Model-Based Calibration Toolbox

For Use with MATLAB®

Computation

Visualization

Programming

CAGE User's Guide

Version 1

The MathWorks

How to Contact The MathWorks:

`www.mathworks.com`	Web
`comp.soft-sys.matlab`	Newsgroup
`support@mathworks.com`	Technical support
`suggest@mathworks.com`	Product enhancement suggestions
`bugs@mathworks.com`	Bug reports
`doc@mathworks.com`	Documentation error reports
`service@mathworks.com`	Order status, license renewals, passcodes
`info@mathworks.com`	Sales, pricing, and general information
508-647-7000	Phone
508-647-7001	Fax
The MathWorks, Inc. 3 Apple Hill Drive Natick, MA 01760-2098	Mail

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

Model-Based Calibration Toolbox CAGE User's Guide

Printing History: July 2002 Online only New for Version 1.1 (Release 13)

Contents

1 Getting Started

2 Feature Calibration Tutorial

3 Tradeoff Calibration Tutorial

4 Data Sets Tutorial

5 Filling Tables from Data Tutorial

6 Using CAGE

7 Normalizers

8 Feature Calibrations

9 Tradeoff Calibrations

10 Data Sets

11 Calibration Manager

12 Surface Viewer

13 Manual Calibration and the History Display

Index

1

Getting Started

The following sections introduce the CAGE browser part of the Model-Based Calibration Toolbox.

About 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 models from the Model Browser part of the Model-Based Calibration toolbox. From these models, CAGE builds steady-state ECU calibrations.

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

Calibrating Lookup Tables Using CAGE

There are two different types of calibration that you can perform using CAGE:

- Feature calibration
- Tradeoff calibration

Feature Calibration

A feature calibration compares a model of an estimated signal with a lookup table (or algebraic collection of tables) which estimates the same signal in the ECU. CAGE finds the optimum calibration for the lookup table(s).

For example, a typical engine subsystem controls the spark angle to produce the peak torque; that is, the Maximum Brake Torque (MBT) spark. Using the Model Browser, you can build a statistically sound model of MBT spark, over a range of engine speeds and relative air charges, or loads. Use the feature calibration to fill a lookup table by comparing the table to the model.

Tradeoff Calibration

A tradeoff calibration fills lookup tables by comparing models of different engine characteristics at key operating points.

For example, there are several models of important engine characteristics, such as torque and Nitrogen Oxides (NOx) emissions. Both models depend on the spark angle. At a particular operating point, a slight reduction of torque may result in a dramatic reduction of NOx emissions. Thus, the calibrator uses the value of the spark angle that gives this reduction in NOx emissions in favor of the spark angle that generates maximum torque.

Comparing Calibrations to Data

You can compare your calibrations to experimental data, for validation.

For example, after completing a calibration, you can import experimental data from a spreadsheet. You can use CAGE to compare your calibration to the data.

Starting CAGE

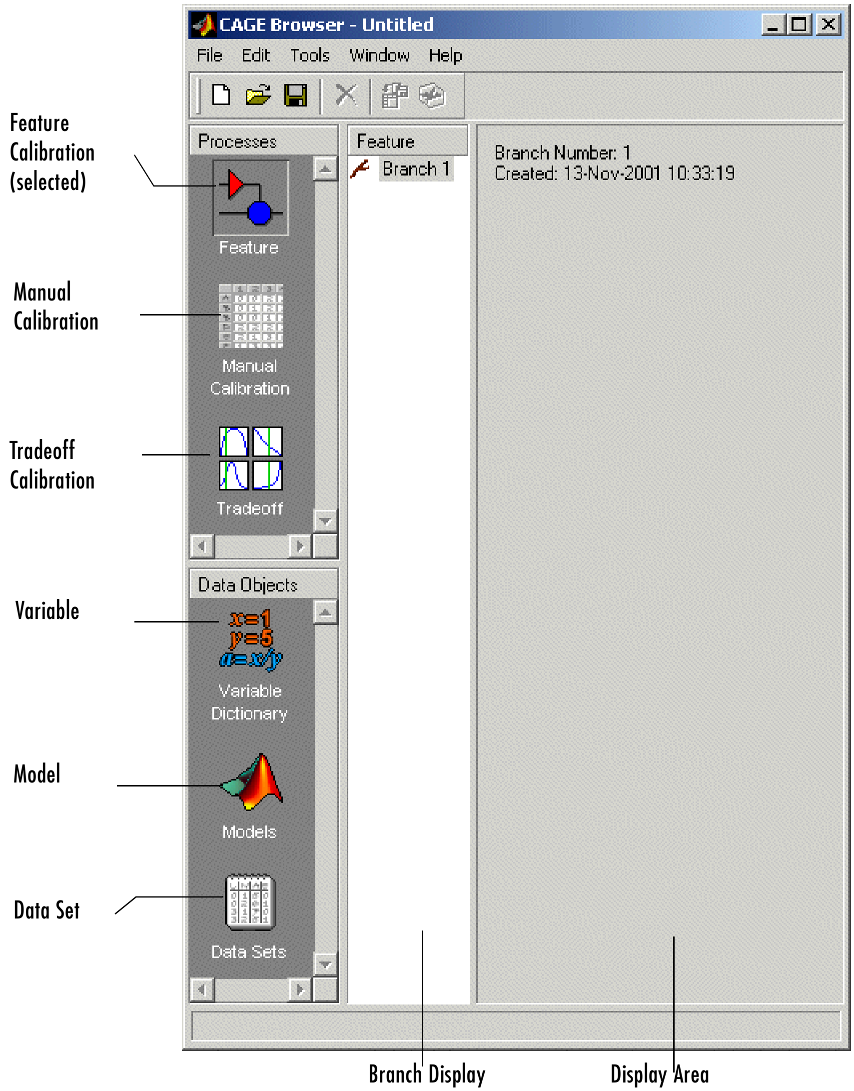
To start the application, type

```
cage
```

The view of CAGE depends on two things:

- The process or object type that you are viewing
- The item you highlight in the branch display (tree)

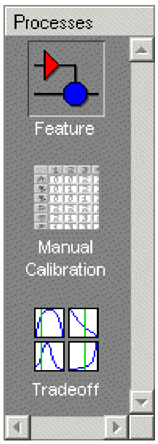
When you open CAGE, it looks like this.


CAGE Browser - Untitled
File Edit Tools Window Help
Feature Calibration (selected)
Manual Calibration
Tradeoff Calibration
Variable
Model
Data Set
Processes
Feature
Manual Calibration
Tradeoff
Data Objects
Variable Dictionary
Models
Data Sets
Feature
Branch 1
Branch Number: 1
Created: 13-Nov-2001 10:33:19
Branch Display
Display Area

CAGE includes a **Processes** pane and a **Data Objects** pane to help you identify the type of calibration you want to do, and the data objects that you intend to use.

Processes

The **Processes** pane enables you to select the type of calibration that you want to perform with CAGE. For example, if you want to calibrate lookup tables by comparing them to a model, select **Feature**.



The **Processes** pane has three buttons:

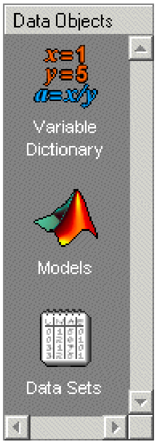
- **Feature** shows the **Feature** view, with any tables or strategies that are associated with that feature.

  For more information, see "Feature Calibrations" on page 8-1.
- **Manual Calibration** enables you to calibrate tables manually. It also acts as a store for all the tables and normalizers in your session.
- **Tradeoff** shows the **Tradeoff** view, with a list of the tables and models to display.

  For more information, see "Tradeoff Calibrations" on page 9-1.

Data Objects

The **Data Objects** pane lets you identify the different objects that you need to perform the calibrations with CAGE. For example, if you want to fill lookup tables by comparing them to models, you need to include models. Select **Models** to view the models in your session.



