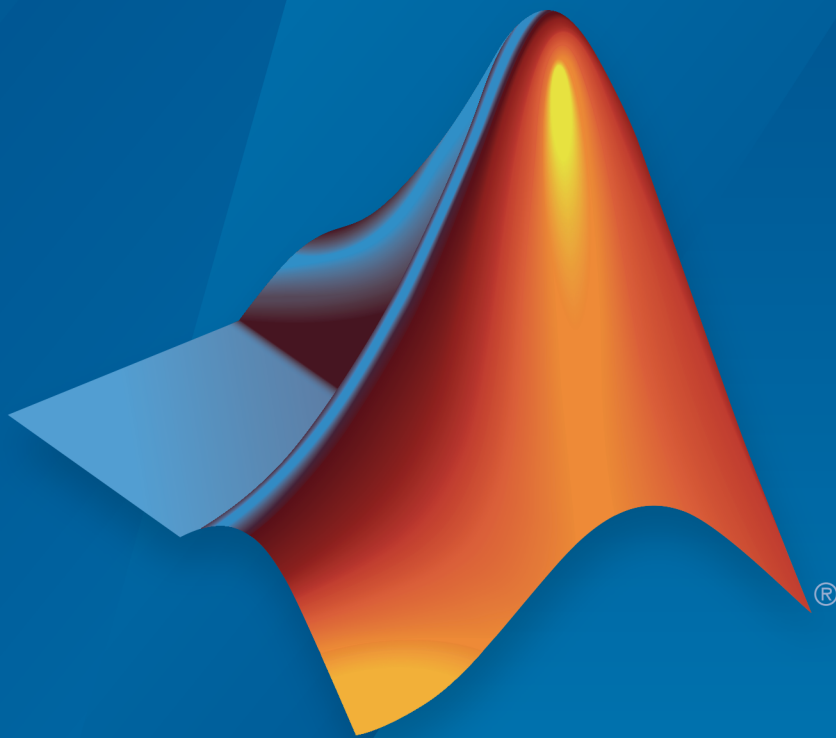


Model-Based Calibration Toolbox™

Getting Started Guide



MATLAB® & SIMULINK®

R2017a

 MathWorks®

How to Contact MathWorks



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



Phone: 508-647-7000



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

Model-Based Calibration Toolbox™ Getting Started Guide

© COPYRIGHT 2005–2017 by The MathWorks, Inc.

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

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

Trademarks

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

Patents

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

Revision History

November 2005	Online only	New for Version 3.0 (Release 14SP3+)
September 2006	Online only	Version 3.1 (Release 2006b)
March 2007	Online only	Version 3.2 (Release 2007a)
September 2007	Online only	Revised for Version 3.3 (Release 2007b)
March 2008	Online only	Revised for Version 3.4 (Release 2008a)
October 2008	Online only	Revised for Version 3.4.1(Release 2008a+)
October 2008	Online only	Revised for Version 3.5 (Release 2008b)
March 2009	Online only	Revised for Version 3.6 (Release 2009a)
September 2009	Online only	Revised for Version 3.7 (Release 2009b)
March 2010	Online only	Revised for Version 4.0 (Release 2010a)
September 2010	Online only	Revised for Version 4.1 (Release 2010b)
April 2011	Online only	Revised for Version 4.2 (Release 2011a)
September 2011	Online only	Revised for Version 4.3 (Release 2011b)
March 2012	Online only	Revised for Version 4.4 (Release 2012a)
September 2012	Online only	Revised for Version 4.5 (Release 2012b)
March 2013	Online only	Revised for Version 4.6 (Release 2013a)
September 2013	Online only	Revised for Version 4.6.1 (Release 2013b)
March 2014	Online only	Revised for Version 4.7 (Release 2014a)
October 2014	Online only	Revised for Version 4.8 (Release 2014b)
March 2015	Online only	Revised for Version 4.8.1 (Release 2015a)
September 2015	Online only	Revised for Version 5.0 (Release 2015b)
March 2016	Online only	Revised for Version 5.1 (Release 2016a)
September 2016	Online only	Revised for Version 5.2 (Release 2016b)
March 2017	Online only	Revised for Version 5.2.1 (Release 2017a)

Introduction

1

Model-Based Calibration Toolbox Product Description	1-2
Key Features	1-2
What Is Model-Based Calibration?	1-3
Designs and Modeling in the Model Browser	1-3
Calibration Generation in CAGE	1-4
Limitations	1-6
Waitbar May Have Transparent Background	1-6

Gasoline Engine Calibration

2

Gasoline Case Study Overview	2-2
Gasoline Calibration Problem Definition	2-2
Case Study Example Files	2-3
Design of Experiment	2-5
Context	2-5
Benefits of Design of Experiment	2-5
Power Envelope Survey Testing	2-5
Create Designs and Collect Data	2-6
Data Collection and Physical Modeling	2-13
Empirical Engine Modeling	2-15
Examine Response Models	2-15
Examine the Test Plan	2-19

Optimization	2-20
Optimization Overview	2-20
View Optimization Results	2-21
Set Up Optimization	2-23
Filling Tables From Optimization Results	2-27

Gasoline Engine Calibration Case Study

3

Gasoline Case Study Overview	3-2
Case Study Introduction	3-2
Why Use Design of Experiment and Engine Modeling?	3-3
Problem Definition	3-4
Introduction to Two-Stage Modeling	3-4
 Designing the Experiment	 3-6
Overview of Design Process	3-6
Specifying Model Inputs	3-6
Creating Designs	3-8
Data Source	3-11
 Selecting Data and Models to Fit	 3-13
 Viewing and Filtering Data	 3-18
View Data	3-18
Filter Data	3-21
 How Is a Two-Stage Model Constructed?	 3-23
 Selecting Local Models	 3-27
 Viewing the Boundary Model	 3-28
 Selecting Global and Two-Stage Models	 3-33
Inspect the Global Models	3-33
Create Multiple Models to Compare	3-33
Create a Two-Stage Model	3-35
Evaluate Other Response Models	3-35
 Using Validation Data	 3-36

Exporting the Models	3-37
Optimized Calibration	3-38
Problem Definition	3-38
Benefits of Automated Calibration	3-39
Importing Additional Models into CAGE	3-41
Setting Up Calibration Tables to Fill	3-44
Altering Variable Ranges	3-44
Setting Up Tables	3-44
Setting Up the Optimization	3-47
Defining Variable Values	3-50
Running the Optimization	3-52
Setting Up the Sum Optimization	3-57
Setting Up the Optimization Initial Values and Objective ..	3-57
Creating Table Gradient Constraints	3-58
Running the Sum Optimization	3-60
Filling Tables with Optimization Results	3-62
MBT Spark Estimator Problem	3-63
What Is an Estimator?	3-63
View the Feature	3-63
View Variables	3-65
Edit and Import Boundary Model	3-66
Use the Feature Fill Wizard	3-67
Inspect Results	3-71
CAGE Import Tool	3-74

Diesel Engine Calibration Case Study

4

Diesel Case Study Overview	4-2
Introduction	4-2
Problem Definition	4-2

Design of Experiment	4-4
Introducing Design of Experiment	4-4
Constraining the Design	4-6
Creating Candidate Designs	4-9
Data Collection	4-10
Modeling	4-11
Overview of Modeling Process	4-11
Optimized Calibration	4-12
Problem Definition	4-12
Benefits of Automated Calibration	4-13
Importing Models of Engine Responses into CAGE	4-15
Defining Additional Variables and Models	4-16
Setting Up Calibration Tables to Fill	4-18
Setting Up the Point Optimization	4-19
Setting Up Constraints	4-22
Defining Variable Values	4-26
Running the Optimization	4-29
Setting Up the Sum Optimization	4-31
Setting Up Initial Values and Sum Objective	4-31
Creating the Brake Specific NOx Constraint	4-32
Setting Weights for the Sum Objective and Constraint	4-33
Set Parameters and Run Optimization	4-35
Filling Tables with Optimization Results	4-36

5 Point-by-Point Diesel Engine Calibration Case Study

5

Point-by-Point Diesel Case Study Overview	5-2
What Is Point-by-Point Modeling?	5-2

Engine Calibration Specifications and Problem Description . .	5-2
Required Tasks	5-4
Create Designs and Models Programmatically	5-5
Overview of Programmatic Design and Modeling	5-5
Creating Designs and Models Programmatically	5-5
Verify and Refine Models	5-7
Open the Project and View Designs	5-7
Analyze and Refine Local Fits	5-7
Point-by-Point Optimization Overview	5-9
Create Point Optimizations in CAGE	5-10
Introduction	5-10
Load Models to Optimize	5-10
Create Part Load Point Optimization	5-11
Create Full-Load Point Optimization	5-15
Create Multiregion Sum Optimization	5-19
Introduction	5-19
Create Data Set from Point Optimization Results	5-19
Create Sum Optimization	5-20
Add Application Point Sets	5-21
Set Up Constraints	5-23
Define Weights and Fixed Variables and Run Optimization . .	5-25
Use Optimization Results	5-27
Introduction	5-27
Create Tables to Fill	5-27
Fill Tables from Optimization Results	5-28
Export Tables	5-29

Composite Models and Modal Optimizations

6

Introducing Composite Models and Modal Optimization . . .	6-2
---	-----

Composite Model and Modal Optimization Projects	6-3
Gasoline Example with Four Cylinder and Eight Cylinder	
Modes	6-3
Diesel Example with Low and High EGR Modes	6-3
Composite Model Example with Separate Tables for Each	
Mode	6-4

Design and Modeling Scripts

7

Introduction to the Command-Line Interface	7-2
Automate Design and Modeling With Scripts	7-3
Processes You Can Automate	7-3
Engine Modeling Scripts	7-5
Understanding Model Structure for Scripting	7-6
Projects and Test Plans for Model Scripting	7-6
Response Model Scripting	7-6
Boundary Model Scripting	7-8
How the Model Tree Relates to Command-Line Objects	7-10

Multi-Injection Diesel Calibration

8

Multi-Injection Diesel Calibration Workflow	8-2
Multi-Injection Diesel Problem Definition	8-2
Engine Calibration Workflow	8-7
Air-System Survey Testing	8-8
Multi-Injection Testing	8-9
Data Collection and Physical Modeling	8-10
Statistical Modeling	8-11
Optimization Using Statistical Models	8-12
Case Study Example Files	8-20

Design of Experiment	8-22
Benefits of Design of Experiment	8-22
Air-System Survey Testing	8-22
Create Designs and Collect Data	8-23
Fit a Boundary Model to Air Survey Data	8-29
Use the Air Survey and Boundary Model to Create the Final Design	8-32
Multi-Injection Testing	8-34
Statistical Modeling	8-35
Examine the Test Plans for Point-by-Point Models	8-35
Examine Response Models	8-40
Optimization	8-42
Optimization Overview	8-42
Set Up Models and Tables for Optimization	8-42
Examine the Point Optimization Setup	8-45
Examine the Point Optimization Results	8-49
Create Sum Optimization from Point Optimization	8-51
Fill Tables from Optimization Results	8-57
Examine the Multiobjective Optimization	8-67

Model Quickstart

9

Empirical Engine Modeling	9-2
Two-Stage Modeling Example	9-3
About the Two-Stage Modeling Example	9-3
About Two Stage Models	9-3
Open the App and Load Data	9-5
Set Up the Model	9-7
Specifying Model Inputs	9-7
Setting Up the Response Model	9-9
Verify the Model	9-11
Verifying the Local Model	9-11
Verifying the Global Model	9-12

Creating the Two-Stage Model	9-13
Comparing the Local Model and the Two-Stage Model	9-14
Response Node	9-14
Export the Model	9-15
Create Multiple Models to Compare	9-17
Methods For Creating More Models	9-17
Creating New Local Models	9-17
Adding New Response Features	9-20
Comparing Models	9-21
Creating New Global Models	9-22

Design of Experiment

10

Design of Experiment	10-2
Why Use Design of Experiment?	10-2
Design Styles	10-3
Create Examples Using the Design Editor	10-3
Set Up a Model and Create a Design	10-5
Set Up Model Inputs	10-5
Open the Design Editor	10-5
Create a New Design	10-6
Create a Constrained Space-Filling Design	10-7
Apply Constraints	10-7
Save Designs	10-12
Create Optimal Designs	10-13
Introducing Optimal Designs	10-13
Start Point Tab	10-14
Candidate Set Tab	10-15
Algorithm Tab	10-17
View Design Displays	10-19

Use the Prediction Error Variance Viewer	10-21
Introducing the Prediction Error Variance Viewer	10-21
Add Points Optimally	10-24
Create Classical Designs	10-27
Adding a Classical Design	10-27
Classical Design Browser	10-28
Setting Up and Viewing a Classical Design	10-29
Use the Design Evaluation Tool	10-33

Data Editor for Modeling

11

Data Manipulation for Modeling	11-2
Load Data	11-3
Opening the Data Editor	11-3
Loading a Data File	11-3
View and Edit the Data	11-5
Viewing Data	11-5
Using Notes to Sort Data for Plotting	11-5
Removing Outliers and Problem Tests	11-6
Reordering and Editing Data	11-7
Create New Variables and Filters	11-9
Adding New Variables	11-9
Applying a Filter	11-9
Sequence of Variables	11-11
Deleting and Editing Variables and Filters	11-11
Store and Import Variables, Filters, and Plot Preferences	11-12
Define Test Groupings	11-13
Match Data to Experimental Designs	11-16
Introducing Matching Data to Designs	11-16
Tolerances and Cluster Information	11-18
Understanding Clusters	11-20

Feature Calibration

12

Feature Calibration	12-2
What Are Feature Calibrations?	12-2
Start CAGE	12-4
Set Up Variables	12-4
Set Up Models	12-7
Set Up a New Feature	12-9
Set Up the Strategy	12-10
Set Up the Tables	12-12
Process For Feature Calibration	12-14
Calibrate the Normalizers	12-15
Calibrate the Tables	12-19
Calibrate the Feature	12-25
Export Calibrations	12-30

Tradeoff Calibration

13

Tradeoff Calibration	13-2
What Is a Tradeoff Calibration?	13-2
Setting Up a Tradeoff Calibration	13-2
Performing the Tradeoff Calibration	13-7

Data Sets

14

Compare Calibrations To Data	14-2
Setting Up the Data Set	14-2
Comparing the Items in a Data Set	14-7
Reassigning Variables	14-14

15

Fill Tables from Data	15-2
Setting Up a Table and Experimental Data	15-2
Filling the Table from the Experimental Data	15-8
Selecting Regions of the Data	15-11
Exporting the Calibration	15-13

Optimization and Automated Tradeoff

16

Optimization and Automated Tradeoff	16-2
Import Models to Optimize	16-2
Optimization Example Problems	16-4
Single-Objective Optimization	16-5
Process Overview	16-5
Using the Create Optimization from Model Wizard	16-6
Setting Constraints and Operating Points	16-9
Running the Optimization	16-12
Using Optimization Results to Fill Tables	16-14
Using a Custom Fill Routine to Fill Tables	16-16
Multiobjective Optimization	16-18
Setting Up and Running the Multiobjective Optimization .	16-18
Optimization Output View	16-23
Selecting Best Solutions	16-28
Sum Optimization	16-30
What Is a Sum Optimization?	16-30
Procedure	16-31
Automated Tradeoff	16-35

Introduction

The following sections introduce Model-Based Calibration Toolbox software.

- “Model-Based Calibration Toolbox Product Description” on page 1-2
- “What Is Model-Based Calibration?” on page 1-3
- “Limitations” on page 1-6

Model-Based Calibration Toolbox Product Description

Model and calibrate engines

Model-Based Calibration Toolbox provides apps and design tools for optimally calibrating complex engines and powertrain subsystems. You can define optimal test plans, automatically fit statistical models, and generate calibrations and lookup tables for complex high-degree-of-freedom engines that would otherwise require exhaustive testing using traditional methods. Calibrations can be optimized at individual operating points or over drive cycles to identify the optimal balance of engine fuel economy, performance, and emissions. Using apps or MATLAB[®] functions, you can automate the calibration process for similar engine types.

Models created with Model-Based Calibration Toolbox can be exported to Simulink[®] to support control design, hardware-in-the-loop testing, and powertrain simulation activities across the powertrain design team. Calibration tables can be exported to ETAS INCA and ATI Vision.

Key Features

- Apps that support the entire workflow: designing experiments, fitting statistical models to engine data, and producing optimal calibrations
- Design-of-Experiments methodology for reducing testing time through classical, space-filling, and optimal design techniques
- Accurate engine modeling with data fitting techniques including Gaussian process, radial basis function, and linear regression modeling
- Boundary modeling to keep optimization results within the engine operating envelope
- Generation of lookup tables from optimizations over drive cycles, models, or test data
- Export of performance-optimized models to Simulink for use in simulation and HIL testing
- Lookup table import and export to ETAS INCA and ATI Vision

What Is Model-Based Calibration?



High accuracy engine models are a key component for reducing calibration effort and engine development time.

The time spent calibrating an engine control unit has been increasing, due to new control actuators. The new actuators give the potential for increased performance, reduced emissions, and improved fuel consumption. It is necessary to apply advanced modeling and optimization techniques to achieve the full benefits available from the addition of new actuators. Advanced modeling techniques offer increased understanding of the complex, nonlinear engine responses. High accuracy models can be used throughout the design process, including the calibration of base maps with the optimal settings for the control parameters, determined by constrained optimizations.

The toolbox has two main user interfaces for model-based calibration workflows:

- Model Browser for design of experiment and statistical modeling
- CAGE Browser for analytical calibration

The Model Browser part of the toolbox is a powerful tool for experimental design and statistical modeling. The models you build with the Model Browser can be imported into the CAGE Browser part of the toolbox to produce optimized calibration tables.

Designs and Modeling in the Model Browser

The Model Browser is a flexible, powerful, intuitive graphical interface for building and evaluating experimental designs and statistical models:

- Design of experiment tools can drastically reduce expensive data collection time.

- You can create and evaluate optimal, space-filling, and classical designs, and constraints can be designed or imported.
- Hierarchical statistical models can capture the nature of variability inherent in engine data, accounting for variation both within and between tests.
- The Model Browser has powerful, flexible tools for building, comparing, and evaluating statistical models and experimental designs.
- There is an extensive library of prebuilt model types and the capability to build user-defined models.
- You can export models to CAGE or to MATLAB or Simulink software.

Starting the Model Browser

To start the application, type

```
mbcmodel
```

at the MATLAB command prompt.

Calibration Generation in CAGE

CAGE (CALibration GEneration) is an easy-to-use graphical interface for calibrating lookup tables for your electronic control unit (ECU).

As engines get more complicated, and models of engine behavior more intricate, it is increasingly difficult to rely on intuition alone to calibrate lookup tables. CAGE provides analytical methods for calibrating lookup tables.

CAGE uses models of the engine control subsystems to calibrate lookup tables. With CAGE, you fill and optimize lookup tables in existing ECU software using Model Browser models. From these models, CAGE builds steady-state ECU calibrations.

CAGE also compares lookup tables directly to experimental data for validation.

Starting the CAGE Browser

To start the application, type

`cage`

at the MATLAB command prompt.

Limitations

Waitbar May Have Transparent Background

The waitbar dialogs that are displayed to inform you of the progress of lengthy operations in the toolbox often display with transparent backgrounds. Instead of the normal window grey, they pick up their background from whatever is on the screen when they are displayed.

This bug is purely cosmetic and will not cause any data loss.

Gasoline Engine Calibration

- “Gasoline Case Study Overview” on page 2-2
- “Design of Experiment” on page 2-5
- “Empirical Engine Modeling” on page 2-15
- “Optimization” on page 2-20

Gasoline Case Study Overview

In this section...
“Gasoline Calibration Problem Definition” on page 2-2
“Case Study Example Files” on page 2-3

Gasoline Calibration Problem Definition

This case study demonstrates how to systematically develop a set of optimal steady-state engine calibration tables using the Model-Based Calibration Toolbox.

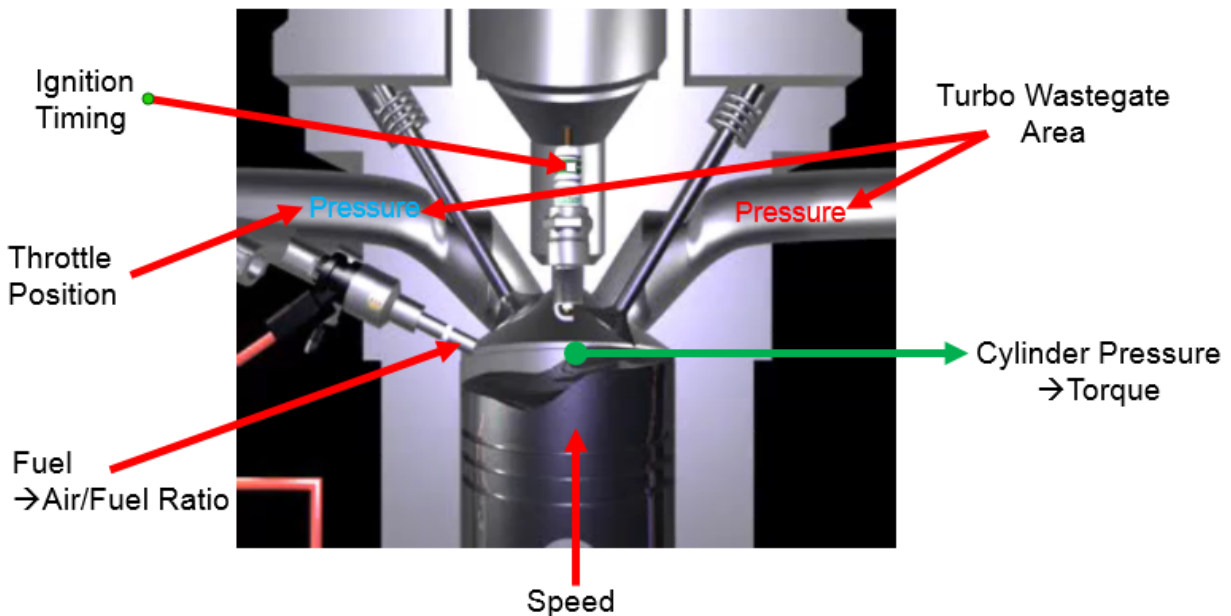
The engine to calibrate is a direct-injected 1.5L spark ignition turbocharged engine with dual cam-phasers and turbocharger wastegate.

The aim of the calibration is to maximize torque at specific speed/load values across the engine's operating range, and meet these constraints:

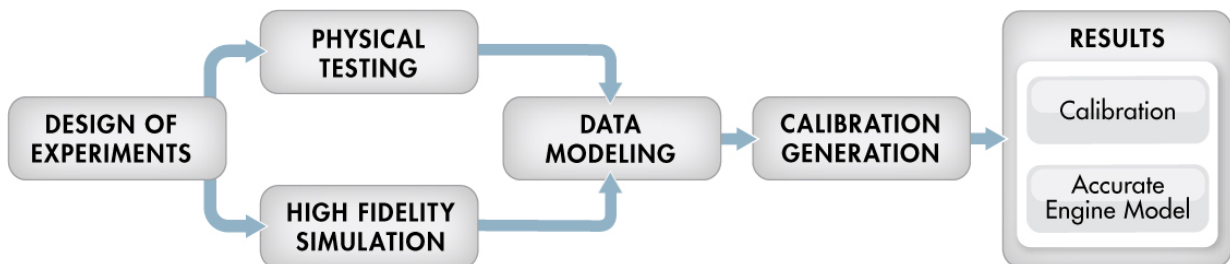
- Limit knock
- Limit residual fraction
- Limit exhaust temperature
- Limit calibration table gradients for smoothness.

The analysis must produce optimal engine calibration tables in speed and load for:

- Spark advance ignition timing (SA)
- Throttle position % (TPP)
- Turbo wastegate area % (WAP)
- Air/Fuel Ratio (Lambda: LAM)
- Intake cam phase (ICP)
- Exhaust cam phase (ECP)
- Torque (TQ)
- Brake-specific fuel consumption (BSFC)
- Boost (MAP)
- Exhaust temperature (TEXH)



This case study illustrates the model-based calibration process.



Case Study Example Files

The following sections guide you through opening example files to view each stage of the model-based calibration process. You can examine:

- 1 Designs, constraints, boundary model, and collected data, in topic “Design of Experiment” on page 2-5.

- 2 Finished statistical models, in topic “Empirical Engine Modeling” on page 2-15.
- 3 Optimization setup and results, and filled calibration tables, in topic “Optimization” on page 2-20.

Use these example files to understand how to set up systematic calibrations for similar problems. For next steps, see “Design of Experiment” on page 2-5.

Tip: Learn how MathWorks® Consulting helps customers develop engine calibrations that optimally balance engine performance, fuel economy, and emissions requirements: see [Optimal Engine Calibration](#).

Design of Experiment

In this section...

“Context” on page 2-5

“Benefits of Design of Experiment” on page 2-5

“Power Envelope Survey Testing” on page 2-5

“Create Designs and Collect Data” on page 2-6

“Data Collection and Physical Modeling” on page 2-13

Context

This topic describes design of experiments for the gasoline one-stage case study. To view the high-level workflow, see “Gasoline Case Study Overview” on page 2-2.

Benefits of Design of Experiment

You use design of experiment to efficiently collect engine data. Testing time (on a dyno cell, or as in this case, using high-fidelity simulation) is expensive, and the savings in time and money can be considerable when a careful experimental design takes only the most useful data. Dramatically reducing test time is increasingly important as the number of controllable variables in more complex engines is growing. With increasing engine complexity, the test time increases exponentially.

Power Envelope Survey Testing

The first stage to solve this calibration problem is to determine the boundaries of the feasible system settings. You need to generate the power envelope to constrain the design points. To do this, data was collected using simulation across a range of speed and torque. The initial survey determined boundaries that produce:

- Acceptable exhaust temperature (not too high to burn piston crowns)
- Achievable torque production
- Acceptable BSFC (not too high)
- Avoids knock

The envelope must include the idle region of low torque and speed, where the cams must be parked. The cam timings are set to zero in the idle design.

The initial survey design and test data provide information about the engine operating envelope. This information was used to create constraints for the final design, to collect detailed data about the engine behavior within those boundaries. You can then use this data to create response models for all the responses you need in order to create an optimal calibration for this engine.

The final design used 238 points for this engine with 4 inputs: speed, load, intake and exhaust cam. You can view the constraints defining the operating envelope by following the steps below.

Create Designs and Collect Data

You can use a space-filling design to maximize coverage of the factors' ranges as quickly as possible, to understand the operating envelope.

To create a design, you need to first specify the model inputs. Open the example file to see how to define your test plan.

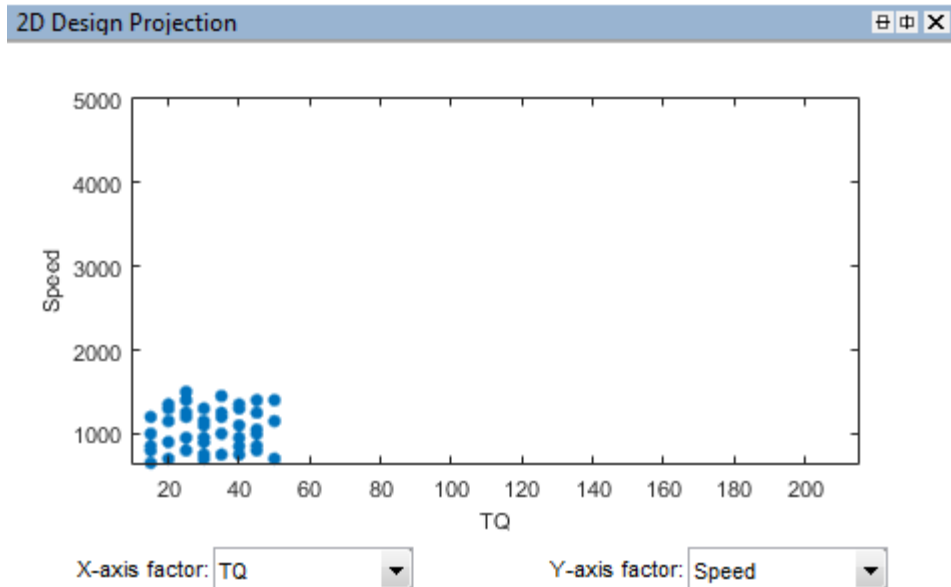
- 1 Open MATLAB. On the **Apps** tab, in the **Math, Statistics and Optimization** group, click **MBC Model Fitting**.
- 2 In the Model Browser home page, in the **Case Studies** list, select **Dual CAM gasoline engine with spark optimized during testing**. Alternatively, select **File > Open Project** and browse to the example file `gasolineOneStage.mat`, found in `matlab\toolbox\mbc\mbctraining`.
- 3 To view how to define your test plan design inputs, in the **All Models** tree, click the top project node, `gasolineOneStage`. In the **Common Tasks** pane, click **Design experiment**. In the New Test Plan dialog box, observe the inputs pane, where you can change the number of model inputs and specify the input symbols, signals and ranges. This example project already has inputs defined, so click **Cancel**.
- 4 Click the first test plan node in the **All Models** tree, `gasolineOneStageDoE`. The test plan view appears.
- 5 Observe the inputs listed on the test plan diagram. Double-click the **Inputs** block to view the ranges and names (symbols) for variables in the Input Factor Set Up dialog box. The design inputs are torque, speed, intake cam, and exhaust cam. The dynamometer test setup is in speed/torque and so the design is in speed/torque.

Close the dialog box by clicking **Cancel**.

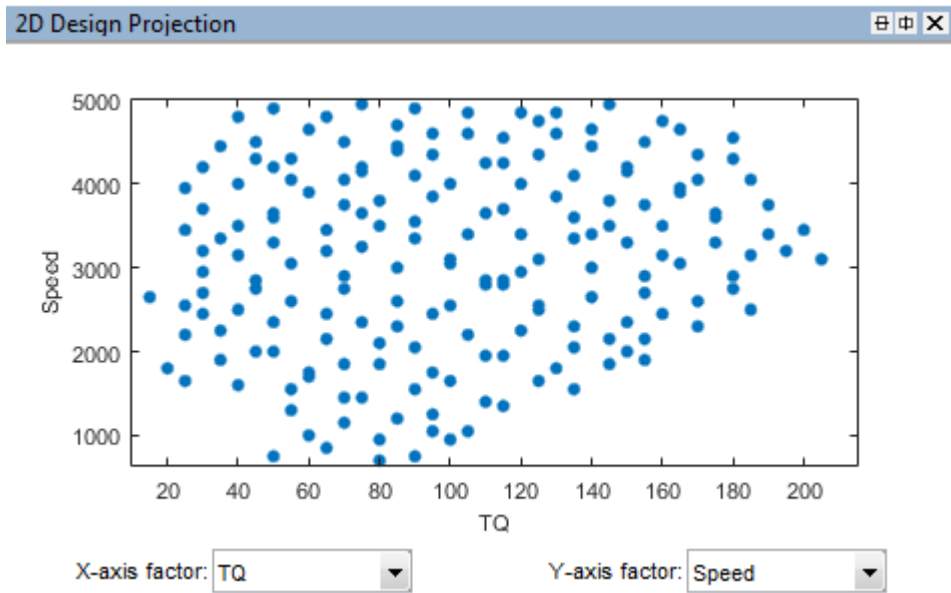
- 6 After setting up inputs, you can create designs. In the **Common Tasks** pane, click **Design experiment**.

The Design Editor opens. Here, you can see how these designs are built.

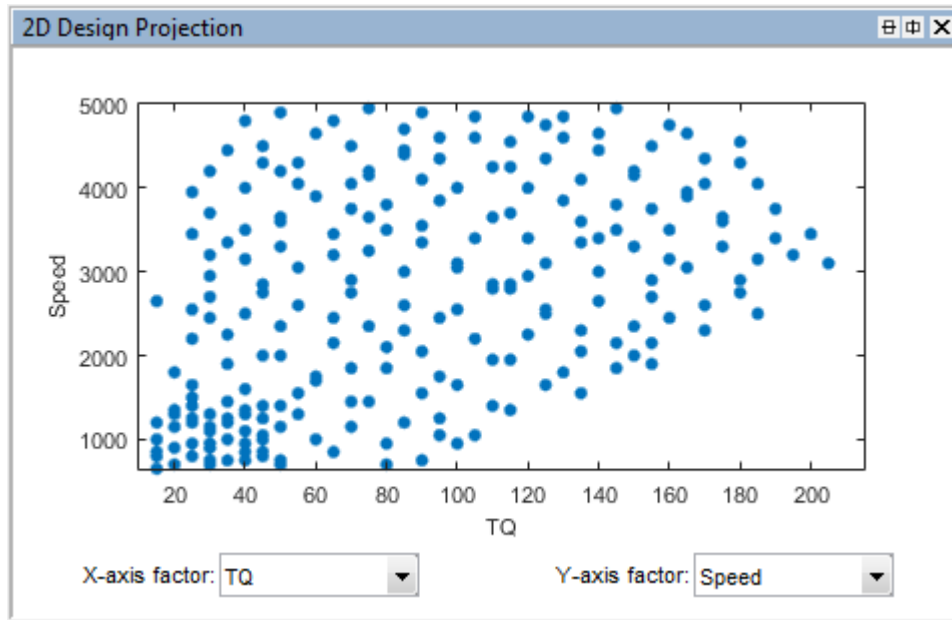
- 7 Click the first design in the tree, **gasolineOneStageIdle**. If you do not see a 2D plot, select **View > Current View > 2D Design Projection**. Then select each design in the tree in turn.
 - The first design in the tree, **gasolineOneStageIdle**, concentrates points in the idle area with cams parked. It uses the Sobol Sequence space-filling design type to maximize coverage. To park the cams, after creating the idle region space-filling design, the cam phaser positions are set to park position (0,0).



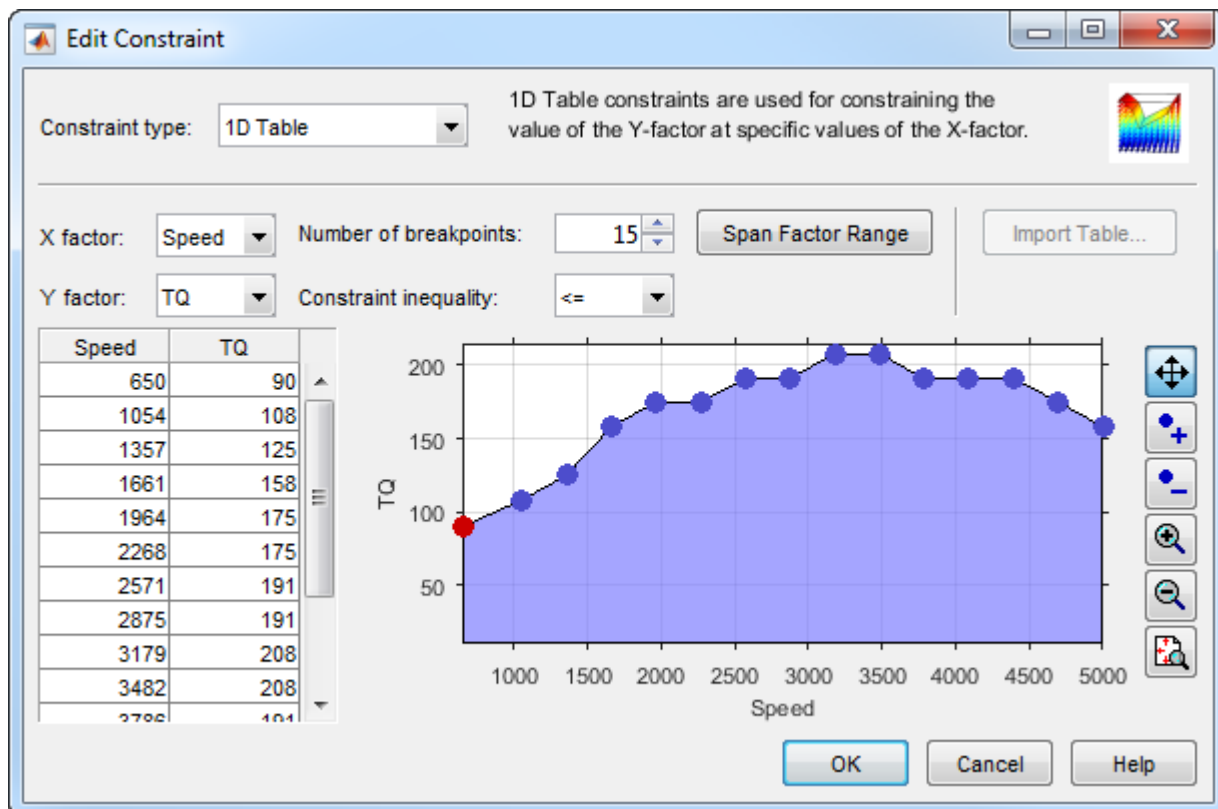
- The second design, **gasolineOneStageNonIdle**, is another Sobol Sequence space-filling design to span the non-idle engine operating envelope.

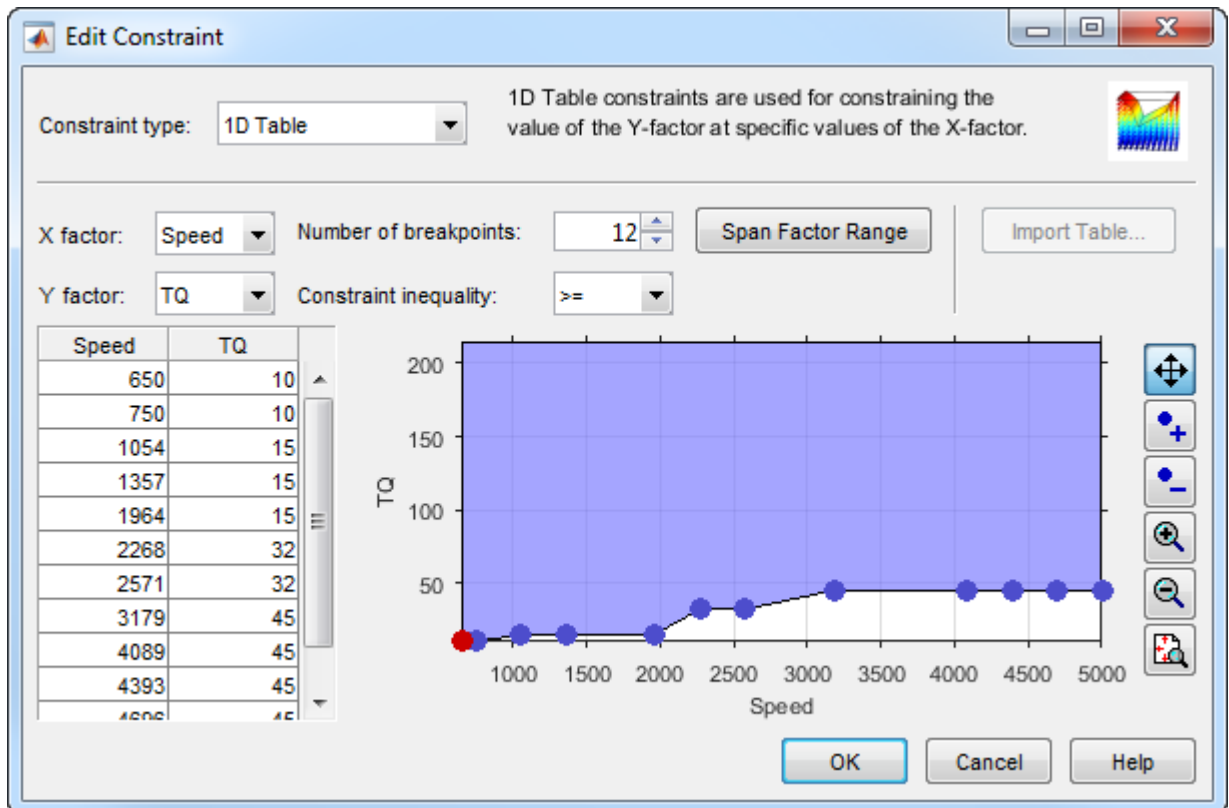


- The final design is called `gasolineOneStageMerged` because it contains the other two merged designs to cover the whole envelope.



- 8 To see how the constraints are set up, select **Edit > Constraints**.
- 9 In the Constraints Manager dialog box, select each torque constraint in turn and click **Edit**. Observe the maximum and minimum torque constraints define the upper and lower boundary of the feasible operating envelope. These constraints were developed prior to this design in initial wide-open throttle and closed-throttle engine performance testing.



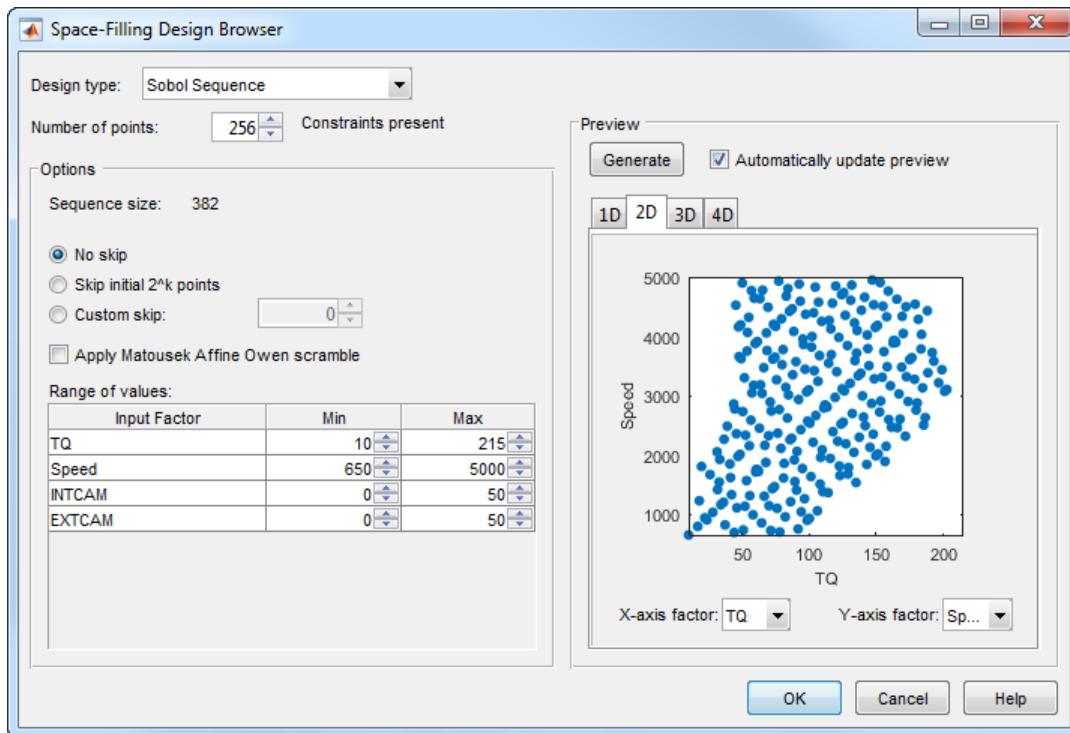


Observe that you can define areas to exclude by dragging points, typing in the edit boxes, or using the Table Editor tab.

To leave the constraint unchanged, click **Cancel**.

- 10 Observe the **Properties** of the selected `gasolineOneStageMerged` design under the tree, listing 2 constraints, and 238 points.
- 11 To experiment with a new design and avoid editing the prior designs, select the root **Designs** node and select **File > New Design**.
- 12 Add the constraints by selecting **Edit > Constraints**. In the Constraints Manager dialog box, click **Import**. Select the torque max and min constraints from the merged design and click **OK**. In the following dialogs, click **OK** to return to the Design Editor.

- 13 See how to construct a similar constrained space-filling design by selecting **Design > Space Filling > Design Browser**, or click the space-filling design toolbar button.
- 14 In the Space-Filling Design Browser, observe the design type is **Sobol Sequence**, and specify a **Number of points**. View the preview of design points. Click **OK**.



- 15 Click `gasolineOneStageMerged`. This design style is **Custom** because the points are rounded, using **Edit > Round Factor**. You might also sort design points to make data collection easier on a dyno. To preserve the space-filling sequence in case you want to add more points later, you can copy a design before rounding or sorting.

`gasolineOneStageMerged` is the final design used to collect data. To make it easier to collect the data, the points are rounded as follows:

- Intake and exhaust cam timings are rounded to 1 degree (2% of range)
- Speed is rounded to 50 RPM (1% of range)
- Torque is rounded to 5 Nm (3% of range)

You can export designs or copy and paste design points into other files to proceed to data collection.

16 Close the Design Editor.

The final `gasolineOneStageMerged` design was used to collect data from the GT-Power model with the Simulink and Simscape™ test harness. The example Model Browser project file `gasolineOneStage.mat` in the `mbctraining` folder contains this data, imported to the Model Browser after the data collection.

Data Collection and Physical Modeling

The toolbox provides the data in the projects for you to explore this calibration example.

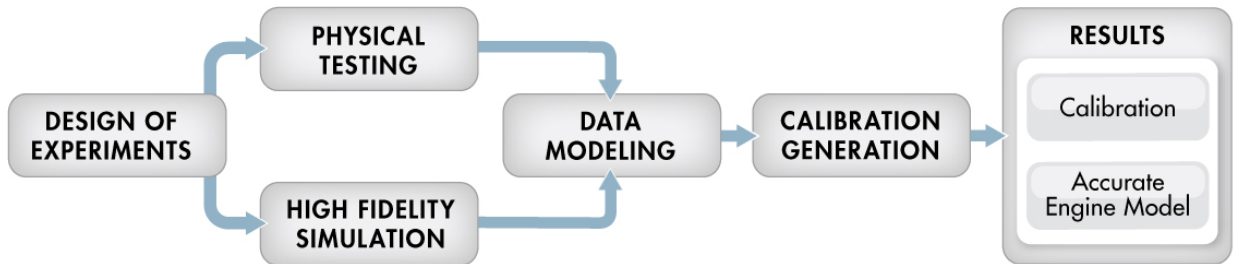
MathWorks collected the data using simulation tools. Control and simulation models were constructed using a Simulink and Stateflow® test harness. Constrained experimental designs were constructed using Model-Based Calibration Toolbox. The points specified in the design were measured using the GT-Power engine simulation tool from Gamma Technologies (see <http://www.gtisoft.com>). The engine to calibrate is a direct-injected 1.5L spark ignition turbocharged engine with dual cam-phasers and turbocharger wastegate. This model is part of a GT-POWER engine library from Gamma Technologies.

To collect the data, Simulink and Stateflow controlled the GT-Power engine model to the desired Design of Experiments points.

Note: Simulation time was reduced from days to minutes using Parallel Computing Toolbox™.

This simulation of 238 design points took 20 minutes to run in parallel on multiple machines in the cloud. Running on a single core, the same simulation takes 3 days. Parallel Computing Toolbox distributed the work to the 225 cores on a cloud computing cluster and showed that this problem scales linearly as you add workers.

The data was used in the next step of model-based-calibration, to create statistical models.



For next steps, see “Empirical Engine Modeling” on page 2-15.

Empirical Engine Modeling

In this section...
“Examine Response Models” on page 2-15
“Examine the Test Plan” on page 2-19

After designing the experiments and collecting the data, as described in “Design of Experiment” on page 2-5, then you can fit statistical models to the data. You can use the toolbox to generate accurate, fast-running models from the measured engine data.

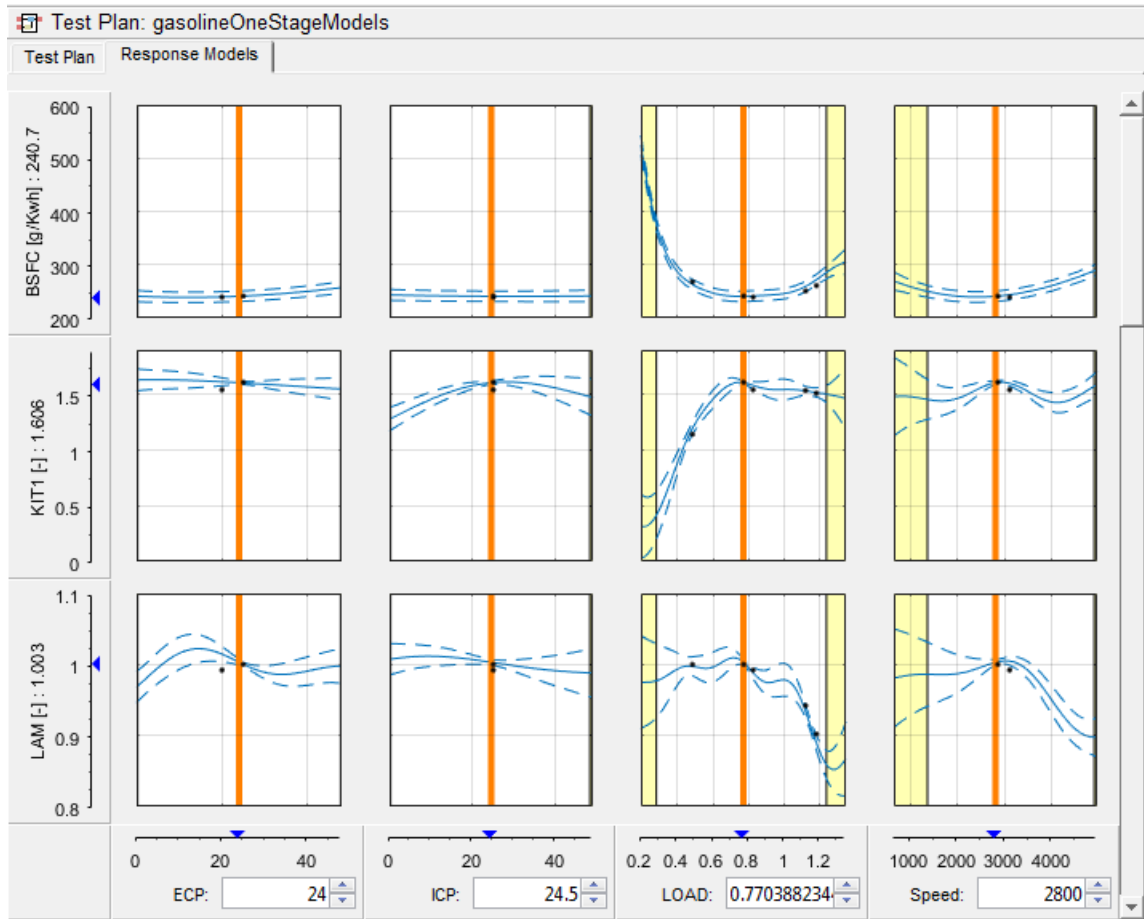
The dynamometer test setup is in speed/torque and so the design is in speed/torque. The model is in speed/load because the production engine controller implementation uses load (derived from air flow) instead of torque tables. The controller uses load because airflow sensors are presently less expensive in mass production than torque meters.

Examine Response Models

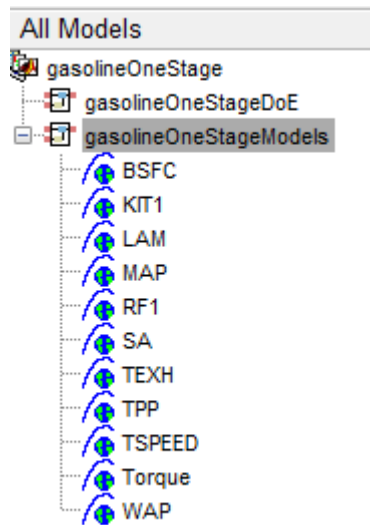
- 1 Open MATLAB. On the **Apps** tab, in the **Math, Statistics and Optimization** group, click **MBC Model Fitting**.

In the Model Browser home page, in the **Case Studies** list, open **Dual CAM gasoline engine with spark optimized during testing**.

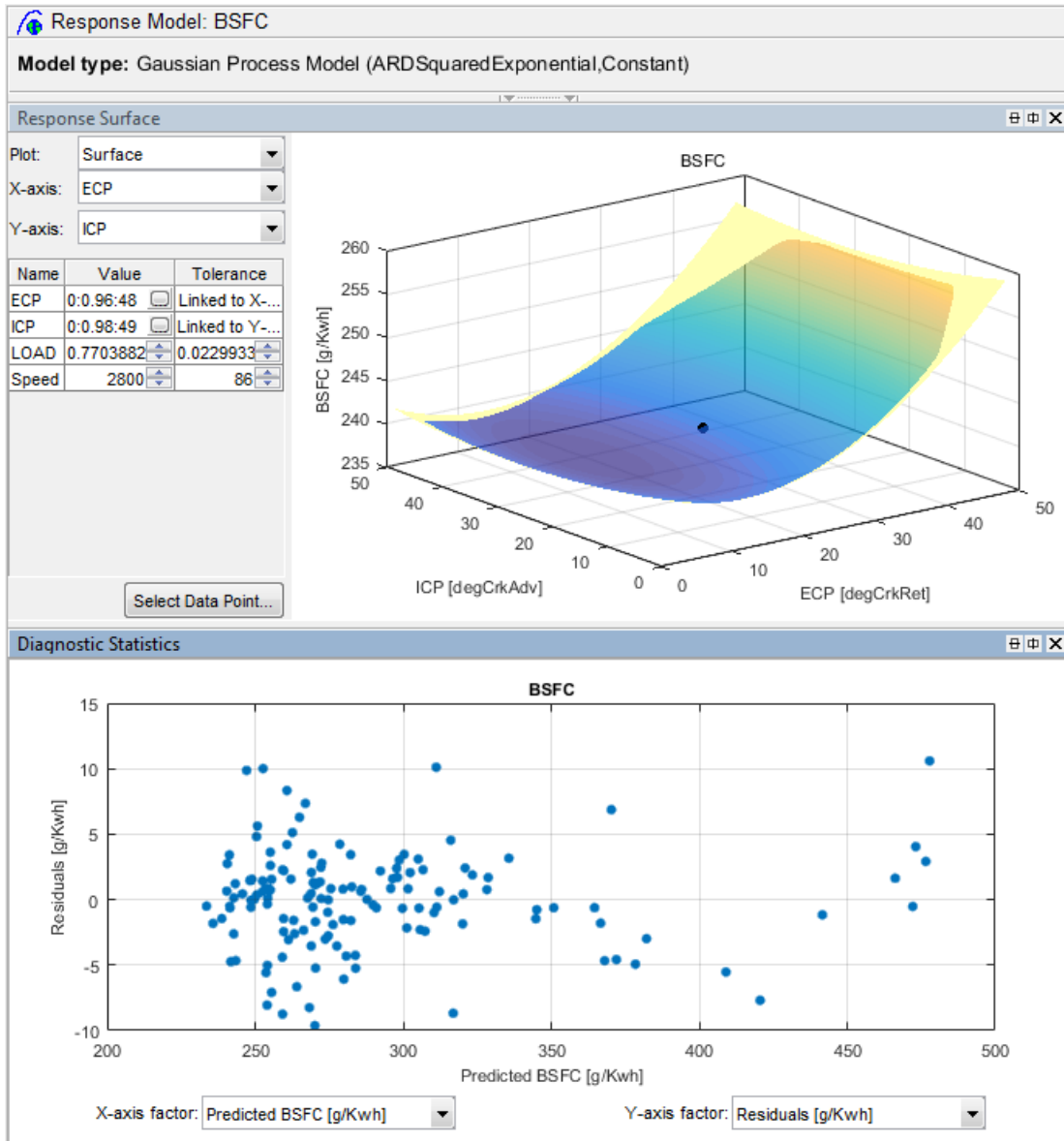
- 2 In the `gasolineOneStage.mat` project, click the second test plan node in the **All Models** tree, `gasolineOneStageModels`.
- 3 To assess high-level model trends, at the test plan node select the **Response Models** tab. After you fit models, the view at the test plan node displays the **Response Models** tab by default. View the cross-section plots of all the response models.



- 4 To view each response model in detail, expand **gasolineOneStageModels** test plan node in the **All Models** tree. Select the first response node under the test plan node, **BSFC**.



- 5 Examine the **Response Surface** plot and the **Diagnostic Statistics** plot.



6 Examine the other responses in the All Models tree.

For details on using the plots and statistics to analyze models, see “Assess High-Level Model Trends” and “Assess One-Stage Models”.

Examine the Test Plan

Examine the model setup.

- 1** At the `gasolineOneStageModels` test plan node, change to the test plan view if necessary by clicking the **Test Plan** tab. The Model Browser remembers selected views.
- 2** Observe the inputs and response model outputs listed on the test plan diagram.
- 3** Double-click the **Inputs** block to view the ranges and names (symbols) for variables on the Input Factor Set Up dialog box.
- 4** Double-click the **Model** block to view that the model class is the default for one-stage models, a `Gaussian Process Model`. When you use the **Fit models** button in the **Common Tasks** pane, and select a `One-stage` template, the toolbox sets the model type to a `Gaussian Process Model`.

For details on setting up one-stage models, see “Fit a One-Stage Model”.

- 5** Click **Cancel** to close the Model Setup dialog box without altering your example models.

For next steps, see “Optimization” on page 2-20.

Optimization

In this section...

“Optimization Overview” on page 2-20

“View Optimization Results” on page 2-21

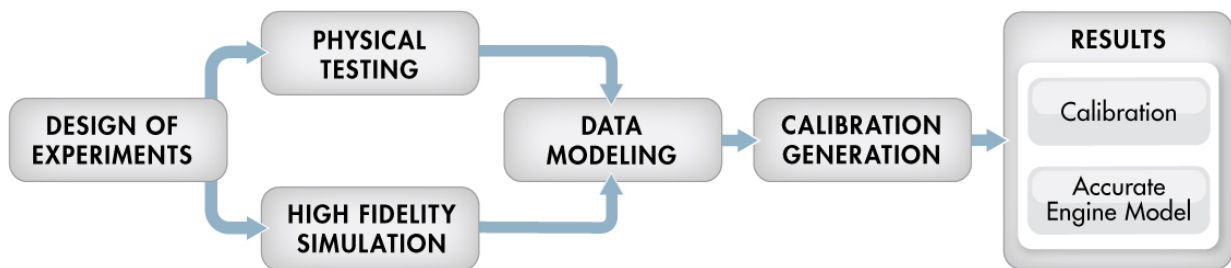
“Set Up Optimization” on page 2-23

“Filling Tables From Optimization Results” on page 2-27

Optimization Overview

After creating statistical models to fit the data, you can use them in optimizations. You can use the accurate statistical engine model to replace the high-fidelity simulation and run much faster, enabling optimization to generate calibrations in feasible times. You use the statistical models to find the optimal configurations of the engine that meet the constraints.

The statistical models described in “Empirical Engine Modeling” on page 2-15 were used in the next step of model-based-calibration, to create optimized calibration tables.



The aim of the calibration is to maximize torque at specific speed/load values across the engine's operating range, and meet these constraints:

- Limit knock (KIT1)
- Limit residual fraction
- Limit exhaust temperature
- Limit calibration table gradients for smoothness.

Turbocharger speed constraints are implicitly included in the boundary model which models the envelope.

The analysis must produce optimal engine calibration tables in speed and load for:

- Spark advance ignition timing (SA)
- Throttle position % (TPP)
- Turbo wastegate area % (WAP)
- Air/Fuel Ratio (Lambda: LAM)
- Intake cam phase (ICP)
- Exhaust cam phase (ECP)
- Torque (TQ)
- Brake-specific fuel consumption (BSFC)
- Boost (MAP)
- Exhaust temperature (TEXH)

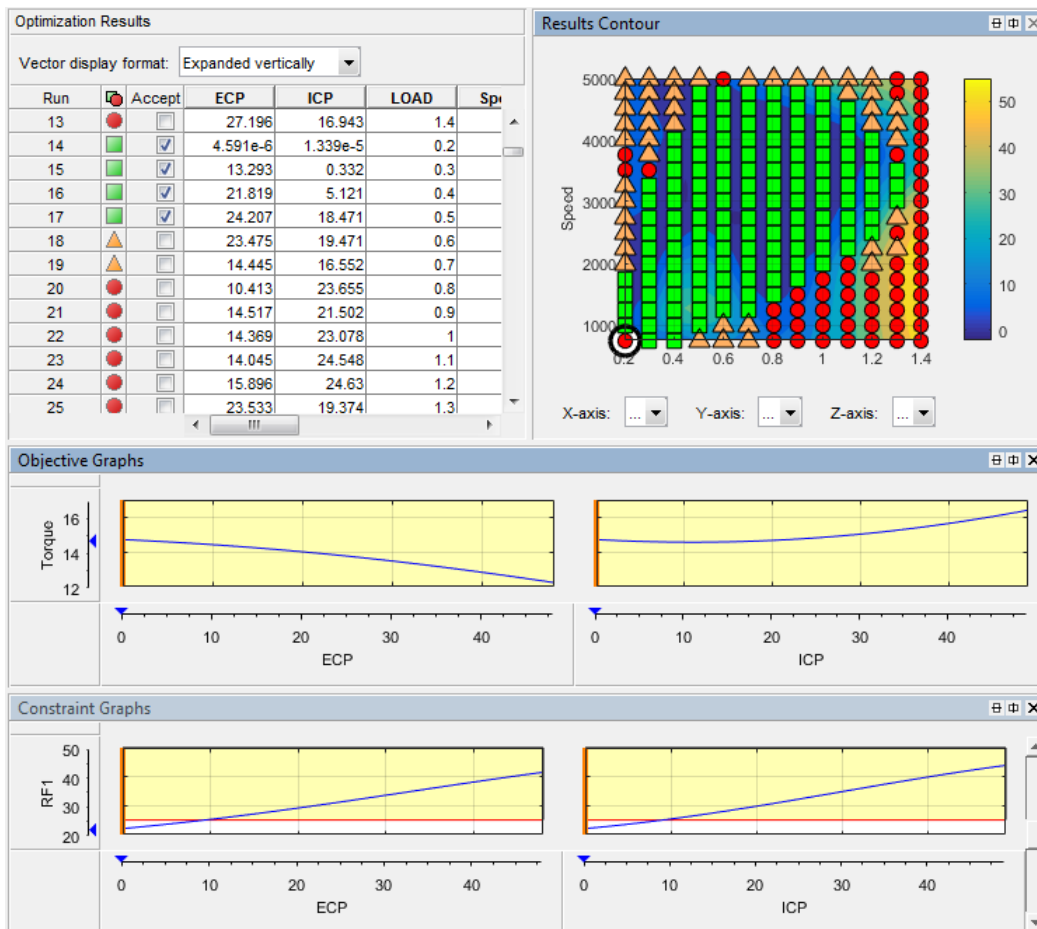
View Optimization Results

- 1 Open MATLAB. On the **Apps** tab, in the **Math, Statistics and Optimization** group, click **MBC Optimization**.
- 2 In the CAGE Browser home page, in the **Case Studies** list, open **Dual CAM gasoline engine with spark optimized during testing**. Alternatively, select **File > Open Project** and browse to the example file `gasolineOneStage.cag`, found in `matlab\toolbox\mbc\mbctraining`.
- 3 In the **Processes** pane, click **Optimization**. Observe two optimizations, `Torque_Optimization` and `Sum_Torque_Optimization`.

Why are there two optimizations? To complete the calibration, you need to start with a point optimization to determine optimal values for the cam timings. You use the results of this point optimization to set up a sum optimization to optimize over the drive cycle and meet gradient constraints to keep the tables smooth.

- 4 In the Optimization pane, ensure `Torque_Optimization` is selected. In the Objectives pane, observe the description: maximize Torque as a function of ECP, ICP, LOAD and Speed.
- 5 In the Constraints pane, observe the constraints: the boundary model of Torque, knock, residual fraction, speed and exhaust temperature. If you want to see how these are set up, double-click a constraint and view settings in the Edit Constraint dialog box.

- 6 In the Optimization Point Set pane, observe the variable values defining the points at which to run the optimization.
- 7 To view the optimization results, in the Optimization pane expand the Torque_Optimization node and select Torque_Optimization_Output.



To learn about analyzing optimization results, see “Choosing Acceptable Solutions”.

- 8 In the Optimization pane, select Sum_Torque_Optimization and compare with the previous point optimization setup. The sum optimization has additional gradient constraints to produce smooth tables, and a single run.

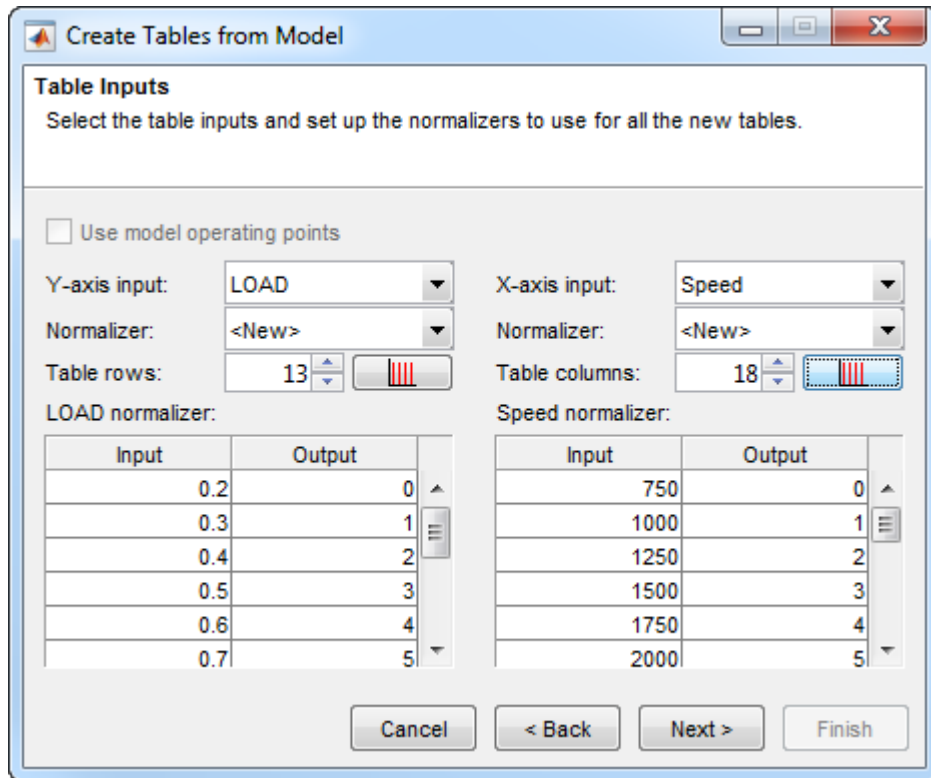
Set Up Optimization

Learn how to set up this optimization.

- 1** To perform an optimization, you need to import the statistical models created in the Model Browser. To see how to import models, in CAGE, select **File > Import From Project**.

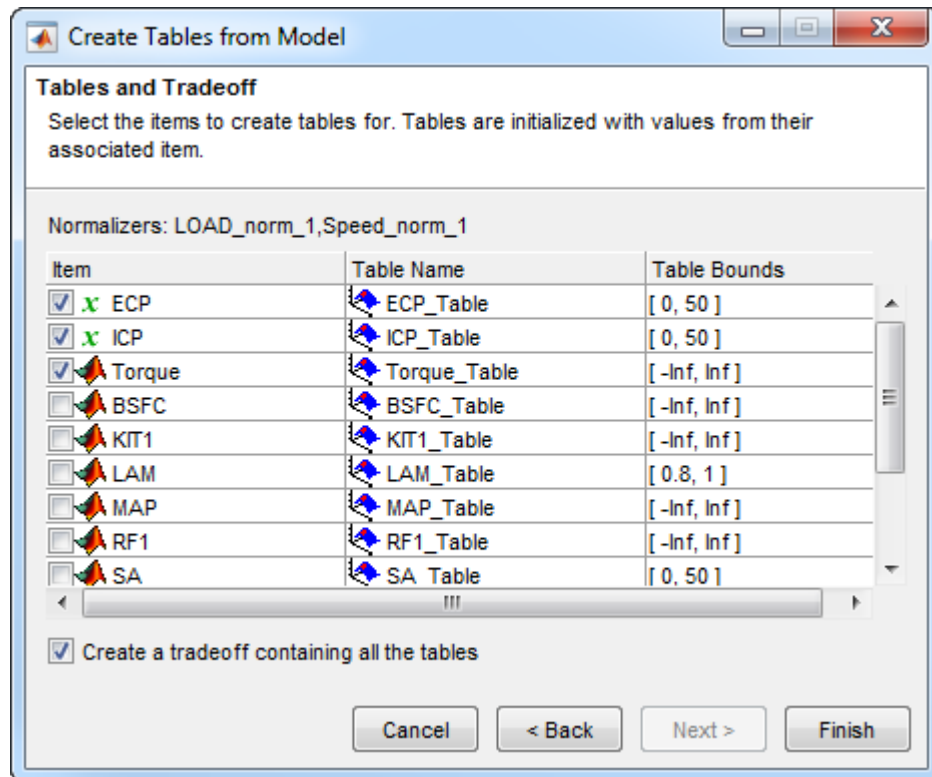
The CAGE Import Tool opens. Here, you can select models to import from a model browser project file or direct from the Model Browser if it is open. However, the toolbox provides an example file with the models already imported, so click **Close** to close the CAGE Import Tool.

- 2** To view the models, click **Models** in the left **Data Objects** pane.
- 3** You need tables to fill with the results of your optimizations. To see all the tables, select the **Tables** view.
- 4** To see how to set up these tables, select **Tools > Create Tables from Model**. The Create Tables from Model wizard appears.
 - a** Select the model **Torque** and click **Next**.
 - b** On the next screen you see the controls for specifying the inputs and size of tables. Click the **Edit breakpoints** buttons to see how to set up row and column values. Click **Next**.



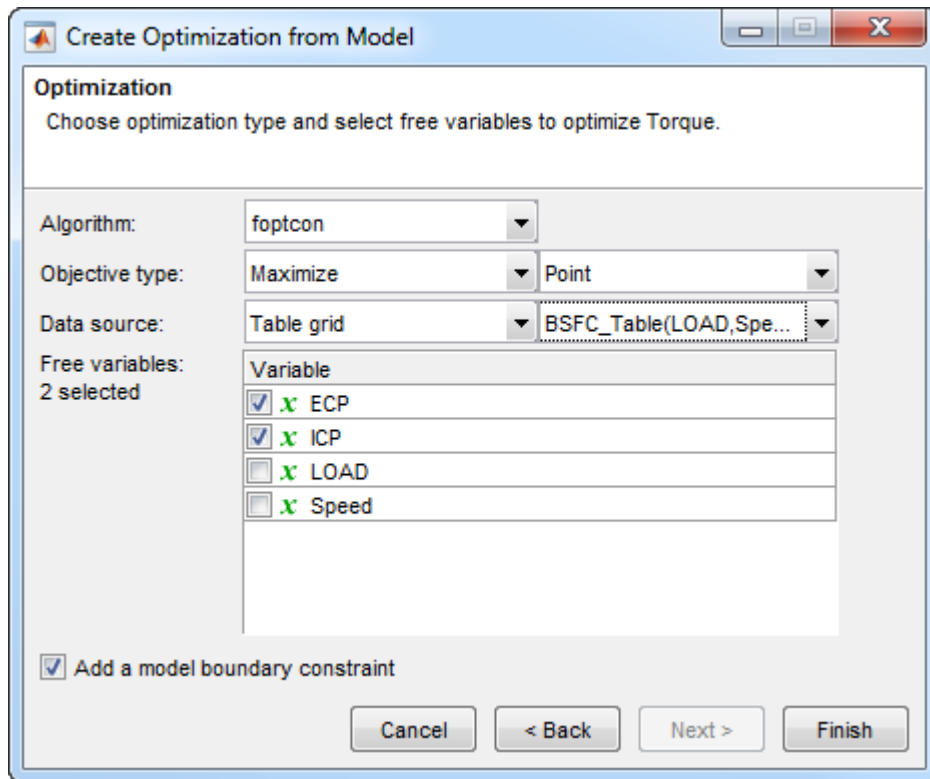
- c On the next screen you can set limits on table values. To edit, double-click **Table Bounds** values.

In this example, bounds on SA, ICP and ECP are set to [0,50], TPP and WAP are [0,100], and LAM is [0.8,1].



Click **Cancel** to avoid creating new unnecessary tables in your example file.

- 5 Learn the easiest way to set up an optimization by selecting **Tools > Create Optimization from Model**.
 - a Select the model **Torque** and click **Next**.
 - b Observe that the default settings will create a point optimization to minimize **Torque**, using 4 free variables, and constrained by a model boundary constraint. To create this example optimization, edit the settings to maximize **Torque**, use the **BSFC** table grid as a data source, and use only **ECP** and **ICP** as free variables. Click **OK** to create the optimization.



- 6 Compare your new optimization with the example Torque_Optimization. To finish the setup you need to add or import the rest of the constraints. In this case, select **Optimization > Constraints > Import Constraints**. Import the constraints from the Torque_Optimization optimization.
- 7 To learn how to set up the sum optimization, from the Torque_Optimization_Output node, select **Solution > Create Sum Optimization**. The toolbox creates a sum optimization for you.
- 8 Compare your new optimization with the example sum optimization, Sum_Torque_Optimization. The example shows you need to add the table smoothness constraints to complete the calibration. To see how to set up table gradient constraints, double-click the constraint GradECP.

To learn more about setting up optimizations and constraints, see:

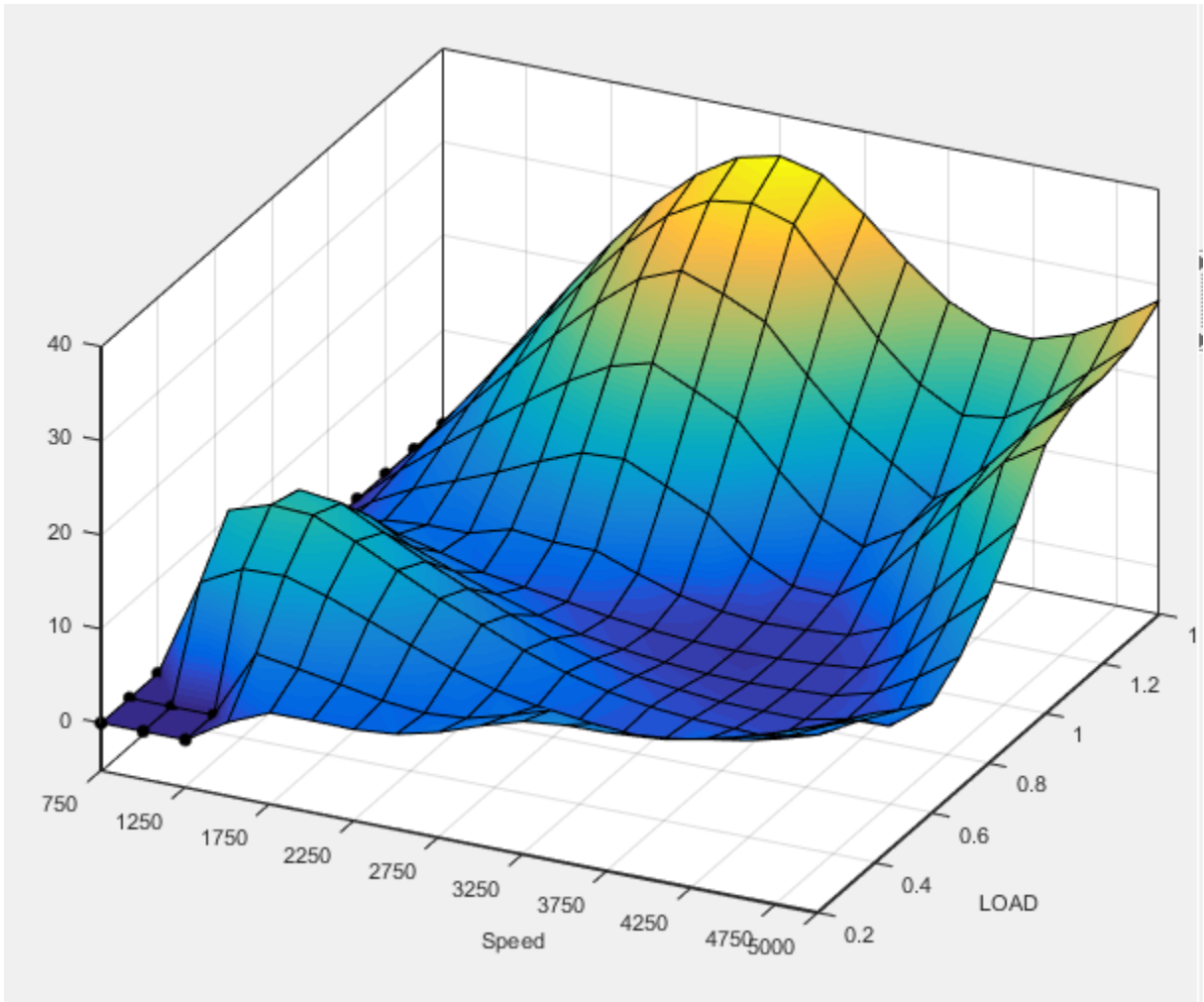
- “Creating Tables from a Model”
- “Creating Optimizations from Models”
- “Edit Objectives and Constraints”
- “Create Sum Optimization from Point Optimization Output”

Filling Tables From Optimization Results

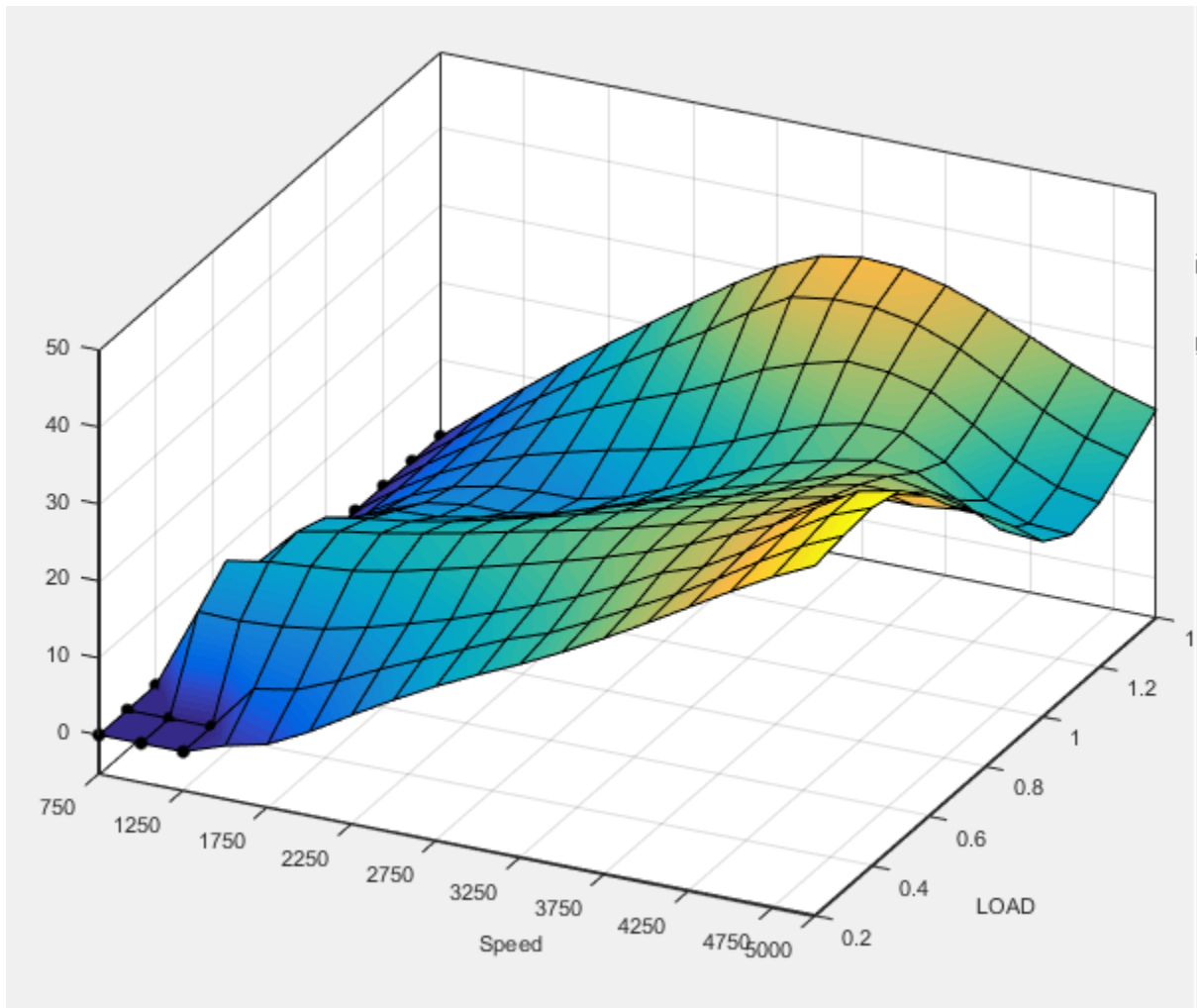
CAGE remembers table filling settings. To view how the example tables are filled:

- 1** Expand the example sum optimization node `Sum_Torque_Optimization` and select the `Sum_Torque_Optimization_Output` node.
- 2** Select **Solution > Fill Tables** (or use the toolbar button) to open the Table Filling from Optimization Results Wizard.
- 3** On the first screen, observe all the tables in the **CAGE tables to be filled list**. Note that the LAM (lambda) table is not filled from the optimization results, because this table was filled from the test data. Click **Next**.
- 4** On the second screen, observe all the tables are matched up with optimization results to fill them with. Click **Next**.
- 5** On the third screen, observe the table filling options. You can either click **Finish** to fill all the tables, or **Cancel** to leave the tables untouched. The example tables are already filled with these settings.

The following plot shows the calibration results for the ICP table. Observe that the idle region has locked cells to keep the cams parked at 0. If you want to lock values like this, to get smooth filled tables, lock the cells before filling from optimization results.



The following plot shows the calibration results for the ECP table.



You can examine all the filled tables in the example project.

To learn more about analyzing and using optimization results, see “Optimization Analysis”.

Related Examples

- “Creating Tables from a Model”

- “Creating Optimizations from Models”
- “Edit Objectives and Constraints”
- “Choosing Acceptable Solutions”
- “Create Sum Optimization from Point Optimization Output”
- “Filling Tables from Optimization Results”

Gasoline Engine Calibration Case Study

This case study provides a step-by-step guide to using the Model-Based Calibration Toolbox product to solve a gasoline engine calibration problem. This section discusses the following topics:

- “Gasoline Case Study Overview” on page 3-2
- “Designing the Experiment” on page 3-6
- “Selecting Data and Models to Fit” on page 3-13
- “Viewing and Filtering Data” on page 3-18
- “How Is a Two-Stage Model Constructed?” on page 3-23
- “Selecting Local Models” on page 3-27
- “Viewing the Boundary Model” on page 3-28
- “Selecting Global and Two-Stage Models” on page 3-33
- “Using Validation Data” on page 3-36
- “Exporting the Models” on page 3-37
- “Optimized Calibration” on page 3-38
- “Importing Additional Models into CAGE” on page 3-41
- “Setting Up Calibration Tables to Fill” on page 3-44
- “Setting Up the Optimization” on page 3-47
- “Defining Variable Values” on page 3-50
- “Running the Optimization” on page 3-52
- “Setting Up the Sum Optimization” on page 3-57
- “Filling Tables with Optimization Results” on page 3-62
- “MBT Spark Estimator Problem” on page 3-63

Gasoline Case Study Overview

In this section...
“Case Study Introduction” on page 3-2
“Why Use Design of Experiment and Engine Modeling?” on page 3-3
“Problem Definition” on page 3-4
“Introduction to Two-Stage Modeling” on page 3-4

Case Study Introduction

This case study demonstrates how to systematically develop a set of optimal steady-state engine calibration tables using the Model-Based Calibration Toolbox product. This case-study uses a 2.2L inline 4 cylinder, naturally aspirated DOHC (Dual Overhead Cams) 4 valve per cylinder spark ignition (SI) engine equipped with dual-independent variable cam-phasing (DIVCP) hardware and electronic throttle.

Optimal steady-state engine calibration tables for intake cam phase, exhaust cam phase, and spark advance are developed as part of the case study process.

This example takes you through the following steps:

- 1** Create a design for your experiment — see “Designing the Experiment” on page 3-6.
- 2** Import the resulting data (taken using the design) and choose responses to model — see “Selecting Data and Models to Fit” on page 3-13
- 3** Examine and filter the data in preparation for modeling, and create the models — see “Viewing and Filtering Data” on page 3-18.
- 4** Evaluate the statistical models based on the data.
 - a** “How Is a Two-Stage Model Constructed?” on page 3-23
 - b** “Selecting Local Models” on page 3-27
 - c** “Viewing the Boundary Model” on page 3-28
 - d** “Selecting Global and Two-Stage Models” on page 3-33
 - e** “Using Validation Data” on page 3-36
- 5** Export these models to the CAGE part of the toolbox to generate optimal calibration tables — see “Exporting the Models” on page 3-37.

- 6** The Model Browser section of the case study involves design of experiment, data handling, and model construction and export. In the CAGE browser section of the case study you use the models to complete the optimization of the calibration tables, see “Optimized Calibration” on page 3-38.

The following sections introduce the benefits of applying model-based calibration methods to solve this case study problem:

- “Why Use Design of Experiment and Engine Modeling?” on page 3-3
- “Problem Definition” on page 3-4
- “Introduction to Two-Stage Modeling” on page 3-4

Why Use Design of Experiment and Engine Modeling?

These approaches can be used to ensure that you develop optimal engine calibrations for complex engines with many controllable variables (such as variable valve timing, variable valve lift, and cylinder deactivation) at minimum cost and time.

Test bed time is expensive, and Design of Experiments methodology can help you choose the most effective points to run to get the maximum information in the shortest time. You can break the exponential dependency between the complexity of the engine (number of inputs) and the cost of testing (number of tests). You can collect the most statistically useful data, and just enough of it to fit the models.

Experimental design test points can be constrained based on previous experience to avoid damaging expensive engine hardware prototypes at unrealistic operating points.

The act of statistically modeling engine data can help identify the effect of interactions between calibration settings and engine performance, which can be vital to understanding how to optimally meet emissions constraints.

Accurate statistical models of engine data can also be used to develop calibration tables that have smooth transitions between the operating range of the engine and the edge regions of calibration tables where the engine will not be operated.

Optimal calibrations can be generated from statistical engine models in a methodical, repeatable process to ensure that maximum performance is achieved subject to emissions, driveability, and material limit constraints.

Problem Definition

The aim of this case study is to produce optimized tables for

- Intake Cam Phase
- Exhaust Cam Phase
- Spark Timing Schedules

as a function of Load and rpm, subject to the following constraints

- Constrain solutions to lie within the boundary constraint model (to keep the engine within its operating region)
- Constrain cam phase solutions so they do not change by more than 10° between table cells (that is, no more than 10° per 500 RPM change and per 0.1 load change).
- Constrain residual fraction $\leq 25\%$ at each drive cycle point (to ensure stable combustion). Residual fraction is the percentage of burned gas mass in the cylinder at intake valve close, relative to the total mass in the cylinder at intake valve close. Constraining maximum residual fraction is a simple and reasonable way of ensuring stable combustion. Residual fraction = $100 * \text{Burned Gas Mass from Last Cycle} / (\text{Burned Gas Mass From Last Cycle} + \text{Fresh Air Mass})$

To produce these tables, you need to make accurate models of the behavior of torque, exhaust temperature, and residual fraction at different values of speed, throttle area, spark, and cam timings. You need engine data to build these models, so the first step is constructing an experimental design to collect the most useful set of points.

Before you can design an experiment you need to set up a two-stage test plan and define your model inputs and model type.

Introduction to Two-Stage Modeling

What is a two-stage test plan? You use a test plan to set up models in the Model Browser. The two stages refer to the way that engine data is often collected. For example, in each test, spark (the local variable) is swept while the other variables (such as speed and load) are held constant — these are referred to as global variables. Each test is taken at a different point in the global variables. Building the statistical models to take into account these individual sweeps makes it possible to incorporate engineering knowledge in the process. You can see plots of torque/spark sweeps, and use variables such as MBT (maximum brake torque) in modeling, rather than solely abstract mathematical

properties of curves. You can then apply previous knowledge about the expected behavior of these variables to help you select good models.

You can easily identify outliers when you can see the sweep in which they were taken. The Model Browser allows you to visualize the data in a way that can help you identify and investigate suspect sweeps, and decide what kind of models will produce the best fit to the shapes of the data. The more controllable variables there are in an engine the more useful it is to have these visual aids to investigate complex data. Constructing models to take into account the way the data is collected helps build good models that you can have more confidence in. Statistically, it is the correct thing to do as it allows you to partition the errors within sweeps and the errors between sweeps separately.

You use a two-stage test plan to build your models because this data is suited to it. Spark is varied as the other variables are held constant, so the data is collected in a hierarchical structure; your models attempt to capture this information. You come to more detail on how this two-stage model is constructed after creating a design and obtaining data.

Designing the Experiment

In this section...

“Overview of Design Process” on page 3-6

“Specifying Model Inputs” on page 3-6

“Creating Designs” on page 3-8

“Data Source” on page 3-11

Overview of Design Process

Creating a design in the Model Browser comprises several steps. You need to open the tool, select a two-stage template, and enter the ranges and names of the input variables. Then you can create an initial design and set up the constraints on the input space. These constraints will be the same for all designs. From this constrained design, a series of child designs can be made with varying numbers of points added and slightly different models used. The final design can be chosen by comparing statistics of the various child designs and considering how many points you can afford to run. These steps are described next.

Specifying Model Inputs

The example project contains finished designs and models in `Gasoline_project.mat`. You can open the example from the home page or from `matlab\toolbox\mbc\mbctraining`.

To see how to set up model inputs, follow these steps.

- 1 In MATLAB, on the **Apps** tab, in the **Math, Statistics and Optimization** group, click **MBC Model Fitting**.
- 2 On the home page, click **Design Experiment**.

The **New Test Plan** dialog box appears.

3



Click the **Two-Stage** test plan icon Two-Stage in the Template pane. A two-stage model fits a model to data with a hierarchical structure.

The models you are building are intended to predict the torque, fuel flow, and manifold pressure of the engine as a function of spark angle at specified operating points defined by the engine's speed, load, and cam timings. The input to the local model is the spark angle.

- 4 Set up your local model input.
 - a Set **Symbol** to **S**.
 - b Set **Signal** to **SPARK**. This is optional and matches the raw data.
 - c Set the range you want to model by changing **Max** to **50** (and leave **Min** at 0).
- 5 Set up your global inputs. The global inputs are the variables that are held constant at each operating point while spark is swept. In this case, these global variables are engine speed, scaled throttle area, intake cam angle, and exhaust cam angle.

By default, there is one input to the global model. Because this engine model has four input factors, you need to edit the input factors as follows:

- a Click the up arrow button to increase the **Number of factors** setting to four.
- b Edit the four factors to create the engine model input. In each case, change the symbols, signal names, and ranges to the following:

Symbol	Signal	Min	Max
N	SPEED	500	6000
L	LOAD	0.05	0.95
ICP	INT_ADV	-5	50
ECP	EXH_RET	-5	50

Load = aircharge/maximum aircharge.

Cam angles are in units of degrees crankshaft, with intake values indicating advance from base timing, and exhaust values indicating retard from base timing.

- c Click **OK** to dismiss the dialog box.

The Model Browser displays the test plan diagram with your specified local and global model inputs, and opens the Design Editor.

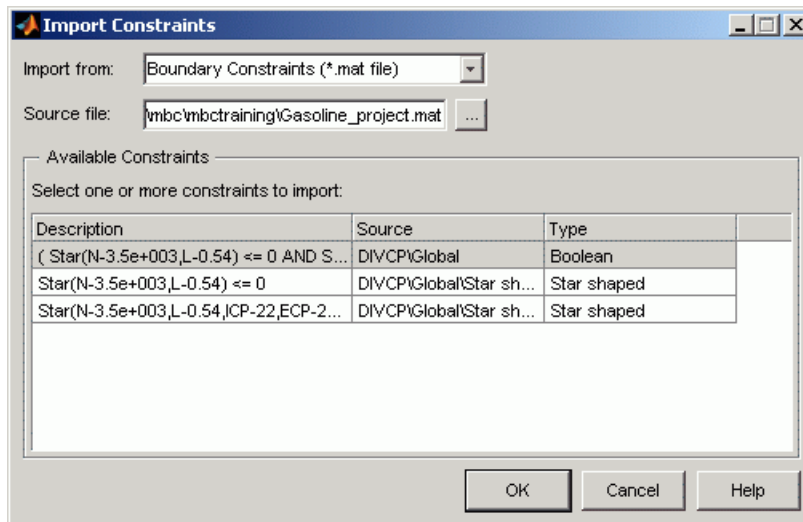
Creating Designs

Now you have set up the modeling test plan you can create an initial design and set up the constraints on the input space — these will be the same for all designs. From this constrained design, a series of child designs can be made with varying numbers of points added and slightly different models used. The final design can be chosen by comparing statistics of the various child designs and considering how many points you can afford to run.

- 1 You will import a boundary model from an example file. In this way, you can use a boundary constraint from a previous investigation on a similar engine to constrain new designs. To locate example files, in MATLAB, change the current folder to `mbctraining`:

```
cd(fullfile(matlabroot, 'toolbox', 'mbc', 'mbctraining'))
```

- 2 In the Design Editor, in the Common Tasks, click **Constraints**.
- 3 In the Constraints Manager dialog box, click **Import**.
- 4 In the Import Constraints dialog box, in the **Import from** list, select **Boundary Constraints (.mat file)**.
- 5 Browse to the file `Gasoline_project.mat`, found in `matlab\toolbox\mbc\mbctraining`.
- 6 Click to select the **Boolean** type constraint. This is the combination of both boundary constraint models.



- 7 Click **OK** to import the boundary constraint.

Click **OK** in the following data matching dialog as all the signal names are automatically selected in this case, and close the Constraints Manager.

- 8 In the Design Editor, in the Common Tasks, click **Create Space Filling Design**.

The Space Filling Design Browser appears. Space-filling designs are best when there is little or no information about the underlying effects of factors on responses. For example, they are most useful when you are faced with a new type of engine, with little knowledge of the operating envelope. These designs do not assume a particular model form. The aim is to spread the points as evenly as possible around the operating space. Space-filling designs are also best for radial basis function models. You can use a mix-and-match approach: start with a space-filling design to survey the space, then continue testing with an optimal design once you have more understanding of the response and constraints. Once you have an idea of what model type will fit the response best, you can optimally add points in the most efficient places for the most robust model fit.


The most important thing to decide is how many design points you want. Testing is expensive and time-consuming, so you need to bear in mind how many points you have time for. When you consider the number of points, you also need to remember

that a sweep will be done at each point and this will take some time. Do you need to allow time to fix problems or redo experimental points that can't be achieved?


- 9 Enter **200** for the **Number of points** and press **Enter**. A space-filling design is constructed, using the sobol sequence method. Click the 3-D and 2-D tabs to examine the preview plots of new design distribution. The preview is identical to the final design.

Note: You will import the example design for this case study in a later step.

Click **OK** when you are satisfied with the design.

- 10 Right-click a view and select **Split View > 3D Design Projection** to view the design points.
- 11 Examine the constrained design space by right-clicking the title bar of a design view and selecting **Current View > 3D Constraints**.
- 12 Add another space-filling design for some points with parked cam phasers. These points are important because we need an accurate model when cams are parked.
 - a Click the  button in the toolbar to add a new design. The view switches to the new design in the tree. Rename it **CAM_Parked**. The design inherits the same constraints as the parent design.
 - b Select **Design > Space Filling > Design Browser**, or click the **Space Filling Design** button on the toolbar.

In the dialog that appears, leave the default choice to replace the current points with a new design and click **OK**.
 - c Enter a number of points of **10**, press **Enter**, and click **OK**.
 - d Display the design as a table, and edit the values of **ICP** and **ECP** in the new design to be zero.
- 13 Select **File > Merge Designs**. Select both your designs in the list, leave the default **Merge to new design** option button selected, and click **OK**. Examine the merged design in the table view to confirm the parked cam points (**ICP** and **ECP** values of **0**) are at the end of the list of design points.
- 14 Add another space-filling design for collecting validation data.

- a Click the  button in the toolbar to add a new design. The view switches to the new design in the tree. You could rename it **Validation**. The design inherits the same constraints as the parent design.
- b Select **Design > Space Filling > Design Browser**, or click the **Space Filling Design** button on the toolbar.

In the dialog that appears, choose to replace the current points with a new design and click **OK**.

- c Enter a number of points of **25**, press **Enter**, and click **OK**.
- d To add a point where the cams are parked, select **Edit > Add Point**.

In the dialog that appears, select **User-specified** from the **Augment method** list, edit **ICP** and **ECP** to zero, and click **OK**.

You can use the Design Editor to make a selection of child designs to compare. When you have chosen the best design you can export it to file. In this case, you can import the example design for this case study. This design was used to collect the data for this case study, and you will later match these design points to data. Import the design as follows:

- 1 Select **Edit > Constraints**. In the Constraints Manager dialog box, click **Import**.
- 2 In the Import Constraints dialog box, in the **Import from** list, select **Design Editor file (.mvd)**.
- 3 Browse to the design file **DIVCP.mvd**, found in **matlab\toolbox\mbc\mbctraining**, select it and click **Open**, then click **OK** to import the design. A new design node appears.
- 4 Rename the imported design (click, and press **F2**) **DIVCP**. You can right-click to change the Current View to examine the design points in 2D and 3D.

