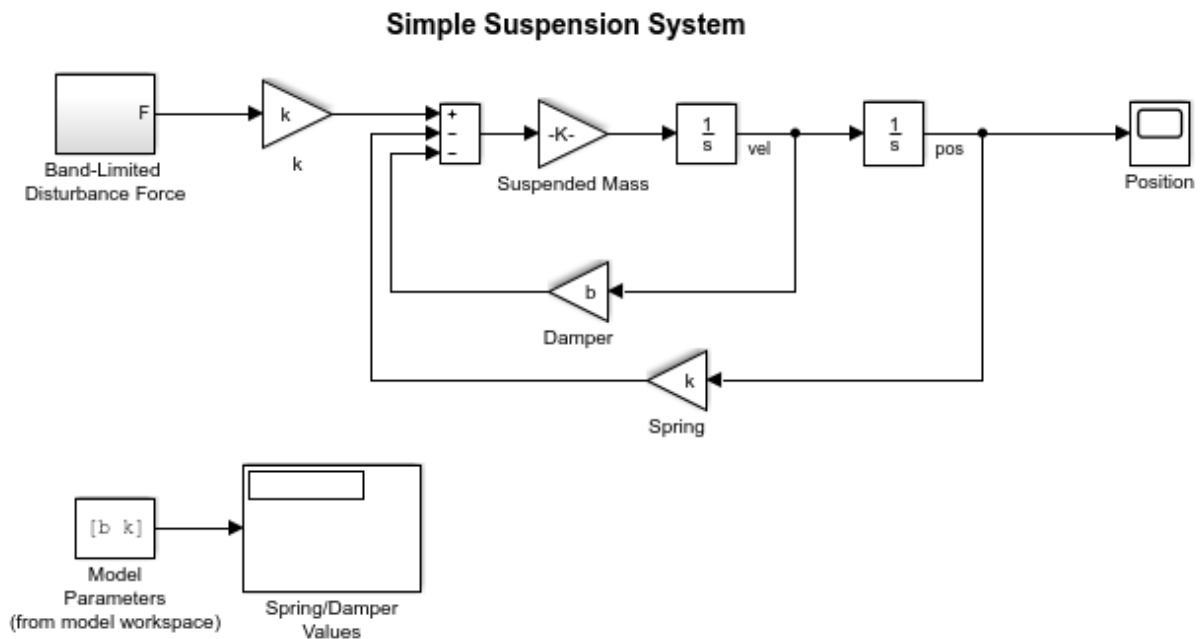
This example shows how to tune model parameters to meet frequency-domain requirements, using the `sdo.optimize` command.

This example requires Simulink® Control Design™.

Suspension Model

Open the Simulink Model.

```
open_system('sdoSimpleSuspension')
```



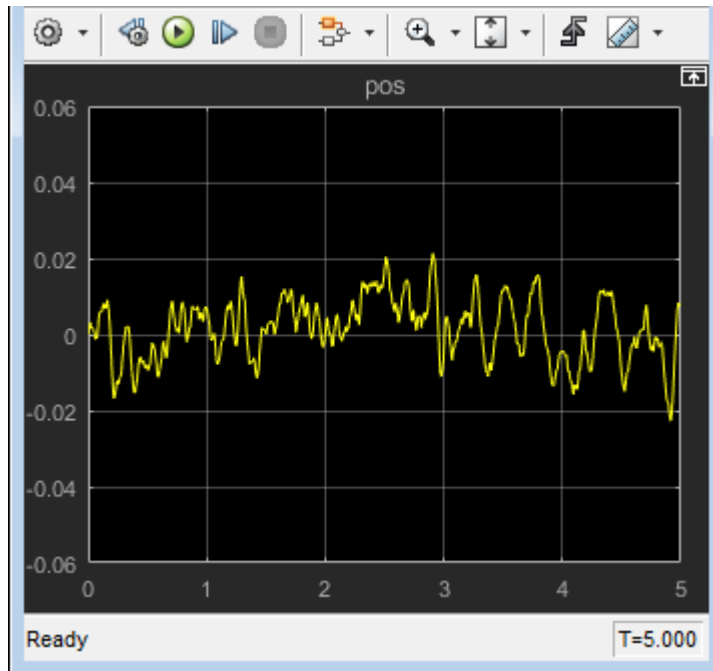
Copyright 2002-2015 The MathWorks, Inc.

Mass-spring-damper models represent simple suspension systems and for this example we tune the system to meet typical suspension requirements. The model implements the

second order system representing a mass-spring-damper using Simulink blocks and includes:

- a Mass gain block parameterized by the total suspended mass, m_0+m_{Load} . The total mass is the sum of a nominal mass, m_0 , and a variable load mass, m_{Load} .
- a Damper gain block parameterized by the damping coefficient, b .
- a Spring gain block parameterized by the spring constant, k .
- two integrator blocks to compute the mass velocity and position.
- a Band-Limited Disturbance Force block applying a disturbance force to the mass. The disturbance force is assumed to be band-limited white noise.

Simulate the model to view the system response to the applied disturbance force.



Design Problem

The initial system has a bandwidth that is too high. This can be seen from the spiky position signal. You tune the spring and damper values to meet the following requirements:

- The -3dB system bandwidth must not exceed 10 rad/s.
- The damping ratio of the system must be less than $1/\sqrt{2}$. This ensures that no frequencies in pass band are amplified by the system.
- Minimize the expected failure rate of the system. The expected failure rate is described by a Weibull distribution dependent on the mass, spring, and damper values.
- These requirements must all be satisfied as the load mass ranges from 0 to 20 kg.

Specify Design Variables

```
DesignVars = sdo.getParameterFromModel('sdoSimpleSuspension',{'b','k'});
DesignVars(1).Minimum = 100;
DesignVars(1).Maximum = 10000;
DesignVars(2).Minimum = 10000;
DesignVars(2).Maximum = 100000;
```

Specify Uncertain Variables

Specify `mLoad`, the load mass, as an uncertain variable. This will ensure the optimal solution is robust to variations in load mass.

```
UncVars = sdo.getParameterFromModel('sdoSimpleSuspension','mLoad');
UncVars_Values = {...
    10; ...
    20};
```

Specify Design Requirements

Specify design requirements to satisfy during optimization.

```
Requirements = struct;
Requirements.Bandwidth = sdo.requirements.BodeMagnitude(...
    'BoundFrequencies', [10 100], ...
    'BoundMagnitudes', [-3 -3]);
Requirements.DampingRatio = sdo.requirements.PZDampingRatio;
```

The reliability requirement will be specified in the optimization objective function described below. The reliability requirement is used to tune the spring and damper values to minimize the expected failure rate over a lifetime of 100e3 miles. The failure rate is computed using a Weibull distribution on the damping ratio of the system. As the damping ratio increases, the failure rate is expected to increase.

Linearization Definition

The design requirements use the bandwidth and damping ratio of the system, these frequency domain characteristics require linearizing the model. Create a simulator for the model and use the simulator to compute the linear systems used by the requirements.

```
Simulator = sdo.SimulationTest('sdoSimpleSuspension');
```

Specify linear systems to compute by specifying the linearization inputs and outputs of the system. The linear system input is the output of the Band-Limited Disturbance Force block and the linear system output is the output of the `x_dot` block (the position signal).

```
IOs(1) = linio('sdoSimpleSuspension/Band-Limited Disturbance Force',1,'input');  
IOs(2) = linio('sdoSimpleSuspension/x_dot',1,'output');
```

Add the linearization IOs to a `sdo.SystemLoggingInfo` object with a logging name, and linearization snapshot time. In this case, the snapshot time is set to 0, the model initial condition.

```
sys1 = sdo.SystemLoggingInfo;  
sys1.Source = 'IOs';  
sys1.LoggingName = 'IOs';  
sys1.LinearizationIOs = IOs;  
sys1.SnapshotTimes = 0;
```

Add the system logging info to the simulator. When the simulator is used to run the model, the linear system defined by the specified linearization IOs is computed and added to the simulation log with the specified logging name.

```
Simulator.SystemLoggingInfo = sys1;
```

Create Optimization Objective Function

Create a function that is called at each optimization iteration to evaluate the design requirements.

Use an anonymous function with one argument that calls `sdoSimpleSuspension_Design`. The `sdoSimpleSuspension_Design` function has arguments for the design parameters, the simulator, the design requirements, the uncertain variables, and uncertain variable values.

```
optimfcn = @(P) sdoSimpleSuspension_Design(P, Simulator, Requirements, UncVars, UncVars_Val
type sdoSimpleSuspension_Design
```

```
function Vals = sdoSimpleSuspension_Design(P, Simulator, Requirements, UncVars, UncVars_Val
%SDOSIMPLESUSPENSION_DESIGN
%
% The sdoSimpleSuspension_Design function is used to evaluate a simple
% suspension system design.
%
% The |P| input argument is the vector of suspension design parameters.
%
% The |Simulator| input argument is a sdo.SimulinkTest object used to
% simulate the |sdoSimpleSuspension| model and log simulation signals.
%
% The |Requirements| input argument contains the design requirements
% used to evaluate the suspension design.
%
% The |UncVars| and |UncVars_Values| arguments specify the uncertain
% variable and uncertain variable values.
%
% The |Vals| return argument contains information about the design
% evaluation that can be used by the |sdo.optimize| function to optimize
% the design.
%
% See also sdoSimpleSuspension_cmddemo

% Copyright 2015 The MathWorks, Inc.

%% Evaluate the suspension reliability
%
%Get the spring and damper design values
allVarNames = {P.Name};
idx         = strcmp(allVarNames, 'k');
k           = P(idx).Value;
idx         = strcmp(allVarNames, 'b');
b           = P(idx).Value;

%Get the nominal mass from the model workspace
```

```
wksp = get_param('sdoSimpleSuspension','ModelWorkspace');
m     = evalin(wksp,'m0');

%The expected failure rate is defined by the Weibull cumulative
%distribution function, 1-exp(-(x/l)^k), where k=3, l is a function of the
%mass, spring and damper values, and x the lifetime.
d     = b/2/sqrt(m*k);
pFailure = 1-exp(-(100e3*d/250e3)^3);

%% Nominal Evaluation
%
% Evaluate the model and requirements with uncertain parameters set to their
% nominal values.

% Simulate the model.
Simulator.Parameters = [P(:);UncVars(:)];
Simulator = sim(Simulator);

% Retrieve logged linearizations.
Sys = find(Simulator.LoggedData,'IOs');

% Evaluate the design requirements.
Cleq_Bandwidth = evalRequirement(Requirements.Bandwidth, Sys.values);
Cleq_DampingRatio = evalRequirement(Requirements.DampingRatio, Sys.values);

%% Uncertain Evaluation
%
% Evaluate the model and requirements for all combinations of uncertain
% parameter values.
for ct=1:size(UncVars_Values,1)
    UncVars(1).Value = UncVars_Values{ct,1};

    % Simulate the model.
    Simulator.Parameters = [P(:);UncVars(:)];
    Simulator = sim(Simulator);

    % Retrieve logged linearizations.
    Sys = find(Simulator.LoggedData,'IOs');

    % Evaluate the design requirements.
    Cleq_Bandwidth_UncVars = evalRequirement(Requirements.Bandwidth, Sys.values);
    Cleq_DampingRatio_UncVars = evalRequirement(Requirements.DampingRatio, Sys.values);

    Cleq_Bandwidth = [Cleq_Bandwidth; Cleq_Bandwidth_UncVars]; %#ok<AGROW>
```

```

    Cleq_DampingRatio = [Cleq_DampingRatio; Cleq_DampingRatio_UncVars]; %#ok<AGROW>
end

%% Return Values.
%
% Collect the evaluated design requirement values in a structure to
% return to the optimization solver.
Vals.F = pFailure;
Vals.Cleq = [...
    Cleq_Bandwidth(:); ...
    Cleq_DampingRatio(:)];
end

```

Optimization Options

Specify optimization options.

```

Options = sdo.OptimizeOptions;
Options.MethodOptions.Algorithm = 'active-set';
Options.MethodOptions.TolGradCon = 1e-06;

```

Optimize the Design

Call `sdo.optimize` with the objective function handle, parameters to optimize, and options.

```
[Optimized_DesignVars, Info] = sdo.optimize(optimfcn, DesignVars, Options);
```

Optimization started 15-Aug-2017 01:23:23

Iter	F-count	f(x)	max constraint	Step-size	First-order optimality
0	5	0.0619897	0.7642		
1	10	0.351266	-0.1277	0.461	1.22
2	15	0.189345	-0.03881	0.188	0.312
3	20	0.0829063	0.01312	0.51	0.425
4	24	0.0365398	0	0.308	1.55
5	28	0.0294977	0	0.0143	0.733
6	32	0.0293065	0	0.000417	0.00142

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.

Related Examples

To learn how to optimize the suspension system using the Response Optimization tool, see "Design Optimization to Meet Frequency-Domain Requirements (GUI)".

Close the model.

```
bdclose('sdoSimpleSuspension')
```

Sensitivity Analysis

- “What is Sensitivity Analysis?” on page 4-2
- “Specify Parameters for Design Exploration” on page 4-4
- “Generate Parameter Samples for Sensitivity Analysis” on page 4-10
- “Specify Time-Domain Requirements” on page 4-28
- “Specify Parameters Requirements” on page 4-50
- “Specify Frequency-Domain Requirements” on page 4-66
- “Evaluate Design Requirements” on page 4-83
- “Analyze Relation Between Parameters and Design Requirements” on page 4-91
- “Use Sensitivity Analysis to Configure Estimation and Optimization” on page 4-99
- “Interact with Plots in the Sensitivity Analysis Tool” on page 4-106
- “Validate Sensitivity Analysis” on page 4-129
- “Store Intermediate Data in the Tool” on page 4-135
- “Use Parallel Computing for Sensitivity Analysis” on page 4-137
- “Use Fast Restart Mode During Sensitivity Analysis” on page 4-144
- “Design Exploration Using Parameter Sampling (GUI)” on page 4-147
- “Identify Key Parameters for Estimation (GUI)” on page 4-168
- “Explore Design Reliability Using Parameter Sampling (GUI)” on page 4-186
- “Design Exploration Using Parameter Sampling (Code)” on page 4-200
- “Identify Key Parameters for Estimation (Code)” on page 4-217

What is Sensitivity Analysis?

Sensitivity analysis is defined as the study of how uncertainty in the output of a model can be attributed to different sources of uncertainty in the model input[1]. In the context of using Simulink Design Optimization software, sensitivity analysis refers to understanding how the parameters and states (optimization design variables) of a Simulink model influence the optimization cost function. Examples of using sensitivity analysis include:

- Before optimization — Determine the influence of the parameters of a Simulink model on the output. Use sensitivity analysis to rank parameters in order of influence, and obtain initial guesses for parameters for estimation or optimization.
- After optimization — Test how robust the cost function is to small changes in the values of optimized parameters.

One approach to sensitivity analysis is local sensitivity analysis, which is derivative based (numerical or analytical). Mathematically, the sensitivity of the cost function with respect to certain parameters is equal to the partial derivative of the cost function with respect to those parameters. The term *local* refers to the fact that all derivatives are taken at a single point. For simple cost functions, this approach is efficient. However, this approach can be infeasible for complex models, where formulating the cost function (or the partial derivatives) is nontrivial. For example, models with discontinuities do not always have derivatives.

Local sensitivity analysis is a one-at-a-time (OAT) technique. OAT techniques analyze the effect of one parameter on the cost function at a time, keeping the other parameters fixed. They explore only a small fraction of the design space, especially when there are many parameters. Also, they do not provide insight about how the interactions between parameters influence the cost function.

Another approach to sensitivity analysis is global sensitivity analysis, often implemented using Monte Carlo techniques. This approach uses a representative (global) set of samples to explore the design space. Use Simulink Design Optimization software to perform global sensitivity analysis using the Sensitivity Analysis tool, or at the command line. The workflow is as follows:

- 1 Sample the model parameters using experimental design principles. That is, for each parameter, generate multiple values that the parameter can assume. Define the parameter sample space by specifying probability distributions for each parameter. You can also specify parameter correlations.

For information about sampling parameters, see “Generate Parameter Samples for Sensitivity Analysis” on page 4-10.

- 2 Define a cost function by creating a design requirement on the model signals.
- 3 Evaluate the requirement (cost function) at each combination of parameter values using Monte Carlo simulations. You can plot the cost function output for the samples to visually analyze trends.
- 4 (Optional) Formally analyze the relation between the evaluated requirement and the samples. Analysis methods include correlation, partial correlation (requires Statistics and Machine Learning Toolbox™ software), and standardized regression. You can configure each analysis method to use either raw or ranked data.

For information about the analysis methods, see “Analyze Relation Between Parameters and Design Requirements” on page 4-91.

References

- [1] Saltelli, A., Ratto, M., Andres, T., Campolongo, F., Cariboni, J., Gatelli, D., Saisana, M., and Tarantola, S. *Global Sensitivity Analysis. The Primer*, John Wiley and Sons, 2008.

See Also

`sdo.analyze` | `sdo.evaluate` | `sdo.sample` | `sdo.scatterPlot`

Related Examples

- “Design Exploration Using Parameter Sampling (GUI)” on page 4-147
- “Design Exploration Using Parameter Sampling (Code)” on page 4-200
- “Identify Key Parameters for Estimation (GUI)” on page 4-168
- “Identify Key Parameters for Estimation (Code)” on page 4-217

More About

- “Generate Parameter Samples for Sensitivity Analysis” on page 4-10
- “Analyze Relation Between Parameters and Design Requirements” on page 4-91
- “Validate Sensitivity Analysis” on page 4-129

Specify Parameters for Design Exploration

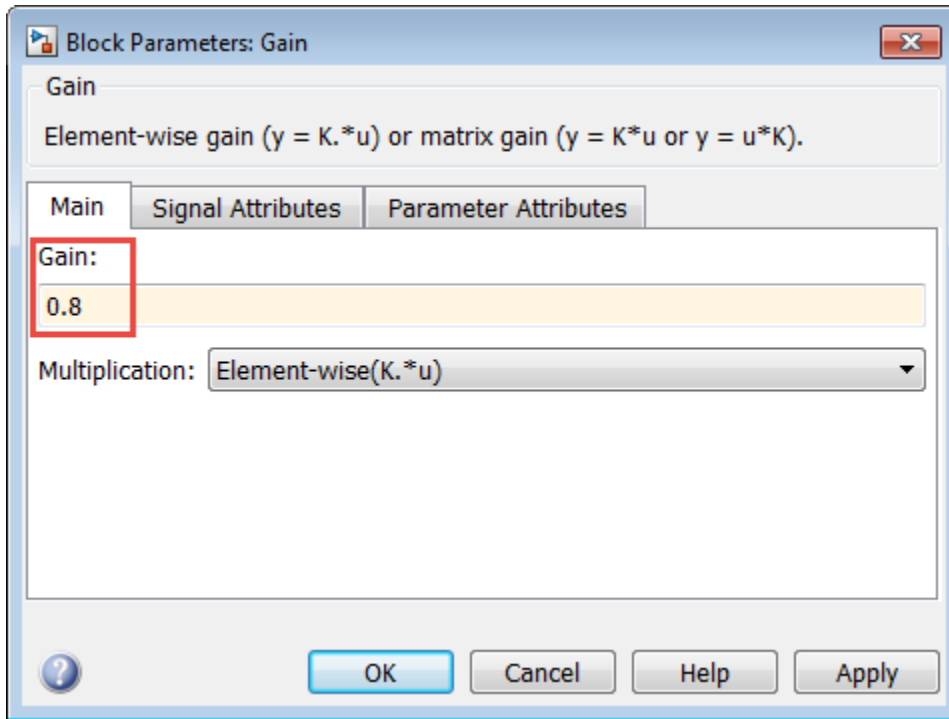
In this section...
“Add Model Parameters as Variables” on page 4-4
“Select Parameters for Design Exploration” on page 4-7

This topic shows how to select parameters of a Simulink model for design exploration in the Sensitivity Analysis tool. After you select the parameters, you generate parameter samples on page 4-10 by varying the parameter values in a specific range, and evaluate your design requirements for each combination of parameter values.

Add Model Parameters as Variables

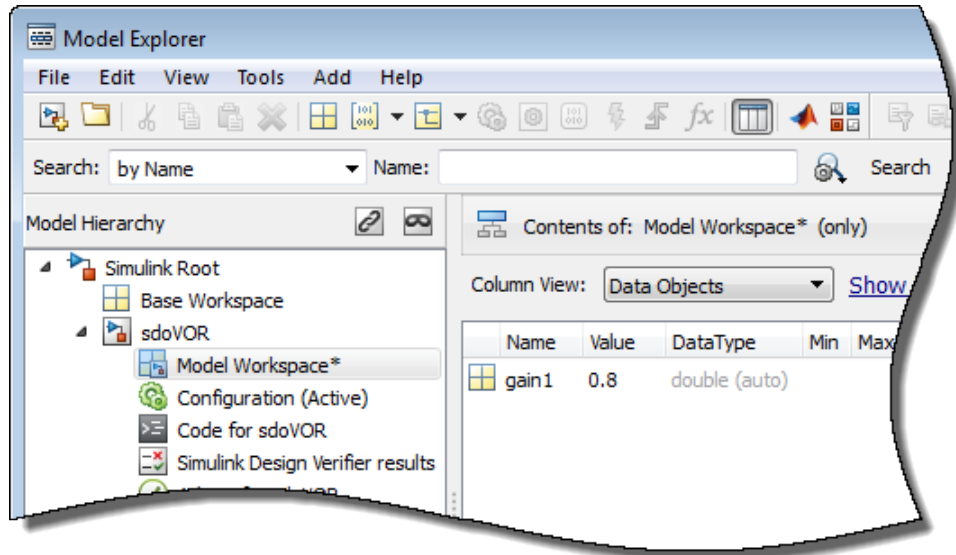
The software can only evaluate variables that are in use by the model. Create variables in the MATLAB or model workspace, and specify your Simulink model or block parameters using these variables. If you have already specified model parameters as variables, select the parameters in the tool on page 4-7.

In this figure, the **Gain** parameter of a Gain block is specified as a numerical value.



To evaluate design requirements using the **Gain** parameter, specify it as variable `gain1`:

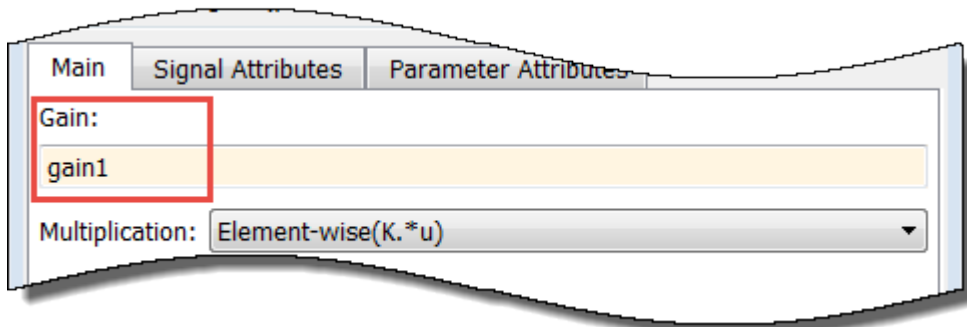
- 1 Create the variable `gain1` in one of the following ways:
 - Add the variables to the model workspace (Simulink), and specify initial values.



- Write initialization code in the **PreloadFcn** callback of the model. For more information, see “Model Callbacks” (Simulink).

```
gain1 = 0.8
```

- 2 Specify the block parameter as variable `gain1` in the Gain block dialog box.



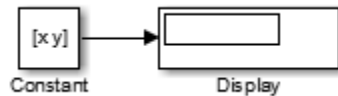
You can now select `gain1` for evaluation. See, “Select Parameters for Design Exploration” on page 4-7.

Specify Independent Parameters

You can also specify independent parameters that do not appear explicitly in the model as variables. However, you cannot use this workflow with Simulink fast restart.

Suppose that a model parameter K_{int} is related to independent parameters x and y such that $K_{int} = x+y$. To add x and y instead of K_{int} :

- Create the independent variables x and y by adding them to the model workspace and specifying initial values.
- The software only allows evaluation of variables that are used by model blocks. To ensure that the software detects x and y for evaluation, add a Constant block to your model, and specify the **Constant value** of the block as $[x \ y]$. Connect the block to a Display block.



- Write code in the **InitFcn** callback of the model that defines the relationship between K_{int} , x , and y . You must first use the `get_param` function to get the variables x and y from the model workspace before you can use them to define K_{int} .

```
wks = get_param(gcs, 'ModelWorkspace')
x = evalin(wks, 'x')
y = evalin(wks, 'y')
Kint = x+y;
```

You can now select x and y for design exploration.

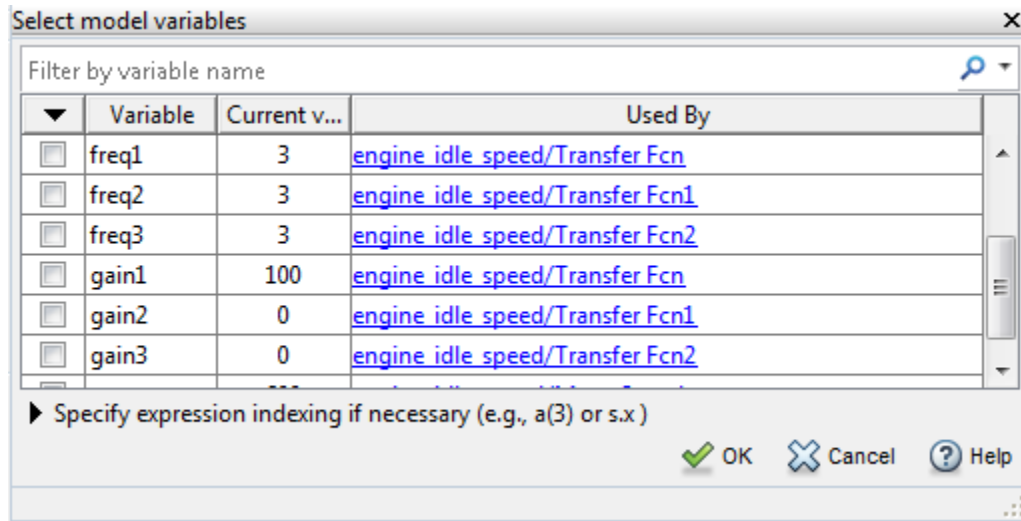
Select Parameters for Design Exploration

In the Sensitivity Analysis tool, in the **Sensitivity Analysis** tab, click **Select Parameters** to open the Select model variables dialog box.

Use this dialog box to select parameters to vary. The table lists the variables that the model uses to set block parameter values. The variables can reside in the model workspace, the base workspace, or a data dictionary.

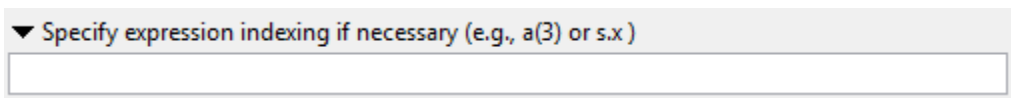
Select variables by clicking the check box next to each variable. If your model contains many variables, filter the list by typing in the **Filter by variable name** field. The **Used**

By column lists all blocks in the model that use the variable. When a variable is used in more than one block, all blocks are listed. To highlight blocks in the model that use the variable, click the block name.



The variables that you select must have a numeric value that uses the data type double. If the value of a variable is not a double number, use these techniques:

- To select a single element or a subset of a matrix or array variable, click **Specify expression indexing if necessary**.



Enter an expression such as `myArray(2)`, which selects the second element of an array variable `myArray`.

After you type the expression, press the **Enter** key to add the variable to the list of model variables.

- To use a variable of a numeric data type other than double, convert the variable to a `Simulink.Parameter` object, which separates a parameter value from its data type. Set the `Value` property to a default double number, and use the `Data Type` property to control the data type.

- To use the value of a `Simulink.Parameter` object, specify the `Value` property. Enter the expression `myParamObj.Value`.
- To use a numeric field of a structure, enter `myStruct.PID.P1`. If you store the structure in a `Simulink.Parameter` object, enter `myStruct.Value.PID.P1`.
- To use one cell of a cell array, enter `myCells{3}`.

You cannot use mathematical expressions such as $a + b$. Sometimes, models have parameters that are not explicitly defined in the model itself. For example, a gain `k` could be defined in the MATLAB workspace as `k = a + b`, where `a` and `b` are not defined in the model but `k` is used. To add these independent parameters, see “Add Model Parameters as Variables” on page 4-4.

See Also

Related Examples

- “What is Sensitivity Analysis?” on page 4-2
- “Generate Parameter Samples for Sensitivity Analysis” on page 4-10

Generate Parameter Samples for Sensitivity Analysis

This topic shows how to generate parameter samples for sensitivity analysis.

You can perform global sensitivity analysis using Simulink Design Optimization software. Using techniques such as design of experiments (DOE) (also referred to as experimental design), you can choose a parameter set for sensitivity analysis.

You generate parameter samples by varying the value of the Simulink model parameters and states of interest according to a specified probability distribution. These parameters and states are collectively referred to as parameters. Each combination of generated parameter values is referred to as a sample or sample point. A collection of samples is referred to as a design space, sample space, or parameter set.

After generating a parameter set, you define a cost function by creating design requirements on the model signals. You then evaluate the cost function for each sample in the parameter set. Then, you analyze the relation between the parameters and requirement to understand how the parameters influence the cost function.

You can generate two kinds of parameter values: random parameter values on page 4-10 or gridded parameter values on page 4-20.

Generate Random Parameter Values

When generating random parameter values, you specify the following characteristics of the parameter space:

- “Number of Samples” on page 4-10
- “Sampling Method” on page 4-11
- “Probability Distribution” on page 4-13
- “Parameter Correlations” on page 4-15

You can specify the characteristics of the parameter space either in the Sensitivity Analysis tool on page 4-15 or at the command line on page 4-18.

Number of Samples

Choose enough samples to yield useful results. However, each model evaluation has a computational expense and can be time intensive. As the number of parameters increases, the number of samples required to explore the design space generally

increases. For correlation or regression analysis, consider using $10Np$ samples, where Np is the number of parameters.

Sampling Method

Specify the method used to generate the samples. You can choose from the following methods:

- **Random** — Random samples are drawn from the probability distributions specified for the parameters.

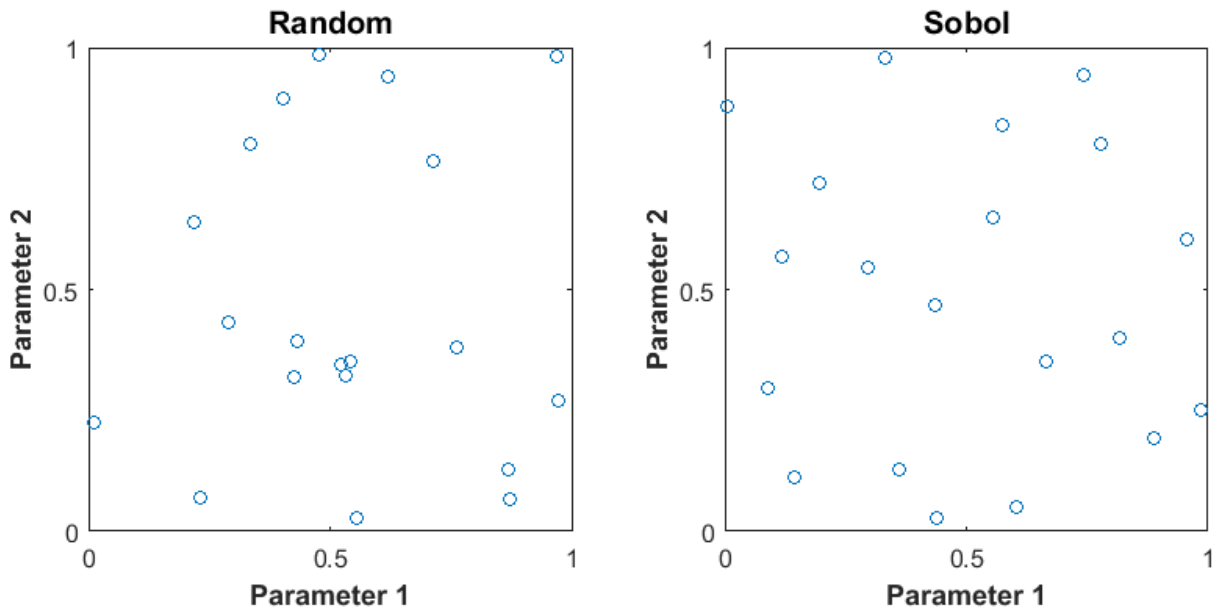
If you specify correlation between parameters, the software uses the Iman-Conover algorithm to impose the parameter correlations.

- **Latin hypercube**— Latin hypercube samples are drawn from the probability distributions specified for the parameters. Use this option for a more systematic space-filling approach than random sampling. The Sobol and Halton quasirandom sampling methods are more space-filling than the Latin hypercube method.

If you specify correlation between parameters, the software uses the Iman-Conover algorithm to impose the parameter correlations.

- **Sobol** — Requires Statistics and Machine Learning Toolbox software. Sobol quasirandom sequences are drawn from the probability distributions specified for the parameters. Use this method for highly systematic space-filling.

The figure shows 20 samples for two parameters. The samples are generated from a uniform distribution, in the interval from 0 to 1. Random sampling can result in large gaps between some samples, and close clustering of other samples. Sobol and Halton quasirandom sampling methods avoid gaps and clustering of samples. If you have many parameters in your parameter set, Sobol sets gives more systematic space filling than Halton quasirandom sets. For more information, see “Generating Quasi-Random Numbers” (Statistics and Machine Learning Toolbox).



If you specify correlation between parameters, the software uses the Iman-Conover algorithm to impose the parameter correlations.

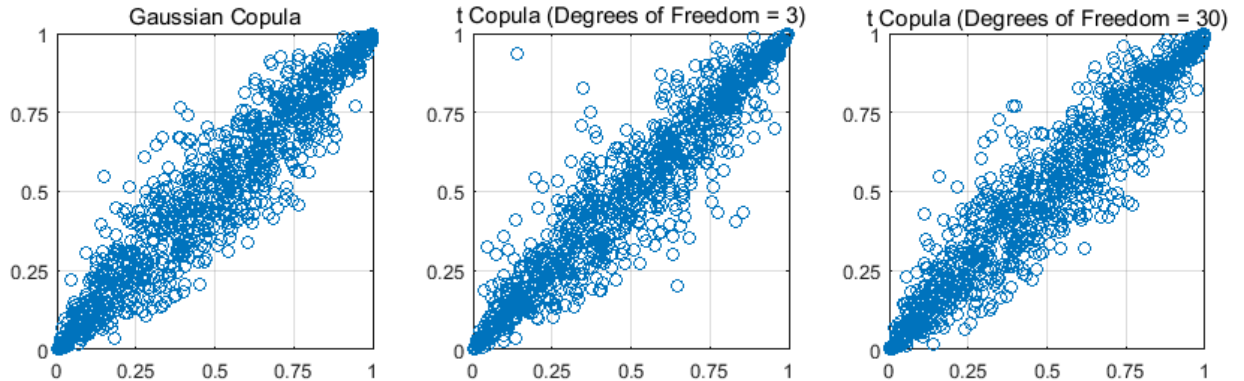
- **Halton** — Requires Statistics and Machine Learning Toolbox software. Halton quasirandom sequences are drawn from the probability distributions specified for the parameters. Like the Sobol method, you can use Halton method for highly systematic space-filling. However, Sobol method gives more systematic space filling if you have many parameters in your parameter set. For more information, see “Generating Quasi-Random Numbers” (Statistics and Machine Learning Toolbox).

If you specify correlation between parameters, the software uses the Iman-Conover algorithm to impose the parameter correlations.

- **Copula**— Requires Statistics and Machine Learning Toolbox software. Random samples are drawn from a copula. Use this option to impose correlations between the parameters using copulas.

You can use either a Gaussian copula (default) or a t copula. Use t copulas when the probability of extreme parameter values is not negligible (distribution is heavy-tailed), and specify the degrees of freedom. As you increase the degrees of freedom, the t copula converges to the Gaussian copula, and the probability of extreme

parameter values becomes negligible. The following figure shows 1000 samples drawn for two parameters in the interval from 0 to 1 using the Gaussian and t copulas.



In comparison to the Gaussian copula, the t copula has more samples that represent the extreme values of the parameters. As the degrees of freedom are increased, the t copula converges to the Gaussian copula.

Specify the correlation type as either Spearman's rank correlation or Kendall's rank correlation.

Probability Distribution

Specify the probability distribution function and related distribution characteristics for each parameter. Use your knowledge of the system (empirical or theoretical) to choose the probability distributions.

Note Simulink Design Optimization software allows you to specify uniform (default), normal, multinomial, piecewise linear, and triangular distributions. For other distributions, you need Statistics and Machine Learning Toolbox software.

Consider the following characteristics of your parameters when choosing a distribution:

Parameter Characteristics	Applicable Distributions
Extends from $-\infty$ to ∞	<ul style="list-style-type: none"> • Normal • Extreme value • Generalized extreme value — Single-ended or from $-\infty$ to ∞, depending on distributional parameter values. • Logistic — heavy tailed compared to normal distribution. • t location-scale — heavy tailed compared to normal distribution.
Bounded at both ends	<ul style="list-style-type: none"> • Uniform • Beta • Binomial — discrete distribution • Multinomial — discrete distribution • Piecewise linear • Triangular

Parameter Characteristics	Applicable Distributions
Extends from 0 to ∞	<ul style="list-style-type: none"> • Birnbaum-Saunders • Burr • Exponential • Gamma • Generalized extreme value — Single-ended or from $-\infty$ to ∞, depending on parameter values. • Inverse Gaussian • Log-logistic • Log-normal • Nakagami • Negative binomial— discrete distribution • Poisson— discrete distribution • Rayleigh • Rician • Weibull
Custom distribution	Piecewise linear

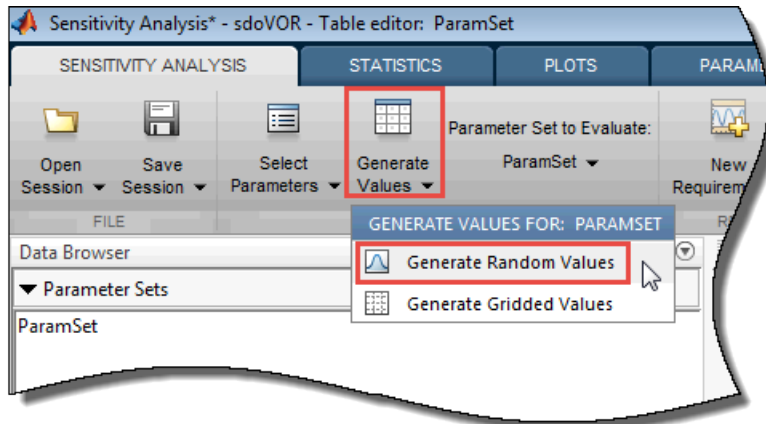
For more information about these distributions, see the “Probability Distributions” (Statistics and Machine Learning Toolbox) category.

Parameter Correlations

Specify the correlation between parameters. The algorithm used to impose the parameter correlations depends on the sampling method. For more information, see “Sampling Method” on page 4-11.

Generate Random Parameter Values in the Tool

In the Sensitivity Analysis tool, after you have selected the parameters in the parameter set on page 4-4, click **Generate Values** and, select **Generate Random Values**.



In the Generate Random Parameter Values dialog box, specify the number of samples, probability distributions, parameter bounds and correlations, and sampling method. For information about how to specify the fields in the dialog box, click **Help**.

Generate Random Parameter Values

Number of Samples:

Overwrite previous values in parameter set when generating new values
 Append to previous values in parameter set when generating new values

Sampling Method:

Parameter	Distribution	Lower	Upper	Cross-Correlated
Delay	Uniform	0.0045	0.0055	<input type="checkbox"/>
Gain	Uniform	0.72	0.88	<input type="checkbox"/>
Tc	Uniform	13.5	16.5	<input type="checkbox"/>

Distribution Plot | Correlation Matrix

Probability Distribution for Delay

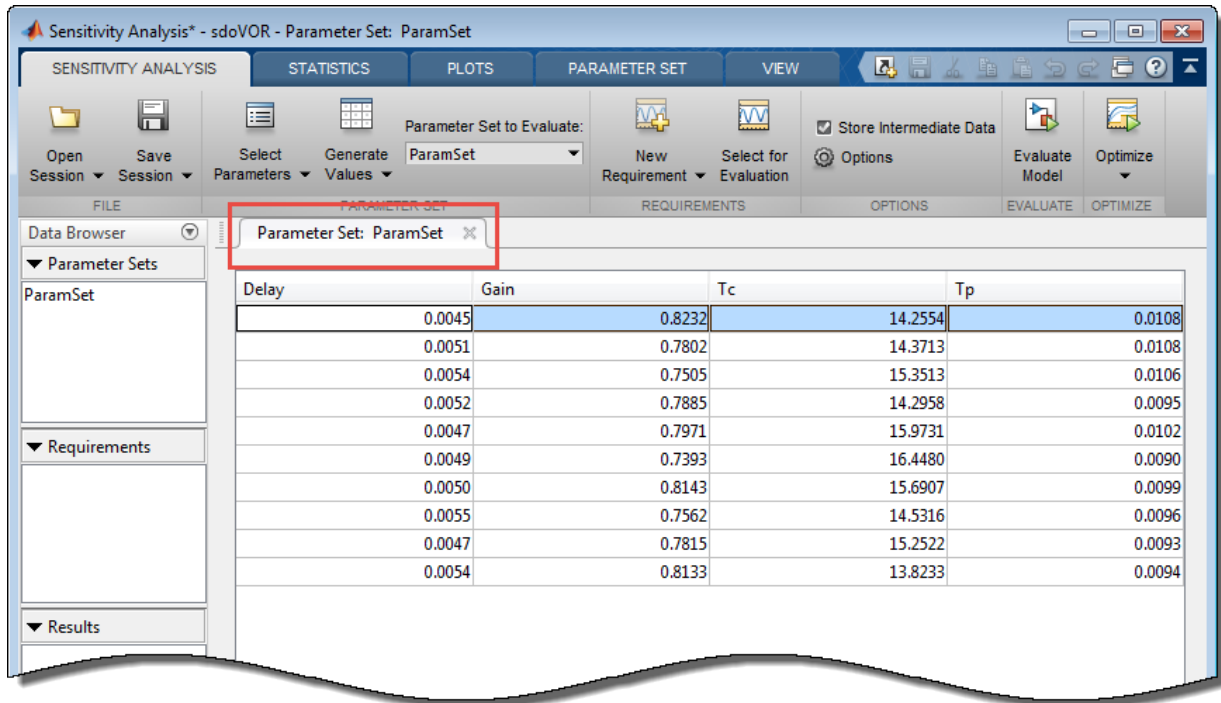
Probability Density

Delay $\times 10^{-3}$

Mean: 0.005
Standard Deviation: 0.000288675

OK Apply **Help**

The generated parameter set and the corresponding parameter set table are displayed in the tool. The number of rows in the parameter set table correspond to the number of samples you specified.



After generating the parameter values, plot them to check if generated parameter values match the intended specifications. This is relevant if you generate a small number of random samples for each parameter set. For more information, see “Inspect the Generated Parameter Set” on page 4-129.

For an example, see “Identify Key Parameters for Estimation (GUI)” on page 4-168.

Generate Random Parameter Values at the Command Line

At the command line, use `sdo.ParameterSpace` to define the parameter space. This object specifies the probability distributions and correlations for the parameters. Use this object as an input to `sdo.sample` for generating parameter values from the specified parameter space.

To generate the random parameter values:

- Specify the number of samples as the second input argument of `sdo.sample`.

- Specify the method used to generate these samples using the `Method` property of an `sdo.SampleOptions` object. Use this object as an input to `sdo.sample` to specify the sampling options.

If the method chosen is 'sobol' or 'halton', specify the `MethodOptions` property of `sdo.SampleOptions`.

If the method chosen is 'copula', specify the choice of copula using the `MethodOptions` property of `sdo.SampleOptions`. Also specify the `RankCorrelation` property of the `sdo.ParameterSpace` object.

- Specify the probability distribution of a parameter using the `ParameterDistributions` property of an `sdo.ParameterSpace` object.
- Specify correlation between parameters, using the `RankCorrelation` property of the `sdo.ParameterSpace`.

After generating the parameter values, plot the generated values to check if they match the desired specifications. This is relevant if you generate a small number of random samples for each parameter set. For more information, see “Inspect the Generated Parameter Set” on page 4-129.

For an example, see “Identify Key Parameters for Estimation (Code)” on page 4-217.

Generate Custom Parameter Values at the Command Line

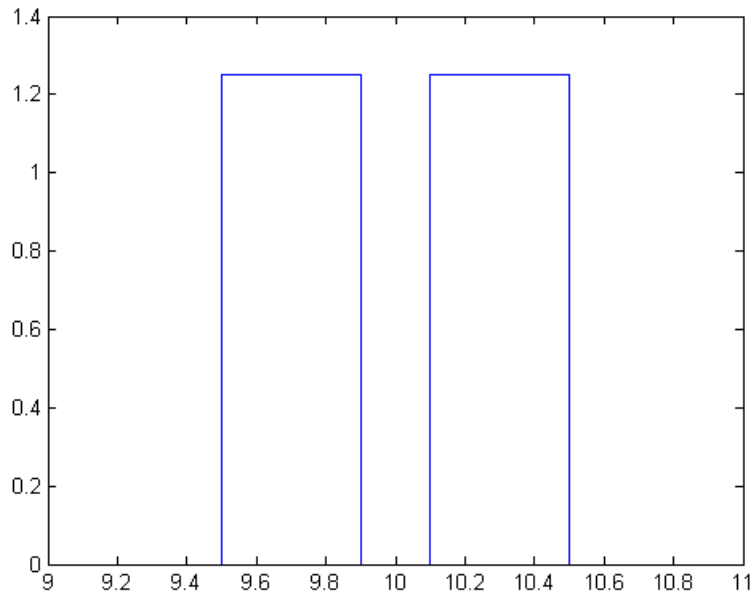
This example shows how to generate random parameter values with a custom distribution when performing sensitivity analysis at the command line. Generate a 1000 samples of a model parameter, `R`, in the 5% range of its nominal value, 10. `R` is a resistor. Resistors of 1% tolerance are removed by the manufacturer, so do not generate `R` values in the 1% range of its nominal value.

- 1 Construct a `param.Continuous` object.

```
R = param.Continuous('R',10);
```

- 2 Create a customized probability distribution, `pdR`, to configure the parameter space.

```
x = [0.95 0.99 1.01 1.05]*R.Value;
F = [0 0.5 0.5 1];
pdR = makedist('PiecewiseLinear','x',x,'Fx',F);
x = linspace(0.9*R.Value,1.1*R.Value,1e3);
plot(x,pdf(pdR,x));
```



`makedist` specifies a piecewise linear distribution for the resistor value, with a “hole” in the 1% range.

- 3 Specify `pdR` as the probability distribution for the `R` parameter in an `sdo.ParameterSpace` object.

```
ps = sdo.ParameterSpace(R, pdR);
```

- 4 Generate 1000 samples.

```
Ns = 1000;  
x = sdo.sample(ps, Ns);
```

- 5 (Optional) Use `sdo.scatterPlot` to visualize the samples and validate the sample space.

Generate Gridded Parameter Values

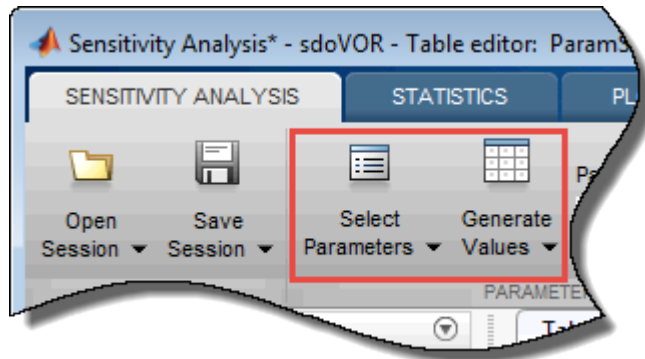
You can generate gridded parameter values in the Sensitivity Analysis tool or at the command line.

Generate Gridded Parameter Values in the Sensitivity Analysis Tool

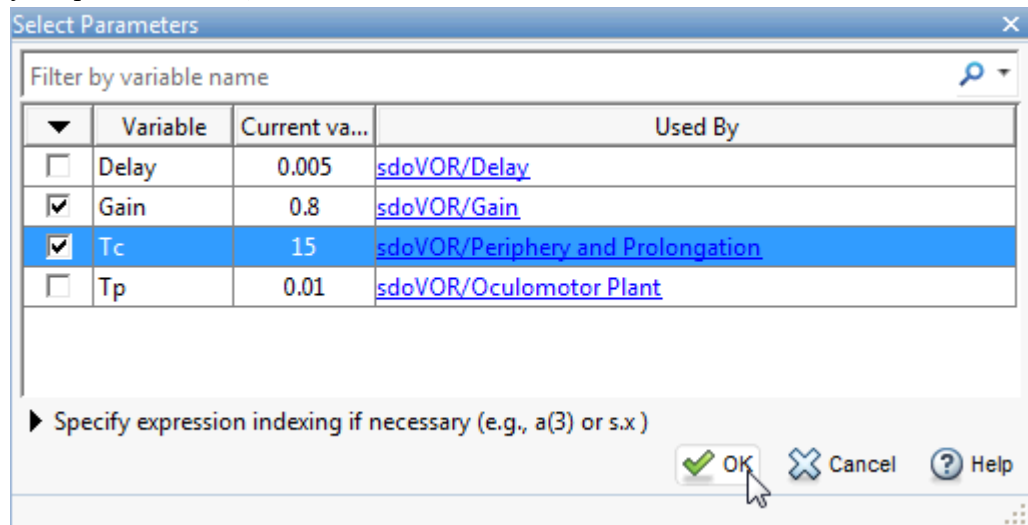
You can generate gridded parameters in the tool after you have created a parameter set.

- 1 Create a parameter set.

In the Sensitivity Analysis tool, in the **Sensitivity Analysis** tab, click **Select Parameters**.

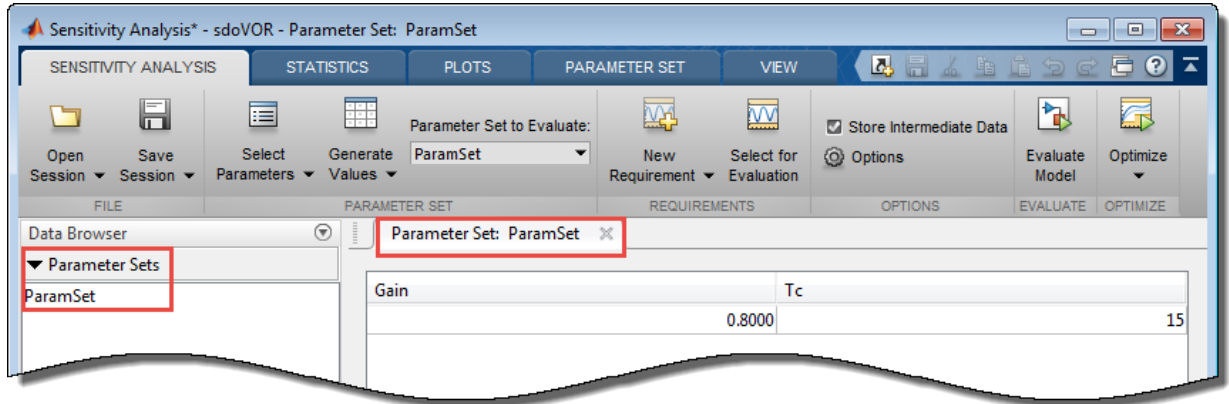


In the Select Parameters dialog box, select all the parameters you want to include in your parameter set, and click **OK**.



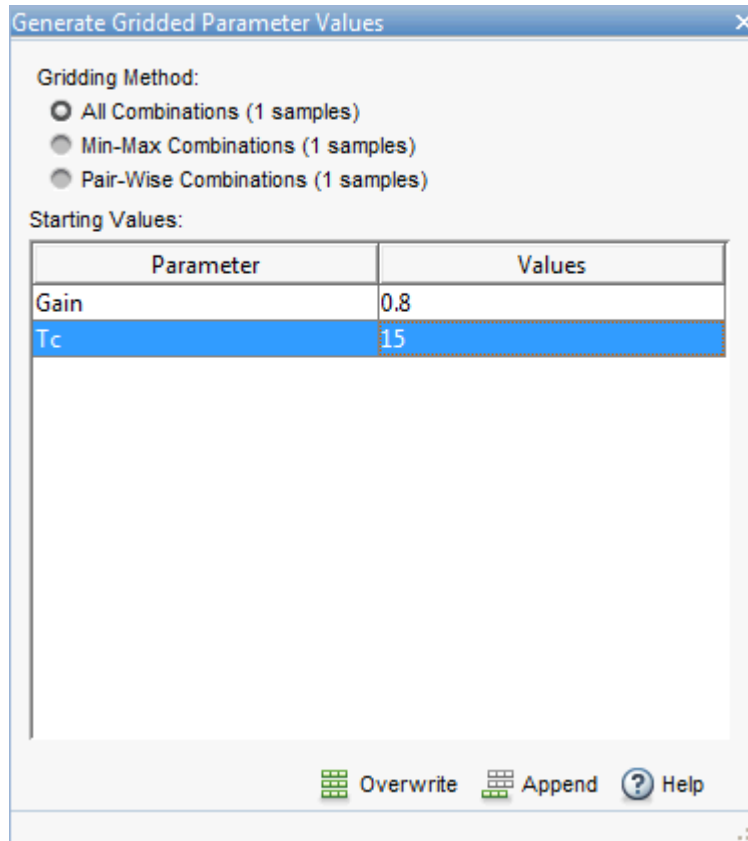
4 Sensitivity Analysis

A ParamSet variable is created in the **Parameter Sets** area of the tool. The current value for each parameter in the parameter set is displayed in a table.



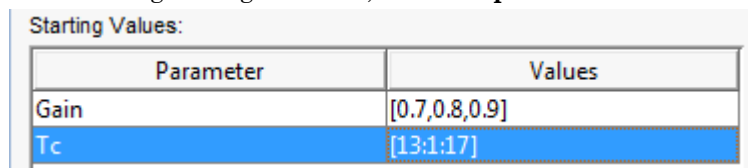
- 2 Specify the starting parameter values and gridding method for grid generation.

In the Sensitivity Analysis tab, click **Generate Values**, and select **Generate Gridded Values** from the drop-down menu.



In the Generate Gridded Parameter Values dialog box, specify the gridding method as **All Combinations**. The tool generates all possible combinations of the values specified in **Values**.

Specify the starting parameter values in **Values**. The values you enter here determine the parameter space. To see the other ways to specify starting parameter values and gridding methods, click **Help**.



- 3 Generate the parameters.

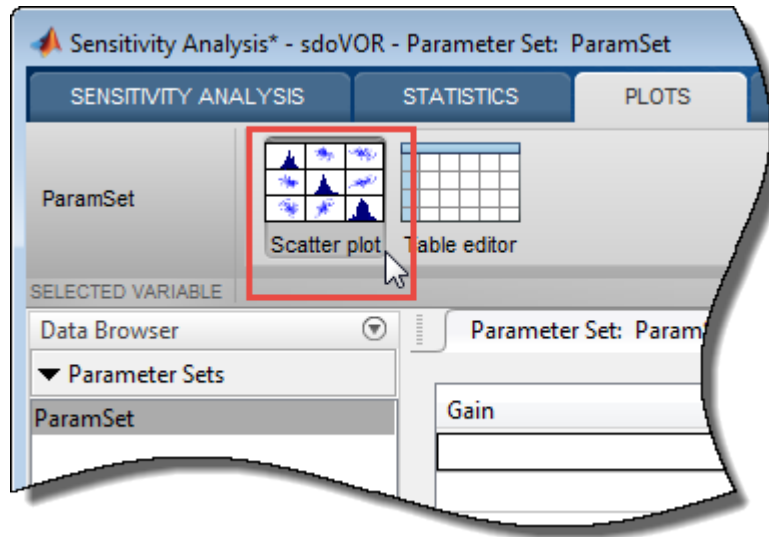
Click **Overwrite**. The parameter set table updates with the generated gridded parameter values.

Parameter Set: ParamSet ✕	
Gain	Tc
0.7000	13
0.8000	13
0.9000	13
0.7000	14
0.8000	14
0.9000	14
0.7000	15
0.8000	15
0.9000	15
0.7000	16
0.8000	16
0.9000	16
0.7000	17
0.8000	17
0.9000	17

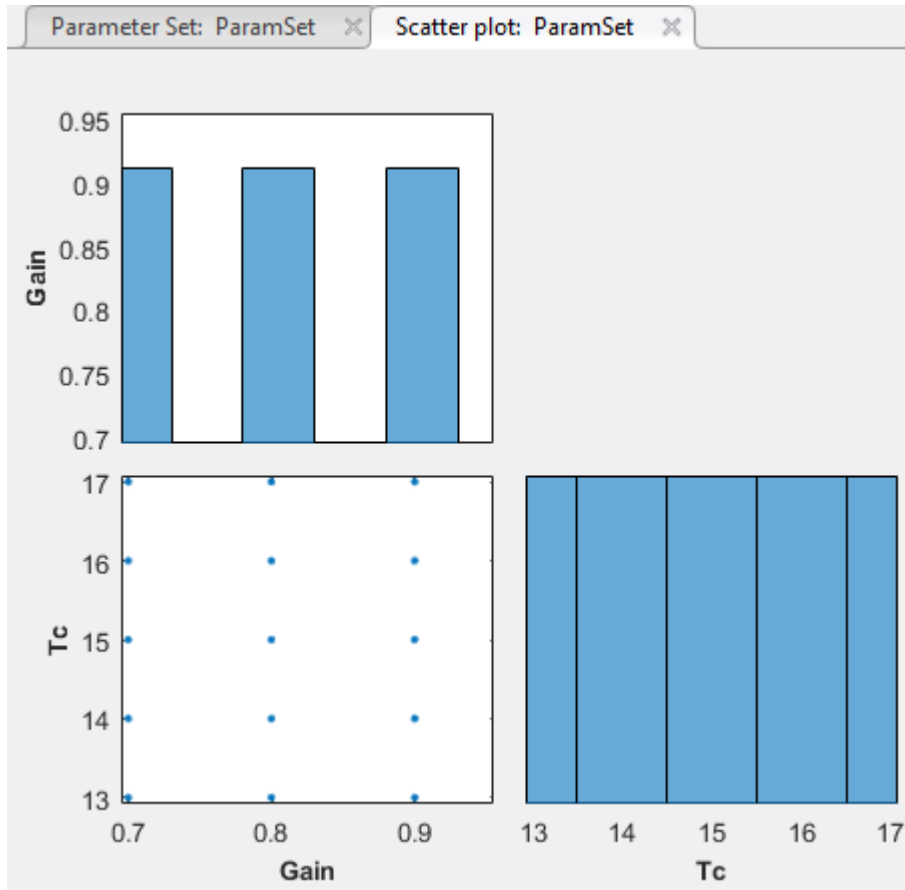
You can also append the generated values to previously generated random or gridded parameter values in the parameter set table. To do so, click **Append**.

- 4 (Optional) Plot the generated values.

In the **Parameter Sets** area of the tool, select **ParamSet**. In the **Plots** tab of the tool, select **Scatter Plot** from the plots gallery.



Plots are generated with histograms of the distribution of the parameter values shown on the diagonals. The off-diagonal plots display the scatter plots between pairs of parameters. To learn more about the plots, see “Interact with Plots in the Sensitivity Analysis Tool” on page 4-106.



Generate Gridded Parameter Values at the Command Line

This example shows how to create a table of gridded parameter values at the command line.

Generate a grid of samples for two model parameters, A and B. Vary A between $[2, 3, 4]$ and B between $[20, 30, 40]$.

- 1 Construct a `param.Continuous` object.

```
A = param.Continuous('A', 1);
B = param.Continuous('B', 10);
```


- 2 Specify the parameter values for grid generation.

```
Avals = [2 3 4];  
Bvals = [20 30 40];
```

- 3 Create a table of gridded parameter values. Specify one column for each parameter, and one row for each sample. The column names must be the same as the parameter names.

```
[Agrid,Bgrid] = meshgrid(Avals,Bvals);  
x = table(Agrid(:),Bgrid(:), 'VariableNames', {'A', 'B'});
```

```
x =
```

A	B
2	20
2	30
2	40
3	20
3	30
3	40
4	20
4	30
4	40

See Also

`sdo.SampleOptions` | `sdo.sample`

Related Examples

- “Inspect the Generated Parameter Set” on page 4-129
- “Design Exploration Using Parameter Sampling (GUI)” on page 4-147
- “Design Exploration Using Parameter Sampling (Code)” on page 4-200
- “Explore Design Reliability Using Parameter Sampling (GUI)” on page 4-186

Specify Time-Domain Requirements

In this section...
“Match Model Outputs to Measured Signals” on page 4-28
“Specify Piecewise-Linear Lower and Upper Bounds” on page 4-33
“Specify Signal Property Requirements” on page 4-35
“Specify Step Response Characteristics” on page 4-38
“Track Reference Signals” on page 4-40
“Impose Elliptic Bound on Phase Plane Trajectory of Two Signals” on page 4-42
“Specify Custom Requirements” on page 4-46

In the Sensitivity Analysis tool, you can specify the following time-domain requirements:

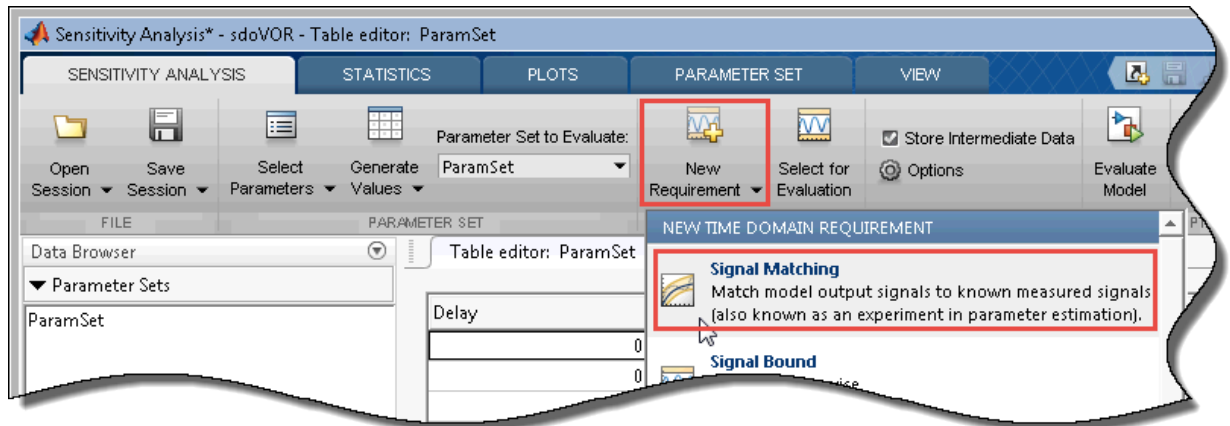
- **Signal Matching** — “Match Model Outputs to Measured Signals” on page 4-28
- **Signal Bound** — “Specify Piecewise-Linear Lower and Upper Bounds” on page 4-33
- **Signal Property** — “Specify Signal Property Requirements” on page 4-35
- **Step Response Envelope** — “Specify Step Response Characteristics” on page 4-38
- **Signal Tracking** — “Track Reference Signals” on page 4-40
- **Ellipse Region Constraint** — “Impose Elliptic Bound on Phase Plane Trajectory of Two Signals” on page 4-42
- **Custom Requirement** — “Specify Custom Requirements” on page 4-46

After you specify the constraints, you can see if the requirements are satisfied by evaluating the design requirements. For more information, see “Evaluate Design Requirements” on page 4-83.

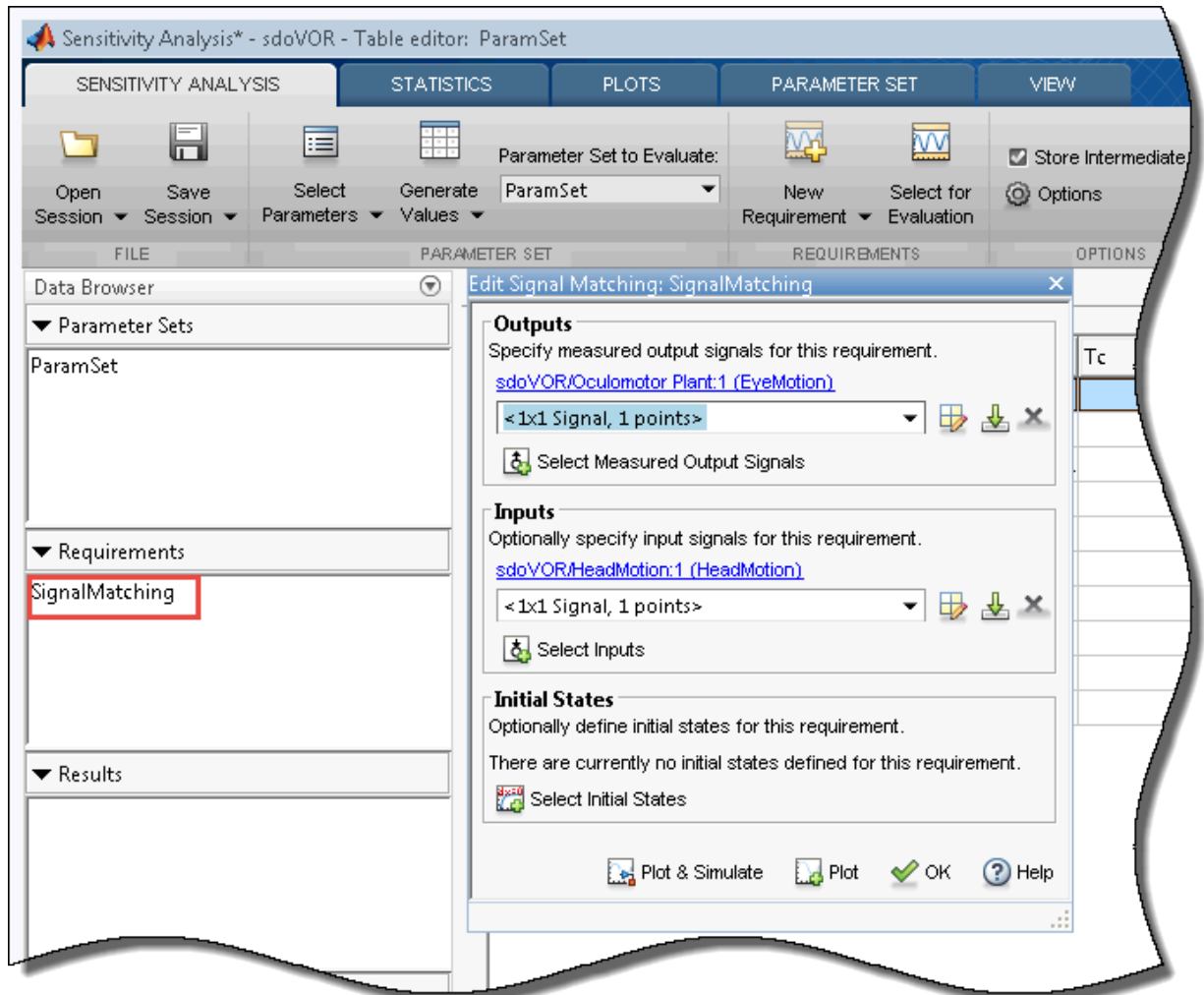
Match Model Outputs to Measured Signals

You can specify a signal matching requirement to match model outputs to measured signals. This requirement is also known as an experiment in the Parameter Estimation tool. To specify a signal matching requirement:



- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Signal Matching**.



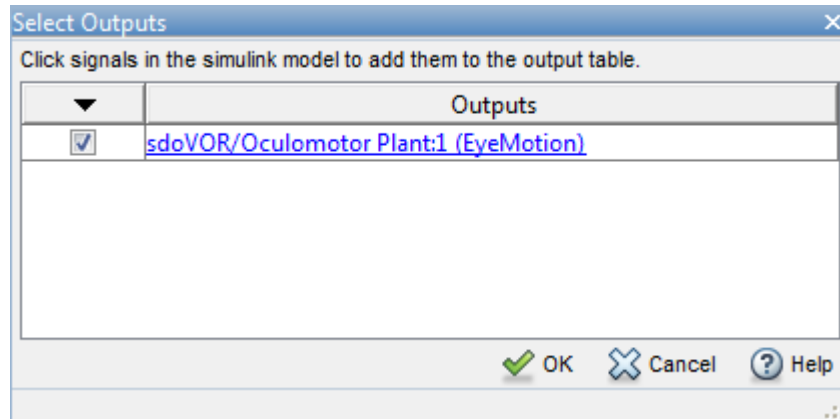
A new signal matching requirement appears in the **Requirements** area of the tool. An Edit Signal Matching dialog box opens where you specify this requirement. You specify model output and input signals, and assign measured data to them. You can also specify initial state values.



- 2 In the **Outputs** panel of the Edit Signal Matching dialog box, select output signals, and import output data. You can select more than one output signal, but you can have only one data set for a signal. If you have multiple data sets, create multiple requirements.
 - a Select the model output signals that you want to add the requirement to.


By default, the root-level model output ports and logged signals are already listed in the **Outputs** panel. To remove existing outputs from the signal matching requirement, click the corresponding . To add other output signals, click  **Select Measured Output Signals**.

The Select Outputs dialog box opens to display the root-level model Output blocks and logged signals.



- b In the Simulink model window, click the signal to add. The Select Outputs dialog box updates to show the new signals. To add these signals as outputs, select the corresponding check boxes. Click **OK**.
- c In the Edit Signal Matching dialog box **Outputs** panel, for each output, import measured output data in one of the following ways:



- Import signal data from spreadsheets or text files — Click , and import the data from the file.
- Import data from the MATLAB workspace — Suppose that the signal data and time data are in the `EyeData` and `time` column vectors in the workspace, respectively. Specify the output data as `[time, EyeData]`.



If your data is stored in a time-series object, *t*, specify the output data as `[t.time, t.outputdata]`.

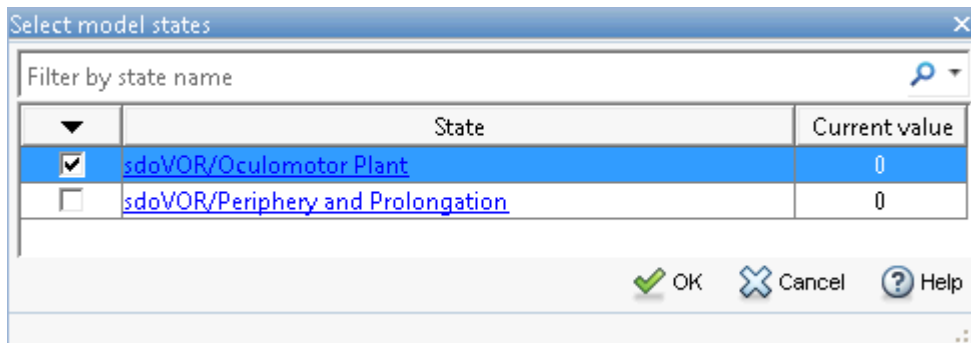
After importing data, to view or edit the data, click .

- 3 (Optional) Select input signals and import input data in the **Inputs** panel.

You can have more than one input signal, but you can have only one data set for a signal. If you have multiple data sets, create multiple requirements. By default, the root-level model input ports are already listed in the **Inputs** panel. Remove or add inputs, and import input data, in the same way as described for the output data.

- 4 (Optional) Specify initial states values in the **Initial States** panel.

By default, the initial conditions specified in the model are used for evaluating the requirement. To specify initial conditions other than the defaults, click **Select Initial States**. In the Select Model States dialog box, select the states to specify, and click **OK**.



The selected states appear in the Edit Signal Matching dialog box **Initial States** panel. Specify the initial states.



- 5 Close the Edit Signal Matching dialog box.

The signal matching requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 6 (Optional) Plot the requirement.
 - a In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.
 - b In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.


Note You can preprocess the data using the preprocess tools in the **Experiment Plot** tab. For more information, see “Preprocess Data” on page 1-16.

Specify Piecewise-Linear Lower and Upper Bounds

To specify upper and lower bounds on a signal:

- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Signal Bound**.

A Create Requirement dialog box opens where you specify upper or lower bounds on a signal. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.


- 2 Select the requirement type from **Type**.
- 3 Specify the edge start and end times and corresponding amplitude in the **Time (s)** and **Amplitude** columns.
- 4 Click  to specify additional bound edges.

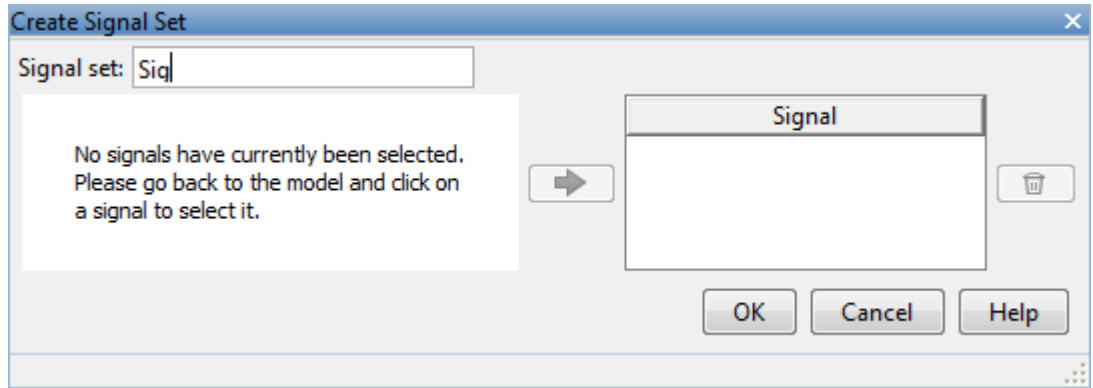
Select a row and click  to delete a bound edge.

- 5 In the **Select Signals to Bound** area, select a logged signal to apply the requirement to.


If you have already selected signals, as described in “Specify Signals to Log” on page 3-12, they appear in the list. Select the corresponding check-box.

If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.
- b In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the Create Requirement dialog box.

- 6 Select the check-box corresponding to the signal, and close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 7 (Optional) Plot the requirement.

- a** In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.
- b** In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.

You can now evaluate the requirement. For more information, see “Evaluate Design Requirements” on page 4-83.

Alternatively, you can add a Check Custom Bounds block to your model to specify piecewise-linear bounds.

Specify Signal Property Requirements

To specify signal property requirements:

- 1** In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Signal Property**. A Create Requirement dialog box opens where you specify signal property requirements. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.
- 2** In the **Specify Property** area, specify a signal property requirement using the **Property** and **Type** lists and the **Bound** box.

Property List

Property	Description	Time weighting available
Signal minimum	Minimum of the signal	No
Signal maximum	Maximum of the signal	No
Signal final value	Last signal value	No
Signal mean	Average of signal value	Yes
Signal median	Middle value of signal	Yes
Signal variance	Variance of signal	Yes
Signal interquartile range	Difference between the 75th and 25th percentiles of signal values	No

Property	Description	Time weighting available
Signal sum	$\sum_{i=t_0}^{t_N} S(i)$, where $S(t_0), \dots, S(t_N)$ is the signal to constrain.	Yes
Signal sum square	$\sum_{i=t_0}^{t_N} S(i)^2$	Yes
Signal sum absolute	$\sum_{i=t_0}^{t_N} S(i) $	Yes

For signal properties where the **Time-Weighted** option is available, you can select it to weight the property computation by the time intervals between samples.

Custom Signal Property

You can add a custom signal property to the **Property** list by editing the function `sdo.requirements.signalPropertyFcns`.

- a** At the MATLAB command prompt, enter `edit sdo.requirements.signalPropertyFcns`.
- b** Add your signal property function to the `FcnData` cell array.


Your signal property function must be on the path.

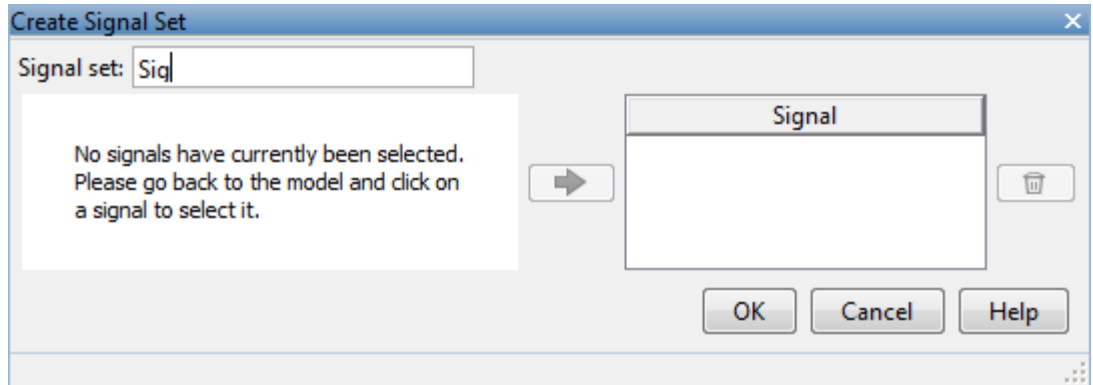
- 3** In the **Select Signals to Bound** area, select the logged signal to which you want to apply the requirement.

The signal selected must have numeric type data (either floating-point or integer). Also, if the property selected is Signal median, Signal variance, or Signal interquartile range, then the signal data must be floating-point (either double or single).


If you have already selected a signal, as described in “Specify Signals to Log” on page 3-12, the signal appears in the list. Select the corresponding check box for that signal.

If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.
- b In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
 - d In **Signal set** field, enter a name for the selected signal set.
- Click **OK**. A new variable, with the specified name, appears in the Create Requirement dialog box.
- 4 Select the check-box corresponding to the signal, and close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

You can now evaluate the requirement. For more information, see “Evaluate Design Requirements” on page 4-83. When you perform evaluation, a positive requirement value indicates that the requirement has been violated.

Specify Step Response Characteristics

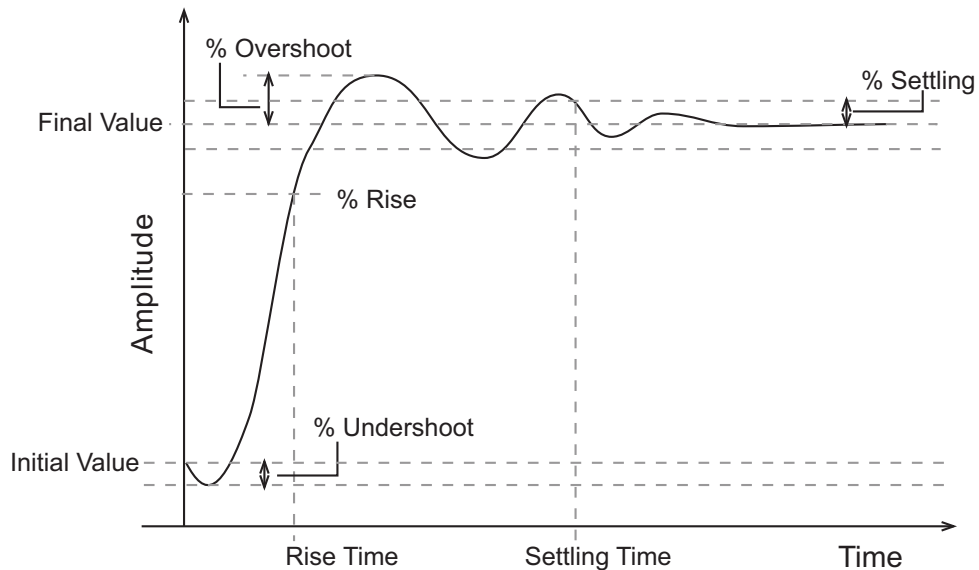
To apply a step response requirement to a signal in your model, specify the step response characteristics as follows:

- 1 Select a step response requirement from the Sensitivity Analysis tool.

In the **New Requirement** drop-down list of the tool, in the **New Time Domain Requirement** section, select **Step Response Envelope**.

A Create Requirement dialog box opens where you specify the step response requirements on a signal. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.

- 2 Specify the step response characteristics:



- **Initial value** — Input level before the step occurs
- **Step time** — Time at which the step takes place
- **Final value** — Input level after the step occurs
- **Rise time** — The time taken for the response signal to reach a specified percentage of the step range. The step range is the difference between the final and initial values.


- **% Rise** — The percentage of the step range used with **Rise time** to define the overall rise time characteristics.
- **Settling time** — Time taken until the response signal settles within a specified region around the final value. This settling region is defined as the final step value plus or minus the specified percentage of the final value.
- **% Settling** — The percentage of the final value that defines the settling range of settling time characteristic specified in **Settling time**.
- **% Overshoot** — The amount by which the response signal can exceed the final value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.
- **% Undershoot** — The amount by which the response signal can undershoot the initial value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.

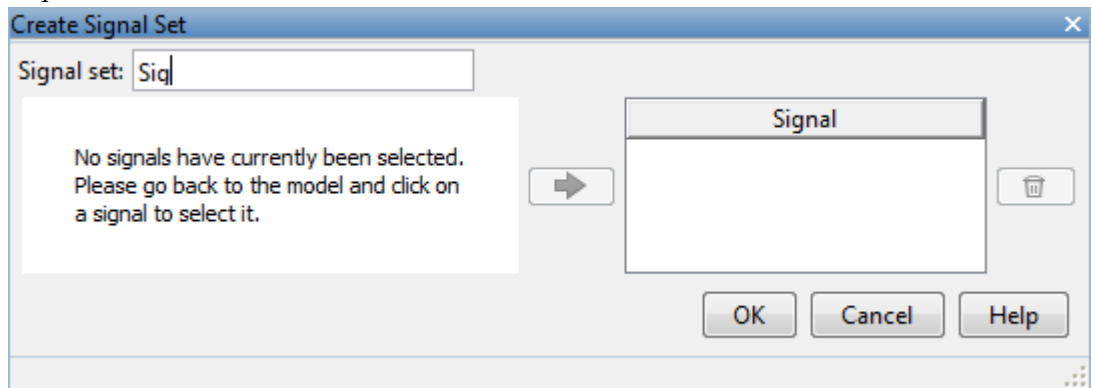
3 Specify the signal to be bound.

To apply this requirement to a model signal, in the **Select Signals to Bound** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, it appears in the list. Select the corresponding check-box.

If you have not selected a signal to log:

- Click . The Create Signal Set dialog box opens where you specify the logged signal.
- In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c** Select the signal and click  to add it to the signal set.
- d** In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the Create Requirement dialog box.

- 4** Select the check-box corresponding to the signal, and close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 5** (Optional) Plot the requirement.

- a** In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.
- b** In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.

You can now evaluate the requirement. For more information, see “Evaluate Design Requirements” on page 4-83.

Alternatively, you can use the Check Step Response Characteristics block to specify step response bounds for a signal.

Track Reference Signals

Use reference tracking to force a model signal to match a desired signal. To track a reference signal:

- 1** In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Signal Tracking**.

A Create Requirement dialog box opens where you specify the reference signal to track. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.

- 2 Define the reference signal by entering vectors, or variables from the workspace, in the **Time vector** and **Amplitude** fields.

Click **Update reference signal data** to use the new amplitude and time vector as the reference signal.


- 3 Specify how the optimization solver minimizes the error between the reference and model signals using the **Tracking Method** list:

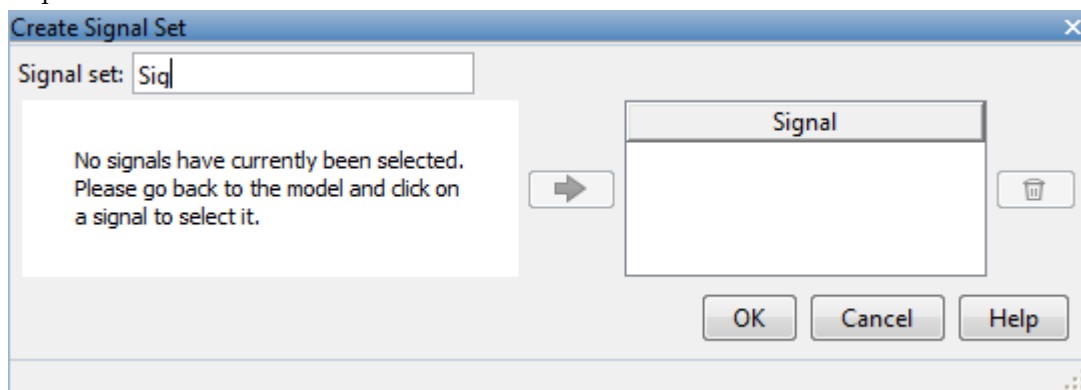
- SSE — Reduces the sum of squared errors
- SAE — Reduces the sum of absolute errors

- 4 In the **Specify Signal to Track Reference Signal** area, select a logged signal to apply the requirement to.

If you already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, they appear in the list. Select the corresponding check-box.

If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.
- b In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.

- d** In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the Create Requirement dialog box.

- 5** Select the check-box corresponding to the signal, and close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 6** (Optional) Plot the requirement.

- a** In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.

- b** In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

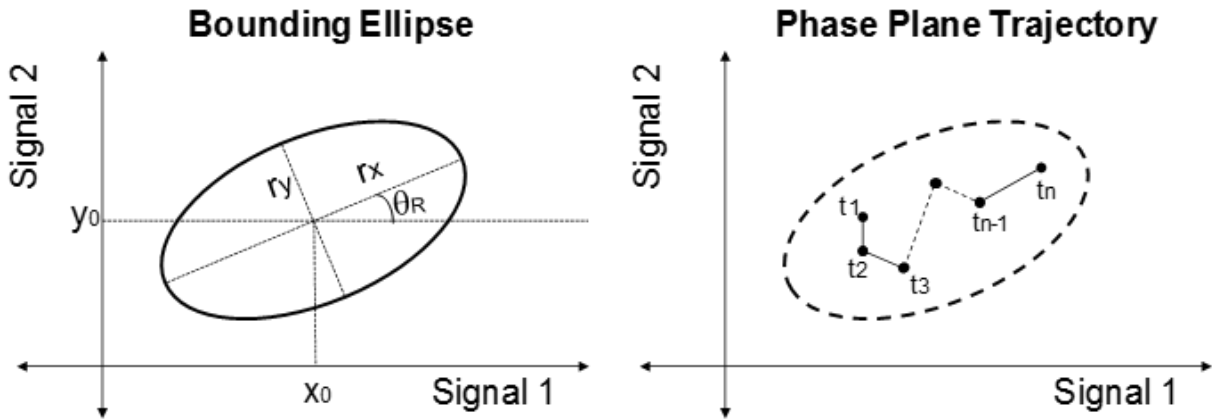
The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.

Alternatively, you can use the Check Against Reference block to specify a reference signal to track.

Impose Elliptic Bound on Phase Plane Trajectory of Two Signals

You can impose an elliptic bound on the phase plane trajectory of two signals in your Simulink model. The phase plane trajectory is a plot of the two signals against each other. You specify the radii, center, and rotation of the bounding ellipse. You also specify whether you require the trajectory of the two signals to lie inside or outside the ellipse.

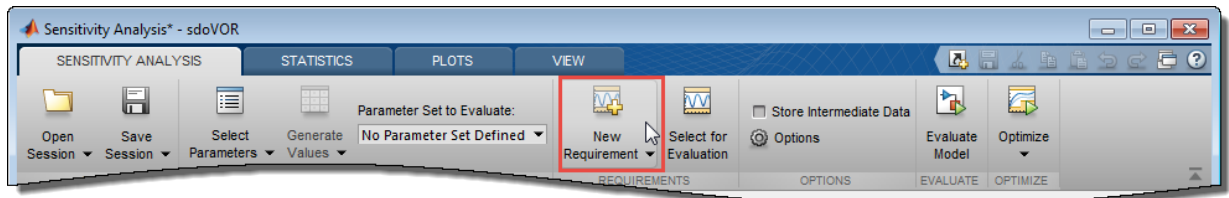
The following image shows the bounding ellipse and an example of the phase plane trajectory of two signals.



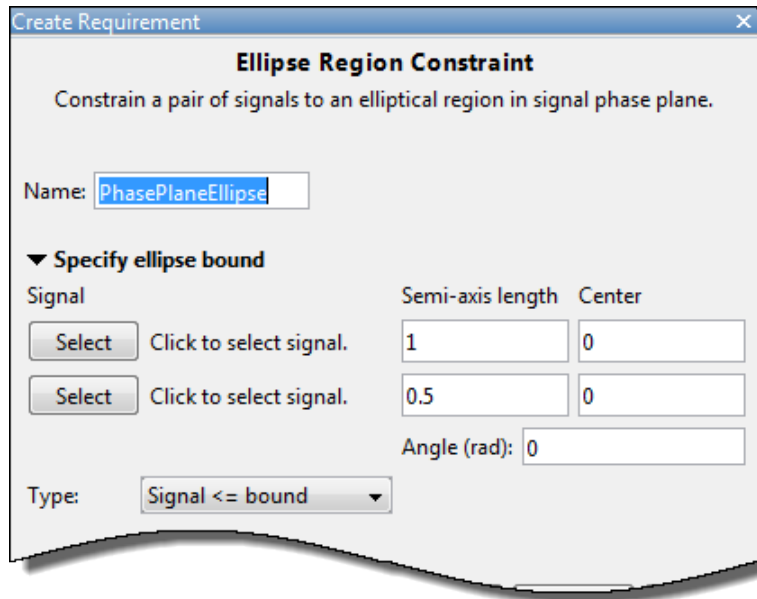
The X-Y plane is the phase plane defined by the two signals. r_x and r_y are the radii of the bounding ellipse along the x and y axes, and θ_R is the rotation of the ellipse about the center. The ellipse center is at (x_0, y_0) . In the image, the phase plane trajectory of the signals lies within the bounding ellipse for all time points t_1 to t_n .

To specify the elliptical bound requirement:

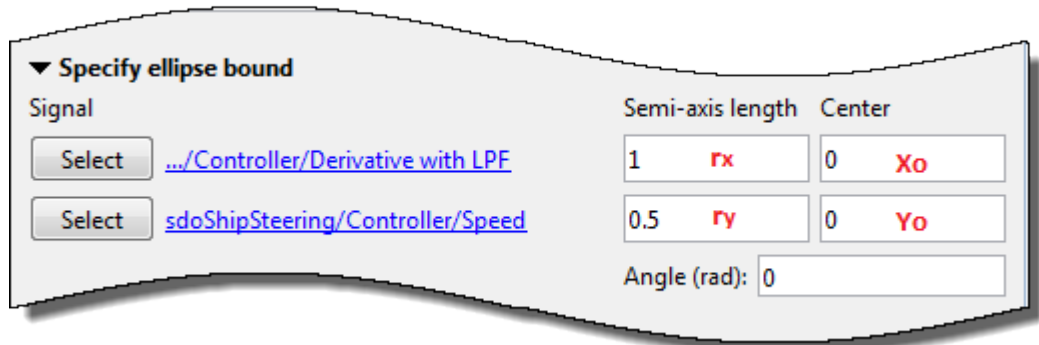
- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Ellipse Region Constraint**.



In the Create Requirement dialog box, specify the signals and elliptical bound. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.




- 1 Specify the two signals that you want to impose the requirement on. The signals define the X-Y plane of the bounding ellipse. To specify the signals, click the corresponding **Select** buttons.



When you click **Select**, the Create Signal Set dialog box opens.



In the Simulink model window, click the signal to which you want to add the requirement. The Create Signal Set dialog box updates with the name of the block and the port number where the selected signal is located. Select the signal, and click  to add it to the signal set.

Once you have specified the logged signal in the Create Signal Set dialog box, the signal appears in the Create Requirement dialog box.

- 2 Specify the radii of the bounding ellipse as real positive finite values in **Semi-axis length**. You specify r_x and r_y that are the x -axis and y -axis radii before any rotation about the ellipse center.
- 3 Specify the location of the center of the bounding ellipse in **Center**. You specify x_0 and y_0 , the x and y coordinates of the center, as real finite values.
- 4 Specify the angle of rotation of the ellipse about its center as a real finite scalar in **Angle (rad)**.
- 5 Specify the bound **Type** as one of the following:
 - ' \leq ' — Ellipse is an upper bound. The phase plane trajectory of the two signals should lie inside or on the ellipse.
 - ' \geq ' — Ellipse is a lower bound. The phase plane trajectory of the two signals should lie outside or on the ellipse.
- 1 Close the Create Requirement dialog box.

The requirement created in the **Requirements** area of the tool is updated with the specified characteristics.

2 (Optional) Plot the requirement.

- a In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.
- b In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.

You can now evaluate the requirement. For more information, see “Evaluate Design Requirements” on page 4-83.

Specify Custom Requirements

You can specify custom requirements, such as minimizing system energy. To specify custom requirements:

1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Custom Requirement**.

A Create Requirement dialog box opens where you specify the reference signal to track. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.

2 Specify the requirement type in the **Type** drop-down menu.

3 Specify the name of the function that contains the custom requirement in **Function**. The field must be specified as a function handle using @. The function must be on the

MATLAB path. Click  to review or edit the function.

If the function does not exist, clicking  opens a template MATLAB file. Use this file to implement the custom requirement. The default function name is myCustomRequirement.

4 (Optional) To prevent the solver from considering specific parameter combinations, select **Error if constraint is violated**. Use this option for parameter-only constraints.

During an optimization iteration, the solver first evaluates requirements with this option selected.

- If the constraint is violated, the solver skips evaluating any remaining requirements and proceeds to the next combination of parameters in the parameter set.
- If the constraint is *not* violated, the solver evaluates the remaining requirements for the current combination of parameter values. If any of the remaining requirements bound signals or systems, then the solver simulates the model.

Note If you select this check box, then do not specify signals or systems to bound. If you *do* specify signals or systems, then this check box is ignored.

5 (Optional) Specify the signal or system, or both, to be bound.

You can apply this requirement to model signals, or a linearization of your Simulink model (requires Simulink Control Design), or both.


Click **Select Signals and Systems to Bound (Optional)** to view the signal and linearization I/O selection area.

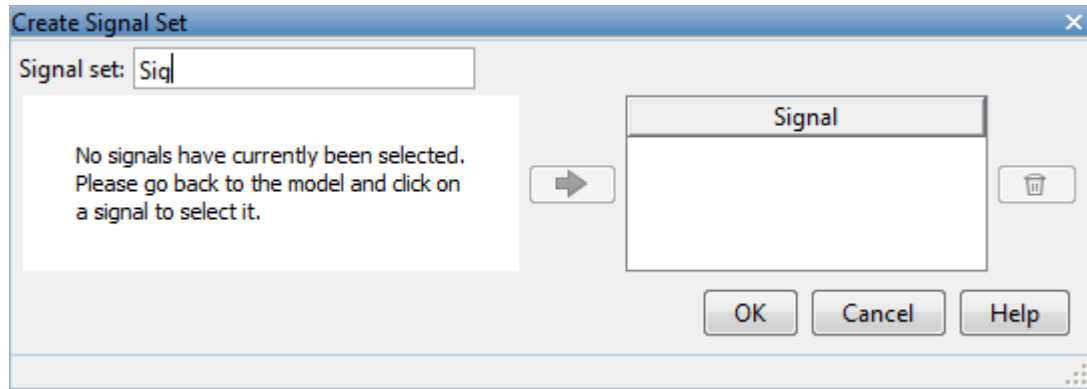
- To apply this requirement to a model signal:

In the **Signal** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, it appears in the list. Select the corresponding check box.