The **Data Objects** pane has these buttons:

- **Variable Dictionary** stores all the variables, constants and formulas in your session. Click **Variable Dictionary** to view and edit any variables in any part of your session.

  For more information, see "Setting Up Your Variable Items" on page 6-7.
- **Models** stores all of the models in your session. Click **Models** to show a graphical display of the models in your session.

  For more information, see "Setting Up Your Models" on page 6-14.
- **Data Sets** enables you to see the data produced by your objects, and enables you to compare this with other data such as experimental data.

  For more information, see "Data Sets" on page 10-1.

For a more detailed overview of CAGE functionality in different views and links to in-depth help on each topic, see "Using CAGE" on page 6-1.

How to Use This Manual

This manual is the CAGE User's Guide. See also the Model Browser User's Guide for information on the other main interface of the Model-Based Calibration Toolbox.

Learning CAGE

There are four tutorial chapters, with worked examples to guide you through using the tools in CAGE:

- "Feature Calibration Tutorial" on page 2-1 describes how to set up and calibrate lookup tables by reference to a model.
- "Tradeoff Calibration Tutorial" on page 3-1 describes how to calibrate lookup tables using tradeoff calibrations.
- "Data Sets Tutorial" on page 4-1 describes how to validate calibrations using experimental data.
- "Filling Tables from Data Tutorial" on page 5-1 describes how to fill lookup tables using experimental data.

Using CAGE

- "Using CAGE" on page 6-1 describes how to set up CAGE sessions before performing calibrations and gives an overview of where in CAGE to find all the functionality for different processes.
- "Normalizers" on page 7-1 describes what normalizers are, and how to space breakpoints in a normalizer.
- "Feature Calibrations" on page 8-1 describes how to calibrate lookup tables by reference to models built using the model browser.
- "Tradeoff Calibrations" on page 9-1 describes how to calibrate lookup tables by adjusting one value to fulfill different objectives.
- "Data Sets" on page 10-1 describes how to use CAGE to compare calibrations to experimental data, and how to use experimental data to fill lookup tables.
- "Calibration Manager" on page 11-1 describes how to use the **Calibration Manager**.
- "Surface Viewer" on page 12-1 describes how to use the **Surface Viewer**.

- "Manual Calibration and the History Display" on page 13-1 describes how to use the **History** viewer, and how to add and delete tables and manually calibrate tables.

Training Material

The files for the tutorial chapters are all contained in the `matlab/toolbox/mbc/mbctraining` directory.

System Requirements

This section lists the following:

- Hardware requirements
- Operating system requirements
- Required MathWorks products
- Optional MathWorks products

Hardware Requirements

The Model-Based Calibration Toolbox has been tested on the following processors:

- Pentium, Pentium Pro, Pentium II, Pentium III, and Pentium IV
- AMD Athlon

Minimum memory:

- 256 MB

Minimum disk space:

- 450 MB for the software and the documentation

Operating System Requirements

The Model-Based Calibration Toolbox is a PC-Windows only product.

The product has been tested on

Microsoft Windows NT, 2000, and 98.

You can see the system requirements for MATLAB online at

`http://www.mathworks.com/products/system.shtml/Windows`

Required MathWorks Products

Model-Based Calibration requires the following other MathWorks products:

- Simulink®
- Optimization Toolbox
- Statistics Toolbox
- Extended Symbolic Math Toolbox

Optional MathWorks Products

The Model-Based Calibration Toolbox can use the following MathWorks product:

- Neural Network Toolbox

2

Feature Calibration Tutorial

This section describes what a feature calibration is, and illustrates the various processes required to perform a feature calibration.

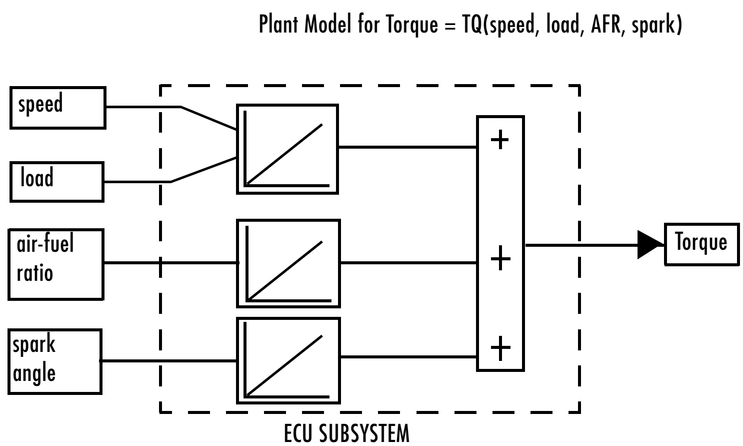
- “What Are Feature Calibrations?” on page 2-3 describes what a feature calibration is.
- “Setting Up Calibrations” on page 2-4 describes the processes required to set up calibrations in CAGE.
- “Creating a Feature Calibration” on page 2-11 describes the various parts of the feature calibration, and how to incorporate them into a calibration in CAGE.
- “Calibrating a Feature” on page 2-17 describes the processes of calibrating the various aspects of a feature.
- “Exporting Calibrations” on page 2-26 describes how to export the calibration when it is completed.

What Are Feature Calibrations?

The feature calibration process within the Model-Based Calibration Toolbox 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.

An example of an ECU subsytem 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.



In this 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.

Setting Up Calibrations

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.

Setting 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** displays all the variables, constants, and formulas in a session.

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:

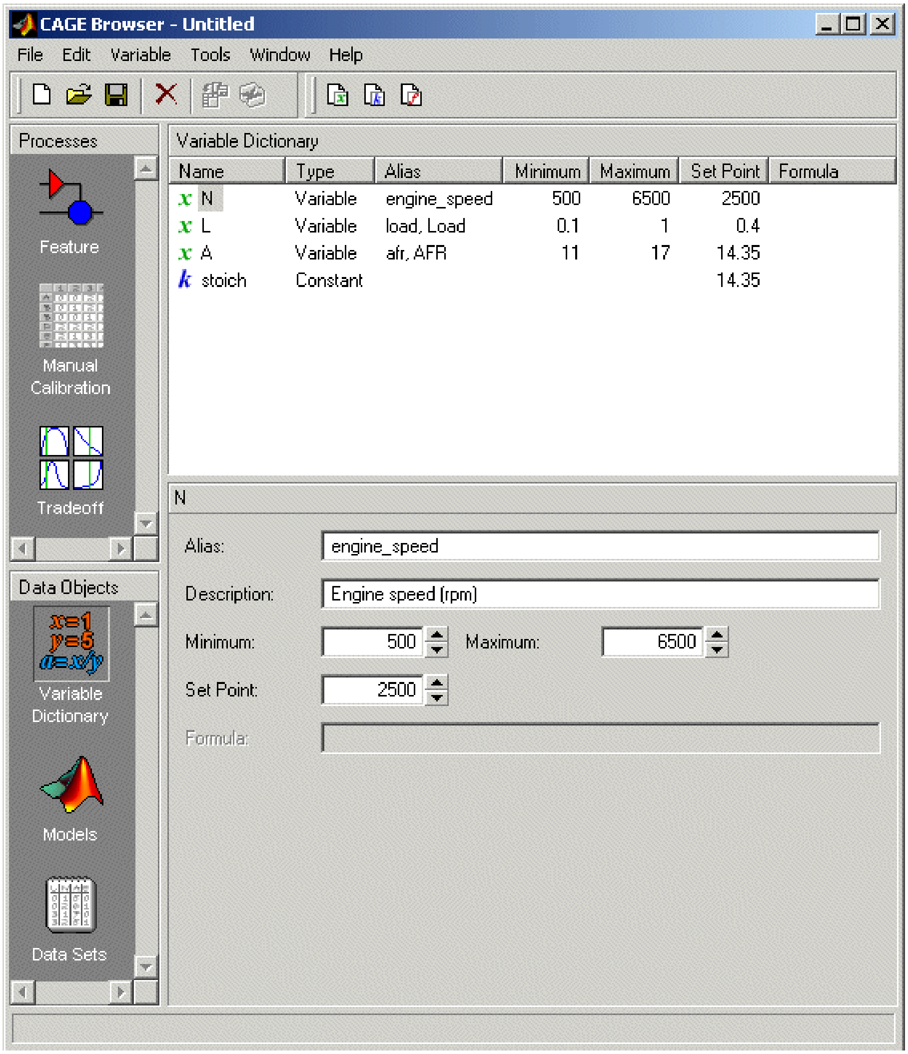
1 Select **File –> Import –> Variable Dictionary**.