If you wanted to **Match selected data to design** when fitting models to data in later steps, you must select a design as best to use it. For this example, do not select any design as best.

- 5 Close the Design Editor.

Data Source

The data was collected using a constrained space-filling design on speed, load, intake cam phase, and exhaust cam phase. The points specified in the design were measured

using the GT-Power engine simulation tool from Gamma Technologies (see <http://www.gtisoft.com>).

Simulink and StateFlow[®] simulation tools controlled the GT-Power model, running on a cluster of 14 desktop PC machines. The simulation time for the testing was 20 hours. The GT-Power model used predictive combustion, which gives good realistic results but is computationally expensive. Ten cycles were run at each spark advance setting after attaining steady-state speed/load conditions.

For next steps, see “Selecting Data and Models to Fit” on page 3-13.

Selecting Data and Models to Fit

Remember that the aim of this case study is to produce optimized tables for:

- Intake cam phase
- Exhaust cam phase
- Spark timing schedules

These tables are all functions of load and rpm, subject to constraints of operating region and residual fraction.

To produce these tables, you need to make accurate models of the behavior of torque, exhaust temperature, and residual fraction at different values of speed, load, spark, and cam timings. You have set the local model input as spark, and the global model inputs as engine speed, load, intake cam phase, and exhaust cam phase. Therefore, the responses you want to model are:

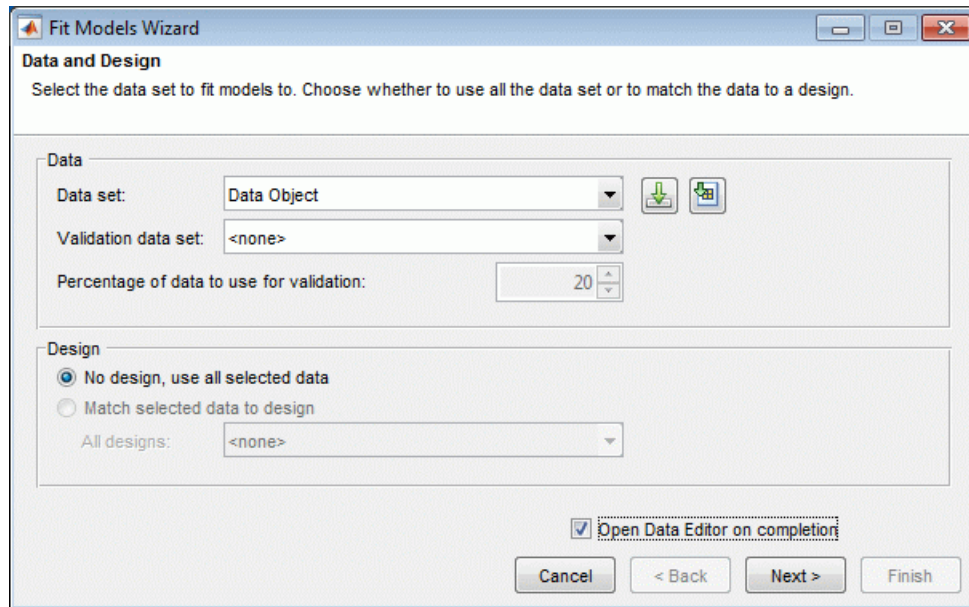
- Torque
- Exhaust Temperature
- Residual fraction

Follow these steps to select data and model types to fit.

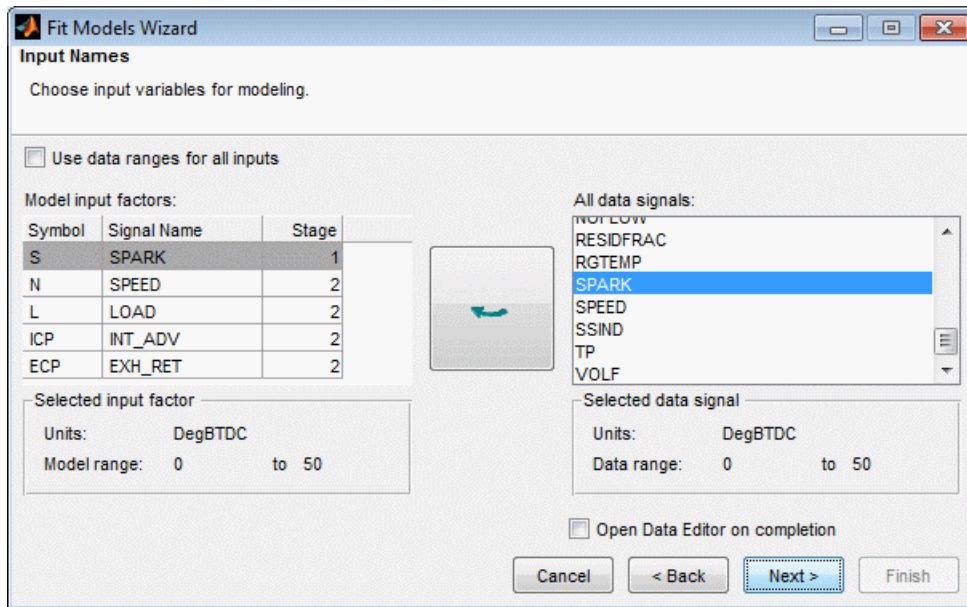
- 1** In the Model Browser, click the test plan node, **Two -Stage**, in the **All Models** tree to go to the test plan view.
- 2** In the **Common Tasks** pane, click **Fit models**.

The Fit Models Wizard opens.

- 3** In the **Data** pane, next to **Data set**, use the browse button to find and select the **DIVCP_Main_DoE_Data.xls** data file, found in `matlab\toolbox\mbc\mbctraining`. This example data file results from the example experimental design. Click **Open**.
- 4** In the Fit Models Wizard, select the **Open Data Editor on completion** check box.

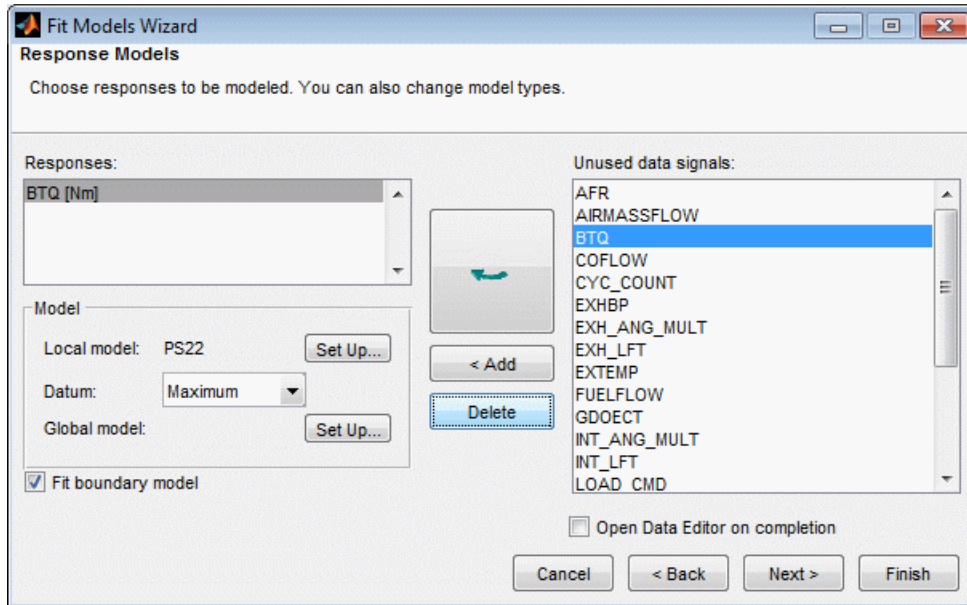


- 5 Click **Next**.
- 6 The wizard tries to match each symbol to a data signal (shown as **Signal Name** in the model input factors list). Make sure **SPARK**, **SPEED**, **LOAD**, **INT_ADV** and **EXH_RET** are selected, as shown.



Click **Next**

- 7 On this screen you select the responses to model. Select BTQ as the first response you want to model and click **Add**.

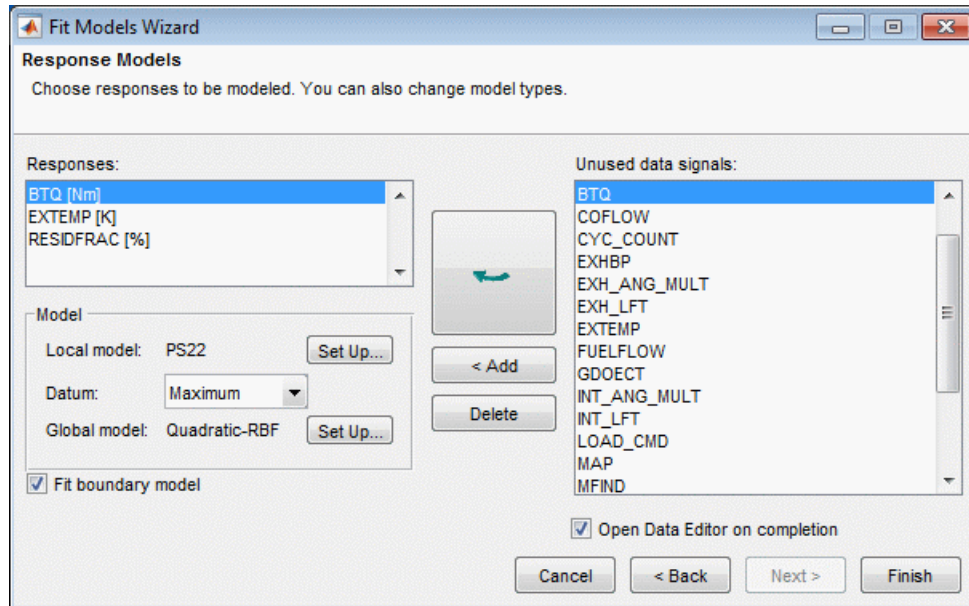


8 Next to **Local model**, click **Set Up**.

The first response you want to model is torque against spark. The shape of torque/spark curves is well understood and you can examine them in the Data Editor in later steps. Polynomial spline curves are useful for fitting these shapes, where different curvature is required above and below the maximum. Therefore, you should set the local model type to polynomial spline. A spline is a curve made up of pieces of polynomial, joined smoothly together. The points of the joins are called knots. In this case, there is only one knot, at the maximum. The location of the knot in this case marks MBT.

- a** In the Local Model Setup dialog box, select **Polynomial Spline** from the **Local Model Class** list.
 - b** Set **Spline Order** to 2 below and 2 above knot.
 - c** Click **OK**.
- 9** In the Fit Models Wizard, select **Maximum** from the **Datum** drop-down menu. In this case, the maximum of the torque/spark curves is MBT (spark angle at maximum brake torque), so this can be a useful feature to model.

- 10 Add two more responses to model. Select **EXTEMP** and click **Add**, then select **RESIDFRAC** and click **Add**.



- 11 Click **Finish**.

The Test Groupings dialog box appears so you can select tests to model.

- 12 In the Test Groupings dialog box, edit the **Tolerance** for **LOAD** to **0.05**. You should see 203 tests defined. Close the test Groupings dialog box.
- 13 Click **OK** to close the Define Test Groupings dialog box.

The Data Editor appears so you can inspect and filter data for modeling. The response models will be built when you close the Data Editor. For next steps, see “Viewing and Filtering Data” on page 3-18.

Viewing and Filtering Data

In this section...
“View Data” on page 3-18
“Filter Data” on page 3-21

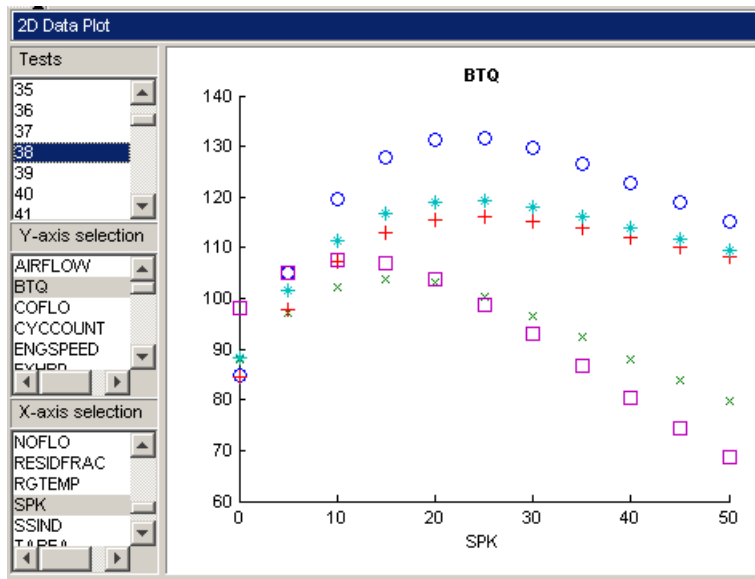
View Data

- 1 Examine the data you imported by following the steps in “Selecting Data and Models to Fit” on page 3-13.

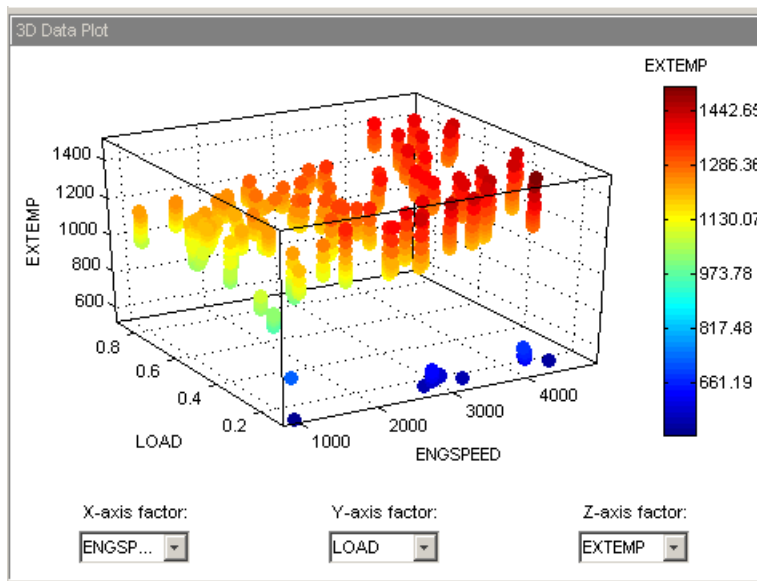
The Data Editor is a powerful tool for displaying and sorting your data. You can use the right-click context menu to split the views, or use the toolbar buttons or the **View** menu. You can choose 2-D plots, 3-D plots, multiple data plots, data tables, and tab views of filters, variables, test filters, and test notes.

For example, if you do not already have a 2-D plot, right-click the title bar of any plot and select **Current View > 2-D Data Plot**.

- 2 Click in the left lists to plot torque (BTQ) against spark (SPARK), then select one or more tests to display.



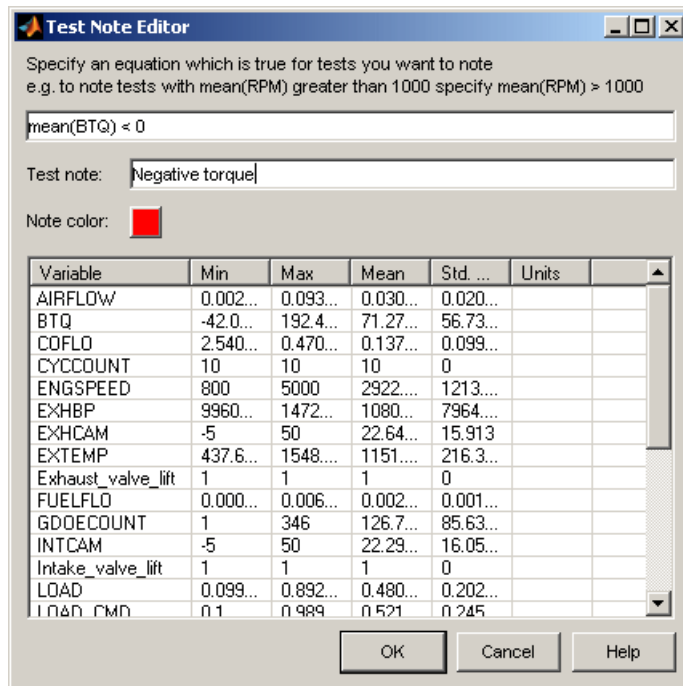
- Right-click and select **Split View > 3-D Data Plot** to split the currently selected view and add a 3-D plot. Select one or more tests to display in the list at the left of the Data Editor, then choose three variables for the axes.



- 4 Click the Test Notes tab. This is empty until you add any test notes.
- 5 Select **Tools > Test Notes > Add**.

The Test Note editor appears.

- a Enter an expression that defines the tests you want to note; for example $\text{mean}(\text{BTQ}) < 0$ will evaluate the mean torque for each test and note those tests where the value is less than zero.
- b Enter the text for this note in the edit box, e.g., Negative Torque.



- c Click **OK** to apply the test note.
- 6 Note that the new test note appears in the test selector pane. You can sort the column for the **Negative Torque** note by clicking the column header (once to sort ascending and once more to sort descending). This allows you to quickly identify which tests satisfy the note definition. Investigate these test points in the other views. Select a test and that test is displayed in the table view, 3-D plot, and multiple data plots views (but not the 2-D plots, which have their own test selection controls). If you select multiple tests, they are all shown in the data plots, but only the first test in the list is highlighted in the table view.

Filter Data

- 1 If you decide certain operating points are unsuitable for modeling, for instance, unstable points on the edge of the engine's operating envelope, you can use the Data Editor tools to help you identify and remove them. You can remove suspect data in the following ways:

- a You can remove individual points by selecting them as outliers in 2D and multiple data plots by clicking, then selecting **Tools > Filters > Remove Outliers**. You can always replace them again with the **Tools** menu.
- b You can define filters to remove all data points that do not fulfill a certain expression. Select **Tools > Filters**. The Filter Editor appears.
- c You can enter an expression that defines the records you want to keep. You can use any MATLAB function for filtering. Enter the following expression, which keeps records with AFR value greater than 14.25:

AFR>14.25

Click **OK**. Observe red records that have been filtered out in the top information bars in the Summary tab. If you have turned on the option to **Allow Editing** in the data table view you can also see removed records in red.

- d Define another filter. Select **Tools > Filters**. The Filter Editor appears.
- e Enter the following expression, which keeps records with RESIDFRAC value less than 35:

RESIDFRAC<35

- f You can exclude whole tests by defining test filters to remove them. Select **Tools > Test Filters**. The Test Filter Editor appears.

You want to keep only those tests with sufficient points to fit the model (at least 5 points).

Enter `length(BTQ)>4` and click **OK**.

- 2 Click the Filters and Test Filters tabs to see whether the filters have been successfully applied and how many records or tests are removed by each filter.

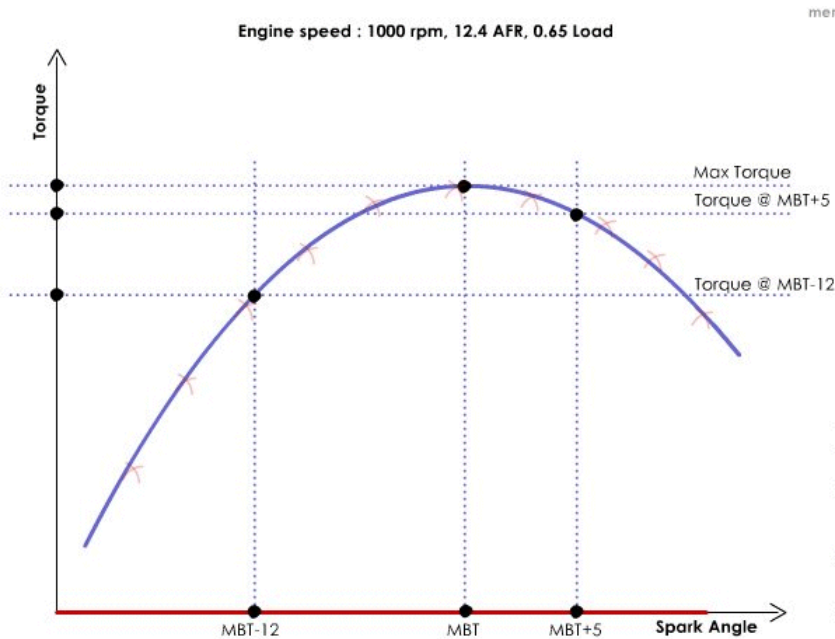
The bars in the Summary tab at the top of the Data Editor display the total numbers and proportion of removed data.

- 3 Close the Data Editor. A dialog appears to check that you want to build the response models and update the **Actual Design** to include all data selected for modeling. Click **Yes**, and the models are created.

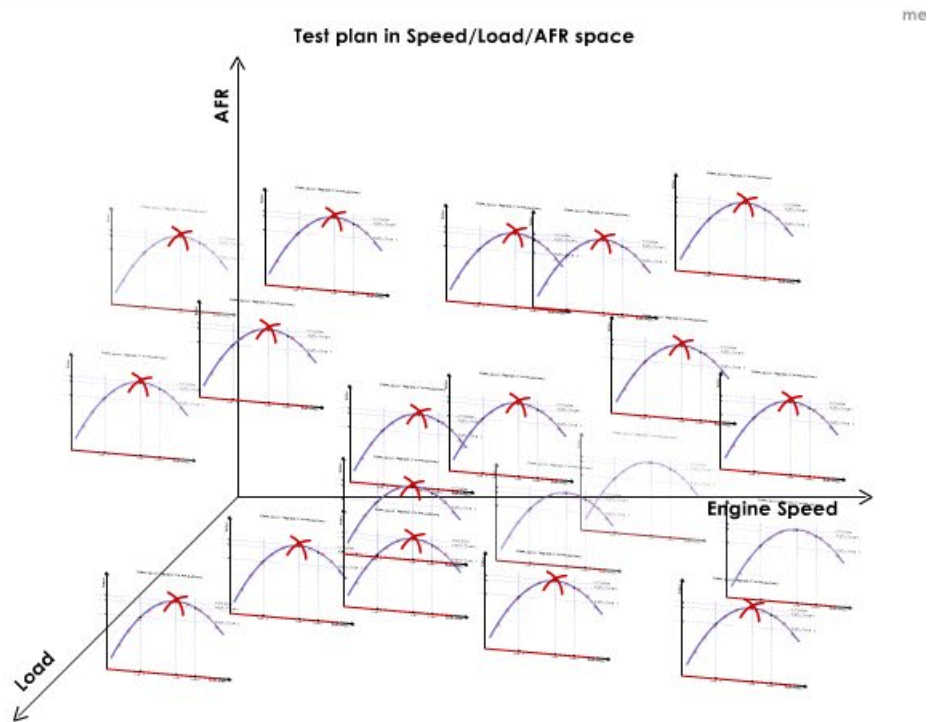
For next steps, see “Selecting Local Models” on page 3-27.

How Is a Two-Stage Model Constructed?

Local models find the best fit of a curve to the data in each test. Each test in this case is a sweep of torque against spark angle, with speed, load, and cams held at a constant value for each sweep. The following illustrates a single sweep with a local model fitted.

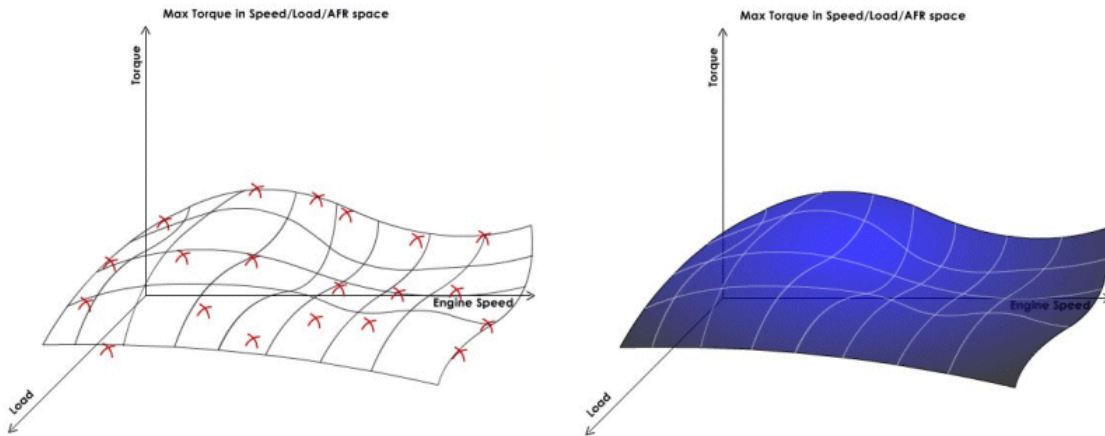


The local models provide the coefficients to generate global models. The equations describing those local model curves have certain coefficients such as max and knot, which for this data are peak torque and MBT spark (the spark angle that generates maximum brake torque).



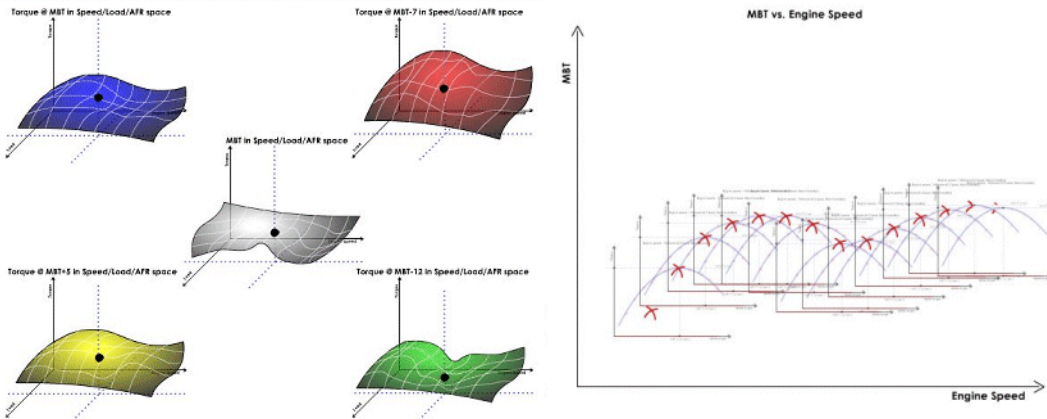
Local models are fitted to each test, in different places across the global space, as illustrated above. Each local model has coefficients for MBT and peak torque, etc. These coefficients become the data to which the global models are fitted. Coefficients such as peak torque and MBT are used to make the second stage of modeling more intuitive; an engineer will have a much better understanding of how a feature such as MBT spark varies through the global factor space than some esoteric curve fit parameter. Familiar variables like these are helpful to engineers trying to decide how well a model describes engine behavior. Better intuitive understanding allows much greater confidence in your models.

Global models are the best fit of a curve to the values of, for example, MBT for each test. This is repeated for each coefficient, producing several global models fitted to different coefficients of the local models. These coefficients are referred to as response features of the local models. The following example shows a global model for maximum torque across the speed/load global space.



The two-stage model is a surface fitted across all the global models, to describe the behavior across all global variables.

It can be useful to think of local and global models as a series of 2-D slices, while the two-stage model fits a 3-D surface across the curves of the global model slices. It is difficult to visualize more dimensions! The following example shows a variety of 3-D plots of global models for properties of the local torque/spark curves (such as MBT, peak torque, and torque a number of degrees before and after MBT), showing how these properties vary across the speed/load global space. The 2-D plot of the global MBT model (on the right) demonstrates how MBT varies with engine speed.



The two-stage model can take values of each coefficient at a certain value of, say, speed, to generate a new curve of torque against spark. This is a slice through the two-stage model surface.

In other words, you can test your two-stage model by comparing it with the local fit and with the data. For example, you can reconstruct a local torque/spark curve at an operating point by taking the values of MBT and peak torque and the curvature from the two-stage model, and then validate this reconstructed curve against the original fit and the data. The two-stage model can also predict responses between tests, for new sweeps at intermediate values for which there is no data. If the two-stage model shows an accurate fit when compared to the local sweeps, this is a good sign that the engine behavior is well described by the model across the global variables.

For more details on two-stage modeling, see “Two-Stage Modeling Example” on page 9-3, and see “Two-Stage Models for Engines” in the Model Browser documentation for more statistical depth.

For next steps, see “Selecting Local Models” on page 3-27.

Selecting Local Models

You should inspect the local and global models in turn, removing outliers if appropriate and trying different model types, before creating a two stage model. If the fit is good at the local and global levels you have the best chance of creating a two-stage model that accurately predicts the engine behavior. First, inspect the local models.

- 1 Select the local model node **PS22** in the model tree, in the **All Models** pane, and the local model view appears on the right.
- 2 Look through the tests to inspect the fits. Use the **Test** controls.
- 3 To quickly identify problem tests, look at the RMSE Plot.

Inspect tests with high error values in the local model view. You can navigate to a test of interest from the RMSE plot by double-clicking a point in the plot to select the test in the other plots. Plot all response features in the RMSE plot and investigate tests with extreme values.

- 4 Consider removing outliers to improve fits if some points are badly distorting the torque spark curve (use right-click or **Outliers** menu). Pay particular attention to end points. For example, for tests where the majority of points are at higher spark angles than the maximum (at MBT), it can improve the fit to remove some of these long “tails”. It can be useful to remove outliers in this region, because there is likely to be knock at spark values much higher than MBT where the engine is less stable. Similarly, as there is no knock in simulation data, points can be collected far in advance of MBT, and it can improve the fit to remove these.
- 5 If removing some outliers does not bring MBT within the range of the data points, consider removing the whole test (use the **Outliers** menu).
- 6 After making changes, check the RMSE plots again for problem tests.
- 7 Check all tests before inspecting the global models. Try looking at the **Local statistics**. Check for high values of **Cond(J)** (e.g., $> 10^8$). High values of this condition indicator can be a sign of numerical instability.

For more advice on inspecting local models, see “Assess Local Models”.

Optionally, to inspect the boundary model, see “Viewing the Boundary Model” on page 3-28.

For next steps, see “Selecting Global and Two-Stage Models” on page 3-33.

Viewing the Boundary Model

Note: The Fit Models Wizard automatically created a boundary model for you.

You can view and create boundary models at the test plan node. A model describing the limits of the operating envelope can be useful when you are evaluating optimization results and global models. The boundary model is useful for viewing global models, so you can see the model areas inside the boundary on plots.


To view the boundary model setup:

- 1 Select the test plan node in the model tree.
- 2 Select **TestPlan > Fit Boundary**.

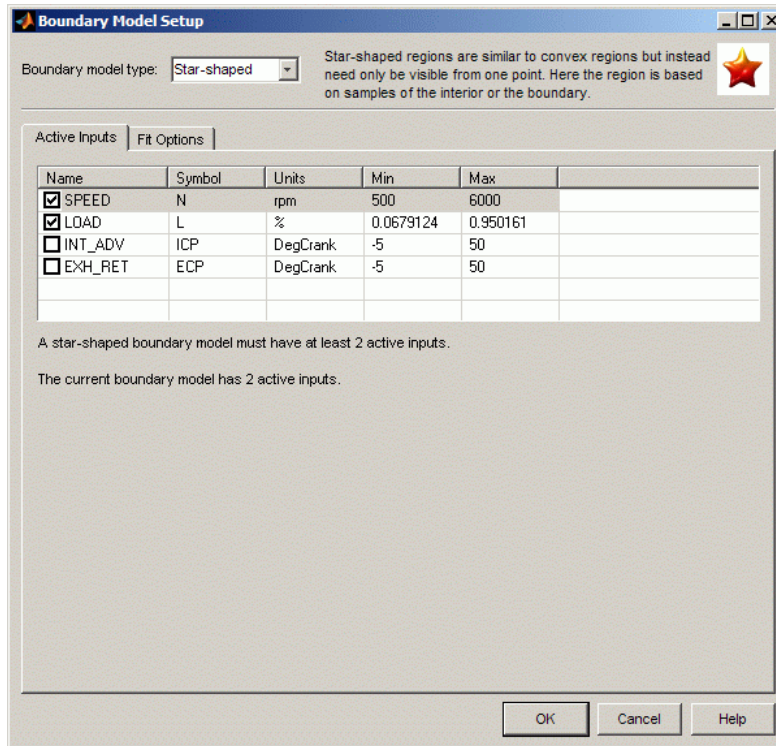
The Boundary Editor opens.

- 3 View the boundary models.

(Optional) The following steps show you how to build two boundary models and combine them, to map the envelope in speed and load, and also in all four inputs.

- 1 (Optional) Select the root Two-Stage node and click New Boundary Model  in the toolbar. A dialog opens where you can choose to build a boundary model of the response, local, or global inputs. In this case, you are interested only in making a model of the global boundary. You are not interested in the **Local** boundary (you know the range of spark already). Select **Global** and click **OK**.

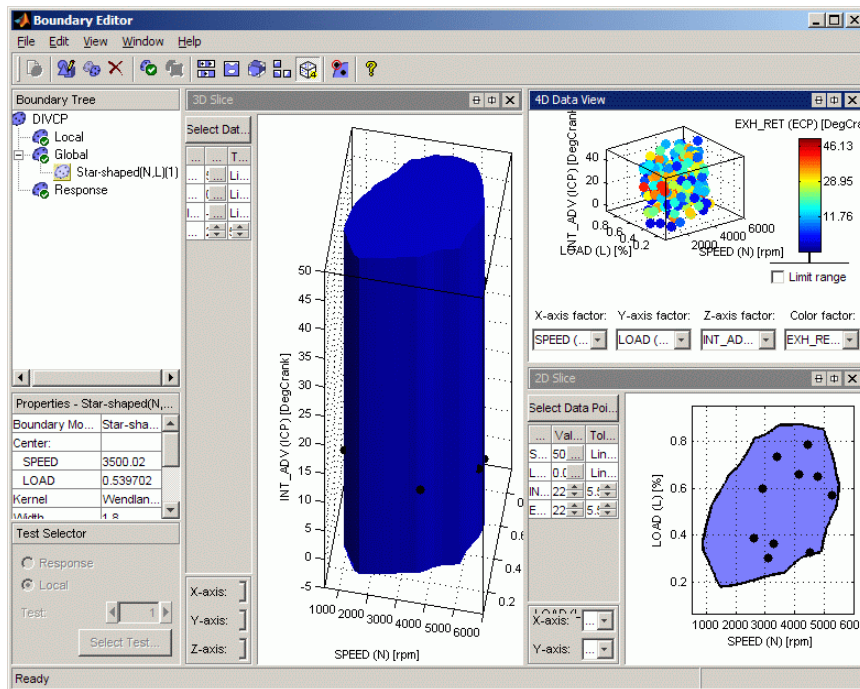
A dialog opens where you can select constraint inputs.



- 2 Leave the **Constraint type** set to **Star shaped** and leave only the **SPEED** and **LOAD** check boxes selected. Click **OK** and the boundary constraint is calculated.

A **Star shaped(N, L)** child node appears under the **Global** node and the view switches to the new model.

- 3 Select **View > Current View > 3D Slice** (or use the toolbar button) to examine the shape of the new boundary model. Drag the axes to rotate the plot. Use the drop-down menus to change the variables plotted. In any view, you can change the position of the plotted slice by altering the variable values in the edit boxes. You can split the views as in the Design and Data Editors using the buttons in the title bars, or the right-click or **View** menu.



4



Click Show Boundary Points in the toolbar. Boundary points are outlined in red. In some projections and at some plot resolutions, points can falsely appear to be outside the boundary. You can alter the number of points plotted in the Value edit boxes, but high resolutions can be time-consuming to plot. You can also check particular points: in the 3D, 2D, and 1D Slice views you can click (and hold) points to see the values of the global variables at that point and the Distance. A distance of 0 means the point is exactly on the boundary, and negative values show the distance inside the boundary. You can use this function to check that enough points are inside or close to the boundary.


5

In a 2D or 3D Slice view, click the button **Select Data Point**. A dialog appears where you can select a data point. When you click **OK** the slice is plotted at the location of the selected data point. You can also double-click points to move the slice to that location.

- 6 Select **View > Current View > Pairwise Projections** (or use the toolbar button). Here you can see pairwise projections to view the boundary across all combinations of factors.

Note: If some points appear to be outside the boundary, select **View > Set Resolution** to increase the resolution of the pairwise plots. Increasing the number of evaluation points displays more detail, but takes longer to calculate.

- 7 On one of the pairwise plots, click and drag to select a small area of points. The area you select is colored yellow across all plots, so you can view how those points are distributed across factors. With some detailed surfaces, areas in the pairwise plots can appear as discrete patches, so this feature is useful for tracking regions across factors. This can help you decide whether the boundary is capturing enough (or too much) detail of the surface.

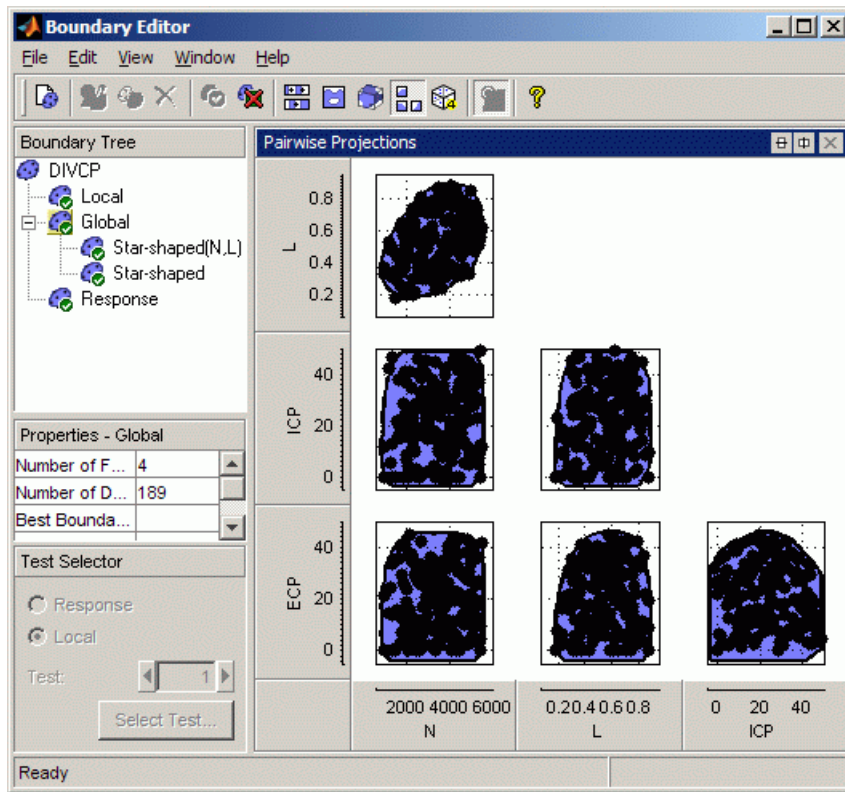
- 8 Select the Global node, and click New Boundary Model  in the toolbar. You can build other boundary models to compare and combine for the most useful model.

From the Global node you will automatically get a global child node, so you do not need to select a level. The Boundary Model Setup dialog box appears.

- 9 This time leave all four inputs selected and click **OK**. A new global boundary constraint is calculated.
- 10 Compare the two global boundary models by selecting the Pairwise view, and selecting each model in the tree in turn.
- 11 If you want to combine boundary models, you select **Edit > Add to Best** and select the parent node to see the combination.

Note: For this example, do not add any models to best.

For more information see “Combining Best Boundary Models”.



- 12 Close the Boundary Editor to return to the Model Browser. Once calculated, boundary models remain part of the test plan unless you delete them.

For next steps, see “Selecting Global and Two-Stage Models” on page 3-33

Selecting Global and Two-Stage Models

In this section...

“Inspect the Global Models” on page 3-33

“Create Multiple Models to Compare” on page 3-33

“Create a Two-Stage Model” on page 3-35

“Evaluate Other Response Models” on page 3-35

Inspect the Global Models

When you are satisfied with the local fits, inspect the global models in turn. You should check trends of global models. Do the trends go in the right direction? Previous engineering knowledge can be applied. The following steps suggest useful plots for investigating trends.

- 1 Expand the local model node (PS22) in the model tree and click **knot**.
- 2 Inspect the **Response Surface** view. Apply engineering knowledge to check trends. Areas outside the boundary model are highlighted yellow to help you focus on the area of interest.
- 3 In the **Diagnostic Statistics** plot, right-click outliers (or any point) to see a plot of the test. You can inspect the shape of the torque/spark curve and see the values of the global variables. This can help you identify problem tests, perhaps on the edge of the stable operating region.

Tip: After identifying problem tests at the global level, return to the local level to decide whether to remove outliers or the whole test.

For more advice on inspecting global models, see “Assess One-Stage Models”.

Create Multiple Models to Compare

The example project contains a finished two-stage model, so you can view the alternative models underneath each global model. For example, expand **knot** and view the selection of child nodes. You can make a selection of models using **Create Alternatives** in the

Common Tasks pane. In the example project, the parent model is already part of a two-stage MLE model, so **Create Alternatives** is not enabled in the example file.

When comparing models using statistics:

- Look for lower RMSE values to indicate better fits.
- Look for lower PRESS RMSE values to indicate better fits without overfitting.

PRESS RMSE is a measure of the predictive power of your models. It is useful to compare PRESS RMSE with RMSE as this may indicate problems with overfitting. RMSE is minimized when the model gets close to each data point; 'chasing' the data will therefore improve RMSE. However chasing the data can sometimes lead to strong oscillations in the model between the data points; this behavior can give good values of RMSE but is not representative of the data and will not give reliable prediction values where you do not already have data. The PRESS RMSE statistic guards against this by testing how well the current model would predict each of the points in the data set (in turn) if they were not included in the regression. To get a small PRESS RMSE usually indicates that the model is not overly sensitive to any single data point.

If the value of PRESS RMSE is much bigger than the RMSE then you are overfitting — the model is unnecessarily complex.

PRESS RMSE can be the most helpful single statistic you can use to search for the best fit relative to the number of terms in the model. However you should not rely on any single statistic, but use a variety of criteria and especially the graphical tools.

- You can add other statistics here to help you select models. Select **Model > Summary Statistics**. In the dialog, select the check box for **AICc** and click **OK**. A column of **AICc** values appears in the model list. This can be a useful statistic for comparison of models as it penalizes over-parameterized models. Over-fitting models can later cause problems with optimization when you use the models to create calibrations. See “Using Information Criteria to Compare Models”.
- After selecting a best model among the alternatives, it can be useful to select **Delete Alternatives** in the Common Tasks pane. This discards all but the model you chose as best. The process of searching for good model fits often results in a large selection of models, and cleaning up the tree reduces file size. It can also save time when saving the file, especially if you have made a change such as removing an outlier at local level, that causes all response feature models to be refitted.

To learn more, see “Create Alternative Models to Compare” and “Choose the Best Model”.

Create a Two-Stage Model

To create a two-stage model, select the local node, then click **Create Two-Stage** in the Common Tasks pane. The example project already contains a finished two-stage model.

- 1 Select the **BTQ** response node in the model tree.
- 2 Inspect the Sweep plots of the two-stage model against the local fits and the data.

To learn more, see “Assess Two-Stage Models”.

Evaluate Other Response Models

To complete the model building you should follow the same process as for modeling torque to select good models of the other response models you created using the Fit Model Wizard: exhaust temperature and residual fraction. These models are required for the optimization problems. Look at the example finished project, `Gasoline_project.mat`, in the `matlab\toolbox\mbc\mbctraining` folder, to examine the example models. You will use these example models in CAGE for the optimization section.

Tip: You can copy outliers you already selected in another model. For example, to copy outlier selections from the torque model to the exhaust temperature model, at the **EXTEMP** local node **POLY2**, select **Outliers > Copy Outliers From**, and select the **BTQ** local model node in the tree (**PS22**, under **BTQ**) .

To assess high-level trends, select the test plan node, and the Response Models tab. Check the model trends for each response in the cross section view. Is the fit acceptable? Do you need more data? Can you improve the fit with the existing data? Pay attention to the boundary model on the plots. Yellow areas are outside the boundary. Focus on the model trends only within the boundary model.

For next steps, see “Using Validation Data” on page 3-36.

Using Validation Data

After you have created models, you can use the validation data.

To view the validation data set,

- 1 Select the top project node in the Model Tree.
- 2 In the Data Sets pane, you can see main data and validation data in the project.

At the test plan node, the validation data set appears in the information pane for the test plan tab, and Validation RMSE is shown the summary statistics for comparison in the bottom list view of response models in the test plan. Check the fit against the validation data.

For next steps, see “Exporting the Models” on page 3-37.

Exporting the Models

You can export models to file, or to CAGE, to generate calibrations.

Tip: For more flexibility, use the CAGE Import Tool instead of exporting from the Model Browser.

- 1 To export all models in the test plan, select the test plan node.
- 2 In the **Common Tasks** pane, click **Generate calibration**.
- 3 In the Export to CAGE dialog box, click to **OK** to accept the default model names.

The CAGE Browser displays the models.

Alternatively you can use **File > Export Models** to choose to export to a file, to the workspace, or to Simulink. This capability makes models easy to share across engineering groups. The models exported depend on the model node you have selected in the tree.

For this example, you will use the models in the CAGE part of the toolbox to produce optimized calibration tables. For next steps, see “Optimized Calibration” on page 3-38.

Optimized Calibration

In this section...
“Problem Definition” on page 3-38
“Benefits of Automated Calibration” on page 3-39

Problem Definition

This section describes the creation and optimization of calibration tables for the gasoline case study. The Model Browser section of this case study covers creating the design for the experiment and creating and evaluating models from the resulting data. You can export your models directly to CAGE; or to Simulink software or to a file, ready to be imported into CAGE for model-based calibration generation. An example file is provided.

The aim of this case study is to produce optimized tables for

- Intake cam phase
- Exhaust cam phase
- Spark timing schedules

as a function of load and rpm, subject to the following constraints

- Constrain solutions to lie within the boundary constraint model
- Constrain cam phase solutions so they do not change by more than 10° between table cells (that is, no more than 10° per 500 RPM change and per 0.1 load change).
- Constrain residual fraction $\leq 25\%$ at each drive cycle point (to ensure stable combustion). Residual fraction is the percentage of burned gas mass in the cylinder at intake valve close, relative to the total mass in the cylinder at intake valve close. Constraining maximum residual fraction is a simple and reasonable way of ensuring stable combustion. Residual fraction = $100 * \text{Burned Gas Mass from Last Cycle} / (\text{Burned Gas Mass From Last Cycle} + \text{Fresh Air Mass})$

CAGE is intended for model-based calibration, although you can still create tables without reference to models if you want. For this case study, you use models produced in the Model Browser to generate calibrations in CAGE. You cover the following steps:

- 1 Load models of engine responses, decide on optimization strategy and define additional models. See “Importing Additional Models into CAGE” on page 3-41.

- 2 Set up tables. See “Setting Up Calibration Tables to Fill” on page 3-44.
- 3 Define optimization objective and constraints. See “Setting Up the Optimization” on page 3-47.
- 4 Set up an operating point set for the optimization. See “Defining Variable Values” on page 3-50.
- 5 Run the optimization and view the results. See “Running the Optimization” on page 3-52
- 6 Duplicate and modify the optimization to create a sum optimization using the previous results as starting points. See “Setting Up the Sum Optimization” on page 3-57
- 7 Fill tables from optimization results. See “Filling Tables with Optimization Results” on page 3-62.
- 8 Use models and optimized tables to fill a spark estimator table. See “MBT Spark Estimator Problem” on page 3-63.

For guidance, you can look at the example finished project:
`Gasoline_optimization.cag`.

Benefits of Automated Calibration

- You can move the table-filling process away from the test bed.
- You can regenerate calibrations when objectives, constraints, or calibration table layouts change, without additional testing.
- You can explore tradeoff possibilities interactively.
- You can produce initial calibrations using engine simulation software, before hardware is available.

CAGE can provide both automatic and interactive calibration optimization. You can trade off multiple objectives, deal with multiple constraints, and you can examine optimizations point-by-point or drive-cycle-based. You can use built-in optimization routines or write your own. You can fill groups of tables simultaneously, and optimize table values and breakpoint settings. CAGE can provide solutions for these example applications:

- Control problems
 - Injection timing and duration
 - EGR valve

- Spark timing
- Dual-independent variable valve timing
- Emissions-constrained BSFC optimization over drive cycles
- Estimation problems
 - Torque
 - Emissions
 - Air flow and manifold pressure
 - Intake valve temperature
 - Borderline spark

Importing Additional Models into CAGE

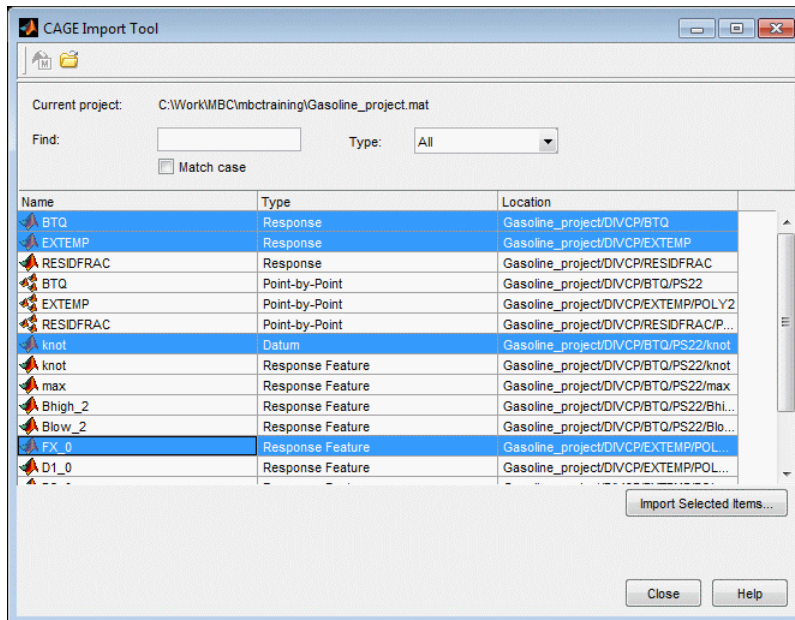
- 1 In the CAGE Browser select **File > Import From Project**. The CAGE Import Tool appears.
- 2 Click the **Import From Project File** button. Locate the example model file created in the Model Browser, `Gasoline_project.mat`, in the `matlab\toolbox\mbc\mbctraining` folder, and click **Open**.
- 3 You can select from a list of models in the file. Select these models by **Ctrl**+clicking in the list:
 - BTQ
 - EXTEMP
 - knot (**Type** = Datum)

In this case the `knot` model is duplicated because the datum model was used twice during modeling. The datum model tracked the maximum of the torque model, that is, MBT (the spark angle at maximum brake torque). This datum model was also used when modeling exhaust temperature and residual fraction, because it can be useful to see MBT on model plots for other factors. This is called a datum link model. You need only one copy of the `knot` model.

- `FX_0` (Check that the **Location** specifies `RESIDFRAC`, *not* `EXTEMP`)

The response feature model `FX_0` is `RESIDFRAC` at MBT. `RESIDFRAC` was constructed using a datum link (in this case `knot`, i.e., MBT) so `FX_0` is residual fraction evaluated at the datum point (MBT).

Check that your selection matches those shown.



- 4 Click the **Import Selected Items** button .
- 5 In the following Import dialog,
 - Double-click to edit the **CAGE Model Name** for the knot model to rename to MBT.
 - Double-click to edit the **CAGE Model Name** for the FX_0 model to rename to RESIDFRACatMBT.
 - Leave the other settings to replace models already in CAGE with the models from the file.
 - Click **OK** to import the models.
- 6 Close the Import Tool and look at the Models view in CAGE. You should see the BTQ, EXTEMP, MBT, and RESIDFRACatMBT models in the list. The selected model is displayed in the other panes.

The objective is to produce optimized tables for spark and cam timings, subject to constraints on operating region, residual fraction, and the gradient of the cam phasers over the calibrated tables. You want the optimization to search for the timings that give the best torque and minimum fuel consumption, subject to the constraints.

You could fix the spark timing to be MBT spark (the spark angle that produces maximum brake torque) by using the MBT spark model as the spark input to the other models in the optimization. With spark fixed at MBT, you can set up an optimization that allows the two valve timings to vary, exploring the space via a gradient descent algorithm to locate the optimal timings. Remember, this is a simplified problem and running an engine at MBT is knock-limited and so is not possible at all operating points. For this example, you will not use the MBT model as you will use spark as a free variable in the optimization. You will return to the MBT model later.

Note you can also:

- Export models directly from the Model Browser to CAGE when both are open.
- Use the CAGE Import Tool to import models directly from the Model Browser.
- Use the CAGE Import Tool to import models and other calibration items (tables, optimizations, tradeoffs, data sets, features) from any project file created in CAGE or the Model Browser. This can help you use existing projects to speed up the setup of new sessions.
- Export models to a file which you can import to CAGE by selecting **File > Import > Model**

Setting Up Calibration Tables to Fill

In this section...

“Altering Variable Ranges” on page 3-44

“Setting Up Tables” on page 3-44

Altering Variable Ranges

Editing variable ranges before creating tables will produce easy-to-read values (e.g., 0.1, 500, 0.2, 1000, etc.) for the table breakpoints when CAGE automatically spaces the normalizer values across the variable ranges.

- 1 Click the Variable Dictionary button to switch to the Variables view.
- 2 Select N, edit the range to 500 for the minimum, 5000 for the maximum, and press **Enter**.
- 3 Select L, edit the range to 0.1 for the minimum, 1 for the maximum, and press **Enter**.

Edit set points of variables as follows.

- 1 Click to select the variable ECP and edit the **Set Point** to 5.
- 2 Click to select the variable ICP and edit the **Set Point** to 40.

Setting Up Tables

You can use the Create Tables From Model wizard helps you quickly create a set of tables with the same axes for all the inputs of a model, and the model response, and any other responses that share the same inputs. This wizard can be useful when creating tables for an optimization, to use when filling tables with optimization results, and for investigating results in the tradeoff views.

- 1 Select **Tools > Create Tables From Model** (or use the toolbar button).

The **Create Tables From Model** Wizard appears.

- 2 Select the BTQ model to base the new tables on.

Click **Next**.

- 3 Select table axes input variables and set up the normalizers to use for the new tables. Select N and L from the **X-** and **Y-axis input** drop-down menus. Leave the number of rows and columns at 10. CAGE automatically initializes the normalizers by spacing the breakpoints evenly across the range of the input variables N (speed) and L (load).

Table Inputs
Select the table inputs and set up the normalizers to use for all the new tables.

Use model operating points

Y-axis input: N
Normalizer: <New>
Table rows: 10

X-axis input: L
Normalizer: <New>
Table columns: 10

N normalizer:

Input	Output
500	0
1000	1
1500	2
2000	3
2500	4
3000	5

L normalizer:

Input	Output
0.068	0
0.166	1
0.264	2
0.362	3
0.46	4
0.558	5

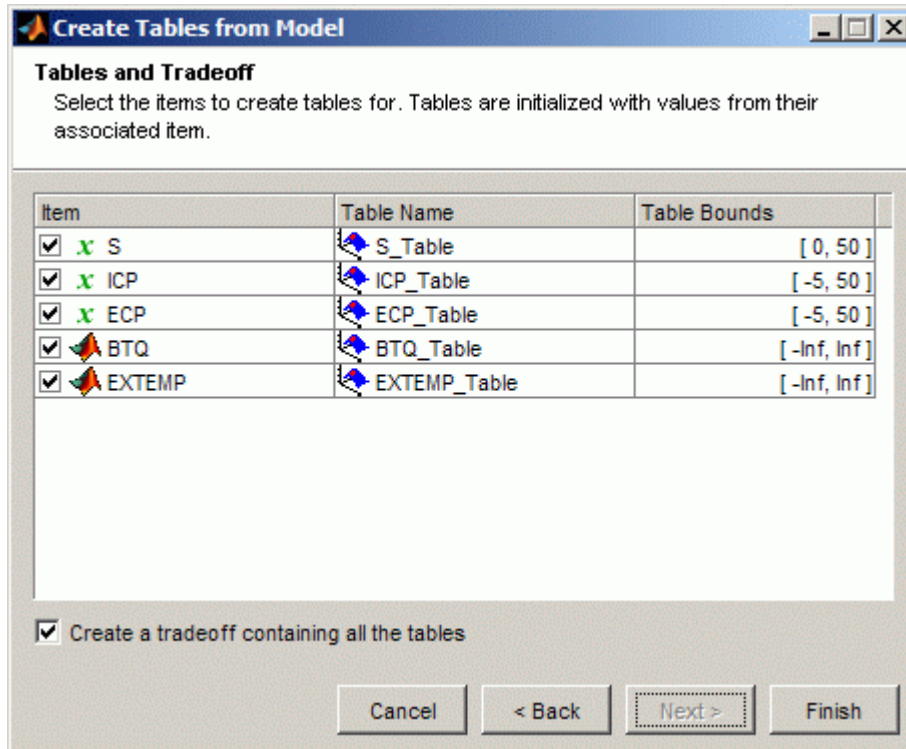
Buttons: Cancel, < Back, Next >, Finish

Click **Next**.

- 4 Select all check boxes to specify which variables and responses to create tables for. CAGE automatically selects the tables for all the models and the response You can create tables for other responses with exactly the same inputs as the primary model.

Note the wizard has defined **Table Bounds** matching the variable bounds for the S, ICP and ECP tables. You can also edit table bounds later in the Table Properties dialog box.

By default you will also create a tradeoff containing all of the new tables. The tradeoff can be very useful for investigating optimization results.



Click **Finish** to create the tables and tradeoff.

You see a dialog listing all the items you have created, and you can choose which (if any) of the items to view next.

Now you have tables for optimized spark and cam timings, ready to fill with optimization results.

Setting Up the Optimization

CAGE provides a flexible optimization environment. You can define the objectives, the constraints, and the points where the optimization is carried out.

The objective is to maximize the weighted sum of torque over a set of [N, L] points subject to a set of constraints. This is a constrained single objective optimization problem.

You solve this problem in two parts:

- 1 Run an initial optimization to explore the problem point-by-point in order to obtain good starting points for the sum optimization.
- 2 Use the solutions from the initial optimization as the start points to run the sum optimization over the drive cycle to find optimal settings of spark (SPK) and cam (ICP, ECP) timing. Use these results to fill calibration tables.

CAGE has several built-in optimization routines and the capacity for you to write your own; in this case study you use `foptcon`. This is a modified version of `fmincon` from the Optimization Toolbox™ product. In CAGE, you can use the algorithm to minimize or maximize.

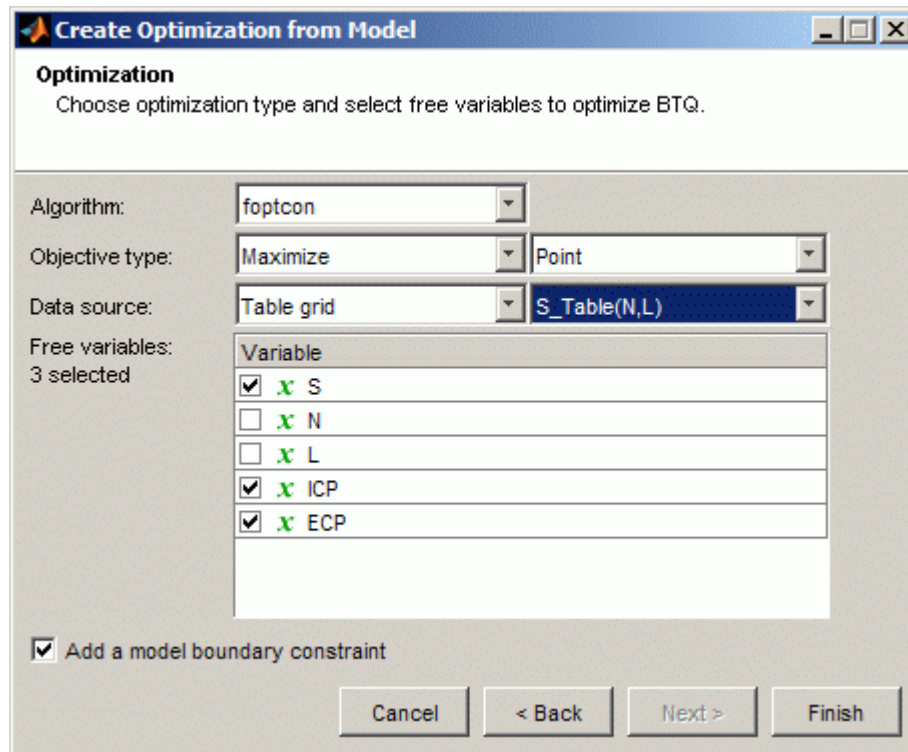
- 1 Select **Tools > Create Optimization From Model** (or use the toolbar button).

The **Create Optimization From Model** Wizard appears.

- 2 Select the BTQ model to maximize in the optimization.

Click **Next**.

- 3 Select the optimization type (algorithm, maximize or minimize, point or sum), data source for optimization, free variables, and boundary constraint, as follows:



- **Algorithm:** Use the default `foptcon` for gradient-based single-objective optimizations.
- **Objective type:** Choose to `Maximize` your model, and leave the default `Point` objective type.
- **Data Source:** Select `Table grid` and `S_Table(N,L)` to define the points where you want to run the optimization from the spark table. You can also set up points in the Optimization view.
- **Free variables:** Leave the selected check boxes of `S`, `ICP`, and `ECP`, the variables you want to optimize from the set of model inputs.

In the preceding step when you selected the table grid as the **Data source**, CAGE automatically removed the table normalizer variables `N` and `L` from the selection of free variables.

- **Add a model boundary constraint:** Leave the check box selected to constrain the optimization within the boundary model associated with your model.

Click **Finish** to create the optimization.

CAGE switches to the Optimization view. Look at the information displayed. Here you can examine and edit the objective and constraint.

The Constraints pane shows the boundary constraint you specified in the wizard.

Add a constraint on the residual fraction response to constrain the optimization to stable combustion regions. Follow these steps:

- 1 Right-click the Constraints pane and select **Add Constraint**. The Edit Constraint dialog box appears.
- 2 Leave the **Constraint type** at the default, **Model**.
- 3 Edit the **Constraint name** to RESIDFRAC.
- 4 Select RESIDFRACatMBT from the list of models on the left and click to select it for the constraint.
- 5 With the RESIDFRACatMBT model selected, enter **25** in the **Constant** edit box and press **Enter**. CAGE closes the dialog box and displays the new constraint in the Optimization view.

Ensure that the constraint **Description** reads RESIDFRACatMBT \leq 25

Defining Variable Values

You need to define the set of points where you want the optimization to run. In this case, you run the optimization over the table breakpoints in the tables. You already defined this in the Create Optimization from Model wizard. In this section you will also set up initial values for spark.

You can edit values manually in the **Input Variable Values** pane in the Optimization view, or you can import values from existing optimization output values, tables or data sets.

To import table initial values to your optimization, you will first create a feature to fill the **S_Table** from the MBT model. You can then use this table to initialize the starting values for your optimization.

To create the feature table,

- 1 Select **File > New > Feature**. CAGE switches to the Feature view.
- 2 Select **Feature > Graphical Strategy Editor**. Three Simulink windows appear.
- 3 In the CAGE project library window, double click the Tables block. The Tables library window opens.
- 4 Drag the **S_Table** table block from the Tables library to the New Feature window.
- 5 Click to select **S_Table**, then **Ctrl+click** the blue **New_Feature** block to connect the two.
- 6 Double-click the blue **New_Feature** block to import the strategy into CAGE.
- 7 In the Feature view, click the **Select Model** button.
- 8 In the Select Model dialog, select MBT and click **OK**.
- 9 Click **Fill Feature** in the toolbar. The Feature Fill Wizard appears.
- 10 Click **Next** three times, then click the **Fill Tables** button. Click **Finish** when the process has finished.
- 11 Expand the **New_Feature** node in the tree and select the **S_Table** node to view the filled table. In the comparison pane you can see where the table limits cause the values to diverge from the feature model at low load.
- 12 Click the **Optimization** button to return to the Optimization view.
- 13 Select **Optimization > Import from Table Values**.

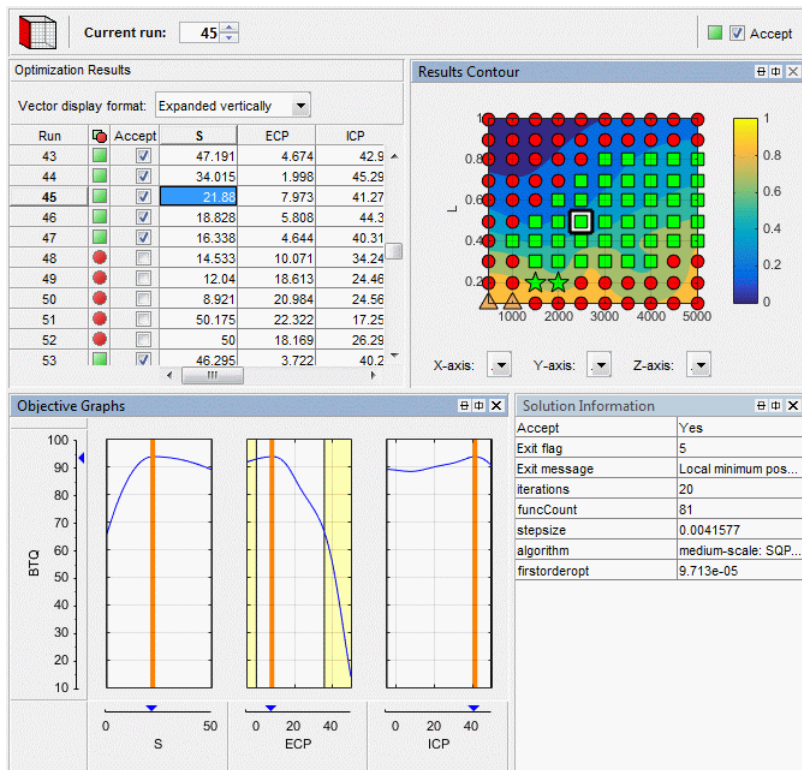
Select the check box to import values for **S**, and select **S_Table** from the **Fill Input With** list. Click **OK** to import the table values of spark into the optimization initial values for **S**.

View the values in the **Input Variable Values** pane in the Optimization view. These values define the variable values at each point where you want the optimization to run. The fixed variable values are the table breakpoints, and the free variable initial values are the filled table values of spark, and the defaults (set point) for the other free variables.

Running the Optimization

You have defined objectives, constraints and a set of operating points. Your optimization is ready to run.

- 1 Click Run Optimization in the toolbar.
- 2 When the optimization is complete, the view switches to the new child node, `BTQ_Optimization_Output`, in the Optimization tree under the `BTQ_Optimization` node. View the results.
- 3 Look through the solutions at different operating points by clicking cells in the Results table or contour plot. Regions that do not meet the constraint are yellow in the plots and tables.



Because this is a single-objective optimization, there is only one solution at each point.

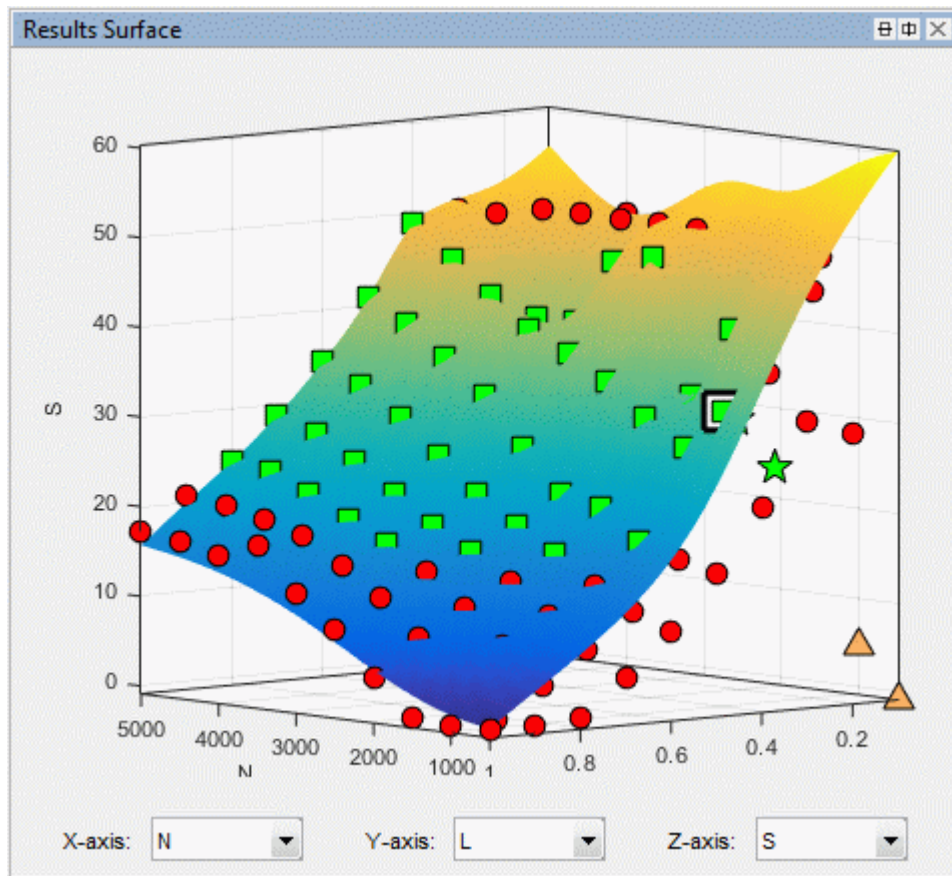
- 4 Observe the colored icons that indicate the Accept status of each solution plotted in the Results Contour view, and of each corresponding row in the Optimization Output Values table.

CAGE automatically selects the Accept check boxes in the table for solutions where the algorithm exit flag indicates success (>0). These solutions show a green square icon in the table and surface. CAGE also highlights unsuccessful solutions for you to investigate.

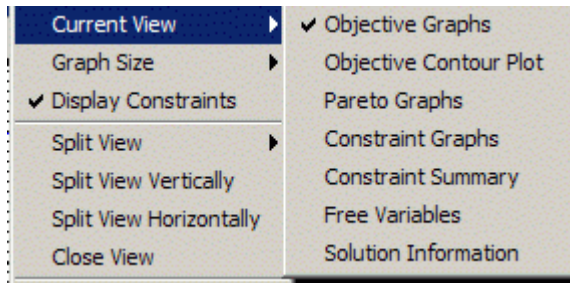
11		<input type="checkbox"/>	4.666
12		<input type="checkbox"/>	44.602
13		<input type="checkbox"/>	35.963
14		<input checked="" type="checkbox"/>	32.128

Click in the table or contour view to select solutions, and other plots (such as the Objective Contours and Objective Graphs) update.

- 5 Right-click the title bar of the Results Contour view and select **Current View > Results Surface**. Observe the Accept icons for the solutions.



- 6 Examine the orange triangle Accept status solutions. This indicates the algorithm exit flag is zero. This means the algorithm terminated because it exceeded limits on the amount of computation allowed (e.g., the algorithm ran out of iterations or function evaluations). You could decide to accept these solutions or you could try changing tolerances and optimizing again. *Do not edit any check boxes yet.*
- 7 Use the plot title bar buttons to split views, or right-click the title bar and use the context menu to try all the available views for analyzing your results. The **Objective Contour** plot of ECP against ICP is useful for analyzing results. Also check the **Objective Graphs** for spark to ensure that maximum torque is obtained.



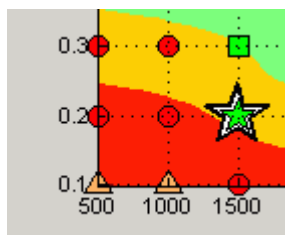
- 8 Export all acceptable results to tables to check surfaces. Click Fill Tables in the toolbar and use the wizard. If you need instructions, see “Filling Tables with Optimization Results” on page 3-62.

Check trends, and don't worry about smoothness. You will apply table gradients constraints to smooth the table surfaces in the next optimization.

- 9 You can change the Accept selections using the check boxes for each solution, or by right-clicking points in the surface view. You can use these Accept selections to choose solutions within the table for use in filling tables, exporting to data sets and importing to other optimization starting values.



Reject points 22 and 32 (clear the check boxes). The points are in the wrong region with respect to adjacent points in the table. Solutions where you have altered the check box status show an asterisk in the table and a star in the Results Surface or Contour plot.



- 10 If you are displaying the Objective Contours plot, you may want to turn off the display of constraints in the plot before running the next sum optimization or viewing the output node for the sum optimization. This will save time displaying

the output views. Right-click the contour plot and toggle **Display Constraints** off. Alternatively close the Objective Contours plot.

Setting Up the Sum Optimization

In this section...

“Setting Up the Optimization Initial Values and Objective” on page 3-57

“Creating Table Gradient Constraints” on page 3-58

“Running the Sum Optimization” on page 3-60

Setting Up the Optimization Initial Values and Objective

The aim of this case study is to produce optimized tables for

- Intake cam phase
- Exhaust cam phase
- Spark timing schedules

as a function of load and RPM, subject to the following constraints:

- Constrain solutions to lie within the boundary constraint model
- Constrain cam phase solutions so they do not change by more than 10° between table cells (that is, no more than 10° per 500 RPM change and per 0.1 load change.)
- Constrain residual fraction $\leq 25\%$ at each drive cycle point

The optimization objective is to maximize the weighted sum of torque over a set of [N, L] points subject to the constraints described.

The initial point-by-point optimization was designed to obtain good starting points for the sum optimization. Now, you can create a sum optimization from the results of that initial optimization to run the sum problem. You use the solutions from the initial optimization as the start points to run the sum optimization over the drive cycle to find optimal settings of spark and cam timings. You use these results to fill calibration tables.

- 1 To create a sum optimization from your point optimization output node, select **Solution > Create Sum Optimization**.

CAGE creates a new optimization named `Sum_BTQ_Optimization`. The optimization has these characteristics:

- The objective matches your original optimization but converted to a sum objective.
- The new optimization has identical constraints to your original optimization.
- Your original fixed and free variables are converted to a sum optimization (a single run with multiple values).
- The new optimization uses only accepted solutions from your original optimization output (all runs with a selected Accept check box). Therefore, the number of accepted solutions you had in the original optimization determines the number of values within the sum optimization run.
- The free variable initial values and fixed variable values are populated from your point optimal results (accepted solutions only).
- The fixed variables have a Weights column with every value set to 1.

In the Optimization view, observe the new sum objective, and the new values in the Input Variable Values pane.

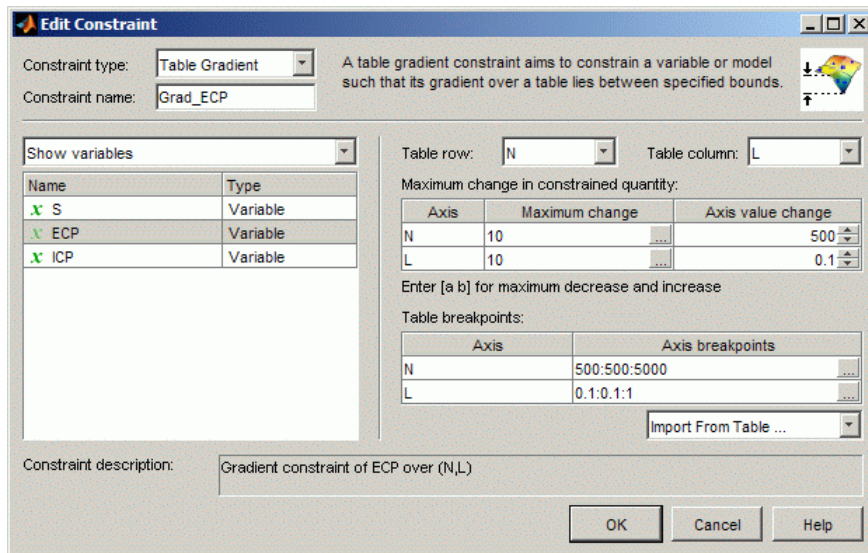
Now, your sum optimization is ready to run. Before you do this, see the next section to create table gradient constraints.

Creating Table Gradient Constraints

To meet engineering constraints it is essential that some calibration tables meet a level of smoothness. For example, large cam phase changes between adjacent table cells cannot be used because the controls cannot physically move that fast. Achieving a level of smoothness is important for variable valve timing calibration problems. Unconstrained optimizations may find solutions where the cam timings change too rapidly across tables. You can apply table gradient constraints to make sure the resulting tables are smooth enough.

To apply a gradient constraint to restrict the change in EXH between adjacent table cells:

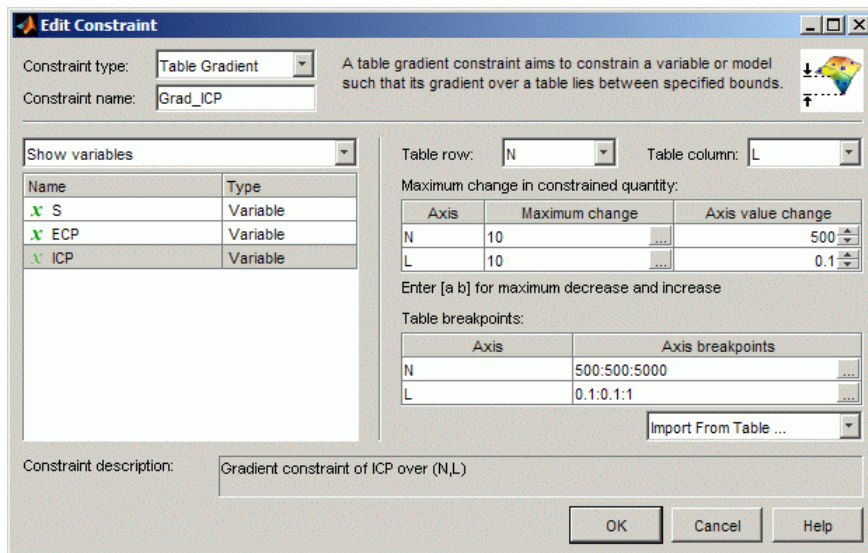
- 1 Right-click in the Constraints pane, and select **Add Constraint**. The Edit Constraint dialog box appears.



- 2 Select **Table Gradient** from the **Constraint type** drop-down menu.
- 3 Enter **Grad_ECP** in the **Constraint name** edit box.
- 4 Select the variable **ECP** to constrain.
- 5 Select the table axes **N** (from the **Table row** drop-down) and **L** (from the **Table column** drop-down).
- 6 Select **ECP_Table** from the **Import From Table** drop-down menu. The breakpoints of this table now appear in the breakpoints edit boxes: **500:500:5000** in **N**, and **0.1:0.1:1** in **L**.
- 7 Apply the gradient constraint across the breakpoints of the table. As shown in the figure, enter **10** in the **N axis Maximum change** edit box, and verify **500** is in the **Axis value change** edit box. This specification constrains the maximum change to 10 degrees **ECP** per 500 change in **N** (the difference between cells, as shown in the axis breakpoints edit box).
- 8 Next, enter **10** in the **L axis row Maximum change** edit box, and verify that **0.1** is shown in the **Axis value change**, to constrain the maximum change to 10 degrees per 0.1 change in **L**.
- 9 Click **OK** to close the **Edit Constraint** dialog box.

Similarly, add another gradient constraint to restrict the change in INT between adjacent table cells as follows:

- 1 Right-click the Grad_ECP constraint in the Constraints pane and select **Duplicate Constraint**. The Edit Constraint dialog appears showing a copy of the settings for your previous constraint.
- 2 Change the **Constraint name** to Grad_ICP.
- 3 Select the variable ICP to constrain.
- 4 Leave the other settings to match the ECP constraint: the table axes, the breakpoints, and the maximum change values. These settings constrain the maximum change to 10 degrees ICP per 500 change in N and per 0.1 change in L.



- 5 Click **OK** to dismiss the Edit Constraint dialog.

Running the Sum Optimization

To run the optimization:

- 1 Click Run Optimization in the toolbar.
- 2 When the optimization is complete, the view switches to the child node, Sum_BTQ_Optimization_Output, in the Optimization tree under the

`Sum_BTQ_Optimization` node. View the results. For guidance see “Interpreting Sum Optimization Output”.

Remember you can check the example project, `Gasoline_optimization.cag`, , found in `matlab\toolbox\mbc\mbctraining`, to view completed examples of the optimizations and filled tables.

Filling Tables with Optimization Results

- 1 From the `Sum_BTQ_Optimization_Output` view, select **Solution > Fill Tables**, or use the toolbar button.

The Table Filling Wizard appears.

- 2 **Shift**-click to multi-select all the tables and click the button to add the tables to the filling list. Click **Next**.
- 3 Verify that the filling factors are correct. CAGE automatically selects factors in the optimization results for table filling, because you used the wizards to create your tables and optimization.

Note that you could also choose the MBT model to fill a table, for example if you wished to compare your optimization results with the spark timings specified by the MBT model. You will fill and optimize tables from models in more detail in the next section.

Click **Next**.

- 4 The last screen is for fill algorithm settings, such as filter rules to use subsets of results, and settings for repeated table filling such as locked cells and extrapolation masks. You can use the default settings.

Make sure the check box **Use acceptable solutions only** is selected, and click **Finish** to fill the tables.

A dialog appears with the message that the tables have been filled successfully. Click **OK**.

- 5 Switch to the **Tables** view to view the filled tables. Click **Tables** in the **Data Objects** pane, and select the filled tables in turn.

MBT Spark Estimator Problem

In this section...

- “What Is an Estimator?” on page 3-63
- “View the Feature” on page 3-63
- “View Variables” on page 3-65
- “Edit and Import Boundary Model” on page 3-66
- “Use the Feature Fill Wizard” on page 3-67
- “Inspect Results” on page 3-71
- “CAGE Import Tool” on page 3-74

What Is an Estimator?

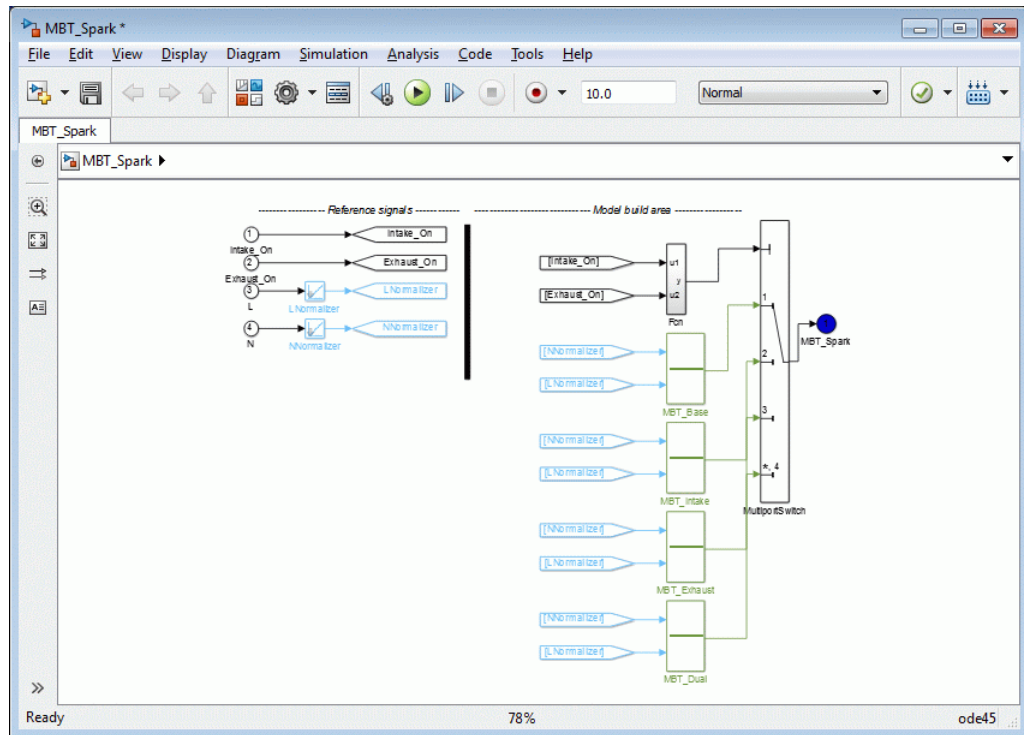
The cam timings optimization results solved a control problem. You can also use CAGE to calibrate estimator problems. Here you will use an MBT model to produce a spark estimator feature which estimates MBT spark when each cam is on or off, using the cam timings found by the optimization.

View the Feature

- 1 Select **File > Open Project** and load the example project, `Gasoline_optimization.cag`, found in `matlab\toolbox\mbc\mbctraining`.
- 2 Click **Feature** in the **Processes** pane to go to the Feature view.

The features `Exhaust_CAM`, `Intake_CAM`, and `MBT_Spark` are in the Feature tree. The `MBT_Start_Feature` was used to provide initial values for the optimization, as described in “Defining Variable Values” on page 3-50.

- 3 Select `MBT_Spark` and view the strategy by selecting **Feature > Graphical Strategy Editor**.
- 4 Examine the strategy model. Observe the Multiport Switch block which switches between MBT tables depending on whether each cam is on or off. The variables `Intake_On` and `Exhaust_On` are used to define whether cams are parked or not. All the MBT tables (with and without cams) share the same speed (N) and load (L) normalizers.



If you wanted to import this strategy model, you would double-click the `MBC_Spark` blue output to parse the strategy into CAGE. This strategy has already been imported into this project so just close the model window.

- 5 Expand the `MBT_Spark` feature in the tree to see the MBT tables that have been created by importing the strategy: `MBT_Base`, `MBT_Intake`, `MBT_Exhaust`, and `MBT_Dual`. These tables share the same normalizers in speed and load.
- 6 Similarly view the strategies for the Exhaust and Intake CAM features. These features switch between parked cams and active cams depending on the threshold value. The CAM features define the cam inputs and switch between the optimal CAM tables (from the optimization results) and the parked values. You can use the linking functionality in the Feature Filling Wizard to connect features and tables to models.

View Variables

You need some variables and constants to define when the cams are parked so the strategy can switch between optimal cam timings and parked cams at a threshold value.

View how this is done:

- 1 Click **Variable Dictionary** in the **Data Objects** pane to switch to the Variable Dictionary view.
- 2 Click to select the new variable `Exhaust_On`.
- 3 Observe the **Minimum** is 0 and the **Maximum** is 1.
- 4 The variable `Intake_On` has the same values.

Also two constants define the parked cam positions:

- 1 Select `Exhaust_Parked`.
- 2 Observe the **Set Point** of this constant is 0.
- 3 Select `Intake_Parked`; this has a **Set Point** of 0.

Edit and Import Boundary Model

To edit the boundary model and import it to CAGE,

- 1 Open the Model Browser (enter `mbcmodel` at the MATLAB command line).
- 2 Load the example project. Select **File > Open Project**, locate and select `Gasoline_project.mat`, found in `matlab\toolbox\mbc\mbctraining`.
- 3 Select the DIVCP test plan node in the tree.
- 4 Select **TestPlan > Boundary Models** (or click **Edit boundary models** in the toolbar).

The Boundary Editor opens.

- 5 Select the **Star shaped** constraint in the tree, and click **Remove boundary model from best** in the toolbar. You need only **Star shaped(N, L)** in the collection of best boundary models.
- 6 Close the Boundary Editor. In the Model Browser, observe the single constraint **Global/Star shaped(N, L)** listed under Boundary Model in the right information pane.
- 7 Return to CAGE and select **File > Import From Project**.

The CAGE Import Tool appears. You can import directly from the Model Browser when it is open, and the CAGE Import Tool automatically displays the available items.

- 8 Select `knot` (the datum model) from the list.


- 9 Click the **Import Selected Items** button.

The Import dialog opens displaying the model you selected for import.

- 10 Double-click the **CAGE Model Name** column cell to edit the name to MBTwithSpeedLoadBoundary, and click **OK** to import the model.
- 11 Close the CAGE Import Tool.

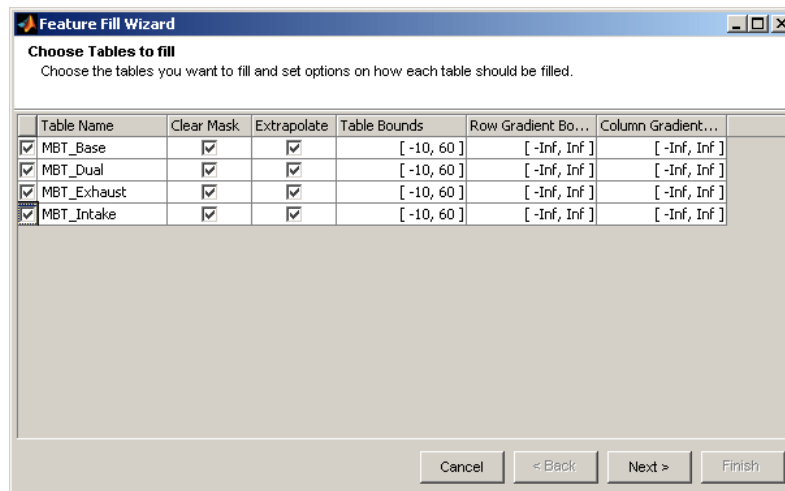
Use the Feature Fill Wizard

You can use the Feature Fill Wizard to fill and optimize the values in tables by reference to the model. You can fill multiple tables at once using the wizard, and you can Fill from the top feature node or from any table node in a feature.

- 1 Click **Feature** in the **Processes** pane to return to the Feature view, then select the feature node MBT_Spark.
- 2 Click  in the toolbar, or select **Feature > Fill**. This opens the Feature Fill Wizard.

Note: The Feature Fill Wizard is modal so you cannot scroll the help after opening the wizard. You might want to review the wizard steps before opening the wizard.

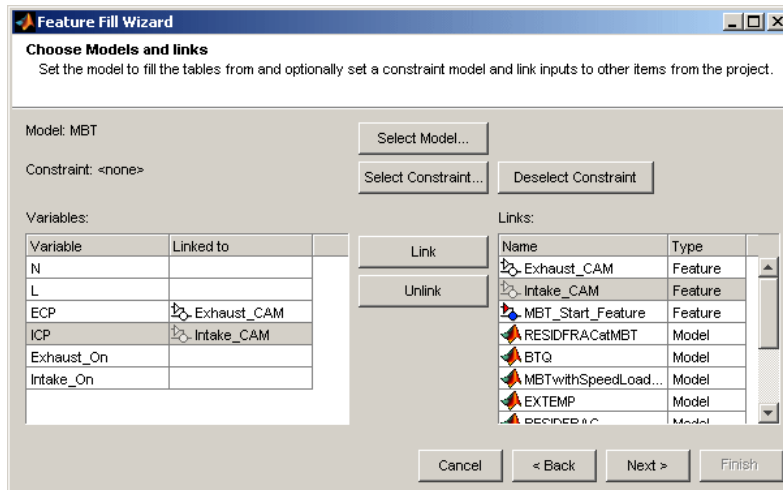
- 3 Select all four table check boxes to fill all tables.



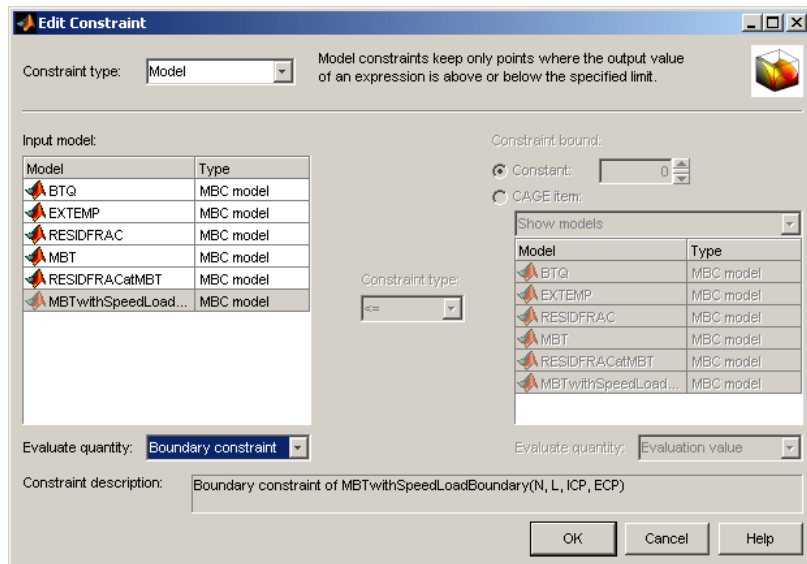
You could also explore setting gradient bounds to constrain table filling for smoothness.

This time leave the other settings at the defaults and click **Next**.

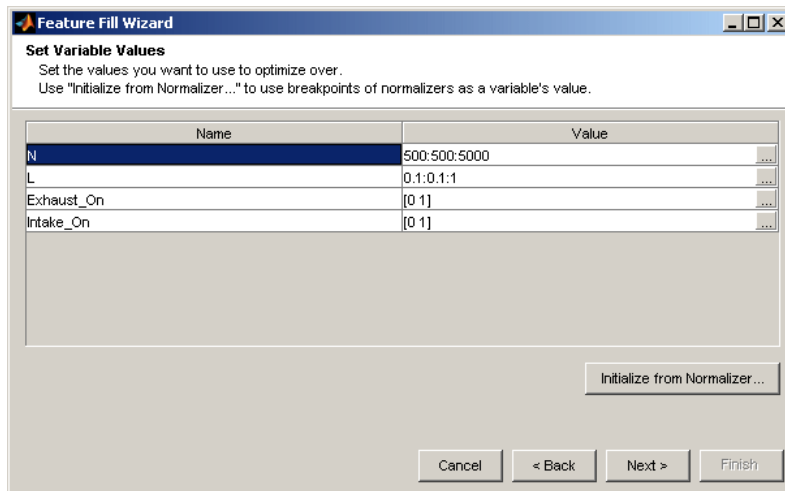
- 4 Choose filling model, constraint, and links.



- a Make sure MBT is the **Model** to fill the tables.
- b Select ECP in the left **Variables** list and the Exhaust_CAM feature in the right **Links** list and click **Link**.
- c Select ICP in the left list and Intake_CAM in the right list and click **Link**.
- d Click **Select** next to Constraint. The Edit Constraint dialog box opens.



- e Select the model MBTwithSpeedLoadBoundary, select **Boundary constraint** as the **Evaluate quantity**, and click **OK**. This boundary model constraint ensures you only fill over speed/load points where the data was collected.
 - f When you return to the Feature Fill Wizard, click **Next**.
- 5 Set values to optimize over.



- Enter 0, 1 in the Exhaust_On Value edit box, and press **Enter**.
- Enter 0, 1 in the Intake_On Value edit box, and press **Enter**.

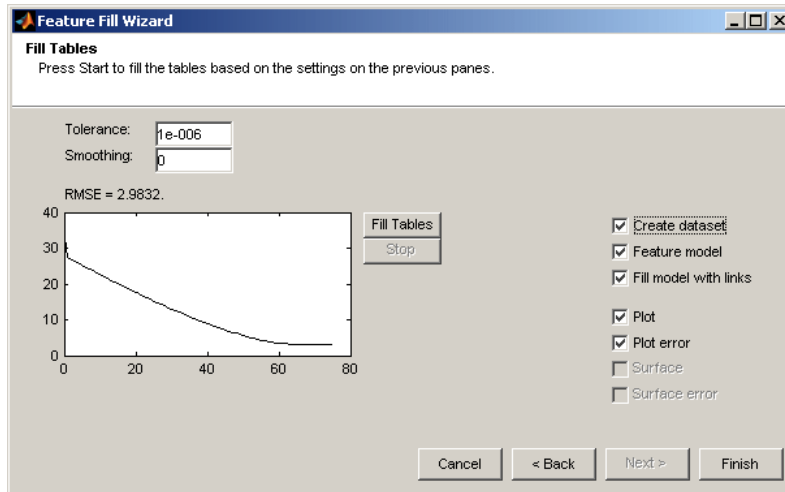
You use these values because the strategy includes the following tables.

Table	Intake_On Value	Exhaust_On Value
MBT_Base (cams parked)	0	0
MBT_Exhaust	0	1
MBT_Intake	1	0
MBT_Dual	1	1

The N and L normalizer values are automatically selected. You can edit normalizers manually, or you can click the **Initialize From Normalizer** button here to reach a dialog where you can select normalizers and interleave values between breakpoints. Interleaving values can minimize interpolation error by adding values between each normalizer value. In this way, you can create a grid of more points than table cells to optimize over. Leave the setting alone for now.

Click **Next**.

- 6 Fill tables and generate plots. Click the **Fill Tables** button. Watch the graph as the optimization progresses.