If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.
- b In the Simulink model window, click the signal to which you want to add a requirement.




The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the Create Requirement dialog box.

- To apply this requirement to a linear system:
 - a Specify the simulation time at which the model is linearized in **Snapshot Times**. For multiple simulation snapshot times, specify a vector.
 - b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box. For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 6 Select the check-box corresponding to the signal or system, and close the Create Requirement dialog box.

The requirement created in the **Requirements** area of the tool is updated with the specified characteristics.

You can now evaluate the requirement. For more information, see “Evaluate Design Requirements” on page 4-83. When you perform evaluation, a positive requirement value indicates that the requirement has been violated.

See Also

Related Examples

- “Specify Parameters Requirements” on page 4-50
- “Specify Frequency-Domain Requirements” on page 4-66
- “Evaluate Design Requirements” on page 4-83
- “Identify Key Parameters for Estimation (GUI)” on page 4-168
- “Design Exploration Using Parameter Sampling (GUI)” on page 4-147

Specify Parameters Requirements

In this section...

“Impose Monotonic Constraint Requirement on Variable” on page 4-50

“Impose Upper Bound on Gradient Magnitude of Variable” on page 4-53

“Specify Linear or Quadratic Function Matching Constraint” on page 4-56

“Specify Requirement on a Vector Property” on page 4-60

“Impose Relational Constraint Between Two Variables” on page 4-63

In the Sensitivity Analysis tool, you can specify the following constraints on Simulink model parameters that are specified as variables:

- **Monotonic Variable** — “Impose Monotonic Constraint Requirement on Variable” on page 4-50
- **Smoothness Constraint** — “Impose Upper Bound on Gradient Magnitude of Variable” on page 4-53
- **Function Matching** — “Specify Linear or Quadratic Function Matching Constraint” on page 4-56
- **Vector Property** — “Specify Requirement on a Vector Property” on page 4-60
- **Relational Constraint** — “Impose Relational Constraint Between Two Variables” on page 4-63

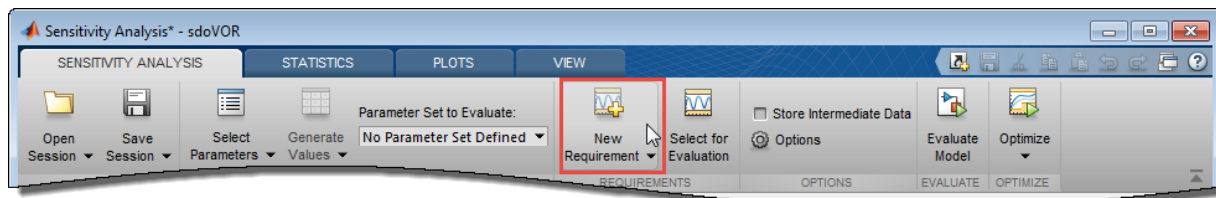
For information about how to specify a model parameter as a variable, see “Add Model Parameters as Variables” on page 4-4.

Impose Monotonic Constraint Requirement on Variable

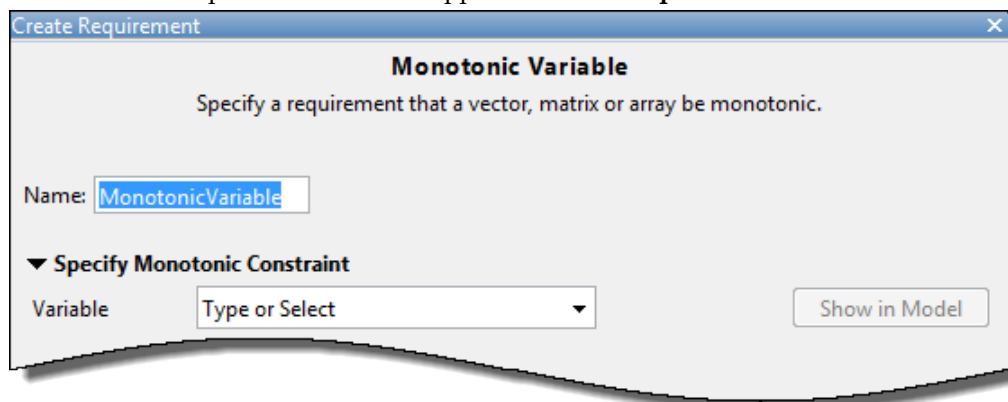
You can impose a monotonic constraint requirement on a variable in your Simulink model. For example, constrain a variable to be monotonically increasing. The variable can be a vector, matrix, or multidimensional array that is a parameter in your model, such as the breakpoints of a lookup table.

To specify the requirement:

- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Monotonic Variable**.



In the Create Requirement dialog box, specify the requirement. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.



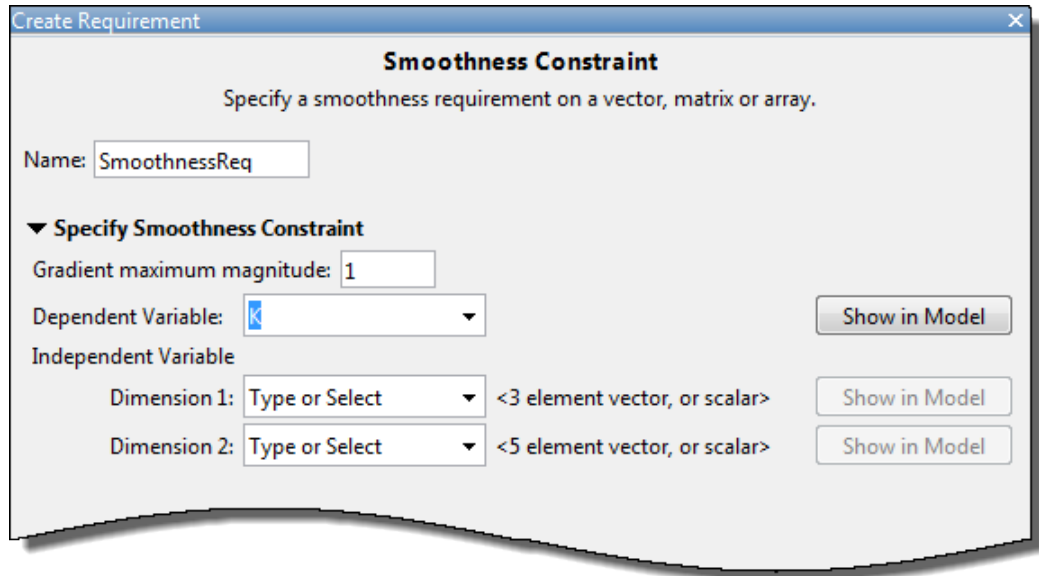
- 2 Specify the name of the variable in **Variable**. The variable must be a vector, matrix, or multidimensional array of data type `double` or `single`.

You can type the name of a nonscalar variable, or select the variable from the drop-down list. The list is prepopulated with all the nonscalar variables in your model. To choose a subset of an array or matrix variable V , type an expression. For example, specify **Variable** as $V(1, :)$ to use the first row of the variable. To use a numeric nonscalar field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$.

Sometimes, models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as $k = a + b$, where a and b are not defined in the model but k is used. To add these independent parameters as variables in the tool, see “Add Model Parameters as Variables” on page 4-4.

- 1 Specify the monotonicity for each dimension of the variable.

After you select the variable, the dialog updates to show **Dimension 1** to **Dimension n**, corresponding to the n dimensions of the variable. For example, for a 2-dimensional variable K of size 3-by-5, the dialog updates as shown.



Specify the monotonicity for the first dimension in **Dimension 1** and for the n^{th} -dimension in **Dimension n** as one of the following options:

- **Strictly increasing** — Each element of the variable is greater than the previous element in that dimension.
- **Increasing** — Each element of the variable is greater than or equal to the previous element in that dimension.
- **Decreasing** — Each element of the variable is less than or equal to the previous element in that dimension.
- **Strictly decreasing** — Each element of the variable is less than the previous element in that dimension.
- **Not constrained** — No constraint exists between the elements of the variable in that dimension.

- 1 Close the Create Requirement dialog box.

The requirement created in the **Requirements** area of the tool is updated with the specified characteristics.

You can now evaluate the requirement. For more information, see “Evaluate Design Requirements” on page 4-83. When you perform evaluation, the tool returns the evaluated requirement value corresponding to each dimension of the variable. A positive requirement value indicates that the requirement has been violated.

Impose Upper Bound on Gradient Magnitude of Variable

You can impose an upper bound on the gradient magnitude of a variable in your Simulink model. The variable can be a vector, matrix, or multidimensional array that is a parameter in your model, such as the data of a lookup table. For example, consider a car engine controller whose gain changes under different operating conditions determined by the car speed. You can use a gradient bound constraint to limit the rate at which the controller gain changes per unit change in vehicle speed.

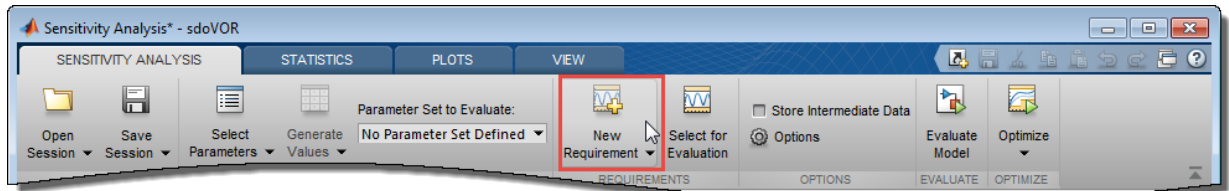
For an N -dimensional variable F that is a function of independent variables x_1, \dots, x_N , the gradient magnitude is defined as:

$$|\nabla F| = \sqrt{\left(\frac{\partial F}{\partial x_1}\right)^2 + \left(\frac{\partial F}{\partial x_2}\right)^2 + \dots + \left(\frac{\partial F}{\partial x_N}\right)^2}$$

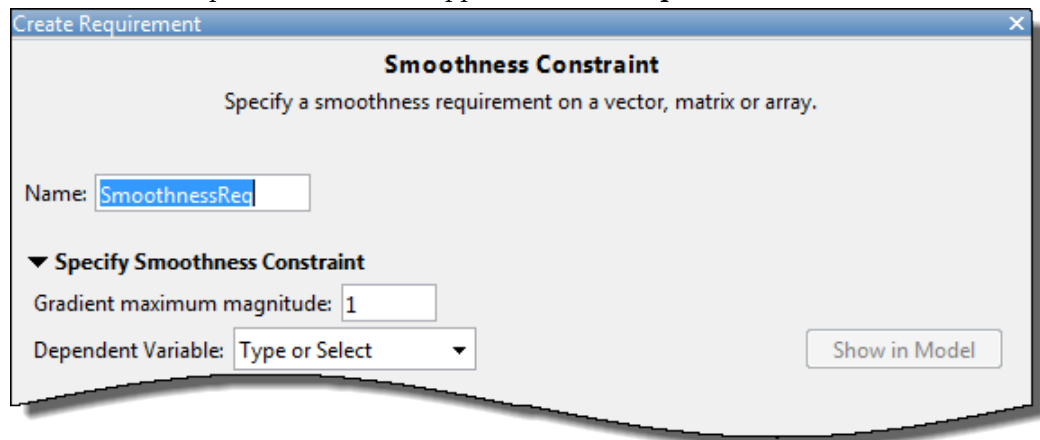
To compute the gradient magnitude, the software computes the partial derivative in each dimension by computing the difference between successive F data in that dimension and dividing by the spacing between the data in that dimension. You specify F and the spacing between the data. The software checks whether the gradient magnitude of the variable data is less than or equal to a specified bound. If the gradient magnitude of the data is greater than the required bound, the variable data is not smooth.

To specify the requirement:

- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Smoothness Constraint**.



In the Create Requirement dialog box, specify the requirement. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.



- 2 Specify the gradient magnitude bound as a nonnegative finite real scalar in **Gradient maximum magnitude**.
- 3 Specify the variable F that you want to impose the requirement on in **Dependent Variable**. The variable must be a vector, matrix, or multidimensional array of data type `double` or `single`. The variable must be a parameter in your model or a constant that you enter.

You can type the name of a nonscalar variable or a constant, or select the variable from the drop-down list. The list is prepopulated with all the nonscalar variables in your model. To choose a subset of an array or matrix variable v , type an expression. For example, specify **Variable** as $v(1, :)$ to use the first row of the variable. To use a numeric nonscalar field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$.

Sometimes, models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as $k = a + b$, where a and b are not defined in the model but k is used. To add these independent parameters as variables in the tool, see “Add Model Parameters as Variables” on page 4-4.

- 1 Specify the spacing between points of **Dependent Variable** data in each dimension in **Independent Variable**.

After you select the **Dependent Variable**, the dialog updates to show **Dimension 1** to **Dimension n**, corresponding to the n dimensions of the dependent variable. For example, for a 1-dimensional variable k , the dialog updates as shown.

The screenshot shows a dialog box titled "Create Requirement" with a sub-title "Smoothness Constraint". The main instruction is "Specify a smoothness requirement on a vector, matrix or array." The "Name" field is "SmoothnessReq". Under the "Specify Smoothness Constraint" section, the "Gradient maximum magnitude" is set to "1". The "Dependent Variable" is set to "k" and has a "Show in Model" button. The "Independent Variable" is set to "Dimension 1: Type or Select" and has a label "<4 element vector, or scalar>" and a "Show in Model" button.

The first dimension specifies the spacing going down the dependent variable data rows, and the second specifies spacing across the columns. The N th dimension specifies the spacing along the N th dimension of dependent variable data. You can specify the independent variables in each dimension as scalars or vectors.

- **Scalars** — Specify the spacing between dependent variable data F in the corresponding dimension as a nonzero scalar. For example, suppose that **Dependent Variable** is two-dimensional, and the spacing between data in the first dimension is 5 and in the second dimension is 2. In the **Independent Variable** section, specify **Dimension 1** as 5 and **Dimension 2** as 2.

- **Vectors** — Specify the coordinates of F data in the corresponding dimension as real, numeric, monotonic vectors. The software uses the coordinates to compute the spacing between the dependent variable data points in the corresponding dimension. The length of the vector must match the length of F in the corresponding dimension. You do not have to specify coordinates with uniform spacing. For example, suppose that F is two-dimensional, and the length of the data in the first and second dimension is 3 and 5, respectively. The coordinates of the data in the first dimension are [1 2 3]. In the second dimension, the spacing is not uniform and the coordinates of the data are [1 2 10 20 30]. In the **Independent Variable** section, specify **Dimension 1** as [1 2 3] and **Dimension 2** as [1 2 10 20 30].

You can also specify the independent variables by typing the name of a variable, or selecting a variable from the drop-down list. The list is prepopulated with all the variables in your model that have the appropriate size. To choose a subset of an array or matrix variable V , type an expression. For example, specify as $V(1, :)$ to use the first row of the variable. To use a numeric field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$.

- 1 Close the Create Requirement dialog box.

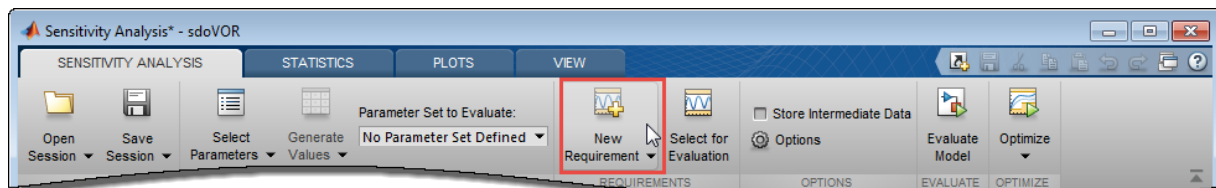
The requirement created in the **Requirements** area of the tool is updated with the specified characteristics.

You can now evaluate the requirement. For more information, see “Evaluate Design Requirements” on page 4-83. When you perform evaluation, the tool returns the evaluated requirement value. A positive requirement value indicates that the requirement has been violated.

Specify Linear or Quadratic Function Matching Constraint

In the tool, you can constrain a variable's values to match a linear or quadratic function. The variable can be a vector, matrix, or a multidimensional array that is a parameter in your model, such as the data of a lookup table in your model. To specify the requirement:

- 1 In Sensitivity Analysis tool, from the **New Requirement** drop-down list, select **Function Matching**.



In the Create Requirement dialog box, specify the requirement. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.

Create Requirement

Function Matching
Specify that values must match a designated function.

Name:

▼ **Specify Function Matching Constraint**

Functional Relation:

Dependent Variable:

Independent Variable

Dimension 1: <4 element vector>

Center and scale independent variables

▼ **Center and Scale Settings**

Use automatic centers and scales

Use custom centers and scales

Independent Variable	Center	Scale
Dimension 1:	<input type="text" value="12"/>	<input type="text" value="2.5820"/>

- 2 Specify the function to be matched. To do so, set **Functional Relation** to one of the following values:

- **Linear** — Data from variable V are fit to a linear function. For example, for a two-dimensional variable with independent variables, X_1 and X_2 , the linear function has the form:

$$V = a_0 + a_1X_1 + a_2X_2$$

The software calculates the fit coefficients a_0 , a_1 , and a_2 and then calculates the sum of squares of the error between the data and the linear function.

- **Quadratic with no cross-terms** — Data are fit to a quadratic function with no cross-terms. For a two-dimensional variable, the pure quadratic function has the form:

$$V = a_0 + a_1X_1 + a_2X_1^2 + a_3X_2 + a_4X_2^2$$

- **Quadratic with all cross-terms** — Variable data are fit to a quadratic function that includes cross-terms. For a two-dimensional variable, the quadratic function has the form:

$$V = a_0 + a_1X_1 + a_2X_1^2 + a_3X_2 + a_4X_2^2 + a_5X_1X_2$$

If the variable is one-dimensional, there are no cross-terms and so the computation is the same as when **Functional relation** is **Quadratic with no cross-terms**.

- 3 Specify the variable V to which you want to apply the requirement in **Dependent Variable**. The variable must be a vector, matrix, or multidimensional array of data type `double` or `single` that is a parameter in your model.

Type the name of a non scalar variable, or select a variable from the drop-down list. The list is prepopulated with all the nonscalar variables in your model. To see where the selected variable is used on your model, click **Show in Model**. To choose a subset of an array or matrix variable A , type an expression. For example, specify $A(1, :)$ to use the first row of the variable. To use a numeric nonscalar field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$.

Sometimes models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as $k = a + b$, where a and b are not defined in the model but k is used. To add these

independent parameters as design variables in the tool, see “Add Model Parameters as Variables” on page 4-4.

- 4 Specify the independent variable vectors used for computing the function in **Independent Variable**. The independent variables are specified as real, numeric, monotonic vectors.

The number of independent variables must equal the number of dimensions of the dependent variable v . For example, you specify two independent variables when v is a matrix, and use three independent variables when v is three-dimensional. The first independent variable vector specifies coordinates going down the rows of v , and the second independent variable vector specifies coordinates going across the columns of v . The n^{th} independent variable vector specifies coordinates along the n^{th} dimension of v . The number of elements in each independent variable vector must match the size of v in the corresponding dimension. The independent variable vectors must be monotonically increasing or decreasing.

You can also specify the independent variables by typing the name of a variable, or selecting a variable from the drop-down list. The list is prepopulated with all the variables in your model that have the appropriate size. To choose a subset of an array or matrix variable A , type an expression. For example, specify $A(1, :)$ to use the first row of the variable. To use a numeric field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$. To use an equally spaced vector, select $[1 \ 2 \ \dots \ N]$ from the drop-down menu.

- 5 Specify whether you want to center and scale the independent variables. When you select the **Center and scale independent variables** option, the independent variable vectors you specify are divided by a scale value after subtracting a center value. Centering can improve numerical conditioning when one or more independent variable vectors have a mean that differs from 0 by several orders of magnitude. Scaling can improve numerical conditioning when independent variable vectors differ from each other by several orders of magnitude.

To specify the center and scale values for each independent variable, expand the **Center and Scale Settings** section, and select one of the following:

- **Use automatic centers and scales** - The center and scale values are the mean and standard deviation for each independent variable. Using the mean and standard deviation values to center and scale the independent variables is the default option.

- **Use custom centers and scales** - Specify the **Center** and **Scale** values for each independent variable. The independent variable vectors are divided by the corresponding **Scale** value after subtracting the value you specify in **Center**.

6 Close the Create Requirement dialog box.

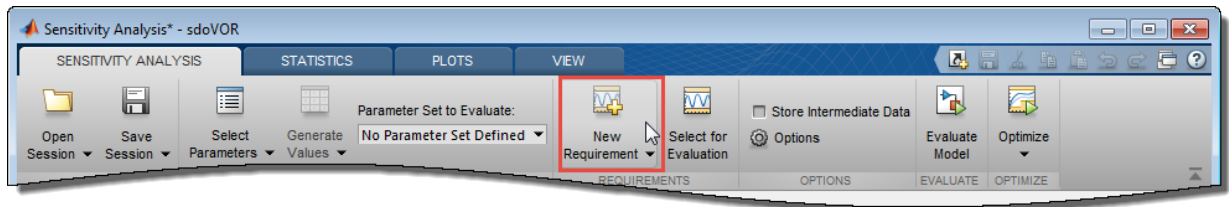
The requirement created in the **Requirements** area of the tool is updated with the specified characteristics.

You can now evaluate the requirement. For more information, see “Evaluate Design Requirements” on page 4-83. When you perform evaluation, the tool returns the evaluated requirement value. The tool computes an error signal that is the difference between the dependent variable data and the specified function of the independent variables. The tool returns the sum of squares of this error as the evaluated requirement value. A positive value indicates that the requirement has been violated, and 0 value indicates that the requirement is satisfied. The closer the value is to 0, the better is the match between the function and dependent variable data.

Specify Requirement on a Vector Property

In the tool, you can specify a requirement on a vector property, such as the mean value of the vector. The vector must be a parameter in your model. To specify the requirement:

1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Vector Property**.



In the Create Requirement dialog box, specify the requirement. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.

1 Specify the vector property in **Property**. For a vector V with N elements, you can specify one of the following properties:

- Vector mean — $mean(V)$
- Vector median — $median(V)$
- Vector variance — $variance(V)$
- Vector inter-quartile range — Difference between the 75th and 25th percentiles of the vector values.

•

• Vector sum — $\sum_{i=1}^N V(i)$

•

• Vector sum of squares — $\sum_{i=1}^N V(i)^2$

•

• Vector sum of absolute values — $\sum_{i=1}^N |V(i)|$

- Vector minimum — $min(V)$

- Vector maximum — $\max(V)$
- 2 Specify the type of requirement you want to impose on the vector property in **Type**. You can set an upper or lower bound on the vector property, or require the property to equal a particular value. You can also choose to maximize or minimize the vector property. For example, to maximize the mean value of your vector, specify **Property** as Vector mean and **Type** as Maximize the property.
 - 3 Specify the value of the bound imposed on the vector property in **Bound**. Specify the bound as a finite real scalar value. For example, if for a vector variable V you require $\text{mean}(V) = 5$, specify **Property** as Vector mean, **Type** as Constrain property to be == the bound, and **Bound** as 5.
 - 1 Specify the name of the variable in **Variable**. The variable must be a vector, matrix, or multidimensional array of data type `double` or `single`.

You can type the name of a nonscalar variable, or select the variable from the drop-down list. The list is prepopulated with all the nonscalar variables in your model. To choose a subset of an array or matrix variable V , type an expression. For example, specify **Variable** as $V(1, :)$ to use the first row of the variable. To use a numeric nonscalar field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$.

Sometimes, models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as $k = a + b$, where a and b are not defined in the model but k is used. To add these independent parameters as variables in the tool, see “Add Model Parameters as Variables” on page 4-4.

- 2 Close the Create Requirement dialog box.

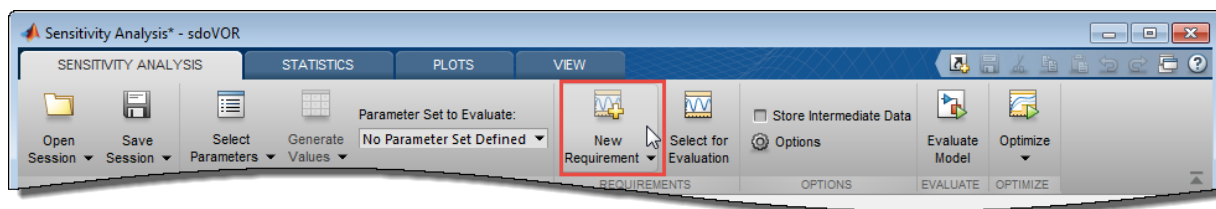
The requirement created in the **Requirements** area of the tool is updated with the specified characteristics.

You can now evaluate the requirement. For more information, see “Evaluate Design Requirements” on page 4-83. When you perform evaluation, the tool returns the evaluated requirement value. A positive requirement value indicates that the requirement has been violated.

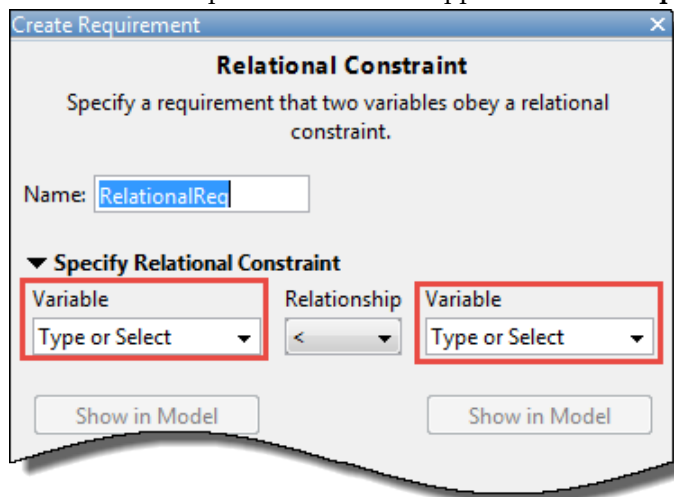
Impose Relational Constraint Between Two Variables

You can impose a relational constraint requirement on a pair of variables in your Simulink model. For example, require that variable a is always greater than variable b . To specify the requirement:

- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Relational Constraint**.



In the Create Requirement dialog box, specify the requirement. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.



- 2 Specify the name of the two variables in **Variable**. The variables can be vectors or arrays but must be the same size.

Type the name of two nonscalar variables, or select the variables from the drop-down lists. The lists are prepopulated with all the nonscalar variables in your model. To see where a selected variable is used on your model, click **Show in Model**. To choose

a subset of an array or matrix variable V , type an expression. For example, specify **Variable** as $V(1, :)$ to use the first row of the variable. To use a numeric nonscalar field x of a structure S , type $S.x$. You cannot use mathematical expressions such as $a + b$.

Sometimes, models have parameters that are not explicitly defined in the model itself. For example, a gain k could be defined in the MATLAB workspace as $k = a + b$, where a and b are not defined in the model but k is used. To add these independent parameters as variables in the tool, see “Add Model Parameters as Variables” on page 4-4.

- 1 Specify the relation between the elements of the two variables as one of the following in **Relationship**:
 - ' $<$ ' — Each data element in the first variable is less than the corresponding element in the second variable.
 - ' $<=$ ' — Each data element in the first variable is less than or equal to the corresponding element in the second variable.
 - ' $>$ ' — Each data element in the first variable is greater than the corresponding element in the second variable.
 - ' $>=$ ' — Each data element in the first variable is greater than or equal to the corresponding element in the second variable.
 - ' $==$ ' — Each data element in the first variable is equal to the corresponding element in the second variable.
 - ' $\sim=$ ' — Each data element in the first variable is not equal to the corresponding element in the second variable.
- 1 Close the Create Requirement dialog box.

The requirement created in the **Requirements** area of the tool is updated with the specified characteristics.

You can now evaluate the requirement. For more information, see “Evaluate Design Requirements” on page 4-83. When you perform evaluation, the tool returns the evaluated requirement value. The interpretation of the evaluated requirement value depends on the requirement **Type**.

Type	Evaluated Requirement Value	
	Requirement is Satisfied	Requirement is Violated
'>' or '<'	Negative number	Positive number, or 0 if the elements are equal
'>=' or '<='	Negative number, or 0 if the elements are equal	Positive number
'=='	0	Non-zero number
'~= '	0	1

See Also

Related Examples

- “Specify Parameters for Design Exploration” on page 4-4
- “Specify Time-Domain Requirements” on page 4-28
- “Specify Frequency-Domain Requirements” on page 4-66

Specify Frequency-Domain Requirements

This topic shows how to specify frequency-domain requirements in the Sensitivity Analysis tool. To specify frequency-domain requirements, you require the Simulink Control Design toolbox.

In this section...
“Specify Lower Bounds on Gain and Phase Margin” on page 4-66
“Specify Piecewise-Linear Lower and Upper Bounds on Frequency Response” on page 4-69
“Specify Bound on Closed-Loop Peak Gain” on page 4-71
“Specify Lower Bound on Damping Ratio” on page 4-72
“Specify Upper and Lower Bounds on Natural Frequency” on page 4-73
“Specify Upper Bound on Approximate Settling Time” on page 4-74
“Specify Piecewise-Linear Upper and Lower Bounds on Singular Values” on page 4-75
“Specify Step Response Characteristics” on page 4-77
“Specify Custom Requirements” on page 4-79

Specify Lower Bounds on Gain and Phase Margin

To specify lower bounds on the gain and phase margin of a linear system:

- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Gain and Phase Margin**. A Create Requirement dialog box opens where you specify upper or lower bounds on the gain and phase margin. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.

Create Requirement ×

Gain and Phase Margin

Specify bounds on the gain and phase margins of a linear system.

Name:

▼ **Specify Margin Bounds**

Gain margin > dB

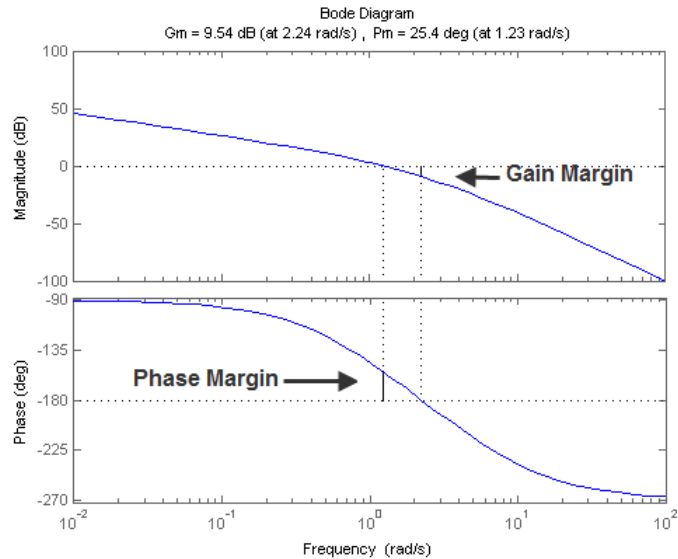
Phase margin > deg

▼ **Select Systems to Bound**

Snapshot Times:

Linearization I/O	
<input type="checkbox"/>	IOs (sdoSimpleSuspension/x_dot:1 [output], sdoSimpleSuspension/Band-...

- 2 Specify bounds on the gain margin or phase margin, or both.



- **Gain margin** — Amount of gain increase or decrease required to make the loop gain unity at the frequency where the phase angle is -180° .
- **Phase margin** — Amount of phase increase or decrease required to make the phase angle -180° when the loop gain is 1.0


To specify a lower bound on the gain margin or phase margin, or both, select the corresponding check box and enter the lower bound value.

- 3 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 4 Close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 5 (Optional) Plot the requirement.


- a In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.
- b In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.

Specify Piecewise-Linear Lower and Upper Bounds on Frequency Response

To specify upper or lower bounds on the magnitude of a system response:

- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Bode Magnitude**. A Create Requirement dialog box opens where you specify the lower or upper bounds on the magnitude of the system response. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.
- 2 Specify the requirement type in the **Type** drop-down list.
- 3 Specify the edge start and end frequencies and corresponding magnitude in the **Frequency** and **Magnitude** columns.
- 4 Insert or delete bound edges.

Click  to specify additional bound edges.


To delete a bound edge, select a row, and click .

- 5 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 6 Close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 7 (Optional) Plot the requirement.
 - a In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.
 - b In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.

Alternatively, you can use the Check Bode Characteristics block to specify bounds on the magnitude of the system response. (Requires Simulink Control Design.)

Specify Bound on Closed-Loop Peak Gain


To specify an upper bound on the closed-loop peak response of a system:

- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Closed-Loop Peak Gain**. A Create Requirement dialog box opens where you specify an upper bound on the closed-loop peak gain of the system. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.
- 2 Specify the upper bound in **Closed-Loop peak gain**.
- 3 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 4 Close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 5 (Optional) Plot the requirement.
 - a In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.

- b** In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.

Alternatively, you can use the Check Nichols Characteristics block to specify bounds on the magnitude of the system response. (Requires Simulink Control Design.)

Specify Lower Bound on Damping Ratio


To specify a lower bound on the damping ratio of the poles of a system:

- 1** In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Damping Ratio**. A Create Requirement dialog box opens where you specify a lower bound on the damping ratio of the system. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.
- 2** Specify the lower bound on the damping ratio in **Damping ratio**.
- 3** In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a** Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 4 Close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 5 (Optional) Plot the requirement.

- a In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.
- b In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.

Alternatively, you can use the Check Pole-Zero Characteristics block to specify a bound on the damping ratio. (Requires Simulink Control Design.)

Specify Upper and Lower Bounds on Natural Frequency


To specify a bound on the natural frequency of the poles of a system:

- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Natural Frequency**. A Create Requirement dialog box opens where you specify a bound on the natural frequency of the system. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.
- 2 Specify a lower or upper bound on the natural frequency in **Natural frequency**.
- 3 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 4 Close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 5 (Optional) Plot the requirement.

- a In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.
- b In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.

Alternatively, you can use the Check Pole-Zero Characteristics block to specify a bound on the natural frequency. (Requires Simulink Control Design.)

Specify Upper Bound on Approximate Settling Time


To specify an upper bound on the approximate settling time of a system:

- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Settling Time**. A Create Requirement dialog box opens where you specify an upper bound on the approximate settling time of the system. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.
- 2 Specify the upper bound on the approximate settling time in **Settling time**.
- 3 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 4 Close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 5 (Optional) Plot the requirement.

- a In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.
- b In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.


Alternatively, you can use the Check Pole-Zero Characteristics block to specify the approximate settling time. (Requires Simulink Control Design.)

Specify Piecewise-Linear Upper and Lower Bounds on Singular Values

To specify piecewise-linear upper and lower bounds on the singular values of a system:

- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Singular Values**. A Create Requirement dialog box opens where you specify the lower or upper bounds on the singular values of the system. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.

- 2 Specify the requirement type using the **Type** drop-down list.
- 3 Specify the edge start and end frequencies and corresponding magnitude in the **Frequency** and **Magnitude** columns, respectively.
- 4 Insert or delete bound edges.

Click  to specify additional bound edges.


Select a row and click  to delete a bound edge.

- 5 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 6 Close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 7 (Optional) Plot the requirement.

- a In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.

- b In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.

Alternatively, you can use the Check Singular Value Characteristics block to specify bounds on the singular value. (Requires Simulink Control Design).

Specify Step Response Characteristics

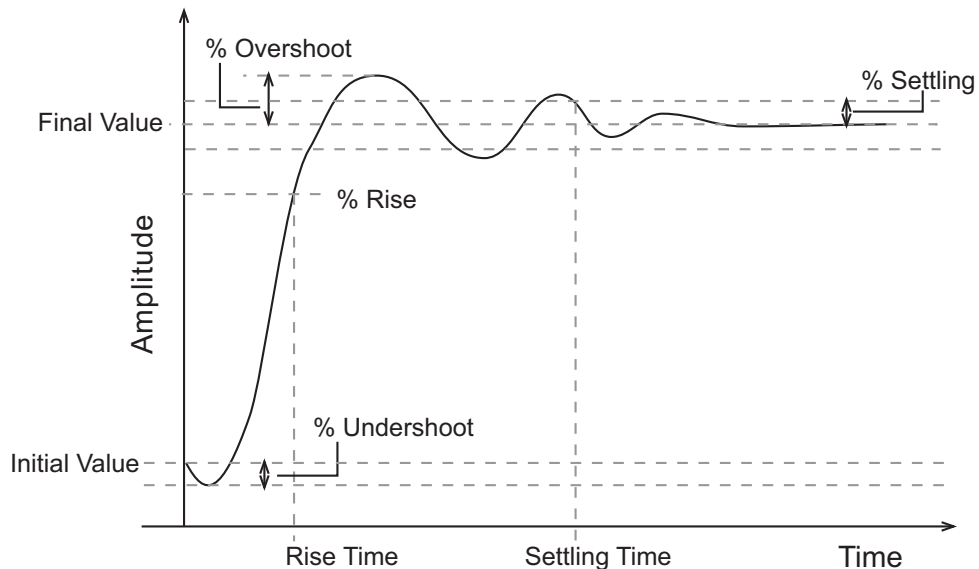
To apply a step response requirement to a linearization of your model (requires Simulink Control Design), specify the step response characteristics as follows:

- 1 Select a step response requirement from the Sensitivity Analysis tool.

In the **New Requirement** drop-down list of the tool, in the **New Frequency Domain Requirement** section, select **Step Response Envelope**.

A Create Requirement dialog box opens where you specify the step response requirements on a signal. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.

- 2 Specify the step response characteristics:




- **Initial value** — Input level before the step occurs
 - **Step time** — Time at which the step takes place
 - **Final value** — Input level after the step occurs
 - **Rise time** — The time taken for the response signal to reach a specified percentage of the step range. The step range is the difference between the final and initial values.
 - **% Rise** — The percentage of the step range used with **Rise time** to define the overall rise time characteristics.
 - **Settling time** — Time taken until the response signal settles within a specified region around the final value. This settling region is defined as the final step value plus or minus the specified percentage of the final value.
 - **% Settling** — The percentage of the final value that defines the settling range of settling time characteristic specified in **Settling time**.
 - **% Overshoot** — The amount by which the response signal can exceed the final value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.
 - **% Undershoot** — The amount by which the response signal can undershoot the initial value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.
- 3 Specify the systems to be bound.

To apply this requirement to a linearization of your Simulink model:

- a** In the **Select Systems to Bound** area, specify the simulation time at which the model is linearized in **Snapshot Times**. For multiple simulation snapshot times, specify a vector.
- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 4 Close the Create Requirement dialog box.

The requirement in the **Requirements** area of the tool is updated with the specified characteristics.

- 5 (Optional) Plot the requirement.

- a In the **Sensitivity Analysis** tab of the tool, in the **Requirements** area, select the requirement.
- b In the **Plots** tab of the tool, select the plot type to generate a graphical display of the requirement.

The plot is populated when you perform evaluation. A positive value indicates that the requirement has been violated.

Alternatively, you can use the Check Step Response Characteristics block to specify step response bounds for a signal.

Specify Custom Requirements


You can specify custom requirements, such as minimizing system energy. To specify custom requirements:

- 1 In the Sensitivity Analysis tool, in the **New Requirement** drop-down list, select **Custom Requirement**.

A Create Requirement dialog box opens where you specify the reference signal to track. A new requirement with the name specified in **Name** appears in the **Requirements** area of the tool.

- 2 Specify the requirement type in the **Type** drop-down menu.
- 3 Specify the name of the function that contains the custom requirement in **Function**. The field must be specified as a function handle using @. The function must be on the

MATLAB path. Click  to review or edit the function.

If the function does not exist, clicking  opens a template MATLAB file. Use this file to implement the custom requirement. The default function name is myCustomRequirement.

- 4 (Optional) To prevent the solver from considering specific parameter combinations, select **Error if constraint is violated**. Use this option for parameter-only constraints.

During an optimization iteration, the solver first evaluates requirements with this option selected.

- If the constraint is violated, the solver skips evaluating any remaining requirements and proceeds to the next combination of parameters in the parameter set.
- If the constraint is *not* violated, the solver evaluates the remaining requirements for the current combination of parameter values. If any of the remaining requirements bound signals or systems, then the solver simulates the model.

Note If you select this check box, then do not specify signals or systems to bound. If you *do* specify signals or systems, then this check box is ignored.

- 5 (Optional) Specify the signal or system, or both, to be bound.

You can apply this requirement to model signals, or a linearization of your Simulink model (requires Simulink Control Design), or both.


Click **Select Signals and Systems to Bound (Optional)** to view the signal and linearization I/O selection area.

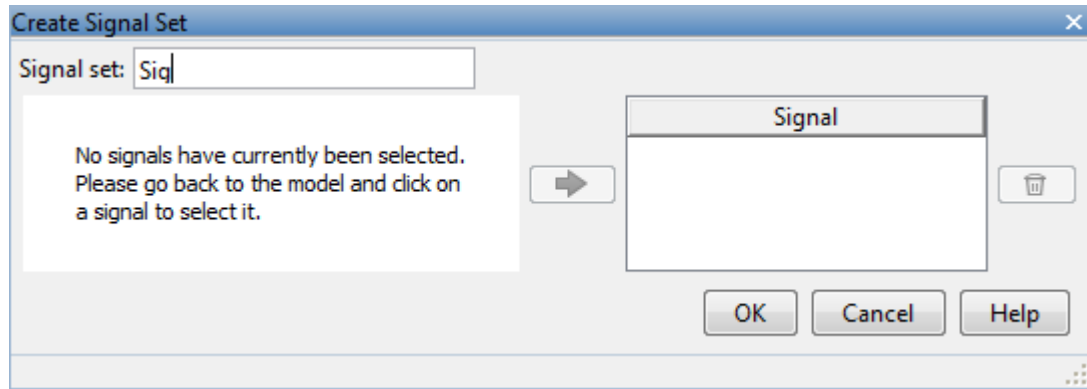
- To apply this requirement to a model signal:

In the **Signal** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, it appears in the list. Select the corresponding check box.

If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.
- b In the Simulink model window, click the signal to which you want to add a requirement.




The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the Create Requirement dialog box.

- To apply this requirement to a linear system:
 - a Specify the simulation time at which the model is linearized in **Snapshot Times**. For multiple simulation snapshot times, specify a vector.
 - b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box. For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 6 Select the check-box corresponding to the signal or system, and close the Create Requirement dialog box.

The requirement created in the **Requirements** area of the tool is updated with the specified characteristics.

You can now evaluate the requirement. For more information, see “Evaluate Design Requirements” on page 4-83. When you perform evaluation, a positive requirement value indicates that the requirement has been violated.

See Also

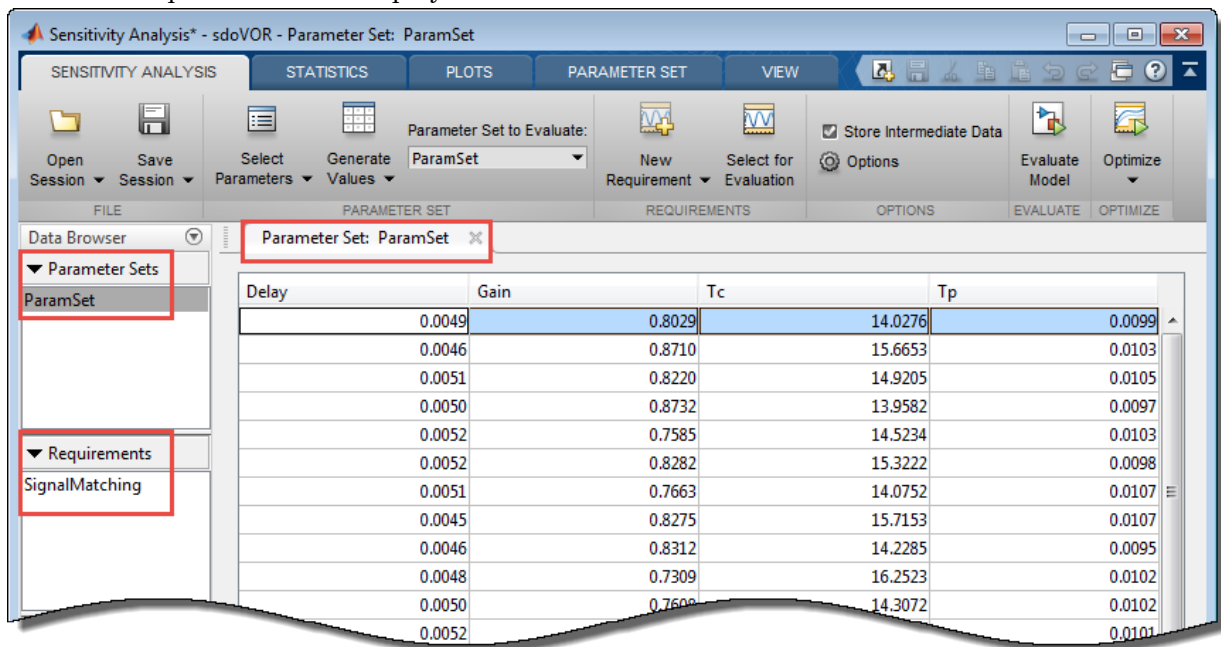
Related Examples

- “Specify Time-Domain Requirements” on page 4-28
- “Evaluate Design Requirements” on page 4-83
- “Design Exploration Using Parameter Sampling (GUI)” on page 4-147
- “Identify Key Parameters for Estimation (GUI)” on page 4-168

Evaluate Design Requirements

This topic shows how to evaluate your design requirements in the Sensitivity Analysis tool. After you generate values for your parameter set and specify design requirements, you can evaluate your design requirements (cost function) at each sample of parameter values in your parameter set.

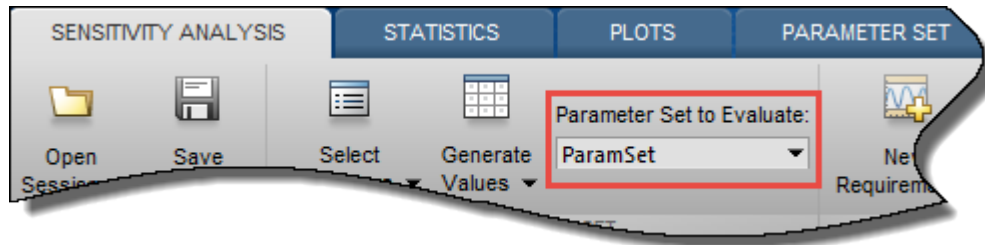
The generated parameter set, the corresponding parameter set table, and specified requirements are displayed in the tool.



To evaluate the design requirements:

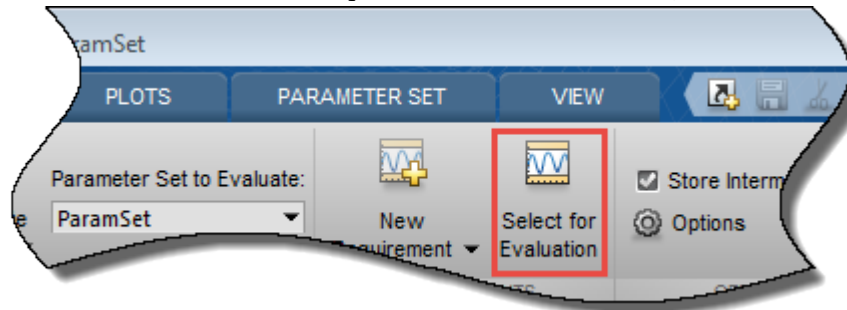
- 1 Select the parameter set to use for evaluation.

If you have multiple parameter sets defined, select the parameter set to use from the **Parameter Set to Evaluate** drop-down list.




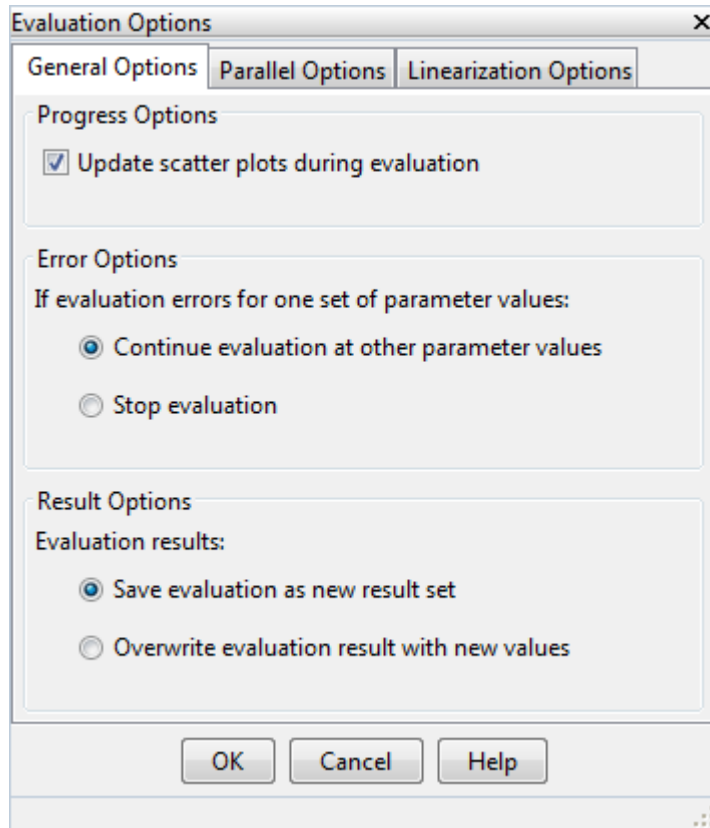
- 2 Select the requirements you want to evaluate.

If you have multiple requirements defined, they are all selected by default for evaluation. To exclude a requirement from evaluation, click **Select for Evaluation**.



- 3 Specify evaluation options.

Click  Options. In the Evaluation Options dialog box, in the **General Options** tab, specify options related to evaluation progress and results. If you have Parallel Computing Toolbox, you can specify options for parallel computing in the **Parallel Options** tab. If you have Simulink Control Design software, you can specify options for linearizing your model in the **Linearization Options** tab. For details about these options, click **Help**.



- 4 Speed up successive evaluations of design requirements that require the same logged signals and parameter values.

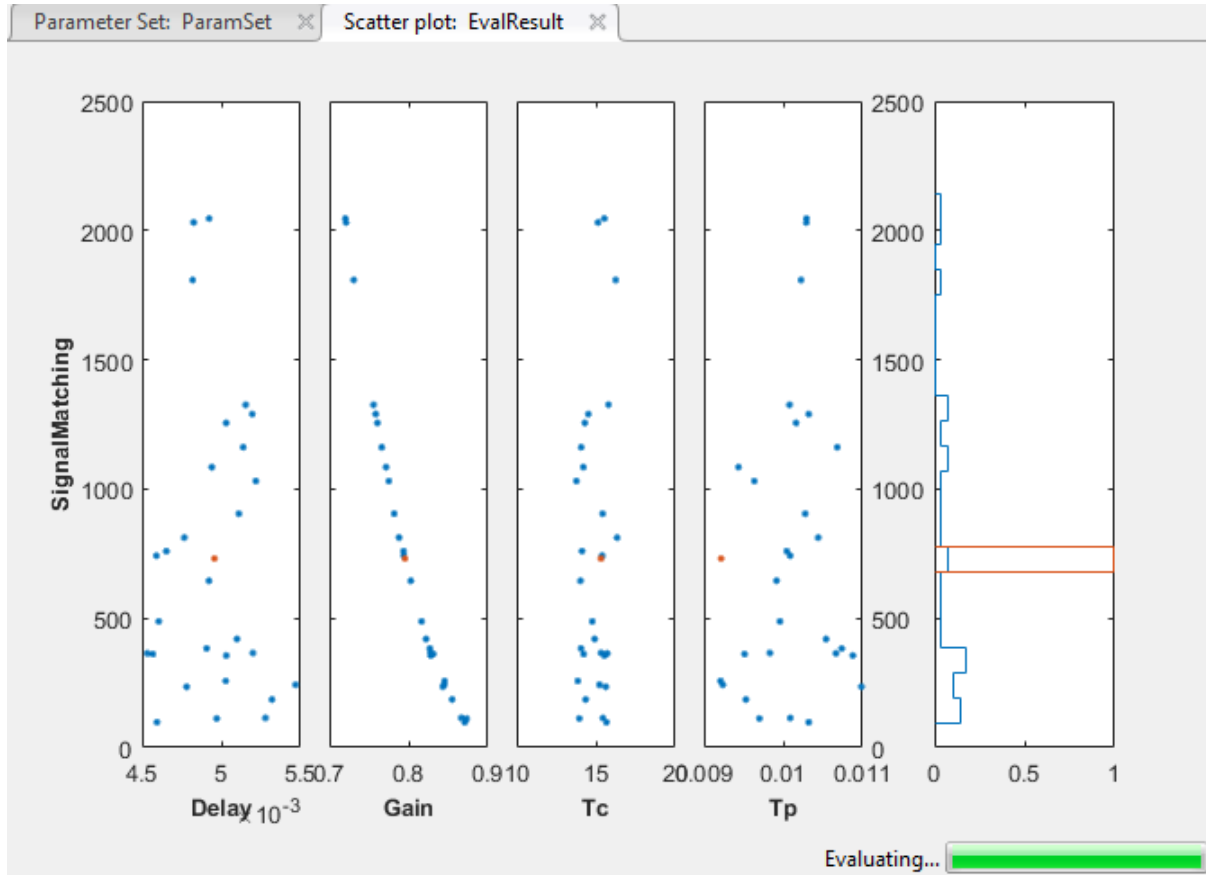
Click **Store Intermediate Data**. For more information, see “Store Intermediate Data in the Tool” on page 4-135.

- 5 Evaluate the design requirements for each sample in your parameter set.

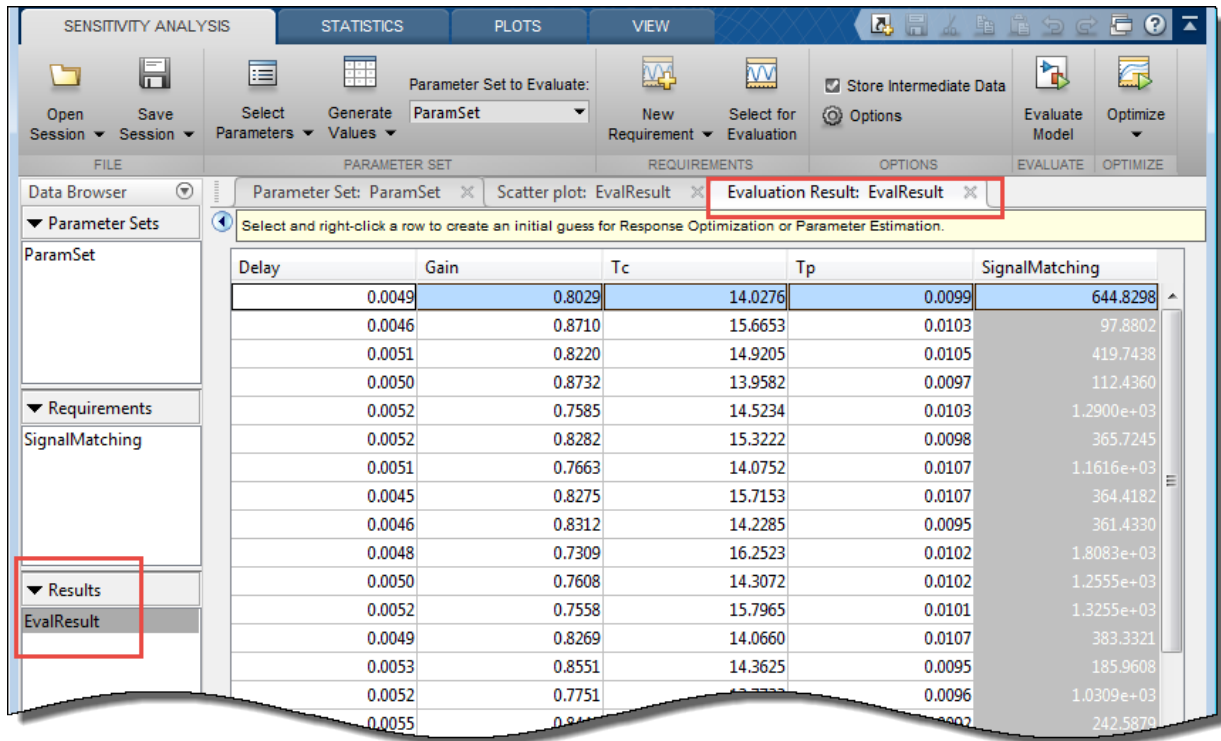
Click **Evaluate Model**. The cost function associated with your design requirement is evaluated using each sample in your parameter set. A sample corresponds to a row of values in the parameter set table.

The tool generates a evaluation result scatter plot. The plot updates to display the evaluated cost function value as a function of each parameter in the parameter set. The

last column of subplots displays histograms of the probability distribution of the evaluated cost function values.



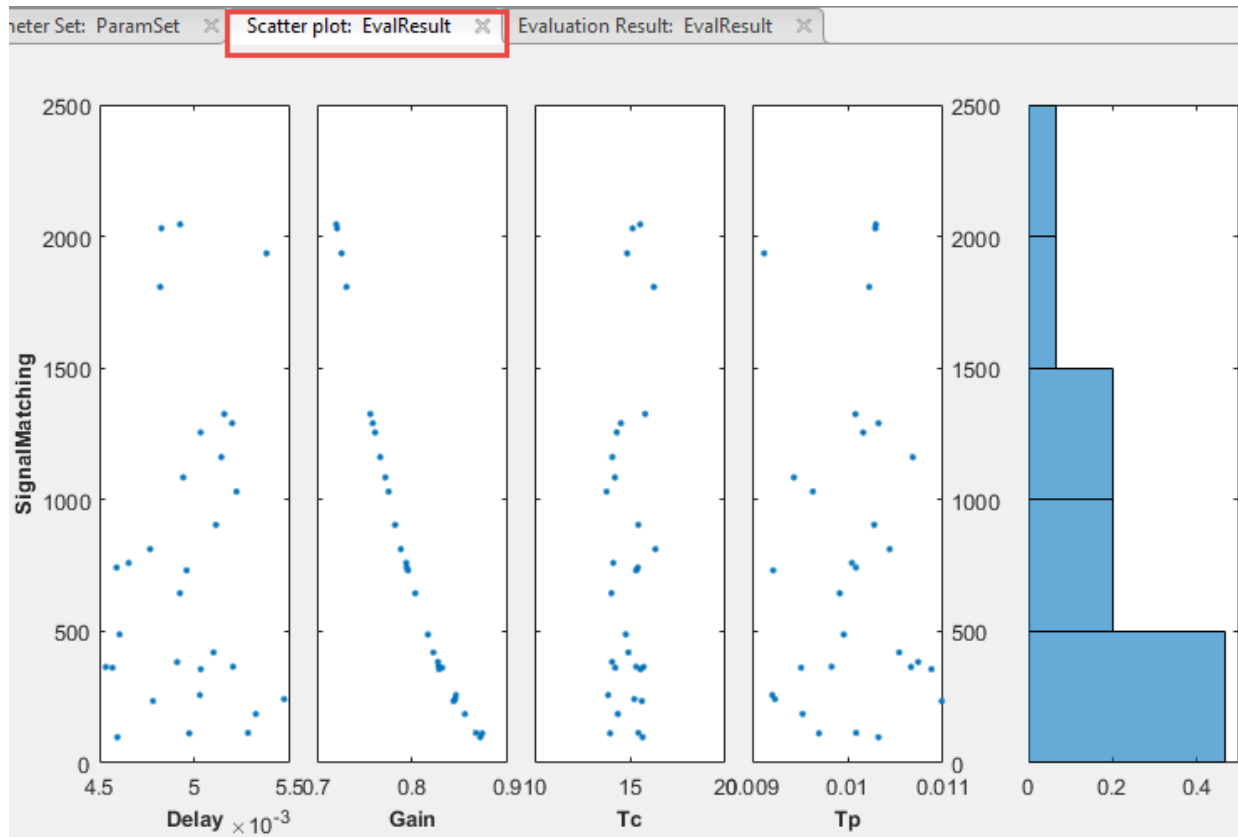
When the evaluation is complete, the tool displays the evaluated requirement (normalized cost function value) and the corresponding parameter values in the **Evaluation Result** table. The tool also creates a new variable with this information in the **Results** area.



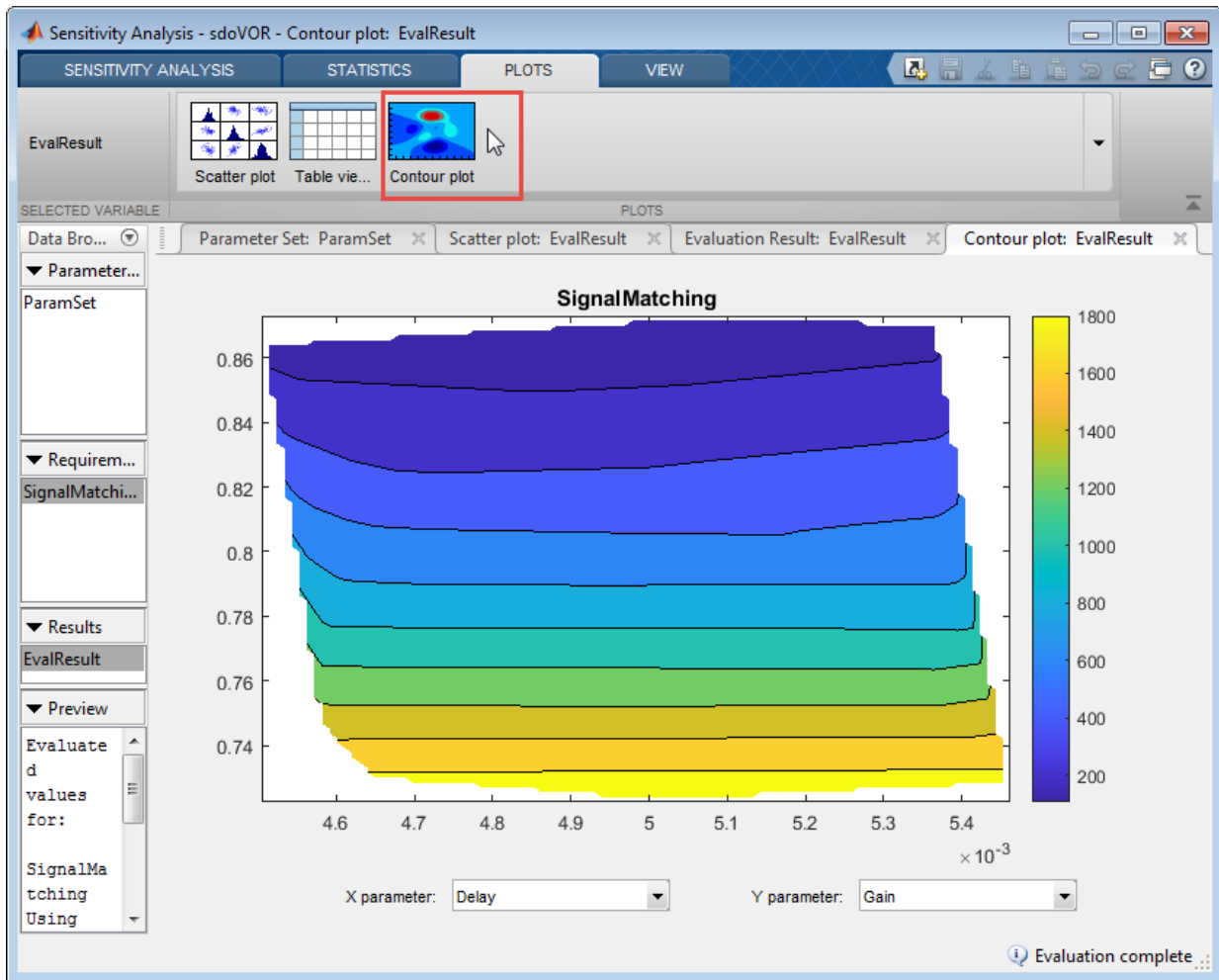
For requirements that involve a bound, a positive requirement value indicates that your requirement was violated, while a negative value indicates that the requirement was satisfied for that sample of parameter values. The parameter sample (row) corresponding to the least requirement value comes closest to satisfying your requirement.

You can employ the evaluation results to configure estimation or optimization of your parameters. For more information, see “Use Sensitivity Analysis to Configure Estimation and Optimization” on page 4-99.

The tool displays the final scatter plots and probability distribution histogram. The number of data points in each scatter plot equals the number of samples (rows) in the parameter set table. Use this plot to visually analyze the relation between parameters and requirements. For example, in this case, the `SignalMatching` requirement looks monotonically related to the `Gain` parameter. To learn more about the plot, see “Interact with Plots in the Sensitivity Analysis Tool” on page 4-106.



You can make a contour plot of the evaluated results. To do so, select the evaluated result in the **Results** area of the tool, and choose a contour plot in the **Plots** tab of the tool.



The contour plot shows that the requirement does not vary systematically as a function of Delay, but it does as a function of Gain.

You can also quantify the relation between parameters and requirements. For more information, see “Analyze Relation Between Parameters and Design Requirements” on page 4-91.

See Also

Related Examples

- “Generate Parameter Samples for Sensitivity Analysis” on page 4-10
- “Specify Time-Domain Requirements” on page 4-28
- “Specify Frequency-Domain Requirements” on page 4-66
- “Analyze Relation Between Parameters and Design Requirements” on page 4-91
- “Validate Sensitivity Analysis” on page 4-129
- “Identify Key Parameters for Estimation (GUI)” on page 4-168

Analyze Relation Between Parameters and Design Requirements

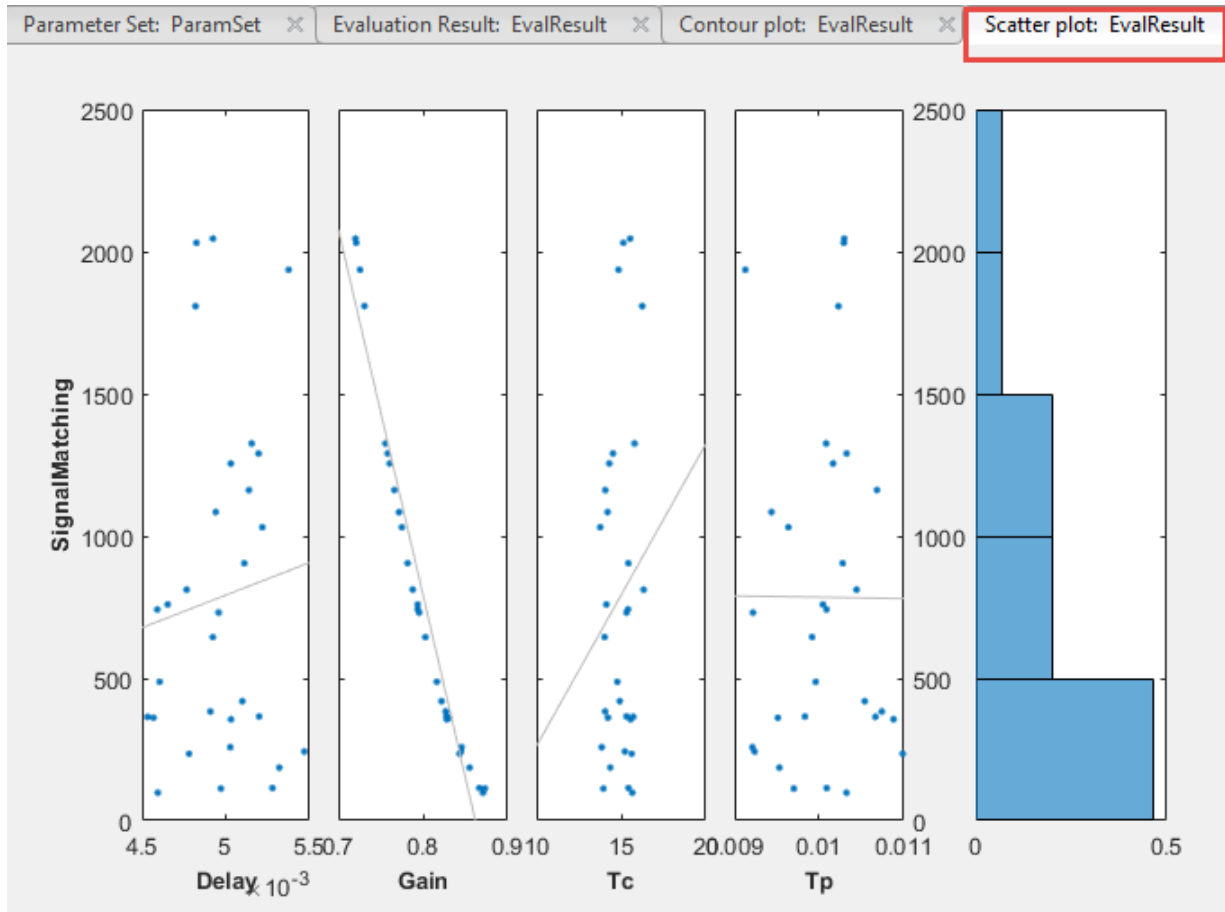
To analyze how the parameters and states (collectively referred to as parameters) of a Simulink model influence the design requirement on the model signals, you first generate samples of the parameters. You then define the cost function by creating a design requirement on the model signals, and evaluate the cost function for each sample. Finally, you analyze the relationship between the parameter variations and the cost function values. You can perform this analysis in the following ways:

- “Visual Analysis” on page 4-91
- “Statistical Analysis” on page 4-93

Visual Analysis

View a plot of the cost function evaluations against the parameter samples to identify trends. This method is informal and provides visual intuition about how the various parameters affect the cost function.

In the Sensitivity Analysis tool, after the evaluation is complete, an evaluation result scatter plot is generated in the tool. The plot displays the evaluated cost function value as a function of each parameter in the parameter set. The last column subplot displays the probability distribution of the evaluated cost function values. You can add a best-fit line to the scatter subplots by right-clicking in the plot, and selecting **Overlay linear fit** in the context menu. In this plot, the best-fit line indicates that the `Gain` parameter has a lot of influence on the requirement.



You can also plot a contour plot of the evaluated results. To learn more about these plots, see “Interact with Plots in the Sensitivity Analysis Tool” on page 4-106. For an example, see “Identify Key Parameters for Estimation (GUI)” on page 4-168.

At the command line, you can use tools such as:

- `sdo.scatterPlot` — Scatter plot of the parameter samples against the cost function evaluation
- `surf`, `mesh`, `contour` — 3-D plot of samples of two parameters against the cost function evaluation

For an example, see “Identify Key Parameters for Estimation (Code)” on page 4-217.

Statistical Analysis

In addition to visually analyzing the effect of parameters on the cost function, you can also compute statistics to quantify the relation.

Obtain summary statistics about the relationship between cost function evaluations and parameters samples. Available analysis methods include:

Method	Description
Correlation on page 4-94	Use to analyze how a model parameter and the cost function output are correlated.
Partial Correlation on page 4-95 (requires Statistics and Machine Learning Toolbox software)	Use to analyze how a model parameter and the cost function are correlated, removing the effects of the remaining parameters.
Standardized Regression on page 4-95	Use when you expect that the model parameters linearly influence the cost function.

For each of these methods, you specify what data to use for the analysis by choosing from the following analysis types:

- Linear analysis, also referred to as Pearson analysis — Uses raw data for analysis. Use linear analysis when you expect a linear relation between the parameters and cost function, and when the residuals about the best-fit line are expected to be normally distributed. Linear analysis is also recommended when the number of samples, and so the number of residual points is large.
- Ranked analysis, also referred to as Spearman analysis and ranked transformation — Uses ranks of data for analysis. Use ranked analysis when you expect a nonlinear monotonic relation between the parameters and the cost function and when the residuals about the best-fit line are not normally distributed. Ranked analysis is also recommended when the number of samples, and so the number of residual points is small.

Linear analysis retains information about intervals between data values, whereas ranked analysis does not. Suppose that you had the following data set:

x_1	x_2	y
9	20	340
5	60	106
2.3	50.4	870.5

Here x_1 and x_2 are model parameters, and y is the cost function. Each row represents a sample and the associated cost function evaluation.

The data is ranked on a per column basis. For example, when you rank the data in column 1 (x_1), which contains the entries 9, 5, and 2.3, the ranked data is equal to 3, 2, and 1. The ranked data set for the samples of x_1 , x_2 and y are as follows:

x_1	x_2	y
3	1	2
2	3	1
1	2	3

The ranked data set can be used for correlation, partial correlation, or standardized regression analysis.

- Kendall — Kendall’s tau rank correlation coefficient is calculated.

Applicable when the analysis method is Correlation. Requires Statistics and Machine Learning Toolbox software.

Correlation Method

Calculates the correlation coefficients, R . Use this method to analyze how a model parameter and the cost function outputs are correlated.

R is calculated as follows:

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

$$C = cov(x, y)$$

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

$$\mu_x = E[x]$$

$$\mu_y = E[y]$$

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 .

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.

Partial Correlation Method

Calculates the partial correlation coefficients, R . This method requires Statistics and Machine Learning Toolbox software. Use this method 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` from the Statistics and Machine Learning Toolbox software.

Standardized Regression Method

Calculates the standardized regression coefficients, R . Use this method when you expect that the model parameters linearly influence the cost function.

R is calculated as follows:


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

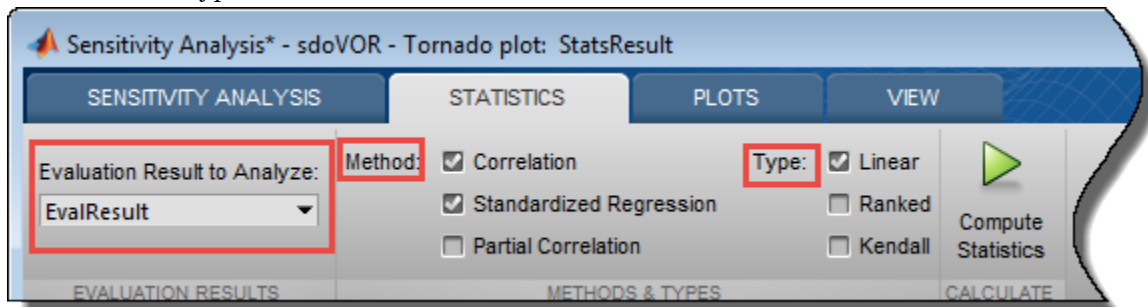
Consider a single sample (x_1, \dots, x_{N_p}) and the corresponding single output, y . b_x is the regression coefficient vector calculated using least squares assuming a linear model

$\hat{y} = b_0 + \sum_{i=1}^{N_p} \hat{b}_{x_i} x_i$. R standardizes each element of b_x by multiplying it with the ratio of the standard deviation of the corresponding x sample (σ_x) to the standard deviation of y (σ_y).

Perform Statistical Analysis

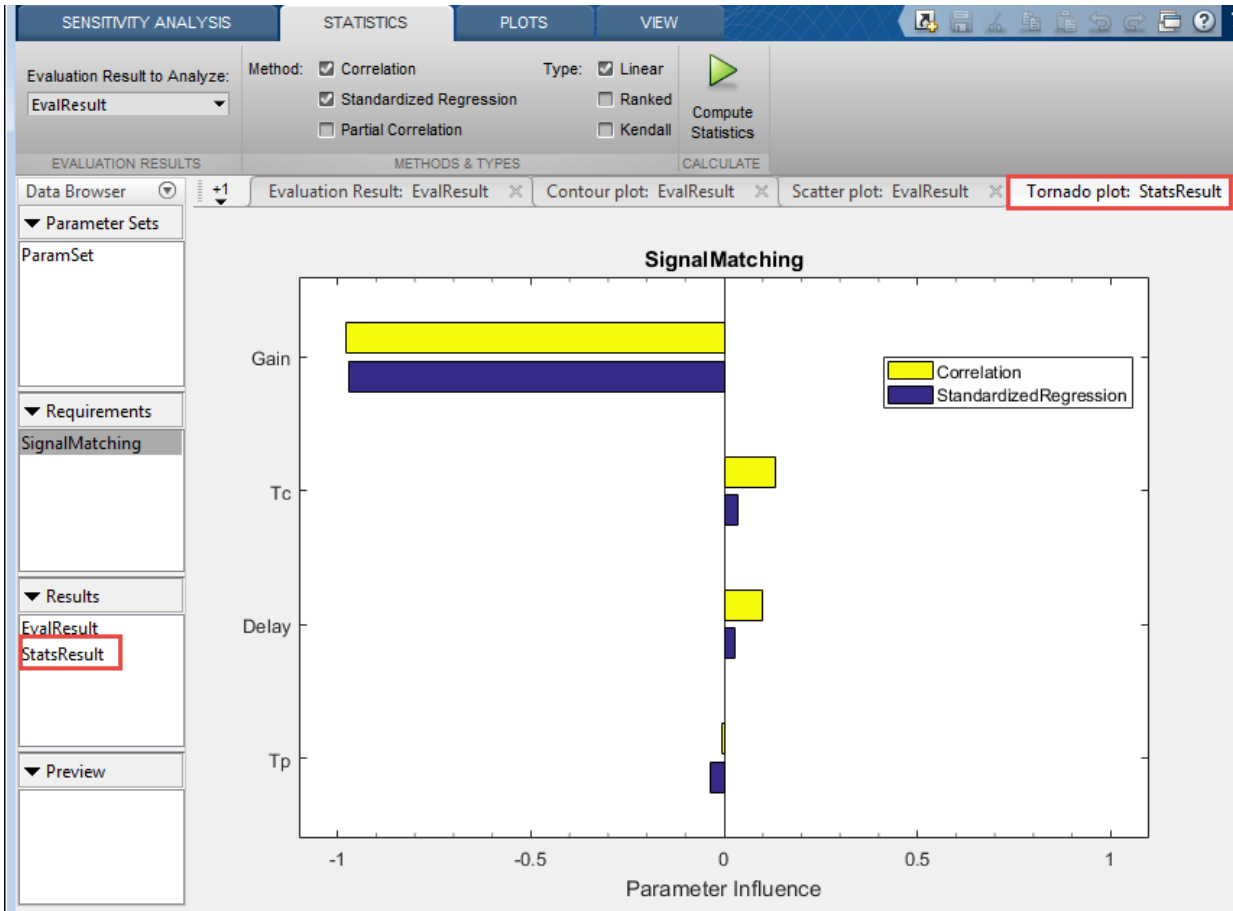
In the Sensitivity Analysis tool, after you have evaluated the design requirements on page 4-83, specify the analysis methods and types in the **Statistics** tab of the tool.

Select the evaluation results you want to analyze in the **Evaluation Results to Analyze** list. After that, you specify the analysis methods and types, and click  **Compute Statistics**. You can compute all applicable combinations of analysis methods and types.



The results of the analysis are returned in the `StatsResult` variable, in the **Results** area of the tool. In this case, the `StatsResult` variable includes the linear (Pearson) correlation coefficients and linear standardized regression coefficients calculated between the cost function and each parameter. To see the coefficients, right-click `StatsResult`, and select **Open** in the context-menu.

A tornado plot is generated that displays the results of the analysis in order of influence of parameters on the cost function. The parameter that most influences the cost function is displayed on the top. As was seen in the results scatter plot, in this tornado plot the Gain parameter has the most influence on the design requirement cost function.



To learn more about tornado plots, see “Interact with Plots in the Sensitivity Analysis Tool” on page 4-106. For an example, see “Identify Key Parameters for Estimation (GUI)” on page 4-168.

At the command line, specify the analysis methods and types using `sdo.analyze`. This function performs linear correlation analysis by default. To specify other analysis methods, use `sdo.AnalyzeOptions`. For an example, see “Identify Key Parameters for Estimation (Code)” on page 4-217.

See Also

`sdo.AnalyzeOptions` | `sdo.analyze` | `sdo.evaluate` | `sdo.sample`

Related Examples

- “What is Sensitivity Analysis?” on page 4-2
- “Evaluate Design Requirements” on page 4-83
- “Identify Key Parameters for Estimation (GUI)” on page 4-168
- “Identify Key Parameters for Estimation (Code)” on page 4-217
- “Use Sensitivity Analysis to Configure Estimation and Optimization” on page 4-99

Use Sensitivity Analysis to Configure Estimation and Optimization

This topic shows how to use the results generated in the Sensitivity Analysis tool to configure parameter estimation or response optimization.

You can use sensitivity analysis to evaluate how the parameters of a Simulink model influence the model output or model design requirements. You first generate samples of the parameters, and then define a cost function by creating a design requirement on the model signals. For more information see, “Generate Parameter Samples for Sensitivity Analysis” on page 4-10, “Specify Time-Domain Requirements” on page 4-28, and “Specify Frequency-Domain Requirements” on page 4-66. You then evaluate the requirement (cost function) for each sample. You can use the evaluated results to configure parameter estimation or response optimization in the following ways:

- 1 Analyze the relationship between the parameters and the evaluated requirement values, and rank parameters in order of influence. For more information, see “Analyze Relation Between Parameters and Design Requirements” on page 4-91. You can then choose to estimate or optimize the more influential parameters.
- 2 Obtain initial guesses for parameter values for estimation or optimization.

After you evaluate the requirements in the Sensitivity Analysis tool, the evaluated requirement (cost function value) and the corresponding parameter values are displayed in the **Evaluation Result** table of the tool. A new variable, `EvalResults`, with this information is created in the **Results** area of the tool. For example, the table below lists the evaluated signal matching requirement and corresponding values for the parameters `Gain`, `Delay`, `Tc`, and `Tp` of the model, `sdoVOR`.

4 Sensitivity Analysis

The screenshot displays the Sensitivity Analysis software interface. The top menu bar includes 'SENSITIVITY ANALYSIS', 'STATISTICS', 'PLOTS', and 'VIEW'. Below the menu bar are several toolbars with icons for file operations (Open, Save), parameter selection (Select Parameters, Generate Values), requirements (New Requirement, Select for Evaluation), options (Store Intermediate Data, Options), and evaluation (Evaluate Model, Optimize). The main window is divided into a 'Data Browser' on the left and a central table. The 'Data Browser' shows a tree view with 'Parameter Sets' (ParamSet), 'Requirements' (SignalMatching), and 'Results' (EvalResult). The central table displays the following data:

Delay	Gain	Tc	Tp	SignalMatching
0.0049	0.8029	14.0276	0.0099	644.8298
0.0046	0.8710	15.6653	0.0103	97.8802
0.0051	0.8220	14.9205	0.0105	419.7438
0.0050	0.8732	13.9582	0.0097	112.4360
0.0052	0.7585	14.5234	0.0103	1.2900e+03
0.0052	0.8282	15.3222	0.0098	365.7245
0.0051	0.7663	14.0752	0.0107	1.1616e+03
0.0045	0.8275	15.7153	0.0107	364.4182
0.0046	0.8312	14.2285	0.0095	361.4330
0.0048	0.7309	16.2523	0.0102	1.8083e+03
0.0050	0.7608	14.3072	0.0102	1.2555e+03
0.0052	0.7558	15.7965	0.0101	1.3255e+03
0.0049	0.8269	14.0660	0.0107	383.3321
0.0053	0.8551	14.3625	0.0095	185.9608
0.0052	0.7751	15.5732	0.0096	1.0309e+03
0.0055			0.0092	242.5379

To extract parameter values to use as an initial guess during estimation or optimization:

- a Sort the evaluated cost function values in ascending order by clicking the evaluated requirement column.

Parameter Set: ParamSet x Scatter plot: EvalResult x Evaluation Result: EvalResult x

Select and right-click a row to create an initial guess for Response Optimization or Parameter Estimation.

Delay	Gain	Tc	TP	SignalMatching
0.0046	0.8710	15.6653	0.0103	97.8802
0.0050	0.8732	13.9582	0.0097	112.4360
0.0053	0.8666	15.4429	0.0101	114.2653
0.0053	0.8551	14.3625	0.0095	185.9608
0.0048	0.8432	15.6278	0.0110	234.8702
0.0055	0.8449	15.2286	0.0092	242.5879
0.0050	0.8456	13.8582	0.0092	257.6335
0.0050	0.8281	15.5501	0.0109	355.8483
0.0046	0.8312	14.2285	0.0095	361.4330
0.0045	0.8275	15.7153	0.0107	364.4182
0.0052	0.8282	15.3222	0.0098	365.7245
0.0049	0.8269	14.0660	0.0107	383.3321
	0.8220		0.0105	419.7438

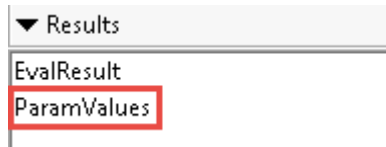
- b To choose the parameter values that minimize the cost function, right-click corresponding row, and select **Extract Parameter Values**.

Delay	Gain	Tc	TP	SignalMatching
0.0046	0.8710	15.6653	0.0103	97.8802
0.0050	0.8732	13.9582	0.0097	112.4360
0.0053	0.8666	15.4429	0.0101	114.2653
0.0053	0.8551	14.3625	0.0095	185.9608
0.0048	0.8432	15.6278	0.0110	234.8702
0.0055	0.8449	15.2286	0.0092	242.5879
0.0050	0.8456	13.8582	0.0092	257.6335
	0.8281		0.0109	355.8483

Context menu options:

- Extract Parameter Values
- Plot all result data

A new variable, ParamValues, is created in the **Results** area of the tool.

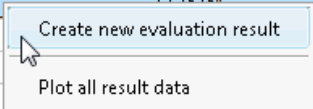


- c When exporting to a Parameter Estimation or Response Optimization tool session on page 4-103, choose this variable to specify the initial guess for parameters.
- 3 To test the robustness of your design during optimization in the Response Optimization tool, specify the values for uncertain parameters. For more information, see “Optimizing Parameters for Robustness” on page 3-201.

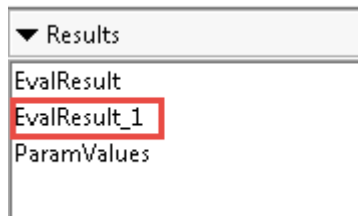
You can specify the values for uncertain parameters using all the parameter values in `EvalResults`. You can also choose a subset of the parameter values:

- a Select the relevant rows of parameter values in the **Evaluation Result** table of the tool.

Delay	Gain	Tc	Tp	SignalMatching /
0.0054	0.8694	15.9519	0.0100	100.2616
0.0046	0.8668	13.9571	0.0098	137.8549
0.0047	0.8687	13.9969	0.0093	138.2714
0.0049	0.8545	13.9865	0.0099	192.0046
0.0052	0.8529	13.7515	0.0102	196.9189
0.0053	0.8503		0.0094	215.8378
0.0052	0.8447		0.0109	222.4661
0.0055	0.8412		0.0095	266.2807
0.0048	0.8406	16.4884	0.0092	268.3061
0.0052	0.8186	15.5515	0.0101	451.3620
	0.8136			521.1111



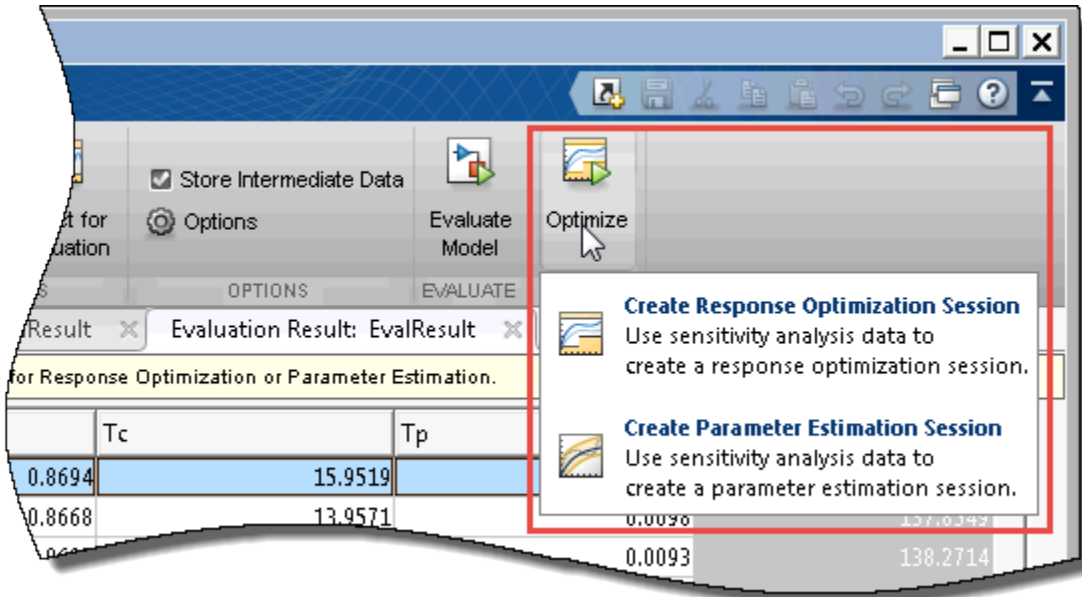
- b Right-click, and select **Create new evaluation result**. A new variable is created in the **Results** area of the tool.



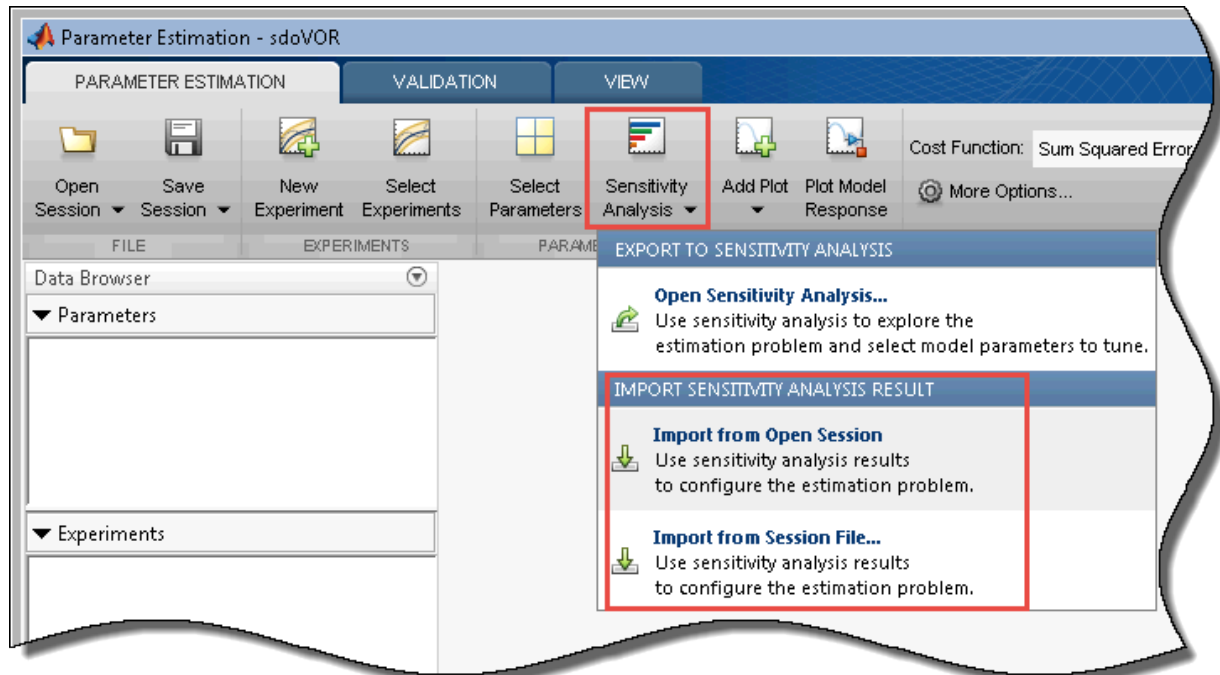
- c When exporting to a Response Optimization tool session on page 4-103, choose this variable to specify the uncertain variables.

Export Sensitivity Analysis Results

You can export results from Sensitivity Analysis tool to the Parameter Estimation and Response Optimization tools. To do so, in the Sensitivity Analysis tool, click **Optimize**. In the drop-down menu, choose the tool to export to.



Alternatively, if you have an open Parameter Estimation or Response Optimization tool session, in these tools, click **Sensitivity Analysis**. In the drop-down menu, choose **Import from Open Session** or **Import from Session file**. The latter option loads results from a previously saved Sensitivity Analysis tool session.



Note

- Only signal matching requirements are exported from Sensitivity Analysis to a Parameter Estimation tool session. In the Parameter Estimation tool, they are referred to as experiments.
- Only requirements other than signal matching requirements are exported from Sensitivity Analysis to a Response Optimization tool session.

See Also

Related Examples

- “What is Sensitivity Analysis?” on page 4-2

- “Design Exploration Using Parameter Sampling (GUI)” on page 4-147
- “Identify Key Parameters for Estimation (GUI)” on page 4-168

Interact with Plots in the Sensitivity Analysis Tool

This topic shows how to interact with and interpret plots generated in the Sensitivity Analysis tool.

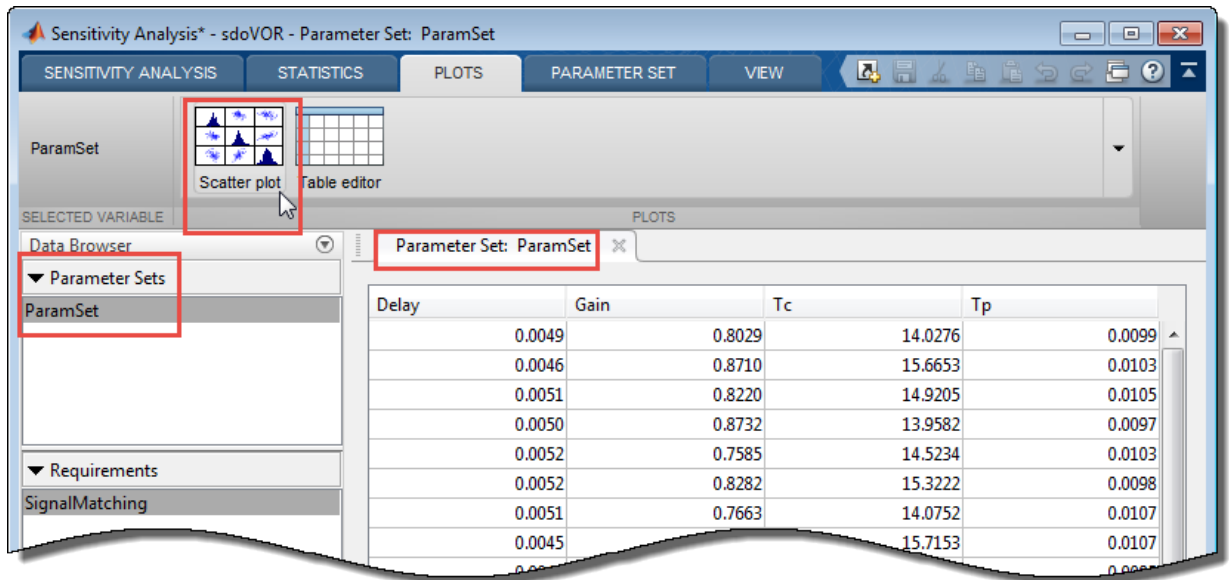
In this section...
“Parameter Set Plots” on page 4-106
“Requirement Plots” on page 4-114
“Evaluated Result Scatter Plots” on page 4-116
“Evaluated Result Contour Plots” on page 4-123
“Statistical Analysis Tornado Plots” on page 4-125

Parameter Set Plots

After you have generated parameter values for sensitivity analysis, you can plot the generated parameter set. For information about parameter generation, see “Generate Parameter Samples for Sensitivity Analysis” on page 4-10.