2 Select the `tutorial.xml` file found in `matlab\toolbox\mbc\mbctraining` and click **Open**.

This imports a set of variables and a constant. In our example, the variable dictionary contains

- `N`, engine speed
- `L`, load
- `A`, AFR
- The stoichiometric constant, `stoich`

Your display should resemble the following.



Adding and Editing Variables and Constants

To add a variable for the spark angle:

1 Click  in the toolbar. This adds a new variable to your dictionary.

2 Select **Edit –> Rename** 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`:

5 Enter `S, spark` in the **Alias** edit box.

6 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.

7 Enter `25` in the **Set Point** edit box to specify the preferred value for `spk`.

For more information about the variables, see “Setting Up Your Variable Items” on page 6-7.

Setting Up Models

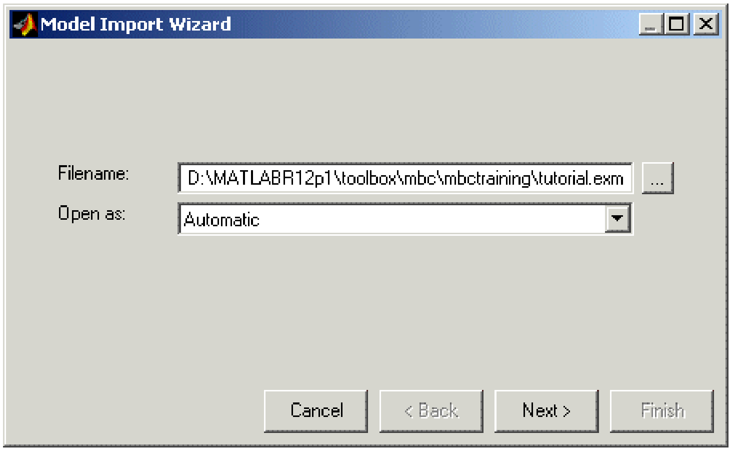
A model in the Model-Based Calibration Toolbox is a function of a set of variables. Typically, you construct a model using the Model Browser; then you can use CAGE to calibrate lookup tables by reference to the model.

The following example uses a model of how torque behaves 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 starts the **Model Import Wizard**.

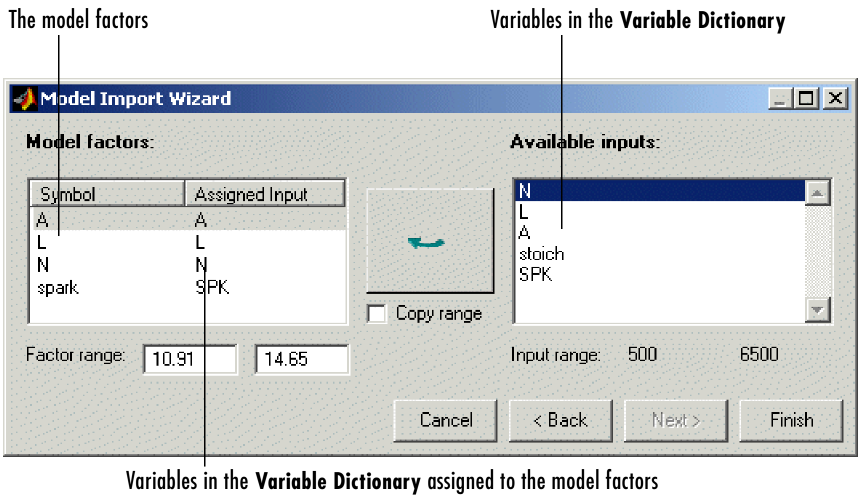


2 Click  to browse for the correct model file.

3 Select the `tutorial.exm` file, found in `matlab\toolbox\mbc\mbctraining` (this is a copy of the torque model built in the Model Browser's quick start tutorial) and click **Open**.

4 To accept the default setting, leave **Open as:** `Automatic` and click **Next**.

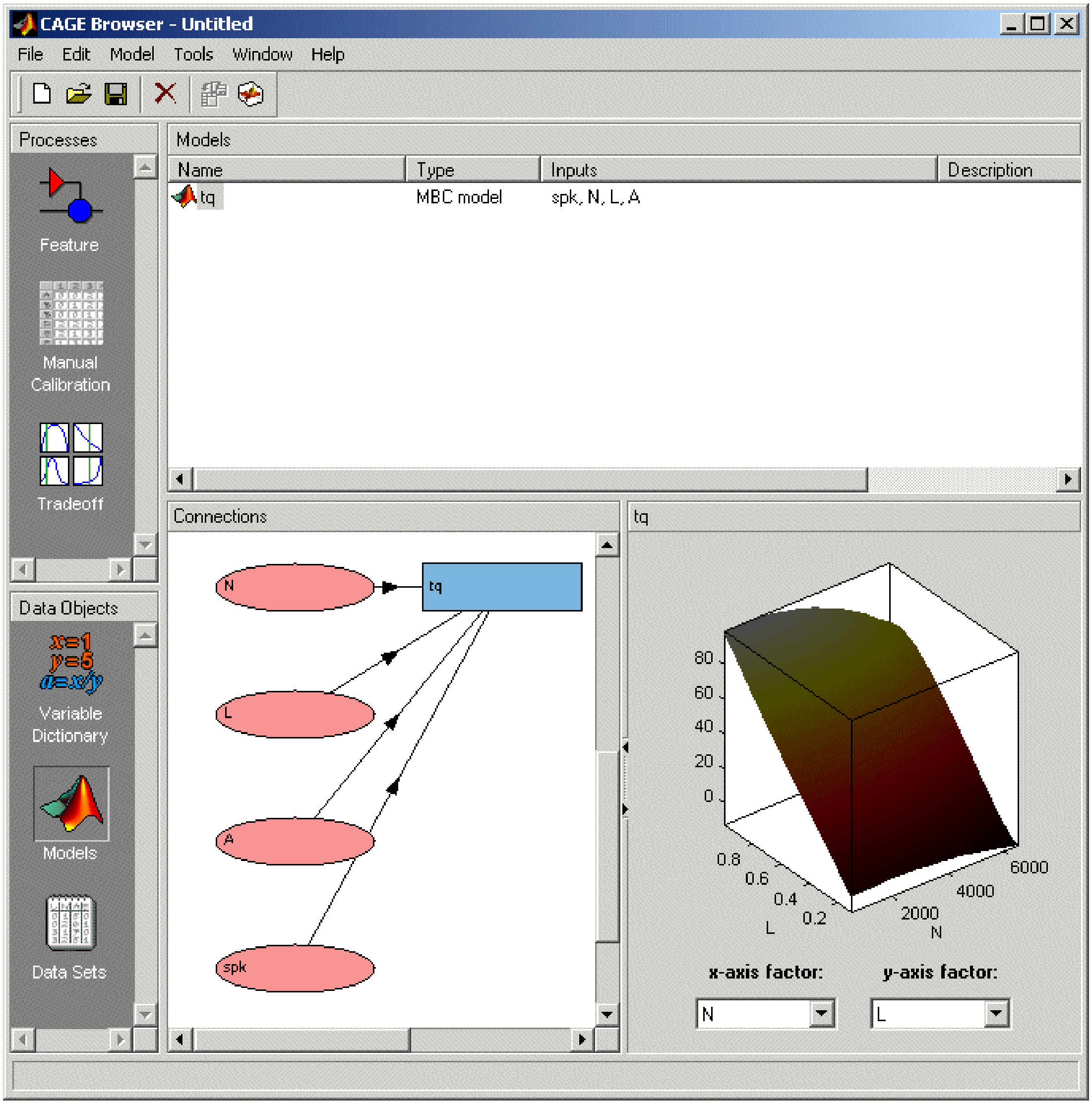
5 There are two models stored in this file, `tq` and `knot`. Highlight `tq` and click **Next** to show the following.