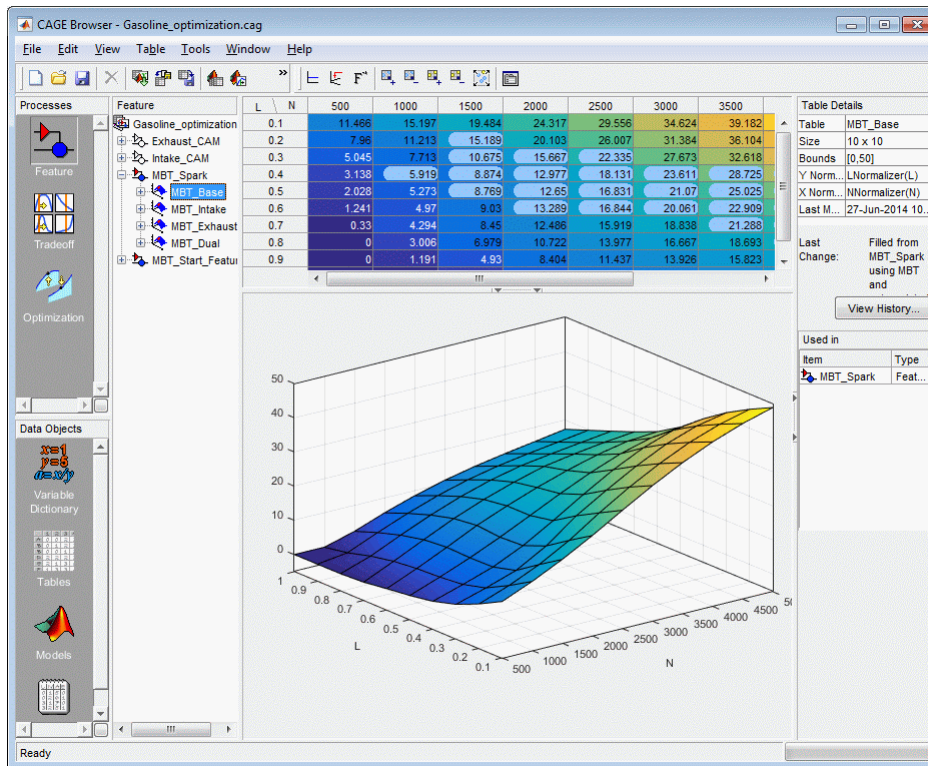


When it is finished, select all enabled check boxes, and click **Finish**. Plots appear summarizing the feature fill data.

Inspect Results

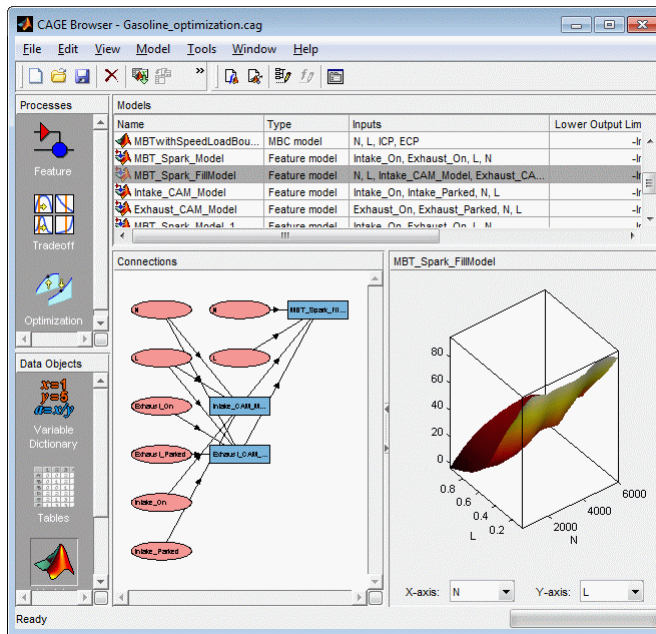
Look at the filled tables, linked models and exported data set.

- 1 In the Feature view, saved fill settings appear in the Feature Fill Settings pane, and the Feature Tables pane shows the tables you filled.
- 2 Select in turn the tables MBT_Base, MBT_Intake, MBT_Exhaust, and MBT_Dual in the feature tree.




Observe the blue mask area of cells in each table — the mask is defined by the limits from the MBTwithSpeedLoadBoundary boundary constraint model you selected. The rest of the table values are extrapolated after the mask cells are filled by the Feature Fill Wizard (specified by the **Extrapolate** check box). Table values are limited to [-10 60] as specified in the **Table Bounds** in the Feature Fill Wizard.

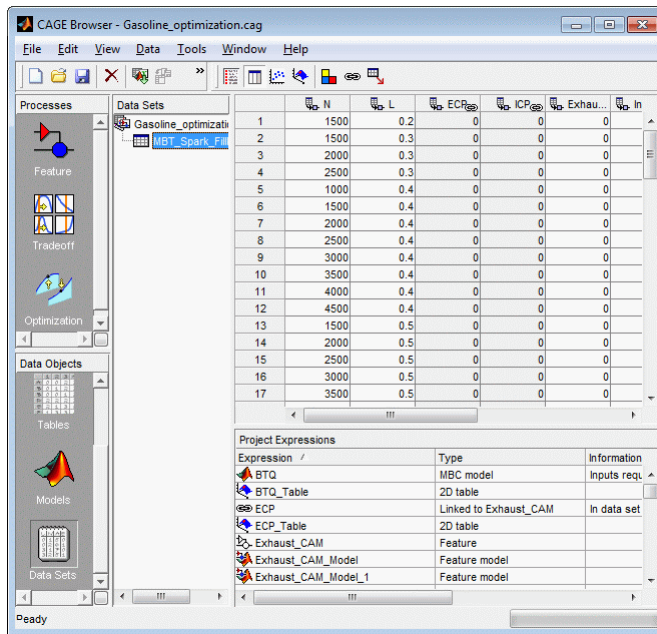
- 3 Click **Models** to select the Models view. Look at the feature model and fill model MBT_SPARK_Model and MBT_SPARK_FillModel. If you can't view the whole Connections diagram of MBT_SPARK_FillModel, right-click and select **Zoom To Fit**.



The linking functionality in the Feature Fill Wizard allows features and tables to be connected to models and any expression to be connected to feature inputs. These links can be made permanent by creating feature models and fill models that are static snapshots of the table on finishing the feature fill wizard (specified by the **Feature model** and **Fill model with links** check boxes on screen 4 of the Feature Fill Wizard).

Notice that the CAM features are converted to feature models (Exhaust_CAM_Model and Intake_CAM_Model are new feature models) and they are connected to MBT_SPARK_FillModel.

- 4 Click **Data Sets** to select the Data Sets view. Select the dataset MBT_SPARK_FillResults to study the gridded data, the model and feature values for the feature fill. Click View Data  in the toolbar to see the data table view. All the links in the feature fill process are defined in this data set — try clicking column headers to see highlighted linked input columns.



CAGE Import Tool

Suppose that you are calibrating a similar engine at a later date. You would build new models for this engine and then want to solve the same problems in CAGE. The CAGE Import Tool allows you to reuse the setup from your old CAGE session. All that is necessary to import new models on top of the existing ones and rerun the optimizations and feature fill problems. For example you could import new BTQ (replace BTQ), knot (replace MBT and MBTwithSpeedLoadBoundary) and EXTEMP models from the example file `Gasoline_Project.mat` as follows:

- 1 Select **File > Import From Project**.

The CAGE Import Tool appears.

- 2 You can choose a project file or import directly from the Model Browser if it is open. If the Model Browser is open, the CAGE Import Tool automatically shows the items available. Use the **Import From Project File** button to select a file.

If you are choosing a project file, a file browser dialog opens. Locate `Gasoline_Project.mat` and click **Open**.

- 3** The CAGE Import Tool displays the available items. Select the items you want to import from the list: BTQ, EXTEMP, and knot.
- 4** Click the **Import Selected Items** button.
- 5** The Import dialog opens displaying the items you selected for import.
 - Double-click the **CAGE Model Name** column cells to edit item names.
 - Choose to replace BTQ, EXTEMP and MBT. For knot, select **Replace** from the **Action** list, then double-click the **CAGE Model Name** column cells to open a dialog to select the correct item, MBT, to replace.
 - Click **OK** to import the items.

Now you can run the optimization again to generate new optimal CAM timings with new models. Export the optimization results to the INTCAM and EXHCAM tables, and use the Feature Fill Wizard to fill the MBT_SPARK strategy using the same steps as before.

Diesel Engine Calibration Case Study

This case study provides a step-by-step guide to using the Model-Based Calibration Toolbox product to solve a diesel engine calibration problem. This section includes the following topics:

- “Diesel Case Study Overview” on page 4-2
- “Design of Experiment” on page 4-4
- “Modeling” on page 4-11
- “Optimized Calibration” on page 4-12
- “Importing Models of Engine Responses into CAGE” on page 4-15
- “Defining Additional Variables and Models” on page 4-16
- “Setting Up Calibration Tables to Fill” on page 4-18
- “Setting Up the Point Optimization” on page 4-19
- “Setting Up Constraints” on page 4-22
- “Defining Variable Values” on page 4-26
- “Running the Optimization” on page 4-29
- “Setting Up the Sum Optimization” on page 4-31
- “Filling Tables with Optimization Results” on page 4-36

Diesel Case Study Overview

In this section...
“Introduction” on page 4-2
“Problem Definition” on page 4-2

Introduction

This case study is an example of a diesel engine control calibration, for a six-cylinder 9.0 L common-rail diesel engine with VGT (variable geometry turbo) and cooled EGR (exhaust gas recirculation). It is applied in an off-road application with a very narrow engine speed range from 1600 to 2200 RPM. The aim of the case study is to produce optimal SOI (start of injection), base fuel, VGT, and EGR calibration schedules as a function of commanded torque and RPM. It involves models for torque, peak pressure, equivalence ratio, exhaust temperature, VGT speed, and EGR mass fraction. The optimization setup in CAGE is based on an 8-mode off-road emission test, approximated to 7 mode points by neglecting the idle operating point of the engine.

The Model Browser part of the example takes you through the following steps:

- 1 Create a design for your experiment “Design of Experiment” on page 4-4
- 2 Create models from the collected data “Modeling” on page 4-11

The Model Browser section of the case study covers design of experiment, data handling, and model construction and export. In the later CAGE browser section of the case study you use the models to complete the optimization of the calibration tables, see “Optimized Calibration” on page 4-12.

Problem Definition

Produce optimal calibration tables in speed and torque for

Best injection timing	soi
Best fuel quantity	basefuelmass
Best fuel pressure	fuelpress
Best VGT	grackmea

Best EGR	egrpos, egrmf
----------	---------------

Minimize mode-weighted brake specific fuel consumption, subject to constraints on

- Turbo speed (vtgrpm)
- Cylinder pressure (pkpress)
- Exhaust equivalence ratio (exheqr)

To solve this problem, you must first use the Model Browser part of the Model-Based Calibration Toolbox product to design an experiment for collecting data, and then create models based on that data. You will use the resulting models in the CAGE Browser part of the toolbox to produce optimal calibration tables.

Design of Experiment

In this section...

“Introducing Design of Experiment” on page 4-4

“Constraining the Design” on page 4-6

“Creating Candidate Designs” on page 4-9

“Data Collection” on page 4-10

Introducing Design of Experiment

Creating a design in the Model-Based Calibration toolbox comprises several steps. First, you need to enter the ranges and names of the variables being used and choose a default model. Then you can create an initial design and set up the constraints on the space. These constraints will be the same for all designs. From this constrained design, you can create a series of child designs adding varying numbers of points and using different construction techniques. You can choose the final design by comparing the statistics of the various child designs, while considering how many test points you can afford to run.

Variables are

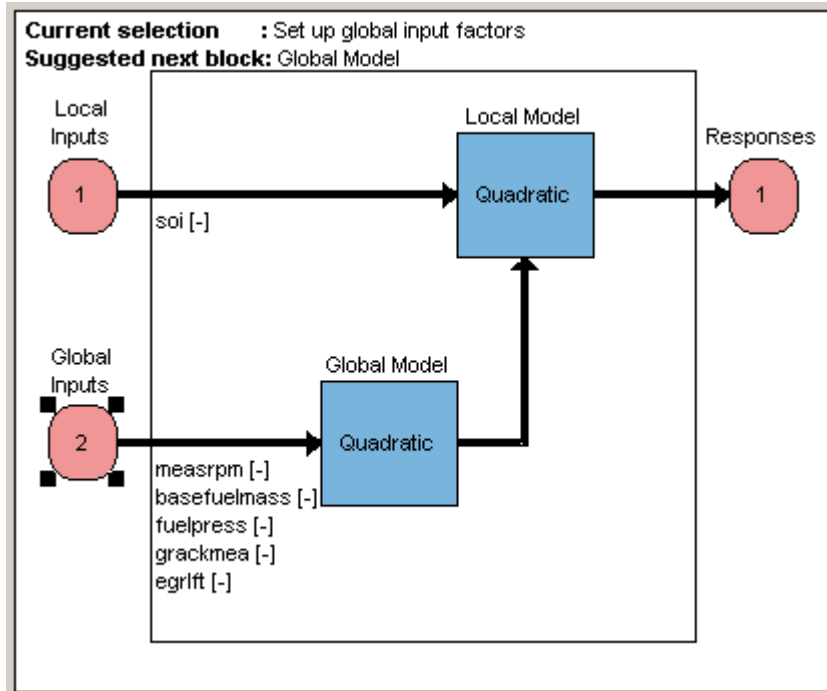
measrpm	Engine speed (rpm)
basefuelmass	Fuel quantity per injection (mg)
fuelpress	Fuel pressure (MPa)
grackmea	VGT rack position (%)
egr1ft	EGR valve position (mm)
soi	Start of injection (deg ATDC)

You need to set up a test plan before you can make designs. This experiment is set up as a two-stage test plan with start-of-injection (SOI) sweeps at the local level and the other five variables at the global level.

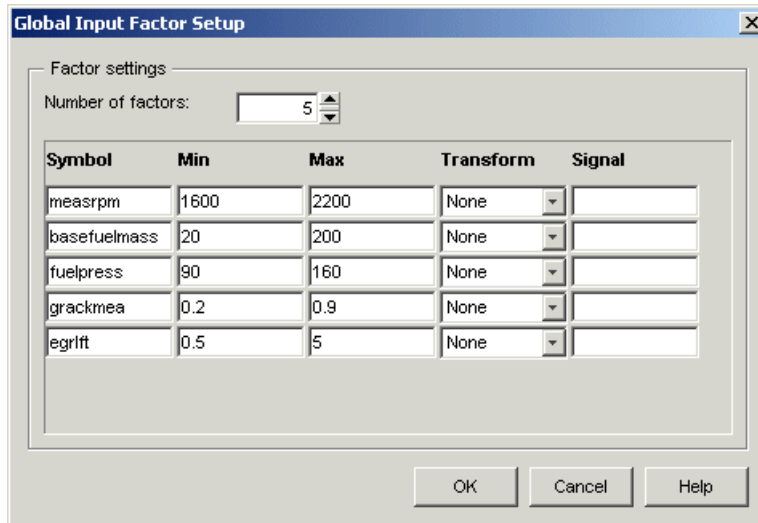
Open the example session with the test plan set up as follows:

- 1 Start the Model Browser by typing `mbcmodel` at the MATLAB command line.

- 2 Select **File > Open Project**. Locate the example session with the test plan set up, Diesel_testplan.mat, in the mbctraining directory and double click to load the project.
- 3 Click the Two-Stage test plan node in the model tree to see the test plan diagram.



- 4 Double-click the Global Inputs block in the diagram to set the ranges of the inputs. You should set up the ranges before designing an experiment. You can enter the ranges in the min/max boxes to include the most extreme values you want to set for each variable. Check the ranges match those shown in the following example, then click **OK**.



- 5 Double-click the Global Model block in the test plan diagram to view the model type. For this exercise, leave the model type at the default, which is a quadratic in all factors. Click **OK** to dismiss the dialog.

Remember that the statistical usefulness of different designs depends on the model type. For example, if you think you need cubic instead of quadratic in EGR, the number of points required rises dramatically and this has a highly adverse effect on the statistical quality of the designs.

Some possible models are

- Cubic polynomial, quadratic in fuel pressure: 41 terms
- Cubic polynomial, quadratic in fuel pressure and EGR: 31 terms

However, you do need to bear in mind that the final model will probably not be either of the possibilities listed here, because some terms will have been removed, or it might even be an RBF (radial basis function). You choose the most suitable model you can in order to construct a design, then when you have collected the data you might find that a different model type produces the best fit.


Constraining the Design

These are the constraints you want to apply to the design space:

- basefuelmass
 - Maximum 200 at 1600 rpm, 175 at 2200 rpm
- fuelpress
 - Range 90 - 110 at 1600 rpm
 - Range 120 - 160 at 2200 rpm
- grackmea
 - Range 0.2 - 0.6 at 1600 rpm
 - Range 0.4 - 0.9 at 2200 rpm

The tables here are very simple: one output value defined at the min and max settings of RPM. The final constraint is a cube within the base fuel mass-fuel pressure-VGT space that moves and changes size as RPM is altered.

To add a constraint to a design,

- 1 First open the Design Editor by right-clicking the Global Model block in the test plan diagram and selecting **Design Experiment**.
- 2 Click the New Design  button in the toolbar or select **File > New Design**. A new node appears.


The new design node is automatically selected. An empty Design Table appears because you have not yet chosen a design, unless you have previous design views from other sessions. The Design Editor retains memory of view settings.

- 3 Select **Edit > Constraints** from the Design Editor menus.
- 4 The Constraints Manager dialog appears. Click **Add**.
- 5 The Edit Constraint dialog with available constraints appears. Make sure the default 1D **Table** is selected from the **Constraint Type** drop-down menu. These are easier to set up than linear constraints, although working out the linear constraint numbers might be worthwhile for larger problems as it is faster.
- 6 You can select the appropriate factors to use. For the first constraint, choose **measrpm**, **basefuelmass**, and the inequality \leq from the menus.

You can define the constraint by typing values in the edit boxes or by moving the large dots (clicking and dragging them) to define a boundary. For this constraint, you want to define two points.

7 Edit the **Number of breakpoints** to 2, and click **Span Factor Range**.

8

Next to the plot, click Move Points , then click and drag the right point (where `measrpm = 2200`) down to `basefuelmass = 175`. You can also enter the values in the **measrpm** and **basefuelmass** edit boxes.

9 Click to select the left point. Make sure the values are 1600 in the **measrpm** edit box and 200 in the **basefuelmass** edit box.

This constraint defines the range of **basefuelmass** in terms of RPM to within these bounds: maximum 200 at 1600 rpm, 175 at 2200 rpm.

10 Click **OK** to return to the Constraints Manager.

11 In the Constraints Manager, click **Duplicate** four times. This saves you setting up tables with only two points for the next constraints. Click to select the first new constraint, then click **Edit**.

You need to add constraints that define each of the following:

fuelpress

- Range 90 - 110 at 1600 rpm
- Range 120 - 160 at 2200 rpm

You achieve this by defining two constraints. In the first, the two table points define a **fuelpress** minimum of 90 at 1600 rpm and a minimum of 120 at 2200 rpm. In the second, the two table points define a **fuelpress** maximum of 110 at 1600 rpm and a maximum of 160 at 2200 rpm.

12 In the Edit Constraint dialog, change the Y factor to **fuelpress** and leave the X factor as **measrpm**.

13 Change the Inequality to \geq .

14 Select the left point (where `measrpm = 1600`) and enter 90 in the **fuelpress** edit box.

15 Select the right point (where `measrpm = 2200`) and enter 120 in the **fuelpress** edit box. Click **OK** to return to the Constraint Manager.

16 Select the next constraint and click **Edit**. Edit the constraint to define a **fuelpress** maximum of 110 at 1600 rpm and a maximum of 160 at 2200 rpm. Click **OK** to return to the Constraint Manager.

17 Complete the other constraints in a similar way.

grackmea

- Range 0.2 - 0.6 at 1600 rpm
 - Range 0.4 - 0.9 at 2200 rpm
- 18** The Constraints Manager should contain all five constraints. Click **OK** to return to the Design Editor.
- 19** Right-click a Design Editor view and select **Current View > 3D Constraints** to view the constrained design space.

Creating Candidate Designs

Number of points


- How many do you have time for? When you consider the number of points, you need to remember that a sweep will be done at each point, and this will take some time.
- Do you need to allow time to fix problems or redo experimental points that can't be achieved due to combustion stability constraints?

Design type

- V-optimal: reduces average prediction error

V-optimal designs are often the preferred choice for engine testing. Optimal designs tend to push points to the edge, so they should give good coverage of the 1600 and 2200 RPM points while also allowing good modeling of the entire experimental region.

Create an optimal design with 65 points to compare to the example design.

- 1** Click Optimal Design  in the toolbar. The Optimal Design dialog opens.
- 2** Enter **65** in the **Total number of points** edit box.
- 3** Select **V-Optimal** from the **Optimality criteria** drop-down menu and click **OK**.
- 4** The Optimizing Design dialog appears. Click **Accept** when iterations are not producing noticeable improvements; that is, the graph becomes very flat.
- 5** Examine the design points and compare to the constraint space by right-clicking the 3D Constraints view and selecting **Split View > 3D Design Projection**.

The final design used contained 65 points, for a quadratic in fuel pressure and EGR lift. V-optimal value = 0.302.

Data Collection

Data was generated by a Ricardo WAVE model using the experimental design. MATLAB and Simulink simulation tools control WAVE. Simulation tools support multiple WAVE processes retrieving test points from a central store. Average simulation time was 8 points (30 engine cycles each) per hour using four processors in parallel. Transient test results were then processed to extract steady-state results.

You can use the toolbox to import test data, view it, sort it into tests, verify ranges, filter out unwanted points, and select data for modeling. For details on any of these processes, see the examples in the gasoline case study section “Viewing and Filtering Data” on page 3-18, and for comprehensive information on data handling in the toolbox, see “Data Import and Processing”. See also the examples in “Data Manipulation for Modeling” on page 11-2.

The example project provided (`Diesel_testplan.mat`) contains the filtered data attached to the test plan.

Modeling

Overview of Modeling Process

Outline of modeling workflow steps:

- 1 Build the models specified in the test plan.
- 2 Review local fits. Is the local model flexible enough?
- 3 Eliminate outliers.
- 4 Try alternative local models.
- 5 Review global fits. Is the global model flexible enough?
- 6 Eliminate outliers.
- 7 Try alternative global models.
- 8 Select best models.
- 9 Clean up tree.
- 10 Create boundary models.
- 11 Export models to file or CAGE.

The toolbox provides example files for you to explore this modeling problem:

- The file `Diesel_testplan.mat`, found in the `mbctraining` directory, contains a test plan where the model inputs and types are set up, and the data is loaded. Try building response models.

For guidance, look at the models in the example finished project, `Diesel_project.mat`.

You use the example models in the CAGE part of the Model-Based Calibration Toolbox product to produce optimized calibration tables. For next steps, see Proceed to the next section for instructions “Optimized Calibration” on page 4-12.

Optimized Calibration

In this section...

“Problem Definition” on page 4-12

“Benefits of Automated Calibration” on page 4-13

Problem Definition

This section describes the creation and optimization of calibration tables for the diesel case study. You use the models created in the Model Browser section of this case study. An example file is provided.

The problem to solve is to produce tables in speed and torque for

Best injection timing	soi
Best fuel quantity	basefuelmass
Best fuel pressure	fuelpress
Best VTG	grackmea
Best EGR	egr1ft

Also, to minimize brake specific fuel consumption, subject to constraints on

- Turbo speed (vtgrpm)
- Cylinder pressure (pkpress)
- Exhaust equivalence ratio (exheqr)

For this case study, you use models produced in the Model Browser to generate calibrations in CAGE. You cover the following steps:

- 1 Load models of engine responses. See “Importing Models of Engine Responses into CAGE” on page 4-15.
- 2 Define additional models and variables required by optimization strategy. See “Defining Additional Variables and Models” on page 4-16.
- 3 Set up tables. See “Setting Up Calibration Tables to Fill” on page 4-18.
- 4 Set up the optimization to provide feasible starting points for the sum problem. See “Setting Up the Point Optimization” on page 4-19.

- 5 Define constraints. See “Setting Up Constraints” on page 4-22.
- 6 Define optimization operating points. See “Defining Variable Values” on page 4-26.
- 7 Run the optimization and view results. See “Running the Optimization” on page 4-29
- 8 Create a sum optimization using the previous solutions as starting points. See “Setting Up the Sum Optimization” on page 4-31
- 9 Fill tables from optimization results. See “Filling Tables with Optimization Results” on page 4-36.

For guidance, you can look at the example finished project, `Diesel_optimization.cag`.

Benefits of Automated Calibration

- You can move the table-filling process away from the test bed.
- You can regenerate calibrations when objectives, constraints, or calibration table layouts change, without additional testing.
- You can explore tradeoff possibilities interactively.
- You can produce initial calibrations using engine simulation software, before hardware is available.

CAGE can provide both automatic and interactive calibration optimization. You can trade off multiple objectives, deal with multiple constraints, and you can examine optimizations point-by-point or over a drive-cycle. CAGE can provide solutions for these example applications:

- Example control applications

Emissions-constrained BSFC optimization over drive cycles producing calibrations such as:

- Optimal fuel injection timing schedule
 - Optimal fuel injection quantity schedule
 - Optimal EGR valve position and EGR mass fraction schedule
 - Optimal spark timing schedule
 - Optimal dual-independent variable valve timing schedules
- Estimation problems
 - Torque
 - Emissions
 - Air flow and manifold pressure
 - Intake valve temperature
 - Borderline spark

Importing Models of Engine Responses into CAGE

- 1 Start CAGE by typing `cage` at the MATLAB command line.
- 2 Select **File > Import > From Project**. The CAGE Import Tool appears.
- 3 To import from a file, click the **Import From Project File** button. Locate the example model file created in the Model Browser, `Diesel_project.mat`, in the `mbctraining` directory, and click **Open**. The import tool loads a list of models in the file.
- 4 Select **Response** from the **Type** drop-down list to filter the model list.
- 5 Select all the response models in the list by **Shift**+clicking.
- 6 Click the button **Import selected items**.
- 7 In the following Import dialog box, click **OK** to import the models.
- 8 Close the Import Tool and observe the Models view in CAGE. You should see the models in the list. The selected model is displayed in the other panes.

You can also:

- Export models directly from the Model Browser to CAGE when both are open.
- Use the CAGE Import Tool to import models directly from the Model Browser.
- Use the CAGE Import Tool to import models and other calibration items (tables, optimizations, tradeoffs, data sets, features) from any project file created in CAGE or the Model Browser. This capability can help you use existing projects to speed up the setup of new sessions.
- Export models to a file, which you can import to CAGE by selecting **File > Import > Model**.

Defining Additional Variables and Models

Looking at the problem definition, you need to define some additional variables and models for this optimization. You want to minimize BSFC, but this is not a model output. You need to fill tables against speed and torque, but torque is not an input for the models. The solution to this is to create function models. You can then use a torque constraint and the BSFC function model in the optimization. To implement the torque constraint, you need to define a new variable for desired torque, `tq_desired`.

Follow these steps:

- 1 Select **File > New > Variable Item > Variable**. The Variable Dictionary view appears.

- a Rename the new variable `tq_desired`.
- b Set the range of this variable to be minimum 0 and maximum 1500.
- c Edit the set point to 600 Nm

- 2 Create a new function model.

- a Copy the following function model definition, ready to paste into the Function Model Wizard:

```
bsfc = 5400 / pi * basefuelmass / tq
```

- b Select **File > New > Function Model**.
- c Paste or enter `bsfc = 5400 / pi * basefuelmass / tq` and click **Next**.

Assuming: base fuel mass [mg/inj], Tq [Nm], 6 cylinders, 4 stroke

- d Select **Automatically assign/create inputs**. It is important to check for typos or this step can create unintended new inputs. Click **Finish**.

The Models view appears.

- 3 This optimization requires a constraint on the air/fuel ratio (AFR). However, as you could not model AFR directly you need to create a function model that relates equivalence ratio to air/fuel ratio. Then you can constrain AFR. To do this, create another new function model.

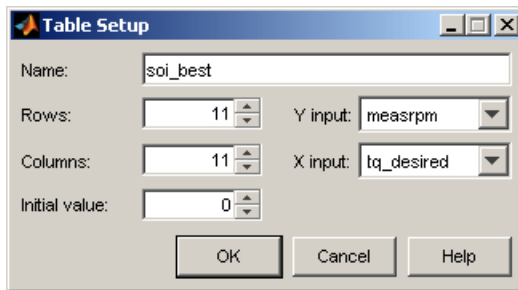
- a Copy the following function model definition, ready to paste into the Function Model Wizard:

`afr = 14.46/eqrexh`

- b** Select **File > New > Function Model**.
 - c** Paste or enter `afr = 14.46/eqrexh` and click **Next**.
 - d** Select **Automatically assign/create inputs** and click **Finish**.
- 4** Select **File > New > Variable Item > Variable**. The Variable Dictionary view appears.
- a** Rename the new variable `afr_min`.
 - b** Set the range of this variable to be minimum 0 and maximum 100
 - c** Edit the set point to 50

Setting Up Calibration Tables to Fill

- 1 Set up a new 2-D table. Select **File > New > 2D Table**.
- 2 Name the table `soi_best`.
- 3 Select `measrpm` and `tq_desired` from the **Y Input** and **X Input** drop-down menus.
- 4 Enter 11 rows and 11 columns.
- 5 Enter 0 for the initial value.



- 6 Click **OK** to create the table.

Look at the **Tables** tree on the left. Note that the normalizers appear as calibratable items in their own right, and as descendants (child nodes) of their tables.

- 7 Once you create a table, you can duplicate it to create more tables that share the same breakpoints. Click `soi_best` in the tree, then select **Edit > Duplicate soi_best**. Repeat until you have four new tables, and rename them as follows:
 - a `basefuelmass_best`
 - b `fuelpress_best`
 - c `grackmea_best`
 - d `egr1ft_best`

Setting Up the Point Optimization

CAGE provides a flexible optimization environment. You can define the objectives, the constraints, and the points where the optimization is carried out.

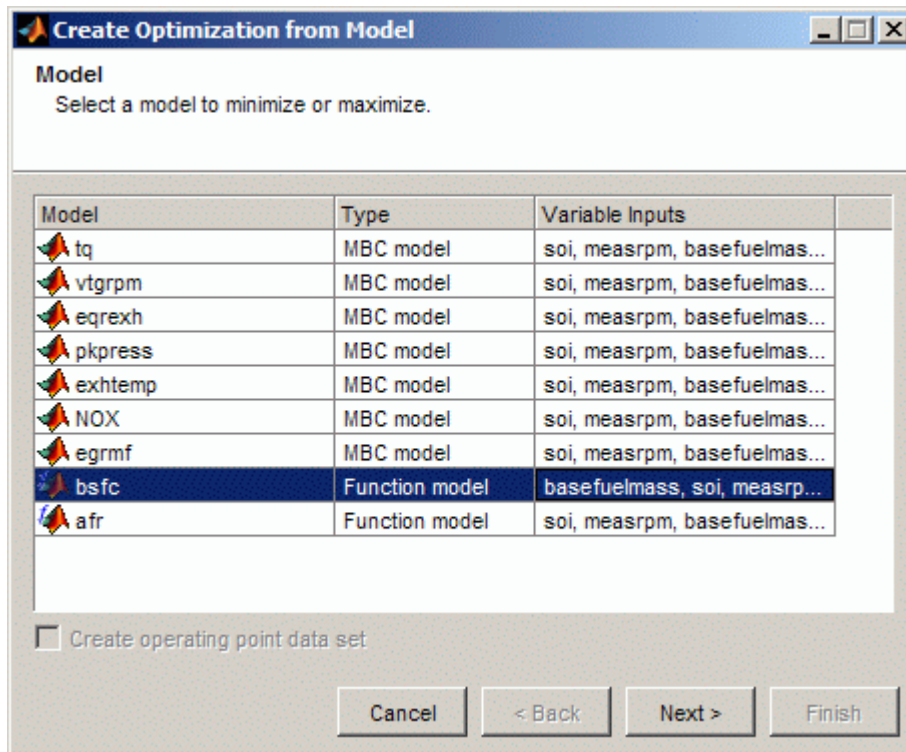
The objective is to minimize weighted BSFC over a set of points, subject to a set of constraints. This is a constrained single objective optimization problem.

You solve this problem in two parts:

- 1 Run an initial point optimization to explore the problem point-by-point in order to obtain good starting points for the sum optimization. You will create the optimization, set up constraints and values and run the optimization.
- 2 Next you can use the solutions from the initial optimization as the start points to run the sum optimization over the drive cycle to find optimal control settings. Use these results to fill calibration tables.

CAGE has several built-in optimization routines and the capacity for you to write your own. In this case study, you use `foptcon`. This algorithm is a modified version of `fmincon` from the Optimization Toolbox product. In CAGE, you can use the algorithm to minimize or maximize an objective function.

- 1 Select **Tools > Create Optimization from Model**, or click the toolbar button. The Create Optimization from Model Wizard appears.
- 2 Select the `bsfc` model to minimize in the optimization.

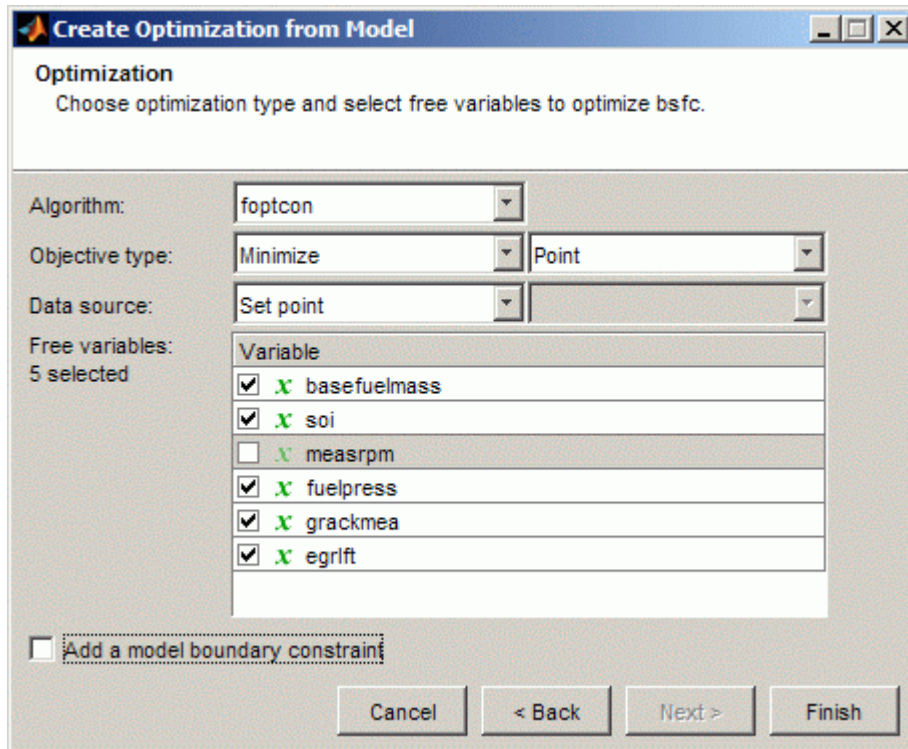


Click **Next**.

3 Select the optimization settings as follows:

- **Free variables:** Clear the check box for `measrpm`. Leave the selected check boxes of the other free variables. These are the variables you want CAGE to optimize from the set of model inputs.
- **Add a model boundary constraint:** Clear the check box. You add another boundary constraint later in the Optimization view.
- Leave the defaults for all the other settings:
 - **Algorithm:** Use the default `foptcon` for gradient-based single-objective optimizations.

- **Objective type:** Leave the default set to Minimize your model, and leave the default Point objective type.
- **Data Source:** Leave the default Set point. You specify points later in the Optimization view.
- Click **Finish** to create the optimization.



CAGE displays the **Optimization** view with your new optimization.

You have not yet set up your constraints, so you can add them next.

Setting Up Constraints

This case study problem has CAGE model constraints on the following quantities (which you will set up next):

- tq_desired
- vtgrpm
- pkpress
- afr
- soi
- grackmea
- fuelpress
- basefuelmass

In the Optimization view, the constraints you will set up next will be shown as follows.

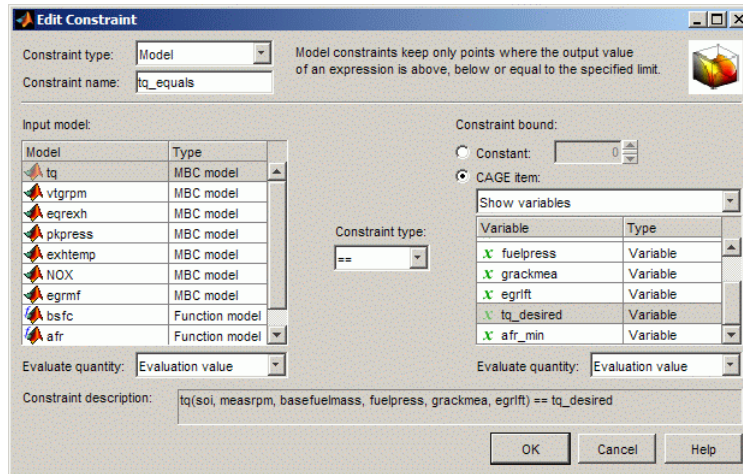
Constraints	
Name	Description
tq_equals	tq(soi, measrpm, basefuelmass, fuelpress, grackmea, egrift) == tq_desired
afr	afr(soi, measrpm, basefuelmass, fuelpress, grackmea, egrift) >= afr_min
vtgrpm	vtgrpm(soi, measrpm, basefuelmass, fuelpress, grackmea, egrift) <= 128000
pkpress	pkpress(soi, measrpm, basefuelmass, fuelpress, grackmea, egrift) <= 18000000
tq_Boundary	Boundary constraint of tq(soi, measrpm, basefuelmass, fuelpress, grackmea, egrift)

You can edit constraint names to aid analysis in other Optimization views (right-click, or double-click to use the Edit Constraint dialog box).

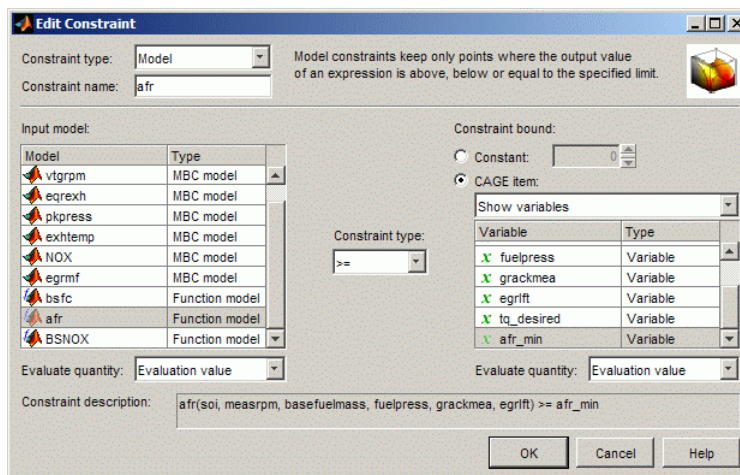
Set up these constraints as follows:

- 1 Right-click in the Constraints pane and select **Add Constraint**. The Constraint Editor appears. Set up the first constraint to `tq == tq_desired` as follows.
 - a Leave the Constraint type as **Model**.
 - b Edit the Constraint name to `tq_equals`.
 - c Select `tq` in the **Input model** list.
 - d Select `==` as the **Constraint type**.
 - e Select the **CAGE item** radio button.
 - f Select **Show variables** from the drop-down list.

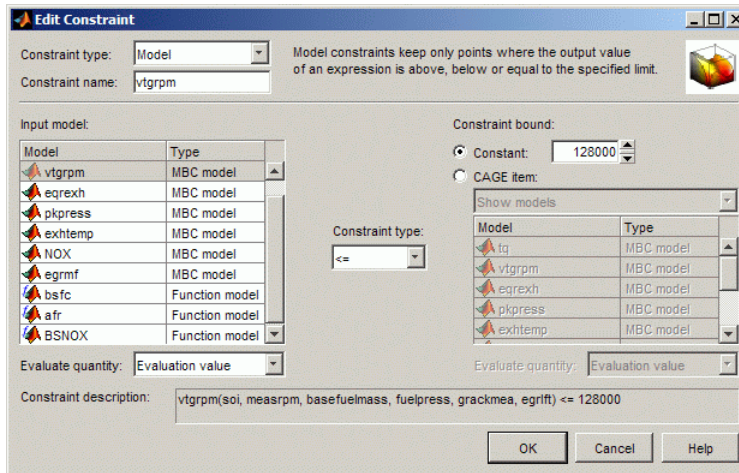
- g Select `tq_desired` in the variable list. The dialog should look as shown in the following figure.



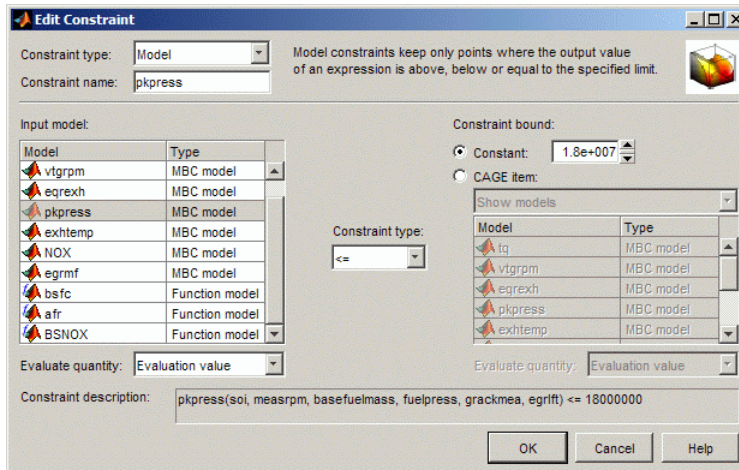
- h Click **OK** to return to the Optimization view.
- 2 Similarly, right-click to add the next constraint as shown: `afr >= afr_min`.



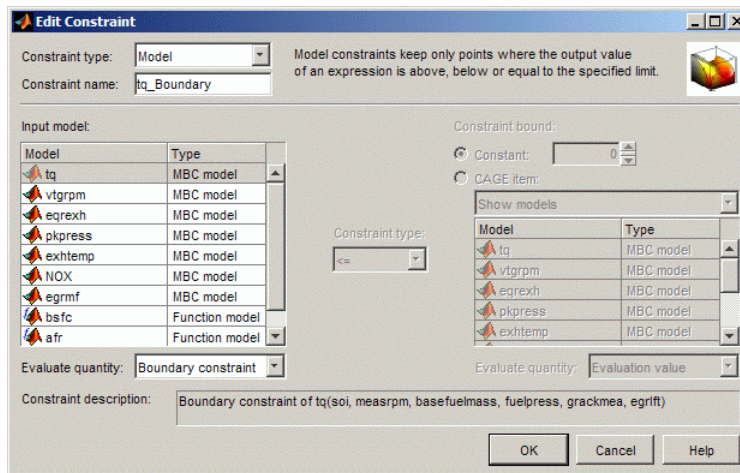
- 3 Right-click again and add the next constraint as shown: `vtgrpm <= 128000`. To do so, enter a value in the **Constant** edit box, rather than select a variable from the CAGE item list.



- 4 Right-click again to set up the constraint pkpress ≤ 18000000 , as shown. This time, enter a value in the **Constant** edit box.



- 5 Repeat the action to set up a boundary model constraint, as shown in the following figure.



- a Leave the Constraint type as Model.
- b Edit the Constraint name to tq_Boundary.
- c Select tq in the **Input model** list.
- d Select Boundary constraint in the **Evaluate quantity** drop-down list.
- e Click **OK** to create the constraint and return to the Optimization view.

Defining Variable Values

You need to define the set of drive cycle points where you want the optimization to run. To do this you use the **Input Variable Values** pane in the Optimization view.

- 1 Increase the **Number of runs** to 7. Click the buttons or enter the value in the box. The number of rows in the fixed and free variables panes increases to 7. The default values are the set point of each variable. Leave the initial values for the free variables at the defaults.
- 2 You can enter or copy values into the **Fixed Variables** pane to define the fixed variable values at each point where you want the optimization to run. You can copy all the variable values from a text file, or from the Help Browser copy each column in turn. Copy and paste the following values into the `measrpm` column in the **Fixed Variables** pane.

```
measrpm
2200
2200
2200
2200
1600
1600
1600
```

- 3 Copy and paste the following values into the `tq_desired` column in the **Fixed Variables** pane.

```
tq_desired
1263
947
632
126
1550
1163
775
```

- 4 Copy and paste the following values into the `afr_min` column in the **Fixed Variables** pane.

```
afr_min  
25.5  
27.75  
30.0  
0  
22  
22.5  
23
```


The **Fixed Variables** pane should look as shown. You have specified the speed and torque values you want for each run, and also the minimum air/fuel ratio (AFR) at each N, TQ point.

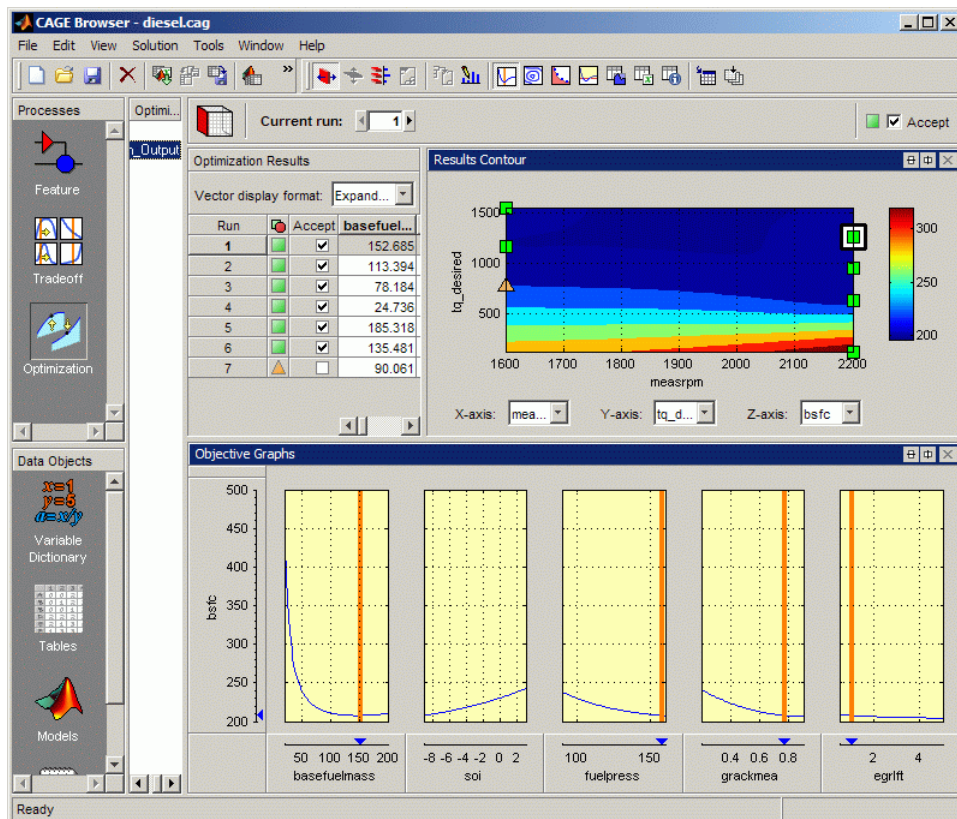
Fixed Variables			
Variable:	measrpm	tq_desired	afr_min
Number of values:	<input type="text" value="1"/>	<input type="text" value="1"/>	<input type="text" value="1"/>
1	2200	1263	25.5
2	2200	947	27.75
3	2200	632	30
4	2200	126	0
5	1600	1550	22
6	1600	1163	22.5
7	1600	775	23

If you wished to constrain the free variables more than the range defined in the variable dictionary, you could select **Optimization > Edit Free Variable Ranges**. Leave the defaults for this problem.

Running the Optimization

Now you can run the optimization. You have set up objectives, constraints and a set of operating points.

- 1 Click Run Optimization  in the toolbar.
- 2 When the optimization is complete, the view switches to the new child node, `bsfc_Optimization_Output`, in the Optimization tree under the `bsfc_Optimization` node. View the results.



- 3 Look through the solutions at different operating points by clicking cells in the output table. Compare with the gasoline example, where regions that do not meet the constraint are yellow in the graphs — in this case the torque equality constraint

causes all the graphs to be yellow, because this constraint produces a feasible contour in the free variable space, and any deviation off this contour is infeasible.

- 4 Split the view to display the constraint graphs, and scroll through them to see each constraint in relation to the solutions.

Setting Up the Sum Optimization

In this section...

“Setting Up Initial Values and Sum Objective” on page 4-31

“Creating the Brake Specific NOx Constraint” on page 4-32

“Setting Weights for the Sum Objective and Constraint” on page 4-33

“Set Parameters and Run Optimization” on page 4-35

Setting Up Initial Values and Sum Objective

Previously, you created the initial point-by-point optimization to obtain good starting points for the sum optimization. Now, you can create the sum problem from that initial optimization. You then use the solutions from the initial optimization as the start points to run the sum optimization over the drive cycle to find optimal settings. Then, you can fill the calibration tables with these results.

- From the `bsfc_Optimization_Output` node of your first optimization, select **Solution > Create Sum Optimization**.

CAGE creates a new optimization called `Sum_bsfc_Optimization`. The optimization has these characteristics:

- The objective matches your original optimization but converted to a sum objective.
- The new optimization has identical constraints to your original optimization.
- Your original fixed and free variables have converted to a sum optimization (a single run with multiple values).
 - The new optimization uses only accepted solutions from your original optimization output (all runs with a selected **Accept** check box).
 - Therefore, the number of accepted solutions you had in the original optimization determines the number of values within the sum optimization run.
- The new optimization imports the free variable initial values and fixed variable values from your point optimal results (accepted solutions only).
- The fixed variables have a **Weights** column with each value set to 1.

In the Optimization view, observe the new sum objective and the new values in the Input Variable Values pane.

Now your sum optimization is ready to run. Before you run it, however, see the next section to create a brake specific NOx constraint.

Creating the Brake Specific NOx Constraint

In this optimization, you will use brake specific NOx to constrain the results.

BSNOx is defined as:

$$BSNOx = \frac{\sum_{i=1}^N w_i NOx_i}{2\pi/60000 \sum_{i=1}^N w_i N_i TQ_i}$$

You implement this constraint as a weighted sum constraint in CAGE by creating a function model that represents the following term of the BSNOx sum:

$$BSNOx_term = \frac{NOx}{2\pi/60000 \sum_{i=1}^N w_i N_i TQ_i}$$

The denominator of this term represents the weighted power (speed * torque) over the drive cycle, which is constant for this optimization. You calculate this constant as follows:

$$\begin{aligned} & 2\pi/60000 \sum_{i=1}^N w_i N_i TQ_i \\ &= 2\pi/60000 \left(\begin{array}{l} 2200 \times 1263 + 2200 \times 947 + 2200 \times 632 + 2200 \times 126 + \\ 1600 \times 1550 + 1600 \times 1163 + 1600 \times 775 \end{array} \right) \\ &= 159.5573 \end{aligned}$$

Therefore, the function model to be created is

$$BSNOx_term = \frac{NOx}{159.5573}$$

Create the function model as follows:

- 1 Copy the following function model definition, ready to paste into the Function Model Wizard:

$$\text{BSNOX} = 3600 * 1000 * \text{NOX} / 159.5573$$

- 2 Select **File > New > Function Model**.
- 3 Paste or enter $\text{BSNOX} = 3600 * 1000 * \text{NOX} / 159.5573$ and click **Next**.

The NOx model is in kilograms per hour, and you want to calculate BSNOx in grams per second, so multiply NOx by 3600*1000.

- 4 Select **Automatically assign/create inputs**. It is important to check for typos or this step can create unintended new inputs. Click **Finish**.

The Models view appears with the new function model. Now, you can use this function model to create a weighted sum constraint.

Setting Weights for the Sum Objective and Constraint

To create a weighted sum constraint:

- 1 Click the Optimization button to return to the Optimization view.
- 2 Select your sum optimization.
- 3 Right-click in the Constraints pane, and select **Add Constraint**. The Edit Constraint dialog box appears.

Edit Constraint

Constraint type: A sum constraint provides a constraint distance value for a weighted sum of a model with reference to a bound

Constraint name:

Constraint	Type
tq	MBC model
vtgrpm	MBC model
eqrexh	MBC model
pkpress	MBC model
exhtemp	MBC model
NOX	MBC model
egrmf	MBC model
bsfc	Function model
afr	Function model
BSNOX	Function model

Constraint type: Constraint bound:

Constraint description:

OK Cancel Help

- 4 Select **Sum Constraint** from the **Constraint type** drop-down menu.
- 5 Enter **BSNOX** in the **Constraint name** edit box.
- 6 Select **BSNOX** from the **Input model** list.
- 7 Enter **3** in the **Constraint bound** edit box, and press **Enter** to create the constraint, and return to the Optimization view. Verify that the new constraint description shows **Weighted sum of BSNOX <= 3**, to constrain the weighted sum to less than 3g/kWh.

Set the weights in the Optimization view as follows:

- 1 Enter **7** in the **Number of values** edit box for the `bsfc_weights` column.
- 2 Enter, or copy and paste, the following values into the `bsfc_weights` column in the **Fixed Variables** pane.

Objective 1 Weights

0.5
0.15
0.15
0.05
0.05
0.05
0.05

These values sum to 1.

- 3 Similarly, enter **7** in the **Number of values** edit box for the `BSNOX_weights` column.
- 4 Copy the following values into the `BSNOX_weights` column in the **Fixed Variables** pane.

Constraint 1 Weights

0.15
0.15
0.15
0.1
0.1

Constraint 1 Weights

0.1

0.1

These values sum to 0.85.

Set Parameters and Run Optimization

To alter parameters and then run the optimization:

- 1 Click Set Up and Run Optimization in the toolbar.
- 2 Change the following parameters in the Optimization Parameters dialog box:
 - **Maximum iterations** = 500
 - **Maximum function evaluations** = 2000
 - **Variable tolerance** = $1e-7$
 - **Function tolerance** = $1e-7$

Click **OK** to close the dialog box, and the optimization runs.

- 3 When the optimization is complete, the view switches to the updated child node, `Sum_bsfc_Optimization_Output`, in the Optimization tree under the `Sum_bsfc_Optimization` node. View the results. For guidance see “Interpreting Sum Optimization Output”.

Filling Tables with Optimization Results

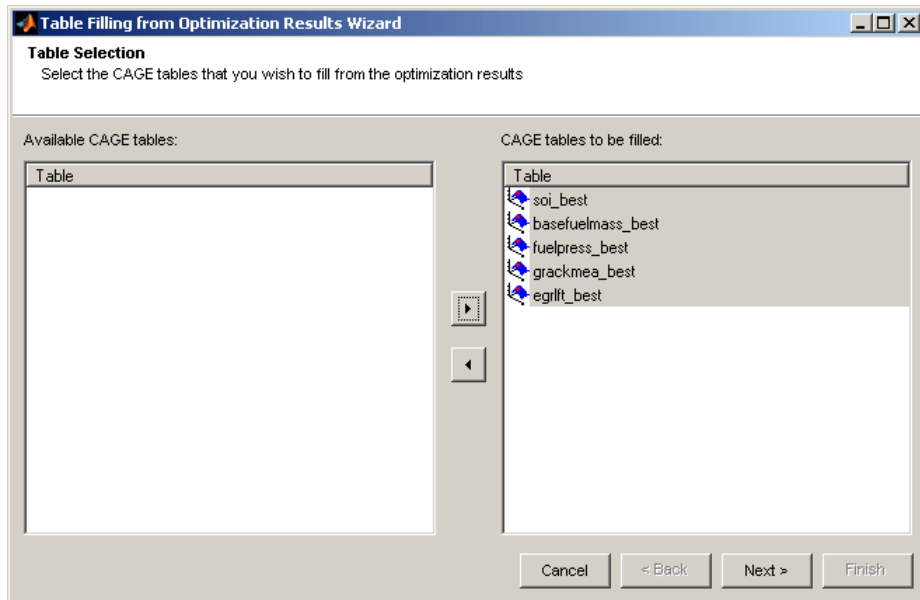
You can use the optimization results to fill the tables you created.

- 1 From the optimization output node, select **Solution > Fill Tables**.

The Table Filling Wizard appears.

- 2 **Shift**-click to multi-select all the following tables and click the button to add all your tables to the filling list:

- soi_best
- basefuelmass_best
- fuelpress_best
- grackmea_best
- egrlft_best

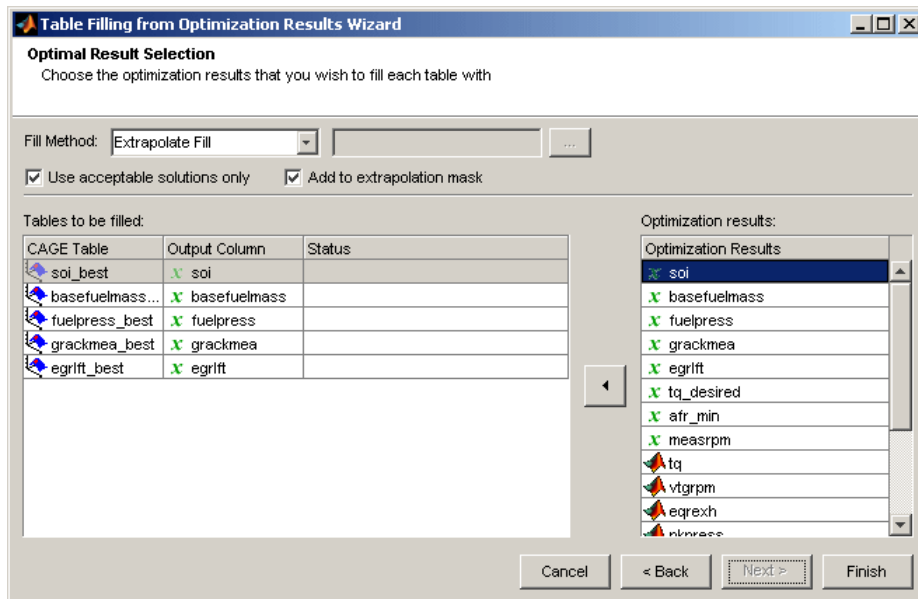


Click **Next**.

- 3 Match the following pairs of an output from the right list of optimization results with a table on the left.

- soi with soi_best
- basefuelmass with basefuelmass_best
- fuelpress with fuelpress_best
- grackmea with grackmea_best
- egrlft with egrlft_best

You can double-click in the right list to select an item, and CAGE automatically moves focus on to the next table in the left list. For example, with the `soi_best` table selected in the left list, double-click `soi` in the right list, then you can double-click `basefuelmass` for the next table, and so on.



4 Click **Finish**.

A dialog appears indicating whether the tables have been filled successfully. Click **Close**.

5 Switch to the **Tables** view to view the filled tables. Click **Tables** in the **Data Objects** pane, and select the tables in turn to view the results.

Look at the example finished project, `Diesel_optimization.cag`.

Point-by-Point Diesel Engine Calibration Case Study

- “Point-by-Point Diesel Case Study Overview” on page 5-2
- “Create Designs and Models Programmatically” on page 5-5
- “Verify and Refine Models” on page 5-7
- “Point-by-Point Optimization Overview” on page 5-9
- “Create Point Optimizations in CAGE” on page 5-10
- “Create Multiregion Sum Optimization” on page 5-19
- “Use Optimization Results” on page 5-27

Point-by-Point Diesel Case Study Overview

In this section...
“What Is Point-by-Point Modeling?” on page 5-2
“Engine Calibration Specifications and Problem Description” on page 5-2
“Required Tasks” on page 5-4

What Is Point-by-Point Modeling?

This case study shows how to use the Model-Based Calibration Toolbox functionality for *point-by-point* engine calibration projects.

Point-by-point functionality allows you to build a model at each operating point of an engine with the necessary accuracy to produce an optimal calibration. You often need point-by-point models for multiple injection diesel engines and gasoline direct-injection engines. You use point-by-point command-line functionality to handle the complexity of developing designs for each operating point.

Point-by-point models can solve calibration problems with increasingly complex engines. Engineers add engine actuators and sensors to Engine Management Systems (EMS) to respond to emerging requirements for fuel economy, performance, and control of engine emissions. In some cases, optimal engine calibration development processes that rely on two-stage modeling can no longer model engine performance responses accurately enough.

Note: Point-by-point models can provide the necessary model accuracy at *measured* operating points. However, such models do not provide estimated responses at other operating points.

Engine Calibration Specifications and Problem Description

The two-stage models in the “Diesel Engine Calibration” case study generated the point-by-point data for this example. The example engine is the same six-cylinder 9.0-L common-rail L common-rail diesel engine with VGT (variable geometry turbo) and cooled EGR (exhaust gas recirculation). The control calibration is for an off-road application with an engine speed range from 1600 rpm to 2200 rpm.

The objective of the case study is to produce optimal calibration schedules as a function of commanded torque and speed for:

- Optimal SOI (start of injection)
- Fuel pressure
- Base fuel
- VGT (variable geometry turbo)
- EGR (exhaust gas recirculation)

While speed and torque define the drive cycle, the engine is tested by controlling speed and total fuel. The engine test procedure adjusts the total fuel to achieve the required torque for each point.

The case study involves models for BSFC (brake-specific fuel consumption), BSNOX (brake-specific NOx emissions), AFR (air/fuel ratio), EGR mass fraction, peak pressure, VGTSPEED, and base fuel.

The optimization setup in CAGE uses an 8-mode off-road emission test as its basis. This setup approximates up to seven mode points by neglecting the idle operating point of the engine.

The engine calibration requires tables in speed and torque for the following variables:

Best injection timing	MAINSOI
Best fuel quantity	MAINFUEL
Best fuel pressure	FUELPRESS
Best VGT position	VGTPPOS
Best EGR position	EGRPOS

To fill these tables, create optimizations to minimize mode-weighted brake-specific fuel consumption, subject to constraints on the following parameters:

- The boundary model defining the data boundary at each operating point (BSFC_Boundary)
- Brake-specific NOx emissions (BSNOX)
- Cylinder pressure (PEAKPRESS) at full load points only
- Air/fuel ratio (AFR)

To solve this problem, follow the steps described in “Required Tasks” on page 5-4

Required Tasks

This example requires you to complete the following tasks:

- Create designs, model, and boundary models at each operating point. See “Create Designs and Models Programmatically” on page 5-5.
- View and analyze models using the Model Browser. See “Verify and Refine Models” on page 5-7.
- Optimize the model. See “Point-by-Point Optimization Overview” on page 5-9.

Creating a point-by-point optimization for the model requires these tasks:

- 1 “Create Point Optimizations in CAGE” on page 5-10
- 2 “Create Multiregion Sum Optimization” on page 5-19
- 3 “Use Optimization Results” on page 5-27

Create Designs and Models Programmatically

In this section...

“Overview of Programmatic Design and Modeling” on page 5-5

“Creating Designs and Models Programmatically” on page 5-5

Overview of Programmatic Design and Modeling

The programmatic part of the case study shows automated creation of designs, models, and boundary models at each operating point. Alternatively, you could use the Model Browser to create models interactively.

This example uses the two-stage models generated in the “Diesel Engine Calibration” case study as a surrogate for an engine dynamometer or engine simulation model. The models generate the point-by-point data for this example. The example shows you how to:

- 1 Generate local designs at each operating point.
- 2 Collect response data at the design points using the diesel case study models.
- 3 Augment the local design points using spacefilling Sobol sequences if you need more design points after initial data collection.
- 4 Create local multiple models to model each of the responses at each operating point.
- 5 Build a point-by-point boundary model to define the data boundary at each operating point for later use in calibration optimization.

Creating Designs and Models Programmatically

To create designs and models in a project:

- 1 View the command-line example instructions for “Point-by-point Modeling for a Diesel Engine” by entering:

```
edit mbcPointByPointCmdLine
```

- 2 Run the example to create designs, a project, response models, and boundary models.
- 3 Save the project to a file before loading it into the Model Browser by entering:

```
project.Save('DieselPointByPoint.mat');
```

After you complete the programmatic design, inspect and refine the fitted models to verify that the model quality is acceptable. You can use the command line to plot diagnostics and remove outliers, but it is easier to use the graphical and statistical tools in the Model Browser. For instructions, see “Verify and Refine Models” on page 5-7.

Verify and Refine Models

In this section...

“Open the Project and View Designs” on page 5-7

“Analyze and Refine Local Fits” on page 5-7

Open the Project and View Designs

Visually inspect your fitted models to verify that the model quality is acceptable. Identify and remove any outliers that affect quality. To verify the models, open the project you created and saved in the previous section (“Create Designs and Models Programmatically” on page 5-5). You can use the command line to plot diagnostics and remove outliers. However, as a best practice, use the graphical and statistical tools in the Model Browser to analyze and refine your models .

- 1 To open the project in the Model Browser, locate `DieselPointByPoint.mat` in the `mbctraining` folder.
- 2 To view the local DOE in the Design Editor:
 - a Select the test plan node in the Model Tree, and the test plan tab.
 - b Right-click the local model block in the test plan diagram, and select **Design Experiment**.

The Design Editor opens.

- 3 View the designs. The local designs for each test are at the top level of the design tree.
- 4 Close the Design Editor.

Analyze and Refine Local Fits

The example projects for this case study include verified models. To view the models, load the Model Browser project file `DieselPointByPoint.mat` from the `mbctraining` folder.

Analyze fits for each response as follows:

- In the Model Browser, select each response node in turn in the Model Tree. Start with BSFC.

- For each response node, look for problem tests and possible outliers. Use the following tools:
 - To find poorly fitted tests, look for tests with high RMSE in the RMSE Plots. Double-click points to view a test in the other plots.
 - Compare alternative model types for the currently selected operating point by looking at the list of **Alternative Local Models**.
 - You can check for tests that are not fitted by clicking **Select Test** to open the Test Selector dialog box.

Point-by-Point Optimization Overview

The CAGE Browser section of the case study covers using the models to create optimized calibration tables. The CAGE browser part of the example takes you through the following steps:

- 1** Create two separate point optimizations for use in combination in a drive cycle optimization. You create two optimizations so you can apply the peak pressure constraint only where it is relevant, at full load conditions.
 - a** Create a constrained point optimization to minimize BSFC (brake-specific fuel consumption) for each part load point.
 - b** Create a constrained point optimization to minimize BSFC for each full load point.
 - c** Export the results to a data set. You use the results from both point optimizations as start points for a sum optimization.
- 2** Create a multiregion constrained drive cycle sum optimization.
 - a** Import the results from both point optimizations as start points.
 - b** Apply the peak pressure constraint only at full load points.
 - c** Add a second drive cycle with different weights.

You can use the second drive cycle to evaluate optimization results at intermediate points. The second drive cycle for objective and constraints includes the measured points (that is, the model operating points) and intermediate speed points.

- 3** Create tables to fill with your optimization results using the Create Tables from Model wizard.
- 4** Fill the tables with your optimization results.
- 5** Export the tables to a calibration tool.

The toolbox provides an example project containing the models, optimizations, results, and filled tables. To view the finished project, load the CAGE browser project file `DieselPointByPoint.cag` from the `mbctraining` folder.

For the next task in solving this point-by-point optimization, see “Create Point Optimizations in CAGE” on page 5-10.

Create Point Optimizations in CAGE

In this section...

“Introduction” on page 5-10

“Load Models to Optimize” on page 5-10

“Create Part Load Point Optimization” on page 5-11

“Create Full-Load Point Optimization” on page 5-15

Introduction

This section takes you through the steps to create two separate point optimizations for use in combination in a drive cycle optimization. You create two optimizations so you can apply the peak pressure constraint only where it is relevant, at full load conditions.

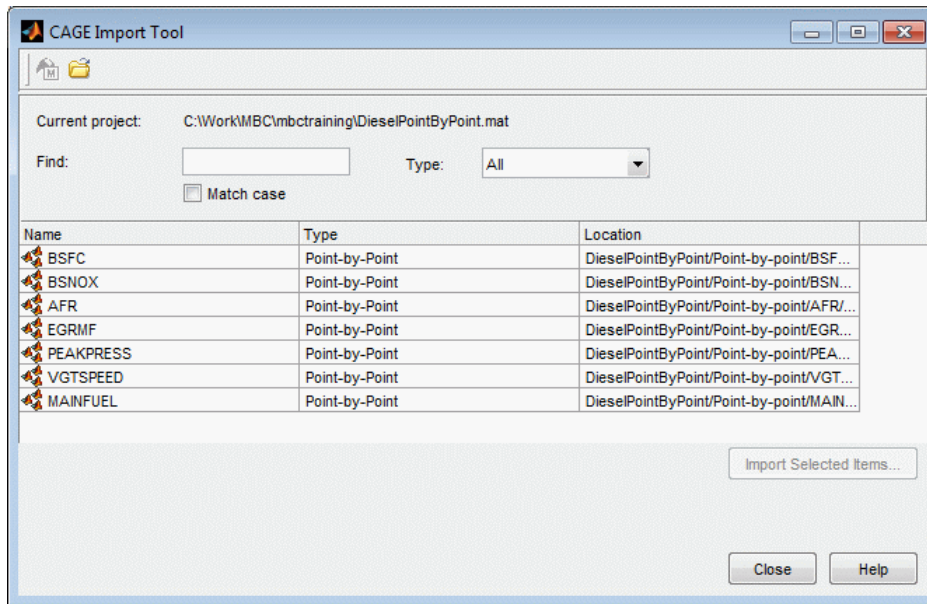
- 1 Create a constrained point optimization to minimize BSFC (brake-specific fuel consumption) for each part load point.
- 2 Create a constrained point optimization to minimize BSFC for each full load point.

You use the results from both point optimizations as start points for a sum optimization.

Load Models to Optimize

To open CAGE and load the models for the optimization, follow these steps:

- 1 Enter `cage` to open the CAGE browser.
- 2 Load models to optimize. Select **File > Import From Project**. The CAGE Import Tool opens.
- 3 Click **Import From Project File**.
- 4 Locate the `mbctraining` folder and open the file `DieselPointByPoint.mat`.



- 5 Press **Shift**+click to select all models in the list, and click **Import Selected Items**. Then, click **OK** in the dialog box that appears next to import all the models. Then, close the Import Tool.

CAGE displays the variable connections for your imported point-by-point models.

Create Part Load Point Optimization

To create the optimization, you perform the following tasks:

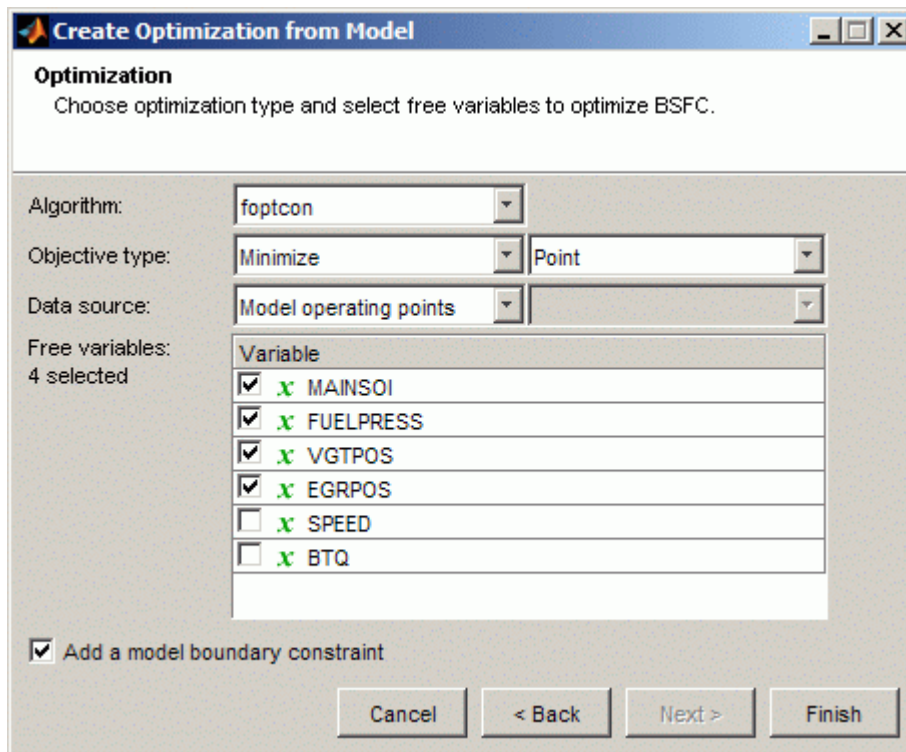
- “Select Model and Parameters for Optimization” on page 5-11
- “Specify Number of Runs” on page 5-13
- “Add a Constraint to the Optimization” on page 5-13
- “Run the Optimization” on page 5-15

Select Model and Parameters for Optimization

- 1 Select **Tools > Create Optimization From Model**.

The Create Optimization From Model wizard opens.

- 2 Select **BSFC** as the model to optimize, and click **Next**.
- 3 Leave all the defaults as shown to create a point optimization that:
 - Minimizes **BSFC**
 - Using the **Model operating points** as the **Data source**
 - With fixed variables **SPEED** and **BTQ**
 - Selects all other variables as free variables for **CAGE** to optimize
 - Adds the boundary model as a constraint



- 4 Click **Finish** to create the optimization.

CAGE displays your optimization. Rename your optimization to **BSFC_PartLoad** and continue to “Specify Number of Runs”.

Specify Number of Runs

Observe the model operating points in the Optimization Point Set pane. Remove two of the runs. Later, your second optimization applies another constraint at the full load points.

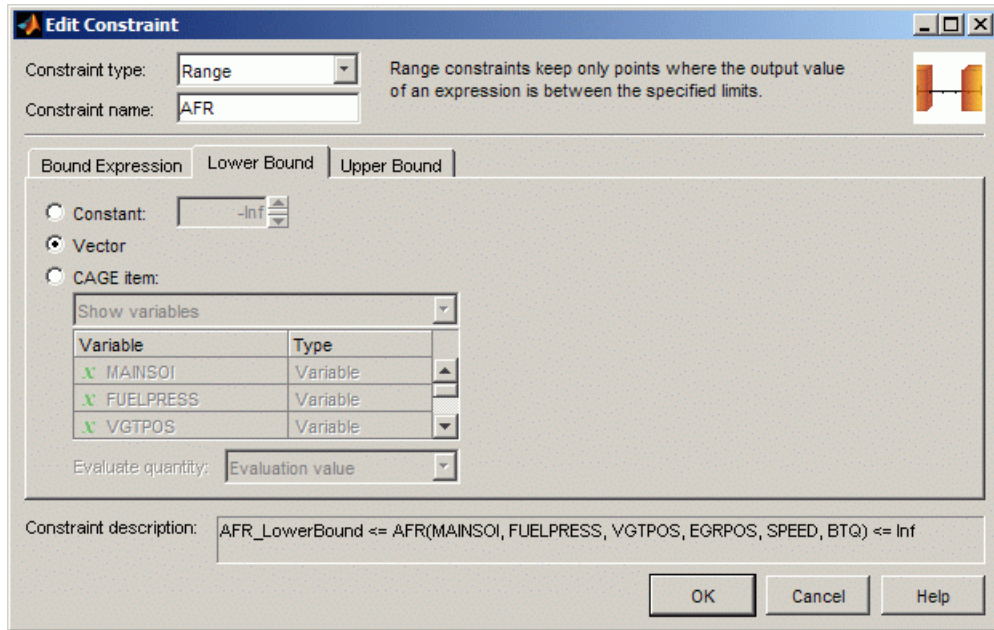
- 1 Right-click the first run, where SPEED = 2200 and BTQ = 1263, and select **Delete Runs**.
- 2 Right-click the fourth run, where SPEED = 1600 and BTQ = 1550, and select **Delete Runs**.

Your optimization now has five runs. Continue to “Add a Constraint to the Optimization”.

Add a Constraint to the Optimization

Add a constraint for air/fuel ratio before running the optimization. Follow these steps:

- 1 Right-click the Constraints pane, and select **Add Constraint**.
- 2 Select **Constraint type Range**.
- 3 Select **Show models** from the drop-down menu on the **Bound Expression** tab.
- 4 Select AFR in the model list.
- 5 Edit the constraint name to AFR.
- 6 Click the **Lower Bound** tab, and select the **Vector** option button. Ensure the **Constraint description** matches the following example, and click **OK**.



- 7 When the variable AFR_LowerBound appears in the Fixed Variables pane, copy and paste the following values into the AFR_LowerBound column.

AFR_LowerBound
27.5
30.0
0.0
20.0
22.5

Verify that your fixed variable values look like those in the following image.

Fixed Variables			
Variable:	SPEED	BTQ	AFR_Lo...
Number of values:	1	1	1
1	2200	947	27.5
2	2200	632	30
3	2200	126	0
4	1600	1163	20
5	1600	775	22.5

Run the Optimization

You can now run your optimization:

- 1 Select **Optimization > Run**.
- 2 After CAGE calculates the optimization and displays the output, view the results.

Create Full-Load Point Optimization

Creating the full-load point optimization requires you to set up the optimization, define runs at full load, add the peak pressure constraint, and define required variables.

- “Set Up the Optimization” on page 5-15
- “Import Variables to Define Constraint and Weights” on page 5-16
- “Add Peak Pressure Constraint and Define Fixed Variable Values” on page 5-17

Set Up the Optimization

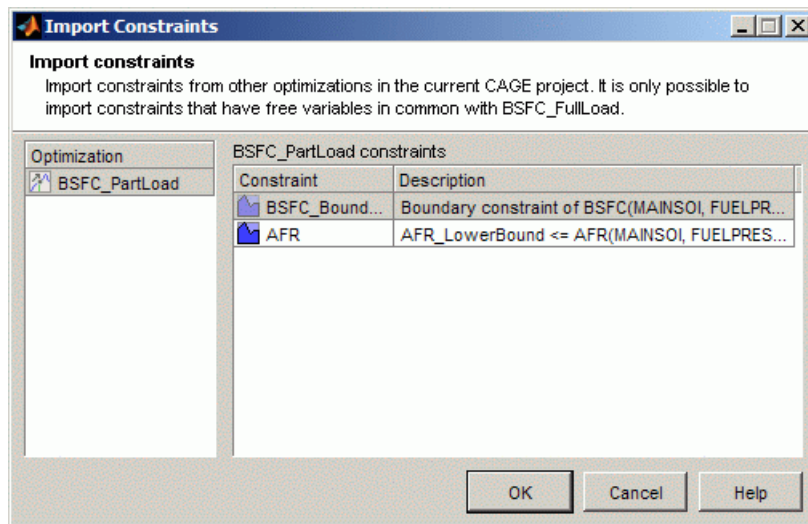
To set up the optimization, follow these steps:

- 1 Select **Tools > Create Optimization From Model**.

The Create Optimization From Model wizard opens.

- 2 Select **BSFC** as the model to optimize, and click **Next**.
- 3 Leave all the defaults, and click **Finish** to create the optimization.
- 4 Rename the new optimization to **BSFC_FullLoad**.
- 5 Delete all but two runs in the Optimization Point Set pane, because this optimization uses only the full-load points. Right-click and delete runs to leave only these two:

- SPEED = 2200 and BTQ = 1263
 - SPEED = 1600, and BTQ = 1550
- 6 Import the AFR constraint from the previous optimization.
 - a Right-click the Constraints pane, and select **Import Constraints**.



- b Select the AFR constraint, and click **OK**.

Import Variables to Define Constraint and Weights

You need some new variables to define a constraint and also to define some weights for the multiregion optimization. To save time you can import the variables and weights from the example project, rather than defining them manually, as follows:

- 1 Select **File > Import From Project**. The CAGE Import Tool opens.
- 2 Click **Import From Project File**.
- 3 Locate the `mbctraining` folder and open the file `DieselPointByPoint.cag`.
- 4 Select **Dataset** from the **Type** drop-down list.
- 5 Select `DriveCycle10` in the list, and click **Import Selected Items**. In the following dialog box, you also import the variables you need for later steps:
 - `BSFC_weights`

- BSNOX_DriveCycle10_weights

Click **OK** to import the data set and variables.

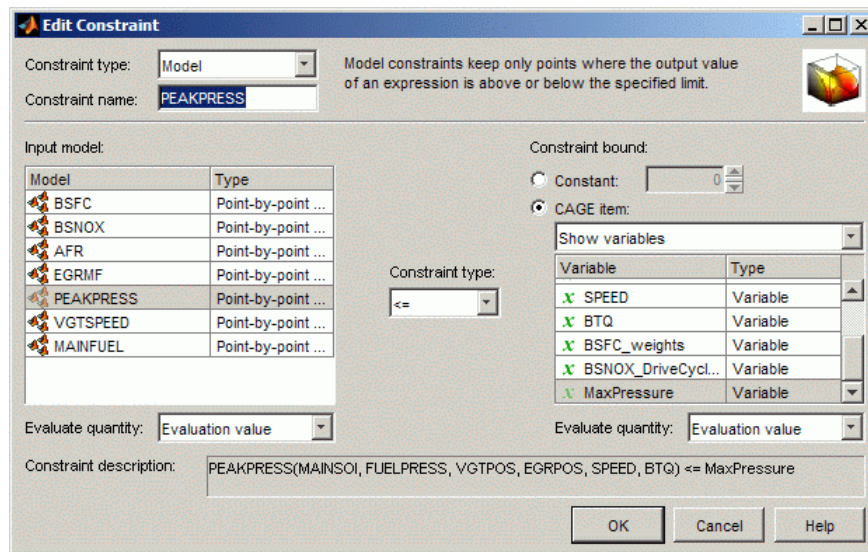
- 6 Select **Variable** from the **Type** drop-down list, select **MaxPressure** in the list, and click **Import Selected Items**. Click **OK** in the following dialog box to import the variable.

Close the Import Tool. You use all the items you imported in later steps.

Add Peak Pressure Constraint and Define Fixed Variable Values

Return to the BSFC_FullLoad optimization node, and add a peak pressure constraint, using the following steps:

- 1 Right-click the Constraints pane, and select **Add Constraint**.
- 2 Select the **PEAKPRESS** model.
- 3 Select the **CAGE item** option button, and then select **Show variables** from the drop-down menu.
- 4 Select **MaxPressure** in the variable list.
- 5 Rename the constraint to **PEAKPRESS**, and click **OK**.



- 6 Copy and paste the following values into the fixed variables columns for AFR_LowerBound and MaxPressure.

AFR_LowerBound	MaxPressure
25.5	15
20	18

When you finish defining all required variables, select **Optimization > Run**. Then, continue next to “Create Multiregion Sum Optimization”.

Create Multiregion Sum Optimization

In this section...

“Introduction” on page 5-19

“Create Data Set from Point Optimization Results” on page 5-19

“Create Sum Optimization” on page 5-20

“Add Application Point Sets” on page 5-21

“Set Up Constraints” on page 5-23

“Define Weights and Fixed Variables and Run Optimization” on page 5-25

Introduction

Creating a multiregion constrained drive cycle optimization requires the following tasks:

- 1 Create the optimization.
- 2 Import the results from both preliminary optimizations as start points.
- 3 Apply one of the constraints only at full load points.
- 4 Add a second drive cycle with different weights to evaluate optimization results at intermediate points.

Create Data Set from Point Optimization Results

You have set up preliminary optimizations to generate suitable start points for the final multiregion sum optimization. To use the results from both previous optimizations as the start points, you can package the results from both optimizations into a single data set. Follow these steps:

- 1 Select the output node of your optimization, `BSFC_PartLoad_Output`.
- 2 Click **Export to data set** in the toolbar.

The **Export To Data Set** dialog box opens.

- 3 Enter `PointResults` for the new data set name, and click **OK**.
- 4 Return to the Optimization view, and select the output node of your second optimization, `BSFC_FullLoad_Output`.

- 5 Click **Export** to data set in the toolbar.

The **Export To Data Set** dialog box opens.

- 6 To add the full load results to the existing data set, select the **Modify existing** option button.
- 7 Select your data set **PointResults** in the list, leave the action set to **Append**, and click **OK**.

Create Sum Optimization

To create the optimization, follow these steps:

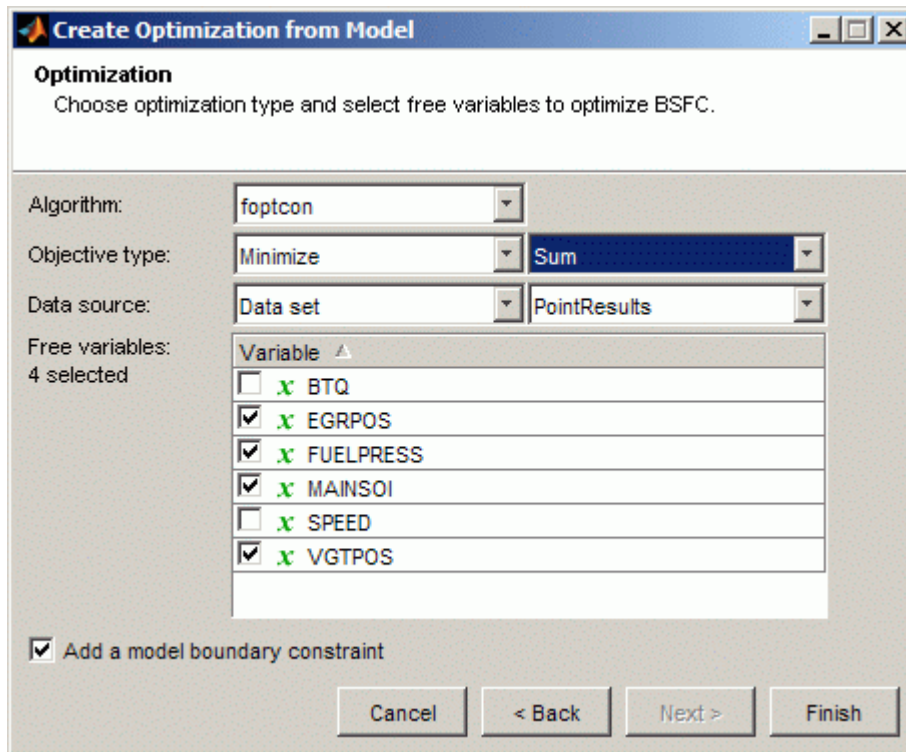
- 1 Select **Tools > Create Optimization From Model**.

The **Create Optimization From Model** wizard opens.

- 2 Select **BSFC** as the model to optimize, and click **Next**.
- 3 Change the **Objective type** from **Point** to **Sum**.
- 4 To use your **PointResults** data set to define the optimization points, change the **Data source** to **Data set**.

Leave the rest of the defaults as shown to create a sum optimization that:

- Minimizes **BSFC**
- With fixed variables **SPEED** and **BTQ**
- Selects all other variables as free variables for **CAGE** to optimize
- Adds the boundary model as a constraint



- 5 Click **Finish** to create the optimization.

Add Application Point Sets

You can use *application point sets* to evaluate constraints and objectives at different operating points than those points specified in the optimization. You can only use application point sets with sum optimizations.

In this case, you want to apply a peak pressure constraint only at full load. You can use an application point set to define the points where you want to apply this constraint.

You also want to evaluate the BSFC objective and a NO_x constraint over a different drive cycle to the optimization points. You can use an application point set to define the secondary drive cycle points. This approach allows you to run an optimization at a subset of points, and evaluate interpolated results across an application point set.

To set up your application point sets, follow these steps:

- 1 To add an application point set to your optimization objective, right-click the objective **BSFC**, and select **Select Application Point Set**

The Select Operating Point Variables dialog box opens.

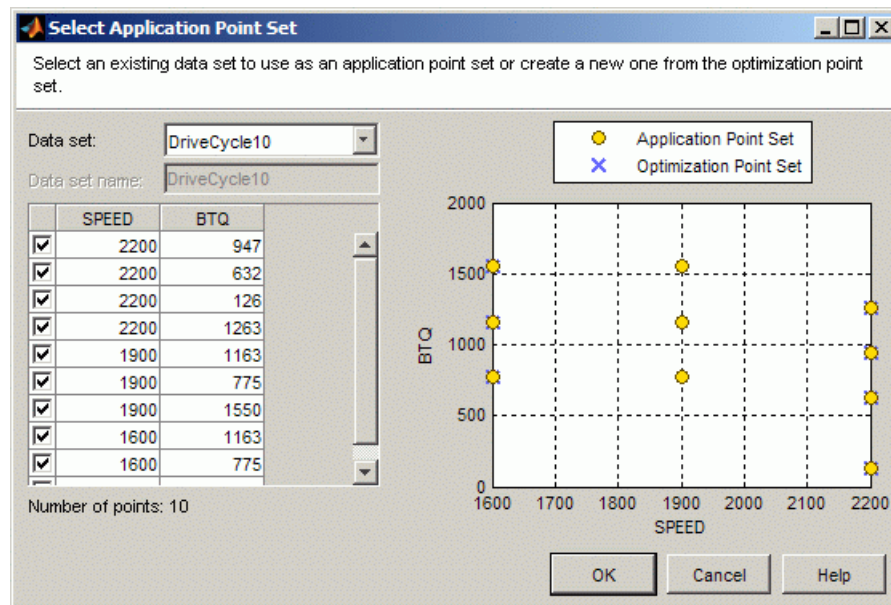
- 2 Define a pair of the fixed variables as the application point variables to use with this optimization. You only select variables once per optimization.

In this case, you only have two fixed variables available (**BTQ** and **SPEED**), so leave the defaults, and click **OK**.

The Select Application Point Set dialog box opens.

- 3 Select the **DriveCycle10** data set you imported earlier. View the plot displaying the application points and optimization points.

Some of the application points coincide with the optimization points, and others lie between them. CAGE extrapolates the optimization results to evaluate the objective at these intermediate points.



- 4 Click **OK** to apply the application point set to the objective.

Set Up Constraints

- “Import Constraints from a Previous Optimization” on page 5-23
- “Apply the Peak Pressure Constraint at Full Load” on page 5-23
- “Add NOx Sum Constraints” on page 5-24

Import Constraints from a Previous Optimization

Import the AFR and PEAKPRESS constraints from the previous optimization.

- 1 Right-click the Constraints pane, and select **Import Constraints**.
- 2 Select the AFR and PEAKPRESS constraints from the BSFC_FullLoad optimization, and click **OK**.

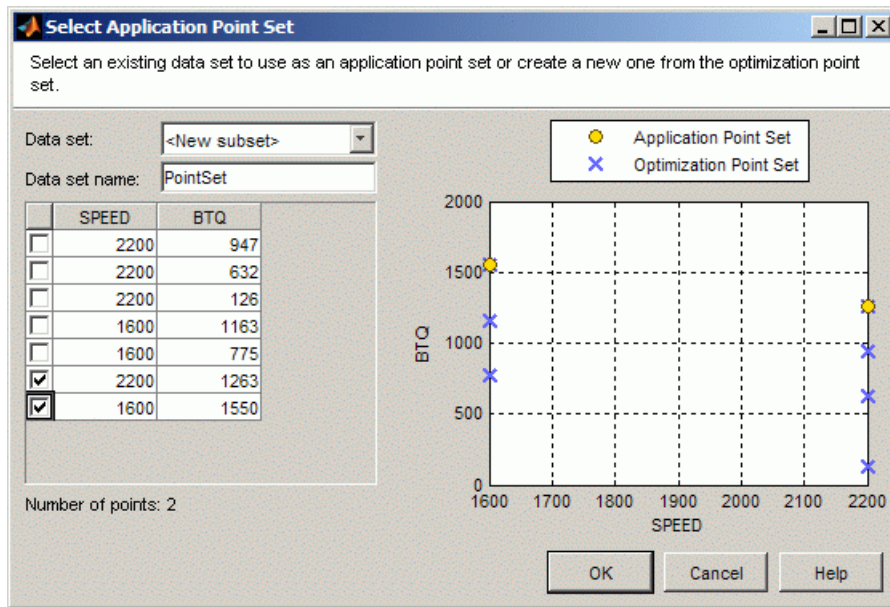
Apply the Peak Pressure Constraint at Full Load

To apply the peak pressure constraint only at full load, add an application point set to the PEAKPRESS constraint.

- 1 Right-click the constraint PEAKPRESS, and select **Select Application Point Set**

The Select Application Point Set dialog box opens.

- 2 Select **New subset** from the **Data set** list. You can use this setting to select a subset of your optimization points for an objective or constraint.
- 3 Select the check boxes for the full load points only. To select points, you can use the check boxes or click points on the plot.



- 4 View the plot displaying the application points where the constraint applies, and click **OK**.

Add NOx Sum Constraints

To add the NOx sum constraints,






- 1 Right-click the Constraints pane, and select **Add Constraint**.
The Edit Constraint dialog box opens.
- 2 Select the **Sum Constraint Constraint type**.
- 3 Select the **BSNOX** model.
- 4 Enter 0.003 for the **Constraint bound** and press **Enter**. CAGE closes the dialog box.
- 5 Ensure that the constraint description shows the weighted sum ≤ 0.003 .
- 6 Right-click the BSNOX constraint, and select **Duplicate**.
The Edit Constraint dialog box opens.
- 7 Rename the constraint to **BSNOX_DriveCycle10**, and click **OK**.

- To evaluate the constraint at different operating points than the points specified in the optimization, right-click the constraint `BSNOX_DriveCycle10`, and select **Select Application Point Set**

The Select Application Point Set dialog box opens.

- Select the `DriveCycle10` data set. View the plot, and click **OK**.

Your sum optimization should contain the following constraints.

Constraints		
Name	Description	Application Point Set
 BSFC_Boundary	Boundary constraint of BSFC(M...	
 AFR	AFR_LowerBound <= AFR(MAI...	
 PEAKPRESS	PEAKPRESS(MAINSOI, FUELPR...	FullLoad(SPEED,BTQ)
 BSNOX	Weighted sum of BSNOX(MAIN...	
 BSNOX_DriveCycle10	Weighted sum of BSNOX(MAIN...	DriveCycle10(SPEED,BTQ;BSNOX_DriveCycle10_weights)

Define Weights and Fixed Variables and Run Optimization

Next, review the Fixed Variables pane. To define new weights and fixed variables, change the default values for `BSFC_weights`, `AFR_LowerBound`, `MaxPressure`, `BSNOX_weights` and `BSNOX_DriveCycle10_weights` as described in the following steps:

- Increase the **Number of values** to 7 for `AFR_LowerBound`, `MaxPressure`, `BSNOX_weights` and `BSNOX_DriveCycle10_weights`.
- Copy and paste the following values into the fixed variables columns.

BSFC_weights	SPEED	BTQ	AFR_LowerBound	Max Pressure	BSNOX_weights	BSNOX_DriveCycle10_weights
0.15	2200.0	947.0	27.5	18	0.15	0.15
0.15	2200.0	632.0	30.0	18	0.15	0.15
0.15	2200.0	126.0	0.0	18	0.15	0.15
0.1	1600.0	1163.0	20.0	18	0.1	0.1
0.1	1600.0	775.0	22.5	18	0.1	0.1

BSFC_weights	SPEED	BTQ	AFR_LowerBound	Max Pressure	BSNOX_weights	BSNOX_DriveCycle10_weights
0.15	2200.0	1263.0	25.5	15	0.15	0.15
0.1	1600.0	1550.0	20.0	18	0.1	0.1

Note: Weights in application point sets override weights in the Fixed Variables pane. If an objective or constraint has an application point set, CAGE applies the weights in the application point set if the name matches, for example “BSNOX_weights”.

3 Select **Optimization > Run**.

Inspect the optimization results before proceeding to the next task, “Use Optimization Results.”

Use Optimization Results

In this section...

“Introduction” on page 5-27

“Create Tables to Fill” on page 5-27

“Fill Tables from Optimization Results” on page 5-28

“Export Tables” on page 5-29

Introduction

This section takes you through the following steps:

- 1 Create tables to fill with your optimization results using the Create Tables from Model wizard.
- 2 Fill the tables with your optimization results.
- 3 Export the tables to a calibration tool.

Create Tables to Fill

You can use the Create Tables from Model wizard to create a set of tables with the same axes for all the inputs of a model, and the model response. You can add tables for any other responses that share the same inputs. You can choose which of these tables to create and select the values for the axes (or normalizers) that all tables share. You can also add all the new tables to a tradeoff.

- 1 Select **Tools > Create Tables From Model** (or use the toolbar button).

The **Create Tables From Model** Wizard opens.

- 2 Select the BSFC model to base the new tables on.

If you are viewing an optimization or an optimization output node, then the wizard automatically selects the model in the first objective. You can use this feature to create tables for the selected optimization.

Click **Next**.

- 3 As you have point-by-point models, CAGE automatically selects the check box to **Use model operating points** for the table normalizers. To edit the normalizers, clear the check box.

- 4 Enter 11 for the **Table rows** and **Table columns**, and click **Next**.
- 5 Select check boxes to specify which variables and responses to create tables for. CAGE automatically selects your free variables and the objective response (BSFC). You can add tables for other responses with the same the same inputs as the primary model (and the same operating points for point-by-point models). Select the check boxes to add AFR, BSNOX and MAINFUEL.
- 6 By default you also create a tradeoff containing all the new tables. The tradeoff can be useful for specifying fill factors for tables and for investigating optimization results. Leave the check box selected to **Create a tradeoff containing all the tables**.
- 7 Click **Finish** to create the tables and tradeoff.

