Simulink[®] Design Optimization[™] 1 Getting Started Guide

MATLAB[®] SIMULINK[®]



How to Contact The MathWorks



(a)

www.mathworks.comWebcomp.soft-sys.matlabNewsgroupwww.mathworks.com/contact_TS.htmlTechnical Support

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



508-647-7000 (Phone) 508-647-7001 (Fax)

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

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

Simulink[®] Design Optimization[™] Getting Started Guide

© COPYRIGHT 1993–2009 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

The 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 September 2009 Online only New for Version 1 (Release 2009a) Revised for Version 1.1 (Release 2009b)

Contents

Product Overview

What You Can Accomplish Using This Product	1-2
Upgrading from an Earlier Release	1-4
Required and Related Products	1-5
Documentation and Demos Accessing Documentation Accessing Demos	1-6 1-6 1-6

Estimating Model Parameters

Types of Data for Parameter Estimation	2-2
Quick Start — Estimating Model Parameters	2-4
Parallel Computing for Parameter Estimation	2-12

Optimizing Model Parameters

3

2

Types of Time-Domain Design Requirements for			
Optimizing Parameters			
Quick Start — Optimizing Model Parameters	3-3		

Optimization-Based Linear Control Design

When to Use Optimization-Based Linear Control 4-2 Design 4-2 Types of Time- and Frequency-Domain Design 4-3 Quick Start — Optimization-Based Linear Control 4-4

Tutorial — Preparing Data for Parameter Estimation Using the GUI

5

About This Tutorial	5-2 5-2
About the Sample Data	5-2
Configuring a Project for Parameter Estimation	5-4
Importing Data into the GUI	5-6
Importing Input Data and Time Vector	5-6
Importing Output Data and Time Vector	5-11
Analyzing Data	5-14
Selecting Data for Estimation	5-16
Selecting Output Data	5-16
Selecting Input Data	5 - 22

Removing Outliers	5-25
Why Remove Outliers	5 - 25
How to Remove Outliers	5-25
Filtering Data	5-29
Filtering Output Data	5-29
Filtering Input Data	5-32
Interpolating Missing Data	5-34
Saving the Project	5-37

Tutorial — Estimating Parameters from Measured Data Using the GUI

About This Tutorial	6-2
Objectives	6-2
About the Model	6-3
Estimating Model Parameters Using Default Estimation	
Settings	6-7
Overview of the Estimation Process	6-7
Specifying Parameters and Estimation Data	6-8
Validating Model Parameters	6-13
Improving Estimation Results Using Parameter	
Bounds	6-20
Strategy for Improving the Estimation Results	6-20
How to Specify Parameter Bounds	6-20
Validating Estimated Model Parameters	6-26

7

About This Tutorial Objectives Objectives Objectives About the Model Objectives Design Requirements Objectives	7-2 7-2 7-2 7-4
Configuring a Model for Optimizing Parameters	7-5
Optimizing Model Parameters to Meet Step Response Requirements	7-8 7-8 7-18 7-21
Refining Model Parameters to Track a Reference Signal	7-25
Saving the Project	7-31

Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the Command Line

About This Tutorial Objectives Objectives Objectives About the Model Objectives Design Requirements Objectives	8-2 8-2 8-2 8-4
Configuring a Model for Optimizing Parameters	8-5
Optimizing Model Parameters to Meet Step Response Requirements	8-8

Refining	Model	Param	eters to	Track a Re	ference	
Signal						8-15

Tutorial — Designing a PID Controller Using Optimization-Based Tuning

About This Tutorial	9-2
Objectives	9-2
About the Model	9-2
Design Requirements	9-4
Configuring a Project for Optimization-Based Control	
Design	9-5
Designing an Initial PID Controller to Meet Bode	
Magnitude and Phase Margins Requirements	9-11
Specifying the Controller Parameters Specifying Bode Magnitude and Phase Margin Design	9-11
Requirements	9-15
Designing the Controller	9-25
Refining the Controller Design to Meet Controller	
Output Bounds	9-32
Saving the Project	9-48

Tutorial — Modeling a System Using Adaptive Lookup Table

10

About This Tutorial	10-2
Objectives	10-2
About the Data	10-2

Overview of Modeling a System Using Adaptive Lookup Table	10-3
Building a Model Using Adaptive Lookup Table Blocks	10-4
Adapting the Lookup Table Values Using Time-Varying I/O Data	10-16

Examples

A

Getting Started		A-2
------------------------	--	-----

Index

Product Overview

- "What You Can Accomplish Using This Product" on page 1-2
- "Upgrading from an Earlier Release" on page 1-4
- "Required and Related Products" on page 1-5
- "Documentation and Demos" on page 1-6

What You Can Accomplish Using This Product

Simulink[®] Design Optimization[™] software is a Simulink[®]-based product that lets you improve designs by estimating and optimizing model parameters using numerical optimization. You can increase model accuracy by using measured data to estimate parameters and then improve system performance by automatically optimizing parameters in your Simulink model.

The software uses methods from the Optimization Toolbox[™] and Genetic Algorithm and Direct Search Toolbox[™] products to estimate and optimize model parameters. You can select multiple parameters for estimation and optimization, and the parameters can be scalars, vectors, matrices or fields of structured variables defined in the MATLAB[®] or model workspace.

You can perform the following types of estimation using the Simulink Design Optimization software:

- Transient Estimation Estimate parameters by comparing simulated model output to measured transient data.
- Initial Condition Estimation Estimate the initial conditions of states from measured data.
- Adaptive Lookup Table Estimation— Estimate the lookup table values using time-varying data from a physical system. The software provides Adaptive Lookup Table blocks to model and estimate such types of systems.

Simulink Design Optimization software also lets you optimize model parameters to meet time-domain design requirements. The software provides a Signal Constraint block where you can graphically specify bounds on signals or a reference signal to track. The software also provides CRMS and DRMS blocks to compute the continuous and discrete root mean square values of signals. Use these blocks with the Signal Constraint block to optimize the root mean square value of the signal. You can also test and optimize the design for robustness by specifying uncertainty bounds on additional model parameters.

When you have the Control System Toolbox[™] software installed, you can also refine controllers by optimizing the controller parameters in the SISO Design Tool. You can perform optimization-based control design for LTI models and Simulink models linearized using the Simulink[®] Control Design[™] software. You can graphically specify both time- and frequency-domain requirements on open- and closed-loop responses. When you use optimization-based control design, the controller parameters can be poles, zeros, and gains or Simulink block parameters.

You can work either in the Graphical User Interface (GUI) or at the command line to estimate and optimize model parameters. New users should start by using the GUI to become familiar with the product. The following operations are available only using the GUI:

- Interactive data preprocessing (see "Data Analysis and Processing")
- Model validation using residual plot (see "Comparing Residuals")
- Optimization-based controller design for time- and frequency-domain requirements (see "Optimization-Based Linear Compensator Design")

Using Simulink Design Optimization software does not require that you have a strong background in optimization theory or practice. As you gain familiarity with the product, you may find it helpful to consult the Optimization Toolbox documentation to learn more about the optimization methods. 1

Upgrading from an Earlier Release

Simulink Design Optimization software combines the functionality of Simulink[®] Parameter Estimation[™] and Simulink[®] Response Optimization[™] products. Prior to R14, Simulink Response Optimization software was called the Nonlinear Control Design Blockset software. Models that are created using Nonlinear Control Design software will not work directly with Simulink Design Optimization software. To convert your Nonlinear Control Design Blockset models to work with Simulink Design Optimization software, use the ncdupdate command, as described in Simulink Design Optimization documentation.

Required and Related Products

Simulink Design Optimization software requires MATLAB technical computing software, Simulink software, and Optimization Toolbox software.

The following table summarizes MathWorks[™] products that extend and complement the Simulink Design Optimization software. For current information about these and other MathWorks products, visit http://www.mathworks.com/products/product_listing/index.html.

Product	Description
Control System Toolbox	Design and analyze control systems.
Genetic Algorithm and Direct Search Toolbox	Solve optimization problems using genetic algorithms, simulated annealing, and direct search.
Neural Network Toolbox [™]	Design and simulate neural networks.
Parallel Computing Toolbox [™]	Perform parallel computations on multicore computers and computer clusters.
Simulink Control Design	Design and analyze control systems in Simulink.
System Identification Toolbox [™]	Create linear and nonlinear dynamic models from measured input-output data.

Documentation and Demos

In this section...

"Accessing Documentation" on page 1-6

"Accessing Demos" on page 1-6

Accessing Documentation

The Simulink Design Optimization documentation contains the following components:

- Getting Started Guide Provides information for mapping your problem to the capabilities of the Simulink Design Optimization software. Step-by-step tutorials walk you through the most common tasks for estimating parameters, optimizing parameters, and designing controllers using optimization methods.
- User's Guide Describes tasks for using the Simulink Design Optimization software.
- Reference Describes commands and blocks for design optimization.
- Release Notes Describes important changes in the current product version and compatibility considerations.

If you are new to using this product, the Getting Started Guide helps you begin using this product quickly. You can follow the steps in the tutorials to perform design optimization using the graphical user interface (GUI) or the MATLAB Command Window.

You can also search or browse the documentation for information about specific design optimization tasks.

Accessing Demos

The Simulink Design Optimization software provides demo files that show you how to estimate and optimize parameters of Simulink models, design compensators using optimization methods, and model systems using Adaptive Lookup Tables. To access demos in the Help browser, type the following command at the MATLAB prompt:

demo('Simulink', 'Simulink Design Optimization')

2

Estimating Model Parameters

- "Types of Data for Parameter Estimation" on page 2-2
- "Quick Start Estimating Model Parameters" on page 2-4
- "Parallel Computing for Parameter Estimation" on page 2-12

Types of Data for Parameter Estimation

You can estimate model parameters and initial conditions of single or multiple input and output Simulink models from *transient data*. You measure *transient data* when the system is not in steady-state to capture the system dynamics expected under normal operating conditions. For example, the response of a system to step or impulse inputs is transient data.

Simulink Design Optimization software lets you estimate model parameters from the following types of data:

- *Time-domain* data Data with one or more input variables u(t) and one or more output variables y(t), sampled as a function of time.
- Time-series data Data stored in time-series objects. For more information, see "Time Series Objects" in the MATLAB documentation.

When you import time-series data for parameter estimation, you must specify the data and time vector as *t.data* and *t.time*, respectively. For more information on how to import data into the GUI, see "Importing Data into the GUI" in the *Simulink Design Optimization User's Guide*.

Simulink Design Optimization does not directly support using complex data for parameter estimation. *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 when you want to estimate parameters of electrical systems. For example, you can use complex data to estimate the magnitude and phase of a system.

If you have complex data, use the MATLAB functions real, and imag to split the data into two data sets that contain the real and imaginary values. You can then import both data sets into the GUI, as described in "Importing Data into the GUI" in the *Simulink Design Optimization User's Guide*, and estimate the parameters by specifying both data sets simultaneously as estimation data.

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

Simulink Design Optimization software estimates model parameters by comparing the transient data with simulation data generated from the Simulink model. Using optimization techniques, the software estimates the parameters and initial conditions of states to minimize a user-selected cost function. The cost function typically calculates a least-square error between the measured and simulated data. To learn more, see "Estimating Model Parameters" in the *Simulink Design Optimization User's Guide*.

Quick Start – Estimating Model Parameters

In this quick start, you get an overview of the typical tasks for estimating model parameters using the Control and Estimation Tools Manager GUI:

- 1 Start a parameter estimation task.
- 2 Import estimation and validation data sets.
- **3** Select parameters to estimate.
- **4** Estimate the parameters from the estimation data.
- **5** Validate the estimated parameters using the validation data set.

Prerequisites for parameter estimation include:

• Simulink model that contains inport or outport blocks, or signal logging

For more information, see "Configuring a Model for Importing Data" in the *Simulink Design Optimization User's Guide*.

• Transient data in the MATLAB workspace

To estimate model parameters:

 Start a parameter estimation task by selecting Tools > Parameter Estimation in the Simulink model window.

	Control and Estimation T	ools Manager	
New estimation task	Workspace	Task settings Title: Subject: Author: Company: Description:	Optional information about the estimation task
		Model: spe_engine_throttle1Open ModelUpdate Task	
	Select the nodes below to config	x v -	
	belett the houes below to conlig	are and run estimations.	1

- **2** Import the input and output data for estimating and validating model parameters.
 - a Select the Transient Data node, and click New.
 - **b** Select the **New Data** node.
 - **c** In the **Input Data** tab, select the **Data** cell corresponding to the model channel, and click **Import**. Select the variable to import in the Data Import dialog box.

	Model of for which import of	data Selec click l	Import		t and mport to t time vector		Select varia to import	able	
🙀 Control and Estimation T	ools Maria	iger				_ 0	×		
File View Help							🙀 Data Imp	ort	X
🗃 🗋 🗳 🛃 🗐							Data 111p		
Workspace Project - spe_engine_t Project - spe_engine_t Project - spe_engine_t Project - spe_engine_t Project - spe_engine_t Project - spe_engine_t	-Assign da Block	a Output Da ata to blocks Name Danne	ita State Dat	:a Time / T	s Weight	Length	Vari n ble N		ytes Class
[iii] New Data Variables	Channe		ie ninput		1	-1-		1x1	8 doubl
Estimation								1x1	8 doubli
Validation Validation New Data node has been adde		port	Pre-proces	i	Plot Data	Clear All		init 1x1 the following columns the following rows to Import Close	selected channel(:
Select the tabbed panels to confi	igure the tr	ansient data s	iet.					-	
	-						Ir	nport variable	

- **d** Select the **Time** /**Ts** cell, and click **Import**. Select the time vector to import in the Data Import dialog box.
- In the **Output Data** tab, repeat steps c-d to import the output data and time vector.

	🐱 Control and Estimation Tool	s Manager				
	File View Help					
	😅 边 😂 🛃 📋					
	📣 Workspace	Input Data Outp	ut Data 🛛 State D	ata		
	Project - spe_engine_throt Estimation Task	Assign data to blo	cks			
	Transient Data	Block Name	Data	Time / Ts	Weight	Length
	-[iii] New Data	spe_engine_t	nrottle1/Input			
Validation _ data node	► 👯 New Data (2)	Channel - 1			1	-/-
	Variables					
		Import	Pre-proce	es, Plo	ot Data	Clear All
		<u> </u>				
	- New Data node has been added to					
	- New Data (2) node has been adde	d to Transient Data.				-
	J Select the tabbed panels to configure	e the transient data	set.			
	belocciate cabboa parteis to contriguit	e che cransieric daca	300			11.

f Repeat steps a-d to import the validation data set.

For more information, see "Importing Data into the GUI" in the *Simulink* Design Optimization User's Guide.

3 Specify parameters to estimate by selecting the **Variables** node, and clicking **Add**. Select the parameters in the Select Parameters dialog box.

Workspace Select Parameters Workspace Select and the settings Select additional parameters to estimate Select additional parameters to estimate Select Parameters Select additional parameters to estimate Select additional parameters Select additional parameters	Control and Estimation Tools Manager File View Help	Select parameters to estimate
	Workspace Estimated Parameters Estimated States Project - spe, engine_thr Estimation Task: Selected parameters Default settings Warkspace Selected parameters Name: Value: Workspace Selected parameters Name: Value: Warkspace Mininum: Name: Value Mininum: Mininum: Validation Add Defete New Data (2) node has been added to Transent Data. Referenced by:	Select additional parameters to estimate Name Size 1x1 1x1 angle_init 1x1 angle_gen 1x1 input_delay 1x1 k 1x1 specify expression (e.g., s.x, a(3), b{2}):

Specify parameters

For more information, see "Specifying Parameters to Estimate" in the *Simulink Design Optimization User's Guide*.

- **4** Estimate the parameters.
 - **a** Select the **Estimation** node, and click **New**.
 - **b** Select the **New Estimation** node.
 - c In the Data Sets tab, select the estimation data set.

	Lists imported data sets	Select data se for estimation			
🙀 Control and Estimation Tools	s Manager				
File View Help					
😅 🗋 🗳 🛃 🔳	Ļ				
Workspace	Data Sets Parameters	s States Estimation			
Project - spe_engine_throt	Data sets used for est	imation	Output data	weights	
Estimation Task	Transient estimation		Block Nan	ne Length	Weight
[ii] New Data		*	_ spe_engi	ne_throttle1/Po	sition
[;;;] New Data (2)	Data Set New Data	Selected	Channel -	1 -/-	1
Variables	New Data New Data (2)				
Estimation					
	Select All	Clear All			
- New Estimation node has been add	ed to Estimation.				A
					-
Select the tab panels to configure yo	our estimation.				

		al) Specify ter bounds					
Control and Estimation Tool	s Manager						<u>- 0 ×</u>
File View Help							
😅 🎦 😂 🛃 📳							
Workspace 	Data Sets Para Estimation paran		Estimation				
Estimation Task	Name	Value	Estimate	Initial Guess	Minimum	Maximum	Typi
liti New Data	J	0.05	V	J	-Inf	+Inf	
New Data (2)	c	40	V	с	-Inf	+Inf	
Variables	input_delay	0.02	V	input_delay	-Inf	+Inf	inpu
Estimation	k	1		k	-Inf	+Inf	
							Þ
teres t	Use Value as	Initial Guess	Reset to	Default Settings	Save a	is Default Sett	ings
- New Estimation node has been add	ed to Estimation.						
Select the tab panels to configure yo	ur estimation.						11.

d In the **Parameters** tab, select the parameters to estimate.

e In the Estimation tab, begin estimation by clicking Start.

File View Help File View Help Image: State State Estimation Image: State State State Image: State State
Select the tab panels to configure your estimation.

f In the **Parameters** tab, examine the estimated parameter values. The Simulink model also gets updated with the estimated parameter values.

Control and Estimation File View Help The View Help Very Space Project - spe_engin Estimation Task Control Transient C Control T							
Workspace Project - spe_engir Estimation Task Transient C Variables State State Variables Estimation Estimation Variables	n Tools Manage	r					_ [] >
Project - spe_engir Stimation Task Transient D Variables Stimation Stima]						
 Image: Transient C Image: Transient	Data Sets Para		Estimation				
→ 🛱 Variables → 🛱 Estimation → 🛱 New Es → C Views → C Nev	Name	Value	Estimate	Initial Guess	Minimum	Maximum	Typical Valu
Estimation	J	0.22364	V	J	0	+Inf	J
New Es ⊡-⊡⊋ Views Nev	c	9.0575		с	0	+Inf	с
E Views	input_delay	0.0089359	V	input_delay	0	0.1	input_dela
_ 🗠 Nev	k	19.896	V	k	0	+Inf	k
	Use V	alue as Initial G	uess Re	set to Default Se	ttings	Save as Default	► Settings
Estimation completed. Select the tab panels to confid	gura vour actimati						

For more information, see "Performing Estimation" in the *Simulink Design Optimization User's Guide*.

- **5** Validate the estimated parameters.
 - **a** Select the Validation node, and click New.
 - **b** Select the **New Validation** node.

		ect validation type	Select va data set	lidation	
🙀 Control and Estimation To	ols Manager				
File View Help					
🗃 🛅 🗳 🛃 🔳					
Workspace	Validation Setup				
Project - spe_engine_thr	Select plot types	•			
Estimation Task	Plot Number	Plot Type		Plot Title	
[;;] New Data	Plot 1 Residua Plot 2 (none)	als			<u> </u>
الله المعنى ا المعنى المعنى	Plot 2 (none) Plot 3 (none)				
	Plot 4 (none)				
📲 🔤 New Estimati	Options				
Views	Validation data set:	New Data (2)		_	
			. [
🔤 🔛 New Validatio	Estimation New Estimation	Plot	1		
	New Escinacion				
			Show Plo	ts	
	aca co validación.				
Configure validation plots.					
		Select to			111
		display validatio	· Ope	ens dation plots	

c Configure the validation plots and the validation data set.

For more information, see "Performing Validation" in the *Simulink* Design Optimization User's Guide.

See Also: Chapter 6, "Tutorial — Estimating Parameters from Measured Data Using the GUI"

Parallel Computing for Parameter Estimation

When you have the Parallel Computing Toolbox software, you can use parallel computing to speed up parameter estimation. When you use parallel computing, the software distributes the independent simulations on multiple MATLAB sessions. Thus, the simulations run in parallel which reduces the estimation time.

Using parallel computing may reduce the estimation time in the following cases:

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

For more information, see "Speeding Up Parameter Estimation Using Parallel Computing" in the *Simulink Design Optimization User's Guide*.

Optimizing Model Parameters

- "Types of Time-Domain Design Requirements for Optimizing Parameters" on page 3-2
- "Quick Start Optimizing Model Parameters" on page 3-3
- "Parallel Computing for Parameter Optimization" on page 3-11

Types of Time-Domain Design Requirements for Optimizing Parameters

You can optimize parameters of Simulink models to meet the following types of time-domain design requirements:

- Step-response characteristics such as overshoot, and rise time.
- Lower and upper bounds on signals
- Reference signal

Simulink Design Optimization software optimizes the model parameters by formulating the time-domain requirements into a constrained optimization problem. It then solves the problem using optimization methods. During the optimization, the software performs the following operations:

- Simulates the Simulink model,
- Compares the simulation data with the constraint objectives and any specified reference signal
- Uses gradient methods to modify selected model parameters to meet the objectives

To learn more, see "Optimizing Model Parameters" in the *Simulink Design Optimization User's Guide*.

Quick Start – Optimizing Model Parameters

In this quick start, you get an overview of the typical tasks for optimizing model parameters to meet time-domain requirements:

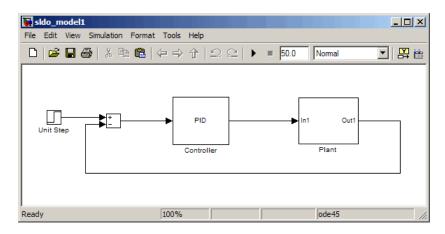
- **1** Specify an input signal in the Simulink system.
- **2** Specify the design requirements.
- **3** Specify parameters to optimize.
- **4** Optimize the parameters.
- **5** Evaluate the optimization results.

Prerequisites for optimizing model parameters include:

- Simulink model
- Time-domain design requirements

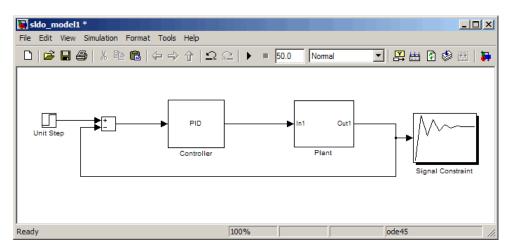
To optimize model parameters:

1 In the Simulink model, specify an input signal to the system. For example, add a Step block.



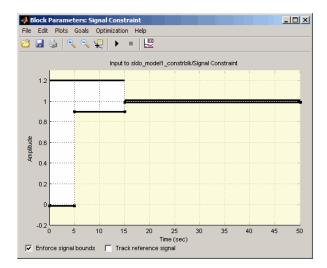
2 Specify the time-domain design requirements:

- a In the Simulink Library Browser, select Simulink Design Optimization.
- **b** Drag and drop the Signal Constraint block into the model.
- **c** Connect the Signal Constraint block to the signal that should meet the design requirements.



d Double-click the Signal Constraint block.

Design requirements appear as line segments in the Block Parameters: Signal Constraint block window. By default, the design requirements are step-response characteristics.



e Double-click the lower yellow region on the plot. Specify the design requirements in the Edit Design Requirement dialog box.

	besign require Design require	,	er time respon:	se bound fron	n 0 to 50 sec		∎ ▼	۷ ۲-		Select design requirement type
r	Segments:							1	L	
	Sta	art	En	ld				Ĩ	l	
	Time	Amplitude	Time	Amplitude	Slope	Weight				Enter design
	0	-0.0100	5	-0.0100	0	1			l	requirement values
	5	0.9000	15	0.9000	0	1			l	
	15	0.9900	50	0.9900	0	1	▼		l	
L					Insert	Delete		1		
Close Help										

For more information, see "Specifying Design Requirements" in the *Simulink Design Optimization User's Guide*.

3 In the Block Parameters window, select **Optimization > Tuned Parameters**, and click **Add**. Select the parameters to optimize in the Add Parameters dialog box.

Lists parameters selected for optimi		Select parameters	
Add Delete	Optimization Settings Name: Kp Value: 1 Initial guess: Kp Minimum: Maximum: nf Typical value: Kp Tuned Referenced by: Controller OK	Add Parameters Select workspace va Kd Kp W0 Zeta Specify expression (OK	riables: e.g., s.x or a(3)):
to optimize			

For more information, see "Specifying Parameters to Optimize" in the Simulink Design Optimization User's Guide.