CAGE automatically assigns variables in the variable dictionary to the model input factors or their aliases (as long as names are exact).

6 Click **Finish** to complete the wizard.

When you complete the wizard, you return to the **Model**s view.



For more information about models, see “Setting Up Your Models” on page 6-14.

Creating a Feature Calibration

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:

1. Select **File –> New –> Feature**.

   This automatically displays the **Feature** pane and creates a new feature.

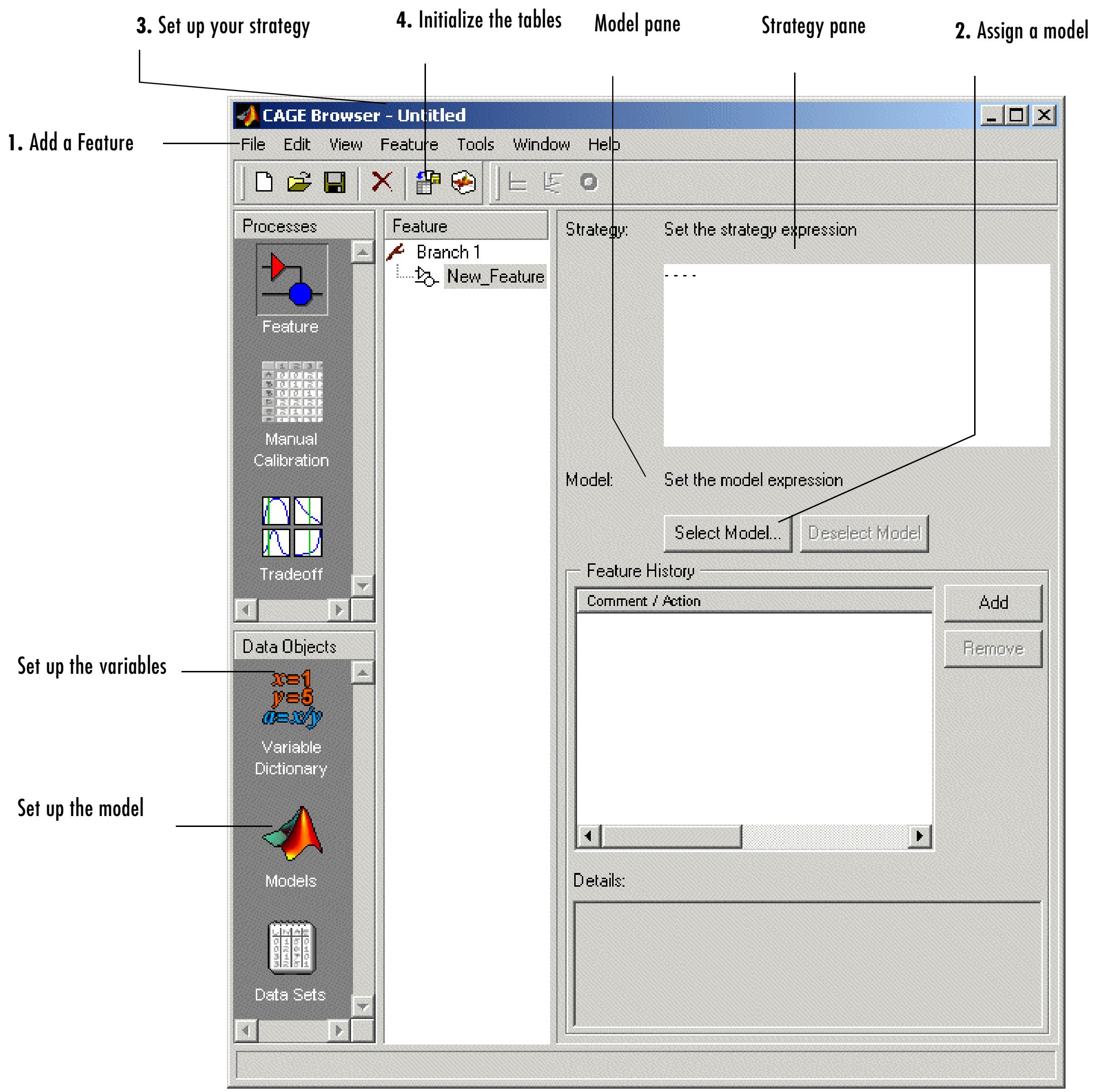
2. Click **Select Model**. This automatically selects `tq`, the torque model, for you (because there is currently only one model in your project).

   You can see the model appear above the **Select Model** button.

3. Create a strategy. For more information, see the next section, "Setting Up the Strategy" on page 2-13.

   A strategy is a collection of tables. The Model-Based Calibration Toolbox uses Simulink® to enable you to graphically specify the collection of tables for a feature.

4. Set up your tables. For more information, see the following section, "Setting Up the Tables" on page 2-14.


3. Set up your strategy
4. Initialize the tables
Model pane
Strategy pane
2. Assign a model
1. Add a Feature
CAGE Browser - Untitled
File Edit View Feature Tools Window Help
Processes
Feature
Manual Calibration
Tradeoff
Feature
Branch 1
New_Feature
Strategy:
Set the strategy expression
Model:
Set the model expression
Select Model...
Deselect Model
Feature History
Comment / Action
Add
Remove
Details:
Data Objects
Set up the variables
x=1
y=5
a=x/y
Variable Dictionary
Set up the model
Models
Data Sets

Setting Up the Strategy

The toolbox uses Simulink to graphically specify the strategy.

Importing a Strategy

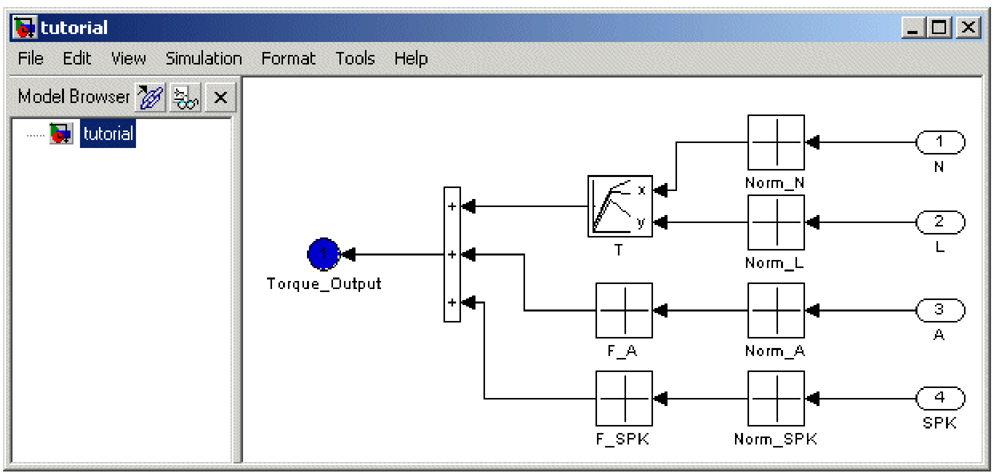
To import a strategy:

1 Select **File –> Import –> Strategy**.

2 Select the file called `tutorial.mdl`, found in `matlab\toolbox\mbc\mbctraining` and click **Open**.

3 This opens the **Import Strategy** dialog box. To view the strategy, click **Manual**.

   This opens the following Simulink window.



   This shows how the strategy is built up.

4 Now double-click the blue circle labeled `Torque_Output`.

Note This shuts down the Simulink window and parses the new feature into the calibration browser.