Fill Tables from Optimization Results

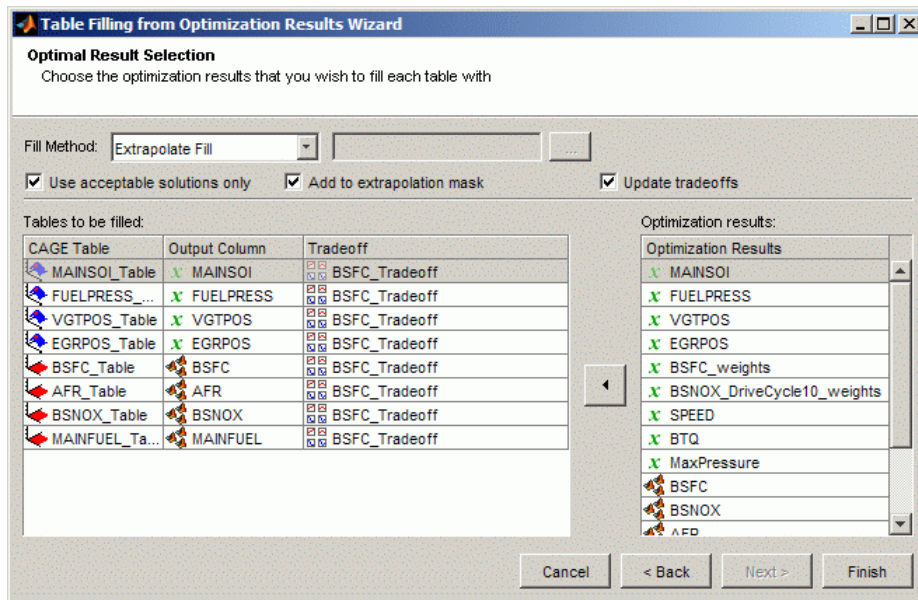
To fill the tables from your optimization results:

- 1 Select the output node of your multiregion sum optimization, BSFC_Optimization_Output.
- 2 Select **Solution > Fill Tables**, or click the toolbar button.

The Table Filling wizard opens.

- 3 **Shift**+click to select all the tables in the list, and click the button to move them to list of tables to fill. Click **Next**.

4



Check that all tables match to the correct filling item, and clear the check box to **Update tradeoffs**.

- 5 Click **Finish** to fill the tables.
- 6 View your optimization results in the Tables view.

Export Tables

To export your calibration data to file or to a calibration tool,

- 1 Select **File > Export > Calibration > Selected Item** or **All Items**.

The Export Calibration Data dialog opens.

- 2 Select the check boxes of the calibration items you want to export.

All tables and normalizers in the project are in the list of calibration items.

- 3 Select the format you want to export to:

- ATI Vision
- ATI Vision MAT file

- INCA DCM file
- Simple CSV file
- Simple MAT file
- Simple MATLAB file

Click **OK** to export your selected items.

- 4 If you selected **ATI Vision**, the ATI Vision Connection Manager dialog opens.

If you select a file format, a file browser appears. Choose a location and filename, and click **Save**.

Composite Models and Modal Optimizations

- “Introducing Composite Models and Modal Optimization” on page 6-2
- “Composite Model and Modal Optimization Projects” on page 6-3

Introducing Composite Models and Modal Optimization

You can use composite models in CAGE to produce optimal calibrations for engines with multiple operating modes.

See the next section for example projects you can use to learn about modal optimizations, “Composite Model and Modal Optimization Projects” on page 6-3.

For more information, see:

- “Creating and Viewing Composite Models in CAGE”
- “Set Up Modal Optimizations”
- “Analyzing Modal Optimization Results”

Composite Model and Modal Optimization Projects

In this section...

“Gasoline Example with Four Cylinder and Eight Cylinder Modes” on page 6-3

“Diesel Example with Low and High EGR Modes” on page 6-3

“Composite Model Example with Separate Tables for Each Mode” on page 6-4

Gasoline Example with Four Cylinder and Eight Cylinder Modes

You can load these example projects found in `matlab\toolbox\mbc\mbctraining`, to view completed examples of composite models, optimizations and filled tables:

- `GasolineComposite.mat`
- `GasolineComposite.cag`

The `GasolineComposite.cag` example shows optimizations created from composite models imported from `GasolineComposite.mat`. The composite models combine responses for two engine operating modes: 4-cylinder mode and 8-cylinder mode. In this example, the modal optimization is a point optimization that selects the best mode for maximizing torque at each operating point.

In this example, the sum optimization has been created from the results of the point optimizations, using the selected best mode at each point. Table gradient constraints are added for ICP and ECP to ensure smooth control and engine response.

For instructions showing examples from this project describing how to create and use composite models and modal optimizations, see:

- “Creating and Viewing Composite Models in CAGE”
- “Creating Modal Optimizations”
- “Viewing and Selecting Modal Optimization Results”
- “Creating Sum Optimizations from Modal Optimizations”
- “Filling Tables for Operating Modes”

Diesel Example with Low and High EGR Modes

You can load these example projects found in `matlab\toolbox\mbc\mbctraining`, to view completed examples of composite models, optimizations and filled tables:

- DieselComposite.mat
- DieselComposite.cag

This diesel example shows the interaction of composite models and point-by-point models. The DieselComposite.cag example shows optimizations created from a composite model imported from DieselComposite.mat. The composite model combines point-by-point response models for two engine operating modes: low and high exhaust gas recirculation (EGR).

Operating points are repeated in different modes. When using modal optimization to choose the best mode at an operating point, you need a single run for each operating point, with the mode as a free variable. Use the Create Optimization from Model wizard to remove repeated operating points as follows:

- 1 To specify an optimization run for each unique operating point, use the **Data Source** option **Unique operating points** in the Create Optimization from Model wizard. This option displays only for composite models created from point-by-point models.
- 2 The wizard automatically selects the mode variable as a free variable when you use the Create Optimization from Model wizard to create a modal optimization from a composite model.

In this example, both modes use the same operating points except for one point. Select **Model > Properties** to view which modes are available for each operating point. See “Creating and Viewing Composite Models in CAGE”.

If you want to run a separate optimization for each mode, use the **Model operating points** option in the Create Optimization from Model wizard. Then, delete the runs for the modes you do not want.

Composite Model Example with Separate Tables for Each Mode

To see an example project where the strategy has separate tables for each mode, load the example project CompositeWith2Tables.cag found in matlab\toolbox\mbc\mbctraining.

In this example project there is a single table for each control variable which stores the value for the best mode. Composite calibration problems of this kind often involve separate optimizations (point and sum) with different free variables and constraints for each mode.

- 1** Each mode has a separate point optimization.
- 2** The results from each mode have been exported to the same data set, using the append option.
- 3** To finish off the calibration the sum optimization provides results for a multimodal drive cycle, using the selected mode at each point. The sum optimization was created as follows:
 - Select the combined point results data set for the optimization points.
 - Import point constraints from the point optimizations.
 - Add table gradient constraints for ICP and ECP. Separate table gradient constraints are required for different modes because the goal is to fill separate tables for each mode.

Add an application point set to restrict table gradient constraints to mode 1 points only. Use the Interpolate by mode option.

Duplicate the table gradient constraints and create an application point set to restrict them to mode 2 points only.
- 4** This example uses table-filling filter rules to fill tables using only the appropriate mode. For details see “Filling Tables for Operating Modes”.

Design and Modeling Scripts

- “Introduction to the Command-Line Interface” on page 7-2
- “Automate Design and Modeling With Scripts” on page 7-3
- “Understanding Model Structure for Scripting” on page 7-6
- “How the Model Tree Relates to Command-Line Objects” on page 7-10

Introduction to the Command-Line Interface

The Model-Based Calibration Toolbox product is a software tool for modelling and calibrating powertrain systems. The command-line interface to the Model-Based Calibration Toolbox product enables the design of experiments and modeling tools available in the toolbox to be accessible from the test bed.

You can use these commands to assemble your specific engine calibration processes into an easy to use script or graphical interface. Calibration technicians and engineers can use the custom interface without the need for extensive training. This system enables:

- Transfer of knowledge from the research and development engineers into the production environment
- Faster calibration
- Improved calibration quality
- Improved system understanding
- Reduced development time

See Model-Based Calibration Toolbox Demos for command-line examples.

Automate Design and Modeling With Scripts

In this section...

“Processes You Can Automate” on page 7-3

“Engine Modeling Scripts” on page 7-5

Processes You Can Automate

The following description illustrates an example engine modeling process you can automate with the command-line Model-Based Calibration Toolbox product. You can assemble the commands for these steps into an easy-to-use script or graphical interface. This is a guideline for some of the steps you can use to model engine data.

- 1 Create or load a project — CreateProject; Load
- 2 Create a new test plan for the project using a template set up in the Model Browser — CreateTestplan
- 3 Create designs that define data points to collect on the test bed — CreateDesign. Work with classical, space-filling or optimal designs: CreateConstraint; CreateCandidateSet; Generate; FixPoints; Augment.
- 4 Create or load a data object for the project and make it editable — CreateData; BeginEdit
- 5 Load data from a file or the workspace — ImportFromFile; ImportFromMBCDataStructure

You can instead specify the required data file when you call CreateData; you must still call BeginEdit before you can then make changes to the data.

- 6 Work with the data:
 - Examine data values — Value
 - Modify the data to remove unwanted records — AddFilter; AddTestFilter
 - Add user-defined variables to the data — AddVariable
 - Add new data — Append
 - Group your data for hierarchical modeling by applying rules — DefineTestGroups; DefineNumberOfRecordsPerTest
 - Export your data to the workspace — ExportToMBCDataStructure

- 7 Save your changes to the data, or discard them — `CommitEdit`; `RollbackEdit`
- 8 Designate which project data object to use for modeling in your test plan — `AttachData`
- 9 Create and evaluate boundary models, either in a project or standalone. You can use boundary models as design constraints. See “Boundary Model Scripting” on page 7-8.
- 10 Create models for the data; these can be one- or two-stage models and can include datum models — `CreateResponse`
- 11 Work with your models:
 - Examine input data and response data — `DoubleInputData`; `DoubleResponseData`
 - Examine predicted values at specified inputs — `PredictedValue`; `PredictedValueForTest`
 - Examine Predicted Error Variance (PEV) at specified inputs — `PEV`; `PEVForTest`
 - Examine and remove outliers — `OutlierIndices`; `OutlierIndicesForTest`; `RemoveOutliers`; `RemoveOutliersForTest`; `RestoreData`
 - Create a selection of alternative models — `CreateAlternativeModels`
 - Choose the best model by using the diagnostic statistics — `AlternativeModelStatistics`; `DiagnosticStatistics`; `SummaryStatistics`
 - Extract a model object from any response object (Model Object), then:
 - Fit to new data (`Fit`)
 - Create a copy of the model, change model type, properties and fit algorithm settings (`CreateModel`; `ModelSetup`; `Type` (for models); `Properties` (for models); `CreateAlgorithm`)
 - Include and exclude terms to improve the model (`StepwiseRegression`)
 - Examine regression matrices and coefficient values (`Jacobian`; `ParameterStatistics`)
 - If you change the model you need to use `UpdateResponse` to replace the new model back into the response in the project.
 - For two-stage test plans, once you are satisfied with the fit of the local and response feature models (and have selected best models from any alternatives you created), you can calculate the two-stage model — `MakeHierarchicalResponse`.
 - Now you can also examine the predicted values and PEV of the two-stage model.

- You can export any of these models to MATLAB or Simulink software — Export

This overview is not an exhaustive list of the commands available. For more information on all available functions, see “Automation Scripting”.

Engine Modeling Scripts

See Model-Based Calibration Toolbox Examples for a selection of command-line script examples. Run the scripts to learn about:

- Loading and Modifying Data
- Designing experiments and constraining designs
- Gasoline engine modeling script to automatically generate a project for the gasoline case study, including:
 - Grouping and filtering data
 - Boundary modeling
 - Response modeling
 - Removing outliers and copying outlier selections
 - Creating alternative models and selecting the best based on statistical results
- Point-by-point diesel engine modeling to automatically generate a project for the diesel case study, including:
 - Defining engine operating points
 - Creating designs for each operating point
 - Augmenting designs to collect more data
 - Building a point-by-point boundary model
 - Create response models using the local multiple model type

Understanding Model Structure for Scripting

In this section...

“Projects and Test Plans for Model Scripting” on page 7-6

“Response Model Scripting” on page 7-6

“Boundary Model Scripting” on page 7-8

Projects and Test Plans for Model Scripting

To use the Model Browser in the Model-Based Calibration Toolbox product, you must understand the structure and functions of the model tree to navigate the views. To use the command-line version of the toolbox, you must understand the same structure and functions, that is, how projects, test plans, and models fit together. The following sections describe the relationship between the different models that you can construct. The diagrams in the following section, “How the Model Tree Relates to Command-Line Objects” on page 7-10, illustrate these relationships.

- Projects can have one or more test plans.
- Projects can have one or more data objects.
- Test plans have no more than one data object.
- Test plans have response objects.
 - If a one-stage test plan, these are simply known as responses.
 - If two-stage test plan, these are hierarchical responses.
- Test plans have boundary tree objects.

Response Model Scripting

A response is a model fitted to some data. These are the types of responses:

- Hierarchical Response (Level 0)

A hierarchical response (also known as a two-stage response) models a `ResponseSignalName` using a local response and one or more response features.

A hierarchical response has one or more different local responses (accessible via the property `LocalResponses`) that provide different possible models of the

ResponseSignalName. One of these must be chosen as the best, and that will then be the local response used subsequently. The response features of each of the local responses are available directly from those local response objects.

- Local Response (Level 1)

The local response consists of models of the **ResponseSignalName** as a function of the local input factors. The local input factors are accessible via the **InputSignalNames** property.

A local response has one or more response features (accessible via the property **ResponseFeatures**) containing the models fitted to those response features of the local model.

- Response (Level 1 or 2)

- For two-stage test plans, response objects model the response features of local responses (**ResponseSignalName** corresponds to the name of the response feature). In this case, the response has a level value of 2.
- For one-stage test plans, response objects simply model the **ResponseSignalName** as a function of the input factors. In this case, the response will have a level value of 1.

All responses can have zero or more alternative responses (accessible via the property **AlternativeResponses**) that provide different possible models of the **ResponseSignalName**. These all retain the same level as the response for which they are an alternative. One of these must be chosen as the best and that will then be the response used subsequently.

See the illustrations in the following section, “How the Model Tree Relates to Command-Line Objects” on page 7-10, for examples of different responses and how they relate to each other.

Note that each response contains a model object (`mbcmodel.model`) that can be extracted and manipulated independently of the project. You can change the model type and settings, fit to new data, examine coefficients, regression matrices and predicted values, and use stepwise functions to include or remove terms. You can change model type, properties and fit algorithm settings. To learn about what you do with a model object, see Model Object. If you change the model, you must use **UpdateResponse** to replace the new model type in the response object in the project. When you use **UpdateResponse** the new model is fitted to the response data. See **UpdateResponse**.

To learn about all the available functions for constructing and working with models, see “Modeling”.

Boundary Model Scripting

You can create and evaluate boundary models either in a project or standalone. You can combine boundary models in the same way as when using the Boundary Editor GUI. You can use boundary models as design constraints.

In a project, the test plan has a `Boundary` property that can contain an `mbcboundary.Tree` object.

```
BoundaryTree = mbcmodel.testplan.Boundary
```

The `BoundaryTree` is a container for all the boundary models you create. The tree is empty until you create boundaries, and if you change the testplan data the toolbox deletes the boundaries.

You can fit boundary models in `mbcmodel` projects using the boundary tree class `mbcboundary.Tree`, or you can fit boundary models directly to data.

To create a boundary model outside of a project, you can either:

- Use the `CreateBoundary` package function:

```
B = mbcboundary.CreateBoundary(Type, Inputs)
```

- Use the `Fit` method to create and fit a boundary to some data `X`:

```
B = mbcboundary.Fit(X, Type)
```

To create a boundary model within a project, use the `CreateBoundary` method of the boundary tree:

```
B = CreateBoundary(Tree, Type)
```

This creates a new boundary model, `B`, from the `mbcboundary.Tree` object, `Tree`. The test plan inputs are used to define the boundary model inputs. The new boundary model is not added to the tree, you must call `Add`.

To create a new boundary model from an existing boundary model, you can use the `CreateBoundary` method of all boundary model types:

```
B = CreateBoundary(B, Type)
```

You can combine boundary models by using the `InBest` property of the boundary tree. This corresponds to combining boundary models in *best* in the Boundary Editor GUI, as described in “Combining Best Boundary Models” in the Model Browser documentation. You can also combine boundary models with logical operators, for use as design constraints or outside projects.

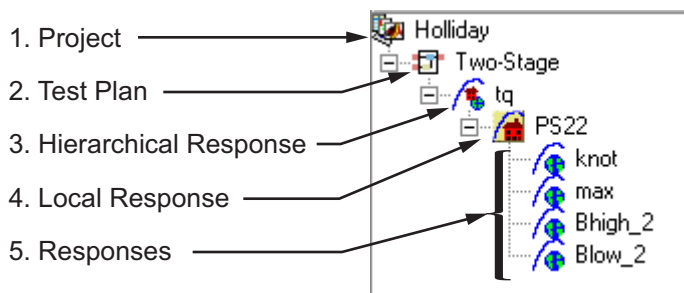
You can change the `ActiveInputs`, `Evaluate`, and use as a `designconstraint`.

For a complete list of boundary model classes, methods and properties, see “Boundary Models”.

How the Model Tree Relates to Command-Line Objects

The tree in the Model Browser displays the hierarchical structure of models. This structure must be understood to use the command-line interface. The following examples illustrate the relationship between projects, test plans and responses in one-stage and two-stage models.

The following is an example of a two-stage model tree.



The elements of the tree correspond to the following objects in the command-line interface:

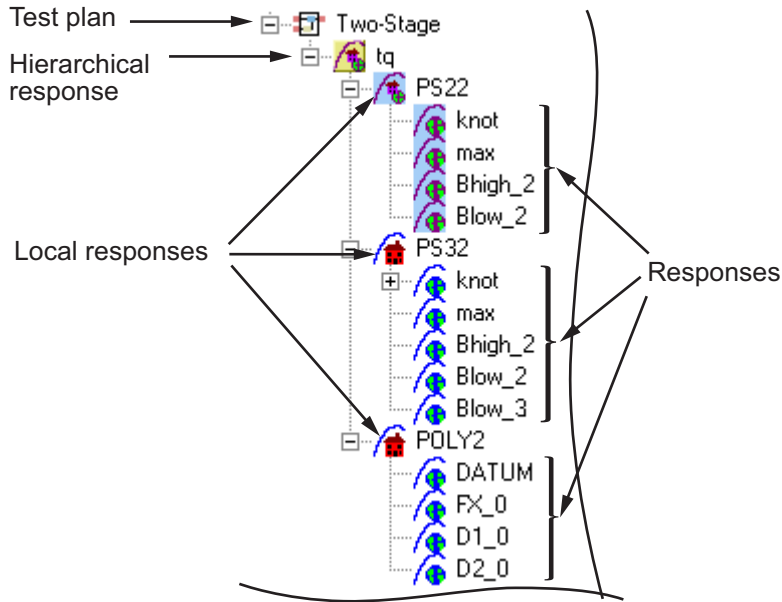
- 1 Project
- 2 Test Plan
- 3 Hierarchical Response
- 4 Local Response
- 5 Responses

The following example illustrates a project containing a one-stage test plan; in the command-line interface this corresponds to a project, one-stage test plan, and a response model.

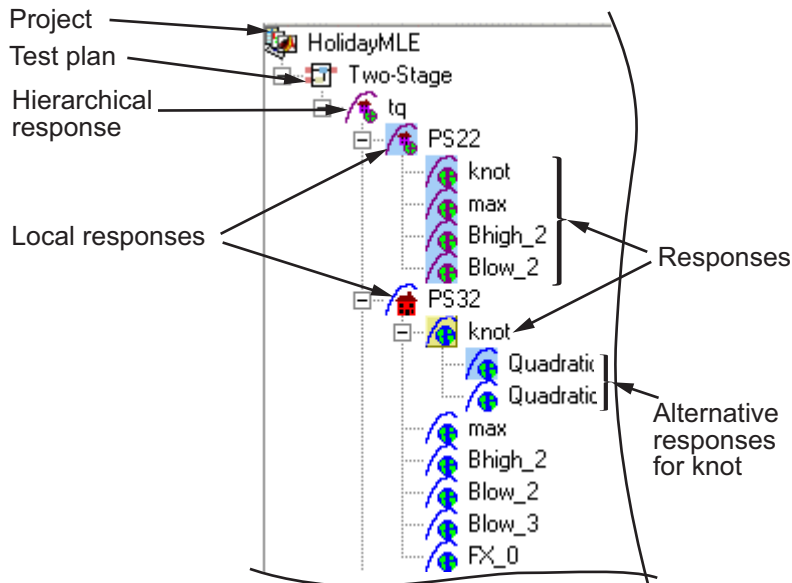


Hierarchical responses can have multiple local responses, as shown in the following example from the Model Browser. In the command-line interface these are accessible via the property `LocalResponses` for a hierarchical response object (`mbcmodel.hierarchicalresponse`). In this example, the local responses are **PS22**, **PS32**, and **POLY2**.

Only one of these local responses can be chosen as best (in this example, **PS22**, indicated by the blue icon) and used to construct the hierarchical response, together with the associated response features of the local response. Each local response object has a set of responses, accessible by the property `ResponseFeatures(Local Response)`.

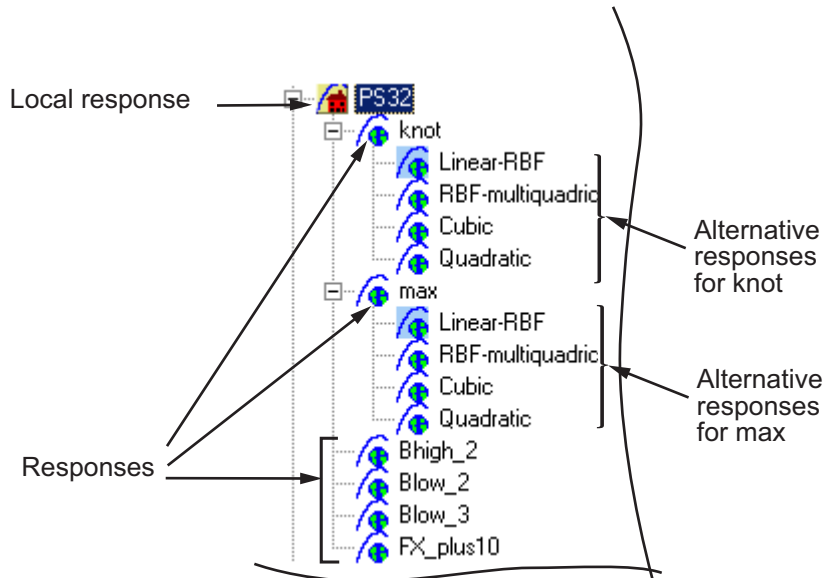


Responses can have zero or more alternative responses, as shown in the following model tree. You call the method `CreateAlternativeModels` on the command line to do the same.



In this example, the alternative responses for the `knot` response are accessible via the property `AlternativeResponses`. You can create alternative responses for any response (including all one-stage responses).

You can use model templates to try alternative model types for several responses. The following example shows the results of using a model template for four alternative responses (`Linear-RBF`, `RBF-multiquadric`, `Cubic`, and `Quadratic`). The model template has been used to create alternative responses for the responses `knot` and `max`. You can call the method `CreateAlternativeModels` to do this in the command-line interface.



One of the alternative responses must be chosen as best for each response (call the method `ChooseAsBest`). In this example, when `Linear-RBF` is chosen as best from the alternatives for the `knot` response, then it is copied to `knot`.

Multi-Injection Diesel Calibration

- “Multi-Injection Diesel Calibration Workflow” on page 8-2
- “Design of Experiment” on page 8-22
- “Statistical Modeling” on page 8-35
- “Optimization” on page 8-42

Multi-Injection Diesel Calibration Workflow

In this section...
“Multi-Injection Diesel Problem Definition” on page 8-2
“Engine Calibration Workflow” on page 8-7
“Air-System Survey Testing” on page 8-8
“Multi-Injection Testing” on page 8-9
“Data Collection and Physical Modeling” on page 8-10
“Statistical Modeling” on page 8-11
“Optimization Using Statistical Models” on page 8-12
“Case Study Example Files” on page 8-20

Multi-Injection Diesel Problem Definition

This case study shows how to systematically develop a set of optimal steady-state engine calibration tables using Model-Based Calibration Toolbox.

The engine to be calibrated is a 3.1L multi-injection combustion ignition engine with common rail, variable-geometry turbocharger (VGT), and cooled exhaust gas recirculation (EGR).

The aim of the calibration is to minimize brake-specific fuel consumption (BSFC) at specific speed/load operating points across the engine’s operating range, and meet these constraints:

- Limit total NO_x emissions.
- Limit maximum turbocharger speed.
- Limit calibration table gradients for smoothness.

The analysis must produce optimal calibration tables in speed and torque for:

- Best main start of injection timing
- Best total injected fuel mass per cylinder per cycle
- Best pilot injection timing relative to main timing
- Best pilot injection fuel mass fraction of total injection mass
- Best exhaust gas recirculation (EGR) position

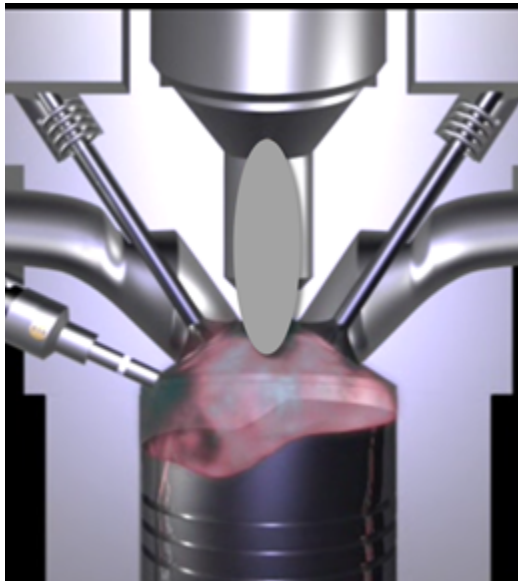
- Best variable geometry turbocharger (VGT) vane position
- Best fuel rail pressure relative to nominal pressure vs. engine speed

These sections explain the objectives of selecting best values for these calibration tables and the effects of these control variables on the engine:

- “Select Main Injection Timing for Efficiency” on page 8-3
- “Select Pilot Injection Timing to Control Noise” on page 8-4
- “Select Main Fuel Mass for Efficiency and Emissions” on page 8-5
- “Select Fuel Pressure for Efficiency and Emissions” on page 8-5
- “Select the Turbocharger Position to Control Air-Charge and EGR” on page 8-6
- “Select the EGR Valve Position to Control Air-Charge and Emissions” on page 8-7

Select Main Injection Timing for Efficiency

You select the injection timing of the main fuel injection to maximize engine efficiency. You aim to make peak cylinder pressure occur slightly after piston top center. You inject fuel just before top-center compression, as shown.



Then you can achieve peak combustion pressure just after top-center expansion.



You also need to adjust injection timing according to speed and other conditions.

- You need to advance (move earlier before piston top center) the start of injection timing with increasing speed and dilution (exhaust gas recirculation or EGR).
- You need to retard the start of injection timing with increased fresh air intake (load).

Select Pilot Injection Timing to Control Noise

You select the timing of the pilot fuel injection to start combustion early before the larger main fuel injection. The pilot fuel injection occurs well before top-center compression and before the main injection.



You can use pilot fuel injection to control combustion noise, because it affects the variability in cylinder pressure.

In this example, pilot fuel injection timing is defined as a crank-angle delta offset before the main injection, and is therefore a relative quantity.

Select Main Fuel Mass for Efficiency and Emissions

The air-fuel ratio (AFR) affects engine efficiency and emissions. A rich AFR causes high engine-out particulates and low engine-out NO_x. You control AFR by changing the main fuel mass for optimal balance between power and emissions.

The AFR of the combustion mixture is determined by the main fuel injection mass for a given amount of fresh air. The amount of air results mainly from EGR valve position, VGT position, intake throttle position, and speed.

Select Fuel Pressure for Efficiency and Emissions

You can use fuel pressure to control fuel droplet size. Reduced fuel droplet size in the combustion chamber reduces particulates, releases more energy from the fuel, and achieves more stable combustion. High fuel pressure decreases fuel droplet size to improve efficiency and emissions.

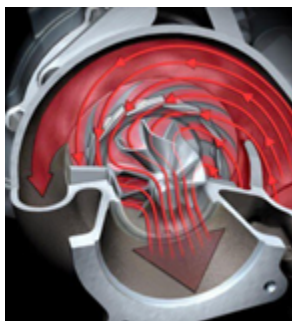
At low loads, you can use lower fuel pressure to decrease fuel pump power losses without much effect on emissions and power efficiency.

In this example, fuel pressure is controlled relative to an engine-speed-dependent base level via a fuel pressure delta, and is therefore a relative quantity.

Select the Turbocharger Position to Control Air-Charge and EGR

You can use the variable-geometry turbocharger (VGT) position to balance fresh air and exhaust gas recirculation for optimal NO_x control at a given power level.

You can change VGT vane position to increase cylinder fresh air due to the turbocharger speed increase. With the vanes closed, the turbocharger moves faster (high VGT speed) and sends a higher load (or boost) of air into the engine. Closing the vanes also increases exhaust gas recirculation (EGR) due to increased backpressure from the closed vanes.



With the vanes open, VGT speed is low and passes through low load (or boost) to the engine.



Select the EGR Valve Position to Control Air-Charge and Emissions

You can use the EGR valve position to control the flow of burned exhaust gases back to the intake manifold.

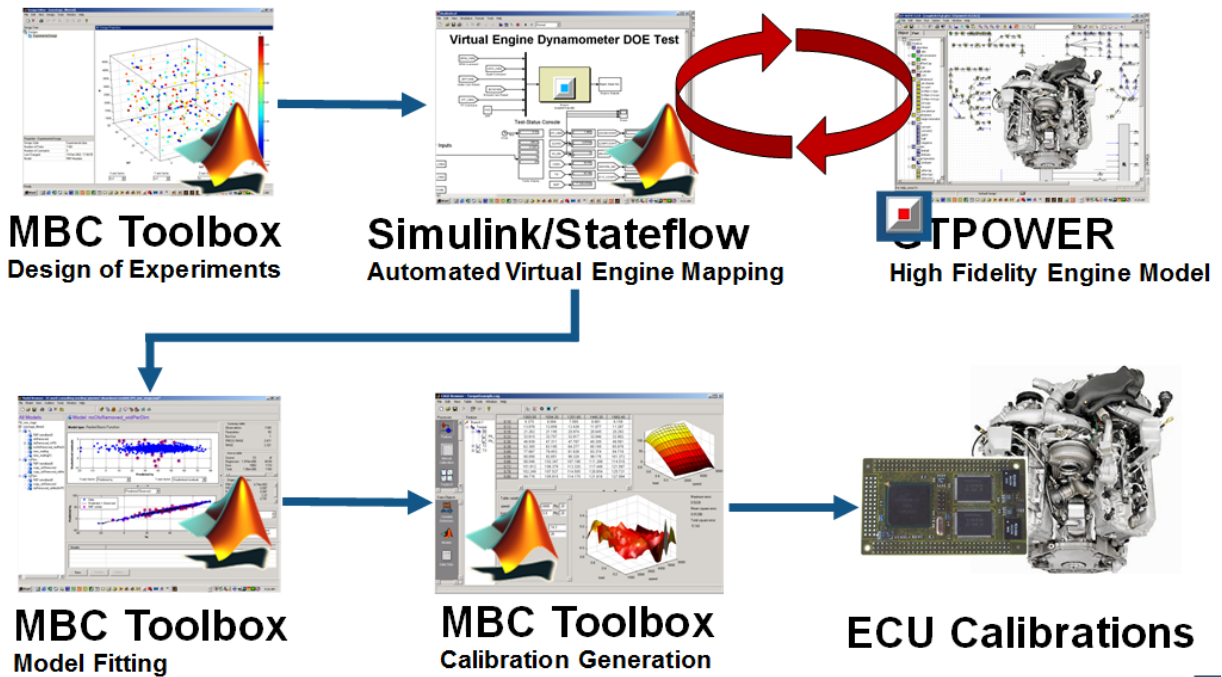
Reburning exhaust gases decreases in-cylinder temperature, resulting in a significant decrease in NOx emissions.

If you select too much EGR for a given amount of injected fuel, then the air-fuel ratio will be rich, causing increased soot emissions. Therefore, you must balance these competing objectives.

In engines, timing is everything. Using the EGR valve and all the other control variables, controlling the engine's air flow is the key to optimizing fuel economy, reducing emissions, and increasing power density.

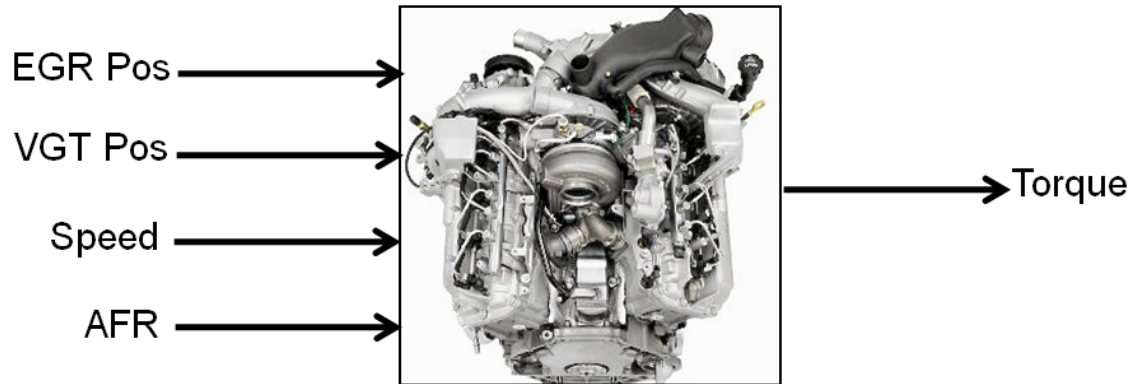
Engine Calibration Workflow

The following graphic illustrates the workflow for the model-based calibration process. The workflow can use a combination of tools: Model-Based Calibration Toolbox, Simulink, Stateflow, third-party high-fidelity simulation tools, and Hardware-in-the-Loop testing for fine tuning calibrations on ECUs.



Air-System Survey Testing

The first step to solve this calibration problem is to determine the boundaries of the feasible air-system settings. To do this, you create an experimental design and collect data to determine air-system setting boundaries that allow positive brake torque production in a feasible AFR range.



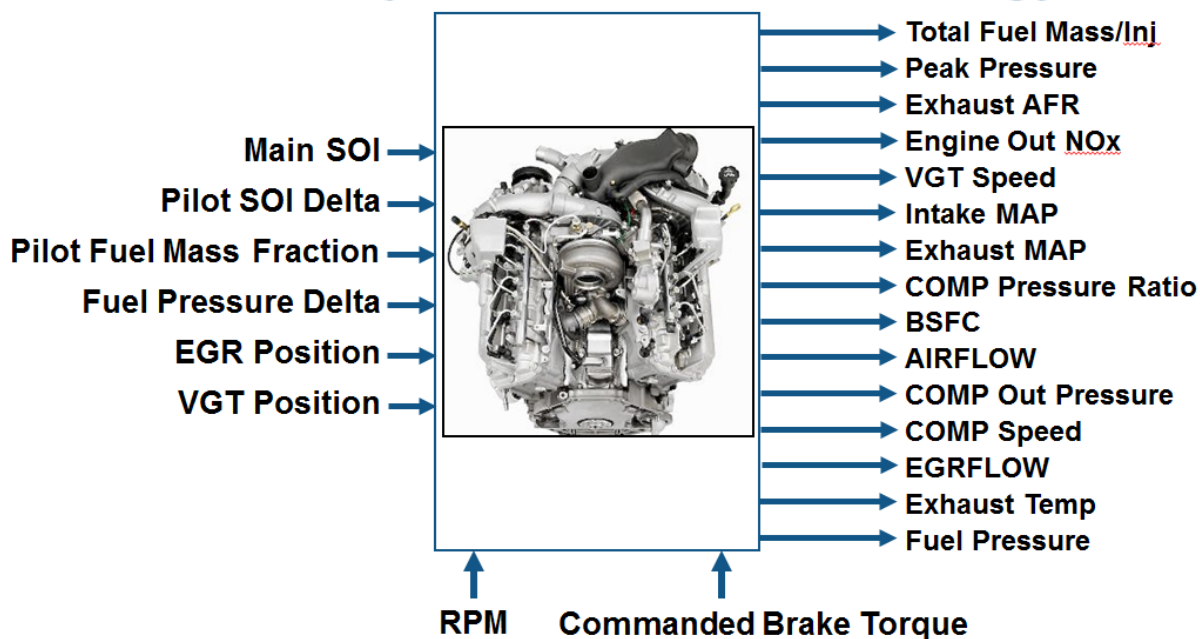
These simplifications were used to conduct the initial study:

- Pilot injection is inactive.
- Main timing is fixed.
- Nominal fuel pressure vs RPM.
- Main fuel mass is moved to match the AFR target.

Fit a boundary model to these design points.

Multi-Injection Testing

After the air-system survey, you have established the boundaries of positive brake torque air-system settings. Now, you can create an experimental design and collect data to gather fuel injection effects within those boundaries. You can then use this data to create response models for all the responses you need to create an optimal calibration for this engine.

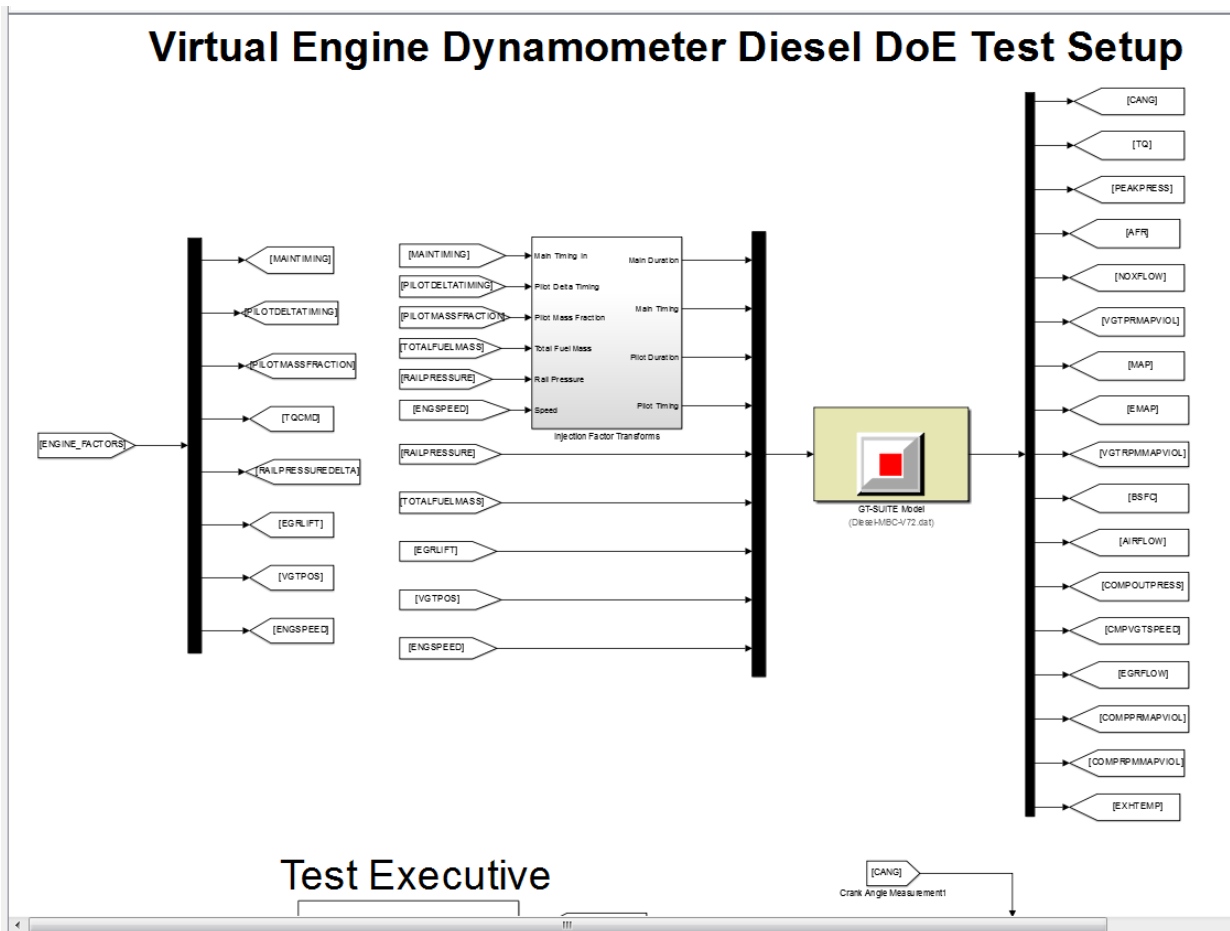


Data Collection and Physical Modeling

The toolbox provides the data for you to explore this calibration example.

MathWorks collected the data using simulation tools. Control and simulation models were constructed using Simulink and Stateflow. Constrained experimental designs were constructed using Model-Based Calibration Toolbox. The points specified in the design were measured using the GT-Power engine simulation tool from Gamma Technologies (see <http://www.gtisoft.com>).

To collect the data, Simulink and Stateflow controlled the torque output of the GT-Power engine model to the desired Design of Experiments points using total fuel mass. This graphic shows the virtual dynamometer test model.

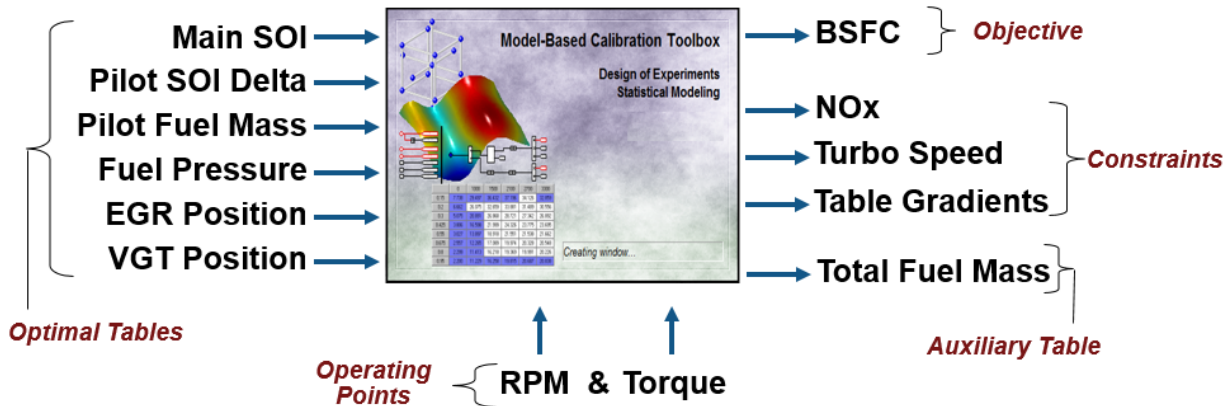


Statistical Modeling

After designing the experiments and collecting the data, you can fit statistical models to the data. You can use the toolbox to generate accurate, fast-running models from the measured engine data.

The following graphic shows the models to define in the toolbox to solve this calibration problem. The graphic shows how the model inputs and output relate to the optimal tables, optimization operating points, objectives and constraints you need to perform the optimization and create the calibration.

I/O of Multi-Inject 3.1L Common Rail Engine Model with Variable Geometry Turbocharger and Cooled EGR



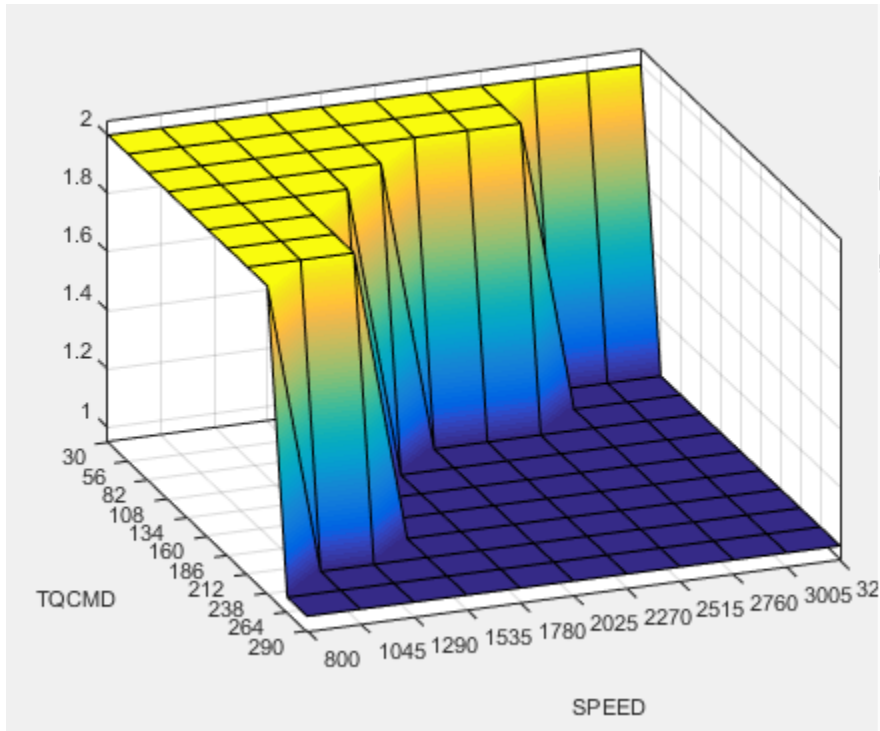
Goal: Minimize BSFC subject to NOx, turbocharger speed, and user-specified table gradient constraints

Optimization Using Statistical Models

After creating statistical models to fit the data, you can use them in optimizations. You can use the accurate statistical engine model to replace the high-fidelity simulation and run much faster, enabling optimizations to generate calibrations.

- 1 Run an optimization to choose whether to use Pilot Injection at each operating point.
- 2 Optimize fuel consumption over the drive cycle, while meeting these constraints:
 - Constrain total NOx
 - Constrain turbocharger speed
 - Constrain smoothness of tables
- 3 Fill lookup tables for all control inputs.

The following plots show a preview of the calibration results.

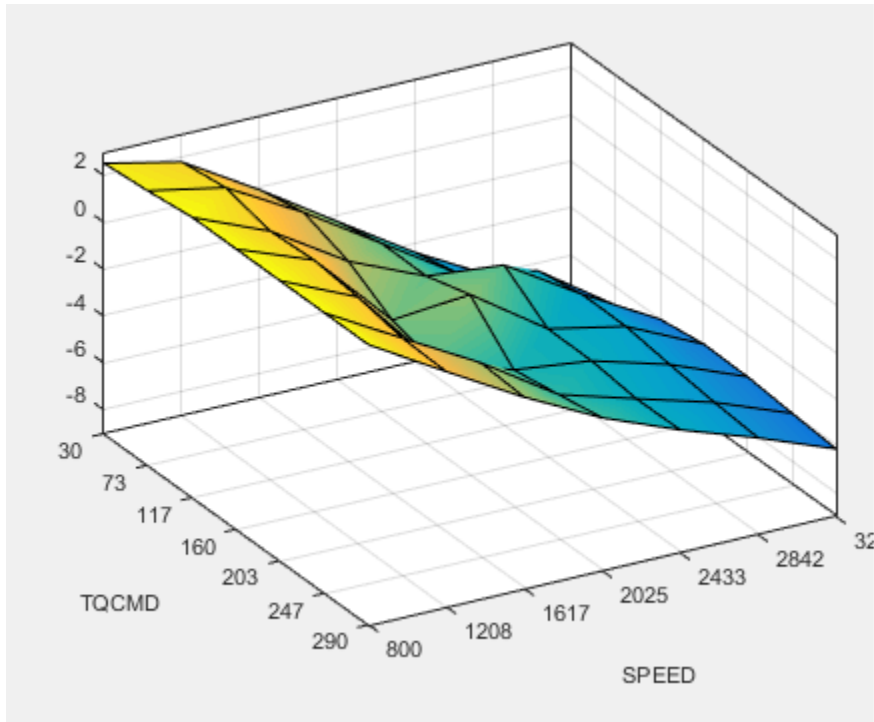


Pilot Mode Table

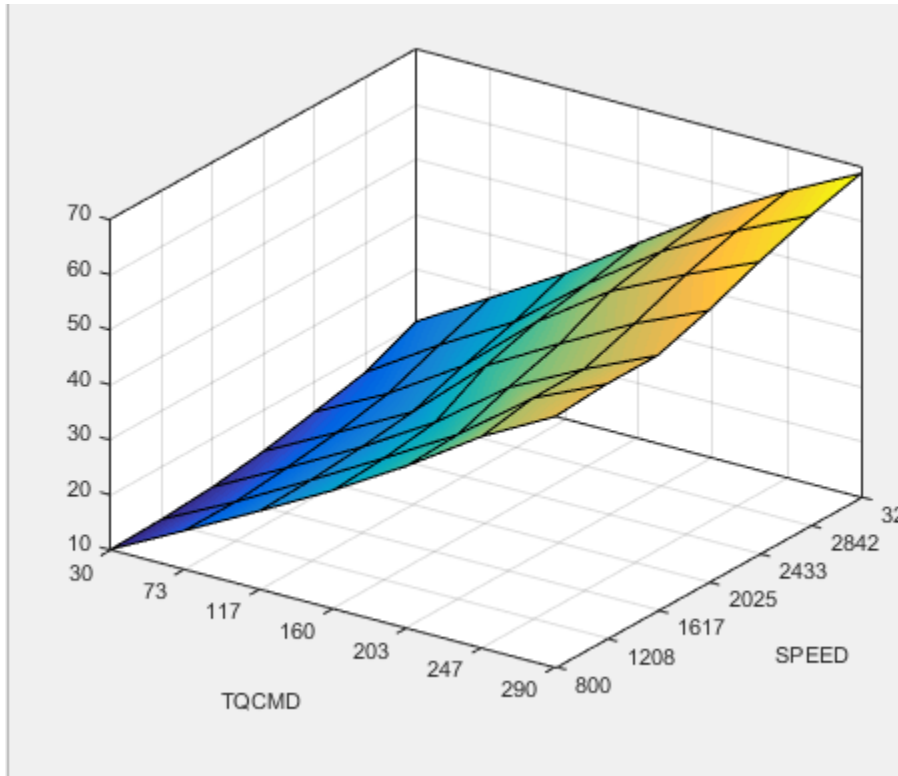
This graphic shows the plot of the table to select the active or inactive pilot mode depending on the speed and commanded torque

You need to fill calibration tables for each control variable described in “Multi-Injection Diesel Problem Definition” on page 8-2, in both pilot modes, active and inactive.

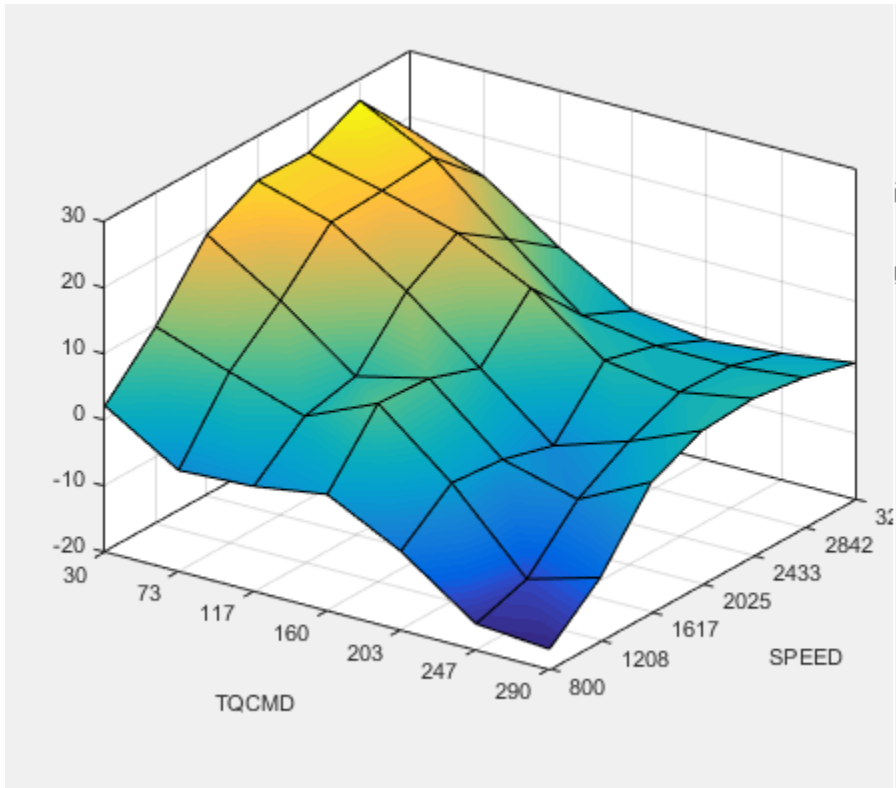
Following are all the pilot active tables.



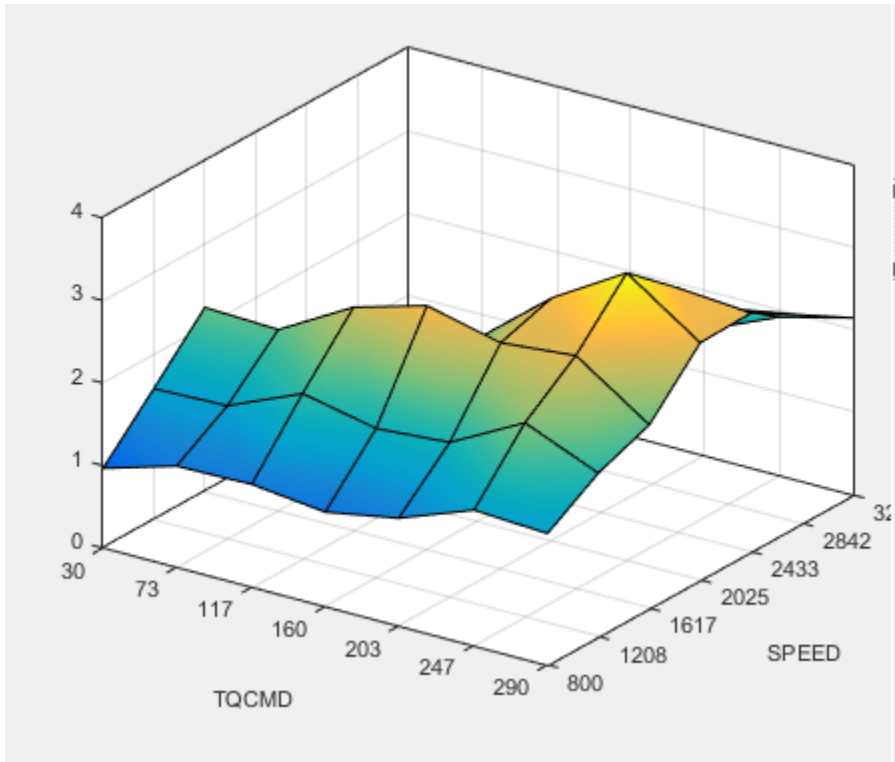
Main Start of Injection (SOI) Timing Table



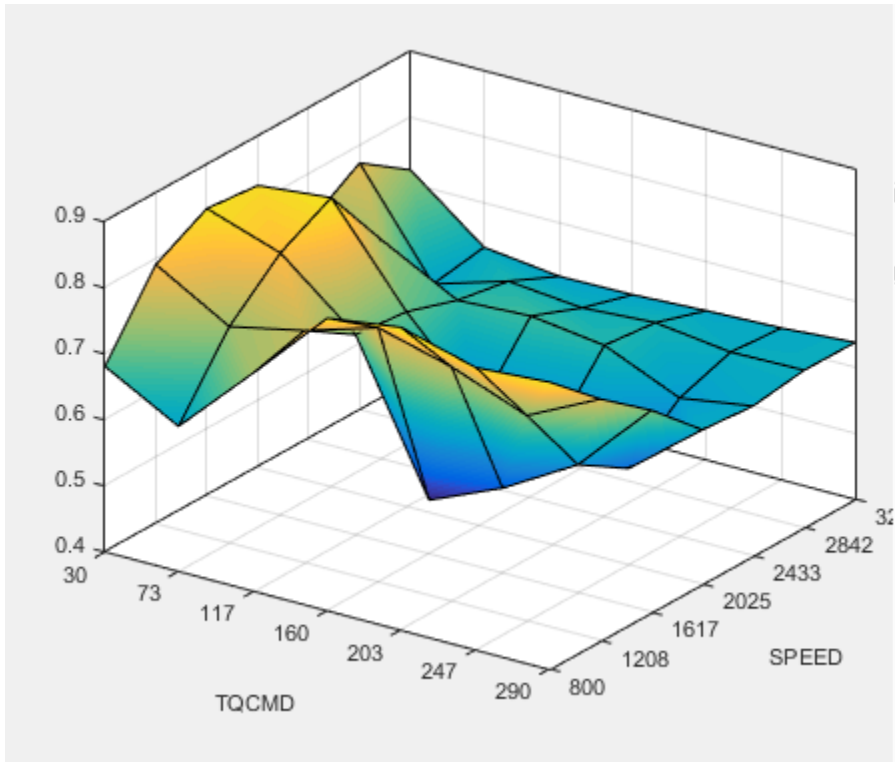
Total Injected Fuel Mass Table



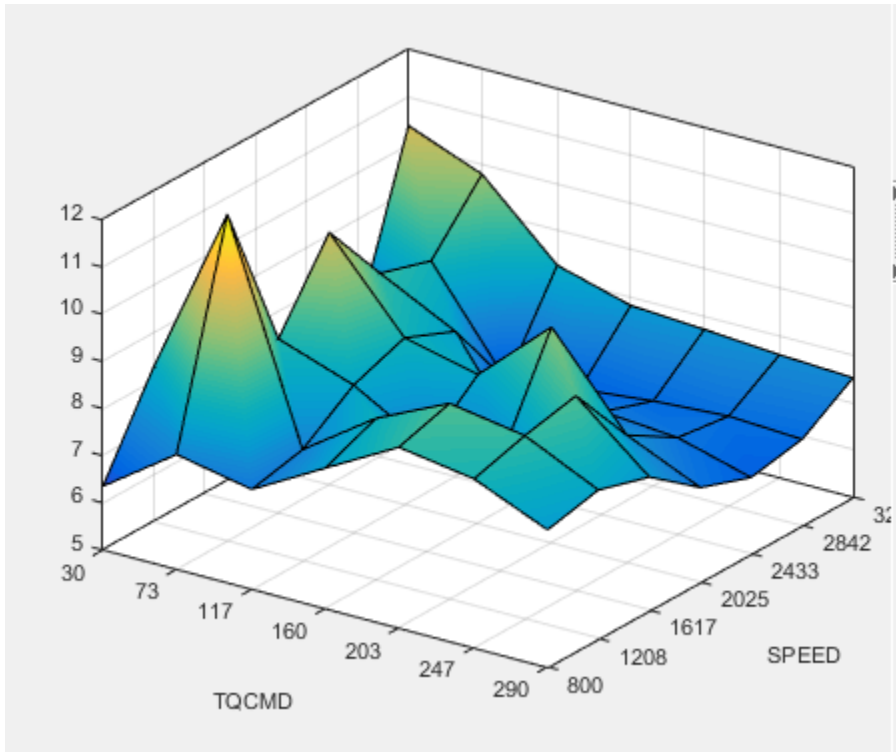
Fuel Pressure Delta Table



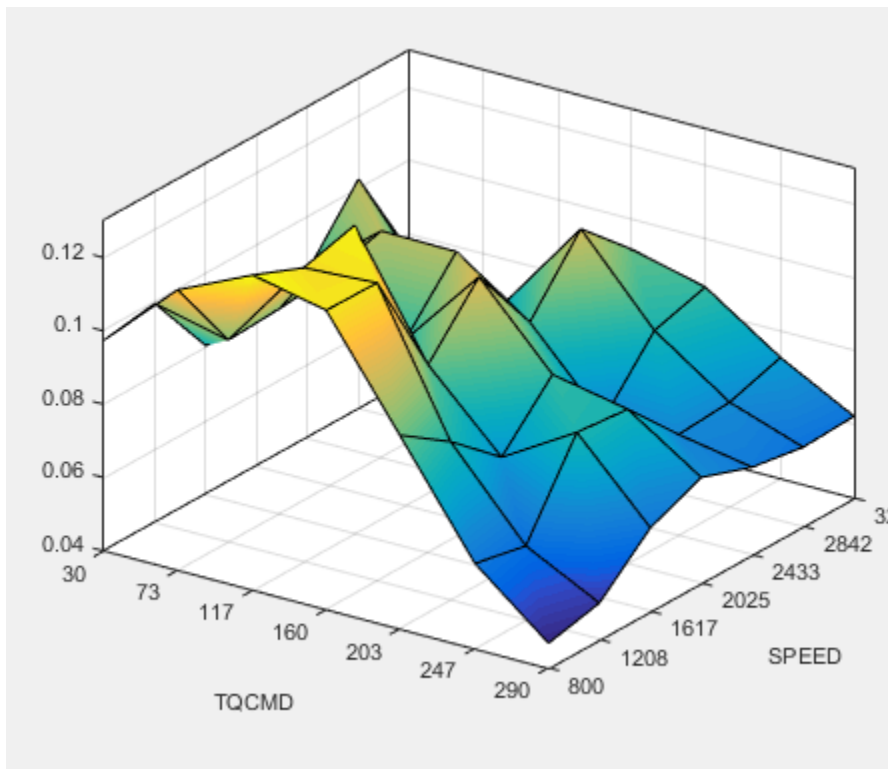
Exhaust Gas Recirculation (EGR) Valve Position Table



Variable-Geometry Turbo (VGT) Position Table



Pilot Injection Timing (Pilot SOI Delta) Table



Pilot Fuel Mass Fraction Table

Case Study Example Files

The following sections guide you through opening example files to view each stage of the model-based calibration process. You can examine:

- 1 Designs, constraints, boundary model, and collected data, in topic “Design of Experiment” on page 8-22.
- 2 Finished statistical models, in topic “Statistical Modeling” on page 8-35.
- 3 Optimization setup and results, and filled calibration tables, in topic “Optimization” on page 8-42.

Use these example files to understand how to set up systematic calibrations for similar problems. For next steps, see “Design of Experiment” on page 8-22.

If you need more help on any stage of calibration, see the tutorials in “Getting Started with Model-Based Calibration Toolbox” for step-by-step examples.

Tip: Learn how MathWorks Consulting helps customers develop engine calibrations that optimally balance engine performance, fuel economy, and emissions requirements: see [Optimal Engine Calibration](#).

Design of Experiment

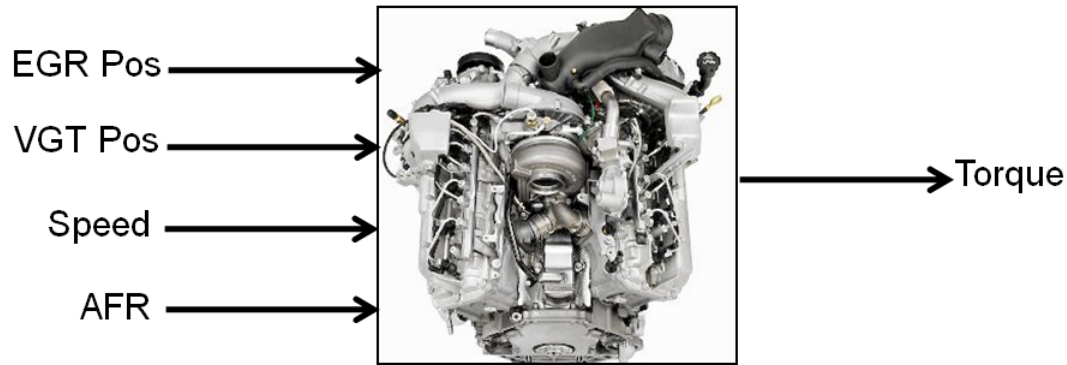
In this section...
“Benefits of Design of Experiment” on page 8-22
“Air-System Survey Testing” on page 8-22
“Create Designs and Collect Data” on page 8-23
“Fit a Boundary Model to Air Survey Data” on page 8-29
“Use the Air Survey and Boundary Model to Create the Final Design” on page 8-32
“Multi-Injection Testing” on page 8-34

Benefits of Design of Experiment

You use design of experiment to efficiently collect engine data. Testing time (on a dyno cell, or as in this case, using high-fidelity simulation) is expensive, and the savings in time and money can be considerable when a careful experimental design takes only the most useful data. Dramatically reducing test time is increasingly important as the number of controllable variables in more complex engines is growing. With increasing engine complexity, the test time increases exponentially.

Air-System Survey Testing

The first stage to solve this calibration problem is to determine the boundaries of the feasible air-system settings. To do this, create an experimental design and collect data to determine air-system setting boundaries that allow positive brake torque production in a feasible AFR range.



These simplifications were used to conduct the initial study:

- Pilot injection is inactive.
- Main timing is fixed.
- Nominal fuel pressure vs RPM.
- Main fuel mass is moved to match the AFR target.

The design process follows these steps:

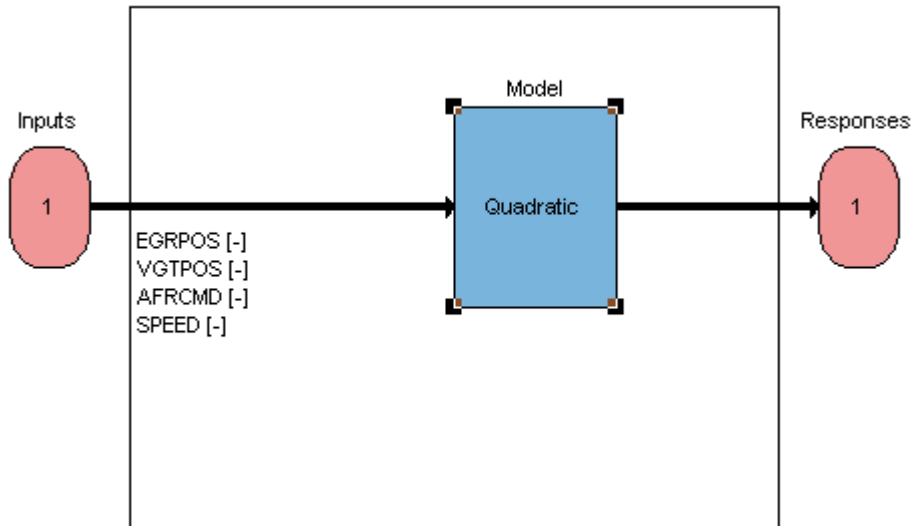
- 1 Set up variable information for the experiment, to define the ranges and names of the variables in the design space.
- 2 Choose an initial model.
- 3 Create a base design that contains the correct constraints.
- 4 Create child designs using varying numbers of points and/or methods of construction.
- 5 Choose the design to run based on the statistics and how many points you can afford to run.

Create Designs and Collect Data

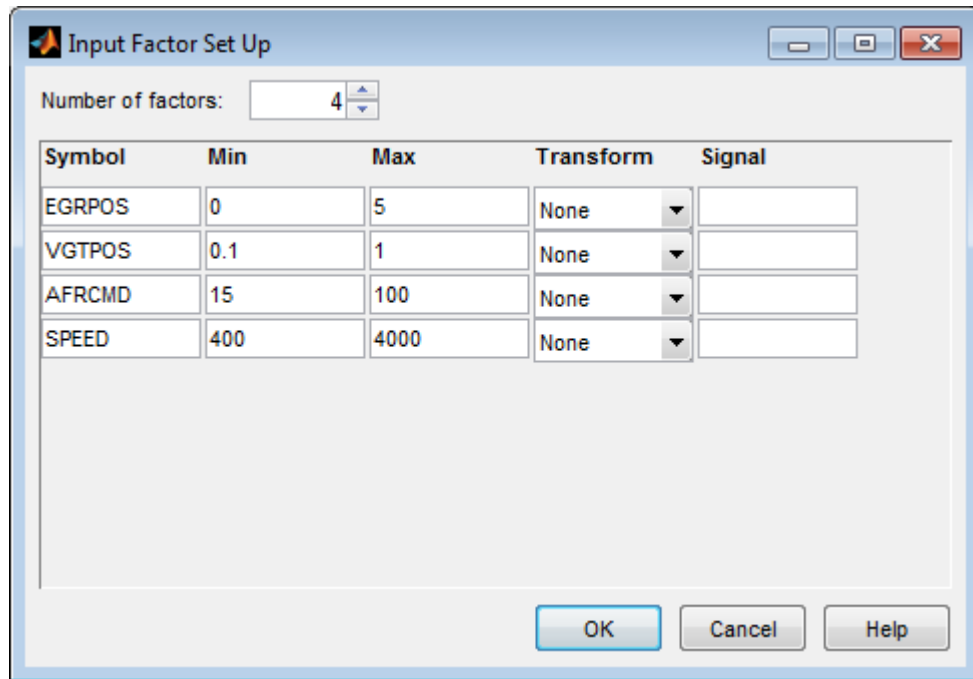
You can use a space-filling design to maximize coverage of the factors' ranges as quickly as possible, to understand the operating envelope.

To create a design, you need to first specify the model inputs. Open the example file to see how to define your test plan.

- 1 In MATLAB, on the **Apps** tab, in the **Math, Statistics and Optimization** group, click **MBC Model Fitting**.
- 2 Select **File > Open Project** and browse to the example file `CI_MultiInject_AirSurvey.mat`, found in `matlab\toolbox\mbc\mbctraining`.
- 3 The Model Browser displays the top project mode in the **All Models** tree, `CI_Multiinject_AirSurvey`.
- 4 To see how to define your test plan, click **Design Experiment**. In the new test plan dialog box, observe the inputs pane, where you can change the number of model inputs and specify the input symbols, signals and ranges. This example project already has inputs defined, so click **Cancel**.
- 5 Click the first test plan node in the **All Models** tree, `AirSurveyDoE`. The test plan view appears.



- 6 Observe the inputs listed on the test plan diagram. Double-click the **Inputs** block to view the ranges and names (symbols) for variables in the Input Factor Set Up dialog box.

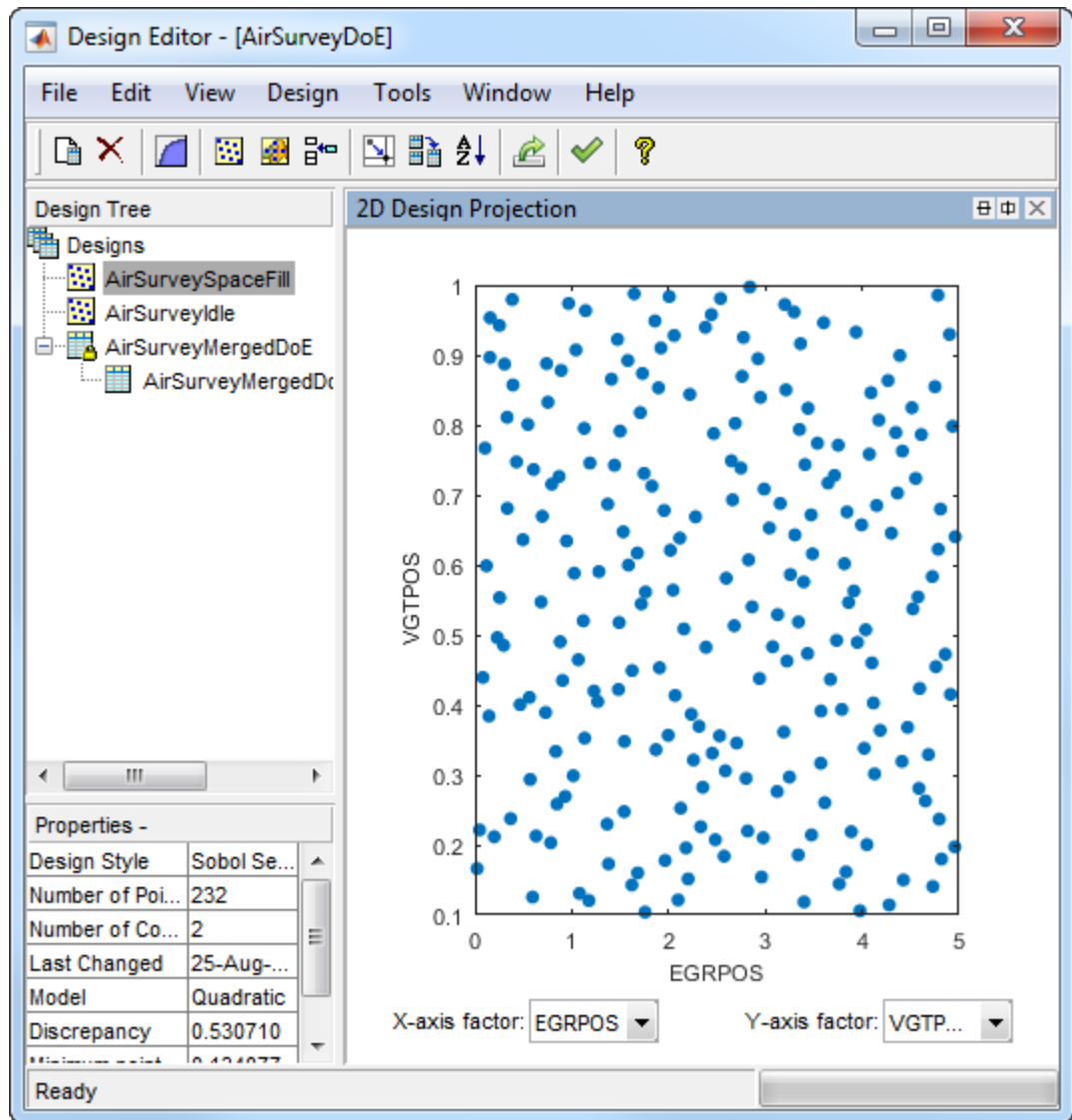


Close the dialog box.

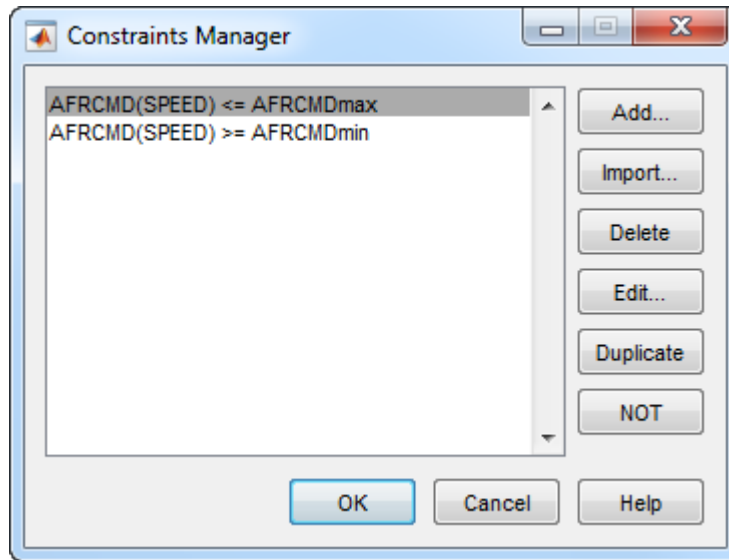
- 7 After setting up inputs, you can create designs. In the **Common Tasks** pane, click **Design experiment**.

The Design Editor opens. Here, you can see how these designs are built:

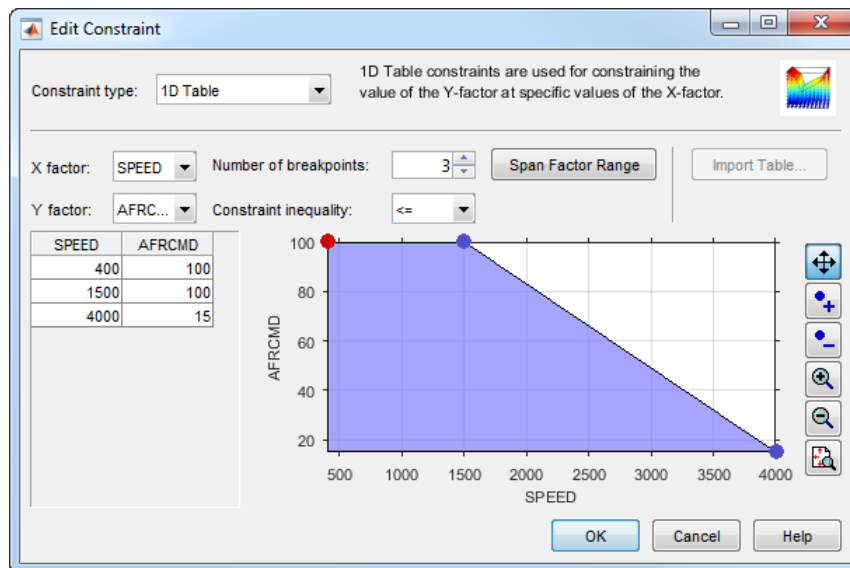
- The final Air Survey design is a ~280 point Sobol Sequence design to span the engine operating space. The Sobol Sequence algorithm generated the space-filling design points. The final design is called **AirSurveyMergedDoE** because it contains two merged designs, one with a constraint and one unconstrained:
 - A 232-point design for overall engine operating range called **AirSurveySpaceFill**
 - A 50-point design low speed/load for an idle region called **AirSurveyIdle**
- 8 Select the **AirSurveySpaceFill** design in the Designs tree, and select **View > Current View > 2D Design Projection**.



- 9 Select **Edit > Constraints** to see how the constraints are set up.

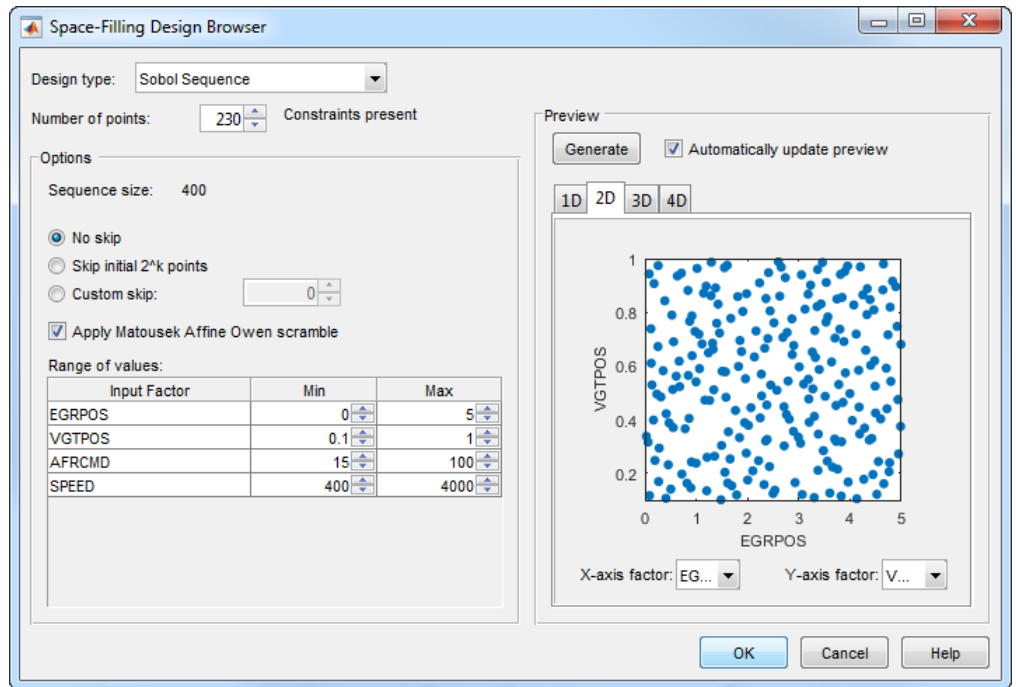


- 10 In the Constraints Manager dialog box, select a constraint and click **Edit**. Observe that you can define areas to exclude by dragging points or typing in the edit boxes.



Click **Cancel** to close the Edit Constraint dialog box and leave the constraint unchanged. Close the Constraints Manager dialog box.

- 11 Observe the **Properties** of the selected **AirSurveySpaceFill** design under the tree, listing 2 constraints, 232 points, and a **Design Style** of **Sobol Sequence**.
- 12 You can see how to construct a design like this by selecting **Design > Space Filling > Design Browser**, or click the space-filling design toolbar button.
 - a • A dialog box asks what you want to do with the existing design points. Click **OK** to replace the points with a new design.
 - In the Space-Filling Design Browser, click **Generate** several times to try different space-filling points. You can specify the **Number of points**, and view previews of design points. Observe the default design type is **Sobol Sequence**.



- Click **Cancel** to close the Space-Filling Design Browser and leave the design unchanged.
- 13 Click the `AirSurveyIdle` design to observe it is also a **Sobol** Sequence design, containing 50 points and no constraints.
 - 14 Click the `AirSurveyMergedDoE` design to observe it is of type **Custom**, and contains 282 points and 2 constraints. This design is formed by merging the previous 2 designs. You can find **Merge Designs** in the **File** menu.
 - 15 Click `AirSurveyMergedDoE_RoundedSorted`, the child design of `AirSurveyMergedDoE`. This design contains the same points but rounded and sorted, using **Edit > Sort Points** and **Edit > Round Factor**. This is the final design used to collect data.
 - 16 Close the Design Editor.

The final air survey design was used to collect data from the GT-Power model with the Simulink and Simscape test harness. The example Model Browser project file `CI_MultiInject_AirSurvey.mat` contains this data, imported to the Model Browser after the air survey. You can also view the data in spreadsheet form in the file `CI_AirSurvey_Data.xlsx` in the `mbctraining` folder.

Fit a Boundary Model to Air Survey Data

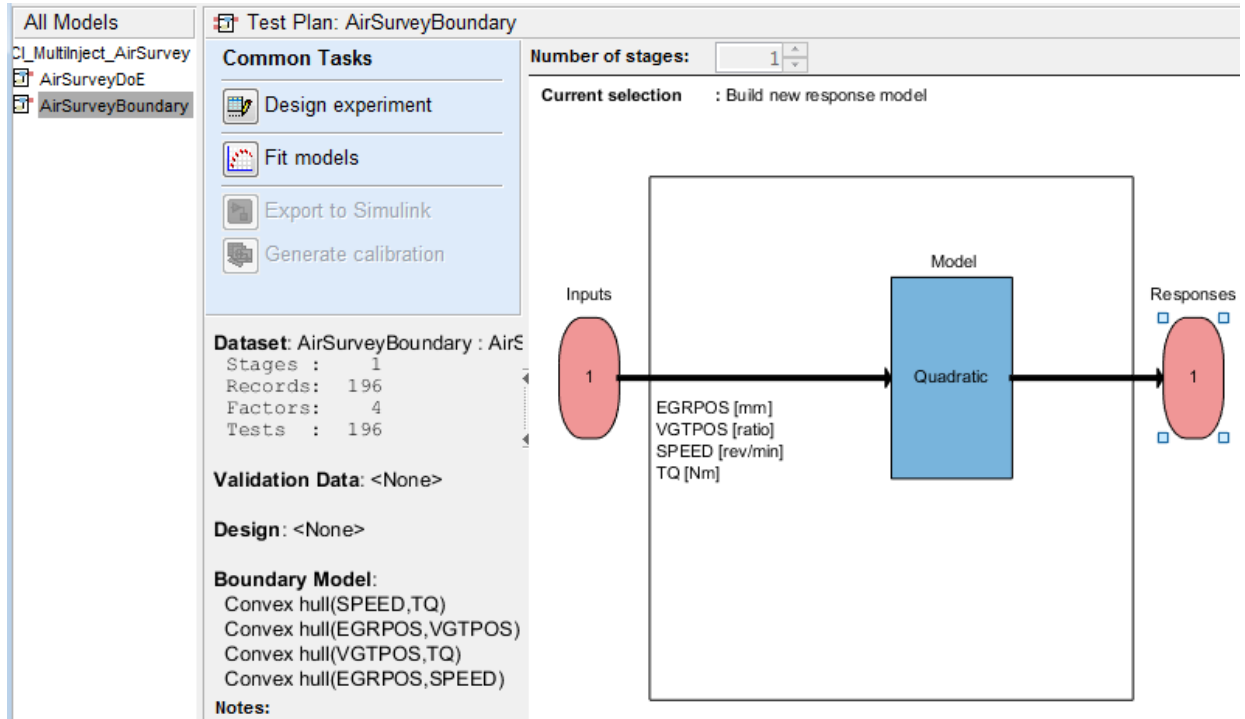
You need to fit a boundary model to the data collected at the air survey design points. The test data from the air survey was used to create a boundary model of the engine operating range. The example Model Browser project file `CI_MultiInject_AirSurvey.mat` contains this boundary model.

- 1 In the Model Browser, click the second test plan node in the **All Models** tree, `AirSurveyBoundary`.

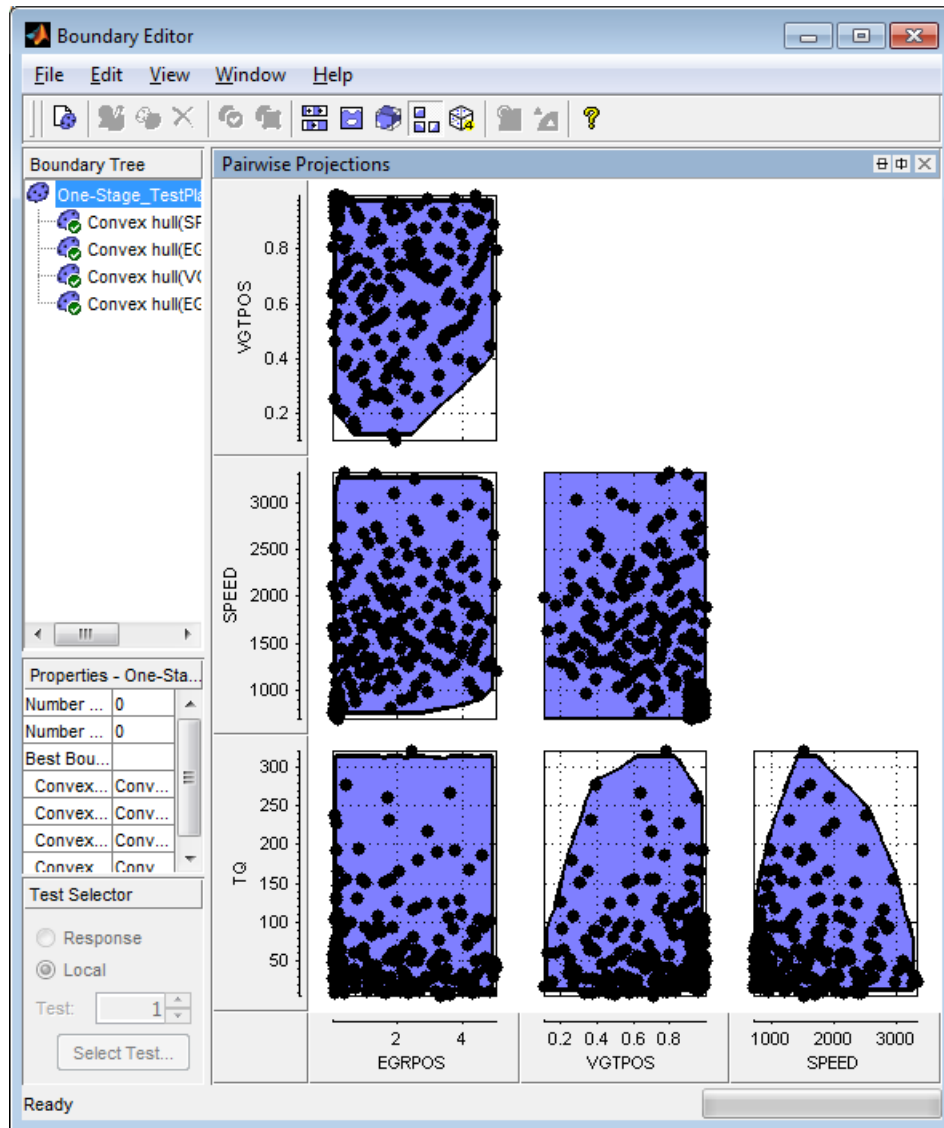
Compare with the `AirSurveyDoE` test plan view. For the boundary modeling test plan,

- Observe an imported data set listed under the **Common Tasks** pane. The second test plan has imported the air survey data in order to fit a boundary model to the data.
- Observe the **Inputs** are different in the test plan diagram. Instead of the `AFRCMD` input in the DoE test plan, there is a `SPEED` input for boundary modeling. `AFRCMD` was used to constrain the design points to collect the air survey data.

To model the boundary before creating the final design, you now need the SPEED input instead.



- 2 In the Model Browser, select **TestPlan** > **Fit Boundary** to open the Boundary Editor.
- 3 Examine the boundary model by selecting **View** > **Current View** > **Pairwise Projections**. The plots show the shape across the inputs.



Use the Air Survey and Boundary Model to Create the Final Design

The initial air survey design and test data provide information about the engine operating envelope, where the feasible AFR range can produce positive brake torque. The resulting data lets you create a boundary model. You can use the boundary model to create the final design avoiding infeasible points, and in later steps to guide modeling and constrain optimizations.

Using the boundary model as a constraint, you can generate the final design to collect detailed data about fuel injection effects within those boundaries. You can then use this data to create response models for all the responses you need in order to create an optimal calibration for this engine.

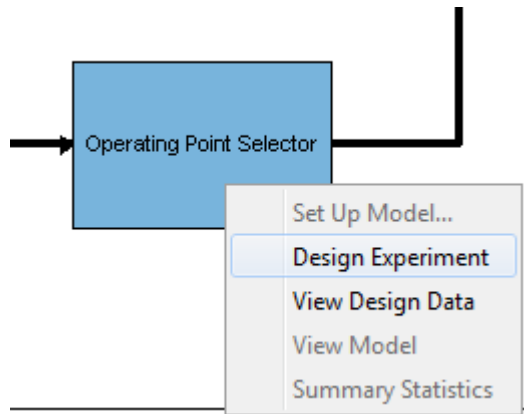
The final Point-by-Point Multi-Injection design (around 7000 points) was generated using a MATLAB script together with the Air Survey boundary model.

- 1** Open the example files `CI_PointbyPointDoE.m` and `createCIPointbyPointDoEs.m` in the `mbctraining` folder to see the script commands that build the final design.

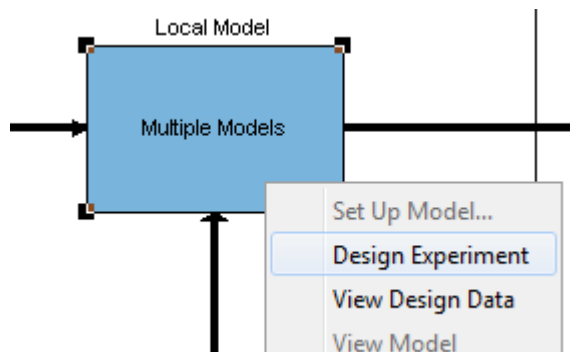
The script keeps only points that lie within the boundary model, and continues generating design points until it reaches the desired number of points. The Sobol space-filling design type keeps filling in spaces for good coverage of the design space.

After generating design points for each test, the script creates a Model Browser project with a point-by-point test plan, and attaches the point-by-point designs to the test plan.

- 2** Open the project `CI_MultiInject_PointbyPoint.mat` to view the project created by the script.
- 3** Select the second test plan node in the tree, and then click the Test Plan tab. In the test plan diagram, right-click the Operating Point Selector block, and select **Design Experiment** to view the designs created by the script.



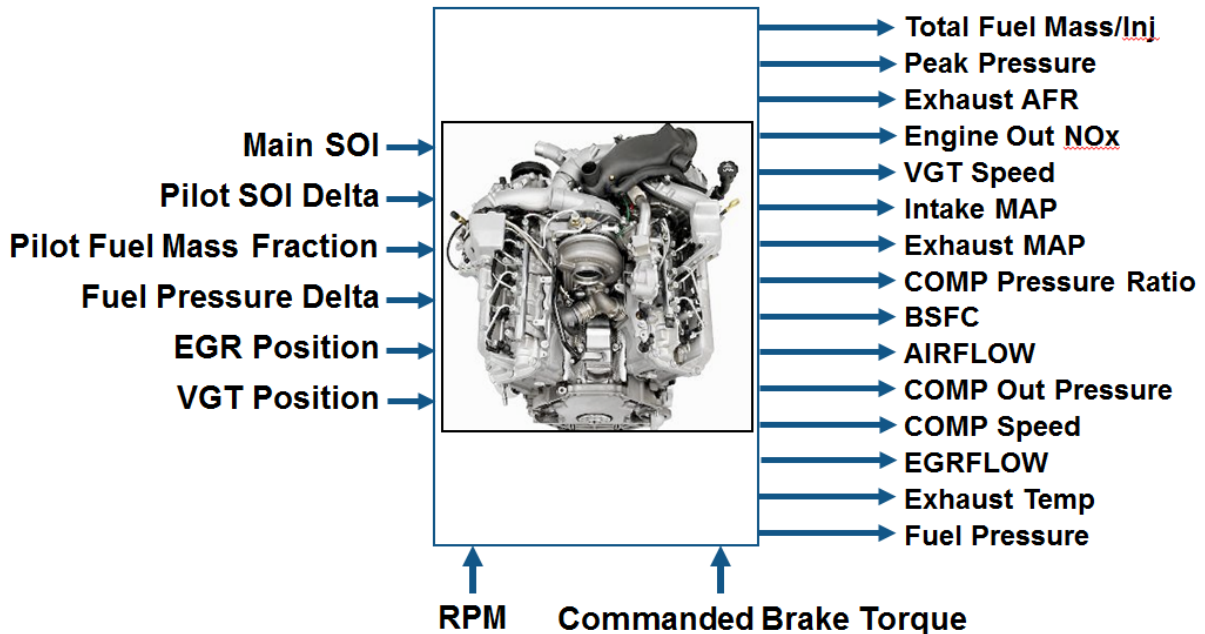
- 4 In the Design Editor, select **Mode Points** in the Design tree, and view the 2D Design Projection.
- 5 In the Design tree, select **Actual Design**. Observe that the boundary model constraint removed points in the corners, so that the actual design points collect data only in the feasible region determined by the initial air survey.
- 6 Return to the Model Browser test plan, right-click the Local Model block, and select **Design Experiment**.



- 7 Click test numbers to view the local design points at each operating point. At each test, the values of **SPEED** and **TQ** are fixed and the space-filling design selects points across the space of the local inputs.

Multi-Injection Testing

The final design was used to collect data for the following inputs and responses.



The toolbox provides the data files for you to explore this calibration example. You can view the data in spreadsheet form in the file `CI_MultiInject_PointByPoint_Data.xls`, and the data is imported to the Model Browser project file `CI_MultiInject_PointbyPoint.mat`.

For details on how the data was collected, see “Data Collection and Physical Modeling” on page 8-10.

For next steps, see “Statistical Modeling” on page 8-35.

Tip: Learn how MathWorks Consulting helps customers develop engine calibrations that optimally balance engine performance, fuel economy, and emissions requirements: see [Optimal Engine Calibration](#).

Statistical Modeling

In this section...

“Examine the Test Plans for Point-by-Point Models” on page 8-35

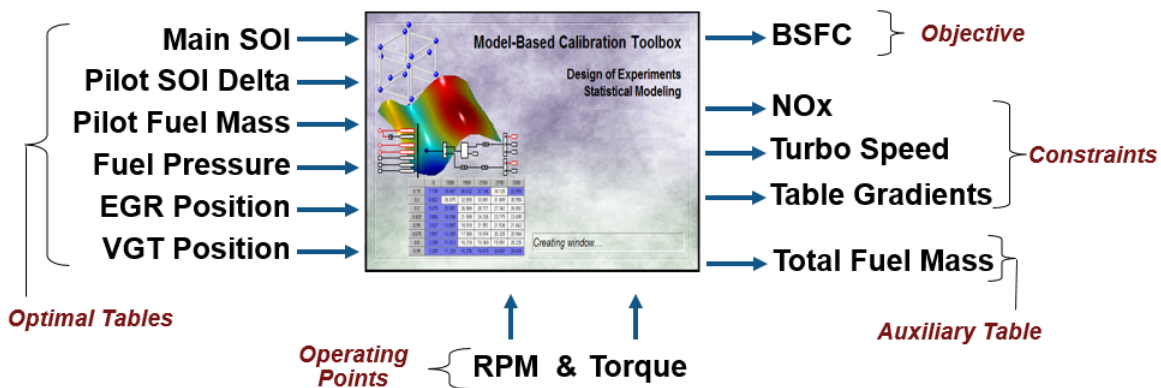
“Examine Response Models” on page 8-40

Examine the Test Plans for Point-by-Point Models

After designing the experiments and collecting the data, you can fit statistical models to the data. You can use the toolbox to generate accurate, fast-running models from the measured engine data.