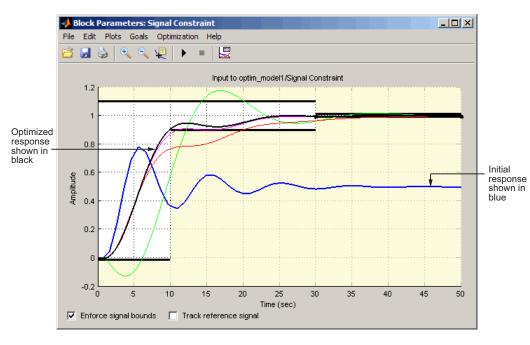
4 In the Block Parameters window, start the optimization by selecting **Optimization > Start**.

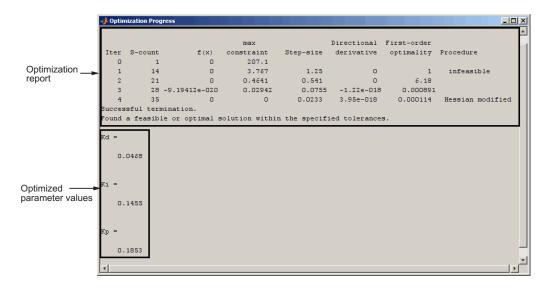
The Optimization Progress window opens where you see the optimization progress.

l Optin	nization Progress					>
Iter	S-count	f (v)	max constraint	Stan_giza	Directional	
0	1	0	207.1	500p-5120	uerivative	opermaney

For more information, see "Running the Optimization" in the *Simulink* Design Optimization User's Guide.

- 5 Evaluate the optimization results after the optimization completes.
 - **a** In the Block Parameters window, compare the response of the system against the design requirements.





b In the Optimization Progress window, view the optimized parameter values.

See Also: Chapter 7, "Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the GUI".

Parallel Computing for Parameter Optimization

When you have the Parallel Computing Toolbox software, you can use parallel computing to speed up optimizing model parameters to meet time-domain design requirements. When you use parallel computing, the software distributes the independent simulations on multiple MATLAB sessions. Thus, the simulations run in parallel which reduces the optimization time.

Using parallel computing may reduce the optimization time in the following cases:

- The model contains a large number of parameters to optimize, and the Gradient descent method is selected for optimization.
- ThePattern 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.

For more information, see "Speeding Up Response Optimization Using Parallel Computing" in the *Simulink Design Optimization User's Guide*.

Optimization-Based Linear Control Design

- "When to Use Optimization-Based Linear Control Design" on page 4-2
- "Types of Time- and Frequency-Domain Design Requirements" on page 4-3
- "Quick Start Optimization-Based Linear Control Design" on page 4-4

When to Use Optimization-Based Linear Control Design

When you have Control System Toolbox software installed, you can design and optimize control systems for LTI models by optimizing controller parameters in the SISO Design Tool. To use optimization methods for linear control design, also known as *optimization-based tuning*, you must already have an initial controller. You can then use optimization-based tuning to refine the controller design to meet additional design requirements. For more information on designing controllers, see the Control System Toolbox documentation.

Note Optimization-based tuning only changes the value of the controller parameters and not the controller structure itself.

Optimization-based tuning provides flexibility in terms of specifying additional design requirements for the controller. When you have a large number of design requirements, you can first design an initial controller by selecting a subset of requirements and subsequently select additional requirements to refine the design.

Optimization-based tuning also provides flexibility in terms of selecting a subset of controller parameters to optimize, and specifying bounds on the controller parameters.

To design linear controllers for Simulink models using optimization-based tuning, you must first linearize the model using the Simulink Control Design software. For more information on linearizing Simulink models, see the Simulink Control Design documentation.

Types of Time- and Frequency-Domain Design Requirements

When you design linear controllers for LTI or Simulink models using the Simulink Design Optimization software, you can specify both time- and frequency-domain requirements on the system response. You can specify design requirements on the following plots:

- Root Locus plot
- Open-Loop and Prefilter Bode plots
- Open-Loop Nichols plot
- Step/Impulse Response plots

For more information, see "Supported Time- and Frequency-Domain Requirements" in the *Simulink Design Optimization User's Guide*.

Simulink Design Optimization software uses the frequency-domain requirements to compute the frequency response of the system. It then uses optimization methods to reduce the distance between the current response and the requirements by modifying the controller parameters. The software does not change the controller structure when optimizing the controller parameters. To learn more, see "Optimization-Based Linear Compensator Design" in the *Simulink Design Optimization User's Guide*.

Quick Start – Optimization-Based Linear Control Design

In this quick start, you get an overview of the typical tasks for optimization-based linear control design using the SISO Design Tool:

- 1 Open a SISO Design Tool session.
- 2 Configure a project for optimization-based control design.
- **3** Specify the controller parameters to design.
- 4 Specify the design requirements.
- **5** Design the controller.
- **6** Evaluate the controller design.

Note The same workflow applies to optimization-based control design for LTI models created at the command line using Control System Toolbox software. To learn how to create LTI models, see "Linear (LTI) Models" in the Control System Toolbox documentation.

Prerequisites for optimization-based linear control design include:

• Simulink Compensator Design Task that contains a linearized version of the Simulink model and, optionally, any response plots you configure.

For more information on how to linearize a Simulink model for control design, see "Designing Compensators" in the Simulink Control Design documentation.

• Time- and frequency-domain design requirements

To design a controller using optimization methods:

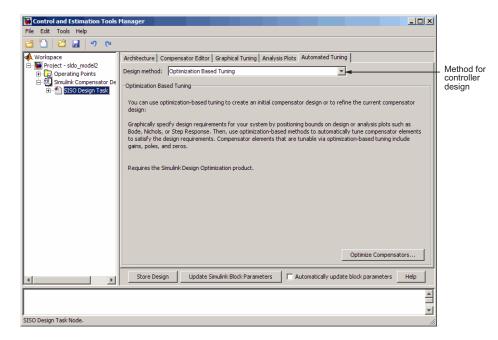
1 Open a SISO Design Tool session by typing the following command at the MATLAB prompt:

sisotool('projectname.mat')
-----------	--------------------

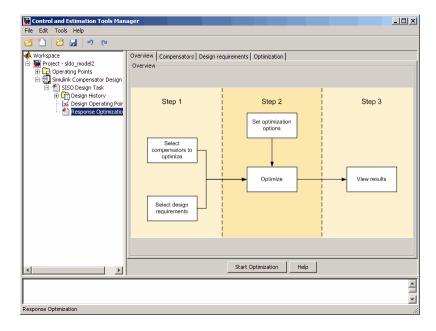
	Control and Estimation Tools	Manager		
SISO Design Task for SISO Design — Tool	Workspace Project - side_model2 Project - si	Project Settings Title: Subject: Created by: Date modified: Simulnik model: Description:	Project - sido_model2	A A
	Project node.			

The command also opens a SISO Design for SISO Design Task window by default and any response plots you configured when you linearized the Simulink model using Simulink Control Design software.

2 Configure a project for optimization-based control design by clicking Optimize Compensators in the Automated Tuning tab of the SISO Design Task.



This action creates a new **Response Optimization** node in the Control and Estimation Tools Manager.



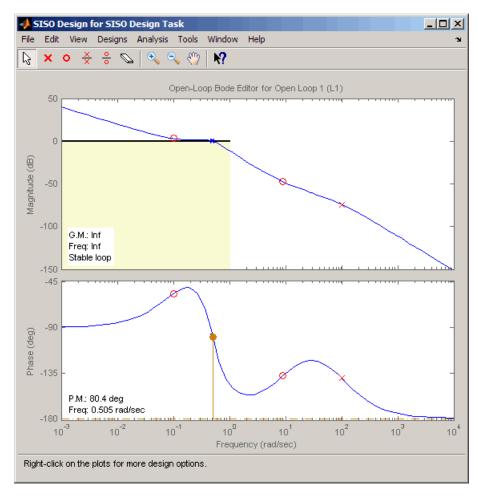
						parame	eter bounds	
Control and Estimation	n To	ools Manager						_ [
ile Edit Tools Help								
ž 🗋 🖬 🖬 🔊	(2						
Workspace		Select compens	npensators Design requirements	Optimization				
Simulink Compens		Coptimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical valu
🖃 🎦 SISO Design			sldo_model2/Controller					
🕀 🛄 Design Hi			Gain	0.1	0.1	-Inf	Inf	0.1
🖂 🚘 Design O			Real Zero	-8.999	-8.999	-Inf	Inf	-8.999
🚹 Response	Ш		Real Zero	-0.10102	-0.10102	-Inf	Inf	-0.10102
	ш		Real Pole	-100	-100	-Inf	Inf	-100
	F	light click on a cor	mpensator name to change its represent.	tion.			Use Value as	: Initial Guess

3 Specify the controller parameters to design in the **Compensators** tab.

- 4 Specify the design requirements.
 - **a** In the **Design requirements** tab, click **Add new design requirement**. Specify the design requirements, for example Bode magnitude lower limit, in the New Design Requirement dialog box.

Control and Estimation Tools Manage	,				X				
File Edit Tools Help		📣 New Desig	n Requirem	ent			_ 🗆 🗙		
Image: State State Image: State	Overview Compensators Design requirements Optimization Select design requirements to satury Image: Compensators Design requirements Image: Compensators Response pixt	Design requirement type:				Bode magnitude lower limit			Requirement type Response type
Cesign Operating Point		Segments:							
Response Optimization		St	art	Er	nd				
		Frequency	Magnitude	Frequency	Magnitude	Slope	Weight .	┝ -	_Requirement
		1	0	10	0	0	1		values
]	Insert	Delete		
					ОК	Cancel	Help		
	Cick the 'Add new design requirement' button or right clok on a plot to add a new de		Add new design	requirement 1	+	- Add requir	ement		
x	Start Optimization	Неір							
Response Optimization					1				
Kesponse Optimization					lle				

In the SISO Design window, the yellow region with the black line segment represents the design requirement on the response plot.



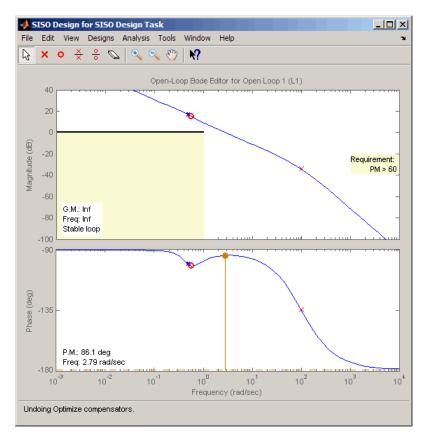
Se to op	Lists all specified design requirements	
Gontrol and Estimation Tools	1anager	
File Edit Tools Help		
Workspace	Overview Compensators Design requirements Optimization	
🕀 🔁 Operating Points	Optimize Response plot	Design requirement
 Simulink Compensator De SISO Design Task 	Open Loop at outport 1 of Controller	
Design History	Open-Loop Bode Editor for Open Loop 1 (L1)	Lower gain limit from 0.001 to 1 rad/sec
L 🗠 Response Optimi	Click the 'Add new design requirement' button or right click on a plot to add a new desig	
I F	Start Optimization Help	
Undo		- -
Undo		1.

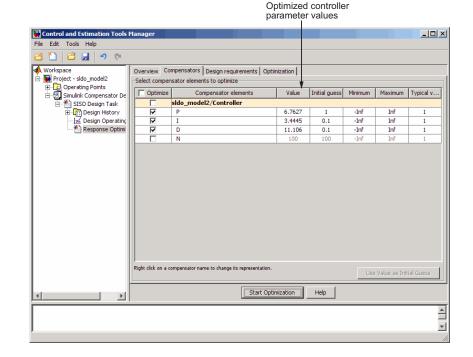
The **Design Requirements** tab also lists the design requirement.

b Repeat step a to specify additional time- and frequency-domain requirements.

- 🙀 Control and Estimation Tools Manager <u>- 🗆 ×</u> File Edit Tools Help 🗃 🛅 🖆 🛃 🔊 🤊 📣 Workspace Overview Compensators Design requirements Optimization Wonspace
 Project - sldo_model2
 Project - sldo_model2
 Project - gloperating Points
 Project - gloperating Points Optimization progress Iteration Eval-Count Cost function Constraint... Step size Procedure Optimization options... 57.12 1.777 0 7 0 8.36 Display options. Design Histor 1 14 0 21 0 1.697 12.1 2 Hessian m... 3 28 0 0 21.7 Optimization progress Constructing optimization problem... Optimization started 10-Nov-2008 14:01:11 ٠ Optimization finished 10-Nov-2008 14:01:21 Successful termination. Found a feasible or optimal solution within the specified Optimization tolerances. status Start Optimization Help ► • 1 ▲ ▼ Redo
- **5** Start the optimization to design the controller by clicking **Start Optimization** in the **Optimization** tab.

- 6 Evaluate the controller design.
 - **a** Examine the system's response in the response plot, for example the Bode plot, to see if it meets the requirements. The system's response must lie in the white region in order to meet the design requirement.





b Examine the controller parameter values in the **Compensator** tab.

7 Write the controller parameter values into the Simulink model. To do so, click Update Simulink Block Parameters in the SISO Design Task node.

See Also: Chapter 9, "Tutorial — Designing a PID Controller Using Optimization-Based Tuning".

5

Tutorial — Preparing Data for Parameter Estimation Using the GUI

- "About This Tutorial" on page 5-2
- "Configuring a Project for Parameter Estimation" on page 5-4
- "Importing Data into the GUI" on page 5-6
- "Analyzing Data" on page 5-14
- "Selecting Data for Estimation" on page 5-16
- "Removing Outliers" on page 5-25
- "Filtering Data" on page 5-29
- "Interpolating Missing Data" on page 5-34
- "Saving the Project" on page 5-37

About This Tutorial

In this section...

"Objectives" on page 5-2

"About the Sample Data" on page 5-2

Objectives

In this tutorial, you learn how to import, analyze, and prepare measured input and output (I/O) data for estimating parameters of a Simulink model.

Note Simulink Design Optimization software estimates parameters from real, time-domain data only.

You learn to perform the following tasks using the GUI:

- Import data from the MATLAB workspace.
- Analyze data quality using a time plot.
- Select a subset of data for estimation.
- Remove outliers.
- Filter high-frequency noise.
- Compute missing data using interpolation.

About the Sample Data

In this tutorial, you use spe_engine_throttle1.mat, which contains I/O data measured from an engine throttle system. The MAT-file includes the following variables:

- input1 Input data samples
- position1 Output data samples
- time1 Time vector

Note The number of input and output data samples must be equal to the length of the corresponding time vector.

The engine throttle system controls the flow of air and fuel mixture to the engine cylinders. The throttle body contains a butterfly valve which opens when a driver presses the accelerator pedal. Opening this valve increases the amount of fuel mixture entering the cylinders, which increases the engine speed. A DC motor controls the opening angle of the butterfly valve in the throttle system. The input to the throttle system is the motor current (in amperes), and the output is the angular position of the butterfly valve (in degrees).

spe_engine_throttle1.mdl contains the Simulink model of the engine throttle system. For more information on building models, see "Creating a Simulink Model" in the Simulink documentation.

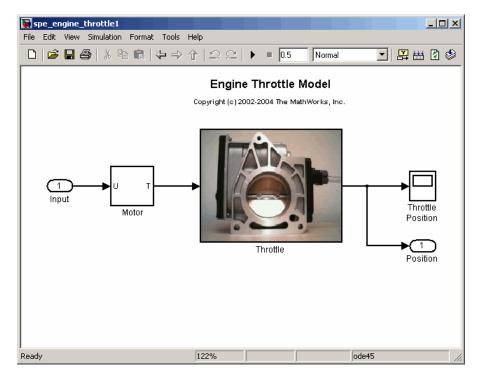
Configuring a Project for Parameter Estimation

To perform parameter estimation, you must first configure a Control and Estimation Tools Manager project.

1 Open the engine throttle system model by typing the following at the MATLAB prompt:

spe_engine_throttle1

This command opens the Simulink model, and loads the data into the MATLAB workspace.



Simulink[®] Model of Engine Throttle System

2 In the Simulink model window, select Tools > Parameter Estimation.

This action opens a new project named **Project - spe_engine_throttle1** in the Control and Estimation Tools Manager GUI. This project contains the **Estimation Task**, as shown in the next figure.

🙀 Control and Estimation Tools Man	ager		
File View Help			
🛅 🗅 🗳 🖬 📋			
📣 Workspace	-Task settings-		
Project - spe_engine_throttle1 Estimation Task	Title:		
😥 🗍 Transient Data	Subject:		
Variables	Author:		
Estimation	Company:		
	Description:		
			
	Model:	spe_engine_throttle1	Open Model Update Task
			<u> </u>
			•
J Select the nodes below to configure and ru	n estimations.		

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

Importing Data into the GUI

In this section ...

"Importing Input Data and Time Vector" on page 5-6

"Importing Output Data and Time Vector" on page 5-11

Importing Input Data and Time Vector

In this portion of the tutorial, you import measured I/O data into the Control and Estimation Tools Manager GUI. Importing data assigns the data to the corresponding model input and output signals.

The model input and output signals are designated with the Inport Input and Outport Position blocks respectively, as shown in the figure Simulink[®] Model of Engine Throttle System on page 5-4. These blocks let you import I/O data into the GUI. To learn more about the blocks, see the Inport and Outport block reference pages in the Simulink documentation.

You must have already configured the parameter estimation project, as described in "Configuring a Project for Parameter Estimation" on page 5-4.

To import input data and time vector into the Control and Estimation Tools Manager GUI:

1 In the Control and Estimation Tools Manger GUI, select **Transient Data** under the **Estimation Task** node, and click **New**.

This action creates a **New Data** node under the **Transient Data** node.

🙀 Control and Estimation Tools I	Manager		
File View Help			
🗚 🗅 😂 🖬 📋			
📣 Workspace	Transient data sets		
🖻 🙀 Project - spe_engine_throttle	Name	Properties	
🖃 🗐 Estimation Task	New Data	· · · ·	
ि - ि Transient Data			
Variables			
🕀 📴 Estimation			
🗄 💭 Validation			
			-
	,		
	Description:		
			-
			_
	New	Delete Edit	
- New Data node has bee	en added to Transient Data.		
			-
Transient data sets are stored in this f	older. Press 'New' to create a new data set		//

2 Select the New Data node.

🙀 Control and Estimation Tools	Manager				_ 🗆 ×	
File View Help						
🗚 🗅 😂 🖬 🔳						
📣 Workspace	Input Data Output Data	a State Data				
Project - spe_engine_throttle	Assign data to blocks					
Estimation Task	Block Name	Data	Time / Ts	Weight	Length	
Itil New Data	spe_engine_throttle	1/Input				
Variables	Channel - 1			1	-/-	
윤-대왕 Estimation 표-대왕 Validation						
			1	1		
		Import	Pre-process	Plot Data	Clear All	
	L					
- New Data node has bee	n added to Trans	ient Data.			<u> </u>	
					•	
Select the tabbed panels to configure	the transient data set.					

3 In the **Input Data** tab, select the **Data** cell for **Channel - 1**, and click **Import**.

This action opens the Data Import dialog box.

🙀 Data Import				×				
Import from: Workspace 💌								
Variable Name	Size	Bytes	Class					
🌐 in1	201x1	1608	double					
🖽 input1	701x1	5608	double					
🗮 input2	500x1	4000	double					
🗰 input3	500x1	4000	double	-				
Assign the following columns to selected channel(s): I Assign the following rows to selected channel(s): [1:701]								
Im	port	Close	Help					

4 In the Data Import dialog box, select input1, and click Import.

This action assigns the input data input1 to the model input signal **spe_engine_throttle1/Input**.

🙀 Control and Estimation Tools	Manager					×	
File View Help							
😁 🗅 📂 🖬 🔳							
📣 Workspace	Input Data Output Data	a State Data					
E-B Project - spe_engine_throttl	Assign data to blocks						
Escination rask	Block Name	Data	Time / Ts	Weight	Length	Ī	
[iii] New Data	spe_engine_throttle						
Variables 	Channel - 1	input1(:,1)		1	701/-	-	
Estimation							
		Import	Pre-process	Plot Data	Clear All		
		-				-	
- New Data node has be	en added to Trans	sient Data.					
						-	
J Select the tabbed panels to configure	the transient data set.					<u> </u>	

- **5** Select the **Time / Ts** cell for **Channel 1**.
- 6 In the Data Import dialog box, select time1, and click Import.

This action assigns the time vector to the model input signal **spe_engine_throttle1/Input**.

🙀 Control and Estimation Tools	lanager			<u>_</u> _×
File View Help				
🛎 🗅 🚅 🖬 🔟				
📣 Workspace	Input Data Output Data State Data	1		
Project - spe_engine_throttle Estimation Task	Assign data to blocks			
Escination Task	Block Name Dat	a Time / Ts	Weight	Length
[iii] New Data	spe_engine_throttle1/Input			
Variables	Channel - 1 input1	(:,1) time1(:,1)	1	701/701
⊕ - R Estimation				
		Import Pre-process	Plot Data	Clear All
	n added to Transient Dat	a.		A V
Select the tabbed panels to configure	ne transient data set.			11.

7 In the Data Import dialog box, click Close.

Importing Output Data and Time Vector

To import output data and time vector into the Control and Estimation Tools Manager GUI:

1 In the Control and Estimation Tools Manger GUI, select the **Output Data** tab of the **New Data** node.

🙀 Control and Estimation Tools 1	Manager				_ D ×	
File View Help						
🛎 🗅 🚘 🖬 🗐						
📣 Workspace	Input Data Output Data	State Data				
E	Assign data to blocks					
E- Transient Data	Block Name	Data	Time / Ts	Weight	Length	
Ital New Data	spe_engine_throttle	1/Position				
Variables	Channel - 1			1		
Estimation						
		Import	Pre-process	Plot Data	Clear All	
- New Data node has bee	n added to Trans	ient Data.				
					•	
Select the tabbed panels to configure I	the transient data set.					

2 Select the **Data** cell for **Channel - 1**, and click **Import**.

This action opens the Data Import dialog box.

🙀 Data Import				x	
Import from: Workspace 💌					
Variable Name	Size	Bytes	Class		
🇮 kexp	1x1	8	double	-	
🗰 position1	701x1	5608	double		
🗰 position2	500x1	4000	double		
🗰 position3	500x1	4000	double	•	
Assign the following columns to selected channel(s): Assign the following rows to selected channel(s): [1:701]					
Import Close Help					

3 In the Data Import dialog box, select position1, and click Import.

This action assigns the output data position1 to the model output signal **spe_engine_throttle1/Position**.

Gontrol and Estimation Tools	Manager		
File View Help			
🛎 🗅 🚅 🖬 🔟			
📣 Workspace	Input Data Output Data State Data		
E-B Project - spe_engine_throttle	Assign data to blocks		
Estimation Task	Block Name Data Time / Ts Weight	Length	
[iii] New Data	spe_engine_throttle1/Position		
Variables	Channel - 1 position1(:,1) 1	701/-	
H - G Estimation			
	Import Pre-process Plot Data	Clear All	
	en added to Transient Data.	×	
Select the tabbed panels to configure the transient data set.			

- 4 Select the Time / Ts cell for Channel 1.
- 5 In the Data Import dialog box, select time1, and click Import.

This action assigns the time vector to the model output signal **spe_engine_throttle1/Position**.

🙀 Control and Estimation Tools	Manager					×
File View Help						
🛎 🗅 🚘 🖬 🔟						
📣 Workspace	Input Data Output Data	State Data				
Project - spe_engine_throttle	Assign data to blocks					
Esumation Task	Block Name	Data	Time / Ts	Weight	Length	1
[iii] New Data	spe_engine_throttle	e1/Position				
Variables	Channel - 1	position1(:,1)	time1(:,1)	1	701/701	
Estimation		Import	Pre-process	Plot Data	Clear All	
- New Data node has bee	en added to Trans	ient Data.				
Load projects						1.

6 In Data Import dialog box, click Close.

You have now imported the I/O data into the Control and Estimation Tools Manager GUI, and assigned the data to the corresponding model signals.

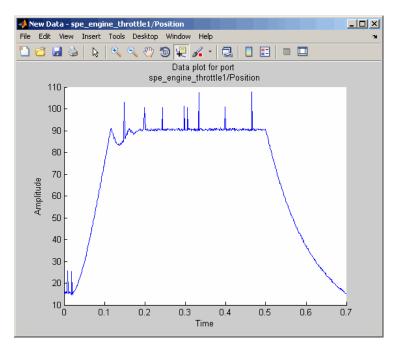
Analyzing Data

In this portion of the tutorial, you analyze the output data quality by viewing the data characteristics on a time plot. Based on the analysis, you decide whether to preprocess the data before estimating parameters. For example, if the data contains noise, you might want to filter the noise from the system dynamics before estimating parameters.