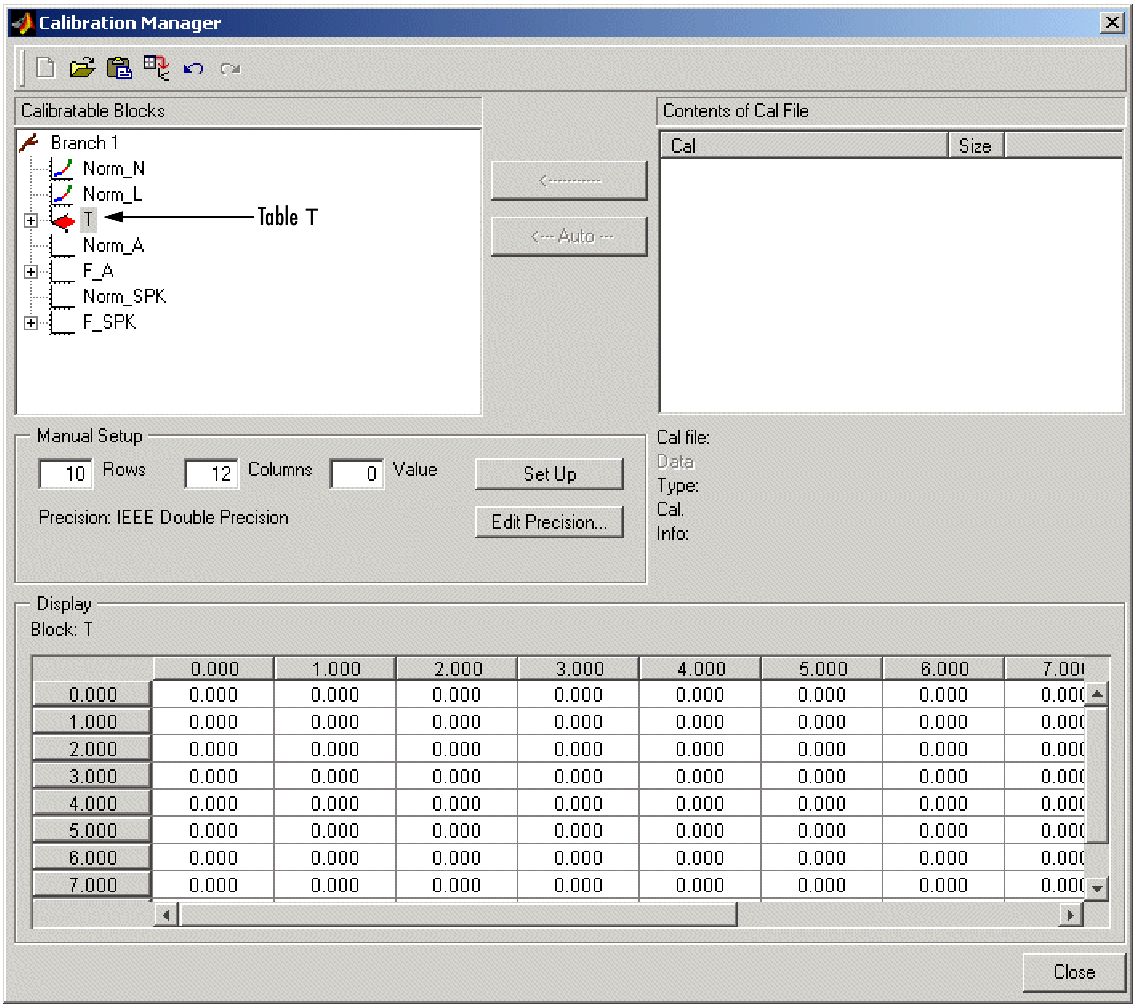
The `New_Feature` is the output of the algebraic equation of tables. You can see this parsed into the **Strategy** pane.

For a more detailed description of the strategies, see “Setting Up Your Strategy” on page 8-6.

Setting Up the Tables

Currently, the lookup tables have neither rows nor columns, so you must set up the tables.

Click  or select **Tools –> Calibration Manager**. The **Calibration Manager** dialog box opens, so you can specify the number of breakpoints for each axis.


Calibration Manager
Calibratable Blocks
Branch 1
Norm_N
Norm_L
T
Table T
Norm_A
F_A
Norm_SPK
F_SPK
<-----------
<--- Auto ---
Contents of Cal File
Cal
Size
Manual Setup
10
Rows
12
Columns
0
Value
Set Up
Precision: IEEE Double Precision
Edit Precision...
Cal file:
Data
Type:
Cal.
Info:
Display
Block: T
0.000 1.000 2.000 3.000 4.000 5.000 6.000
0.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000
1.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000
2.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000
3.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000
4.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000
5.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000
6.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000
7.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000
Close

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 in each cell to be 0.

4 Click **Set Up** to change the **Display** pane to show the table is set up.

5 Follow the same procedure for the F_A table. In other words,

   a Highlight the F_A node.

   b Set the number of rows to be 10.

   c Set the value in each cell to be 0.

   d Click **Set Up**.

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.

Calibrating a Feature

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:

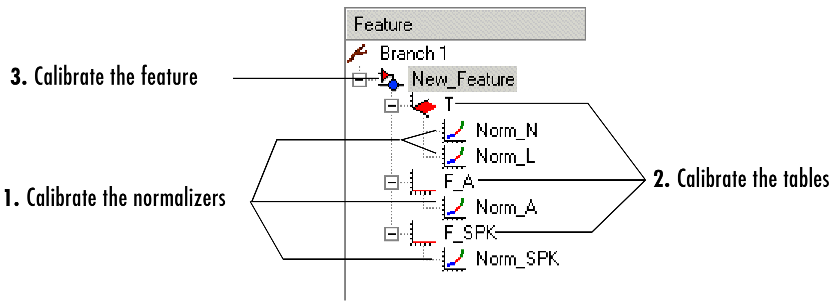
1 Calibrate the normalizers.

2 Calibrate the tables.

3 Calibrate the feature as a whole.

These steps are described in the next sections.

Click the expand icon, ⊞, to expand the nodes and display all the tables and normalizers in the feature.