The following graphic shows the models to define in the toolbox to solve this calibration problem. The graphic shows how the model inputs and output relate to the optimal tables, optimization operating points, objectives and constraints you need to perform the optimization and create the calibration.

I/O of Multi-Inject 3.1L Common Rail Engine Model with Variable Geometry Turbocharger and Cooled EGR



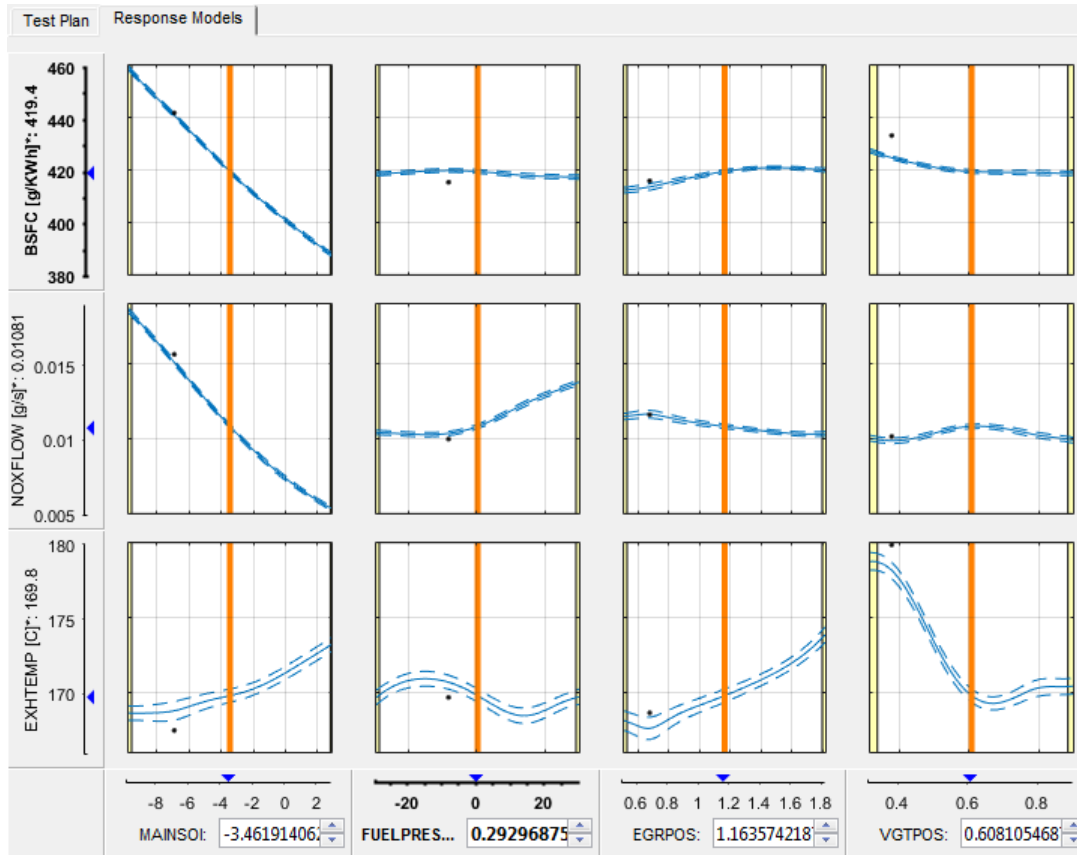
Goal: Minimize BSFC subject to NOx, turbocharger speed, and user-specified table gradient constraints

The toolbox provides the data for you to explore this calibration example. For details on how the data was collected, see “Data Collection and Physical Modeling” on page 8-10.

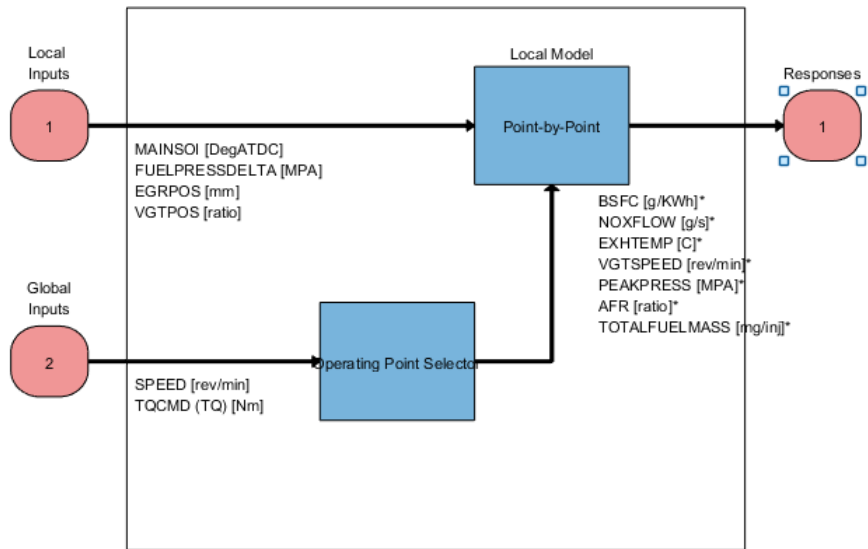
Examine the model setup.

- 1 In MATLAB, on the **Apps** tab, in the **Math, Statistics and Optimization** group, click **MBC Model Fitting**.
- 2 In the Model Browser home page, in the **Case Studies** list, select **Multi-injection diesel tested with pilot injection on and off**. Alternatively, select **File > Open Project** and browse to the example file `CI_MultiInject_PointbyPoint.mat`, found in `matlab\toolbox\mbc\mbctraining`.
- 3 The Model Browser remembers views, so change to the test plan view if necessary, by clicking the `PilotInactivePointbyPoint` test plan node in the **All Models** tree.

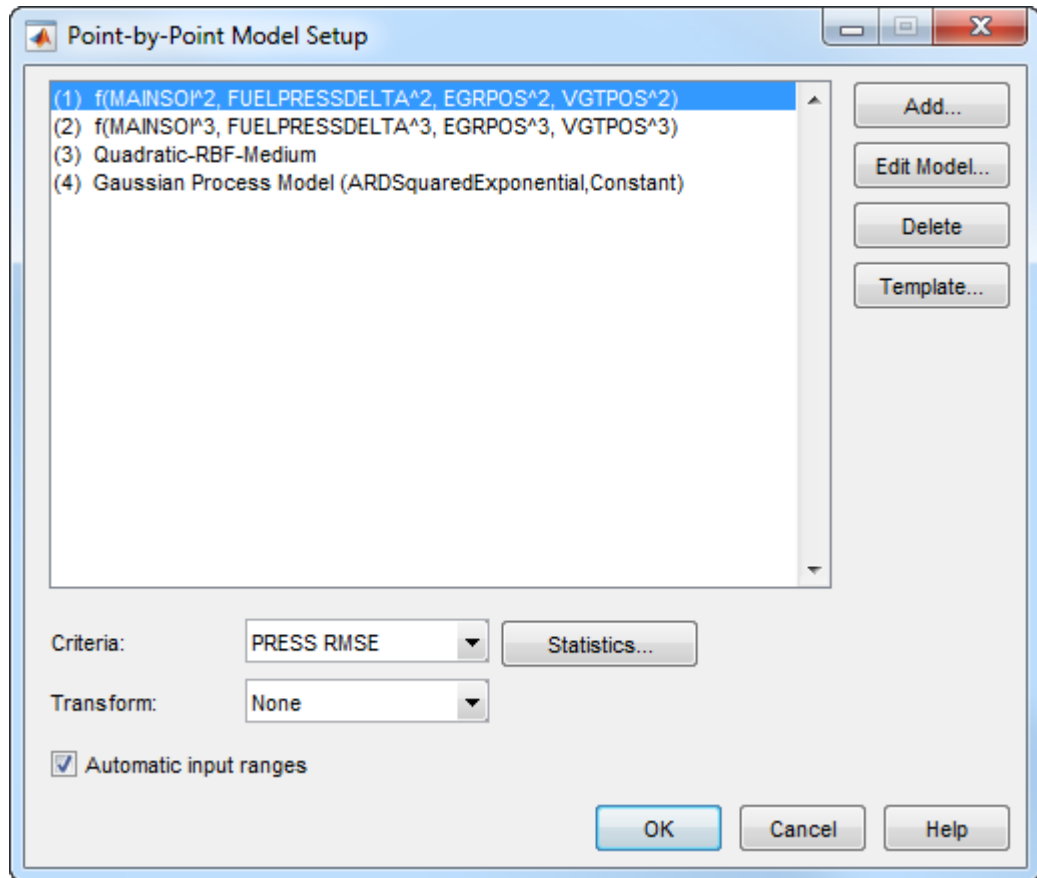
The **Response Models** tab shows the cross-section view of the responses. Here you can assess high-level model trends.



4 Select the **Test Plan** tab.



- 5 Observe the inputs and response model outputs listed on the test plan diagram. This is a Point-by-Point test plan. The Global Inputs **SPEED** and **TQCMD** select the operating points for each model.
- 6 Double-click the **Inputs** blocks to view the ranges and names (symbols) for variables on the Input Factor Set Up dialog box.
- 7 Double-click the **Local Model** block to view that the Point-by-Point Model Setup dialog. View the list of alternative models to fit at each operating point for a point-by-point test plan. When you use the **Fit models** button in the **Common Tasks** pane, and select a **Point-by-Point** template, the toolbox selects this list of models. This model setup fits four alternative model types at each operating point, and selects the best model based on the **Criteria** setting, in this case **PRESS RMSE**.



- 8 Click **Cancel** to close the Model Setup dialog box without altering your example models.
- 9 Similarly, examine the `PilotActivePointbyPoint` test plan. This test plan has the same response models and model type setup, but two more local inputs for the pilot injection timing and mass, `PILOTDELTASOI` and `PILOTFMF`. The data used to fit the models is also different, with the two additional test factors. Observe the **Dataset** information under the **Common Task** pane in the test plan view.

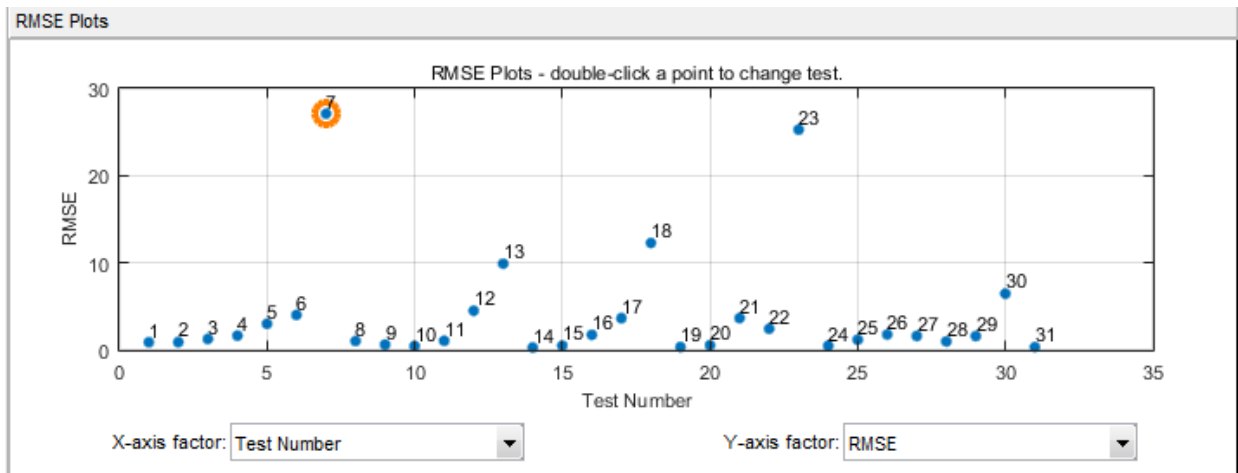
For details on setting up point-by-point models, see “Fit a Point-by-Point Model”.

Examine Response Models

- 1 Expand the `PilotInactivePointByPoint` test plan node in the **All Models** tree and select the nodes for each response name, e.g., `BSFC`.
- 2 Click through tests to see the model selected as best at each point.

The toolbox tries to select the best model for each operating point based on the selected statistical criteria. However, you should always verify the model choices. To search for the best fit you must examine each model, consider removing outliers, and compare with alternative fits. The example models show the results of this process.

- 3 Look at the RMSE Plots. These plots can help you identify problem tests to investigate. Double-click to view a test of interest (with larger RMSE) in the other plots.



- 4 Choose a model to use at a particular operating point by selecting the **Best Model** check box in the list of **Alternative Local Models**.

Alternative Local Models						
Name	Observations	Parameters	Box-Cox	PRESS RMSE	RMSE	Best Model
Quadratic	59	8	1	89.554	80.283	<input type="checkbox"/>
Cubic	59	12	1	74.319	65.344	<input type="checkbox"/>
Quadratic-RBF-Medium	59	33	1	44.751	27.093	<input checked="" type="checkbox"/>
GPM-ARDSquaredExpo...	59	8.259	1	94.567	76.468	<input type="checkbox"/>

For details on analyzing point-by-point models, see “Analyze Point-by-Point Models and Choose the Best”.

For next steps, see “Optimization” on page 8-42.

Tip: Learn how MathWorks Consulting helps customers develop engine calibrations that optimally balance engine performance, fuel economy, and emissions requirements: see [Optimal Engine Calibration](#).

Optimization

In this section...

“Optimization Overview” on page 8-42

“Set Up Models and Tables for Optimization” on page 8-42

“Examine the Point Optimization Setup” on page 8-45

“Examine the Point Optimization Results” on page 8-49

“Create Sum Optimization from Point Optimization” on page 8-51

“Fill Tables from Optimization Results” on page 8-57

“Examine the Multiobjective Optimization” on page 8-67

Optimization Overview

After creating the statistical models to fit the data, you can use them in optimizations. You can use the accurate statistical engine model to replace the high-fidelity simulation and run much faster, enabling optimization to generate calibrations in feasible times.

- 1 Run an optimization to choose whether to use Pilot Injection at each operating point.
- 2 Optimize fuel consumption over the drive cycle, and meet these constraints:
 - Constrain total NO_x
 - Constrain turbocharger speed
 - Constrain smoothness of tables
- 3 Fill lookup tables for all control inputs.

Set Up Models and Tables for Optimization

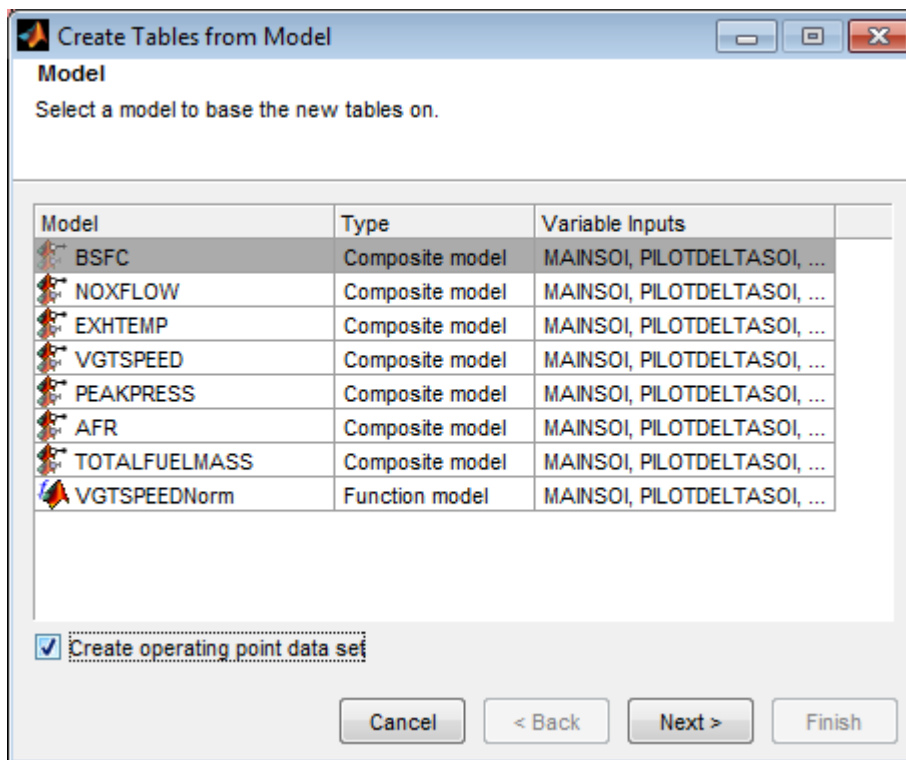
To perform an optimization, you need to import the statistical models created earlier in the Model Browser.

- 1 In MATLAB, on the **Apps** tab, in the **Math, Statistics and Optimization** group, click **MBC Optimization**.
- 2 To see how to import models, on the home page, click **Import From Project**

The CAGE Import Tool opens. Here, you can select models to import from a model browser project file or direct from the Model Browser if it is open. However, the

toolbox provides an example file with the models already imported, so click **Close** to close the CAGE Import Tool.

- 3 On the home page, in the Case Studies list, select the multi-injection example, or select **File > Open Project** and browse to the example file `CI_MultiInject.cag`, found in `matlab\toolbox\mbc\mbctraining`.
- 4 Click **Models** in the left **Data Objects** pane to view the models.
- 5 To see how to set up tables to fill with the results of your optimizations, select **Tools > Create Tables from Model**. The Create Tables from Model wizard appears.



- 6 Select the model BSFC and the **Create operating point data set** check box, and click **Next**.
- 7 If you left the defaults, you would use the model operating points for the table breakpoints. However, you need to create different tables to the operating points.
 - a Clear the **Use model operating points** check box.

- b Select SPEED and TQCMD for the axis inputs.
- c Enter 11 for the table rows and columns.

Table Inputs
Select the table inputs and set up the normalizers to use for all the new tables.

Use model operating points

Y-axis input: SPEED X-axis input: TQCMD

Normalizer: <New> Normalizer: <New>

Table rows: 11 Table columns: 11

SPEED normalizer:

Input	Output
800	0
1045	1
1290	2
1535	3
1780	4
2025	5

TQCMD normalizer:

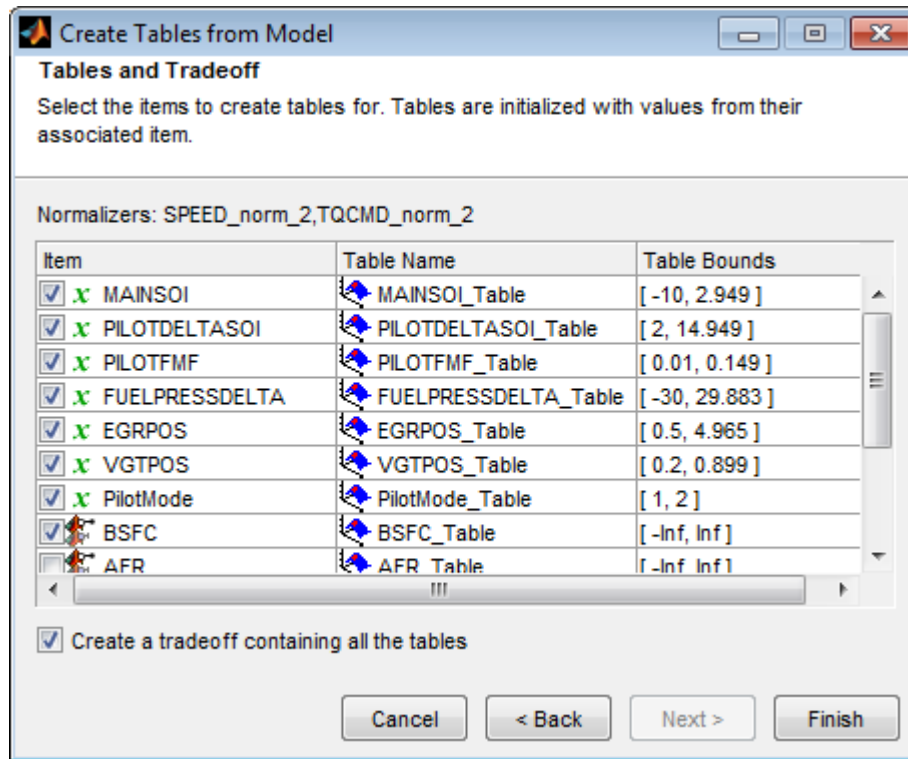
Input	Output
30	0
56	1
82	2
108	3
134	4
160	5

Buttons: Cancel, < Back, Next >, Finish

Click **Next**.

- 8 View the last screen to see how many tables CAGE will create for you if you finish the wizard.

Click **Cancel** to avoid creating new unnecessary tables in your example file. The example file already contains all the tables needed for your calibration results.

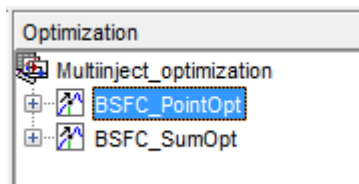


- 9 Select the **Tables** view to see all the tables. Observe there are tables named `_wPilot` and `_noPilot`, that will be filled by optimization results. The goal is to fill separate tables for each mode, pilot injection active and no pilot injection.

Examine the Point Optimization Setup



The example file `CI_MultiInject.cag` shows the results of the multistage process to finish this calibration.

- 1 Click **Optimization** in the left **Data Objects** pane to view the two optimizations.

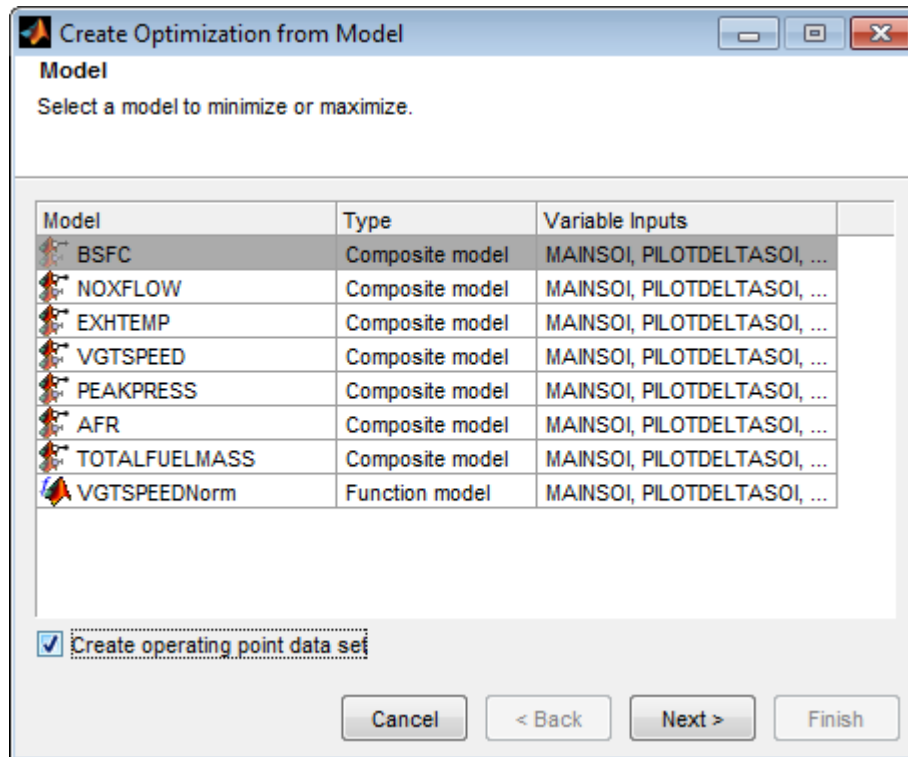


Why are there two optimizations? To complete the calibration, you need to start with a point optimization to determine whether to use Pilot Injection at each operating point. You use the results of this point optimization to set up the sum optimization to optimize fuel consumption over the drive cycle, and meet these constraints:

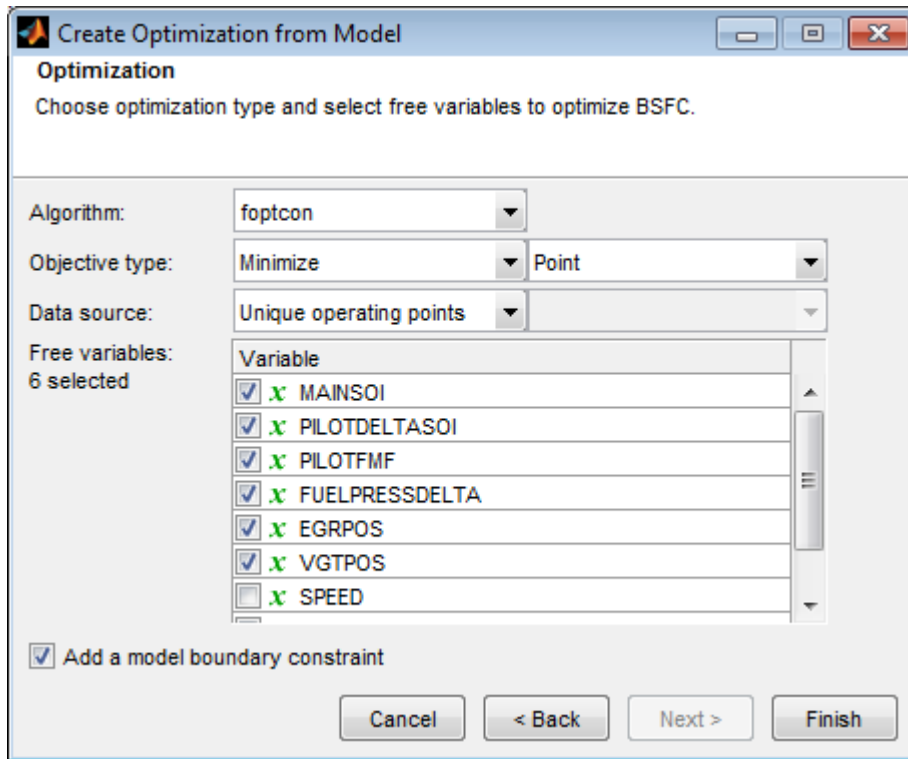
- Constrain total NO_x
 - Constrain turbocharger speed
 - Constrain smoothness of tables
- 2 Select the **BSFC_PointOpt** optimization to view the setup.
 - 3 View the two objectives in the Objectives pane. These are set up to minimize BSFC and PEAKPRESS. Double-click an objective to view its settings.

Objectives		
Name	Description	Type
 BSFC	BSFC(MAINSOI, PILOTDELTAOI, PILOTFMF, FUELPRESSDELTA, EGRPOS, VGTPOS, SPEED, TQCMD, PilotMode)	Minimize
 PEAKPRESS	PEAKPRESS(MAINSOI, PILOTDELTAOI, PILOTFMF, FUELPRESSDELTA, EGRPOS, VGTPOS, SPEED, TQCMD, PilotMode)	Minimize

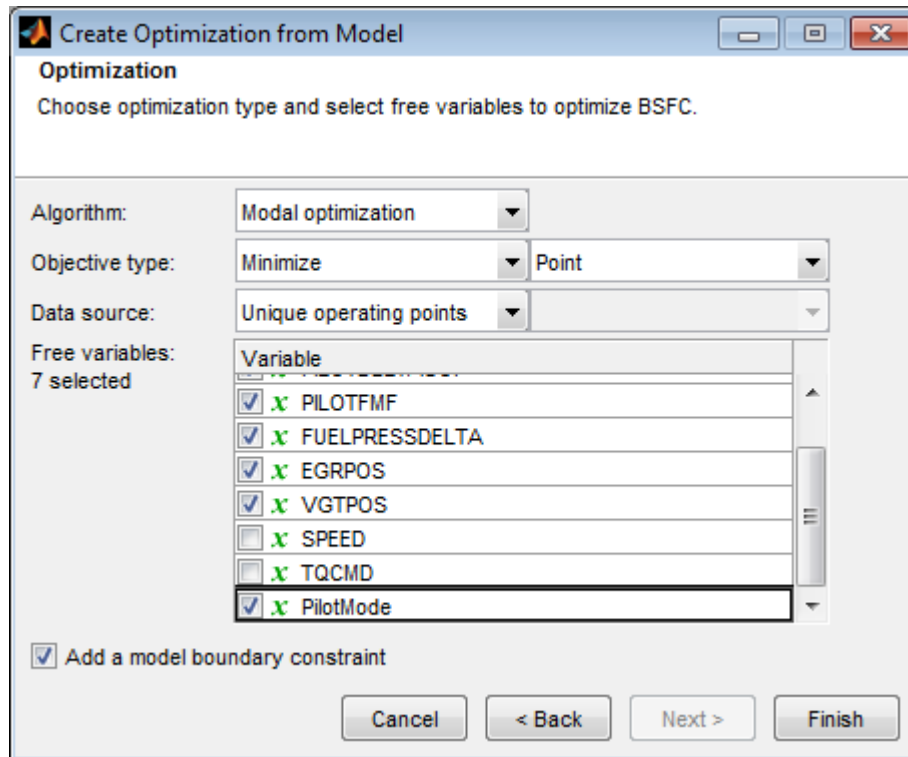
- 4 Observe that the Constraints pane shows a boundary model constraint.
- 5 Learn the easiest way to set up an optimization like this by selecting **Tools > Create Optimization from Model**.



- 6 Select the model **BSFC** and the **Create operating point data set** check box, and click **Next**.
- 7 Observe that the default settings will create a point optimization to minimize **BSFC** at the model operating points, using six free variables and not **SPEED**, **TQCMD**, or **PilotMode**, and constrained by a model boundary constraint.



- 8 You want the optimization to identify which pilot mode (active pilot injection or no pilot injection) is best to use at each operating point. To do this, you must select **Modal optimization** from the **Algorithm** list.
- 9 You want to include **PilotMode** as a free variable, so the optimization can identify which mode is best to use at each operating point. From the **Free variables** list, select the check box to include **PilotMode**.



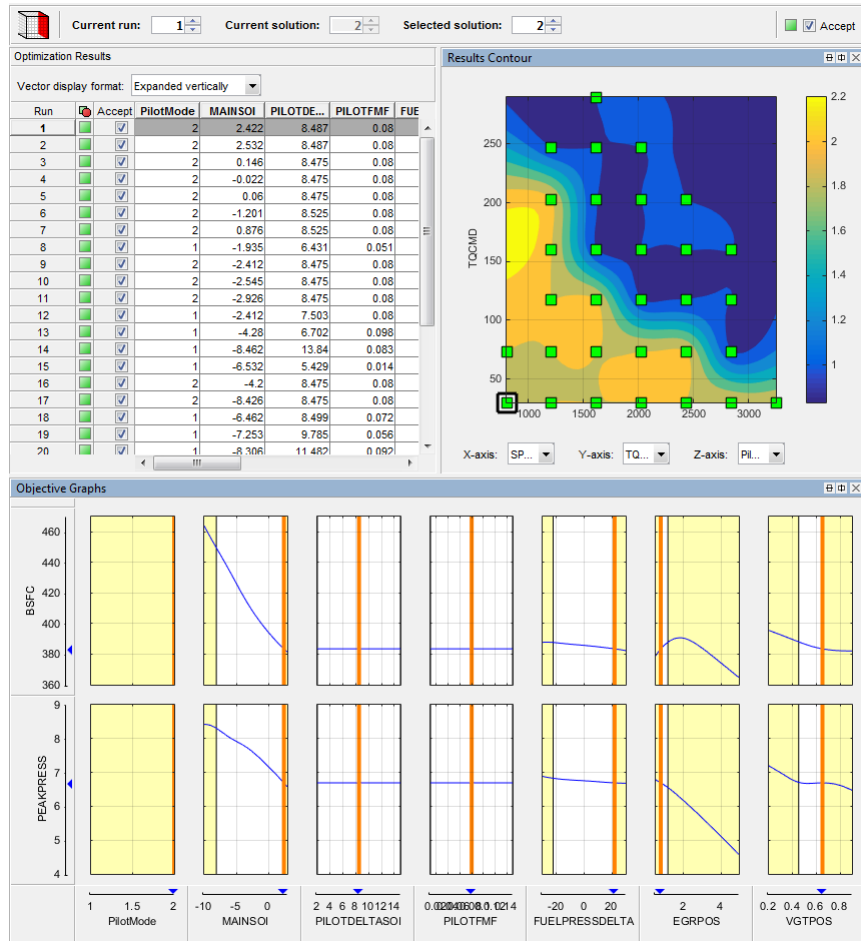
- 10 Click **Finish** to create the new optimization.
- 11 Compare your new optimization with the example BSFC_PointOpt. Notice the example has a second objective, to minimize PEAKPRESS. To see how to set this up, right-click the Objectives pane and select **Add Objective**.

Examine the Point Optimization Results

The example file CI_MultiInject.cag shows the results of the multistage process to finish this calibration.

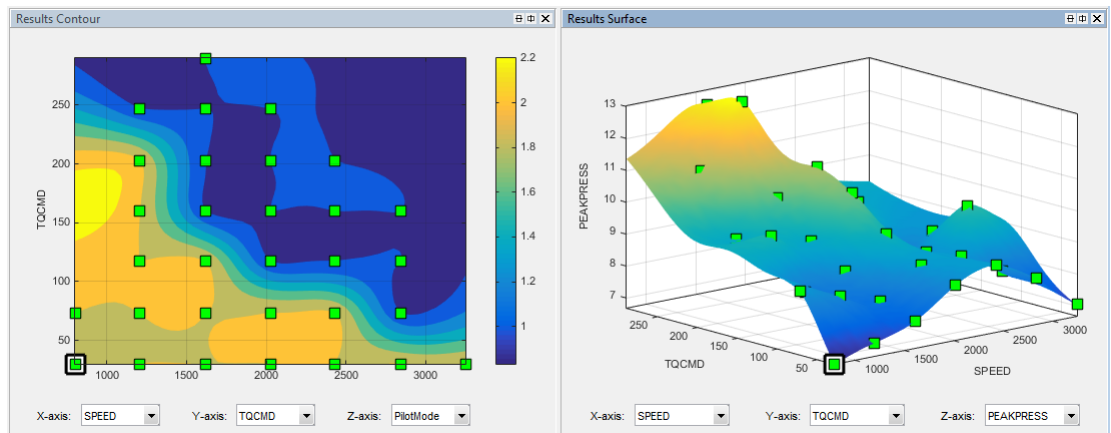
- 1 Expand the first optimization node, BSFC_PointOpt, and select the output node underneath, BSFC_PointOpt_output.
- 2 Select **View > Selected Solution** (or the toolbar button) to view which pilot mode the optimization selected as best at each operating point. You can see the selected

value of 1 or 2 at each point in the **Results Contour** plot and the **PilotMode** column in the **Optimization Results** table.



- Review the Results Contour plot to see which mode has been selected across all operating points. Use this view to verify the distribution of mode selection.
- Click to select a point in the table or Results Contour, and you can use the **Selected solution** controls at the top to alter which mode is selected at that point. You might want to change selected mode if another mode is also feasible at that point. For example, you can change the mode to make the table more smooth.

- 5 Use the other objectives to explore the results. For example, you might want to manually change the selected mode based on an extra objective value. It can be useful to view plots of the other objective values at your selected solutions.
 - a To display another plot simultaneously, right-click the Results Contour title bar and select **Split View**.
 - b Plot the objective BSFC on the Results Surface and observe the difference when you change a selected solution.
 - c Plot PEAKPRESS on the Results Surface and observe the difference when you change a selected solution.



- 6 To see both solutions for a particular operating point, use the Pareto Slice view. You can inspect the objective value (and any extra objective values) for each solution. If needed, you can manually change the selected mode to meet other criteria, such as the mode in adjacent operating points, or the value of an extra objective.

For details on tools for choosing a mode at each operating point, see “Analyzing Modal Optimization Results”.

Create Sum Optimization from Point Optimization

The point optimization results determine whether to use pilot injection at each operating point. When you are satisfied with all selected solutions for your modal optimization, you can make a sum optimization over all operating points. The pilot injection mode must be fixed in the sum optimization to avoid optimizing too many combinations of operating modes.

The results of the point optimization were used to set up the sum optimization to optimize fuel consumption over the drive cycle. To see how to do this:

- 1 From the point optimization output node, BSFC_PointOpt_output, select **Solution > Create Sum Optimization**.

The toolbox automatically creates a sum optimization for you with your selected best mode for each operating point. The create sum optimization function converts the modal optimization to a standard single objective optimization (foptcon algorithm) and changes the **Mode Variable** to a fixed variable.

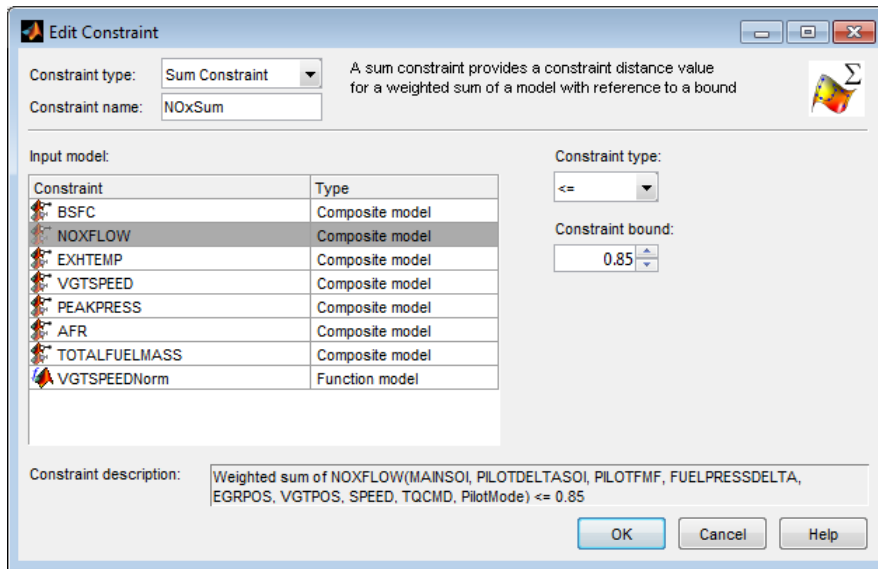
- 2 Compare your new optimization with the example sum optimization, BSFC_SumOpt. The example shows you need to set up more constraints to complete the calibration.

Constraints			
Name	Description	Application Point Set	Status
BSFC_Boundary	Boundary constraint of BSFC(M...		
NOxSum	Weighted sum of NOXFLOW(M...		
VGTSpeedMax	VGTSPEEDNorm(MAINSOI, PILO...		
MainSOI_wPilot	Gradient constraint of MAINSOI ...	PilotActive(SPEED,TQCMD;PilotMode)	
MainSOI_noPilot	Gradient constraint of MAINSOI ...	PilotInactive(SPEED,TQCMD;PilotMode)	
EGRPos_wPilot	Gradient constraint of EGRPOS ...	PilotActive(SPEED,TQCMD;PilotMode)	
EGRPos_noPilot	Gradient constraint of EGRPOS ...	PilotInactive(SPEED,TQCMD;PilotMode)	
FuelPressDelta_w...	Gradient constraint of FUELPRE...	PilotActive(SPEED,TQCMD;PilotMode)	
FuelPressDelta_n...	Gradient constraint of FUELPRE...	PilotInactive(SPEED,TQCMD;PilotMode)	
VGTPos_wPilot	Gradient constraint of VGTPOS ...	PilotActive(SPEED,TQCMD;PilotMode)	
VGTPos_noPilot	Gradient constraint of VGTPOS ...	PilotInactive(SPEED,TQCMD;PilotMode)	
PilotDeltaSOI_wPilot	Gradient constraint of PILOTDEL...	PilotActive(SPEED,TQCMD;PilotMode)	
PilotFMF_wPilot	Gradient constraint of PILOTFM...	PilotActive(SPEED,TQCMD;PilotMode)	

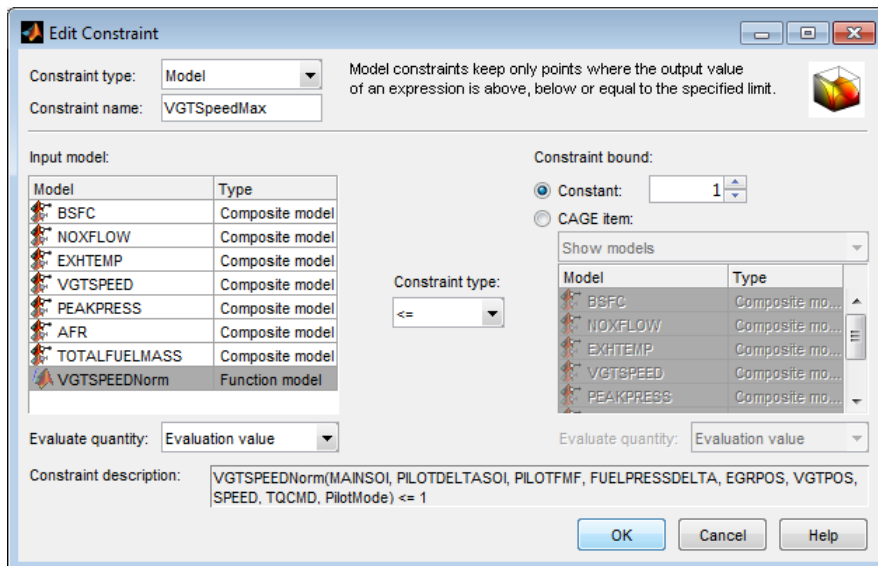
The additional constraints make the optimization meet these requirements:

- Constrain total NOx
 - Constrain maximum turbocharger speed
 - Constrain smoothness of tables with gradient constraints
- 3 Import all required constraints to your new optimization by selecting **Optimization > Constraints > Import Constraints**. Select the example sum optimization and import all but the first boundary model constraint.
 - 4 Double-click the additional constraints to open the Edit Constraint dialog box and view the setup.

This sum constraint controls the total NOx.

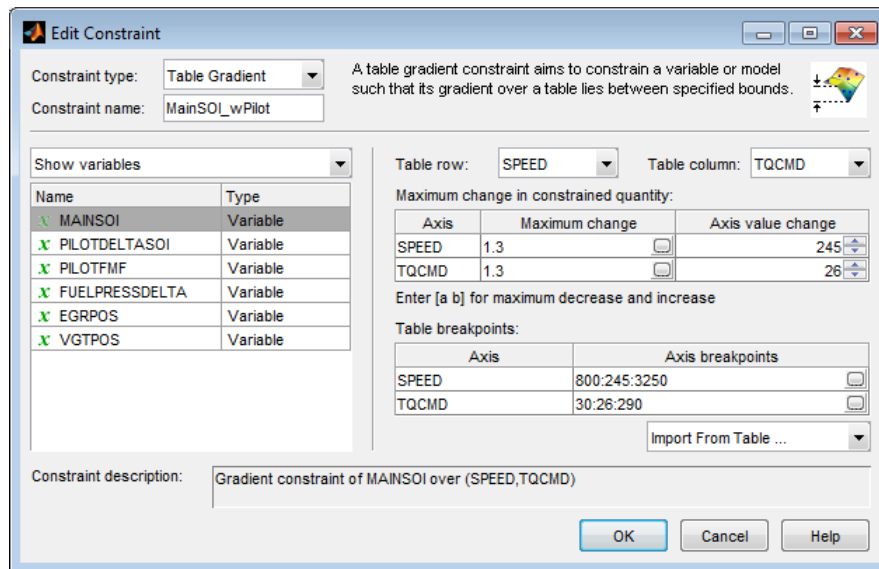


This constraint controls maximum turbocharger speed.



Observe that this constraint does not use the VGTSPEED model directly, but instead uses a function model VGTSPEEDNorm. You can examine this function model in the Models view. The function model scales the constraint to help the optimization routines, which have problems if constraints have very different sizes. VGTSPEED is of order of 165000, while the other constraints are of the order of 1, so the function model VGTSPEEDNorm normalizes VGTSPEED by dividing it by 165000.

The following constraint controls the gradient across the MAINSOI table.

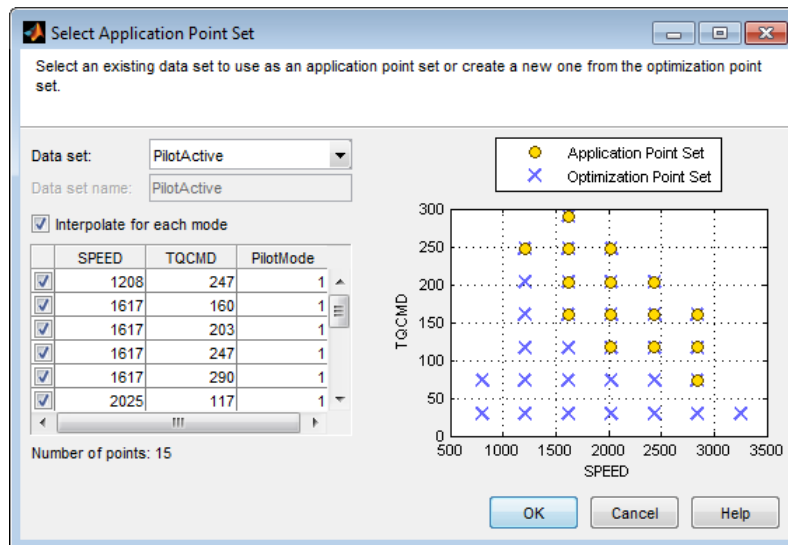


5 In the Optimization view, in the Constraints pane, observe that:

- The gradient constraints are in pairs, one with pilot and one with no pilot. Separate table gradient constraints are required for different modes because the goal is to fill separate tables for each mode.
- All the gradient constraints have an entry in the **Application Point Set** column.

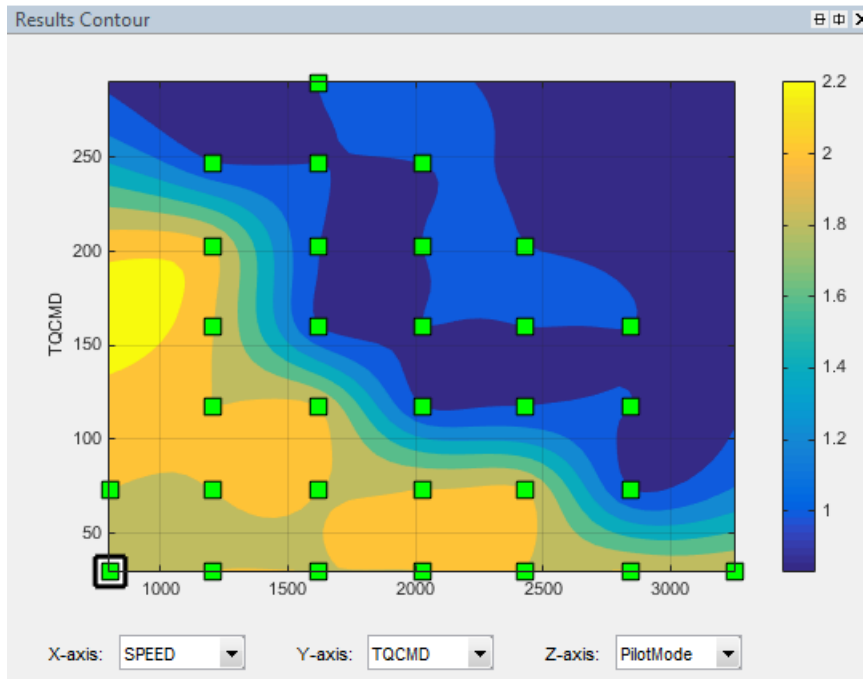
Constraints			
Name	Description	Application Point Set	St
BSFC_Boundary	Boundary constraint of BSFC(M...		
NOxSum	Weighted sum of NOXFLOW(M...		
VGTSpeedMax	VGTSPEEDNorm(MAINSOI, PILO...		
MainSOI_wPilot	Gradient constraint of MAINSOI ...	PilotActive(SPEED,TQC...	
MainSOI_noPilot	Gradient constraint of MAINSOI ...	PilotInactive(SPEED,TQ...	
EGRPos_wPilot	Gradient constraint of EGRPOS ...	PilotActive(SPEED,TQC...	
EGRPos_noPilot	Gradient constraint of EGRPOS ...	PilotInactive(SPEED,TQ...	
FuelPressDelta_w...	Gradient constraint of FUELPRE...	PilotActive(SPEED,TQC...	
FuelPressDelta_n...	Gradient constraint of FUELPRE...	PilotInactive(SPEED,TQ...	
VGTPos_wPilot	Gradient constraint of VGTPOS ...	PilotActive(SPEED,TQC...	
VGTPos_noPilot	Gradient constraint of VGTPOS ...	PilotInactive(SPEED,TQ...	
PilotActiveSOI_wPilot	Gradient constraint of PILOTDE...	PilotActive(SPEED,TQC...	
PilotActiveSOI_noPilot	Gradient constraint of PILOTDE...	PilotInactive(SPEED,TQ...	

- 6 Right-click the table gradient constraint **MainSOI_wPilot** and click **Select Application Point Set** to see how these are set up.

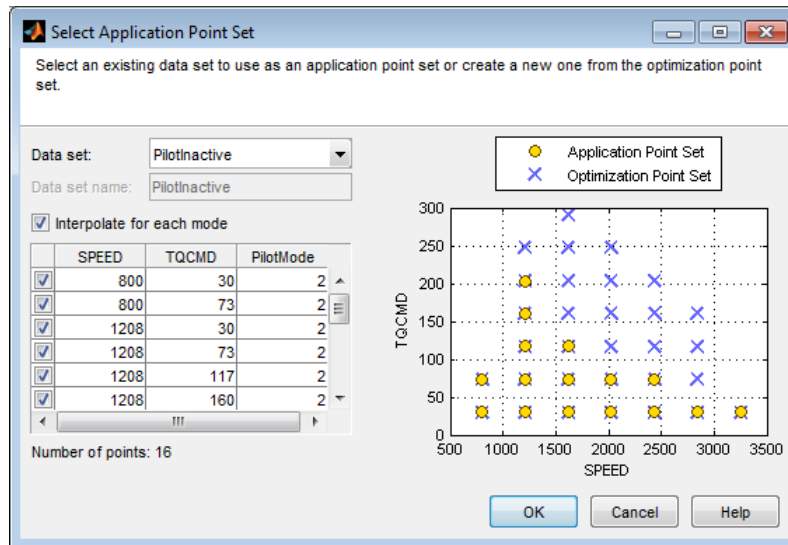


- 7 Observe the **Interpolate for each mode** option is selected. This application point set restricts the table gradient constraint to **PilotMode 1** (active) points only.

You can create an application point set like this by selecting **New subset** and then choosing a subset of the optimization points by clicking in the plot or table. The application points here correspond to the operating points where the point optimization determined that the pilot mode should be active (**PilotMode = 1**). For comparison, here is the results contour plot for the point optimization results.



- 8 Compare the results contour plot with the application points for the next table gradient constraint, **MainSOI_noPilot**. These application points restrict the table gradient constraint to **PilotMode 2** points only, where the pilot is inactive.

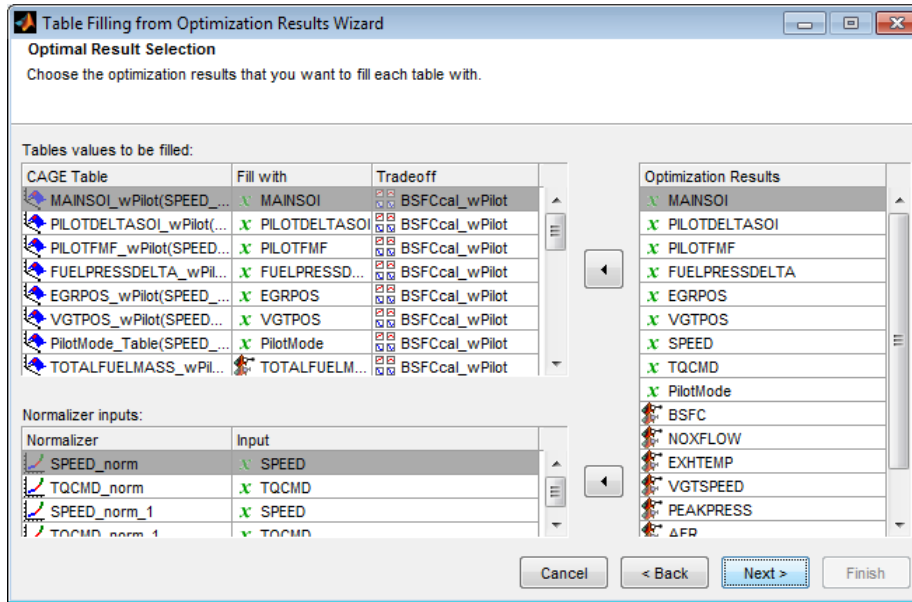


Your sum optimization now contains all the required constraints and is ready to run. Next, view the results in the example sum optimization.

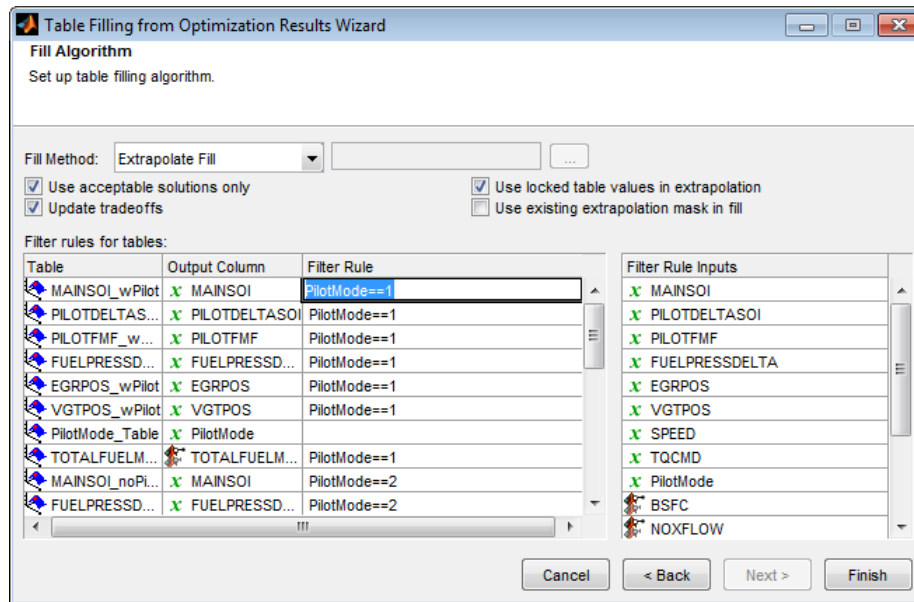
Fill Tables from Optimization Results

CAGE remembers table filling settings. To view how the example tables are filled:

- 1 Expand the example sum optimization node **BSFC_SumOpt** and select the **Output** node.
- 2 Select **Solution > Fill Tables** (or use the toolbar button) to open the **Table Filling from Optimization Results Wizard**.
- 3 On the first screen, observe all the tables in the **CAGE tables to be filled list**. Click **Next**.
- 4 On the second screen, observe all the tables are matched up with optimization results to fill them with. Click **Next**.

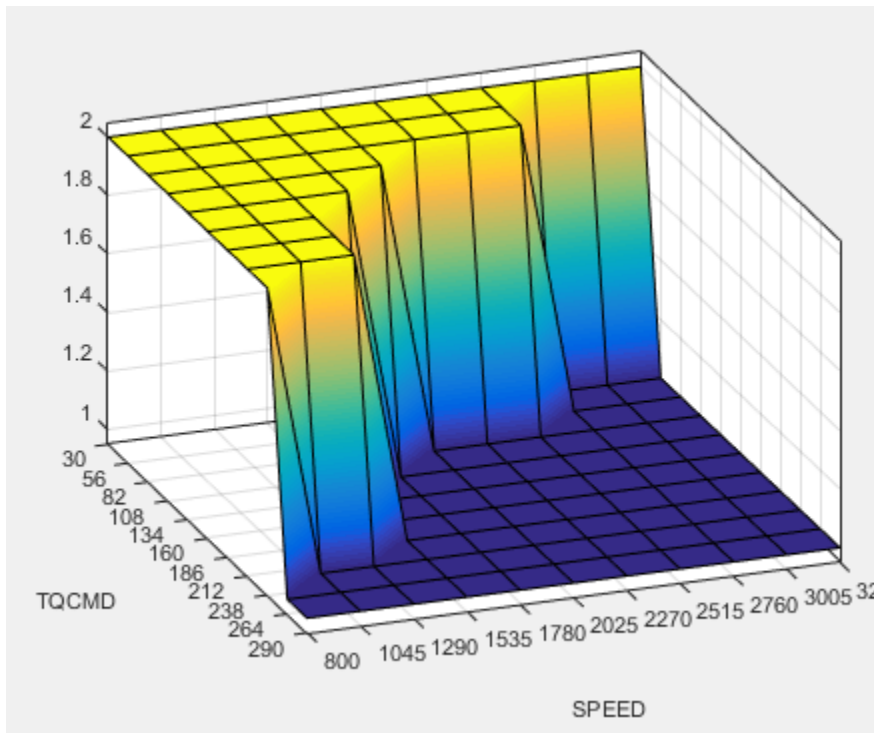


- On the third screen, observe how some tables have a **Filter Rule** set up so that they are filled only with results where PilotMode is 1 or 2. You can create a filter rule like this by entering PilotMode==1 in the **Filter Rule** column.



You can either click **Finish** to fill all the tables, or **Cancel** to leave the tables untouched. The example tables are already filled with these settings.

The following plots show the calibration results.

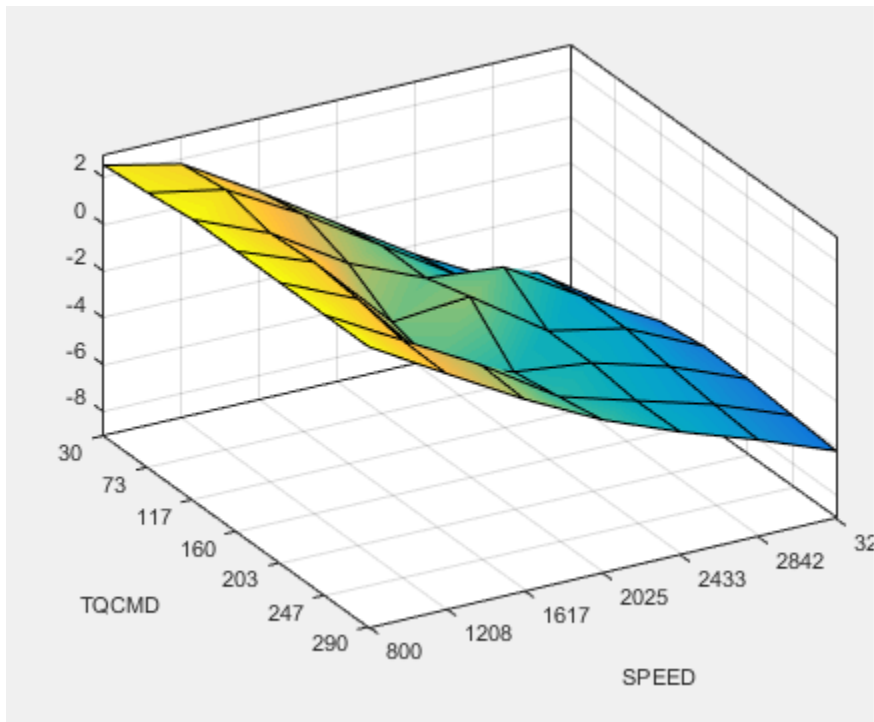


Pilot Mode Table

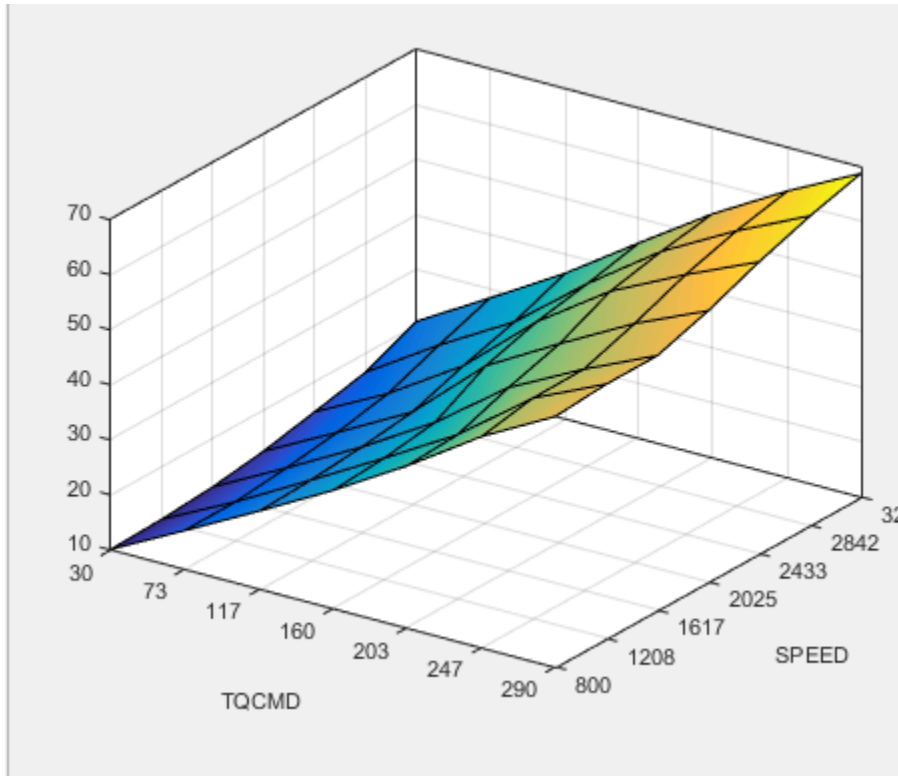
This graphic shows the plot of the table to select the active or inactive pilot mode depending on the speed and commanded torque

You need to fill calibration tables for each control variable described in “Multi-Injection Diesel Problem Definition” on page 8-2, in both pilot modes, active and inactive.

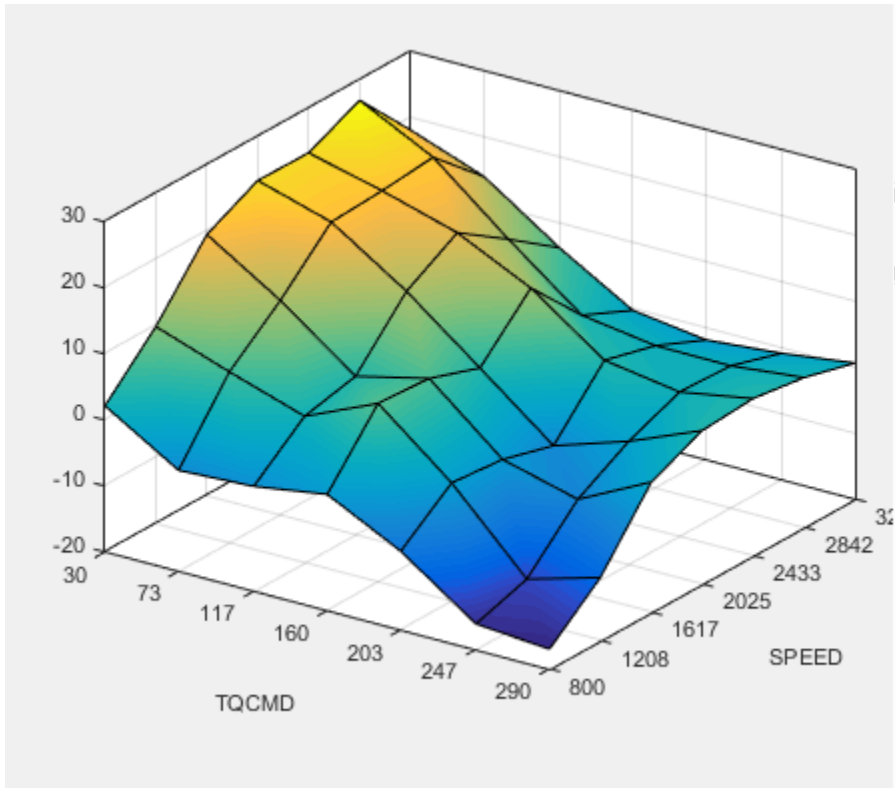
Following are all the pilot active tables.



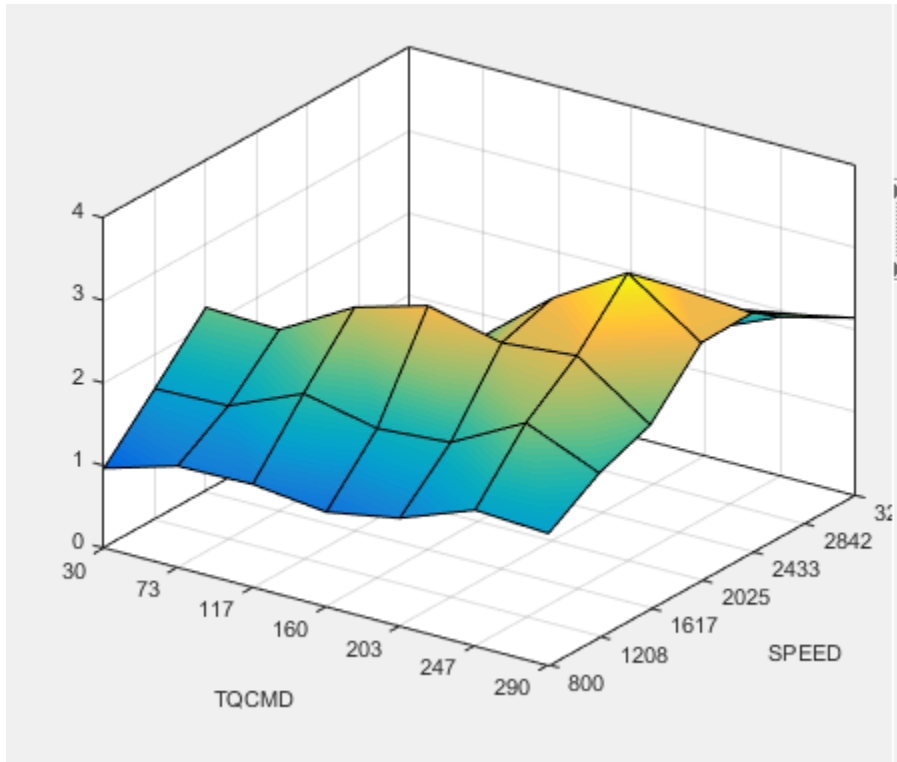
Main Injection Timing (SOI) Table



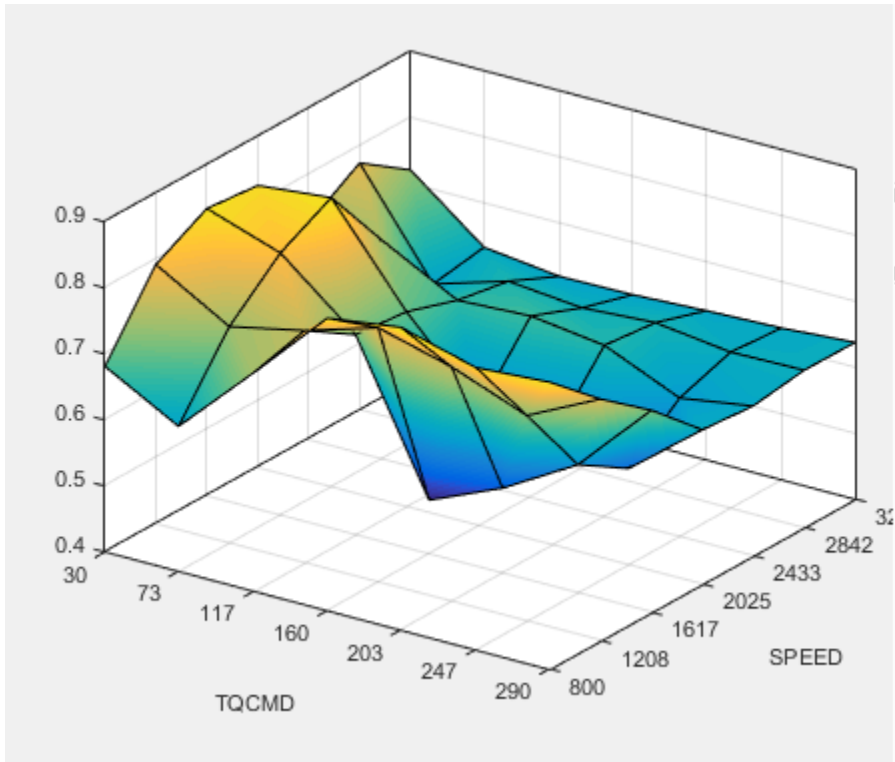
Total Injected Fuel Mass Table



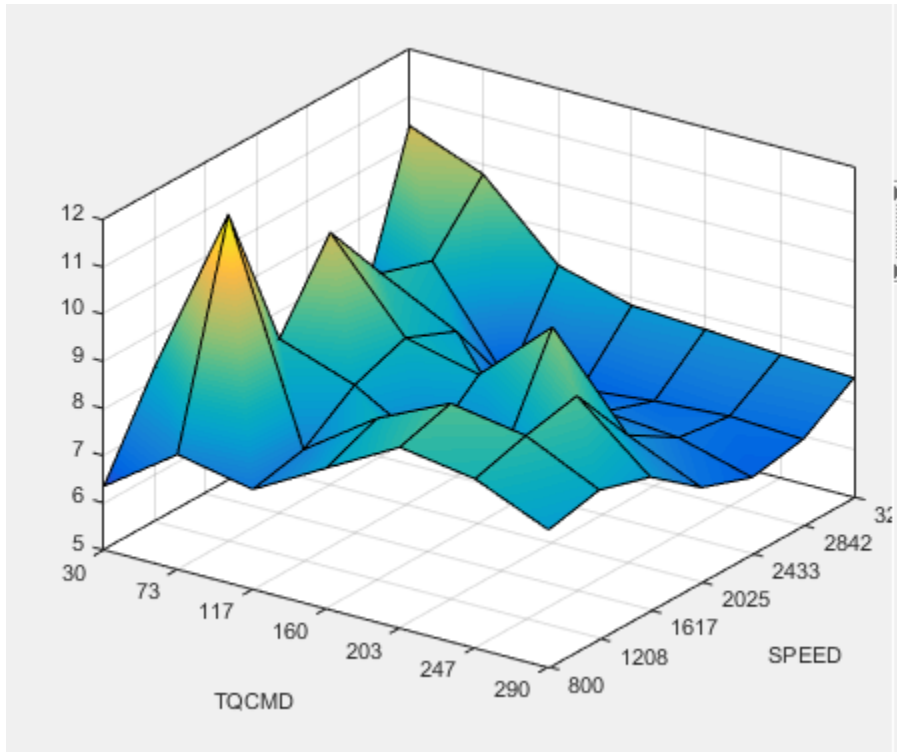
Fuel Pressure Delta Table



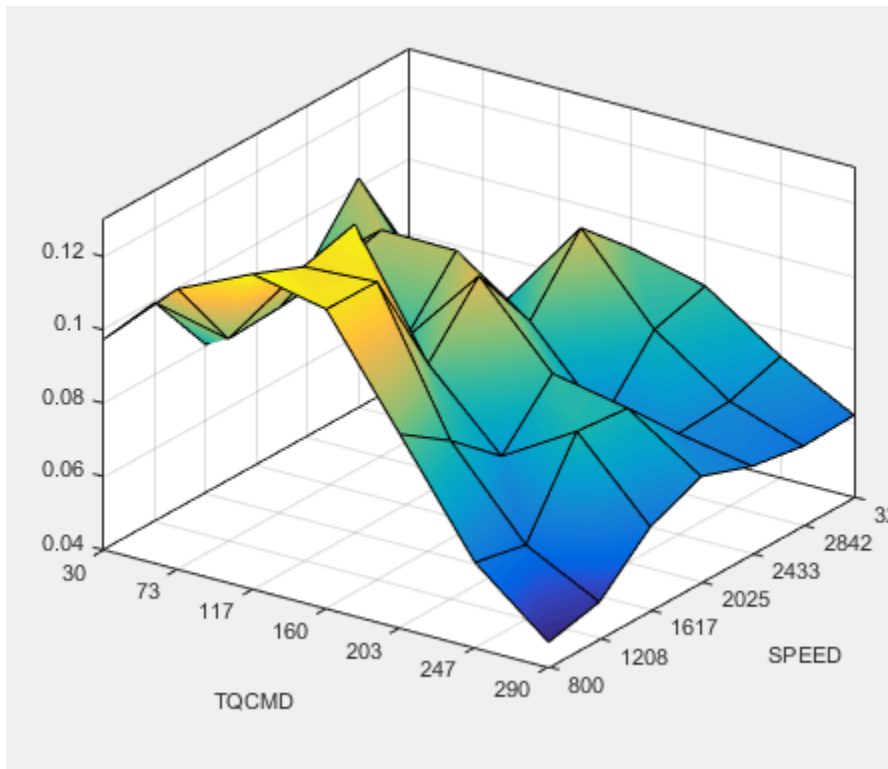
Exhaust Gas Recirculation (EGR) Valve Position Table



Variable-Geometry Turbo (VGT) Position Table



Pilot Injection Timing (Pilot SOI Delta) Table



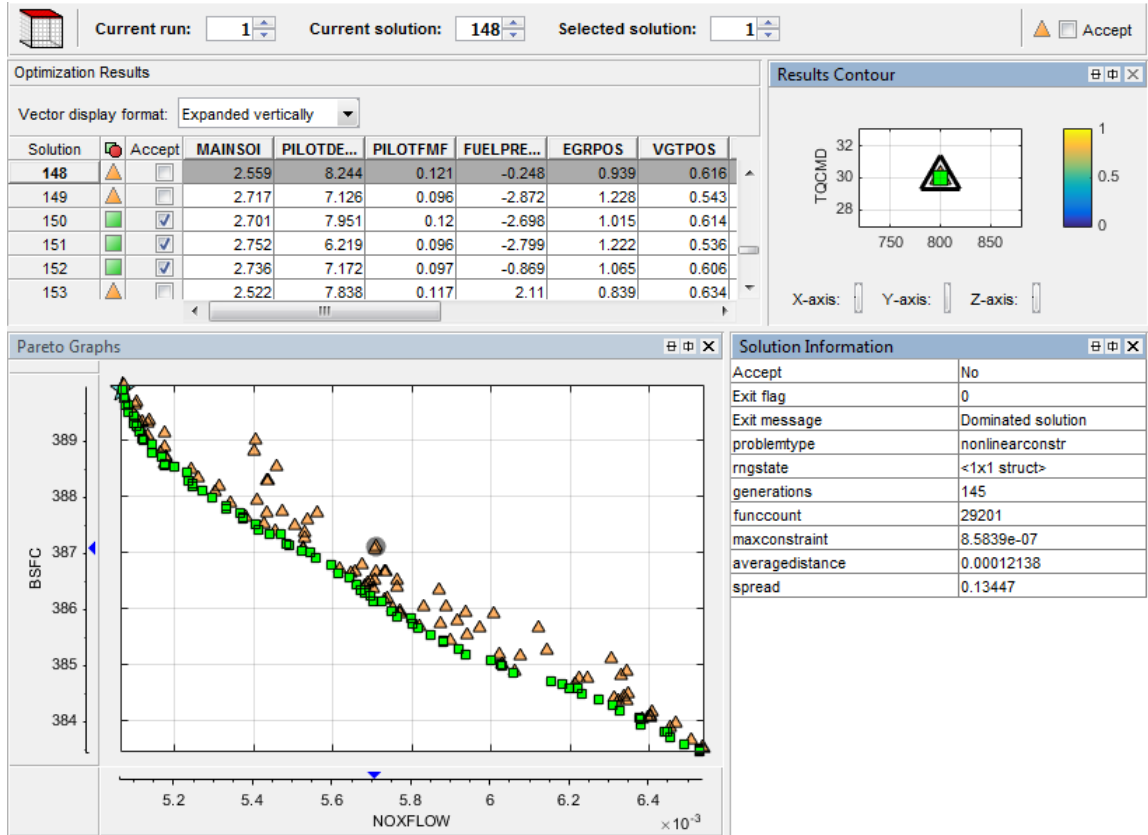
Pilot Fuel Mass Fraction Table

Examine the Multiobjective Optimization

The example file `CI_MultiInject.cag` also shows an example multiobjective optimization. This can be useful for calibrations where you want to minimize more than one objective at a time, in this case, BSFC and NOX. The multiobjective optimization uses the `gamultiobj` algorithm.

- 1 Select the `BSFC_NOX` node to see how the optimization is set up. Observe the 2 objectives and the optimization information: Multiobjective optimization using a genetic algorithm.
- 2 Expand the `BSFC_NOX` node and select the `BSFC_NOX_Output` to view the results.

- Examine the Pareto Graphs. It can be useful to display the Solution Information view at the same time to view information about a selected solution. You might want to select a dominated solution (orange triangle) over a pareto solution (green square) to trade off desired properties.



- Select the Sum_BSFC_NOX node to see how the sum optimization is set up. The sum optimization was created from the point optimization results. Observe the 2 objectives and all the constraints.
- Expand the Sum_BSFC_NOX node and select the Sum_BSFC_NOX_Output to view the results.

Tip: Learn how MathWorks Consulting helps customers develop engine calibrations that optimally balance engine performance, fuel economy, and emissions requirements: see [Optimal Engine Calibration](#).

Model Quickstart

This section discusses the following topics:

- “Empirical Engine Modeling” on page 9-2
- “Two-Stage Modeling Example” on page 9-3
- “Open the App and Load Data” on page 9-5
- “Set Up the Model” on page 9-7
- “Verify the Model” on page 9-11
- “Export the Model” on page 9-15
- “Create Multiple Models to Compare” on page 9-17

Empirical Engine Modeling

MBC Model Fitting provides tools for building, evaluating and comparing statistical models fitted to engine data.

- To get started with empirical engine modeling, follow the steps in this workflow: “Fit a One-Stage Model”.
- To view an example project with engine data and finished models, see “Gasoline Engine Calibration”.

To get started with two-stage modeling, follow the steps in this example: “Two-Stage Modeling Example” on page 9-3.

Related Examples

- “Fit a One-Stage Model”
- “Gasoline Engine Calibration”
- “Multi-Injection Diesel Calibration”

Two-Stage Modeling Example

About the Two-Stage Modeling Example

This two-stage modeling example shows you how to use MBC Model Fitting to create a statistical model of an automobile engine that predicts the torque generated by the engine as a function of spark angle and other variables. The toolbox provides example engine data. The instructions show you how to load data, fit a statistical model to the data, examine and verify the fit, and export the model.

In the normal modeling process, you would create many different models for one project and compare them to find the best solution. The tutorial also provides a quick guide to fitting and comparing multiple models.

To get started with two-stage modelling, follow the steps in these sections in order:

- 1 “About Two Stage Models” on page 9-3
- 2 “Open the App and Load Data” on page 9-5
- 3 “Set Up the Model” on page 9-7
- 4 “Verify the Model” on page 9-11
- 5 “Export the Model” on page 9-15
- 6 “Create Multiple Models to Compare” on page 9-17

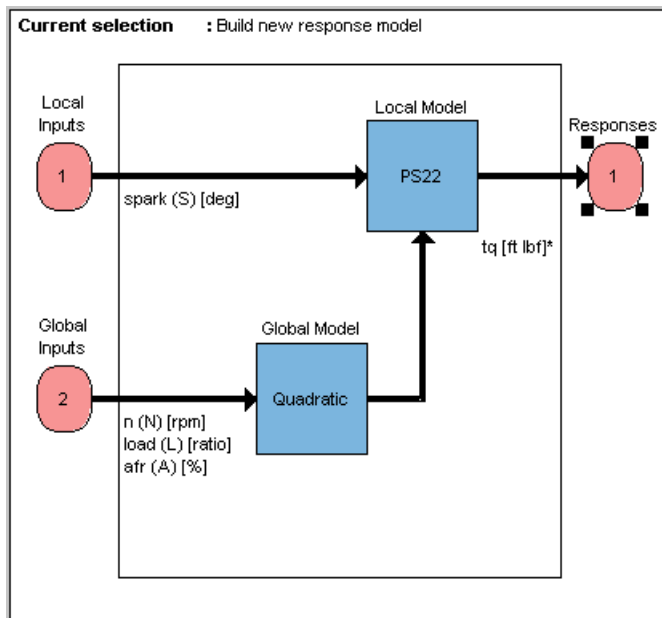
About Two Stage Models

Following is an explanation of how two-stage models are constructed and how they differ from one-stage models.

This tutorial is a step-by-step guide to constructing a single two-stage model for modeling engine brake torque as a function of spark, engine speed, load, and air/fuel ratio. One-stage modeling fits a model to all the data in one process, without accounting for the structure of the data. When data has an obvious hierarchical structure (as here), two-stage modeling is better suited to the task.

The usual way for collecting brake torque data is to fix engine speed, load, and air/fuel ratio within each test and sweep the spark angle across a range of angles. For this experimental setup, there are two sources of variation. The first source is variation within tests when the spark angle is changed. The second source of variation is between

tests when the engine speed, load, and air/fuel ratio are changed. The variation within a test is called local, and the variation between tests, global. Two-stage modeling estimates the local and global variation separately by fitting local and global models in two stages. A local model is fitted to each test independently. The results from all the local models are used to fit global models across all the global variables. Once the global models have been estimated, they can be used to estimate the local models' coefficients for any speed, load, and air/fuel ratio. The relationship between the local and global models is shown in the following block diagram, as you will see in the Model Browser.



For next steps, see “Open the App and Load Data” on page 9-5.

Open the App and Load Data

- 1 In MATLAB, on the **Apps** tab, in the **Math, Statistics and Optimization** group, click **MBC Model Fitting**.
- 2 If you have never used the toolbox before, the User Information dialog appears. If you want, you can fill in any or all of the fields: your name, company, department, and contact information, or you can click **Cancel**. The user information is used to tag comments and actions so that you can track changes in your files (it does not collect information for MathWorks).

Note You can edit your user information at any time by selecting **File > Preferences**.

- 3 When you finish with the User Information dialog, click **OK**.

The Model Browser window appears.

In this window, the left pane, **All Models**, shows the hierarchy of the models currently built in a tree. At the start, only one node, the project, is in the tree. As you build models, they appear as child nodes of the project. The right panes change, depending on the tree node selected. You navigate to different views by selecting different nodes in the model tree.

Load the example data file `holliday.xls`:

- 1 From the startup project node view, in the **Common Tasks** pane, click **Import data**.
- 2 In the Select File to Import dialog box, browse to the file `holliday.xls` in the `mbctraining` directory. Click **Open** or double-click the file.

The Data Editor opens.

- 3 View plots of the data in the Data Editor by selecting variables and tests in the lists on the left side. Have a look through the data to get an idea of the shape of curve formed by plotting torque against spark.

Use the editor to prepare your data before model fitting. For more details on using the Data Editor, see “Data Manipulation for Modeling”.

- 4 Close the Data Editor to accept the data and return to the Model Browser. Notice that the new data set appears in the **Data Sets** pane.

This data is from Holliday, T., “The Design and Analysis of Engine Mapping Experiments: A Two-Stage Approach,” Ph.D. thesis, University of Birmingham, 1995.

For next steps, see “Set Up the Model” on page 9-7.

Set Up the Model

In this section...

“Specifying Model Inputs” on page 9-7

“Setting Up the Response Model” on page 9-9

Specifying Model Inputs

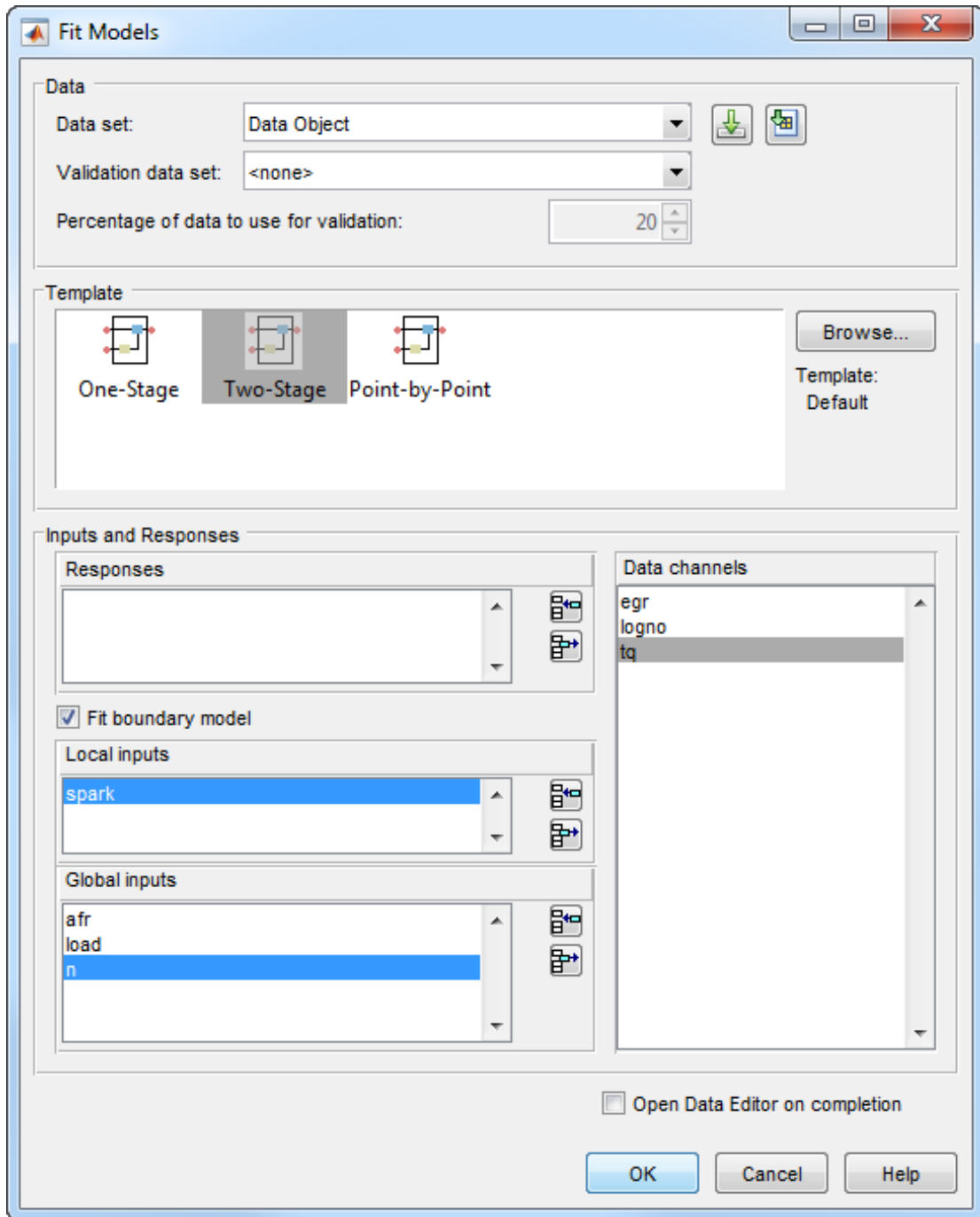
You can use the imported data to create a statistical model of an automobile engine that predicts the torque generated by the engine as a function of spark angle and other variables.

- 1 In the Model Browser project node view, in the **Common Tasks** pane, click **Fit models**.
- 2 In the Fit Models dialog box, observe that the **Data Object** you imported is selected in the **Data set** list.
- 3 Click the **Two-Stage** test plan icon in the **Template** pane.
- 4 In the **Inputs and Responses** pane, select data channels to use for the responses you want to model.

The model you are building is intended to predict the torque generated by an engine as a function of spark angle at a specified operating point defined by the engine's speed, air/fuel ratio, and load. The input to the local model is therefore the spark angle, and the response is torque.

The inputs to the global model are the variables that determine the operating point of the system being modeled. In this example, the operating point of the engine is determined by the engine's speed in revolutions per minute (rpm - often called N), load (L), and air/fuel ratio (afr).

- a Select **spark** in the **Data channels** list and click the button to add it to the **Local inputs** list.
- b Select **n**, **load** and **afr** in the **Data channels** list and click the button to add them to the **Global inputs** list.



- 5 Leave the responses empty, and click **OK**. You will set up the response later.

The default name of the new test plan, **Two-Stage**, appears in the Model Browser tree, in the **All Models** pane.

Setting Up the Response Model

To achieve the best fit to torque/spark sweeps, you need to change the local model type from the default. The type of a local model is the shape of curve used to fit the test data, for example, quadratic, cubic, or polynomial spline curves. In this example, you use polynomial spline curves to fit the test data. A spline is a curve made up of pieces of polynomial, joined smoothly together. The points of the joins are called knots. In this case, there is only one knot. These polynomial spline curves are very useful for torque/spark models, where different curvature is required above and below the maximum.

To change from the default models and specify polynomial spline as the local model type,

- 1 In the Model Browser, select the test plan node **Two-Stage**, and in the **Common Tasks** pane, click **Fit models**. A dialog box asks if you want to change all the test plan models. Click **Yes**.
- 2 In the Fit Models Wizard, click **Next** to continue using the currently selected data.
- 3 The next screen shows the model inputs you already selected. Click **Next**.
- 4 To choose the response, on the Response Models screen, select **tq** and click **Add**.
- 5 Edit the **Local model** type by clicking **Set Up**.

The Local Model Setup dialog box appears.

- a Select **Polynomial Spline** from the **Local Model Class** list.
 - b Edit the **Spline Order Below knot** to 2, and leave **Above knot** set to 2.
 - c Click **OK** to dismiss the dialog box.
- 6 Select **Maximum** under **Datum**. Only certain model types with a clearly defined maximum or minimum can support datum models. See “Add Response Models and Datum Models”.
 - 7 Click **Finish**.

The Model Browser calculates local and global models using the test plan models you just set up.

Notice that the new name of the local model class, PS (for polynomial spline) 2, 2 (for spline order above and below knot) now appears on a new node in the tree in the **All Models** pane, called PS22.

For next steps, see “Verify the Model” on page 9-11.

Verify the Model

In this section...

- “Verifying the Local Model” on page 9-11
- “Verifying the Global Model” on page 9-12
- “Creating the Two-Stage Model” on page 9-13
- “Comparing the Local Model and the Two-Stage Model” on page 9-14
- “Response Node” on page 9-14

Verifying the Local Model

The first step is to check that the local models agree well with the data:

- 1 If necessary, select PS22 (the local node) on the Model Browser tree.

The **Local Model** view appears, displaying the local model fitting the torque/spark data for the first test and diagnostic statistics that describe the fit. The display is flexible in that you can drag, open, and close the divider bars separating the regions of the screen to adjust the view.

- 2 View local model plots and statistics. The Sweep Plot shows the data being fitted by the model (blue dots) and the model itself (line). The red spot shows the position of the polynomial spline knot, at the datum (maximum) point.
- 3 Look for problem tests with the RMSE Plots. The plot shows the standard errors of all the tests, both overall and by response feature. Navigate to a test of interest by double-clicking a point in the plot to select the test in the other plots in the local model view.
- 4 In the Diagnostic Statistics plot pane, click the **Y-axis factor** pop-up menu and select **Studentized residuals**.
- 5 Scroll through local models test by test using the **Test** arrows at the top left, or by using the **Select Test** button.
- 6 Select Test 588. You see a data point outlined in red. This point has automatically been flagged as an outlier.
- 7 Right-click the plot and select **Remove Outliers**. Observe that the model is refitted without the outlier.

Both plots have right-click pop-up menus offering various options such as removing and restoring outliers and confidence intervals. Clicking any data point marks it in red as an outlier.

You can use the **Test Notes** pane to record information on particular tests. Each test has its own notes pane. The test numbers of data points with notes recorded against them are colored in the global model plots, and you can choose the color using the **Test Number Color** button in the **Test Notes** pane. Quickly locate tests with notes by clicking **Select Test**.

To learn more about the plots and statistics, see “Assess Local Models”.

Verifying the Global Model

The next step is to check through the global models to see how well they fit the data:

- 1 Expand the PS22 local node on the Model Browser tree by clicking the plus sign (+) to the left of the icon. Under this node are four response features of the local model. Each of these is a feature of the local model of the response, which is torque.
- 2 Select the first of the global models, `knot`.

You see a dialog box asking if you want to update fits, because you removed an outlier at the local level. Click **Yes**.

The **Response Feature** view appears, showing the fit of the global model to the data for `knot`. Fitting the local model is the process of finding values for these coefficients or *response features*. The local models produce a value of `knot` for each test. These values are the data for the global model for `knot`. The data for each response feature come from the fit of the local model to each test.

Use the plots to assess model fits.

- 3 Select the response feature `Bhigh_2`. One outlier is marked. Points with an absolute studentized residual value of more than 3 are automatically suggested as outliers (but included in the model unless you take action). You can use the right-click menu to remove suggested outliers (or any others you select) in the same way as from the Local Model plots. Leave this one. If you zoom in on the plot (**Shift**-click-drag or middle-click-drag) you can see the value of the studentized residual of this point more clearly. Double-click to return to the previous view.

Note Never remove outliers as a matter of course. However, this tutorial shows you how the toolbox helps you to do this when required. The default outlier selection criterion is a studentized residual greater than 3, to bring your attention to possible outliers, but you should never remove data without good reasons. Remove enough points and the model will simply interpolate the data and become useless for prediction. You can customize the criteria for outlier selection.

- 4 Select the other response features in turn: `max` and `Blow_2`. You will see that `Blow_2` has a suggested outlier with a very large studentized residual; it is a good distance away from all the other data points for this response feature. All the other points are so clustered that removing this one could greatly improve the fit of the model to the remaining points, so remove it.

To learn more about the plots and statistics, see “Assess One-Stage Models”.

Creating the Two-Stage Model

Recall how two-stage models are constructed: two-stage modeling partitions the variation separately between tests and within tests, by fitting local and global models separately. A model is fitted to each test independently (local models). These local models are used to generate global models that are fitted across all tests.

For each sweep (test) of spark against torque, you fit a local model. The local model in this case is a spline curve, which has the fitted response features of `knot`, `max`, `Bhigh_2`, and `Blow_2`. The result of fitting a local model is a value for `knot` (and the other coefficients) for each test. The global model for `knot` is fitted to these values (that is, the `knot` global model fits `knot` as a function of the global variables). The values of `knot` from the global model (along with the other global models) are then used to construct the two-stage model

The global models are used to reconstruct a model for the local response (in this case, torque) that spans all input factors. This is the two-stage model across the whole global space, derived from the global models.

After you are satisfied with the fit of the local and global models, it is time to construct a two-stage model from them.



- 1 Return to the Local Model view by clicking the local node `PS22` in the Model Browser tree.

- 2 To create a two-stage model, click **Create Two-Stage** in the Common Tasks pane.

Comparing the Local Model and the Two-Stage Model

- 1 Now the plots in the **Local Model** view show two lines fitted to the test data. Scroll through the tests using the left/right arrows or the **Select Test** button at the top left. The plots now show the fit of the two-stage model for each test (green circles and line), compared with the fit of the local model (blue line) and the data (blue dots). Zoom in on points of interest by **Shift**-click-dragging or middle-click-dragging. Double-click to return the plot to the original size.

Compare how close the two-stage model fit is to both the data and the local fit for each test.

- 2 Notice that the local model icon has changed (from the local  icon showing a house, to a two-stage icon  showing a house and a globe) to indicate that a two-stage model has been calculated.

Response Node

Click the Response node (tq) in the Model Browser tree.

Now at the Response node in the Model Browser tree (tq), which was previously blank, you see plots showing you the fit of the two-stage model to the data. You can scroll through the tests, using the arrows at top left, to view the two-stage model against the data for groups of tests.

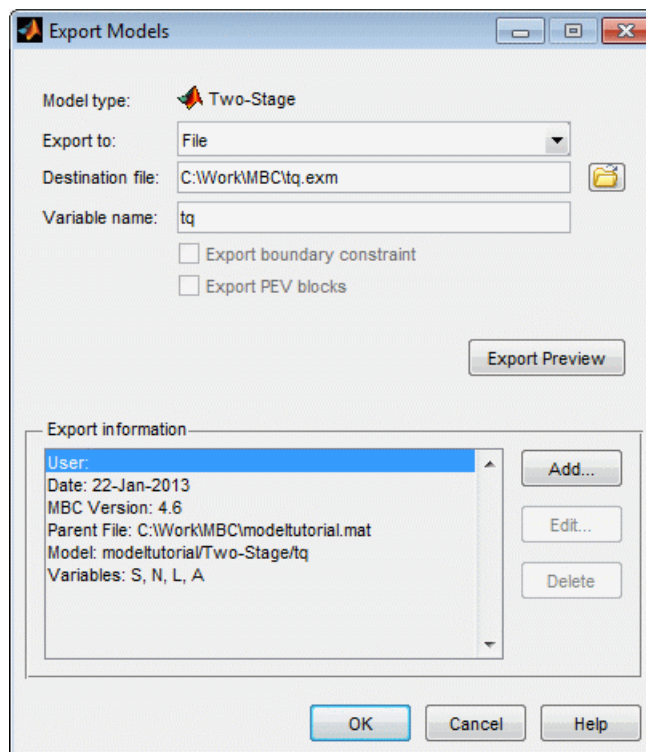
You have now completed setting up and verifying a two-stage model.