The tool displays the generated parameter set and the corresponding parameter set table. The number of rows in the parameter set table correspond to the number of samples you specified. To plot the generated parameters in the tool:

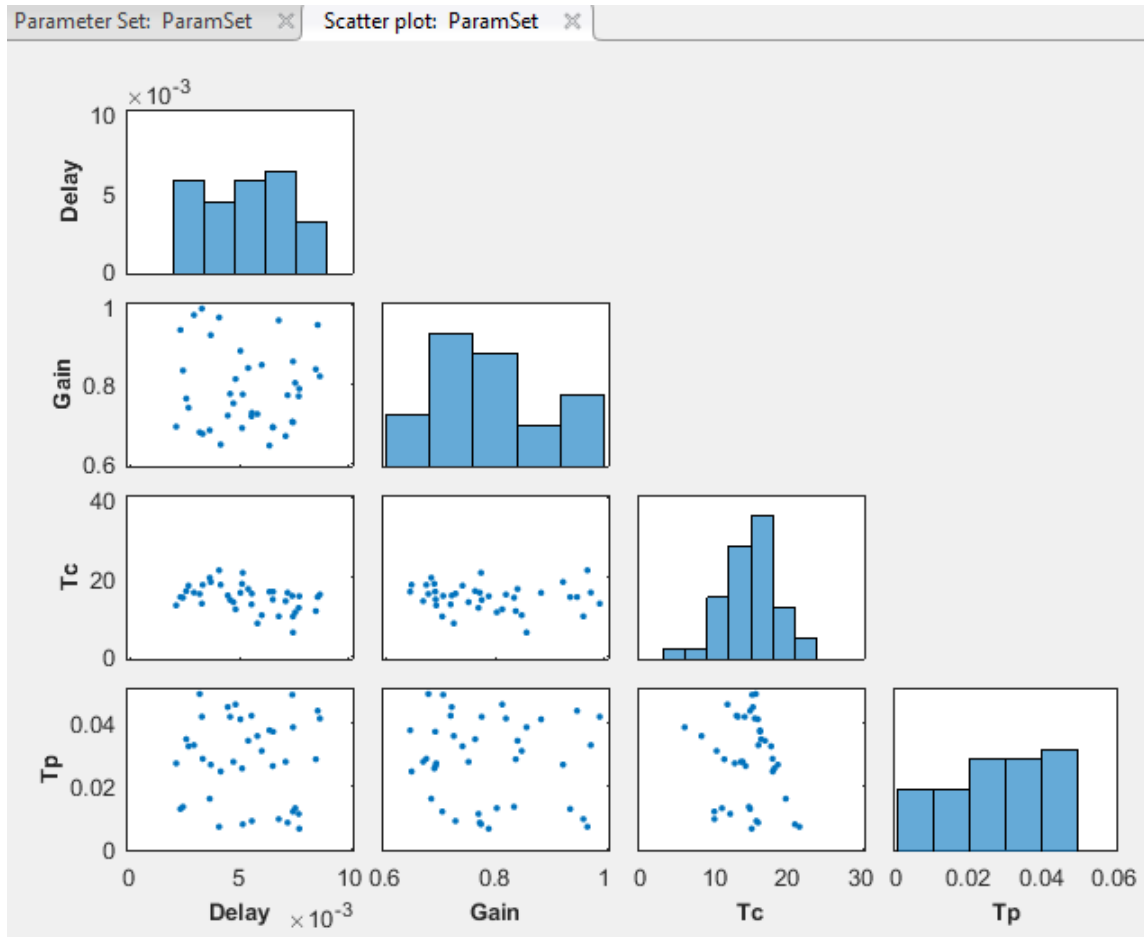
- 1 Select the generated parameter set in the **Parameter Sets** area of the tool.



2 On the **Plots** tab, select **Scatter Plot**.

Alternatively, right-click the parameter set, and select **Plot** in the drop-down menu.

The diagonal subplots display the histograms of generated parameter values. The off-diagonal subplots are pair-wise scatter plots of the parameters. The number of data points in each scatter plot equals the number of rows in the parameter set table.



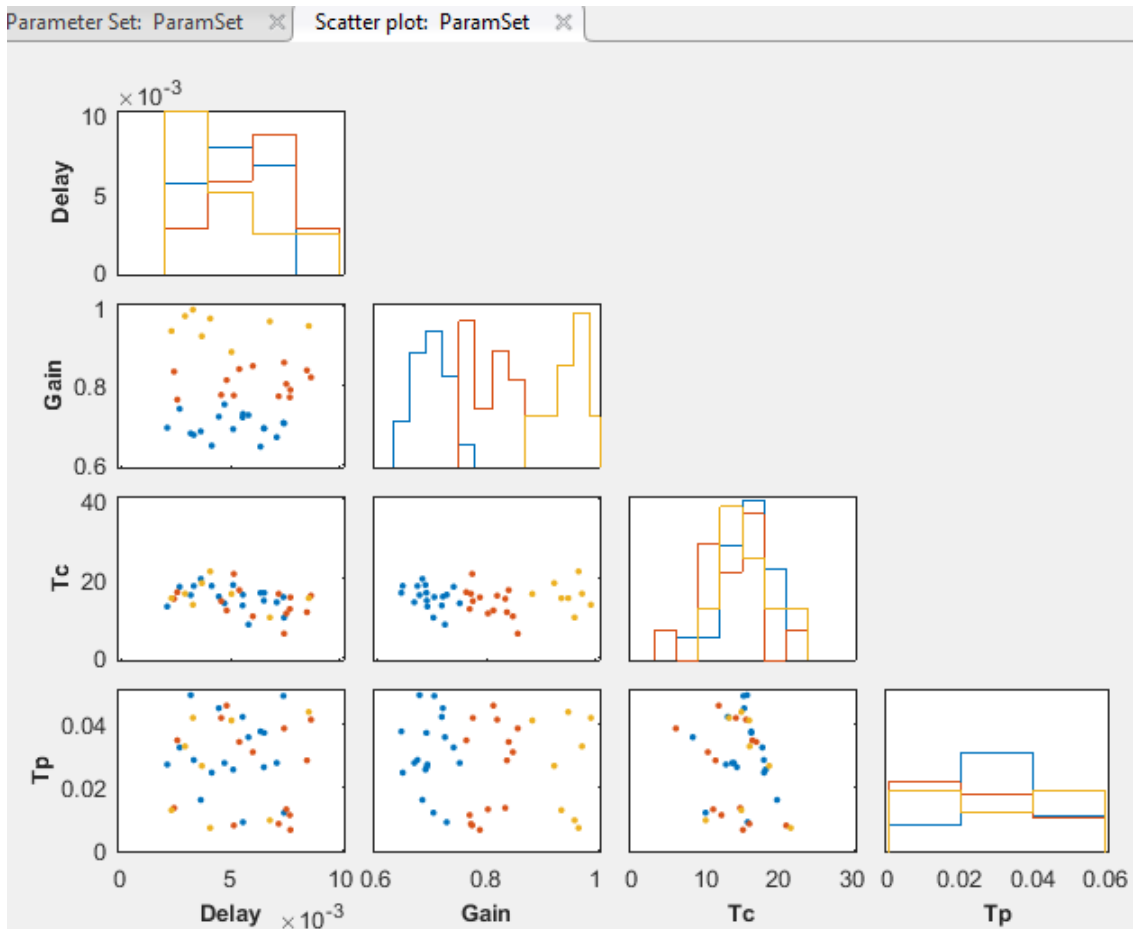
You can inspect the histograms to ensure that the generated parameter values match the desired parameter distributions within the constraints of a finite sample size. Inspect the off-diagonal scatter plots to ensure that any specified correlations between parameters are present. For more information, see “Inspect the Generated Parameter Set” on page 4-129.

- 3 To access additional plot features, right-click in the white area of any scatter plot.

You can choose from the following options:

- **Variables** — Select the parameters to plot.
- **Groups** — Select grouping variables for the plots, and configure how the groups are displayed.

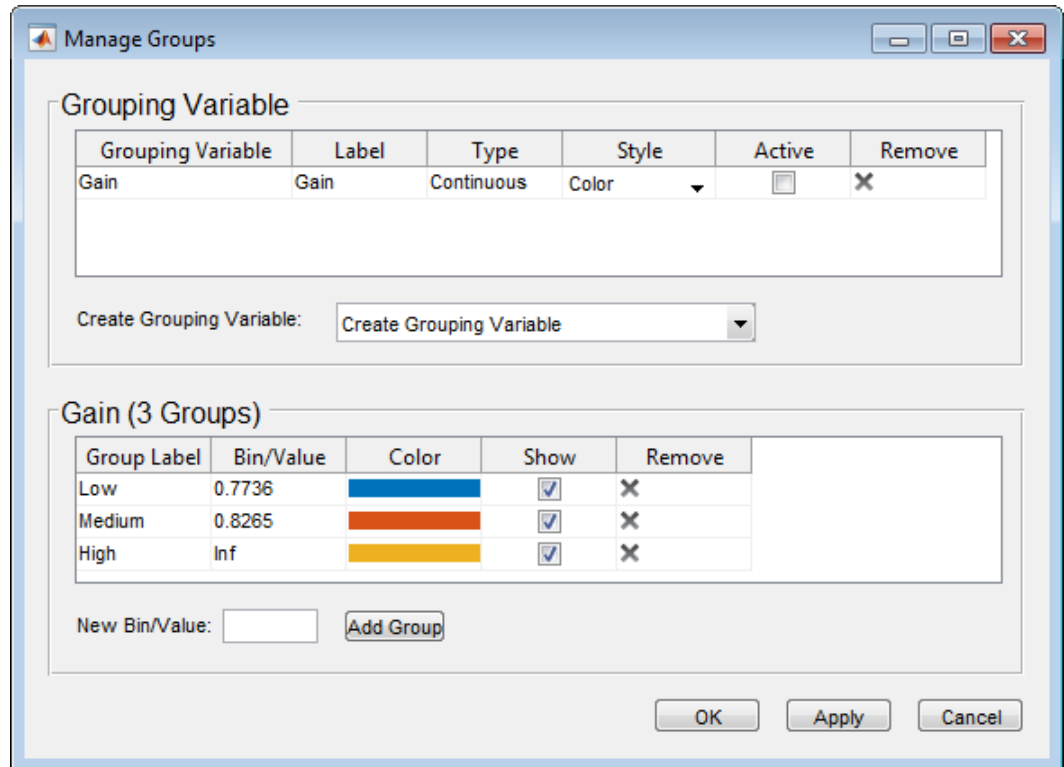
To select a parameter as a grouping variable, click **Groups > New Grouping Variable**. For example, the following plot is generated when the grouping variable is Gain.



The tool creates three groups based on low, medium, and high values of the grouping variable. The tool computes these grouping values, but you can change them in the **Manage Groups** dialog box. The second diagonal plot shows the

distribution of the gain values in the low (blue), medium (red), and high (yellow) groups. The other diagonal plots show the distribution of the remaining parameters when the corresponding gain value is low, medium, or high. The off-diagonal scatter plots show points belonging to the same group using the same color.

You can demarcate the groups based on marker size and marker type instead of color, add more groups corresponding to the grouping variable, and change the grouping values. You can also add more grouping variables. To do so, click **Groups > Manage Groups**.

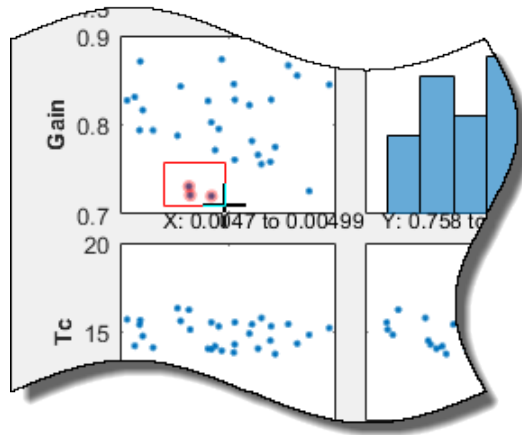


In the **Manage Groups** dialog box, you configure how the groups are displayed. You can perform tasks such as:

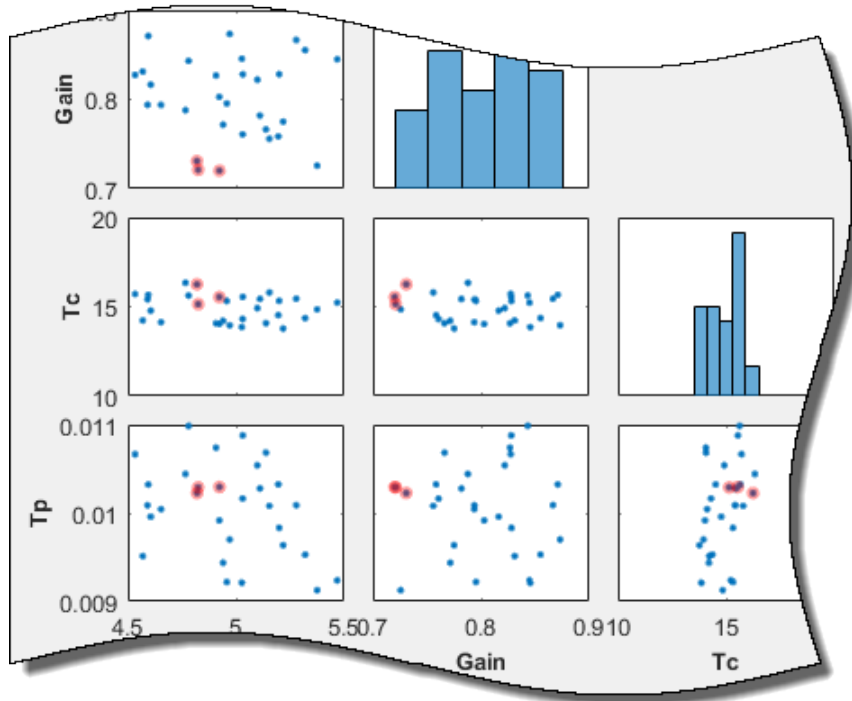
- Select the plotting **Style** as either `Color`, `MarkerSize`, or `MarkerType`. In the plots, the tool uses the selected style to demarcate the groups corresponding to a grouping variable.
- Select whether a grouping variable is **Active**. If a grouping variable is inactive, the scatter plot points are not demarcated in groups corresponding to that variable. To delete a grouping variable, click **✕** in the corresponding **Remove** column.
- Add more grouping variables using the **Create Grouping Variable** drop-down list.
- For a grouping variable, specify the range of values for each group in the **Bin/Value** column. For example, currently the dialog box shows that the `Gain` values in the groups are:
 - **Low** — below 0.7736
 - **Medium** — 0.7736–0.8265
 - **High** — above 0.8265

To change the **Low** group values to be 0.79 or lower, type 0.79 in the corresponding row of the **Bin/Value** column.

- Add more groups corresponding to a grouping variable. For example, to add a group with values from 0.8265 through 0.9, type 0.9 in **New Bin/Value**, and click **Add Group**.
- **Upper triangle plot** — Plot the off-diagonal subplots above the diagonal in addition to the existing plots.
- **Marginal Box Plots** — Requires Statistics and Machine Learning Toolbox software. Plot box plots for each of the parameters in the parameter set, and choose the position of the plots.
- **Histograms** — Plot the probability distribution of the parameters, and choose the position of the plots.
- **Kernel Density Plots** — Requires Statistics and Machine Learning Toolbox software. Plot the probability distribution of the parameters using a kernel density estimator, and choose the position of the plots. For more information, see “Kernel Distribution” (Statistics and Machine Learning Toolbox).
- **Overlay linear fit** — Plot the best-fit line on the scatter subplots. You can choose to plot the best-fit lines for one, all, a row, or a column of scatter subplots.
- **Enable brushing/data selection** — Enable selection of data points in the scatter subplots.



When you highlight parameter values in one plot, the values corresponding to other parameters from the same row in the parameter set table are also highlighted. In addition, the rows in the parameter set table that correspond to these values are highlighted.



Delay	Gain	Tc	TP	
0.0049		0.8029	14.0276	0.0099
0.0046		0.8710	15.6653	0.0103
0.0051		0.8220	14.9205	0.0105
0.0050		0.8732	13.9582	0.0097
0.0052		0.7585	14.5234	0.0103
0.0052		0.8282	15.3222	0.0098
0.0051		0.7663	14.0752	0.0107
0.0045		0.8275	15.7153	0.0107
0.0046		0.8312	14.2285	0.0095
0.0048		0.7309	16.2523	0.0102
0.0050		0.7608	14.3072	0.0102
0.0052		0.7558	15.7965	0.0101
0.0049		0.8269	14.0660	0.0107
0.0053		0.8551	14.3625	0.0095
0.0052		0.7751	13.7733	0.0096
0.0055		0.8449	15.2286	0.0092
0.0050		0.8281	15.5501	0.0109
0.0048		0.7211	15.1398	0.0103
0.0046		0.8163	14.7772	0.0100
0.0051		0.7819	15.4333	0.0103
0.0053		0.8666	15.4429	0.0101
0.0049		0.7202	15.5371	0.0103
0.0046		0.7940	15.4074	0.0101
0.0048		0.7879	16.3355	0.0104

To remove the highlighting, invert the selection to all other data points in the plot, or disable the feature, right-click the highlighted data points, and choose from the context-menu.

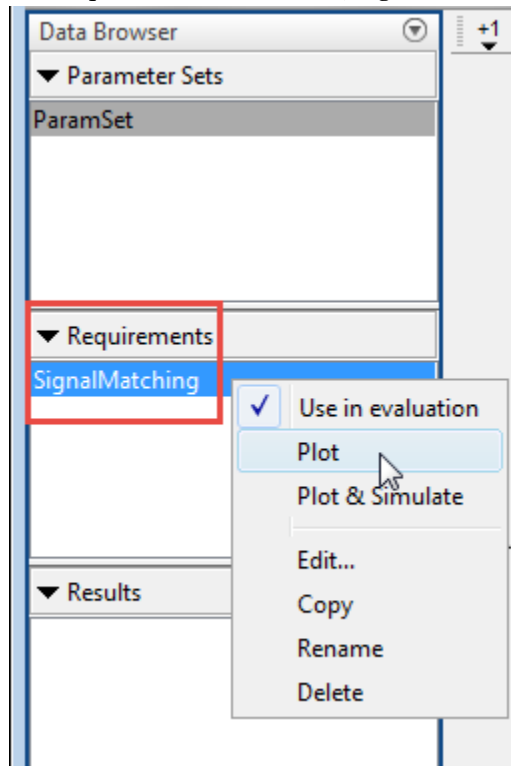
- **Pop-out plot** — View a subplot in a new window.

Requirement Plots

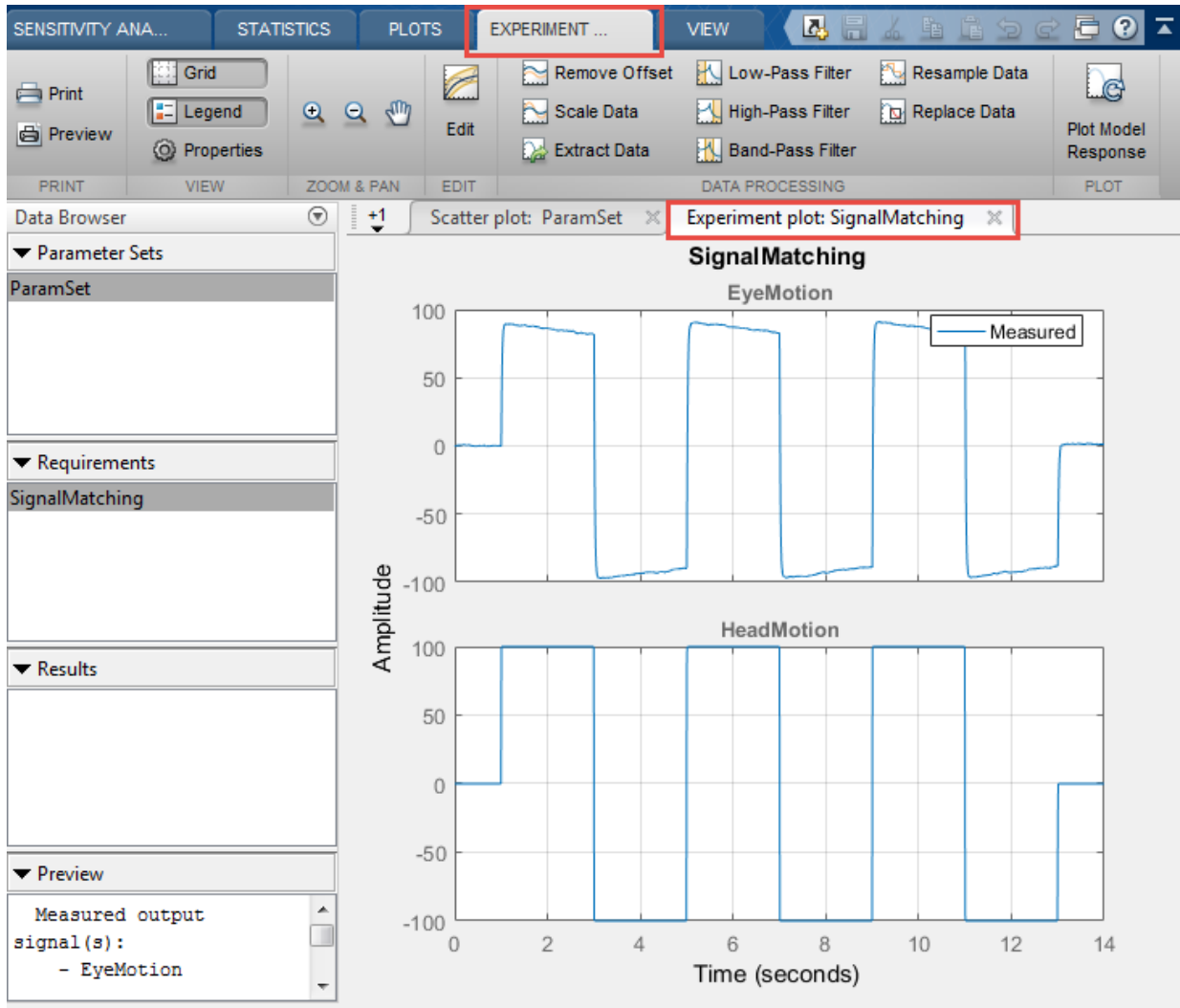
After you have specified design requirements, you can plot the requirements and associated model response. For information about specifying the requirements, see

“Specify Time-Domain Requirements” on page 4-28 and “Specify Frequency-Domain Requirements” on page 4-66.

The specified requirements are displayed in the **Requirements** area of the tool. To plot the requirement in the tool, right-click the requirement, and select **Plot**.



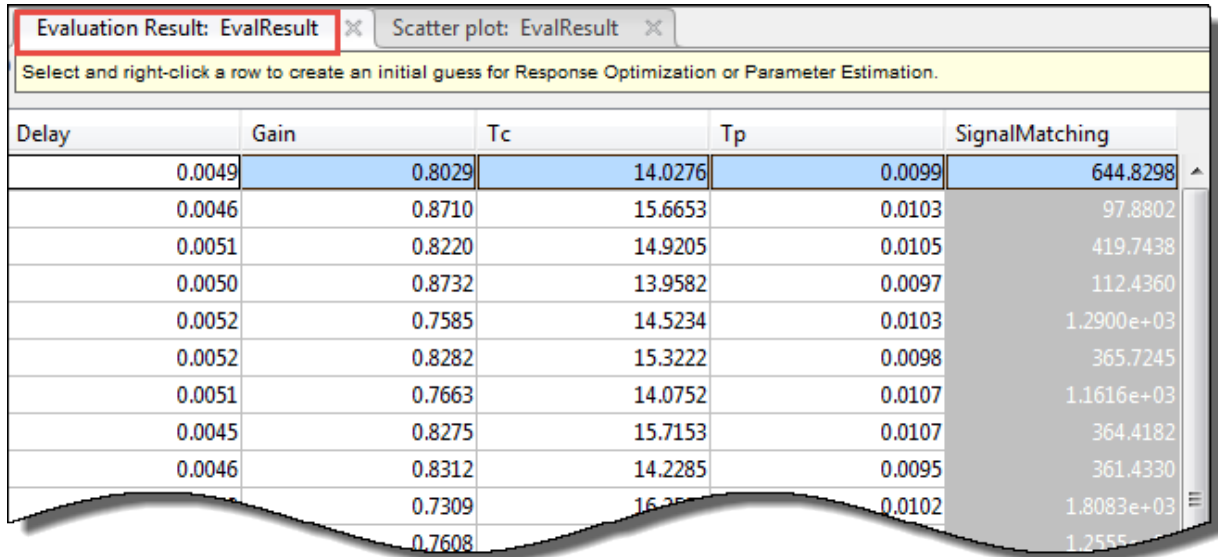
Alternatively, select the requirement, and in the **Plots** tab of the tool, select the plot type. A plot is generated and a new tab associated with the plot appears in the tool. In the new tab, you can perform additional tasks such as preprocessing imported data on page 1-16 (for signal matching requirement only), zooming, and plotting the associated model response. The model response is the signal or system on which the requirement is applied. The response is plotted using the parameter values specified in the model workspace and is not updated during evaluation.



Evaluated Result Scatter Plots

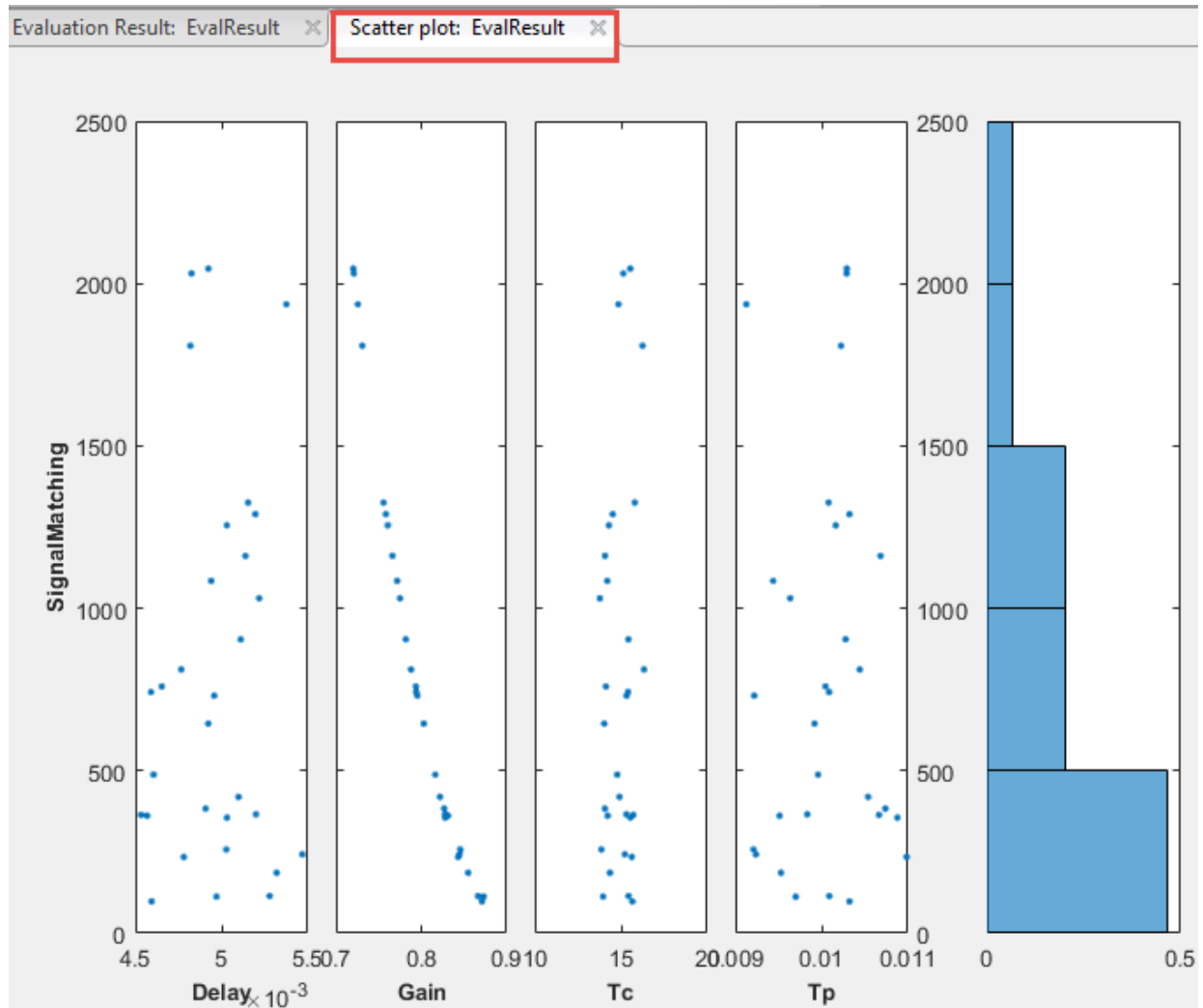
After you have evaluated your design requirements on page 4-83, an evaluation results table lists the samples in the parameter set and the corresponding evaluated requirement (cost function) values. For requirements that involve a bound, a positive

requirement value indicates that your requirement was violated, while a negative value indicates that the requirement was satisfied for that sample of parameter values.



Delay	Gain	Tc	Tp	SignalMatching
0.0049	0.8029	14.0276	0.0099	644.8298
0.0046	0.8710	15.6653	0.0103	97.8802
0.0051	0.8220	14.9205	0.0105	419.7438
0.0050	0.8732	13.9582	0.0097	112.4360
0.0052	0.7585	14.5234	0.0103	1.2900e+03
0.0052	0.8282	15.3222	0.0098	365.7245
0.0051	0.7663	14.0752	0.0107	1.1616e+03
0.0045	0.8275	15.7153	0.0107	364.4182
0.0046	0.8312	14.2285	0.0095	361.4330
	0.7309	16.2555	0.0102	1.8083e+03
	0.7608			1.2555

An evaluation result plot is also generated. The scatter subplots display the evaluated requirement (cost function value) as a function of each parameter in the parameter set. The number of points in each scatter plot equals the number of rows in the parameter set. The last column of subplots displays histograms of the probability distribution of the evaluated cost function values.



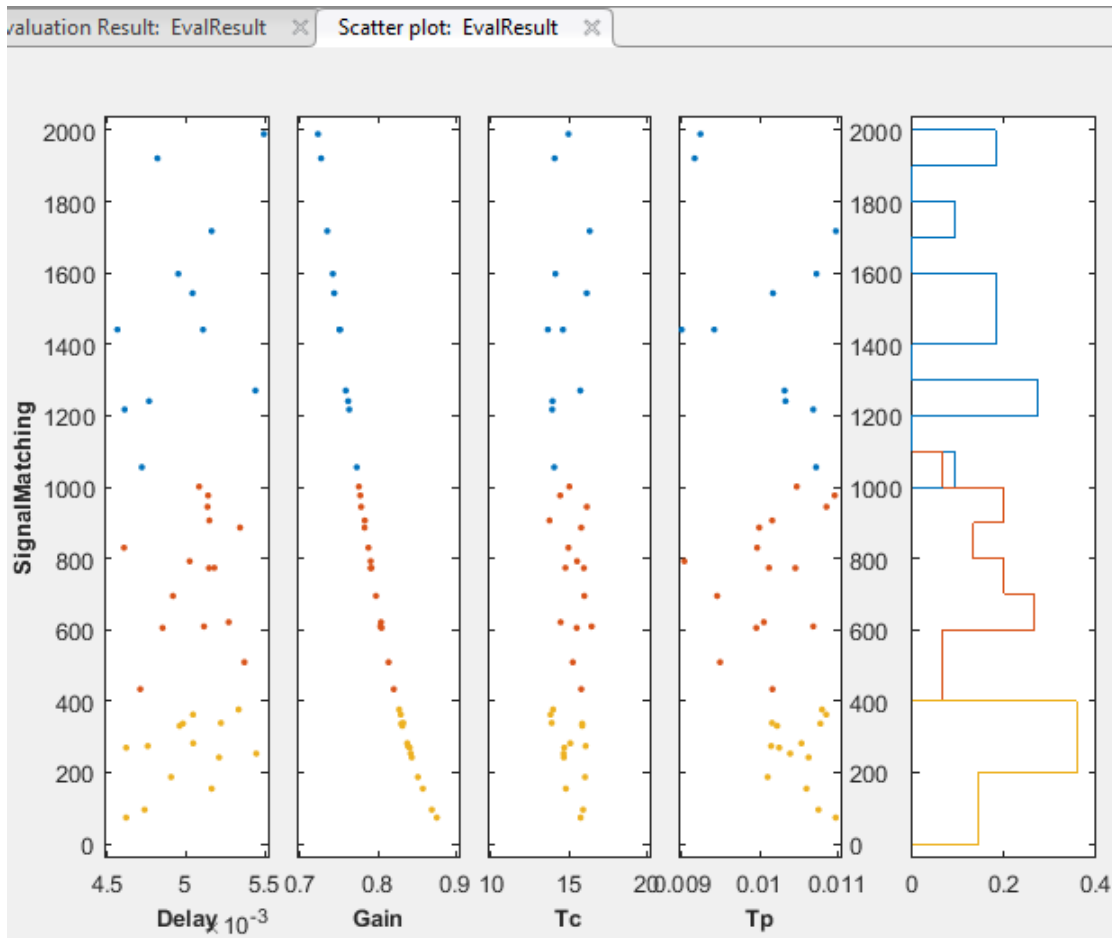
Use this plot to visually analyze the relation between parameters and requirements on page 4-91. For example, in this case, the `SignalMatching` requirement looks monotonically related to the `Gain` parameter.

You can also plot best-fit lines on the scatter subplots. To do so, and to access additional plot features, right-click in the white area of any scatter subplot.

You can choose from the following options:

- **X-Variables** — Select the parameters and requirements to use as x-variables in the scatter subplots.
- **Y-Variables** — Select the parameters and requirements to use as y-variables in the scatter subplots.
- **Grouping** — Select grouping variables for the subplots, and configure how the groups are displayed.

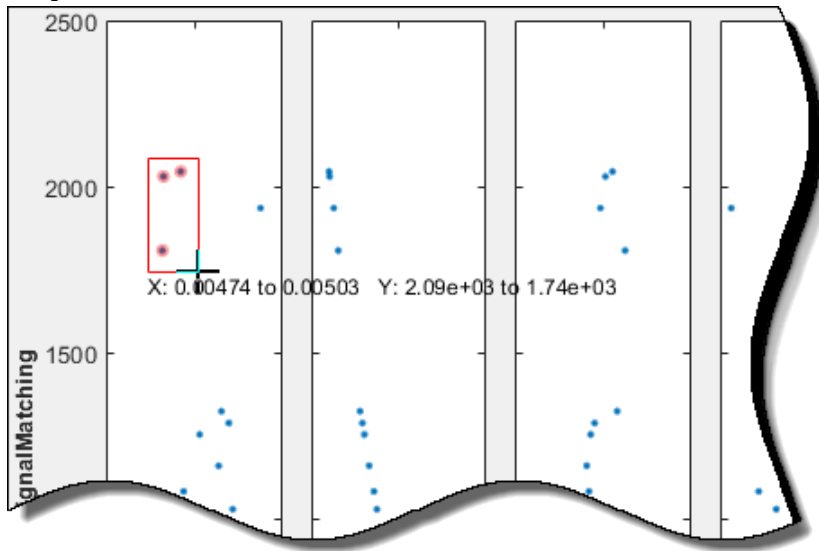
To select a parameter or evaluated requirement as a grouping variable, click **Groups > New Grouping Variable**. For example, the following plot is generated when the grouping variable is `Gain`. The tool creates three groups based on low, medium, and high values of the grouping variable. The tool computes these grouping values, but you can change them in the **Manage Groups** dialog box. The scatter subplots display the evaluated requirement values when the corresponding gain value is low (blue), medium (red), and high (yellow). The histogram plots the probability distribution of the evaluated requirement corresponding to the groups.



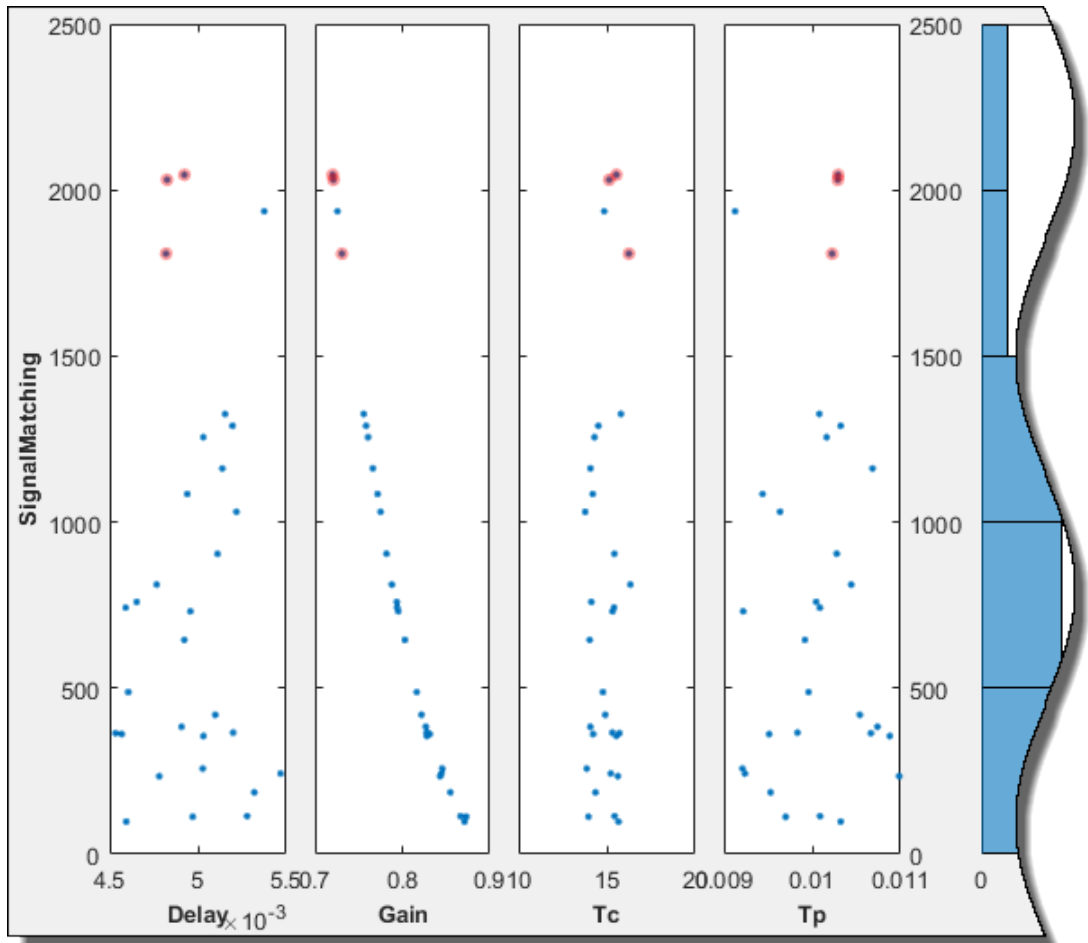
You can demarcate the groups based on marker size and marker type instead of color, add more groups corresponding to the grouping variable, and change the grouping values. You can also add more grouping variables. To do so, click **Groups > Manage Groups**. For more information, see “Parameter Set Plots” on page 4-106.

- **Marginal Box Plots** — Requires Statistics and Machine Learning Toolbox software. Plot box plots for each of the parameters in the parameter set, and choose the position of the plots.
- **Histograms** — Plot the probability distribution of the parameters, and choose the position of the plots.

- **Kernel Density Plots** — Requires Statistics and Machine Learning Toolbox software. Plot the probability distribution of the parameters using a kernel density estimator, and choose the position of the plots. For more information, see “Kernel Distribution” (Statistics and Machine Learning Toolbox).
- **Overlay linear fit** — Plot the best-fit line on the scatter subplots. You can choose to plot the best-fit lines for one, all, a row, or a column of scatter plots.
- **Enable brushing/data selection** — Enable selection of data points in the scatter subplots.



When you highlight parameter values in one scatter subplot, the values corresponding to other parameters from the same row in the evaluated results table are also highlighted. In addition, the rows in the evaluated results table that correspond to these values are highlighted.



Delay	Gain	Tc	Tp	SignalMatching
0.0049	0.8029	14.0276	0.0099	644.8298
0.0046	0.8710	15.6653	0.0103	97.8802
0.0051	0.8220	14.9205	0.0105	419.7438
0.0050	0.8732	13.9582	0.0097	112.4360
0.0052	0.7585	14.5234	0.0103	1.2900e+03
0.0052	0.8282	15.3222	0.0098	365.7245
0.0051	0.7663	14.0752	0.0107	1.1616e+03
0.0045	0.8275	15.7153	0.0107	364.4182
0.0046	0.8312	14.2285	0.0095	361.4330
0.0048	0.7309	16.2523	0.0102	1.8083e+03
0.0050	0.7608	14.3072	0.0102	1.2555e+03
0.0052	0.7558	15.7965	0.0101	1.3255e+03
0.0049	0.8269	14.0660	0.0107	383.3321
0.0053	0.8551	14.3625	0.0095	185.9608
0.0052	0.7751	13.7733	0.0096	1.0309e+03
0.0055	0.8449	15.2286	0.0092	242.5879
0.0050	0.8281	15.5501	0.0109	355.8483
0.0048	0.7211	15.1398	0.0103	2.0309e+03
0.0046	0.8163	14.7772	0.0100	487.9127
0.0051	0.7819	15.4333	0.0103	904.3621
0.0053	0.8666	15.4429	0.0101	114.2653
0.0049	0.7202	15.5371	0.0103	2.0458e+03
0.0046	0.7940	15.4074	0.0101	742.1060
0.0048	0.7879	16.3355	0.0104	811.8904

To remove the highlighting, invert the selection to all other data points in the plot, or disable the feature, right-click the highlighted data points, and choose from the context-menu.

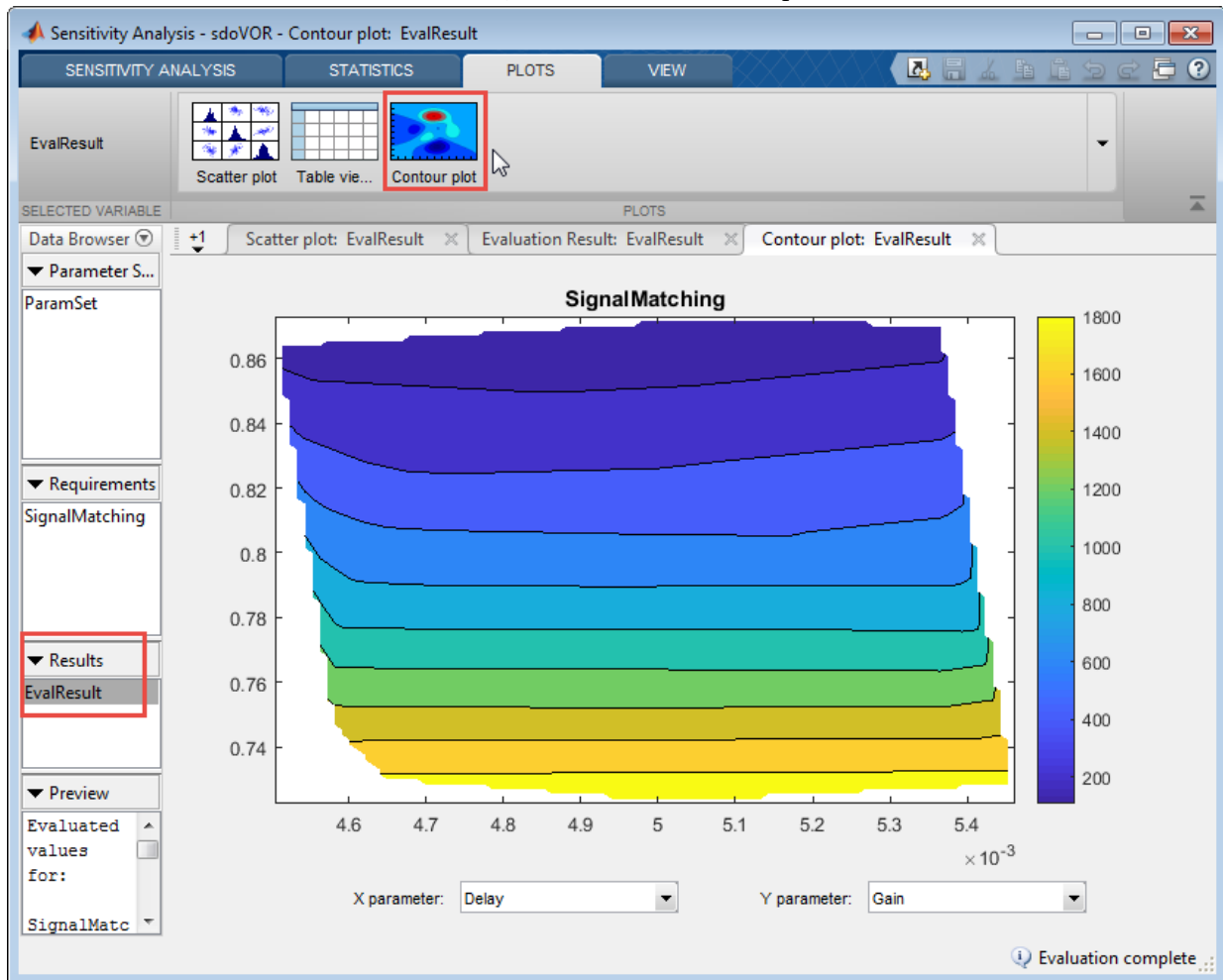
- **Pop-out plot** — View a subplot in a new window.

Evaluated Result Contour Plots

After you have evaluated your design requirements, an evaluation results table lists the samples in the parameter set and the corresponding evaluated requirement (cost

function) values. For information about evaluation, see “Evaluate Design Requirements” on page 4-83.

You can plot a contour plot of the evaluated results. To do so, select the evaluated result in the **Results** area of the tool, and choose a contour plot in the **Plots** tab of the tool.



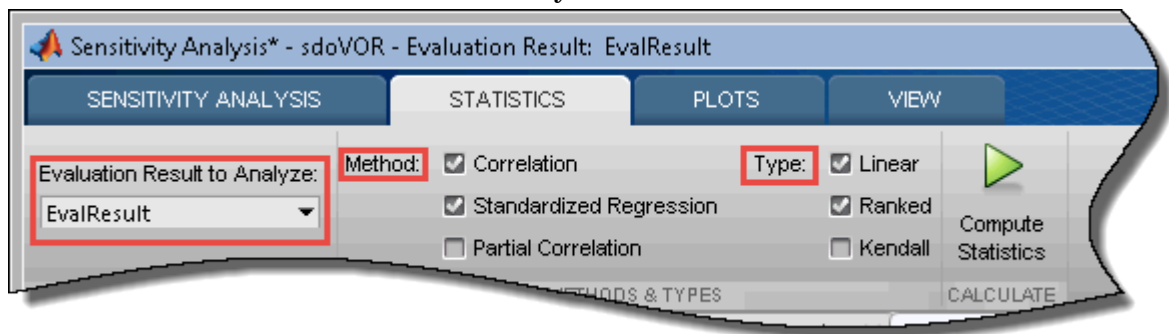
Use this plot to visually analyze the relation between parameters and design requirements. Select the parameters to plot in the **X parameter** and **Y parameter** drop-down lists. The evaluated requirement value is plotted as a function of these parameters.

Statistical Analysis Tornado Plots

After you have evaluated the design requirements for each parameter, you can perform statistical analysis to analyze how the parameters of your Simulink model influence the requirements.

To generate a tornado plot ranking the influence of parameters on requirements:

- 1 In the **Statistics** tab of the tool, select the evaluation results you want to analyze in the **Evaluation Results to Analyze** list.



- 2 Specify the statistical analysis methods.

You can choose to calculate a correlation coefficient, standardized regression coefficient, and partial correlation coefficient (requires Statistics and Machine Learning Toolbox software).

For more information, see “Analyze Relation Between Parameters and Design Requirements” on page 4-91.

- 3 For each of these methods, specify what data to use for the analysis. You can choose from **Linear** (Pearson), **Ranked** (Spearman), and **Kendall** analysis types. **Kendall** is applicable when the analysis method is **Correlation**, and requires Statistics and Machine Learning Toolbox software.

You can compute all applicable combinations of analysis methods and types.

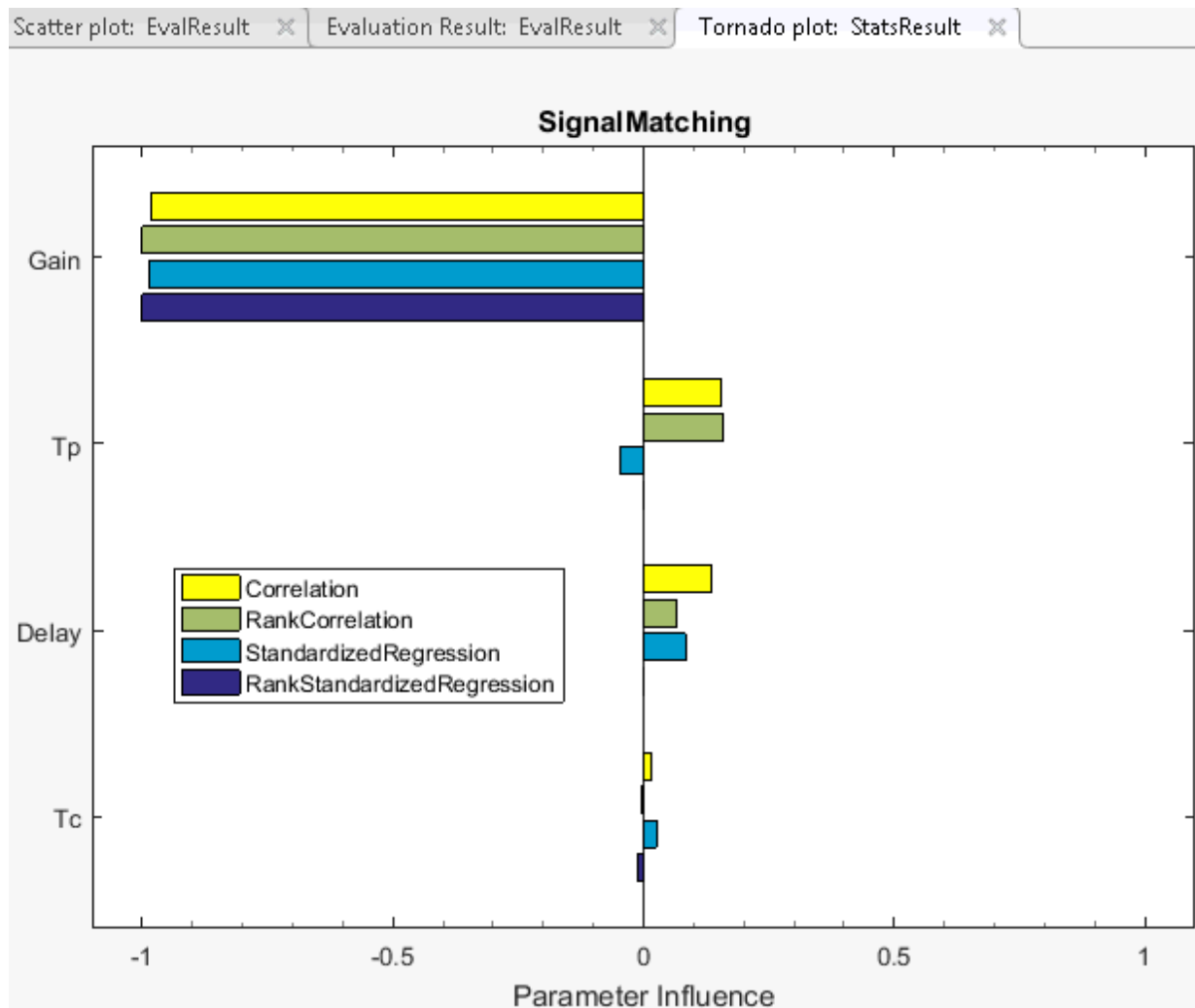
- 4 Calculate the coefficients, and generate a tornado plot.

Click  **Compute Statistics**.

The resulting tornado plot displays the calculated coefficients for each specified analysis method and type. The coefficients are plotted in order of influence of parameters on the cost function. The parameter with the greatest magnitude of influence on the cost function is displayed on the top, giving the plot a tornado shape. When more than one type of coefficient is calculated, the tornado plot sorts the parameters based on the first calculated coefficient. The coefficients are calculated in the following order:

- Correlation
- Rank correlation
- Kendall correlation
- Partial correlation
- Rank partial correlation
- Standardized regression
- Rank Standardized Regression

In this tornado plot, the parameters are sorted based on the Correlation coefficient. For all calculated coefficients, the `Gain` parameter has the most influence on the design requirement cost function.



See Also

Related Examples

- “Generate Parameter Samples for Sensitivity Analysis” on page 4-10

- “Specify Time-Domain Requirements” on page 4-28
- “Specify Frequency-Domain Requirements” on page 4-66
- “Evaluate Design Requirements” on page 4-83
- “Analyze Relation Between Parameters and Design Requirements” on page 4-91

Validate Sensitivity Analysis

You can validate sensitivity analysis by checking generated parameter values, evaluation results, and analysis results.

Inspect the Generated Parameter Set

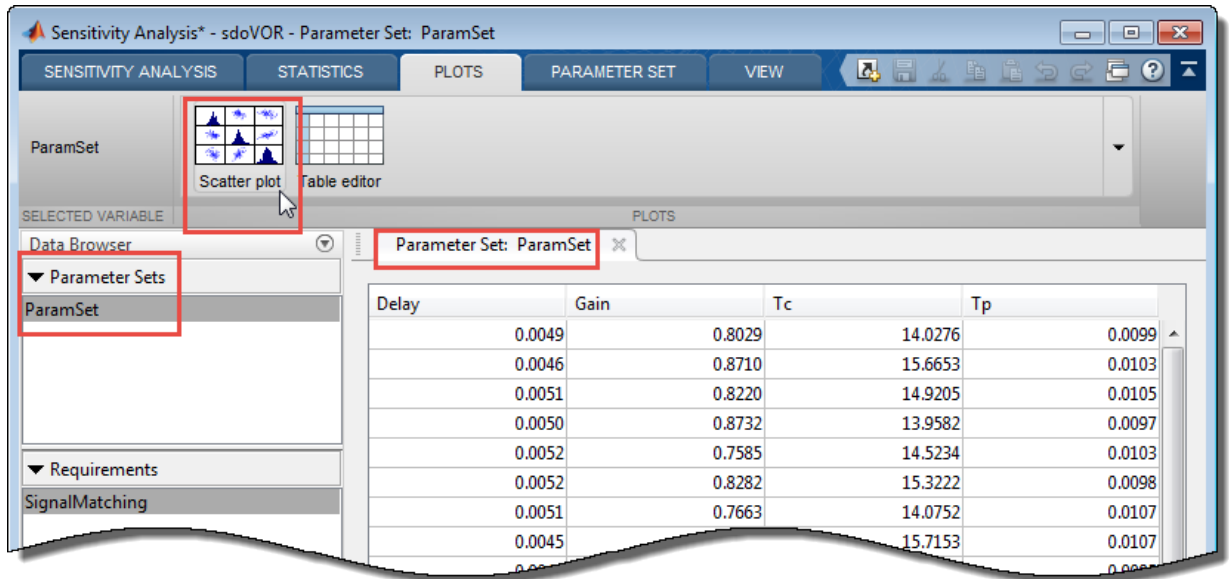
To perform sensitivity analysis, you select model parameters for evaluation, and generate a representative set of parameter values to explore the design space. You create the parameter set by specifying parameter distributions such as normal or uniform. You can also specify correlations between parameters. For more information, see “Generate Parameter Samples for Sensitivity Analysis” on page 4-10. After generating the parameter values, plot them to check if generated parameter values match the desired specifications. This is particularly important if you generate a small number of random samples for each parameter set.

If you see a discrepancy in the generated parameters and the specified distribution and correlations, you can try one of the following:

- Generate the random samples again, until you achieve the specified distributions and correlations.
- Increase the sample size at the expense of increasing the evaluation time.
- Specify different sampling methods. Use Latin hypercube sampling method for a more systematic space-filling approach than random sampling. If you have Statistics and Machine Learning Toolbox software, use the Sobol and Halton quasirandom sampling methods for a more space-filling approach than the Latin hypercube method.

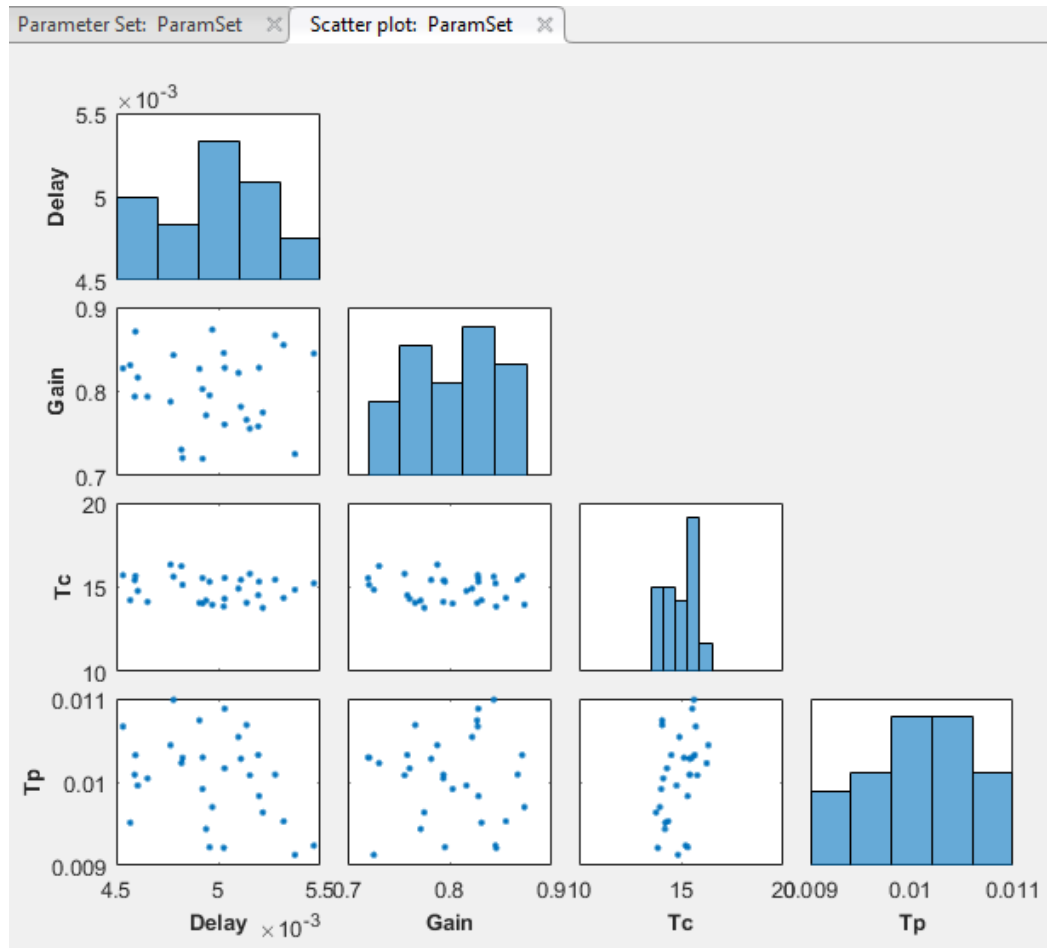
To plot the generated parameters in the Sensitivity Analysis tool:

- 1 Select the generated parameter set in the **Parameter Sets** area of the tool.



2 In the **Plots** tab, select **Scatter Plot**.

The generated plot displays histograms of generated values for each parameter on the diagonal, and the pairwise scatter plot of the parameters on the off-diagonals. For more information about the scatter plot, see “Interact with Plots in the Sensitivity Analysis Tool” on page 4-106.



- 3 Inspect the histograms to ensure that the generated parameter values match the intended parameter distributions. Inspect the off-diagonal scatter plots to ensure that any specified correlations between parameters are present.

To plot the generated parameter values at the command line, use `sdo.scatterPlot`. Use functions such as `mean` to check the sample statistics.

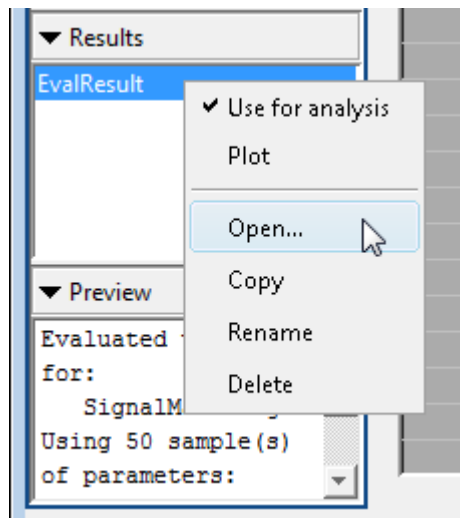
Check Evaluation Results

After generating a parameter set, you define a cost function by creating a design requirement on the model signals. You then evaluate the cost function at each set of parameter values on page 4-83. To validate the evaluation results, inspect the evaluated cost function values. If the cost function evaluations contain NaN values, that could indicate an issue.

To check for NaN values in the Sensitivity Analysis tool after the evaluation is complete:

- 1 Open the evaluation results table if it is not already open.

In the **Results** area of the tool, right-click the evaluated result, and select **Open** in the menu.



In the **Evaluation Results** table, each row of the table lists the parameter set values and the corresponding evaluated design requirement cost function values.

- 2 Sort the evaluated requirement values in descending order. To do so, click twice on the evaluated requirement column. Any NaN values are listed at the top of the evaluated requirement column.

Data Browser

Scatter plot: EvalResult Evaluation Result: EvalResult

Select and right-click a row to create an initial guess for Response Optimization or

Delay	Gain	Tc	Tp	SignalMatching
0.0053	0.7739	14.2199	0.0106	NaN
0.0052	0.7219	14.0517	0.0106	2.0227e+03
0.0049	0.7286	15.3662	0.0106	1.8622e+03
0.0049	0.7321	16.0591	0.0109	1.7803e+03
0.0051	0.7390	13.9969	0.0100	1.6656e+03
0.0050	0.7408	13.7279	0.0096	1.6389e+03
0.0053	0.7439	13.7345	0.0102	1.5719e+03
0.0054	0.7422	16.4884	0.0109	1.5718e+03
0.0054	0.7460	15.0856	0.0109	1.5100e+03
0.0052	0.7515	13.7533	0.0103	1.4276e+03
0.0055	0.7558	15.7445	0.0092	1.3312e+03
0.0054	0.7590	15.8247	0.0090	1.2768e+03
0.0055	0.7608	13.7515	0.0098	1.2625e+03
0.0049	0.7605	13.7515	0.0105	1.2598e+03
0.0055	0.7605	13.7515	0.0105	1.2474e+03

Parameter Sets

ParamSet

Requirements

SignalMatching

Results

EvalResult

- 3 Inspect the parameter values that resulted in the NaN values for evaluated requirements. If you do not expect an NaN result for that row of parameter values, investigate your model further.

To view the evaluated results at the command line, inspect the cost function evaluation output of `sdo.evaluate`.

Perform Sensitivity Analysis with Different Parameter Set

After evaluation, you analyze the effect of the parameters on the design requirements, and identify most influential parameters. For more information, see “Analyze Relation Between Parameters and Design Requirements” on page 4-91. To validate the analysis results, generate a different parameter set and reevaluate the design requirements. If the analysis results are not consistent, consider increasing the number of samples in your parameter set.

See Also

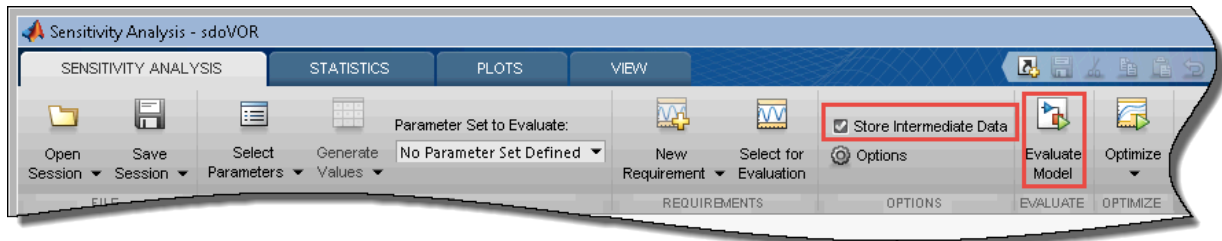
Related Examples

- “What is Sensitivity Analysis?” on page 4-2
- “Generate Parameter Samples for Sensitivity Analysis” on page 4-10
- “Analyze Relation Between Parameters and Design Requirements” on page 4-91

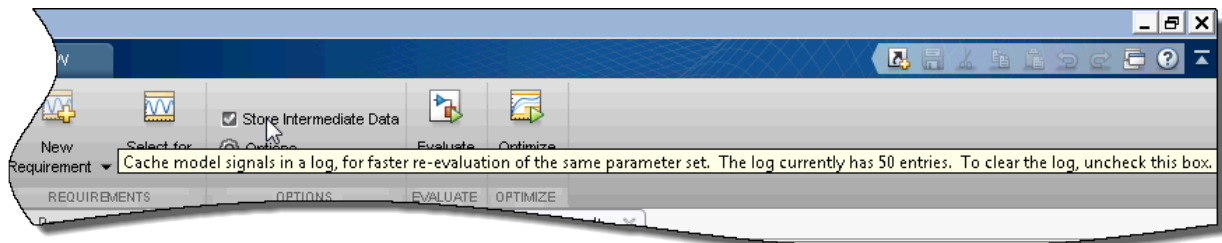
Store Intermediate Data in the Tool

This topic shows how to speed up successive evaluations of design requirements in the Sensitivity Analysis tool by storing intermediate data during evaluation. Use this option when memory usage is not a concern. Choosing this option results in a memory-time trade-off.

To store intermediate data during evaluation, in the **Sensitivity Analysis** tab, click **Store Intermediate Data**.



After creating a parameter set and specifying design requirements, you can evaluate your model. When you click **Evaluate Model**, the tool saves the logged signals that are used to compute the design requirements at each combination of parameter values in your parameter set. The logged signal values are stored in the RAM. After the evaluation is complete, you can view the number of logged entries by hovering over **Store Intermediate Data**.



If you now evaluate your model for a different design requirement that uses the same logged signals and parameter set values, the tool uses the stored signal values. As a result, successive evaluations take less time.

If your system is running out of memory during evaluation, stop the evaluation, and clear **Store Intermediate Data**. Doing so will delete the stored data from memory.

See Also

Related Examples

- “Specify Parameters for Design Exploration” on page 4-4
- “Generate Parameter Samples for Sensitivity Analysis” on page 4-10
- “Specify Time-Domain Requirements” on page 4-28
- “Specify Frequency-Domain Requirements” on page 4-66
- “Evaluate Design Requirements” on page 4-83
- “Ways to Speed Up Design Optimization Tasks”

Use Parallel Computing for Sensitivity Analysis

In this section...

“Configure Your System for Parallel Computing” on page 4-137

“Model Dependencies” on page 4-137

“Perform Sensitivity Analysis Using Parallel Computing (GUI)” on page 4-138

“Perform Sensitivity Analysis Using Parallel Computing (Code)” on page 4-142

“Troubleshooting” on page 4-143

Configure Your System for Parallel Computing

To perform global sensitivity analysis, you sample the model parameters and states, define a cost function by creating a design requirement on the model signals, and evaluate the cost function for each sample. Evaluating the model for many samples can be time consuming. You can speed up evaluation in the Sensitivity Analysis tool, or at the command line, using parallel computing on multicore processors or multiprocessor networks.

When you evaluate the cost function with the parallel computing option enabled, the software uses the available parallel pool. If none is available, and **Automatically create a parallel pool** is selected in your Parallel Computing Toolbox preferences, the software starts a parallel pool using the settings in those preferences. To open a parallel pool that uses a specific cluster profile, use:

```
parpool(MyProfile);
```

`MyProfile` is the name of a cluster profile.

For information regarding creating a cluster profile, see “Create and Modify Cluster Profiles” (Parallel Computing Toolbox).

Model Dependencies

Model dependencies are any referenced models, data such as model variables, S-functions, or additional files necessary to run the model. Before starting the parallel model evaluation, verify that the model dependencies are complete. Otherwise, you may get unexpected results.

Making Model Dependencies Accessible to Remote Workers

When you use parallel computing, the Simulink Design Optimization software helps you identify model dependencies. To do so, the software uses the Simulink Manifest Tools. The dependency analysis may not find all the files required by your model. To learn more, see “Scope of Dependency Analysis” (Simulink). If your model has dependencies that are undetected or inaccessible by the parallel pool workers, then add them to the list of model dependencies.

The dependencies 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.
- Path dependencies: the paths to the model dependencies are added to the paths of the parallel pool workers. If you are working in a multi-platform scenario, ensure that the paths are compatible across platforms.


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 dependent files on the local computer.

Perform Sensitivity Analysis Using Parallel Computing (GUI)

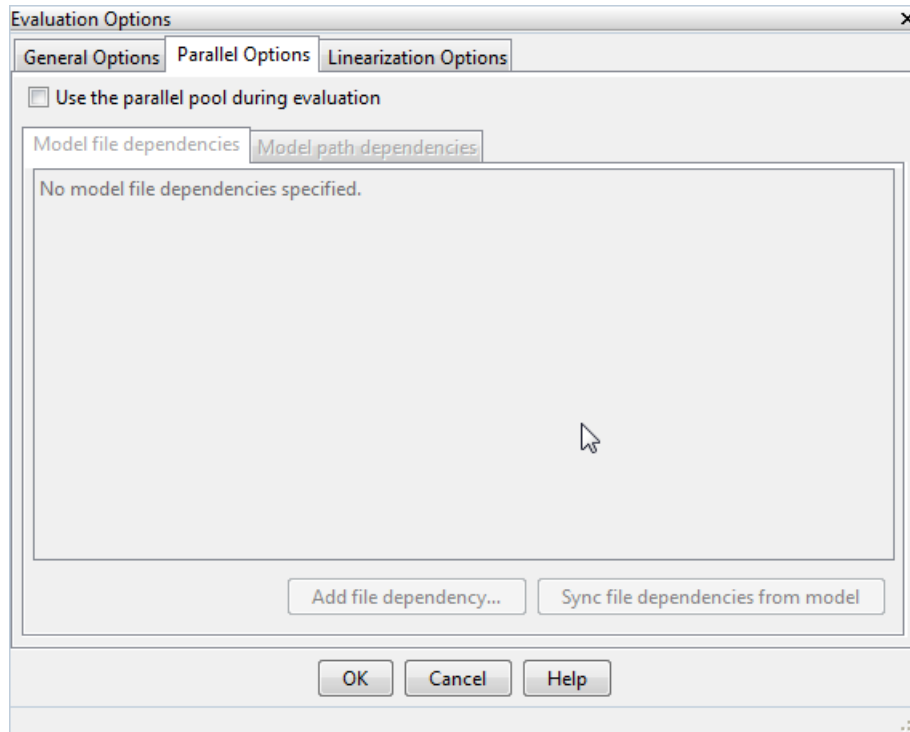
To perform sensitivity analysis using parallel computing in the Sensitivity Analysis tool:

- 1 Ensure that the software can access parallel pool workers that use the appropriate cluster profile.

For more information, see “Configure Your System for Parallel Computing” on page 4-137.

- 2 Open the Sensitivity Analysis tool for the Simulink model.
- 3 Specify the parameter set, generate parameter samples, and specify the requirements for sensitivity analysis. For example, see “Design Exploration Using Parameter Sampling (GUI)” on page 4-147 and “Identify Key Parameters for Estimation (GUI)” on page 4-168.
- 4 On the **Sensitivity Analysis** tab, click  **Options** to open the **Evaluation Options** dialog box.

5 Select the **Parallel Options** tab.

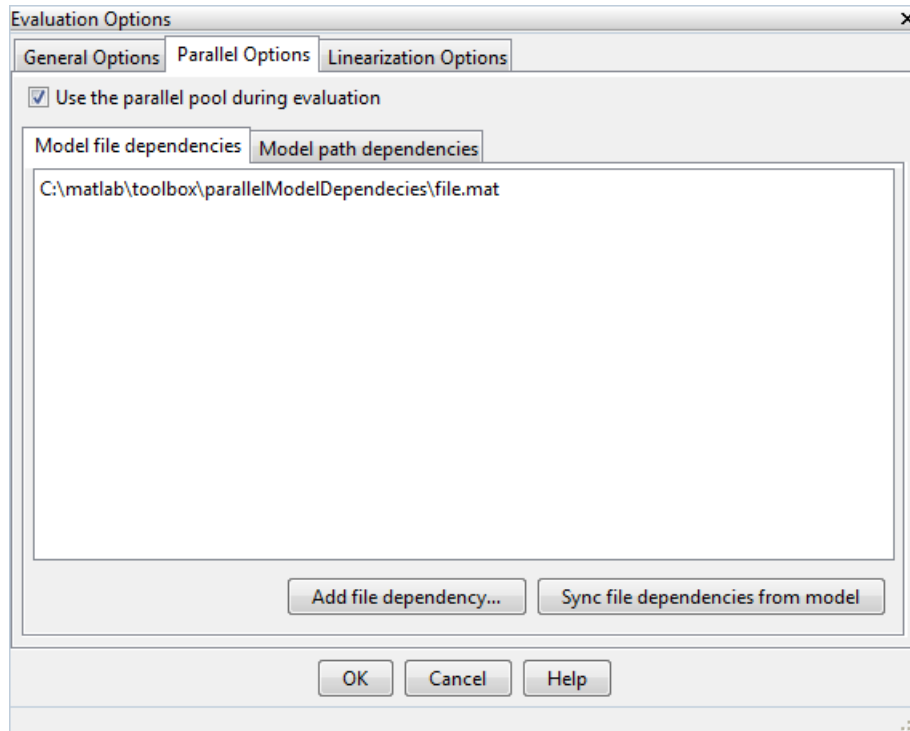


6 Select the **Use the parallel pool during evaluation** check box.

This option checks for dependencies in your Simulink model. The file dependencies are displayed in the **Model file dependencies** list box, and corresponding path to the files in **Model path dependencies**. The files listed in **Model file dependencies** are copied to the remote workers.

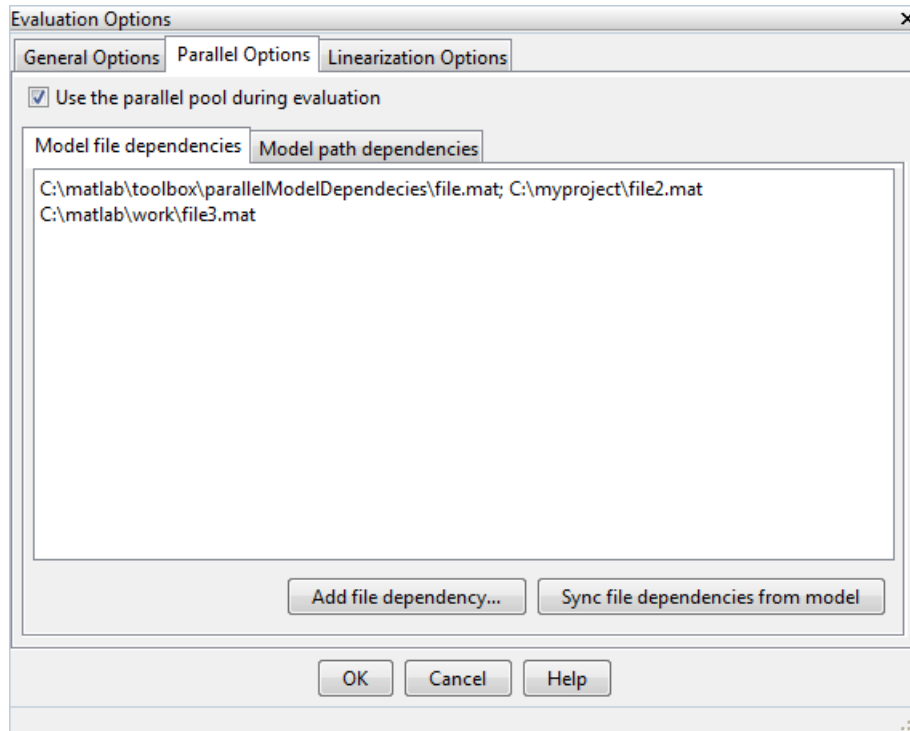
Note The automatic dependencies check may not detect all the dependencies in your model.

For more information, see “Model Dependencies” on page 4-137. In this case, add the undetected dependencies manually.



- 7 Add any file dependencies that the automatic check does not detect.

Specify the files in the **Model file dependencies** list box separated by semicolons or on separate lines.



Alternatively, click **Add file dependency** to open a dialog box, and select the file to add.

Note If you do not want to copy the files to the remote workers, delete all entries in the **Model file dependencies** list box. Populate the **Model path dependencies** list box by clicking the **Sync path dependencies from model**, and add any undetected path dependencies. In addition, in the list box, update the paths on local drives to make them accessible to remote workers. For example, change `C:\` to `\\ \hostname\C$`.

- 8 If you modify the Simulink model, resync the dependencies to ensure that any new dependencies are detected. Click **Sync file dependencies from model** in the **Parallel Options** tab to rerun the automatic dependency check for your model.

This action updates the **Model file dependencies** list box with any new file dependency found in the model.

- 9 Click **OK**.
- 10 In the Sensitivity Analysis tool, click **Evaluate** to perform sensitivity analysis using parallel computing. The design requirements are evaluated for each combination of parameter values in your parameter set.

Perform Sensitivity Analysis Using Parallel Computing (Code)

To evaluate a model using parallel computing:

- 1 Ensure that the software can access parallel pool workers that use the appropriate cluster profile.

For more information, see “Configure Your System for Parallel Computing” on page 4-137.

- 2 Open the model.
- 3 Specify the cost function and generate parameter samples for sensitivity analysis. For example, see “Design Exploration Using Parameter Sampling (Code)” on page 4-200.
- 4 Enable parallel computing using an evaluation option set.

```
opt = sdo.EvaluateOptions;  
opt.UseParallel = true;
```

- 5 Find the model dependencies.

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

Note `sdo.getModelDependencies` may not detect all the dependencies in your model. For more information, see “Model Dependencies” on page 4-137. In this case, add the undetected dependencies manually.

- 6 Modify files to include any file dependencies that `sdo.getModelDependencies` does not detect.

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

Note If you do not want to copy the files to the remote workers, add any undetected path dependencies to `dirs` and update the paths on local drives to make them accessible to remote workers. See `sdo.getModelDependencies` for more details.

- 7 Add the file dependencies for evaluation.

```
opt.ParallelFileDependencies = files;
```

- 8 Specify the name of the model to be evaluated in parallel.

```
opt.EvaluatedModel = modelname;
```

- 9 Evaluate the model.

```
[pOpt,opt_info] = sdo.evaluate(fcn,samples,opt);
```

Troubleshooting

Why Don't I See the Evaluation Speed up I Expected Using Parallel Computing?

When you evaluate a model that does not require a large number of evaluations or does not take long to simulate, you might not see a speedup in the evaluation time. In such cases, the overhead associated with creating and distributing the parallel tasks outweighs the benefits of running the evaluation in parallel.

See Also

`parpool` | `sdo.EvaluateOptions` | `sdo.evaluate` | `sdo.getModelDependencies`

More About

- “Ways to Speed Up Design Optimization Tasks”
- “Store Intermediate Data in the Tool” on page 4-135

Use Fast Restart Mode During Sensitivity Analysis

This topic shows how to speed up sensitivity analysis using Simulink fast restart. You can use the fast restart feature to speed up sensitivity analysis of tunable parameters (Simulink) of a model.

Fast restart enables you to perform iterative simulations without compiling a model or terminating the simulation each time. Using fast restart, you compile a model only once. You can then tune parameters and simulate the model again without spending time on compiling. Fast restart associates multiple simulation phases with a single compile phase to make iterative simulations more efficient. You see a speedup of design optimization tasks using fast restart in models that have a long compilation phase. See “How Fast Restart Improves Iterative Simulations” (Simulink).

When you enable fast restart, you can only change tunable properties of the model during simulation. For more information about the limitations, see “Factors Affecting Fast Restart” (Simulink).

You can perform sensitivity analysis using fast restart in the Sensitivity Analysis tool on page 4-144 or at the command line on page 4-145.

Sensitivity Analysis Tool Workflow for Fast Restart

To evaluate a model using fast restart in the Sensitivity Analysis tool:

- 1 Open the Simulink model.
- 2 Enable fast restart in the model.

Click **Fast Restart**  in the model window.

- 3 Open the Sensitivity Analysis tool for the model.
- 4 Specify the parameter set, generate parameter samples, and specify the requirements for sensitivity analysis. Optionally, specify evaluation settings. For example, see “Design Exploration Using Parameter Sampling (GUI)” on page 4-147 and “Identify Key Parameters for Estimation (Code)” on page 4-217.
- 5 Click **Evaluate** to evaluate the model in fast restart mode. The design requirements are evaluated for each combination of parameter values in your parameter set.
- 6 Disable fast restart.

In the model window, click **Fast Restart**



Command-Line Workflow for Fast Restart

You can use sensitivity analysis to evaluate which model parameters most influence a cost function. You can use these parameters during parameter estimation or response optimization. Suppose that you want to use sensitivity analysis to reduce the number of parameters that you need to estimate to fit a model.

To evaluate the model in fast restart mode using a cost function aimed at parameter estimation:

- 1 Open the Simulink model.
- 2 Specify the model parameter values, `params`, to estimate and generate parameter samples, `params_samples`. For an example, see “Identify Key Parameters for Estimation (Code)” on page 4-217.
- 3 Create an experiment object, `Exp`.

```
Exp = sdo.Experiment('model');
```

Store the measured input-output data in `Exp`. For an example, see “Identify Key Parameters for Estimation (Code)” on page 4-217.

- 4 Create a model simulator from the experiment.

```
Simulator = createSimulator(Exp);
```

`Simulator` is an `sdo.SimulationTest` object.

Note You must create a simulation scenario with logging information before configuring the model for fast restart. You cannot modify logging information once the model has been compiled for fast restart.

- 5 Configure the model and simulator for fast restart.

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

- 6 Create a cost function, `myCostfcn`, and pass `Simulator` to the cost function as an input. For more information, see “Write a Cost Function” on page 2-62. In the cost function, the simulator configured for fast restart is used to update the model parameters, simulate the model, and log signals.

Use an anonymous function with one argument that calls `myCostfcn`.

```
evalfcn = @(param) myCostfcn(param, Simulator, Exp);
```

7 Evaluate the model.

```
[param_opt, opt_info] = sdo.evaluate(evalfcn, param, param_samples);
```

8 Restore the simulator fast restart settings.

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

The fast restart workflow is similar for sensitivity analysis that identifies design variables using a cost function aimed at response optimization. See “Use Fast Restart Mode During Response Optimization” on page 3-226.

Troubleshooting

Why Don't I See the Evaluation Speedup I Expected Using Fast Restart?

You see a speedup of design optimization tasks using fast restart in models that have a long compilation phase. If the compilation phase of your model is not long, you do not see a significant change in estimation speed.

See Also

`fastRestart` | `sdo.SimulationTest` | `sdo.evaluate`

More About

- “Ways to Speed Up Design Optimization Tasks”
- “Store Intermediate Data in the Tool” on page 4-135

Design Exploration Using Parameter Sampling (GUI)

This example shows how to sample and explore a design space using the Sensitivity Analysis tool. You explore the design of a Continuously Stirred Tank Reactor (CSTR) to minimize product concentration variation and production cost. The design must also account for the uncertainty in the temperature and concentration of the input feed to the reactor.

You explore the CSTR design by characterizing model parameters using probability distributions. You use the distributions to generate random samples and perform Monte-Carlo evaluation of the design at these sample points. You then create plots to visualize the design space and select the best design. You then use the best design as an initial guess for optimization of the design.

Continuously Stirred Tank Reactor (CSTR) Model

Continuously Stirred Tank Reactors (CSTRs) are common in the process industry. The Simulink model, `sd0CSTR`, models a jacketed diabatic (i.e., non-adiabatic) tank reactor described in [1]. The CSTR is assumed to be perfectly mixed, with a single first-order exothermic and irreversible reaction, $A \rightarrow B$. A , the reactant, is converted to B , the product.

In this example, you use the following two-state CSTR model, which uses basic accounting and energy conservation principles:

$$\frac{dC_A}{dt} = \frac{F}{A * h} (C_{feed} - C_A) - r * C_A$$

$$\frac{dT}{dt} = \frac{F}{A * h} (T_{feed} - T) - \frac{H}{c_p \rho} r - \frac{U}{c_p * \rho * h} (T - T_{cool})$$

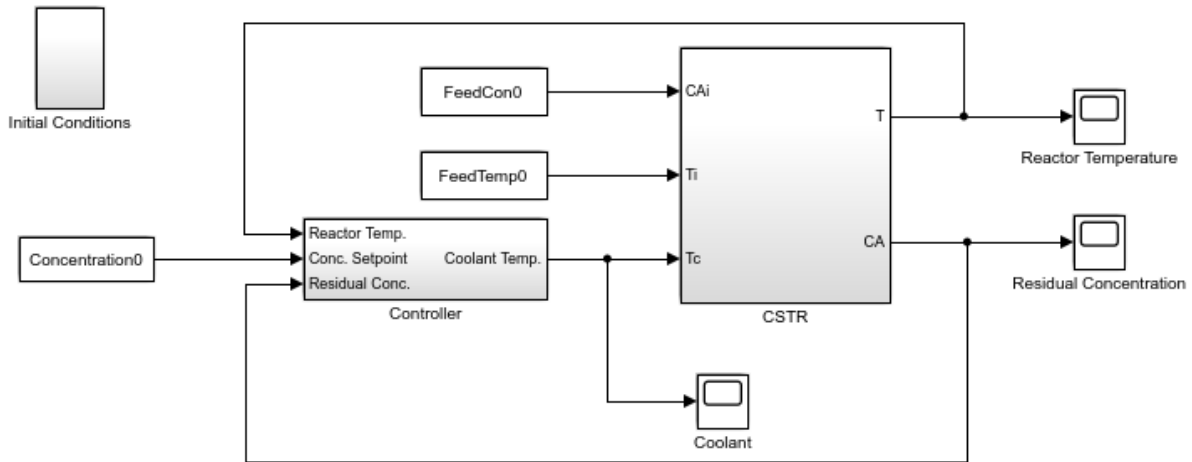
$$r = k_0 * e^{-\frac{E}{RT}}$$

- C_A , and C_{feed} - Concentrations of A in the CSTR and in the feed [kgmol/m³]
- T , T_{feed} , and T_{cool} - CSTR, feed, and coolant temperatures [K]
- F and ρ - Volumetric flow rate [m³/h] and the density of the material in the CSTR [1/m³]
- h and A - Height [m] and heated cross-sectional area [m²] of the CSTR.

- k_0 - Pre-exponential non-thermal factor for reaction $A \rightarrow B$ [1/h]
- E and H - Activation energy and heat of reaction for $A \rightarrow B$ [kcal/kgmol]
- R - Boltzmann's gas constant [kcal/(kgmol * K)]
- c_p and U - Heat capacity [kcal/K] and heat transfer coefficients [kcal/(m² * K * h)]

Open the Simulink model.

```
open_system('sdoCSTR');
```



Copyright 2012-2015 The MathWorks, Inc.

CSTR Design Problem

Assume that the CSTR is cylindrical, with the coolant applied to the base of the cylinder. Tune the CSTR cross-sectional area, A , and CSTR height, h , to meet the following design goals:

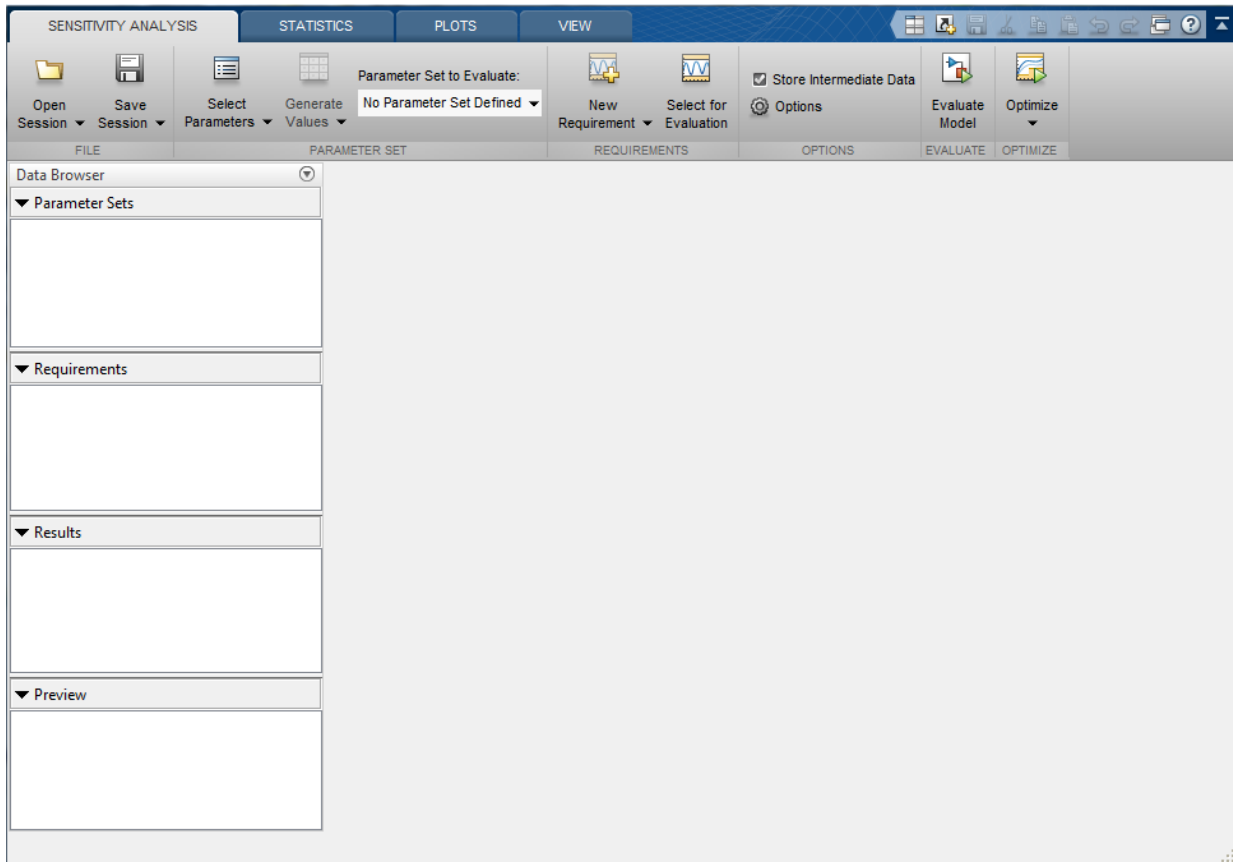
- Minimize the variation in residual concentration, C_A . Variations in the residual concentration negatively affect the quality of the CSTR product. Minimizing the variations also improves CSTR profit.

- Minimize the mean coolant temperature T_{cool} . Heating or cooling the jacket coolant is expensive. Minimizing the mean coolant temperature improves CSTR profit.

The quality of the feed to the reactor can differ amongst suppliers. Thus, the design must allow for variations in the supply feed concentration, C_{feed} , and feed temperature, T_{feed} . The quality of the feed differs from supplier to supplier and also varies within each supply batch.

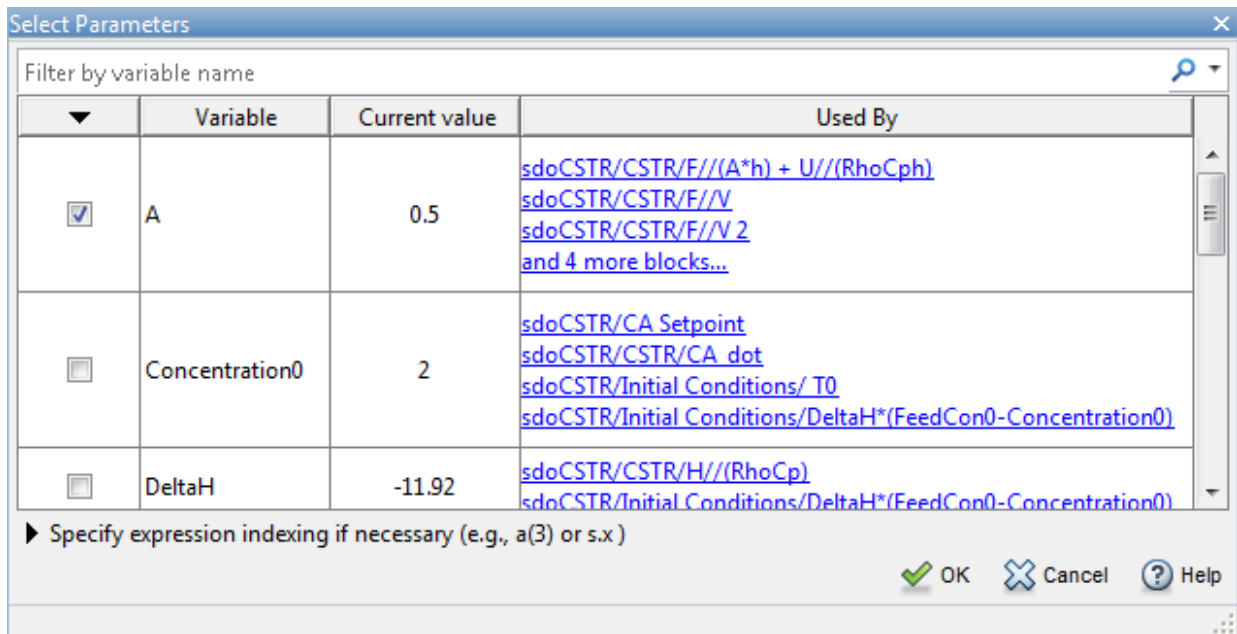
Specify Design Variables

Open the Sensitivity Analysis Tool. In the Simulink model editor, click **Analysis->Sensitivity Analysis**. The tool opens with an empty Sensitivity Analysis session.



Create a parameter set that includes the CSTR design variables A and h and the feed variation parameters $FeedConc0$ and $FeedTemp0$. You randomly generate multiple values for these parameters to evaluate the CSTR design.

- In the **Sensitivity Analysis** tab, in the **Select Parameters** drop-down menu, select **New**.
- In the dialog box, select A , $FeedConc0$, $FeedTemp0$, and h .
- Click **OK**. An empty parameter set, $ParamSet$ is created in the **Parameter Set** area of the tool browser.



Specify the parameter distributions and correlations. $ParamSet$ will be populate with parameter values selected randomly from the specified distributions:

- Sample A from a uniform distribution with lower bound 0.2 m^2 and upper bound 2 m^2 .
- Sample h from a uniform distribution with lower bound 0.5 m and upper bound 3 m .
- Sample $FeedConc0$ from a normal distribution with mean 10 kgmol/m^3 and standard deviation 0.5 kgmol/m^3 .

- Sample `FeedTemp0` from a normal distribution with mean 295 K and standard deviation 3 K.
- Specify `FeedCon0` and `FeedTemp0` as negatively correlated with covariance 0.6.

To generate 100 parameter values using the above distribution and correlation information, click **Generate Values**, and select **Generate Random Values**. For repeatability of the example reset the random number generator.

```
rng('default')
```

In the **Generate Random Parameter Values** dialog box, specify the following:

- Set the number of samples to 100
- For parameter `A` select **Uniform** distribution, set the lower bound to 0.2 and upper bound to 2.
- For parameter `FeedCon0` select **Normal** distribution, set `mu` to 10 and `sigma` to 0.5, and check `cross-correlated`.
- For parameter `FeedTemp0` select **Normal** distribution, set `mu` to 295 and `sigma` to 3, and check `cross-correlated`.
- For parameter `h` select **Uniform** distribution, set the lower bound to 0.5 and upper bound to 3.
- In the **Correlation Matrix** tab, set the `FeedCon0`, `FeedTemp0` covariance to -0.6.
- Click **OK** to generate the parameter values.