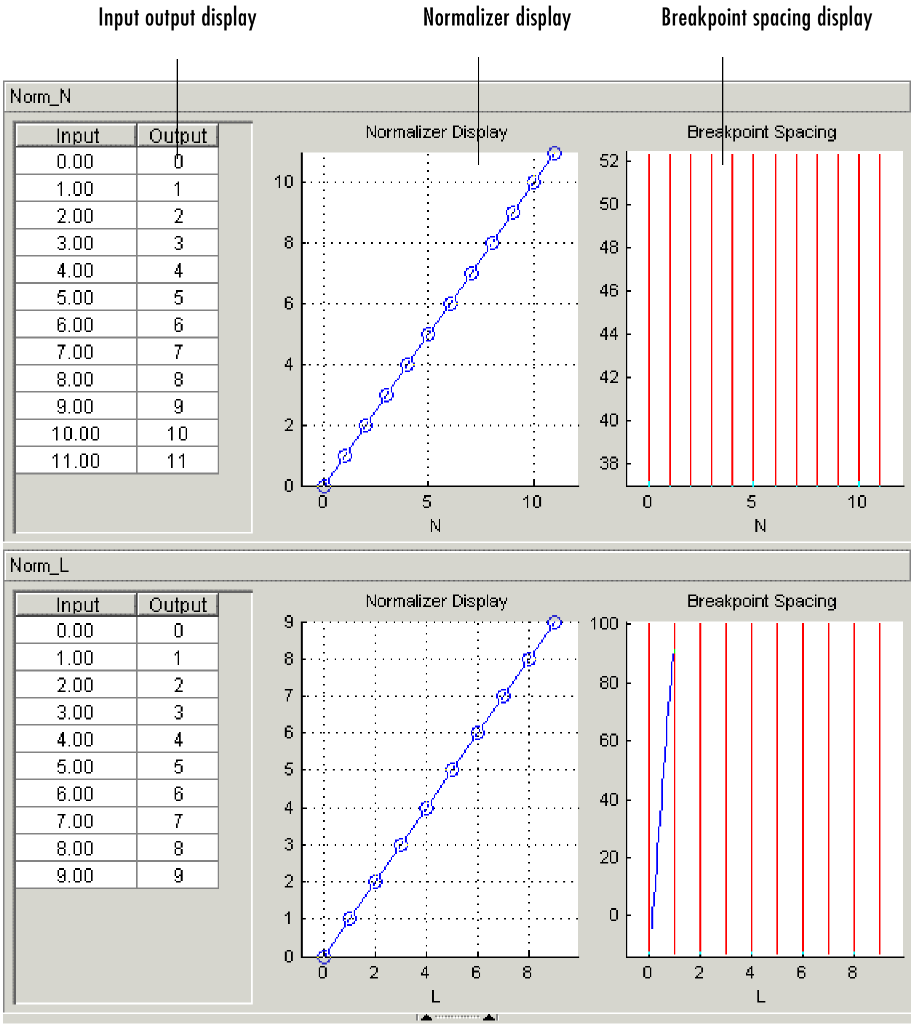


Each node in the display has a different view and different operations.

Calibrating 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 “Normalizer View” on page 7-14.

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  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  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.

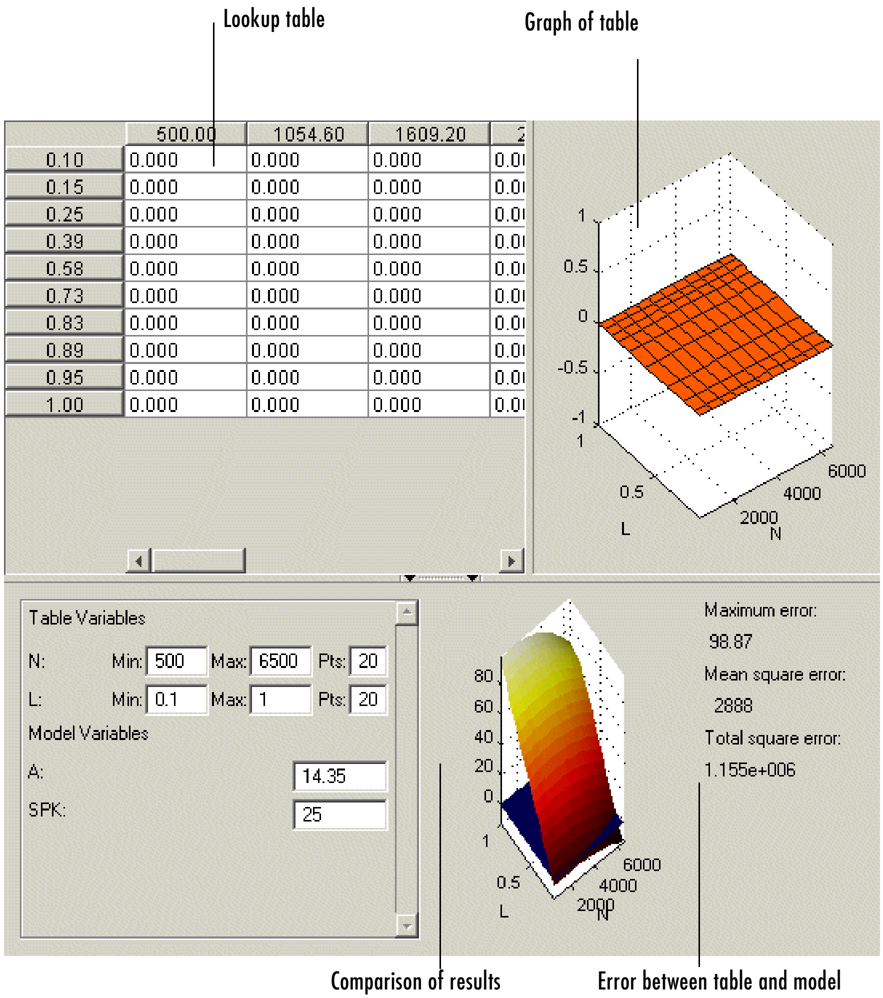
   For more information about calibrating the normalizers, see “Normalizers” on page 7-1.

You can now calibrate the lookup tables; this is described in the next section.

Calibrating 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 three panes: the table, the graph, and the comparison-of-results pane.

To fill the table with values of the model at the appropriate operating points:

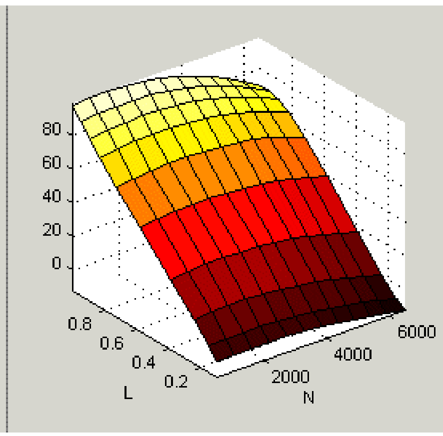
1 Click on the toolbar.

This opens a dialog box that suggests the set points of `AFR` and `SPK` as appropriate values for evaluating the model over the range of `N` and `L`.

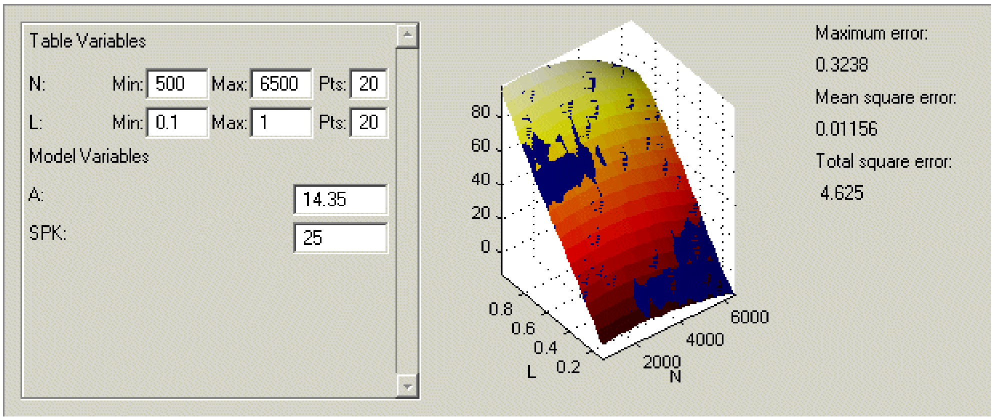
2 Click **OK**.

The following view shows the table filled with values of the model.

	500.00	1054.60	1609.20
0.10	-3.837	-3.190	-2.865
0.15	2.495	3.117	3.380
0.25	12.899	13.673	14.047
0.39	28.748	29.753	30.359
0.58	48.652	50.061	51.010
0.73	64.542	66.105	67.180
0.83	76.353	77.852	78.853
0.89	84.193	85.556	86.416
0.95	90.974	92.151	92.823
1.00	97.626	98.548	98.903



The following comparison-of-results pane shows just how good a fit the strategy is to the model.



The model is represented by the multicolored surface and the strategy is the blue surface.

The table T is now filled with values of the model at these operating points.

For more information about the process of filling tables, see "Calibrating the Tables" on page 8-12.