For next steps, see “Export the Model” on page 9-15 or “Create Multiple Models to Compare” on page 9-17.

Export the Model

All models created in the Model Browser are exported using the **File** menu. A model can be exported to the MATLAB workspace, to a file, to CAGE, or to a Simulink model.

- 1 Click the `tq` node in the model tree.
- 2 Choose **File > Export Models**. The Export Model dialog box appears.
- 3 Choose **File** from the **Export to** pop-up menu. This saves the work as a file for use within the Model-Based Calibration Toolbox product, for instance, to create calibrations in the CAGE Browser.
- 4 In the **Export to** edit box, select the destination of the file. You can do this by typing directly in the edit box, or using the Browse button if you want to locate a directory or use an existing file.



- 5 Click **OK** to export the models to file.

To import models into CAGE to create calibrations, use the CAGE Import Tool instead for more flexibility.

For next steps, see “Create Multiple Models to Compare” on page 9-17.

Create Multiple Models to Compare

In this section...

“Methods For Creating More Models” on page 9-17

“Creating New Local Models” on page 9-17

“Adding New Response Features” on page 9-20

“Comparing Models” on page 9-21

“Creating New Global Models” on page 9-22

Methods For Creating More Models

After you have fitted and examined a single model, you normally want to create more models to search for the best fit. You can create individual new models or use the **Create Alternatives** common task to create a selection of models at once, or create a template to save a variety of model settings for reuse.

To use the Model Template dialog box to quickly create a selection of different child nodes to compare, click **Create Alternatives** in the Common Tasks pane. The following exercises show you examples of these processes.

Note: You need to complete the previous tutorial sections to have a complete two-stage model as a starting point. See “Empirical Engine Modeling” on page 9-2.

Creating New Local Models

To follow these examples, you need to create the initial models by following the previous sections in “Empirical Engine Modeling” on page 9-2.

- 1 As an example, select the **tq** response node and click **New Local Model** in the Common Tasks pane.

The Local Model Setup dialog appears.

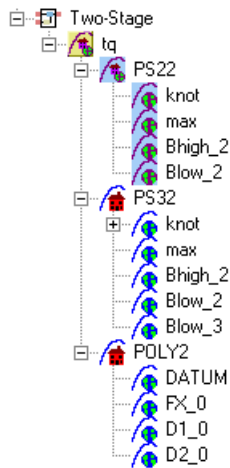
- 2 Select a **Polynomial Spline**, and edit the spline order to **3** below the knot and **2** above. Click **OK**.

A new set of local models (and associated response feature models) is calculated.

- 3 Return to the parent **tq** response node , and click **New Local Model** again, in the Common Tasks pane.
- 4 Select a **Polynomial** with an order of 2 in the Local Model Setup dialog. Click **OK**.

A new set of local models and response feature models is calculated.

Now you have three alternative local models to compare: two polynomial splines (order 3,2 and order 2,2) and a polynomial (order 2), as shown.



You can select the alternative local models in turn and compare their statistics. For an example, follow these steps:

- 1 Select the new local model node **PS32**.
- 2 Select test 587 in the **Test** edit box.
- 3 In the **Local statistics** pane, observe the value of RMSE (root mean squared error) for the current (i^{th}) test.

The RMSE value is our basic measure of how closely a model fits some data, which measures the average mismatch between each data point and the model. This is why you should look at the RMSE values as your first tool to inspect the quality of the fit — high RMSE values can indicate problems.

- 4 Now select the local model node **POLY2** and see how the value of RMSE changes.

Observe that the shape of the torque/spark sweep for this test is better suited to a polynomial spline model than a polynomial model. The curve is not symmetrical because curvature differs above and below the maximum (marked by the red cross at the datum). This explains why the value of RMSE is much lower for PS32 (the polynomial spline) than for the POLY2 (polynomial) for this test. The polynomial spline is a better fit for the current test.

- 5 Look through some other tests and compare the values of RMSE for the different local models. To choose the most suitable local model you must decide which fits the majority of tests better, as there are likely to be differences among best fit for different tests.
- 6 To help you quickly identify which local models have the highest RMSE, indicating problems with the model fit, check the RMSE Plots.
 - a Use the plot to help you identify problem tests. Use the drop-down menus to change the display. For example, select `s_knot` to investigate the error values for knot (MBT), or `RMSE` to look at overall error.
 - b You can navigate to a test of interest from the RMSE Plots by double-clicking a point in the plot to select the test in the other plots.
- 7 Look at the value of `Local RMSE` reported in the **Pooled Statistics** pane on the right (this is pooled between all tests). Now switch between the POLY2 and the PS32 local models again and observe how this value changes.
- 8 You can compare these values directly by selecting the parent `tq` response node, when the Local RMSE is reported for each child local model in the list at the bottom.

When all child models have a two-stage model calculated, you can also compare two-stage values of RMSE here. Remember, you can see statistics to compare the list of child models of the response node in this bottom list pane.

When comparing models, look for lower RMSE values to indicate better fits. However, remember that a model that interpolates between all the points can have an RMSE of zero but be useless for predicting between points. Always use the graphical displays to visually examine model fits and beware of “overfitting” — chasing points at the expense of prediction quality. You will return to the problem of overfitting in a later section when you have two-stage models to compare.

Adding New Response Features

Recall that two-stage models are made up of local models and global models. The global models are fitted to the response features of the local models. The response features available are specific to the type of local model. You can add different response features to see which combination of response features makes the best two-stage model as follows:

1 Select the local model node **PS32**.

2 Select **File > New Response Feature**.

A dialog appears with a list of available response features.

3 Select $f(x+\text{datum})$ from the list and enter **-10** in the **Value** edit box. Click **OK**.

A new response feature called **FX_less10** is added under the **PS32** local model. Recall that the datum marks the maximum, in this case maximum torque. The spark angle at maximum torque is referred to as maximum brake torque (MBT). You have defined this response feature ($f(x+\text{datum})$) to measure the value of the model (torque) at $(-10 + \text{MBT})$ for each test. It can be useful to use a response feature like this to track a value such as maximum brake torque (MBT) minus 10 degrees of spark angle. This response feature is not an abstract property of a curve, so engineering knowledge can then be applied to increase confidence in the models.

4 Select the local node **PS32**, and click **Create Two-Stage** in the Common Tasks pane. The Model Selection window opens, because you now need to choose 5 of the 6 response features to form the two-stage model.

In the Model Selection window, observe four possible two-stage models in the Model List. This is because you added a sixth response feature. Only five (which must include **knot**) are required for the two-stage model, so you can see the combinations available and compare them. Note that not all combinations of five response features can completely describe the shape of the curve for the two-stage model, so only the possible alternatives are shown.

5 Close the Model Selection window and click **Yes** to accept one of the models as best.

Notice that the response features chosen to calculate the two-stage model are highlighted in blue, and the unused response feature is not highlighted.

6 Select the **tq** response node to see a comparison of the statistics of both two-stage models (your original **PS22** and the new **PS32**).

Remember that the **POLY2** local model has no two-stage model yet; no two-stage statistics are reported for **POLY2** in the bottom list pane. You cannot fully compare

the two-stage models until every local model in the test plan has a two-stage model calculated.

- 7 To calculate the two-stage model for POLY2, in the Common Tasks pane, click **Create Two-Stage**.

Comparing Models

- 1 Now you have three two-stage models. Select the **tq** response node and look at the statistics, particularly Local RMSE and Two-Stage RMSE reported in the list of child models at the bottom.

- Look for lower RMSE values to indicate better fits.
- Look for lower PRESS RMSE values to indicate better fits without overfitting. PRESS RMSE is a measure of the predictive power of your models.

It is useful to compare PRESS RMSE with RMSE as this may indicate problems with overfitting. RMSE is minimized when the model gets close to each data point; “chasing” the data will therefore improve RMSE. However, chasing the data can sometimes lead to strong oscillations in the model between the data points; this behavior can give good values of RMSE but is not representative of the data and will not give reliable prediction values where you do not already have data. The PRESS RMSE statistic guards against this by testing how well the current model would predict each of the points in the data set (in turn) if they were not included in the regression. To get a small PRESS RMSE usually indicates that the model is not overly sensitive to any single data point.

If the value of PRESS RMSE is much bigger than the RMSE, then you are overfitting - the model is unnecessarily complex. For a fuller description of the meaning of overfitting, and how RMSE and PRESS can help you select good models, see “Choose the Best Model”. “Choose the Best Model”. As a rule of thumb, if you have about 100 data points, you should aim for a PRESS RMSE no more than 5% larger than the RMSE (remember here you have only 27 tests).

- Look for lower T^2 values. A large T^2 value indicates that there is a problem with the response feature models.
- Look for large negative log likelihood values to indicate better fits.

See “Pooled Statistics” for more on T^2 and log likelihood.

- 2 To compare all three two-stage models simultaneously, select **Model > Selection Window**. Here you can see the same statistics to compare the models in the bottom list, but you can also make use of a variety of views to look for the best fit:
 - You can plot the models simultaneously on the Tests, Residuals and Cross Section views (**Shift-** or **Ctrl-**click to select models in the list)
 - You can view each model in the Response Surface view as a surface; movie, contour or multiline plot, and as a table
- 3 You can select a model and click **Assign Best** in the Model Selection window, or double-click a model to assign it as best.
- 4 When you close the Model Selection window and return to the Model Browser, the model you selected as best is copied to the parent response node, **tq**.

Creating New Global Models

In this example, you have not yet searched for the best global model types. You would normally do this before creating and comparing two-stage models. For the purpose of this tutorial, you have already created two-stage models and used RMSE to help you identify better models. The principle is the same at each level in the model tree: add new child models and choose the best. You can create any number of child nodes to search for the best global model fit for each response feature in your tree.

- 1 Select the local node **POLY2**.
- 2 To create a selection of alternatives for each response feature node, in the Common Tasks pane, click **Build Global Models**.
- 3 In the Model Template dialog box, click **New**, then click **OK**.
- 4 Observe the default list of a variety of model types, then click **OK**. It is worth trying the default model settings for a quick exploration of the trends in the data.
- 5 In the Model Selection dialog box, leave the default selection criterion for automatically choosing the best child node, and click **OK**.

The toolbox builds the models and selects the best using your selection criteria.

Note: The toolbox automatically builds models in parallel if you have Parallel Computing Toolbox.

- 6 Assess all the fits in the Alternative Models list in case you want to choose an alternative as a better fit.

- 7 Notice that the child node model assigned as best is highlighted in blue in the Alternative Models list and the model tree. The local node has changed from the two-stage icon back to the local model icon (a red house). This is because you have changed the response feature models, and so you need to recalculate the two-stage model using the new global models for the response features.

When you have chosen best global models for all your response features, you need to recalculate the two-stage model.

- 8 When you have chosen a best model among alternatives, it can be useful to clean up the rejected models by selecting **Delete Alternatives** in the Common Tasks pane. You can also select **File > Clean Up Tree**. This deletes all rejected child models where best models have been chosen; only the child nodes selected as best remain.

You can use the Model Template dialog box to create and save templates of model types you often want to build. Creating a template containing a list of all the models you want is a very efficient way to quickly build a selection of alternative model child nodes for many global models. Use these techniques to find models well suited to the data for each of your global models. See “Create Alternative Models to Compare”.

Related Examples

- “Fit a One-Stage Model”
- “Gasoline Engine Calibration”
- “Multi-Injection Diesel Calibration”

Design of Experiment

This section discusses the following topics:

- “Design of Experiment” on page 10-2
- “Set Up a Model and Create a Design” on page 10-5
- “Create a Constrained Space-Filling Design” on page 10-7
- “Save Designs” on page 10-12
- “Create Optimal Designs” on page 10-13
- “View Design Displays” on page 10-19
- “Use the Prediction Error Variance Viewer” on page 10-21
- “Create Classical Designs” on page 10-27
- “Use the Design Evaluation Tool” on page 10-33

Design of Experiment

In this section...

“Why Use Design of Experiment?” on page 10-2

“Design Styles” on page 10-3

“Create Examples Using the Design Editor” on page 10-3

Why Use Design of Experiment?

With today's ever-increasing complexity of models, design of experiment has become an essential part of the modeling process. The Design Editor within the Model-Based Calibration Toolbox product is crucial for the efficient collection of engine data. Dyno-cell time is expensive, and the savings in time and money can be considerable when a careful experimental design takes only the most useful data. Dramatically reducing test time is growing more and more important as the number of controllable variables in more complex engines is growing. With increasing engine complexity, the test time increases exponentially.

The traditional method of collecting large quantities of data by holding each factor constant in turn until all possibilities have been tested is an approach that quickly becomes impossible as the number of factors increases. A full factorial design (that is, testing for torque at every combination of speed, load, air/fuel ratio, and exhaust gas recirculation on a direct injection gasoline engine with stratified combustion capability) is not feasible for newer engines. Simple calculation estimates that, for recently developed engines, to calibrate in the traditional way would take 99 years!

With a five-factor experiment including a multiknot spline dimension and 20 levels in each factor, the number of points in a full factorial design quickly becomes thousands, making the experiment prohibitively expensive to run. The Design Editor solves this problem by choosing a set of experimental points that allow estimation of the model with the maximum confidence using just a fraction of the number of experimental runs; for the preceding example just 100 optimally chosen runs is more than enough to fit the model. Obviously, this approach can be advantageous for any complex experimental design, not just engine research.

The Design Editor offers a systematic, rigorous approach to the data collection stage. When you plan a sequence of tests to be run on an example engine, you can base your design on engineering expertise and existing physical and analytical models. During

testing, you can compare your design with the latest data and optimize the remaining tests to get maximum benefit.

The Design Editor provides prebuilt standard designs to allow a user with a minimal knowledge of the subject to quickly create experiments. You can apply engineering knowledge to define variable ranges and apply constraints to exclude impractical points. You can increase modeling sophistication by altering optimality criteria, forcing or removing specific design points, and optimally augmenting existing designs with additional points.

Design Styles

The Design Editor provides the interface for building experimental designs. You can make three different styles of design: classical, space-filling, and optimal.

Space-filling designs are better when there is low system knowledge. In cases where you are not sure what type of model is appropriate, and the constraints are uncertain, space-filling designs collect data in such a way as to maximize coverage of the factors' ranges as quickly as possible. See “Create a Constrained Space-Filling Design” on page 10-7.

Optimal designs are best for cases with high system knowledge, where previous studies have given confidence on the best type of model to be fitted, and the constraints of the system are well understood. See “Create Optimal Designs” on page 10-13.

Classical designs (including full factorial) are very well researched and are suitable for simple regions (hypercube or sphere). Engines have complex constraints and models (high-order polynomials and splines). See “Create Classical Designs” on page 10-27.

You can augment any design by adding points. Working in this way allows new experiments to enhance the original, rather than simply being a second attempt to gain the necessary knowledge.

Create Examples Using the Design Editor

Follow the steps in the following sections in order. The instructions guide you through constructing space-filling, optimal, and classical designs; how to apply constraints to designs, and how to compare designs using the prediction error variance (PEV) viewer and Design Evaluation tool.

- 1 Choose a model to design an experiment for, enter the Design Editor, and construct an space-filling design. Then construct and apply two different constraints to this

design and view the results. Often you would design constraints before constructing a design, but for the purposes of this tutorial you make constraints last so you can view the effects on your design.

See

- “Set Up a Model and Create a Design” on page 10-5
 - “Create a Constrained Space-Filling Design” on page 10-7
- 2** Create an optimal design. See “Create Optimal Designs” on page 10-13.
 - 3** After you create designs, use the displays and tools to examine the properties of the design, save the design, and make changes.
 - “View Design Displays” on page 10-19
 - “Save Designs” on page 10-12
 - “Use the Prediction Error Variance Viewer” on page 10-21
 - 4** Create a classical design, and use the Prediction Error Variance Viewer to compare it with the optimal design. You can also use the Design Evaluation tool to view all details of any design.

See

- “Create Classical Designs” on page 10-27
- “Use the Design Evaluation Tool” on page 10-33

For more details on functionality in the Design Editor, see “Design of Experiments”.

Related Examples

- “Set Up a Model and Create a Design” on page 10-5
- “Create a Constrained Space-Filling Design” on page 10-7
- “Apply Constraints” on page 10-7
- “Create Optimal Designs” on page 10-13
- “Create Classical Designs” on page 10-27
- “Save Designs” on page 10-12

More About

- “Design of Experiments”

Set Up a Model and Create a Design

In this section...

“Set Up Model Inputs” on page 10-5

“Open the Design Editor” on page 10-5


“Create a New Design” on page 10-6

Set Up Model Inputs

You must first specify model inputs for which to design an experiment.

- 1 Start the Model Browser part of the toolbox by typing `mbcmodel` at the MATLAB command line.
- 2 From the startup project node view, in the **Common Tasks** pane, click **Design experiment**.

The **New Test Plan** dialog box appears.

- 3 Click the **Two-Stage** test plan icon  in the Template pane. A two-stage model fits a model to data with a hierarchical structure.
- 4 There is only one input to the global model by default. Click the up arrow button to increase the **Number of factors** setting to 3.
- 5 Change the symbols of the three input factors to N, L, and A. This matches the global factors modeled in the tutorial “Empirical Engine Modeling” on page 9-2: speed (n), load (L), and air/fuel ratio (A).
- 6 Click **OK** to dismiss the dialog box.

The Model Browser displays the test plan diagram with your specified model inputs.

Open the Design Editor


In the test plan view, in the **Common Tasks** pane, click **Design experiment**.

The Design Editor window appears.

Alternatively, to open the Design Editor, you can also use either of the following methods:

- Right-click the global model in the diagram and choose **Design Experiment**, as shown.
- You can also access the Design Editor by selecting the menu item **TestPlan > Design Experiment**.

Create a New Design

- 1 Click the  button in the toolbar or select **File > New**. A new node called appears.
- 2 The new node is automatically selected. An empty Design Table appears (or any view you last used in the Design Editor) because you have not yet chosen a design. For this example you create an optimal design for the default global model.

You can change the model for which you are designing an experiment from within the Design Editor window by selecting **Edit > Model**.

- 3 Rename the new node **Space Fill** (you can edit the names by clicking again on a node when it is already selected, or by pressing **F2**, as when selecting to rename in Windows Explorer).

For next steps, see “Create a Constrained Space-Filling Design” on page 10-7.

Create a Constrained Space-Filling Design

Space-filling designs should be used when there is little or no information about the underlying effects of factors on responses. For example, they are most useful when you are faced with a new type of engine, with little knowledge of the operating envelope. These designs do not assume a particular model form. The aim is to spread the points as evenly as possible around the operating space. These designs literally fill out the n -dimensional space with points that are in some way regularly spaced. These designs can be especially useful with nonparametric models such as radial basis functions (a type of neural network).

- 1 In the Design Editor, with the new design selected (that you created in “Open the Design Editor” on page 10-5), select **Design > Space Filling > Design Browser**, or click the Space Filling Design button on the toolbar.

The Space Filling Design Browser appears.

- 1 Leave the default Sobol Sequence in the **Design type** list, and the default **Number of points**.
- 2 Use the Preview tabs to view 2-D and 3-D previews.
- 3 Click **OK** to calculate the space-filling design and return to the main Design Editor.

Apply Constraints

In many cases, designs might not coincide with the operating region of the system to be tested. For example, a conventional stoichiometric AFR automobile engine normally does not operate with high exhaust gas recirculation (EGR) in a region of low speed (n) and low load (l). You cannot run 15% EGR at 800 RPM idle with a homogeneous combustion process. There is no point selecting design points in impractical regions, so you can constrain the candidate set for test point generation. Only optimal designs have candidate sets of points; classical designs have set points, and space-filling designs distribute points between the coded values of (1, -1).

You would usually set up constraints *before* making designs. Applying constraints to classical and space-filling designs simply removes points outside the constraint. Constraining the candidate set for optimal designs ensures that design points are optimally chosen within the area of interest only.

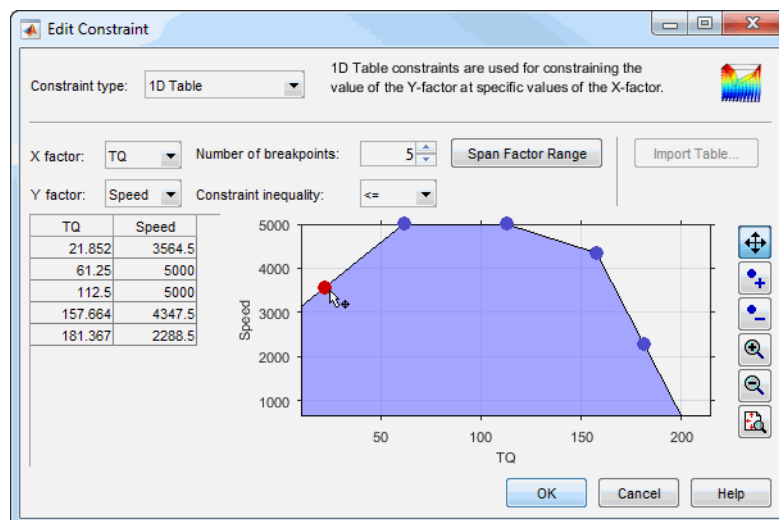
Designs can have any number of geometric constraints placed upon them. Each constraint can be one of four types: an ellipsoid, a hyperplane, a 1-D lookup table, or a 2-D lookup table.

To add a constraint to your currently selected design:

- 1 Select **Edit > Constraints** from the Design Editor menus.
- 2 The Constraints Manager dialog appears. Click **Add**.

The Constraint Editor dialog with available constraints appears. Leave the default **1D Table** in the **Constraint type** list.

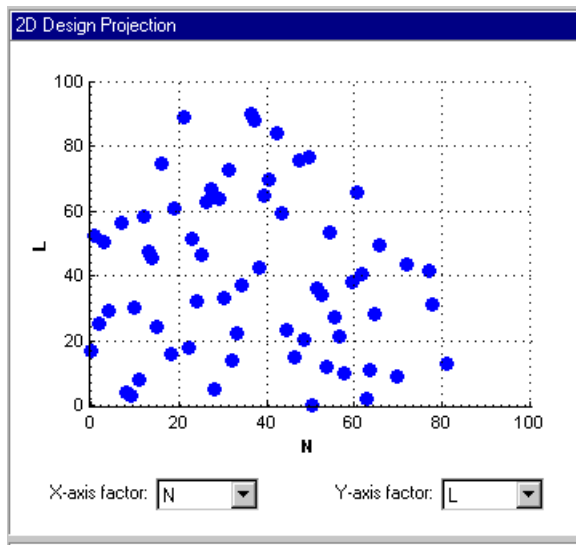
- 3 You can select the appropriate factors to use. For this example, choose speed (N) and air/fuel ratio (A) for the X and Y factors.
- 4 Move the large dots (click and drag them) to define a boundary. The Constraint Editor should look something like the following.



- 5 Click **OK**.

Your new constraint appears in the Constraint Manager list box. Click **OK** to return to the Design Editor. A dialog appears because there are points in the design that fall outside your newly constrained candidate set.

- 6 Click **Continue** to remove the points outside the constraint. Note that fixed points are not deleted by this process.

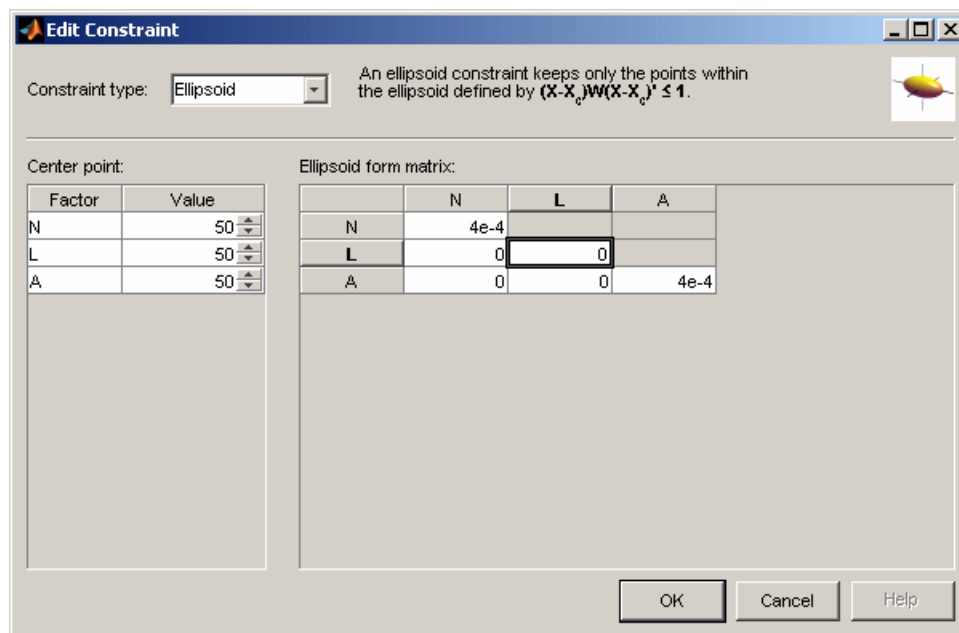


Plot the 2-D projection of the hypercube, and observe the effects of the new constraint on the shape of the design, as shown in the preceding example.

- 7 Right-click the display pane to reach the context menu, and select **Current View > 3D Constraints**.

These views are intended to give some idea of the region of space that is currently available within the constraint boundaries.

- 8 Return to the Constraint Editor, choose **Edit > Constraint**, and click **Add** in the Constraint Manager.
- 9 Add an ellipsoid constraint. Choose **Ellipsoid** from the drop-down menu of constraint types.



Enter 0 as the value for the **L** diagonal in the table, as shown. This will leave **L** unconstrained (a cylinder). The default ellipsoid constraint is a sphere. To constrain a factor, if you want a radius of r in a factor, enter $1/(r^2)$. For this example, leave the other values at the defaults. Click **OK** to apply the constraint.

- 10 Click **OK**, click **OK** again in the Constraint Manager, and click **Continue** to remove design points outside the new candidate set (or **Replace** if you are constraining an optimal design). Examine the new constraint 3-D constraints plot.

Both constraints are applied to this design.

For next steps, see “Save Designs” on page 10-12.

To try other design styles, see “Create Optimal Designs” on page 10-13 and “Create Classical Designs” on page 10-27.

Related Examples

- “Design of Experiment” on page 10-2
- “Save Designs” on page 10-12
- “Create Optimal Designs” on page 10-13
- “View Design Displays” on page 10-19
- “Create Classical Designs” on page 10-27

More About

- “Design of Experiments”

Save Designs

To save your designs with your project and close the Design Editor, select **File > Save and Close** or the toolbar button.

To export your design to a file:

- 1 Choose **File > Export Design**. The selected design *only* is exported.

Export to options:

- **Comma separated format file** exports the matrix of design points to a CSV (comma-separated values) file. You can include factor symbols and/or convert to coded values by selecting the check boxes.
 - **Design Editor file** generates a Design Editor file (.mvd).
 - **Workspace** exports the design matrix to the workspace. You can convert design points to a range of [-1, 1] by selecting the check box.
- 2 Choose a CSV file.
 - 3 Name the destination file by typing `Designtutorial` in the edit box.
 - 4 Click **OK** to save the file.

For more details on functionality in the Design Editor, see “Design of Experiments”.

Related Examples

- “Design of Experiment” on page 10-2
- “Create a Constrained Space-Filling Design” on page 10-7
- “Create Optimal Designs” on page 10-13
- “View Design Displays” on page 10-19
- “Create Classical Designs” on page 10-27

More About

- “Design of Experiments”

Create Optimal Designs

In this section...


“Introducing Optimal Designs” on page 10-13

“Start Point Tab” on page 10-14

“Candidate Set Tab” on page 10-15

“Algorithm Tab” on page 10-17

Introducing Optimal Designs

In the Design Editor, with a new design selected, create an optimal design by clicking the  button in the toolbar, or choose **Design > Optimal**.

Optimal designs are best for cases with high system knowledge, where previous studies have given confidence on the best type of model to be fitted, and the constraints of the system are well understood.

The optimal designs in the Design Editor are formed using the following process:

- An initial starting design is chosen at random from a set of defined candidate points.
- m additional points are added to the design, either optimally or at random. These points are chosen from the candidate set.
- m points are deleted from the design, either optimally or at random.
- If the resulting design is better than the original, it is kept.

This process is repeated until either (a) the maximum number of iterations is exceeded or (b) a certain number of iterations has occurred without an appreciable change in the optimality value for the design.

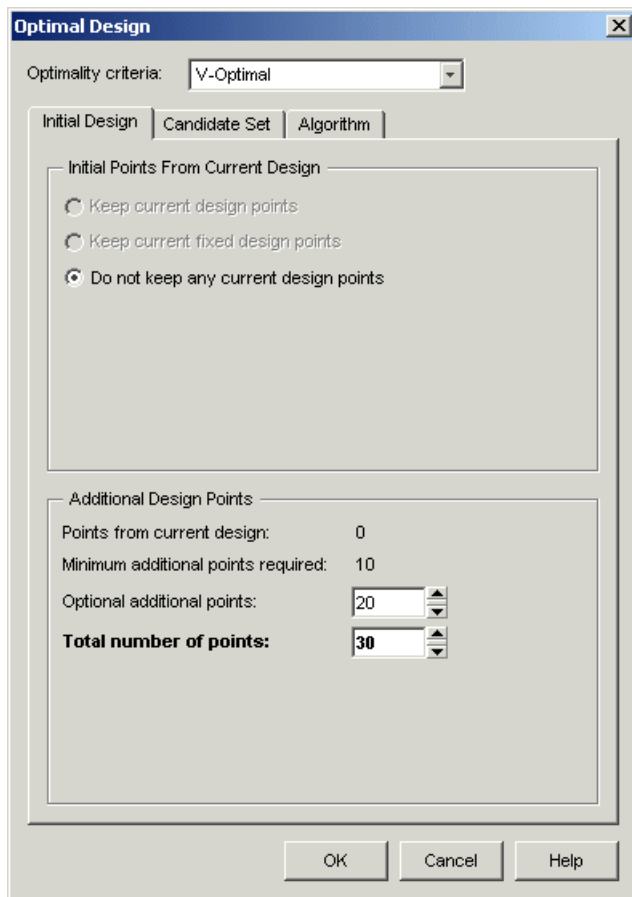
The Optimal Design dialog consists of several tabs that contain the settings for three main aspects of the design:

- Starting point and number of points in the design
- Candidate set of points from which the design points are chosen
- Options for the algorithm that is used to generate the points

Start Point Tab

The **Start Point** tab allows you to define the composition of the initial design: how many points to keep from the current design and how many extra to choose from the candidate set.

- 1 Leave the optimality criteria at the default to create a V-Optimal design.
- 2 Increase the total number of points to 30 by clicking the **Optional additional points** up/down buttons or by typing directly into the edit box. You can edit the additional points and/or the total number of points.



Candidate Set Tab

The **Candidate Set** tab allows you to set up a candidate set of potential test points. This typically ranges from a few hundred points to several hundred thousand.

- 1** Choose **Grid** for this example. Note that you could choose different schemes for different factors.
- 2** This tab also has buttons for creating plots of the candidate sets. Try them to preview the grid.
- 3** Notice that you can see 1-D, 2-D, 3-D, and 4-D displays (the fourth factor is color, but this example only uses three factors) at the same time as they appear in separate windows (see example following). Look at a display window while changing the number of levels for the different factors. See the effects of changing the number of levels on different factors, then return them all to the default of 21 levels.

Select variables in this list

Choose grid from this list

Optimality criteria: V-Optimal

Initial Design Candidate Set Algorithm

Generation algorithm: Grid

Allow replicated points in design

Options

View coded values

Equally spaced levels

Minimum N value: 0

Maximum N value: 100

Number of levels for N: 21

User-specified levels

0.5:100

Display

Display a maximum of 2500 points

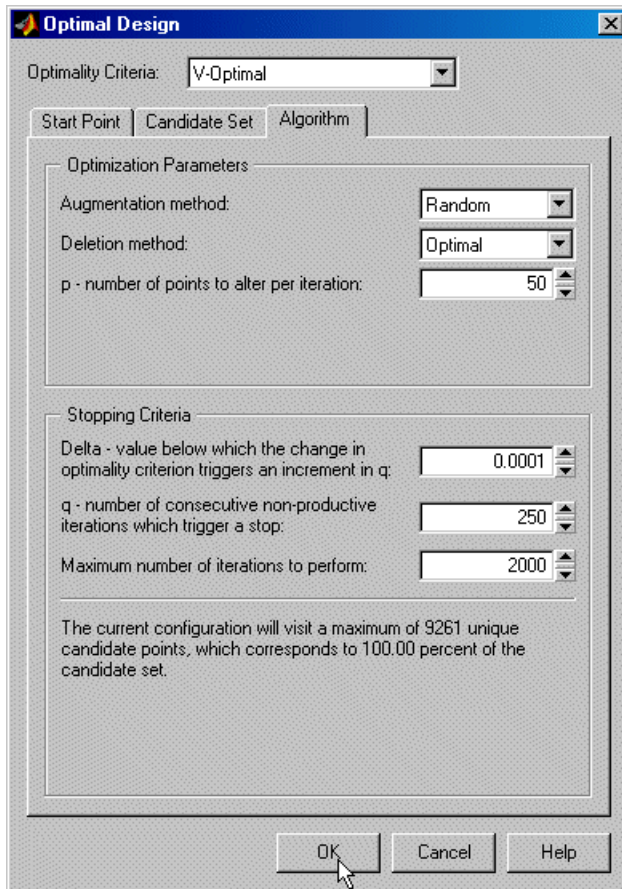
0 constraints will be applied to this candidate set.

OK Cancel Help

Open display windows with these buttons.

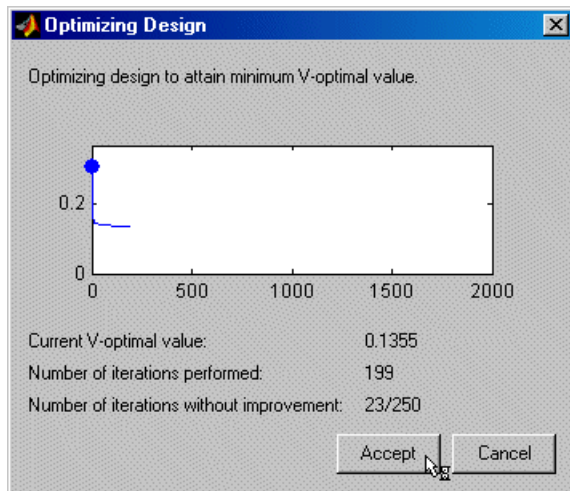
Change the number of levels of the selected variable here.

Algorithm Tab



- 1 Leave the algorithm settings at the defaults and click **OK** to start optimizing the design.

When you click the **OK** button on the Optimal Design dialog, the Optimizing Design dialog appears, containing a graph. This dialog shows the progress of the optimization and has two buttons: **Accept** and **Cancel**. **Accept** stops the optimization early and takes the current design from it. **Cancel** stops the optimization and reverts to the original design.



- 2 Click **Accept** when iterations are not producing noticeable improvements; that is, the graph becomes very flat.

For next steps, see “View Design Displays” on page 10-19, “Use the Prediction Error Variance Viewer” on page 10-21, and “Create Classical Designs” on page 10-27.

View Design Displays

When you click **Accept** button, you return .

In the Design Editor, after you create a design, it shows the **Design Table** view of the design or other views if you previously viewed designs. In the context menu, available by right-clicking on the title bar, you can change the view of the design to 1-D, 2-D, 3-D, 4-D, and pairwise design projections, 2-D and 3-D constraint views, and the table view (also under the **View** menu). This menu also allows you to split the display either horizontally or vertically so that you simultaneously have two different views on the current design. You can also use the toolbar buttons to do this. The split can be merged again. After splitting, each view has the same functionality; that is, you can continue to split views until you have as many as you want. When you click a view, its title bar becomes blue to show it is the active view.

The currently available designs are displayed on the left in a tree structure. For details, see “The Design Tree”.

Display Options

The Design Editor can display multiple design views at once, so while working on a design you can keep a table of design points open in one corner of the window, a 3-D projection of the constraints below it and a 2-D or 3-D plot of the current design points as the main plot.

The current view and options for the current view are available either through the context menu or the **View** menu on the Design Editor window.

- 1 Change the main display to 3-D Projection view.
- 2 You can rotate the projection with click-drag mouse movement. View your design in several projections (singly, or simultaneously by dividing the pane) by using the right-click context menu in the display pane.

Related Examples

- “Design of Experiment” on page 10-2
- “Create a Constrained Space-Filling Design” on page 10-7
- “Create Optimal Designs” on page 10-13
- “Create Classical Designs” on page 10-27

- “Use the Prediction Error Variance Viewer” on page 10-21
- “Use the Design Evaluation Tool” on page 10-33

More About

- “Design of Experiments”

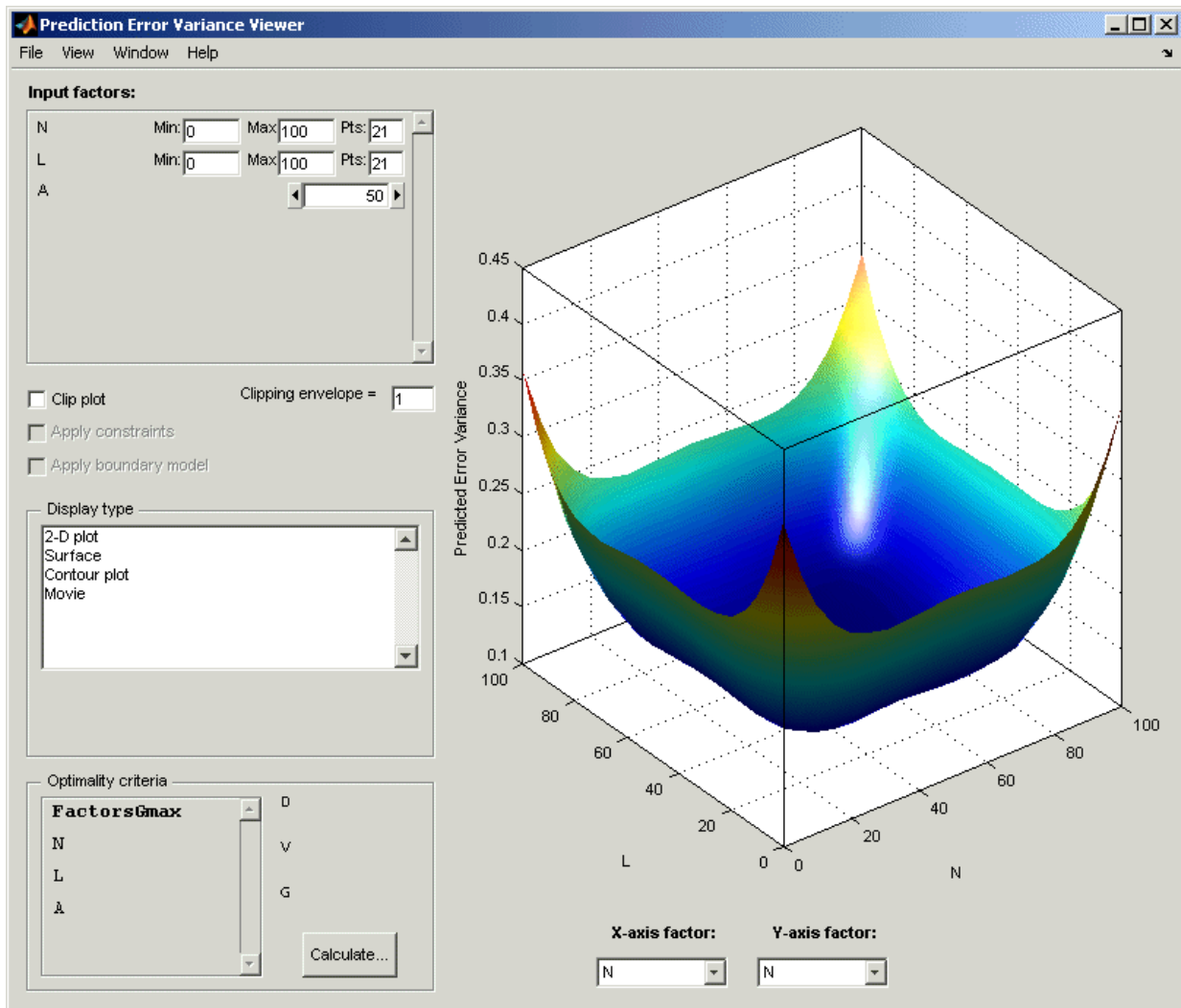
Use the Prediction Error Variance Viewer

In this section...
“Introducing the Prediction Error Variance Viewer” on page 10-21
“Add Points Optimally” on page 10-24

Introducing the Prediction Error Variance Viewer

A useful measure of the quality of a design is its prediction error variance (PEV). The PEV hypersurface is an indicator of how capable the design is in estimating the response in the underlying model. A bad design is either not able to fit the chosen model or is very poor at predicting the response. The Prediction Error Variance Viewer is only available for linear models. The Prediction Error Variance Viewer is not available when designs are rank deficient; that is, they do not contain enough points to fit the model. Optimal designs attempt to minimize the average PEV over the design region.

With an optimal design selected (as constructed in “Create Optimal Designs” on page 10-13), select **Tools > Prediction Error Variance Viewer**.



The default view is a 3-D plot of the PEV surface.

This shows where the response predictions are best. This example optimal design predicts well in the center and the middle of the faces (one factor high and the other midrange), but in the corners the design has the highest error. Look at the scale to see

how much difference there is between the areas of higher and lower error. For the best predictive power, you want low PEV (close to zero).

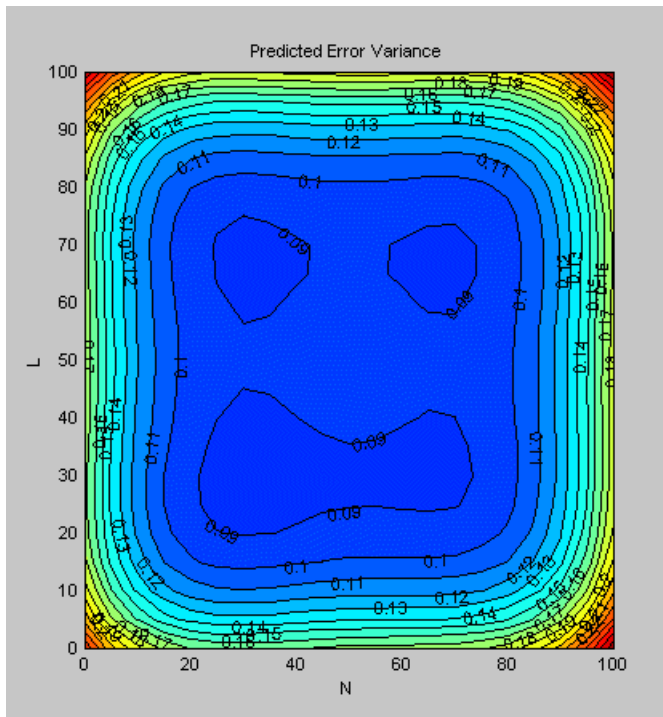
You can examine PEV for designs and models. The two are related in this way:

Accuracy of model predictions (model PEV)=Design PEV * MSE (Mean Square Error in measurements).

You can think of the design PEV as multiplying the errors in the data. The smaller the PEV, the greater the accuracy of your final model. You can read more about the calculation of PEV in “Prediction Error Variance”.

Try the other display options.

- The **View** menu has many options to change the look of the plots.
- You can change the factors displayed in the 2-D and 3-D plots. The pop-up menus below the plot select the factors, while the unselected factors are held constant. You can change the values of the unselected factors using the buttons and edit boxes in the **Input factors** list, top left.
- The **Movie** option shows a sequence of surface plots as a third input factor's value is changed. You can change the factors, replay, and change the frame rate.
- You can change the number, position, and color of the contours on the contour plot with the **Contours** button, as shown.



Add Points Optimally

You can further optimize the optimal design by returning to the Optimal Design dialog, where you can delete or add points optimally or at random. The most efficient way is to delete points *optimally* and add new points *randomly* — these are the default algorithm settings. Only the existing points need to be searched for the most optimal ones to delete (the least useful), but the entire candidate set has to be searched for points to add optimally.

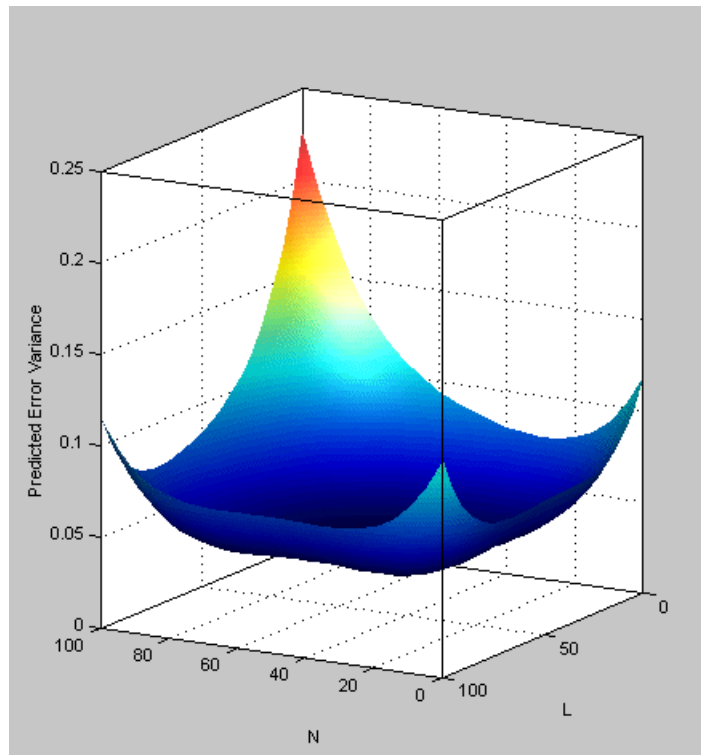
To strengthen the current optimal design:

- 1 Return to the Design Editor window.
- 2 Click the Optimal Design button in the toolbar again to reenter the dialog, and add 60 more points. Keep the existing points (which is the default).

- 3 Click **OK** and watch the optimization progress, then click **Accept** when the number of iterations without improvement starts increasing.
- 4 View the improvements to the design in the main displays.
- 5 Once again select **Tools > Prediction Error Variance Viewer** and review the plots of prediction error variance and the new values of optimality criteria in the optimality frame (bottom left). The shape of the PEV projection might not change dramatically, but note the changes in the scales as the design improves. The values of D, V, and G optimality criteria will also change (you have to click **Calculate** to see the values).

To see more dramatic changes to the design, return to the Design Editor window (no need to close the Prediction Error Variance Viewer).

- 1 Split the display so you can see a 3-D projection at the same time as a Table view.
- 2 You can sort the points to make it easier to select points in one corner. For example, to pick points where N is 100 and L is 0,
 - a Select **Edit > Sort Points**.
 - b Choose to sort by N only (reduce the number of sort variables to one) and click **OK**.
- 3 Choose **Edit > Delete Point**.
- 4 Using the Table and 3-D views as a guide, in the Delete Points dialog, pick six points to remove along one corner. Add the relevant point numbers to the delete list by clicking the add (>) button.
- 5 Click **OK** to remove the points. See the changes in the main design displays and look at the new Surface plot in the Prediction Error Variance Viewer (see the example following).



Related Examples

- “Design of Experiment” on page 10-2
- “Create a Constrained Space-Filling Design” on page 10-7
- “Create Optimal Designs” on page 10-13

More About

- “Design of Experiments”

Create Classical Designs

In this section...

“Adding a Classical Design” on page 10-27

“Classical Design Browser” on page 10-28


“Setting Up and Viewing a Classical Design” on page 10-29

Adding a Classical Design

Classical designs (including full factorial) are very well researched and are suitable for simple regions (hypercube or sphere).

- 1 In the Design Editor window, click to select the Optimal design in the design tree (as constructed in “Create Optimal Designs” on page 10-13).
- 2 Add a new design. Use the first toolbar button, or select **File > New**.

A new child node appears in the tree, called **Optimal_1**. Notice that the parent node now has a padlock on the icon. This indicates it is locked. This maintains the relationship between designs and their child nodes. The tree arrangement lets you try different operations starting from a basic design, then select the most appropriate one to use. The hierarchy allows clear viewing of the effects of changes on designs. The locking of parent designs also gives you the ability to easily reverse out of changes by retreating back up the tree.

- 3 Select the new design node in the tree. Notice that the display remains the same — all the points from the previous design remain, to be deleted or added to as necessary. The new design inherits all its initial settings from the currently selected design and becomes a child node of that design.
- 4 Rename the new node **Classical** by clicking again or by pressing **F2**.
- 5 Click the  button in the toolbar or select **Design > Classical > Design Browser**.

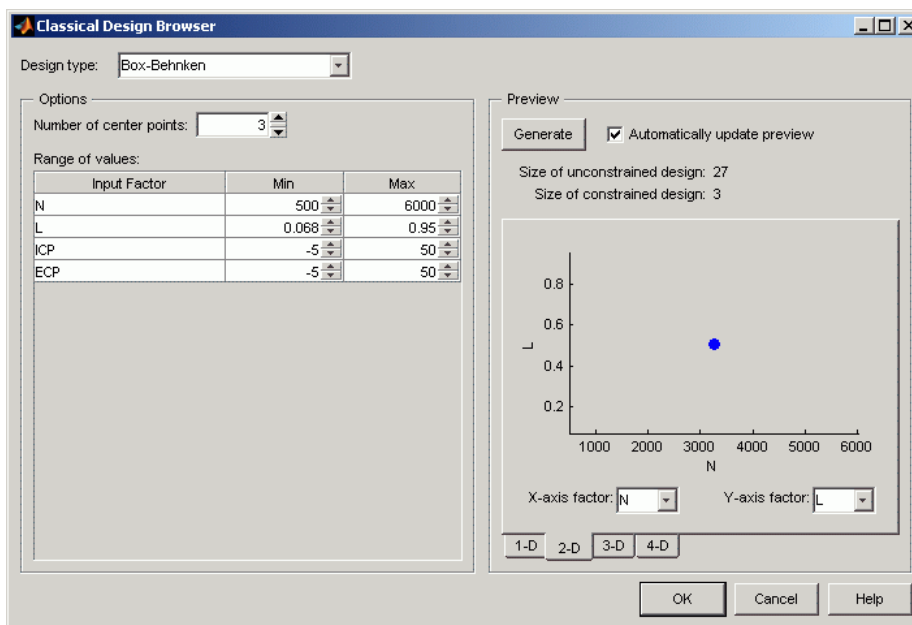
Note In cases where the preferred type of classical design is known, you can go straight to one of the five options under **Design > Classical**. Choosing the **Design Browser** option allows you to see graphical previews of these same five options before making a choice.

A dialog appears because there are already points from the previous design. You must choose between replacing and adding to those points or keeping only fixed points from the design.

- 6 Choose the default, replace current points with a new design, and click **OK**.

The Classical Design Browser appears.

Classical Design Browser



In the **Design Style** drop-down menu, there are five classical design options:

- Central Composite

Generates a design that has a center point, a point at each of the design volume corners, and a point at the center of each of the design volume faces. You can choose a ratio value between the corner points and the face points for each factor and the number of center points to add. You can also specify a spherical design. Five levels are used for each factor.

- **Box - Behnken**

Similar to Central Composite designs, but only three levels per factor are required, and the design is always spherical in shape. All the design points (except the center point) lie on the same sphere, so there should be at least three to five runs at the center point. There are no face points. These designs are particularly suited to spherical regions, when prediction at the corners is not required.

- **Full Factorial**

Generates an n -dimensional grid of points. You can choose the number of levels for each factor and the number of additional center points to add.

- **Plackett Burman**

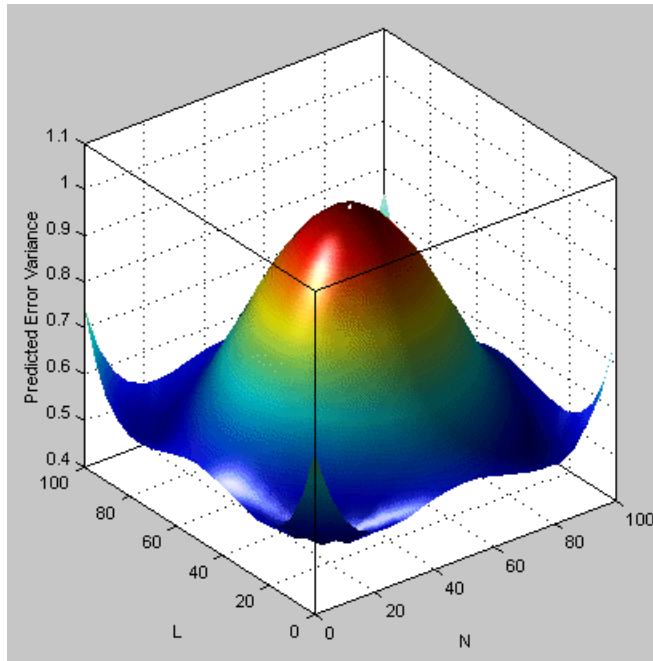
These are “screening” designs. They are two-level designs that are designed to allow you to work out which factors are contributing any effect to the model while using the minimum number of runs. For example, for a 30-factor problem this can be done with 32 runs.

- **Regular Simplex**

These designs are generated by taking the vertices of a k -dimensional regular simplex (k = number of factors). For two factors a simplex is a triangle; for three it is a tetrahedron. Above that are hyperdimensional simplices. These are economical first-order designs that are a possible alternative to Plackett Burman or full factorials.

Setting Up and Viewing a Classical Design

- 1 Choose a **Box - Behnken** design.
- 2 Reduce the number of center points to 1.
- 3 View your design in different projections using the tabs under the display.
- 4 Click **OK** to return to the Design Editor.
- 5 Use the Prediction Error Variance Viewer to see how well this design performs compared to the optimal design created previously; see the following illustration.



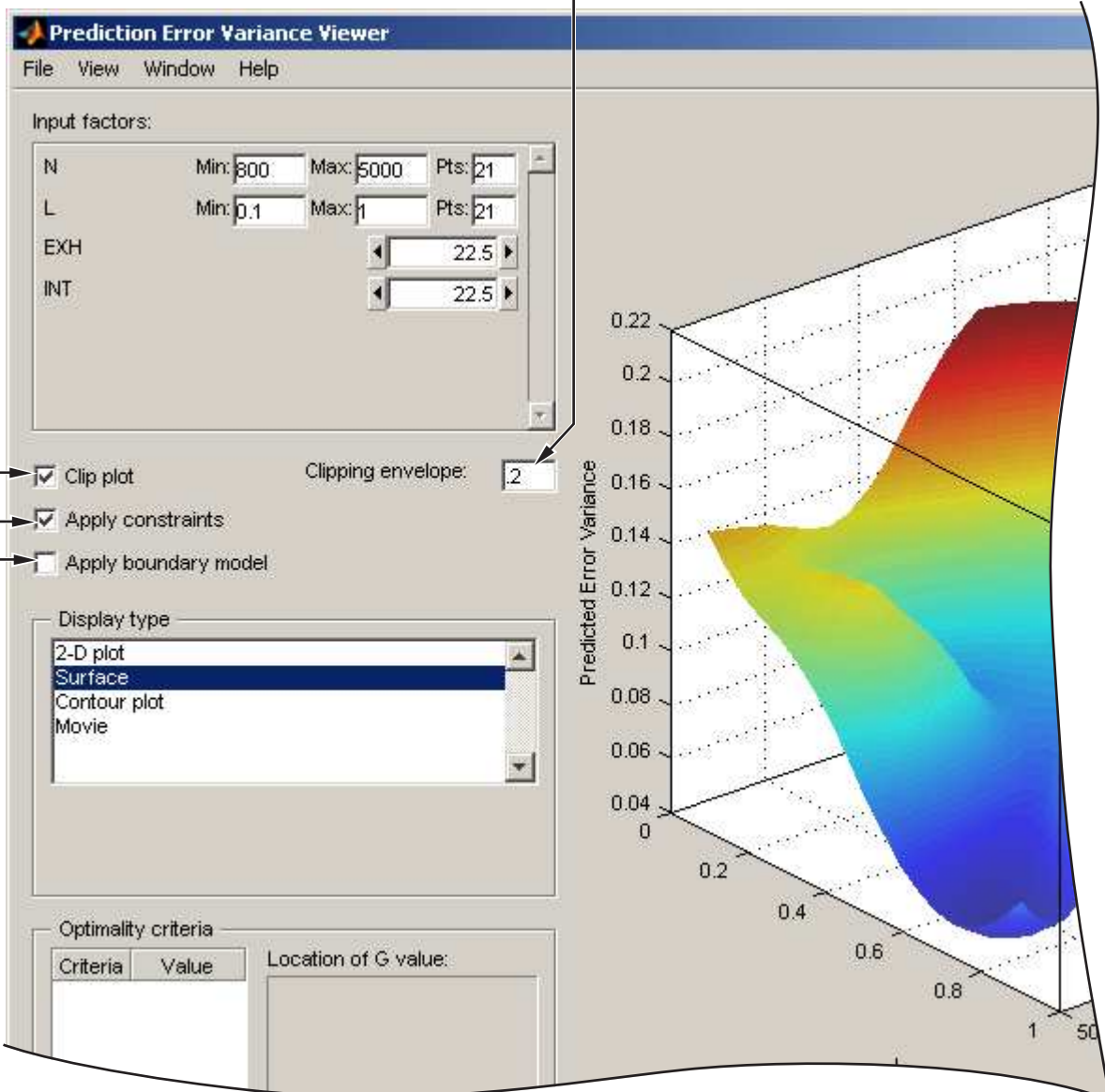
As you can see, this is not a realistic comparison, as this design has only 13 points (you can find this information in the bottom left of the main Design Editor display), whereas the previous optimal design had 100, but this is a good illustration of leverage. A single point in the center is very bad for the design, as illustrated in the Prediction Error Variance Viewer surface plot. This point is crucial and needs far more certainty for there to be any confidence in the design, as every other point lies on the edge of the space. This is also the case for Central Composite designs if you choose the spherical option. These are good designs for cases where you are not able to collect data points in the corners of the operating space.

If you look at the PEV surface plot, you should see a spot of white at the center. This is where the predicted error variance reaches 1. For surfaces that go above 1, the contour at 1 shows as a white line, as a useful visual guide to areas where prediction error is large.

- 1 Select **Movie**, and you see this white contour line as the surface moves through the plane of value 1.
- 2 Select the **Clip Plot** check box. Areas that move above the value of 1 are removed. The following example explains the controls.

Turn PEV value clipping on and off here

Change PEV clipping value here



Apply boundary model clipping here

Apply design constraint clipping here

Related Examples

- “Design of Experiment” on page 10-2
- “Create a Constrained Space-Filling Design” on page 10-7
- “Create Optimal Designs” on page 10-13
- “View Design Displays” on page 10-19

More About

- “Design of Experiments”

Use the Design Evaluation Tool

The Design Evaluation Tool is available for linear models only. See also “Global Model Class: Multiple Linear Models”.

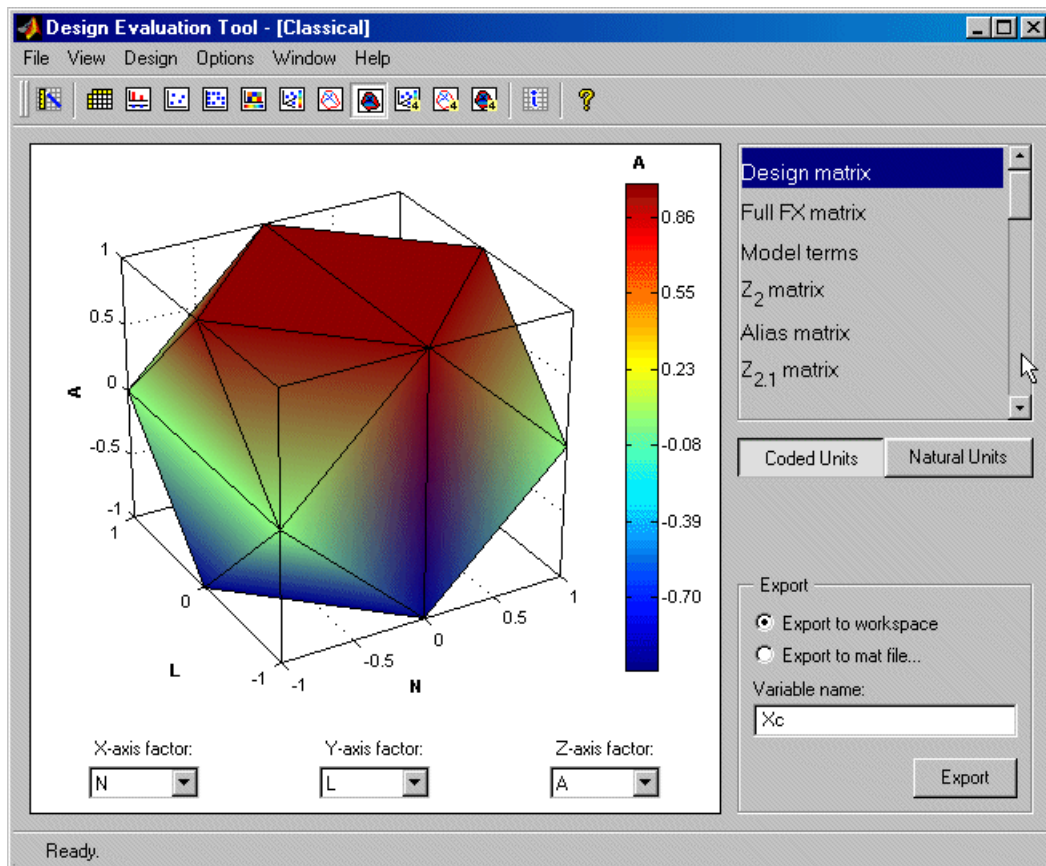
- 1 In the Design Editor, after creating a classical design (in “Create Classical Designs” on page 10-27) select **Tools > Evaluate Designs**.
- 2 Choose the **Box - Behnken** design and click **OK** in the Select Designs dialog.

The Design Evaluation Tool displays a large amount of statistical information about the design.

- 3 Select **Hat Matrix** from the list on the right.
- 4 Click the **Leverage Values** button.

Note that the leverage of the central point is **1.00** (in red) and the leverage of all other points is less than this. The design would clearly be strengthened by the addition of more central points. Obviously, this is a special case, but for any kind of design, the Design Evaluation Tool is a powerful way to examine properties of designs.

- 5 Select **Design Matrix** from the list box.
- 6 Click the **3D Surface** button in the toolbar.



This illustrates the spherical nature of the current design. As usual, you can rotate the plot by clicking and dragging with the mouse.

There are many other display options to try in the toolbar, and in-depth details of the model terms and design matrices can all be viewed. You can export any of these to the workspace or a `.mat` file using the **Export** box.

For a description of all the information available here, see “Use the Design Evaluation Tool” on page 10-33.

Improving the Design

To strengthen the current Box-Behnken design near the center region:

- 1 Close the Design Evaluation Tool.
- 2 Return to the Design Editor window.
- 3 Select **Design > Classical > Box-Behnken**.
- 4 Click **OK** to replace the current points with a new design.
- 5 Increase the number of center points and click **OK**.
- 6 Once again select **Tools > Prediction Error Variance Viewer** and review the plots of prediction error variance and the new values of optimality criteria in the optimality frame (bottom left).
- 7 Review the leverage values of the center points. From the Design Editor window, use **Tools > Evaluate Design** and go to **Hat Matrix**.
- 8 Try other designs from the Classical Design Browser. Compare Full Factorial with Central Composite designs; try different options and use the Prediction Error Variance Viewer to choose the best design.

Note You cannot use the Prediction Error Variance Viewer if there are insufficient points in the design to fit the model. For example, you cannot fit a quadratic with less than three points, so the default Full Factorial design, with two levels for each factor, must be changed to three levels for every factor before you can use the Prediction Error Variance Viewer.

- 9 When you are satisfied, return to the Design Editor window and choose **Edit > Select as Best**. You will see that this design node is now highlighted in blue in the tree. This can be applied to any design.

When you are creating designs before you start modeling, the design that you select as best is the one used to collect data.

Related Examples

- “Design of Experiment” on page 10-2
- “Create a Constrained Space-Filling Design” on page 10-7
- “Create Optimal Designs” on page 10-13
- “View Design Displays” on page 10-19

- “Create Classical Designs” on page 10-27

More About

- “Design of Experiments”

Data Editor for Modeling

This section discusses the following topics:

- “Data Manipulation for Modeling” on page 11-2
- “Load Data” on page 11-3
- “View and Edit the Data” on page 11-5
- “Create New Variables and Filters” on page 11-9
- “Store and Import Variables, Filters, and Plot Preferences” on page 11-12
- “Define Test Groupings” on page 11-13
- “Match Data to Experimental Designs” on page 11-16

Data Manipulation for Modeling

For empirical engine modeling in the Model Browser, you first need to load, process and select data for modeling. This tutorial shows you how to use the Data Editor for loading data, creating new variables, and creating constraints for that data.

You can load data from files (Microsoft® Excel® files, MATLAB files, text files) and from the MATLAB workspace. You can merge data in any of these forms with previously loaded data sets (providing there is no conflict in the form of the data) to produce a new data set. Test plans can use only one data set, so the merging function allows you to combine records and variables from different files in one model.

You can define new variables, apply filters to remove unwanted data, and apply test notes to filtered tests. You can store and retrieve these user-defined variables and filters for any data set, and you can store plot settings. You can change and add records and apply test groupings, and you can match data to designs. You can also write your own data loading functions.

The following tutorial is a step-by-step guide. Follow the steps in these sections in order:

- “Load Data” on page 11-3
- “View and Edit the Data” on page 11-5
- “Create New Variables and Filters” on page 11-9
- “Store and Import Variables, Filters, and Plot Preferences” on page 11-12
- “Define Test Groupings” on page 11-13
- “Match Data to Experimental Designs” on page 11-16

For comprehensive help on using data in the Model Browser, see “Data Manipulation for Modeling”.

Load Data

In this section...

“Opening the Data Editor” on page 11-3

“Loading a Data File” on page 11-3

Opening the Data Editor

You can create, copy, rename and delete data objects from the Project view in the Model Browser.

To enter the Data Editor and create a new data object, from the Project node, select **Data > New Data** (or click the New Data Object toolbar button).

The Data Editor appears.

There are no plots until some data has been loaded. The views shown depend on whether you have previously looked at the Data Editor, as it retains memory of your previous layout.

By default the new data object is called **Data Object**. Select **File > Rename Data** to enter a new name.

Loading a Data File

- 1 Click the import data from file button in the toolbar.
- 2 In the dialog box, find and select the **Holliday.xls** data file in the **matlab \toolbox\mbc\mbctraining** folder. Double-click to load the file, or select it and click **Open**.

You can also enter file pathnames in the edit box. The list contains the file types recognized by the Model Browser (**Excel files**, **Delimited text files**, **MATLAB data files**, etc). Leave this at the default, and the toolbox tries to determine what type of file is selected by looking at the file extension.

- 3 In Data Variable Editor, edit the variables names and units if you want. Double-click or use the Edit or Remove buttons for selected variables. View the summary showing the total number of records and variables imported, and each variable's range, mean, standard deviation, and units. Click **OK**.

View the data you loaded in the Data Editor.

View and Edit the Data

In this section...

“Viewing Data” on page 11-5

“Using Notes to Sort Data for Plotting” on page 11-5

“Removing Outliers and Problem Tests” on page 11-6

“Reordering and Editing Data” on page 11-7

Viewing Data

You can split the views to display several plots at once. Use the right-click context menus, the toolbar buttons, or the **View** menu to split views. You can choose 2-D plots, 3-D plots, multiple data plots, data tables, and tabs showing summary, statistics, variables, filters, test filters, and test notes.

For more details about each view, see “View and Edit Data” in the *Model-Based Calibration Toolbox Model Browser User's Guide*.

You can use test notes to investigate problem data and decide whether some points should be removed before modeling. The following steps cover using notes and views to sort and investigate your data.

Using Notes to Sort Data for Plotting

- 1 Right-click a view and select **Current View > Multiple Data Plot**.
- 2 Right-click the new view and select **Add Plot**.

The Plot Variables Setup dialog appears.

- 3 Select **spark** and click to add to the **X Variable** box, then select **tq** and click to add to the **Y Variable** box. Click **OK** to create the plot.
- 4 Click in the **Tests** list to select a test to plot (or **Shift**-click, **Ctrl**-click, or click and drag to select multiple tests).
- 5 Select **Tools > Test Notes**.
- 6 In The Test Note Editor, enter $\text{mean}(tq) < 10$ in the top edit box to define the tests to be noted, and enter **Low torque** in the Test Note edit box. Leave the note color at the default and click **OK**.

- 7 Click the Test Notes tab to view your note definition.
- 8 In the Test Selector pane on the left, observe that all the tests that satisfy the condition $\text{mean}(\text{tq}) < 10$ show **Low torque** next to them. Click the column header to sort the tests that meet the note condition to the top or bottom of the list.
- 9 Now create some more views.
 - Right-click a view and select **Split View > Data Table**.
 - Right-click a view and select **Split View > 3D Plot**.
- 10 In the Test Selector pane, click particular tests with the **Low torque** note.

Notice that when you select a test here, the same test is plotted in the multiple data plots, the 3D data plot, and highlighted in the data table. You can use the notes in this way to easily identify problem tests and decide whether you should remove them.

Removing Outliers and Problem Tests

- 1 Click a point on the **Multiple Data Plots** view.

The point is outlined in red on the plot, and highlighted in the data table. You can remove points you have selected as outliers by selecting **Tools > Remove Data** (or use the keyboard shortcut **Ctrl+A**). Select **Tools > Restore Data** (or use the keyboard shortcut **Ctrl+Z**) to open a dialog where you can choose to restore any or all removed points.

You can remove individual points as outliers, or you can remove records or entire tests with filters.

- 2 For example, after examining all the **Low torque** noted tests, you could decide they should be filtered out.
 - a Select **Tools > Test Filters**.
 - b In the Test Filter Editor, enter $\text{mean}(\text{tq}) > 10$ to keep all tests where the mean torque is greater than 10, and click **OK**.
 - c Click the Test Filters tab and observe the new test filter results show it is successfully applied and the number of records removed.

Similarly, you can use filters to remove individual records rather than entire tests, which you will cover in a later section “Applying a Filter” on page 11-9.

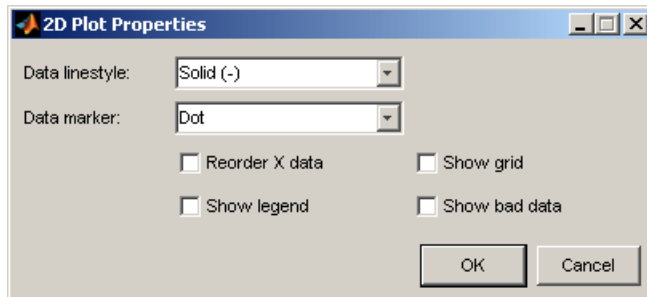
- 3 To view removed data in the table view, right-click and select **Allow Editing**. Removed records are red. To view removed data in the 2-D and Multiple Data Plots, select **Properties** and select the box **Show removed data**.

Reordering and Editing Data

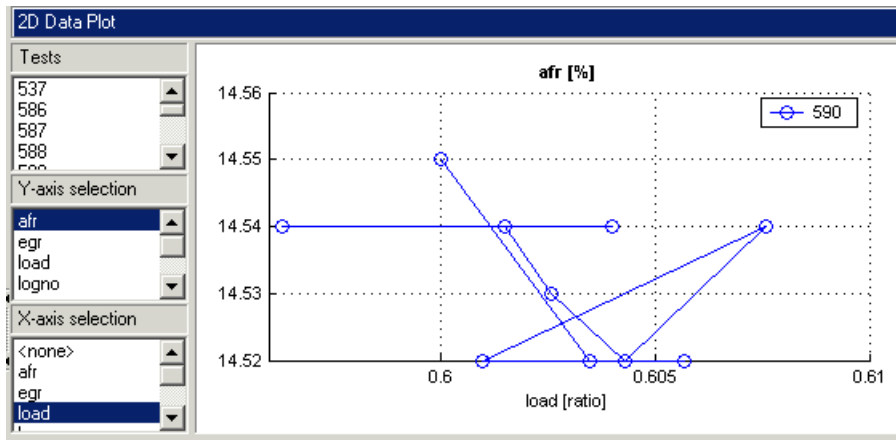
To change the display, right-click a 2-D plot and select **Properties**. You can alter grid and plot settings including lines to join the data points.

Reorder X Data in the Plot Properties dialog can be useful when record order does not produce a sensible line joining the data points. For an illustration of this:

- 1 Ensure you are displaying a 2-D plot. You can right-click on any plot and select **Current Plot > 2-D Plot**, or use the context menu split commands to add new views.
- 2 Right-click on a 2-D plot and select **Properties** and choose **solid** from the **Data Linestyle** drop-down menu, as shown below. Click **OK**.

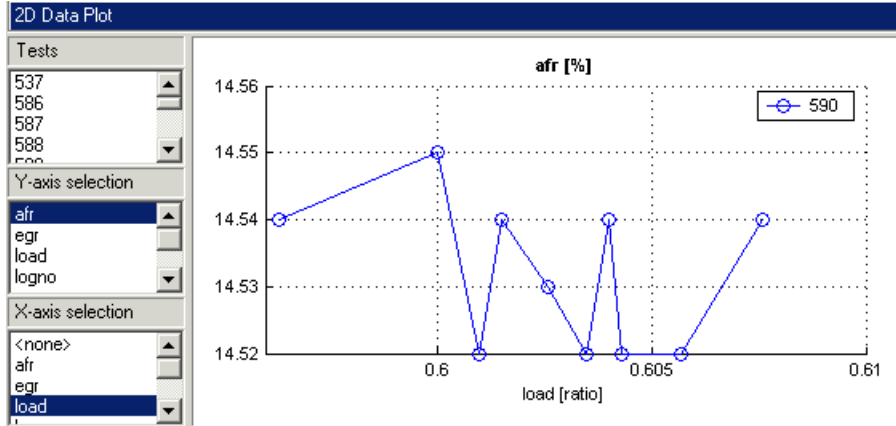


- 3 Choose **af r** for the *y*-axis.
- 4 Choose **Load** for the *x*-axis.
- 5 Select test 590. You must use the test controls contained within the 2-D plot. The **Tests** pane on the left applies to other views: tables and 3-D and multiple data plots.



- 6 Right-click and select **Properties** and choose **Reorder X Data**. Click **OK**.

This command replots the line from left to right instead of in the order of the records, as shown.



- 7 Right-click and select **Split Plot > Data Table** to split the currently selected view and add a table view. You can select particular test numbers in the **Tests** pane on the left of the Data Editor. You can right-click to select **Allow Editing**, and then you can double-click cells to edit them.

Create New Variables and Filters

In this section...

“Adding New Variables” on page 11-9

“Applying a Filter” on page 11-9

“Sequence of Variables” on page 11-11

“Deleting and Editing Variables and Filters” on page 11-11

Adding New Variables

You can add new variables to the data set.

- 1 Select **Tools > Variables**, or click the toolbar button.

In the Variable Editor, you can define new variables in terms of existing variables. Define the new variable by writing an equation in the edit box at the top.

- 2 Define a new variable called **POWER** that is defined as the product of two existing variables, **tq** and **n**, by entering **POWER=tq*n**, as seen in the example following. You can also double-click variable names and operators to add them, which can be useful to avoid typing mistakes in variable names, which must be exact including case.
- 3 Click **OK** to add this variable to the current data set.
- 4 View the new variable in the Data Editor in the Variables tab at the top. You can also now see **7 + 1 variables** in the summary tab.

Applying a Filter

A filter is a constraint on the data set you can use to exclude some records. You use the Filter Editor to create new filters.

- 1 Choose **Tools > Filters**, or click the toolbar button.

In the Filter Editor, define the filter using logical operators on the existing variables.

- 2 Keep all records with speed (**n**) greater than 1000. Type **n** (or double-click on the variable **n**), then type **>1000**.
- 3 Click **OK** to impose this filter on the current data set.

- 4 View the new filter in the Data Editor in the Filters tab at the top. Here you can see a list of your user-defined filters and how many records are removed by the new filter. You can also now see **141/270 records** in the summary tab and a red section in the bar illustrating the records removed by the filter.

Sequence of Variables

You can change the order of user-defined variables in the Variable Editor list using the up and down arrow buttons.

Select **Tools > Variables**.

Example:

- 1 Define two new variables, **New1** and **New2**. Note that you can use the buttons to add or remove a list item to create or delete variables in this view. Click the button to 'Add item' to add a new variable, and enter the definitions shown.

Notice that **New2** is defined in terms of **New1**. New variables are added to the data in turn and hence **New1** must appear in the list before **New2**, otherwise **New2** is not well defined.

- 2 Change the order by clicking the down arrow in the Variable Editor to produce this erroneous situation. Click **OK** to return to the Data Editor and in the Variables tab you can see the error message that there is a problem with the variable.
- 3 Use the arrows to order user-defined variables in legitimate sequence.

Deleting and Editing Variables and Filters

You can delete user-defined variables and filters.


Example:

- 1 To delete the added variable **New1**, select it in the Variables tab and press the **Delete** key.
- 2 You can also delete variables in the Variable Editor by clicking the Remove Item button.

Similarly, you can delete filters by selecting the unwanted filter in the Filters tab and using the **Delete** key.

Store and Import Variables, Filters, and Plot Preferences

You can store and import plot preferences, user-defined variables, filters, and test notes so they can be applied to other data sets loaded later in the session, and to other sessions.

- Select **Tools > Import Expressions**
- Click the toolbar button 

The Data Editor remembers your plot type settings and when reopened will display the same types of views. You can also store your plot layouts to save the details of your Multiple Data Plots, such as which factors to display, line style, grid, etc.

In the Import Variables, Filters, and Editor Layout dialog box, use the toolbar buttons to import variables, filters and plot layouts. Import from other data sets in the current project, or from MBC project files, or from files exported from the Data Editor.

To use imported expressions in your current project, select items in the lists and click the toolbar button to apply in the data editor.

To store expressions in a file, in the Data Editor, select **Tools > Export Expressions** and select a file name.

For more, see “Using Data”.

Define Test Groupings

The Define Test Groupings dialog collects records of the current data object into groups; these groups are referred to as *tests*.


The dialog is accessed from the Data Editor in either of these ways:

- Using the menu **Tools > Test Groups**

- Using the toolbar button 


When you enter the dialog using the `holliday` data, a plot is displayed as the variable `logno` is automatically selected for grouping tests.

Select another variable to use in defining groups within the data.


- 1 Select `n` in the **Variables** list.
- 2 Click the  button to add the variable (or double-click `n`).

The variable `n` appears in the list view on the left. You can now use this variable to define groups in the data. The maximum and minimum values of `n` are displayed. The **Tolerance** is used to define groups: on reading through the data, when the value of `n` changes by more than the tolerance, a new group is defined. You change the **Tolerance** by typing directly in the edit box.

You can define additional groups by selecting another variable and choosing a tolerance. Data records are then grouped by `n` or by this additional variable changing outside their tolerances.

- 3 Clear the box **Group by** for `logno`. Notice that variables can be plotted without being used to define groups.
- 4 Add `load` to the list by selecting it on the right and clicking .
- 5 Change the `load` tolerance to 0.01 and watch the test grouping change in the plot.
- 6 Clear the **Group By** check box for `load`. Now this variable is plotted without being used to define groups.

The plot shows the scaled values of all variables in the list view (the color of the tolerance text corresponds to the color of data points in the plot). Vertical pink bars show the tests (groups). You can zoom the plot by **Shift**-click-dragging or middle-click-dragging the mouse; zoom out again by double-clicking.

- 7 Select **load** in the list view (it becomes highlighted in blue) and remove it from the list by clicking the  button.
- 8 Double-click to add **spark** to the list, and clear the **Group By** check box. Select **logno** as the only grouping variable.

It can be helpful to plot the local model variable (in this case **spark**) to check you have the correct test groupings. The plot shows the sweeps of **spark** values in each test while speed (**n**) is kept constant. Speed is only changed between tests, so it is a global variable. Try zooming in on the plot to inspect the test groups; double-click to reset.

One-stage data defines one test per record, regardless of any other grouping. This is required if the data is to be used in creating one-stage models.

Sort records before grouping allows you to reorder records in the data set. Otherwise, the groups are defined using the order of records in the original data object.

Show original test groups displays the original test groupings if any were defined.

Test number variable contains a list of all the variables in the current data set. Any of these could be selected to number the tests.

- 9 Make sure **logno** is selected for the **Test number variable**.

This changes how the tests are displayed in the rest of the Model Browser. Test number can be a useful variable for identifying individual tests in Model Browser and Data Editor views (instead of 1,2,3...) if the data was taken in numbered tests and you want access to that information during modeling.

If you chose **none** from the **Test number variable** list, the tests would be numbered 1,2,3 and so on in the order in which the records appear in the data file. With **logno** chosen, you will see tests in the Data Editor listed as 586, 587 etc.

Every record in a test must share the same test number to identify it, so when you are using a variable to number tests, the value of that variable is taken in the first record in each test.

Test numbers must be unique, so if any values in the chosen variable are the same, they are assigned new test numbers for the purposes of modeling (this does not change the underlying data, which retains the correct test number or other variable).

- 10** Click **OK** to accept the test groupings defined and close the dialog.

In the Data Editor summary tab, view the number of tests.

The number of records shows the number of values left (after filtration) of each variable in this data set, followed by the original number of records. The color coded bars also display the number of records removed as a proportion of the total number. The values are collected into a number of tests; this number is also displayed. The variables show the original number of variables plus user-defined variables.

Match Data to Experimental Designs

In this section...

“Introducing Matching Data to Designs” on page 11-16

“Tolerances and Cluster Information” on page 11-18

“Understanding Clusters” on page 11-20

Introducing Matching Data to Designs


You can use an example project to illustrate the process of matching experimental data to designs.

Experimental data is unlikely to be identical to the desired design points. You can use the Design Match view in the Data Editor to compare the actual data collected with your experimental design points. Here you can select data for modeling. If you are interested in collecting more data, you can update your experimental design by matching data to design points to reflect the actual data collected. You can then optimally augment your design (using the Design Editor) to decide which data points it would be most useful to collect, based on the data obtained so far.

You can use an iterative process: make a design, collect some data, match that data with your design points, modify your design accordingly, then collect more data, and so on. You can use this process to optimize your data collection process in order to obtain the most robust models possible with the minimum amount of data.

- 1 To see the data matching functions, in the Model Browser, select **File > Open Project** and browse to the file `Data_Matching.mat` in the `matlab\toolbox\mbc\mbctraining` directory.
- 2 Click the **Spark Sweeps** node in the model tree to change to the test plan view.

Here you can see the two-stage test plan with model types and inputs set up. The global model has an associated experimental design (which you could view in the Design Editor). You are going to use the Data Editor to examine how closely the data collected so far matches to the experimental design.

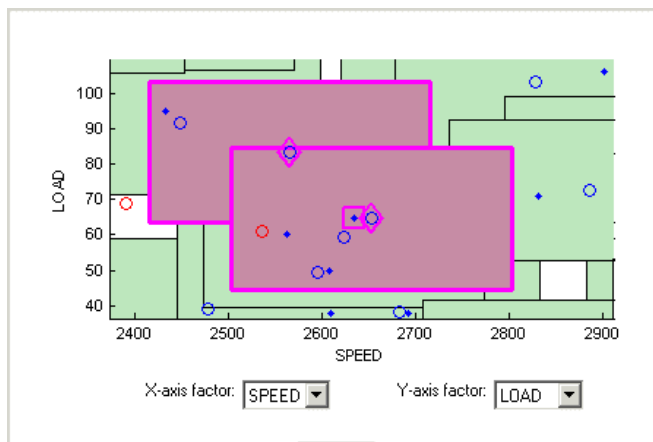
- 3 Click the Edit Data button () in the toolbar.

The Data Editor appears.

- 4 You need a **Design Match** view to examine design and data points. Right-click a view in the Data Editor and select **Current View > Design Match**.

In the Design Match you can see colored areas containing points. These are “clusters” where closely matching design and data points have been selected by the matching algorithm.

Tolerance values (derived initially from a proportion of the ranges of the variables) are used to determine if any data points lie within tolerance of each design point. Data points that lie within tolerance of any design point are matched to that cluster. Data points that fall inside the tolerance of more than one design point form a single cluster containing all those design and data points. If no data points lie within tolerance of a design point, it remains unmatched and no cluster is plotted.



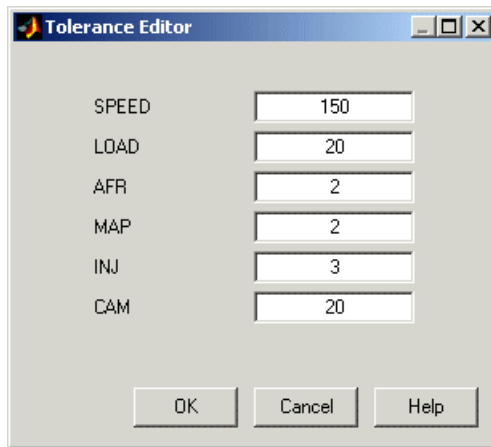
Notice the shape formed by overlapping clusters. The example shown outlined in pink is a single cluster formed where a data point lies within tolerance of two design points.

Note that on this plot you can see other unselected points that appear to be contained within this cluster. You need to track points through other factor dimensions using the axis controls to see where points are separated beyond tolerance. You will do this in a later step of this tutorial, “Understanding Clusters” on page 11-20.

Tolerances and Cluster Information

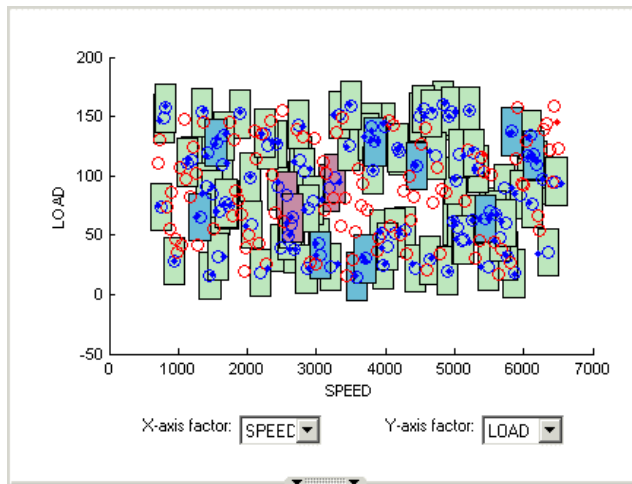
- 1 To edit tolerance values, select **Tolerances** in the context menu.

The Tolerance Editor appears. Here you can change the size of clusters in each dimension. Observe that the **LOAD** tolerance value is currently 100. This accounts for the elongated shape (in the **LOAD** dimension) of the clusters in the current plot, because this tolerance value is a high proportion of the total range of this variable.

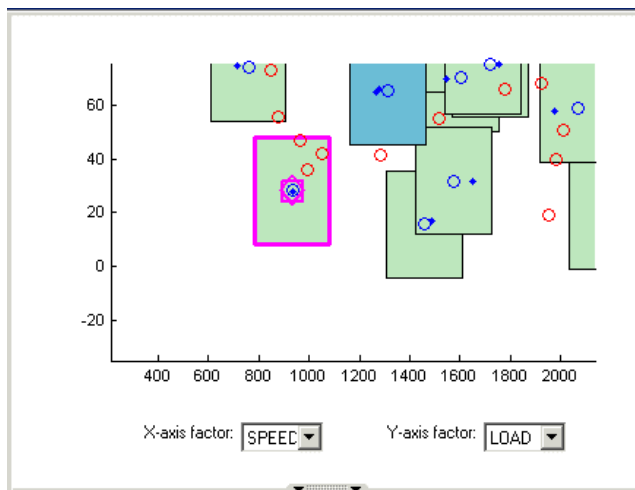


- 2 Click the **LOAD** edit box and enter 20, as shown. Click **OK**.

Notice the change in shape of the clusters in the **Design Match** view.



- 3 Shift click (center-click) and drag to zoom in on an area of the plot, as shown. You can double-click to return to the full size plot.



- 4 Click a cluster to select it. Selected points or clusters are outlined in pink. If you click and hold, you can inspect the values of global variables at the selected points (or for all data and design points if you click on a cluster). You can use this information to help you decide on suitable tolerance values if you are trying to match points.

- 5 Notice that the **Cluster Information** list shows the details of all data and design points contained in the selected cluster. You use the check boxes here to select or exclude data or design points. Click different clusters to see a variety of points. The list shows the values of global variables at each point, and which data and design points are within tolerance of each other. Your selections here determine which data will be used for modeling, and which design points will be replaced by actual data points.

Note All data points with a selected check box will be used for modeling. All data points with a cleared check box will be removed from the data set, and not seen in any other views. This design match cluster view is the only place you can restore these excluded data to the data set.

Understanding Clusters

If you are not interested in collecting more data, then there is no need to make sure the design is modified to reflect the actual data. All data (except those you exclude by clearing the check boxes) will be used for modeling.

However, if you want your new design (called **Actual Design**) to accurately reflect what data has been obtained so far, for example to collect more data, then the cluster matching is important. All data points with a selected check box will be added to the new **Actual Design**, except those in red clusters. The color of clusters indicates what proportion of selected points it contains as follows:

- Green clusters have equal numbers of selected design and selected data points. The data points will replace the design points in the **Actual Design**.

Note that the color of all clusters is determined by the proportion of *selected* points they contain; excluded points (with cleared check boxes) have no effect. Your check box selections can change cluster color.

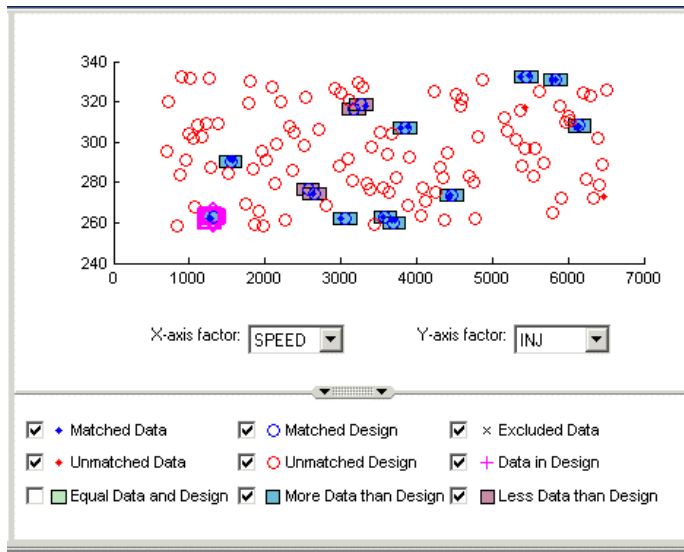
- Blue clusters have more data points than design points. All the data points will replace the design points in the **Actual Design**.
- Red clusters have more design points than data points. These data points will not be added to your design as the algorithm cannot choose which design points to replace, so you must manually make selections to deal with red clusters if you want to use these data points in your design.

If you don't care about the **Actual Design** (for example, if you do not intend to collect more data) and you are just selecting data for modeling, then you can ignore red clusters. The data points in red clusters are selected for modeling.

- 1** Right-click the **Design Match** and select **Select Unmatched Data**. Notice that the remaining unmatched data points appear in the list. Here you can use the check boxes to select or exclude unmatched data in the same way as points within clusters.
- 2** Select a cluster, then use the drop-down menu to change the **Y-Axis factor** to INJ. Observe the selected cluster now plotted in the new factor dimensions of SPEED and INJ.

You can use this method to track points and clusters through the dimensions. This can give you a good idea of which tolerances to change in order to get points matched. Remember that points that do not form a cluster may appear to be perfectly matched when viewed in one pair of dimensions; you must view them in other dimensions to find out where they are separated beyond the tolerance value. You can use this tracking process to decide whether you want particular pairs of points to be matched, and then change the tolerances until they form part of a cluster.

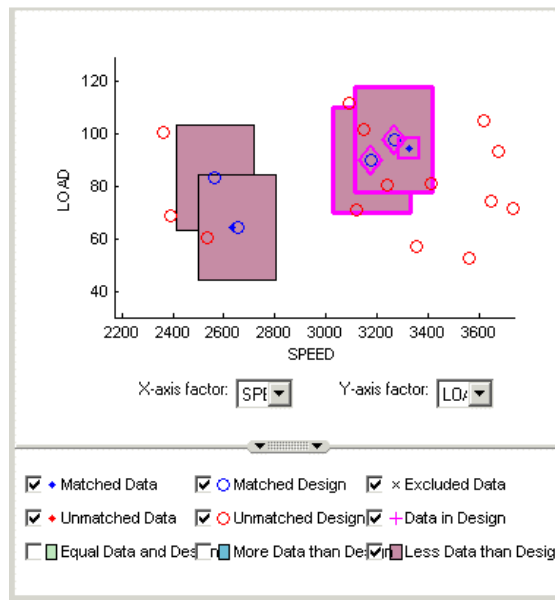
- 3** Clear the **Equal Data and Design** check box in the **Design Match** view. You control what is plotted using these check boxes.



This removes the green clusters from view, as shown. These clusters are matched; you are more likely to be interested in unmatched points and clusters with uneven numbers of data and design points. Removing the green clusters allows you to focus on these points of interest. If you want your new **Actual Design** to accurately reflect your current data, your aim is to get as many data points matched up to design points as possible; that is, as few red clusters as possible.

- 4 Clear the check box for **More Data than Design**. You may also decide to ignore blue clusters, which contain more data points than design points. These design points will be replaced by all data points within the cluster. An excess of data points is unlikely to be a concern.

However, blue clusters may indicate that there was a problem with the data collection at that point, and you may want to investigate why more points than expected were collected.



- 5 Select one of the remaining red clusters. Both of these have two design points within tolerance of a single data point.
- 6 Choose one of the design points to match to the data point, then clear the check box of the other design point. The cleared design point remains unchanged in the design. The selected design point will be replaced by the matched data point.

Notice that the red cluster disappears. This is because your selection results in a cluster with an equal number of selected data and design points (a green cluster) and your current plot does not display green clusters.

- 7 Repeat for the other red cluster.

Now all clusters are green or blue. There are two remaining unmatched data points.

- 8 Clear the **Unmatched Design** check box to locate the unmatched data points. Select **Unmatched Design** check box again — you need to see design points to decide if any are close enough to the data points that they should be matched.
- 9 Locate and zoom in on an unmatched data point. Select the unmatched data point and a nearby design point by clicking, then use the axis drop-down menus to track the candidate pair through the dimensions. Decide if any design points are close

enough to warrant changing the tolerance values to match the point with a design point.

- 10** Recall that you can right-click the **Design Match** and select **Select Unmatched Data** to display the remaining unmatched data points in the **Cluster Information** list. Here you can use the check boxes to select or exclude these points. If you leave them selected, they will be added to the **Actual Design**.

These steps illustrate the process of matching data to designs, to select modeling data and to augment your design based on actual data obtained. Some trial and error is necessary to find useful tolerance values. You can select points and change plot dimensions to help you find suitable values. If you want your new **Actual Design** to accurately reflect your experimental data, you need to make choices to deal with red clusters. Select which design points in red clusters you want to replace with the data points. If you do not, then these data points will not be added to the new design.

When you are satisfied that you have selected all the data you want for modeling, close the Data Editor. At this point, your choices in the Design Match view will be applied to the data set and a new design called **Actual Design** will be created. All the changes are determined by your check box selections for data and design points.

All data points with a selected check box are selected for modeling. Data points with cleared check boxes are excluded from the data set. Changes are made to the existing design to produce the new **Actual Design**. All selected data will be added to your new design, except those in red clusters. Selected data points that have been matched to design points (in green and blue clusters) replace those design points.

All these selected data points become fixed design points (red in the Design Editor) and appear as **Data in Design** (pink crosses) when you reopen the Data Editor.

This means these points will not be included in clusters when matching again. These fixed points will also not be changed in the Design Editor when you add points, though you can unlock fixed points if you want. This can be very useful if you want to optimally augment a design, taking into account the data you have already obtained.

See “Match Data to Designs” for more information.

For more information on all aspects of data handling for modeling, see “Data Import and Processing”.

Feature Calibration

This section includes the following topics:

Feature Calibration

In this section...

“What Are Feature Calibrations?” on page 12-2

“Start CAGE” on page 12-4

“Set Up Variables” on page 12-4

“Set Up Models” on page 12-7

“Set Up a New Feature” on page 12-9

“Set Up the Strategy” on page 12-10

“Set Up the Tables” on page 12-12

“Process For Feature Calibration” on page 12-14

“Calibrate the Normalizers” on page 12-15

“Calibrate the Tables” on page 12-19

“Calibrate the Feature” on page 12-25

“Export Calibrations” on page 12-30

What Are Feature Calibrations?