4 Sensitivity Analysis

Number of Samples:

Overwrite previous values in parameter set when generating new values
 Append to previous values in parameter set when generating new values

Sampling Method:

Parameter	Distribution							Cross-Correlated
FeedCon0	Normal	mu:	10	sigma:	0.5			<input checked="" type="checkbox"/>
FeedTemp0	Normal	mu:	295	sigma:	3			<input checked="" type="checkbox"/>
h	Uniform	Lower:	0.5	Upper:	3			<input type="checkbox"/>

Distribution Plot Correlation Matrix

	FeedCon0	FeedTemp0
FeedCon0	1	-0.6000
FeedTemp0	-0.6000	1

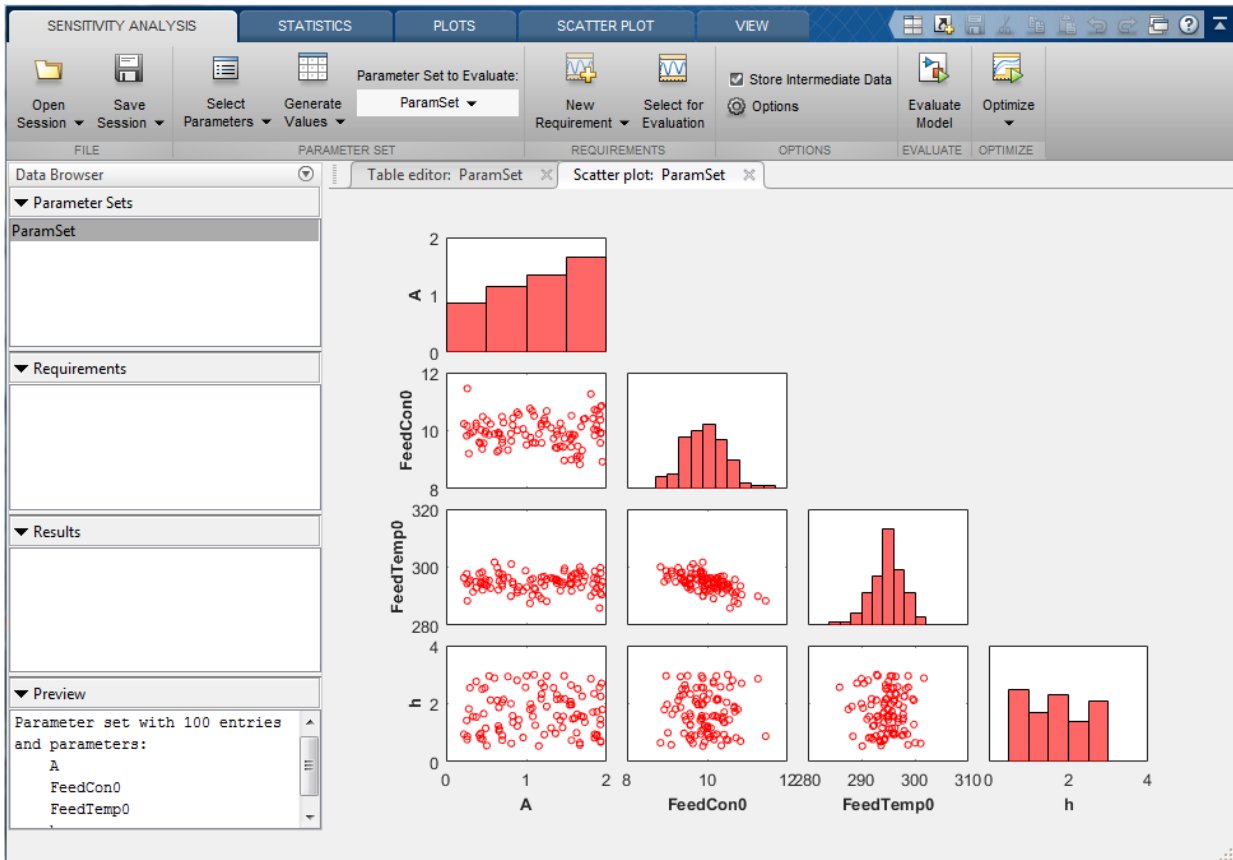
OK Apply Help

The ParamSet table is updated with the generated parameter values. Note that you can manually edit the generated parameter values in the ParamSet table.

To plot the parameter set click ParamSet in the **Parameter Sets** area of the tool browser. In the **Plots** tab, select **Scatter Plot** in the plots gallery. The plot shows the histogram of the generated parameters on the diagonal and pair-wise parameter values on the off-diagonal.

Note that due to the random number generator the specific values in the plots and tables below may differ from what you get when running the example.

Each marker on the plot represents one row of the ParamSet table, with each row being simultaneously displayed on all the plots. The correlation between FeedCon0 and FeedTemp0 can be seen on the plot.



Specify Requirements for Evaluation

The CSTR design is required to minimize the variation in residual concentration and minimize the mean coolant temperature. Select **New Requirement** and click **Signal Property** to create a requirement to minimize the residual concentration variation.

In the **Create Requirement** dialog box, specify the following fields:

- In the **Property** drop-down list, select **Signal Variance**.
- In the **type** drop-down list, select **Minimize**.
- In the **Select Signals** area, select a logged signal to apply the requirement to. To do so click **+**. A **Create Signal Set** dialog box opens where you specify the logged signal. In the Simulink model, click the signal at the CA output of the CSTR block. The dialog box now displays this signal. Add the signal to the signal set and click **OK**.



Create Requirement [x]

Signal Property
Specify a requirement on a signal property such as the mean value.

Name:

▼ **Specify Signal Property**

Property: Time-Weighted

Type:

Bound:

▼ **Select Signals**

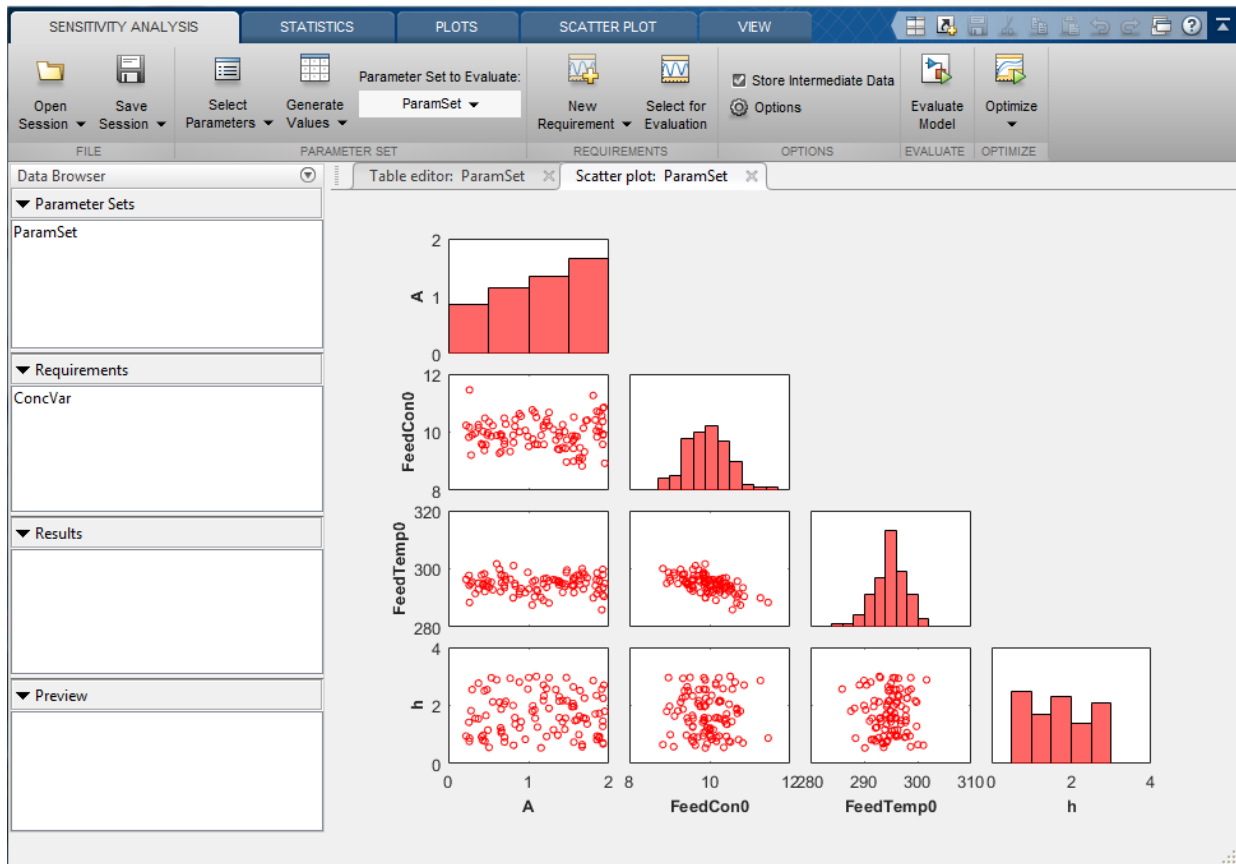
	Signal
<input checked="" type="checkbox"/>	Sig (sdoCSTR/CSTR:2)

[+]
[-]

- Close the Signal Property requirement dialog by clicking the x in the upper right dialog corner.

Close the **Create Requirement** dialog box. A new requirement, SignalProperty is listed in the **Requirements** area of the tool browser

- Right-click SignalProperty, select **Rename**; rename the requirement to ConcVar.



Select **New Requirement** and click **Signal Property** to create a requirement to minimize the coolant mean (output of block sdoCSTR/Controller) temperature.

In the **Create Requirement** dialog box, specify the following fields:

- In the **Property** drop-down list, select **Signal Mean**.
- In the **Type** drop-down list, select **Minimize**.
- In the **Select Signals** area, add the sdoCSTR/Controller signal to the requirement.

Create Requirement

Signal Property

Specify a requirement on a signal property such as the mean value.

Name:

▼ **Specify Signal Property**

Property: Time-Weighted

Type:

Bound:

▼ **Select Signals**

	Signal
<input type="checkbox"/>	Sig (sdoCSTR/CSTR:2)
<input checked="" type="checkbox"/>	Sig1 (sdoCSTR/Controller:1)

Close the **Create Requirement** dialog box. A new requirement, `SignalProperty` is created in the **Requirements** area of the tool browser. Rename the requirement `CoolMean`.

Evaluate

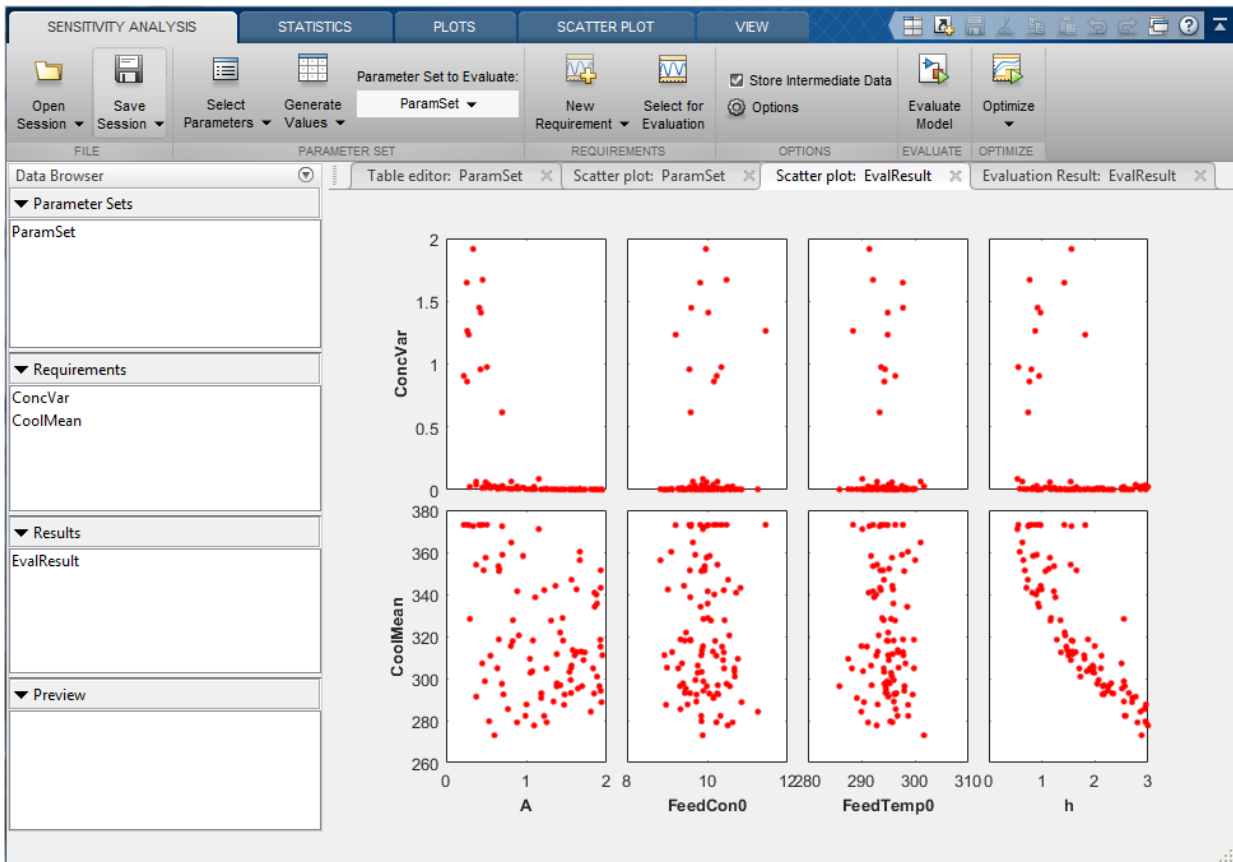
In the **Sensitivity Analysis** tab, click **Select for Evaluation**. By default, all requirements are selected to be evaluated. Click **Evaluate Model** to evaluate the `ConcVar` and `CoolMean` requirements for each row of parameter values in `ParamSet`. Note you can speed up evaluation by using parallel computing if you have the Parallel Computing Toolbox (TM), or by using fast restart. For more information, see "Use

Parallel Computing for Sensitivity Analysis" and "use Fast Restart Mode During Sensitivity Analysis" in the Simulink Design Optimization™ documentation.

A results scatter plot showing each parameter vs each requirement is updated during model evaluation. At the end of evaluation a table with the evaluation results is created, each row in the evaluation result table contains values for A, FeedCon0, FeedTemp0, h and the resulting requirement values ConcVar and CoolMean. The evaluation results are stored in the EvalResult variable in the **Results** area of the tool.

The screenshot displays the Simulink Design Optimization tool interface. The top menu bar includes 'SENSITIVITY ANALYSIS', 'STATISTICS', 'PLOTS', and 'VIEW'. Below the menu is a toolbar with icons for 'Open Session', 'Save Session', 'Select Parameters', 'Generate Values', 'Parameter Set to Evaluate', 'New Requirement', 'Select for Evaluation', 'Store Intermediate Data', 'Options', 'Evaluate Model', and 'Optimize'. The main workspace is divided into several panes. On the left, there is a 'Data Browser' pane with sections for 'Parameter Sets', 'Requirements', 'Results', and 'Preview'. The 'Results' section is expanded to show 'EvalResult'. The central pane displays a table editor with the following data:

A	FeedCon0	FeedTemp0	h	ConcVar	CoolMean
1.6665	9.1027	298.7040	0.5815	0.0078	360.1810
1.8304	10.4202	294.1778	1.7051	0.0017	304.7684
0.4286	9.5560	294.4233	0.8015	0.9576	373
1.8441	10.0500	296.2681	2.3446	0.0015	292.8644
1.3382	9.7277	296.0506	2.5454	0.0026	289.0413
0.3756	10.1518	295.0374	2.7658	0.0392	291.3596
0.7013	9.6998	297.5851	0.9037	0.0300	358.7545
1.1844	10.2450	294.7875	2.1729	0.0036	292.9212
1.9235	10.3697	291.0346	1.4615	0.0017	315.1435
1.9368	10.8559	290.4815	2.6991	0.0019	288.7562
0.4837	9.9029	294.8352	2.5439	0.0151	298.6608
1.9471	8.9308	295.0001	1.7965	0.0011	311.0358
1.9229	9.5802	299.8118	1.2818	0.0016	318.4055
1.0737	10.6773	290.3048	1.8478	0.0046	303.4633
1.6405	9.4639	294.5618	2.5084	0.0016	295.2404
0.4554	10.4805	292.1669	0.7694	1.6702	373
0.9592	10.0620	291.7999	0.8329	0.0170	358.2039
1.8483	10.7183	291.3615	0.8200	0.0035	340.7548
1.6260	9.0196	295.9212	0.9953	0.0025	342.2727
1.9271	9.9024	298.0002	2.2453	0.0014	294.1091
1.3803	9.3961	295.2950	2.3061	0.0021	297.6694
0.2643	11.4540	288.4227	0.8747	1.2637	373

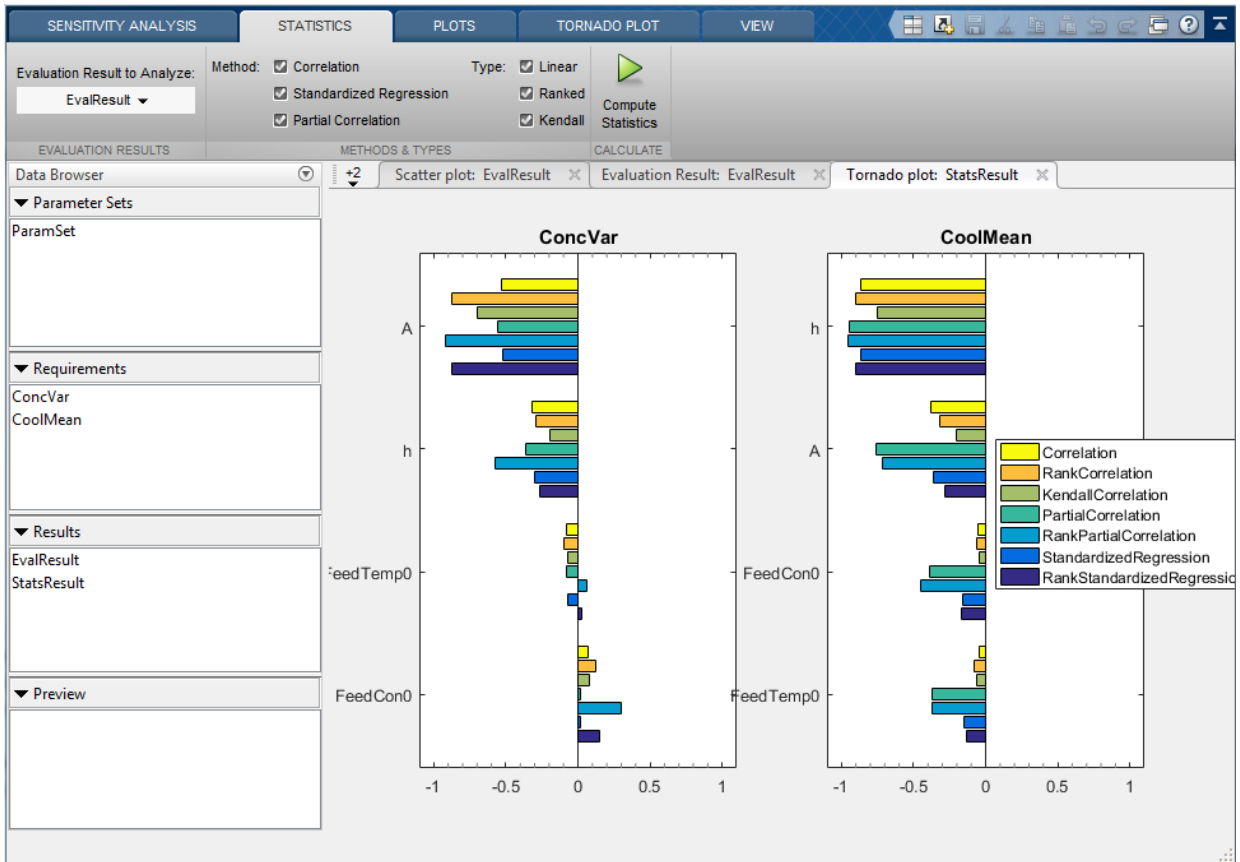


Analyze the Evaluation Results

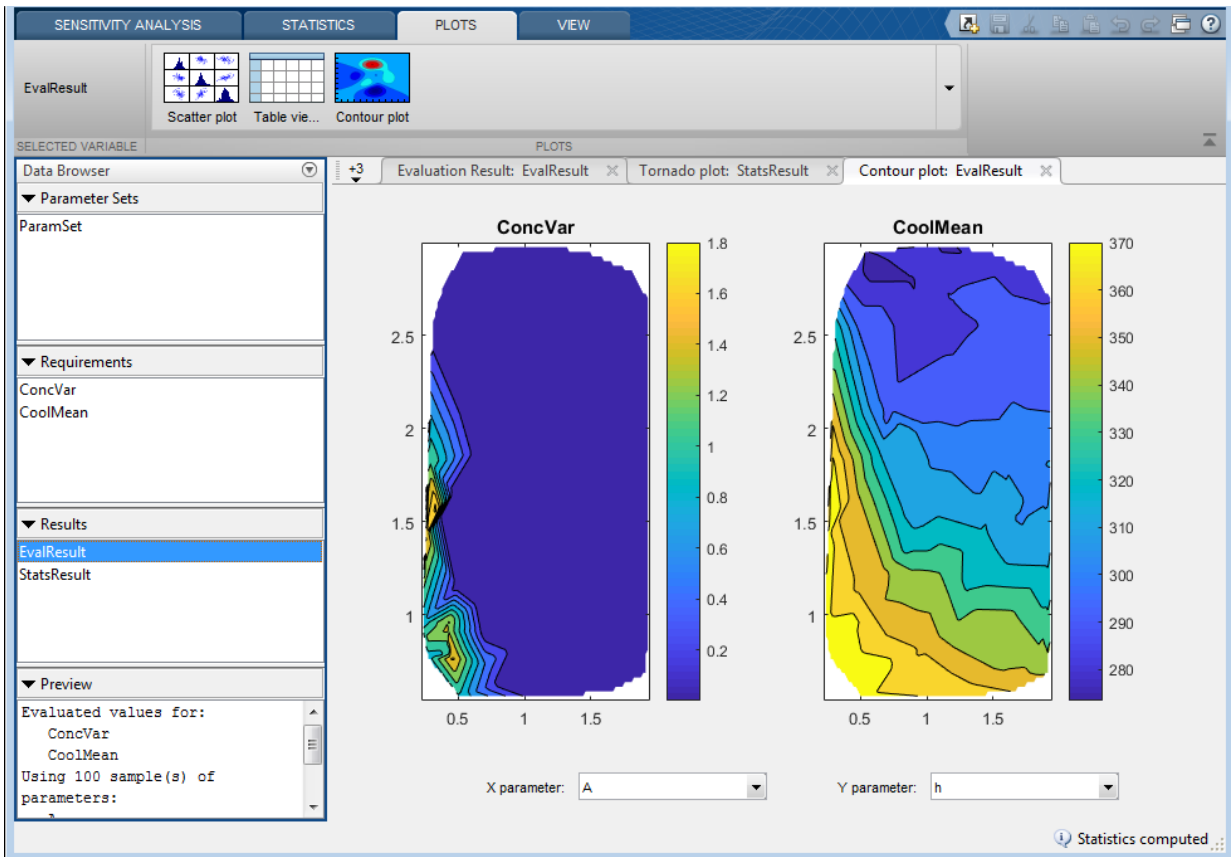
The results scatter plot for EvalResult shows that CoolMean is inversely correlated with h (increasing h decreases CoolMean) and that low values of h can result in high values for ConcVar. The plot shows that low values of A can result in high values for ConcVar, but it is not clear from the plot how A is correlated with ConcVar or CoolMean or which parameter influences ConcVar the most. To investigate further, in the **Statistics** tab, select all the analysis methods and types and click **Compute Statistics**. This performs analysis on the evaluation results and creates a tornado plot. The tornado plot shows the influence of each parameter on each requirement:

4 Sensitivity Analysis

- h is inversely correlated with CoolMean, and is the parameter that influences CoolMean the most.
- A is inversely correlated with CoolMean.
- FeedCon0 and FeedTemp0 are inversely correlated with CoolMean.
- A is inversely correlated with ConcVar, and is the parameter that influences ConcVar the most.
- h is inversely correlated with ConcVar.
- FeedCon0 and FeedTemp0 have mixed correlation with ConcVar, but have minimal correlation with ConcVar.



The analysis shows that choosing a large h , to reduce $CoolMean$ and choosing a large A to reduce $ConcVar$ would appear to be a good design choice. You can confirm this by creating a contour plot of $CoolMean$ and $CoolVar$ versus h and A . Select $EvalResult$ from the **Results** area of the tool browser, and in the **Plots** tab, in the plots gallery click **Contour plot**. On the contour plot select h for the **Y parameter**, note that large h medium values of A give low values for both $ConcVar$ and $CoolMean$.



Choose an Initial Guess for Optimization

Sort the Evaluation Result table by decreasing h , and select a row that has low $ConcVar$ and $CoolMean$ values. Right-click the selected row and click **Extract Parameter Values**. The extracted parameter values are saved in the `ParamValues` variable in the

Results area of the tool browser. These parameter values are used as the initial guess for optimization.

The screenshot shows the Sensitivity Analysis tool interface. The main window displays a table of results. The table has the following columns: A, FeedCon0, FeedTemp0, h, ConcVar, and CoolMean. The row with the following values is highlighted in blue: A=0.6029, FeedCon0=9.8876, FeedTemp0=301.6883, h=2.8791, ConcVar=0.0268, and CoolMean=273.0267.

A	FeedCon0	FeedTemp0	h	ConcVar	CoolMean
1.0971	10.5093	292.8640	2.9977	0.0228	277.6852
1.0021	9.5104	293.0473	2.9602	0.0045	287.6597
1.4709	8.9741	298.7273	2.9576	0.0021	287.4703
1.2535	10.2275	295.7842	2.9567	0.0077	279.2062
0.5364	9.8572	295.4055	2.9541	0.0376	279.6695
0.7708	9.3332	296.4149	2.9324	0.0076	285.4186
0.6029	9.8876	301.6883	2.8791	0.0268	273.0267
1.8036	11.2630	289.9894	2.8568	0.0035	284.2235
0.8848	10.6303	291.2480	2.8008	0.0265	279.1187
0.3756	10.1518	295.0374	2.7658	0.0392	291.3596
1.1850	10.1538	289.1535	2.7047	0.0035	290.7058
1.9368	10.8559	290.4815	2.6991	0.0019	288.7562
1.5567	9.4412	294.5130	2.6388	0.0018	293.1237
1.2221	10.3301	296.7435	2.5784	0.0056	282.2222
1.9104	10.5637	285.9125	2.5633	0.0015	296.3316
0.9897	9.8505	298.7508	2.5609	0.0070	282.2279
1.3382	9.7277	296.0506	2.5454	0.0026	289.0413
0.2971	9.9012	295.5474	2.5444	0.0228	328.2981
0.4837	9.9029	294.8352	2.5439	0.0151	298.6608
1.6405	9.4639	294.5618	2.5084	0.0016	295.2404
0.4495	9.3714	298.2479	2.5008	0.0135	307.1411
1.8441	10.0500	296.2681	2.3446	0.0015	292.8644

Optimize

Use the data in the Sensitivity Analysis tool to create an optimization problem to optimize A and h. In the **Sensitivity Analysis** tab click **Optimize**, and select **Create Response Optimization Session**. This opens a dialog to import data from Sensitivity Analysis to Response Optimization tool.

- Select both the ConcVar and CoolMean requirements to import.
- Select ParamValues to import as design variables for optimization.

- Select `EvalResult` to import as uncertain variables to use during optimization.
- Click **OK** to import the data to the Response Optimization tool

Select requirements to import:

Import	Requirement	Description
<input checked="" type="checkbox"/>	ConcVar	Minimize signal 'var'.
<input checked="" type="checkbox"/>	CoolMean	Minimize signal 'mean'.

Select parameter values to import as design variables:

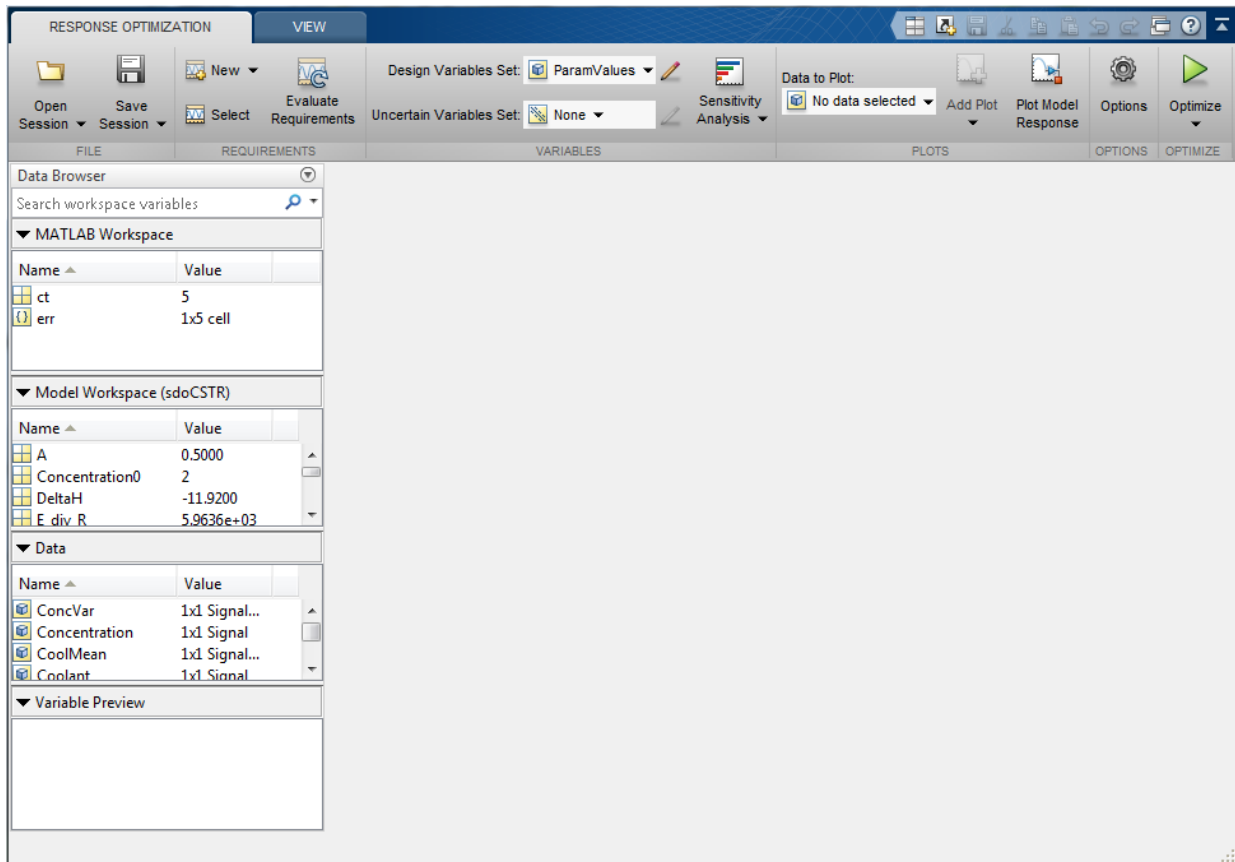
Import	Parameter Values	Description
<input checked="" type="checkbox"/>	ParamValues	Values for - A, FeedCon0, FeedTemp0, h

Select parameter values to import as uncertain variables:

Import	Parameter Values	Description
<input checked="" type="checkbox"/>	EvalResult	100 values for - A, FeedCon0, FeedTemp0, h

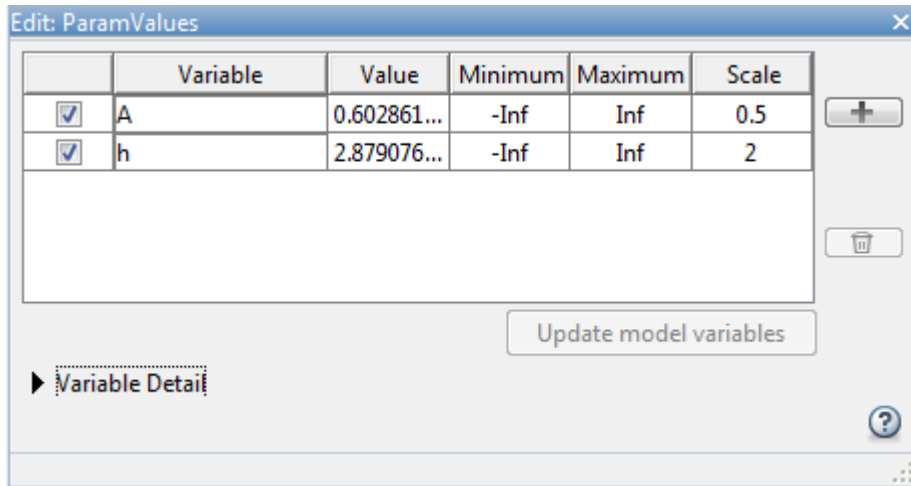
OK Cancel Help

4 Sensitivity Analysis

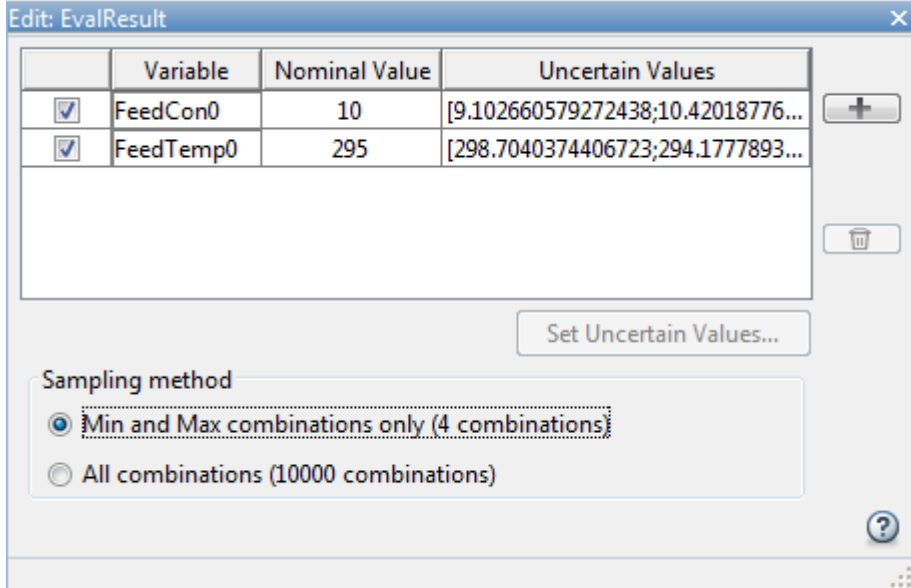


Configure the Response Optimization tool to optimize the CSTR design:

- Click the pencil icon to edit the `ParamValues` design variable set, and remove the `FeedCon0` and `FeedTemp0` variables from the design variable set.



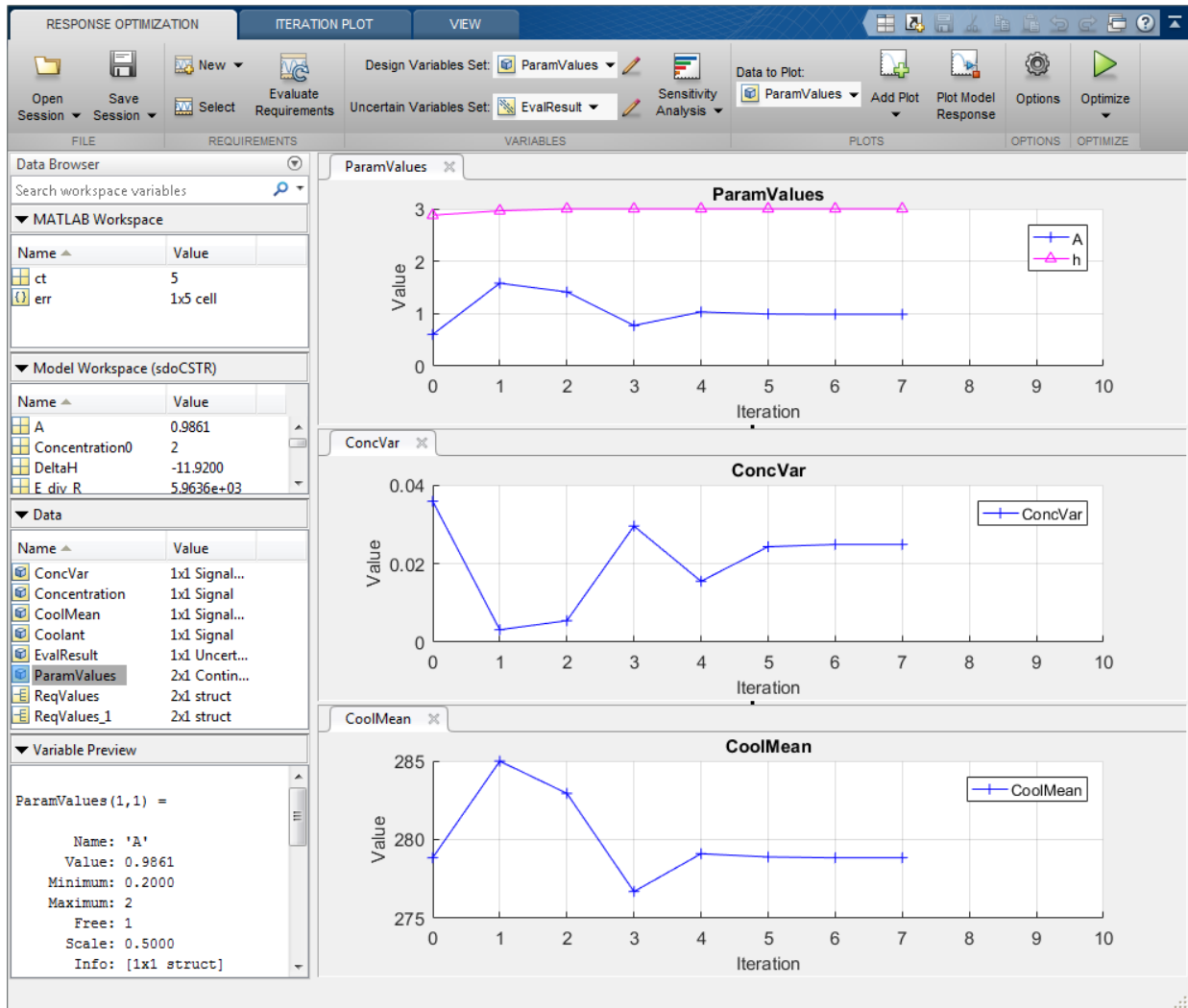
- Select EvalResult as the uncertain variable set, click the pencil icon to edit EvalResult, and remove A and h from the uncertain variable set.



4 Sensitivity Analysis

Add iteration plots to see how the variables `ParamValues` (A and h), and optimization requirements `ConcVar` and `CoolMean` change during optimization.

- Select the variables in the `Data to Plot` drop-down list, and select `Iteration Plot` in the `Add Plot` drop-down list.
- Click **Optimize**.



The optimization minimizes `CoolMean` and `ConcVar` in the presence of varying `FeedCon0` and `FeedTemp0`.

Related Examples

To learn how to explore the CSTR design space using the `sdo.evaluate` command, see "Design Exploration using Parameter Sampling (Code)".

References

[1] Bequette, B.W. *Process Dynamics: Modeling, Analysis and Simulation*. 1st ed. Upper Saddle River, NJ: Prentice Hall, 1998.

Close the model

```
bdclose('sdoCSTR')
```

See Also

More About

- "Design Exploration Using Parameter Sampling (Code)" on page 4-200
- "Use Parallel Computing for Sensitivity Analysis" on page 4-137
- "Use Fast Restart Mode During Sensitivity Analysis" on page 4-144
- "Generate Parameter Samples for Sensitivity Analysis" on page 4-10
- "Analyze Relation Between Parameters and Design Requirements" on page 4-91
- "Validate Sensitivity Analysis" on page 4-129

Identify Key Parameters for Estimation (GUI)

This example shows how to use sensitivity analysis to narrow down the number of parameters that you need to estimate when fitting a model. This example uses a model of the vestibulo-ocular reflex, which generates compensatory eye movements.

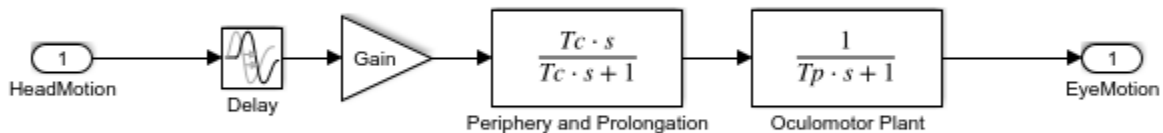
Model Description

The vestibulo-ocular reflex (VOR) enables the eyes to move at the same speed and in the opposite direction as the head, so that vision is not blurred when the head moves during normal activity. For example, if the head turns to the right, the eyes turn to the left at the same speed. This happens even in the dark. In fact, the VOR is most easily characterized by measurements in the dark, to ensure that eye movements are predominantly driven by the VOR.

Head rotation is sensed by organs in the inner ears, known as semicircular canals. These detect head motion and transmit signals about head motion to the brain, which sends motor commands to the eye muscles, so that eye movements compensate for head motion. We would like to use eye movement data to estimate model parameters for these various stages. The model we will use is shown below. There are four parameters in the model: Delay, Gain, T_c , and T_p .

```
open_system('sdoVOR')
```

Vestibulo-Ocular Reflex



Copyright 2013 The MathWorks, Inc.

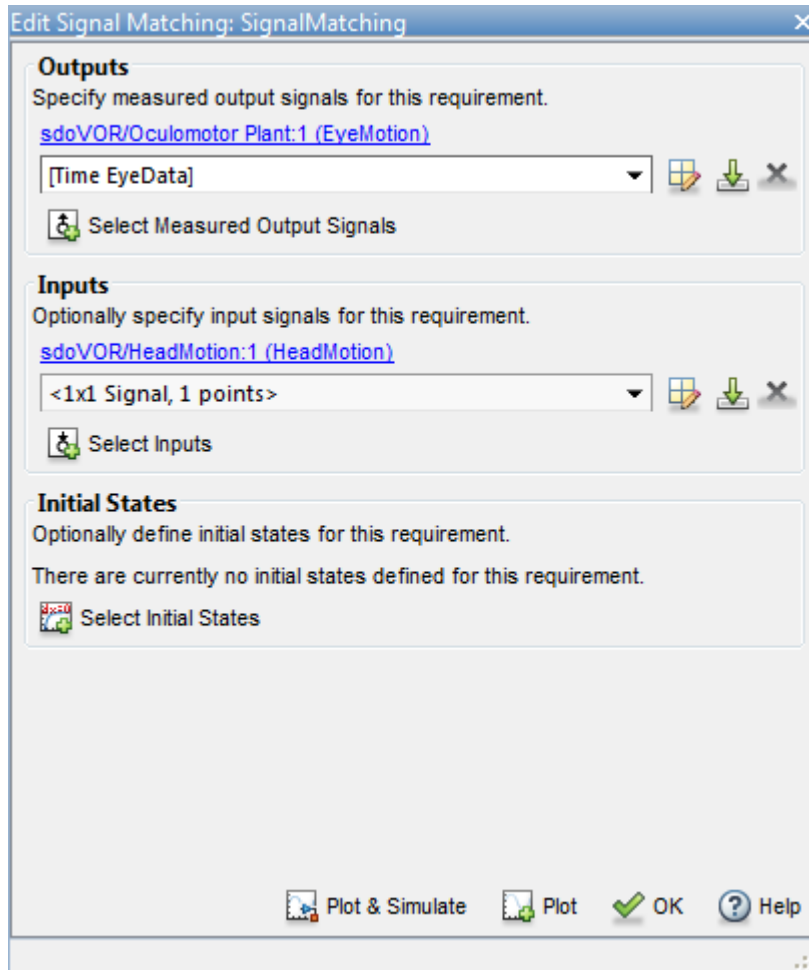
The file `sdoVOR_Data.mat` contains uniformly sampled data of stimulation and eye movements. If the VOR were perfectly compensatory, then a plot of eye movement data,

when flipped vertically, would overlay exactly on top of a plot of head motion data. Such a system would be described by a gain of 1 and a phase of 180 degrees. However, real eye movements are close, but not perfectly compensatory.

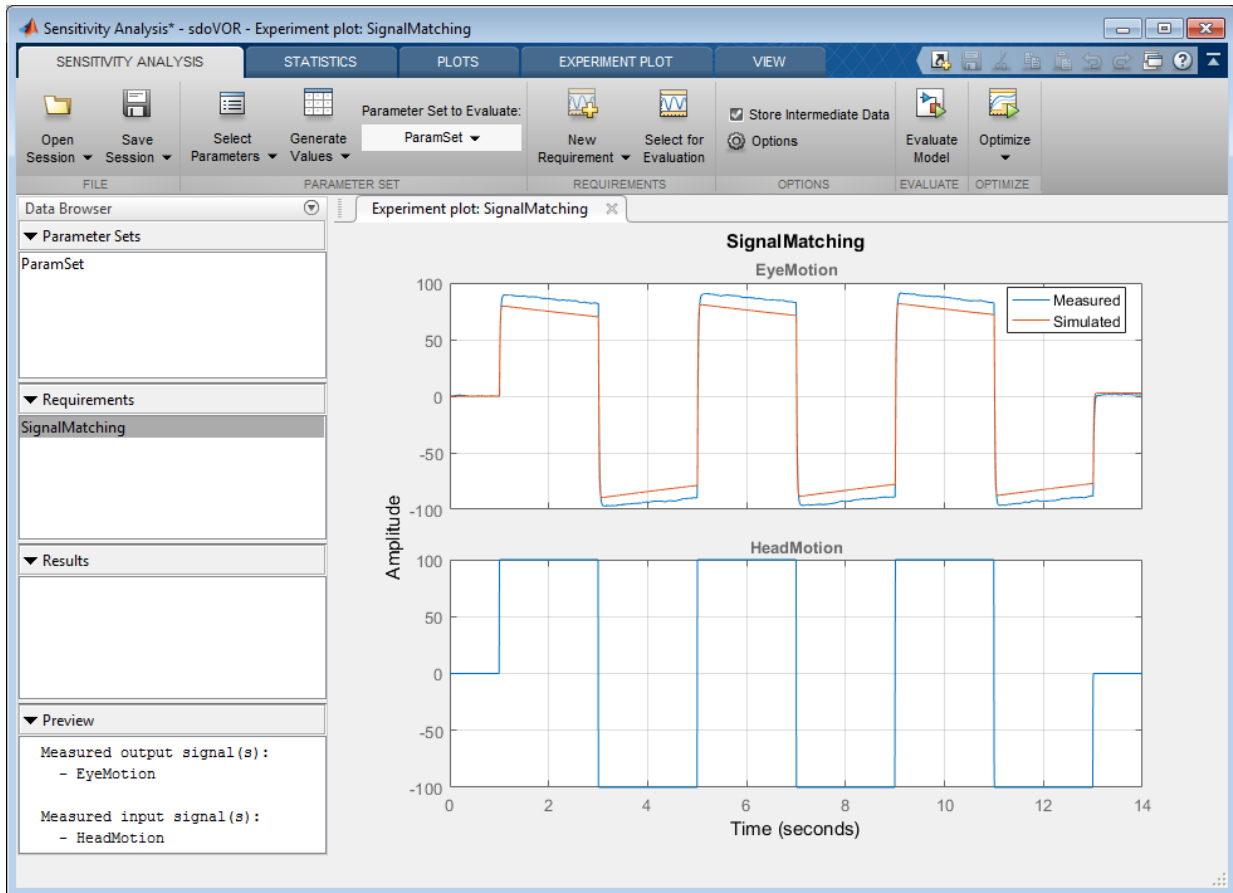
```
load sdoVOR_Data.mat; % Column vectors: Time HeadData EyeData
```

We will use Sensitivity Analysis UI to see how well the model output fits the data, and explore which model parameters have the most influence on the goodness of fit. To open Sensitivity Analysis UI, navigate to the **Analysis** menu in the Simulink model and click **Sensitivity Analysis...**

To associate the data with the model, click **New Requirement** and select a Signal Matching requirement. This specifies an objective function consisting of the sum of squared error between the data and model output. In the Signal Matching dialog, specify the output as [Time EyeData], and specify the input as [Time HeadData].

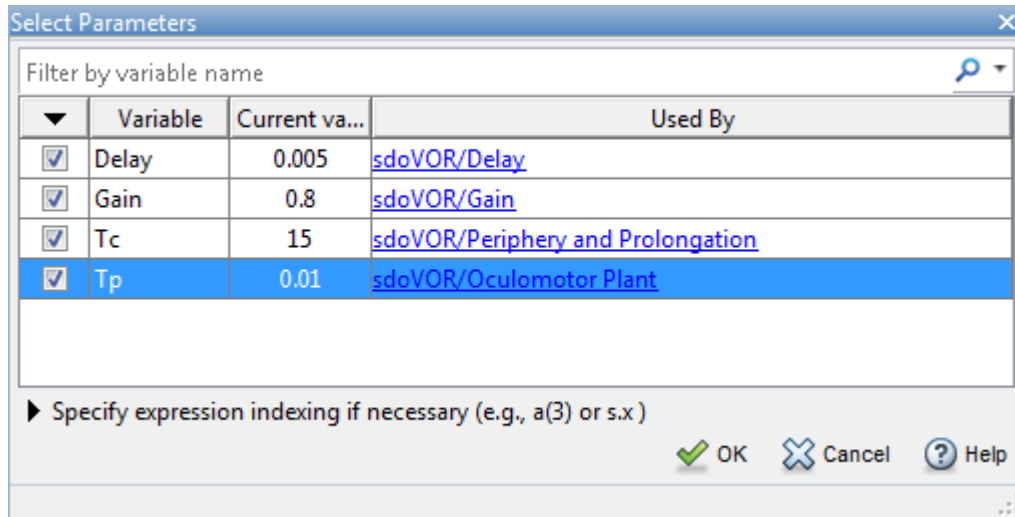


To view the eye movement data, navigate to the data browser on the left side of the UI, right-click the `SignalMatching` requirement, and select `Plot & Simulate`. The bottom plot shows the stimulation, consisting of a series of pulses. The top plot shows eye movement data, which resembles but does not exactly match the stimulation. It also shows that the model simulated output does not match the eye movement data, because model parameters need to be estimated.



Explore the Design Space

The model attempts to capture the phenomena which cause the difference between head movements and eye movements. Here we will explore the design space formed by the model parameters. To specify the parameters to explore in Sensitivity Analysis UI, click **Select Parameters** and create a new parameter set. Select all model parameters: Delay, Gain, Tc and Tp.



Explore the design space by generating parameter values. Click **Generate Values** and select random values. For repeatability of the example, reset the random number generator.

```
rng('default')
```

Since there are 4 parameters, we will generate 40 samples.

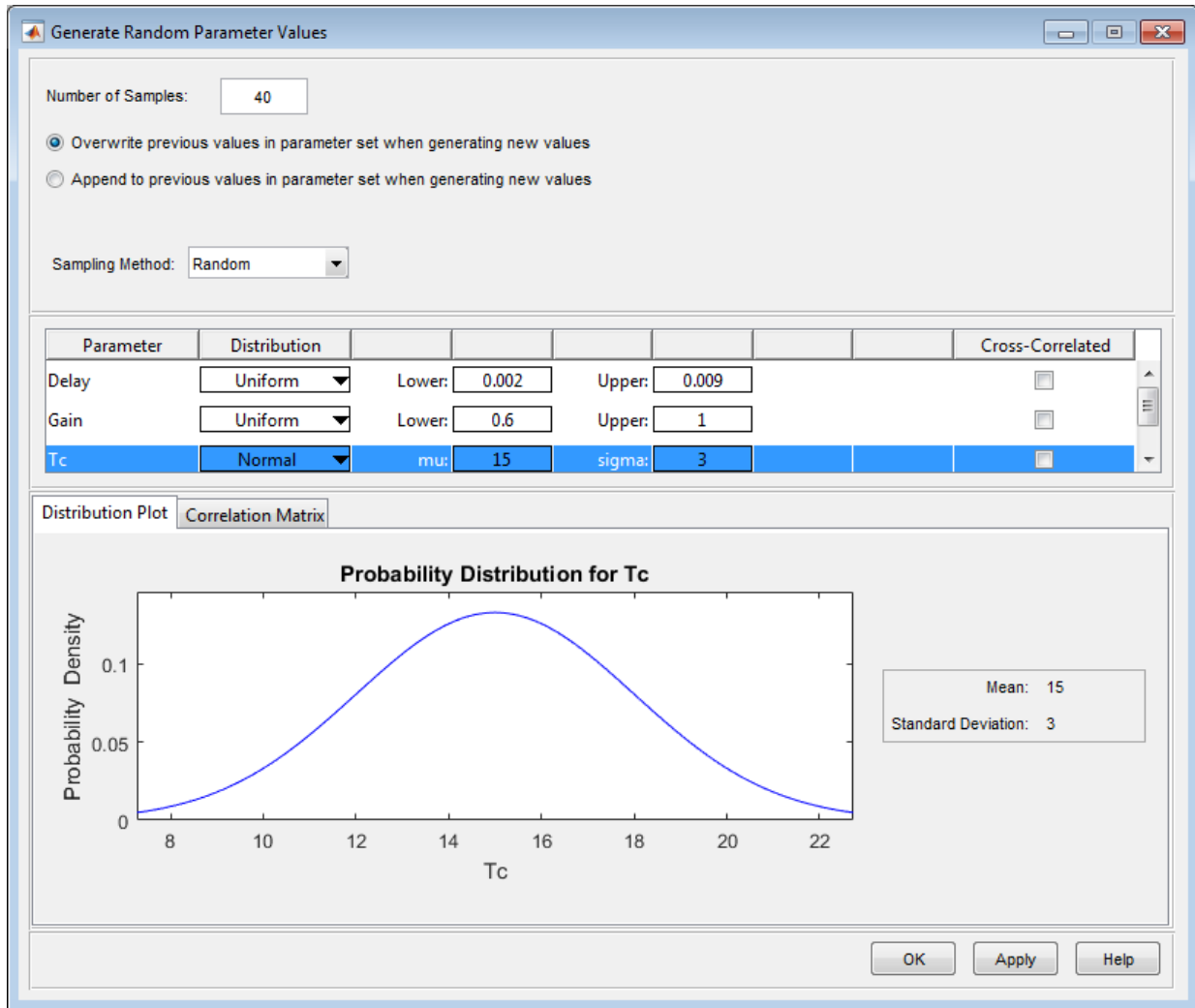
The `Delay` parameter models the fact that there is some delay in communicating the signals from the inner ear to the brain and eyes. This delay is due to the time needed for chemical neurotransmitters to traverse the synaptic clefts between nerve cells. Based on the number of synapses in the vestibulo-ocular reflex, this delay is expected to be around 5 ms. We will model it with a uniform distribution with a lower bound of 2 ms and an upper bound of 9 ms.

The `Gain` parameter models the fact that in the dark, the eyes do not move quite as much as the head. We will model it with a uniform distribution with a lower bound of 0.6 and an upper bound of 1.

The `Tc` parameter models the dynamics associated with the semicircular canals, as well as some additional neural processing. The canals are high-pass filters, because after a subject has been put into rotational motion, the neurally active membranes in the canals slowly relax back to resting position, so the canals stop sensing motion. Thus, after the stimulation undergoes transition edges, eye movements tend to depart from the

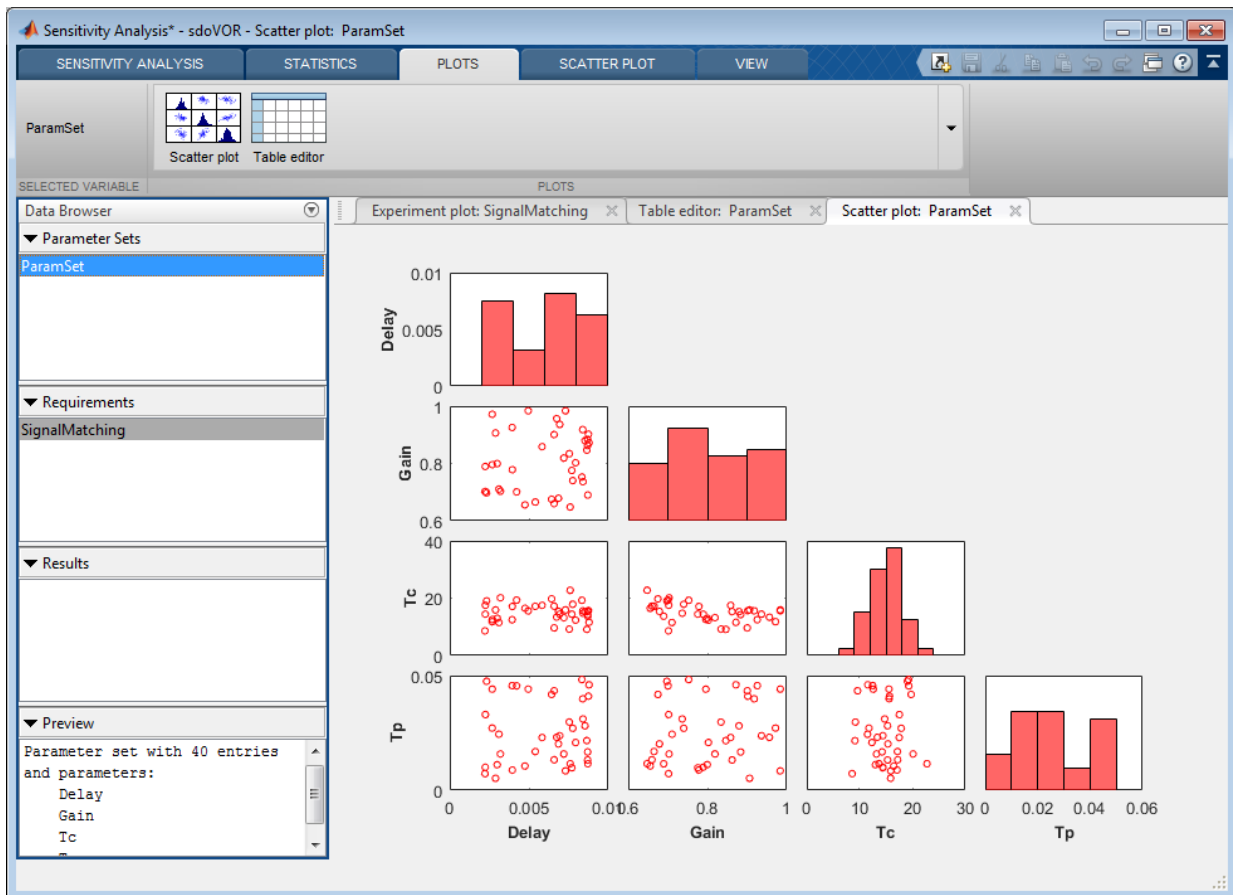
stimulation over time. Based on mechanical characteristics of the canals, combined with additional neural processing which prolongs this time constant to improve the accuracy of the VOR, we will model T_C with a normal (i.e., bell curve) distribution with a mean of 15 seconds and a standard deviation of 3 seconds.

Finally, the T_P parameter models the dynamics of the oculomotor plant, i.e. the eye and the muscles and tissues attached to it. The plant can be modeled by two poles, however it is believed that the pole with the larger time constant is cancelled by precompensation in the brain, to enable the eye to make quick movements. Thus in the plot, when the stimulation undergoes transition edges, the eye movements follow with only a little delay. We will model T_P with a uniform distribution with a lower bound of 0.005 seconds and an upper bound of 0.05 seconds.



When the sample values are generated, they appear in a table in the Sensitivity Analysis UI. To plot them, select `ParamSet` in the data browser, click the **Plots** tab, and make a scatter plot. The sampling above used default options, and these are reflected in the scatter plot. For parameters modeled by a uniform distribution, the histograms appear approximately uniform. However, parameter `Tc` was modeled by a normal distribution, and its histogram has a bell curve profile. If *Statistics and Machine Learning Toolbox™*

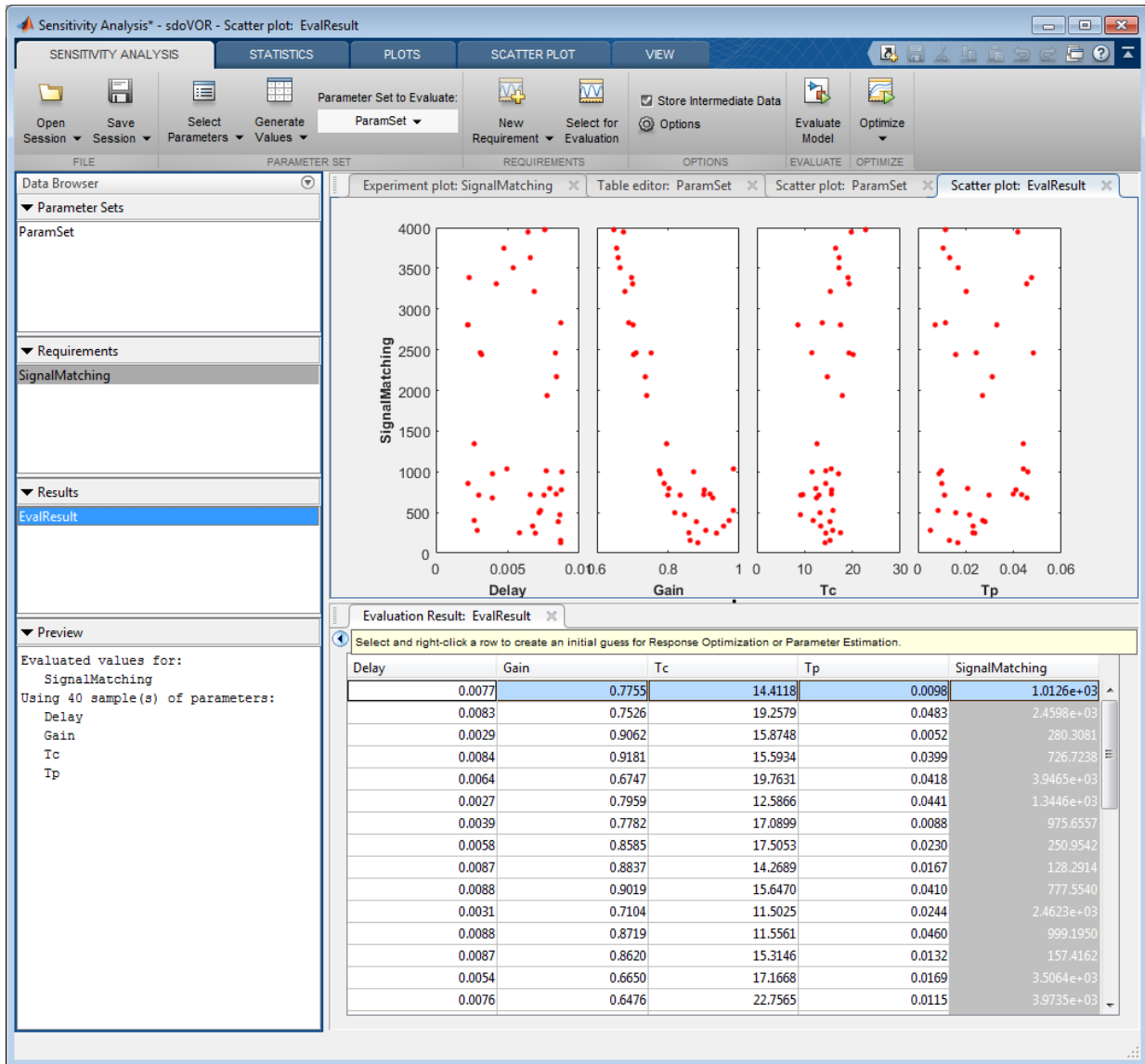
is available, many other distributions may be used, and sampling can be done using Sobol or Halton low-discrepancy sequences. The off-diagonal plots show scatter plots between pairs of different variables. Since we did not specify cross-correlations between parameters, the scatter plots appear uncorrelated. However, if parameters were believed to be correlated, this can be specified using Correlation Matrix tab in the dialog for generating random parameter values.



Evaluate the Model

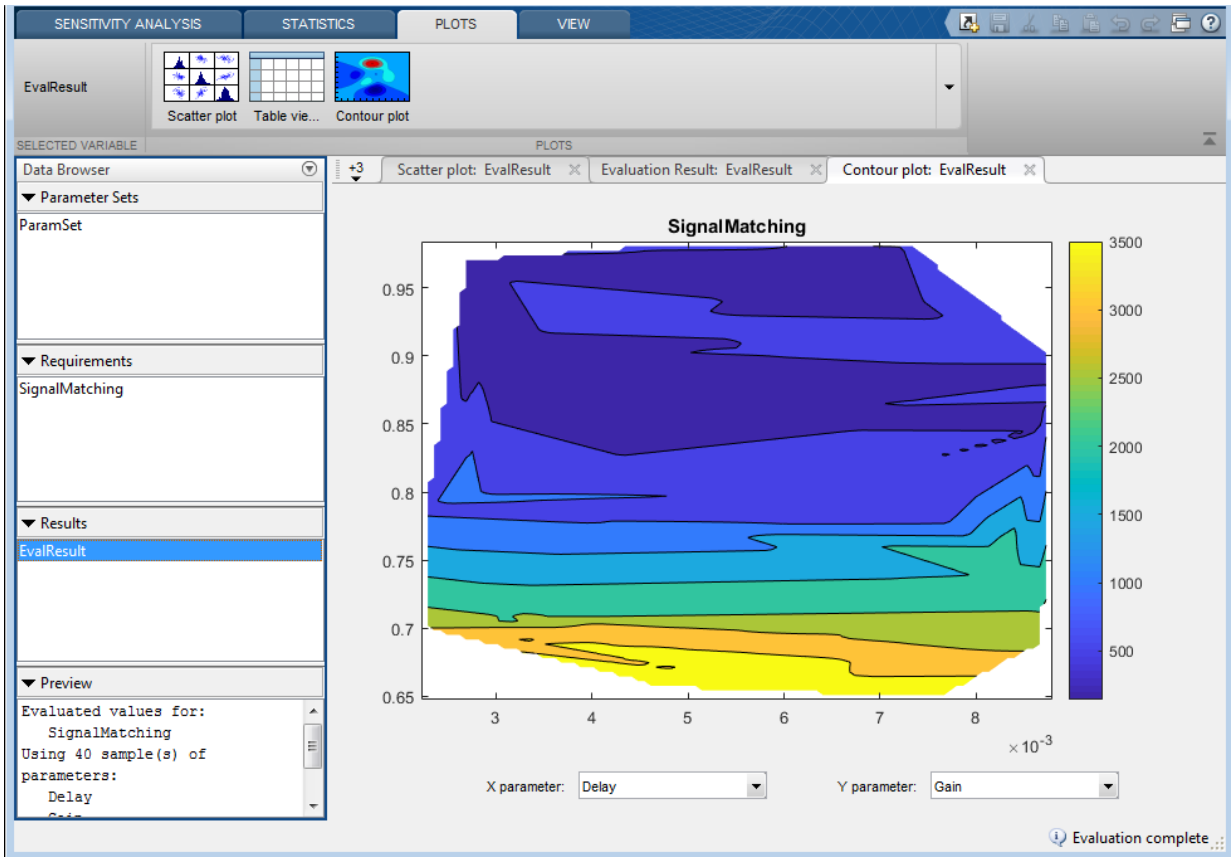
Now that we have generated values for the parameter set and specified a requirement (SignalMatching), we can evaluate the model. In the **Sensitivity Analysis** tab, click **Evaluate Model**.

The model is run once for each set of parameter values, and the results scatter plot is updated as new computations become available. Evaluation could also be sped up using parallel computing. After evaluation is complete, all results are also displayed in a table.



From the scatter plot of evaluation results, the SignalMatching requirement seems to vary systematically as a function of Gain and Tc, but not Delay or Tp. Something similar can be seen in a contour plot. Select the EvalResults variable in the data

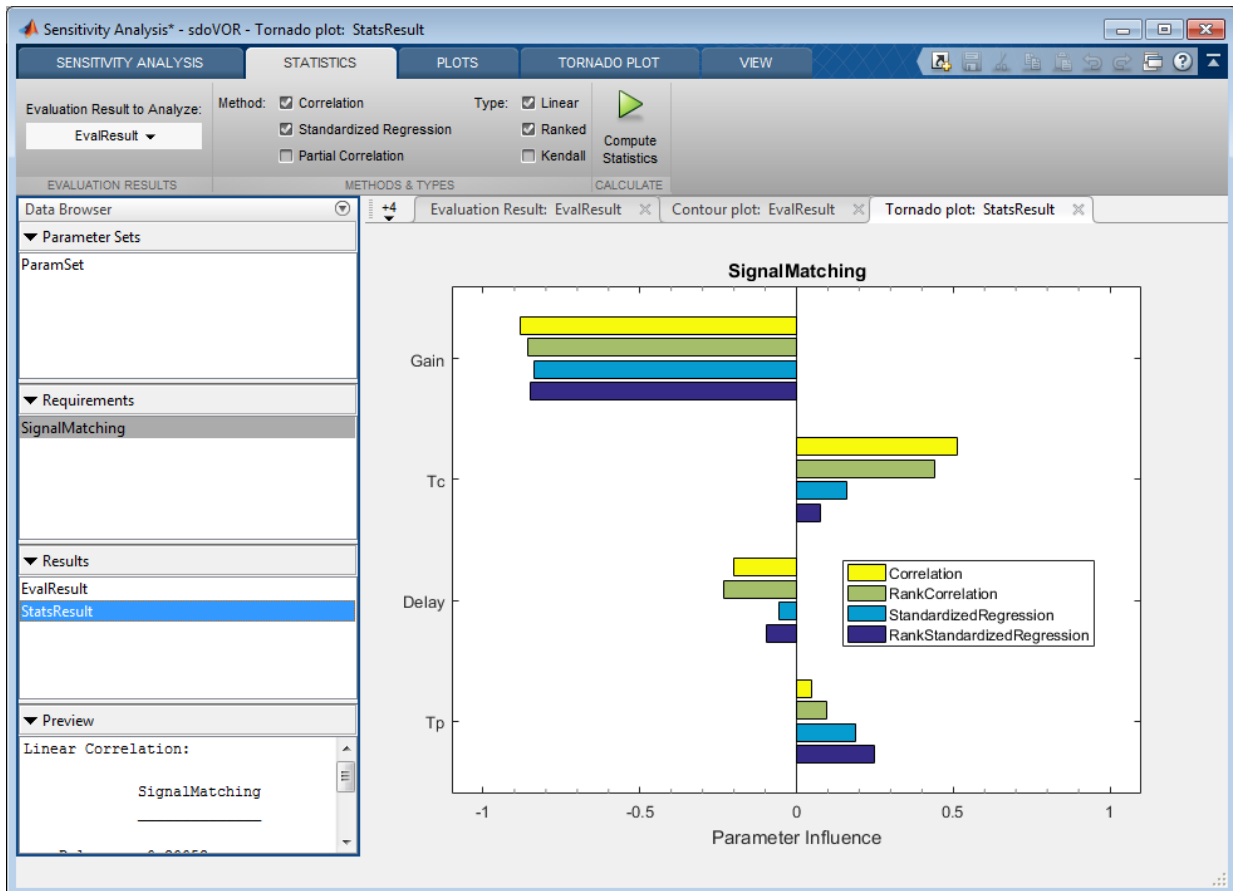
browser, click the **Plots** tab, and make a contour plot. The requirement does not vary systematically from left to right as a function of `Delay`, but it does vertically as a function of `Gain`.



Statistical Analysis

We can use statistical analysis to quantify how much each parameter influences the requirement. Click the **Statistics** tab and select both correlation and standardized regression; and both linear and ranked analysis types. If Statistics and Machine Learning Toolbox is available, partial correlation and Kendall correlation can also be selected. Click **Compute Statistics** to carry out the calculations and show a tornado plot. The tornado plot displays results from top to bottom in order of which parameter

most influences the requirement. The statistical values range from -1 to 1, where the magnitude indicates how much the parameter influences the requirement, and the sign indicates whether an increase in the parameter value corresponds to an increase or decrease in the requirement value. By most measures, this SignalMatching requirement is more sensitive to Gain and Tc, and less sensitive to Delay and Tp.



Select Parameters for Estimation

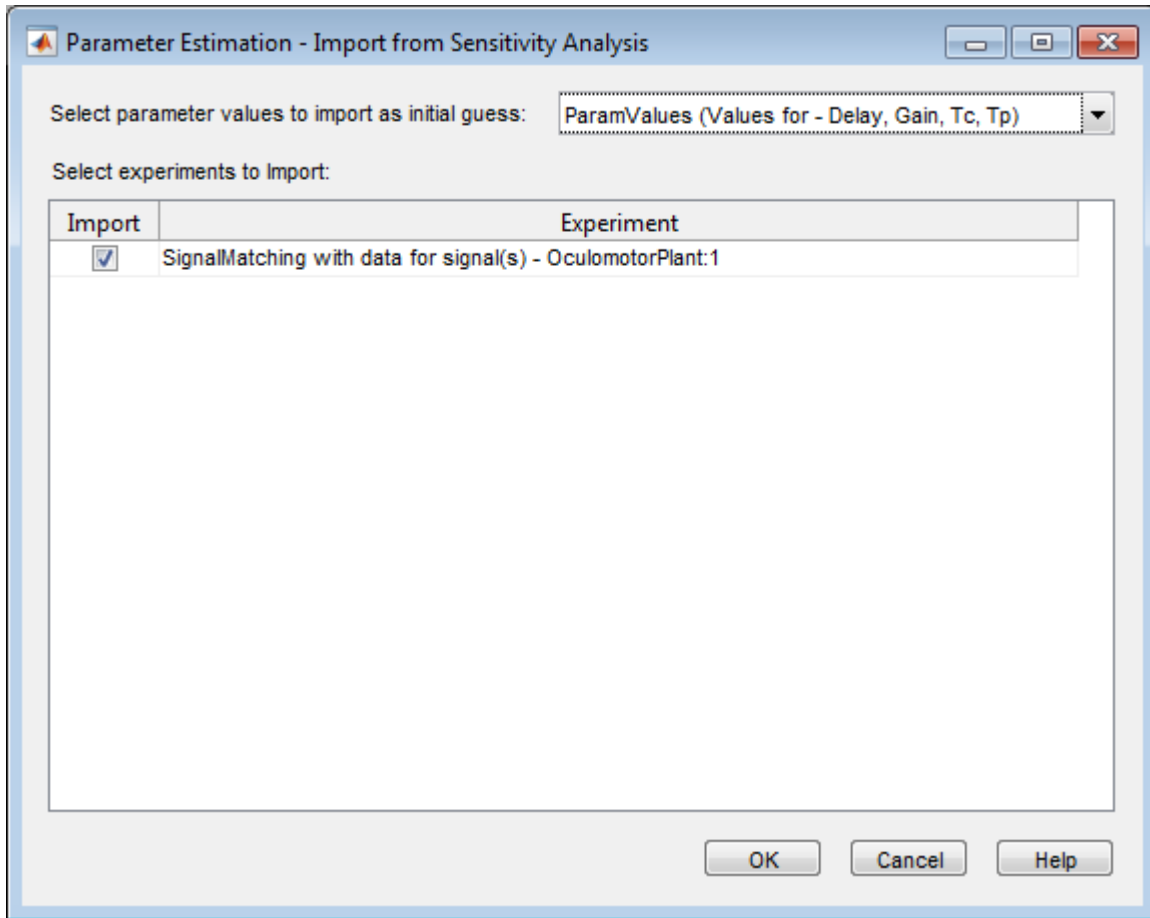
For parameter estimation, we need to specify starting values for the parameters. Click the evaluation results table and click the SignalMatching column header to sort results. Select the row of parameter values that minimizes the SignalMatching

requirement. Right-click on the row and extract these parameter values. A new variable, ParamValues, is shown in the data browser.

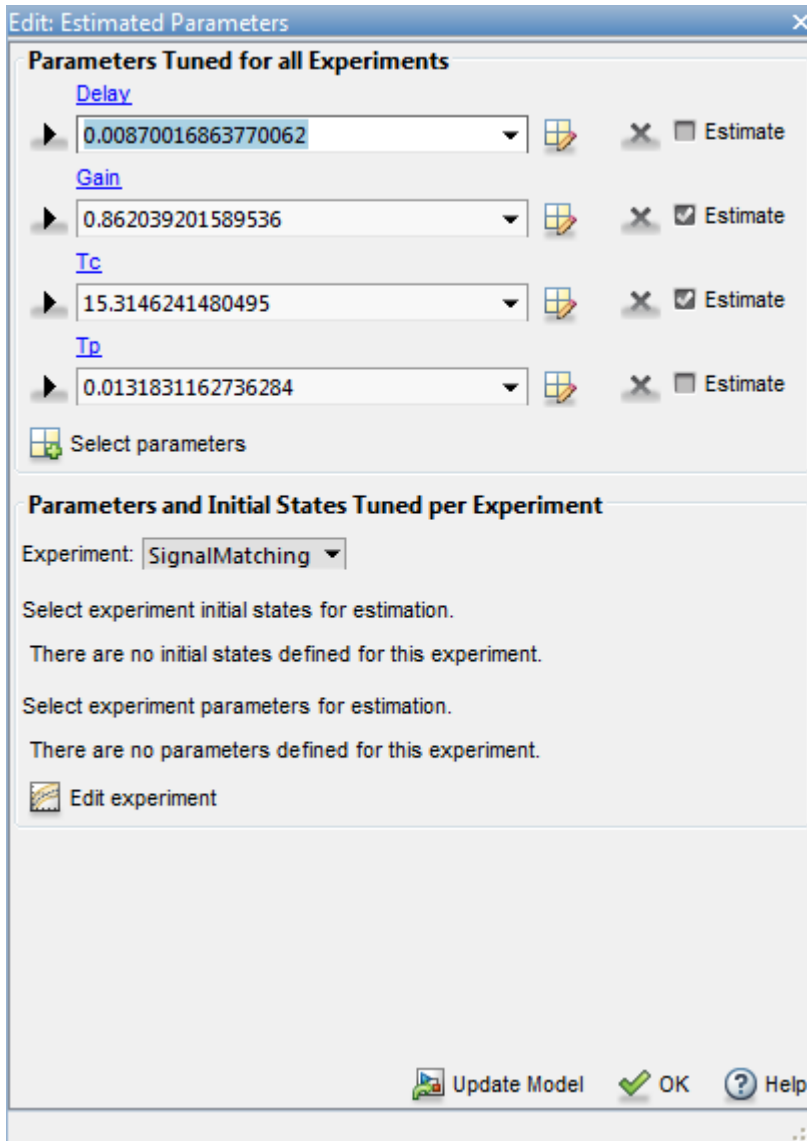
The screenshot shows the 'Sensitivity Analysis* - sdoVOR - Evaluation Result: EvalResult' window. The interface includes a menu bar with 'SENSITIVITY ANALYSIS', 'STATISTICS', 'PLOTS', and 'VIEW'. Below the menu bar is a toolbar with icons for 'Open Session', 'Save Session', 'Select Parameters', 'Generate Values', 'Parameter Set to Evaluate: ParamSet', 'New Requirement', 'Select for Evaluation', 'Store Intermediate Data', 'Options', 'Evaluate Model', and 'Optimize'. The main window is divided into several panes. On the left is the 'Data Browser' with sections for 'Parameter Sets' (containing 'ParamSet'), 'Requirements' (containing 'SignalMatching'), 'Results' (containing 'EvalResult', 'ParamValues', and 'StatsResult'), and 'Preview'. The central pane displays a table with the following data:

Delay	Gain	Tc	Tp	SignalMatching
0.0077	0.7755	14.4118	0.0098	1.0126e+03
0.0083	0.7526	19.2579	0.0483	2.4598e+03
0.0029	0.9062	15.8748	0.0052	280.3081
0.0084	0.9181	15.5934	0.0399	726.7238
0.0064	0.6747	19.7631	0.0418	3.9465e+03
0.0027	0.7959	12.5866	0.0441	1.3446e+03
0.0039	0.7782	17.0899	0.0088	975.6557
0.0058	0.8585	17.5053	0.0230	250.9542
0.0087	0.8837	14.2689	0.0167	128.2914
0.0088	0.9019	15.6470	0.0410	777.5540
0.0031	0.7104	11.5025	0.0244	2.4623e+03
0.0088	0.8719	11.5561	0.0460	999.1950
0.0087	0.8620	15.3146	0.0132	157.4162
0.0054	0.6650	17.1668	0.0169	3.5064e+03
0.0076	0.6476	22.7565	0.0115	3.9735e+03
0.0030	0.7993	12.9993	0.0111	713.5724
0.0050	0.9839	15.5620	0.0441	1.0352e+03
0.0084	0.7362	14.7525	0.0311	2.1674e+03
0.0075	0.8341	9.2009	0.0297	714.3145
0.0087	0.6895	13.6831	0.0115	2.8307e+03
0.0066	0.9005	9.6160	0.0434	719.8047
0.0022	0.7020	17.5211	0.0330	2.8054e+03

To transition from sensitivity analysis to parameter estimation, navigate to the **Sensitivity Analysis** tab, click **Optimize**, and open a parameter estimation session. In the dialog that appears, specify that you want to use the parameter values in ParamValues, and the SignalMatching requirement.



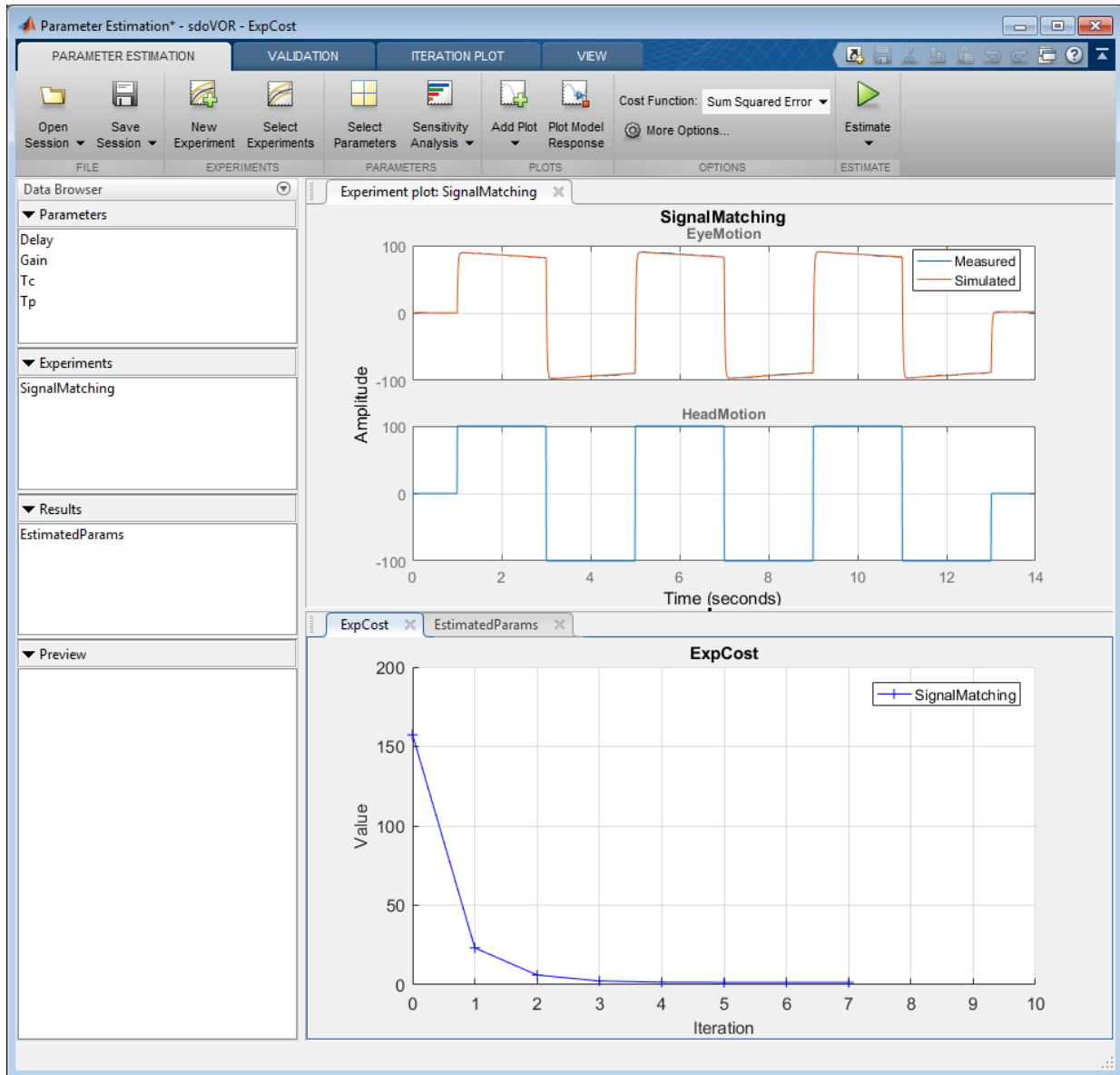
Since we found above that parameters $Gain$ and T_c have the most influence on the value of `SignalMatching`, we would like to estimate only these two parameters, since the time for estimation increases with the number of parameters being estimated. In Parameter Estimation UI, click **Select Parameters** and select only $Gain$ and T_c for estimation.



Since the experiment definition has been imported from `SignalMatching` and the parameter values have been imported from `ParamValues`, we have everything needed for estimation. Click **Estimate** to carry out parameter estimation for `Gain` and `Tc`.

Because we are only estimating the two most influential parameters, estimation converges quickly and the model output closely matches the data. As was the case with model evaluations in sensitivity analysis, parallel computing could be used to speed up estimation.

4 Sensitivity Analysis



In summary, Sensitivity Analysis UI was used to explore the parameter design space and determine that two parameters, Gain and Tc, were substantially more influential than

the others. A start point for estimation was also determined. This start point and the requirement of obtaining a good fit to experimental data were imported into Parameter Estimation UI. Estimation completed quickly because only two parameters needed to be estimated, and the model output fit the data with very little residual error.

Close the model

```
bdclose('sdoVOR')
```

See Also

Related Examples

- “Identify Key Parameters for Estimation (Code)” on page 4-217
- “Use Parallel Computing for Sensitivity Analysis” on page 4-137
- “Use Fast Restart Mode During Sensitivity Analysis” on page 4-144
- “Generate Parameter Samples for Sensitivity Analysis” on page 4-10
- “Analyze Relation Between Parameters and Design Requirements” on page 4-91
- “Validate Sensitivity Analysis” on page 4-129

Explore Design Reliability Using Parameter Sampling (GUI)

This example shows how to use the Sensitivity Analysis tool to explore the behavior of a PI controller for a DC motor. The controller is susceptible to variations caused by component tolerances, and the impact on controller reliability is explored.

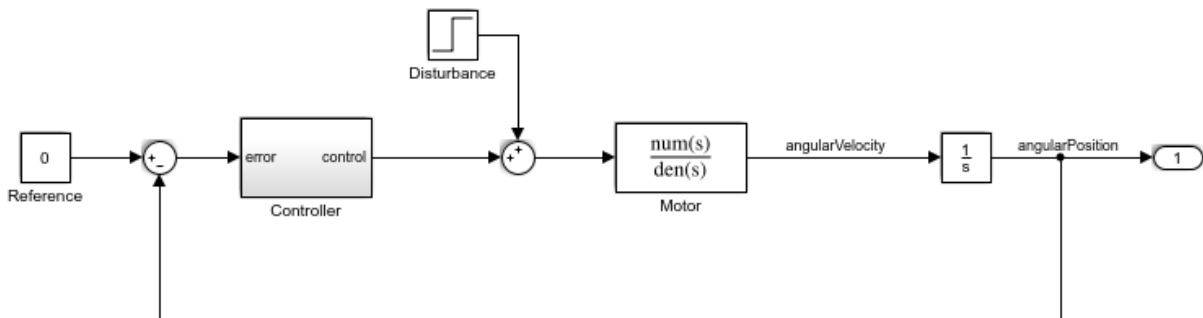
You explore the controller reliability by characterizing the components using probability distributions. You use the distributions to generate random samples and perform Monte-Carlo evaluation of the controller design at these sample points. You evaluate the impact of the component tolerances on the controller behavior, and use statistical analysis to determine which components have the most influence on whether the controller meets its requirements. This analysis guides the selection of component tolerances.

This example requires Statistics and Machine Learning Toolbox™.

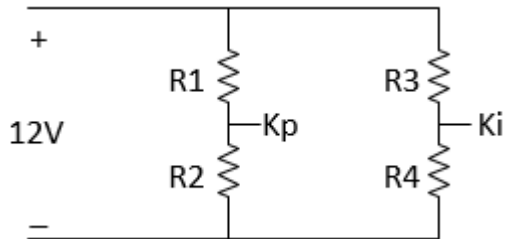
Implementation of Controller for DC Motor

The controller enables the DC motor's angular position to match a desired reference value. The load on the motor is subject to disturbances, and the controller needs to reject these disturbances. The Simulink model can be used to probe how well the controller rejects a step disturbance at 1 second.

```
open_system('sdoMotorPosition');
```



The gains of the PI controller, K_p and K_i , are set using resistors in the circuit below:

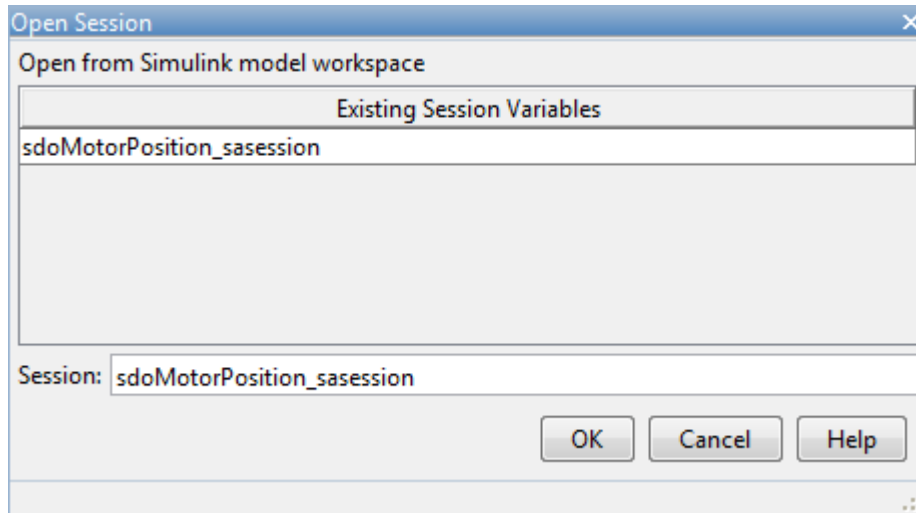


The resistances R_1 through R_4 are 47 kOhm, 180 kOhm, 10 kOhm and 10 kOhm respectively. These were chosen to set K_p and K_i to values that enable the controller to meet the requirements for disturbance rejection. However in practice, the actual resistor values will differ from the nominal ones, within a tolerance. This raises concern about whether the actual controller will still satisfy the requirements. To explore the effect of different resistance values, use the Sensitivity Analysis tool. In the Simulink **Analysis** menu, click **Sensitivity Analysis...** to open the tool.

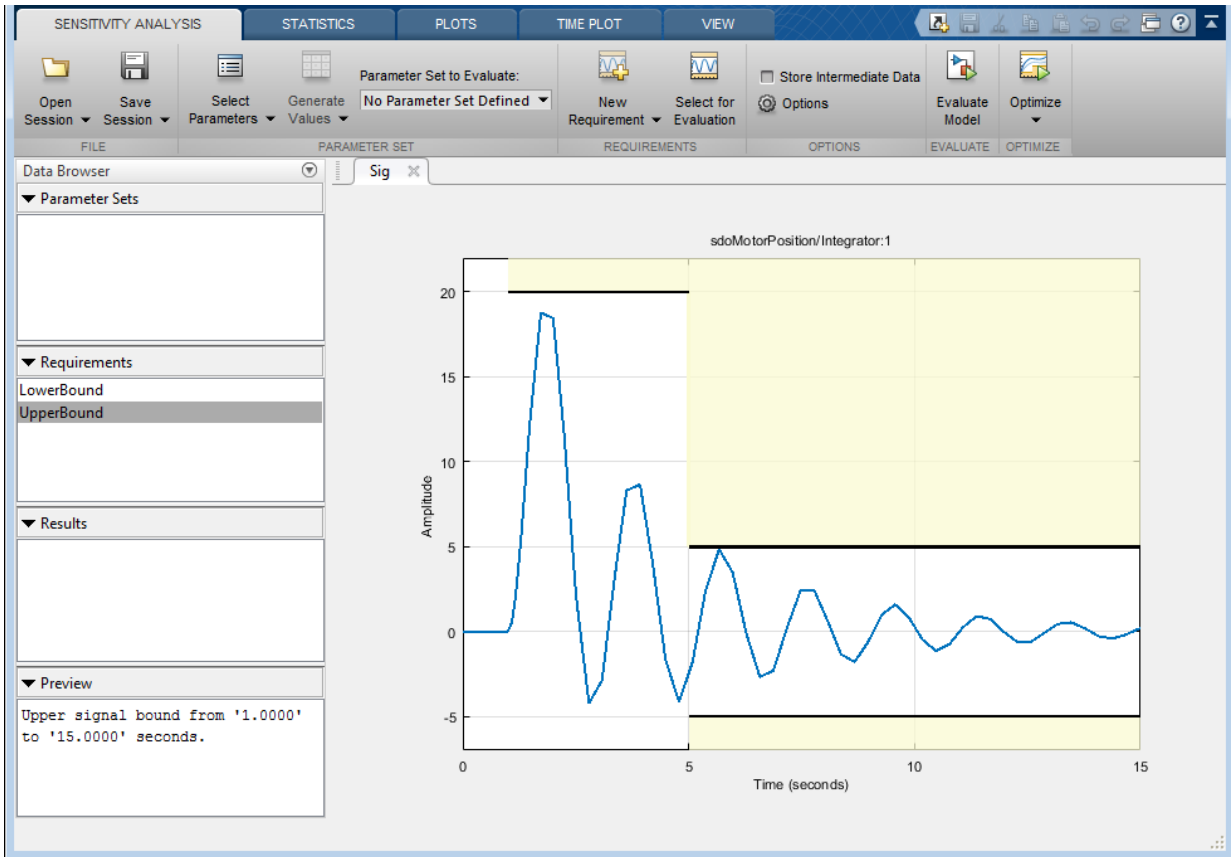
Design Requirements

The controller needs to maintain the motor at a reference position in the presence of disturbances. If a step disturbance occurs, the motor needs to deviate no more than 20 degrees, and needs to settle back to within 5 degrees of the reference position by 4 seconds after the disturbance.

Load previously specified design requirements for disturbance rejection. In the tool, click **Open Session** and select **Open from model workspace** in the drop-down menu.