You must have already imported the data into the Control and Estimation Tools Manager GUI, as described in "Importing Data into the GUI" on page 5-6. If you have not imported the data, click here.

To plot the output data on a time plot, select the position1(:,1) cell in the **Output Data** tab, and click **Plot Data**.

🙀 Control and Estimation Tools I	Manager					- 🗆 🗵
File View Help						
🗚 🗅 😂 🖬 🗐						
📣 Workspace	Input Data Output Data	State Data				
Project - spe_engine_throttle	Assign data to blocks					
Estimation Task	Block Name	Data	Time / Ts	Weight	Length	
I	spe_engine_throttle					
Variables	Channel - 1	position1(:,1)	time1(:,1)	1	701/701	
Estimation						
۲		Import	Pre-process	Plot Data	Clear A	
- New Data node has bee	en added to Trans	ient Data.				*
Load projects						



This action plots the measured output data position1(:,1), as shown in the next figure.

The time plot shows the output data in response to a step input, as described in "About the Sample Data" on page 5-2. The plot shows a rapid decrease in the response after t = 0.5 s because the system is shut down. To focus parameter estimation on the time period when the system is active, you select the data samples between t = 0 s and t = 0.5 s, as described in "Selecting Data for Estimation" on page 5-16 section of this tutorial.

The spikes in the data indicate *outliers*, defined as data values that deviate from the mean by more than three standard deviations. They may be caused by measurement errors or sensor problems. The response also contains noise. Before estimating model parameters from this data, you remove the outliers and filter the noise, as described in "Removing Outliers" on page 5-25, and "Filtering Data" on page 5-29 sections of this tutorial.

Tip You can also plot the input data on a time plot by selecting the input1(:,1) cell in the **Input Data** tab, and clicking **Plot Data**.

Selecting Data for Estimation

In this section...

"Selecting Output Data" on page 5-16

"Selecting Input Data" on page 5-22

Selecting Output Data

In this portion of the tutorial, you select a subset of I/O data for estimation. As described in "Analyzing Data" on page 5-14, the system is shut down at t = 0.5 s. To focus the estimation on the time period before t = 0.5 s, you exclude the data samples beyond t = 0.5 s. This operation selects the data between t = 0 s and t = 0.5 s for estimation.

You must have already imported the data into the Control and Estimation Tools Manager GUI, as described in "Importing Data into the GUI" on page 5-6. If you have not imported the data, click here.

To select the portion of data between t = 0 s and t = 0.5 s:

In the Control and Estimation Tools Manager window, select the New Data node under the Transient Data node.

2 Select the position1(:,1) cell in the Output Data tab, and click **Pre-process**.

This action opens the Data Preprocessing Tool GUI.

Data Preprocessing Tool Modify data fromthrottle1/f		C existing dataset		
Data Editing				
Raw data Modified data		Exclusion Rules Detrend/Filtering		
Time (seconds)	position1(:,1)			
0.0	15.116	F Bounds		
1.0e-3	15.964			
2.0e-3	15.241	Exclude X: <= 🔽 Exclude X: >= 🔽		
3.0e-3	15.993			
4.0e-3	15.376	Exclude Y: <= 🔻 Exclude Y: >= 👻		
5.0e-3	15.997			
6.0e-3	15.380			
7.0e-3	15.100	Outliers		
8.0e-3	15.649			
9.0e-3	25.592			
1.0e-2	15.944	Window length: 10		
1.100e-2	15.238			
1.200e-2	15.817	Confidence limits (%): 95		
1.300e-2	15.385			
		MATLAB expression abs(x)>1		
Grey back Excluded by Red text Manually e:	Exclude Graphically	Flatines Window: 5		
Missing Data Handling				
Remove rows where all v data is excluded or missing Interpolate missing values using interpolation method zoh v				
		Add Close Help		

The **Data Editing** area of the Data Preprocessing Tool GUI shows the output data and time vector in the **position1(:,1)** and **Time (seconds)** columns, respectively. The Data Preprocessing Tool GUI lets you perform the following types of preprocessing operations:

- Excluding data
- Detrending and filtering data
- Handling missing data
- **3** To exclude the output data beyond t = 0.5 s:
 - a In the Exclusion Rules tab, select the Bounds check box.

b In the Exclude X >= field, where X corresponds to the time vector, select > from the drop-down list. Enter 0.5 in the adjacent field to specify the upper limit of the data to select for estimation.

The Data Preprocessing Tool GUI resembles the next figure.

🙀 Data Preprocessing Tool	for Dataset: New Data	X		
Modify data fromthrottle1/F		C existing datasetw Data V O new dataset Dataset1		
Modilly data from [throttle1)F	vosicion 💽 👘 while results to:			
Data Editing				
Raw data Modified data		Exclusion Rules Detrend/Filtering		
Time (seconds)	position1(:,1)	✓ Bounds		
0.0	15.116	Je bounds		
1.0e-3	15.964			
2.0e-3	15.241	Exclude X: <= Exclude X: > 0.5		
3.0e-3	15.993			
4.0e-3	15.376	Exclude Y: <= 💌 Exclude Y: >= 💌		
5.0e-3	15.997			
6.0e-3	15.380			
7.0e-3	15.100	Coutliers		
8.0e-3	15.649			
9.0e-3	25.592	Window length: 10		
1.0e-2	15.944	Windowicigen. 10		
1.100e-2	15.238			
1.200e-2	15.817	Confidence limits (%): 95		
1.300e-2	15.385			
/	·	MATLAB expression abs(x)>1		
Grey back Excluded by	Exclude Graphically			
Red text Manually ex		Flatlines Window: 5		
Missing Data Handling				
Remove rows where all 🕐 data is excluded or missing 🦳 Interpolate missing values using interpolation method 🛛 zoh				
		Add Close Help		

c Click Add.

This action adds a new **Dataset1** node under the **Transient Data** node in the Control and Estimation Tools Manager GUI. The **Dataset1** node contains the modified output data position1(:,1)* in the **Output Data** tab.

당 Control and Estimation Tools	anager	_ 🗆 ×
File View Help		
📸 🗅 🛸 🖬 🗐		
📣 Workspace	Input Data Output Data State Data	
🖻 🕞 Project - spe_engine_throttk	Assign data to blocks	
Estimation Task	Block Name Data Time / Ts	Weight Length
[iii] New Data	spe_engine_throttle1/Position	
[;;;] Dataset1	Channel - 1 position1(:,1)* time1(:,1)*	1 701/701
Variables		
🗄 🔂 Validation		
	Import Pre-proces	ss Plot Data Clear All
Select the tabbed panels to configure I	e transient data set.	

This operation also replaces the output data samples beyond t = 0.5 s in position1(:,1)* with NaNs. You can view the NaNs by selecting the **Modified data** tab in the Data Preprocessing Tool GUI, as shown in the next figure.

Data Preprocessing Modify data fromthrot	Tool for Dataset: New Data	C existing dataset Dataset1 🔽 🕝 new dataset Dataset2
Data Editing		
Raw data Modified dat	ia	Exclusion Rules Detrend/Filtering
Time (seconds)	position1(:,1)*	
0.497	90.331	✓ Bounds
0.498	90.323	
0.499	90.816	Exclude X: <=
0.500	90.890	
0.501	NaN	Exclude Y: <=
0.502	NaN	
0.503	NaN	
0.504	NaN	Outliers
0.505	NaN	
0.506	NaN	
0.507	NaN	Window length: 10
0.508	NaN	
0.509	NaN	Confidence limits (%): 95
0.510	NaN	
		MATLAB expression abs(x)>1
	ed by a rule Exclude Graphically	Flatines Window: 5
Missing Data Handling		
Remove rows where	all 💌 data is excluded or missing	Thterpolate missing values using interpolation method zoh
		Add Close Help

- 4 To remove the NaNs:
 - a In the Control and Estimation Tools Manager GUI, select the **Output Data** tab of the **Dataset1** node.
 - **b** Select the position1(:,1)* cell, and click **Pre-process**.

This action updates the Data Preprocessing Tool GUI with the selected data position1(:,1)*.

c In the Missing Data Handling area, select the
 Remove rows where all a data is excluded or missing check box.

dify data fromthrottle	1/Position 💌 Write results I	s to: C existing dataset Dataset I 🖉 C new dataset Dataset2
Raw data Modified data]	Exclusion Rules Detrend/Filtering
Time (seconds)	position1(:,1)**	
).487	90.604	Bounds
).488	90.533	
).489	90.105	Exclude X: <= Exclude X: >= Exclude X: >= Exclude X: = Ex
).490	90.592	
).491	90.72	Exclude Y: <=
).492	90.648	
).493	90.772	
).494	90.432	Coutliers
).495	90.562	
).496	90.206	Window length: 10
.497	90.331	Window length. 110
.498	90.323	
.499	90.816	Confidence limits (%): 95
.500	90.890	•
		MATLAB expression abs(x)>1
Grey back Excluded by a rule Exclude Graphically Red text Manually excluded Flatlines Window:		
ssing Data Handling		

Tip You can view the results of this operation in the Modified data tab.

Add

Close

Help

d In the **Write results to** area, select the **existing dataset** Dataset

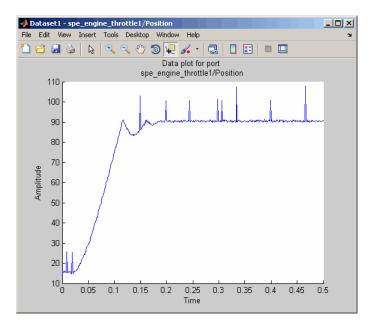
Modify data fromthrottle1/Position 💌	Write results to:	existing dataset	Dataset1 💌	🔿 new dataset	Dataset2

e Click Add.

The Update table data dialog box appears. Click **Yes** to overwrite the position1(:,1)* data set with the modified data.

5 To plot the data, select the position1(:,1)* cell in the **Output Data** tab of the **Dataset1** node, and click **Plot Data**.

The selected output data from t = 0 s to t = 0.5 s is shown in the next figure.



Selecting Input Data

After you select the output data between t = 0 s and t = 0.5 s, as described in "Selecting Output Data" on page 5-16, you must also select the corresponding input data samples. This operation makes the number of I/O data samples equal.

- 1 In the Control and Estimation Tools Manger GUI, select the **New Data** node under the **Transient Data** node.
- 2 In the Input Data tab, select the input1(:,1) cell, and click Pre-process.

This action updates the Data Preprocessing Tool GUI with the selected data input1(:,1).

- **3** To exclude the data beyond t = 0.5 s:
 - a In the Exclusion Rules tab, select the Bounds check box.
 - b In the Exclude X >= field, where X corresponds to the time vector, select > from the drop-down list. Enter 0.5 in the adjacent field to specify the upper limit of the input data to select for estimation.
 - c In the Write results to area, verify that the existing dataset Dataset option remains selected.
 - d Click Add.

This action adds the modified data input1(:,1)* to the **Input Data** tab of the **Dataset1** node. This operation also replaces the input data samples beyond t = 0.5 s in input1(:,1)* with NaNs.

- 4 To remove the NaNs:
 - a In the Control and Estimation Tools Manager GUI, select the input1(:,1)* cell in the Input Data tab of the Dataset1 node, and click Pre-process.

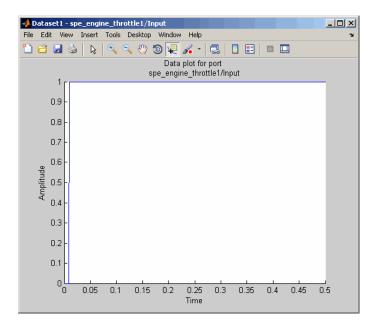
This action updates the Data Preprocessing Tool GUI with the selected data input1(:,1)*.

- **b** In the Missing Data Handling area, select the **Remove rows where** all data is excluded or missing check box.
- c In the Write results to area, verify that the
 existing dataset □ataset1 option remains selected.
- d Click Add.

The Update table data dialog box appears. Click **Yes** to overwrite the input1(:,1)* data set with the modified data.

5 To plot the data, select the input1(:,1)* cell in the Input Data tab of the Dataset1 node, and click Plot Data.

The selected input data from t = 0 s to t = 0.5 s is shown in the next figure.



Removing Outliers

In this section...

"Why Remove Outliers" on page 5-25

"How to Remove Outliers" on page 5-25

Why Remove Outliers

Outliers are 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.

Removing outliers replaces the data samples containing outliers with NaNs, which represent missing data. You interpolate the missing data values in a subsequent operation, as described in "Interpolating Missing Data" on page 5-34.

How to Remove Outliers

In this portion of the tutorial, you remove outliers from the output data. You must have already selected a subset of the data, as described in "Selecting Data for Estimation" on page 5-16. If you have not done this preparation, click here.

1 In the Control and Estimation Tools Manger, select the position1(:,1)* cell in the **Output Data** tab of the **Dataset1** node, and click **Pre-process**.

This action updates the Data Preprocessing Tool GUI with the selected data position1(:,1)*.

2 In the Exclusion Rules tab, select the Outliers check box.

By default, the **Window length** and **Confidence limits** fields are set to 10 and 95 respectively. The **Window length** field specifies the number of successive data samples the software uses to compute the mean and standard deviation. The **Confidence limits** field specifies the threshold number for identifying outliers. In this example, the mean and standard deviation of 10 successive data samples are computed, and data values that exceed 95% of standard deviation are identified as outliers.

Data Preprocessing Too			
10dify data fromthrottle1/	Position Vrite results to:	existing dataset Dataset1 O new dataset Dataset2	
Data Editing			
Raw data Modified data		Exclusion Rules Detrend/Filtering	
Time (seconds)	position1(:,1)**		
		Bounds	
8.0e-3	15.649		
9.0e-3	NaN		
1.0e-2	15.944	Exclude X: <= Exclude X: >= Exclude X: >= Exclude X: >= Exclude X: >= Exclude X: >= Exclude X: = Exc	
1.100e-2	15.238		
1.200e-2	15.817	Exclude Y: <=	
1.300e-2	15.385		
1.400e-2	16.11	✓ Outliers	
1.500e-2	15.879	Je Oddiers	
1.600e-2	15.748		
1.700e-2	15.260	Window length: 10	
1.800e-2	14.688		
1.900e-2	NaN	Confidence limits (%): 95	
2.0e-2	15.391		
2.100e-2	14.742	MATLAB expression abs(x)>1	
Grey back Excluded by a rule Red text Manually excluded Exclude Graphically Flattines Window: 5			
Missing Data Handling			
Remove rows where al	data is excluded or missing	☐ Interpolate missing values using interpolation method	
		Add Close Hep	

Note The data samples containing outliers are replaced with NaNs.

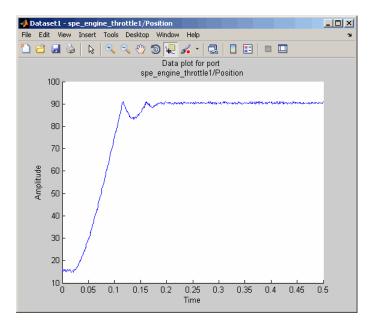
3 In the Write results to area, verify that the existing dataset Dataset1 verify option remains selected.

4 Click Add.

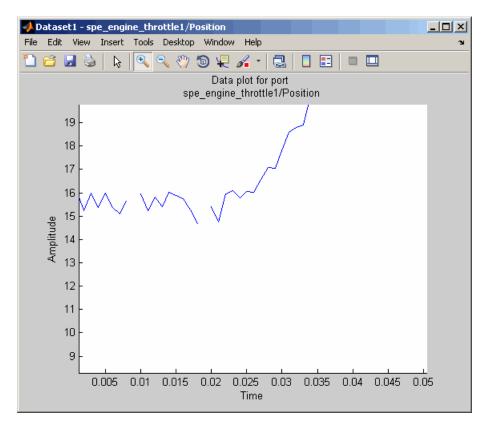
The Update table data dialog box appears. Click **Yes** to overwrite the position1(:,1)* data set with the modified data.

5 To plot the data, select the position1(:,1)* cell in the **Output Data** tab of the **Dataset1** node, and click **Plot Data**.

The spikes, which indicate outliers, no longer appear on the time plot, as shown in the next figure.



The missing data samples, represented by NaNs, appear as gaps on the time plot. To see an example, zoom in to the bottom-left corner of the plot. As shown in the next figure, the data values corresponding to t = 0.009 s and t = 0.019 s are missing.



Filtering Data

In this section ...

"Filtering Output Data" on page 5-29

"Filtering Input Data" on page 5-32

Filtering Output Data

In this portion of the tutorial, you filter the noise, and remove any periodic trends from the output data. To avoid relative phase shift between the I/O data, you must also apply the same filter to the input data.

You must have already removed outliers from the output data, as described in "Removing Outliers" on page 5-25. If you have not done this preparation, click here.

1 In the Control and Estimation Tools Manager window, select the position1(:,1)* cell in the **Output Data** tab of the **Dataset1** node, and click **Pre-process**.

This action updates the Data Preprocessing Tool GUI with the selected data position1(:,1)*.

2 In the **Detrend/Filtering** tab:

a Select the Filtering check box.

By default, First order is selected as the filter type. To learn more about the filters, see "Filtering Data".

b Specify 0.001 in the **First order filter with time constant** field.

This field specifies the time constant for the first-order filter.

Tip For calculating the time constant, you can visually inspect the time plot to determine the frequency components.

🙀 Data Preprocessing Tool	for Dataset: Dataset1	×
Modify data from throttle1/F	Position 👻 Write results to	: O existing dataset Dataset1 V • new dataset Dataset2
. ,		
Data Editing		
Raw data Modified data		Exclusion Rules Detrend/Filtering
Time (seconds)	position1(:,1)*	
0.0	15.116	Detrending Constant
1.0e-3	15.964	C Straight Line
2.0e-3	15.241	0.000
3.0e-3	15.993	
4.0e-3	15.376	Filtering Select filter type First order
5.0e-3	15.997	
6.0e-3	15.380	
7.0e-3	15.100	First order filter with time constant 0.001
8.0e-3	15.649	
9.0e-3	NaN	
1.0e-2	15.944	
1.100e-2	15.238	
1.200e-2	15.817	
1.300e-2	15.385	
	-	
Grey back Excluded by	u a rula	
Citey back Excluded by	Exclude Graphically	1
Red text Manually ex	xcluded	
Missing Data Handling		
Prissing Data nanding		
Remove rows where	data is excluded or miss	ing Interpolate missing values using interpolation method zoh
		Add Close Help

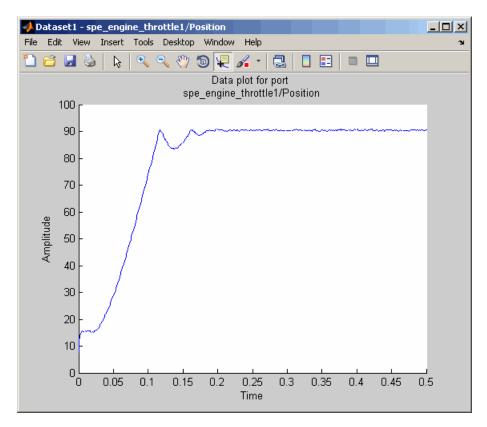
3 In the Write results to area, verify that the existing dataset Dataset **Dataset1** verify that the existing dataset

4 Click Add.

The Update table data dialog box appears. Click **Yes** to overwrite the position1(:,1)* data set with the modified data.

5 To plot the data, select the position1(:,1)* cell in the Output Data tab of the Dataset1 node, and click Plot Data.

The noise is filtered and the output data appears smooth, as shown in the next figure.



Filtering Input Data

After you filter the output data, as described in "Filtering Output Data" on page 5-29, you must also filter the input data with the same filter.

1 In the Control and Estimation Tools Manager window, select the input1(:,1)* cell in the Input Data tab of the Dataset1 node, and click Pre-process.

This action updates the Data Preprocessing Tool GUI with the selected data $input1(:,1)^*$.

- 2 In the **Detrend/Filtering** tab:
 - **a** Select the **Filtering** check box.

By default, First order is selected as the filter type.

- **b** Specify 0.001 in the **First order filter with time constant** field.
- **3** In the Write results to area, verify that the existing dataset Dataset1 verify option remains selected.
- 4 Click Add.

The Update table data dialog box appears. Click **Yes** to overwrite the input1(:,1)* data set with the modified data.

- U × 📣 Dataset1 - spe_engine_throttle1/Input File Edit View Insert Tools Desktop Window Help 🛅 🗃 😹 🖕 🔍 🔍 🖤 🕲 🐙 🔏 - 😓 🔲 🖽 💷 💷 Data plot for port spe_engine_throttle1/Input 1 0.9 0.8 0.7 0.6 0.6 Amplitude 0.4 0.3 0.2 0.1 0 L 0 0.15 0.2 0.25 0.3 0.35 0.45 0.05 0.1 0.4 0.5 Time
- 5 To plot the data, select the input1(:,1)* cell in the Input Data tab of the Dataset1 node, and click Plot Data.

Interpolating Missing Data

In this portion of the tutorial, you interpolate the output data to compute the missing values created when removing outliers, as described in "Removing Outliers" on page 5-25. The interpolation operation uses the known data values to compute the missing data values.

You must have already filtered the noise, as described in "Filtering Data" on page 5-29. If you have not already done this preparation, click here.

1 In the Control and Estimation Tools Manager GUI, select the position1(:,1)* cell in the **Output Data** tab of the **Dataset1** node, and click **Pre-process**.

This action updates the Data Preprocessing Tool GUI with the selected data position1(:,1)*.

2 In the Missing Data Handling area, select the Interpolate missing values using interpolation method check box.

By default, zoh is selected as the interpolation method. This method fills the missing data sample with the data value immediately preceding it.

😺 Data Preprocessing Tool	for Dataset: Dataset1	×	
Modify data fromthrottle1/f	Position 💌 Write results to:	existing dataset Dataset1 C new dataset Dataset2	
-Data Editing			
Raw data Modified data		Evolution Dates I and a literative D	
		Exclusion Rules Detrend/Filtering	
Time (seconds)	position1(:,1)**	Bounds	
0.0	15.116		
1.0e-3	15.964		
2.0e-3	15.241	Exclude X: <= 💌 Exclude X: >= 💌	
3.0e-3	15.993		
4.0e-3	15.376	Exclude Y: <=	
5.0e-3	15.997		
6.0e-3	15.380	_	
7.0e-3	15.100	Cutliers	
8.0e-3	15.649		
9.0e-3	15.649	Window length: 10	
1.0e-2	15.944	windowiengch: jro	
1.100e-2	15.238		
1.200e-2	15.817	Confidence limits (%): 95	
1.300e-2	15.385		
		MATLAB expression abs(x)>1	
Grey back Excluded by a rule Red text Manually excluded			
Missing Data Handling			
Remove rows where all v data is excluded or missing V Interpolate missing values using interpolation method zoh			
		Add Close Help	

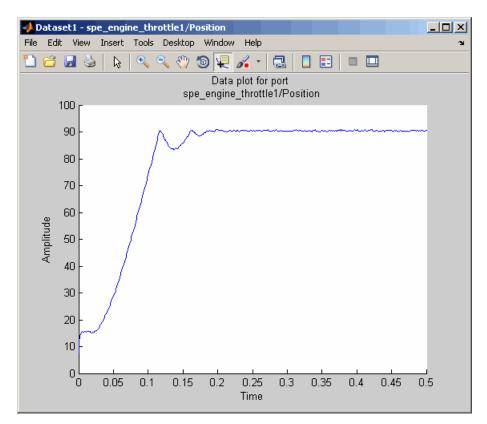
Tip You can view the interpolated data samples in the Modified data tab.

- **3** In the Write results to area, verify that the existing dataset Dataset **Dataset1** verify option remains selected.
- 4 Click Add.

The Update table data dialog box appears. Click **Yes** to overwrite the position1(:,1)* data set with the modified data.

5 To plot the data, select the position1(:,1)* cell in the **Output Data** tab of the **Dataset1** node, and click **Plot Data**.

The new estimation data, prepared by removing outliers, filtering noise, and interpolating missing data, is shown the next figure.



Saving the Project

After you prepare the data, you can delete the data in the **New Data** node, rename the prepared data, and save the session. To skip the data preparation steps, click here.