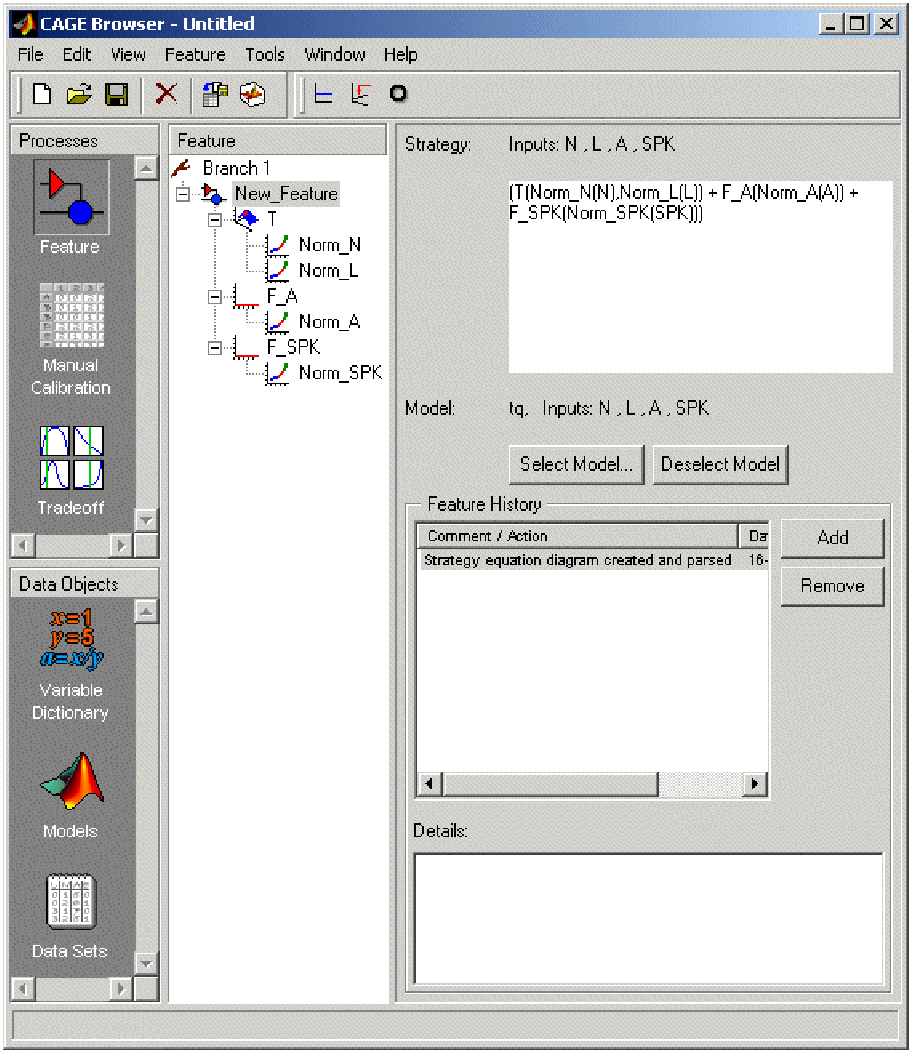
Now you must fill the tables F_A and F_SPK and their normalizers. The tables are modifiers for AFR and the spark angle respectively. These steps are described in the next section.

Calibrating the Feature

A feature is a strategy (which is a collection of tables) and a model. Currently the torque table, T, is filled with values of 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 New_Feature node.



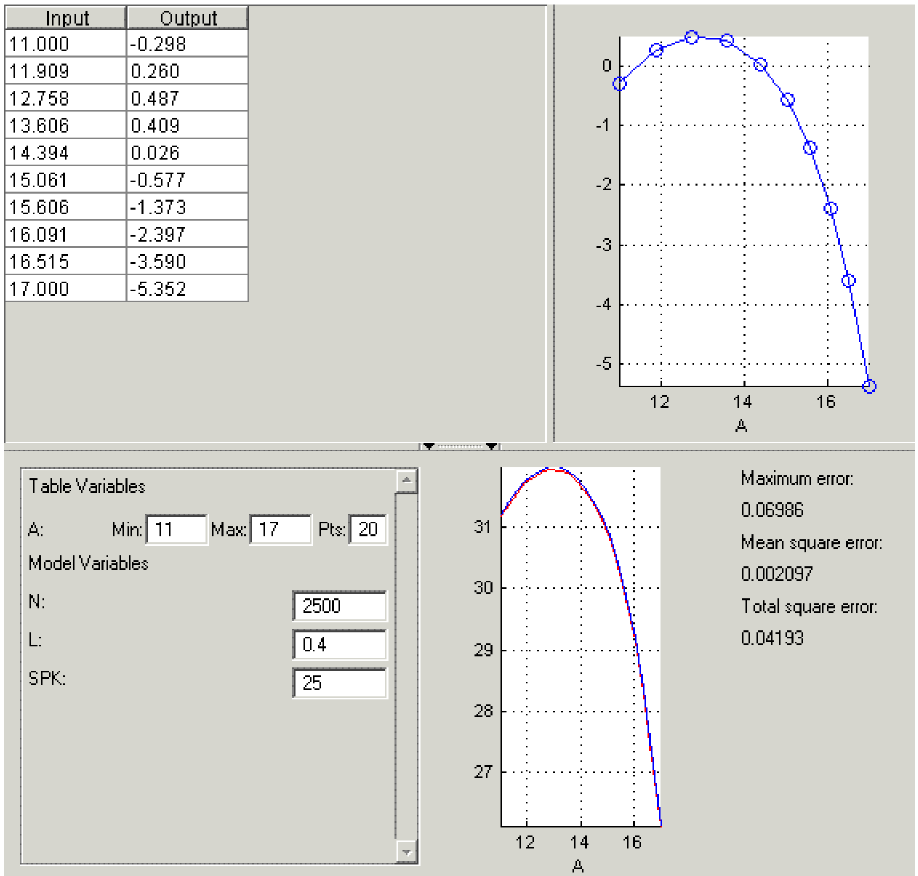
To calibrate all the tables and their normalizers:

1 Select **Feature –> Fill** to open a dialog box.

2 Confirm the variable ranges and the table-filling order by clicking **OK** in the **Feature Filling Options** dialog box.

All three tables and normalizers are filled.

As the model and the feature are four-dimensional objects, it is difficult to fully view a comparison between the feature and the model. A meaningful comparison is shown in the lower half of the following figure (select the `F_A` node in the branch display). The equation *model* = *strategy* is rearranged so that the table is compared to the model and the remainder of the strategy.


Input	Output
11.000	-0.298
11.909	0.260
12.758	0.487
13.606	0.409
14.394	0.026
15.061	-0.577
15.606	-1.373
16.091	-2.397
16.515	-3.590
17.000	-5.352

This display shows that the range of the normalizer for `F_A` is 11 to 17, the range of AFR. The lower pane shows a comparison between the red strategy and a slice through the model, over the range of AFR.

You can use CAGE to improve on these results. CAGE can run an optimization routine over the feature to minimize the total square error between the model and the feature.

To optimize the feature:

1 Select the `New_Feature` node

2 Click . This opens a dialog box, suggesting ranges for the variables.

3 To confirm the default variable ranges and table-filling order, click **OK** in the dialog box.

This reduces the error between the feature and the model.

To view this reduction in error, select the `F_A` node in the branch display.

Notice that the mean square error between the model and the feature over this range of values is 0.001348, which is less than the 0.002097 previously obtained.

This completes the calibration of the torque feature.

For more information about calibrating features, see “Calibrating the Feature Node” on page 8-38.

You now need to export the calibration for the ECU.

Exporting Calibrations

To export your feature:

1 Select the `New_Feature` node in the branch display.

2 Select **File –> Export –> Calibration**.

3 Choose the type of file you want to save your calibrations as. You can choose from
   - **Comma Separated Value (.csv)**
   - **MAT-file (.mat)**
   - **M-file script**

4 For the purposes of this tutorial, select **Comma Separated Value (.csv)**.

5 Enter `tutorial.csv` as the file name and click **Save**.

This exports the successful calibration, ready for the ECU.

Note that 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.

3

Tradeoff Calibration Tutorial

This tutorial describes what a tradeoff calibration is and illustrates the various processes required to perform a tradeoff calibration.