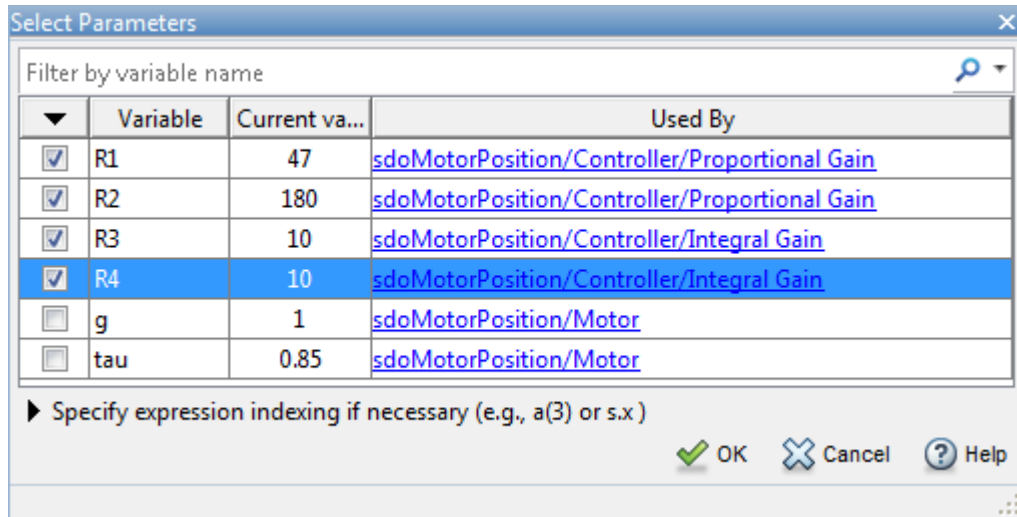


You can plot the requirements and verify that they are met when the resistances have the nominal values. In the **Results** area in the data browser, right-click on the LowerBound requirement, and select **Plot and Simulate**. Do the same for the UpperBound requirement.



Parameter Sampling

The motor position satisfies the disturbance rejection requirements, when the resistances are at their nominal values. However in practice, the actual resistor values will differ from the nominal ones, and we need to determine whether the controller will still meet the requirements. Click **Select Parameters** and make a new parameter set. This creates ParamSet in the **Parameter Sets** area of the tool. Specify that R1, R2, R3, and R4 are in the parameter set, and click **OK**.



Click **Generate Values** and generate random values. For repeatable results, reset the state of the random number generator in MATLAB®.

```
rng('default')
```

In the Generate Random Parameters dialog box, specify 500 samples to generate.

Specify the probability distribution for each parameter. Standard precision resistors match their nominal component value within a tolerance of 5%. This could be modeled using a uniform probability distribution. However, because resistors that measure within 1% of the nominal value are separated out and sold as higher-priced precision resistors, the 5% resistors can be more accurately modeled by a probability distribution with a well that excludes values within 1% of nominal. This can be modeled using a piecewise linear probability distribution if Statistics and Machine Learning Toolbox™ is available.

Specify the distribution of R1 as piecewise linear with 4 points. Specify the x values as [0.95 0.99 1.01 1.05] times 47 (the nominal value of the resistor). Specify the Fx values as [0 0.5 0.5 1]; these are the values of the cumulative distribution function corresponding to each x value. Similarly, set the distributions of R2, R3 and R4 to piecewise linear with 4 points, the x values as [0.95 0.99 1.01 1.05] times the nominal values (180, 10, and 10, respectively), and the Fx values as [0 0.5 0.5 1].

Number of Samples:

Overwrite previous values in parameter set when generating new values
 Append to previous values in parameter set when generating new values

Sampling Method:

Parameter	Distribution	# points:	x:	Fx:	Cross-Correlated
R1	Piecewise Linear	<input type="text" value="4"/>	x: [44.65 46.53...]	Fx: [0 0.5 0.5 1]	<input type="checkbox"/>
R2	Piecewise Linear	<input type="text" value="4"/>	x: [171 178.21...]	Fx: [0 0.5 0.5 1]	<input type="checkbox"/>
R3	Piecewise Linear	<input type="text" value="4"/>	x: [9.5 9.9 10.1...]	Fx: [0 0.5 0.5 1]	<input type="checkbox"/>
R4	Piecewise Linear	<input type="text" value="4"/>	x: [9.5 9.9 10.1...]	Fx: [0 0.5 0.5 1]	<input checked="" type="checkbox"/>

Distribution Plot Correlation Matrix

Probability Distribution for R4

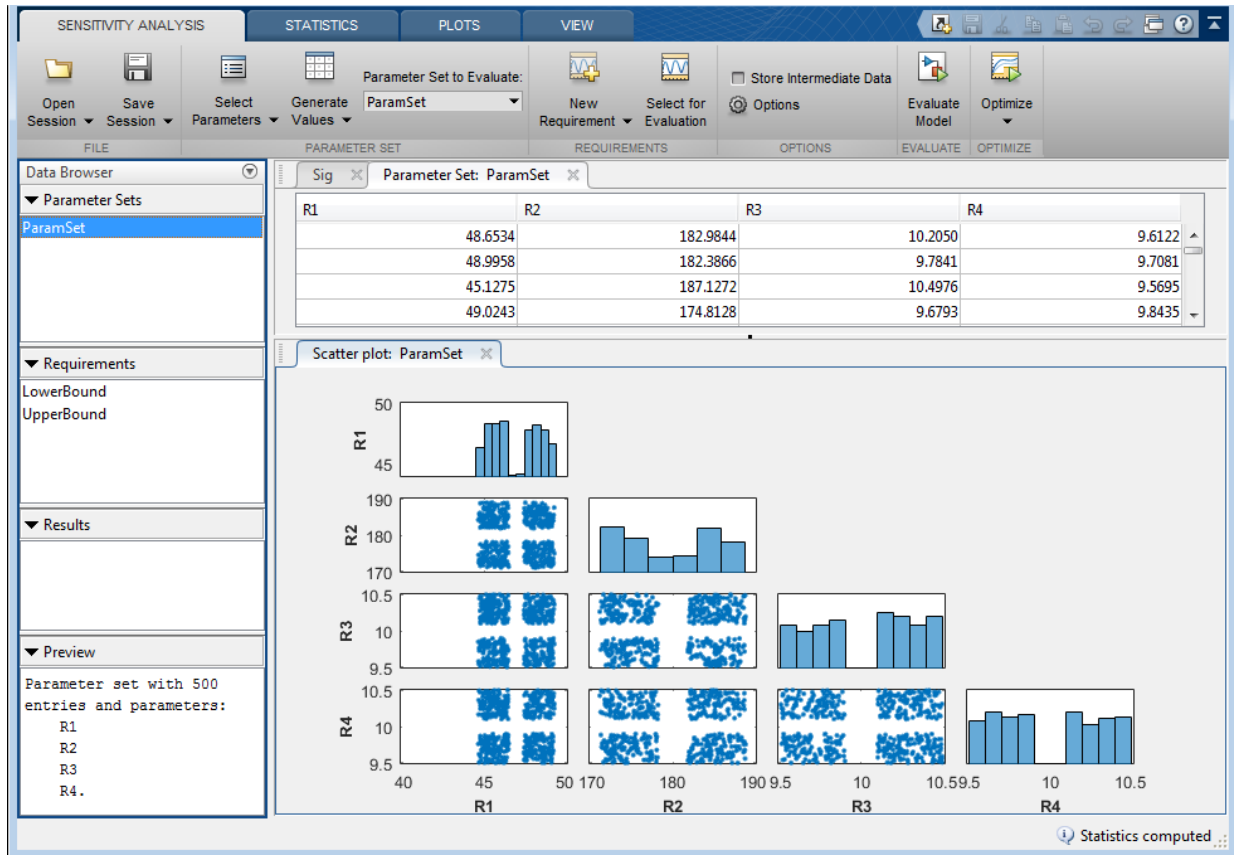
Mean: 10
Standard Deviation: 0.321455

500 values generated

Click **OK** to generate parameter values. The generated values are stored in the `ParamSet` variable in the **Parameter Set** area of the tool. (Note that due to the random number generator, the specific values in the table below may differ from what you get when running the example.)

To plot the parameter set click `ParamSet` in the **Parameter Sets** area of the tool browser. In the **Plots** tab, select **Scatter Plot** in the plot gallery. The plot shows the histogram of the generated parameters on the diagonal and pair-wise parameter scatter plots off the diagonal. Each marker on the plot represents one row of the `ParamSet`

table, with each row being simultaneously displayed on all the scatter plots. You can use the **View** tab to arrange the layout of the table and plot so they are both visible.



Evaluate Requirements with 5% Components

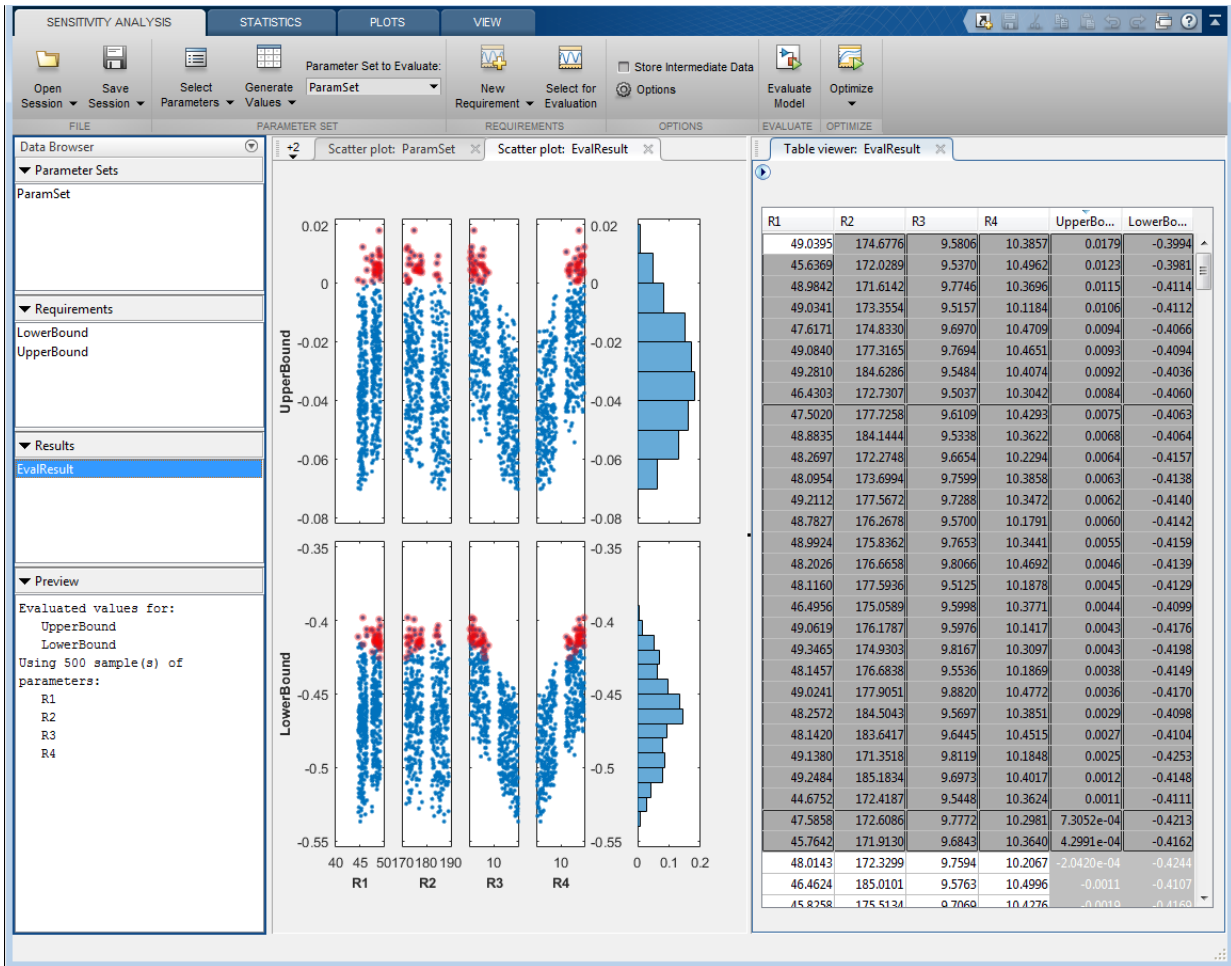
Evaluate the requirements for each row of parameter values in the table to see if the requirements are satisfied. In the **Sensitivity Analysis** tab, click **Select for Evaluation**. By default, all requirements are selected to be evaluated. Click **Evaluate Model** to evaluate the UpperBound and LowerBound requirements for each row of parameter values in ParamSet. Note you can speed up evaluation by using parallel computing if you have the Parallel Computing Toolbox™, or by using fast restart. For more information, see "Use Parallel Computing for Sensitivity Analysis" and "Use Fast

Restart Mode During Sensitivity Analysis" in the Simulink Design Optimization™ documentation.

A results scatter plot showing each requirement vs. each parameter is updated during model evaluation. At the end of evaluation a table with the evaluation results is shown. Each row in the evaluation result table contains values for R1, R2, R3, R4 and the resulting requirement values `UpperBound` and `LowerBound`. The evaluation results are stored in the `EvalResult` variable in the **Results** area of the tool. You can use the **View** tab to arrange the layout of the table and plot so they are both visible.

You can sort the evaluation results table by clicking on the column headers in the table. The `LowerBound` requirement is still met, as indicated by the fact that all evaluation results for the signal bound requirement are negative. That is not the case for the `UpperBound` requirement, which has several positive values. By selecting the rows of the table with these positive values, you can also see the corresponding points highlighted in the scatter plot.

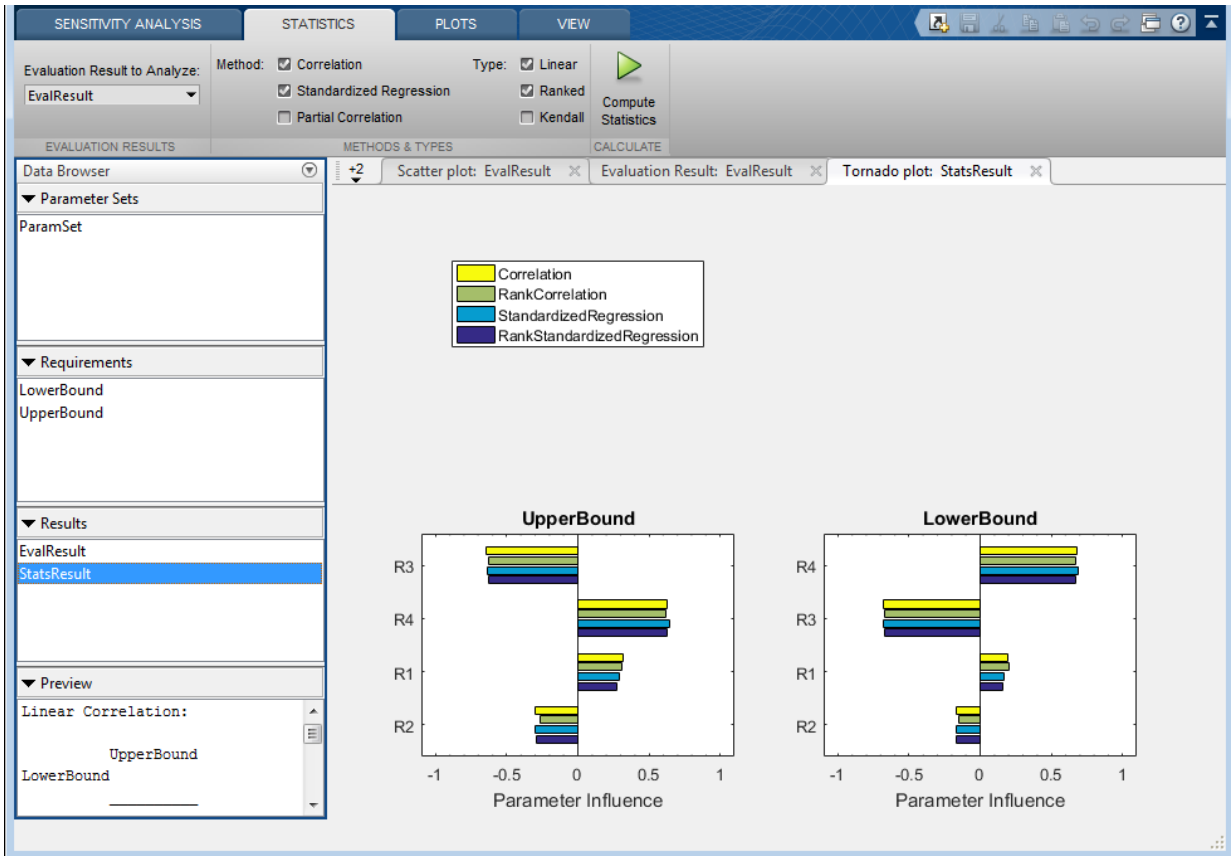
4 Sensitivity Analysis



Analyze the Evaluation Results

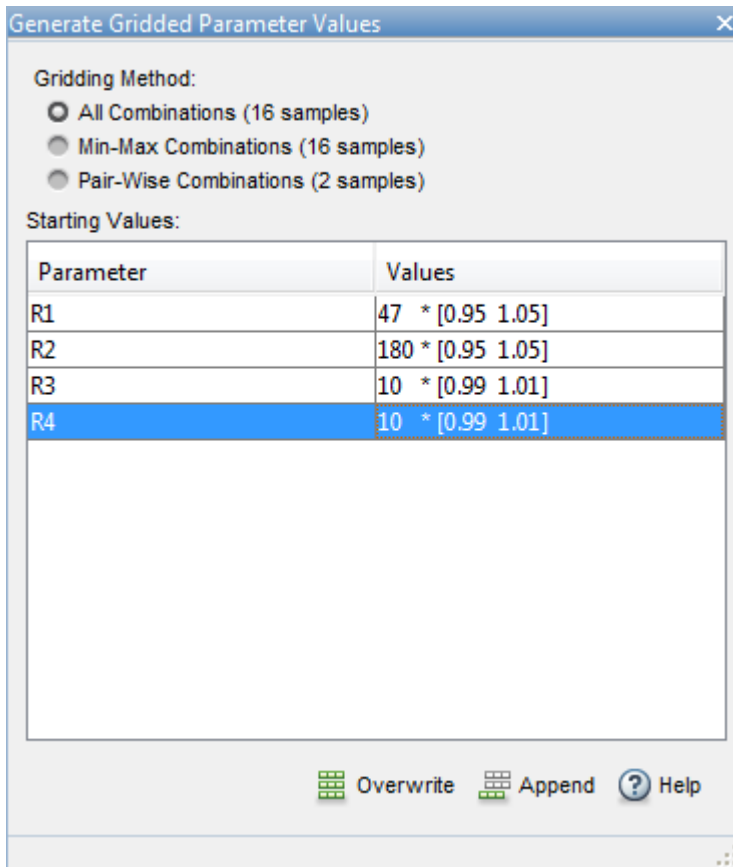
Using 5% tolerance components resulted in violation of the UpperBound requirement. Precision components with 1% tolerance would satisfy the design requirements, but they are more costly, so it is desirable to use only as many as necessary. You can use statistical analysis to identify the components that most influence the design requirements.

In the **Statistics** tab, select a variety of analyses to be done, including **Correlation** and **Standardized Regression** methods, and **Linear** and **Ranked** types of processing. Click **Compute Statistics**. The analysis result is stored in `StatsResult` in the **Results** area of the tool, and a tornado plot shows the analysis results. For each requirement, the tornado plot shows the most influential parameters at the top, and the others in decreasing order of the magnitude of their influence on the requirement. For the `UpperBound` requirement, R3 and R4 have the most influence, so we will try replacing these by higher precision 1% components.

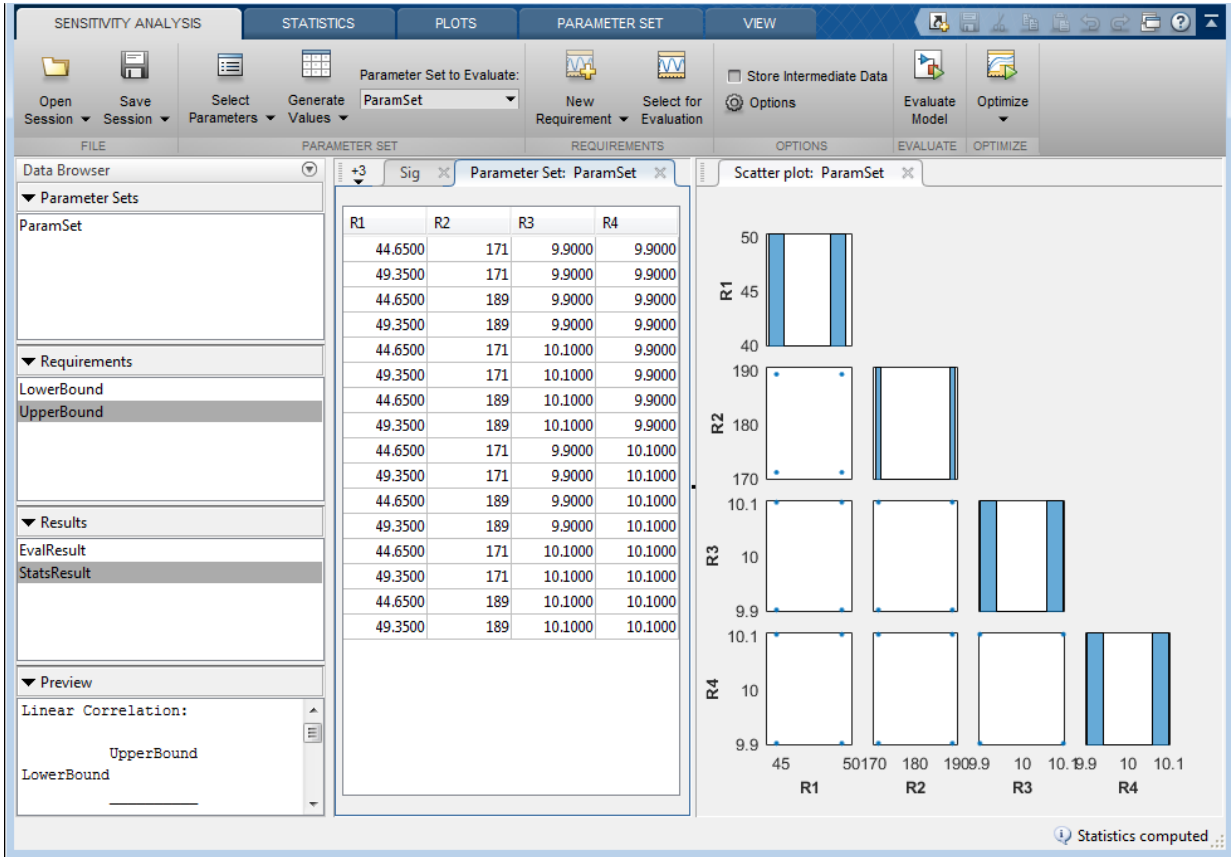


Evaluate Requirements with Mixed Components

Explore the use of 1% component tolerances only for resistors R3 and R4. In the **Sensitivity Analysis** tab, click **Generate Values** and generate gridded values. For R1 and R2, specify that the nominal value is to be perturbed by plus-and-minus 5%. For R3 and R4, specify that the nominal value is to be perturbed by plus-and-minus 1%.

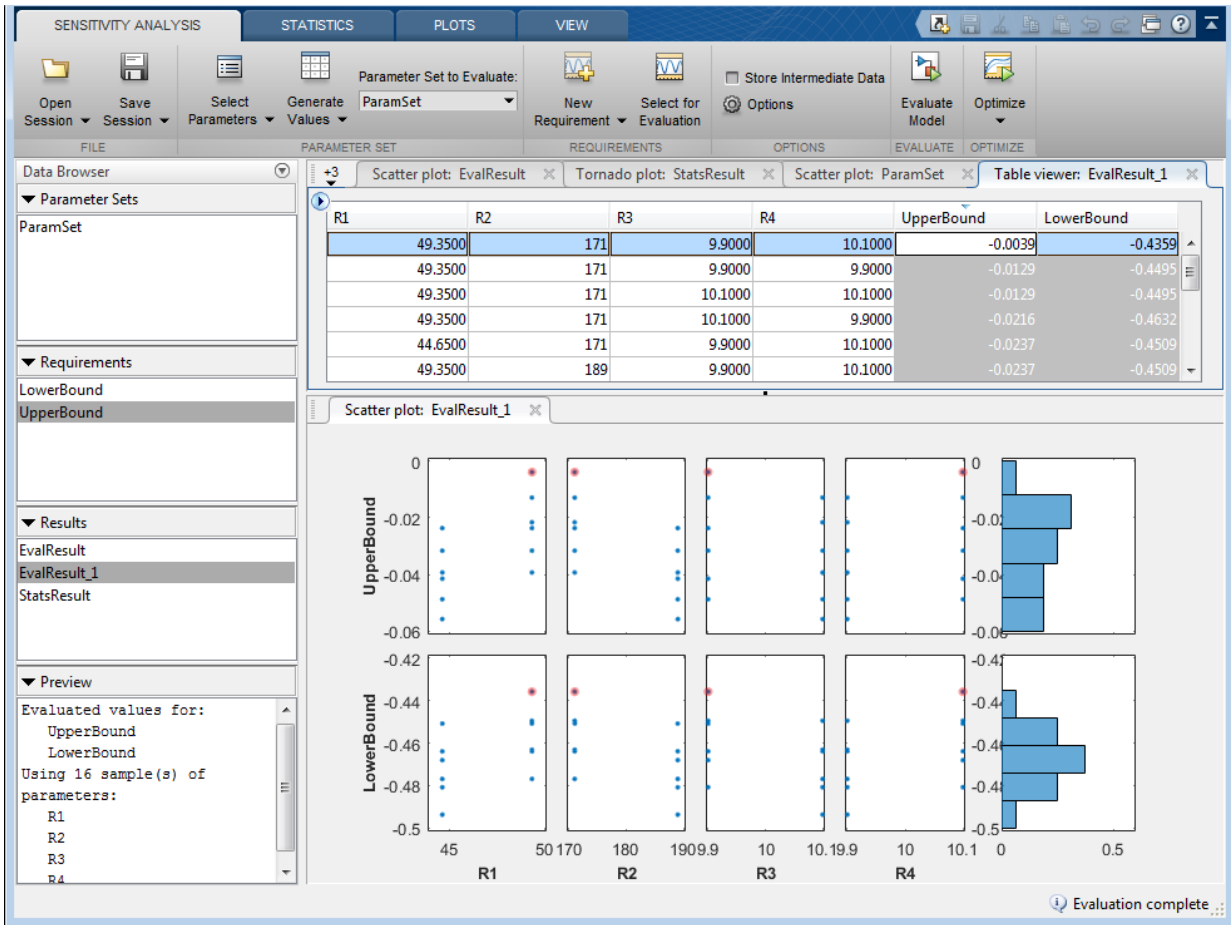


Click **Overwrite** to generate the new parameter values. To plot the parameter set click ParamSet in the **Parameter Sets** area of the tool browser. In the **Plots** tab, select **Scatter Plot** in the plot gallery.



In the **Sensitivity Analysis** tab, click **Evaluate Model**. The requirements are evaluated for each row in the table of parameter values, and results are stored in EvalResults_1 shown in the **Results** area of the tool. The evaluation results scatter plot and the evaluation results table show that both requirements are met for all combinations of component values.

4 Sensitivity Analysis



The Sensitivity Analysis tool was used to explore the effect of standard precision components on the design requirements of a PI controller. With standard precision components, some requirements were found to be violated. Statistical analysis was used to identify which parameters most influence the requirements. The analysis resulted in replacement of only two of the four components with most costly high-precision components.

Close the model.

```
bdclose('sdoMotorPosition')
```

See Also

Related Examples

- “Specify Parameters for Design Exploration” on page 4-4
- “Generate Parameter Samples for Sensitivity Analysis” on page 4-10
- “Analyze Relation Between Parameters and Design Requirements” on page 4-91
- “Validate Sensitivity Analysis” on page 4-129
- “Interact with Plots in the Sensitivity Analysis Tool” on page 4-106

Design Exploration Using Parameter Sampling (Code)

This example shows how to sample and explore a design space. You explore the design of a Continuously Stirred Tank Reactor to minimize product concentration variation and production cost. The design includes feed stock uncertainty.

You explore the CSTR design by characterizing design parameters using probability distributions. You use the distributions to generate random samples in the design space and perform Monte-Carlo evaluation of the design at these sample points. You then create plots to visualize the design space and select the best design. You can then use the best design as an initial guess for optimization of the design.

You can also use the sampled design space and Monte-Carlo evaluation output to analyze the influence of design parameters on the design, see "Sensitivity Analysis for Parameter Estimation (Code)"

Continuously Stirred Tank Reactor (CSTR) Model

Continuously Stirred Tank Reactors (CSTRs) are common in the process industry. The Simulink model, `sdoCSTR`, models a jacketed diabatic (i.e., non-adiabatic) tank reactor described in [1]. The CSTR is assumed to be perfectly mixed, with a single first-order exothermic and irreversible reaction, $A \rightarrow B$. A , the reactant, is converted to B , the product.

In this example, you use the following two-state CSTR model, which uses basic accounting and energy conservation principles:

$$\frac{dC_A}{dt} = \frac{F}{A * h} (C_{feed} - C_A) - r * C_A$$

$$\frac{dT}{dt} = \frac{F}{A * h} (T_{feed} - T) - \frac{H}{c_p \rho} r - \frac{U}{c_p * \rho * h} (T - T_{cool})$$

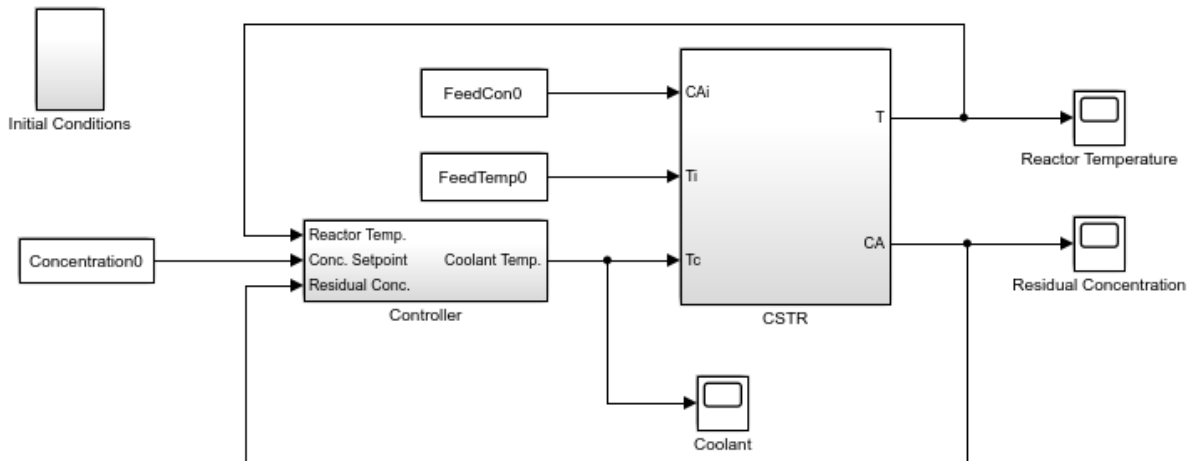
$$r = k_0 * e^{\frac{-E}{RT}}$$

- C_A , and C_{feed} - Concentrations of A in the CSTR and in the feed [kgmol/m³]
- T , T_{feed} , and T_{cool} - CSTR, feed, and coolant temperatures [K]
- F and ρ - Volumetric flow rate [m³/h] and the density of the material in the CSTR [1/m³]

- h and A - Height [m] and heated cross-sectional area [m²] of the CSTR.
- k_0 - Pre-exponential non-thermal factor for reaction $A \rightarrow B$ [1/h]
- E and H - Activation energy and heat of reaction for $A \rightarrow B$ [kcal/kgmol]
- R - Boltzmann's gas constant [kcal/(kgmol * K)]
- c_p and U - Heat capacity [kcal/K] and heat transfer coefficients [kcal/(m² * K * h)]

Open the Simulink model.

```
open_system('sdoCSTR');
```



Copyright 2012-2015 The MathWorks, Inc.

CSTR Design Problem

Assume that the CSTR is cylindrical, with the coolant applied to the base of the cylinder. Tune the CSTR cross-sectional area, A , and CSTR height, h , to meet the following design goals:

- Minimize the variation in residual concentration, C_A . Variations in the residual concentration negatively affect the quality of the CSTR product. Minimizing the variations also improves CSTR profit.

- Minimize the mean coolant temperature T_{cool} . Heating or cooling the jacket coolant temperature is expensive. Minimizing the mean coolant temperature improves CSTR profit.

The design must allow for variations in the quality of supply feed concentration, C_{feed} , and feed temperature, T_{feed} . The CSTR is fed with feed from different suppliers. The quality of the feed differs from supplier to supplier and also varies within each supply batch.

Specify Design Variables

Select the following model parameters as design variables:

- Cylinder cross-sectional area A
- Cylinder height h

```
p = sdo.getParameterFromModel('sdoCSTR', {'A', 'h'});
```

Limit the cross-sectional area to a range of [0.2 2] m².

```
p(1).Minimum = 0.2;  
p(1).Maximum = 2;
```

Limit the height to a range of [0.5 3] m.

```
p(2).Minimum = 0.5;  
p(2).Maximum = 3;
```

Sample the Design Space

Create a parameter space for the design variables. The parameter space characterizes the allowable parameter values and combinations of parameter values.

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

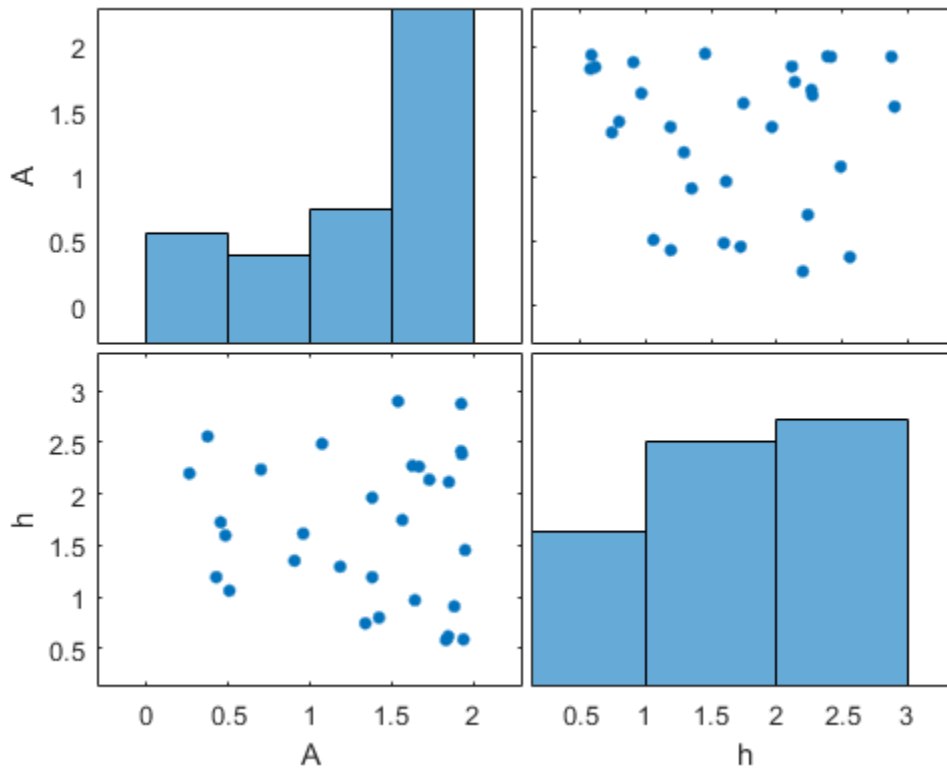
The parameter space uses default uniform distributions for the design variables. The distribution lower and upper bounds are set to the design variable minimum and maximum value respectively.

Use the `sdo.sample` function to generate samples from the parameter space. You use the samples to evaluate the model and explore the design space.


```
rng default; % For reproducibility
pSmpl = sdo.sample(pSpace,30);
```

Use the `sdo.scatterPlot` command to visualize the sampled parameter space. The scatter plot shows the parameter distributions on the diagonal subplots and pairwise parameter combinations on the off diagonal subplots.

```
figure, sdo.scatterPlot(pSmpl)
```



Specify Uncertain Variables

Select the feed concentration and feed temperature as uncertain variables. You evaluate the design using different values of feed temperature and concentration.

```
pUnc = sdo.getParameterFromModel('sdoCSTR',{'FeedCon0','FeedTemp0'});
```

Create a parameter space for the uncertain variables. Use normal distributions for both variables. Specify the mean as the current parameter value. Specify a variance of 5% of the mean for the feed concentration and 1% of the mean for the temperature.

```
uSpace = sdo.ParameterSpace(pUnc);  
uSpace = setDistribution(uSpace, 'FeedCon0', makedist('normal', pUnc(1).Value, 0.05*pUnc(1).Value));  
uSpace = setDistribution(uSpace, 'FeedTemp0', makedist('normal', pUnc(2).Value, 0.01*pUnc(2).Value));
```

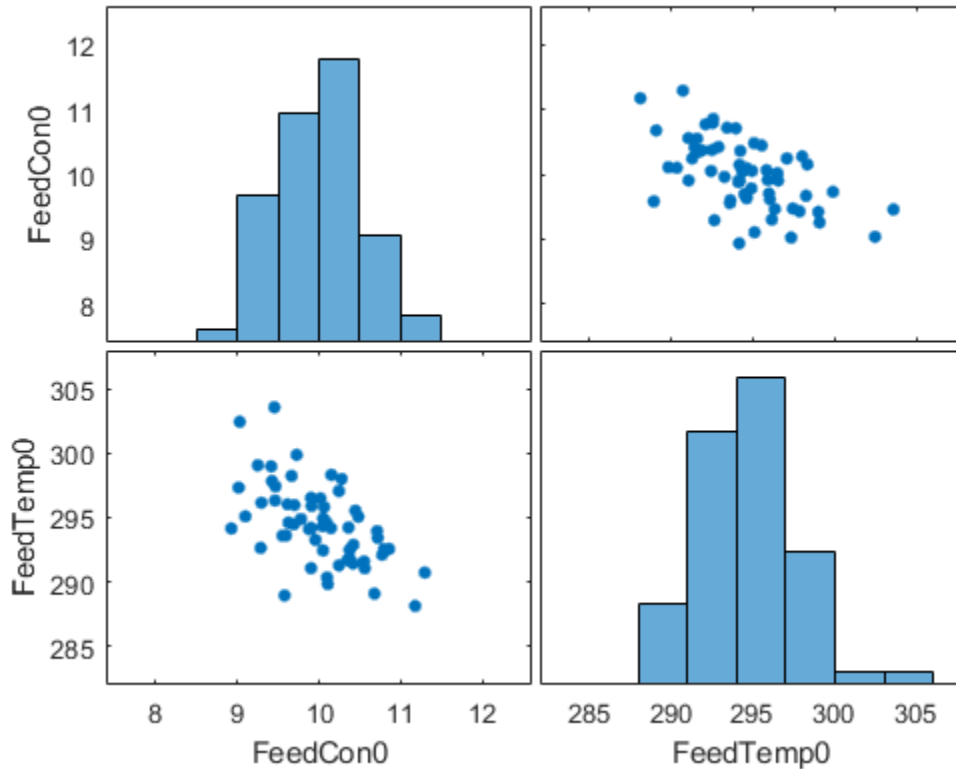
The feed concentration is inversely correlated with the feed temperature. Add this information to the parameter space.

```
uSpace.RankCorrelation = [1 -0.6; -0.6 1];
```

The rank correlation matrix has a row and column for each parameter with the (i,j) entry specifying the correlation between the i and j parameters.

Sample the parameter space. The scatter plot shows the correlation between concentration and temperature.

```
uSmpl = sdo.sample(uSpace, 60);  
sdo.scatterPlot(uSmpl)
```



Ideally you want to evaluate the design for every combination of points in the design and uncertain spaces, which implies $30 \times 60 = 1800$ simulations. Each simulation takes around 0.5 sec. You can use parallel computing to speed up the evaluation. For this example you instead only use the samples that have maximum & minimum concentration and temperature values, reducing the evaluation time to around 1 min.

```
[~,iminC] = min(uSmpl.FeedCon0);
[~,imaxC] = max(uSmpl.FeedCon0);
[~,iminT] = min(uSmpl.FeedTemp0);
[~,imaxT] = max(uSmpl.FeedTemp0);
uSmpl = uSmpl(unique([iminC,imaxC,iminT,imaxT]) , :)
```

```
uSmpl =
```

4x2 table

FeedCon0	FeedTemp0
9.4555	303.58
11.175	288.13
11.293	290.73
8.9308	294.16

Create Evaluation Function

Create a function that evaluates the design for a given sample point in the design space. The design is evaluated on how well it minimizes the variation in residual concentration and mean coolant temperature.

Specify Design Requirements

Evaluating a point in the design space requires logging model signals. Logged signals are used to evaluate the design requirements.

Log the following signals:

- CSTR concentration, available at the second output port of the `sdoCSTR/CSTR` block

```
Conc = Simulink.SimulationData.SignalLoggingInfo;  
Conc.BlockPath      = 'sdoCSTR/CSTR';  
Conc.OutputPortIndex = 2;  
Conc.LoggingInfo.NameMode = 1;  
Conc.LoggingInfo.LoggingName = 'Concentration';
```

- Coolant temperature, available at the first output of the `sdoCSTR/Controller` block

```
Coolant = Simulink.SimulationData.SignalLoggingInfo;  
Coolant.BlockPath      = 'sdoCSTR/Controller';  
Coolant.OutputPortIndex = 1;  
Coolant.LoggingInfo.NameMode = 1;  
Coolant.LoggingInfo.LoggingName = 'Coolant';
```

Create and configure a simulation test object to log the required signals.

```
simulator = sdo.SimulationTest('sdoCSTR');  
simulator.LoggingInfo.Signals = [Conc,Coolant];
```

Evaluation Function

Use an anonymous function with one argument that calls the `sdoCSTR_design` function.

```
evalDesign = @(p) sdoCSTR_design(p, simulator, pUnc, uSmpl);
```

The `evalDesign` function:

- Has one input argument that specifies the CSTR dimensions
- Returns the optimization objective value

The `sdoCSTR_design` function uses a `for` loop that iterates through the sample values specified for the feed concentration and temperature. Within the loop, the function:

- Simulates the model using the current design point, feed concentration, and feed temperature values
- Calculates the residual concentration variation and coolant temperature costs

To view the objective function, type `edit sdoCSTR_design`.

```
type sdoCSTR_design
```

```
function design = sdoCSTR_design(p, simulator, pUnc, smplUnc)
%SDOCSTR_DESIGN
%
% The sdoCSTR_design function is used to evaluate a CSTR design.
%
% The |p| input argument is the vector of CSTR dimensions.
%
% The |simulator| input argument is a sdo.SimulinkTest object used to
% simulate the |sdoCSTR| model and log simulation signals.
%
% The |pUnc| input argument is a vector of parameters to specify the CSTR
% input feed concentration and feed temperature. The |smplUnc| argument is
% a table of different feed concentration and temperature values.
%
% The |design| return argument contains information about the design
% evaluation that can be used by the |sdo.optimize| function to optimize
% the design.
%
% see also sdo.optimize, sdoExampleCostFunction
%
```

```
% Copyright 2012-2013 The MathWorks, Inc.

%% Model Simulations and Evaluations
%
% For each value in |smpUnc|, configure and simulate the model. Use
% the logged concentration and coolant signals to compute the design cost.
%
costConc = 0;
costCoolant = 0;
for ct=1:size(smpUnc,1)
    %Set the feed concentration and temperature values
    pUnc(1).Value = smpUnc{ct,1};
    pUnc(2).Value = smpUnc{ct,2};

    %Simulate model
    simulator.Parameters = [p; pUnc];
    simulator = sim(simulator);
    logName = get_param('sdoCSTR','SignalLoggingName');
    simLog = get(simulator.LoggedData,logName);

    %Compute Concentration cost based on the standard deviation of the
    %concentration from a nominal value.
    Sig = find(simLog,'Concentration');
    costConc = costConc+10*std(Sig.Values-2);

    %Compute coolant cost based on the mean deviation from room
    %temperature.
    Sig = find(simLog,'Coolant');
    costCoolant = costCoolant+abs(mean(Sig.Values - 294))/30;
end

%% Return Total Cost
%
% Compute the total cost as a sum of the concentration and coolant costs.
%
design.F = costConc + costCoolant;

%%
% Add the individual cost terms to the return argument. These are not used
% by the optimizer, but included for convenience.
design.costConc = costConc;
design.costCoolant = costCoolant;
end
```

Evaluate

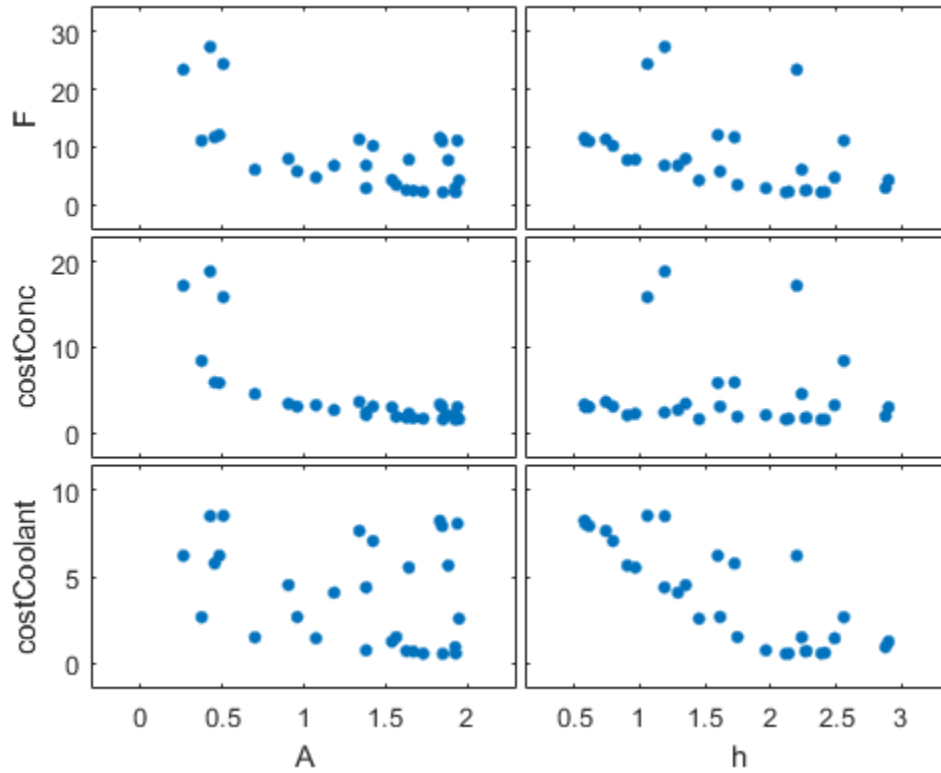
Use the `sdo.evaluate` command to evaluate the model at the sample design points.

```
y = sdo.evaluate(evalDesign,p,pSmpl);
```

```
Model evaluated at 30 samples.
```

View the results of the evaluation using a scatter plot. The scatter plot shows pairwise plots for each design variable (A,h) and design cost. The plot include the total cost, F, as well as coolant and concentration costs, `costCoolant` and `costConc` respectively.

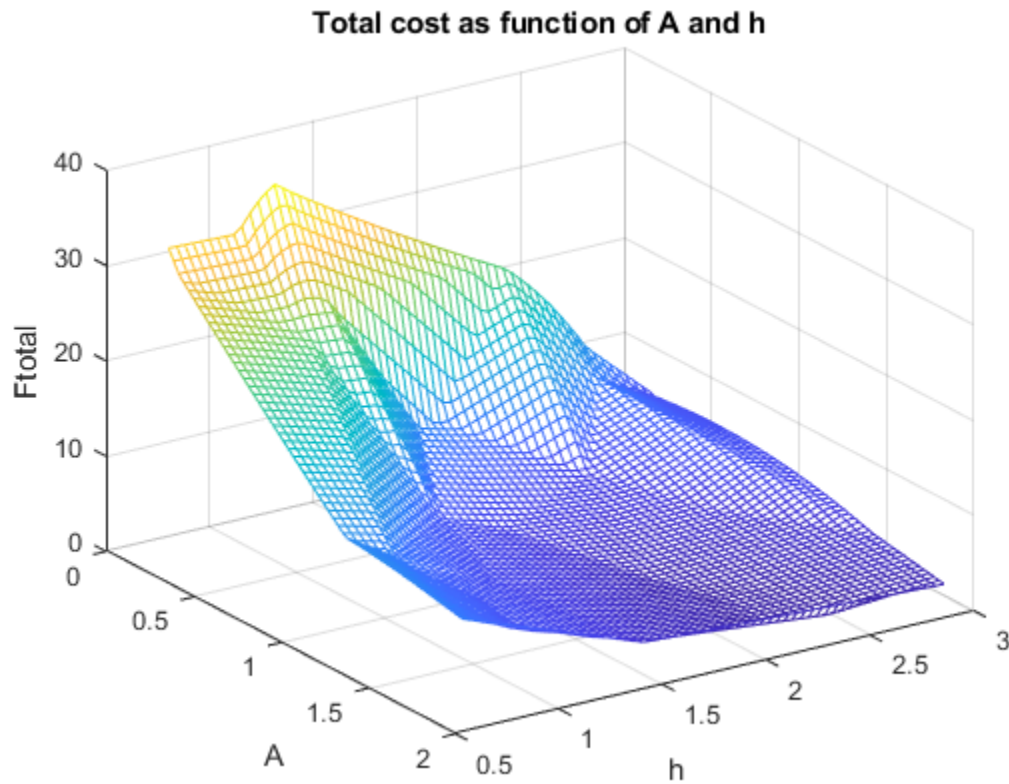
```
sdo.scatterPlot(pSmpl,y);
```



The plot shows that larger cross-sectional areas result in lower total costs. However it is difficult to tell how the height influences the total cost.

Create a mesh plot showing the total cost as a function of A and h.

```
Ftotal = scatteredInterpolant(pSmpl.A,pSmpl.h,y.F);
xR = linspace(min(pSmpl.A),max(pSmpl.A),60);
yR = linspace(min(pSmpl.h),max(pSmpl.h),60);
[xx,yy] = meshgrid(xR,yR);
zz = Ftotal(xx,yy);
mesh(xx,yy,zz)
view(56,30)
title('Total cost as function of A and h')
zlabel('Ftotal')
xlabel(p(1).Name), ylabel(p(2).Name);
```

The plot shows that high values of A and h result in lower costs. The best design in the sampled space corresponds to the design with the lowest cost value.

```
[~,idx] = min(y.F);
pBest = [y(idx,:), pSmpl(idx,:)]
```

```
pBest =
```

```
1x5 table
```

F	costConc	costCoolant	A	h
---	----------	-------------	---	---

2.1052 1.5757 0.52953 1.8483 2.1158

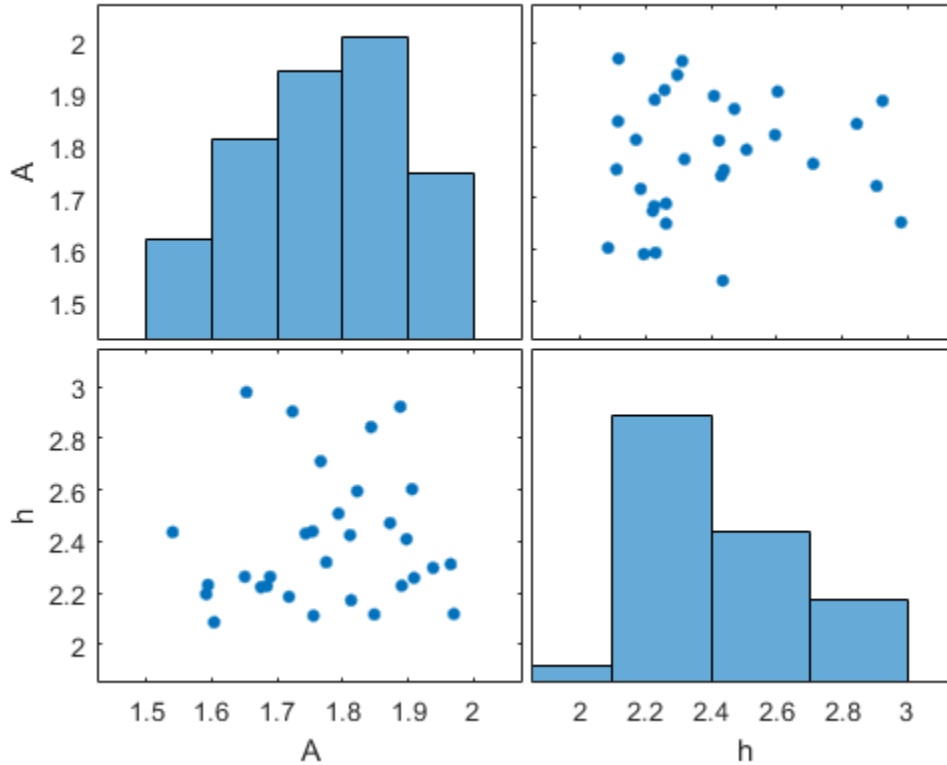
Refine the Design Space

The total cost surface plot shows that low cost designs are designs with A in the range [1.5 2] and h in the range [2 3]. Modify the parameter space distributions for A and h and resample the design space to focus on this region.

```
pSpace = setDistribution(pSpace, 'A', makedist('uniform', 1.5, 2));  
pSpace = setDistribution(pSpace, 'h', makedist('uniform', 2, 3));  
pSmpl = sdo.sample(pSpace, 30);
```

Add the pBest found earlier to the new samples so that it is part of the refined design space.

```
pSmpl = [pSmpl; pBest(:, 4:5)];  
sdo.scatterPlot(pSmpl)
```



Evaluate using Refined Design Space

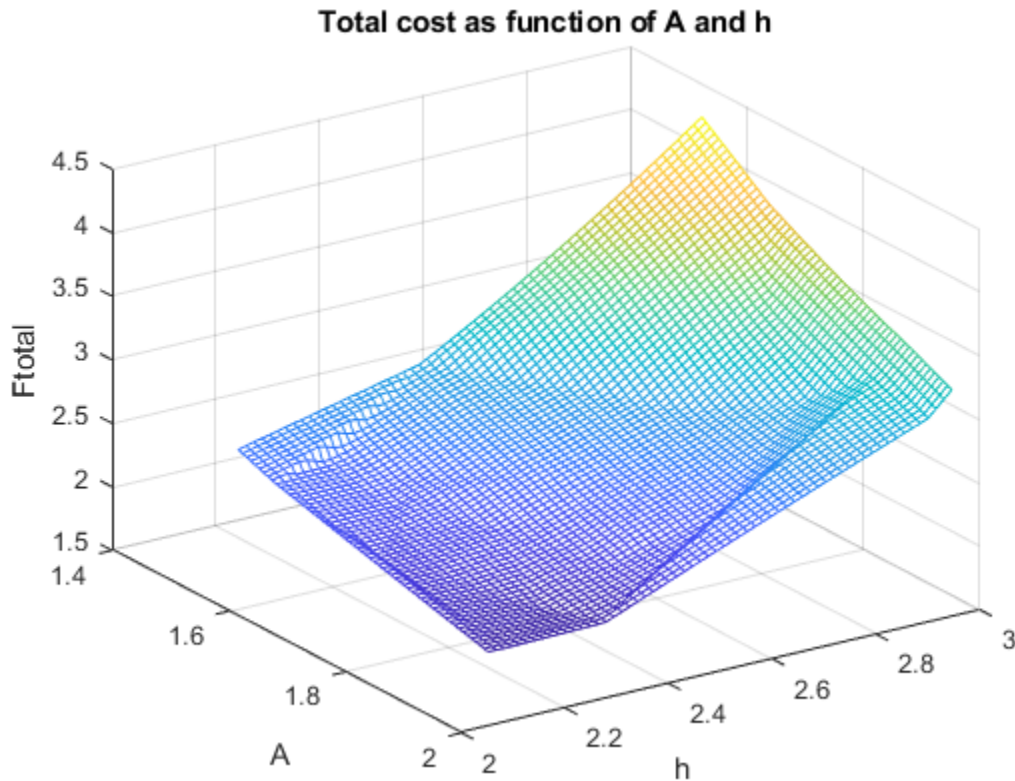
```
y = sdo.evaluate(evalDesign,p,pSmpl);
```

Model evaluated at 31 samples.

Create a mesh plot for this section of the design space. The surface indicates that better designs are near the $A = 1.9$, $h = 2.1$ point.

```
Ftotal = scatteredInterpolant(pSmpl.A,pSmpl.h,y.F);
xR = linspace(min(pSmpl.A),max(pSmpl.A),60);
yR = linspace(min(pSmpl.h),max(pSmpl.h),60);
[xx,yy] = meshgrid(xR,yR);
zz = Ftotal(xx,yy);
```

```
mesh(xx,yy,zz)
view(56,30)
title('Total cost as function of A and h')
zlabel('Ftotal')
xlabel(p(1).Name), ylabel(p(2).Name);
```



Find the best design from the new design space and compare with the best design point found earlier.

```
[~,idx] = min(y.F);
pBest = [pBest; [y(idx,:), pSmpl(idx,:)]]
```

```
pBest =
```

2x5 table

F	costConc	costCoolant	A	h
2.1052	1.5757	0.52953	1.8483	2.1158
1.979	1.4838	0.49528	1.9695	2.1174

The best design in the refined design space is better than the design found earlier. This indicates that there may be better designs in the same region and warrants refining the design space further. Alternatively you can use the best design point as an initial guess for optimization.

Related Examples

To learn how to explore the CSTR design space using the Sensitivity Analysis Tool, see "Design Exploration using Parameter Sampling (GUI)".

To learn how to optimize the CSTR design using the `sdo.optimize` command, see "Design Optimization with Uncertain Variables (Code)".

To learn how to analyze the influence of design parameters on the design using the `sdo.analyze` command, see "Sensitivity Analysis for Parameter Estimation (Code)".

References

[1] Bequette, B.W. *Process Dynamics: Modeling, Analysis and Simulation*. 1st ed. Upper Saddle River, NJ: Prentice Hall, 1998.

Close the model

```
bdclose('sdoCSTR')
```

See Also

More About

- "Design Exploration Using Parameter Sampling (GUI)" on page 4-147
- "Use Parallel Computing for Sensitivity Analysis" on page 4-137

- “Use Fast Restart Mode During Sensitivity Analysis” on page 4-144
- “Generate Parameter Samples for Sensitivity Analysis” on page 4-10
- “Analyze Relation Between Parameters and Design Requirements” on page 4-91
- “Validate Sensitivity Analysis” on page 4-129

Identify Key Parameters for Estimation (Code)

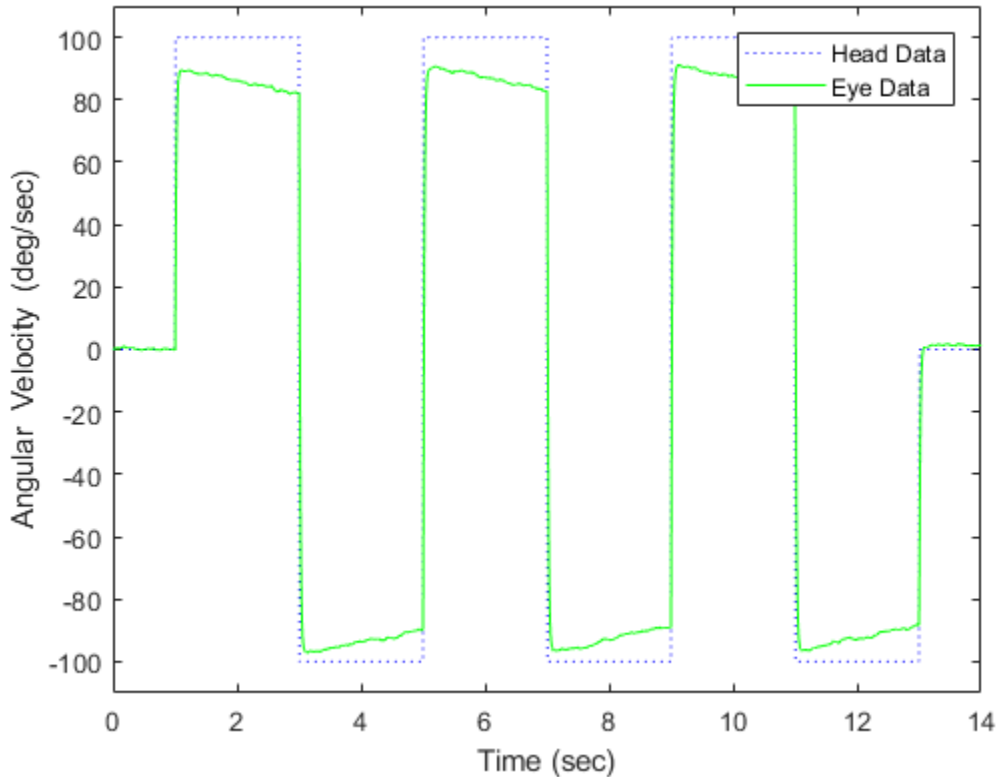
This example shows how to use sensitivity analysis to narrow down the number of parameters that you need to estimate to fit a model. This example uses a model of the vestibulo-ocular reflex, which generates compensatory eye movements.

Model Description

The vestibulo-ocular reflex (VOR) enables the eyes to move at the same speed and in the opposite direction as the head, so that vision is not blurred when the head moves during normal activity. For example, if the head turns in one direction, the eyes turn in the opposite direction, with the same speed. This happens even in the dark. In fact, the VOR is most easily characterized by measurements in the dark, to ensure that eye movements are predominantly driven by the VOR.

The file `sdoVOR_Data.mat` contains uniformly sampled data of stimulation and eye movements. If the VOR were perfectly compensatory, then a plot of eye movement data, when flipped vertically, would overlay exactly on top of a plot of head motion data. Such a system would be described by a gain of 1 and a phase of 180 degrees. However, when we plot the data in the file `sdoVOR_Data.mat`, the eye movements are close, but not perfectly compensatory.

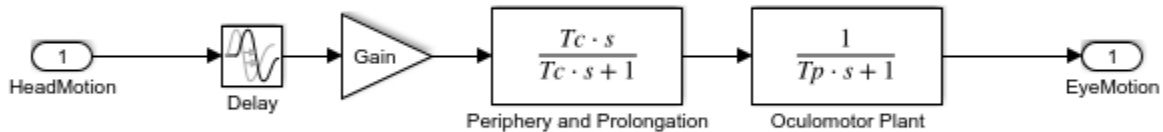
```
load sdoVOR_Data.mat; % Column vectors: Time HeadData EyeData
figure
plot(Time, HeadData, ':b', Time, EyeData, '-g')
xlabel('Time (sec)')
ylabel('Angular Velocity (deg/sec)')
ylim([-110 110])
legend('Head Data', 'Eye Data')
```



The eye movement data does not perfectly overlay the head motion data, and this can be modeled by several factors. Head rotation is sensed by organs in the inner ears, known as semicircular canals. These detect head motion and transmit signals about head motion to the brain, which sends motor commands to the eye muscles, so that eye movements compensate for head motion. We would like to use this eye movement data to estimate the parameters in the models for these various stages. The model we will use is shown below. There are four parameters in the model: Delay, Gain, T_c , and T_p .

```
model_name = 'sdoVOR';  
open_system(model_name)
```


Vestibulo-Ocular Reflex



Copyright 2013 The MathWorks, Inc.

The Delay parameter models the fact that there is some delay in communicating the signals from the inner ear to the brain and the eyes. This delay is due to the time needed for chemical neurotransmitters to traverse the synaptic clefts between nerve cells. Based on the number of synapses involved in the vestibulo-ocular reflex, this delay is expected to be around 5 ms. For estimation purposes, we will assume it is between 2 and 9 ms.

```
Delay = sdo.getParameterFromModel(model_name, 'Delay');
Delay.Value = 0.005; % seconds
Delay.Minimum = 0.002;
Delay.Maximum = 0.009;
```

The Gain parameter models the fact that the eyes do not move quite as much as the head does. We will use 0.8 as our initial guess, and assume it is between 0.6 and 1.

```
Gain = sdo.getParameterFromModel(model_name, 'Gain');
Gain.Value = 0.8;
Gain.Minimum = 0.6;
Gain.Maximum = 1;
```

The Tc parameter models the dynamics associated with the semicircular canals, as well as some additional neural processing. The canals are high-pass filters, because after a subject has been put into rotational motion, the neurally active membranes in the canals slowly relax back to resting position, so the canals stop sensing motion. Thus in the plot above, after the stimulation undergoes transition edges, the eye movements tend to depart from the stimulation over time. Based on mechanical characteristics of the canals, combined with additional neural processing which prolongs this time constant to improve

the accuracy of the VOR, we will estimate the Tc parameter to be 15 seconds, and assume it is between 10 and 30 seconds.

```
Tc = sdo.getParameterFromModel(model_name, 'Tc');
Tc.Value = 15;
Tc.Minimum = 10;
Tc.Maximum = 30;
```

Finally, the Tp parameter models the dynamics of the oculomotor plant, i.e. the eye and the muscles and tissues attached to it. The plant can be modeled by two poles, however it is believed that the pole with the larger time constant is cancelled by precompensation in the brain, to enable the eye to make quick movements. Thus in the plot, when the stimulation undergoes transition edges, the eye movements follow with only a little delay. For the Tp parameter, we will use 0.01 seconds as our initial guess, and assume it is between 0.005 and 0.05 seconds.

```
Tp = sdo.getParameterFromModel(model_name, 'Tp');
Tp.Value = 0.01;
Tp.Minimum = 0.005;
Tp.Maximum = 0.05;
```

Collect these parameters into a vector.

```
v = [Delay Gain Tc Tp];
```

Compare Measured Data to Initial Simulated Output

Create an Experiment object. Specify HeadData as input.

```
Exp = sdo.Experiment(model_name);
Exp.InputData = timeseries(HeadData, Time);
```

Associate eye movement data with model output.

```
EyeMotion = Simulink.SimulationData.Signal;
EyeMotion.Name = 'EyeMotion';
EyeMotion.BlockPath = [model_name '/Oculomotor Plant'];
EyeMotion.PortType = 'outport';
EyeMotion.PortIndex = 1;
EyeMotion.Values = timeseries(EyeData, Time);
```

Add EyeMotion to the experiment.

```
Exp.OutputData = EyeMotion;
```

Use the data's timing characteristics in the model.

```
stop_time = Time(end);
set_param(gcs, 'StopTime', num2str(stop_time));
dt = Time(2) - Time(1);
set_param(gcs, 'FixedStep', num2str(dt))
```

Create a simulation scenario using the experiment, and obtain the simulated output.

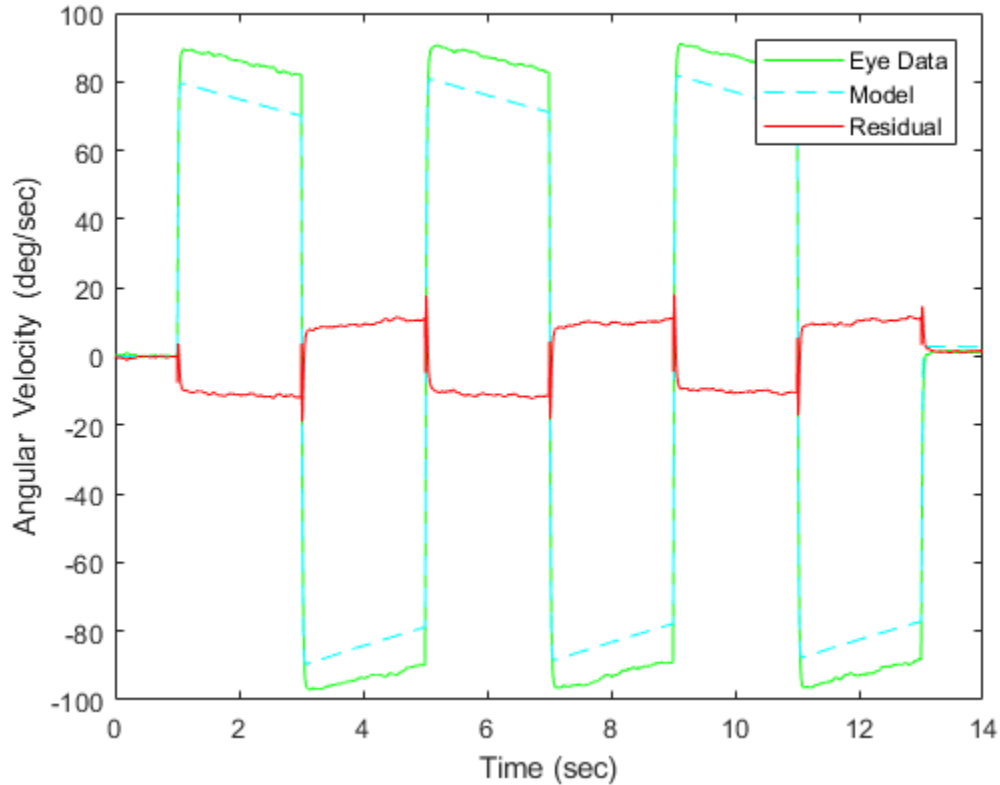
```
Exp = setEstimatedValues(Exp, v); % use vector of parameters/states
Simulator = createSimulator(Exp);
Simulator = sim(Simulator);
```

Search for the model_residual signal in the logged simulation data.

```
SimLog = find(Simulator.LoggedData, ...
    get_param(model_name, 'SignalLoggingName') );
EyeSignal = find(SimLog, 'EyeMotion');
```

The model output does not match the data very well, as shown by the residual, which we can compute by calling the objective function.

```
estFcn = @(v) sdoVOR_Objective(v, Simulator, Exp, 'Residuals');
Model_Error = estFcn(v);
plot(Time, EyeData, '-g', ...
    EyeSignal.Values.Time, EyeSignal.Values.Data, '--c', ...
    Time, Model_Error.F, '-r');
xlabel('Time (sec)');
ylabel('Angular Velocity (deg/sec)');
legend('Eye Data', 'Model', 'Residual');
```



The objective function used above is defined in the file "sdoVOR_Objective.m".

```
type sdoVOR_Objective.m
```

```
function vals = sdoVOR_Objective(v, Simulator, Exp, Method)
% Compare model output with data
%
% Inputs:
%   v - vector of parameters and/or states
%   Simulator - used to simulate the model
%   Exp - Experiment object
%   Method - 'SSE' for scalar output, 'Residuals' for vector of residuals
```

```

% Copyright 2014-2015 The MathWorks, Inc.

% Requirement setup
req = sdo.requirements.SignalTracking;
req.Type = '==';
req.Method = Method;

% If Residuals requested, keep on same scale as signals, for plotting
switch Method
    case 'Residuals'
        req.Normalize = 'off';
end

% Simulate the model
Exp = setEstimatedValues(Exp, v); % use vector of parameters/states
Simulator = createSimulator(Exp, Simulator);
Simulator = sim(Simulator);

% Compare model output with data
SimLog = find(Simulator.LoggedData, ...
    get_param(Exp.ModelName, 'SignalLoggingName') );
OutputModel = find(SimLog, 'EyeMotion');
Model_Error = evalRequirement(req, OutputModel.Values, Exp.OutputData.Values);
vals.F = Model_Error;

```

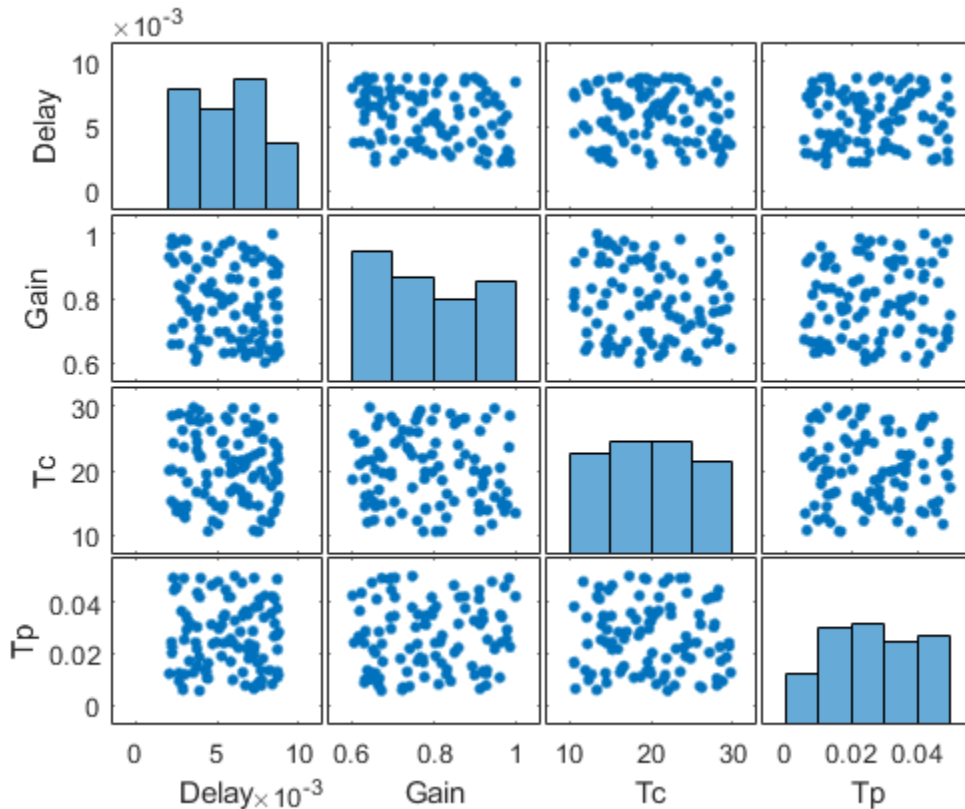
Sensitivity Analysis

Create an object to sample the parameter space.

```
ps = sdo.ParameterSpace([Delay ; Gain ; Tc ; Tp]);
```

Generate 100 samples from the parameter space.

```
rng default; % for reproducibility
x = sdo.sample(ps, 100);
sdo.scatterPlot(x);
```



The sampling above used default options, and these are reflected in the plots above. Parameter values were selected at random from distributions that were uniform over the range of each parameter. Consequently, the histogram plots along the diagonal appear approximately uniform. If Statistics and Machine Learning Toolbox™ is available, many other distributions may be used, and sampling can be done using Sobol or Halton low-discrepancy sequences.

The off-diagonal plots above show scatter plots between pairs of different variables. Since we did not specify a RankCorrelation matrix in `ps`, the scatter plots do not indicate correlations. However, if parameters were believed to be correlated, this can be specified using the RankCorrelation property of `ps`.

For sensitivity analysis, it is simpler to use a scalar objective, so we will specify the sum of squared errors, "SSE":

```
estFcn = @(v) sdoVOR_Objective(v, Simulator, Exp, 'SSE');
y = sdo.evaluate(estFcn, ps, x);
```

Model evaluated at 100 samples.

Evaluation could also be sped up using parallel computing.

Obtain the standardized regression coefficients.

```
opts = sdo.AnalyzeOptions;
opts.Method = 'StandardizedRegression';
sensitivities = sdo.analyze(x, y, opts);
```

Other types of analysis include correlation and, if Statistics and Machine Learning Toolbox is available, partial correlation.

We can view the analysis results.

```
disp(sensitivities)
```

	F
Delay	0.01303
Gain	-0.90873
Tc	-0.044395
Tp	0.19919

For standardized regression, parameters that highly influence the model output have sensitivity magnitudes close to 1. On the other hand, less influential parameters have smaller sensitivity magnitudes. We see that this objective function is sensitive to changes in the Gain and Tp parameters, but much less sensitive to changes in the Delay and Tc parameters.

You can validate sensitivity analysis results by resampling and reevaluating the objective function for the samples. You can also use engineering intuition for a quick analysis. For example, in this model, the time constant T_C ranges from 10 to 30 seconds. Even the minimum value of 10 seconds is large compared to the 2-second duration for which the head motion stimulation is held at constant velocity. Therefore, T_C is not expected to affect the output greatly. However, even when this kind of intuition is not

readily available in other models, sensitivity analysis can help highlight which parameters are influential.

Based on the results of sensitivity analysis, designate the `Delay` and `Tc` parameters as fixed when optimizing. This reduction in the number of free parameters speeds up optimization.

```
Delay.Free = false;  
Tc.Free = false;
```

Optimization

We can use the minimum from sensitivity analysis as the initial guess for optimization.

```
[fval, idx_min] = min(y.F);  
Delay.Value = x.Delay(idx_min);  
Gain.Value = x.Gain(idx_min);  
Tc.Value = x.Tc(idx_min);  
Tp.Value = x.Tp(idx_min);  
%  
v = [Delay Gain Tc Tp];  
opts = sdo.OptimizeOptions;  
opts.Method = 'fmincon';
```

As was the case with model evaluations in sensitivity analysis, parallel computing could be used to speed up the optimization.

```
vOpt = sdo.optimize(estFcn, v, opts);  
disp(vOpt)
```

```
Optimization started 15-Aug-2017 00:06:27
```

Iter	F-count	f(x)	max constraint	Step-size	First-order optimality
0	5	13.4798	0		
1	18	12.2052	0	0.129	305
2	30	11.1441	0	0.0648	790
3	41	10.0493	0	0.0843	290
4	46	9.23607	0	0.0758	286
5	51	8.76122	0	0.0183	10.1
6	56	8.75862	0	0.00184	0.476
7	57	8.75862	0	8.41e-05	0.476

```
Local minimum possible. Constraints satisfied.
```


fmincon stopped because the size of the current step is less than the selected value of the step size tolerance and constraints are satisfied to within the selected value of the constraint tolerance.

(1,1) =

```
Name: 'Delay'  
Value: 0.0038  
Minimum: 0.0020  
Maximum: 0.0090  
Free: 0  
Scale: 0.0078  
Info: [1x1 struct]
```

(1,2) =

```
Name: 'Gain'  
Value: 0.9012  
Minimum: 0.6000  
Maximum: 1  
Free: 1  
Scale: 1  
Info: [1x1 struct]
```

(1,3) =

```
Name: 'Tc'  
Value: 16.6833  
Minimum: 10  
Maximum: 30  
Free: 0  
Scale: 16  
Info: [1x1 struct]
```

(1,4) =

```
Name: 'Tp'  
Value: 0.0157  
Minimum: 0.0050  
Maximum: 0.0500  
Free: 1
```

```
Scale: 0.0156  
Info: [1x1 struct]
```

```
1x4 param.Continuous
```

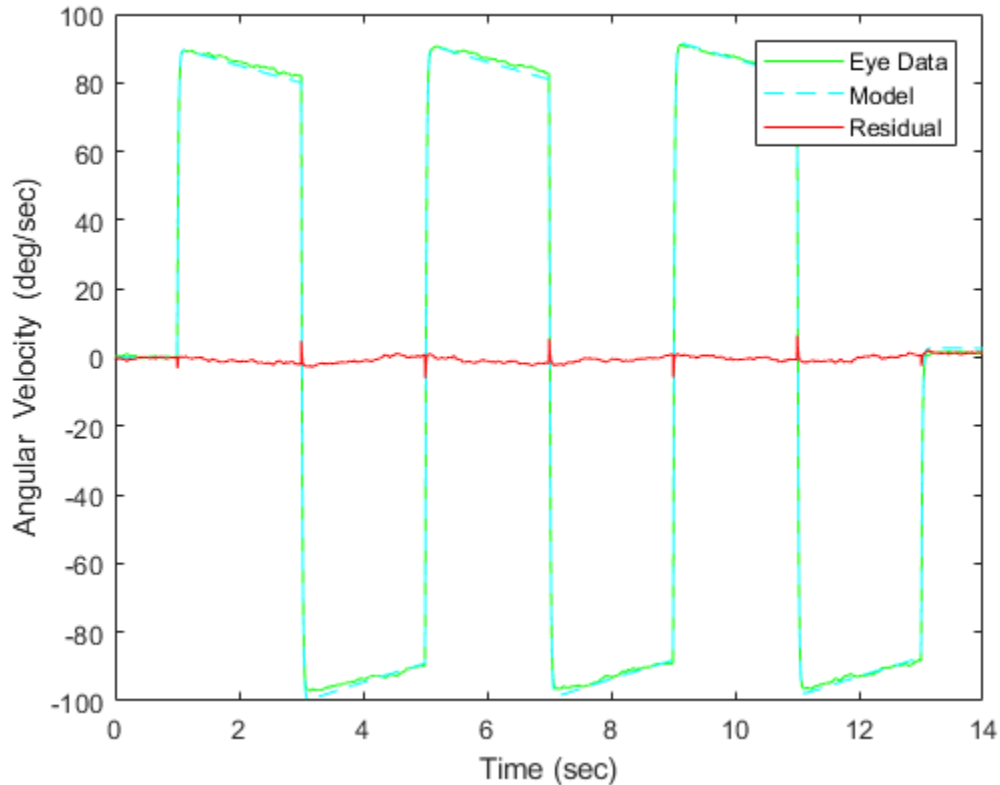
Visualizing Result of Optimization

Obtain the model response after estimation. Search for the `model_residual` signal in the logged simulation data.

```
Exp = setEstimatedValues(Exp, vOpt);  
Simulator = createSimulator(Exp, Simulator);  
Simulator = sim(Simulator);  
SimLog = find(Simulator.LoggedData, ...  
    get_param(model_name, 'SignalLoggingName') );  
EyeSignal = find(SimLog, 'EyeMotion');
```

Comparing the measured eye data with the optimized model response shows that the residuals are much smaller.

```
estFcn = @(v) sdoVOR_Objective(v, Simulator, Exp, 'Residuals');  
Model_Error = estFcn(vOpt);  
plot(Time, EyeData, '-g', ...  
    EyeSignal.Values.Time, EyeSignal.Values.Data, '--c', ...  
    Time, Model_Error.F, '-r');  
xlabel('Time (sec)');  
ylabel('Angular Velocity (deg/sec)');  
legend('Eye Data', 'Model', 'Residual');
```



Close the model

```
bdclose(model_name)
```

See Also

Related Examples

- “Identify Key Parameters for Estimation (GUI)” on page 4-168
- “Use Parallel Computing for Sensitivity Analysis” on page 4-137
- “Use Fast Restart Mode During Sensitivity Analysis” on page 4-144

- “Generate Parameter Samples for Sensitivity Analysis” on page 4-10
- “Analyze Relation Between Parameters and Design Requirements” on page 4-91
- “Validate Sensitivity Analysis” on page 4-129

Optimization-Based Control Design

- “Overview of Optimization-Based Compensator Design” on page 5-2
- “Time-Domain Design Requirements in Simulink” on page 5-4
- “Frequency-Domain Design Requirements in Simulink” on page 5-15
- “Time- and Frequency-Domain Requirements in Control System Designer App” on page 5-35
- “Time-Domain Simulations in Control System Designer App” on page 5-40
- “Design Optimization-Based Controllers for LTI Systems” on page 5-41
- “Optimize LTI System to Meet Frequency-Domain Requirements” on page 5-42
- “Design Linear Controllers for Simulink Models” on page 5-65

Overview of Optimization-Based Compensator Design

You can design optimization-based controllers for Simulink models to meet time and frequency-domain design requirements, as described in “Design Optimization to Meet Time- and Frequency-Domain Requirements (GUI)” on page 3-99.

If you have Control System Toolbox software installed, you can also design and optimize control systems by tuning controller elements or parameters within a SISO Design Task in the Control and Estimation Tools Manager. You can tune elements or parameters such as poles, zeros, and gains within any controller in the system and optimize the open and closed loop responses to meet time- and frequency-domain requirements.

Optimize the responses of systems in the SISO Design Task to meet both time- and frequency-domain performance requirements by graphically constraining signals:

- Add frequency-domain design requirements to plots such as root-locus, Nichols, and Bode in the SISO Design Task graphical tuning editor called SISO Design Tool.
- Add time-domain design requirements to plots such as step or impulse response (when displayed within the Linear System Analyzer as part of a SISO Design Task).

You can use optimization methods in **Control System Designer** app to tune both command-line LTI models as well as Simulink models:

- Create an LTI model using the Control System Toolbox command-line functions and enter `controlSystemDesigner` at the command line to open Control System Designer. For an example, see “Optimize LTI System to Meet Frequency-Domain Requirements” on page 5-42.
- Use a Simulink Compensator Design task (from Simulink Control Design software) to automatically analyze the model and then create a SISO Design Task for a linearized version of the model. You can then use the optimization techniques in the SISO Design Task to tune the response of the linearized Simulink model. For an example, see “Design Optimization-Based PID Controller for Linearized Simulink Model (GUI)”.

Note When using response optimization within a SISO Design Task you cannot add uncertainty to system parameters.

When using a SISO Design Task, Simulink Design Optimization software automatically sets the model's simulation start and stop time and you cannot directly change them. By

default, the simulation starts at 0 and continues until the SISO Design Task determines that the dynamics of the model have settled out. In addition, when the design requirements extend beyond this point, the simulation continues to the extent of the design requirements. Although you cannot directly adjust the start or stop time of the simulation, you can adjust the design requirements to extend further in time and thus force the simulation to continue to a certain point.

Time-Domain Design Requirements in Simulink

In this section...

“Specify Piecewise-Linear Lower and Upper Bounds” on page 5-4

“Specify Step Response Characteristics” on page 5-5


“Track Reference Signals” on page 5-8

“Specify Custom Requirements” on page 5-10

“Edit Design Requirements” on page 5-12

Specify Piecewise-Linear Lower and Upper Bounds

To specify upper and lower bounds on a signal:


- 1 In the Response Optimization tool, select **Signal Bound** in the **New** drop-down list. A window opens where you specify upper or lower bounds on a signal.
- 2 Specify a requirement name in the **Name** box.
- 3 Select the requirement type using the **Type** list.
- 4 Specify the edge start and end times and corresponding amplitude in the **Time (s)** and **Amplitude** columns.
- 5 Click  to specify additional bound edges.

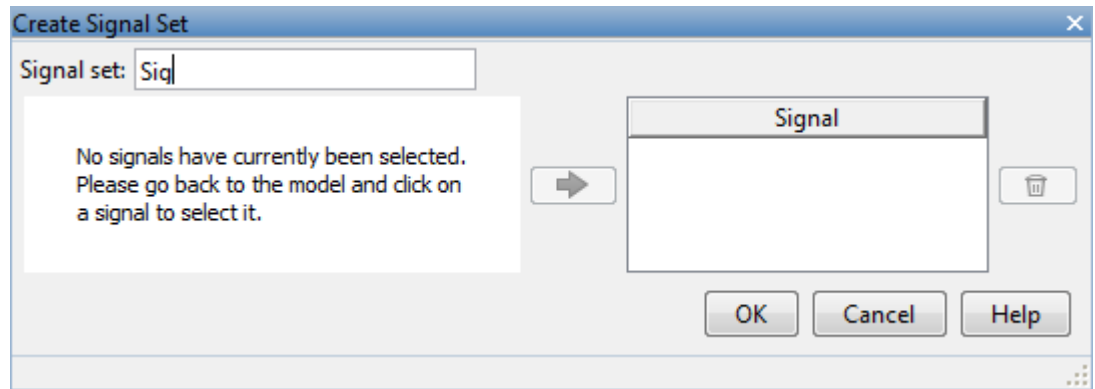
Select a row and click  to delete a bound edge.

- 6 In the **Select Signals to Bound** area, select a logged signal to apply the requirement to.


If you have already selected signals, as described in “Specify Signals to Log” on page 3-12, they appear in the list. Select the corresponding check-box.

If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.
- b In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

7 Click **OK**.

A variable with the specified requirement name appears in the **Data** area of the tool. A graphical display of the requirement also appears in the Response Optimization tool window.

8 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can add a Check Custom Bounds block to your model to specify piecewise-linear bounds.

Specify Step Response Characteristics

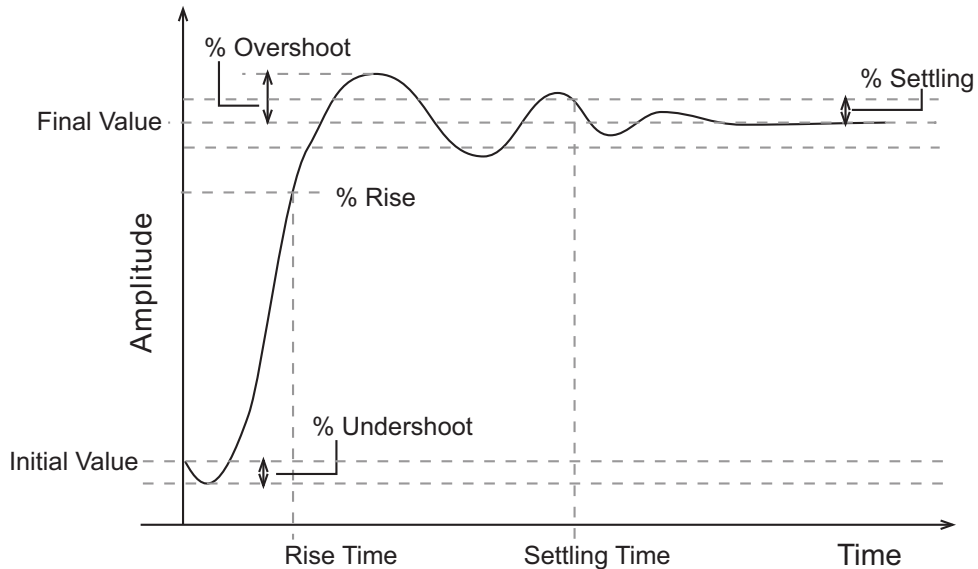
To apply a step response requirement to a signal in your model, specify the step response characteristics as follows:

- 1 Select a step response requirement from the Response Optimization tool.

In the **New** drop-down menu of the tool, in the **New Time Domain Requirement** section, select **Step Response Envelope**.

A Create Requirement dialog box opens where you specify the step response requirements on a signal.

- 2 Specify a requirement name in the **Name** field of the dialog box.
- 3 Specify the step response characteristics:




- **Initial value** — Input level before the step occurs
- **Step time** — Time at which the step takes place
- **Final value** — Input level after the step occurs
- **Rise time** — The time taken for the response signal to reach a specified percentage of the step range. The step range is the difference between the final and initial values.
- **% Rise** — The percentage of the step range used with **Rise time** to define the overall rise time characteristics.
- **Settling time** — Time taken until the response signal settles within a specified region around the final value. This settling region is defined as the final step value plus or minus the specified percentage of the final value.

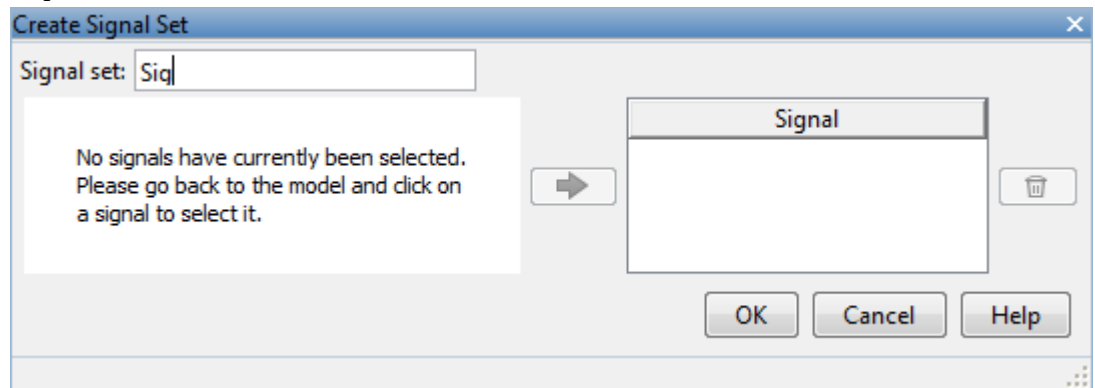
- **% Settling** — The percentage of the final value that defines the settling range of settling time characteristic specified in **Settling time**.
 - **% Overshoot** — The amount by which the response signal can exceed the final value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.
 - **% Undershoot** — The amount by which the response signal can undershoot the initial value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.
- 4 Specify the signal to be bound.

To apply this requirement to a model signal, in the **Select Signals to Bound** area, select a logged signal to which you will apply the requirement.

If you have already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, it appears in the list. Select the corresponding check-box.

If you have not selected a signal to log:

- Click . The Create Signal Set dialog box opens where you specify the logged signal.
- In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- Select the signal and click  to add it to the signal set.

- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

Alternatively, you can use the Check Step Response Characteristics block to specify step response bounds for a signal.

See Also

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

Track Reference Signals

Use reference tracking to force a model signal to match a desired signal. To track a reference signal:


- 1 In the Response Optimization tool, select **Signal Tracking** in the **New** drop-down list. A window opens where you specify the reference signal to track.
- 2 Specify a requirement name in the **Name** box.
- 3 Define the reference signal by entering vectors, or variables from the workspace, in the **Time vector** and **Amplitude** fields.

Click **Update reference signal data** to use the new amplitude and time vector as the reference signal.

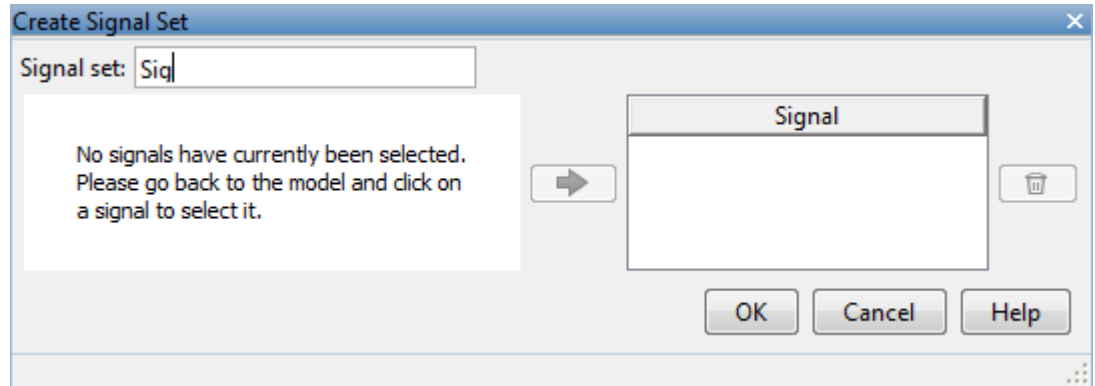
- 4 Specify how the optimization solver minimizes the error between the reference and model signals using the **Tracking Method** list:
 - SSE — Reduces the sum of squared errors
 - SAE — Reduces the sum of absolute errors
- 5 In the **Specify Signal to Track Reference Signal** area, select a logged signal to apply the requirement to.

If you already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, they appear in the list. Select the corresponding check-box.


If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.

- b In the Simulink model window, click the signal to which you want to add a requirement.



The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

- e Select the check-box corresponding to the signal and click **OK**.

A variable with the specified requirement name appears in the **Data** area of the tool. A graphical display of the signal bound also appears in the Response Optimization tool window.

Note When tracking a reference signal, the software ignores the maximally feasible solution option. For more information on this option, in the **Response Optimization** tab, click **Options > Optimization Options**, and click **Help**.


Alternatively, you can use the Check Against Reference block to specify a reference signal to track.


See Also

“Design Optimization to Track Reference Signal (GUI)”

Specify Custom Requirements

You can specify custom requirements, such as minimizing system energy. To specify custom requirements:

- 1 In the Response Optimization tool, in **New** drop-down menu, select **Custom Requirement**. The Create Requirement dialog box opens where you specify the custom requirement.
- 2 Specify a requirement name in **Name**.
- 3 Specify the requirement type in the **Type** drop-down menu.
- 4 Specify the name of the function that contains the custom requirement in **Function**. The field must be specified as a function handle using @. The function must be on the MATLAB path. Click  to review or edit the function.