- **1** In the Control and Estimation Tools Manger GUI, select the **New Data** node under the **Transient Data** node.
- 2 Right-click the New Data node, and select Delete.
- **3** Select the **Dataset1** node under the **Transient Data** node.
- 4 Right-click the **Dataset1** node, and select **Rename**. Specify Estim_Data_Prep as the name of the new estimation data set.

The Control and Estimation Tools Manager GUI resembles the next figure.

🙀 Control and Estimation Tools Mana	ger				
File View Help					
唑 🗅 🚅 🖬 🔳					
📣 Workspace	Input Data Output Da	ta State Data			
Project - spe_engine_throttle1	Assign data to blocks				
Estimation Task	Block Name	Data	Time / Ts	Weight	Length
	spe_engine_thrott	le1/Input			
Variables	Channel - 1	input1(:,1)*	time1(:,1)***	1	501/501
요. Estimation 요 California					
		Import	Pre-process	Plot Data	Clear All
,					
1					×
					<u> </u>
Select the tabbed panels to configure the tra	ansient data set.				

- 5 Save the Control and Estimation Tools Manager project:
 - **a** In the Control and Estimation Tools Manager GUI, select **File > Save**.

This action opens the Save Projects dialog box.

b In the Save Projects dialog box, click **OK**.

📣 Save Projects	
Projects:	
Project - spe_engine_throttle1	_
	-
OK Cancel	Help
	neib

c In the Save Projects window, specify spe_engine_throttle1p.mat in the File name field, and click Save.

The action saves the project as a MAT-file.

To learn how to estimate parameters from this data, see Chapter 6, "Tutorial — Estimating Parameters from Measured Data Using the GUI".

Tutorial — Estimating Parameters from Measured Data Using the GUI

- "About This Tutorial" on page 6-2
- "Estimating Model Parameters Using Default Estimation Settings" on page 6-7
- "Improving Estimation Results Using Parameter Bounds" on page 6-20
- "Validating Estimated Model Parameters" on page 6-26

About This Tutorial

In this section...

"Objectives" on page 6-2

"About the Model" on page 6-3

Objectives

In this tutorial, you learn how to estimate parameters of a single-input single-output (SISO) Simulink model from measured input and output (I/O) data.

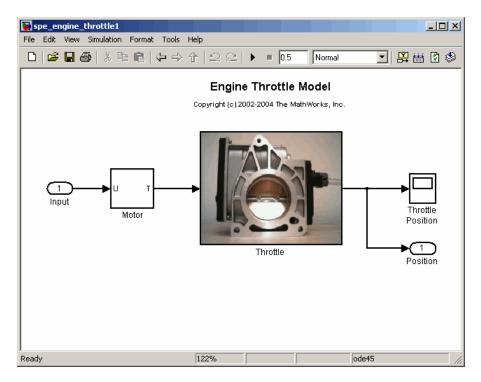
Note Simulink Design Optimization software estimates parameters from real, time-domain data only.

You learn to perform the following tasks using the GUI:

- Load a saved project containing data.
- Estimate model parameters using default settings.
- Validate the model, and refine the estimation results.

About the Model

In this tutorial, you use the spe_engine_throttle1 Simulink model. This model represents an engine throttle system, as shown in the next figure.

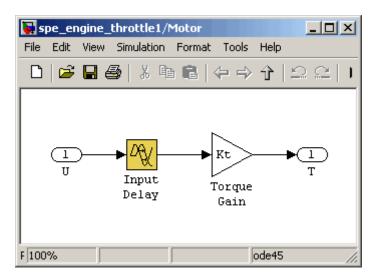


The throttle system controls the flow of air and fuel mixture to the engine cylinders. The throttle body contains a butterfly valve which opens when a driver presses the accelerator pedal. Opening this valve increases the amount of fuel mixture entering the cylinders, which increases the engine speed. A DC motor controls the opening angle of the butterfly valve in the throttle system. The models for these components are described in "Motor Subsystem" on page 6-4, and "Throttle Subsystem" on page 6-5.

The input to the throttle system is the motor current (in amperes), and the output is the angular position of the butterfly valve (in degrees).

Motor Subsystem

The Motor subsystem contains the DC motor model. To open the model, double-click the corresponding block.

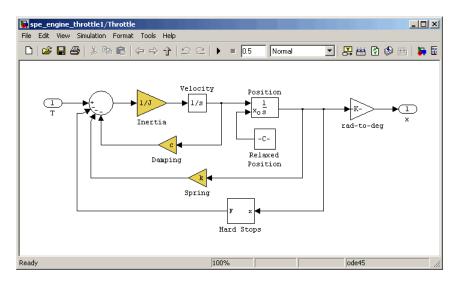


The following table describes the variables, parameters, equation, input, and output of the Motor subsystem.

Variables	U is the input current to the motor.
	T is the torque applied by the motor.
Parameters	$K_{\rm t}$ is the torque gain of the motor, represented by Kt in the model.
	${\tt t}_{\rm d}$ is the input time delay of the motor, represented by <code>input_delay</code> in the model.
Equation	The torque applied by the motor is described in the following equation:
	$T(t) = K_t U(t - t_d)$
	where t is time.
Input	U
Output	Т

Throttle Subsystem

The Throttle subsystem contains the butterfly valve model. To open the model, right-click the corresponding block, and select Look Under Mask.



The Hard Stops block models the value angular position limit of 15° to 90° .

The following table describes the variables, parameters, states, differential equations, inputs, and outputs of the Throttle subsystem.

Variables	T is the torque applied by the DC motor.
	θ is the angular position of the valve, represented by x in the model.
	$T_{hardstop}$ is the torque applied by the hard stop.
Parameters	J is the inertia.
	c is the viscous friction.
	k is the spring constant.
States	θ is the angular position.
	$\dot{\theta}$ is the angular velocity.

Equations	The mathematical system for the butterfly valve is described in the following equation: $J\ddot{\theta} + c\dot{\theta} + k\theta = T + T_{hardstop}$ where $15^{\circ} \le \theta \le 90^{\circ}$, with initial conditions $\theta_0 = 15^{\circ}$, and $\dot{\theta}_0 = 0$. The torque applied by the Hard Stops block is described in the following equation: $T_{hardstop} = \begin{cases} 0, & 15^{\circ} \le \theta \le 90^{\circ} \\ K(90^{\circ} - \theta), \theta > 90^{\circ} \\ K(15^{\circ} - \theta), \theta < 15^{\circ} \end{cases}$ where K is the gain of the Hard Stops block.
Input	<i>T</i>
Output	heta

Estimating Model Parameters Using Default Estimation Settings

In this section ...

"Overview of the Estimation Process" on page 6-7

"Specifying Parameters and Estimation Data" on page 6-8

"Validating Model Parameters" on page 6-13

Overview of the Estimation Process

Simulink Design Optimization software uses optimization techniques to estimate model parameters. In each optimization iteration, the model is simulated with the current parameter values. The error between the simulated and measured output is computed and minimized. The estimation is complete when the optimization method finds a local minimum.

You perform the following tasks to estimate the model parameters:

- "Specifying Parameters and Estimation Data" on page 6-8
- "Validating Model Parameters" on page 6-13

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

Specifying Parameters and Estimation Data

To specify parameters and estimation data:

1 Load the preconfigured project spe_engine_throttle1p.mat.

This MAT-file contains the estimation data. To learn more about the data and how to prepare it, see Chapter 5, "Tutorial — Preparing Data for Parameter Estimation Using the GUI".

To load the preconfigured project:

a Open the engine throttle system Simulink model by typing the following at the MATLAB prompt:

spe_engine_throttle1

b In the Simulink model window, select **Tools > Parameter Estimation**.

This action opens a new project named **Project - spe_engine_throttle1** in the Control and Estimation Tools Manager GUI.

c In the Control and Estimation Tools Manager GUI, select File > Load. Browse to the *matlabroot*\toolbox\sldo\sldodemos\estim directory, and select spe_engine_throttle1p.mat.

The Confirm Node Replace dialog box appears. Click **Yes** to load the project into the Control and Estimation Tools Manager GUI.

The Control and Estimation Tools Manager GUI opens as shown in the next figure.

🙀 Control and Estimation Tools Mana	ger				
File View Help					
🛎 🗅 🚅 🖬 🔟					
📣 Workspace	Input Data Output Dat	a State Data			
Project - spe_engine_throttle1	Assign data to blocks				
Estimation Task	Block Name	Data	Time / Ts	Weight	Length
Estim_Data_Prep	spe_engine_throttle1/Input				
Variables	Channel - 1	input1(:,1)*	time1(:,1)***	1	501/501
Estimation ⊕_ Government Ball Station					
		Import	Pre-process	Plot Data	Clear All
p					
					<u> </u>
					•
Select the tabbed panels to configure the transient data set.					

- 2 Specify parameters for estimation.
 - **a** Select the Variables node under the Estimation Task node.

🙀 Control and Estimation Tools Manag	er 📃			
File View Help				
🛎 🗅 🚅 🖬 🗐				
Workspace Project - spe_engine_throttle1 Transient Data Transient Data	Estimated Parameters Estimated States Selected parameters Default settings Name: Value: Initial guess: Minimum: Maximum: Typical value: Referenced by: Referenced by: Add Delete	A A		
		▲ ▼		
J Select the tab panels to configure your estima	ition parameters and states.	 //.		

b Click **Add**.

This action opens the Select Parameters dialog box, which shows the model parameters for the Simulink model.

c Select the parameters J, c, input_delay, and k by pressing the Ctrl key while clicking each name, and then click OK.

This action adds the selected parameters to the **Estimated Parameters** tab.

Select Parameters		2	
Select additional parameters to estimate			
Name	Size		
3	1×1		
Kt	1×1		
angle_init	1×1		
angle_open	1×1		
c	1×1		
input_delay	1×1		
k	1×1		
		Ŧ	
Specify expression (e.g., s.x, a(3)), b{2}):		
OK N Cancel H	telp A		

Tip When estimating a large number of parameters, you can select a subset of parameters to estimate. To learn more, see the Inverted Pendulum Parameter Estimation demo.

d Select the Estimation node, and click New.

This action adds a **New Estimation** node under the **Estimation** node.

e Select the New Estimation node.

f In the **Parameters** tab, select the **Estimate** check box for all the parameters.

This action specifies the selected parameters for estimation.

File View Help			
Workspace Project - spe_engine_throttle1 Estimation Task Image: Transient Data Image: T			
- New Estimation node has been added to Estimation.			
Select the tab panels to configure your estimation.			

The **Parameters** tab, as shown in the previous figure, displays the following information for each parameter:

• **Value**: Current parameter value. By default, it is equal to the value specified in the Simulink model.

During estimation, the optimization method might change a parameter value to minimize the error between the measured and simulated output.

• **Initial Guess**: Initial parameter value. By default, it is equal to the value in the **Value** field and is used at the start of estimation.

- Minimum and Maximum: Bounds on the parameter value. By default, they are set to -Inf, and +Inf, respectively. This means that the optimization method searches for a solution in the range [-Inf, +Inf].
- **Typical Value**: Order of magnitude of the parameter value. By default, it is equal to the value in the **Initial Guess** field.
- 3 Specify data for estimation by checking the **Selected** check box for **Estim_Data_Prep** in the **Data Sets** tab.

🙀 Control and Estimation Tools Mana	ger			_ 🗆 ×
File View Help				
😁 🗅 🚅 🖬 📋				
Workspace Project - spe_engine_throttle1 Estimation Task Transient Data Uii Estim_Data_Prep Variables New Estimation New Estimation Extination Variables Variables Variables Variables Variables Variables Variables Variables Variables Variables Variables Variables Variables Variables Variables Variables Variables Variables	Data Sets Parameters States Estimation Data sets used for estimation Transient estimation Data Set Selected Estim_Data_Prep	Output data weigh	its	Weight
- New Estimation node has 1				
- New Estimation node has 1	peen added to istimation.			.
Load projects				1

Validating Model Parameters

After you specify parameters and data for estimation, as described in "Specifying Parameters and Estimation Data" on page 6-8, estimate the parameters and analyze the results to determine if the model needs to be refined.

- 1 Create plots to view the estimation results.
 - **a** Select the Views node under the Estimation node, and click New.

This action creates a New View node under the Views node.

- **b** Select the **New View** node.
- **c** In the **View Setup** tab, select the following plots from the drop-down list in the **Plot Type** column:
 - For **Plot 1**, select Measured and simulated.
 - For Plot 2, select Cost function.

🙀 Control and Estimation Tools	1anager				
File View Help					
🗚 🗅 🚅 🖬 🗐					
📣 Workspace	View Setup				
Project - spe_engine_throttle	Select plot types				
🗄 📲 Estimation Task	Plot Number Plot	Туре	Plot Title		
[iii] Fatin_Data_Pre	Plot 1 Measured and simul				
Variables	Plot 2 ((none)				
🖃 📴 Estimation	Plot 3 Cost function				
- By New Estimation	Plot 4 Measured and simu Plot 5 Parameter sensitivi				
Views	Plot 6 Parameter sensitiv				
E Validation	Residuals				
	(none)				
	I				
	Options				
	Estimation Plot 1				
	New Estimation				
Show Plots					
- New Estimation node has been added to Estimation.					
- New View node has been added to Views.					
Configure dynamic views.					

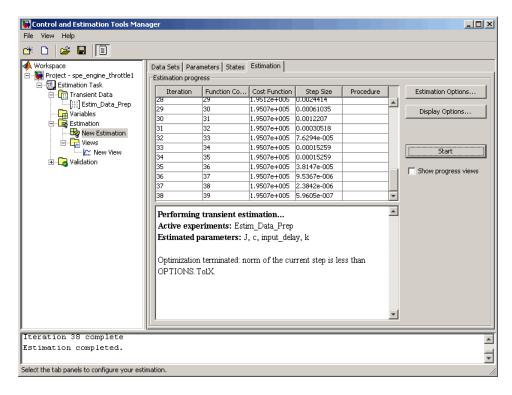
🙀 Control and Estimation Tools	Manager			
File View Help				
📸 🗅 🛸 🖬 🔟				
📣 Workspace	View Setup			
Project - spe_engine_throttle	Select plot types			
🖻 📲 Estimation Task	Plot Number	Plot Type	Plot Title	
Estim_Data_Pre	Plot 1	Measured and simulated		
Variables	Plot 2	Cost function		
Estimation	Plot 3	(none)		
🛶 🔤 New Estimation	Plot 4	(none)		
E 🕞 Views	Plot 5 Plot 6	(none)		
🛛 🕍 New View		(none)		
	Options Estimation New Estimation	Plot 1 Plot 2	5	
- New Estimation node has been added to Estimation.				
- New View node has been				
- New View house has bee	in added to views	•		
Configure dynamic views.			-	

d In the **Options** area, select the **Plot 1** and **Plot 2** check boxes, and click **Show Plots**.

Clicking **Show Plots** opens the following figures:

- New View Plot 1 (Measured and simulated): Displays the measured output data Estim_Prep_Data. During estimation, this plot updates to display the simulated response at each iteration.
- New View Plot 2 (Cost function): Displays the error between the measured and simulated output, computed at each iteration. By default, the error is computed using *sum- of- squared- errors*, which is a least-squares method.
- 2 Estimate the parameters by selecting the Estimation tab of the New Estimation node, and clicking Start.

The **Estimation** tab updates at each iteration, and provides information about the estimation progress. When the estimation completes, the **Estimation** tab resembles the next figure.



Note The results of the optimization may differ slightly because of different numerical precision across platforms.

The table displays the following information for each iteration:

• Cost Function: Error between the simulated and measured output.

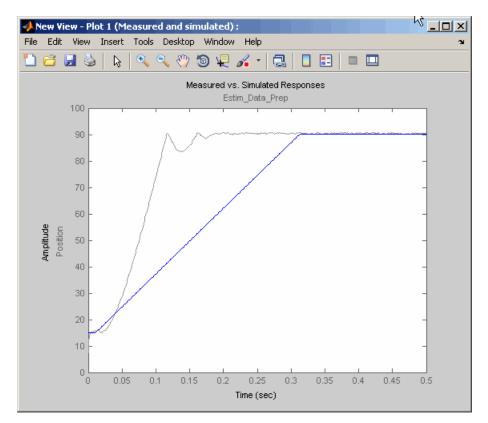
During estimation, the default optimization method Nonlinear least squares, lsqnonlin, minimizes the cost function by changing the parameter values. As shown in the previous figure, the cost function decreases by only 27% from 2.69e5 to 1.95e5.

- **Step Size**: Displacement of the optimization method in the current search direction when minimizing the cost function. A final value close to zero indicates that the method has found a local minimum. As shown in the previous figure, the final step size in the last iteration is 5.96e-7.
- **3** View the estimated parameter values in the **Value** column of the **Parameters** tab.

Control and Estimation Tools Manager								
File View Help								
🗳 🗅 🞏 🖬 🔳								
📣 Workspace	Data Sets Para	meters States	Estimation					
E- Noject - spe_engine_throttle	Estimation parameters							
Estimation Task	Name	Value	Estimate	Initial Guess	Minimum	Maximum	Typical Value	
Estim_Data_Pre	J	0.0017013	~	J	-Inf	+Inf	J	
Variables	c	39.994	V	с	-Inf	+Inf	с	
Estimation	input_delay	0.00016981	~	input_delay	-Inf	+Inf	input_delay	
- By New Estimation	k	0.99889	v	k	-Inf	+Inf	k	
E- 🔁 Views								
🔤 🗠 New View								
🗄 🔂 Validation								
		Lice Veli	ie as Initial Gue	cc Decet	: to Default Set	tipae S	ave as Default Se	ttipge
		Use vai	ie en miliai auc	Neset	ron perault per	ungs 2	ave as per duit ac	ounga.
Iteration 37 complete								<u> </u>
Estimation completed.								•
1								•
Select the tab panels to configure you	r estimation.							11.

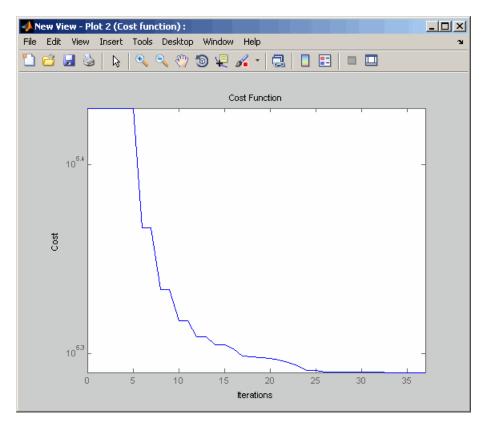
4 Examine the simulated response plot to see how well the simulated output matches the measured output.

The simulated response with the estimated parameters is shown in blue on the New View - Plot 1 (Measured and simulated) plot, and significantly differs from the measured data. This difference indicates that the estimated parameters are not accurate.



5 Examine the cost function plot to see how the cost function changes during the estimation.

The cost function values shown in step 2 are plotted on the New View - Plot 2(Cost function) plot. The cost function decreases by only 27% from its initial value, and in conjunction with the measured and simulated output plot, indicates that the estimated parameters are not accurate.



To improve the estimation results, you apply bounds on the parameter values, as described in "Improving Estimation Results Using Parameter Bounds" on page 6-20.

Improving Estimation Results Using Parameter Bounds

In this section...

"Strategy for Improving the Estimation Results" on page 6-20

"How to Specify Parameter Bounds" on page 6-20

Strategy for Improving the Estimation Results

Analyzing the Measured and Simulated and Cost function plots, as described in "Validating Model Parameters" on page 6-13, you see that the model parameters estimated using the default estimation settings are not accurate. You must run additional estimations to improve the accuracy of the model. There are several techniques to improve the accuracy of the estimated parameters. For more information, see "How to Specify Estimation Options in the GUI".

In this portion of the tutorial, you improve the results by specifying bounds on parameter values. This technique restricts the region in which the optimization method searches for a local minima.

Based on physical insight, you know the following characteristics of the engine throttle system:

- All the parameter values are positive.
- Maximum time delay of the system, represented by input_delay, is 0.1 s.

Therefore, you specify 0 as the minimum value for all parameters, and 0.1 as the maximum value of input_delay.

After you estimate the parameters, analyze the results using the Measured and Simulated and Cost function plots.

How to Specify Parameter Bounds

You must have already estimated the parameters using default settings, as described in "Estimating Model Parameters Using Default Estimation Settings" on page 6-7.

To improve the estimation results by specifying parameter bounds:

- 1 Select the **Parameters** tab of the **New Estimation** node.
- **2** Specify the minimum value for each parameter by double-clicking the corresponding **Minimum** cell, and replacing Inf with 0.
- **3** Specify the maximum value for input_delay by double-clicking the corresponding **Maximum** cell and replacing +Inf with 0.1.

The **Parameters** tab resembles the next figure.

🗅 😅 🖶 🗐							
Workspace	Data Sets Para	meters States	Estimation				
Project - spe_engine_throttle	Estimation parar	neters					
Estimation Task	Name	Value	Estimate	Initial Guess	Minimum	Maximum	Typical Value
Estim_Data_Pre	3	0.0017013	~	J	0	+Inf	J
Variables	c	39.994	v	с	0	+Inf	c
Estimation	input_delay	0.00016981	~	input_delay	0	0.1	input_delay
New Estimation	k	0.99889		k	0	+Inf	k
		Use Valu	ue as Initial Gue	ess Reset	: to Default Set	tings 5.	ave as Default Setting
ration 37 complete		Use Valu	ue as Initial Gue	Reset	: to Default Set	tings S	ave as Default Setting

4 Select the Estimation tab, and click Start.

The **Estimation** tab updates at each iteration, and provides information about the estimation progress. When the estimation completes, the **Estimation** tab resembles the next figure.

₩ Control and Estimation Tools N File View Help	1anager						
🗥 🗋 😅 🖬 📳	Data Sets Param	ators Statos F	stimation				
- W Project - spe_engine_throttle	Estimation progre		· · · · · · · · · · · · · · · · · · ·				
🖃 🗐 Estimation Task	Iteration Function Count Cost Function Step Size Procedure						
🖃 🛅 Transient Data	Iteration 35	Function Count	Cost Function 239.96	5tep Size	Procedure		Estimation Options
[iii] Estim_Data_Pre	36	37	239.96	0.01434			Diselas Ostisas
Variables	37	38	239.96	0.003585		-	Display Options
Estimation	38		239.96	0.00089624		1	
New Estimation	39		239.96	0.00022406			
🖻 📴 Views	40	41	239.95	5.6015e-005			Charl
🖳 🗠 New View	41	42	239.95	5.6015e-005			Start
🗄 🔂 Validation	42		239.95	1.4004e-005			E chamarana a
	43		239.94	3.5009e-006			Show progress views
	44	45	239.94	3.5009e-006			
	45	46	239.94	8.7524e-007		-	
۲ ۲	Active exper Estimated pa		_Data_Prep c, input_delay,	k ent step is less t	han		
الغد المستحد الت						_	
Iteration 45 complete							
Estimation completed.							
iave projects							

Note The results of the optimization may differ slightly because of different numerical precision across platforms.

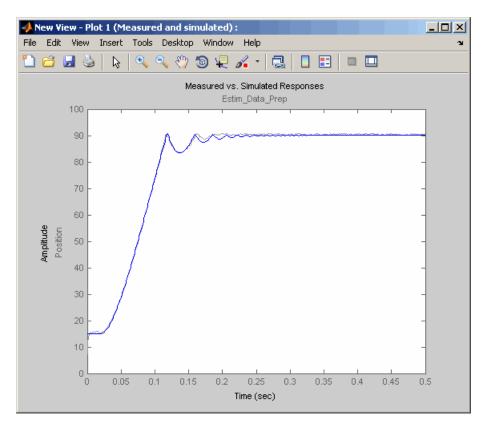
The cost function decreases by 99% from 1.95e5 to 239.94. The final step size 8.75e-7 is close to 0, which indicates that the optimization method has found a local minimum.

Workspace Import - spe_engine_throttle	Data Sets Para		Estimation				
🖃 🗐 Estimation Task	Name	Value	Estimate	Initial Guess	Minimum	Maximum	Typical Value
🖃 🛄 Transient Data	J	0.22295		Inicial Gdess	0	+Inf	
[;;;;] Estim_Data_Pre		9.0178	- -	, ,	0	+Inf	c
Variables	input delay	0.0090554	- -	input delay	0	0.1	input delay
New Estimation	<u>k</u>	20.216	<u> </u>	k	0	+Inf	k
				- 1			
			ie as Initial Gue	ss Reset	to Default Set	tinas S.	ave as Default Setting

5 View the estimated parameter values in the **Value** column of the **Parameters** tab.

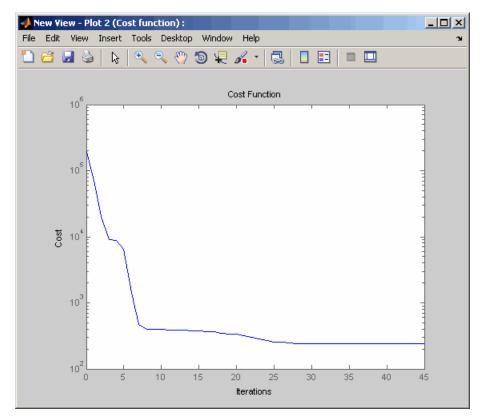
6 Examine the simulated response plot to see how well the simulated output matches the measured output.