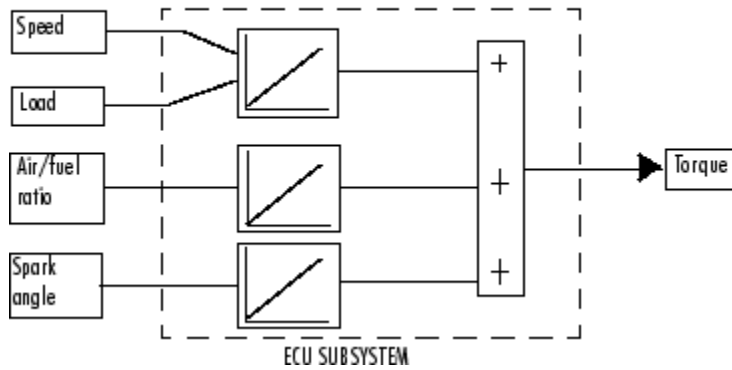
The feature calibration process within the Model-Based Calibration Toolbox product calibrates an estimator, or feature, for a control subsystem in an electronic control unit (ECU). These features are usually algebraic collections of one or more tables. You use the features to estimate signals in the engine that are unmeasurable, or expensive to measure, and are important for engine control. The toolbox can calibrate the ECU subsystem by directly comparing it with a plant model of the same feature.

There are advantages to feature calibration compared with simply calibrating using experimental data. Data is noisy (that is, there is measurement error) and this can be smoothed by modeling; also models can make predictions for areas where you have no data. This means you can calibrate more accurately while reducing the time and effort required for gathering experimental data.

An example of an ECU subsystem control feature estimates the value of torque, depending on the four inputs: speed, load, air/fuel ratio (AFR), and spark angle.

A diagram of this ECU subsystem example follows.

Plant Model for Torque = $TQ(\text{speed, load, AFR, spark})$



In this tutorial example, there are three lookup tables:

- A speed-load table
- A modifier, or table, for AFR
- A modifier for spark angle

This tutorial takes you through the various steps required to set up this feature and then calibrate it using CAGE. You will use CAGE to fill the tables by comparing them with a torque engine model.

The model is a copy of the torque model built in the Model Browser's Quick Start tutorial using engine data. This illustrates how you can use the Model-Based Calibration Toolbox product to map engine behavior and transfer this information to engine calibrations. You can construct a model using the Model Browser; then you can use CAGE to calibrate lookup tables by reference to the model.

Start CAGE

Start CAGE by typing

```
cage
```

at the MATLAB prompt.

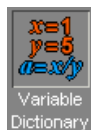
Note If you have a CAGE session open, select **File > New > Project**.

Before you can perform a calibration, you must set up the variable dictionary and models that you want to use.

Set Up Variables

To set up the variables and constants that you want to use in your calibration,

- 1 Click **Variable Dictionary** in the **Data Objects** pane of CAGE.



The **Variable Dictionary** view displays all the variables, constants, and formulas in a session. This is empty until you add some variable items.

There are two ways in which you can set up variables:

- Import a variable dictionary
- Add variables and constants to your session

After setting up your variables and constants, you can export the variable dictionary to use in other calibrations.

Importing a Variable Dictionary

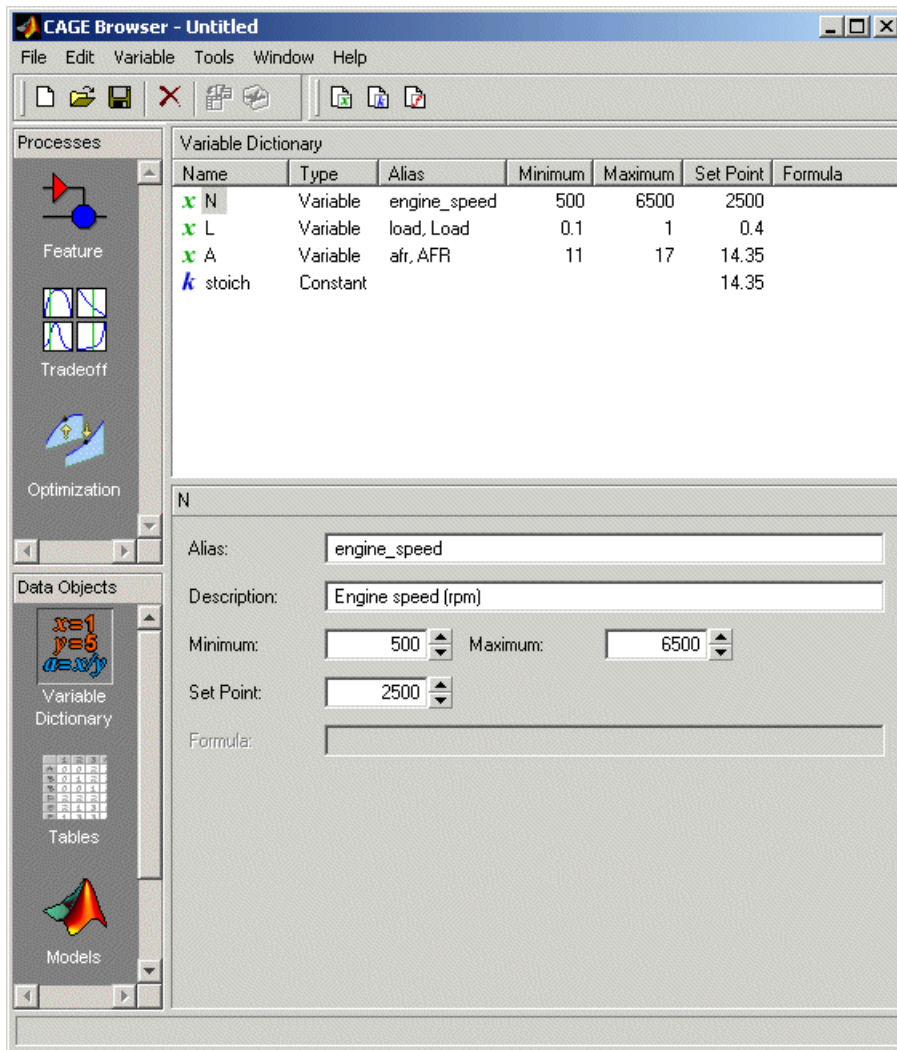
To import a variable dictionary,

- 1 Select **File > Import > Variable Dictionary**.
- 2 Select the `tutorial.xml` file found in `matlab\toolbox\mbc\mbctraining` and click **Open**. CAGE automatically switches to the Variable Dictionary view.

This imports a set of variables and a constant. In this example, the variable dictionary contains


- The stoichiometric constant, `stoch`
- `N`, engine speed
- `L`, load
- `A`, AFR

Your display should resemble the following.



Adding and Editing Variables and Constants

To add a variable for the spark angle,

- 1 Click New Variable  in the toolbar. This adds a new variable to your dictionary.

- 2 Right-click the new variable and select **Rename** (or press **F2**) to rename the variable.
- 3 Enter SPK as the name.
- 4 Set the range of the variable by entering -5 as the **Minimum** and 50 as the **Maximum**.

The variable dictionary enables you to specify different names for the same variable, and also give descriptions of variables. For example, the variable `spk` might be referred to as `S` or `spark` in other models.

To ensure that CAGE recognizes an instance of `S` or `spark` as the same as `spk`, specify the aliases of SPK:

- 1 Enter `S`, `spark` in the **Alias** edit box.
- 2 Enter `Spark advance (deg)` in the **Description** edit box.

Note The **Variable Dictionary** is case sensitive: `s` and `S` are different.

The variable dictionary enables you to specify a preferred value for a variable. For example, in the preferred value of the variable, `AFR` is set as the stoichiometric constant 14.35.

- 1 Select `SPK` and enter 25 in the **Set Point** edit box to specify the preferred value.

Set Up Models

A model in the Model-Based Calibration Toolbox product is a function of a set of variables. Typically, you construct a model using the Model Browser; then you can use CAGE to calibrate lookup tables by reference to the model.

The following example uses a model of the behavior of torque with varying spark angle, air/fuel ratio, engine speed, and load.

Importing a Model

To import a model built using the Model Browser,

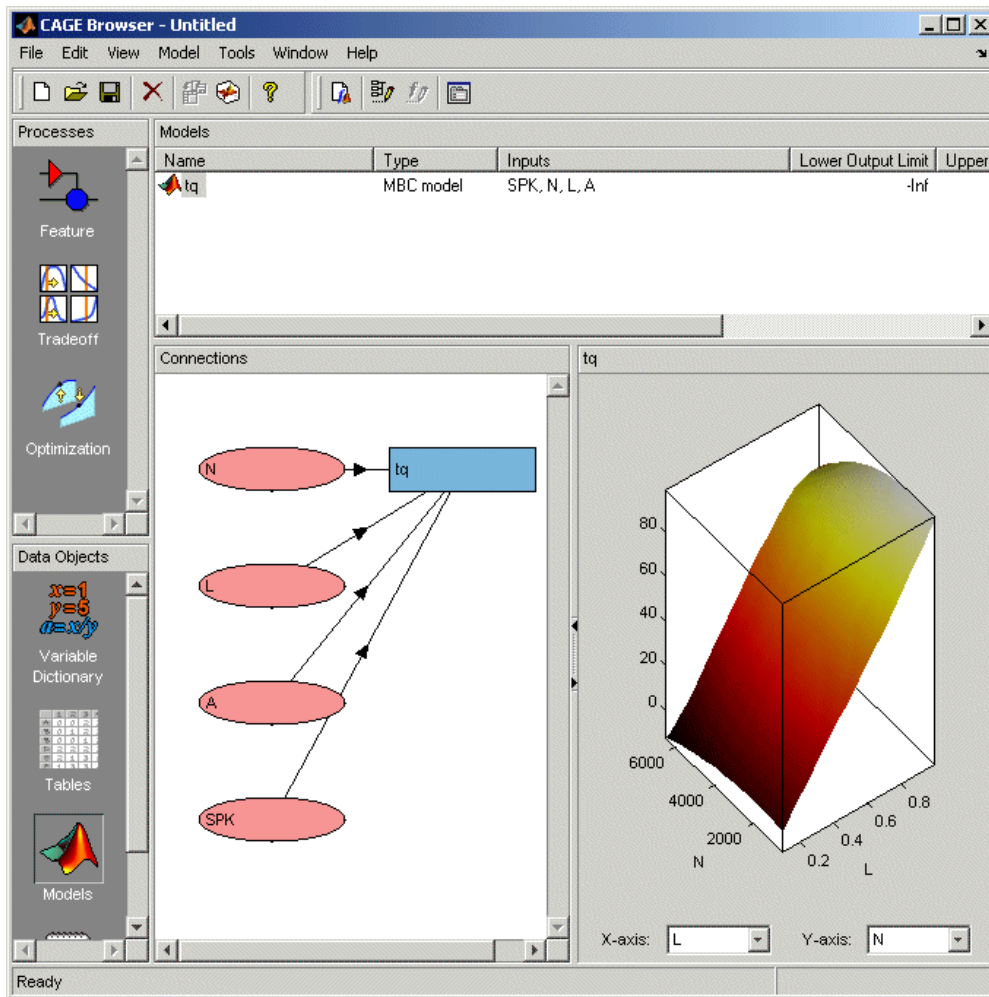
- 1 Select **File > Import > Model**, which opens a file browser.

- 2 Browse to `matlab\toolbox\mbc\mbctraining`, select the `tutorial.exm` file (this is a copy of the torque model built in the Model Browser's Empirical Engine Modeling tutorial), and click **Open**. The Model Import Wizard appears.
- 3 There are two models stored in this file, `tq` and `knot`. Highlight `tq` and select the check box to **Automatically assign/create inputs**.

CAGE automatically assigns variables in the variable dictionary to the model input factors or their aliases (as long as names are exact). If names are not exact you can select variables manually using the wizard.

- 4 Click **Finish** to complete the wizard.

CAGE switches to the **Models** view, as shown following.



For more information about models, see “Setting Up Models” in the CAGE documentation.

Set Up a New Feature

The feature calibration process calibrates an algebraic collection of lookup tables, or *strategy*, by comparing the tables to the model.

When you have set up the variables and models, you can set up the feature as follows:

- 1 Select **File > New > Feature**.

CAGE automatically displays the **Feature** view and creates a new feature.

- 2 Select **Feature > Select Filling Item**. This opens the Select Filling Item dialog box. Select **tq** (currently the only model in your project) and click **OK**.
- 3 Create a strategy. For instructions, see the next section, “Set Up the Strategy” on page 12-10.

A strategy is a collection of tables. The Model-Based Calibration Toolbox product uses Simulink software to enable you to graphically specify the collection of tables for a feature.

- 4 After you have created a strategy, the next step is to set up your tables. For more information, see the section, “Set Up the Tables” on page 12-12.

Set Up the Strategy

The toolbox uses Simulink to graphically specify the strategy.

Importing a Strategy

To import a strategy,

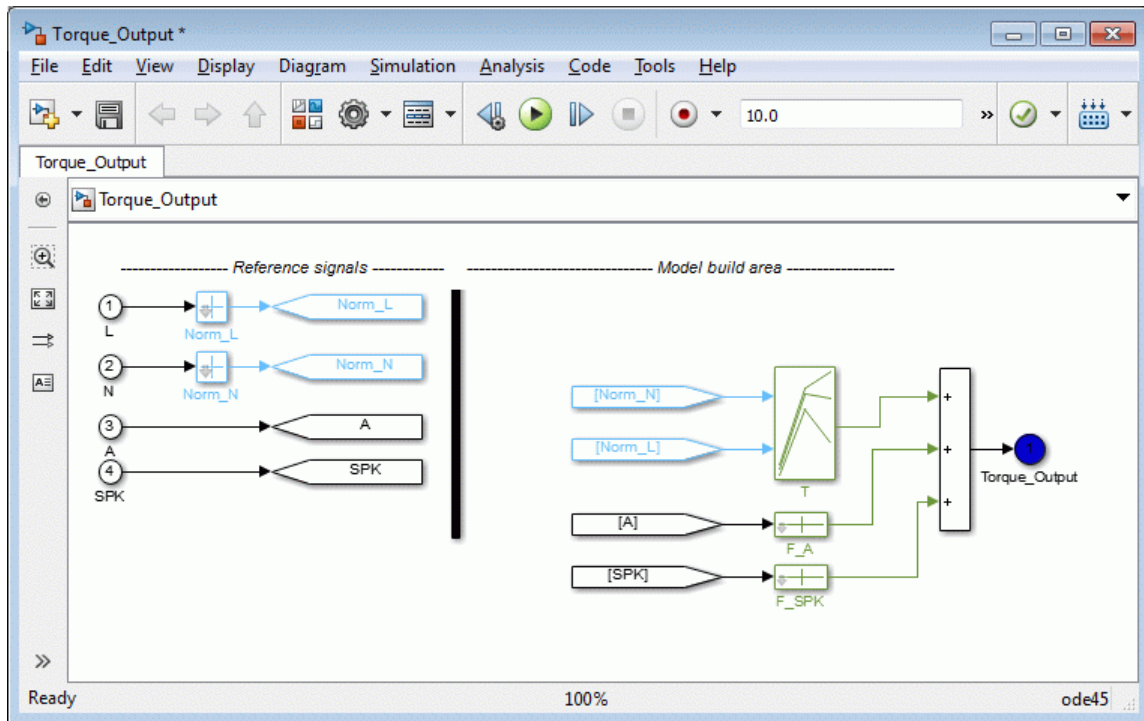
- 1 Select **File > Import > Strategy**.
- 2 Select the model file called **tutorial**, found in `matlab\toolbox\mbc\mbctraining`, and click **Open**.

CAGE imports the strategy.

- 3 To view the strategy, select **Feature > Graphical Strategy Editor**.

This opens the `Torque_Output` strategy in a Simulink window, and also the libraries for your project and other blocks for use with CAGE.

View the `Torque_Output` strategy.



The blocks show how the strategy is built up.

- 4 Close the Simulink windows.

CAGE displays the new `Torque_Output` feature parsed from Simulink as the output of the algebraic equation of tables. You can see this parsed into the **Strategy** pane as follows:

$$\text{Torque_Output} = T(\text{Norm_N}(N), \text{Norm_L}(L)) + F_A(A) + F_SPK(\text{SPK})$$

- 5 Select **View > Full Strategy Display** to turn off the full description and see this simplified expression:

$$\text{Torque_Output} = T + F_A + F_SPK$$

This shows the collection of tables that makes up the new feature — a torque table `T` (with normalizers in speed `N` and load `L`) combined with modifier tables depending


on the values of air/fuel ratio and spark. You will fill these tables by using CAGE to compare them with the torque model.

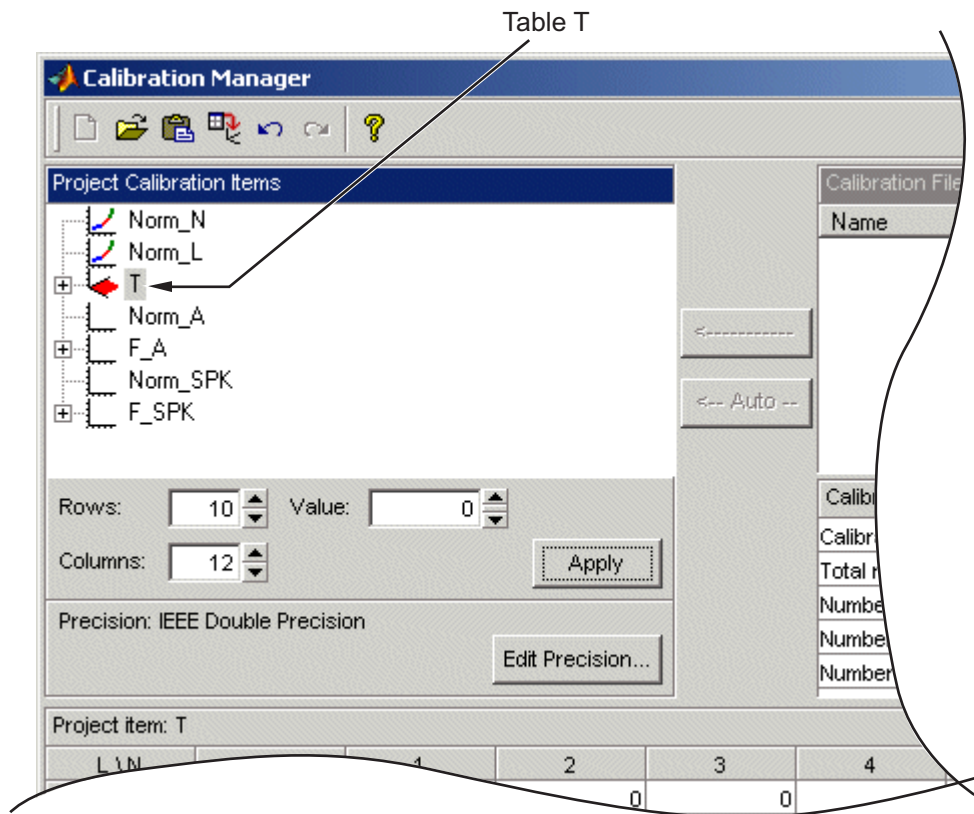
- 6 Click the plus next to `Torque_Output` to expand the Feature tree and observe the tables created by importing the strategy: `T`, `F_A`, and `F_SPK`. Expand each table node in turn to view the normalizers of each. You will define the sizes of the tables next.

For a more detailed description of strategies, see “What Is a Strategy?” in the CAGE documentation.

Set Up the Tables

Currently, the lookup tables have neither rows nor columns, so you must set up the tables.

Click Calibration Manager  or select **Tools > Calibration Manager**. The Calibration Manager dialog box opens, so you can specify the number of breakpoints for each axis.



To set up table T,

- 1 Highlight the table T by clicking T in the tree hierarchy.
- 2 Enter 10 as the number of rows and 12 as the number of columns. This determines the size of each normalizer.
- 3 Set the Value for each cell set to 0.
- 4 Click **Apply**, and click **Continue** in the dialog to change the size of the table. The pane changes to show the table is set up.
- 5 Follow the same procedure for the F_A table:
 - a Highlight the F_A node.
 - b Set the number of rows to be 10 and press Enter.

- c Leave the value for each cell set to 0.
 - d Click **Apply**.
- 6 Repeat step 5 for F_SPK.

Note The icons change as you initialize each table or function.

- 7 Click **Close** to leave the Calibration Manager.


After completing these steps, you can calibrate the lookup tables.

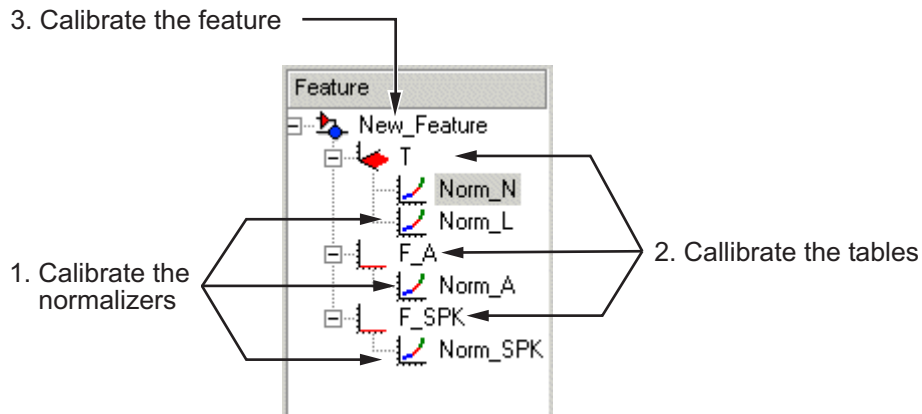
Process For Feature Calibration

The feature contains both a strategy (which is a collection of tables) and a model. You can use CAGE to fill the lookup tables using the model as a reference.

These are the three steps to calibrate a feature, described in these sections:

- 1 Calibrate the normalizers.
- 2 Calibrate the tables.
- 3 Calibrate the feature as a whole.

Click the expand icon, , to expand the nodes and display all the tables and normalizers in the feature.

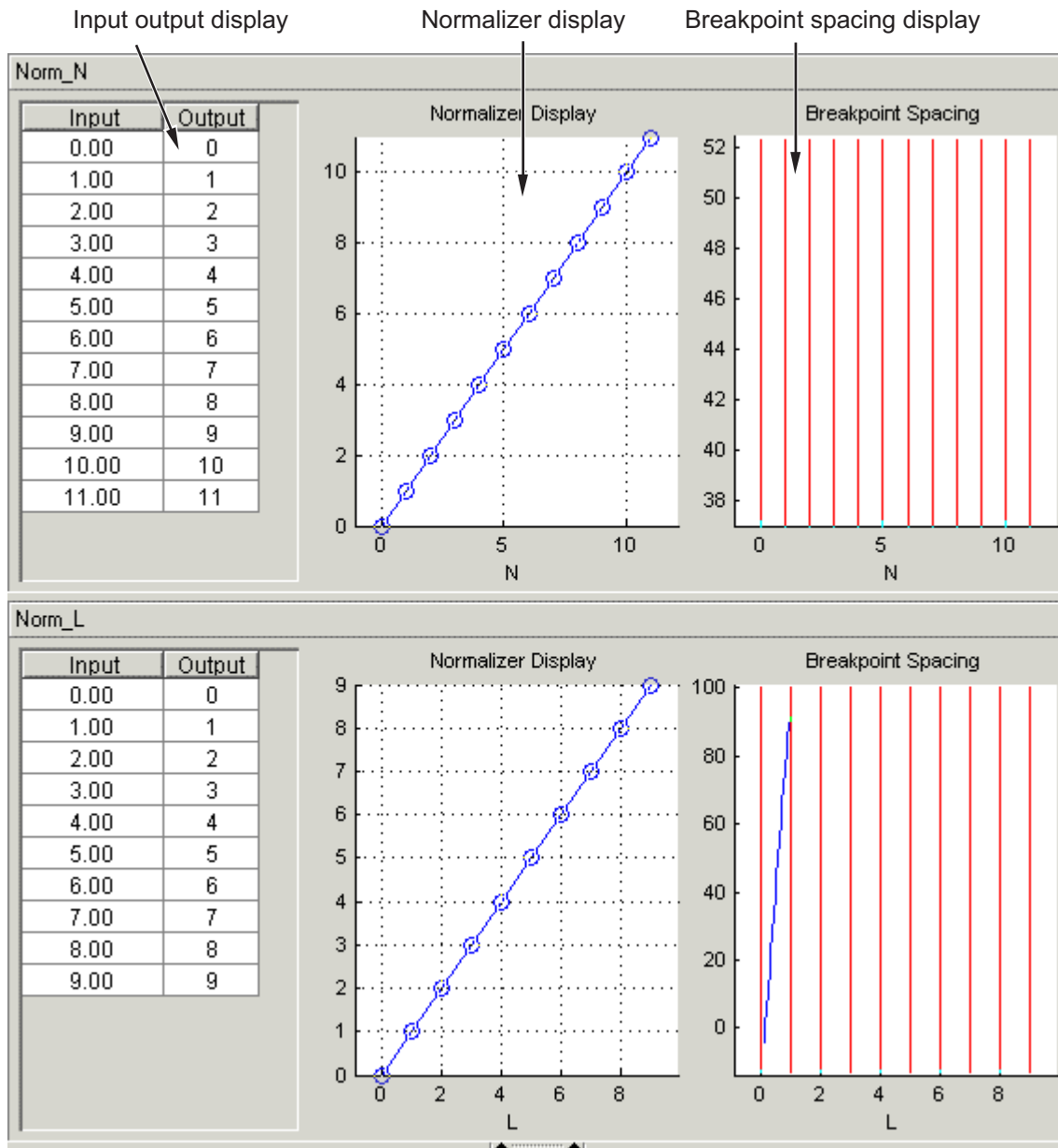


Each node in the display has a different view and different operations.

Calibrate the Normalizers

Normalizers are the axes for the lookup tables. Currently, Norm_N has 12 breakpoints; the other normalizers have 10 breakpoints each. This section describes how to set values for the normalizers Norm_N and Norm_L, based on the torque model, tq .

To display the Normalizer view, select the normalizer Norm_N in the branch display.



The Normalizer view has two panes, **Norm_N** and **Norm_L**.

In each pane, you see

- An input/output table
- A normalizer display
- A breakpoint spacing display

In both Normalizer panes, the Input Output table and the **Normalizer Display** show the position of the breakpoints.

The **Breakpoint Spacing** display shows a blue slice through the model with the break points overlaid as red lines.

For a more detailed description of the Normalizer view, see “Table Normalizers” in the CAGE documentation.

Placing the Breakpoints Automatically

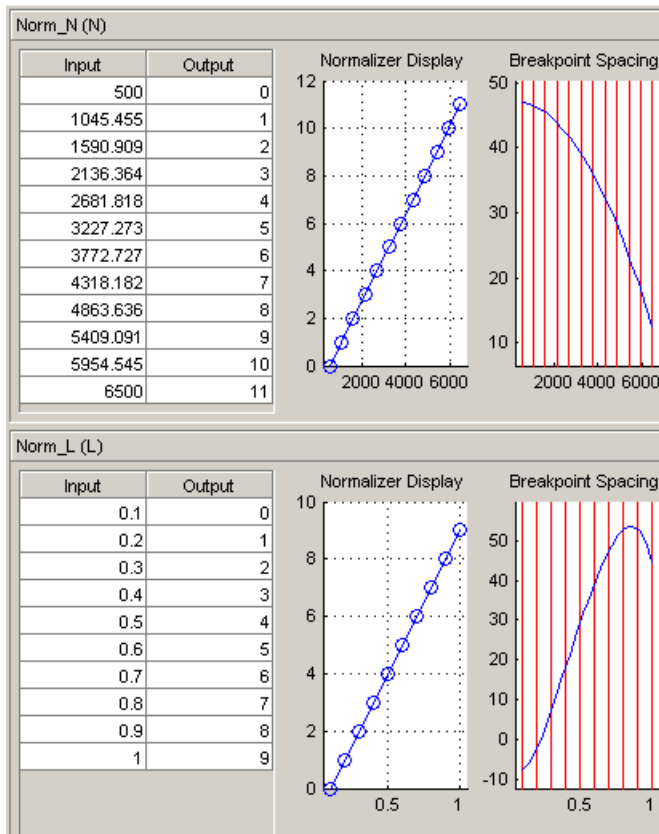
You now must space the breakpoints across the range of each variable. For example, Norm_N takes values from 500 to 6500, the range of the engine speed.

To space the breakpoints evenly throughout the data values,

- 1 Click Initialize  in the toolbar. Alternatively, select **Normalizer > Initialize**.

This opens a dialog box that suggests ranges for Norm_N and Norm_L.

- 2 To accept the default ranges of values of the data, click **OK**.



A better fit between model and table can often be achieved by spacing the breakpoints nonlinearly.

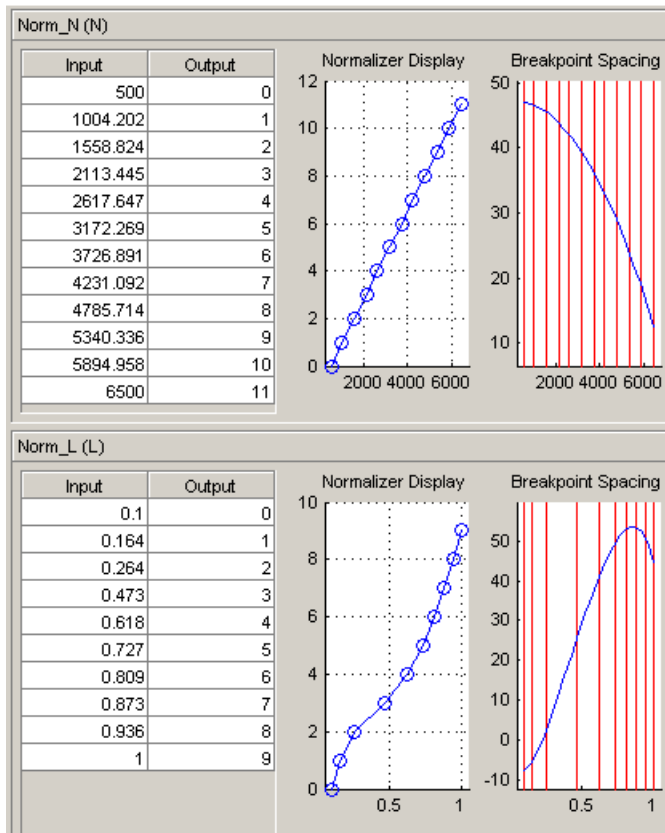
- 1 Click Fill  in the toolbar. Alternatively, select **Normalizer > Fill**.

This opens a dialog box that suggests ranges for Norm_N and Norm_L. It also suggests values for AFR and SPK; these values are the set points for AFR and SPK.

- 2 To accept the values in the dialog box, click **OK**.

This ensures that the majority of the breakpoints are where the model is most curved. The table now has most values where the model changes most. So, with the same number of breakpoints, the table is a better match to the model.

For more information about calibrating the normalizers, see “About Normalizers” in the CAGE documentation.



You can now calibrate the lookup tables; this is described in the next section.

Calibrate the Tables

The lookup tables currently have zero as the entry for each cell. This section demonstrates how to fill the table T with values of torque using the torque model, tq .

To view the Table display, click the T node.

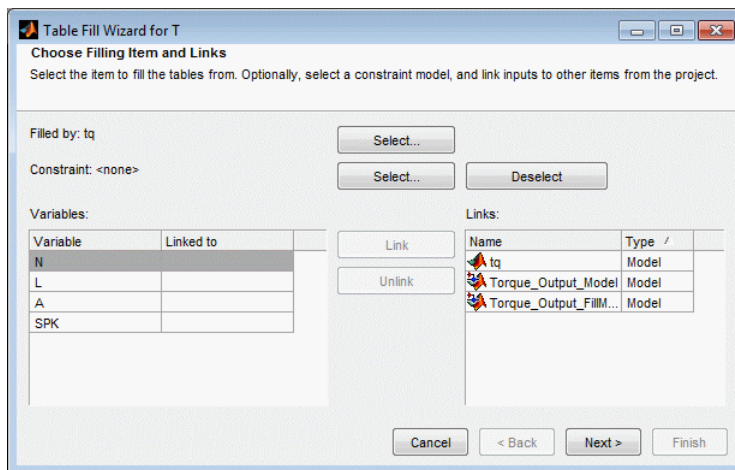
This view has two panes: the table and the graph of the table.

To fill the table with values of the model at the appropriate operating points,

- 1 Click Fill  on the toolbar.

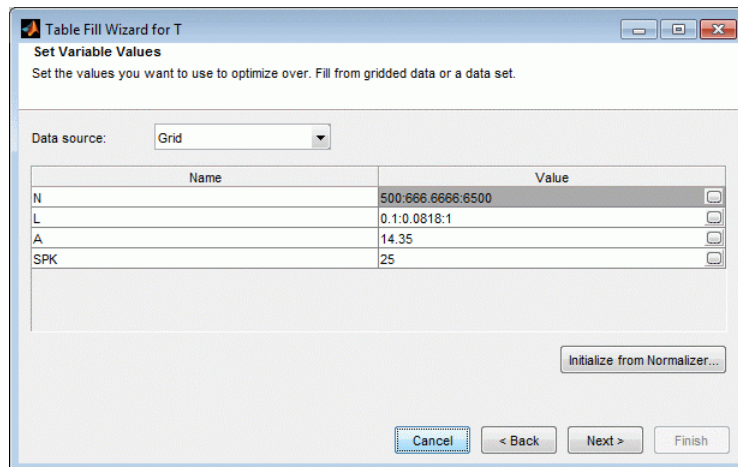
This opens the Feature Fill Wizard.

- 2 Observe that the `tq` model is selected to fill the table. Here you could also set up constraints, for example using a boundary model to constrain filling to table areas where data was collected, and you can link other models or features to inputs.



Leave the settings at the defaults and click **Next**.

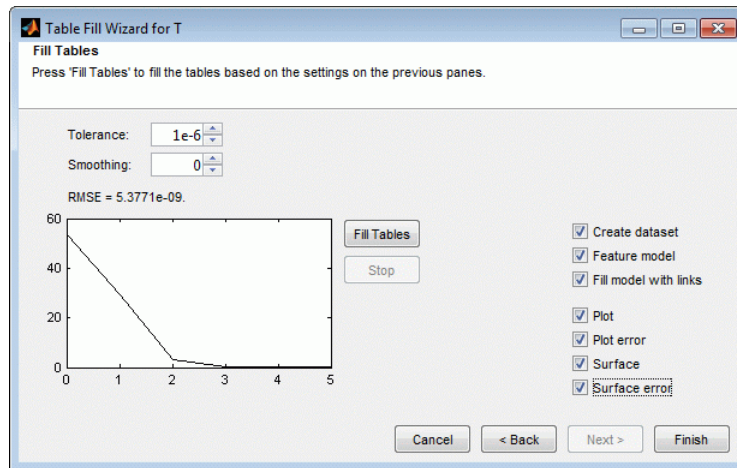
- 3 On this screen you can set variable values for optimizing over.



By default the table's normalizer breakpoints (here N and L) and the set points of the other variables (A and SPK) are selected. You can select different normalizers, and edit values to optimize over a range rather than at a single point. You can edit values directly in the **Value** edit box or click the button on the right in the **Value** box to open the Vector Editor dialog box. If you choose a range of values, CAGE fills the table using the average model value at each cell. If you choose **Initialize from Normalizer**, you can use the **Number of values between breakpoints** setting to add values between normalizer breakpoints to optimize over a finer grid than the number of table cells.

Leave the settings at the defaults and click **Next**.

- 4 Click **Fill Tables**. The graph shows the progress of the optimization.
- 5 Select all the check boxes.

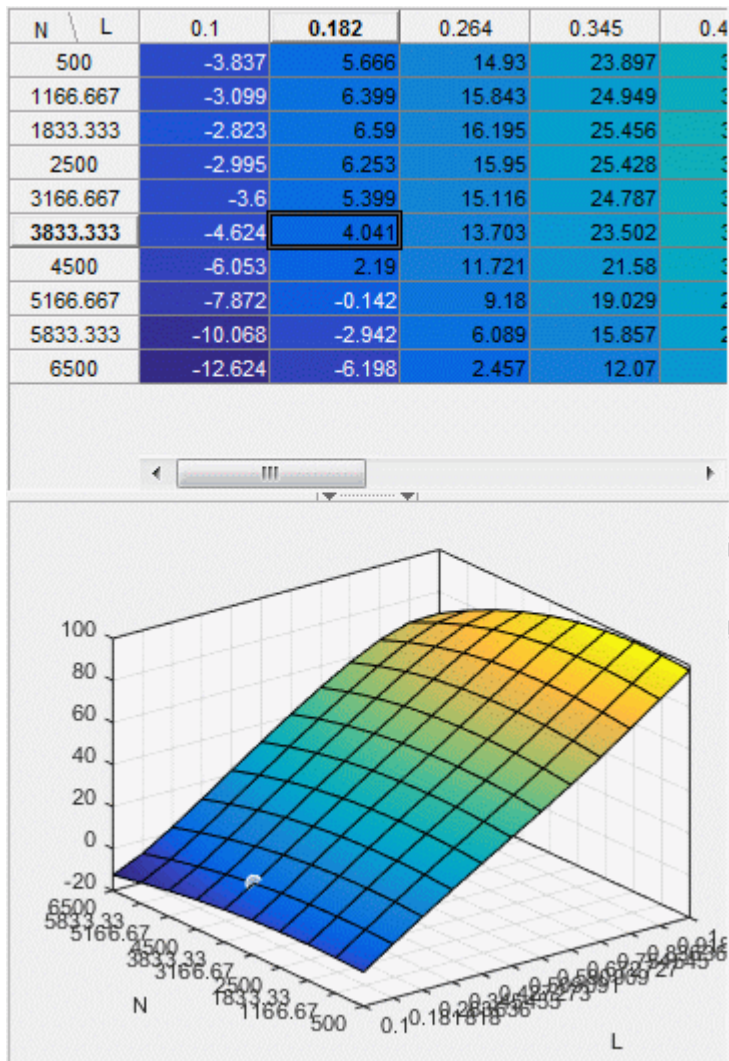


6 Click **Finish**.

CAGE creates plots of the filled table surface and the error between the table values and the model.

7 Click **OK**.

The following view shows the table filled with values of the model.



The table T is now filled with optimized values compared to the model at these operating points. CAGE runs an optimization routine over the feature to minimize the total square error between the model and the feature.

For more information about the process of filling tables, see “Optimize Table Values” in the CAGE documentation

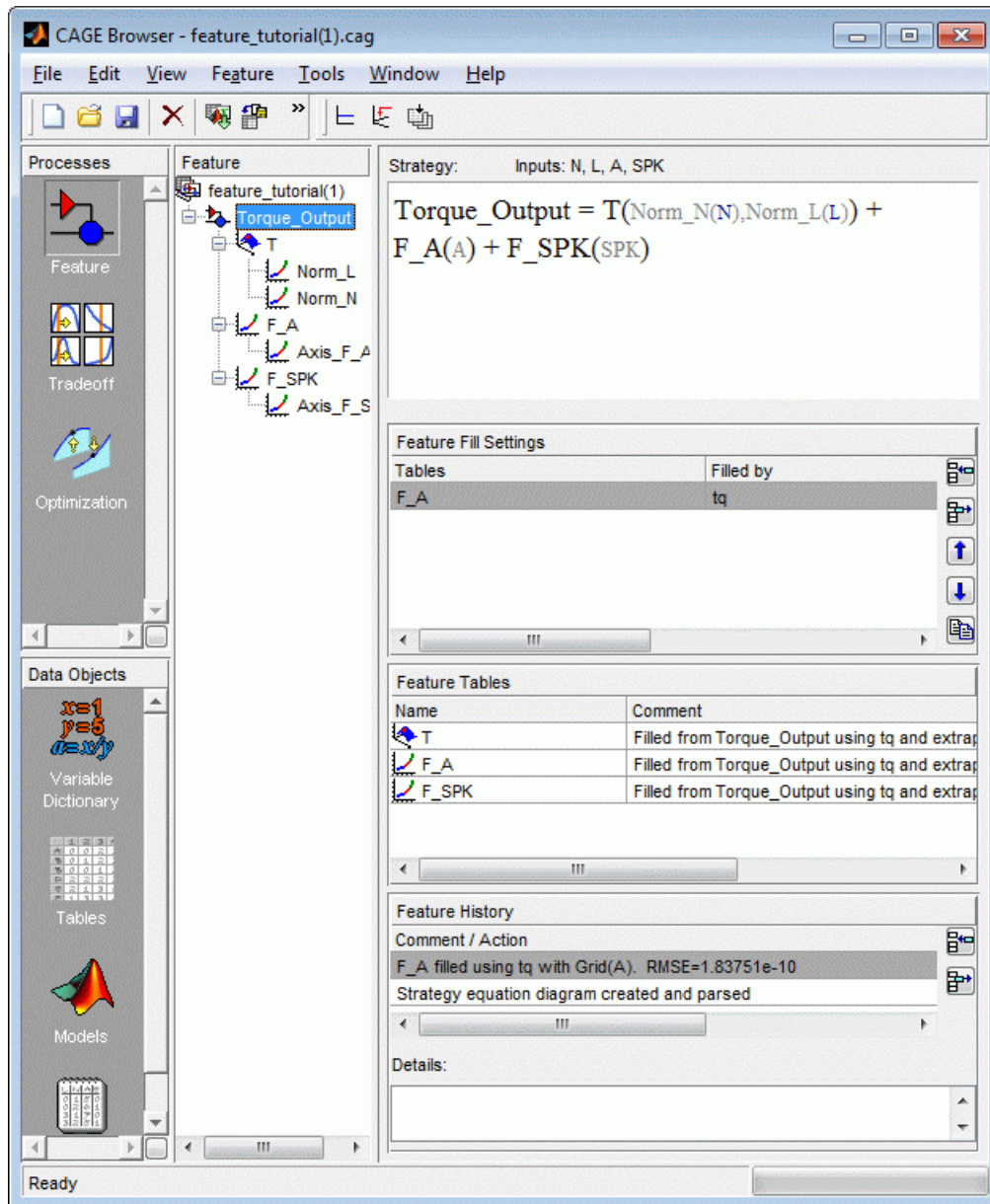
Now you must fill the tables F_A and F_SPK and their normalizers. The tables are modifiers for AFR and the spark angle respectively. These steps are described in the next section.

Calibrate the Feature

A feature is a strategy (which is a collection of tables) and a model. Currently the torque table, T, is filled with optimized values compared to the torque model, tq. You must now calibrate the normalizers and tables for F_A and F_SPK.

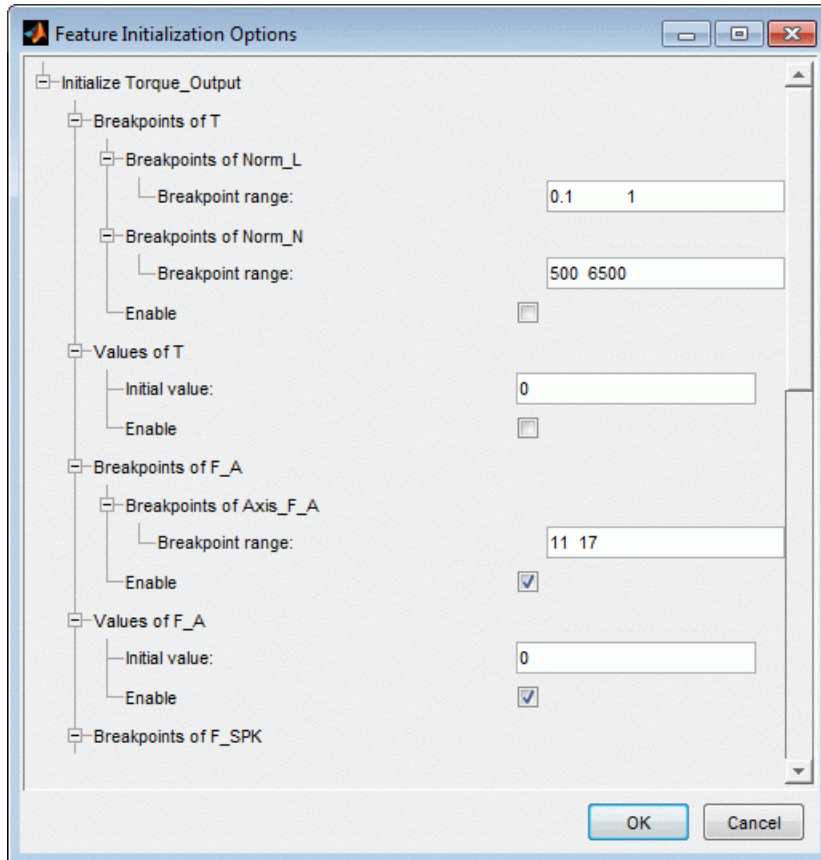
You could calibrate the normalizers and then the tables for F_A and F_SPK in turn. However, CAGE enables you to calibrate the entire feature in one procedure.

To view the Feature view following, click the Torque_Output node.




To calibrate all the tables and their normalizers,


- 1 Select **Feature > Initialize** (or use the Initialize toolbar button). The Feature Initialization Options dialog appears.
- 2 Clear the Enable check boxes for Breakpoints of T, and Values of T, as shown.



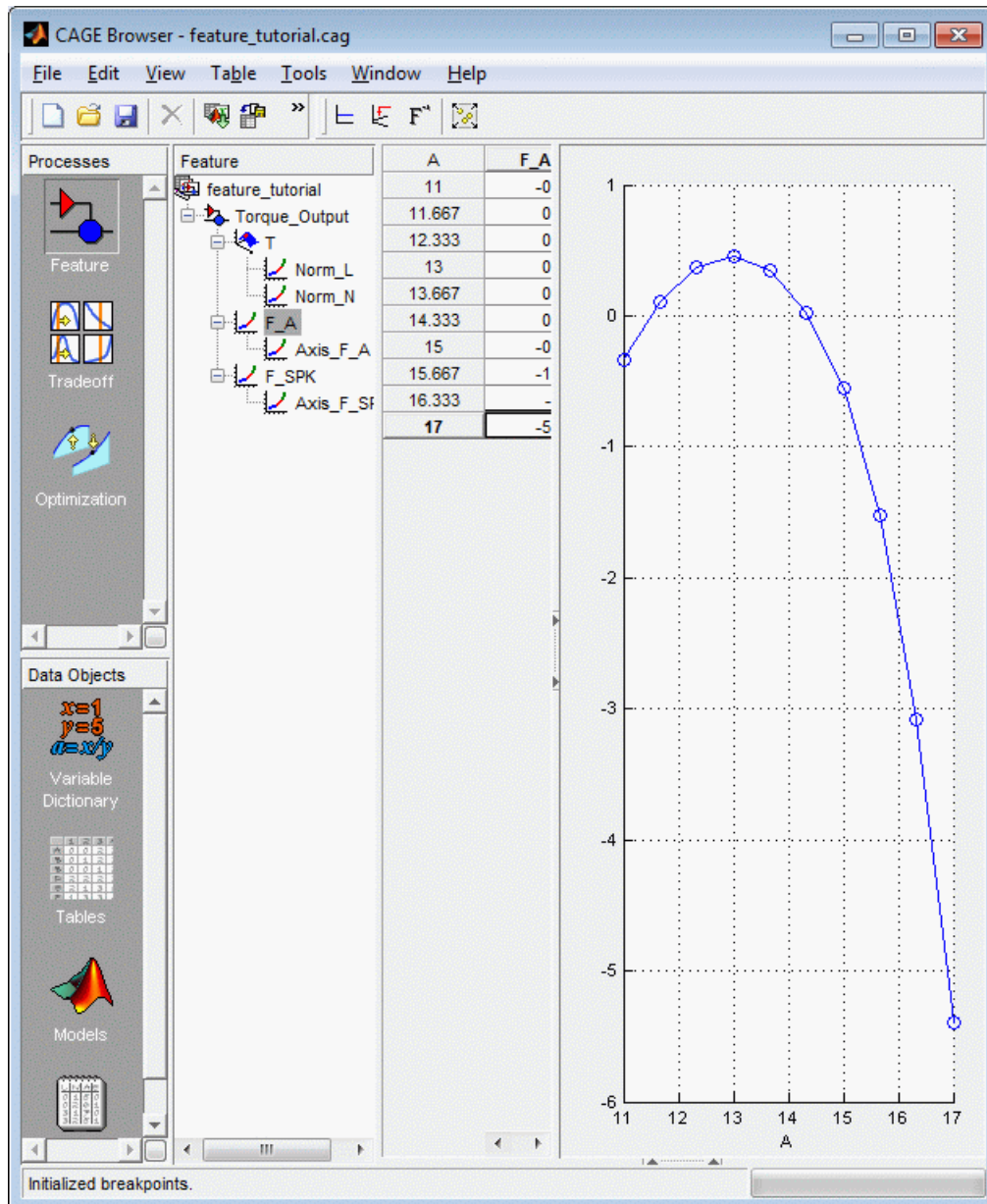
You have already optimized the breakpoints and table values for table T, so you only want to initialize the other tables F_A and F_SPK.

Click **OK**.

- 3 Select **Feature > Fill Feature** (or use the Fill  toolbar button) to open the Feature Fill Wizard.
- 4 Select only the F_A table to fill, and follow the steps in the wizard to fill this table:

- a** Click **Next** 3 times.
 - b** Click **Fill Tables**.
 - c** Click **Finish**.
- 5** Select the F_A table node to view the results.
- 6** Select F_SPK table node and click Fill . Repeat the wizard steps to fill the table.

All three tables and normalizers are filled.



This display shows that the range of the normalizer for `F_A` is 11 to 17, the range of AFR.

This completes the calibration of the torque feature.

For more information about calibrating features, see “Optimize Table Values” in the CAGE documentation.

You can now export the calibration.

Export Calibrations

To export your feature,

- 1 Select the `Torque_Output` node in the branch display.
- 2 Select **File > Export > Calibration > Selected Item**.
- 3 Choose the type of file you want to save your calibrations as. For the purposes of this tutorial, in the **Export to** list, select **Simple CSV file**, a comma separated value file. Click **OK**.
- 4 Enter `feature_tutorial.csv` as the file name and click **Save**.

This exports the calibration.

Note that when you choose to export **Selected Item** rather than **All Items**, what you export depends on which node is highlighted:

- Selecting a normalizer node outputs the values of the normalizer.
- Selecting a table node outputs the values of the table and its normalizers.
- Selecting a feature outputs the whole feature (all tables and normalizers).
- Selecting a branch node outputs all the features under the branch.

You have now completed the feature calibration tutorial.

Tradeoff Calibration

This section includes the following topics:

Tradeoff Calibration

In this section...
“What Is a Tradeoff Calibration?” on page 13-2
“Setting Up a Tradeoff Calibration” on page 13-2
“Performing the Tradeoff Calibration” on page 13-7

What Is a Tradeoff Calibration?

A tradeoff calibration is the process of filling lookup tables by balancing different objectives.

Typically there are many different and conflicting objectives. For example, a calibrator might want to maximize torque while restricting nitrogen oxides (NOX) emissions. It is not possible to achieve maximum torque and minimum NOX together, but it is possible to trade off a slight reduction in torque for a reduction of NOX emissions. Thus, a calibrator chooses the values of the input variables that produce this slight loss in torque over the values that produce the maximum value of torque.

This tutorial takes you through the various steps required for you to set up this tradeoff, and then to calibrate the lookup table for it.

Setting Up a Tradeoff Calibration

- “Creating a Tradeoff” on page 13-2
- “Adding Tables to a Tradeoff Calibration” on page 13-5
- “Displaying the Models” on page 13-6

Creating a Tradeoff

Start CAGE by typing

```
cage
```

at the MATLAB prompt.

Before you can calibrate the lookup tables, you must set up the calibration.

- 1 Select **File > Open Project** (or the toolbar button) to choose the `tradeoffInit.cag` file, found in the `matlab\toolbox\mbc\mbctraining` directory, then click **OK**.

The `tradeoffInit.cag` project contains two models and all the variables necessary for this tutorial. For information about how to set up models and variables, see “Calibration Setup” in the CAGE documentation.

- 2 To create a tradeoff calibration, select **File > New > Tradeoff**.

This takes you to the **Tradeoff** view. You need to add tables and display models to the tradeoff, which are described step by step in the following sections:

- “Adding Tables to a Tradeoff Calibration” on page 13-5.
- “Displaying the Models” on page 13-6 describes how you display the models of torque and NOX emissions.

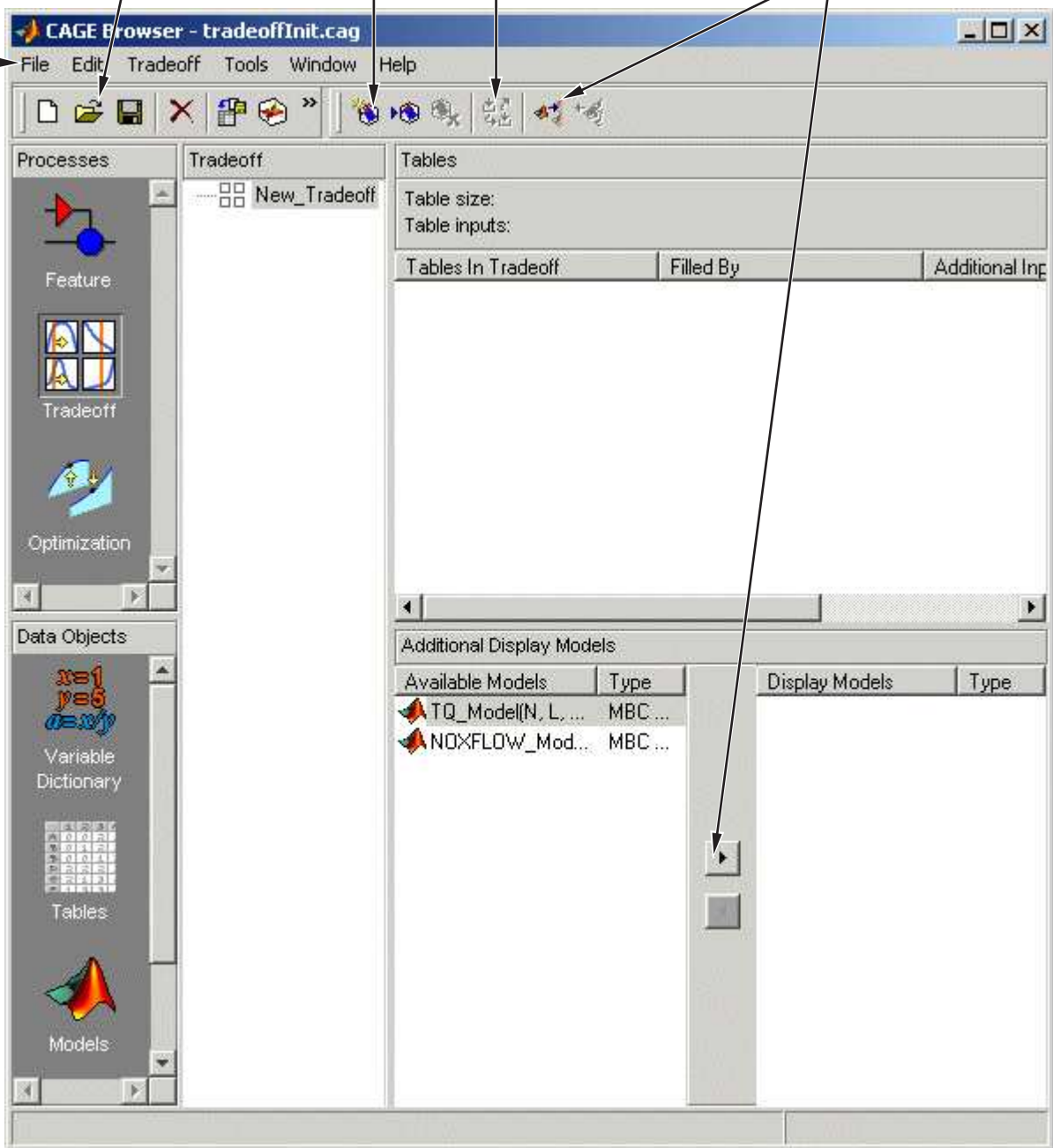
1. Open the project file

2. Add a new tradeoff

3. Add a new table

3. Select item to fill table.

5. Display the models



Adding Tables to a Tradeoff Calibration

The models of torque and NOX are in the current session. You must add the lookup table to calibrate.

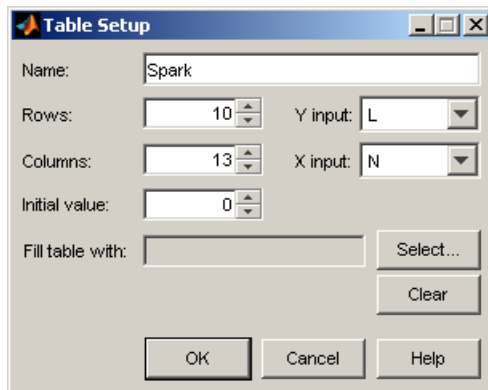
Both models have five inputs. The inputs for the torque and NOX models are

- Exhaust gas recycling (EGR)
- Air/fuel ratio (AFR)
- Spark angle
- Speed
- Load

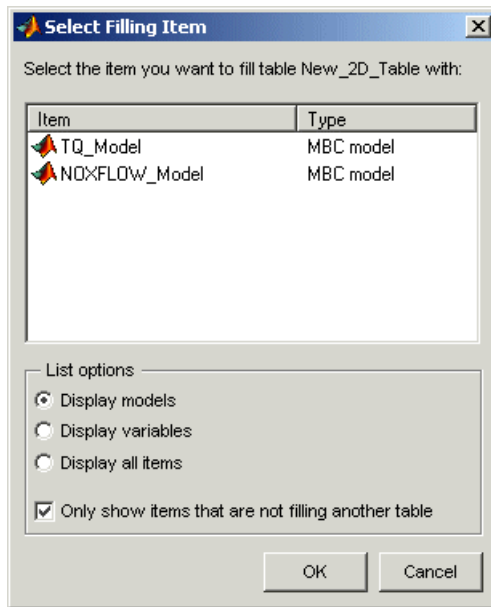
For this tutorial, you are interested in the spark angle over the range of speed and load.

To generate a lookup table for the spark angle,

- 1 Click  (Add New Table) in the toolbar. This opens the Table Setup dialog.



- 2 Enter **Spark** as the table **Name**.
- 3 Check that **N** is the **X input** and **L** is the **Y input** (these are selected automatically as the first two variables in the current Variable Dictionary).
- 4 Enter **10** as the size of the load axis (**Rows**).
- 5 Enter **13** as the size of the speed axis (**Columns**).
- 6 Click **Select** to open the dialog Select Filling Item.



Select the radio button **Display variables**, then select **SPK** to fill the table and click **OK**.

- 7 Click **OK** to close the Table Setup dialog.

Before you can perform the calibration, you must display the models.

Displaying the Models

For this tutorial, you are comparing values of the torque and NOX models. Thus, you need to display these models.

The method that you use to fill the lookup table is

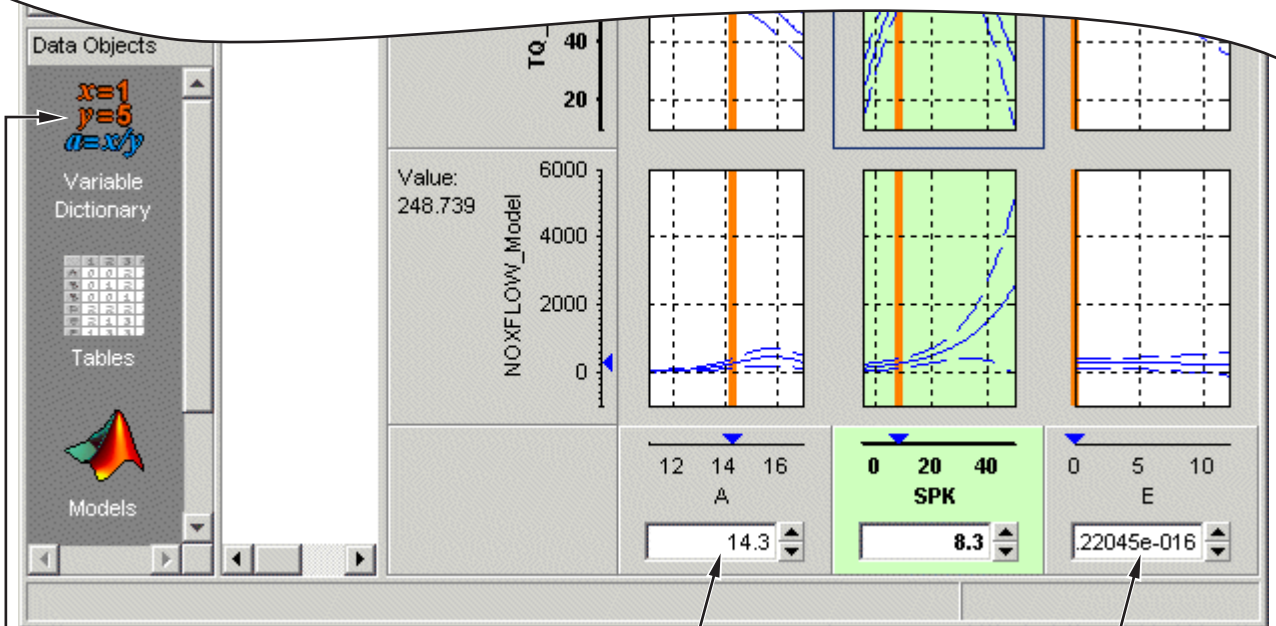
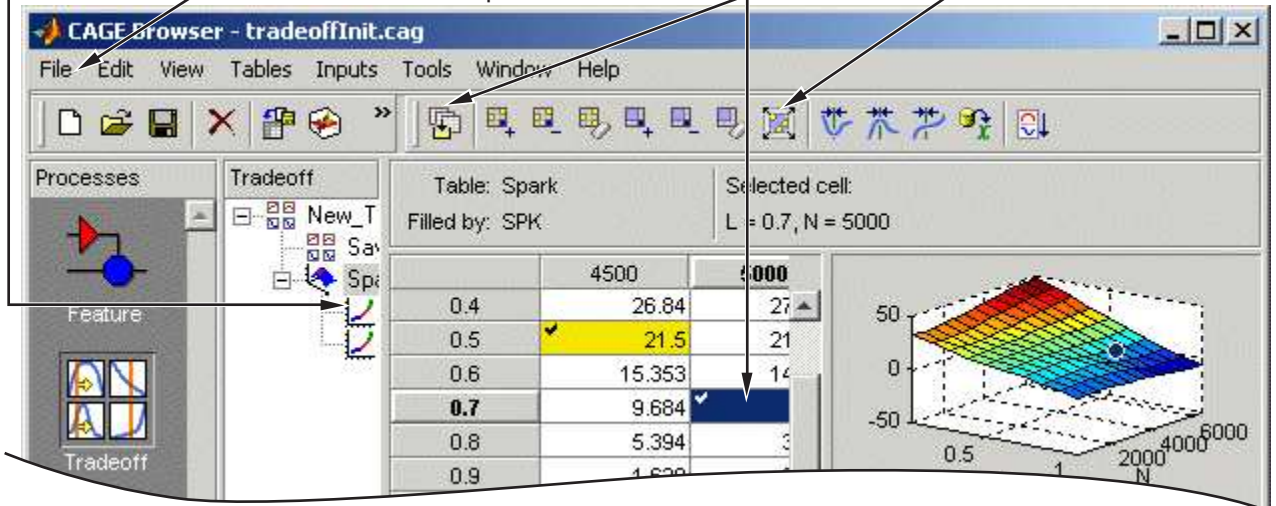
- Obtain the maximum possible torque.
- Restrict NOX to below 250 g/hr at any operating point.

To perform the tradeoff calibration, follow the instructions in the next four sections:

- 1 Check the normalizers.
- 2 Set values for the other variables, AFR and EGR.
- 3 Fill key operating points with values for spark angle.
- 4 Fill the table by extrapolation.

Once you have completed the calibration, you can export the calibration for use in the electronic control unit.

1. Check the normalizers.
2. Set values for the other variables, either individually for each operating point or in the Variable Directory.
3. Trade off torque and **NOX** to find values of spark to fill key operating points in the table.
4. Fill the table by extrapolation.
5. Export the calibration.

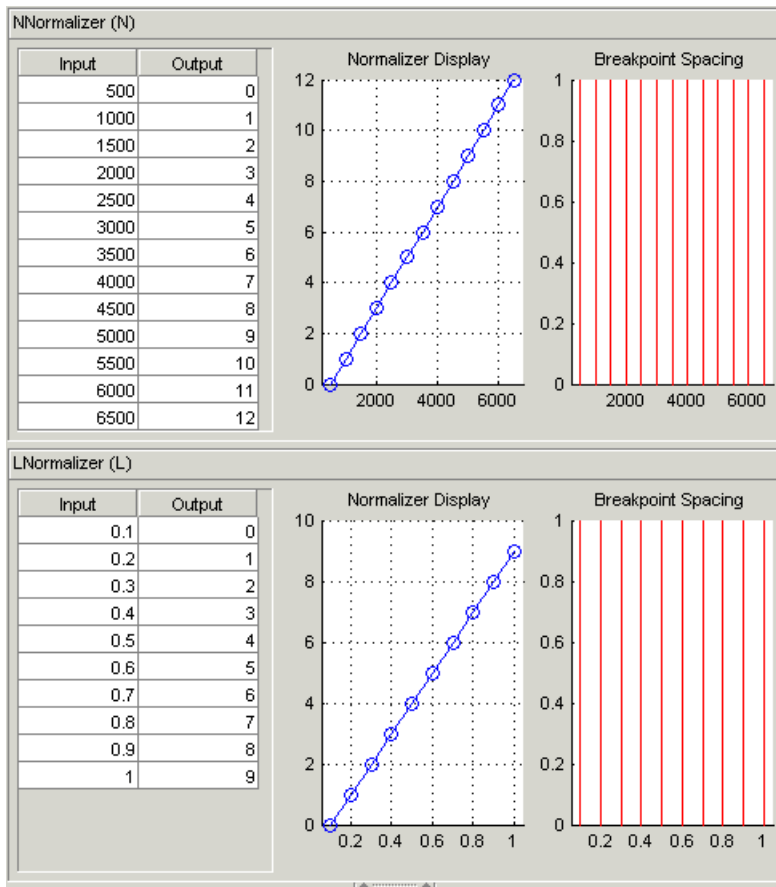


2. Set values for the other variables, either individually for each operating point or in the Variable Directory.

Checking the Normalizers

A normalizer is the axis of the lookup table (which is the collection of breakpoints). The breakpoints of the normalizers are automatically spaced over the ranges of speed and load. These define the operating points that form the cells of the tradeoff table.

Expand the Tradeoff tree by clicking the plus sign in the display, so you can see the Spark table and its normalizers Speed and Load. Click to highlight either normalizer to see the normalizer view. A tradeoff calibration does not compare the model and the table directly, so you cannot space the breakpoints by reference to the model.



Setting Values for Other Variables

At each operating point, you must fill the values of the spark table. Both of the models depend on spark, AFR (labeled A, in the session), and EGR (labeled E in the session). You could set the values for AFR and EGR individually for each operating point in the table, but for simplicity you will set constant values for these model inputs.

To set constant values of AFR and EGR for all operating points,

- 1 Click **Variable Dictionary** in the **Data Objects** pane.
- 2 Click A and edit the **Set Point** to **14.3**, the stoichiometric constant, and press **Enter**.
- 3 Click E and change the **Set Point** to **0** and press **Enter**.

You have set these values for every operating point in your tradeoff table. You can now fill the spark angle lookup table. The process is described next.

- 4 Click **Tradeoff** in the **Processes** pane to return to the tradeoff view.
- 5 Highlight the **Spark** table node in the Tradeoff tree display.
- 6 In the lower pane, check that the value for A is **14.3**, and the value for E is **0**, as shown in the following example. You leave these values unchanged for each operating point.

For each operating point you change the values of spark to trade off the torque and NOX objectives; that is, you search for the best value of spark that gives acceptable torque within the emissions constraint. The following example illustrates the controls you use, and there are step-by-step instructions in the following section.

1. Highlight the **Spark** node.

2. Select operating points in the **Spark** tradeoff table.

The screenshot shows the Tradeoff Calibration software interface. At the top left, the 'Tradeoff' menu is open, and the 'Spark' node is highlighted. The main window displays a table titled 'Table: Spark' with columns for '500', '1000', and '1500'. The '500' column is selected, and the value '0' is entered in the first row. To the right of the table is a 3D plot showing a green surface representing the tradeoff between variables. Below the table, there are three 2D plots for 'TQ_Model', 'NOXFLOW_Mode', and 'A'. The 'TQ_Model' plot shows a blue curve with a vertical orange line at approximately 14.3. The 'NOXFLOW_Mode' plot shows a blue curve with a vertical orange line at approximately 0. The 'A' plot shows a blue curve with a vertical orange line at approximately 14.3. At the bottom, there are three input fields for 'A', 'SPK', and 'E'. The 'A' field is set to 14.3, the 'SPK' field is set to 0, and the 'E' field is set to 0. The 'SPK' field is highlighted in green.

	500	1000	1500
0.1	0	0	0
0.2	0	0	0
0.3	0	0	0
0.4	0	0	0
0.5	0	0	0

Value: 5.48987

Value: 0.72677

12 14 16
A
14.3

0 20 40
SPK
0

0 5 10
E
0

3. Change values of **SPK** to trade off torque and NOX emissions at each operating point.

4. Leave A and E at the set point values.

Filling Key Operating Points

You now fill the key operating points in the lookup table for spark angle.

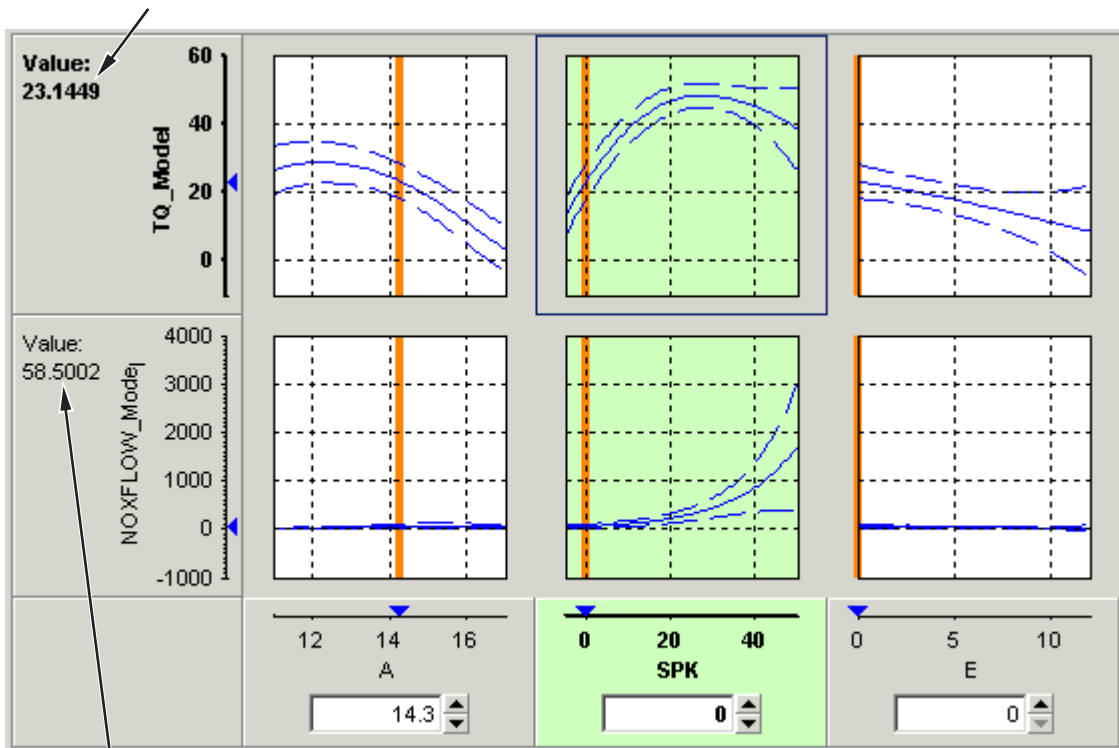
The upper pane displays the lookup table, and the lower pane displays the behavior of the torque and NOX emissions models with each variable.

The object is to maximize the torque and restrict NOX emissions to below 250 g/hr.

Determining the Value of Spark

At each operating point, the behavior of the model alters. The following display shows the behavior of the models over the range of the input variables at the operating point selected in the table, where speed (N) is 4500 and load (L) is 0.5. You can show confidence intervals by selecting **View > Display Confidence Intervals**.


Value of the torque model

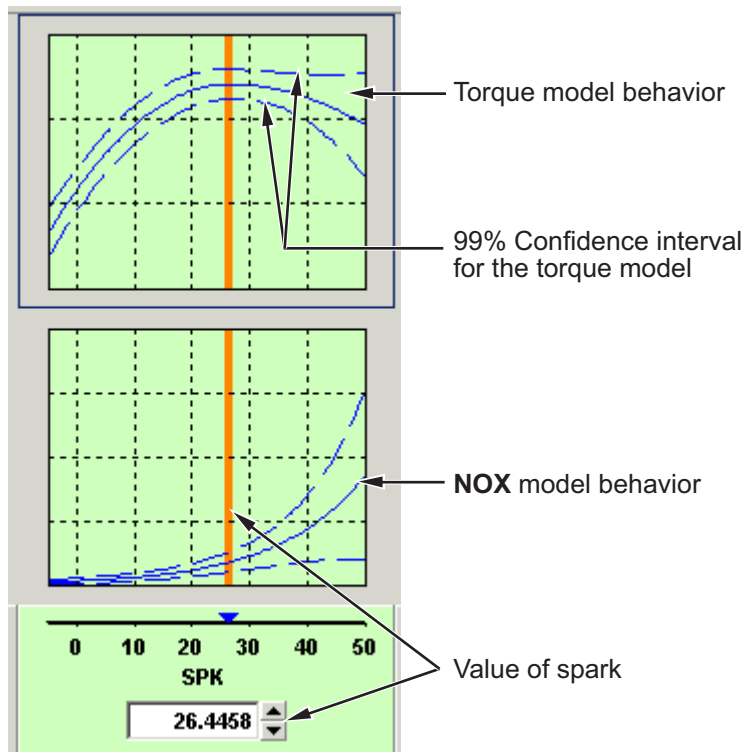


Value of the **NOX** model

The top three graphs show how the torque model varies with the AFR (labeled A), the spark angle (SPK), and the EGR (E), respectively. The lower panes show how the NOX emissions model varies with these variables.

You are calibrating the Spark table, so the two spark (SPK) graphs are green, indicating that these graphs are directly linked to the currently selected lookup table.

- 1 Select the operating point $N = 4500$ and $L = 0.5$ in the lookup table.
- 2 Now try to find the spark angle that gives the maximum torque and restricts NOX emissions to below 250 g/hr. You can change the value of spark by clicking and dragging the orange line on the SPK graphs, or by typing values into the SPK edit box. You can change the values of any of the other tradeoff variables in the same way, but as you have already set constant values for A and E you should not change these. Try different values of spark and look at the resulting values of the torque and NOX models.
- 3 Click to select the top SPK - TQ_Model graph (TQ_Model row, SPK column). When selected the graph is outlined as shown in the following example.
- 4 Now click 'Find maximum of output' () in the toolbar. This calculates the value of spark that gives the maximum value of torque. The following display shows the behavior of the two models when the spark angle is 26.4458, which gives maximum torque output.




At this operating point, the maximum torque that is generated is 48.136 when the spark angle is 26.4989 . However, the value of NOX is 348.968 , which is greater than the restriction of 250 g/hr. Clearly you have to look at another value of spark angle.


- 5 Click and drag the orange bar to change to a lower value of spark. Notice the change in the resulting values of the torque and NOX models.
- 6 Enter 21.5 as the value of **SPK** in the edit box at the bottom of the SPK column.

The value of the NOX emissions model is now 249.154 . This is within the restriction, and the value of torque is 47.2478 .

At this operating point, this value of 21.5 degrees is acceptable for the spark angle lookup table, so you want to apply this point to your table.

- 7 Press **Ctrl+T** or click  (Apply table filling values) in the toolbar to apply that value to the spark table.

This automatically adds the selected value of spark to the table and turns this cell yellow. It is blue when selected, yellow if you click elsewhere. Look at the table legend to see what this means: yellow cells have been added to the extrapolation mask, and the tick mark indicates you saved this input value by applying it from the tradeoff. You can use the **View** menu to choose whether to display the legend.

- 8 Now repeat this process of finding acceptable values of spark at four more operating points listed in the table following. In each case,
- Select the cell in the spark table at the specified values of speed and load.
 - Enter the value of spark given in the table (the spark angles listed all satisfy the requirements).
 - Press **Ctrl+T** or click  (Apply table filling values) in the toolbar to apply that value to the spark table.


Speed, N	Load, L	Spark Angle, SPK
2500	0.3	25.75
3000	0.8	10.7
5000	0.7	8.2
6000	0.2	41.3

After you enter these key operating points, you can fill the table by extrapolation. This is described in the next section.

Filling the Table by Extrapolation

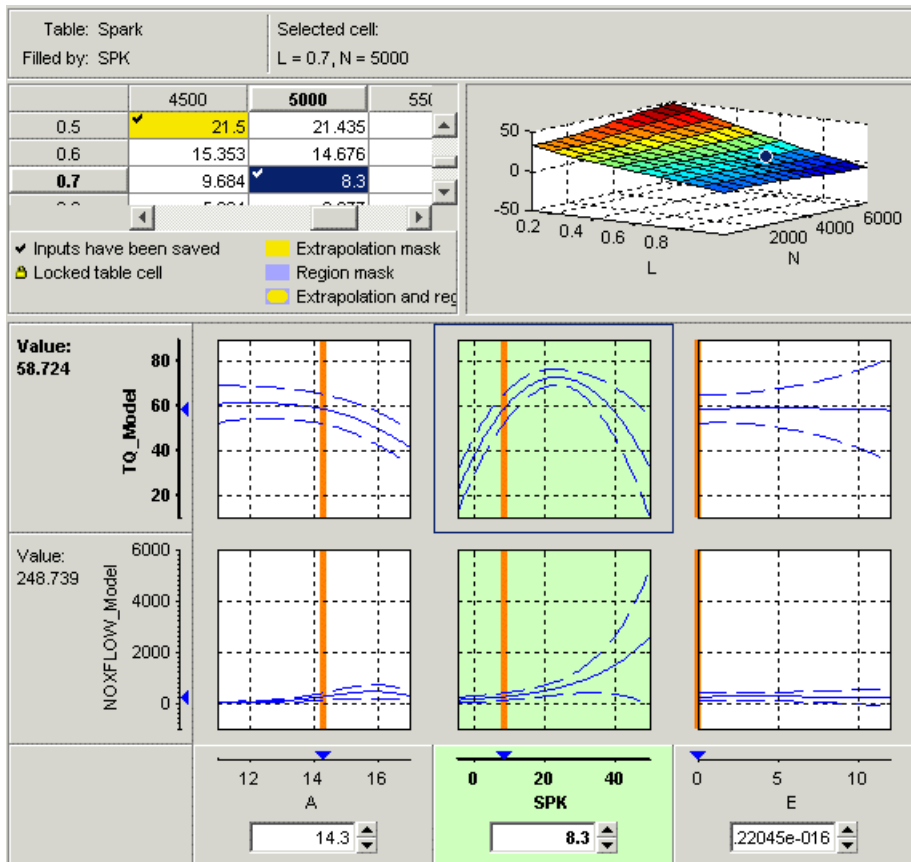
When you have calibrated several key operating points, you can produce a smooth extrapolation of these values across the whole table.

When you apply the value of the spark angle to the lookup table, the selected cell is automatically added to the extrapolation mask. This is why the cell is colored yellow. The extrapolation mask is the set of cells that are used as the basis for filling the table by extrapolation.

Click  in the toolbar to fill the table by extrapolation.

The lookup table is filled with values of spark angle.

The following figure displays the view after extrapolation over the table.



Note Not all the points in the lookup table will necessarily fulfill the requirements of maximizing torque and restricting the NOx emissions.

You could use these techniques to further improve the calibration and trade off torque and NOx to find the best values for each cell in the spark table.

For a more detailed description of tradeoff calibrations, see "Tradeoff Calibration".

You can now export this calibration to file.

Exporting Calibrations

To export your table and its normalizers,

- 1** Select the **Spark** node in the branch display.
- 2** Select **File > Export > Calibration**.
- 3** Choose the file type you want for your calibration. For the purposes of this tutorial, select **Comma Separated Value (.csv)**.
- 4** Enter **tradeoff.csv** as the file name and click **Save**.

This exports the spark angle table and the normalizers, **Speed** and **Load**.

You have now completed the tradeoff calibration tutorial.

Data Sets

This section includes the following topics:

Compare Calibrations To Data

In this section...

- “Setting Up the Data Set” on page 14-2
- “Comparing the Items in a Data Set” on page 14-7
- “Reassigning Variables” on page 14-14

Setting Up the Data Set

- “Tutorial Overview” on page 14-2
- “Opening an Existing Calibration” on page 14-3
- “Importing Experimental Data into a Data Set” on page 14-3
- “Adding an Item to a Data Set” on page 14-6

Tutorial Overview

You can use the **Data Sets** view in CAGE to compare features, tables, and models with experimental data. You can use data sets to plot the features, tables, etc., as tabular values or as plots on a graph.

Data sets enable you to view the data at a set of operating points. You can determine the set of operating points yourself, using Build Grid. Alternatively, you can import a set of experimental data taken at a series of operating points. These operating points are not the same as the breakpoints of your tables.

This tutorial takes you through the basic steps required to compare a completed feature calibration to a set of experimental data.

Start CAGE by typing

```
cage
```

at the MATLAB prompt.

To set up the data set tutorial, you need to

- 1 Open an existing calibration.
- 2 Import the experimental data.

- 3 Add the **Torque** feature to the data set.

Your data set contains all the input factors and output factors required. As the imported data contains various operating points, this information is also included in the data set.

The next sections describe these processes in more detail.

Opening an Existing Calibration

For this tutorial, use the file `datasettut.cag`, found in the `matlab\toolbox\mbc\mbctraining` directory.

To open this file,

- 1 Select **File > Open Project**.
- 2 In the file browser, select `datasettut.cag` and click **Open**.

This opens a file that contains a complete calibrated feature with its associated models and variables. This particular feature is a torque calibration, using a torque table (labeled T1) and modifiers for spark (labeled T2) and air/fuel ratio (labeled T3).

For information about completing a feature calibration, see “Feature Calibration”.

- 3 Select **File > New > Data Set** to add a new data set to your session.

This automatically switches you to the **Factor Information** pane of the data set display.

Importing Experimental Data into a Data Set

To import data into a data set,

- 1 Select **File > Import > Data > File**.
- 2 In the file browser, select `meas_tq_data.xls` from the `mbctraining` directory, and click **Open**.

This set of data includes six columns of data, the test cell settings for engine speed (RPM), and the measured values of torque (`tqmeas`), engine speed (`nmeas`), air/fuel ratio (`afrmeas`), spark angle (`spkmeas`), and load (`loadmeas`).

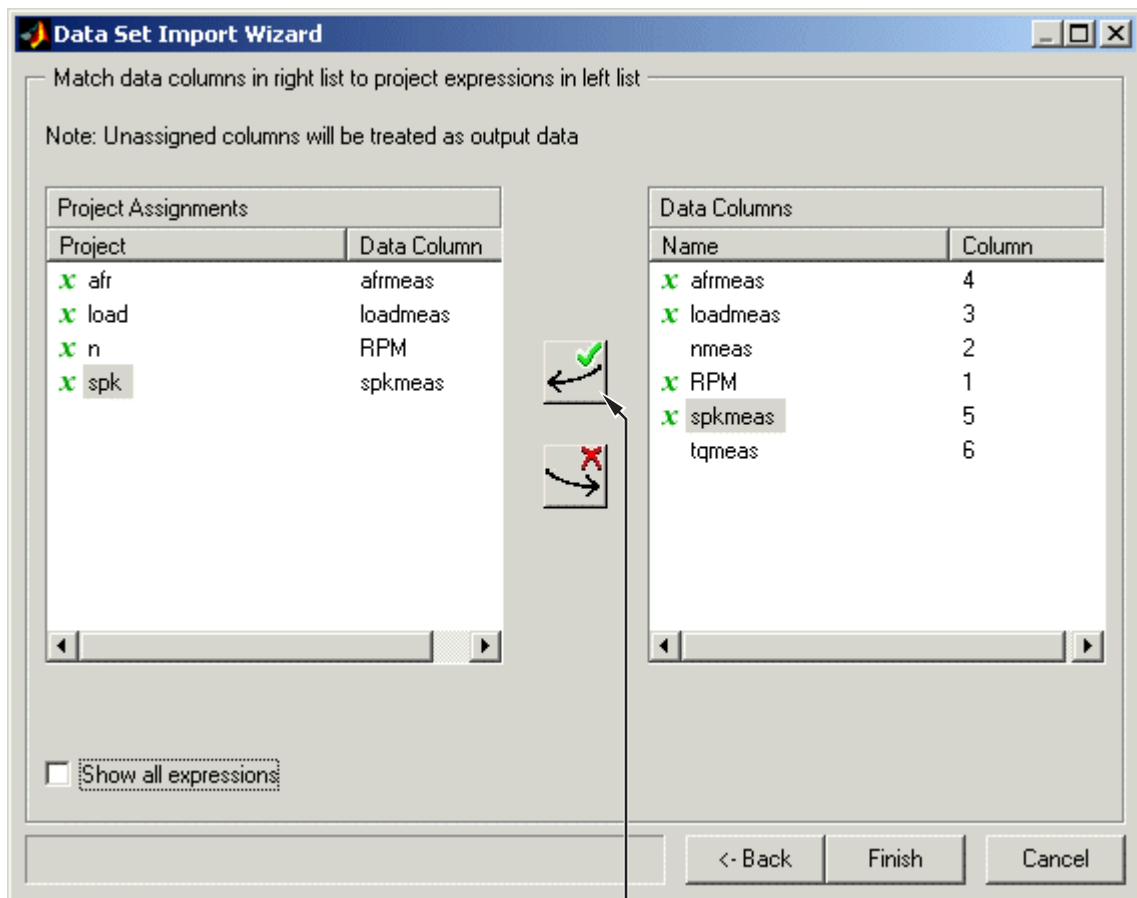
- 3 The Data Set Import Wizard asks which of the columns of data you would like to import. Click **Next** to import them all.

The following screen asks you to associate variables in your project with data columns in the data.

- 4 Highlight `afr` in the **Project Assignments** column and `afrmeas` in the **Data Column**, then click the assign button, shown.




- 5 Repeat this to associate `load` with `loadmeas`, `n` with `RPM`, and `spk` with `spkmeas`. The dialog box should be the same as shown.



Assign button

- 6 Click **Finish** to close the dialog box.

Note If you need to reassign any inputs after closing this dialog you can click  in the toolbar or select **Data > Assign**.

Adding an Item to a Data Set

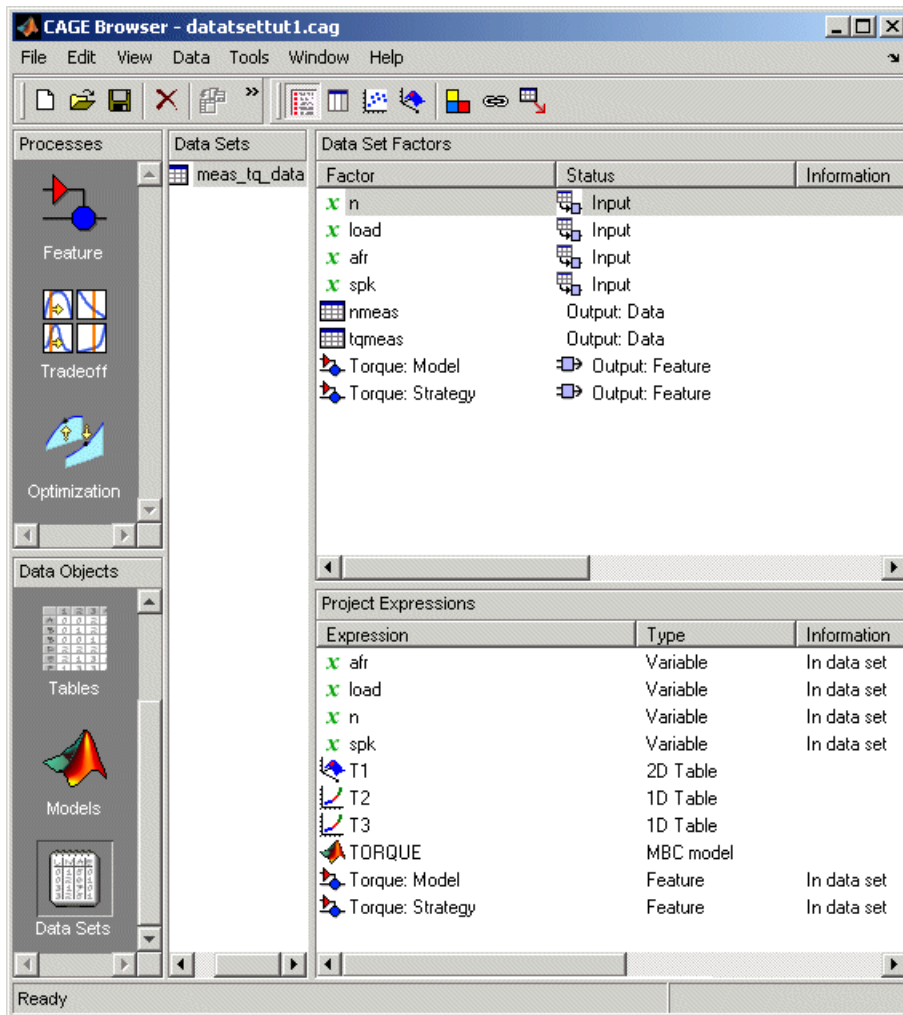
To add the Torque feature to the data set,

- 1 Highlight the Torque feature in the lower list of **Project Expressions**.
- 2 Select **Data > Factors > Add to Data Set**.

This adds two objects to the data set: **Torque: Model** and **Torque: Strategy**. These two objects make up the Torque feature.

- **Torque: Model** is the model used as a reference point to calibrate the feature.
- **Torque: Strategy** is the values of the feature at these operating points.

When these steps are complete, the list of factors includes four input factors and four output factors, as shown.










Comparing the Items in a Data Set

- “Viewing the Data Set as a Table” on page 14-8
- “Viewing the Data Set as a Plot” on page 14-9
- “Displaying the Error” on page 14-10
- “Coloring the Display” on page 14-12

Viewing the Data Set as a Table

By viewing the data set, you can compare experimental data with calibrations or models in your project.

Click  in the toolbar to view the data set as a table of values.








	 n	 load	 afr	 spk	nmeas	tqmeas	 Torque: Model	 Torque: Strategy
1	2235	0.549	9.5	0.1	2247	66.7	71.666	66.079
2	3591	0.454	13.2	0.1	3613	54.1	47.163	46.891
3	4946	0.651	12	0.1	4974	73.7	47.573	79.256
4	881	0.648	11.9	5.7	881	75.8	99.23	80.211
5	2234	0.441	13.3	0.1	2247	55.9	51.256	45.152
6	3591	0.747	10.9	0.1	3612	90	92.837	105.586
7	4947	0.541	9.7	0.1	4973	62.8	57.76	57.587
8	881	0.622	9.9	0.1	884	72.1	76.198	60.926
9	1219	0.333	14	0.1	1224	41.8	33.226	21.318
10	1558	0.382	12	0.1	1567	49.4	40.487	31.957
11	1896	0.209	10.7	3.3	1906	28.5	3.492	4.197
12	2234	0.284	9.8	3.2	2245	36	23.063	19.891
13	2574	0.407	13.4	3	2588	49.9	49.629	44.794
14	2914	0.595	11.5	3.1	2929	70.5	84.68	82.229
15	3251	0.781	12.3	3.1	3268	90.5	117.424	117.259
16	3589	0.668	13.5	3	3608	77.1	87.987	96.408
17	3930	0.452	11.9	3.1	3952	52.7	46.511	51.722
18	4268	0.235	10.9	3	4293	27.7	5.253	3.085
19	4606	0.194	12	3.2	4633	21.3	-2.088	-5.771

In the table, the input cells are white and the output cells are grey. Select the **Torque: Strategy** column header to see the view shown. The selected column turns blue and the column headers of the strategy's inputs (**n**, **load**, **afr** and **spk**) turn cream. Column headers are always highlighted in this way when they are associated with the currently selected column (such as model inputs, strategy inputs or linked columns).

In addition to viewing the columns, you can use data sets to create a column that shows the difference between two columns:

- 1 Select the **tqmeas** and **Torque: Strategy** columns by using **Ctrl+click**.
- 2 Select **Create Error** from the right-click menu on either column header.

This creates another column that is the difference between **tqmeas** and **Torque: Strategy**. Note that all the columns that are inputs to this new column have highlighted headers.

	 n	 load	 afr	 spk	nmeas	tqmeas	 Torque: Model	 Torque: Strategy	 tqmeas minus Torque
1	2235	0.549	9.5	0.1	2247	66.7	71.666	66.079	-0.621
2	3591	0.454	13.2	0.1	3613	54.1	47.163	46.891	-7.209
3	4946	0.651	12	0.1	4974	73.7	47.573	79.256	5.556
4	881	0.648	11.9	5.7	881	75.8	99.23	80.211	4.411
5	2234	0.441	13.3	0.1	2247	55.9	51.256	45.152	-10.748
6	3591	0.747	10.9	0.1	3612	90	92.837	105.586	15.586
7	4947	0.541	9.7	0.1	4973	62.8	57.76	57.587	-5.213
8	881	0.622	9.9	0.1	884	72.1	76.198	60.926	-11.174
9	1219	0.333	14	0.1	1224	41.8	33.226	21.318	-20.482
10	1558	0.382	12	0.1	1567	49.4	40.487	31.957	-17.443
11	1896	0.209	10.7	3.3	1906	28.5	3.492	4.197	-24.303
12	2234	0.284	9.8	3.2	2245	36	23.063	19.891	-16.109
13	2574	0.407	13.4	3	2588	49.9	49.629	44.794	-5.106
14	2914	0.595	11.5	3.1	2929	70.5	84.68	82.229	11.729
15	3251	0.781	12.3	3.1	3268	90.5	117.424	117.259	26.759
16	3589	0.668	13.5	3	3608	77.1	87.987	96.408	19.308
17	3930	0.452	11.9	3.1	3952	52.7	46.511	51.722	-0.978
18	4268	0.235	10.9	3	4293	27.7	5.253	3.085	-24.615
19	4606	0.194	12	3.2	4633	21.3	-2.088	-5.771	-27.071

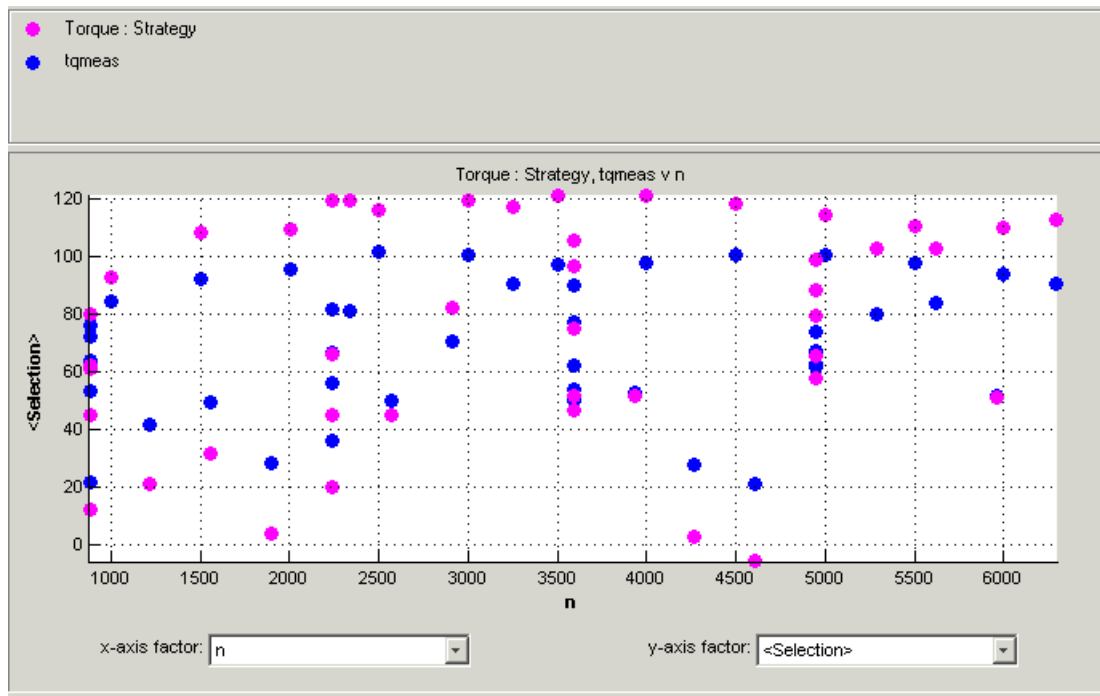
The error column is simply the difference between `tqmeas` and `Torque: Strategy`. This provides a simple way of comparing the feature and the measured data.