If the function does not exist, clicking  opens a template MATLAB file. Use this file to implement the custom requirement. The default function name is `myCustomRequirement`.

- 5 (Optional) To prevent the solver from considering specific parameter combinations, select **Error if constraint is violated**. Use this option for parameter-only constraints.

During an optimization iteration, the solver first evaluates requirements with this option selected.

- If the constraint is violated, the solver skips evaluating any remaining requirements and proceeds to the next iteration.
- If the constraint is *not* violated, the solver evaluates the remaining requirements for the current iteration. If any of the remaining requirements bound signals or systems, then the solver simulates the model.

For more information, see “Skip Model Simulation Based on Parameter Constraint Violation (GUI)” on page 3-190.

Note If you select this check box, then do not specify signals or systems to bound. If you *do* specify signals or systems, then this check box is ignored.

- 6 (Optional) Specify the signal or system, or both, to be bound.

You can apply this requirement to model signals, or a linearization of your Simulink model (requires Simulink Control Design), or both.


Click **Select Signals and Systems to Bound (Optional)** to view the signal and linearization I/O selection area.

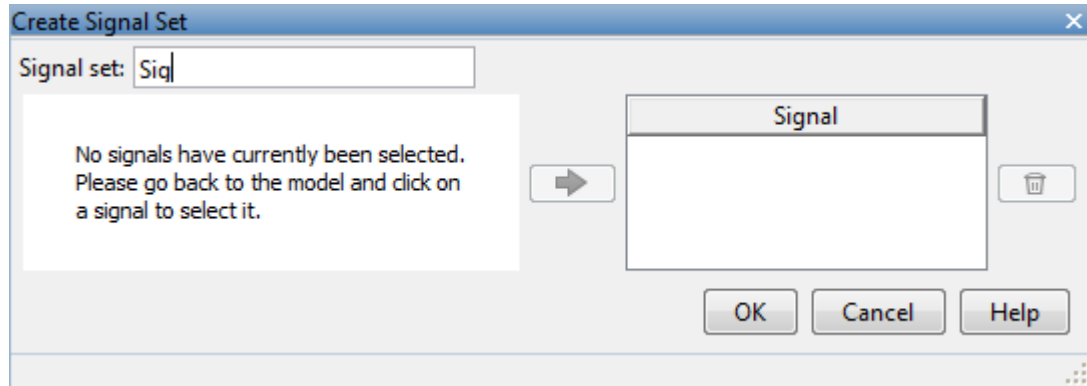
- To apply this requirement to a model signal:

In the **Signal** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, it appears in the list. Select the corresponding check box.

If you have not selected a signal to log:

- Click . A Create Signal Set dialog box opens where you specify the logged signal.
- In the Simulink model window, click the signal to which you want to add a requirement.




The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- Select the signal and click  to add it to the signal set.
- In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

- To apply this requirement to a linear system:
 - a Specify the simulation time at which the model is linearized in **Snapshot Times**. For multiple simulation snapshot times, specify a vector.
 - b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box. For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

7 Click **OK**.

A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.

See Also

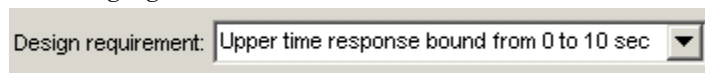
- “Design Optimization to Meet a Custom Objective (GUI)” on page 3-115
- “Design Optimization to Meet a Custom Objective (Code)” on page 3-133

Edit Design Requirements

The Edit Design Requirement dialog box allows you to exactly position constraint segments and to edit other properties of these constraints. The dialog box has two main components:

- An upper panel to specify the constraint you are editing
- A lower panel to edit the constraint parameters

The upper panel of the Edit Design Requirement dialog box resembles the image in the following figure.



In the **Control System Designer** app in Control System Toolbox, you can edit design requirements from the analysis plots. The **Design requirement** drop-down list will contain all the requirements on that plot.

Edit Design Requirement Dialog Box Parameters

The particular parameters shown within the lower panel of the Edit Design Requirement dialog box depend on the type of constraint/requirement. In some cases, the lower panel contains a grid with one row for each segment and one column for each constraint parameter. The following table summarizes the various constraint parameters.

Edit Design Requirement Dialog Box Parameters

Parameter	Found in	Description
Time	Upper and lower time response bounds on step and impulse response plots	Defines the time range of a segment within a constraint/requirement.
Amplitude	Upper and lower time response bounds on step and impulse response plots	Defines the beginning and ending amplitude of a constraint segment.
Slope (1/s)	Upper and lower time response bounds	Defines the slope, in 1/s, of a constraint segment. It is an alternative method of specifying the magnitude values. Entering a new Slope value changes any previously defined magnitude values.
Final value	Step response bounds	Defines the input level after the step occurs.
Rise time	Step response bounds	Defines a constraint segment for a particular rise time.
% Rise	Step response bounds	The percentage of the step's range used to describe the rise time.
Settling time	Step response bounds	Defines a constraint segment for a particular settling time.
% Settling	Step response bounds	The percentage of the final value that defines the settling region used to describe the settling time.
% Overshoot	Step response bounds	The percentage amount by which the signal can exceed the final value before settling.
% Undershoot	Step response bounds	Defines the constraint segments for a particular percent undershoot.

Frequency-Domain Design Requirements in Simulink

In this section...

“Specify Lower Bounds on Gain and Phase Margin” on page 5-15

“Specify Piecewise-Linear Lower and Upper Bounds on Frequency Response” on page 5-17

“Specify Bound on Closed-Loop Peak Gain” on page 5-19

“Specify Lower Bound on Damping Ratio” on page 5-21

“Specify Upper and Lower Bounds on Natural Frequency” on page 5-23

“Specify Upper Bound on Approximate Settling Time” on page 5-25

“Specify Piecewise-Linear Upper and Lower Bounds on Singular Values” on page 5-27

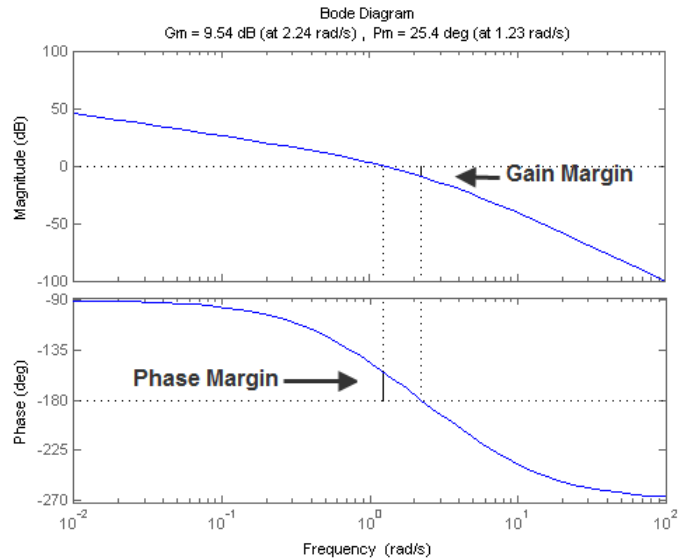
“Specify Step Response Characteristics” on page 5-29

“Specify Custom Requirements” on page 5-31

Specify Lower Bounds on Gain and Phase Margin

To specify lower bounds on the gain and phase margin of a linear system:

- 1 In the Response Optimization tool, select **Gain and Phase Margin** in the **New** list. A window opens where you specify lower bounds on the gain and phase margin of your linear system.
- 2 Specify a requirement name in **Name**.
- 3 Specify bounds on the gain margin or phase margin, or both.



- **Gain margin** — Amount of gain increase or decrease required to make the loop gain unity at the frequency where the phase angle is -180° .
- **Phase margin** — Amount of phase increase or decrease required to make the phase angle -180° when the loop gain is 1.0


To specify a lower bound on the gain margin or phase margin, or both, select the corresponding check box and enter the lower bound value.

- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

5 Click **OK**.

A variable with the specified requirement name appears in the **Data** area of the tool. A graphical display of the requirement also appears in the Response Optimization tool window.

6 (Optional) In the graphical display, you can:


- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Gain and Phase Margins block to specify bounds on the gain and phase margin. (Requires Simulink Control Design.)

Specify Piecewise-Linear Lower and Upper Bounds on Frequency Response

To specify upper or lower bounds on the magnitude of a system response:

- 1** In the Response Optimization tool, select **Bode Magnitude** in the **New** list. A window opens where you specify the lower or upper bounds on the magnitude of the system response.
- 2** Specify a requirement name in the **Name** box.
- 3** Specify the requirement type using the **Type** list.
- 4** Specify the edge start and end frequencies and corresponding magnitude in the **Frequency** and **Magnitude** columns.
- 5** Insert or delete bound edges.

Click  to specify additional bound edges.


Select a row and click  to delete a bound edge.

- 6 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

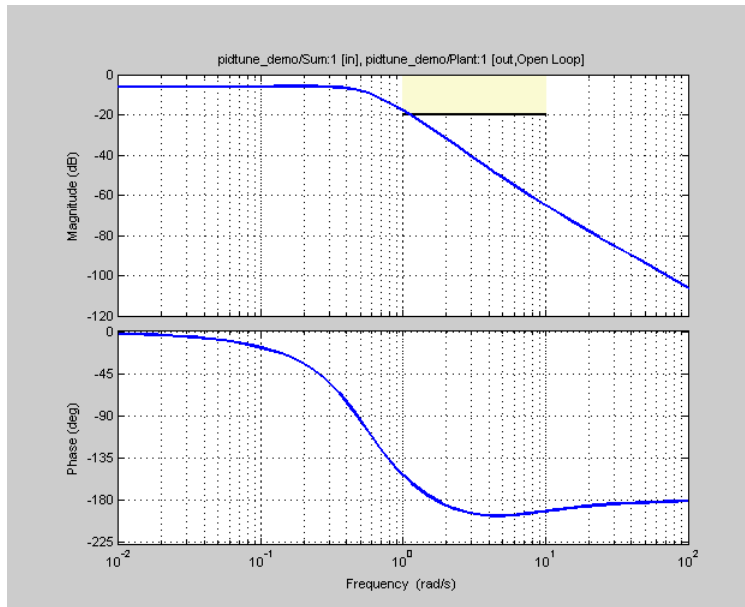
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 7 Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool window. A graphical display of the requirement also appears in the Response Optimization tool window.



8 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Bode Characteristics block to specify bounds on the magnitude of the system response. (Requires Simulink Control Design.)

Specify Bound on Closed-Loop Peak Gain


To specify an upper bound on the closed-loop peak response of a system:

- 1 In the Response Optimization tool, select **Closed-Loop Peak Gain** in the **New** list. A window opens where you specify an upper bound on the closed-loop peak gain of the system.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify the upper bound on the closed-loop peak gain in the **Closed-Loop peak gain** box.
- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a** Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b** Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

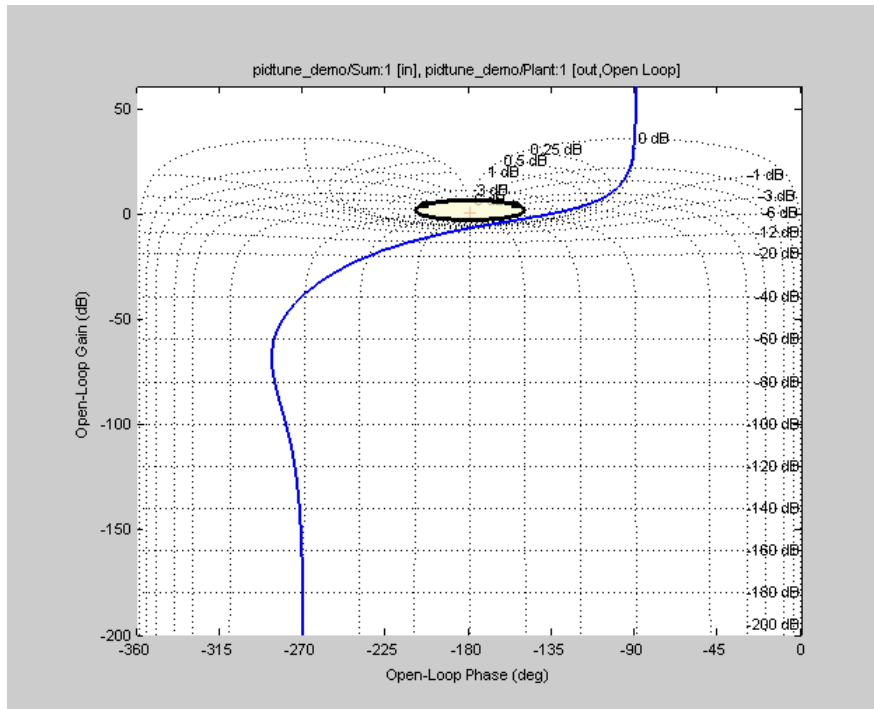
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 5** Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool window. A graphical display of the requirement also appears in the Response Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Nichols Characteristics block to specify bounds on the magnitude of the system response. (Requires Simulink Control Design.)

Specify Lower Bound on Damping Ratio

To specify a lower bound on the damping ratio of the system:


- 1 In the Response Optimization tool, select **Damping Ratio** in the **New** list. A window opens where you specify a lower bound on the damping ratio of the system.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify the lower bound on the damping ratio in the **Damping ratio** box.

- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

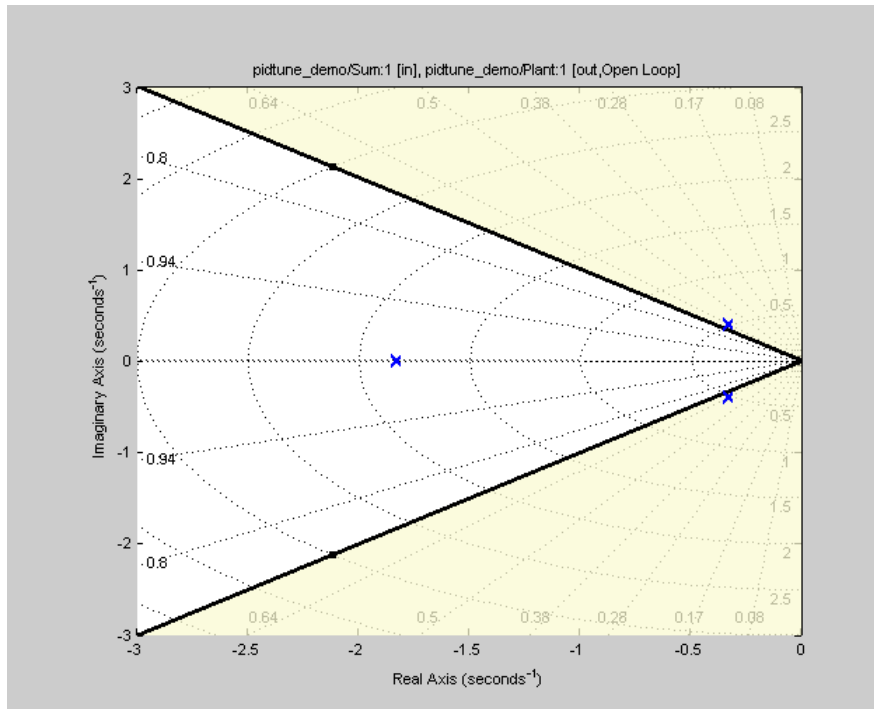
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 5 Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Pole-Zero Characteristics block to specify a bound on the damping ratio. (Requires Simulink Control Design.)

Specify Upper and Lower Bounds on Natural Frequency

To specify a bound on the natural frequency of the system:


- 1 In the Response Optimization tool, select **Natural Frequency** in the **New** list. A window opens where you specify a bound on the natural frequency of the system.
- 2 Specify a requirement name in the **Name** box.
- 3 Specify a lower or upper bound on the natural frequency in the **Natural frequency** box.

- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

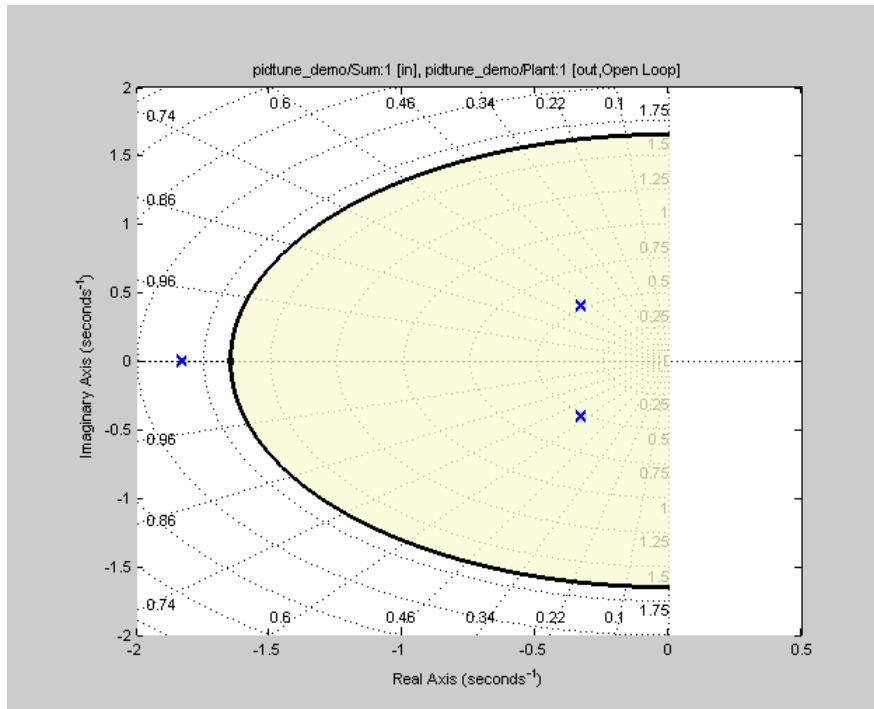
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 5 Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Pole-Zero Characteristics block to specify a bound on the natural frequency. (Requires Simulink Control Design.)

Specify Upper Bound on Approximate Settling Time

To specify an upper bound on the approximate settling time of the system:


- 1 In the Response Optimization tool, select **Settling Time** in the **New** list. A window opens where you specify an upper bound on the approximate settling time of the system.
- 2 Specify a requirement name in the **Name** box.

- 3 Specify the upper bound on the approximate settling time in the **Settling time** box.
- 4 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

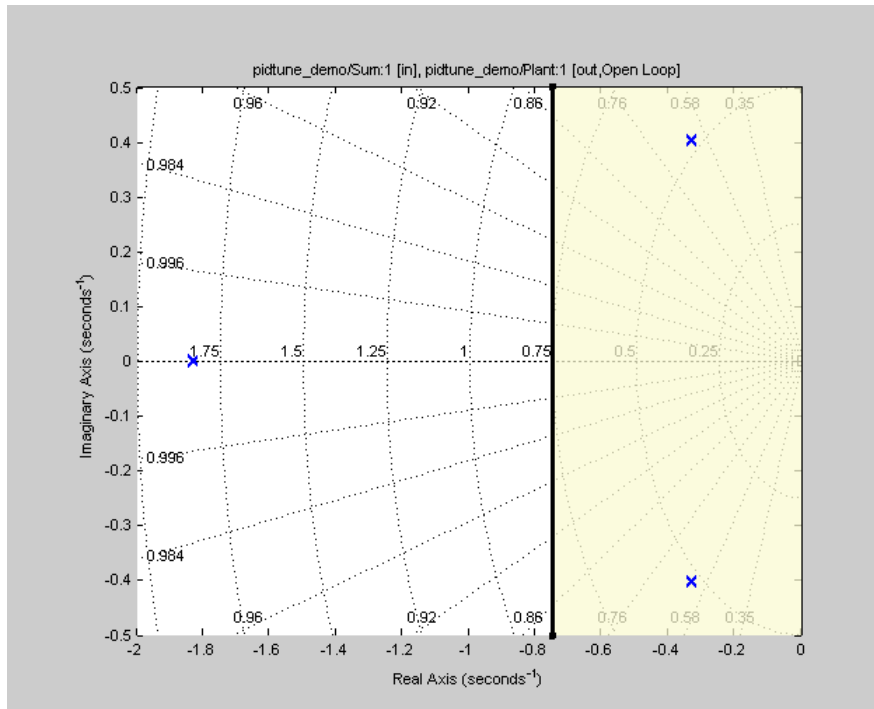
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 5 Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.



6 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17


Alternatively, you can use the Check Pole-Zero Characteristics block to specify the approximate settling time. (Requires Simulink Control Design.)

Specify Piecewise-Linear Upper and Lower Bounds on Singular Values

To specify piecewise-linear upper and lower bounds on the singular values of a system:

- 1 In the Response Optimization tool, select **Singular Values** in the **New** list. A window opens where you specify the lower or upper bounds on the singular values of the system.
- 2 Specify a requirement name in the **Name** box.

- 3 Specify the requirement type using the **Type** list.
- 4 Specify the edge start and end frequencies and corresponding magnitude in the **Frequency** and **Magnitude** columns, respectively.
- 5 Insert or delete bound edges.

Click  to specify additional bound edges.


Select a row and click  to delete a bound edge.

- 6 In the **Select Systems to Bound** section, select the linear systems to which this requirement applies.

Linear systems are defined by snapshot times at which the model is linearized and sets of linearization I/O points defining the system inputs and outputs.

- a Specify the simulation time at which the model is linearized using the **Snapshot Times** box. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

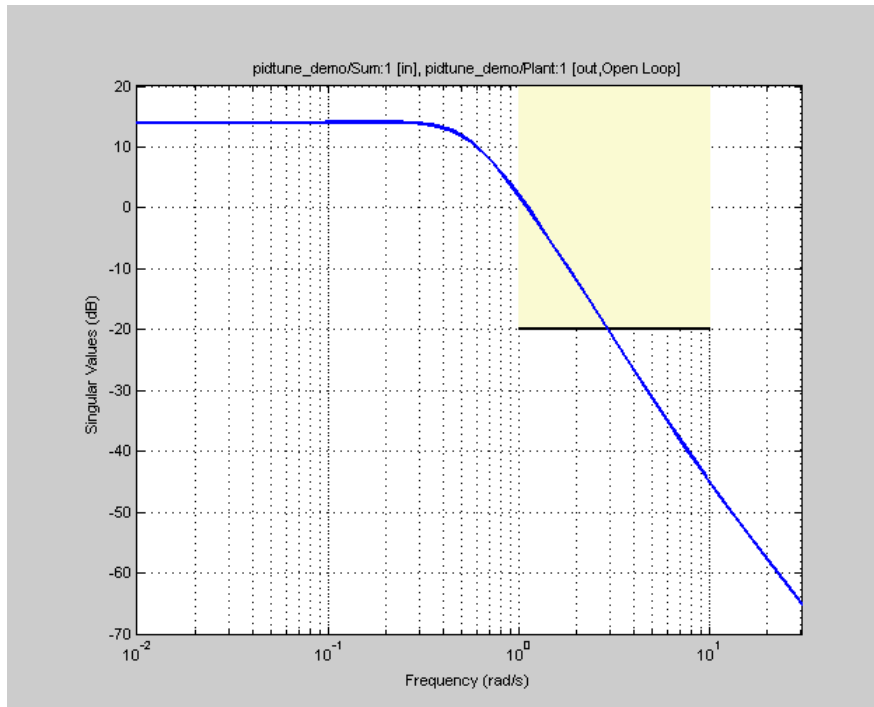
If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

- 7 Click **OK**.

A new variable with the specified name appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.



8 (Optional) In the graphical display, you can:

- “Move Constraints Graphically” on page 3-16
- “Position Constraints Exactly” on page 3-17

Alternatively, you can use the Check Singular Value Characteristics block to specify bounds on the singular value. (Requires Simulink Control Design).

Specify Step Response Characteristics

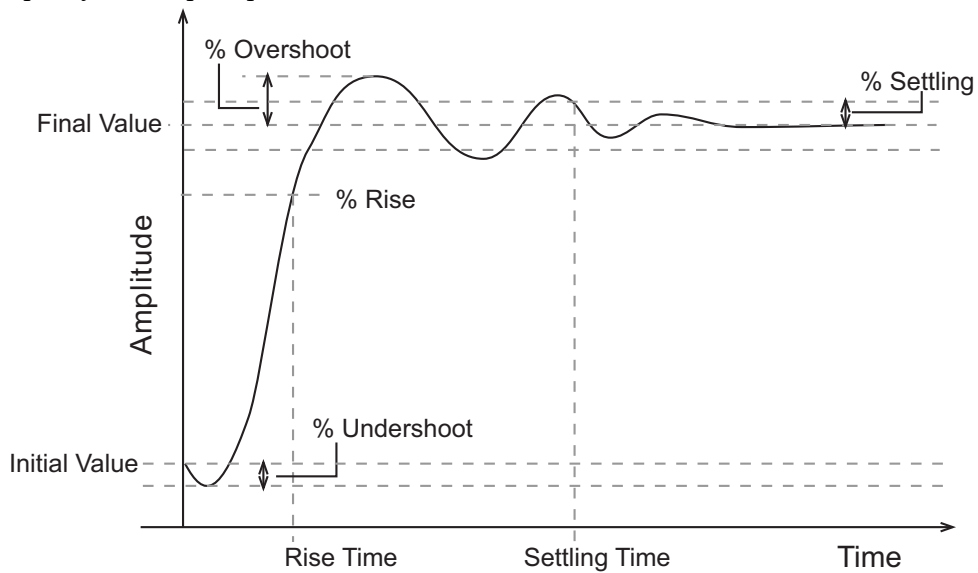
To apply a step response requirement to a linearization of your model (requires Simulink Control Design), specify the step response characteristics as follows:

- 1 Select a step response requirement from the Response Optimization tool.

In the **New** drop-down menu of the tool, in the **New Frequency Domain Requirement** section, select **Step Response Envelope**.

A Create Requirement dialog box opens where you specify the step response requirements.

- 2 Specify a requirement name in the **Name** field of the dialog box.
- 3 Specify the step response characteristics:




- **Initial value** — Input level before the step occurs
- **Step time** — Time at which the step takes place
- **Final value** — Input level after the step occurs
- **Rise time** — The time taken for the response signal to reach a specified percentage of the step range. The step range is the difference between the final and initial values.
- **% Rise** — The percentage of the step range used with **Rise time** to define the overall rise time characteristics.
- **Settling time** — Time taken until the response signal settles within a specified region around the final value. This settling region is defined as the final step value plus or minus the specified percentage of the final value.
- **% Settling** — The percentage of the final value that defines the settling range of settling time characteristic specified in **Settling time**.

- **% Overshoot** — The amount by which the response signal can exceed the final value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.
 - **% Undershoot** — The amount by which the response signal can undershoot the initial value. This amount is specified as a percentage of the step range. The step range is the difference between the final and initial values.
- 4 Specify the systems to be bound.

To apply this requirement to a linearization of your Simulink model:

- a In the **Select Systems to Bound** area, specify the simulation time at which the model is linearized in **Snapshot Times**. For multiple simulation snapshot times, specify a vector.
- b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box.

For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.


For more information on linearization, see “What Is Linearization?” (Simulink Control Design).


Alternatively, you can use the Check Step Response Characteristics block to specify step response bounds for a signal.

Specify Custom Requirements

You can specify custom requirements, such as minimizing system energy. To specify custom requirements:

- 1 In the Response Optimization tool, in **New** drop-down menu, select **Custom Requirement**. The Create Requirement dialog box opens where you specify the custom requirement.
- 2 Specify a requirement name in **Name**.

- 3 Specify the requirement type in the **Type** drop-down menu.
- 4 Specify the name of the function that contains the custom requirement in **Function**. The field must be specified as a function handle using @. The function must be on the MATLAB path. Click  to review or edit the function.

If the function does not exist, clicking  opens a template MATLAB file. Use this file to implement the custom requirement. The default function name is `myCustomRequirement`.

- 5 (Optional) To prevent the solver from considering specific parameter combinations, select **Error if constraint is violated**. Use this option for parameter-only constraints.

During an optimization iteration, the solver first evaluates requirements with this option selected.

- If the constraint is violated, the solver skips evaluating any remaining requirements and proceeds to the next iteration.
- If the constraint is *not* violated, the solver evaluates the remaining requirements for the current iteration. If any of the remaining requirements bound signals or systems, then the solver simulates the model.

For more information, see “Skip Model Simulation Based on Parameter Constraint Violation (GUI)” on page 3-190.

Note If you select this check box, then do not specify signals or systems to bound. If you *do* specify signals or systems, then this check box is ignored.

- 6 (Optional) Specify the signal or system, or both, to be bound.

You can apply this requirement to model signals, or a linearization of your Simulink model (requires Simulink Control Design), or both.


Click **Select Signals and Systems to Bound (Optional)** to view the signal and linearization I/O selection area.

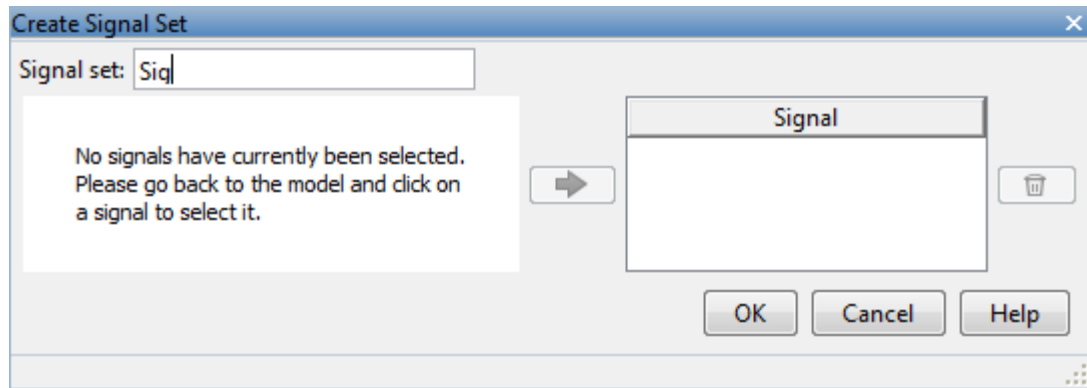
- To apply this requirement to a model signal:

In the **Signal** area, select a logged signal to which you will apply the requirement.


If you have already selected a signal to log, as described in “Specify Signals to Log” on page 3-12, it appears in the list. Select the corresponding check box.

If you have not selected a signal to log:

- a Click . A Create Signal Set dialog box opens where you specify the logged signal.
- b In the Simulink model window, click the signal to which you want to add a requirement.




The Create Signal Set dialog box updates and displays the name of the block and the port number where the selected signal is located.

- c Select the signal and click  to add it to the signal set.
- d In **Signal set** field, enter a name for the selected signal set.

Click **OK**. A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool.

- To apply this requirement to a linear system:
 - a Specify the simulation time at which the model is linearized in **Snapshot Times**. For multiple simulation snapshot times, specify a vector.
 - b Select the linearization input/output set from the **Linearization I/O** area.

If you have already created a linearization input/output set, it appears in the list. Select the corresponding check box.

If you have not created a linearization input/output set, click  to open the Create linearization I/O set dialog box. For more information on using this dialog box, see “Create Linearization I/O Sets” on page 3-86.

For more information on linearization, see “What Is Linearization?” (Simulink Control Design).

7 Click **OK**.

A new variable, with the specified name, appears in the **Data** area of the Response Optimization tool. A graphical display of the requirement also appears in the Response Optimization tool window.

See Also

- “Design Optimization to Meet a Custom Objective (GUI)” on page 3-115
- “Design Optimization to Meet a Custom Objective (Code)” on page 3-133

Time- and Frequency-Domain Requirements in Control System Designer App

In this section...

“Root Locus Diagrams” on page 5-35

“Open-Loop and Prefilter Bode Diagrams” on page 5-37

“Open-Loop Nichols Plots” on page 5-37

“Step/Impulse Response Plots” on page 5-38

Root Locus Diagrams

- “Settling Time” on page 5-35
- “Percent Overshoot” on page 5-35
- “Damping Ratio” on page 5-36
- “Natural Frequency” on page 5-36
- “Region Constraint” on page 5-37

Settling Time

If you specify a settling time in the continuous-time root locus, a vertical line appears on the root locus plot at the pole locations associated with the value provided (using a first-order approximation). In the discrete-time case, the constraint is a curved line.

It is required that $\text{Re}\{pole\} < -4.6 / T_{\text{settling}}$ for continuous systems and

$\log(\text{abs}(pole)) / T_{\text{discrete}} < -4.6 / T_{\text{settling}}$ for discrete systems. This is an approximation of the settling time based on second-order dominant systems.

Percent Overshoot

Specifying percent overshoot in the continuous-time root locus causes two rays, starting at the root locus origin, to appear. These rays are the locus of poles associated with the percent value (using a second-order approximation). In the discrete-time case, the constraint appears as two curves originating at (1,0) and meeting on the real axis in the left-hand plane.

The percent overshoot $p.o$ constraint can be expressed in terms of the damping ratio, as in this equation:

$$p.o. = 100e^{-\pi\zeta/\sqrt{1-\zeta^2}}$$

where ζ is the damping ratio.

Damping Ratio

Specifying a damping ratio in the continuous-time root locus causes two rays, starting at the root locus origin, to appear. These rays are the locus of poles associated with the damping ratio. In the discrete-time case, the constraint appears as curved lines originating at (1,0) and meeting on the real axis in the left-hand plane.

The damping ratio defines a requirement on $-\text{Re}\{pole\} / \text{abs}(pole)$ for continuous systems and on

$$r = \text{abs}(pSys)$$

$$t = \text{angle}(pSys)$$

$$c = -\log(r) / \sqrt{(\log(r))^2 + t^2}$$

for discrete systems.

Natural Frequency

If you specify a natural frequency, a semicircle centered around the root locus origin appears. The radius equals the natural frequency.

The natural frequency defines a requirement on $\text{abs}(pole)$ for continuous systems and on

$$r = \text{abs}(pSys)$$

$$t = \text{angle}(pSys)$$

$$c = \sqrt{(\log(r))^2 + t^2} / Ts_{model}$$

for discrete systems.

Region Constraint

Specifies an exclusion region in the complex plane, causing a line to appear between the two specified points with a shaded region below the line. The poles must not lie in the shaded region.

Open-Loop and Prefilter Bode Diagrams

- “Gain and Phase Margins” on page 5-37
- “Upper Gain Limit” on page 5-37
- “Lower Gain Limit” on page 5-37

Gain and Phase Margins

Specify a minimum phase and or a minimum gain margin.

Upper Gain Limit

You can specify an upper gain limit, which appears as a straight line on the Bode magnitude curve. You must select frequency limits, the upper gain limit in decibels, and the slope in dB/decade.

Lower Gain Limit

Specify the lower gain limit in the same fashion as the upper gain limit.

Open-Loop Nichols Plots

- “Phase Margin” on page 5-37
- “Gain Margin” on page 5-38
- “Closed-Loop Peak Gain” on page 5-38
- “Gain-Phase Requirement” on page 5-38

Phase Margin

Specify a minimum phase amount.

While displayed graphically at only one location around a multiple of -180 degrees, this requirement applies to phase margin regardless of actual phase (i.e., it is interpreted for all multiples of -180).

Gain Margin

Specify a minimum gain margin.

While displayed graphically at only one location around a multiple of -180 degrees, this requirement applies to gain margin regardless of actual phase (i.e., it is interpreted for all multiples of -180).

Closed-Loop Peak Gain

Specify a peak closed-loop gain at a given location. The specified value can be positive or negative in dB. The constraint follows the curves of the Nichols plot grid, so it is recommended that you have the grid on when using this feature.

While displayed graphically at only one location around a multiple of -180 degrees, this requirement applies to gain margin regardless of actual phase (i.e., it is interpreted for all multiples of -180).

Gain-Phase Requirement

Specifies an exclusion region for the response on the Nichols plot. The response must not pass through the shaded region.

This only applies to the region (phase and gain) drawn.

Step/Impulse Response Plots

- “Upper Time Response Bound” on page 5-38
- “Lower Time Response Bound” on page 5-38
- “Step Response Bound” on page 5-38

Upper Time Response Bound

You can specify an upper time response bound for step and impulse responses.

Lower Time Response Bound

You can specify a lower time response bound for step and impulse responses.

Step Response Bound

For a step response plot, you can also specify a step response bound design requirement.

To define a step response bound requirement, specify the following step response parameters:

- **Final value** — Final steady-state value
- **Rise time** — Time required to reach the specified percentage, % **Rise**, of the **Final value**.
- **Settling time** — Time at which the response enters and stays within the settling percentage, % **Settling**, of the **Final value**.
- % **Overshoot** — Maximum percentage overshoot above the **Final value**.
- % **Undershoot** — Maximum percentage undershoot below the **Initial value**.

In the **Control System Designer** app, step response plots always use an **Initial value** and a **Step time** of 0.

See Also

Related Examples

- “Design Optimization-Based Controllers for LTI Systems” on page 5-41
- “Optimize LTI System to Meet Frequency-Domain Requirements” on page 5-42
- “Design Optimization-Based PID Controller for Linearized Simulink Model (GUI)”

Time-Domain Simulations in Control System Designer App

When performing optimization-based tuning in the **Control System Designer** app, Simulink Design Optimization software automatically sets the model simulation start and stop times; you cannot directly change them. By default, the simulation starts at 0 and continues until the app determines that the dynamics of the model have settled out. In addition, when the design requirements extend beyond this point, the simulation continues to the extent of the design requirements. Although you cannot directly adjust the start or stop time of the simulation, you can adjust the design requirements to extend further in time and thus force the simulation to continue to a certain point.

See Also

Related Examples

- “Optimize LTI System to Meet Frequency-Domain Requirements” on page 5-42

Design Optimization-Based Controllers for LTI Systems

This topic shows how to design optimization-based linear controllers for an LTI model.

To design an optimization-based linear controller:

- 1 Create and import a linear model into the **Control System Designer** app. You can create an LTI model at the MATLAB command line, as described in “Create an LTI Plant Model” on page 5-43.
- 2 Create a Control System Designer session, as described in “Open the Control System Designer App” on page 5-44.
- 3 In the **Tuning Methods** drop-down list, select `Optimization Based Tuning` to open the Response Optimization window. For more information, see “Open Optimization Based Tuning Method” on page 5-47 .
- 4 In the Response Optimization window, select the **Compensators** tab to select and configure the compensator elements you want to tune during the response optimization. For more information, see “Select Tunable Compensator Elements” on page 5-50.
- 5 In the **Design requirements** tab, select the design requirements you want the system to satisfy. For more information, see “Add Design Requirements” on page 5-51.
- 6 In the Response Optimization window, click **Start Optimization**. The optimization progress results are displayed in the **Optimization** tab. The **Compensators** pane contains the new, optimized compensator element values. For more information, see “Optimize the System Response” on page 5-60.

See Also

Related Examples

- “Optimize LTI System to Meet Frequency-Domain Requirements” on page 5-42

Optimize LTI System to Meet Frequency-Domain Requirements

In this section...

- “Design Requirements” on page 5-42
- “Create an LTI Plant Model” on page 5-43
- “Open the Control System Designer App” on page 5-44
- “Open Optimization Based Tuning Method” on page 5-47
- “Select Tunable Compensator Elements” on page 5-50
- “Add Design Requirements” on page 5-51
- “Optimize the System Response” on page 5-60
- “Create and Display the Closed-Loop System” on page 5-63

This example shows how to use frequency-domain design requirements to optimize the response of an LTI system in the **Control System Designer** app.

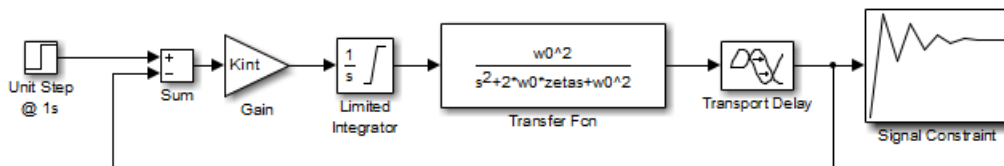
When used with Control System Toolbox software, you can place Simulink Design Optimization design requirements or constraints on plots in the **Control System Designer** app. You can include design requirements for response optimization in the frequency-domain and time-domain.

You can specify frequency-domain design requirements to optimize response signals for any model that you design in the **Control System Designer** app, such as:

- Command-line LTI models created with the Control System Toolbox commands
- Simulink models that have been linearized using Simulink Control Design software

Design Requirements

In this example, you use a linearized version of the Simulink model, `srotut1`.



You use optimization methods to design a compensator so that the closed-loop system meets the following design specifications when you excite the system with a unit step input:

- Maximum 30-second settling time
- Maximum 10% overshoot
- Maximum 10-second rise time
- Limit of ± 0.7 on the actuator signal

Create an LTI Plant Model

In the `srotut1` model, the plant model is composed of a gain, a limited integrator, a transfer function, and a transport delay block.

Design the compensator for the open-loop transfer function of the linearized `srotut1` model. The linearized `srotut1` plant model is composed of the gain, an unlimited integrator, the transfer function, and a Padé approximation to the transport delay.

To create an open-loop transfer function based on the linearized `srotut1` model, enter the following commands.

```
w0 = 1;
zeta = 1;
Kint = 0.5;
Tdelay = 1;
[delayNum, delayDen] = pade(Tdelay, 1);
integrator = tf(Kint, [1 0]);
transfer_fcn = tf(w0^2, [1 2*w0*zeta w0^2]);
delay_block = tf(delayNum, delayDen);
open_loopTF = integrator*transfer_fcn*delay_block;
```

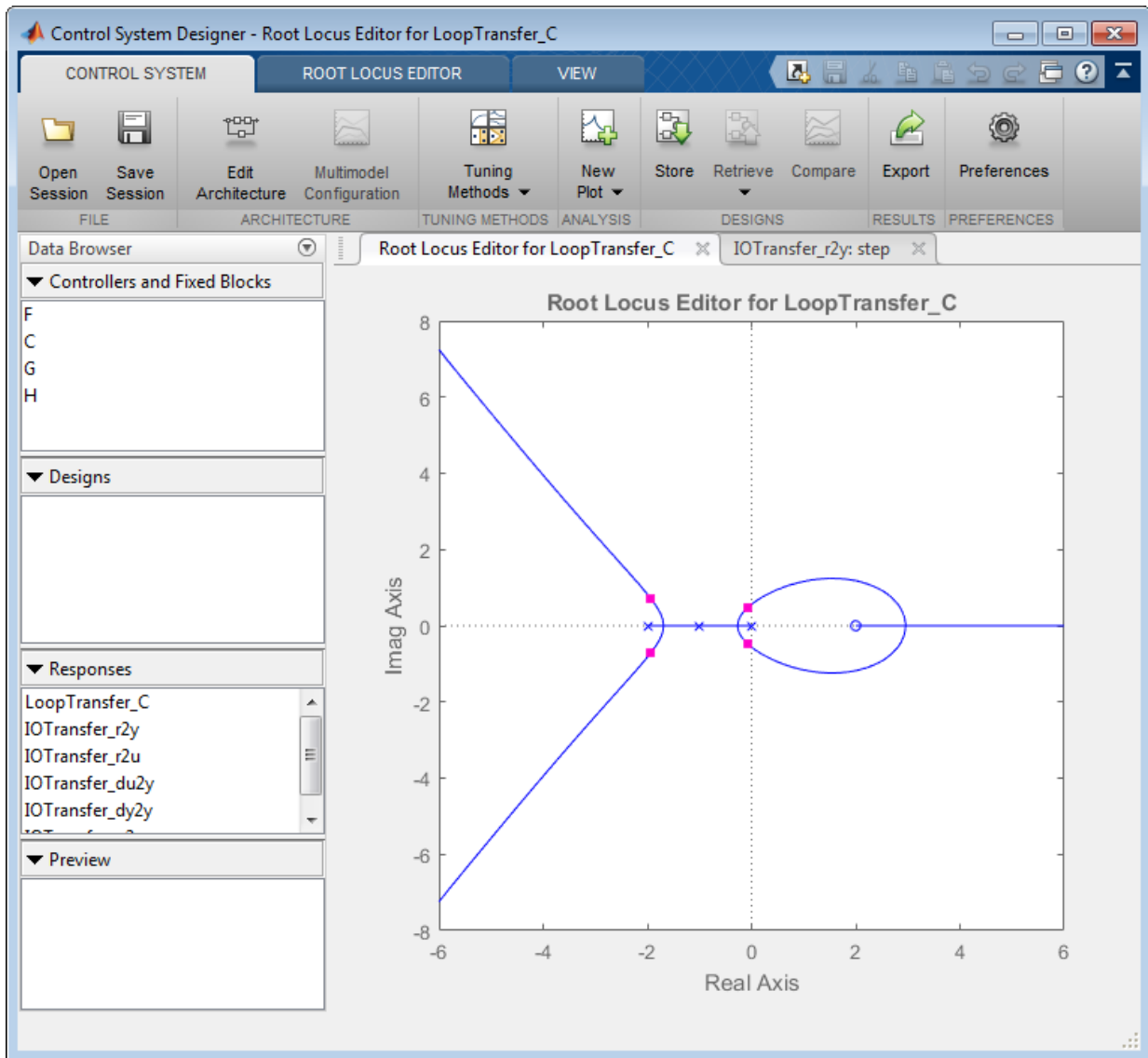
If the plant model is an array of models (Control System Toolbox), the controller is designed for a nominal model only. You can also analyze the control design for the remaining models in the array. For more information, see “Multimodel Control Design” (Control System Toolbox).

Tip You can directly linearize the Simulink model using Simulink Control Design software.

Open the Control System Designer App

This example uses a root locus diagram to design the response of the open-loop transfer function, `open_loopTF`. To create a **Control System Designer** app session with a root locus plot for the open-loop transfer function, use the following command:

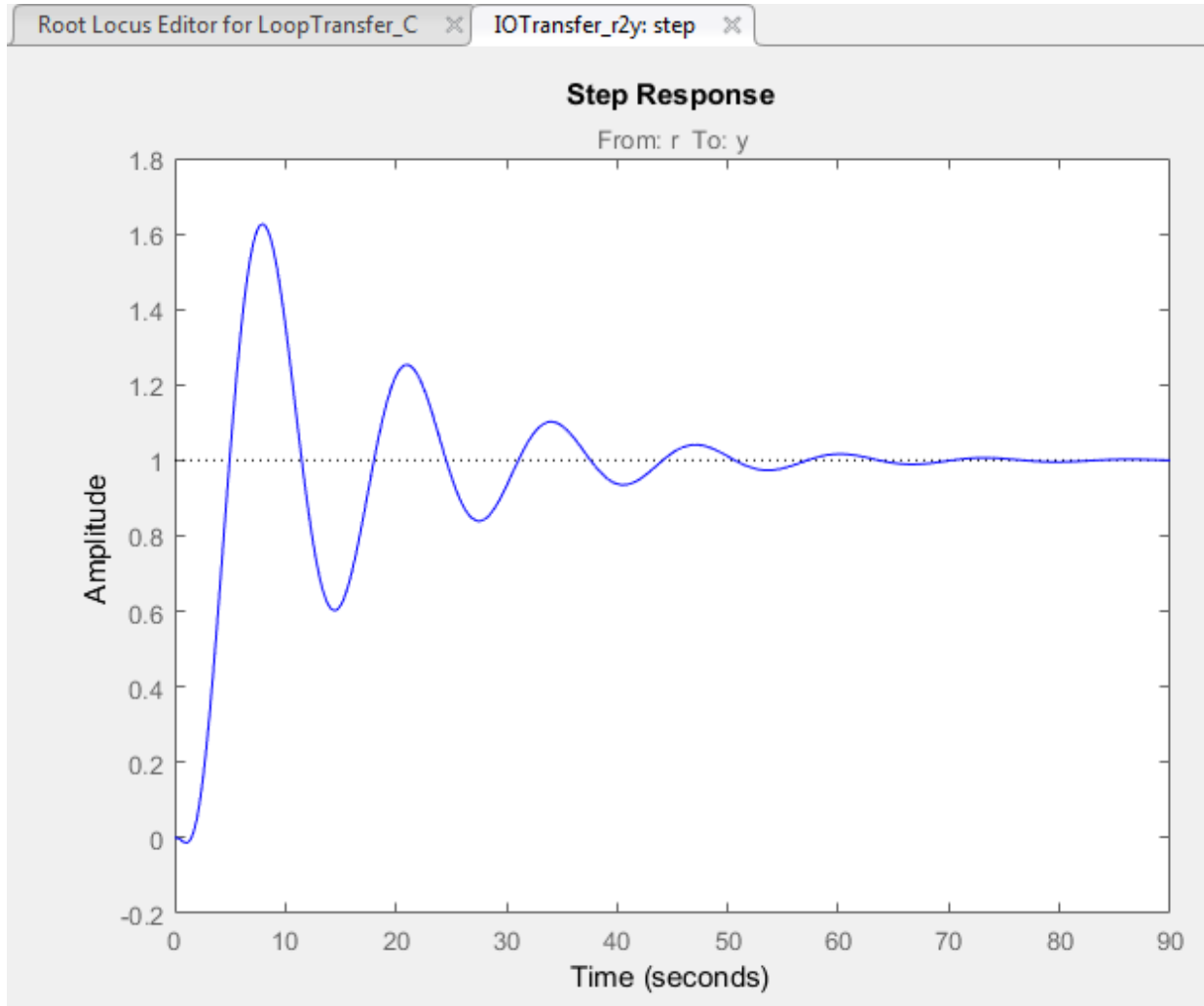
```
controlSystemDesigner('rlocus',open_loopTF)
```

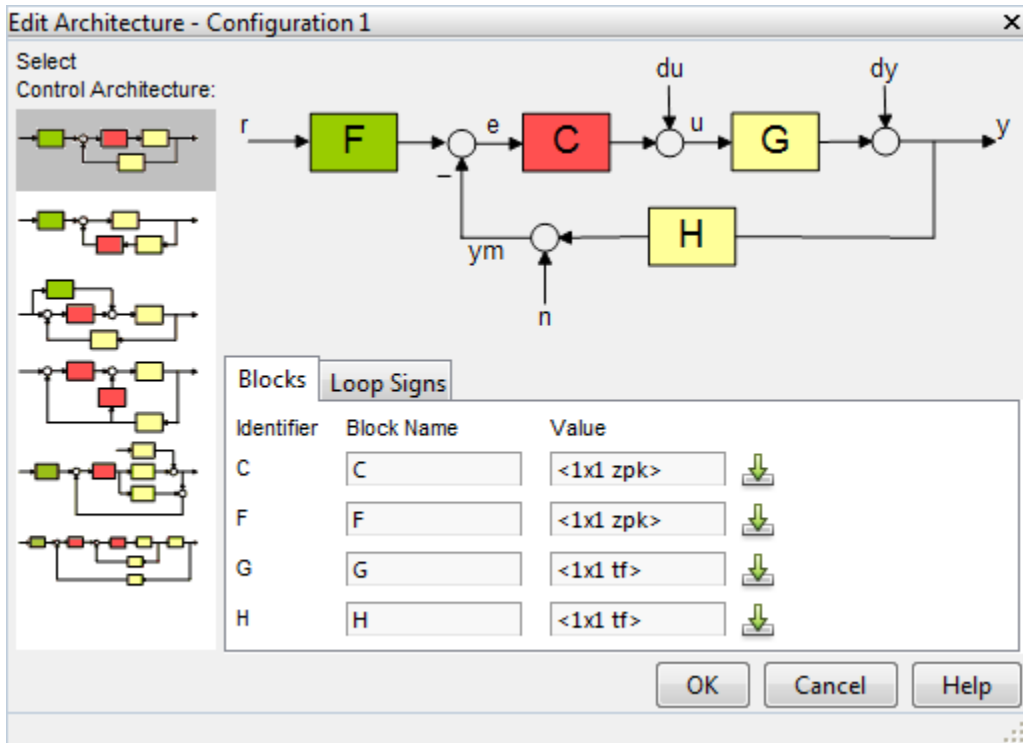
The **Control System Designer** app opens, and a **Root Locus Editor** is displayed. The app lets you design controllers for single-input, single-output (SISO) systems in

MATLAB and Simulink. For more information, see the “Classical Control Design” (Control System Toolbox) category.

The app also displays the step response plot of the system. The plot shows the response of the closed-loop system from r (input to the prefilter, F) to y (output of the plant model, G).



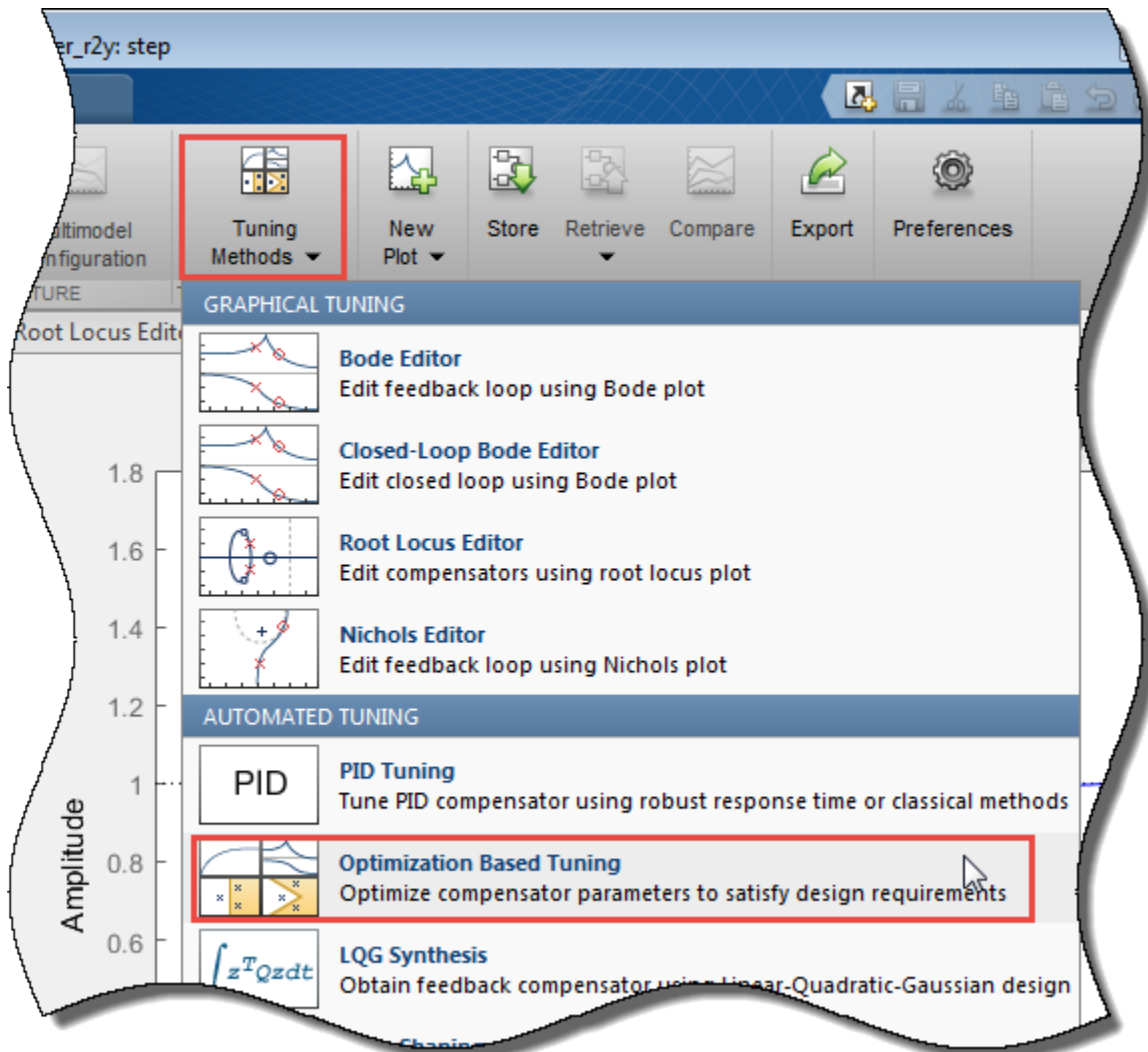
To choose the architecture for the control system you are designing, in the app click **Edit Architecture**. This example uses the default architecture. In this system, the plant model, G , is the open-loop transfer function `open_loopTF`. The prefilter, F , and the sensor, H , are set to 1, and the compensator, C , is the compensator that is designed using response optimization methods.



Open Optimization Based Tuning Method

There are several possible methods for designing a SISO system; this example uses an automated approach that uses response optimization methods.

To create a response optimization task, in the **Tuning Methods** drop-down list, select Optimization Based Tuning.

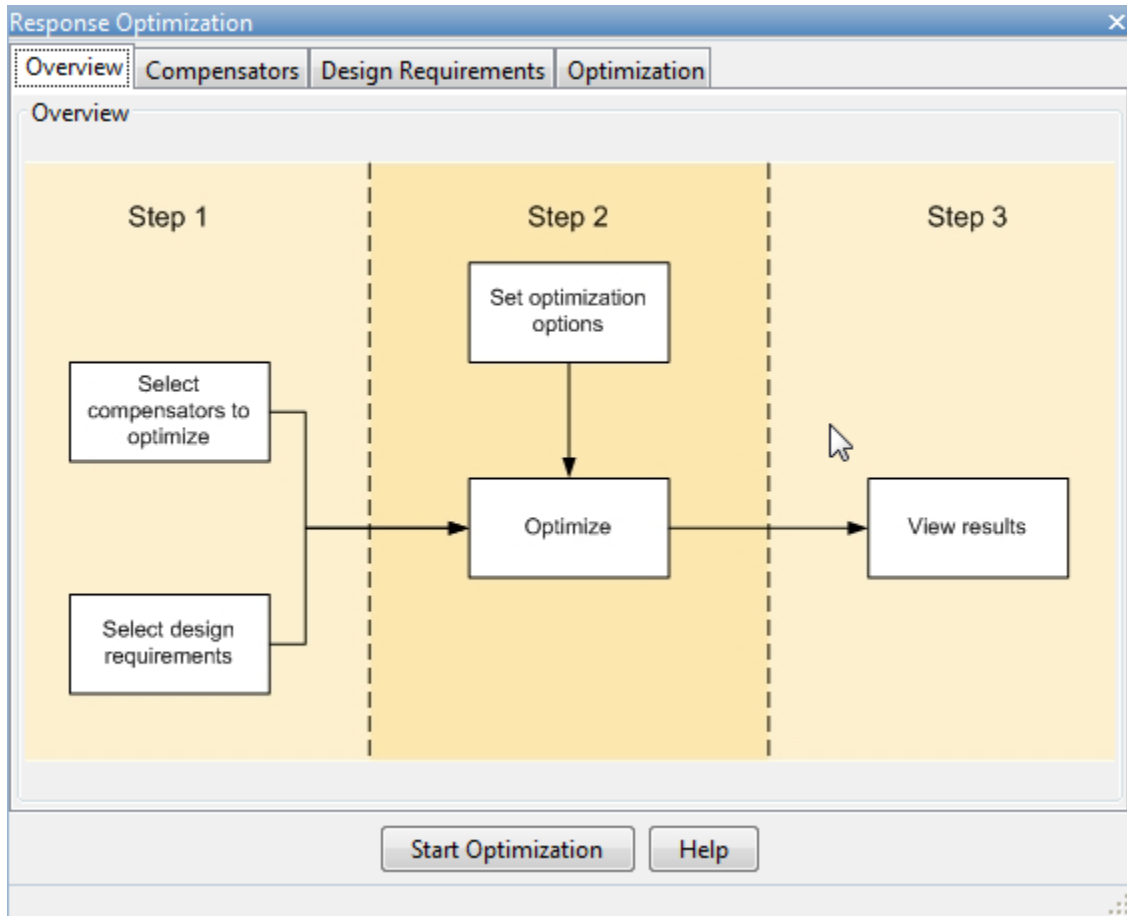


The Response Optimization window has four tabs. Except for the first tab, each tab corresponds to a step in the response optimization process:

- **Overview** — Schematic diagram of the response optimization process.

- **Compensators** — Select and configure the compensator elements that you want to tune. See “Select Tunable Compensator Elements” on page 5-50.
- **Design requirements** — Select the design requirements that you want the system to meet after tuning the compensator elements. See “Add Design Requirements” on page 5-51.
- **Optimization** — Configure optimization options, and view the progress of the response optimization. See “Optimize the System Response” on page 5-60.

Note When optimizing responses in the app, you cannot add uncertainty to parameters or compensator elements.



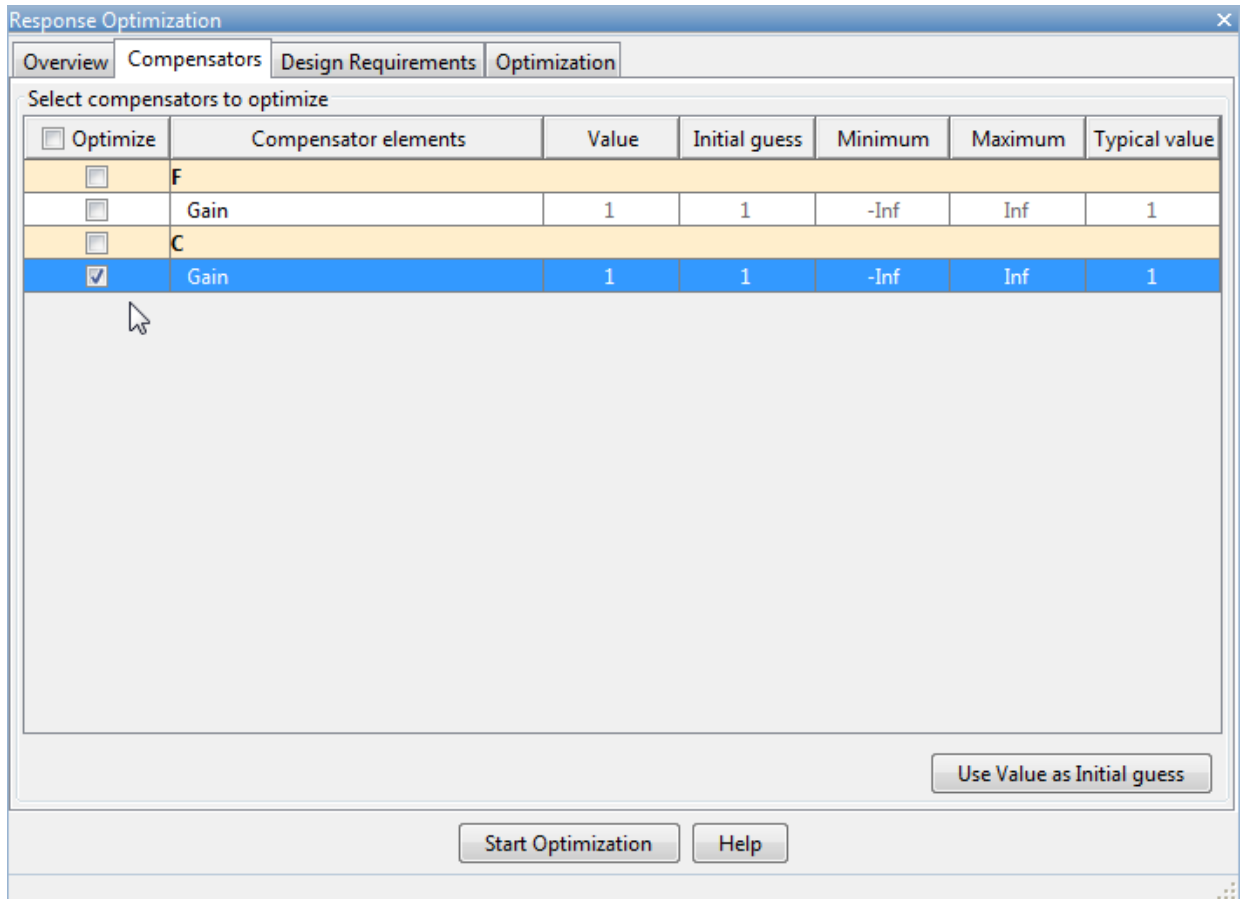
Select Tunable Compensator Elements

You can tune compensator elements or parameters within compensators in your system to meet the design requirements you specify.

To specify the compensator elements to tune:

- 1 In the Response Optimization window, select the **Compensators** tab.
- 2 In the **Compensators** tab, select the check boxes in the **Optimize** column that correspond to the compensator elements to tune.

In this example, select **Gain** in the compensator **C**.



Add Design Requirements

You can use both frequency-domain and time-domain design requirements to tune parameters in a control system.

This example uses the design specifications described in “Design Requirements” on page 5-42. Create design requirements to meet these specifications:

- “Settling Time Design Requirement” on page 5-52

- “Overshoot Design Requirement” on page 5-54
- “Rise Time Design Requirement” on page 5-55
- “Actuator Limit Design Requirement” on page 5-57

After you add the design requirements, you can select a subset of requirements for controller design, as described in “Select the Design Requirements to Use During Response Optimization” on page 5-60. In the **Design requirements** tab of the Response Optimization window, you can create design requirements and select the requirements you want to use for optimization.

Settling Time Design Requirement

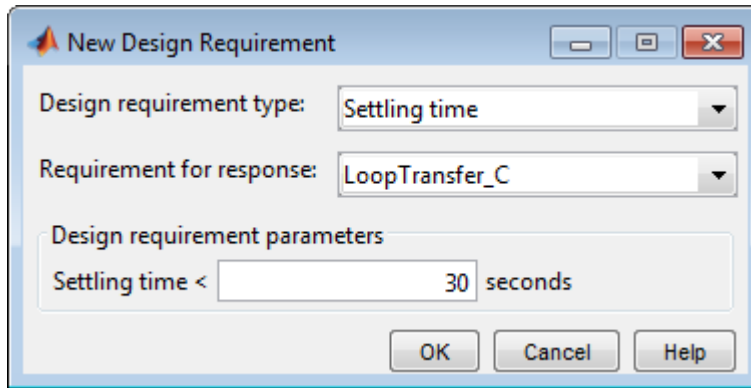
The first design requirement is to have a settling time of 30 seconds or less. This specification can be represented on a root locus diagram as a constraint on the real parts of the poles of the open-loop system.

To add the settling time design requirement:

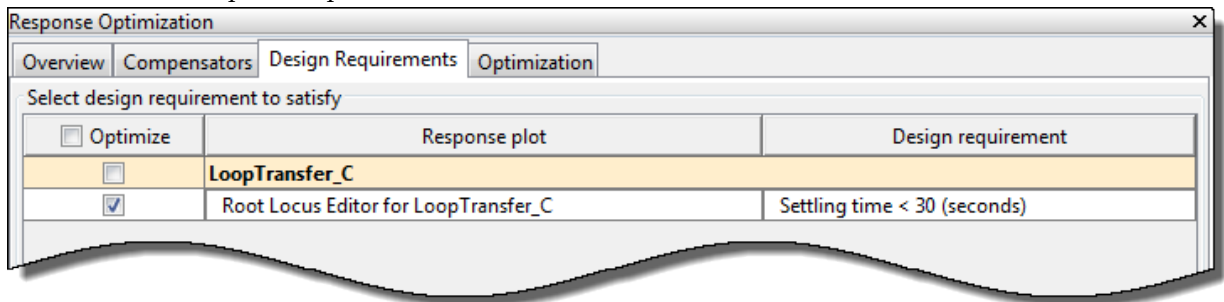
- 1 In the **Design requirements** tab, click **Add new design requirement**. A New Design Requirement dialog box opens.

In this dialog box, you can specify new design requirements, and add them to a new or existing plot.

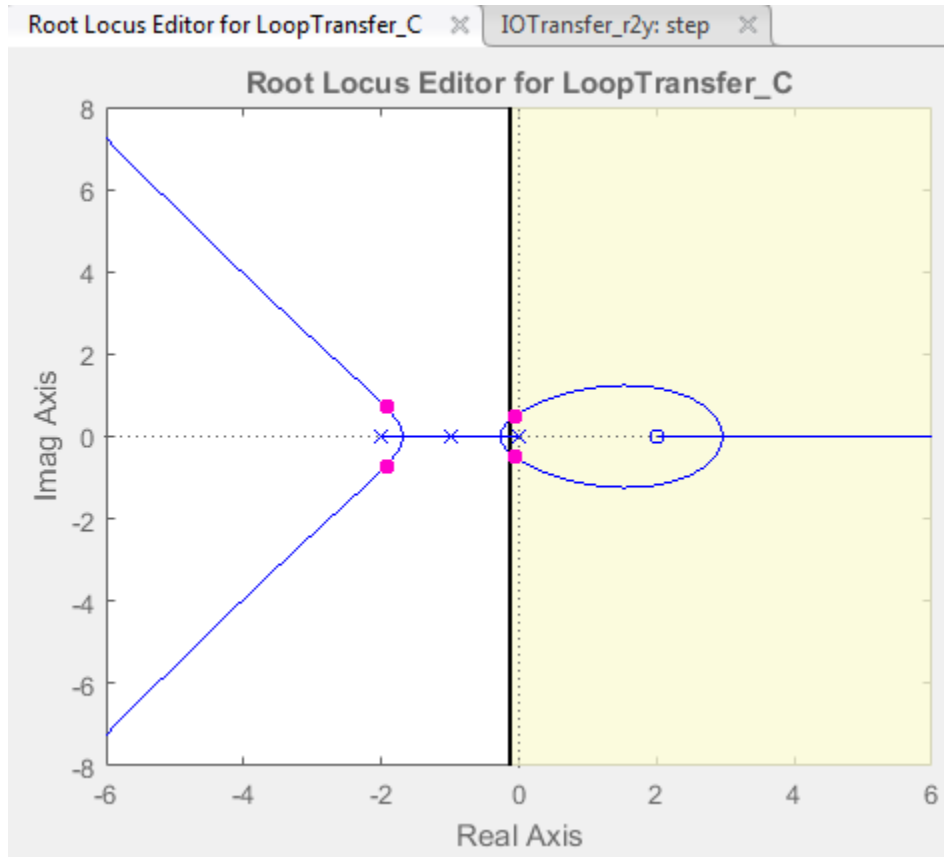
- 2 Add a design requirement to the existing root locus diagram.
 - a In the **Design requirement type** drop-down list, select `Settling time`.
 - b In the **Requirement for response** drop-down list, select `LoopTransfer_C`.
 - c Specify **Settling time** as 30 seconds.
 - d Click **OK**.



The settling time design requirement is listed in the **Design Requirements** tab of the Response Optimization window.



In the app, the design requirement appears on the root locus plot as a vertical line.



Overshoot Design Requirement

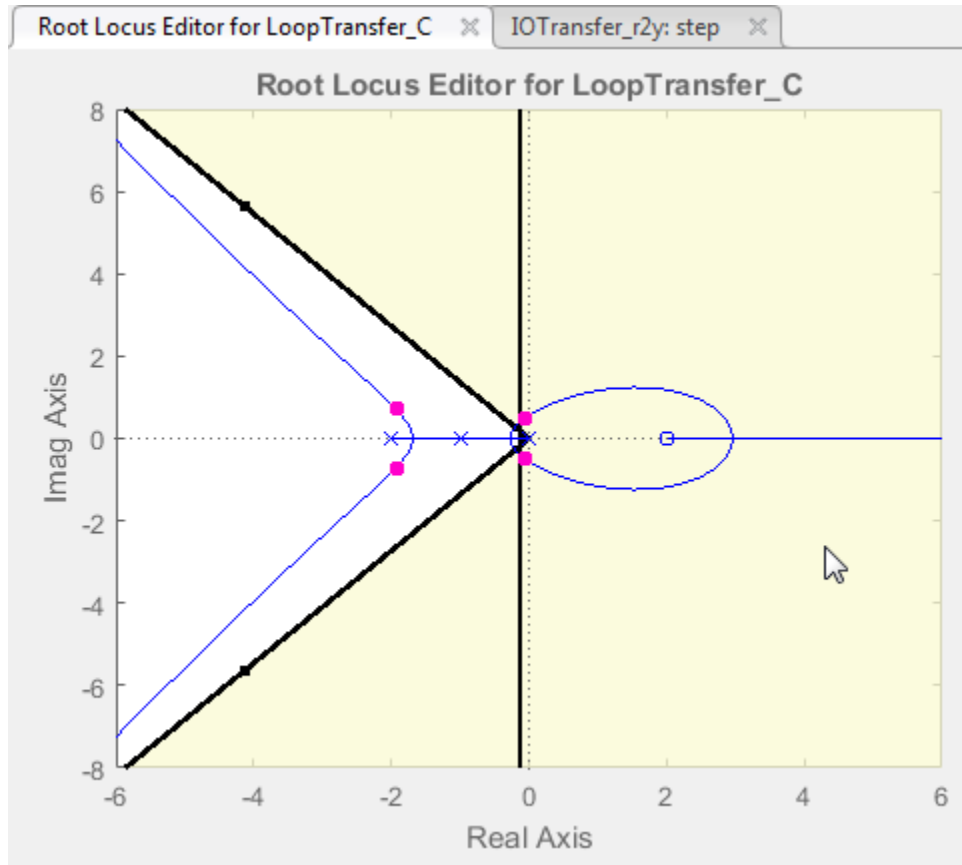
The second design requirement is to have a percentage overshoot of 10% or less. This requirement is related to the damping ratio on a root locus diagram. In addition to adding a design requirement with the **Add new design requirement** button, you can also right-click directly on the plots to add the requirement.

To add this design requirement:

- 1 In the **Control System Designer** app, right-click within the white space of the root-locus diagram. Select **Design Requirements > New** to open the New Design Requirement dialog box.

- 2 In the **Design requirement type** drop-down list, select Percent overshoot.
- 3 Specify **Percent overshoot** as 10.
- 4 Click **OK**.

In the app, the design requirement appears on the root-locus plot as two lines radiating at an angle from the origin.

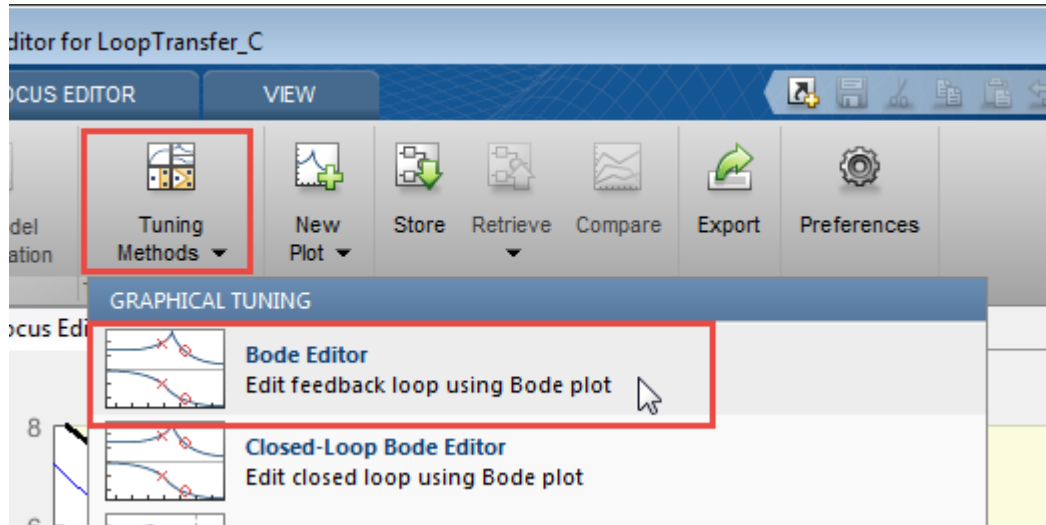


Rise Time Design Requirement

The third design requirement is to have a rise time of 10 seconds or less. This requirement corresponds to a lower limit on a Bode Magnitude diagram.

To add this design requirement:

- 1 In the app, in the **Tuning Methods** drop-down list, select **Bode Editor**.

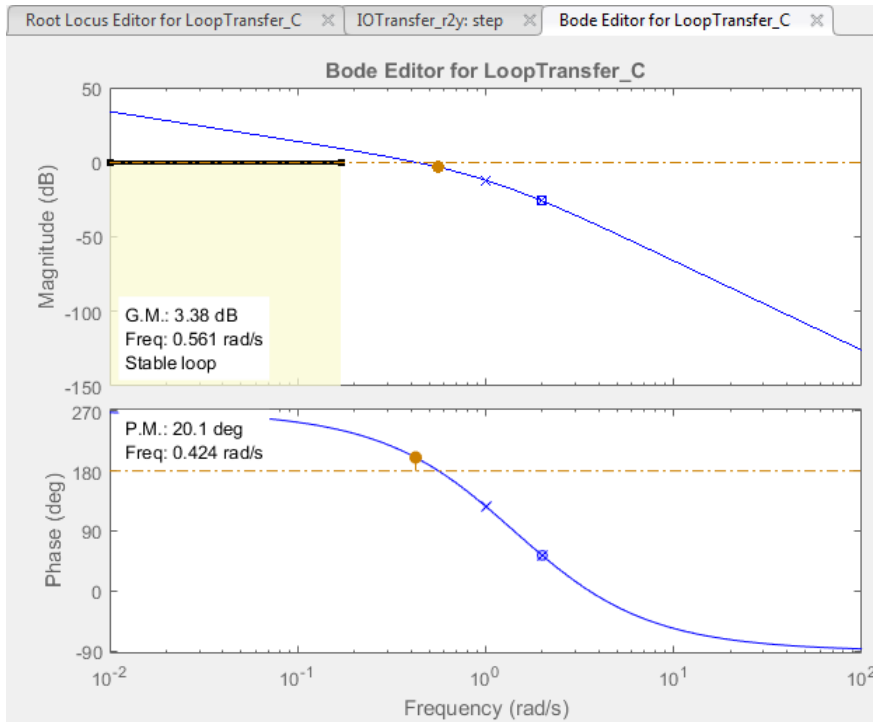


- 2 In the Select Response to Edit dialog box, specify **Select Response to Edit** as `LoopTransfer_C`, and click **Plot**.

A Bode plot is displayed in a **Bode Editor**.

- 3 Right-click within the white space of the open-loop Bode plot, and select **Design Requirements** > **New**, to open the New Design Requirement dialog box.
- 4 Specify the design requirement to represent the rise time, and add it to the new Bode plot.
 - a In the **Design requirement type** drop-down list, select `Lower gain limit`.
 - b Specify the **Frequency** range as `1e-2` to `0.17`.
 - c Specify the **Magnitude** range as `0` to `0`.
 - d Click **OK**.

The design requirement appears on the plot as a horizontal line.



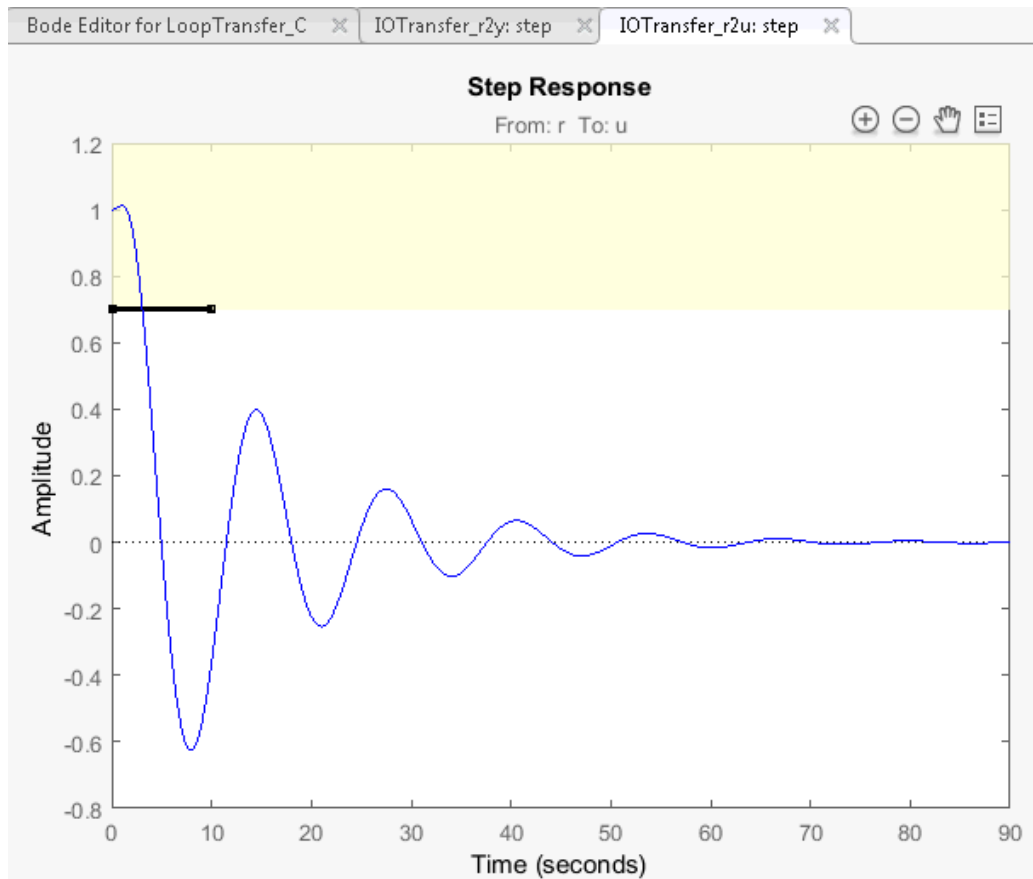
Actuator Limit Design Requirement

The fourth design requirement is to limit the actuator signal to within ± 0.7 .

To add this design requirement:

- 1 In the Response Optimization window, in the **Design requirements**, click **Add new design requirement**. A New Design Requirement dialog box opens.
- 2 Create a time-domain design requirement to represent the upper limit on the actuator signal, and add it to a new step response plot:
 - a In the **Design requirement type** drop-down list, select Step response upper amplitude limit.
 - b In the **Requirement for response** drop-down list, select IOTransfer_r2u.
 - c Specify the **Time** range as 0 to 10.

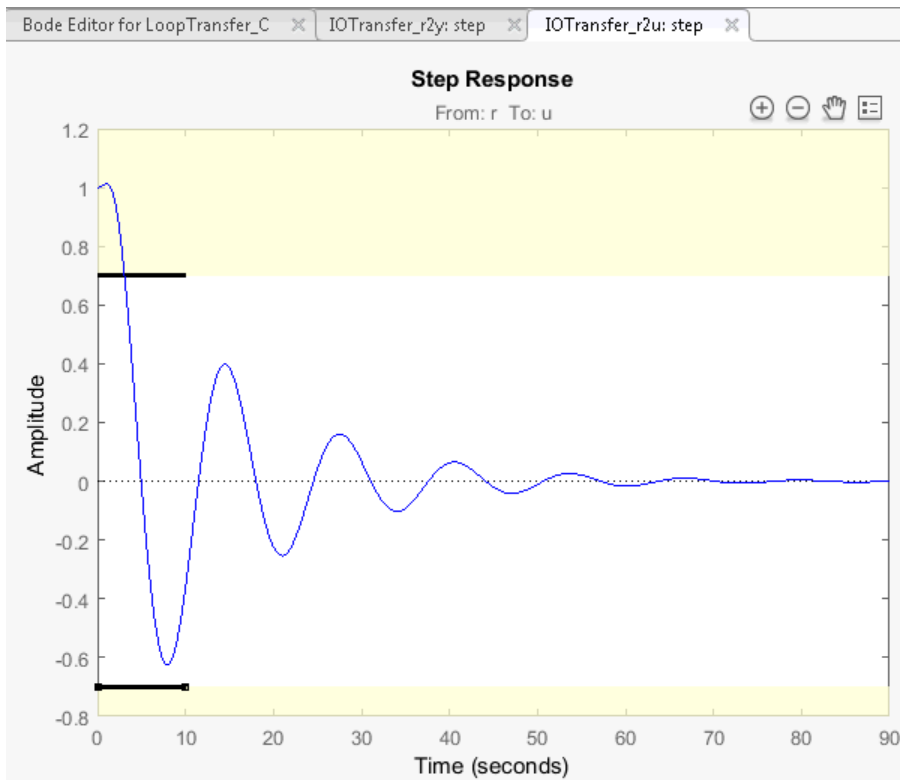
- d Specify the **Amplitude** range as 0.7 to 0.7.
- e Click **OK**. A second step response plot for the closed-loop response from r to u is generated in the app. The plot contains a horizontal line representing the upper limit on the actuator signal.
- f To extend this limit for all times (to $t = \infty$), right-click in the yellow shaded area, and select **Extend to inf**.



To add the corresponding design requirement for the lower limit on the actuator signal:

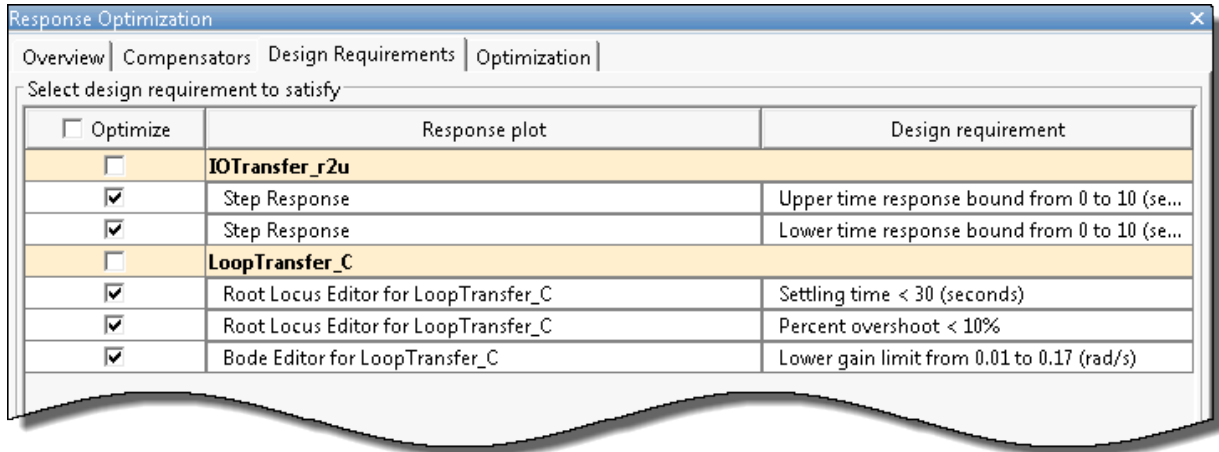
- 1 In the Response Optimization window, in the **Design requirements**, click **Add new design requirement**. A New Design Requirement dialog box opens.

- 2 Create a time-domain design requirement to represent the lower limit on the actuator signal, and add it to the step response plot:
 - a In the **Design requirement type** drop-down list, select Step response lower amplitude limit.
 - b In the **Requirement for response** drop-down list, select IOTransfer_r2u.
 - c Specify the **Time** range as 0 to 10.
 - d Specify the **Amplitude** range as -0.7 to -0.7 .
 - e Click **OK**. The step response plot now contains a second horizontal line representing the lower limit on the actuator signal.
 - f To extend this limit for all times (to $t = \infty$), right-click in the yellow shaded area of the design requirement, and select **Extend to inf**.



Select the Design Requirements to Use During Response Optimization

The table in the **Design requirements** tab lists all the specified design requirements. Select the design requirements you want to use in the response optimization. This example uses all the current design requirements.

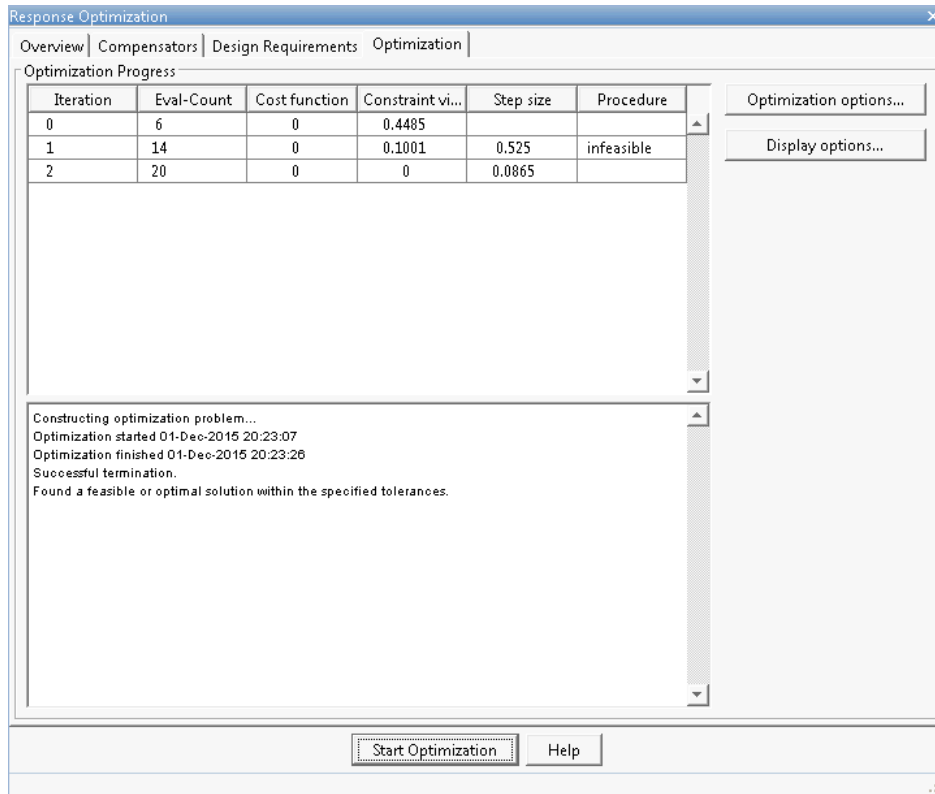


Optimize the System Response

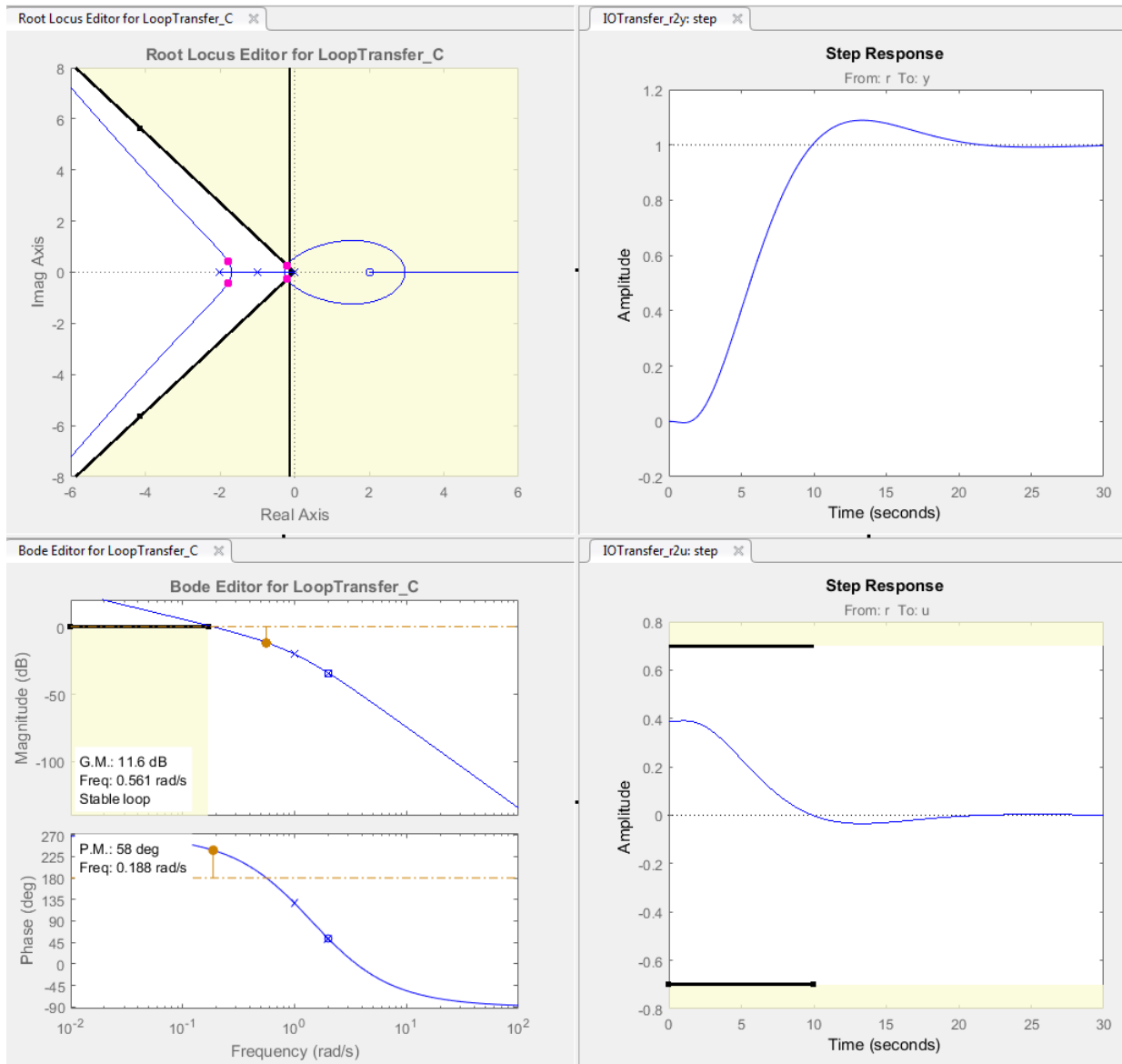
After you select the compensator elements to tune and add design requirements, you can optimize the system response.

To optimize the response of the system, in the **Optimization** tab of the Response Optimization window, click **Start Optimization**.

The **Optimization** tab displays the progress of the optimization.



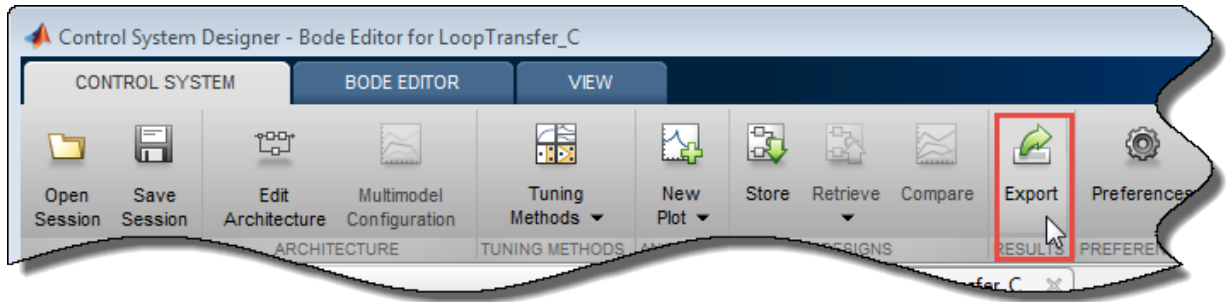
The status message indicates that the optimization solver found a solution that meets the design requirements within the tolerances. Verify that the design requirements are satisfied.



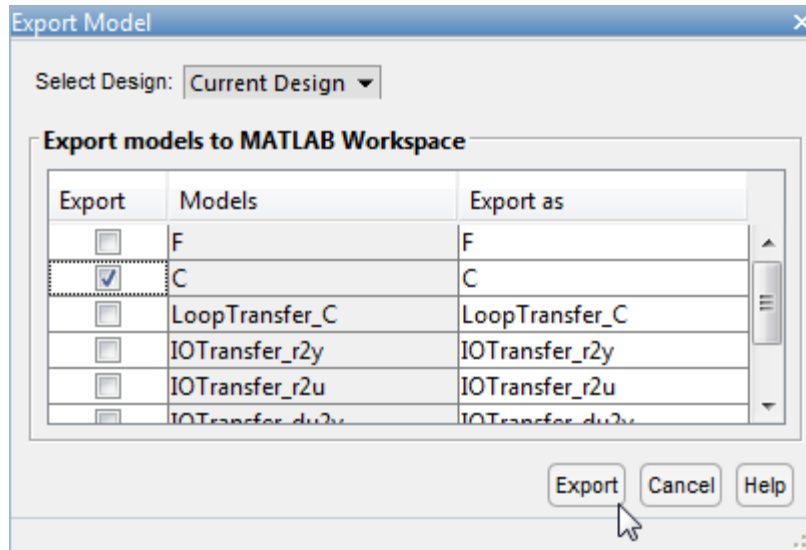
Create and Display the Closed-Loop System

After designing a compensator, you can export it to the MATLAB workspace and create a model of the full closed-loop system. To export the tuned compensator:

- 1 In the app, select **Export**.