The simulated response plot, shown in blue on the New View - Plot 1 (Measured and simulated) plot, is overlaid on the measured output data. The simulated output closely matches the measured data.



7 Examine the cost function plot to see how the cost function changes during the estimation.

The cost function values shown in step 4 are plotted on the New View - Plot 2(Cost function) plot. The cost function also decreases by 99% from its initial value, and in conjunction with the measured and simulated response plot indicates a good fit of the simulated response with the measured data.



Validating Estimated Model Parameters

After you estimate the model parameters, as described in "Improving Estimation Results Using Parameter Bounds" on page 6-20, validate the model using another data set *(validation data)*. A good match between the simulated response and the validation data indicates that you have not overfitted the model.

To validate the estimated parameters using a validation data set:

1 Load the preconfigured project spe_engine_throttle1_importValData1.mat.

This MAT-file contains a validation data set already imported into the GUI, and the estimated parameters as described in "Improving Estimation Results Using Parameter Bounds" on page 6-20.

Tip To learn how to import data and time vector into the Control and Estimation Tools Manager GUI, see "Importing Data into the GUI" on page 5-6.

To load the preconfigured project:

- a In the Control and Estimation Tools Manager GUI, select File > Load.
- **b** Browse to the *matlabroot*\toolbox\sldo\sldodemos\estim directory, and select spe_engine_throttle1_importValData1.mat.

The Confirm Node Replace dialog box appears. Click **Yes** to load the project into the Control and Estimation Tools Manager GUI.

The Control and Estimation Tools Manager GUI now has a node that contains the validation data, as shown in the next figure.

5 🗅 🗳 🖬 🗐		1	5			
Workspace		Output Dat a to blocks	a State Data			
Estimation Task		Name	Data	Time / Ts	Weight	Length
	spe_en	gine_throttl	e1/Input			
Image: Provide the second	Channe		input2(:,1)	time2(:,1)	1	500/500
		Import	Pre-p	rocess	Plot Data	Clear All

The validation data contains the input data, output data and time vector in the MATLAB variables input2, position2 and time2 respectively.

- 2 Plot the measured and simulated response, and residuals.
 - **a** Select the Validation node, and click New.

This action creates a New Validation node.

- **b** Select the **New Validation** node.
- In the Validation Setup tab, select the following plots in the Plot Type column.
 - For **Plot 1**, select Measured and simulated.
 - For Plot 2, select Residuals.
- d In the Options area, select the Plot 1, and Plot 2 check boxes.

e Select Validation Data 1 from the Validation data set drop-down list.

The Validation Setup tab resembles the next figure.

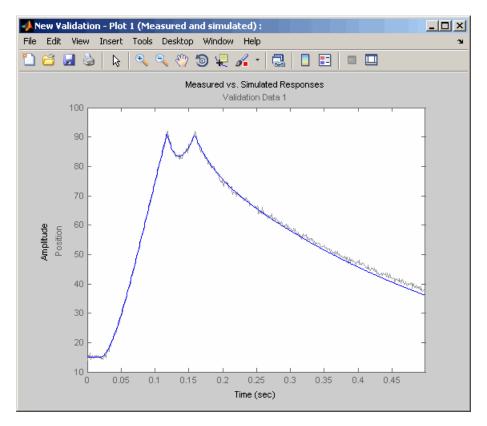
Control and Estimation Tools Manager					_ 🗆 X
🕂 🗅 🖙 🖬 🗐	Validation Setup				
Project - spe_engine_throttle1	Select plot types				
Image: Transient Data Image:	Plot 2 Res Plot 3 (no	Plot Type asured and simulated siduals ne) ne)		Plot Ti	tle
Estimation	Plot 5 (no	ne) ne)			
白- 🕞 Validation	· · · · · · · · · · · · · · · · · · ·	/alidation Data 1			
	Estimation New Estimation	Plot 1	Plot 2		
		[Show Plots		
 New Data node has been adde New Validation node has bee Configure validation plots. 					×

f Click Show Plots.

This action opens the following figures:

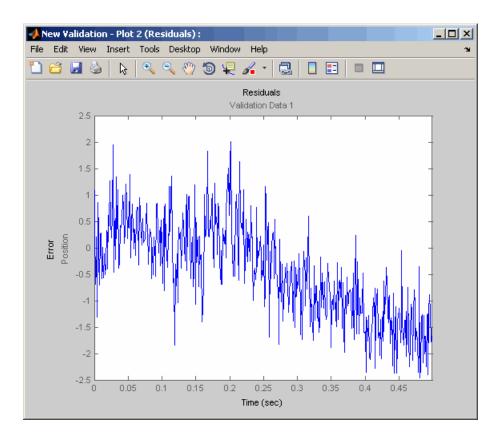
- New Validation Plot 1 (Measured and simulated): Displays the validation data and simulated response.
- New Validation Plot 2(Residuals): Displays the difference between the measured data and simulated response.
- **3** Examine the simulated response plot to see how well the simulated output matches the validation data.

The simulated response, as shown in blue on the New Validation - Plot 1 (Measured and simulated) plot, is overlaid on the measured output data, and closely matches the measured data Validation Data 1.



4 Examine the residuals plot to compare the difference between the simulated response and measured data.

The difference between the simulated and measured data, as shown in the New Validation - Plot 2(Residuals) plot, varies between 2 and -2.5. The residuals lie within 6% of the maximum output variation, and do not display any systematic patterns. This indicates a good fit between the simulated output and measured data.



- 5 Save the Control and Estimation Tools Manager project.
 - a In the Control and Estimation Tools Manager GUI, select File > Save.

This action opens the Save Projects dialog box.

- **b** In the Save Projects dialog box, click **OK**.
- **c** In the Save Projects window, specify the name of the project in the **File name** field, and click **Save**.

This action saves the project as a MAT-file.

Tip You can load a project by selecting **File > Load** in the Control and Estimation Tools Manager GUI. To view the estimated parameter values, select the **Parameters** tab of **New Estimation** node.



Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the GUI

- "About This Tutorial" on page 7-2
- "Configuring a Model for Optimizing Parameters" on page 7-5
- "Optimizing Model Parameters to Meet Step Response Requirements" on page 7-8
- "Refining Model Parameters to Track a Reference Signal" on page 7-25
- "Saving the Project" on page 7-31

About This Tutorial

In this section...

"Objectives" on page 7-2

"About the Model" on page 7-2

"Design Requirements" on page 7-4

Objectives

In this tutorial, you learn how to use the GUI to optimize the parameters of a Simulink model to meet time-domain design requirements.

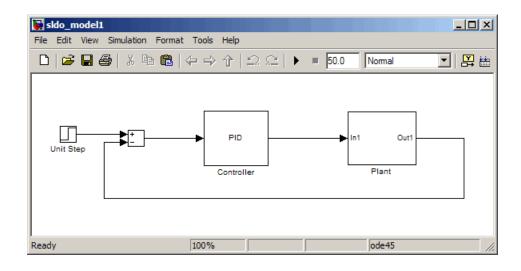
You accomplish the following tasks using the GUI:

- Specify step response requirements on the model's output.
- Specify a reference signal for the model's output to track.
- Optimize and refine the parameters to meet the design requirements.

Tip To learn how to optimize model parameters to meet time-domain design requirements using command line functions, see Chapter 8, "Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the Command Line".

About the Model

In this tutorial, you use the Simulink model named sldo_model1, as shown in the next figure.

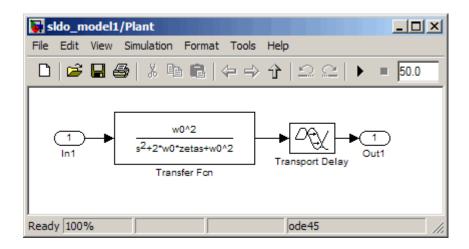


Tip To open the model, type sldo_model1 at the MATLAB command prompt.

The model contains a Controller block, which is a PID controller. This block controls the output of the Plant subsystem. The Unit Step block applies a step input to the system.

Note This tutorial uses a step input to produce the model's response and optimize the model parameters to meet step response requirements. You can also use other types of inputs, such as ramp, and optimize parameters to meet requirements on the model's response to these inputs.

Double-click the Plant subsystem to open it. The plant is a second-order system with delay. It contains Transfer Function and Transport Delay blocks, as shown in the next figure.



To learn more about the blocks, see Transfer Fcn and Transport Delay block reference pages.

Design Requirements

The model's output must meet the following design requirements:

- Overshoot less than 10%
- Rise time less than 10 seconds
- Settling time less than 30 seconds
- Track a reference signal specified by 1-e(-0.3*t), where t is time

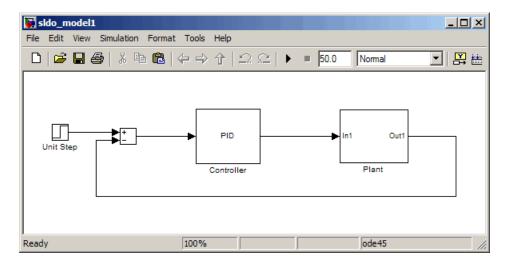
Configuring a Model for Optimizing Parameters

In this portion of the tutorial, you configure a Simulink model for optimizing parameters:

1 Open the sldo_model1 model, if it is not already open, by typing the model name at the MATLAB prompt:

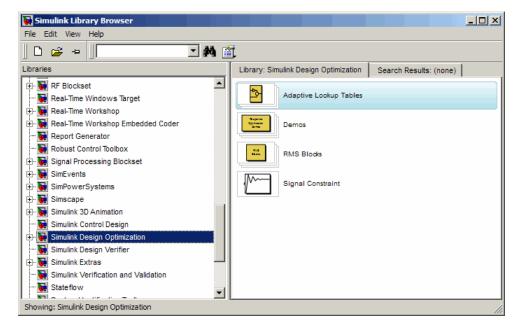
sldo_model1

The Simulink model opens, as shown in the next figure.



To learn more about the model, see "About the Model" on page 7-2.

- **2** Add a Signal Constraint block to the model.
 - **a** In the Simulink model, select **View > Library Browser** to open the Simulink Library Browser.



b In the Libraries pane, select Simulink Design Optimization.

c Drag and drop the Signal Constraint block into the Simulink model window.

To learn more about the block, see the Signal Constraint block reference page.

- 🙀 sldo_model1 * - D × File Edit View Simulation Format Tools Help 💽 🔛 📇 👔 🍪 🔛 🖡 🗋 😂 🖬 🚳 👗 🖻 🛍 (수 수 수 | 으 으) ► = 50.0 Normal tĒ PID Out1 Unit Step Plant Controller Signal Constraint Ready 100% ode45
- **d** Connect the Signal Constraint block to the model's output, as shown in the next figure.

Note You must connect the Signal Constraint block to the signal to which you want to add design requirements. To learn more, see "Constraining Model Signals" in the *Simulink Design Optimization User's Guide*.

Optimizing Model Parameters to Meet Step Response Requirements

In this section...

"Specify Time-Domain Design Requirements" on page 7-8

"Specifying Parameters to Optimize" on page 7-18

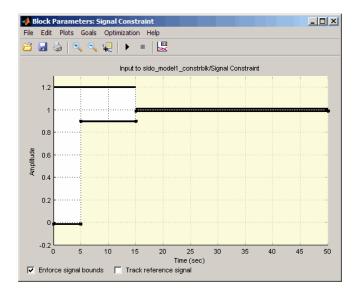
"Optimizing the Parameters" on page 7-21

Specify Time-Domain Design Requirements

In this portion of the tutorial, you specify the rise time, setting time and overshoot requirements that the model's response should satisfy.

You must have already configured the Simulink model, as described in "Configuring a Model for Optimizing Parameters" on page 7-5.

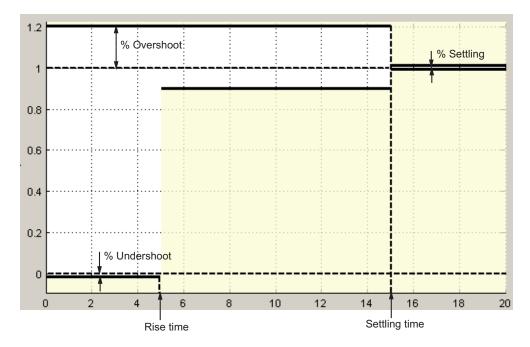
To specify the design requirements:



1 Double-click the Signal Constraint block to open the Block Parameters: Signal Constraint window.

Tip To view the line segment that extends from 0 to 15 seconds, set the upper limit of the **Amplitude** axis to 1.3. To do so, right-click in the plot, and select **Axes Limits**.

The Block Parameters window shows an amplitude versus time plot. The line segments map to the step response requirements shown in the next figure.



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

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

To specify the design requirements described in "Design Requirements" on page 7-4, you must modify the time and amplitude values of the line segments, as described in the next steps.

7

2 Double-click the lower yellow region shown on the plot.

This action opens the Edit Design Requirement dialog box, as shown in the following figure.

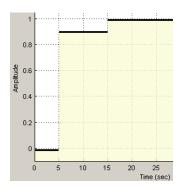
-	📕 Edit Desig	n Requirem	ent							
(Design require	ement: Lowe	er time respon	se bound from	n 0 to 50 sec	•				
٦	Design requirement parameters									
S	Segments:									
	Sta	art	En	d						
	Time	Amplitude	Time	Amplitude	Slope	Weight				
	0	-0.0100	5	-0.0100	0	1				
Į	5	0.9000	15	0.9000	0	1				
Į	15	0.9900	50	0.9900	0	1 🔻				
	InsertDelete									
					Clos					

The rows in the dialog box define the *lower bound* on the signal. The time span, displayed in the **Design requirement** drop-down list, is same as the simulation time specified in the Simulink model.

The rows correspond to the following three line segments in the Block Parameters window:

- First row Lower-line segment that extends from 0 to 5 seconds.
- Second row Lower-line segment that extends from 5 to 15 seconds.
- Third row Lower-line segment that extends from 15 to 50 seconds.

The default amplitude and time values specified in the rows appear in the Block Parameters window, as shown in the next figure.

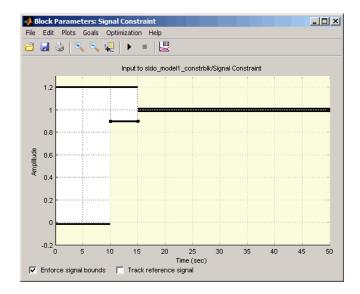


- **3** Specify the rise time requirement:
 - **a** In the first row of the Edit Design Requirement dialog box, double-click the **Time** field in the **End** column.
 - **b** Type 10, and press **Enter**.

The first row now looks like the following figure.

5	itart	Er	nd			
Time	Amplitude	Time	Amplitude	mplitude Slope Weight		
	-0.0100	10	-0.0100	0	1	

This action also updates the Block Parameters window to show the rise time requirement. This requirement is represented by the line segment at the bottom left of the window that extends from 0 to 10 seconds, as shown in the next figure.



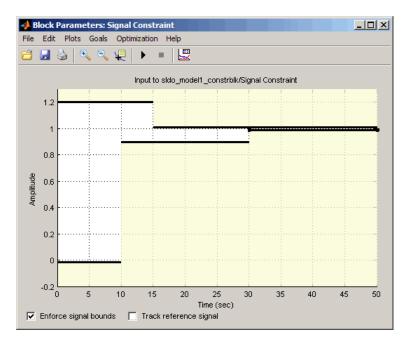
- **4** Specify the settling time requirement:
 - **a** In the second row of the Edit Design Requirement dialog box, double-click the **Time** field in the **End** column.

b Type **30**, and press **Enter**.

The second row looks like the following figure.

Start End		nd				
Time	Amplitude	Time	Amplitude	Slope	Weight	
0	-0.0100	10	-0.0100	0	1	
10	0.9000	30	0.9000	0	1	

This action also updates the Block Parameters window to show the settling time requirement on the lower bound. This requirement is represented by the line segment that extends from 10 to 30 seconds, as shown in the next figure.



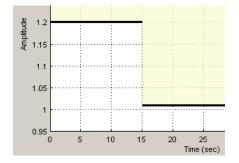
c In the Edit Design Requirements dialog box, select Upper time response bound from 0 to 50 sec in the **Design Requirement** drop-down list.

1	🙏 Edit Desig	n Requirem	ent			_ 🗆 X				
	Design require	ement: Uppe	r time respons	e bound from	0 to 50 sec	T				
Г	Design requirement parameters									
	Segments:									
	Sta	art	Er	nd						
	Time Amplitude		Time	Amplitude	Slope	Weight				
	0	1.2000	15	1.2000	0	1				
	15	1.0100	50	1.0100	0	1				
	Insert Delete									
					Close	Help				

The rows in the dialog box define the *upper bound* on the signal, and correspond to the following two line segments in the Block Parameters window:

- First row Upper-line segment that extends from 0 to 15 seconds.
- Second row Upper-line segment that extends from 15 to 50 seconds.

The default amplitude and time values specified in the rows appear in the Block Parameters window, as shown in the next figure.

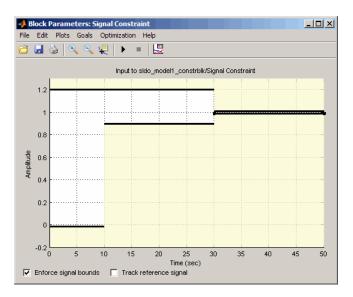


- d In the first row, double-click the Time field in the End column.
- e Type 30, and press Enter.

The first row looks like the following figure.

Start E		nd			
Time	Amplitude	Time	Amplitude	Slope	Weight
0	1.2000	30	1.2000	0	1

This action also updates the Block Parameters window to show the settling time requirement on the upper-bound. This requirement is represented by the line segment that extends from 0 to 30 seconds, as shown in the next figure.



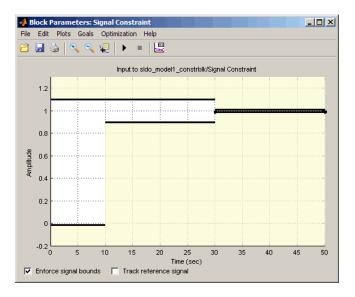
- **5** Specify the overshoot requirement:
 - a In the Edit Design Requirement dialog box, verify that Upper time response bound from 0 to 50 sec is selected in the Design Requirement drop-down list.
 - **b** In the first row, double-click the **Amplitude** field in the **Start** column.
 - c Type 1.1, and press Enter.

- d Double-click the Amplitude field of the End column.
- e Type 1.1, and press Enter.

The first row now looks like the following figure.

Start End			nd			
1	Time	Amplitude	Time Amplitude		Slope Weigh	
	0	1.1000	30	1.1	0	1

The Block Parameters window updates to show the overshoot requirement. This requirement is represented by the amplitude value of 1.1 for the line segment that extends from 0 to 30 seconds, as shown in the following figure.



When you optimize the model parameters, the model's final response must lie in the white region bounded by the line segments in order to meet the design requirements.

6 In Edit Design Requirement dialog box, click Close.

Specifying Parameters to Optimize

In this portion of the tutorial, you specify the model parameters to optimize. When you optimize the model parameters, the software modifies the parameter values to meet the specified design requirements.

Before you specify the parameters to optimize, you can specify the design requirements, as described in "Specify Time-Domain Design Requirements" on page 7-8.

To specify the parameters to optimize:

 In the Block Parameters window, select Optimization > Tuned Parameters.

This action opens the Tuned Parameters dialog box.

📣 Tuned Parameters		_ 🗆 ×
Tuned parameters	Optimization Settings Name: Value: Initial guess: Minimum: Maximum: Typical value: Typical value: Referenced by:	
Add Delete	OK Cancel	▲ ▼ Help

2 In the Tuned Parameters window, click **Add**.

This action opens the Add Parameters dialog box.

📣 Add Parameter	s X III
Select workspace v	variables:
Kd Ki Kp w0 zeta	
2010	•
Specify expression	(e.g., s.x or a(3)):
0	K Cancel Help

3 In the Add Parameters dialog box, select the PID controller parameters Kd, Ki and Kp, and click **OK**.

This action adds Kd, Ki and Kp to the Tuned Parameters window.

📣 Tuned Parameters			
Tuned parameters	Optimization Setti Name: Value: Initial guess: Minimum:	ngs Kd 0 Kd Inf	
	Maximum: Typical value: I✓ Tuned Referenced by:	inf Ka	
Xdd] Delete	Controller		A V
			OK Cancel Help

The **Optimization Settings** area displays the following parameter settings:

- Value Current parameter value
- Initial Guess Initial parameter value
- Minimum and Maximum Parameter bounds
- Typical Value Scaling factor for the parameter

To learn more about the parameter settings, see "Changing Tuned Parameter Specifications" in the *Simulink Design Optimization User's Guide*. **4** To limit the parameters to positive values, enter the minimum value of each parameter as **0** in the corresponding **Minimum** field.

After you enter these values, the Tuned Parameters window resembles the next figure.

📣 Tuned Parameters			_ 🗆 🗵
Tuned parameters Kd Ki Kp	Optimization Settir Name: Value: Initial guess: Minimum: Maximum: Typical value:	ngs Kp 1 kp D Inf	
Add Delete	Tuned Referenced by:		×
		OK Cancel	Help

5 Click **OK** to apply the parameter settings and close the Tuned Parameters window.

Optimizing the Parameters

In this portion of the tutorial, you optimize the model parameters to meet the specified design requirements.

You must have already specified the design requirements and the parameters to optimize, as described in "Specify Time-Domain Design Requirements" on page 7-8, and "Specifying Parameters to Optimize" on page 7-18, respectively.

To optimize the model parameters:

1 In the Block Parameters window, click **Optimization > Start** to start the optimization.

This action opens the optimization progress window as shown in the next figure.

📣 Optin	nization Progress				
Iter 0	S-count 1	f(x) 0	max constraint 207.1	Step-size	First-order optimality
•					

• During each optimization iteration, the software simulates the model, and the default optimization method Gradient descent modifies the controller parameters to reduce the distance between the simulated response and the design requirement line segments. To learn more about the optimization method, see "Selecting Optimization Methods" in the Simulink Design Optimization User's Guide.

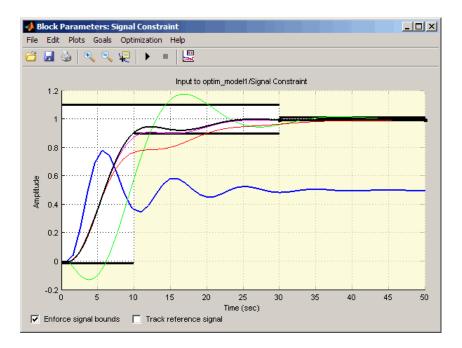
- Optimization Progress - 🗆 🗡 . max Directional First-order Iter S-count f(x) constraint Step-size derivative optimality Procedure Ω 1 0 207.1 14 0 3.767 1.25 0 infeasible 1 1 21 0 0.4641 0.541 0 6.18 2 28 -9.19412e-020 0.02942 0.0755 0.000891 3 -1.22e-018 4 35 Π Π 0.0233 3.95e-018 0.000114 Hessian modified Successful termination. Found a feasible or optimal solution within the specified tolerances. Kd = 0.0468 Ki = 0.1455 Kp = 0.1853
- After the optimization completes, the Optimization Progress window resembles the next figure.

The message Successful termination indicates that the optimization method found a solution that meets the design requirements within the parameter bounds. For more information about the outputs displayed in the optimization progress window, see "Displaying Iterative Output" in the Optimization Toolbox documentation.

The optimized parameter values are displayed in the Optimization Progress window, and also written into the Simulink model.

2 In the Block Parameters window, examine the model's response to see if the response meets the step response requirements.

The final response of the system with the optimized parameter values appears in black, as shown in the next figure.



The plot also displays the initial response of the system in blue. The remaining responses show the simulated response at each optimization iteration.

The optimized response of the system lies in the white region bounded by the design requirement line segments and thus meets the specified requirements.

Next, you refine the model parameters by specifying a reference signal that the system response must track, as described in "Refining Model Parameters to Track a Reference Signal" on page 7-25.