Viewing the Data Set as a Plot

- 1 Click  or select **View > Plot** to view the data set as a plot.

The lower pane lists all the output expressions in the data set and in the project.

- 2 Use **Ctrl+click** to select `tqmeas` and `Torque: Strategy` from the lower list.



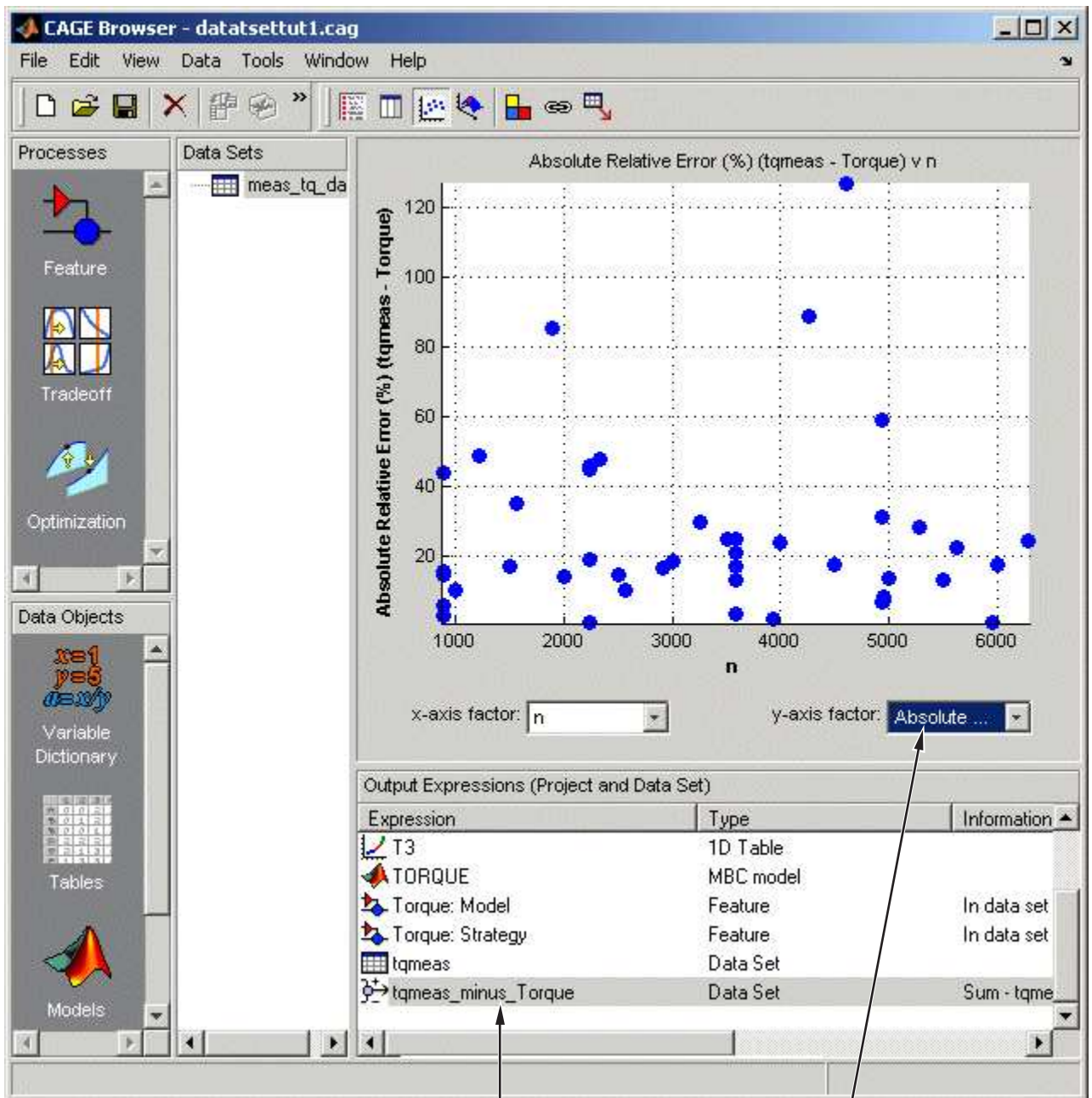
3 Change the **x-axis factor** to n from the drop-down menu.

This displays the calibrated values of torque from the feature, and the measured values of torque from the experimental data, against the test cell settings for engine speed.

Clearly there is some discrepancy between the two.

Displaying the Error

View the error between the calibrated and measured values of torque.



1. Select tqmeas_minus_Torque.

2. Select Absolute Relative Error (tqmeas - Torque).

- 1 Select `tqmeas_minus_Torque` from the lower list (**Output Expressions**).
- 2 For the **y-axis factor**, select **Absolute Relative Error (tqmeas - Torque)** from the drop-down menu.

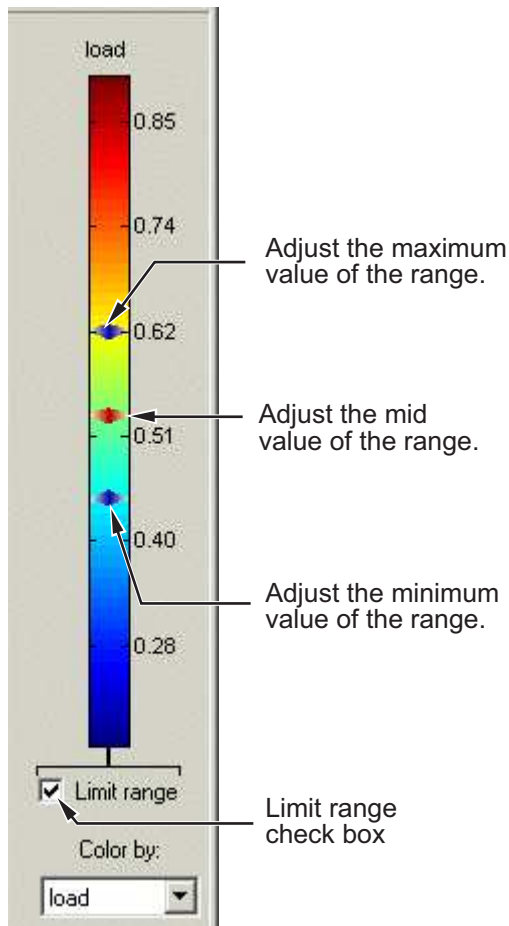
As you can see, there seems to be no particular correlation between engine speed and the error in the calibration.

Coloring the Display

- 1 Select **Color by Value** from the right-click menu on the graph.
- 2 From the **Color by** drop-down menu, select **load**.

In this display, you can see that some of the low values of load display a high error.

Limiting the Range of the Colors



To view the colors in more detail, you can limit the range of the colors:

- 1 Select the **Limit range** box (or you could right-click the graph and select **Restrict Color to Limits**).
- 2 Set the minimum value of the color range to be as low as possible by dragging the minimum value down.
- 3 Set the maximum value of the color range to be around 0.4.


As the low values of load are causing large errors, it would be wise to reexamine the calibration, particularly at small values of `load`.

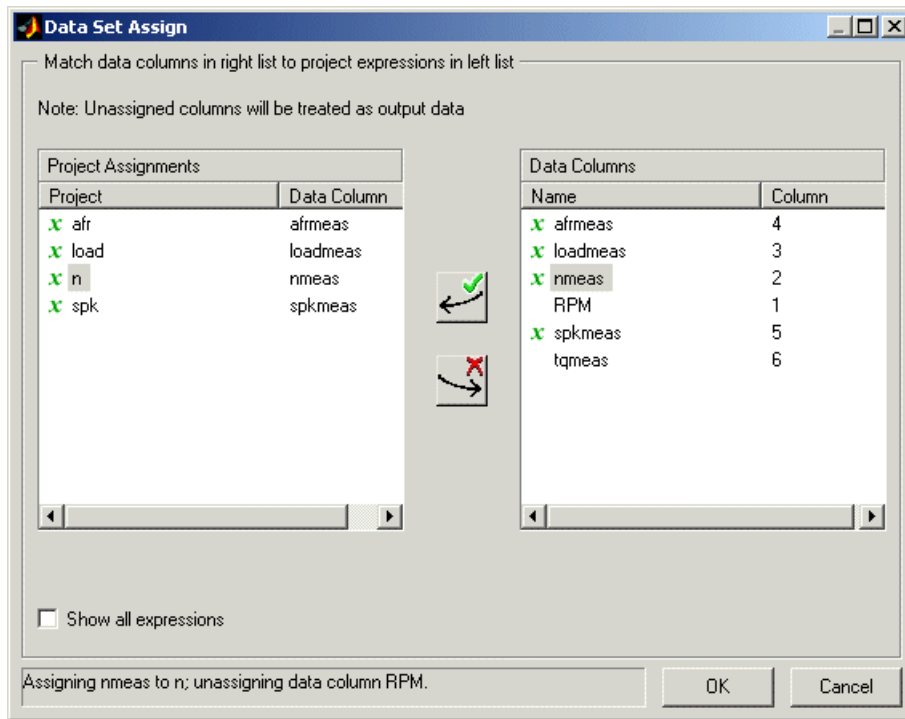
Reassigning Variables

You can alter the data set by changing which variables are used for project expressions.

Instead of using the test cell settings for the engine speed (RPM), you might want to use the measured values of engine speed (`nmeas`). So you have to reassign the variable `n` to `nmeas`.

To reassign `n`,

- 1 Click  or select **Data > Assign**.
- 2 In the dialog that appears, select `n` from the **Project Assignments** pane and `nmeas` from the **Data Columns** pane.
- 3 Click the assign button.



You can now compare your calibration with your experimental data again, using the techniques described.

For more information about the functionality of data sets, see “Data Sets in CAGE”.

You have now completed the data sets tutorial.

Filling Tables from Data

This section includes the following topics:

Fill Tables from Data

In this section...

“Setting Up a Table and Experimental Data” on page 15-2

“Filling the Table from the Experimental Data” on page 15-8

“Selecting Regions of the Data” on page 15-11

“Exporting the Calibration” on page 15-13

Setting Up a Table and Experimental Data

- “Tutorial Overview” on page 15-2
- “Adding Variables” on page 15-3
- “Adding a New Table” on page 15-4
- “Importing Experimental Data” on page 15-6

Tutorial Overview

If you are considering a straightforward strategy, you might want to fill tables directly from experimental data. For example, a simple torque strategy fills a lookup table with values of torque over a range of speed and relative air charge, or load. You can use CAGE to fill this strategy (which is a set of tables) by referring to a set of experimental data. You can also fill tables with the output of optimizations.

This tutorial takes you through the steps of calibrating a lookup table for torque, based on experimental data.

- This section describes the steps required to set up CAGE in order to calibrate a table by reference to a set of data.
- “Filling the Table from the Experimental Data” on page 15-8 describes the process of filling the lookup table.
- “Selecting Regions of the Data” on page 15-11 describes how you can select some of the data for inclusion when you fill the table.
- “Exporting the Calibration” on page 15-13 describes how to export your completed calibration.

Start CAGE by typing

cage

at the MATLAB prompt.

If you already have a CAGE session open, select **File > New Project**.

First you will set up a blank table ready for filling using experimental data or optimization output.

The steps that you need to follow to set up the CAGE session are

- 1 Add the variables for speed and load by importing a variable dictionary.
- 2 Add a new table to your session.
- 3 Import your experimental data.

The next sections describe each of these processes in detail.

Adding Variables

Before you can add tables to your session, you must add variables to associate with the normalizers or axes.

To add a variable dictionary,

- 1 Select **File > Import > Variable Dictionary**.
- 2 Select `table_filling_tutorial.xml` from the `matlab\toolbox\mbc\mbctraining` directory.

This loads a variable dictionary into your session. The variable dictionary includes the following:

- N, the engine speed
- L, the relative air charge
- A, the air/fuel ratio (AFR)
- `stoich`, the stoichiometric constant

You can now add a table to your session.

Adding a New Table

You must add a table to fill.

To add a new table,

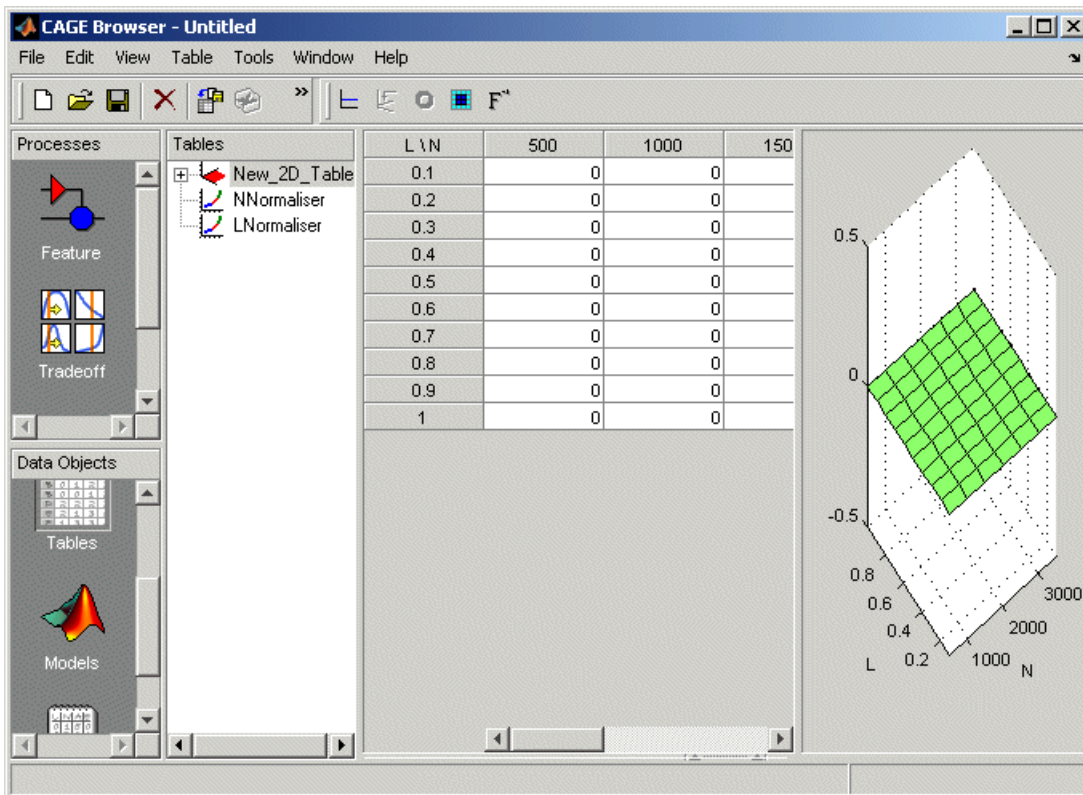
- 1 Select **File > New > 2D table**.

This opens a dialog box that asks you to specify the variable names for the normalizers. As you can see in the dialog controls, accepting the defaults will create a table with ten rows and ten columns with an initial value of 0 in each cell.

- 2 Change the number of columns to 7.
- 3 Select L as the variable for normalizer **Y** and N as the variable for normalizer **X**, then click **OK**.


Note In CAGE, a 2-D table is defined as a table with two inputs.

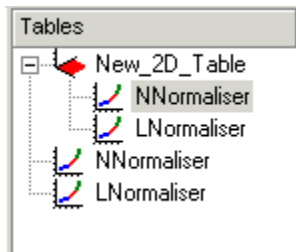
CAGE takes you to the **Tables** view, where you can see the following.



Inspecting the Values of the Normalizers

CAGE has automatically initialized the normalizers by spacing the breakpoints evenly across the range of values for the engine speed (N) and load (L). The variable ranges are found in the variable dictionary. Switch to the Normalizer view to inspect the normalizers.

Expand the table branch by clicking , and select `NNormalizer` as shown.



This displays the two normalizers for the table.

You have an empty table with breakpoints over the ranges of the engine speed and load, which you can fill with values based on experimental data.

Importing Experimental Data

To fill a table with values based on experimental data, you must add the data to your session. If you want to fill a table with the output of an optimization, the output appears automatically in the Data Sets view as a new data set called `Exported_Optimization_Data` when you select the Export to Data Set toolbar button. For this tutorial you need to import some experimental data.

CAGE uses the **Data Sets** view to store grids of data. Thus, you need to add a data set to your session as well.

Select **File > New > Data Set** to add a data set to your session. This changes the view to the **Data Set** view.

You can now import experimental data into the data set:

- 1 Select **File > Import > Data**.
- 2 In the file browser, select `meas_tq_data.csv` from the `matlab\toolbox\mbc\mbctraining` directory and click **Open**.

This set of data includes six columns of data: the test cell settings for engine speed (RPM), and the measured values of torque (`tqmeas`), engine speed (`nmeas`), air/fuel ratio (`afmeas`), spark angle (`spkmeas`), and load (`loadmeas`).

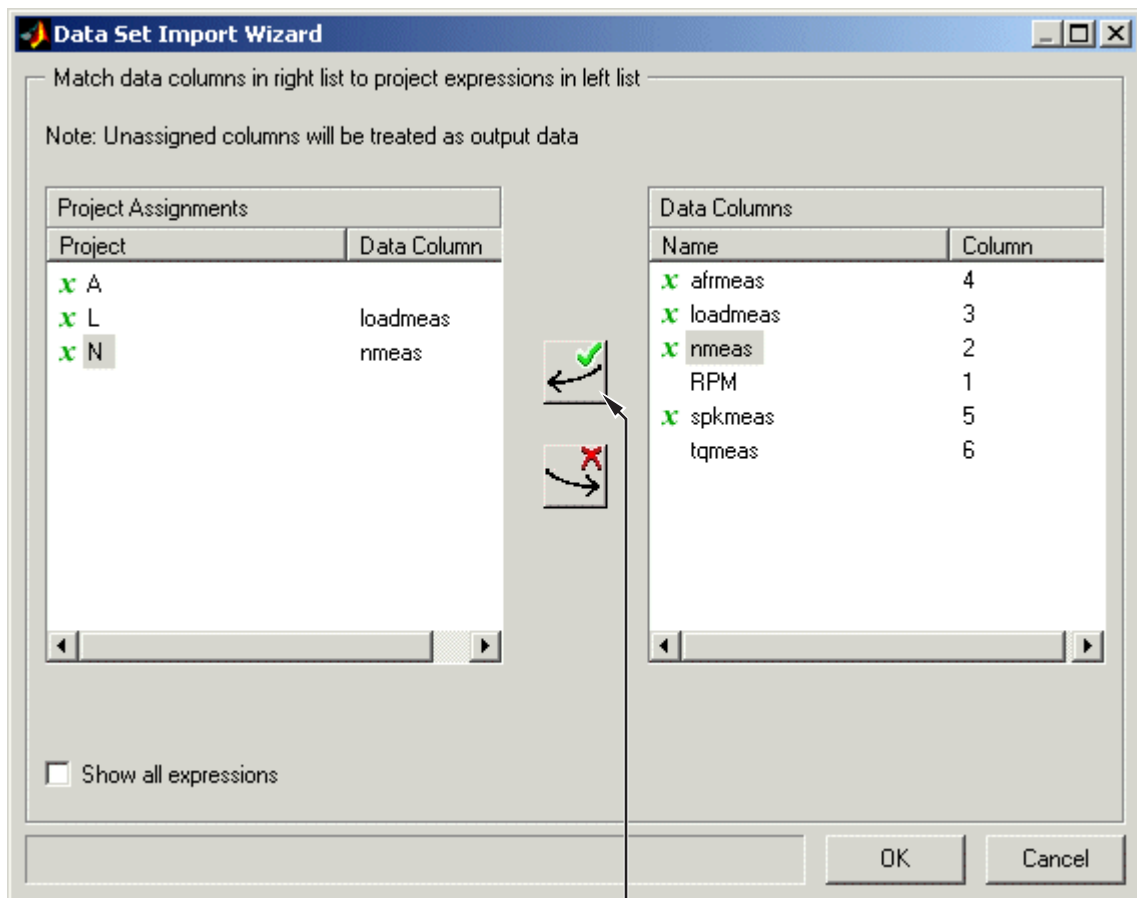
- 3 This opens the Data Set Import Wizard. The first screen asks which of the columns of data you want to import. Click **Next** to import them all.

The following screen asks you to associate variables in your project with data columns in the data.

- 4 Highlight N in the **Project Assignments** column and nmeas in the **Data Column**, then click the assign button, shown.



- 5 Repeat this to associate L with loadmeas. The dialog box should be the same as the following.



Assign button

- 6 Click **Finish** to close the dialog box.

You now have an empty table and some experimental data in your session. You are ready to fill the table with values based on this data.


Filling the Table from the Experimental Data

You have an empty table and the experimental data in your session. You can now fill the table with values based on your data.

The data that you have imported is a series of measured values of torque at a selection of different operating points. These operating points do not correspond to the values of the breakpoints that you have specified. The lookup table has a range of engine speed from 500 revolutions per minute (rpm) to 3500 rpm. The range of the experimental data is far greater.

CAGE extrapolates the values of the experimental data over the range of your table. Then it fills the table by selecting the torque values of the extrapolation at your breakpoints.

To fill the table with values based on the experimental data,

- 1 To view the **Table Filler** display, click  (Fill Table From Data Set) in the toolbar in the **Data Sets** view; or select **View > Table Filler**.

You can use this display to specify the table you want to fill and the factor you want to use to fill it.

- 2 In the lower pane, select `New_2D_Table` from the **Table to fill** list.
- 3 Select `tqmeas` from the **Factor to fill table** list. This is the data that you want to use to fill the table.
- 4 Select `N` from the **x-axis factor** list and `L` from the **y-axis factor** list. Your session should be similar to the following display.

An operating point from the experimental data (a blue dot)

A breakpoint in your lookuptable (a cross)

Filling table New_2D_Table, from factor tqmeas

L (table axis)

N (table axis)

x-axis factor: N

y-axis factor: L

Filling table New_2D_Table with output factor tqmeas from meas_tq_data

Table to fill	
Table	Inputs
New_2D_Table	N, L

Factor to fill table	
Factor	Information
RPM	
spkmeas	
tqmeas	

Table filling rules (optional)

Click and drag over Data Set plot to create rules

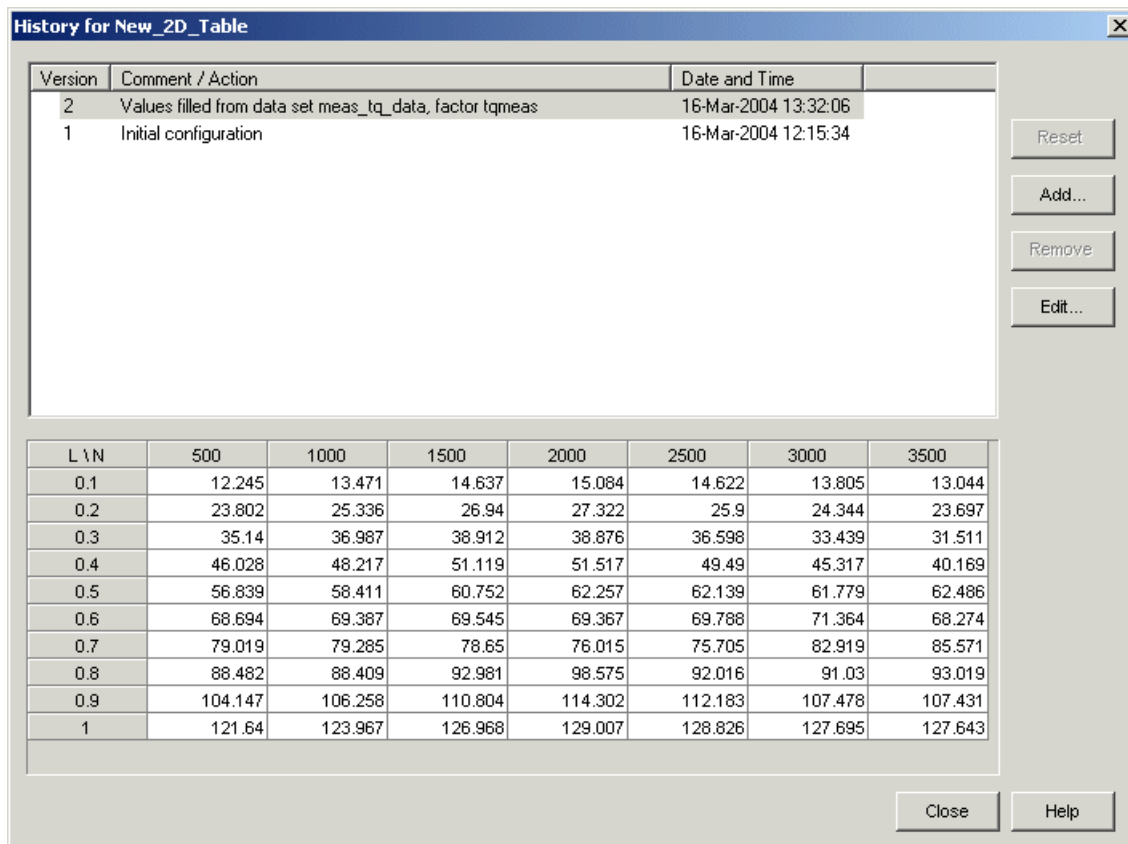
Show table history after fill

Fill Table

The upper pane displays the breakpoints of your table as crosses and the operating points where there is data as blue dots. Data sets display the points in the experimental data, not the values at the breakpoints. You can inspect the spread of the data compared to the breakpoints of your table before you fill the table.

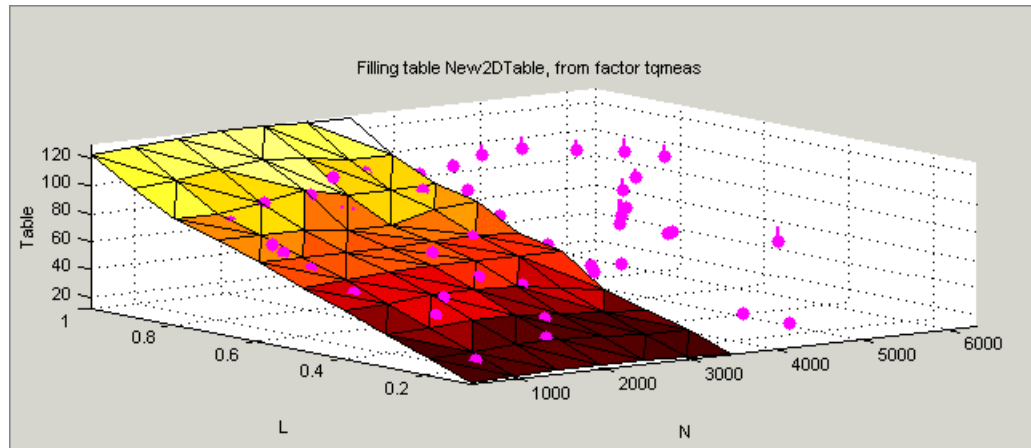
- 5 To view the table after it is filled, ensure that the **Show table history after fill** box, at the bottom left, is selected.
- 6 To fill the table with values of `tqmeas` extrapolated over the range of the normalizers, click **Fill Table**.

This opens the History dialog box, shown.



- 7 Click **Close** to close the History dialog box and return to the **Table Filler** display.

- 8 To view the graph of your table, as shown, select **Data > Plot > Surface**.



This display shows the table filled with the experimental points overlaid as purple dots.

The table has been calibrated by extrapolating over the values of your data and filling the values that the data predicts at your breakpoints.

Notice that the range of the table is smaller than the range of the data, as the table only has a range from 500 rpm to 3500 rpm.

The data outside the range of the table affects the values that the table is filled with. You can exclude the points outside the range of the table so that only points in the range that you are interested in affect the values in the table.

Selecting Regions of the Data

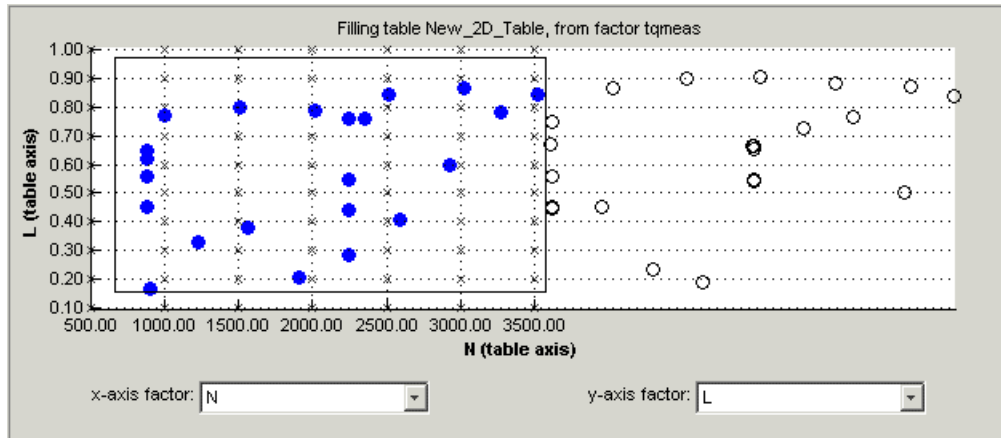
You can ignore points in the data set when you fill your lookup table.

For example, in this tutorial the experimental data ranges over values that are not included in the lookup table. You want to ignore the values of engine speed that are greater than the range of the table.

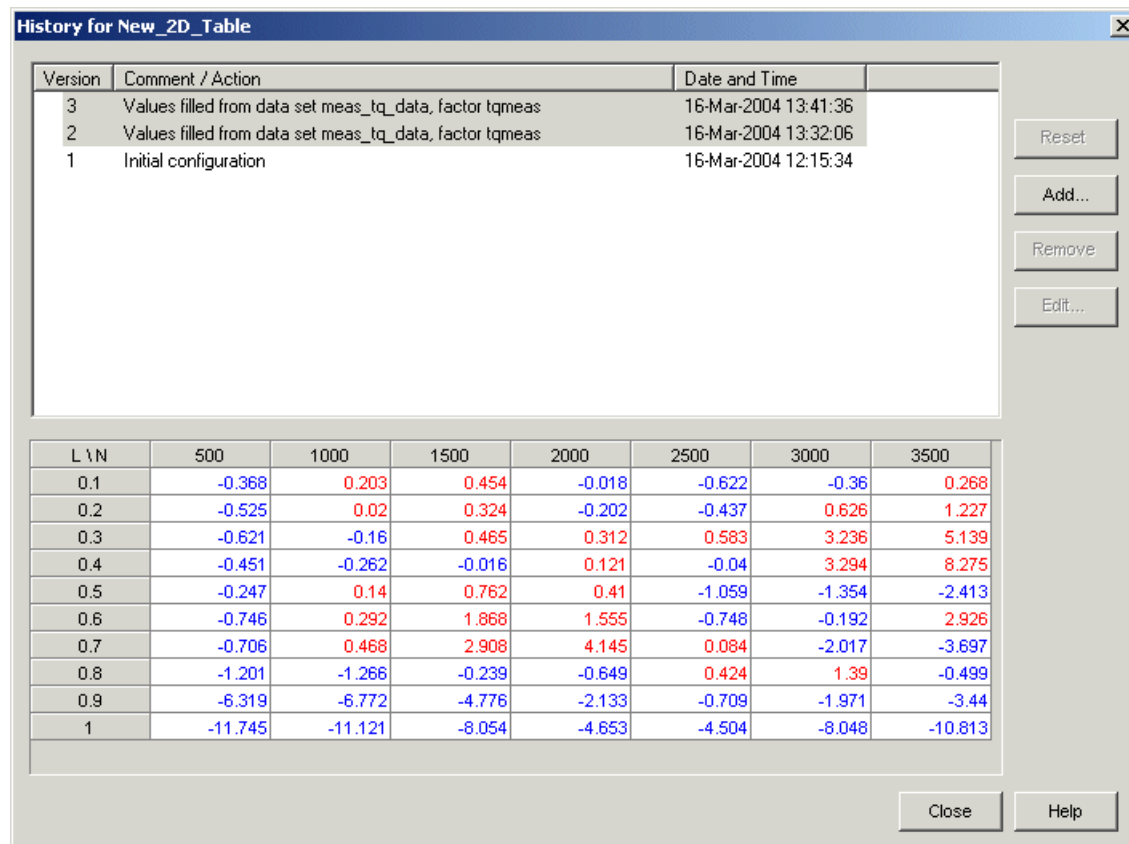
To ignore points in the data set,

- 1 Select **Data > Plot > Data Set**. This returns you to the view of where the breakpoints lie in relation to the experimental data.

- 2 To define the region that you want to include, left-click and drag the plot. Highlight all the points that are included in your table range, as shown.



- 3 To fill the table based on an extrapolation over these data points only, click **Fill Table**. This opens the History display again.
- 4 In the History display, select version 2 and 3, using **Ctrl+click**. The following display shows a comparison between the table filled with two different extrapolations.



- 5 Click **Close** to close the History viewer.
- 6 Select **Data > Plot > Surface** to view the surface again.

The display of the surface now shows the table filled only by reference to the data points that are included in the range of the table.

You have filled a lookup table with values taken from experimental data.

Exporting the Calibration

To export the calibration,

- 1 To highlight the table that you want to export, you must first click **Tables**, shown.



- 2 Highlight the New_2D_Table.
- 3 Select **File > Export > Calibration > Selected Item**.
- 4 Choose the type of file you want to save your calibrations as. For the purposes of this tutorial, select **Comma Separated Value (.csv)**.
- 5 Enter `table_filling_tutorial.csv` as the file name and click **Save**.

This exports the calibration.

You have now completed this tutorial.

Optimization and Automated Tradeoff

This section includes the following topics:

- “Optimization and Automated Tradeoff” on page 16-2
- “Single-Objective Optimization” on page 16-5
- “Multiobjective Optimization” on page 16-18
- “Sum Optimization” on page 16-30
- “Automated Tradeoff” on page 16-35

Optimization and Automated Tradeoff

In this section...

“Import Models to Optimize” on page 16-2

“Optimization Example Problems” on page 16-4

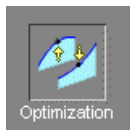
Import Models to Optimize

To open the CAGE browser and set up your optimization:

- 1 Start the CAGE Browser part of the Model-Based Calibration Toolbox product by typing

`cage`

at the MATLAB prompt.
- 2 To reach the Optimization view, click the **Optimization** button in the **Processes** pane.



You can use the **Optimization** view to set up, run, view, and export optimizations. You must also set up optimizations here in order to use them for automated tradeoff.

When you first open the **Optimization** view both panes are blank until you create an optimization. After you set up your optimizations, the left **Optimization** pane shows a tree hierarchy of your optimizations, and the right hand panes display details of the optimization selected in the tree, as with other CAGE processes.

For any optimization, you need one or more models. You can run an optimization at a single point, or you can supply a set of points to optimize. The steps required are as follows:

- 1 Import a model or models.
- 2 Set up a new optimization.

The following tutorial guides you through this process to evaluate this optimization problem:

MaxTQ (SPK, N, L)

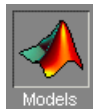
That is, find the maximum of the torque model (TQ) as a function of spark (SPK), engine speed (N), and load (L). You will use the NOXFLOW model to constrain these optimization problems.

For any optimization, you need to load one or more models. You can use the CAGE Import tool to import models from Model Browser projects (see “Import Models and Calibration Items Using CAGE Import Tool” in the CAGE documentation). For this tutorial you can load a CAGE project from the `mbctraining` directory that contains two models for the optimization problems. Load the project containing models to optimize as follows:

- 1 Select **File > Open Project** (or the toolbar button) to choose the `tradeoffInit.cag` file, found in the `matlab\toolbox\mbc\mbctraining` directory, then click **OK**.

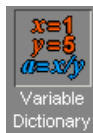
The `tradeoffInit.cag` project contains two models and all the variables necessary for this tutorial. For more information about how to set up models and variables, see “Calibration Setup” in the CAGE documentation.

- 2 CAGE displays the Models view. You can view your models at any time by clicking the **Models** button in the **Data Objects** pane.



Observe that the project you have opened contains two models: `TQ_Model` and `NOXFLOW_Model`. In this tutorial you use these models to optimize torque values subject to emissions constraints.

- 3 To view the items in the Variable Dictionary, click the **Variable Dictionary** button in the **Data Objects** pane.



The **Variable Dictionary** view appears, displaying the variables, constants, and formulas in the current project. The project already has the relevant variables defined, so you do not need to import a variable dictionary. Note that the variables have ranges and set points defined.

Optimization Example Problems

In this tutorial you will use optimization to find solutions to the following problems:

- A single-objective optimization to find maximum values of torque, subject to a constraint to keep NOX emissions below a specified level. You will export the output and use it to fill a table.
- A multiobjective optimization to maximize torque and minimize NOX emissions.
- A sum optimization to maximize torque while minimizing NOX, weighted to give more importance to idle speed.
- Using any of your optimizations to run an automated tradeoff. Once you have set up an optimization you can apply it to a tradeoff.

To work through these examples, follow the steps in the these sections in order:

- 1 “Single-Objective Optimization” on page 16-5
- 2 “Multiobjective Optimization” on page 16-18
- 3 “Sum Optimization” on page 16-30
- 4 “Automated Tradeoff” on page 16-35

Note: Start with “Single-Objective Optimization” on page 16-5.

Single-Objective Optimization

In this section...

“Process Overview” on page 16-5

“Using the Create Optimization from Model Wizard” on page 16-6

“Setting Constraints and Operating Points” on page 16-9

“Running the Optimization” on page 16-12

“Using Optimization Results to Fill Tables” on page 16-14

“Using a Custom Fill Routine to Fill Tables” on page 16-16

Process Overview

The following sections describe these stages:

- 1** Using the Create Optimization from Model wizard to choose
 - A model for your objective
 - The optimization settings:
 - Which algorithm to use
 - Maximize or minimize
 - Point or sum objective
 - Operating points for your optimization
 - What free variables to use
- 2** Using the Optimization view to choose
 - A model, type, and value for your constraint
- 3** Running the optimization, examining the output, exporting to a data set, and using the output to fill a table

Note: To follow these steps, you must first load models. See “Import Models to Optimize” on page 16-2.

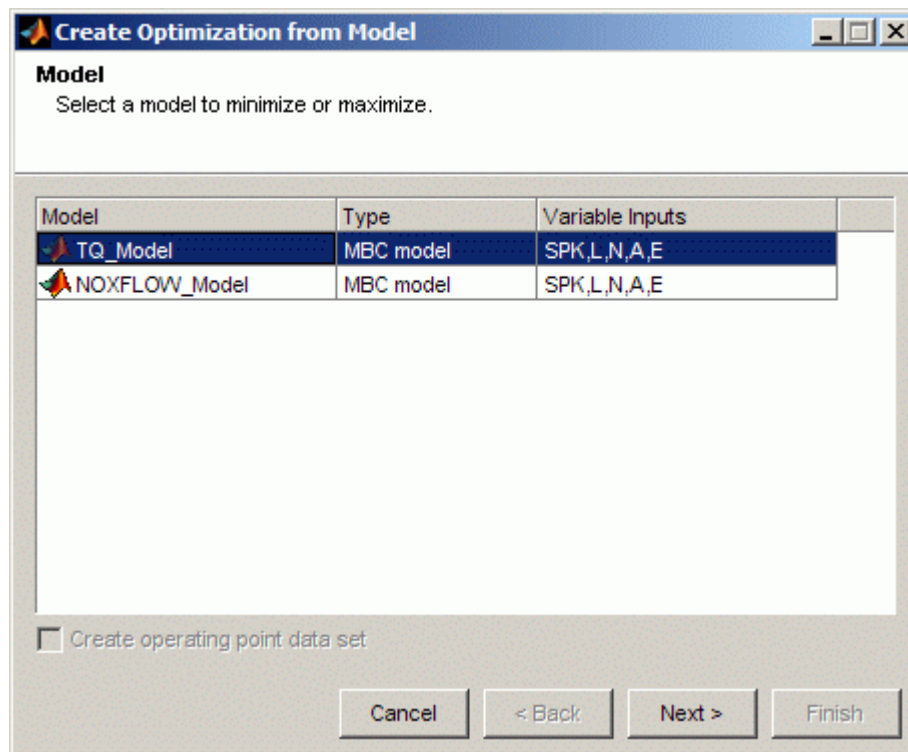
Using the Create Optimization from Model Wizard

To create your optimization,

- 1 Select **Tools > Create Optimization From Model** (or use the toolbar button).

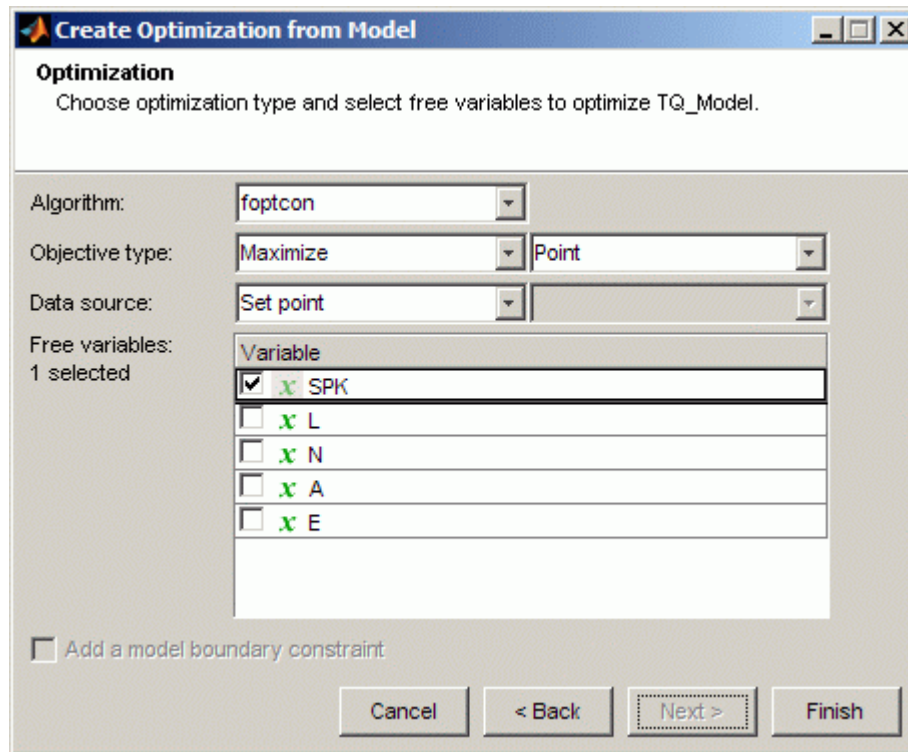
The **Create Optimization From Model** Wizard appears. If you are viewing a model, then the wizard automatically selects the current model.

- 2 Select TQ_Model as the model you want to optimize.



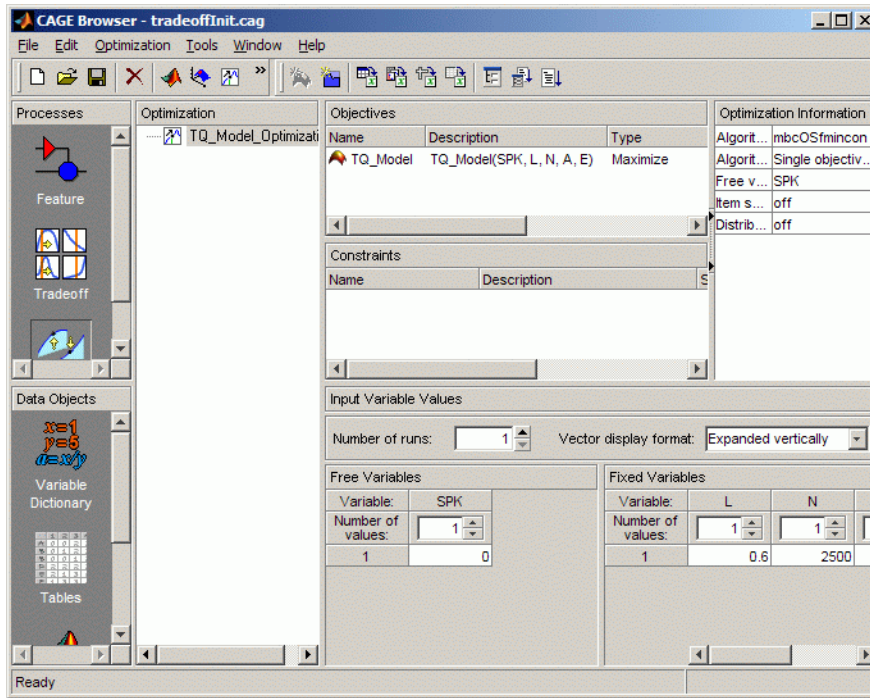
Click **Next**.

- 3 On this page of the wizard you select the optimization settings.
 - a Select **Maximize** for the **Objective type**.
 - b Clear the check boxes for all the **Free variables** except SPK.



Leave the other settings at the defaults, and click **Finish** to create the optimization.


You see the CAGE Browser Optimization view. A new branch named `TQ_Optimization` appears in the Optimization tree. View the new optimization. Your CAGE browser should look like the following example.



In the **Objectives** pane you can see the **Description** TQ_Model (SPK, L, N, A, E) and the **Type** is Maximize.

In the **Optimization Information** pane you can see listed the default algorithm name mbcOSfmincon, free variable SPK, and if you hover the mouse you can read the full description Single objective optimization subject to constraints.

The **Constraints** pane is empty as you have not yet added any constraints.

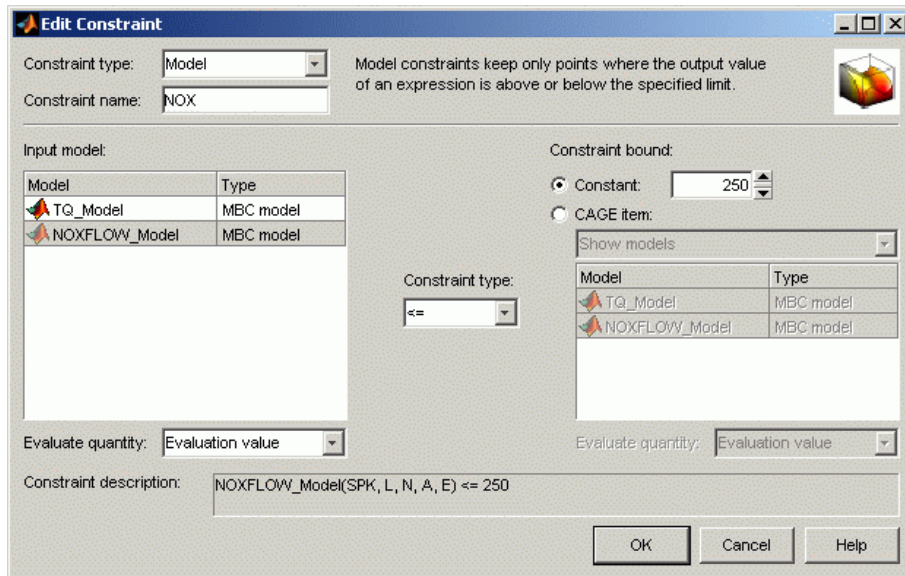
Note that the toolbar button Run Optimization () is enabled, because your optimization setup has provided enough information to start an optimization.

Your optimization is ready to run, but would run at a single point (the set points of the variables is the default). To finish setting up your optimization you need to specify a constraint and edit the points where you want the optimization to run.

Setting Constraints and Operating Points

- 1 Right-click the **Constraints** pane and select **Add Constraint**.

The Edit Constraint dialog appears.



- a Leave the **Constraint type** drop-down menu at the default, **Model**.
- b Edit the **Constraint name** to **NOX**.
- c Select **NOXFLOW_Model** from the **Input model** list.
- d Make sure the inequality is **<=**, and enter **250** in the **Constant** edit box as the maximum value for the constraint, as shown above.
- e Press **Enter**.

You return to the CAGE Browser Optimization view. Make sure the **Description** **NOXFLOW_Model(SPK, L, N, A, E) <= 250** appears in the **Constraints** pane.

- 2 You can use the **Optimization Point Set** panes to define a set of operating points for the optimization. Note that you do not have to have an operating point set; if you

do not, the optimization will run at a single point of your choosing (the set points of variables is the default).

You can use the Create Optimization from Model wizard to choose the points where you want to run the optimization, or you can set up points later in the Optimization view. In both cases you can choose to use points from a suitable data set or table grid if they exist in your project.

Running the optimization requires the selected models to be evaluated (many times over) and hence values are required for all the model input factors (L, N, A, E, and SPK). The defaults of the fixed variables (L, N, A, E) are their set points, as shown in the **Fixed Variables** pane. You have chosen SPK as a free variable, so the optimization will choose different values for SPK in trying to find the best. The default initial value for a free variable is the set point, as shown in the **Free Variables** pane.

To define the set of operating points for the optimization,

- a In the **Optimization Point Set** pane, increase the **Number of runs** to 6. Notice 6 rows appear in both fixed and free variables panes, all containing the default set point values of each variable.
- b Enter, or copy and paste, these values into the N column of the **Fixed Variables** pane. (Select all N rows before pasting):

N
1000
1000
3000
3000
6000
6000

- c Enter, or copy and paste, these values into the L column of the **Fixed Variables** pane:

L
0.1
0.8

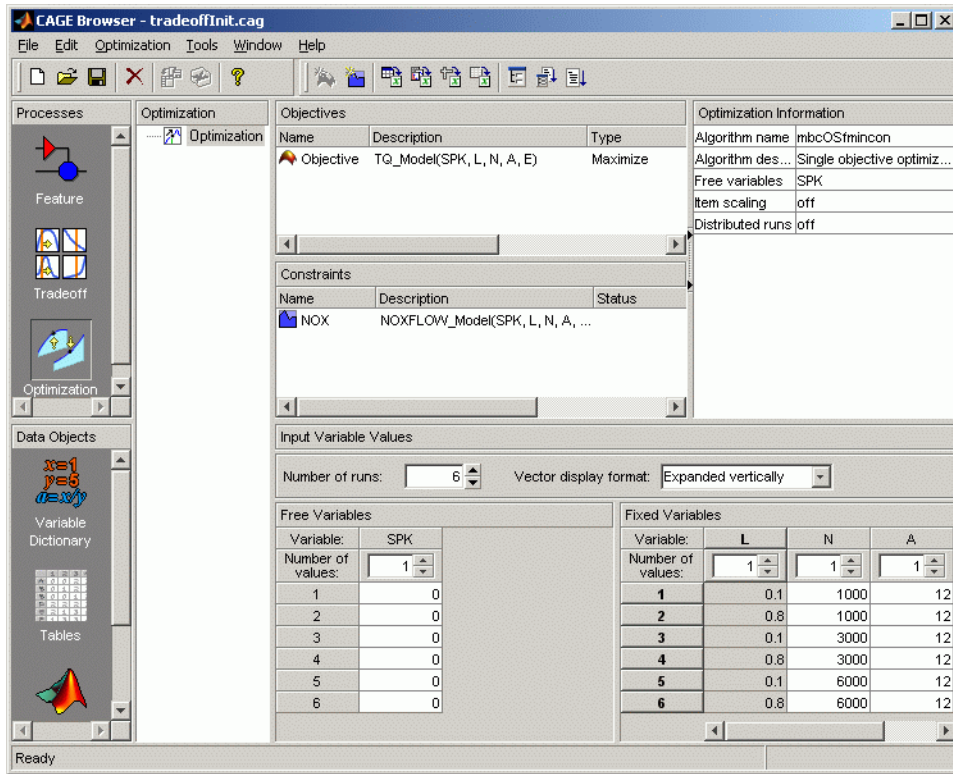
L
0.1
0.8
0.1
0.8

The **Fixed Variables** pane should look as shown.


Variable:	L	N	A	E
Number of values:	1	1	1	1
1	0.1	1000	12	5
2	0.8	1000	12	5
3	0.1	3000	12	5
4	0.8	3000	12	5
5	0.1	6000	12	5
6	0.8	6000	12	5

Leave the other fixed variables and the free variable values at the defaults. If you wished to restrict the range of the free variables, you could select **Optimization > Edit Free Variable Ranges**. The default is the range of the variable as defined in the Variable Dictionary. For this example, leave the default.

- 3 Your CAGE Browser should now look like the following example, with an objective, constraint, and set of operating points. The optimization is ready to run.



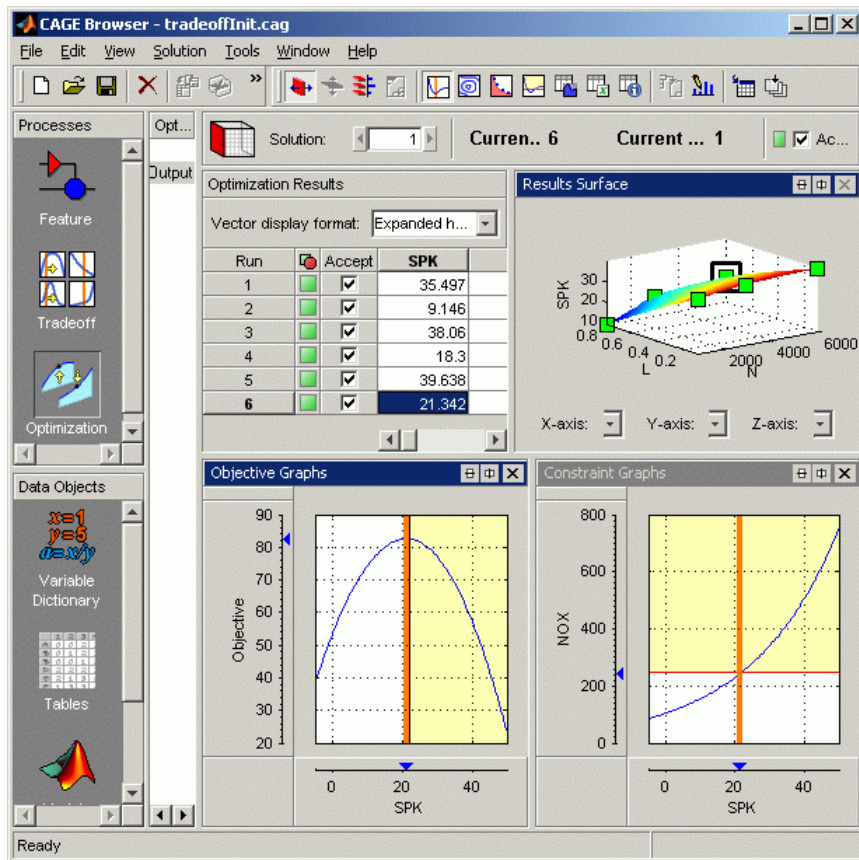
Running the Optimization

- 1 Click Run Optimization () in the toolbar.

The optimization runs, showing progress messages as each point is evaluated until the optimization is complete. On completion of the optimization, a new node appears in the Optimization tree.

- 2 The view switches to the new node `TQ_Model_Optimization_Output` where you can view the optimization results.
- 3 The optimization output view retains a memory of previous layout. If you have not used these views before, try the buttons and right-click context menus in the view title bars to add **Constraint Graphs** to examine your results.

- 4 This single-objective optimization produces one best solution for each point in the operating point set. To view solutions at particular operating points, either:
 - Click the cells of the table, or
 - Click the points in the **Results Surface** or **Results Contour** plot.
- 5 Select **Run 6** and examine the results.



For more information, see “Analyzing Point Optimization Output” in the CAGE User's Guide documentation.

Using Optimization Results to Fill Tables

As an example, to use these optimization results to fill a table, first create a new table as follows:

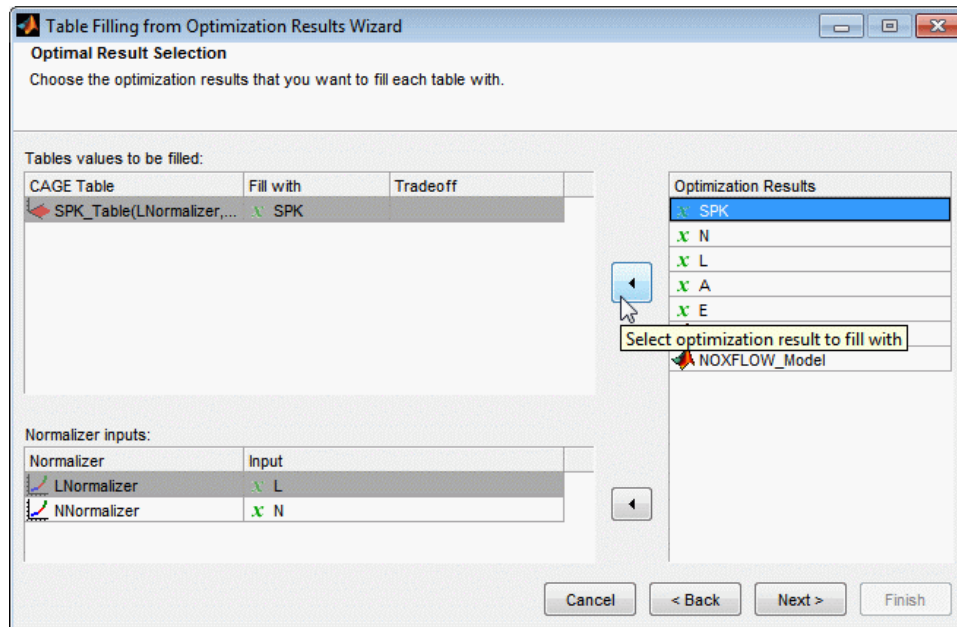
- 1 Build a SPK table in N and L. Select **File > New > 2-D Table**.
- 2 Leave 10 in the **Rows** and **Columns** edit boxes and 0 in the **Initial Value** edit box.
- 3 Use the drop-down menus to select L and N for the Y and X inputs.
- 4 Rename the table to SPK_Table.
- 5 Click **OK**. Your CAGE browser switches to the **Tables** view. CAGE has automatically initialized the normalizers to space breakpoints evenly over the ranges of N and L.

There are two methods for filling tables with optimization results.

- 1 Click the **Optimization** button in the **Processes** pane to return to the Optimization view
- 2 Click the plus to expand the TQ_Model_Optimization node, and select the TQ_Model_Optimization_Output node.
- 3 Select **Solution > Fill Tables** (or the toolbar button **Fill tables using optimal settings**).

The Table Filling wizard appears.

- 4 Select the SPK_Table table and click the button to add it to the list of tables to be filled. Click **Next**.
- 5 Select the SPK_Table table, and SPK in the list of optimization results, and click the button to select it to fill the table, as shown.




- 6 Click **Next**, then **Finish**.

You see a dialog reporting successful table filling.

- 7 Leave the **View selected item** check box selected to switch to the **Tables** view, then click **Close**. Examine the new spark table.

The other method of filling tables with optimization output uses Data Sets.

- 1 From the **Optimization_Output** optimization output node, click **Export to Data Set** () in the toolbar (or select **Solution > Export to Data Set**). Click **OK** in the **Export to Data Set** dialog box to accept the defaults.
- 2 CAGE displays the **Data Sets** view. Click **View Data** in the toolbar to see the table of optimization results contained in the new data set **New_Dataset**.

You can now use this data set (or any optimization results) to fill tables, as you can with any data set.

- 3 Click  (**Fill Table From Data Set**) in the toolbar.

- 4 Choose to fill the spark table with the SPK optimization output by selecting them in the two lists:
 - a Select `SPK_Table` in the **Table to fill** list.
 - b Select the SPK optimization output in the **Factor to fill table** list.
 - c Click the button **Fill Table** at the bottom right.
- 5 The Table History dialog box appears to show you that CAGE filled the table from the data set. Click **Close** to dismiss the dialog box.
- 6 To see the filled table surface and the optimization output spark values on the same plot, right-click the display and select **Surface**. Recall you already filled the spark table from the optimization results with the table filling wizard, so the table should look unchanged.

See also “Fill Tables from Data” on page 15-2 for more details on using data sets to fill tables.

In the next section you will use a custom fill routine to fill the table.

Using a Custom Fill Routine to Fill Tables

It can be useful to create your own custom fill function to fill tables from the results of an optimization. Some example situations are:

- You have your own smoothing strategy for certain regions of your look-up tables
- Implementation of an alternative method to the two fill methods supplied
- You want to produce some customized output

You can use a custom fill routine to fill the `SPK_Table` table from the optimization results.

- 1 Create a custom fill function. For this example, you can use the supplied example, `griddataTableFill.m`, which can be found in the `mbctraining` directory. Copy `griddataTableFill.m` to a directory away from the MATLAB root directory, and make sure this directory is on the MATLAB path (or change the current directory to the location where you copied the file).
- 2 At the optimization output node, select **Solution > Fill Tables**.
- 3 The wizard retains a memory so the `SPK_Table` table is already selected to be filled. Click **Next**.

- 4 Similarly, **SPK** is already selected from the list of optimization results to fill the table. Click **Next**.
- 5 Select **Custom** from the **Fill Method** drop down menu. Use the file selector, or enter the name of the fill function you wish to use to fill your tables. In this case, select or enter `griddataTableFill`, and press **Enter**. Note that this function must be on the MATLAB path.
- 6 Click **Finish** to fill the `SPK_Table` table.
- 7 You see a dialog reporting successful table filling. Click **Close**.

In the next section you will add a multiobjective optimization to this project. For next steps, see “Multiobjective Optimization” on page 16-18.

Multiobjective Optimization

In this section...
“Setting Up and Running the Multiobjective Optimization” on page 16-18
“Optimization Output View” on page 16-23
“Selecting Best Solutions” on page 16-28

Note: To follow these steps, you must first complete the previous steps in the tutorial to load models and create an optimization. See “Optimization and Automated Tradeoff” on page 16-2.

Setting Up and Running the Multiobjective Optimization

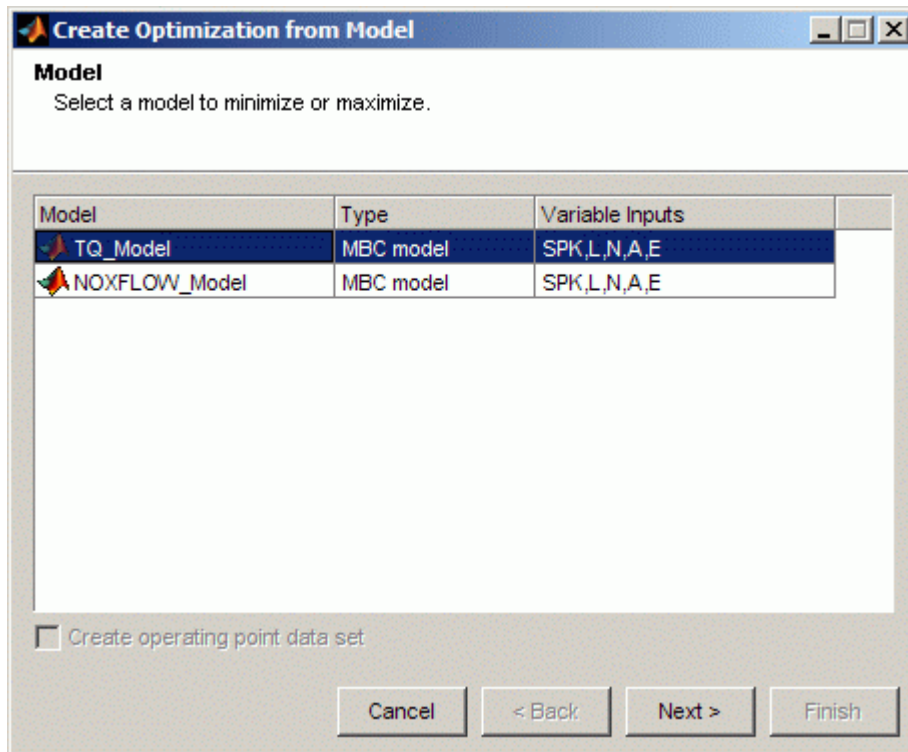
In this optimization you will construct a search for values of spark that maximize values of torque while minimizing values of NOX at a series of (L, N, A, E) points.

To create your optimization,

- 1 Select **Tools > Create Optimization From Model** (or use the toolbar button).

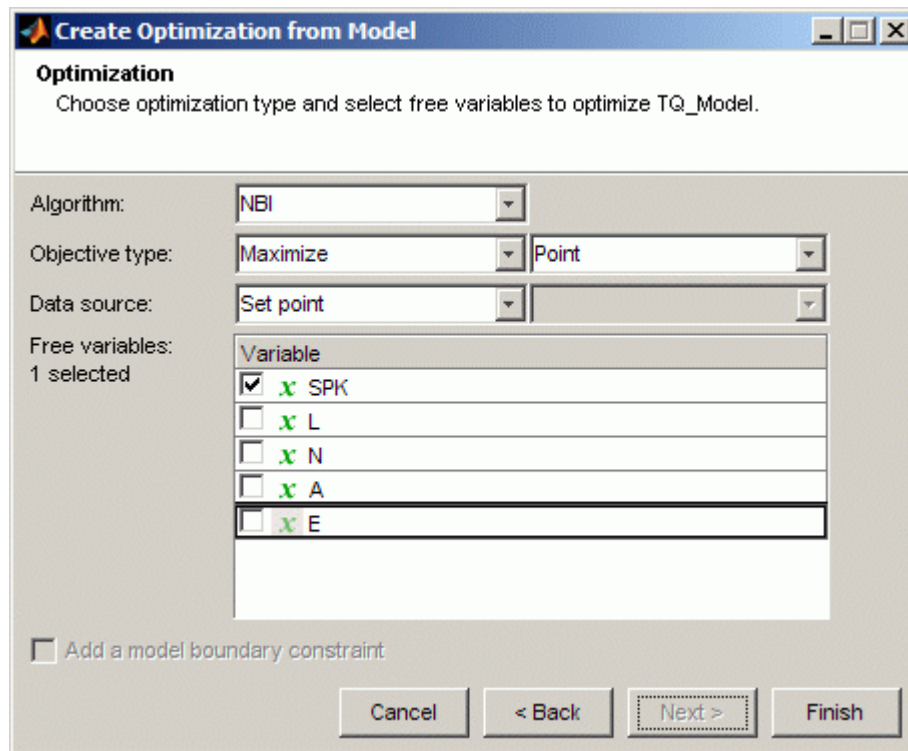
The **Create Optimization From Model** Wizard appears.

- 2 Select TQ_Model1 as the first model you want to optimize.



Click **Next**.

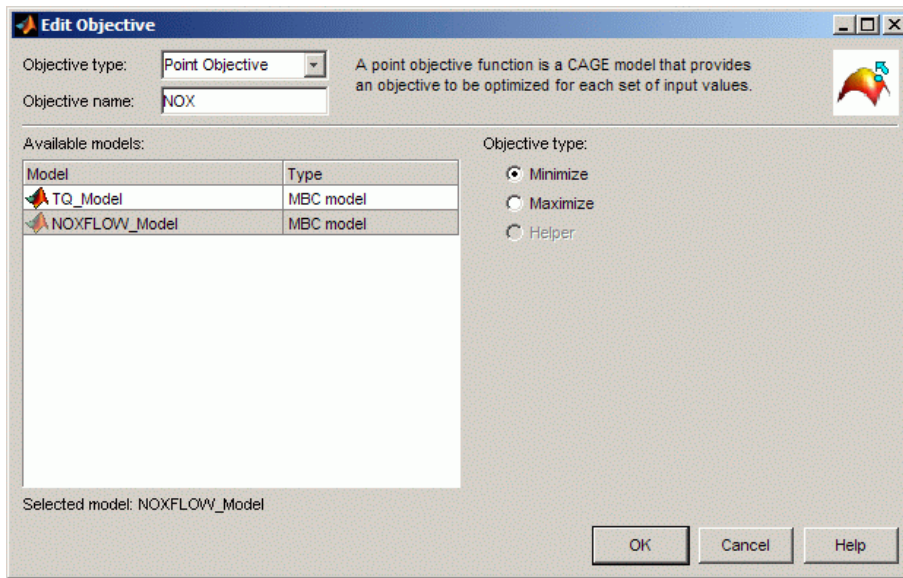
- 3** On this page of the wizard you select the optimization settings.
 - a** Select **NBI** for the **Algorithm**. You must use the NBI algorithm to solve multiobjective optimizations.
 - b** Select **Maximize** for the **Objective type**.
 - c** Clear the check boxes for all the **Free variables** except **SPK**.
 - d** Click **Finish** to create the optimization.



You see the CAGE Browser Optimization view. A new branch named `TQ_Model_Optimization_1` appears in the Optimization tree.

You need to set up your second objective. In the Objectives pane you see a status message informing you that you need to specify a model for the second objective.

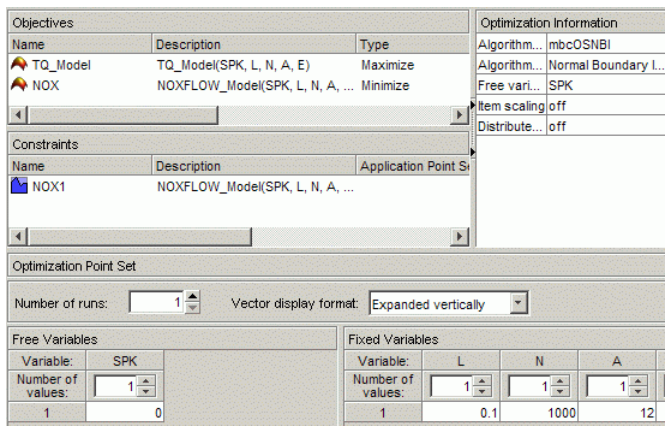
- 1 Double-click `Objective2` to edit the objective.
- 2 Edit the Objective name to `NOX`, and specify `NOXFLOW_Model1` and **Minimize**. Click **OK**.



Import the NOX constraint from your previous optimization.

- 1 Right-click in the Constraints pane and select **Import Constraints**.
- 2 In the Import Constraints dialog box, select the NOX constraint from your previous optimization and click **OK**.


Your CAGE browser should look like the following example.



Your optimization has objectives and a constraint set up and is ready to run. However unless you edit the fixed variable values it will run at a single point, the set point of the variables.

- 1 In the **Optimization Point Set** pane, increase the **Number of runs** to **6**. Notice 6 rows appear in both fixed and free variable values panes, all containing the default set point values of each variable.
- 2 Select **Optimization > Import From Output**. The Import From Output dialog box appears.
 - a Select the previous single objective optimization node, **TQ_Model_Optimization_Output**, in the top list to import values from this optimization.
 - b Clear the check boxes for SPK, A, and E, to leave only N and L selected for import, and click **OK**.
- 3 View these values in the N and L columns in the **Fixed Variables** pane.


Number of runs: <input type="text" value="6"/>		Vector display format: <input type="text" value="Expanded vertically"/>				
Free Variables		Fixed Variables				
Variable:	SPK	Variable:	L	N	A	E
Number of values:	<input type="text" value="1"/>	Number of values:	<input type="text" value="1"/>	<input type="text" value="1"/>	<input type="text" value="1"/>	<input type="text" value="1"/>
1	0	1	0.1	1000	12	5
2	0	2	0.8	1000	12	5
3	0	3	0.1	3000	12	5
4	0	4	0.8	3000	12	5
5	0	5	0.1	6000	12	5
6	0	6	0.8	6000	12	5

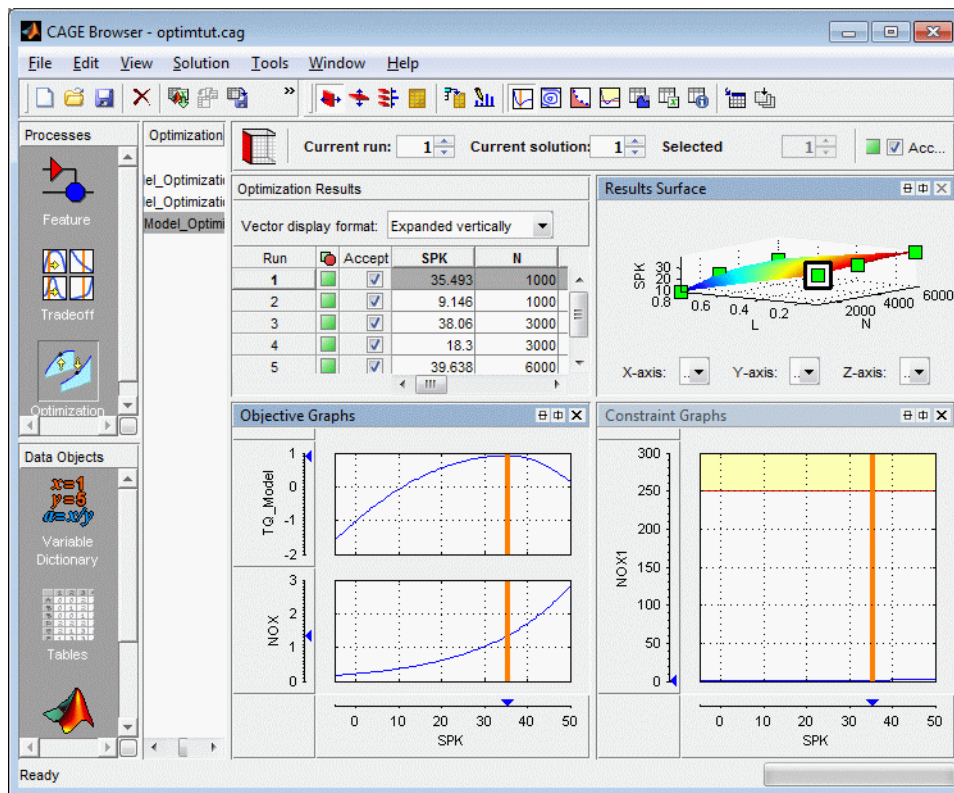
- 4 Click Run Optimization () in the toolbar.

The optimization runs, showing progress messages as each point is evaluated until the optimization is complete. A new node, **TQ_Model_Optimization_1_Output**, appears under **TQ_Model_Optimization_1** in the Optimization tree.

Optimization Output View

The view switches to the TQ_Model_Optimization_1_Output node in the Optimization tree where you can examine the optimization output.

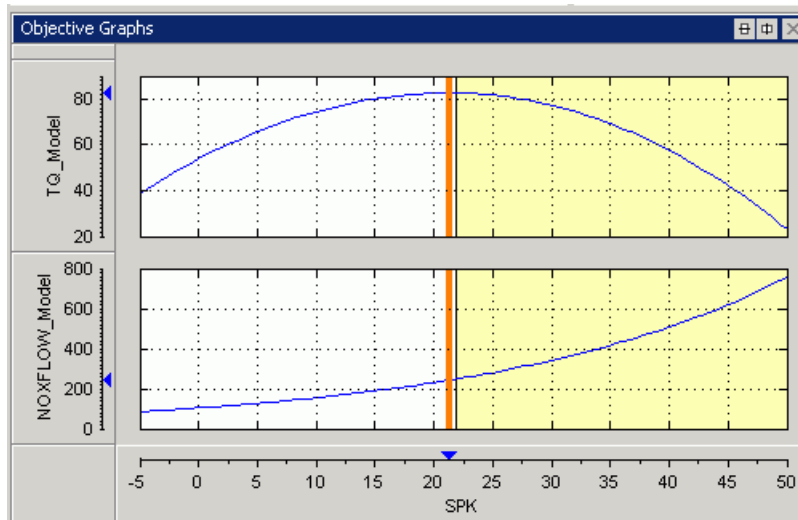
The toolbar buttons determine which view is displayed. The default is the Solution Slice (). The Solution Slice shows one solution at all operating points. That is, you can see a table (and surface plot) of all operating points at once, and you can scroll through the solutions using the **Current solution** controls at the top. At the start all 6 operating points show solution 1. Change solution to 2, and you see the second solution for all 6 operating points, and so on. As this is a multiobjective optimization, there are several solutions for each operating point.



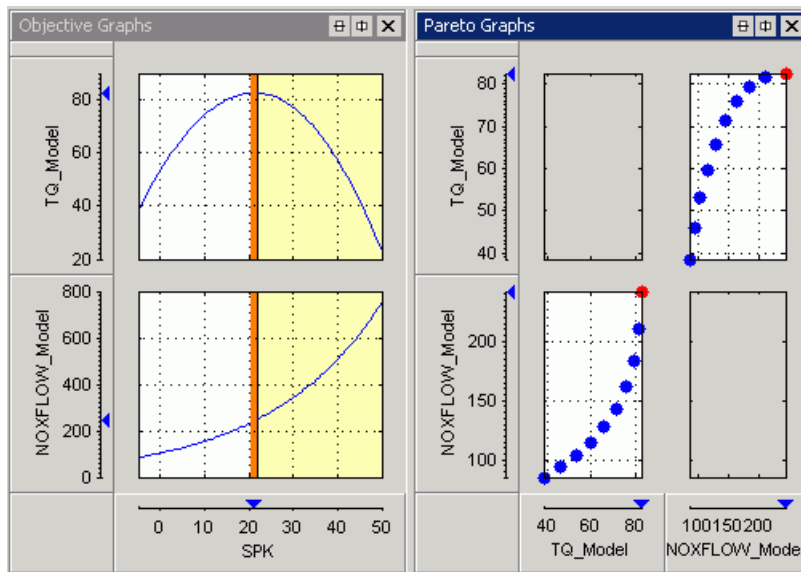
The graphs show the objective functions at the currently selected operating point (highlighted in the table and Results Surface view), with the solution value shown in red.

Note that before you run an optimization you can specify how many solutions you want the optimization to find, using the Set Up and Run Optimization toolbar button.


- 1 For an example point, click in the table to select operating point 6, and enter **10** in the **Current solution** edit box. Observe the constraint you applied in the objective function graphs, as shown in the example. Areas in yellow are excluded by constraints. Similarly, if you use a boundary constraint model exported from the Model Browser as a constraint, areas outside the boundary appear in optimization graphs as yellow areas. Note that for some problems the optimization might fail to find a value within the constraints (depending on the constraints and starting values) in which case you might need to run the optimization again to find valid solutions. Choosing more suitable starting values and changing your settings to make constraints less stringent can help in these cases. See “Analyzing Point Optimization Output” in the CAGE User's Guide documentation.



- 2 Right-click the Objective Graphs view and select **Split View > Pareto Graphs**.



The view splits to show both objective and pareto graphs. You can right-click and select **Graph Size** to adjust how many plots can be displayed. In the Pareto Graphs view you can see all solutions found by the optimization at the selected operating point (the selected solution is highlighted in red). Try clicking different points in the pareto graph to see the different solutions in the objective graphs.

- 3 Click Pareto Slice () in the toolbar. This changes the table to display all solutions at a single operating point. You can scroll through the operating points using the **Run** buttons at the top. The pareto graphs always show the currently selected solution in red. Click in the table or the graph to select different solutions.


Recall that the first example, a single-objective optimization, produced a single solution at each point, so you could not view the Pareto Slice. The Pareto Slice is useful to show you the set of optimal tradeoff solutions when using multiobjective optimizations, as in this case. You can use these plots to help you select the best solution for each operating point. As you can see, this example trades off NOX emissions for torque, so it is a judgment call to choose the best depending on your priorities. You will select best solutions in a later section, “Selecting Best Solutions” on page 16-28.

- 4 Click Weighted Pareto Slice () in the toolbar.

Run: Current run: <none> Current solution: 5


Optimization Results

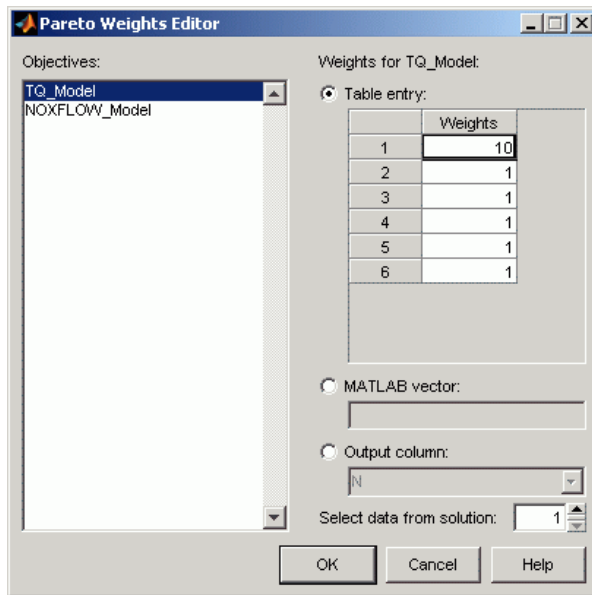
Vector display format:

Solution	 Accept	TQ_Model	NOXFLO...
1	<input checked="" type="checkbox"/>	119.626	124.037
2	<input checked="" type="checkbox"/>	143.645	135.766
3	<input checked="" type="checkbox"/>	166.544	149.986
4	<input checked="" type="checkbox"/>	187.918	167.587
5	<input checked="" type="checkbox"/>	207.213	189.77
6	<input checked="" type="checkbox"/>	223.797	217.909
7	<input checked="" type="checkbox"/>	237.154	253.132
8	<input checked="" type="checkbox"/>	247.031	295.968
9	<input checked="" type="checkbox"/>	253.294	346.526
10	<input checked="" type="checkbox"/>	255.571	404.93

This table view displays a weighted sum objective output across all operating points for each solution.

The value in the NOXFLOW_Model column in the first row shows the weighted sum of the solution 1 values of NOX across all 6 operating points. The second row shows the weighted sum of solution 2 NOX values across all 6 operating points, and so on. This can be useful, for example, for evaluating total emissions across a drive cycle. The default weights are unity (1) for each operating point.

- 5 You can alter these weights by clicking Edit Pareto Weights () in the toolbar. The Pareto Weights Editor appears.



Here you can select models, and select weights for any operating point, by clicking and editing, as shown in the example above. The same weights are applied to each solution to calculate the weighted sums. Click **OK** to apply new weights, and the weighted sums are recalculated.

You can also specify weights with a MATLAB vector or any column in your optimization output by selecting the other radio buttons. If you select **Output column** you can also specify which solution; for example you could choose to use the values of spark from solution 5 at each operating point as weights. Click **Table Entry** again, and you can then view and edit these new values.

Note Weights applied in the Weighted Pareto Slice do not alter the results of your optimization as seen in other views. You can use the weighted sums to investigate your results only. You need to perform a sum optimization if you want to optimize using weighted operating points.

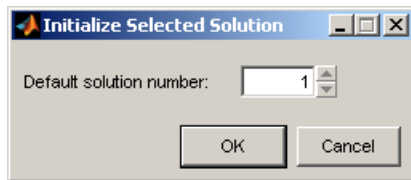
Selecting Best Solutions

In a multiobjective optimization, there is more than one possible optimal solution at each operating point. You can export all solutions, or you can select a solution for each point. You can use the Selected Solution Slice to collect and export those solutions you have decided are optimal at each operating point.


Once you have enabled the Selected Solution Slice, you can use the plots in the Pareto Slice and Solution Slice to help you select best solutions for each operating point. These solutions are saved in the Selected Solution Slice. You can then export your chosen optimization output for each point from the Selected Solution Slice, or use your chosen optimization output to fill tables.

- 1 In order to choose a single solution at each operating point, you need to enable the Selected Solution Slice. Select **Solution > Selected Solution > Initialize**.

A dialog called Initialize Selected Solution appears. Click **OK**.




The default initializes the first solution for each operating point as the selected solution.


- 2 Click the Selected Solution Slice button () which is now enabled in the toolbar. Observe that the **Current solution** number at the top is not editable, and is initially solution 1 for each operating point you click in the table. You must decide which solution is best for each point. You can select solutions in the **Selected solution** edit box, either here in the Selected Solution Slice or in the Pareto Slice or Solution Slice views.
- 3 In the Selected Solution Slice view, choose a solution for run 6. Enter **6** in the **Current run** edit box, and use the **Selected solution** controls to click through the available solutions and view each solution in the objective and pareto graphs. For example, to select solution 7 as best for the current run, enter **7** in the **Selected solution** edit box.

Leave your chosen solution selected and the table remembers your selection for each run.

If you want to select best solutions from the Pareto Slice or Solution Slice views, either:

- Click Select Solution () in the toolbar.
 - Select the menu item **Solution > Selected Solution > Select Current Solution**.
- 4 The Selected Solution Slice view collects all your selected solutions together in one place. For example, you might want to select solution 7 for the first operating point, and solution 6 best for the second, and so on.
 - 5 In order to use one solution per point from your optimization output to fill tables, you should repeat this process to select a suitable solution for all operating points. Then use the Table Filling From Optimization Results Wizard (**Solution > Fill Tables**) as before — the table is filled with your selected solution for each run.

Alternatively you could fill from a data set containing the selected solutions. To

do this, click Export to Data Set () and click **OK** in the dialog box. Go to the **Data Sets** view (click **Data Sets** in the **Data Objects** pane) to see that the table of optimization results is contained in a new data set. You could use these optimization results to fill tables. Both these table-filling methods are described in “Using Optimization Results to Fill Tables” on page 16-14.

Note that the table in the current view is exported to the data set. If you want to export your selected best solutions for each operating point, make sure you display the Selected Solution Slice before exporting the data. If you export from the Pareto Slice, the new data set contains all solutions at the single currently selected operating point set. If you export from the Solution Slice the new data set will contain the current solution at all operating points.

Recall that the previous example was a single-objective optimization and therefore only had one solution per operating point. In that case the optimization results could be exported directly from the Solution Slice, as there was no choice of solutions to be selected. See “Single-Objective Optimization” on page 16-5.

In the next tutorial section you will duplicate this NBI optimization example and alter it to create a sum optimization. For next steps, see “Sum Optimization” on page 16-30.

Sum Optimization

What Is a Sum Optimization?

In this exercise you will use a copy of the NBI optimization from the last example to create a sum optimization.

Up to this point, you have found the optimal values of each objective function at each point of an operating point set individually. A sum optimization finds the optimal value of a weighted sum of each objective function over all free variables simultaneously. The weighted sum is taken over each operating point in the run, and the weights can be edited.

A sum optimization problem without constraints, for M operating points, is the same as M individual optimizations; the minimum of the sum is the same as the sum of the minima.

To illustrate this, consider the following example:

Say the objective function is $f(a,b)$. Consider a to be the free variable and b to be the fixed variable. To set up a sum optimization, at different values of b , we want to find $[a_1, a_2, a_3 \dots a_M]$ which minimizes:

$$w_1*f(a_1,b_1) + w_2*f(a_2,b_2) + w_3*f(a_3,b_3) + \dots + w_M*f(a_M,b_M), [1]$$

where w_1, w_2 etc. are the weights.

There are two ways of viewing this problem. It can be viewed as a big optimization problem in an M -dimensional space for the vector $[a_1, a_2, a_3, \dots a_M]$ as shown in [1]. Alternatively, as each element of the sum depends on its own subset of the free variables, the problem can be written as M separate optimization problems, as in [2]:

$$\min w_i*f(a_i,b_i) \text{ for } i=1:M, [2]$$

Once the M individual problems are solved, the weighted sum can be constructed to get the answer.

When there are sum constraints present, it is not true that a M -point sum optimization problem is equivalent to solving M individual optimizations. In this case, all points must

be evaluated together to find the optimal solution that meets the sum constraints across all of the points.

In a sum optimization, the objectives are typically sum objectives. You can use a mixture of point and model sum constraints in a sum optimization. The following instructions describe these settings.

Procedure

Note: To follow these steps, you must first complete the previous steps in the tutorial to load models and create optimizations. See “Optimization and Automated Tradeoff” on page 16-2.

- 1 Right-click the `TQ_Model_Optimization_1` node in the Optimization tree and select **Duplicate TQ_Model_Optimization_1**.

A copy of the `TQ_Model_Optimization_1` node called `TQ_Model_Optimization_2` appears in the tree. You do not need an existing NBI optimization to create a sum optimization. This example is used for convenience and also to illustrate copying optimizations. This feature can be useful when you are trying different settings to improve your optimizations while keeping previous attempts for comparison.

You use this copy to create a sum optimization. This means that instead of performing the optimization at each point individually, the optimization takes the sum of solutions at all points into consideration. You can apply different weights to operating points, allowing more flexibility for some parts of the optimization.

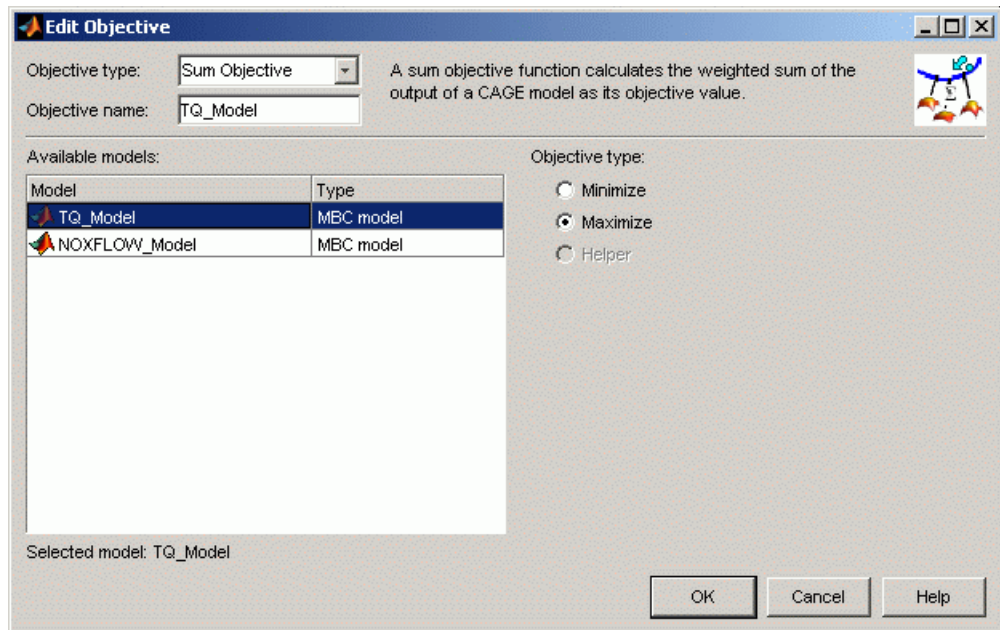
- 2 Select the node `TQ_Model_Optimization_2` and select **Edit > Rename** (or press **F2**). Edit the name to read `SUM_NBI`.

You will edit all the objectives in your existing optimization to be sum objectives.

- 3 Double-click `TQ_Model` (or right-click and select **Edit Objective**). The Edit Objective dialog appears.

To set up your new objective,

- a Select **Sum Objective** from the **Objective Type** drop-down menu.

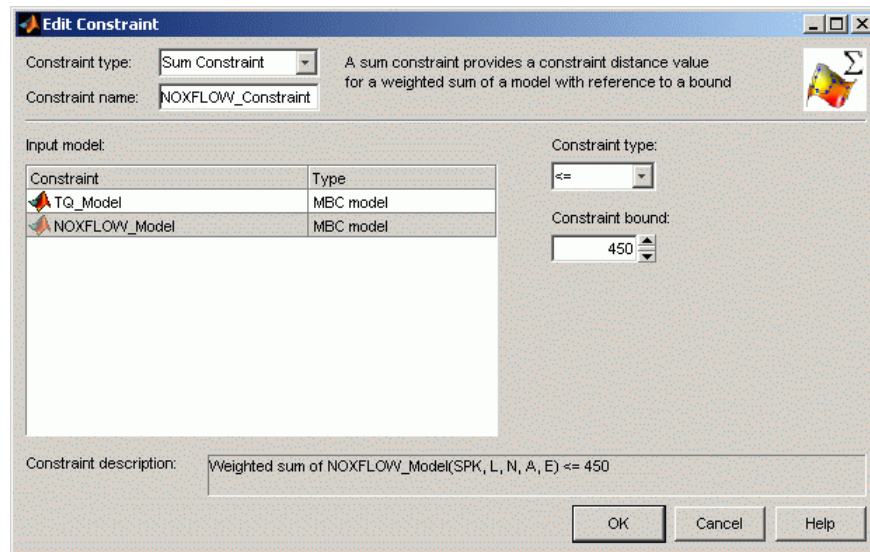


- b Select TQ_Model and make sure **Maximize** is selected. Torque has a strong correlation with fuel consumption so this sort of problem could be useful for a fuel consumption study.
- c Click **OK** to finish editing the objective.
- 4 Repeat to edit NOXFLOW_Model. Set up a sum objective to minimize NOXFLOW.
- 5 You can also edit your constraint to be a sum constraint. You can use a mixture of point and sum constraints.

To set up your sum constraint,


- a Double-click the constraint NOX1 and the Constraint Editor appears.
- b Select **Sum Constraint** from the **Constraint Type** drop-down menu, then select NOXFLOW_Model under **Input Model**.
- c Edit the **Constraint name** to NOXFLOW_Constraint to help you later distinguish it from the NOXFLOW_Model in the Optimization view fixed variables pane.

- d Make sure the inequality is \leq , and enter 450 in the constraint bound edit box as shown.



- e Press **Enter** to dismiss the dialog box and return to the Optimization view.
- 6 You want to perform a sum optimization across a series of operating points. Select **Optimization > Convert to Single Run**. Look at the change in the **Variable Values** panes. Instead of 6 separate runs you now have one run containing 6 operating points. **Number of runs** is now 1, and **Number of Values** is 6 for all variables (you use these controls to set the number of operating points within each run). Each solution will be a sum across all the points in the run.
- 7 Look in the **Fixed Variables** pane for the weights columns, last columns on the right. Enter 5 in the first row (run 1, point 1) for **TQ_Model_weights**, **NOX_weights**, and **NOXFLOW_Constraint_weights**, to weight the first operating point by five, as shown.

Fixed Variables								
Variable:	TQ_Model_weights	L	N	A	E	NOXFLOW_Model_weights	NOXFLOW_Constraint_wei...	
Number of values:	6	6	6	6	6	6	6	
1	(1)	5	0.1	1000	12	5	5	5
	(2)	1	0.8	1000	12	5	1	1
	(3)	1	0.1	3000	12	5	1	1
	(4)	1	0.8	3000	12	5	1	1
	(5)	1	0.1	6000	12	5	1	1
	(6)	1	0.8	6000	12	5	1	1

- 8 You have modified your objectives and constraint for a sum optimization, which is ready to run. Click Run Optimization () in the toolbar.
- 9 There is a wait notice as the optimization runs. There are no progress messages as points are evaluated because sum optimizations do not evaluate points individually.

When the optimization is complete, examine the results at the output node. Select the Pareto Slice table. Look at the objective and constraint results for the ten solutions (number of solutions is specified in the Optimization Parameters dialog). In the Pareto Slice table, negative constraint values are within the constraint. Look at the constraint summary view, where the left and right information corresponds to the constraint inequality; the left value is the distance from the constraint.

For more information see “Interpreting Sum Optimization Output” in the CAGE User’s Guide documentation.

For next steps, see “Automated Tradeoff” on page 16-35.

Automated Tradeoff


Once you have set up an optimization you can fill tables in a tradeoff using automated tradeoff. You can select cells and fill them from the results of an optimization. The cells you select in the tradeoff table define the operating point set for the optimization.

Set up a tradeoff as follows (also described in “Setting Up a Tradeoff Calibration” on page 13-2).

Note: To follow these steps, you must first complete the previous steps in the tutorial to load models and create an optimization. See “Optimization and Automated Tradeoff” on page 16-2.

- 1 Select **File > New > Tradeoff**.

This takes you to the **Tradeoff** view. You need to add tables to the tradeoff.

- 2 Click  (Add New Table). This opens the Table Setup dialog.
- 3 Enter **Spark** as the table **Name**.
- 4 Select **L** as the **Y input** and **N** as the **X input**.
- 5 Click **Select** to open the Select Filling Item dialog.

- a Select the radio button to **Display variables**.

- b Click to select **SPK**.

- c Click **OK** to return to the Table Setup dialog.


- 6 Leave **10** as the size of the rows and columns (the speed and load axes), and **0** as the initial value, and click **OK**.

A new **Spark** table appears in the **Tradeoff** tree. CAGE has automatically spaced the normalizers evenly over the ranges of **N** and **L**.


- 7 Click to expand the **New_Tradeoff** tree and select the **Spark** table node to view the new table.

You need to select the cells where you want to apply automated tradeoff. Create a region within the table:

- 1 Highlight a rectangle of cells in the **SPK** table by clicking and dragging. Note that a large region can take a very long time to evaluate. Try four cells to start with.

- 2 Click  (or right-click and select **Extrapolation Regions > Add Selection**) to define the region. The cells become light blue.

To use automated tradeoff on the cells in the defined region,

- 1 Select **Inputs > Automated Tradeoff** or click the toolbar button .

The Automated Tradeoff dialog appears, showing a list of available optimizations in your session that are set up and ready to run.

- 2 Select your **TQ_Model_Optimization_1** multiobjective optimization (of **Type: Normal Boundary Intersection Algorithm**) to apply to the tradeoff and click **OK**.
- 3 Click **OK** in the following dialog to optimize only the table cells that are in the region, rather than all cells.

The automated tradeoff optimization runs. The results appear in the selected cells in the table, as shown in the example. You could optimize several regions and then use these results to extrapolate across the whole table.