- 2 In the Export Model dialog box, select **C**, the compensator you designed, and click **Export**.



At the command line, enter the following command to create the closed-loop system, CL, from the open-loop transfer function, open_loopTF, and the compensator, C:

```
CL = feedback(C*open_loopTF,1)
```

The following model is returned:

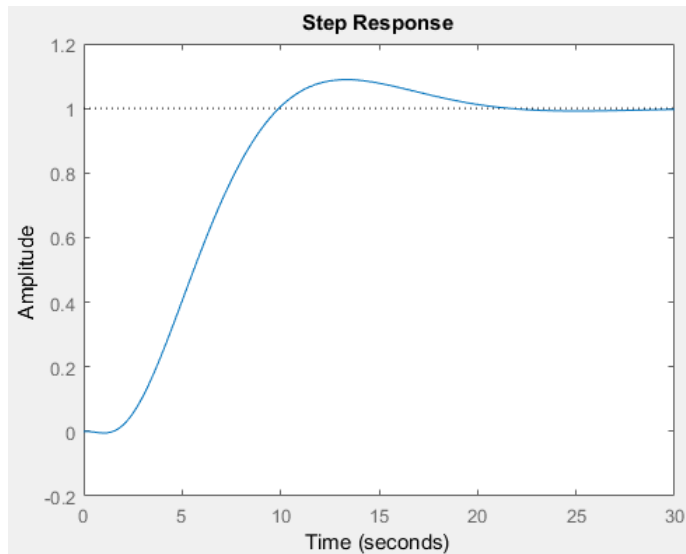
CL =

$$\frac{-0.19414 (s-2)}{(s^2 + 0.409s + 0.1136) (s^2 + 3.591s + 3.418)}$$

Continuous-time zero/pole/gain model.

To create a step response plot of the closed-loop system, enter the following command.

```
step(CL);
```



See Also

More About

- “Design Optimization-Based Controllers for LTI Systems” on page 5-41
- “Time-Domain Simulations in Control System Designer App” on page 5-40

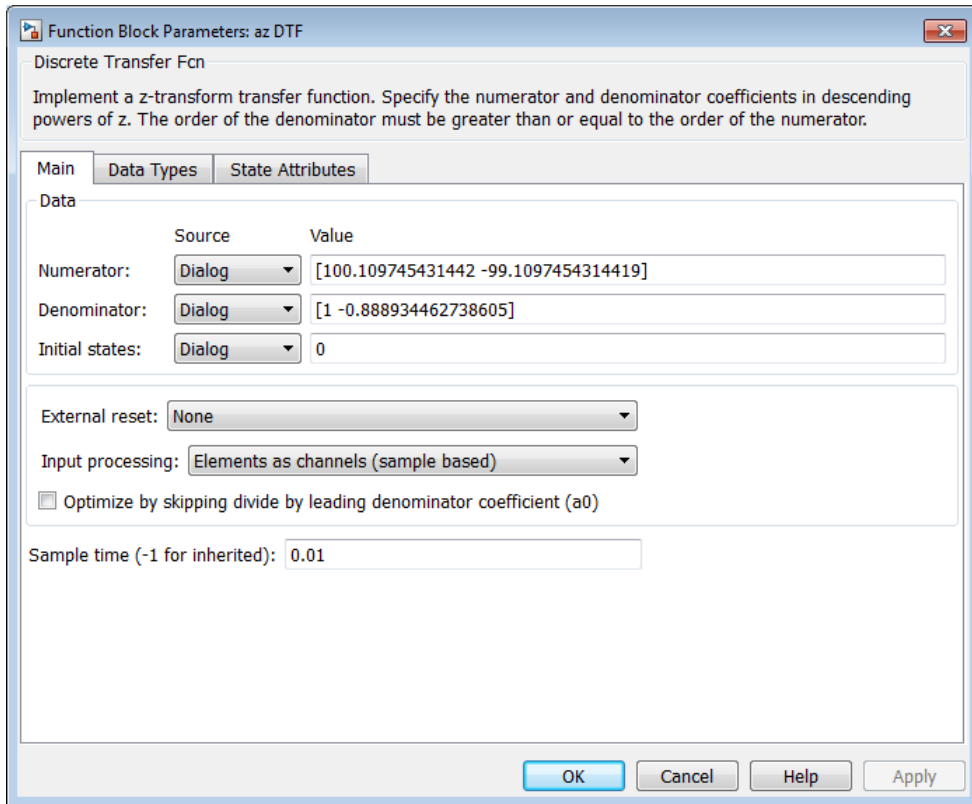
Design Linear Controllers for Simulink Models

When you have Control System Toolbox and Simulink Control Design software, you can perform frequency-domain optimization of Simulink models.

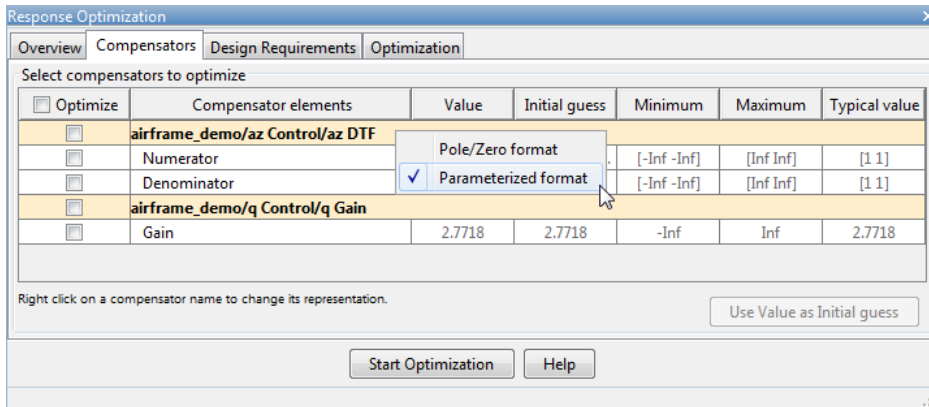
You can use Simulink Control Design software to configure the **Control System Designer** app with compensators, inputs, outputs, and loops computed from a Simulink model. For more information, see topics in the “Classical Control Design” (Simulink Control Design) category.

After you configure the **Control System Designer** app, use Simulink Design Optimization software to optimize the controller parameters of the linearized Simulink model. For an example of optimization-based control design for a model linearized using Simulink Control Design software, see “Design Optimization-Based PID Controller for Linearized Simulink Model (GUI)”.

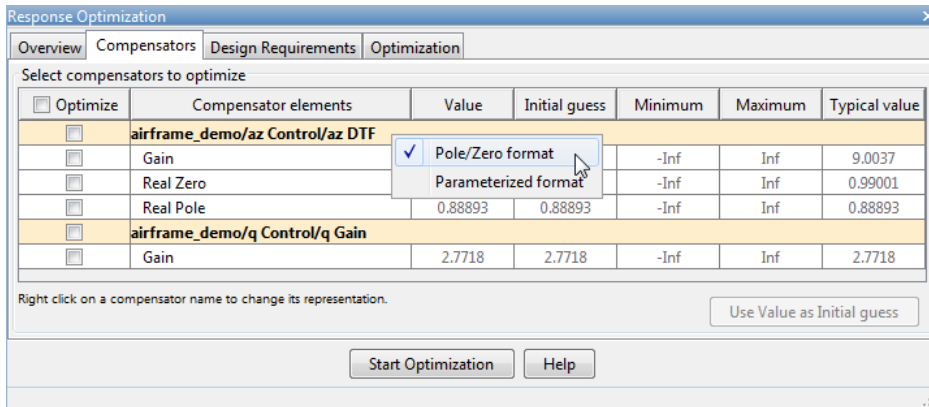
When tuning compensators derived from Simulink Control Design software, the tuning of compensators from a Simulink model is done through the masks of the Simulink blocks representing each compensator. When selecting parameters to optimize, you can tune the compensator in the pole, zero, or gain format, or in a format consistent with the Simulink block mask.



Mask of a Simulink Compensator Block



Response Optimization Compensators Tab — Parameterized Format



Response Optimization Compensators Tab — Pole/Zero Format

Note You cannot change the compensator format if the compensator is not a Simulinkblock.

See Also

Related Examples

- “Design Optimization-Based PID Controller for Linearized Simulink Model (GUI)”

Lookup Tables

- “What are Adaptive Lookup Tables?” on page 6-2
- “How to Estimate Lookup Table Values” on page 6-5
- “Estimate Constrained Values of a Lookup Table” on page 6-6
- “Estimate Lookup Table Values from Data” on page 6-21
- “Building Models Using Adaptive Lookup Table Blocks” on page 6-35
- “Selecting an Adaptation Method” on page 6-39
- “Model Engine Using n-D Adaptive Lookup Table” on page 6-41
- “Using Adaptive Lookup Tables in Real-Time Environment” on page 6-55
- “Design Optimization Using Lookup Table Requirements for Gain Scheduling (Code)” on page 6-56
- “Design Optimization Using Lookup Table Requirements for Gain Scheduling (GUI)” on page 6-73

What are Adaptive Lookup Tables?

Lookup Tables

Lookup tables store numeric data in a multidimensional array format. In the simpler two-dimensional case, lookup tables can be represented by matrices. Each element of a matrix is a numerical quantity, which can be precisely located in terms of two indexing variables. At higher dimensions, lookup tables can be represented by multidimensional matrices, whose elements are described in terms of a corresponding number of *indexing variables*.

Lookup tables provide a means to capture the dynamic behavior of a physical (mechanical, electronic, software) system. The behavior of a system with M inputs and N outputs can be approximately described by using N lookup tables, each consisting of an array with M dimensions.

You usually generate lookup tables by experimentally collecting or artificially creating the input and output data of a system. In general, you need as many indexing parameters as the number of input variables. Each indexing parameter may take a value within a predetermined set of data points, which are called the *breakpoints*. The set of all breakpoints corresponding to an indexing variable is called a *grid*. Thus, a system with M inputs is gridded by M sets of breakpoints. The software uses the breakpoints to locate the array elements, where the output data of the system are stored. For a system with N outputs, the software locates the N array elements and then stores the corresponding data at these locations.

After you create a lookup table using the input and output measurements as described previously, you can use the corresponding multidimensional array of values in applications without having to remeasure the system outputs. In fact, you need only the input data to locate the appropriate array elements in the lookup table because the software reads the approximate system output from the data stored at these locations. Therefore, a lookup table provides a suitable means of capturing the input-output mapping of a *static* system in the form of numeric data stored at predetermined array locations.

Adaptive Lookup Tables

Statically defined lookup tables, as described in “Lookup Tables” on page 6-2, cannot accommodate the *time-varying* behavior (characteristics) of a physical plant. Static

lookup tables establish a permanent and static mapping of input-output behavior of a physical system. Conversely, the behavior of actual physical systems often varies with time due to wear, environmental conditions, and manufacturing tolerances. With such variations, the static mapping of input-output behavior of a plant described by the lookup table may no longer provide a valid representation of the plant characteristics.

Adaptive lookup tables incorporate the time-varying behavior of physical plants into the lookup table generation and maintenance process while providing all of the functionality of a regular lookup table.

The adaptive lookup table receives the input and output measurements of a plant's behavior, which are then used to dynamically create and update the content of the underlying lookup table. In addition to requiring the input data to create the lookup table, the adaptive lookup table also uses the output data of the plant to recalculate the table values. For example, you can collect the output data of the plant by placing sensors at appropriate locations in a physical system.

The software uses the input measurements to locate the array elements by comparing these input values with the breakpoints defined for each indexing variable. Next, it uses the output measurements to recalculate the numeric value stored at these array locations. However, unlike a regular table, which only stores the array data before the actual use of the lookup table, the adaptive table continuously improves the content of the lookup table. This continuous improvement of the table data is referred to as the adaptation process or learning process.

The adaptation process involves statistical and signal processing algorithms to recapture the input-output behavior of the plant. The adaptive lookup table always tries to provide a valid representation of the plant dynamics even though the plant behavior may be time varying. The underlying signal processing algorithms are also robust against reasonable measurement noise and they provide appropriate filtering of noisy output measurements.

See Also

Adaptive Lookup Table (1D Stair-Fit) | Adaptive Lookup Table (2D Stair-Fit) | Adaptive Lookup Table (nD Stair-Fit)

Related Examples

- “Model Engine Using n-D Adaptive Lookup Table” on page 6-41

More About

- “About Lookup Table Blocks” (Simulink)
- “Building Models Using Adaptive Lookup Table Blocks” on page 6-35

How to Estimate Lookup Table Values

You can use lookup table Simulink blocks to approximate a system's behavior, as described in “About Lookup Table Blocks” (Simulink). After you build your system using lookup tables, you can use Simulink Design Optimization software to estimate the table values from measured I/O data.

Estimating lookup table values is an example of estimating parameters which are matrices or multi-dimensional arrays. The workflow for estimating parameters of a lookup table consist of the following tasks:

- 1 Creating a Simulink model using lookup table blocks.
- 2 Importing the measured input and output (I/O) data from which you want to estimate the table values.
- 3 Analyzing and preparing the I/O data for estimation.
- 4 Estimating the lookup table values.
- 5 Validating the estimated table values using a validation data set.

See Also

Related Examples

- “Estimate Lookup Table Values from Data” on page 6-21
- “Estimate Constrained Values of a Lookup Table” on page 6-6

Estimate Constrained Values of a Lookup Table

In this section...

“Objectives” on page 6-6

“About the Data” on page 6-6

“Lookup Table Output” on page 6-6

“Estimate the Monotonically Increasing Table Values Using Default Settings” on page 6-8

“Validate the Estimation Results” on page 6-16

Objectives

This example shows how to estimate constrained values of a lookup table in the Parameter Estimation tool. Apply monotonically increasing constraints to the lookup table output values, and use the Parameter Estimation tool to estimate the table values.

About the Data

In this example, use `lookup_increasing.mat`, which contains the measured I/O data for estimating the lookup table values. The MAT-file includes the following variables:

- `xdata1` — Input data consisting of 602 uniformly sampled data points in the range $[-5, 5]$.
- `ydata1` — Output data corresponding to the input data samples.
- `time1` — Time vector.

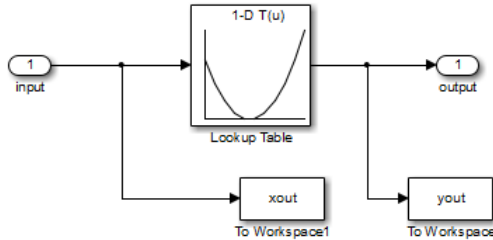
Use the I/O data to estimate monotonically increasing output values of the lookup table in the `lookup_increasing` Simulink model.

Lookup Table Output

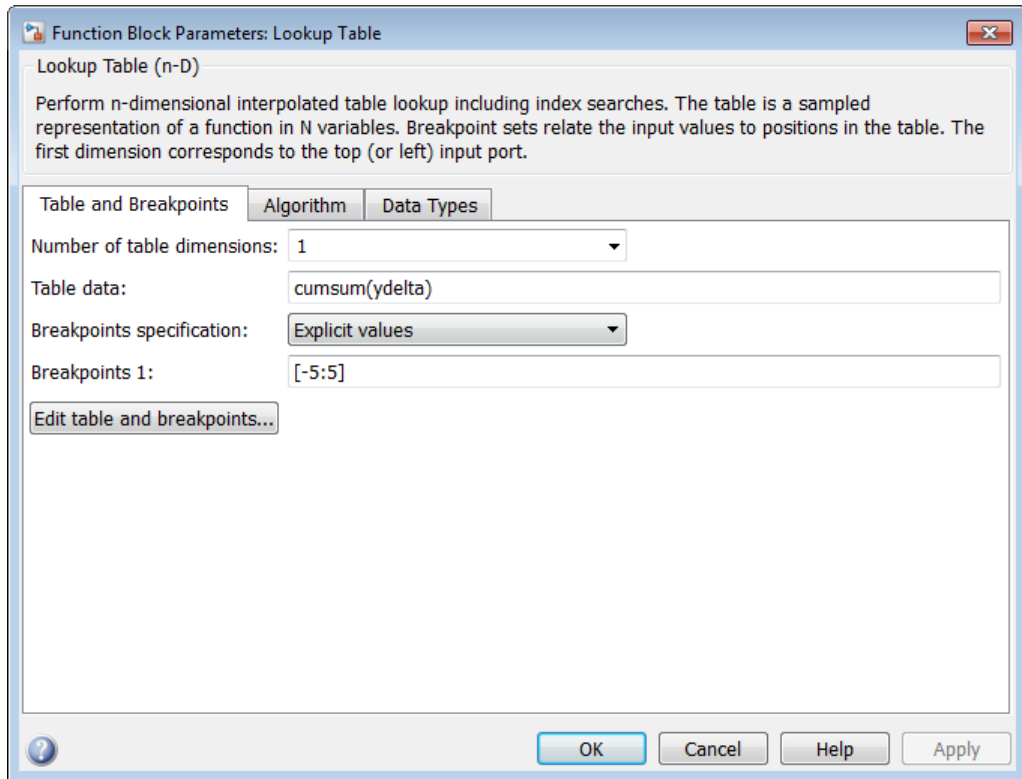
- 1 Open the lookup table model by typing the following command at the MATLAB prompt:

```
lookup_increasing
```

This command opens the Simulink model, and loads the estimation data in the MATLAB workspace.



- 2 View the table output values by double-clicking the Lookup Table block.



The table contains 11 output values at breakpoints $[-5:5]$, specified in the Function Block Parameters dialog box. To learn more about how to specify the table values, see “Enter Breakpoints and Table Data” (Simulink).

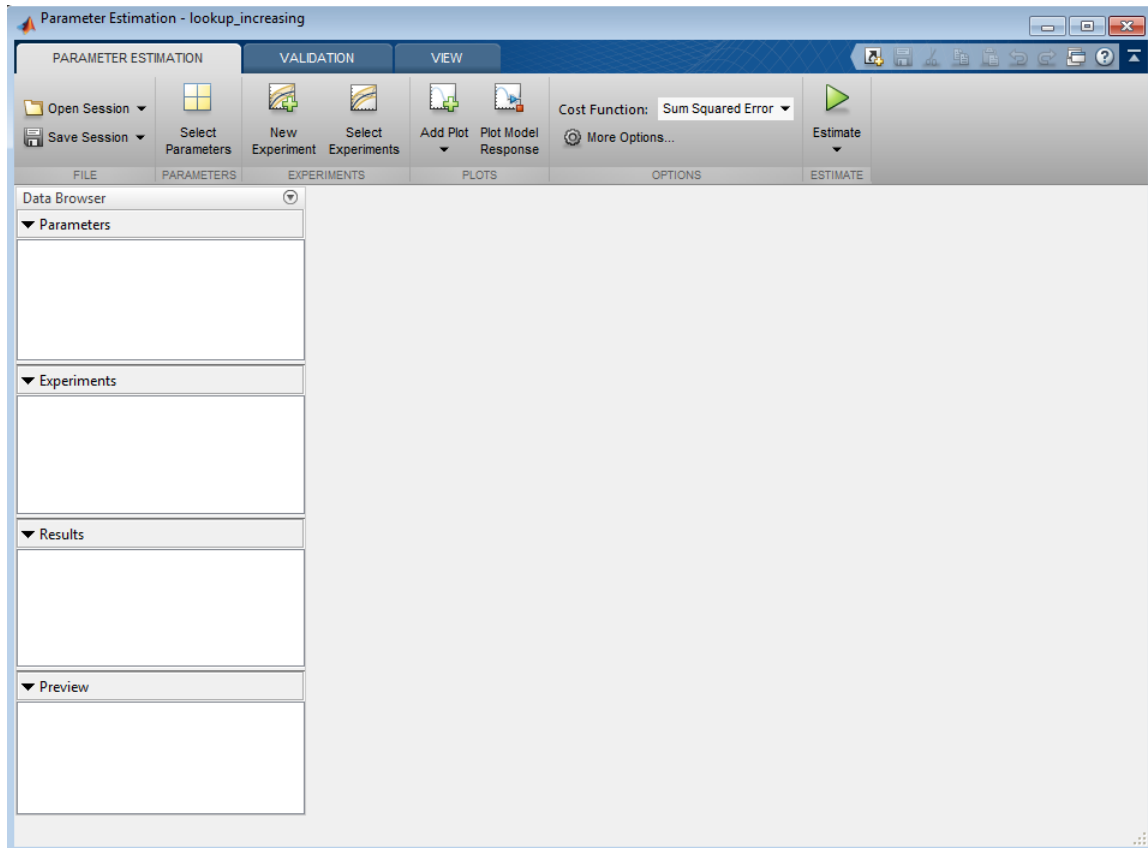
The **Table data** field shows that the table output values are the cumulative sum of the values stored in variable `ydelta`. Thus, if y_n are the 11 table output values, `ydelta` is $(y_1, y_2-y_1, y_3-y_2, \dots, y_{11}-y_{10})$. The initial `ydelta` values are loaded from `lookup_increasing.mat`.

The initial table output values are not monotonically increasing. To ensure monotonically increasing table output values, the difference between adjacent table output values should be positive. To do so, estimate `ydelta` in the Parameter Estimation tool using the measured I/O estimation data, and constrain `ydelta(2:end)` to be positive during estimation.

Estimate the Monotonically Increasing Table Values Using Default Settings

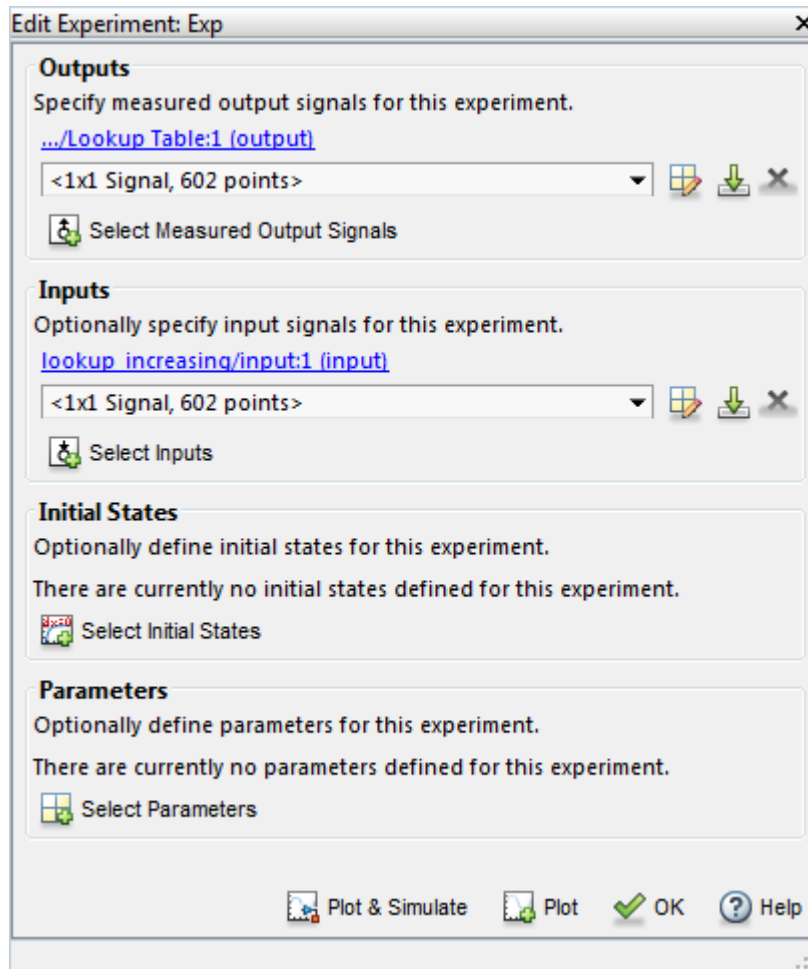
- 1 Open a parameter estimation session.

In the Simulink model, select **Analysis > Parameter Estimation** to open a session with the name `lookup_increasing` in the Parameter Estimation tool.



2 Create an experiment and import the I/O data.

On the **Parameter Estimation** tab, click **New Experiment**. Type [time1, ydata1] in **Outputs** and [time1, xdata1] in **Inputs** of the Edit Experiment dialog box. Click **OK**. A new experiment with name Exp is created in the **Experiments** area of the tool. Rename the experiment EstimationData by right-clicking the default experiment name, Exp, and selecting Rename. For more information, see “Import Data for Parameter Estimation” on page 1-6.

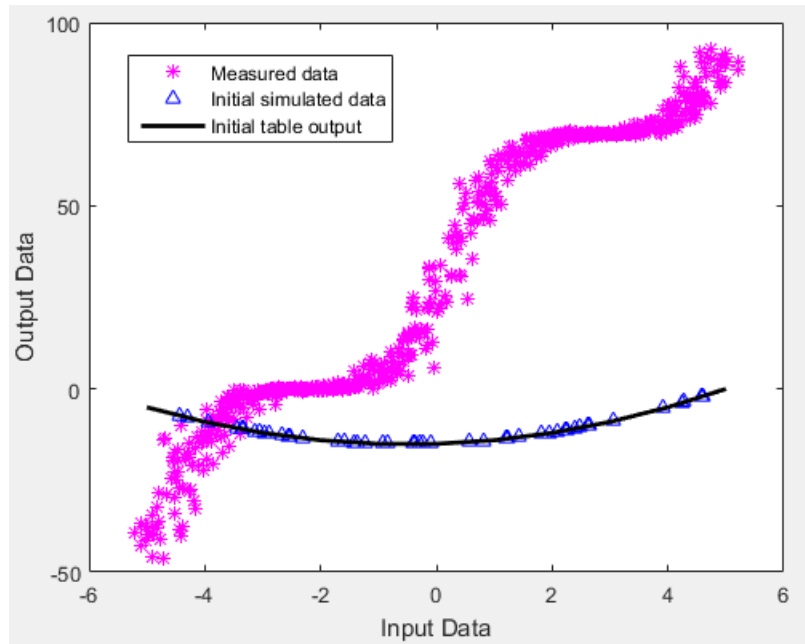


- 3 Run an initial simulation to view the measured data, simulated model values, and the initial table values by typing the following commands at the MATLAB prompt.

```

sim('lookup_increasing')
figure(1); plot(xdata1,ydata1,'m*',xout,yout,'b^')
hold on; plot(-5:5,cumsum(ydelta),'k','LineWidth',2)
xlabel('Input Data'); ylabel('Output Data');
legend('Measured data','Initial simulated data','Initial table output')

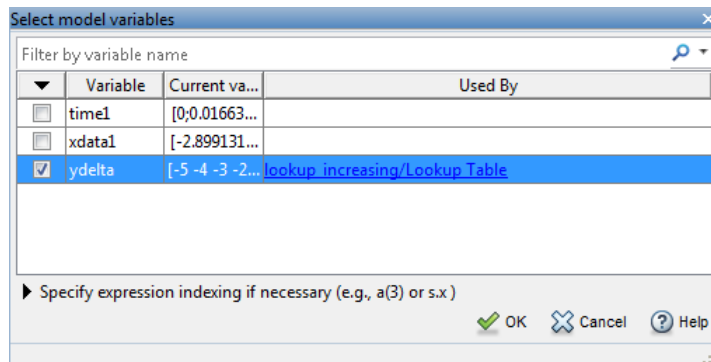
```



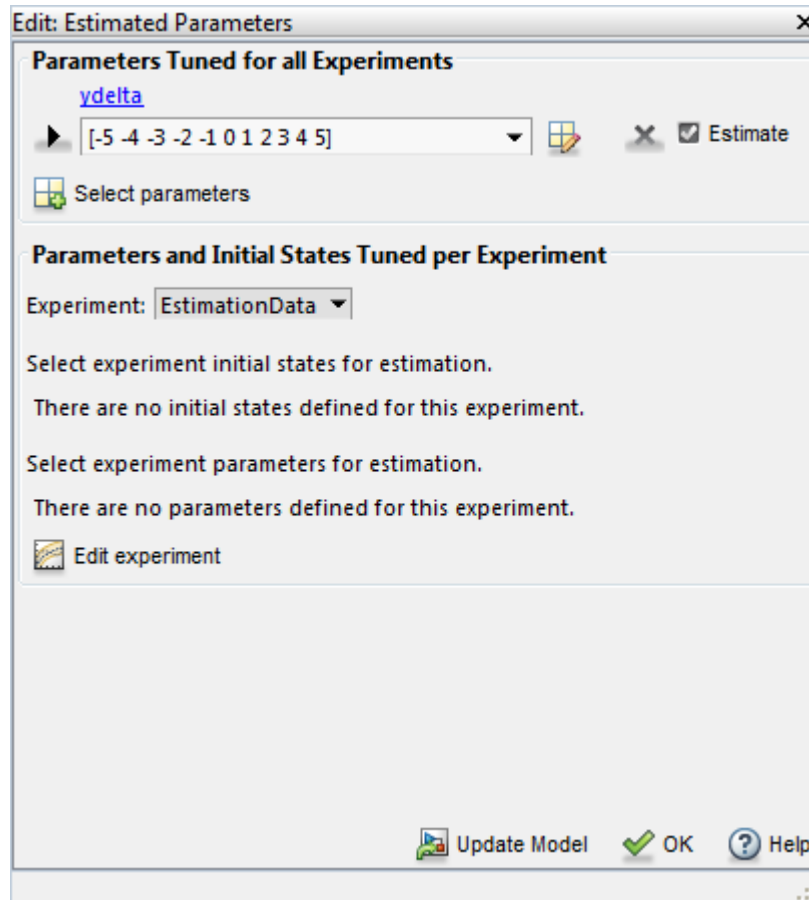
The initial table output values and simulated data do not match the measured data.

4 Select parameter for estimation.

On the **Parameter Estimation** tab, click **Select Parameters**. The Edit: Estimated Parameters dialog box opens. In the **Parameters Tuned for all Experiments** panel, click **Select parameters** to open the Select Model Variables dialog box. Check the box next to `ydelta`, and click **OK**.

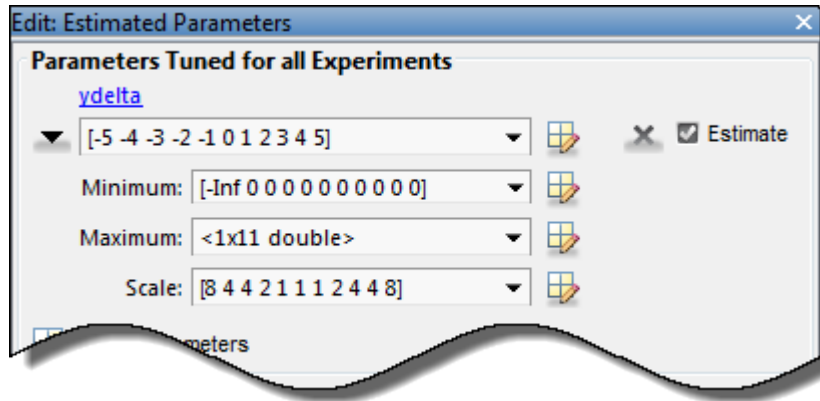


The `ydelta` values are selected for estimation by default in the Edit: Estimated Parameters dialog box.



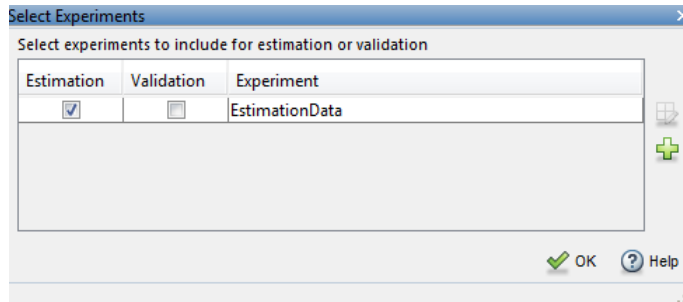
- 5 Apply a monotonically increasing constraint on the table output values. For more details about the table, see “Lookup Table Output” on page 6-6.

In the Edit: Estimated Parameters dialog box, click the arrow next to the `ydelta` values. In the expanded menu, set **Minimum** `ydelta` values to `[-Inf, zeros(1, 10)]`. Thus, while the first value in `ydelta` can be anything, subsequent values which are the difference between adjacent table output values, must be positive.



- 6 Select EstimationData experiment for estimation.

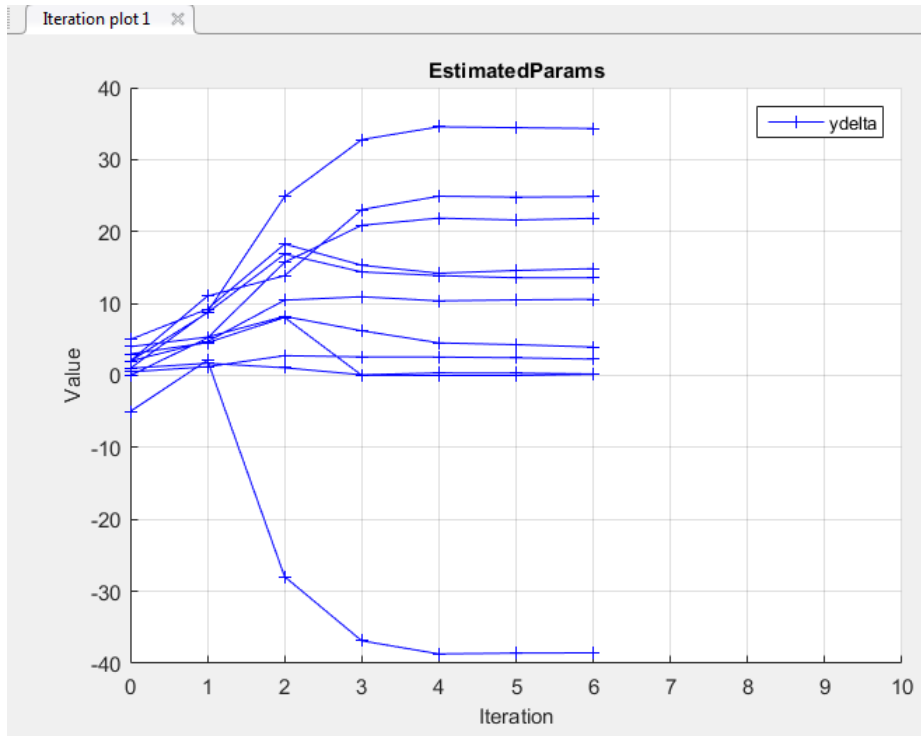
On the **Parameter Estimation** tab, click **Select Experiment**. By default, EstimationData is selected for estimation. If not, check the box under the **Estimation** column, and click **OK**.



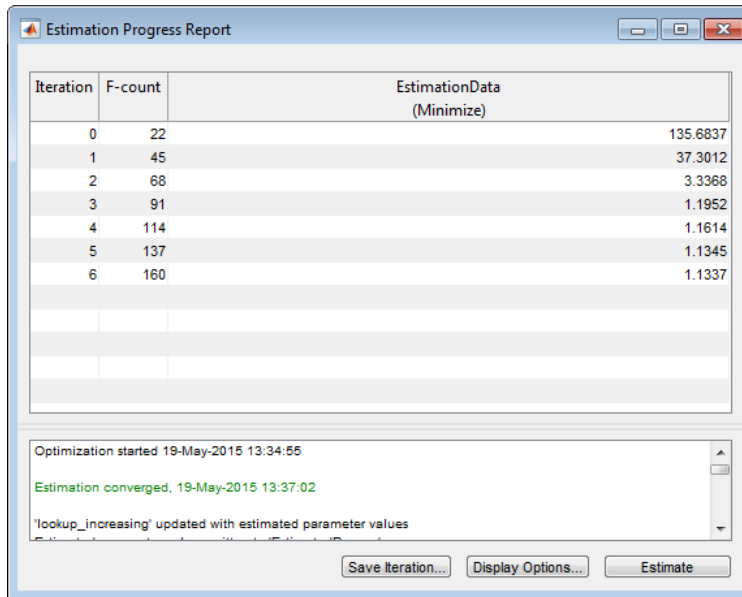
- 7 Estimate the table values using default settings.

On the **Parameter Estimation** tab, click **Estimate**.

The **Parameter Trajectory** plot shows the change in the parameter values at each iteration.



The Estimation Progress Report shows the iteration number, number of times the objective function is evaluated, and value of the cost function at the end of each iteration.



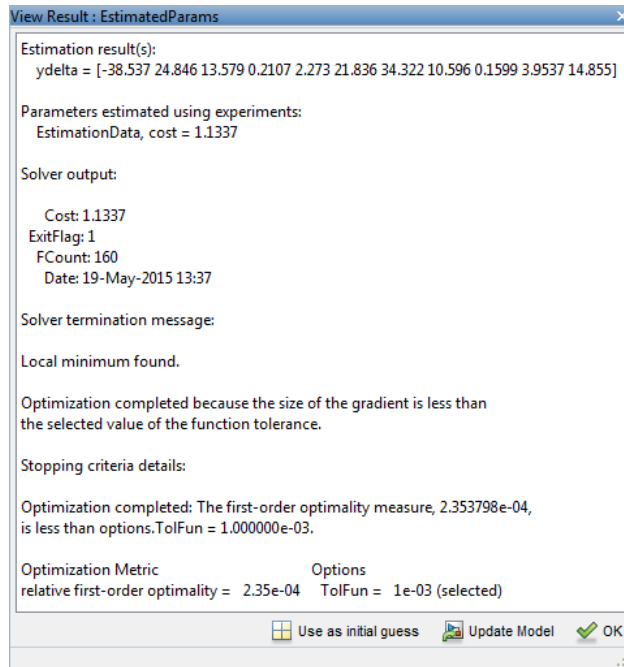
The screenshot shows a window titled "Estimation Progress Report" with a table of iteration data and a log of optimization progress. The table has three columns: "Iteration", "F-count", and "EstimationData (Minimize)". The data shows a decreasing trend in the EstimationData value over six iterations. Below the table, a log shows the optimization starting at 19-May-2015 13:34:55 and converging at 19-May-2015 13:37:02. At the bottom of the window, there are three buttons: "Save Iteration...", "Display Options...", and "Estimate".

Iteration	F-count	EstimationData (Minimize)
0	22	135.6837
1	45	37.3012
2	68	3.3368
3	91	1.1952
4	114	1.1614
5	137	1.1345
6	160	1.1337

Optimization started 19-May-2015 13:34:55
Estimation converged, 19-May-2015 13:37:02
"lookup_increasing" updated with estimated parameter values

Save Iteration... Display Options... Estimate

The estimated parameters are saved in a new variable, `EstimatedParams`, in the **Results** area of the tool. To view the estimated parameters, right-click `EstimatedParams` and select **Open**.



The estimated `ydelta(2:end)` values are positive. Thus, the output of the table, which is the cumulative sum of the values stored in `ydelta`, is monotonically increasing.

Validate the Estimation Results

After you estimate the table values, as described in “Estimate the Monotonically Increasing Table Values Using Default Settings” on page 6-8, you use another measured data set to validate and check that you have not over-fit the model. You can plot and examine the following plots to validate the estimation results:

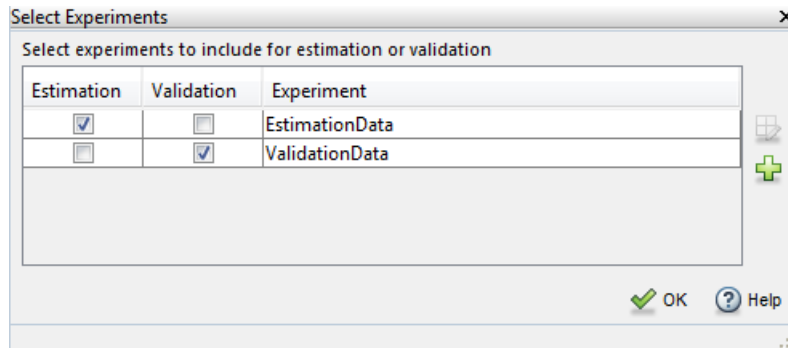
- Residuals plot
 - Measured and simulated data plots
- 1 Create an experiment to use for validation and import the validation I/O data.

On the **Parameter Estimation** tab, click **New Experiment**. Type `[time2, ydata2]` in **Outputs** and `[time2, xdata2]` in **Inputs** of the Edit

Experiment dialog box. Name the experiment `ValidationData` by right-clicking the default experiment name, `Exp`, in the **Experiments** area of the tool, and selecting **Rename**. For more information, see “Import Data for Parameter Estimation” on page 1-6.

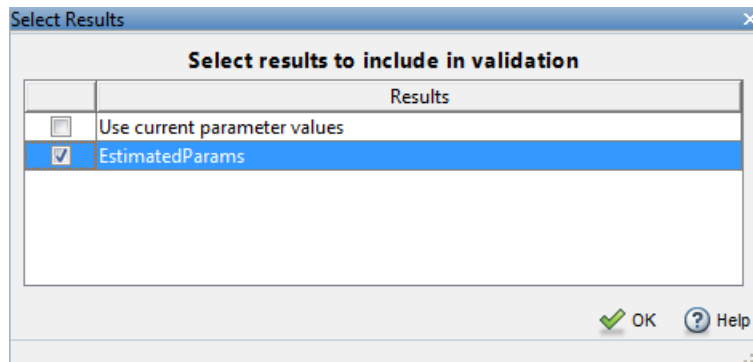
- 2 Select the experiment for validation.

Click **Select Experiments** on the **Parameter Estimation** tab. The `ValidationData` experiment is selected for estimation by default. Clear **Estimation** and select the box for **Validation**.



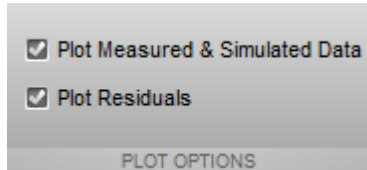
- 3 Select the results to validate.

On the **Validation** tab, click **Select Results to Validate**. Clear **Use current parameter values**, select `EstimatedParams`, and click **OK**.



- 4 Select the plots to display during validation.

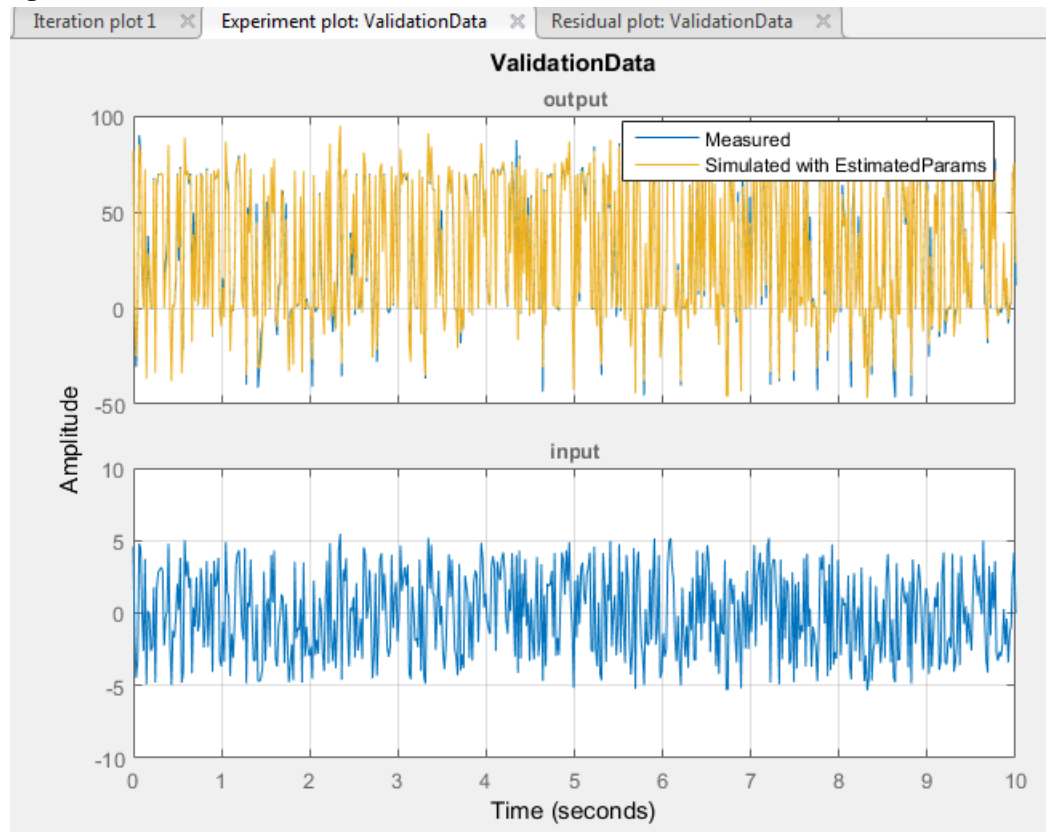
The Parameter Estimation tool displays the experiment plot after validation by default. Add the residuals plot by selecting the corresponding box on the **Validation** tab.



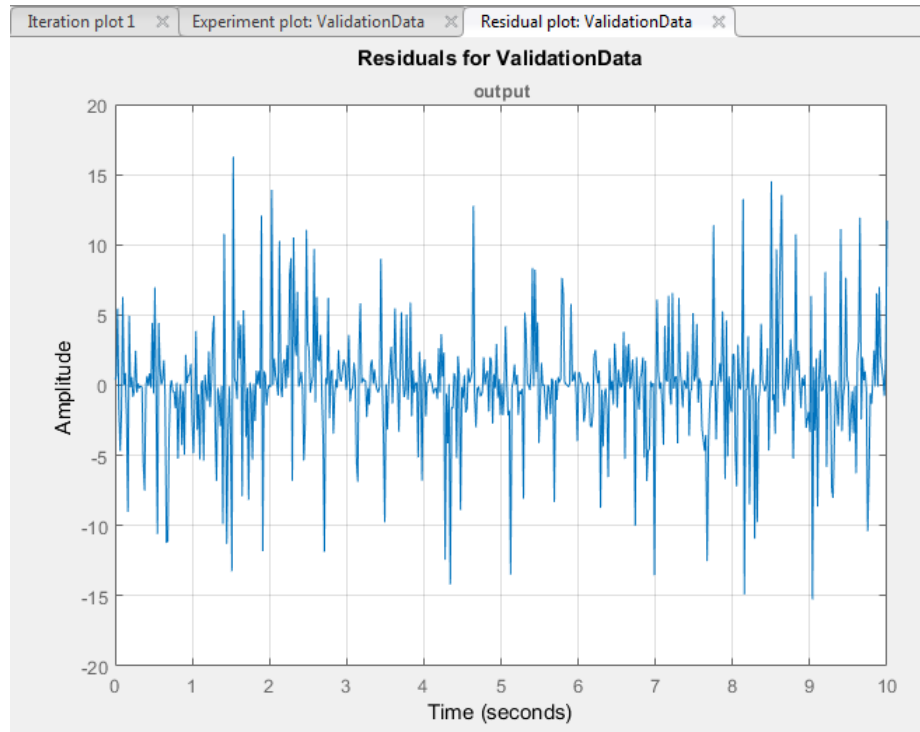
Click **Validate**.

5 Examine the plots.

- a The experiment plot shows the data simulated using estimated parameters agrees with the measured validation data.



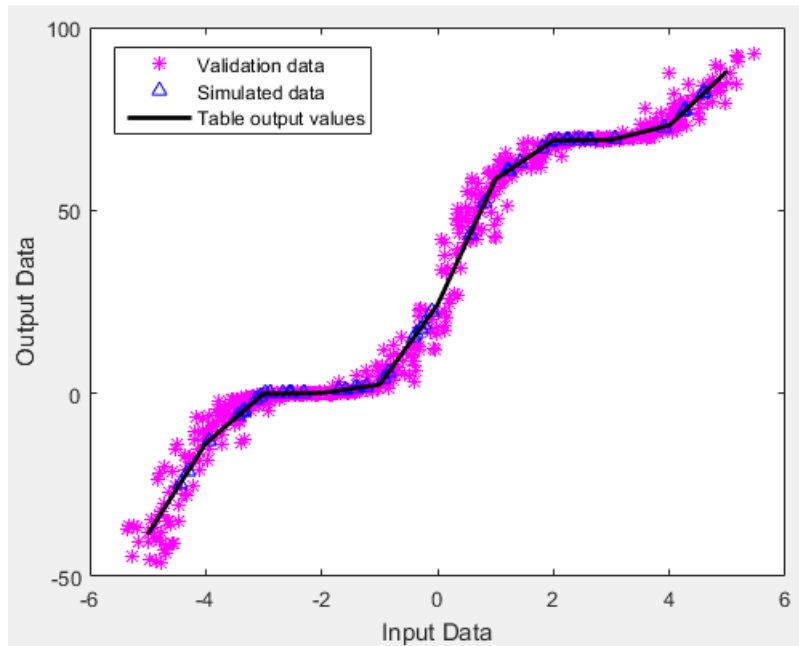
- b** To view the residuals plot, click **Residual plot: ValidationData** tab.



The residuals, which show the difference between the simulated and measured data, lie within 15% of the maximum output variation. This indicates a good match between the measured and simulated table data values.

- c** Plot and examine the validation data, simulated data, and estimated table values.

```
sim('lookup_increasing')
figure(2); plot(xdata2,ydata2,'m*',xout,yout,'b^')
hold on; plot(-5:5,cumsum(ydelta),'k','LineWidth', 2)
xlabel('Input Data'); ylabel('Output Data');
legend('Validation data','Simulated data','Table output values');
```



The table output values match both the measured data and the simulated table values. The table output values cover the entire range of input values, which indicates that all the lookup table values have been estimated.

See Also

Related Examples

- “Estimate Lookup Table Values from Data” on page 6-21

Estimate Lookup Table Values from Data

In this section...

“Objectives” on page 6-21

“About the Data” on page 6-21

“Open a Parameter Estimation Session” on page 6-21

“Estimate the Table Values Using Default Settings” on page 6-23

“Validate the Estimation Results” on page 6-30

Objectives

This example shows how to estimate lookup table values from time-domain input-output (I/O) data in the Parameter Estimation tool.

About the Data

In this example, use the I/O data in `lookup_regular.mat` to estimate the values of a lookup table. The MAT-file includes the following variables:

- `xdata1` — Consists of 63 uniformly-sampled input data points in the range [0,6.5]
- `ydata1` — Consists of output data corresponding to the input data samples
- `time1` — Time vector

Use the I/O data to estimate the lookup table values in the `lookup_regular` Simulink model. The lookup table in the model contains ten values, which are stored in the MATLAB variable `table`. The initial table values comprise a vector of 0s. To learn more about how to model a system using lookup tables, see “Guidelines for Choosing a Lookup Table” (Simulink).

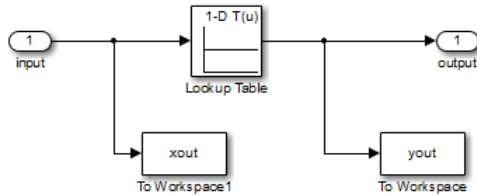
Open a Parameter Estimation Session

To estimate the lookup table values, open a Parameter Estimation session.

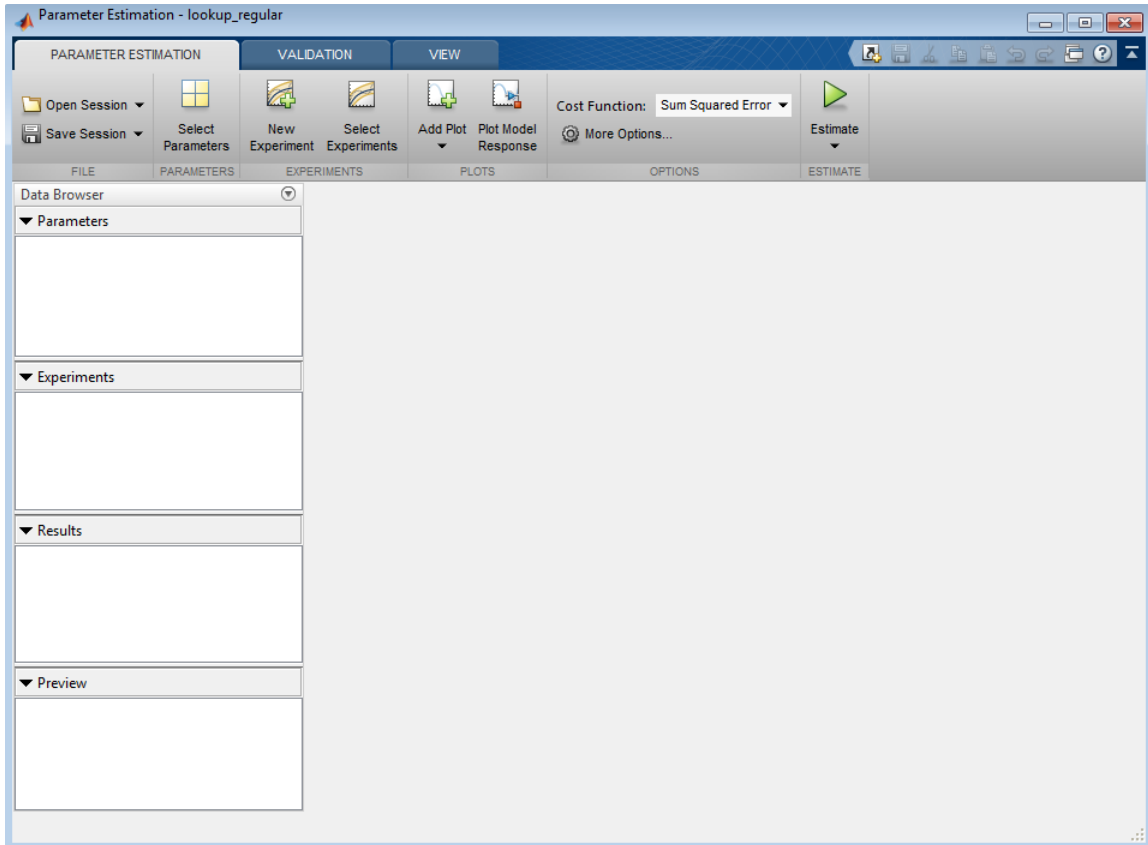
- 1 Open the lookup table model by typing the following command at the MATLAB prompt:

```
lookup_regular
```

This command opens the Simulink model, and loads the estimation data into the MATLAB workspace.



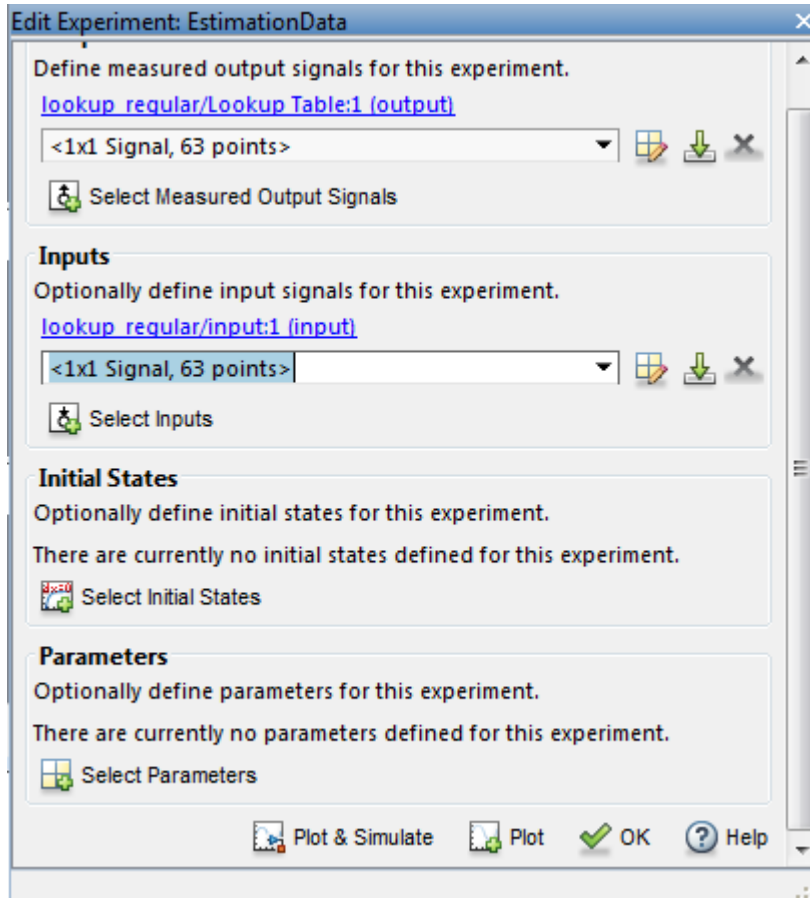
- 2 In the Simulink model, select **Analysis > Parameter Estimation** to open a new session with name **lookup_regular** in the Parameter Estimation tool.



Estimate the Table Values Using Default Settings

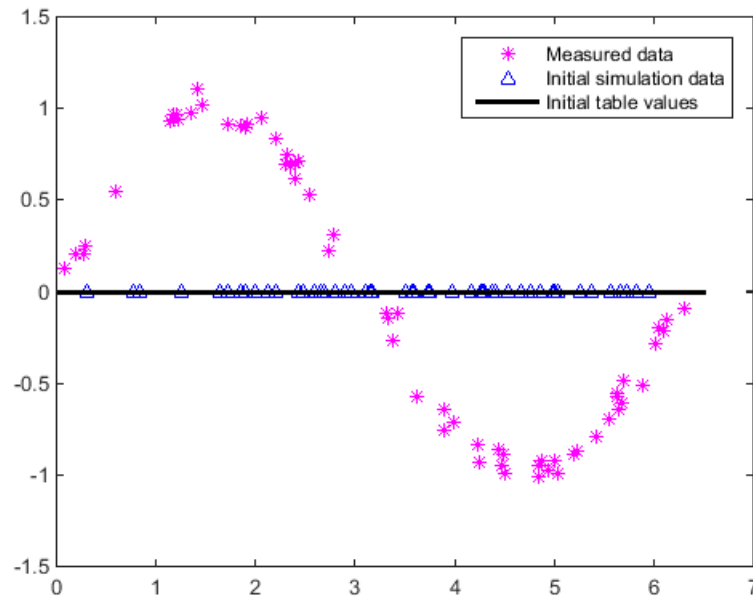
Use the following steps to estimate the lookup table values.

- 1 Create a new experiment by clicking **New Experiment** on the **Parameter Estimation** tab. Name it `EstimationData`. Then import the I/O data, `xdata1` and `ydata1`, and the time vector, `time1`, into the experiment. To do this open the experiment editor by right-clicking `EstimationData` and selecting **Edit...** Type `[time1,ydata1]` in the output dialog box and `[time1,xdata1]` in the input dialog box in the experiment editor. For more information, see “Import Data for Parameter Estimation” on page 1-6. After you import the data the experiment looks as follows:



- 2 Run an initial simulation to view the I/O data, simulated output, and the initial table values. To do so, type the following commands at the MATLAB prompt:

```
sim('lookup_regular')
figure(1); plot(xdata1,ydata1, 'm*', xout, yout,'b^')
hold on; plot(linspace(0,6.5,10), table, 'k', 'LineWidth', 2);
legend('Measured data','Initial simulation data','Initial table values');
```



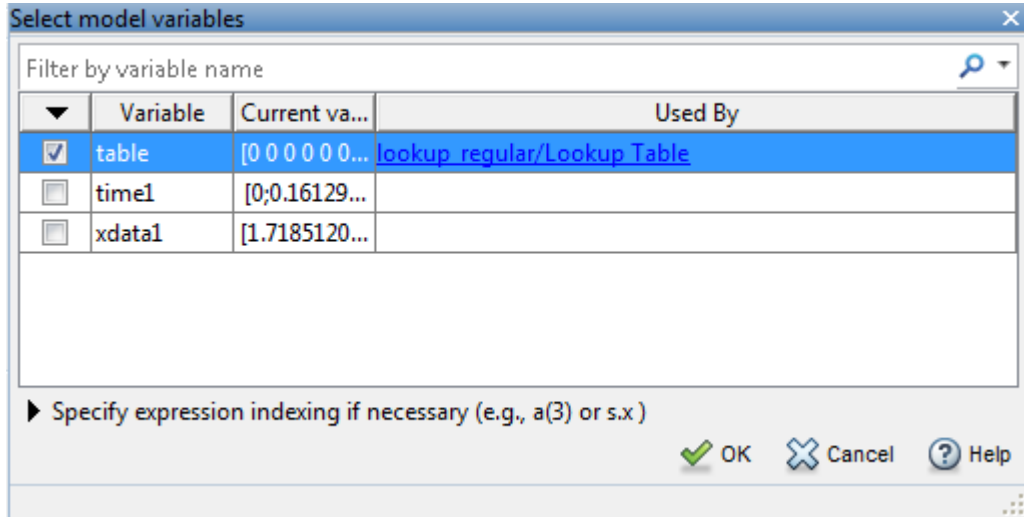
The x-axis and y-axis of the figure represent the input and output data, respectively. The figure shows the following plots:

- Measured data — Represented by the magenta stars (*).
- Initial table values — Represented by the black line.
- Initial simulation data — Represented by the blue deltas (Δ).

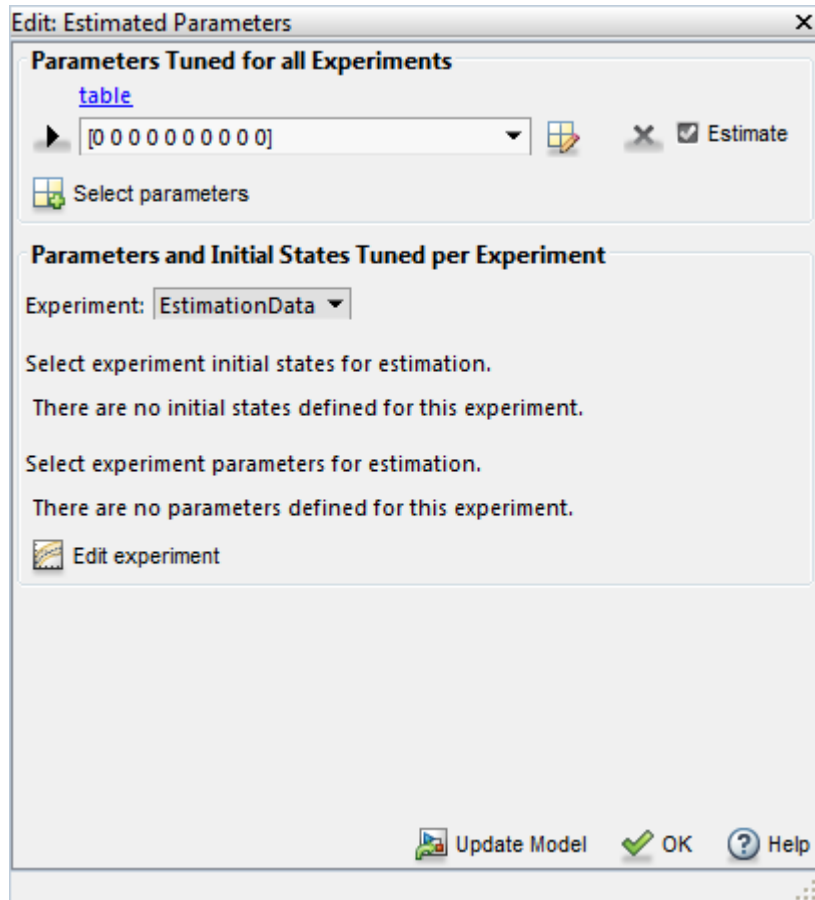
You can see that the initial table values and simulated data do not match with the measured data.

- 3 To select the table values to estimate, on the **Parameter Estimation** tab, click the **Select Parameters** button to open the **Edit:Estimated Parameters** dialog. In the

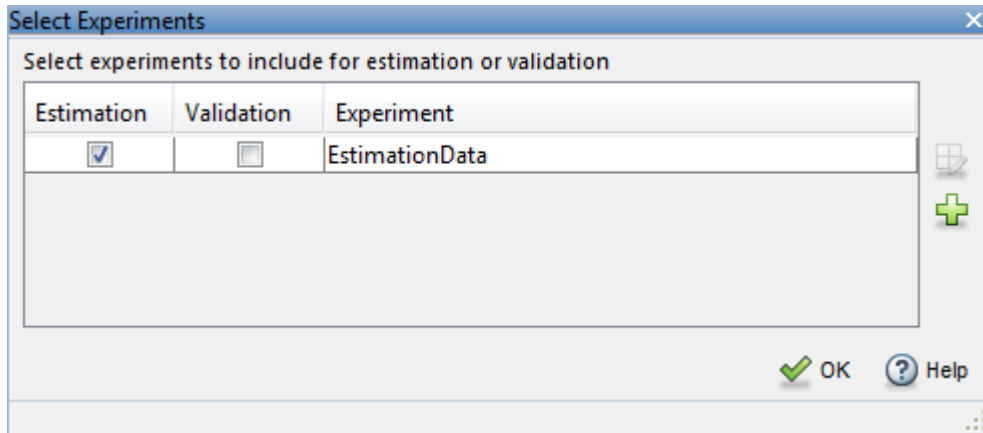
Parameters Tuned for all Experiments panel, click **Select parameters** to launch the Select Model Variables dialog. Check the box next to table, and click **OK**.



The **Edit:Estimated Parameters** window now looks as follows. The table values are selected for estimation by default.

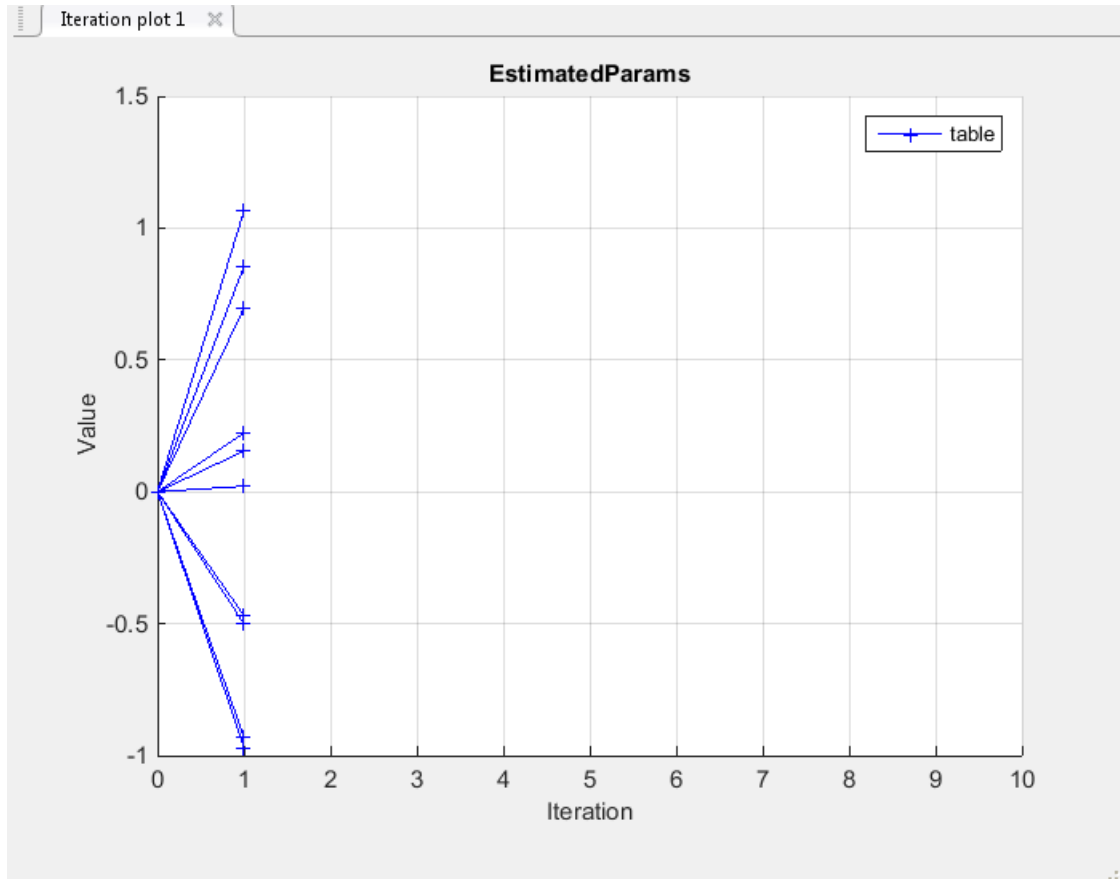


- 4 On the **Parameter Estimation** tab, click **Select Experiment**. `EstimationData` is selected for estimation by default. If not, check the box under the **Estimation** column, and click **OK**.

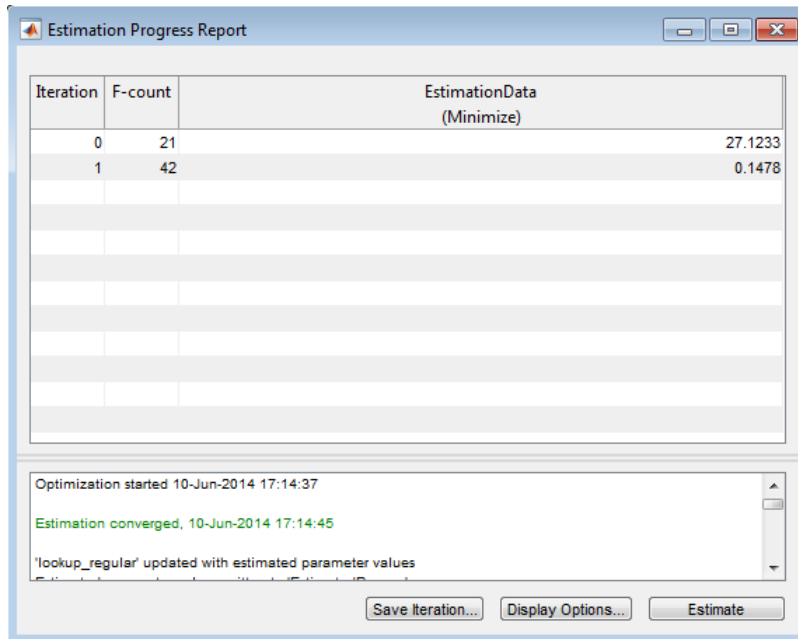


- 5 To estimate the table values using the default settings, on the **Parameter Estimation** tab, click **Estimate** to open the **Parameter Trajectory** plot and **Estimation Progress Report** window. The **Parameter Trajectory** plot shows the change in the parameter values at each iteration.

After the estimation converges, the **Parameter Trajectory** plot looks like this:



The **Estimation Progress Report** shows the iteration number, number of times the objective function is evaluated, and the value of the cost function at the end of each iteration. After the estimation converges, the **Estimation Progress Report** looks like this:



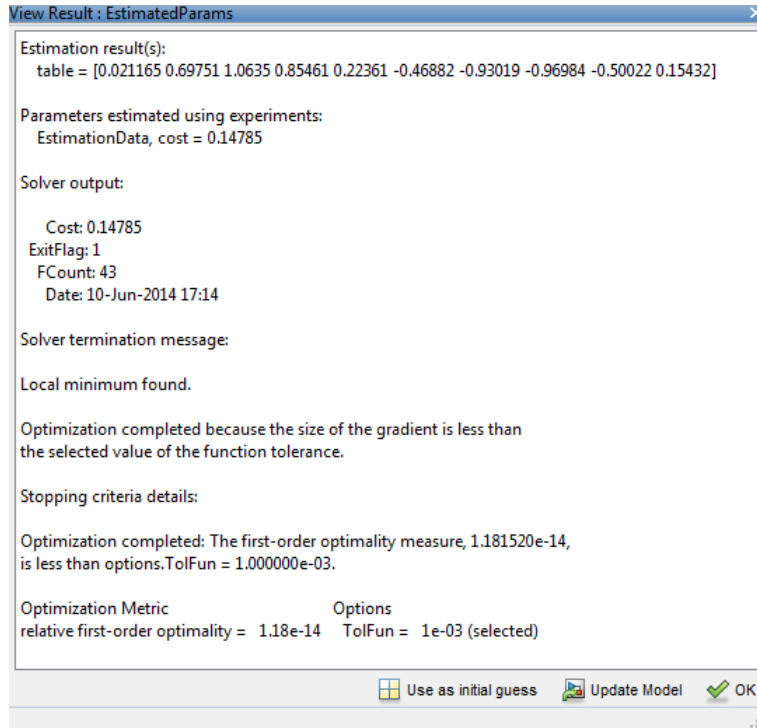
The screenshot shows a window titled "Estimation Progress Report" with a table and a log. The table has three columns: "Iteration", "F-count", and "EstimationData (Minimize)". The log shows the optimization starting at 10-Jun-2014 17:14:37, converging at 10-Jun-2014 17:14:45, and updating 'lookup_regular' with estimated parameter values. At the bottom are buttons for "Save Iteration...", "Display Options...", and "Estimate".

Iteration	F-count	EstimationData (Minimize)
0	21	27.1233
1	42	0.1478

Optimization started 10-Jun-2014 17:14:37
Estimation converged, 10-Jun-2014 17:14:45
'lookup_regular' updated with estimated parameter values

Save Iteration... Display Options... Estimate

The estimated parameters are saved in `EstimatedParams` in the **Results** section of the **Data Browser** pane on the left. To view the results, right-click on `EstimatedParams` and then select **Open**. The report resembles the following.



This report includes the estimated parameter values, the final value of the cost function, and other optimization results. You can see that the optimization stopped when the size of the gradient, $1.18\text{e-}14$ was less than the criteria value, $1\text{e-}3$.

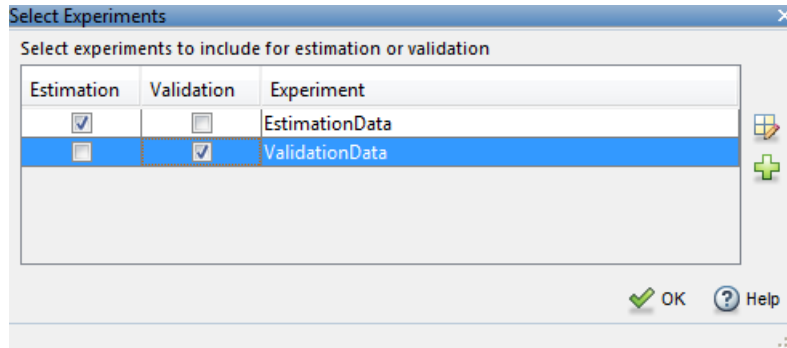
Validate the Estimation Results

After you estimate the table values, as described in “Estimate the Table Values Using Default Settings” on page 6-23, you must use another data set to validate that you have not over-fitted the model. You can plot and examine the following plots to validate the estimation results:

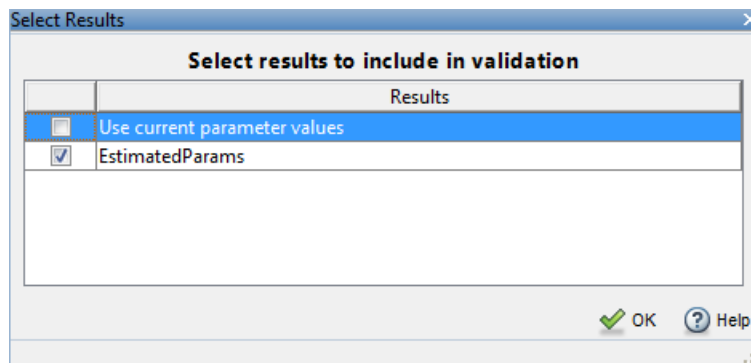
- Residuals plot
- Measured and simulated data plots

To validate the estimation results:

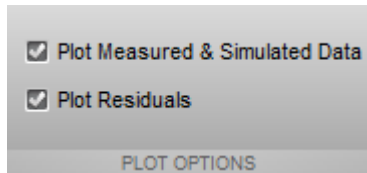
- 1 Create a new experiment to use for validation. Name it `ValidationData`. Import the validation I/O data, `xdata2` and `ydata2`, and time vector, `time2` in the `ValidationData` experiment. To do this open the experiment editor by right-clicking `ValidationData` and selecting **Edit...** Then, type `[time2, ydata2]` in the output dialog box and `[time2, xdata2]` in the input dialog box in the experiment editor. For more information, see “Import Data for Parameter Estimation” on page 1-6.
- 2 To select the experiment for validation, on the **Parameter Estimation** tab, click **Select Experiments**. The `ValidationData` experiment is selected for estimation by default. Deselect the box for estimation and check it for validation.



- 3 To select results to use, on the **Validation** tab, click **Select Results to Validate**. Deselect `Use current parameter values` and select `EstimatedParams`, and click **OK**.



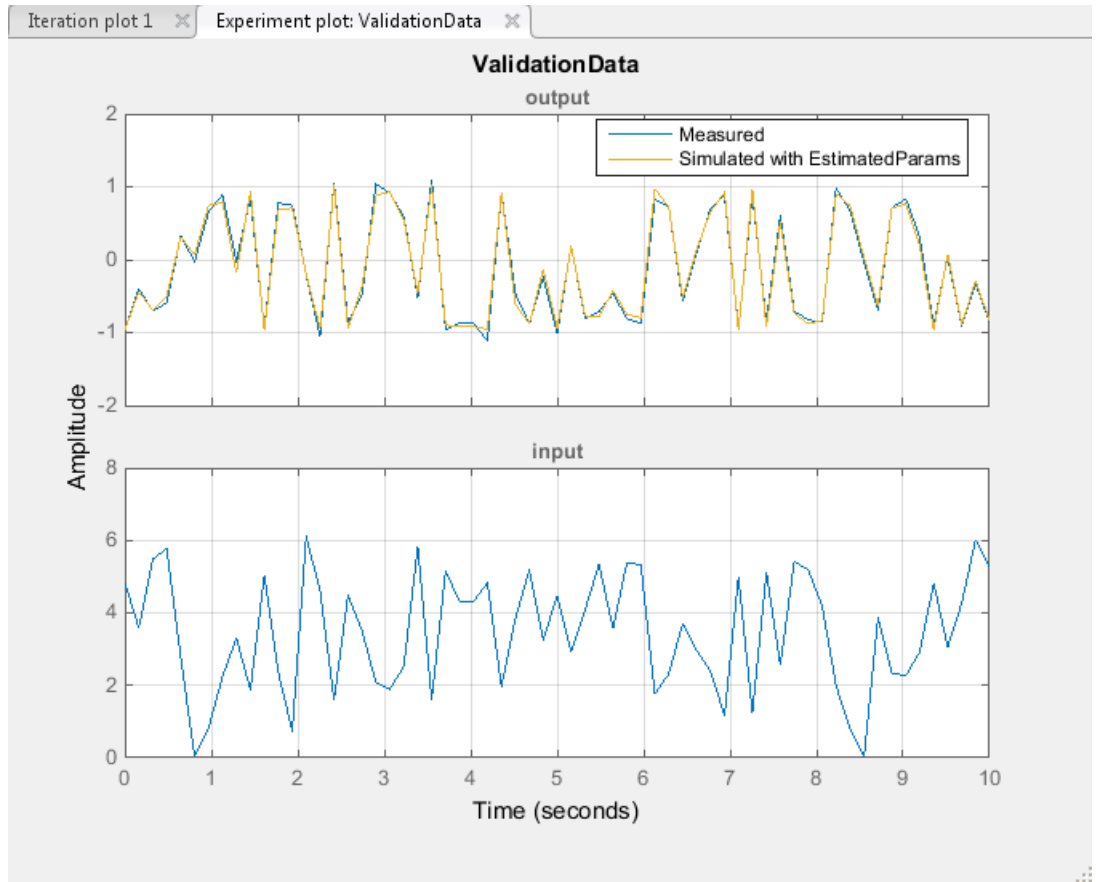
- 4 The Parameter Estimation tool, by default, displays the experiment plot after validation. Add the residuals plot by checking the corresponding box on the **Validation** tab.



To start validation, on the **Validation** tab, click **Validate**.

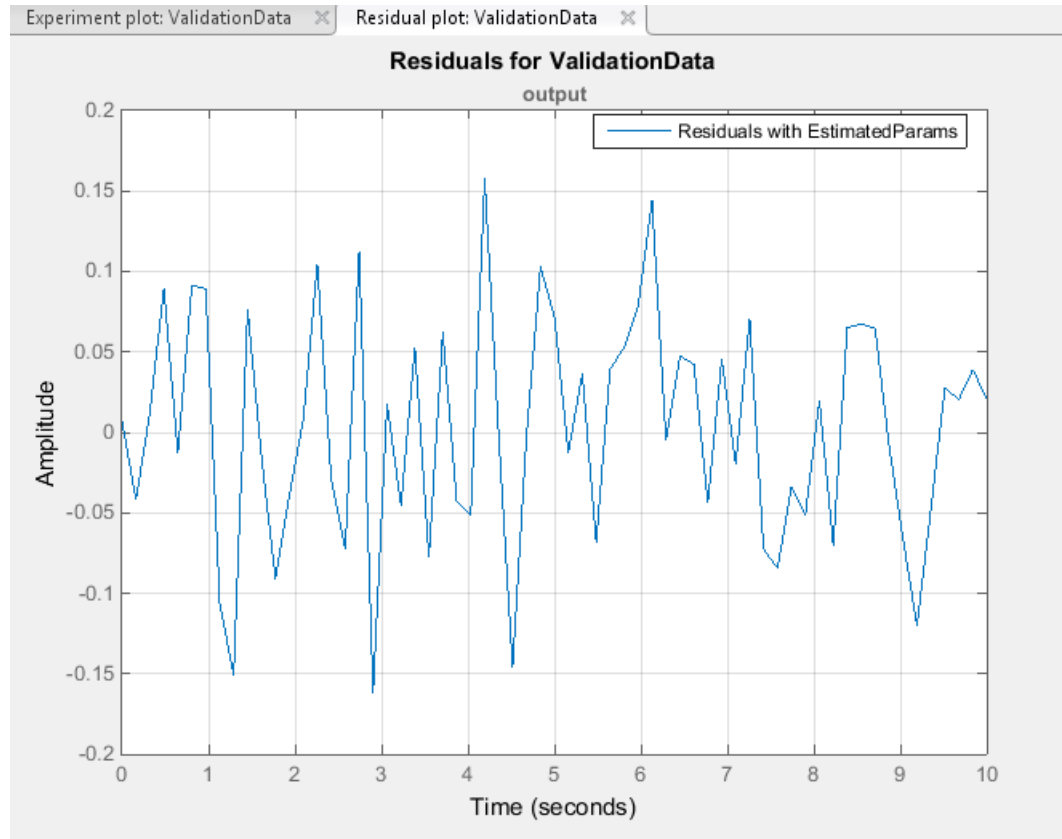
5 Examine the plots

a Experiment plot



You can see that the data simulated using the estimated parameters agrees with the measured validation data.

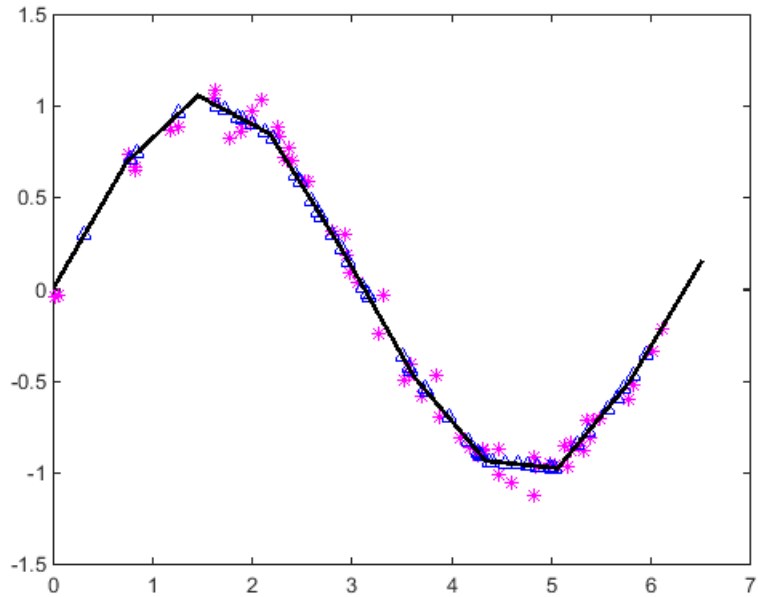
- b** Click Residual plot: ValidationData to open the residuals plot.



The residuals, which show the difference between the simulated and measured data, lie in the range $[-0.15, 0.15]$ — within 15% of the maximum output variation. This indicates a good match between the measured and the simulated table data values.

- c** Plot and examine the estimated table values against the validation data set and the simulated table values by typing the following commands at the MATLAB prompt.

```
sim('lookup_regular')
figure(2); plot(xdata2,ydata2, 'm*', xout, yout, 'b^')
hold on; plot(linspace(0,6.5,10), table, 'k', 'LineWidth', 2)
```



The plot shows that the table values, displayed as the black line, match both the validation data and the simulated table values. The table data values cover the entire range of input values, which indicates that all the lookup table values have been estimated.

See Also

Related Examples

- “Estimate Constrained Values of a Lookup Table” on page 6-6

Building Models Using Adaptive Lookup Table Blocks

Simulink Design Optimization software provides blocks for modeling systems as adaptive lookup tables. You can use the adaptive lookup table blocks to create lookup tables from measured or simulated data. You build a model using the adaptive lookup table blocks, and then simulate the model to adapt the lookup table values to the time-varying I/O data. During simulation, the software uses the input data to locate the table values, and then uses the output data to recalculate the table values. The updated table values are stored in the adaptive lookup table block. For more information, see “What are Adaptive Lookup Tables?” on page 6-2.

The Adaptive Lookup Table library has the following blocks:

- Adaptive Lookup Table (1D Stair-Fit) — One-dimensional adaptive lookup table
- Adaptive Lookup Table (2D Stair-Fit) — Two-dimensional adaptive lookup table
- Adaptive Lookup Table (nD Stair-Fit) — Multidimensional adaptive lookup table

Note Use the Adaptive Lookup Table (nD Stair-Fit) block to create lookup tables of three or more dimensions.

To access the Adaptive Lookup Tables library:

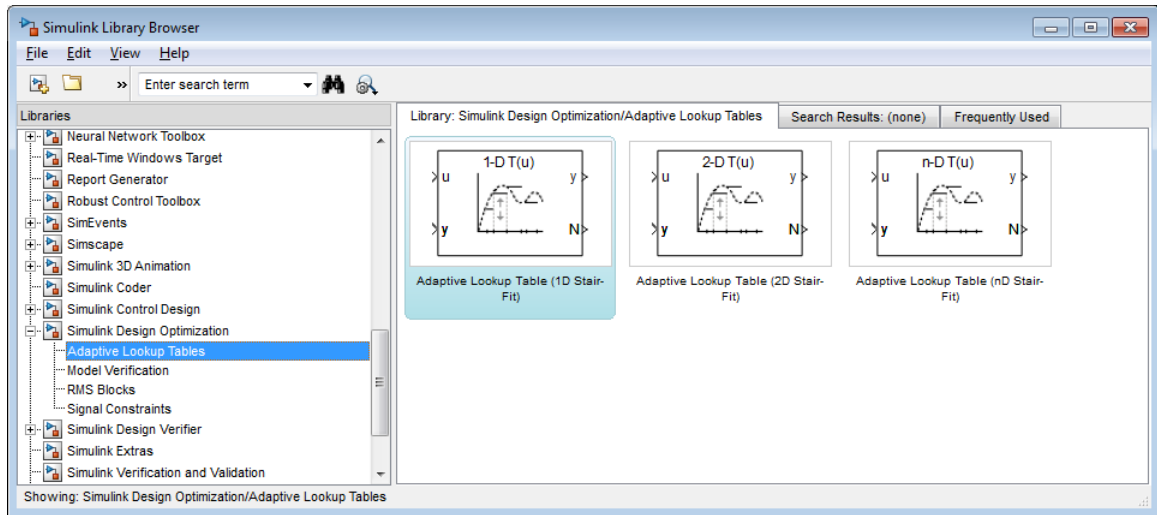
- 1 Open the Simulink Library Browser.

At the MATLAB prompt, enter `slLibraryBrowser`.

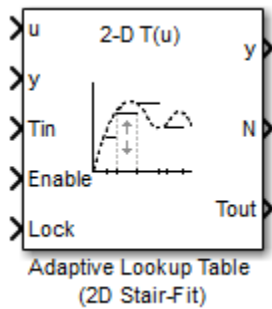
- 2 Open the Simulink Design Optimization library.

In the **Libraries** pane, expand the **Simulink Design Optimization** node.

- 3 In the Simulink Design Optimization library tree, click **Adaptive Lookup Tables**.



By default, the Adaptive Lookup Table blocks have two inputs and outputs. You can display additional inputs and outputs in a block by selecting the corresponding options in the Function Block Parameters dialog box. To learn more about the options, see the block reference pages.



Adaptive Lookup Table Block Showing Inputs and Outputs

The 2-D Adaptive Lookup Table block has the following inputs and outputs:

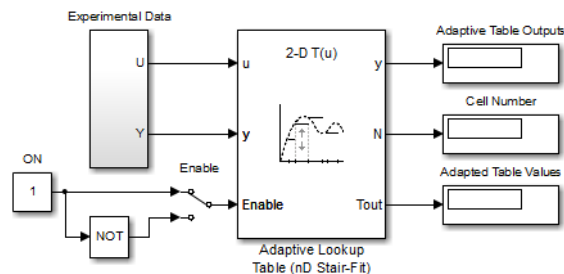
- u and y — Input and output data of the system being modeled, respectively.

For example, to model an engine's efficiency as a function of engine rpm and manifold pressure, specify u as the rpm, y as the pressure, and y as the efficiency signals.

- T_{in} — The initial table data.
- $Enable$ — Signal to enable, disable, or reset the adaptation process.
- $Lock$ — Signal to update only specified cells in the table.
- y — Value of the cell currently being adapted.
- N — Number of the cell currently being adapted.
- T_{out} — Values of the adapted table data.

For more information on how to use adaptive lookup tables, see “Model Engine Using n-D Adaptive Lookup Table” on page 6-41.

A typical Simulink diagram using an adaptive lookup table block is shown in the next figure.



Simulink Diagram Using an Adaptive Lookup Table

In this figure, the Experiment Data block imports a set of experimental data into Simulink through MATLAB workspace variables. The initial table is specified in the block mask parameters. When the simulation runs, the initial table begins to adapt to new data inputs and the resulting table is copied to the block's output.

See Also

Adaptive Lookup Table (1D Stair-Fit) | Adaptive Lookup Table (2D Stair-Fit) | Adaptive Lookup Table (nD Stair-Fit)

Related Examples

- “Model Engine Using n-D Adaptive Lookup Table” on page 6-41

More About

- “What are Adaptive Lookup Tables?” on page 6-2
- “Selecting an Adaptation Method” on page 6-39

Selecting an Adaptation Method

You specify the algorithm using the **Adaptation Method** drop-down list in the Function Block Parameters dialog box of an adaptive lookup table block. This section discusses the details of these algorithms.

Sample Mean

Sample mean provides the average value of n output data samples and is defined as:

$$\hat{y}(n) = \frac{1}{n} \sum_{i=1}^n y(i)$$

where $y(i)$ is the i^{th} measurement collected within a particular *cell*. For each input data u , the sample mean at the corresponding cell is updated using the output data measurement, y . Instead of accumulating n samples of data for each cell, a recursive relation is used to calculate the sample mean. The recursive expression is obtained by the following equation:

$$\hat{y}(n) = \frac{1}{n} \left[\sum_{i=1}^{n-1} y(i) + y(n) \right] = \frac{n-1}{n} \left[\frac{1}{n-1} \sum_{i=1}^{n-1} y(i) \right] + \frac{1}{n} y(n) = \frac{n-1}{n} \hat{y}(n-1) + \frac{1}{n} y(n)$$

where $y(n)$ is the n^{th} data sample.

Defining *a priori estimation error* as $e(n) = y(n) - \hat{y}(n-1)$, the recursive relation can be written as:

$$\hat{y}(n) = \hat{y}(n-1) + \frac{1}{n} e(n)$$

where $n \geq 1$ and the initial estimate $\hat{y}(0)$ is arbitrary.

In this expression, only the number of samples, n , for each cell—rather than n data samples—is stored in memory.

Sample Mean with Forgetting

The adaptation method “Sample Mean” on page 6-39 has an *infinite memory*. The past data samples have the same weight as the final sample in calculating the sample mean.

Sample mean (with forgetting) uses an algorithm with a *forgetting factor* or **Adaptation gain** that puts more weight on the more recent samples. This algorithm provides robustness against initial response transients of the plant and an adjustable speed of adaptation. Sample mean (with forgetting) is defined as:

$$\begin{aligned}\hat{y}(n) &= \frac{1}{\sum_{i=1}^n \lambda^{n-i}} \sum_{i=1}^n \lambda^{n-i} y(i) \\ &= \frac{1}{\sum_{i=1}^n \lambda^{n-i}} \left[\sum_{i=1}^{n-1} \lambda^{n-i} y(i) + y(n) \right] = \frac{s(n-1)}{s(n)} \hat{y}(n-1) + \frac{1}{s(n)} y(n)\end{aligned}$$

where $\lambda \in [0,1]$ is the **Adaptation gain** and $s(k) = \sum_{i=1}^k \lambda^{k-i}$.

Defining *a priori estimation error* as $e(n) = y(n) - \hat{y}(n-1)$, where $n \geq 1$ and the initial estimate $\hat{y}(0)$ is arbitrary, the recursive relation can be written as:

$$\hat{y}(n) = \hat{y}(n-1) + \frac{1}{s(n)} e(n) = \hat{y}(n-1) + \frac{1-\lambda}{1-\lambda^n} e(n)$$

A small value of λ results in faster adaptation. A value of 0 indicates short memory (last data becomes the table value), and a value of 1 indicates long memory (average all data received in a cell).

Model Engine Using n-D Adaptive Lookup Table

In this section...

“Objectives” on page 6-41

“About the Data” on page 6-41

“Building a Model Using Adaptive Lookup Table Blocks” on page 6-42

“Adapting the Lookup Table Values Using Time-Varying I/O Data” on page 6-51

Objectives

In this example, you learn how to capture the time-varying behavior of an engine using an n-D adaptive lookup table. You accomplish the following tasks using the Simulink software:

- Configure an adaptive lookup table block to model your system.
- Simulate the model to update the lookup table values dynamically.
- Export the adapted lookup table values to the MATLAB workspace.
- Lock a specific cell in the table during adaptation.
- Disable the adaptation process and use the adaptive lookup table as a static lookup table.

About the Data

In this example, you use the data in `vedata.mat` which contains the following variables measured from an engine:

- `X` — 10 input breakpoints for intake manifold pressure in the range [10,100]
- `Y` — 36 input breakpoints for engine speed in the range [0,7000]
- `Z` — 10x36 matrix of table data for engine volumetric efficiency

To learn more about breakpoints and table data, see “Anatomy of a Lookup Table” (Simulink).

The output volumetric efficiency of the engine is time varying, and a function of two inputs—intake manifold pressure and engine speed. The data in the MAT-file is used to generate the time-varying input and output (I/O) data for the engine.