Refining Model Parameters to Track a Reference Signal

In this portion of the tutorial, you optimize the model parameters to track a reference signal specified by the equation

 $y = 1 - \exp(-0.3 \times t)$

You must have already optimized the model parameters to meet the step response requirements, as described in "Optimizing Model Parameters to Meet Step Response Requirements" on page 7-8.

To optimize the parameters to track a reference signal:

1 In the Block Parameters window, select **Goals > Desired Response**.

This action opens the Desired Response dialog box.

📣 Desired Response	
Specify reference signal	
C Specify step response characteristics	
Reference signal	
Time vector:	
Amplitude:	
OK Cancel Help	Apply

2 Specify the reference signal:

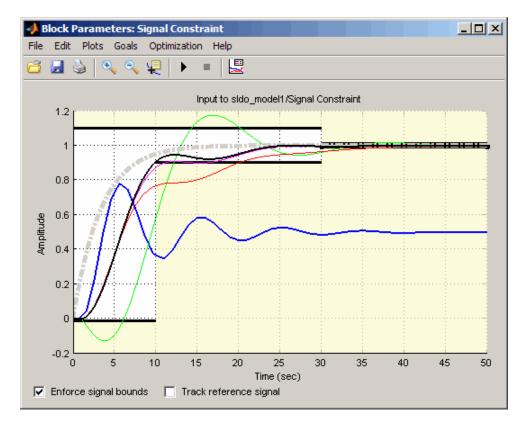
- **a** In the **Time vector** field of the Desired Response dialog box, enter linspace(0,50,200).
- **b** In the **Amplitude** field, enter 1-exp(-0.3*linspace(0,50,200)).

After you make these changes, the Desired Response dialog box resembles the following figure.

📣 Desired Response	_ 🗆 🗙
Specify reference signal	
C Specify step response characteristics	
Reference signal	
Time vector: inspace(0,50,200)	
Amplitude: 1-pxp(-0.3*linspace(0,50,200))	
OK Cancel Help	Apply

c Click **OK** to add the reference signal to the block and close the Desired Response dialog box.

The Block Parameters window updates to display the reference signal, shown as the gray dashed curve, in the next figure.



3 In the Block Parameters window, select the **Track reference signal** check box.

Enforce signal bounds 🛛 🔽 Track reference signal

4 In the Signal Constraint block window, select **Optimization > Start** to start the optimization.

This action restarts the optimization using the optimized parameters values computed in "Optimizing Model Parameters to Meet Step Response Requirements" on page 7-8. The Optimization Progress window opens, as shown in the next figure.

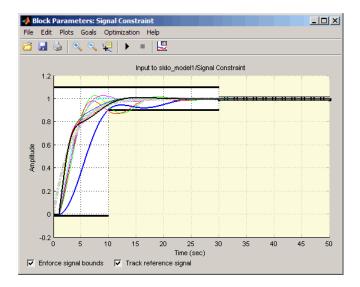
📣 Optin	nization Progress					
			max		Divertional	First-order
	S-count	£ ()	constraint			
		2.1547	Constraint	Step-Size	derivative	optimality
۲ I	1	2.1547	U			
4						Þ

- At each iteration, the optimization method modifies the controller parameters to minimize the error between the simulated response and the reference signal. Simultaneously, the method satisfies the step response requirements by reducing the distance between the simulated response and the design requirement line segments.
- After the optimization completes, the Optimization Progress window resembles the following figure.

📣 Optin	nization Progress						_ 🗆 ×
			max		Directional	First-order	<u> </u>
Iter	S-count	f(x)	constraint	Step-size	derivative	optimality	Procedure
0	1	2.1547	0				
1	20	0.561528	0.02738	0.622	-7.04	74	
2	36	0.438343	0.03301	0.142	-1.13	1.84	
3	49	0.360718	0	0.296	-0.802	3.3	
4	60	0.271571	0	0.173	-0.64	2.69	
5	71	0.178588	0	0.321	-0.468	1.76	
6	82	0.1574	0	0.147	-0.215	1.28	
7	91	0.146542	0	0.0941	-0.168	0.824	
8	100	0.139688	0	0.0443	-0.208	0.37	
9	109	0.135451	0	0.0185	-0.294	0.102	
10	116	0.133726	0	0.0256	-0.0779	0.109	
-	imize furthe on tolerance		ptimization Opt	ions and de	crease the pa	rameter and/o	r
Kd = 1.	5821						
Ki = 0.	2876						
Kp = 0.	6311						_

The message, Optimization terminated due to slow progress in parameter or objective values appears in the Optimization Progress window. This message indicates that the optimization method cannot minimize the error between the simulated response and the reference signal any further. You must examine the optimized response of the model to check if the response tracks the reference signal, as described in the next step. **5** In the Block Parameters window, examine the model's response to see how well the response tracks the reference signal.

The model's response before you began the optimization appears as the blue curve. After the optimization completes, the final response, shown as the black curve, tracks the reference signal.



Note To refine the parameters further to track the reference signal more closely, change the parameter and function tolerances, as described in "Selecting Optimization Termination Options" in the *Simulink Design Optimization User's Guide*.

Saving the Project

After you optimize the model parameters, you can save the model with the design requirements and optimized parameter values in a response optimization project.

To save a response optimization project:

1 In the Signal Constraint block window, select **File > Save As**.

This action opens the Save Project dialog box.

A Save Project
Save optimization project as:
MATLAB workspace variable:
sldo_model1_final_optim
O Model workspace variable:
sldo_model1_final_optim
O MAT-file:
sldo_model1_final_optim.mat sl
Save and reload project with Simulink model
OK Cancel Help

- **2** In the Save Project dialog box, select the **Model workspace variable** option, and enter a variable name.
- 3 Click OK to save the project.

Tip You can load the response optimization project by loading the Simulink model.

Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the Command Line

- "About This Tutorial" on page 8-2
- "Configuring a Model for Optimizing Parameters" on page 8-5
- "Optimizing Model Parameters to Meet Step Response Requirements" on page 8-8
- "Refining Model Parameters to Track a Reference Signal" on page 8-15

About This Tutorial

In this section...

"Objectives" on page 8-2

"About the Model" on page 8-2

"Design Requirements" on page 8-4

Objectives

In this tutorial, you learn how to use Simulink Design Optimization functions to optimize the parameters of a Simulink model to meet time-domain design requirements.

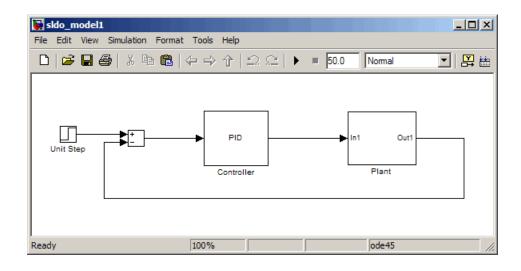
You accomplish the following tasks using the command line:

- Specify step response requirements on the model's output.
- Specify a reference signal for the model's output to track.
- Optimize the parameters to meet the design requirements.

Tip To learn how to optimize model parameters to meet time-domain design requirements using the GUI, see Chapter 7, "Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the GUI".

About the Model

In this tutorial, you use the Simulink model named sldo_model1, as shown in the next figure.

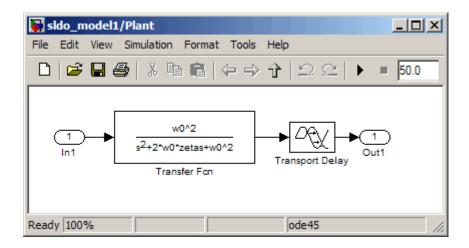


Tip To open the model, type sldo_model1 at the MATLAB command prompt.

The model contains a Controller block, which is a PID controller. This block controls the output of the Plant subsystem. The Unit Step block applies a step input to the system.

Note This tutorial uses a step input to produce the model's response and optimize the model parameters to meet step response requirements. You can also use other types of inputs, such as ramp, and optimize parameters to meet requirements on the model's response to these inputs.

Double-click the Plant subsystem to open it. The plant is modeled as a second-order system with delay. It contains Transfer Function and Transport Delay blocks, as shown in the next figure.



To learn more about the blocks, see Transfer Fcn, and Transport Delay block reference pages.

Design Requirements

The models' output must meet the following design requirements:

- Overshoot less than 10%
- Rise time less than 10 seconds
- Settling time less than 30 seconds
- Track a reference signal specified by $y = 1 \exp(-0.3 \times t)$, where t is time

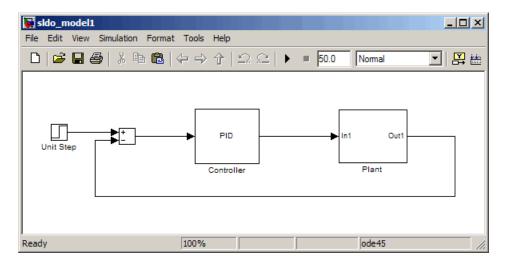
Configuring a Model for Optimizing Parameters

To optimize parameters of a Simulink model, you must first configure the model:

1 Open the sldo_model1 model, if it is not already open, by typing the model name at the MATLAB prompt:

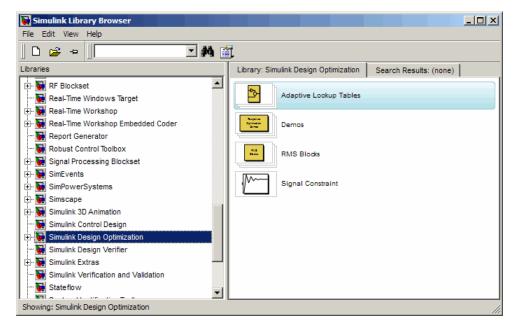
sldo_model1

The Simulink model opens, as shown in the next figure.



To learn more about the model, see "About the Model" on page 8-2.

- **2** Add a Signal Constraint block to the model.
 - **a** In the Simulink model, select **View > Library Browser** to open the Simulink Library Browser.



b In the Libraries pane, select Simulink Design Optimization.

c Drag and drop the Signal Constraint block into the Simulink model window.

To learn more about the block, see Signal Constraint block reference page.

- 🙀 sldo_model1 * - D × File Edit View Simulation Format Tools Help 💽 🔛 📇 👔 🍪 🔛 🖡 🗋 😂 🖬 🚳 👗 🖻 🛍 (수 수 수 | 으 으) ► = 50.0 Normal tĒ PID Out1 Unit Step Plant Controller Signal Constraint Ready 100% ode45
- **d** Connect the Signal Constraint block to the model's output, as shown in the next figure.

Note You must connect the Signal Constraint block to the signal to which you want to add design requirements. To learn more, see "Constraining Model Signals" in the *Simulink Design Optimization User's Guide*.

Optimizing Model Parameters to Meet Step Response Requirements

In this portion of the tutorial, you use Simulink Design Optimization commands to optimize the model parameters to meet step response requirements.

You must have already configured your model for optimization, as described in "Configuring a Model for Optimizing Parameters" on page 8-5.

To optimize the model parameters to meet step response requirements:

1 Create a new project using the newsro command.

```
proj=newsro('sldo_model1',{'Kd','Ki','Kp'})
```

This command creates a new project object, proj, that contains instructions to optimize the parameters Kp, Ki, and Kd.

```
Name: 'sldo_model1'
Parameters: [3x1 ResponseOptimizer.Parameter]
OptimOptions: [1x1 sroengine.OptimOptions]
Tests: [1x1 ResponseOptimizer.SimTest]
Model: 'sldo_model1'
```

Response Optimization Project.

2 Retrieve the signal constraint object for the project, proj, using the findconstr command.

```
constr=findconstr(proj,'sldo_model1/Signal Constraint')
```

This command retrieves the following signal constraint object:

```
ConstrEnable: 'on'
isFeasible: 1
CostEnable: 'off'
Enable: 'on'
Name: 'Signal Constraint'
SignalSize: [1 1]
LowerBoundX: [3x2 double]
```

```
LowerBoundY: [3x2 double]
LowerBoundWeight: [3x1 double]
UpperBoundX: [2x2 double]
UpperBoundY: [2x2 double]
UpperBoundWeight: [2x1 double]
ReferenceX: []
ReferenceY: []
ReferenceWeight: []
```

Signal Constraint.

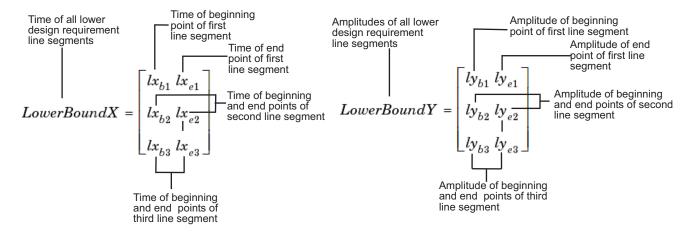
The LowerBoundX, LowerBoundY, UpperBoundX, and UpperBoundY properties of the constr object represent the numeric values of design requirements on a signal.

Tip You can view the default design requirements by double-clicking the Signal Constraint block in the sldo_model1 Simulink model.

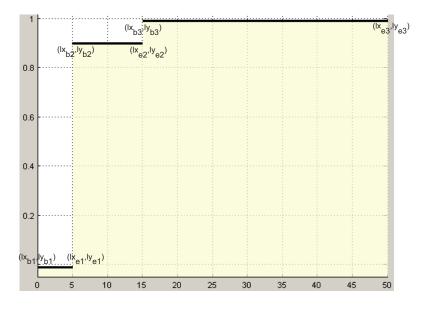
The LowerBoundX, and LowerBoundY properties define the *lower bound*, and the UpperBoundX and UpperBoundY define the *upper bound* on the signal. These properties are matrices, whose elements represent the values of the bounds as shown in the following table.

Property Name	Purpose
LowerBoundX	Time values of the lower-bound line segments
LowerBoundY	Amplitude values of the lower-bound line segments
UpperBoundX	Time values of the upper-bound line segments
UpperBoundY	Amplitude values of the upper-bound line segments

For example, the elements of the LowerBoundX, and LowerBoundY matrices map to the following values.



The line segments that correspond to the elements of LowerBoundX, and LowerBoundY matrices appear in the next figure.



Similarly, the elements of UpperBoundX and UpperBoundY map to the time and amplitude values of the upper-bound line segments, respectively. To specify the step response requirements, as described in "Design Requirements" on page 8-4, you must edit these properties as shown in the next step.

Note When you add or modify design requirements using functions, the Block Parameters window does not update to display the new requirements.

To view the values of the LowerBoundY matrix, type the following command:

constr.LowerBoundY

3 Specify the step response requirements using the following commands:

```
% Rise time requirement
constr.LowerBoundX=[0 10;10 15;15 50];
% Settling time requirement on lower line segments;
constr.LowerBoundX=[0 10;10 30;30 50];
% Settling time requirement on upper line segments
constr.UpperBoundX=[0 30;30 50];
% Overshoot requirement on upper bound
constr.UpperBoundY=[1.1 1.1;1.01 1.01];
```

Tip You can type constr.*PropertyName* to view the updated values of the properties.

- **4** Specify the parameters to optimize:
 - a Retrieve the vector of parameter objects from the project object, proj, using the findpar command.

```
param(1)=findpar(proj,'Kd');
param(2)=findpar(proj,'Ki');
param(3)=findpar(proj,'Kp');
```

b To view the properties of a parameter object, type the object name. For example:

```
param(1)
```

This command returns the properties of param(1) object.

```
Name: 'Kd'
Value: O
InitialGuess: O
Minimum: -Inf
Maximum: Inf
TypicalValue: O
ReferencedBy: {Ox1 cell}
Description: ''
Tuned: 1
```

Tuned parameter.

• To limit the parameter to positive values, enter the minimum value for each parameter as 0 using the following commands:

```
param(1).Minimum=0;
param(2).Minimum=0;
param(3).Minimum=0;
```

Tip To view the updated value of the Minimum property for Kd, type param(1).Minimum.

5 Optimize the parameters using the optimize command:

```
opt=optimize(proj)
```

This command returns the following results:

			max		Directional	First-order	
Iter	S-count	f(x)	constraint	Step-size	derivative	optimality	Procedure
0	1	0	207.1				
1	14	0	3.767	1.25	0	1	infeasible
2	21	0	0.4641	0.541	0	6.18	
3	28	-9.19412e-020	0.02942	0.0755	-1.22e-018	0.000891	
4	35	0	0	0.0233	3.95e-018	0.000114	Hessian modified

```
Successful termination.

Found a feasible or optimal solution within the specified tolerances.

Kd =

0.0468

Ki =

0.1455

Kp =

0.1853

ans =

Cost: 0

X: [4x1 double]

ExitFlag: 1

Iteration: 4
```

- At each optimization iteration, the software simulates the model and the default optimization method Gradient descent modifies the controller parameters to reduce the distance between the simulated response and the design requirements. To learn more about the optimization method, see "Selecting Optimization Methods" in the *Simulink Design Optimization User's Guide*.
- After the optimization completes, you see the messages Successful termination, and ExitFlag:1 at the MATLAB prompt. These messages indicate that the optimization method found a solution that meets the design requirements within the parameter bounds. For more information about the outputs displayed during the optimization, see "Displaying Iterative Output" in the Optimization Toolbox documentation.

The optimized values of the parameters, Kp, Ki and Kd, are displayed at the command line, and are also written into the Simulink model.

Next, you refine the parameters by specifying a reference signal, as described in "Refining Model Parameters to Track a Reference Signal" on page 8-15.

Refining Model Parameters to Track a Reference Signal

In this portion of the tutorial, you optimize the model parameters to track a reference signal specified by the equation

 $y = 1 - \exp(-0.3 \times t)$

You must have already optimized the model parameters to meet the step response requirements, as described in "Optimizing Model Parameters to Meet Step Response Requirements" on page 8-8.

To optimize the parameters to track a reference signal:

1 Specify the reference signal to track using the following commands:

```
% Enable reference tracking
constr.CostEnable='on';
% Specify the time values of the reference signal
constr.ReferenceX=linspace(0,50,200)';
% Specify the amplitude values of the reference signal
constr.ReferenceY=1-exp(-0.3*linspace(0,50,200))';
```

Tip To verify that the reference signal has been added to the signal constraint object, type constr=findconstr(proj,'sldo_model1/Signal Constraint'). The ReferenceX and ReferenceY properties now contain the reference signal.

- 2 Initialize the parameters in proj for optimization.
 - **a** Set the initial guess of the parameters using the initpar command.

```
% Copy parameter values from Simulink model to the project
initpar(proj)
```

b Update the scaling factor of the parameters with the initial parameter values using the following commands.

% Update scaling factor with the initial parameter value param(1).TypicalValue = param(1).InitialGuess;

```
param(2).TypicalValue = param(2).InitialGuess;
param(3).TypicalValue = param(3).InitialGuess;
```

To learn more about the scaling factor, see "Changing Tuned Parameter Specifications" in the *Simulink Design Optimization User's Guide*.

When you optimize the model parameters after this initialization, the optimization method uses the updated parameter settings to recompute new parameter values.

3 Start the optimization using the optimize command.

```
opt=optimize(proj)
```

This command returns the following results:

			max		Directional	First-order	
Iter	S-count	f(x)	constraint	Step-size	derivative	optimality	Procedure
0	1	2.1547	0				
1	20	0.561528	0.02738	0.622	-7.04	74	
2	36	0.438343	0.03301	0.142	-1.13	1.84	
3	49	0.360718	0	0.296	-0.802	3.3	
4	60	0.271571	0	0.173	-0.64	2.69	
5	71	0.178588	0	0.321	-0.468	1.76	
6	82	0.1574	0	0.147	-0.215	1.28	
7	91	0.146542	0	0.0941	-0.168	0.824	
8	100	0.139688	0	0.0443	-0.208	0.37	
9	109	0.135451	0	0.0185	-0.294	0.102	
10	116	0.133726	0	0.0256	-0.0779	0.109	

Optimization terminated due to slow progress in parameter or objective values. To optimize further, go to Optimization Options and decrease the parameter and/or function tolerances.

Kd = 1.5821 Ki =

0.2876

Kp =

```
0.6311
ans =
Cost: 0.1337
X: [4x1 double]
ExitFlag: 5
Iteration: 10
```

- At each iteration, the optimization method modifies the controller parameters to minimize the error between the simulated response and the reference signal. Simultaneously, the method satisfies the step response requirements by reducing the distance between the simulated response and the design requirement line segments.
- After the optimization completes, you see the message Optimization terminated due to slow progress in parameter or objective values. This message indicates that the optimization method cannot minimize the error between the simulated response and the reference signal any further. If you want to refine the parameters further to track the reference signal more closely, change the parameter and function tolerances, as described in "Selecting Optimization Termination Options" in the Simulink Design Optimization User's Guide.

Tip To view the updated response of the model, connect a **Scope** block to the model and simulate the model to view the response.

4 Save the project by typing the following command:

```
save('proj');
```

Tutorial — Designing a PID Controller Using Optimization-Based Tuning

- "About This Tutorial" on page 9-2
- "Configuring a Project for Optimization-Based Control Design" on page 9-5
- "Designing an Initial PID Controller to Meet Bode Magnitude and Phase Margins Requirements" on page 9-11
- "Refining the Controller Design to Meet Controller Output Bounds" on page 9-32
- "Saving the Project" on page 9-48

About This Tutorial

In this section ...

"Objectives" on page 9-2

"About the Model" on page 9-2

"Design Requirements" on page 9-4

Objectives

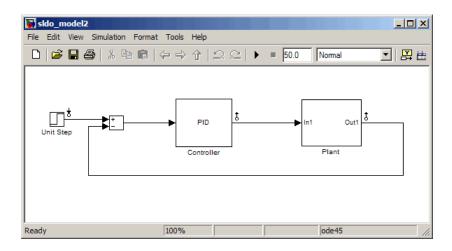
In this tutorial, you learn to use optimization methods to design a PID controller to meet frequency-domain design requirements on a system's response.

You accomplish the following tasks using the GUI:

- Specify frequency-domain Bode magnitude and phase margin requirements.
- Design an initial controller to meet the frequency-domain requirements.
- Refine the initial controller design to limit the controller's output signal.

About the Model

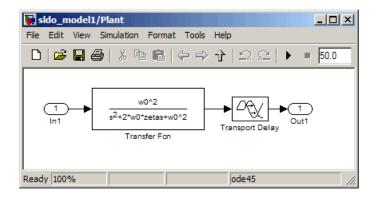
In this tutorial, you use the Simulink model named sldo_model2, as shown in the next figure.



The model contains a Controller block, which is a PID Controller. This block controls the output of the Plant subsystem.

Using the Simulink Control Design software has linearized the Simulink model at the operating point specified in the model. The sldo_model2.mat file contains a preconfigured SISO Design Tool session saved after linearizing the model. To learn more about linearizing Simulink models for control design, see "Designing Compensators" in the *Simulink Control Design User's Guide*.

Double-click the Plant subsystem to open it. The plant is modeled as a second-order system with delay. It contains Transfer Function and Transport Delay blocks, as shown in the next figure.



To learn more about the blocks, see Transfer Fcn and Transport Delay block reference pages.

Design Requirements

The compensator you design in this tutorial must meet the following design requirements:

- Bode lower magnitude bound of 0 in the frequency range 1e-3 to 1 rad/sec
- Phase margin greater than 60 degrees
- Controller output bounds in the range [-250 550]

9

Configuring a Project for Optimization-Based Control Design

To design a linear controller for a Simulink model, you must first configure a Control and Estimation Tools Manager project.

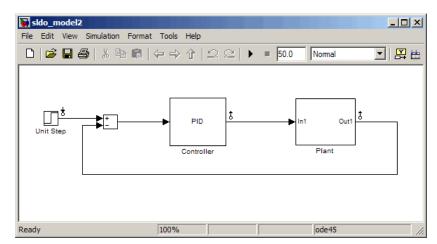
1 Open a SISO Design Tool session for the linearized Simulink model by typing the following command at the MATLAB prompt:

```
sisotool('sldo_model2.mat')
```

Note sldo_model2.mat file contains a preconfigured SISO Design Tool session. This session was saved after Simulink Control Design software linearized the sldo_model2 Simulink model.

This command opens the following windows:

• Simulink model

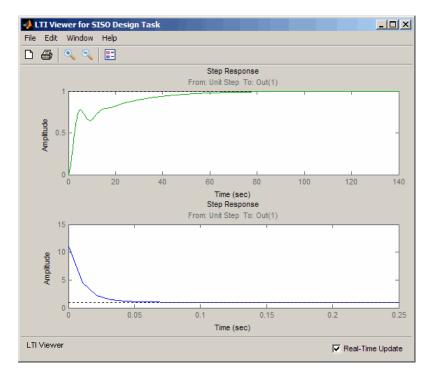


To learn more about the model, see "About the Model" on page 9-2.