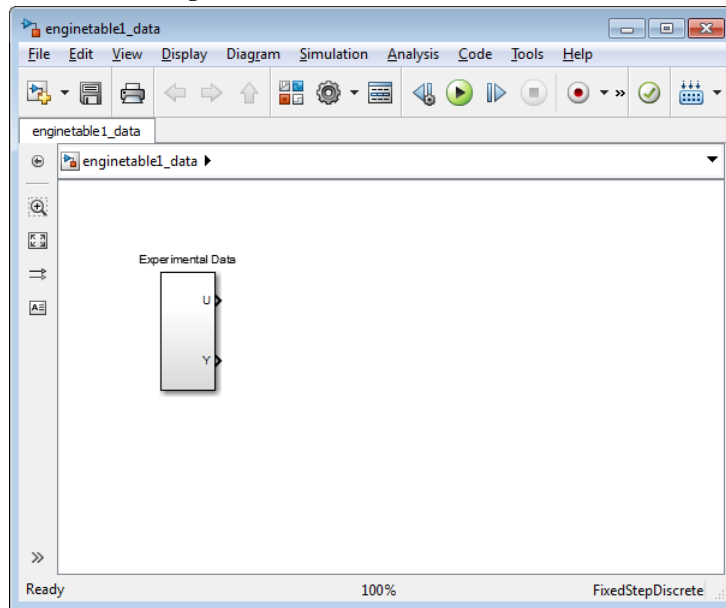
Building a Model Using Adaptive Lookup Table Blocks

In this portion of the tutorial, you learn how to build a model of an engine using an Adaptive Lookup Table block.

- 1 Open a preconfigured Simulink model by typing the model name at the MATLAB prompt:

```
enginetable1_data
```

The Experimental Data subsystem in the Simulink model generates time-varying I/O data during simulation.



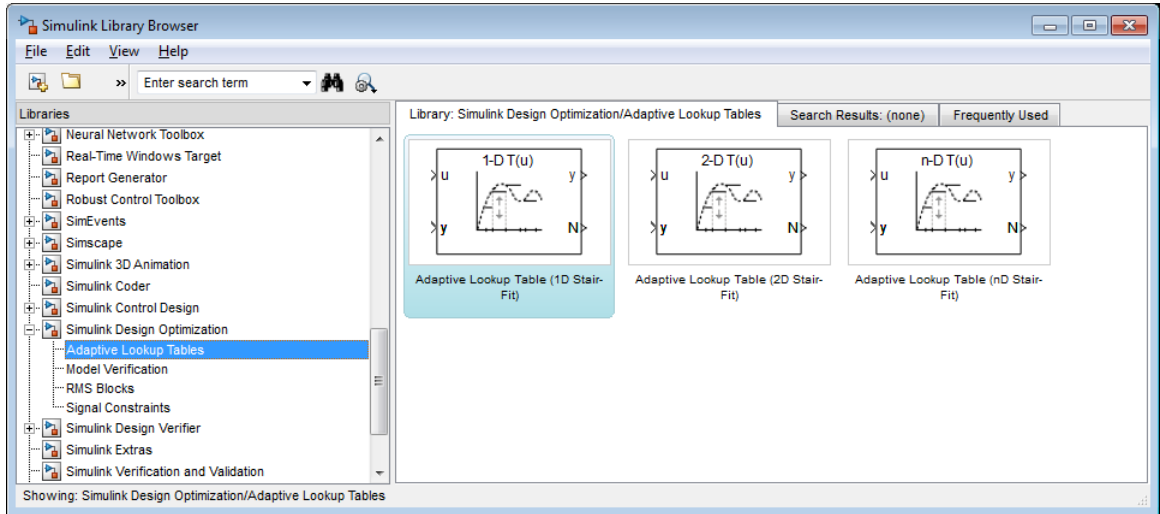
This command also loads the variables X , Y and Z into the MATLAB workspace. To learn more about this data, see “About the Data” on page 6-41.

- 2 Add an Adaptive Lookup Table block to the Simulink model.
 - a Open the Simulink Library Browser.

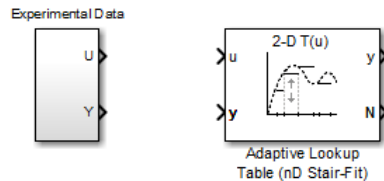
At the MATLAB prompt, enter `slLibraryBrowser`.
 - b Open the Simulink Design Optimization library.

In the **Libraries** pane, expand the **Simulink Design Optimization** node.

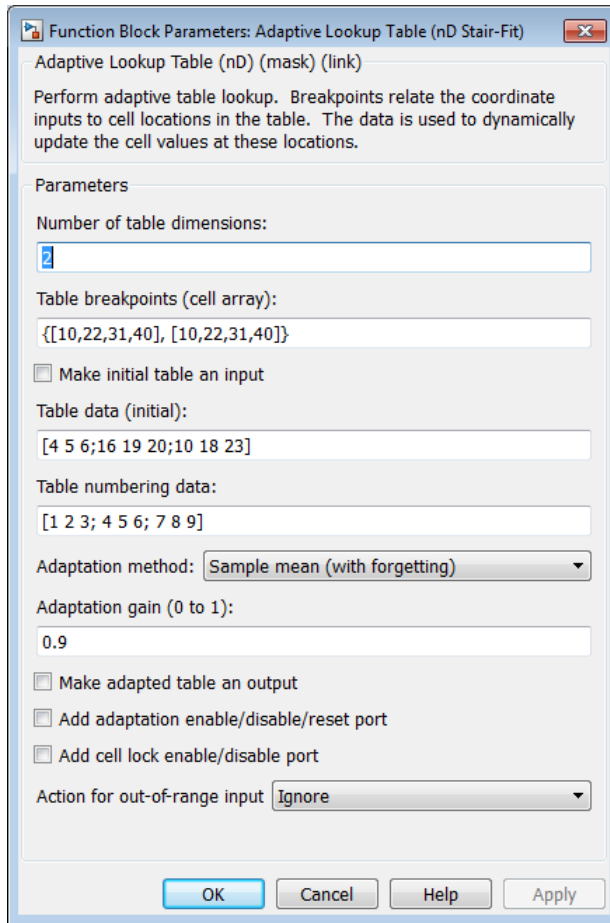
- c In the Simulink Design Optimization library tree, click **Adaptive Lookup Tables**.



- d Drag and drop the Adaptive Lookup Table (nD Stair-Fit) block from the Adaptive Lookup Tables library to the Simulink model window.



- 3 Double-click the Adaptive Lookup Table (nD Stair-Fit) block to open the Function Block Parameters: Adaptive Lookup Table (nD Stair-Fit) dialog box.



- 4 In the Function Block Parameters dialog box:
 - a Specify the following block parameters:
 - **Table breakpoints (cell array)** — Enter `{[X; 110], [Y; 7200]}` to specify the range of input breakpoints.
 - **Table data (initial)** — Enter `rand(10,36)` to specify random numbers as the initial table values for the volumetric efficiency.
 - **Table numbering data** — Enter `reshape(1:360,10,36)` to specify a numbering scheme for the table cells.

- b** Verify that `Sample mean (with forgetting)` is selected in the **Adaptation method** drop-down list.
- c** Enter `0.98` in the **Adaptation gain (0 to 1)** field to specify the *forgetting factor* for the `Sample mean (with forgetting)` adaptation algorithm.

An adaptation gain close to 1 indicates high robustness of the lookup table values to input noise. To learn more about the adaptation gain, see “Sample Mean with Forgetting” on page 6-39 in “Selecting an Adaptation Method” on page 6-39.

- d** Select the **Make adapted table an output** check box.

This action adds a new port named `Tout` to the Adaptive Lookup Table block. You use this port to plot the table values as they are being adapted.

- e** Select the **Add adaptation enable/disable/reset port** check box.

This action adds a new port named `Enable` to the Adaptive Lookup Table block. You use this port to enable or disable the adaptation process.

- f** Select the **Add cell lock enable/disable port** check box.

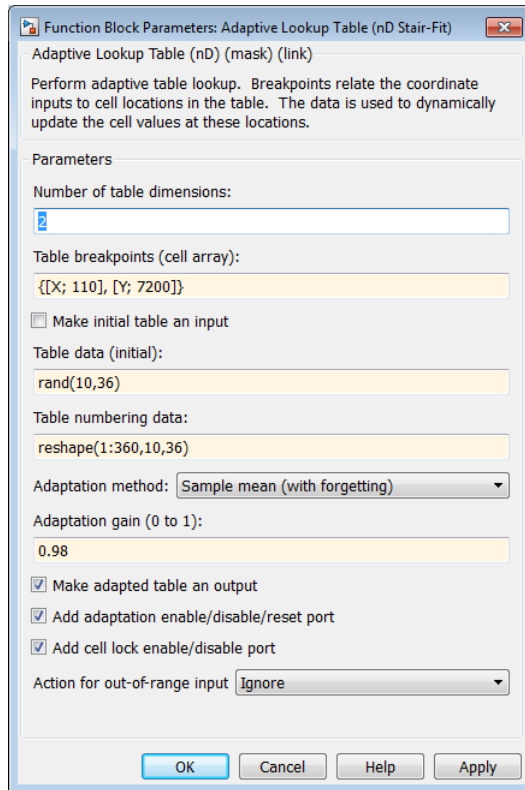
This action adds a new port named `Lock` to the Adaptive Lookup Table block. You use this port to lock a cell during the adaptation process.

- g** Verify that `Ignore` is selected in the **Action for out-of-range** drop-down list.

This selection specifies that the software ignores any time-varying inputs outside the range of input breakpoints during adaptation.

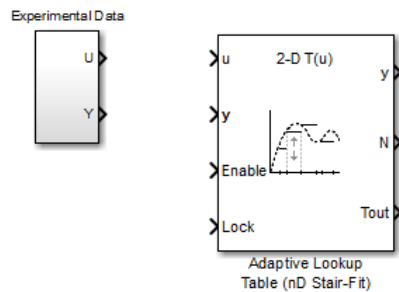
Tip To learn more, see Adaptive Lookup Table (nD Stair-Fit) block reference page.

After you configure the parameters, the block parameters dialog box looks like the following figure.

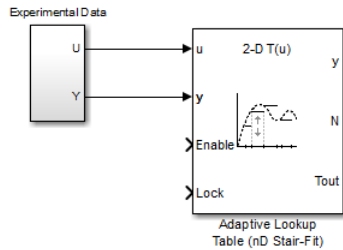


- h Click **OK** to close the Function Block Parameters dialog box.

The Simulink model now looks similar to the following figure.

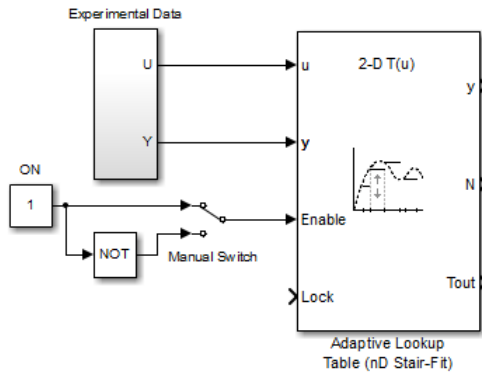


- Assign the input and output data to the engine model by connecting the U and Y ports of the Experimental Data block to the u and y ports of the Adaptive Lookup Table block, respectively.



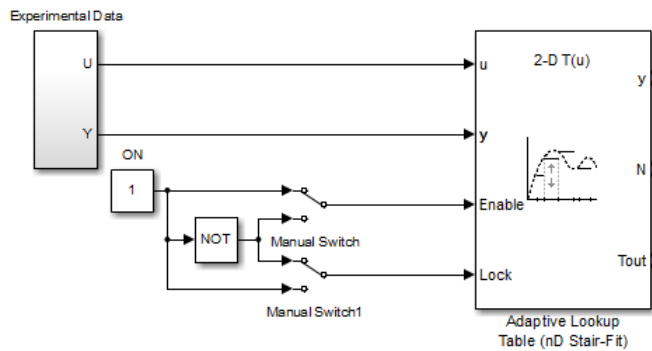
Tip To learn how to connect blocks in the Simulink model window, see “Block and Signal Line Shortcuts and Actions” (Simulink).

- Design a logic using Simulink blocks to enable or disable the adaptation process. Connect the logic to the Adaptive Lookup Table block, as shown in the following figure.

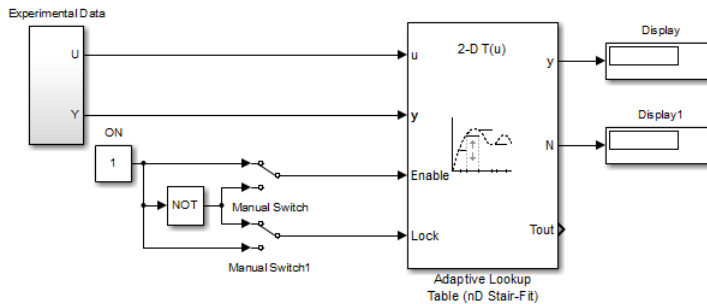


This logic outputs an initial value of 1 which enables the adaptation process.

- Design a logic to lock a cell during adaptation. Connect the logic to the Adaptive Lookup Table block, as shown in the following figure.



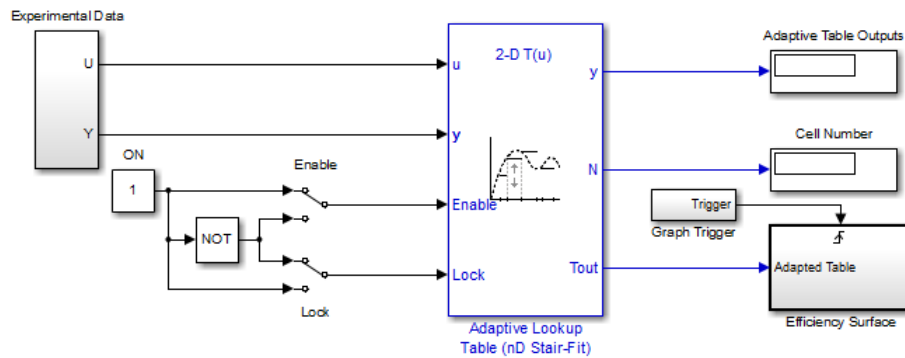
- 8 In the Simulink Library Browser, select the **Simulink** > **Sinks** library, and drag Display blocks to the model window. Connect the blocks, as shown in the following figure.



During simulation, the Display blocks show the following:

- Display block — Shows the value of the current cell being adapted.
 - Display1 block — Shows the number of the current cell being adapted.
- 9 Write a MATLAB function to plot the lookup table values as they adapt during simulation.

Alternatively, type `enginetable` at the MATLAB prompt to open a preconfigured Simulink model. The `Efficiency Surface` subsystem contains a function to plot the lookup table values, as shown in the next figure.

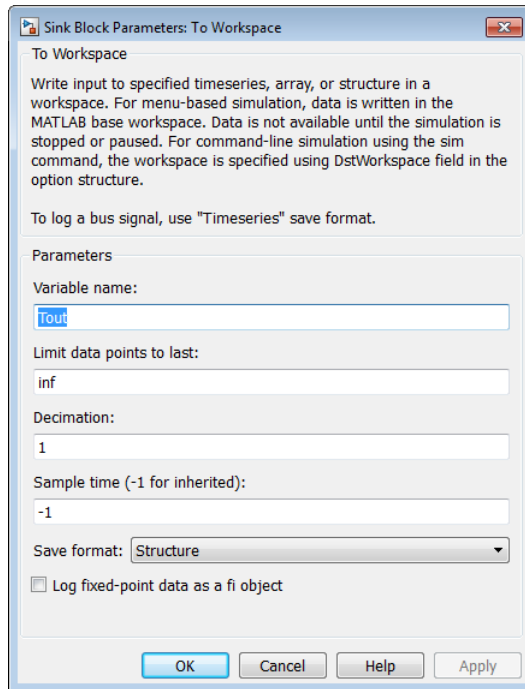


10 Connect a To Workspace block to export the adapted table values:

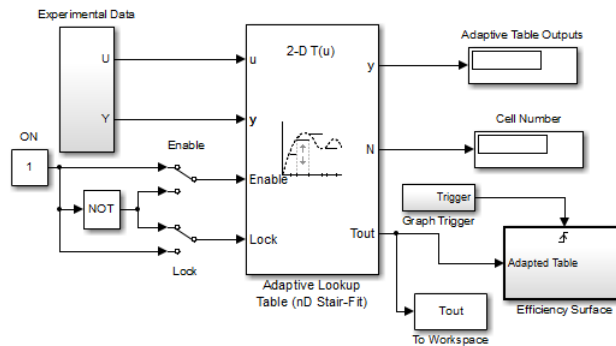
- a In the Simulink Library Browser, select the **Simulink** > **Sinks** library, and drag the To Workspace block to the model window.

To learn more about this block, see the To Workspace block reference page in the Simulink documentation.

- b Double-click the To Workspace block to open the Sink Block Parameters dialog box, and type `Tout` in the **Variable name** field.



- c Click **OK**.
- d Connect the To Workspace block to the adaptive lookup table output signal `Tout`, as shown in the next figure.



You have now built the model for updating and viewing the adaptive lookup table values. You must now simulate the model to start the adaptation, as described in “Adapting the Lookup Table Values Using Time-Varying I/O Data” on page 6-51.

Adapting the Lookup Table Values Using Time-Varying I/O Data

In this portion of the tutorial, you learn how to update the lookup table values to adapt to the time-varying input and output values.

You must have already built the Simulink model, as described “Building a Model Using Adaptive Lookup Table Blocks” on page 6-42.

To perform the adaptation:

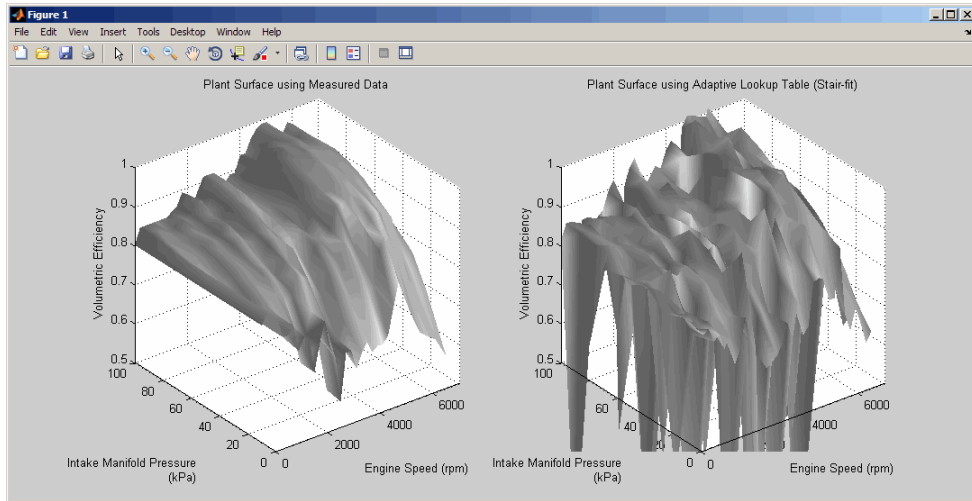
- 1 In the Simulink Editor, specify the simulation time as `inf`.

The simulation time of infinity specifies that the adaptation process continues as long as the input and output values of the engine change.

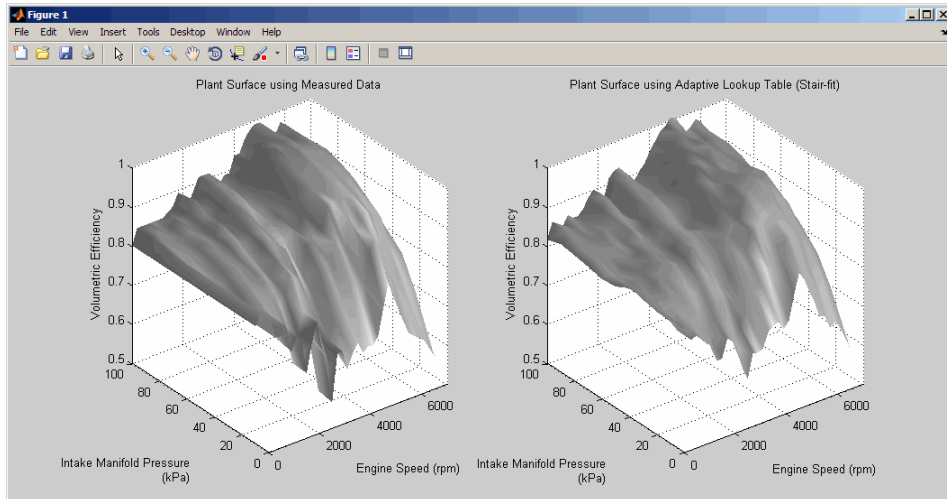
- 2 In the Simulink Editor, select **Simulation > Run** to start the adaptation process.

A figure window opens that shows the volumetric efficiency of the engine as a function of the intake manifold pressure and engine speed:

- The left plot shows the measured volumetric efficiency as a function of intake manifold pressure and engine speed.
- The right plot shows the volumetric efficiency as it adapts with the time-varying intake manifold pressure and engine speed.



During simulation, the lookup table values displayed on the right plot adapt to the variations in the I/O data. The left and the right plots resemble each other after a few seconds, as shown in the next figure.



Tip During simulation, the `Cell Number` and `Adaptive Table Outputs` blocks in the Simulink model display the cell number, and the adapted lookup table value in the cell, respectively.

- 3 Pause the simulation by selecting **Simulation > Pause**.

This action also exports the adapted table values `Tout` to the MATLAB workspace.

Note After you pause the simulation, the adapted table values are stored in the `Adaptive Lookup Table` block.

- 4 Examine that the left and the right plots match. This resemblance indicates that the table values have adapted to the time-varying I/O data.
- 5 Lock a table cell so that only one cell adapts. You may find this feature useful if a portion of the data is highly erratic or otherwise difficult for the algorithm to handle.
 - a Select **Simulation > Run** to restart the simulation.
 - b Double-click the `Lock` block. This action toggles the switch and feeds the output of the `ON` block to the `Lock` input port of the `Adaptive Lookup Table (nD Stair-Fit)` block.

You can view the number of the locked cell in the `Cell Number` block in the Simulink model.

- 6 After the table values adapt to the time-varying I/O data, you can continue to use the `Adaptive Lookup Table` block as a static lookup table:
 - a In the Simulink model window, double-click the `Enable` block. This action toggles the switch, and disables the adaptation.
 - b Select **Simulation > Run** to restart the simulation, if it is not already running.

During simulation, the `Adaptive Lookup Table` block works like a static lookup table, and continues to estimate the output values as the input values change. You can see the current lookup table value in the `Adaptive Table Outputs` block in the Simulink model window.

Note After you disable the adaptation, the `Adaptive Lookup Table` block does not update the stored table values, and the figure that displays the table values does not update.

See Also

Adaptive Lookup Table (nD Stair-Fit)

More About

- “What are Adaptive Lookup Tables?” on page 6-2

Using Adaptive Lookup Tables in Real-Time Environment

You can use experimental data from sensor measurements collected by running various tests on a system in real time. The measured data is then sent to the adaptive table block to generate a lookup table describing the relation between the system inputs and output.

You can also use the Adaptive Lookup Table block in a real-time environment, where some time-varying properties of a system need to be captured. To do so, generate C code using Simulink Coder™ code generation software that can then be run in Simulink Real-Time™ or dSPACE® software. Because you can start, stop, or reset the adaptation if you want, use logic to enable the adaptation of the table data only when it is desired. The cell number output `N`, and the `Enable` and `Lock` inputs facilitate this process. Use the `Enable` input to start and stop the adaptation and the `Lock` input to update only one of the table cells. The `Lock` input combined with some logic using the cell number output `N` provide the means for updating only the desired table cells during a simulation run.

See Also

Adaptive Lookup Table (1D Stair-Fit) | Adaptive Lookup Table (2D Stair-Fit) | Adaptive Lookup Table (nD Stair-Fit)

Related Examples

- “Model Engine Using n-D Adaptive Lookup Table” on page 6-41

More About

- “What are Adaptive Lookup Tables?” on page 6-2

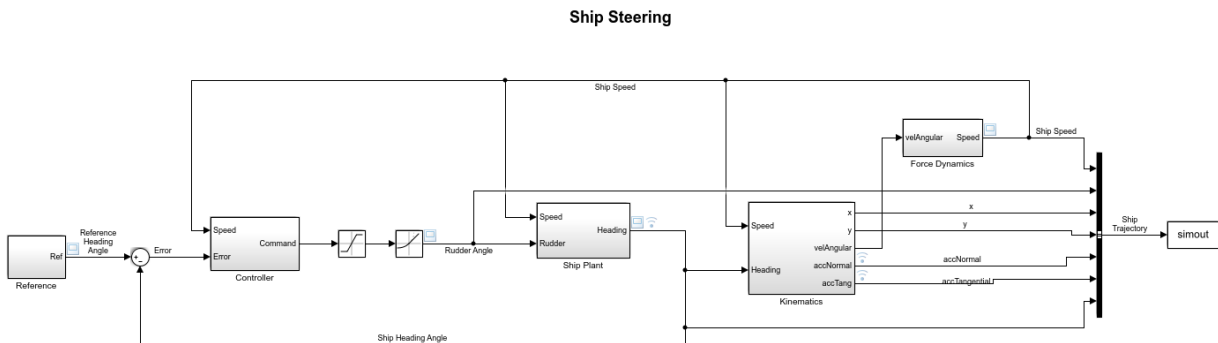
Design Optimization Using Lookup Table Requirements for Gain Scheduling (Code)

This example shows how to tune parameters in a lookup table in a model that uses gain scheduling to adjust the controller's response to a plant that varies. Model tuning uses the `sdo.optimize` command.

Ship Steering Model

Open the Simulink Model.

```
mdl = 'sdoShipSteering';
open_system(mdl)
```



Copyright 2016 The MathWorks, Inc.

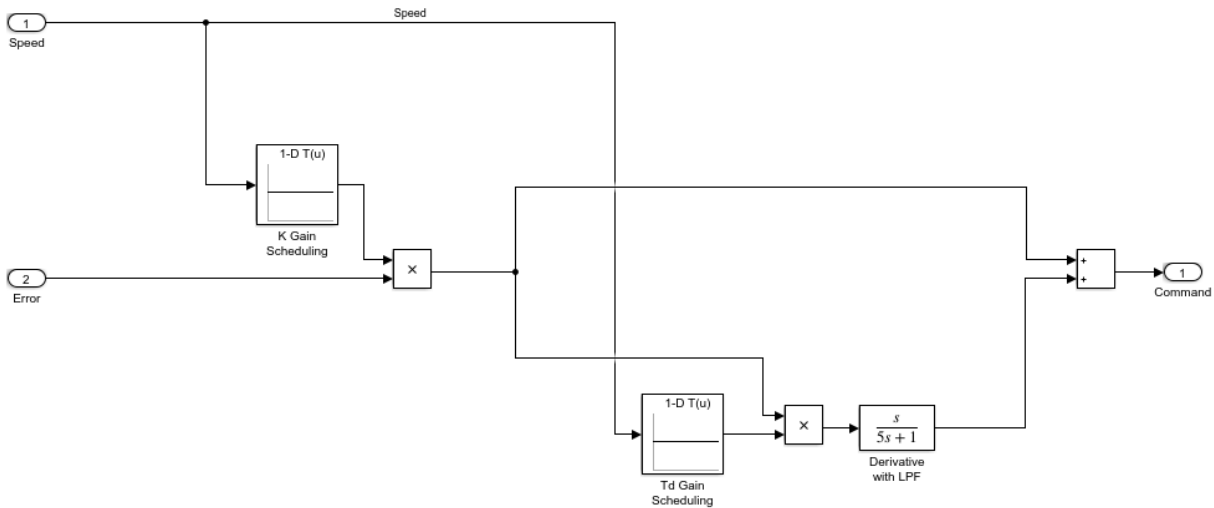
This model implements the Nomoto model which is commonly used for ship steering. The dynamic characteristics of a ship vary significantly with factors such as the ship speed. Therefore the rudder controller should also vary with speed, in order to meet requirements for steering the ship.

To keep the ship on course, a control loop compares the ship's heading angle with the reference heading angle, and a PD controller sends command signals to the rudder. The Ship Plant block implements the Nomoto model, a second order system whose parameters vary with the ship's speed. The ship is initially traveling at its maximum speed of 15 m/s, but it will slow down when the reference trajectory specifies a turn in the water. This turning, along with the force of the engine, is used by the Force

Dynamics block to compute the speed of the ship over time. The Kinematics block computes the trajectory of the ship.

Open the Controller block.

```
open_system([mdl '/Controller'])
```



When the speed changes, the ship plant also changes. Therefore the PD controller gains need to change, and speed is used as the scheduling variable. The controller is in the form $K(1 + sT_d)$ where K is the overall gain and T_d is the time constant associated with the derivative term. Gain scheduling is implemented via lookup tables, and the table data are specified by K and T_d . These are vectors which specify different values for different speeds. The different speeds are specified in the lookup table breakpoint vectors bpK and bpT_d .

Design Problem

The reference specifies that at 200 seconds, the ship should turn 180 degrees and reverse course. One requirement is that the ship heading angle needs to match the reference heading angle within an envelope. For the safety and comfort of passengers, a second requirement is that the total acceleration of the ship needs to stay within a bound of 0.25 g's, where 1 g is the acceleration of gravity at Earth's surface, 9.8 m/s/s.

The controller parameter vectors K and T_d will be design variables, and will be tuned to try to meet the requirements. If it is not possible to meet both requirements, then the lookup table breakpoints bpK and bpT_d will also be used as design variables. In that case, we will need to specify an additional requirement that bpK and bpT_d must be monotonically strictly increasing, because this is required for breakpoint vectors in Simulink lookup tables.

Specify Design Requirements

Specify the requirements which must be satisfied. First, the ship should follow the reference trajectory. Since the reference is essentially a step change from 0 to 180 degrees, you specify a step response envelope for the ship heading angle.

```
Requirements = struct;
Requirements.StepResponseEnvelope = sdo.requirements.StepResponseEnvelope(...
    'StepTime',      200, ... (seconds)
    'RiseTime',      75, ...
    'SettlingTime',  200, ...
    'PercentRise',   85, ... (%)
    'PercentOvershoot', 5, ...
    'PercentSettling', 1, ...
    'FinalValue',   pi ... (radians)
);
```

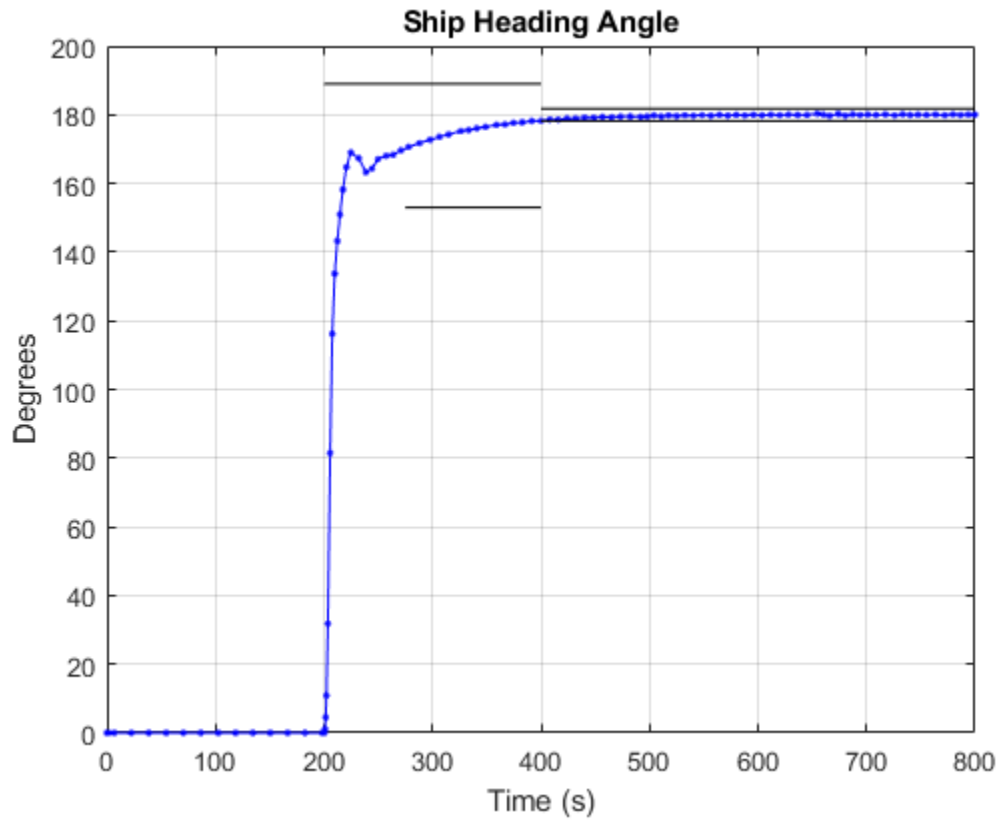
The second requirement is that for the safety and comfort of passengers, the total acceleration should not exceed 0.25 g's at any time. The total acceleration is composed of two components, a tangential component along the direction of the ship's motion, and a normal (horizontal) component. The requirement that total acceleration not exceed 0.25 g's corresponds to requiring that in the phase plane of tangential and normal acceleration, this ship's trajectory remain within a circle of radius 0.25×9.8 .

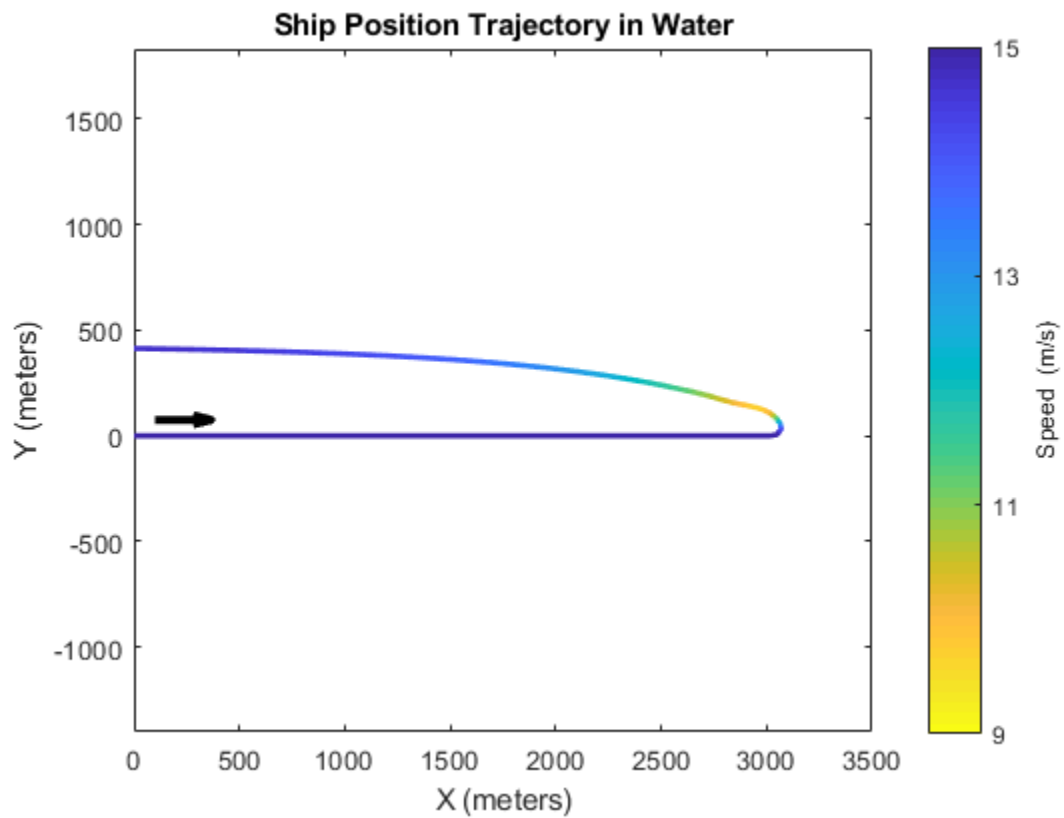
```
accelGravity = 9.8;
Requirements.Comfort = sdo.requirements.PhasePlaneEllipse(...
    'Radius', 0.25*[1 1] * accelGravity );
```

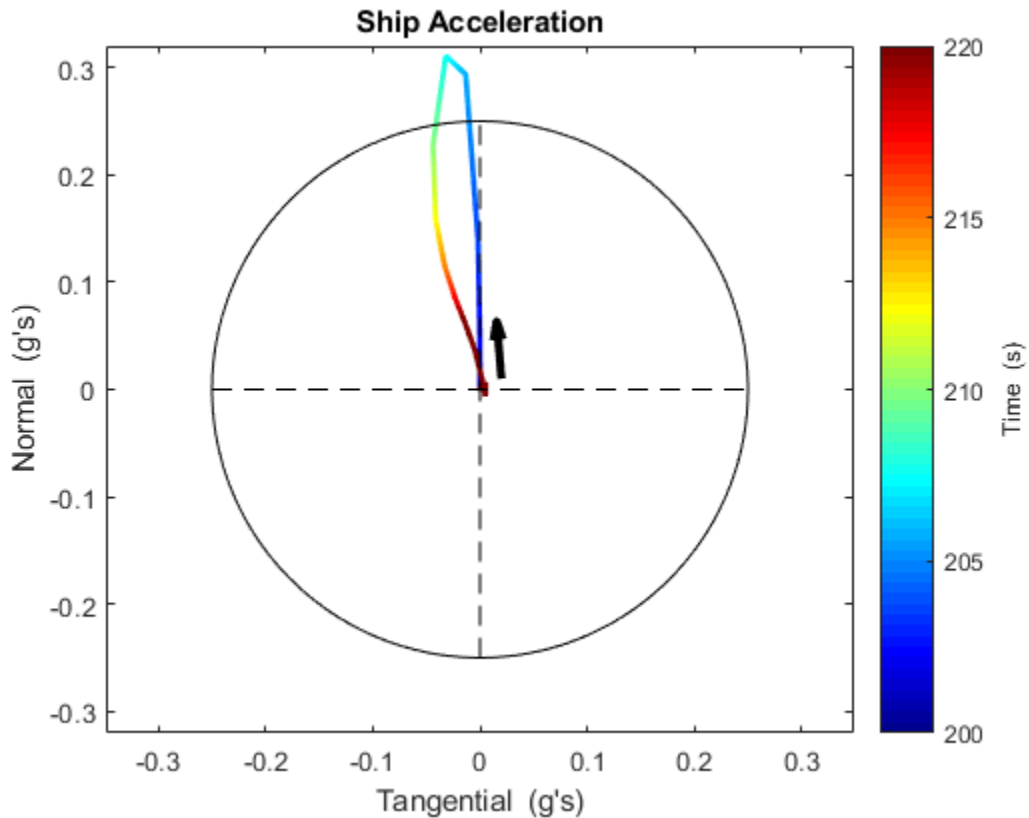
Examine the behavior of the ship with the initial, untuned controller parameters, to see whether the requirements are satisfied. Use the function `sdoShipSteeringPlots` to plot the ship's behavior. The first plot below shows that the ship's heading angle stays within the tolerance bounds of the step response envelope, satisfying the first requirement. The second plot shows the trajectory of the ship in the water. The black arrow indicates the starting position and direction of motion. The ship turns 180 degrees and the diameter is approximately 415 meters. The third plot shows the ship acceleration in the phase plane of tangential and normal acceleration. The black arrow indicates the starting point and

direction of evolution over time. The total acceleration exceeds the bound near 208 seconds, so the requirement for passenger safety and comfort is not satisfied.

```
hPlots = sdoShipSteeringPlots mdl, Requirements;
```







Specify Design Variables

Specify the design variables to be tuned by the optimization routine in order to satisfy the requirements. Specify the gains of the PD controller, K and T_d , as design variables. For initial values, use -0.1 for all entries in the K vector, and 50 for all entries in the T_d vector. If the requirements can't all be satisfied, then later the breakpoint vectors bpK and bpT_d can also be design variables.

```
DesignVars = sdo.getParameterFromModel mdl, {'K', 'Td'});
DesignVars(1).Value = -0.1*[1 1 1 1];
DesignVars(2).Value = 50*[1 1 1 1];
sdo.setValueInModel(mdl, DesignVars);
```

Create Optimization Objective Function

Create an objective function that will be called at each optimization iteration to evaluate the design requirements as the design variables are tuned. This cost function has input arguments for the model, design variables, simulator (defined below), and design requirements. The cost function uses the maximum of the Comfort requirements at all times when it is computed, in order to consolidate requirement evaluation results to a scalar so the number of elements is the same regardless of time steps taken by the Simulink solver.

```
type sdoShipSteering_Design
```

```
function Vals = sdoShipSteering_Design mdl, P, Simulator, Requirements)
%SDOSHIPSTEERING_DESIGN Objective function for ship steering
%
% Function called at each iteration of the optimization problem.
%
% The function is called with the model named mdl, a set of parameter
% values, P, a Simulator, and the design Requirements to evaluate. It
% returns the objective value and constraint violations, Vals, to the
% optimization solver.
%
% See the sdoExampleCostFunction function and sdo.optimize for a more
% detailed description of the function signature.
%
% See also sdoShipSteering_cmddemo

% Copyright 2016 The MathWorks, Inc.

%% Model Evaluation

% Simulate the model.
Simulator.Parameters = P;
Simulator = sim(Simulator);

% Retrieve logged signal data.
SimLog = find(Simulator.LoggedData, get_param(mdl, 'SignalLoggingName'));
Heading = find(SimLog, 'Heading');
NormalAccel = find(SimLog, 'NormalAccel');
TangenAccel = find(SimLog, 'TangenAccel');

% Evaluate the step response envelope requirement
Cleg_StepResponseEnvelope = evalRequirement(Requirements.StepResponseEnvelope, Heading.
```

```

% Evaluate the safety/comfort requirement on total acceleration.
Cleq_Comfort = evalRequirement(Requirements.Comfort, NormalAccel.Values, TangenAccel.Values);

%% Return Values.

% Collect the evaluated design requirement values in a structure to return
% to the optimization solver.
Vals.Cleq = Cleq_StepResponseEnvelope(:);
Vals.Cleq = [Vals.Cleq ; Cleq_Comfort];

% Evaluate monotonic variable requirement
if isfield(Requirements, 'Monotonic')
    Cleq_bpK = evalRequirement(Requirements.Monotonic, P(3).Value);
    Cleq_bpTd = evalRequirement(Requirements.Monotonic, P(4).Value);
    Vals.Cleq = [Vals.Cleq ; Cleq_bpK ; Cleq_bpTd];
end

end

```

The cost function requires a Simulator to run the model. Create the simulator and add model signals to log, so their values are available to the cost function.

```

Simulator = sdo.SimulationTest(mdl);

% Ship Heading Angle
Heading = Simulink.SimulationData.SignalLoggingInfo;
Heading.BlockPath = 'sdoShipSteering/Ship Plant';
Heading.LoggingInfo.LoggingName = 'Heading';
Heading.LoggingInfo.NameMode = 1;
Heading.OutputPortIndex = 1;

% Normal Acceleration
NormalAccel = Simulink.SimulationData.SignalLoggingInfo;
NormalAccel.BlockPath = 'sdoShipSteering/Kinematics';
NormalAccel.LoggingInfo.LoggingName = 'NormalAccel';
NormalAccel.LoggingInfo.NameMode = 1;
NormalAccel.OutputPortIndex = 4;

% Tangential Acceleration
TangenAccel = Simulink.SimulationData.SignalLoggingInfo;
TangenAccel.BlockPath = 'sdoShipSteering/Kinematics';
TangenAccel.LoggingInfo.LoggingName = 'TangenAccel';
TangenAccel.LoggingInfo.NameMode = 1;
TangenAccel.OutputPortIndex = 5;

```

```
% Collect logged signals
Simulator.LoggingInfo.Signals = [...
    Heading ; ...
    NormalAccel ; ...
    TangenAccel ];
```

Optimize Lookup Table Data

Before optimizing, set up the simulator for fast evaluation by enabling Simulink Fast Restart.

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

During optimization, the Simulink solver may generate a warning if the size of the time step becomes too small. Temporarily suppress this warning.

```
warnState = warning('query', 'Simulink:Engine:SolverMinStepSizeWarn');
warning('off', 'Simulink:Engine:SolverMinStepSizeWarn');
```

To optimize, define a handle to the cost function that uses the Simulator and Requirements defined above. Use an anonymous function that takes one argument (the design variables) and calls the objective function. Finally, call `sdo.optimize` to optimize the design variables to try to meet the requirements.

```
optimfcn = @(P) sdoShipSteering_Design mdl, P, Simulator, Requirements);
[Optimized_DesignVars, Info] = sdo.optimize(optimfcn, DesignVars);
```

```
Optimization started 15-Aug-2017 01:37:10
```

Iter	F-count	f(x)	max constraint	Step-size	First-order optimality
0	17	0	0.6127		
1	34	0	0.3215	0.865	0.321
2	57	0	0.3079	0.174	0.308
3	103	0	0.3079	0.00127	0.308

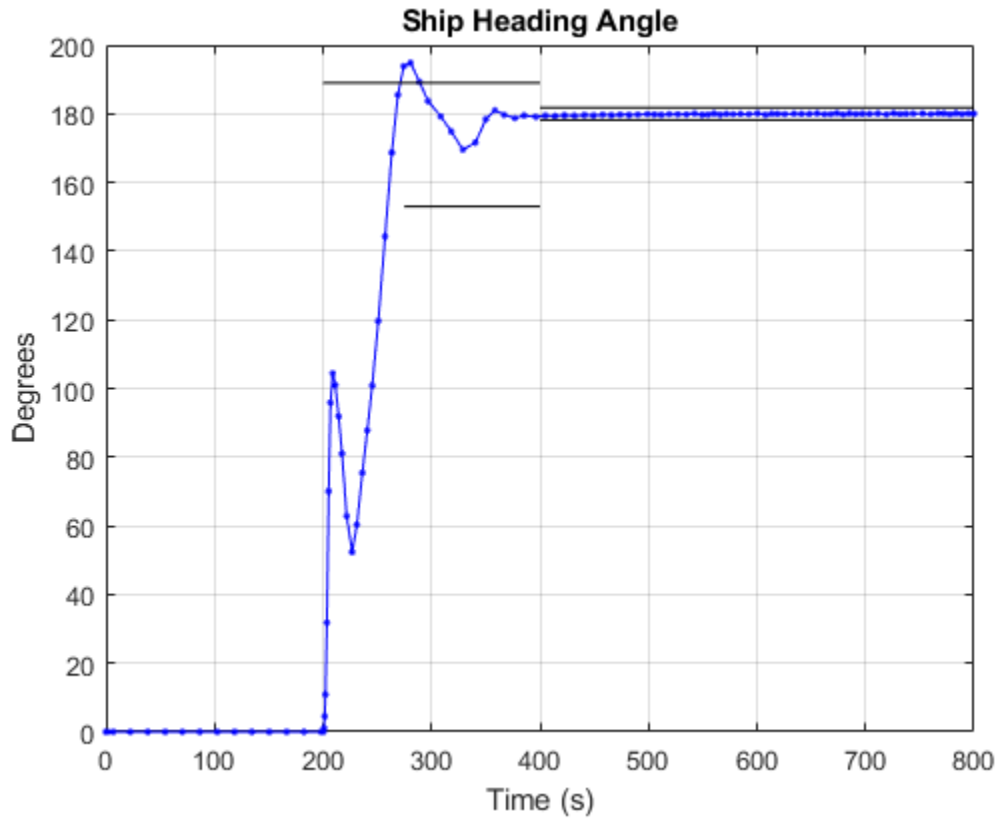
```
Converged to an infeasible point.
```

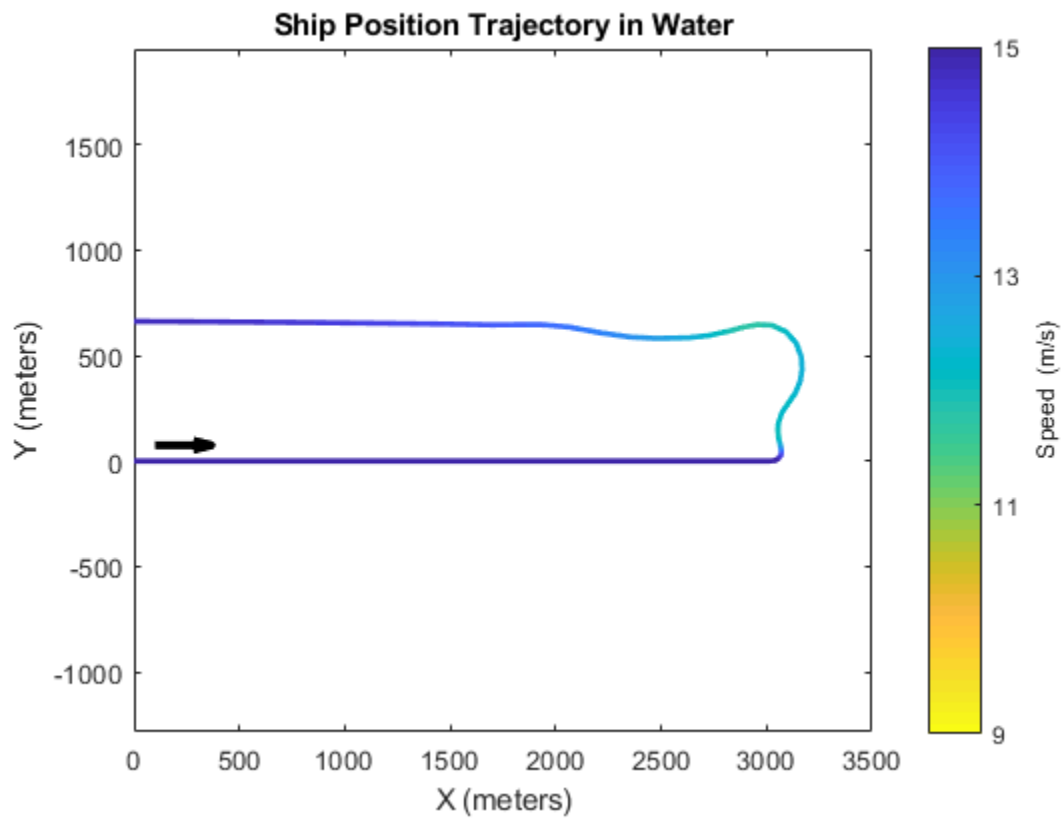
```
fmincon stopped because the size of the current step is less than
the selected value of the step size tolerance but constraints are not
satisfied to within the selected value of the constraint tolerance.
```

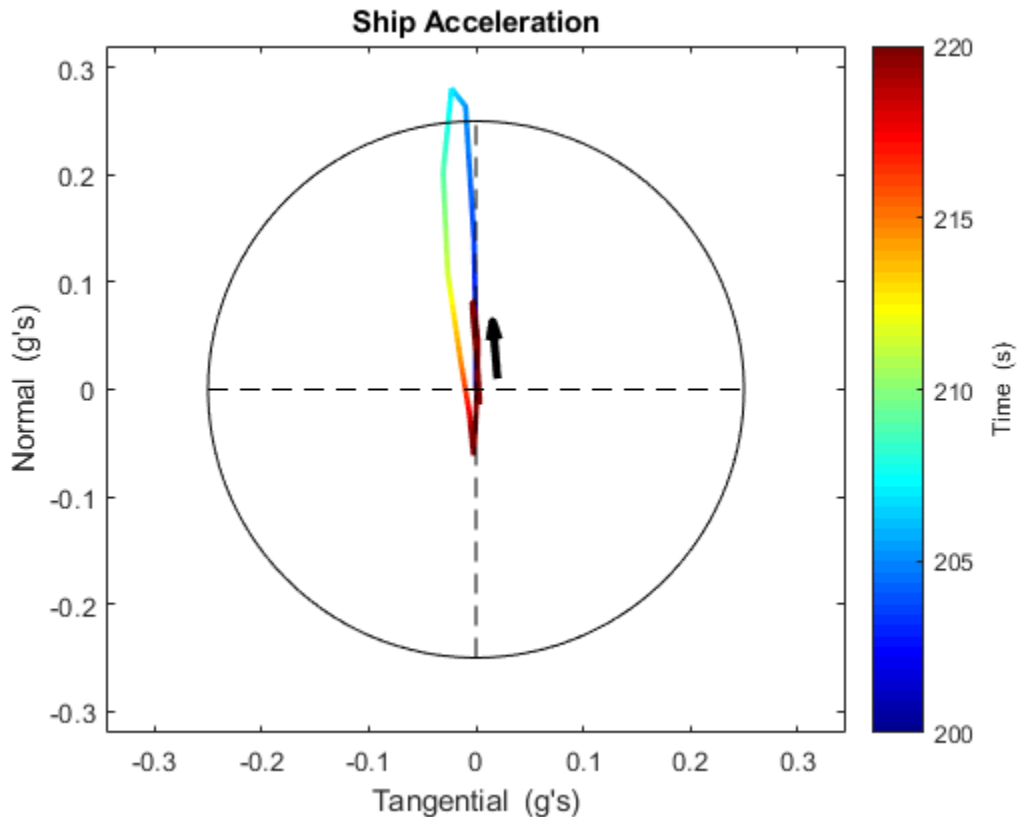
The display indicates that the optimizer was not able to satisfy all requirements. Try the optimized design variables in the model and plot the results. The heading angle is not

within the required step response envelope, and the total acceleration still exceeds the allowable level during part of the turn. In addition, the turning diameter has increased to 660 meters, so the turn is not as tight as with untuned gains.

```
sdo.setValueInModel mdl, Optimized_DesignVars);  
hPlots = sdoShipSteeringPlots(mdl, Requirements, hPlots);
```







Optimize Lookup Table Data and Breakpoints

To try to meet the design requirements, use the optimization result from above as the start point, and tune additional variables. Add breakpoints `bpK` and `bpTd` as design variables. The ship's maximum speed is 15 m/s, and during turning it may slow to 60% of the maximum speed, or 9 m/s. Set the breakpoint initial values to be equally spaced between 9 and 15 m/s. Constrain the breakpoint minimum values to 9 m/s, and constrain the breakpoint maximum values to 15 m/s.

```
DesignVars = Optimized_DesignVars;
DesignVars(3:4) = sdo.getParameterFromModel mdl, {'bpK', 'bpTd'};

% Set initial values
DesignVars(3).Value = [9 11 13 15];
```

```
DesignVars(4).Value = [9 11 13 15];

% Constrain min and max values
DesignVars(3).Minimum = 9;
DesignVars(3).Maximum = 15;
DesignVars(4).Minimum = 9;
DesignVars(4).Maximum = 15;

% Set values in the model
sdo.setValueInModel mdl, DesignVars);
```

Breakpoints in the Simulink lookup table block must be strictly monotonically increasing. Add this to the design requirements.

```
Requirements.Monotonic = sdo.requirements.MonotonicVariable;
optimfcn = @(P) sdoShipSteering_Design(mdl, P, Simulator, Requirements);
```

Optimize the model by tuning all four design variables, to see if all requirements can be met.

```
[Optimized_DesignVars, Info] = sdo.optimize(optimfcn, DesignVars);
```

```
Optimization started 15-Aug-2017 01:37:37
```

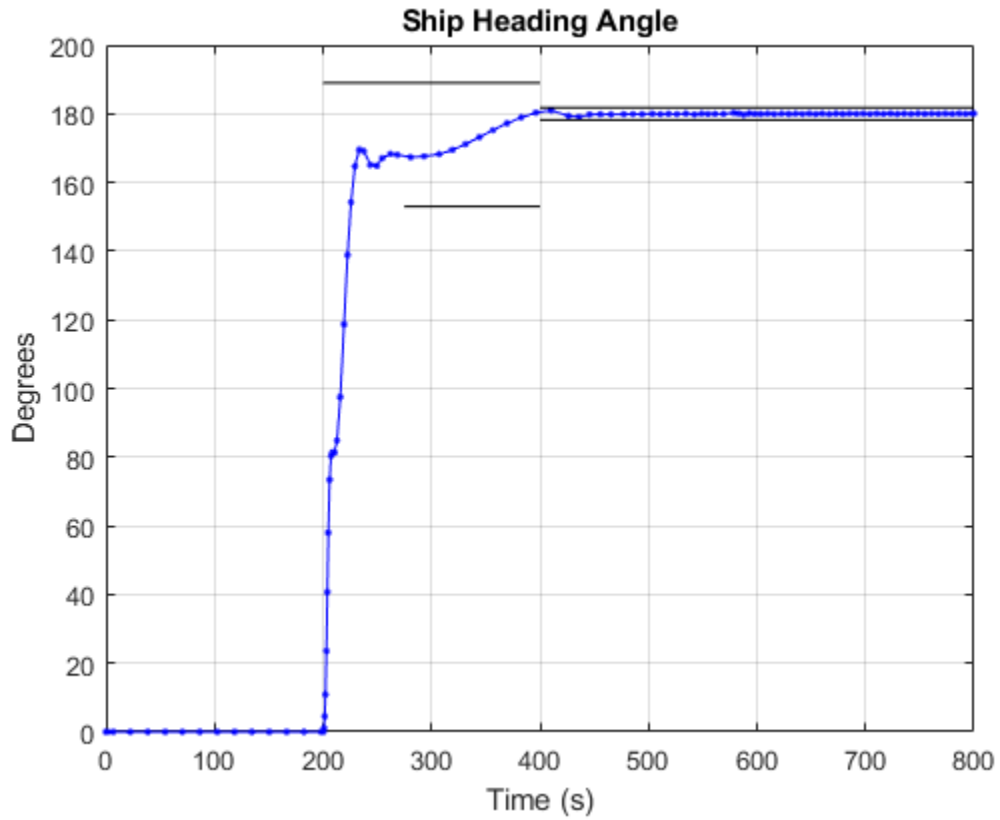
Iter	F-count	f(x)	max constraint	Step-size	First-order optimality
0	29	0	0.3079		
1	61	0	0.1432	0.148	1.01
2	94	0	0.03626	0.0858	0.597
3	123	0	0.01911	0.0548	0.0859
4	153	0	0.007837	0.0341	0.00627
5	183	0	0	0.0256	0.000903

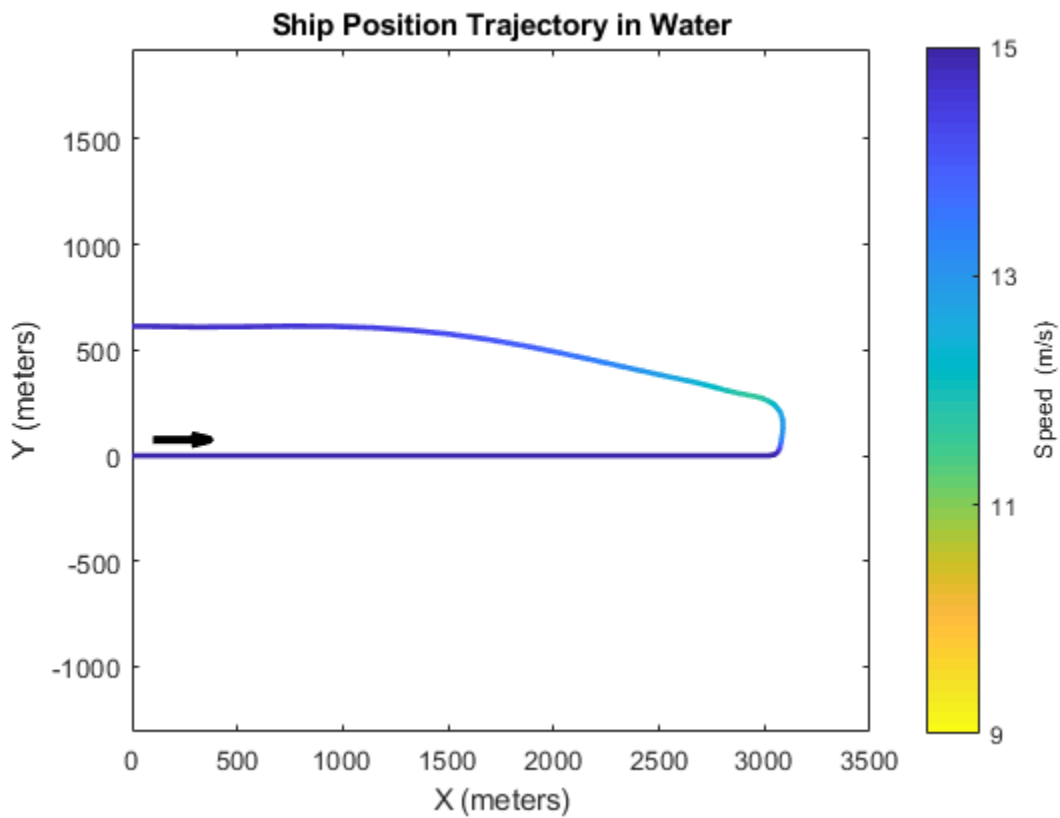
```
Local minimum found that satisfies the constraints.
```

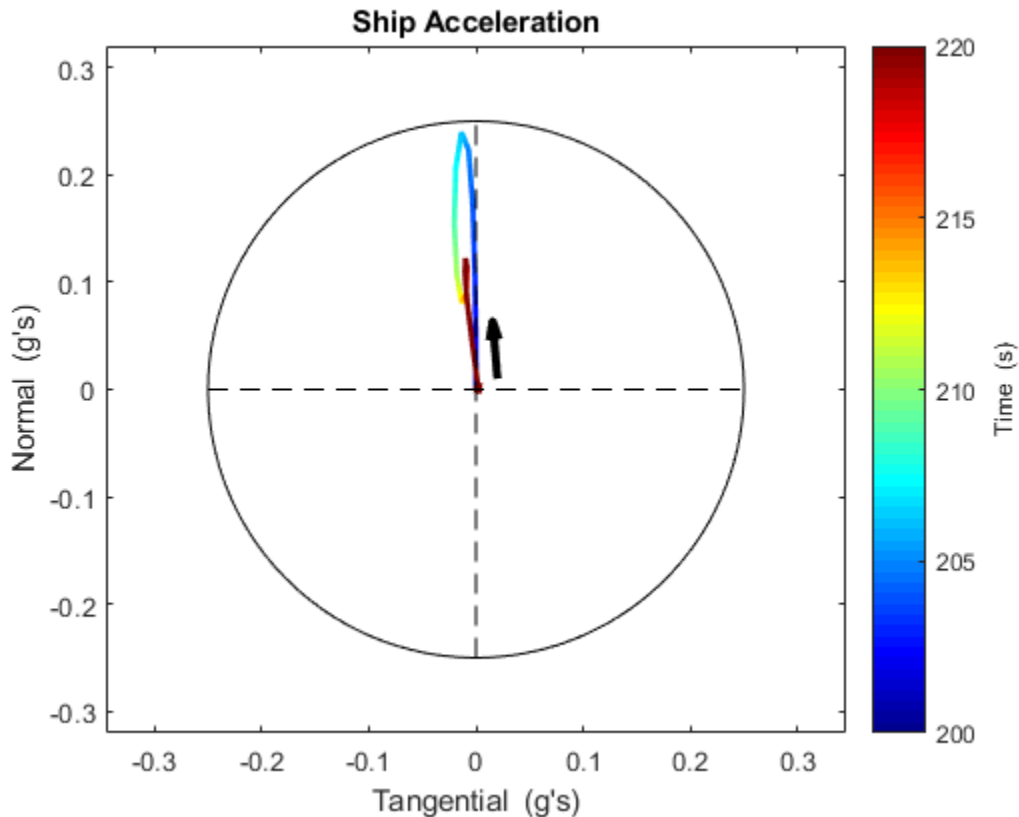
```
Optimization completed because the objective function is non-decreasing in  
feasible directions, to within the selected value of the optimality tolerance,  
and constraints are satisfied to within the selected value of the constraint tolerance.
```

The display indicates that the optimizer was able to satisfy all requirements. Try the optimized design variables in the model and plot the results. In the plots below, the heading angle is within the step response envelope, and the total acceleration is within the allowed range of 0.25 g's. In addition, the turning diameter is 615 meters, which is tighter than when breakpoints were not tuned.

```
sdo.setValueInModel mdl, Optimized_DesignVars);
hPlots = sdoShipSteeringPlots(mdl, Requirements, hPlots);
```







In this example, the ship plant varied with the ship speed, so the controller gains also needed to vary. Gain scheduling was implemented using lookup tables. By tuning the gains and breakpoint values in the controller, the ship was able to follow the reference heading angle, while also constraining total acceleration to ensure a safe and comfortable ride for passengers.

Related Examples

To learn how to optimize the lookup tables in the gain scheduled controller using the Response Optimization tool, see "Design Optimization Using Lookup Table Requirements for Gain Scheduling (GUI)".

```
% Close the model and figures, and restore state of warnings.
fastRestart(Simulator, 'off'); % restore fast restart settings
```

```
bdclose mdl)
close (hPlots)
warning(warnState); % restore state of warning
```

See Also

```
sdo.requirements.FunctionMatching | sdo.requirements.MonotonicVariable
| sdo.requirements.PhasePlaneEllipse |
sdo.requirements.PhasePlaneRegion |
sdo.requirements.RelationalConstraint |
sdo.requirements.SmoothnessConstraint
```

Related Examples

- “Design Optimization Using Lookup Table Requirements for Gain Scheduling (GUI)” on page 6-73
- “Estimate Lookup Table Values from Data” on page 6-21

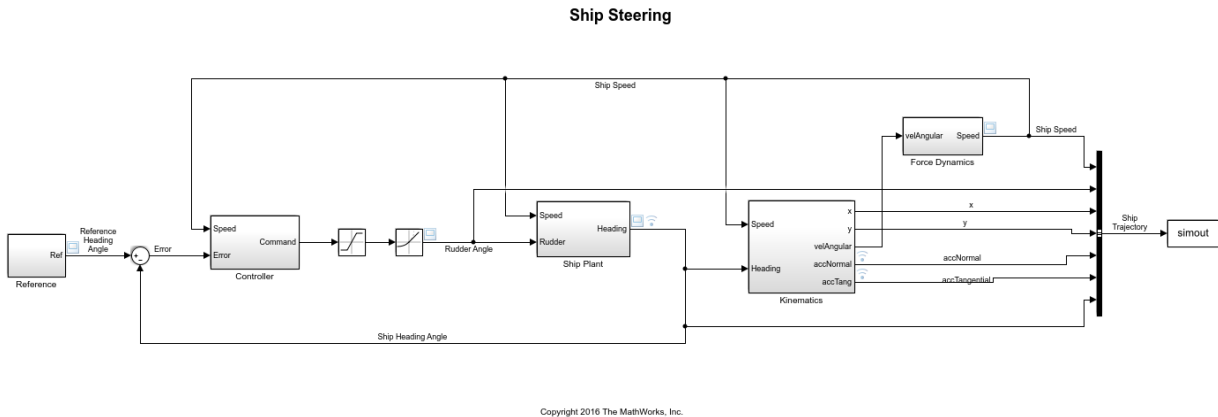
Design Optimization Using Lookup Table Requirements for Gain Scheduling (GUI)

This example shows how to tune parameters in a lookup table in a model that uses gain scheduling to adjust the controller's response to a plant that varies. Model tuning uses the Response Optimization tool.

Ship Steering Model

Open the Simulink Model.

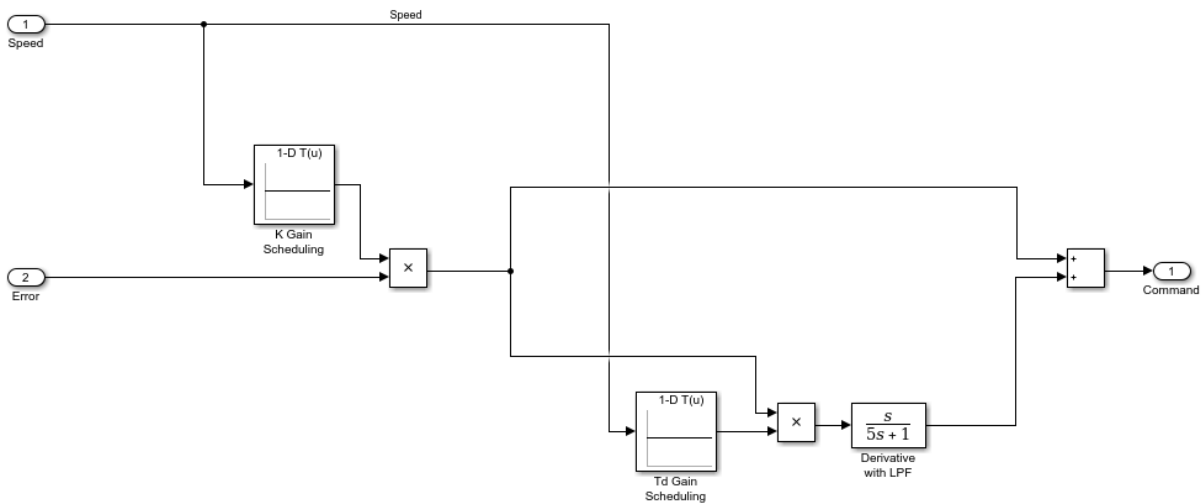
```
open_system('sdoShipSteering')
```



This model implements the Nomoto model which is commonly used for ship steering. The dynamic characteristics of a ship vary significantly with factors such as the ship speed. Therefore the rudder controller should also vary with speed, in order to meet requirements for steering the ship.

To keep the ship on course, a control loop compares the ship's heading angle with the reference heading angle, and a PD controller sends command signals to the rudder. The Ship Plant block implements the Nomoto model, a second order system whose parameters vary with the ship's speed. The ship is initially traveling at its maximum speed of 15 m/s, but it will slow down when the reference trajectory specifies a turn in the water. This turning, along with the force of the engine, is used by the Force Dynamics block to compute the speed of the ship over time. The Kinematics block computes the trajectory of the ship.

Open the Controller block by double clicking it.



When the speed changes, the ship plant also changes. Therefore the PD controller gains need to change, and speed is used as the scheduling variable. The controller is in the form $K(1 + sT_d)$ where K is the overall gain and T_d is the time constant associated with the derivative term. Gain scheduling is implemented via lookup tables, and the table data are specified by K and T_d . These are vectors which specify different values for different speeds. The different speeds are specified in the lookup table breakpoint vectors bpK and bpT_d .

Design Problem

The reference specifies that at 200 seconds, the ship should turn 180 degrees and reverse course. One requirement is that the ship heading angle needs to match the reference heading angle within an envelope. For the safety and comfort of passengers, a second requirement is that the total acceleration of the ship needs to stay within a bound of 0.25 g's, where 1 g is the acceleration of gravity at Earth's surface, 9.8 m/s/s.

The controller parameter vectors K and T_d will be design variables, and will be tuned to try to meet the requirements. If it is not possible to meet both requirements, then the lookup table breakpoints bpK and bpT_d will also be used as design variables. In that case, we will need to specify an additional requirement that bpK and bpT_d must be

monotonically strictly increasing, because this is required for breakpoint vectors in Simulink lookup tables.

Open the Response Optimization Tool

In the Simulink model **Analysis** menu, select **Response Optimization**.

Specify Design Requirements

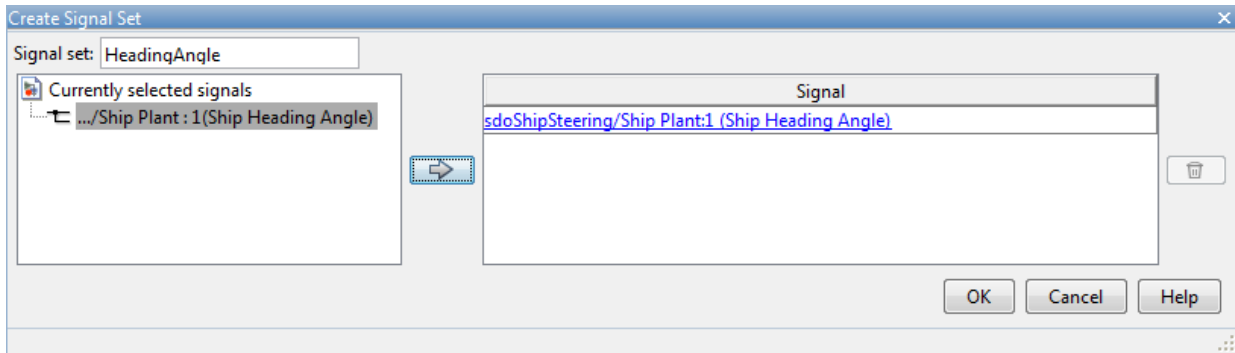
Specify the requirements which must be satisfied. First, the ship should follow the reference trajectory. Since the reference is essentially a step change from 0 to 180 degrees, you specify a step response envelope for the ship heading angle. In the toolstrip, click **New** and select **Step Response Envelope**. Set the initial value to 0 and the final value to pi radians. Set the step time to 200 seconds. Set the rise time as 75 seconds and the rise percent to 85%. Set the settling time to 200 seconds and the settling percent to 1%. Set the percent overshoot to 5%. To specify that this requirement applies to the ship heading, click +.

The screenshot shows a 'Create Requirement' dialog box with the following fields and controls:

- Name:** StepRespEnvelope
- Specify Step Response Characteristics:**
 - Initial value: 0
 - Final value: 3.1416
 - Step time: 200 seconds
 - Rise time: 75 seconds
 - % Rise: 85
 - Settling time: 200 seconds
 - % Settling: 1.0000
 - % Overshoot: 5
 - % Undershoot: 1.0000
- Select Signals to Bound:**

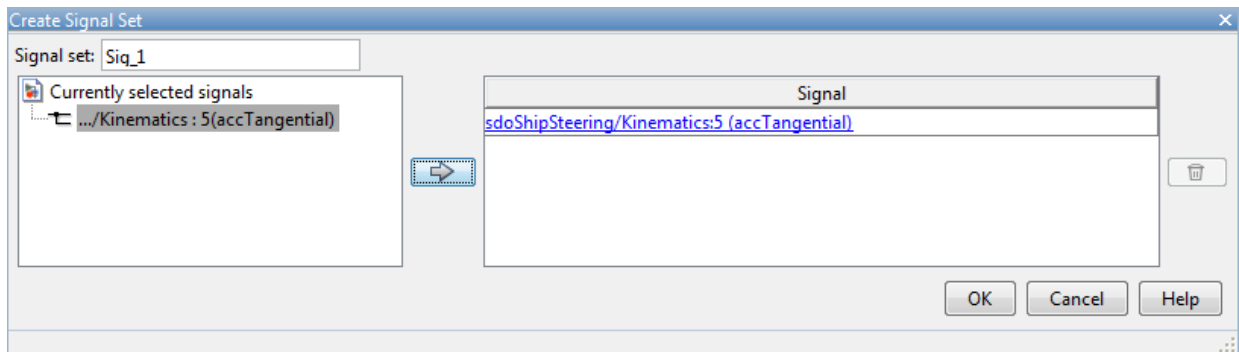
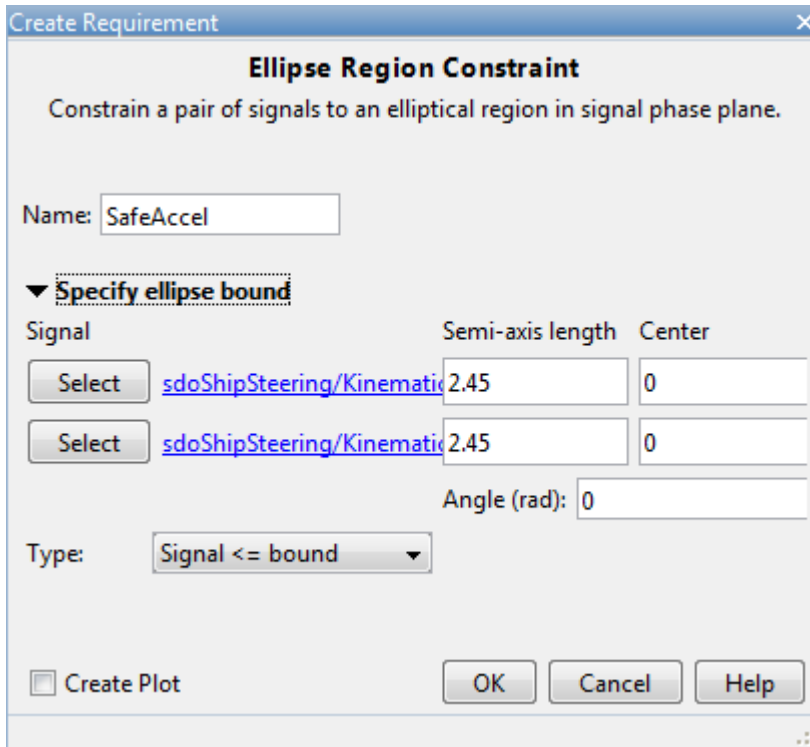
	Signal	
<input checked="" type="checkbox"/>	HeadingAngle (sdoShipSteering/Ship Plant:1)	
- Create Plot
- Buttons: OK, Cancel, Help

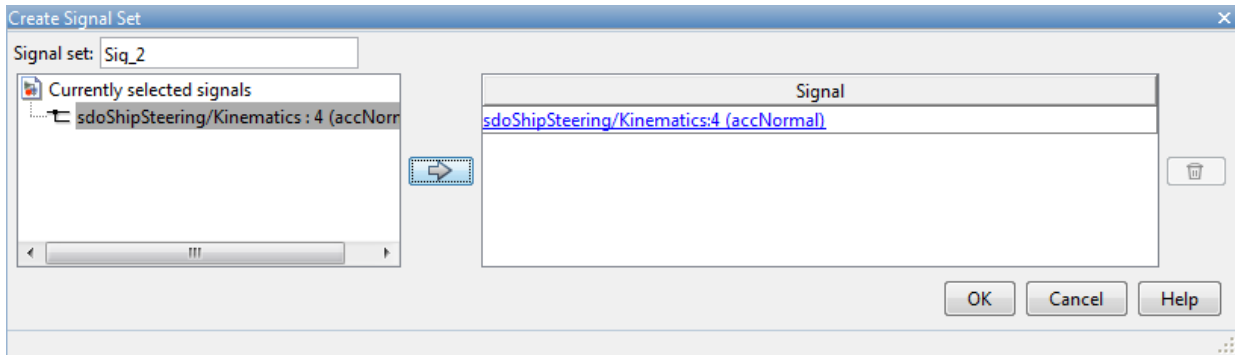
In the Simulink model click the ship heading signal, which is the output of the **Ship Plant** block. Select this signal in the Create Signal Set dialog, and click the arrow button to make it the designated signal, and click OK.



The second requirement is that for the safety and comfort of passengers, the total acceleration should not exceed 0.25 g's at any time. The total acceleration is composed of two components, a tangential component along the direction of the ship's motion, and a normal (horizontal) component. The requirement that total acceleration not exceed 0.25 g's corresponds to requiring that in the phase plane of tangential and normal acceleration, this ship's trajectory remain within a circle of radius 0.25×9.8 .

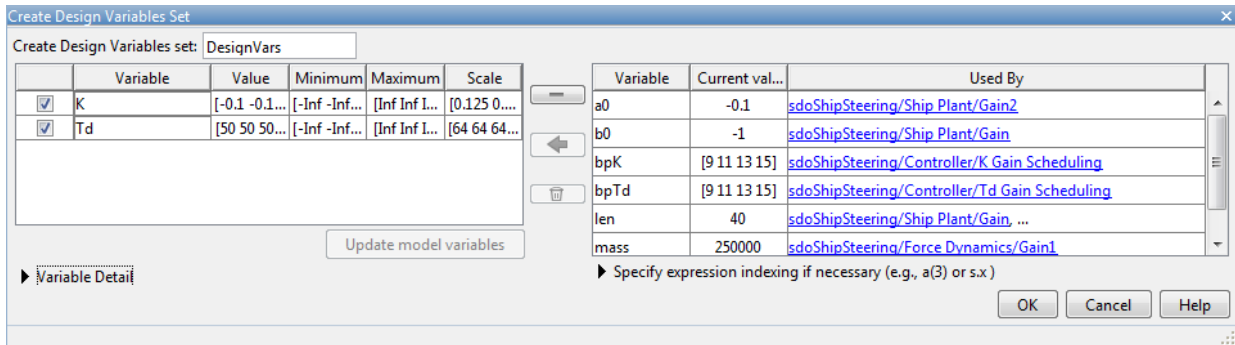
In the toolbar, click **New** and select **Ellipse Region Constraint**. Specify the name as SafeAccel, and the semi-axis length for both signals as $0.25 \times 9.8 = 2.45$. To specify that the requirement applies to the tangential acceleration of the ship, click the Select button. In the Simulink model click the tangential acceleration signal, which is output from the **Kinematics** block. Select this signal in the Create Signal Set dialog, and click the arrow button to make it the designated signal, and click OK. Similarly, to specify that the requirement applies to the normal acceleration of the ship, in the Ellipse Region Constraint dialog click the other Select button, and use the Create Signal Set dialog to specify the normal acceleration signal.





Specify Design Variables

Specify the design variables to be tuned by the optimization in order to satisfy the requirements. In the toolstrip, click the selection box next to **Design Variables Set** and then click New. Select the gains of the PD controller, K and T_d , and click the arrow button to designate them as design variables. Use -0.1 for all entries in the value of the K vector, and use 50 for all entries in the value of the T_d vector, and click OK. If the requirements can't all be satisfied, then later the breakpoint vectors bpK and bpT_d can also be tried as design variables.

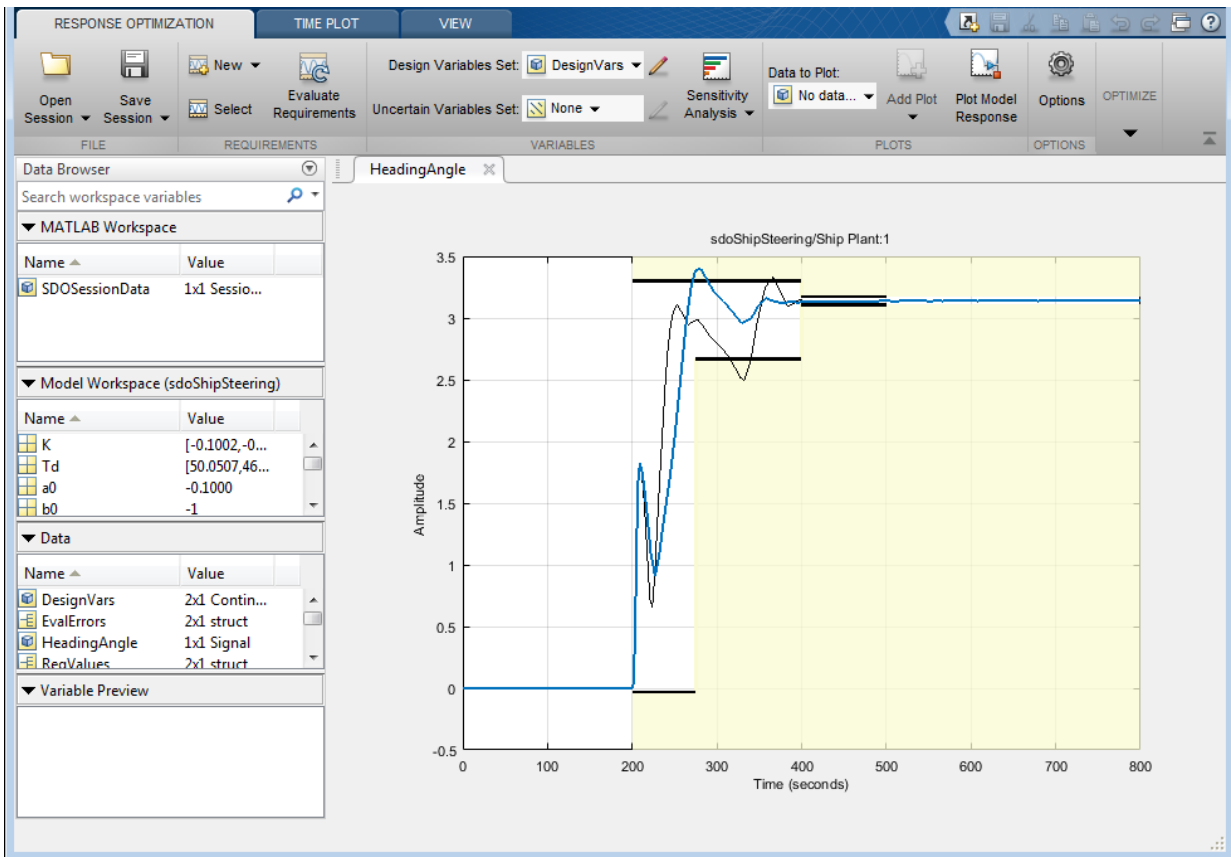


Optimize Lookup Table Data

During optimization, the Simulink solver may generate a warning if the size of the time step becomes too small. Temporarily suppress this warning.

```
warnState = warning('query', 'Simulink:Engine:SolverMinStepSizeWarn');
warning('off', 'Simulink:Engine:SolverMinStepSizeWarn');
```

In the Response Optimization tool, click **Optimize**. The ship heading angle does not meet the required step response envelope, as can be seen in the step response plot in Response Optimization tool and in the Optimization Progress dialog, where the value at the last iteration is still positive, which indicates violation of the requirement. The requirement for safe acceleration is also not met, as seen in the Optimization Progress dialog, where the value at the last iteration is also positive.



Iteration	F-count	StepRespEnvelope (<=0)	SafeAccel (<=0)
0	17	-1.1269e-04	0.6127
1	34	0.0665	0.3215
2	57	0.0307	0.3079
3	103	0.0307	0.3079

Optimization started 30-Dec-2016 16:14:35

Optimization failed to converge, 30-Dec-2016 16:15:01
 The optimizer encountered 2 errors during the optimization. Details of the errors have been written to 'EvalErrors' in the Design Optimization workspace.

Save Iteration... Display Options... Optimize

Optimize Lookup Table Data and Breakpoints

To try to meet the design requirements, use the optimization result from above as the start point, and tune additional variables. Add breakpoints bpK and $bpTd$ as design variables. The ship's maximum speed is 15 m/s, and during turning it may slow to 60% of the maximum speed, or 9 m/s. Set the breakpoint initial values to be equally spaced between 9 and 15 m/s. Constrain the breakpoint minimum values to 9 m/s, and constrain the breakpoint maximum values to 15 m/s.

6 Lookup Tables

Variable	Value	Minimum	Maximum	Scale
<input checked="" type="checkbox"/> K	[-0.1002...	[-Inf -Inf...	[Inf Inf I...	[0.125 0...
<input checked="" type="checkbox"/> Td	[50.0506...	[-Inf -Inf...	[Inf Inf I...	[64 64 6...
<input checked="" type="checkbox"/> bpK	[9 11 13 ...	[9 9 9 9]	[15 15 15...	[16 16 1...
<input checked="" type="checkbox"/> bpTd	[9 11 13 ...	[9 9 9 9]	[15 15 15...	[16 16 1...

Variable	Current value	Used By
a0	-0.1	sdoShipSteering/Ship Plant/Gain2
b0	-1	sdoShipSteering/Ship Plant/Gain
len	40	sdoShipSteering/Ship Plant/Gain, ...
mass	250000	sdoShipSteering/Force Dynamics/Gain1
speed_0	15	sdoShipSteering/Force Dynamics/Fmotor, ...

Breakpoints in the Simulink lookup table block must be strictly monotonically increasing. Add this to the design requirements.

Monotonic Variable
Specify a requirement that a vector, matrix or array be monotonic.

Name:

▼ **Specify Monotonic Constraint**

Variable: <1-dimensional>

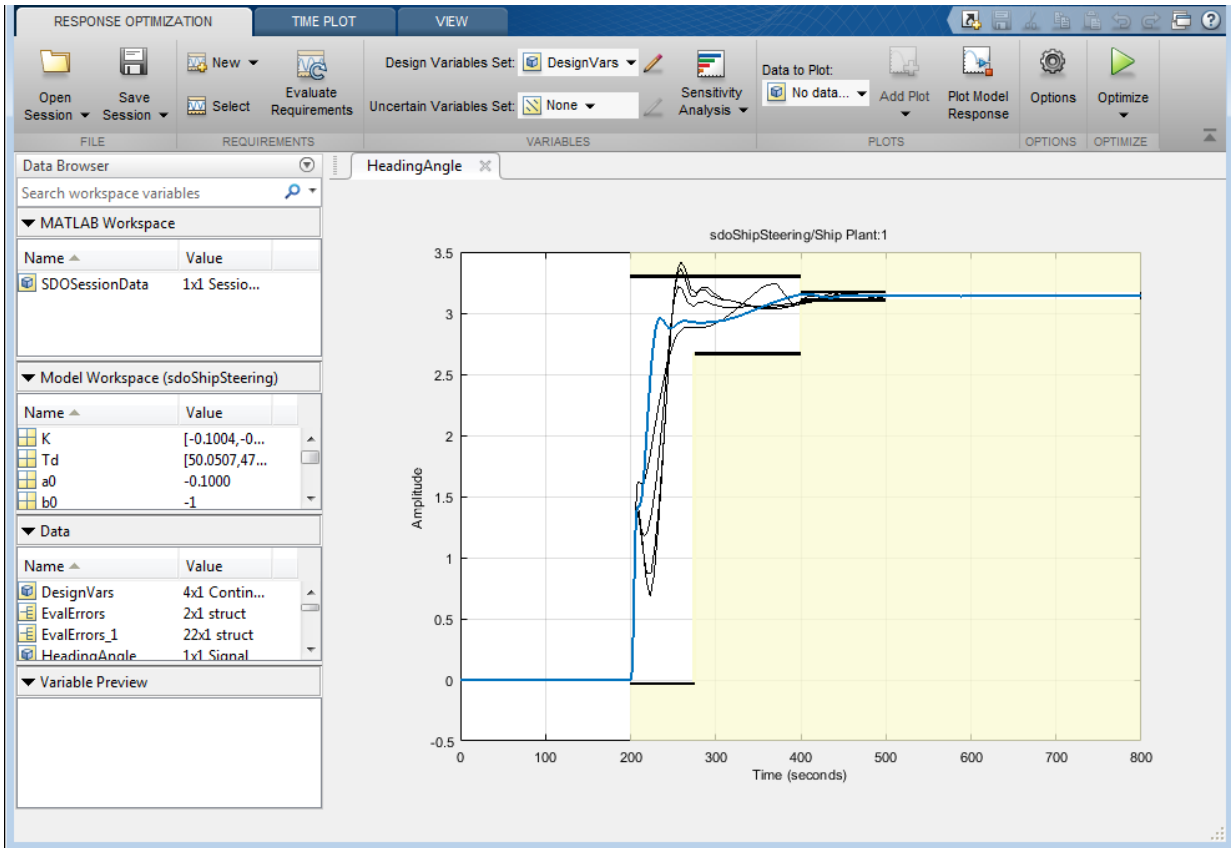
Direction

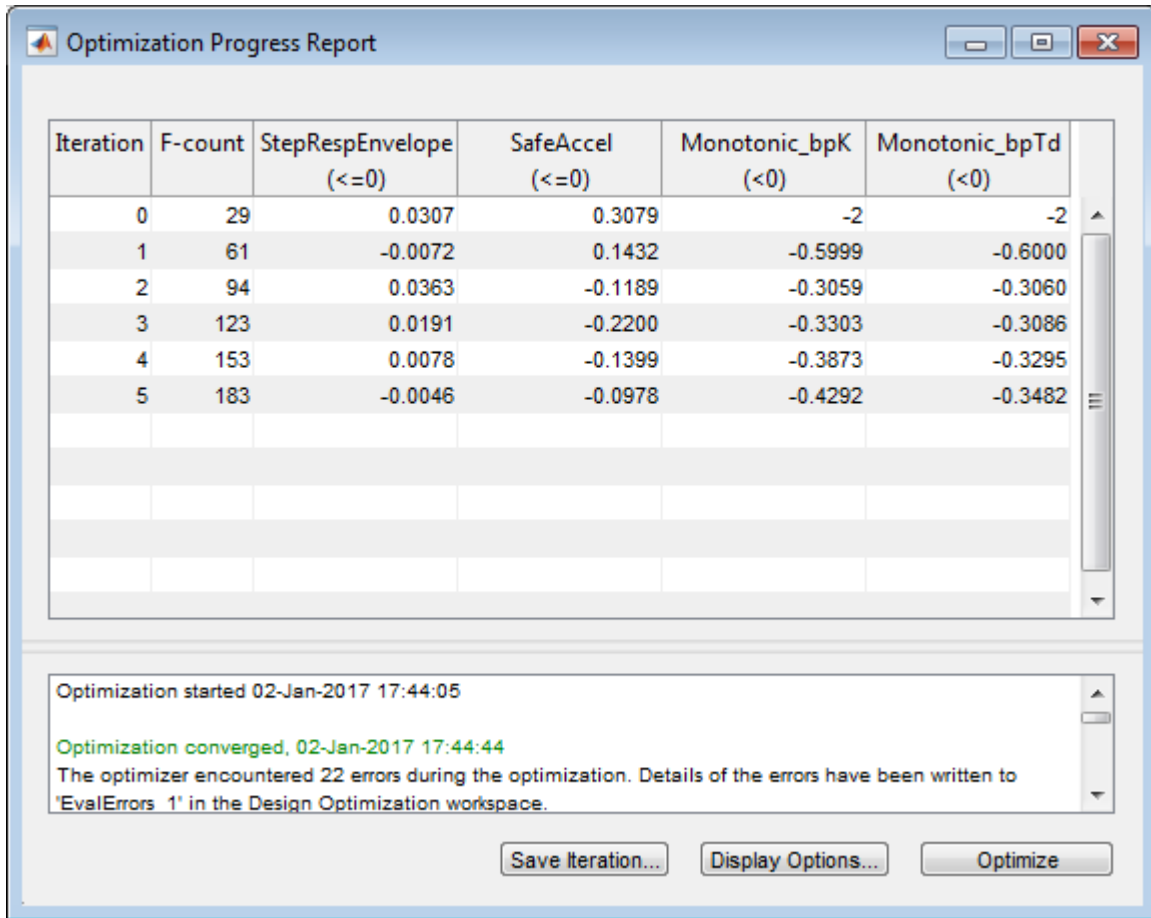
Dimension 1:

The screenshot shows a dialog box titled "Create Requirement" with a close button (X) in the top right corner. The main heading is "Monotonic Variable" and the instruction below it says "Specify a requirement that a vector, matrix or array be monotonic." The "Name:" field contains "Monotonic_bpTd". Under the "Specify Monotonic Constraint" section, the "Variable" dropdown is set to "bpTd" and is followed by the text "<1-dimensional>" and a "Show in Model" button. The "Direction" section has a "Dimension 1:" dropdown set to "Strictly increasing". At the bottom left, there is a checkbox labeled "Create Plot" which is currently unchecked. At the bottom right, there are three buttons: "OK", "Cancel", and "Help".

In the Response Optimization tool, click **Optimize**. This time the ship heading angle meets the required step response envelope, as can be seen in the step response plot in Response Optimization tool and in the Optimization Progress dialog, where the value at the last iteration is negative, which indicates the requirement is satisfied. The requirement for safe acceleration is also satisfied, as seen in the Optimization Progress dialog, where the value at the last iteration is also negative. Similarly, the lookup table breakpoints satisfy the monotonic requirements.

6 Lookup Tables





In this example, the ship plant varied with the ship speed, so the controller gains also needed to vary. Gain scheduling was implemented using lookup tables. By tuning the gains and breakpoint values in the controller, the ship was able to follow the reference heading angle, while also constraining total acceleration to ensure a safe and comfortable ride for passengers.

Related Examples

To learn how to optimize the lookup tables in the gain scheduled controller using the `sdo.optimize` command, see "Design Optimization Using Lookup Table Requirements for Gain Scheduling (Code)".

```
% Close the model and restore state of warnings.  
bdclose('sdoShipSteering')  
warning(warnState); % restore state of warning
```

See Also

Related Examples

- “Design Optimization Using Lookup Table Requirements for Gain Scheduling (Code)” on page 6-56
- “Estimate Lookup Table Values from Data” on page 6-21
- “Specify Time-Domain Design Requirements in the Tool” on page 3-19
- “Specify Variable Requirements in the Tool” on page 3-39