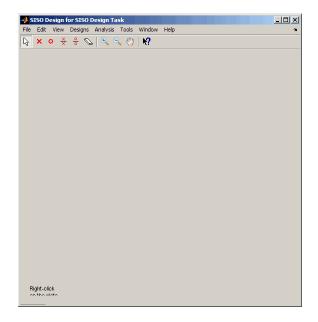
• Control and Estimation Tools Manager GUI, which contains a SISO **Design Task** node under the **Simulink Compensator Design Task** node.

Gontrol and Estimation Tools	Manager	
File Help		
🗃 🛅 🖬 🖬		
Workspace Project = sido_model2 Grant Operating Points Simulink Compensator De E SISO Design Task	- Project Settings Title: Subject: Created by: Date modified: Simulink model: Description:	Project - sldo_model2 10-Nov-2008 sldo_model2
×>		×
Project node.		

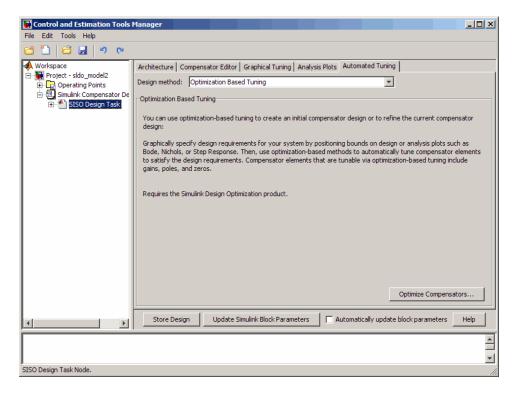
- LTI Viewer for SISO Design Task window, which contains the following plots:
 - Closed-loop step response of the system in the top plot
 - Output of the Controller block in the bottom plot



• A blank SISO Design for SISO Design Task window

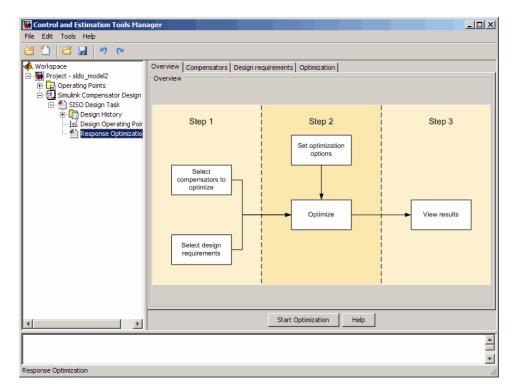


2 In the Control and Estimation Tools Manager GUI, select the Automated Tuning tab in the SISO Design Task node.



3 Click **Optimize Compensators**.

This action creates a new Response Optimization node.



Designing an Initial PID Controller to Meet Bode Magnitude and Phase Margins Requirements

In this section...

"Specifying the Controller Parameters" on page 9-11

"Specifying Bode Magnitude and Phase Margin Design Requirements" on page 9-15

"Designing the Controller" on page 9-25

Specifying the Controller Parameters

In this portion of the tutorial, you specify the controller parameters to design.

You must have already configured a project for control design, as described in "Configuring a Project for Optimization-Based Control Design" on page 9-5.

To specify the controller parameters to design:

1 In the Control and Estimation Tools Manager GUI, select the **Compensators** tab in the **Response Optimization** node.

Workspace	Overview Cor	npensators Design requirements	Optimization				
Project - sldo_model2	Select compen	sator elements to optimize					
	C Optimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical value
SISO Design		sldo_model2/Controller					
🕂 🛄 Design Hi		Gain	0.1	0.1	-Inf	Inf	0.1
🚽 📈 Design O		Real Zero	-8.999	-8.999	-Inf	Inf	-8.999
		Real Zero	-0.10102	-0.10102	-Inf	Inf	-0.10102
		Real Pole	-100	-100	-Inf	Inf	-100
				-100	-111	Δ1 H	-100
	Right click on a co	mpensator name to change its represent.		-100	117		: Initial Guess

The controller parameters appear as poles and zeros in the **Compensator** elements column and represent the following:

- Gain Overall gain of the controller
- Real zeros Zeros resulting from the differentiator and integrator
- Real pole Pole resulting from the low-pass filter of the differentiator

Tip To view the structure of the Controller block, right-click the block and select Look Under Mask.

2 For convenience, change the PID controller parameters to Simulink block mask parameters format. To do so, right-click the sldo_model2/Controller column, and select Parameterized format.

🙀 Control and Estimation Tools Man	ager							_ 🗆 🗵
File Edit Tools Help								
🥶 🛅 😂 🛃 🤊 🖻								
📣 Workspace	Overview Co	ompensators Design requirements	Opti	mization				
🖻 🙀 Project - sldo_model2	Select compe	nsator elements to optimize						
Operating Points	Opti	Compensator elements		Value	Initial gu	Minimum	Maximum	Typical
는 • • • · · · · · · · · · · · · · · · ·		sldo model2/Controller						.,,
		Gain	V P	ole/Zero for	mat	-Inf	Inf	0.1
Design Operating Poir		Real Zero	P	arameterize	d format 🔪	-Inf	Inf	-8.999
Response Optimizatio		Real Zero		-0.10102	-0.10102	-Inf	Inf	-0.10102
		Real Pole		-100	-100	-Inf	Inf	-100
	Right click on a (compensator name to change its represent			Help	Use	/alue as Initia	al Guess
		Start	opami	20001	ricip			
								^
1								v
Response Optimization								//

This action displays the controller parameters as Simulink block mask parameters P, I, and D, as shown in the next figure. To learn more about mask parameters, see "Mask Parameters" in the Simulink documentation.

Control and Estimation Tools Manager							
🗂 🗋 🖬 🔄 🔍							
📣 Workspace	Overview Comp	ensators Design requirements Optimiz	zation				
E Project - sldo_model2	Select compensat	tor elements to optimize					
🕂 🔁 Operating Points	C Optimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical value
SISO Design Task		ildo_model2/Controller					
🗈 🛅 Design History		P	1	1	-Inf	Inf	1
Design Operating Point		I	0.1	0.1	-Inf	Inf	1
Cesponse Optimization		D	0.1	0.1	-Inf	Inf	1
		N	100	100	-Inf	Inf	1
	Right click on a com	pensator name to change its representation.				Jse Value as I	nitial Guess
	Right click on a com		ptimization	Help	t	Jse Value as I	nitial Guess
4	Right click on a corr		optimization	Help	t	Jse Value as I	
<u>4</u> }	Right click on a com		ptimization	Help		Jse Value as I	
<u>د</u>	Right click on a com		ptimization	Help		Jse Value, as I	nitial Guess

The **Compensators** tab displays the following parameter settings:

- Value Current controller parameter value
- Initial Guess Initial controller parameter value
- Minimum and Maximum Controller parameter bounds
- Typical Value Scaling factor for the controller parameter

To learn more about the parameter settings, see "Changing Tuned Parameter Specifications" in the *Simulink Design Optimization User's Guide*.

le Edit Tools Help							
5 🗋 🖆 🛃 🤊 🤊							
Workspace	Overview Com	pensators Design requirements Optimiz	ation				
Project - sldo_model2	Select compens	ator elements to optimize					
⊕ - 🔁 Operating Points 는 - 톈, Simulink Compensator Design Task	C Optimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical v
SISO Design Task		sldo_model2/Controller					
🕀 🛄 Design History		P	1	1	-Inf	Inf	1
Design Operating Point		I	0.1	0.1	-Inf	Inf	1
Response Optimization		D	0.1	0.1	-Inf	Inf	1
		N	100	100	-Inf	Inf	1
	Right click on a co	npensator name to change its representation.					
	Right click on a cost	npensator name to change its representation.			U	Jse Value as Ir	nitial Guess
	Right click on a cor		ptimization	Help		Jse Value as Ir	nitial Guess
	Right click on a con		ptimization	Help		Jse Value as Ir	nitial Guess

 ${\bf 3}$ In the ${\bf Optimize}$ column, select the check boxes for P, I, and D.

Specifying Bode Magnitude and Phase Margin Design Requirements

In this portion of the tutorial, you specify the Bode magnitude and phase margin requirements that the controller must satisfy.

Before you specify the design requirements, you can specify the controller parameters to design, as described in "Specifying the Controller Parameters" on page 9-11.

To specify the Bode design requirements:

 In the Response Optimization node of the Control and Estimation Tools Manager GUI, select the Design requirements tab.

Control and Estimation Tools Manage		
File Edit Tools Help		
Sec. 19 (19 (19 (19 (19 (19 (19 (19 (19 (19		
Workspace Project - sido_model2 Operating Points SISO Design Task Design Tistory Izd Design Operating Point Response Optimization	Overview Compensators Design requirements Optimization Select design requirements to satisfy Optimize Response plot	Design requirement
	Click the 'Add new design requirement' button or right click on a plot to add a new design require Show	
	Start OptimizationHelp	
Response Optimization		

2 Click Add new design requirement.

This action opens the New Design Requirement dialog box.

🥠 New Design Requirem	ent				_ 🗆 🗙
Design requirement type:	Step	response	bounds		~
Requirement for response: Closed Loop from Unit Step to Plant					Plant 💌
Design requirement parame	eters –				
Initial value:	0	Final va	lue:		1
Step time:	0				
Rise time:	5	% Rise:			80
Settling time:	10	% Settli	ng:		1.0000
% Overshoot: 10	.0000	% Unde	rshoot:		1
-		ок	Canc	el	Help

- **3** Specify the Bode magnitude lower limit requirement:
 - **a** In the New Design Requirement dialog box, select Bode magnitude lower limit from the **Design requirement type** drop-down list.

This action updates the New Design Requirement dialog box, as shown in the next figure.

-1	New Design Requirement					_ 🗆 🗵
٥	Design requirement type:			Bode magni	tude lower limi	t 💌
R	Requirement for response:			Loop fro	m Unit Step to	Controller 💌
Г	Design require	ement parame	ters			
1	Segments:					
	Sta	art	Er	nd		
	Frequency	Magnitude	Frequency	Magnitude	Slope	Weight
	1	0	10	0	0	1
	Insert Delete					
				ОК	Cancel	Help

- **b** Select Open Loop at outport 1 of Controller from the **Requirement for response** drop-down list.
- c In the Frequency field of the Start column, enter 1e-3.

d In the **Frequency** field of the **End** column, enter 1.

The New Design Requirement dialog box resembles the following figure.

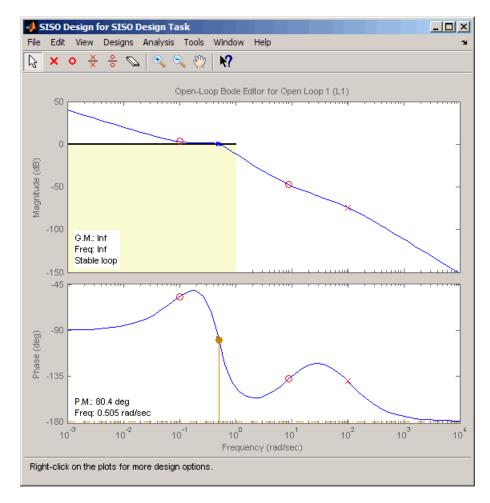
-	New Design Requirement					
0	Design requirement type:			Bode magnitude lower limit		
F	Requirement for response:			en Loop a	at outport 1 of	Controller 💌
Г	Design require	ement parame	ters			
	Segments:					
	Sta	art	Er	nd		
	Frequency	Magnitude	Frequency	Magnitude	Slope	Weight
	1.0000e-03	0	1	0	0	1
	Insert Delete					Delete
				ОК	Cancel	Help

e Click OK to close the New Design Requirement dialog box.

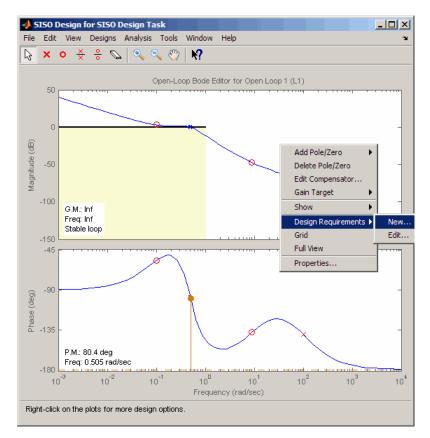
The Bode lower magnitude limit is added to the **Design requirements** tab, as shown in the next figure.

Control and Estimation Tools I	lanager				
File Edit Tools Help					
🗂 🗋 🔂 🚽 🤊 💌					
📣 Workspace	Overview Compensators Design requirements Optimization				
🖻 🙀 Project - sldo_model2	Select design requirements to satisfy				
Operating Points Simulink Compensator De	Coptimize Response plot	Design requirement			
E SISO Design Task	Open Loop at outport 1 of Controller				
🕀 🛄 Design History	Open-Loop Bode Editor for Open Loop 1 (L1)	Lower gain limit from 0.001 to 1 rad/sec			
🖂 🖂 Design Operating					
Response Optimi					
) Click the 'Add new design requirement' button or right click on a plot to add a new desig	n requirement.			
	Show Plo	ts Add new design requirement			
	Start Optimization Help				
		•			
Undo		h			

The SISO Design for SISO Design task window also updates to show the Bode plot with the design requirement displayed as the black line segment.



- **4** Add the phase margin requirement:
 - **a** In the SISO Design window, right-click the white area on the top plot, and select **Design requirement > New**.



This action opens the New Design Requirement dialog box, as shown in the next figure.

📣 New Design Requirement							
Design requirement type: Upper gain limit							
-Design requi	Design requirement parameters						
Segments:							
S	tart	Er	nd				
Frequency	Magnitude	Frequency	Magnitude	Slope	Weight		
1	0	10	0	0	1		
1							
	Insert Delete						
				1			
			OK	Cancel	Help		

b In the New Design Requirement dialog box, select Gain & phase margins from the **Design requirement type** drop-down list.

The New Design Requirement dialog box updates to display the **Gain Margin >** and **Phase Margin >** options, as shown in the next figure.

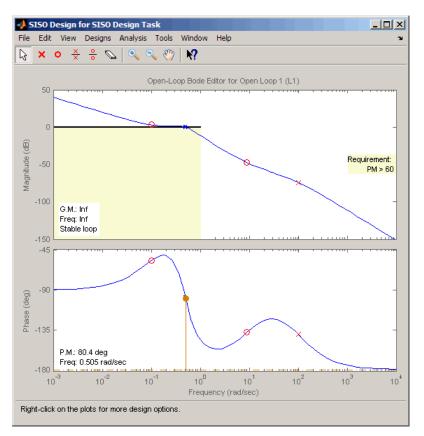
📣 New Design Requirement 📃 🔲 🗙					
Design requirement type:in & Phase margins					
Design requirement p	arameters –				
Gain margin >			20		
Phase margin >			30		
OK	Cano	el He	lp		

• Select the **Phase margin** > check box, and enter 60 in the adjacent field. Then, click **OK** to close the dialog box.

📣 New Design Requirement 📃 🔲 🗙					
Design requirement type:in & Phase margins 💌					
Design requirement pa	arameters				
Gain margin >	20				
Phase margin >	60				
ОК	Cancel Help				

The **Design requirements** tab in the Control and Estimation Tools Manager GUI updates. It now displays the phase margin requirement, as shown in the next figure.

Sontrol and Estimation Tools	1anager	
File Edit Tools Help		
🖆 🗋 🖨 🛃 🔊 🍋		
Workspace	Overview Compensators Design requirements Optimization	
🖻 💽 Project - sldo_model2	Select design requirements to satisfy	
Operating Points Image: Simulink Compensator Deligible	Coptimize Response plot	Design requirement
SISO Design Task	Open Loop at outport 1 of Controller	
🕂 📆 Design History	Open-Loop Bode Editor for Open Loop 1 (L1)	Lower gain limit from 0.001 to 1 rad/sec
🔤 🔛 Design Operating	Open-Loop Bode Editor for Open Loop 1 (L1)	Requirement: PM > 60
	Click the 'Add new design requirement' button or right click on a plot to add a new desig	
	Show Plo	ts Add new design requirement
•	Start Optimization Help	>
		-
1		
		/



The SISO Design window also updates to display the phase margin requirement.

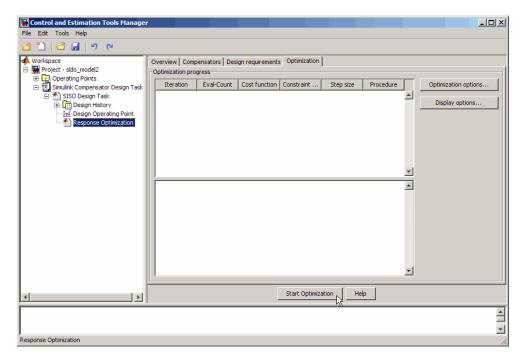
Designing the Controller

In this portion of the tutorial, you design the controller to meet the Bode magnitude and phase requirements.

You must have already specified the controller parameters to design, as described in "Specifying the Controller Parameters" on page 9-11, and the design requirements, as described in "Specifying Bode Magnitude and Phase Margin Design Requirements" on page 9-15.

To design the controller:

1 In the **Optimization** tab, click **Start Optimization**.



The **Optimization** tab updates as shown in the next figure.

Control and Estimation Tools	Manager
File Edit Tools Help	
😅 🛅 😂 🛃 🤊 🍋	
Workspace Project - sldo_model2 Simulink Compensator De SISO Design Task SISO Design Task Company SISO Design History Response Optimi	Overview Compensators Design requirements Optimization Optimization progress Iteration Eval-Count Cost funct Constrain Step size Procedure Optimization options 0 7 0 57.12 Image: Comparison options Display options 0 7 0 57.12 Image: Comparison options Display options 0 7 0 11:44:04 Image: Comparison option started 12:Nov 2008 11:44:04 Image: Comparison option started 12:Nov 2008 11:44:04
	Stop Optimization Help
Response Optimization	

- At every optimization iteration, the default optimization method Gradient descent reduces the distance between the current response and the magnitude requirement line segment by modifying the controller parameters. Simultaneously, the software also computes the phase margin and reduces the distance between the current response and the phase margin. To learn more about the optimization method, see "Selecting Optimization Methods" in the *Simulink Design Optimization User's Guide*.
- After the optimization completes, the **Optimization** tab displays the optimization iterations and status, as shown in the next figure.

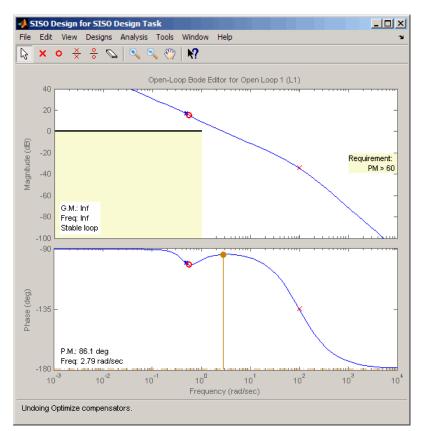
1 🗋 🗗 🖬 🤊 🤊								
Workspace	Overview Com		esign requirem	ents Optimiza	ition			
Operating Points Simulink Compensator	Iteration	Eval-Count	Cost function	Constraint	Step size	Procedure		Optimization options
E SISO Design Task	0	7	0	57.12				
E Design History	1	14	0	1.777	8.36			Display options
🚽 Design Opera	2	21	0	1.697	12.1	Hessian m		
Response Opl	3	28	0	0	21.7			
	Successful ter tolerances.	mination. Fou	nd a feasible o	r optimal solutic	on within the s	pecified	T	
				Start Optimiz		Help		

The message Successful termination indicates that the optimization method found a solution that meets the design requirements. For more information about the outputs displayed in the **Optimization progress** table, see "Displaying Iterative Output" in the Optimization Toolbox documentation.

- **2** Examine the controller parameters and the system's response:
 - **a** View the optimized parameter values in the **Value** field of the **Compensator** tab.

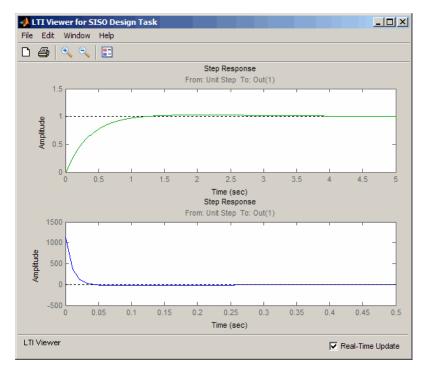
b Examine the system's response on the following plots:





- The top plot shows that the magnitude of the system, displayed as the blue curve, lies outside the yellow region. This plot indicates that the system has met the Bode magnitude requirement.
- The bottom plot displays the phase margin (P.M.) value of 86.1 degrees. This indicates that the system has met the phase margin design requirement of >60 degrees.





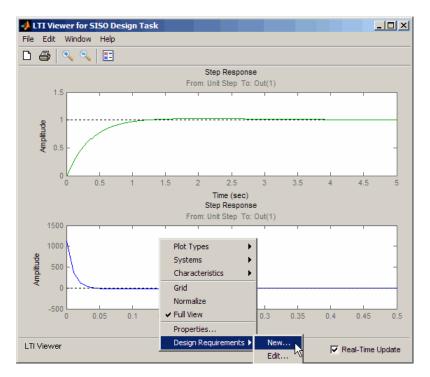
- The top plot shows that the closed-loop response of the system is stable. The system with the designed controller thus meets both the magnitude and phase margin requirements.
- The bottom plot in the LTI Viewer shows that the peak value of the controller's output is 1000, which is very large and can cause damage to the plant. To limit the controller output, you apply lower and upper bounds on the signal, as described in "Refining the Controller Design to Meet Controller Output Bounds" on page 9-32.

Refining the Controller Design to Meet Controller Output Bounds

In this portion of the tutorial, you refine the controller to satisfy bounds on the controller's output.

You must have already designed an initial controller, as described in "Designing an Initial PID Controller to Meet Bode Magnitude and Phase Margins Requirements" on page 9-11. To tune the compensator parameters to meet the bounds on the controller's signal:

- **1** Add the upper-bound on the controller's output:
 - **a** In the LTI Viewer, right-click the white area on the bottom plot, and select **Design requirement > New**.



This action opens the New Design Requirement dialog box.

📣 New Design Requi	rement						
Design requirement type	Design requirement type: Step response bounds						
Design requirement par	ameters						
Initial value:	0 Final value:	1					
Step time:	0						
Rise time:	5 % Rise:	80					
Settling time:	10 % Settling:	1.0000					
% Overshoot: 10.0	000 % Undershoot:	1					
	OK Cancel	Help					

b In the New Design Requirement dialog box, select Upper time response bound from the **Design requirement type** drop-down list.

	New Design Requirement							
	Design requirement type: Upper time response bound							
Г	Design requirement parameters							
	Segments:							
	Sta	art	Er	nd				
	Time	Amplitude	Time Amplitude		Slope	Weight		
	0	1	10	1	0	1		
	Insert Delete							
				ОК	Cancel	Help		

- c In the Time field of the End column, enter Inf.
- d In the Amplitude field of the Start column, enter 550.

e In the Amplitude field of the End column, enter 550.

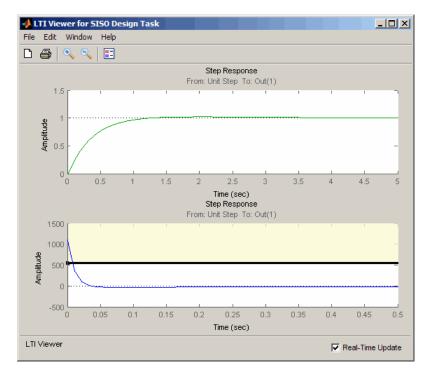
The New Design Requirements dialog box resembles the following figure.

	New Design Requirement						
Desig	n require	ement type: Up	per time resp	onse bound		T	
-		ement parame	ters				
Segme							
	Sta	art	Er	nd			
Т	ïme	Amplitude	Time	Amplitude	Slope	Weight	
	0	550	Inf	550	0	1	
P	InsertDelete						
					Insert	Delete	

 $f \quad {\rm Click} \ OK \ {\rm to} \ {\rm close} \ {\rm the} \ {\rm dialog} \ {\rm box}.$

The **Design requirements** tab in the Control and Estimation Tools Manager GUI updates. It now displays the upper-bound requirement, as shown in the next figure.

Control and Estimation	Tools Manager		
🖆 🛍 🖆 🔙 🤊	(°		
Workspace	Overview Com	pensators Design requirements Optimization	
🖻 💽 Project - sldo_model2	-Select design re	quirements to satisfy	
Operating Points Simulink Compens	Optimize	Response plot	Design requirement
SISO Design		Closed Loop from Unit Step to Controller	
🕀 🛄 Design Hi		Step Response	Upper time response bound from 0 to 10 sec
- 🛃 Design O		Open Loop at outport 1 of Controller	
esponse	V	Open-Loop Bode Editor for Open Loop 1 (L1)	Lower gain limit from 0.001 to 1 rad/sec
		Open-Loop Bode Editor for Open Loop 1 (L1)	Requirement: PM > 60
	Click the 'Add new	design requirement' button or right click on a plot to add a new design rec	uirement.
		Show Pl	ots Add new design requirement
		Start Optimization Help	
		Linear opening the property of	
			<u> </u>
			•



The LTI Viewer also updates to show the design requirement, as shown in the next figure.

- **2** Add the lower-bound on the controller's output:
 - a Right-click the white area in the bottom plot in the LTI Viewer, and select Design requirement > New.

This action opens the New Design Requirement dialog box.

b Select Lower time response bound from the Design requirement type drop-down list.

The New Design Requirement dialog box updates to show the lower bound, as shown in the next figure.

🥠 New Design Requirement							
Design requirement type: Lower time response bound							
Design requirement parameters							
Segments:							
Start End							
Time	Amplitude	Time	Time Amplitude		Weight		
0	1	10	1	0	1		
Insert Delete							
			ОК	Cancel	Help		
	Design require Design require Segments: Sta Time	Design requirement type: Design requirement parame Segments: Start Time Amplitude	Design requirement type: Lower time resp Design requirement parameters Segments: Start En Time Amplitude Time	Design requirement type: Lower time response bound Design requirement parameters Segments: Start End Time Amplitude Time Amplitude	Design requirement type: Lower time response bound Design requirement parameters Segments: Start End Time Amplitude 0 1 10 1 0 1		

- c In the Time field of the End column, enter Inf.
- d In the Amplitude field of the Start column, enter -250.

e In the Amplitude field of the End column, enter -250.

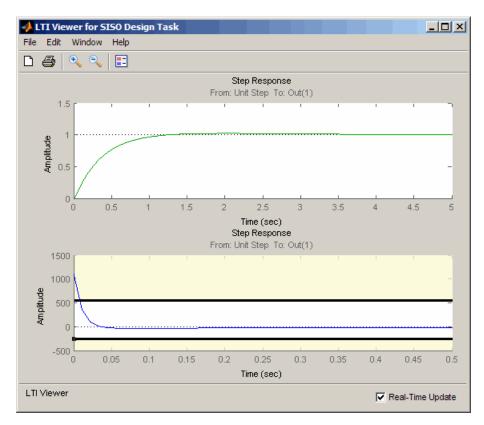
The New Design Requirement dialog box resembles the next figure. Click **OK** to close the dialog box.

📣 New Design Requirement							
Design requirement type: Lower time response bound							
-Design require	ment parame	ters					
Segments:							
Sta	art						
Time	Amplitude	Time Amplitude		Slope	Weight		
0	-250	Inf	-250	0	1		
J							
	Insert Delete						
<u> </u>							
			ОК	Cancel	Help		

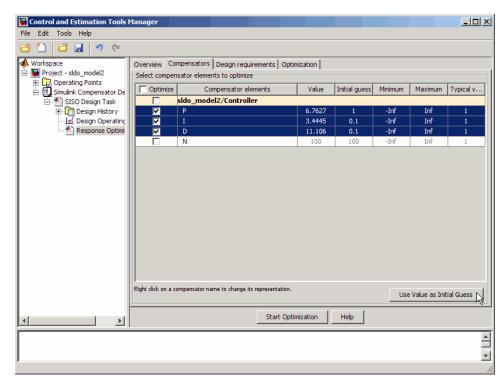
The **Design requirements** tab in the Control and Estimation Tools Manager GUI updates. It now displays the lower-limit requirement, as shown in the next figure.

Control and Estimation	Tools Manager						
File Edit Tools Help							
🖆 🗋 🖨 🔒 🔊	3						
Workspace		pensators Design requirements Optimization					
Project - sldo_model2	Select design re	quirements to satisfy					
Operating Points	Optimize	Response plot	Design requirement				
SISO Design		Closed Loop from Unit Step to Controller					
🕕 🛄 Design Hi	V	Step Response	Upper time response bound from 0 to 10 sec				
🖂 Design O		Step Response	Lower time response bound from 0 to 10 sec				
Response		Open Loop at outport 1 of Controller					
	V	Open-Loop Bode Editor for Open Loop 1 (L1)	Lower gain limit from 0.001 to 1 rad/sec				
		Open-Loop Bode Editor for Open Loop 1 (L1)	Requirement: PM > 60				
I							
	Click the 'Add new	design requirement' button or right click on a plot to add a new design rec	uirement.				
		Show Pl	ots Add new design requirement				
			Add new design requirement				
		Start Optimization Help	1				
		Start Optimization Help					
J			•				
			///				

The LTI Viewer also updates to show the lower-bound on the controller's output, as shown in the next figure.



- **3** Optimize the parameters to meet the design requirements on the controller output:
 - **a** In the **Compensators** tab, select the rows containing P, I, and D, and click **Use Value as Initial Guess**.



When you run the optimization again, the optimization method uses the updated parameter values as the starting point for refining the values.

Clicking **Use Value as Initial Guess** updates the values in the **Initial Guess** column, as shown in the next figure.

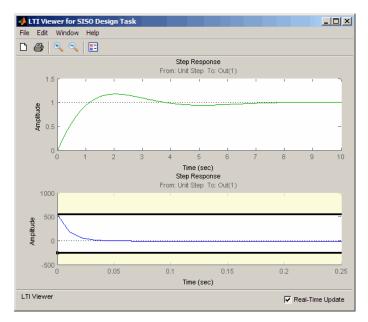
🙀 Control and Estimation Tools I	Manager						_ 🗆 ×
File Edit Tools Help							
😅 🗋 😂 🛃 🔊 💌							
📣 Workspace	Overview Cor	npensators Design requirements Optin	nization				
🖻 💽 Project - sldo_model2	Select compen	sator elements to optimize					
Derating Points	Optimize	Compensator elements	Value	Initial guess	Minimum	Maximum	Typical v
SISO Design Task		sldo_model2/Controller		·			
🕀 📺 Design History		P	6.7627	6.7627	-Inf	Inf	1
		I	3.4445	3.4445	-Inf	Inf	1
Response Optimi	V	D	11.106	11.106	-Inf	Inf	1
		N	100	100	-Inf	Inf	1
	Right click on a co	mpensator name to change its representation.			Use	: Value as Init	ial Guess
		Start Optir	mization	Help			
							▲
Response Optimization							

- **b** In the **Optimization** tab, click **Start Optimization**.
 - At every optimization iteration, the optimization method reduces the distance between the current response and the upper and lower bounds on the signal.
 - After the optimization completes, the **Optimization** tab resembles the next figure.

Control and Estimation Tools	Manager						
File Edit Tools Help							
🗂 🗋 🖬 🔊 🤊							
Workspace Project - sldo_model2 Operating Points Simulink Compensator De SISO Design Task Design Pistory E Design Operating Response Optimi	Overview Compensators D Optimization progress Iteration Eval-Count 0 14 1 28 Constructing optimization pdf Constructing optimization pdf Constructing optimization pdf	Cost funct 0 0	Constrain 1.031 2.914e-014	Step size 5.67	Procedure		Optimization options
	Optimization faither of the second se	-2008 13:29:2 v-2008 13:29:	27		specified Help	_	Ŀş
						_	
							_

9

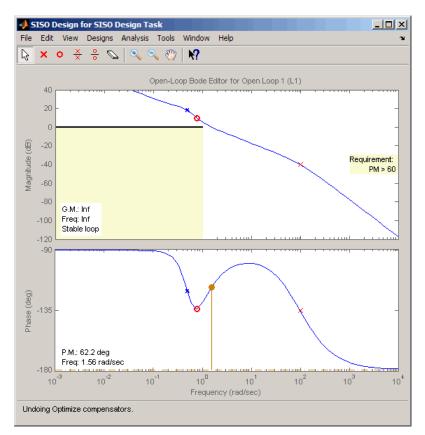
- 4 Examine the parameter values refined controller and the system's response:
 - **a** Examine the following response plots:
 - The LTI viewer



The bottom plot shows that the controller output lies between 550 and -250, and thus meets the design requirement on the controller's output.

Note You must also check that the closed-loop response, shown in the top plot, remains stable after refining the controller design.





The plots show that after refining the design, the system continues to meet the magnitude and phase margin requirements specified in "Design Requirements" on page 9-4.

9

- 🙀 Control and Estimation Tools Manage _ U × File Edit Tools Help 🖆 🗋 🖬 🔚 🔊 (° 📣 Workspace Overview Compensators Design requirements Optimization 🖻 🙀 Project - sldo_model2 Select compensator elements to optimize Operating Points
 Simulink Compensator De C Optimize Initial guess Minimum Compensator elements Value Maximum Typical v.. sldo_model2/Controller 🖃 🎦 SISO Design Task 🗄 🛄 Design History ~ P 6.706 6.7627 -Inf Inf √ 🛃 Design Operating I 3.4445 3.4445 -Inf Inf 1 1 🕙 Response Optimi D 5.4329 11.106 -Inf Inf 1 П Ν -Inf 100 Right click on a compensator name to change its representation. Start Optimization Help Þ -
- b View the optimized controller parameter values in the Value field in the Compensators tab.

5 Select the SISO Design Task node, and click Update Simulink Block Parameters.

This action writes the optimized controller parameter values to the Controller block in the Simulink model.

Saving the Project

To save a project with the optimized controller parameters:

1 In Control and Estimation Tools Manager GUI, select File > Save.

This action opens the Save Projects dialog box.

📣 Save Projects	
Projects:	
Project - sldo_model2	
	-
, 	
OK Cancel	Help

- 2 In the Save Projects dialog box, select **Project sldo_model2**, and click **OK** to open the Save Projects window .
- 3 In the Save Projects window, enter sldo_model2_optimized.mat in the File name field, and click Save.

The action saves the project as a MAT-file.

Tip You can reload this project by typing sisotool('sldo_model2_optimized') at the MATLAB prompt.

10

Tutorial — Modeling a System Using Adaptive Lookup Table

- "About This Tutorial" on page 10-2
- "Building a Model Using Adaptive Lookup Table Blocks" on page 10-4
- "Adapting the Lookup Table Values Using Time-Varying I/O Data" on page 10-16

About This Tutorial

In this section...

"Objectives" on page $10\mathchar`2$

"About the Data" on page 10-2

"Overview of Modeling a System Using Adaptive Lookup Table" on page 10-3

Objectives

In this tutorial, 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.
- Disable the adaptation process and use the adaptive lookup table as a static lookup table.

About the Data

In this tutorial, you use the data in vedata.mat which contains the following variables measured from the 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" in the Simulink documentation.

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.

Overview of Modeling a System Using Adaptive Lookup Table

The process for modeling a system using adaptive lookup table consists of the following tasks:

- "Building a Model Using Adaptive Lookup Table Blocks" on page 10-4.
- "Adapting the Lookup Table Values Using Time-Varying I/O Data" on page 10-16.

Simulink Design Optimization software provides blocks for modeling systems as adaptive lookup tables. 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. To learn more about adaptive lookup tables, see "Adaptive Lookup Tables".

Building a Model Using Adaptive Lookup Table Blocks

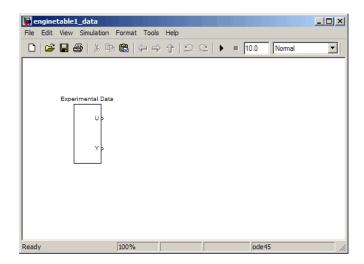
In this portion of the tutorial, you learn how to build a model of an engine using an Adaptive Lookup Table block.

1 Open a preconfigured Simulink model by typing the model name at the MATLAB prompt:

enginetable1_data

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 10-2.

The Experimental Data subsystem in the Simulink model generates the time-varying I/O data during simulation.

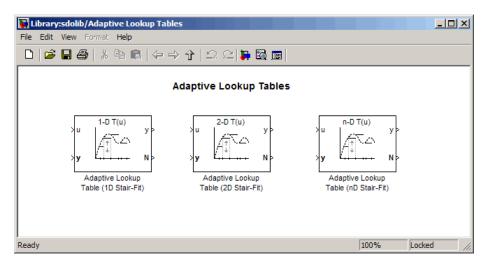


2 Open the Simulink Design Optimization library by typing the following command at the MATLAB prompt:

sdolib

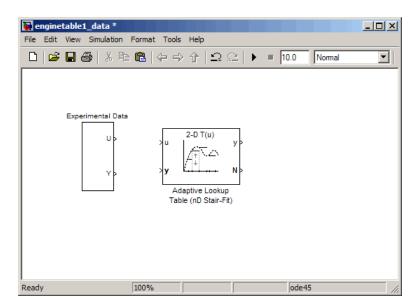
🙀 Library:sdolib			
File Edit View Format Help			
🗋 🗅 📂 🖬 🚭 🕺 🖧 🖻 🕞 🛛	수 수 수 으 으 5		
	Simulink Design	Optimization	
Signal Constraint Attach this block to the signals to be constrained in your SIMULINK model.	RMS Blocks Use these blocks to build signal energy constraints	Adaptive Lookup Tables	Response Optimization Demos Double click on this block to see available demos
Ready			100% Locked //

3 Double-click the Adaptive Lookup Tables block to open the Adaptive Lookup Tables library.



Simulink Design Optimization software provides three Adaptive Lookup Table blocks. In this tutorial, you use the Adaptive Lookup Table (nD Stair-Fit) block to model your system. To learn more about the blocks, see "Block Reference".

4 Drag and drop the Adaptive Lookup Table (nD Stair-Fit) block from the Simulink Design Optimization library to the Simulink model window.



5 Double-click the Adaptive Lookup Table (nD Stair-Fit) block to open the Function Block Parameters: Adaptive Lookup Table (nD Stair-Fit) dialog box.

Function Block Parameters: Adaptive Lookup Table (nD Stair-Fit)	x
Adaptive Lookup Table (nD) (mask) (link)	
Perform adaptive table lookup. Breakpoints relate the coordinate inputs to cell locations in the table. The data is used to dynamically update the cell values at these locations.	
Parameters	
Number of table dimensions:	
2	
Table breakpoints (cell array):	
{[10,22,31,40], [10,22,31,40]}	
☐ Make initial table an input	
Table data (initial):	
[4 5 6; 16 19 20; 10 18 23]	
Table numbering data:	
[1 2 3; 4 5 6; 7 8 9]	
Adaptation method: Sample mean (with forgetting)	
Adaptation gain (0 to 1):	
0.9	
Make adapted table an output	
Add adaptation enable/disable/reset port	
Add cell lock enable/disable port	
Action for out-of-range input Ignore	
	-
OK Cancel Help Apply	

- **6** 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" in "Selecting an Adaptation Method".

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 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.

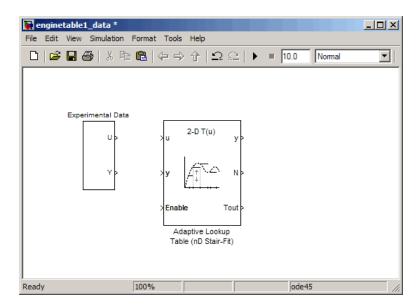
Tip To learn more about the Adaptive Lookup Table (nD Stair-Fit) block parameters, see the Adaptive Lookup Table (nD Stair-Fit) block reference page.

After you configure the parameters, the block parameters dialog box looks like the following figure.

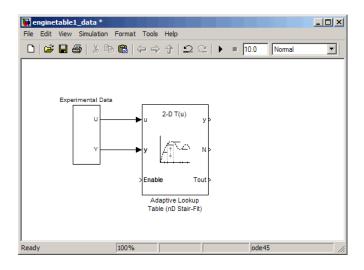
🙀 Function Block Parameters: Adaptive Lookup Table (nD Stair-Fit)	x
Adaptive Lookup Table (nD) (mask) (link)	
Perform adaptive table lookup. Breakpoints relate the coordinate inputs to cell locations in the table. The data is used to dynamically update the cell values at these locations.	
Parameters	
Number of table dimensions:	
2	
Table breakpoints (cell array):	
{[X; 110], [Y; 7200]}	
Make initial table an input	
Table data (initial):	
rand(10,36)	
Table numbering data:	
reshape(1:360,10,36)	
Adaptation method: Sample mean (with forgetting)]
Adaptation gain (0 to 1):	
0.98	
Make adapted table an output	
Add adaptation enable/disable/reset port	
Add cell lock enable/disable port	
Action for out-of-range input Ignore	1
OK Cancel Help Apply	

7 Click OK to close the Function Block Parameters dialog box.

The Simulink model now looks similar to the following figure.



8 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 "Connecting Blocks in the Model Window" in the Simulink documentation.

- **9** Design a logic that enables or disables the adaptation process:
 - **a** Open the Simulink Library Browser by typing simulink at the MATLAB prompt.
 - **b** In the **Libraries** area of the Simulink Library Browser window, select **Commonly Used Blocks**.
 - c Drag the following blocks to the Simulink model window:
 - Constant block
 - Logical Operator block

To learn more about the blocks, see the Constant, and Logical Operator block reference pages in the Simulink documentation.

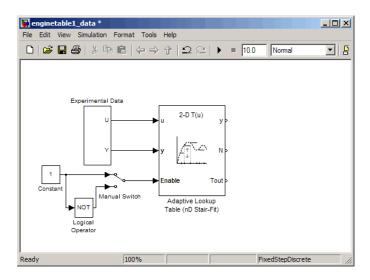
d In the **Libraries** area of the Simulink Library Browser window, select **Signal Routing**, and drag the Manual Switch block to the Simulink model window.

To learn more about the block, see the Manual Switch block reference page in the Simulink documentation.

- **e** Double-click the Logical Operator block to open the Function Block Parameters: Logical Operator dialog box.
- **f** Select NOT from the **Operator** drop-down list.

Function Block Parameters: Logical Operator
Logical Operator
Logical operators. For a single input, operators are applied across the input vector. For multiple inputs, operators are applied across the inputs.
Main Signal Attributes
Operator: AND
Number of i OR
2 NAND NOR
Icon shape
Sample time (-1 for inherited):
-1
OK Cancel Help Apply

g Click OK.

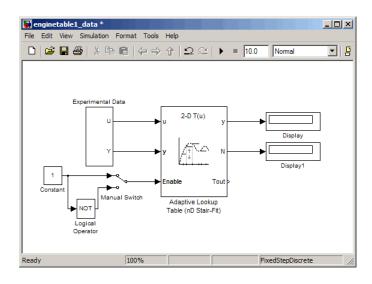


h Connect the blocks to create a logic, as shown in the next figure.

This logic outputs an initial value of 1 which enables the adaptation process.

10 Add Display blocks to view the output of the adaptation process:

- **a** In the **Libraries** area of the Simulink Library Browser window, select **Sinks**, and drag the Display block to the Simulink model window.
- **b** Drag another Display block from the Simulink Library Browser window to the model window.
- Connect the Display blocks to the y and N ports of the Adaptive Lookup Table block, as shown in the next figure.



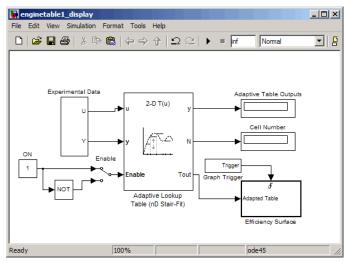
During simulation, the Display blocks show the following:

- Display block Shows the value of the current cell being adapted.
- Display1 block Shows the number of the current cell being adapted.

11 Open a preconfigured Simulink model that already contains a plot to display the lookup table values as they adapt during simulation. To do so, type the following model name at the MATLAB prompt:

enginetable1

The model opens, as shown in the next figure.



This model already contains a subsystem named Efficiency Surface that generates a plot of the lookup table values as they adapt during simulation.

Tip You can also write your own MATLAB function to plot the adapted table values during simulation.

You have now configured the adaptive lookup table parameters and 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 10-16.

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 10-4. To open a saved model that already contains the block parameters and display blocks, type the following model name at the MATLAB prompt:

enginetable1

To perform the adaptation:

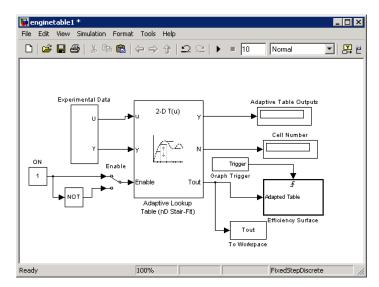
- 1 Connect a To Workspace Simulink block to export the adapted table values:
 - **a** Open the Simulink Library Browser, if it is not already open, by typing simulink at the MATLAB prompt.
 - **b** In the **Libraries** area of the Simulink Library Browser window, select **Sinks**, and drag the To Workspace block to the **enginetable1** Simulink model window.

To learn more about this block, see the To Workspace block reference page in the Simulink documentation. **c** Double-click the To Workspace block to open the Sink Block Parameters dialog box, and type Tout in the **Variable name** field.

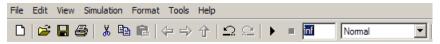
😽 Sink Block Parameters: To Workspace
To Workspace
Write input to specified array or structure in MATLAB's main workspace. Data is not available until the simulation is stopped or paused.
Parameters
Variable name:
Tout
Limit data points to last:
inf
Decimation:
1
Sample time (-1 for inherited):
-1
Save format: Structure
Log fixed-point data as an fi object
OK Cancel Help Apply

d Click OK.

• Connect the To Workspace block to the adaptive lookup table output signal Tout, as shown in the next figure.



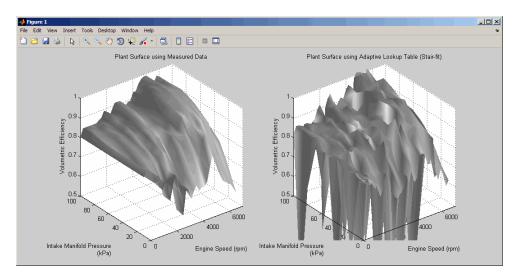
2 In the Simulink model window, enter inf as the simulation time.



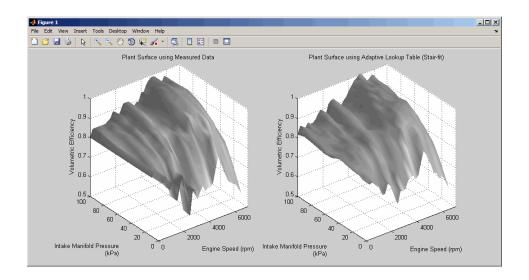
The simulation time of infinity specifies that the adaptation process continues as long as the input and output values of the engine change. **3** In the Simulink model window, select **Simulation > Start** to start the adaptation process.

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.

- **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** 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.

- **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 Manual Switch block.

This action toggles the switch to \checkmark , and disables the adaptation.

b Select **Simulation > Start** to restart the simulation.

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.



Examples

Use this list to find examples in the documentation.

Getting Started

Chapter 5, "Tutorial — Preparing Data for Parameter Estimation Using the GUI" Chapter 6, "Tutorial — Estimating Parameters from Measured Data Using the GUI" Chapter 7 "Tutorial — Optimizing Parameters to Moet Time Domain

Chapter 7, "Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the GUI"

Chapter 8, "Tutorial — Optimizing Parameters to Meet Time-Domain Requirements Using the Command Line"

Chapter 9, "Tutorial — Designing a PID Controller Using Optimization-Based Tuning"

Chapter 10, "Tutorial — Modeling a System Using Adaptive Lookup Table"

Index

A

add Signal Constraint block 7-5

B

block parameters window 7-10

C

configure for parameter optimization 7-5 configuring control design 9-5 Control and Estimation Tools Manager 9-5

D

design requirement line segments 7-8 design requirements time-domain 7-4 designing optimization-based linear controller 9-25

E

Edit Design Requirement dialog box rows 7-11

F

findconstr function example 8-8 findpar function example 8-11

L

linear control design 9-5

Ν

newsro function example 8-8

0

optimization-based control design 9-1 optimize function example 8-12

Ρ

parameter settings 7-20 plots 9-15

R

response optimization tutorial using functions 8-1 tutorial using GUI 7-1

S

signal constraints objects 8-8 Simulink Design Optimization product related products 1-5 required products 1-5 SISO Design Task 9-5 specify reference signal using GUI 7-25 specifying controller parameters 9-11 specifying design requirements 9-15

T

time-domain design requirements 8-4 track reference signal 8-15