- "What Is a Tradeoff Calibration?" on page 3-3 describes what a tradeoff calibration is.
- "Creating a Tradeoff Calibration" on page 3-4 describes the processes required to set up a tradeoff calibration.
- "Performing the Tradeoff Calibration" on page 3-8 describes the processes of calibrating the various parts of a tradeoff.
- "Exporting Calibrations" on page 3-18 describes how to export the calibration when it is completed.

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.

Creating a Tradeoff Calibration

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**.

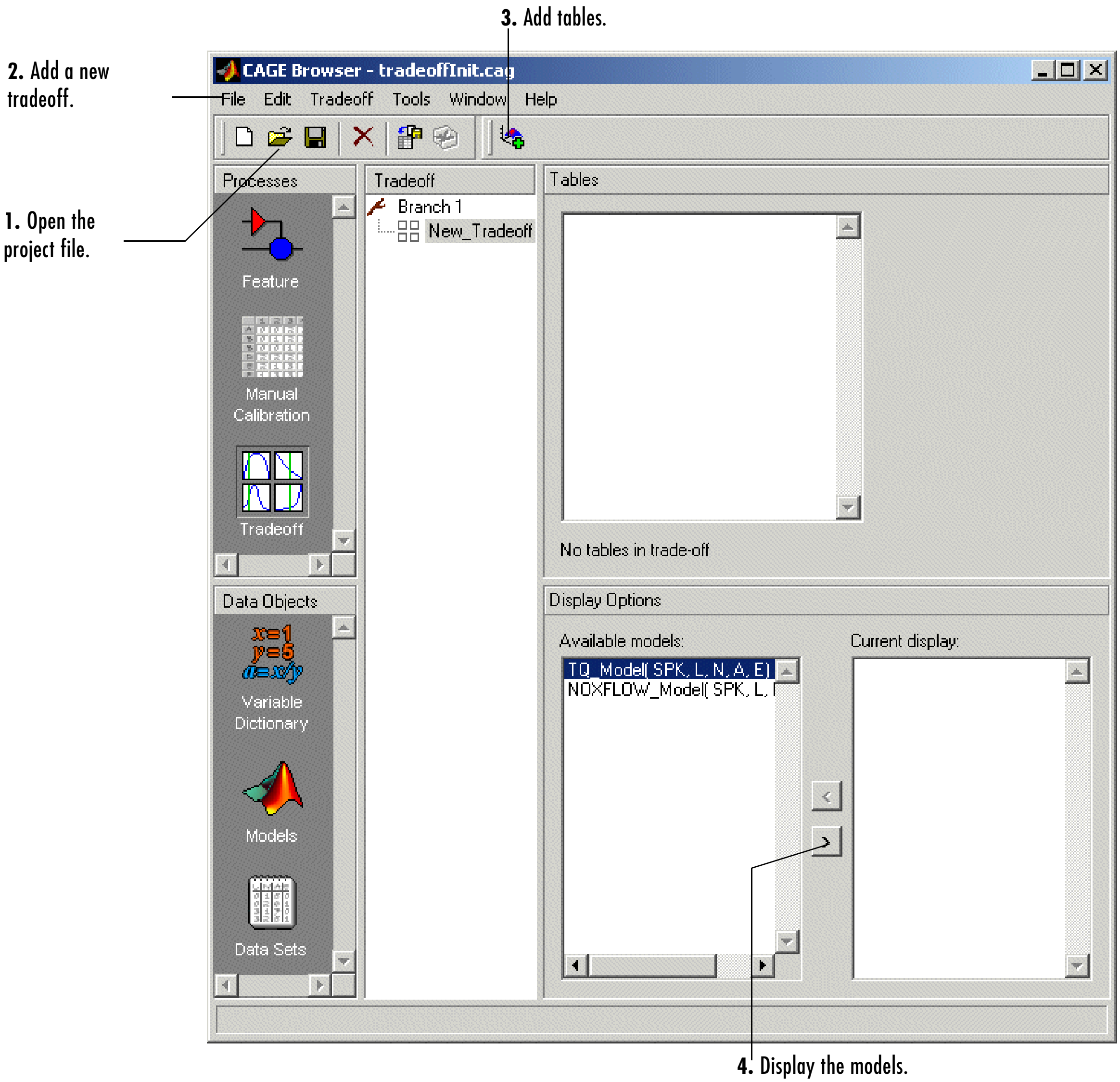
   The `tradeoffInit.cag` project contains two models and all the variables necessary for this tutorial. For information about how to set up models and variables, see “Using CAGE” on page 6-1.

To create a tradeoff calibration:

2 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 3-6.
   - “Displaying the Models” on page 3-7 describes how you display the models of torque and NOx emissions.


3. Add tables.
2. Add a new tradeoff.
1. Open the project file.
4. Display the models.
CAGE Browser - tradeoffInit.cag
File Edit Tradeoff Tools Window Help
Processes
Feature
Manual Calibration
Tradeoff
Data Objects
Variable Dictionary
Models
Data Sets
Tradeoff
Branch 1
New_Tradeoff
Tables
No tables in trade-off
Display Options
Available models:
TQ_Model(SPK, L, N, A, E)
NOXFLOW_Model(SPK, L,
Current display:

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.

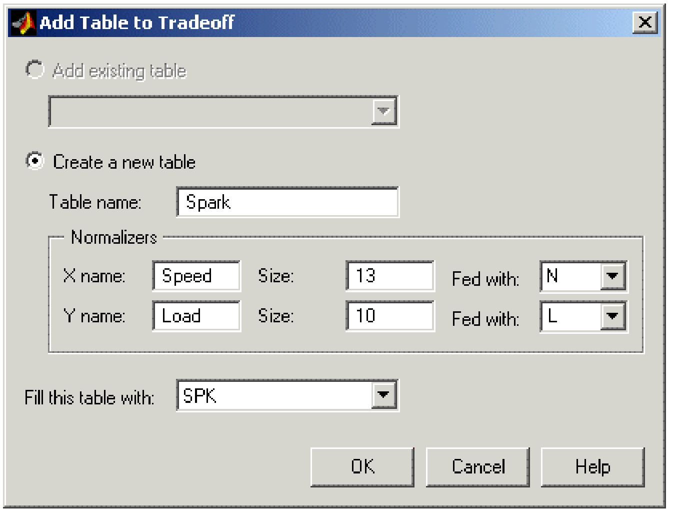
Both models have five inputs. The inputs for the torque and NOx models are

- 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 . This opens the **Add Table to Tradeoff** dialog.



2 Enter `Spark` as the `Table Name`.

3 Enter `Speed` as the **X Name** and `Load` as the **Y name**.

4 Enter `13` as the size of the speed axis.

5 Enter `10` as the size of the load axis.

6 Fill the `Speed` axis with `N` and the `Load` axis with `L`.

7 Fill the table with `SPK` (spark angle).

8 Click **OK**.

Before you can perform the calibration, you must display the models.

Displaying the Models

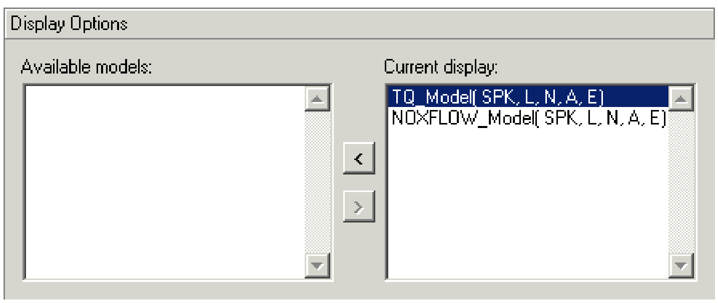
For this tutorial, you are comparing values of the torque and NOx models. Thus, you need to display these models.

To display both models:

1 Highlight `TQ_Model` and click > to include the model in the current display.

2 Repeat for `NOXFLOW_Model`.

The **Display Options** pane following shows both models selected for display.



You can now calibrate the tradeoff.

Performing the Tradeoff Calibration

You now fill the lookup table for spark angle by trading off gain in torque for reduction in NOx emissions.

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:

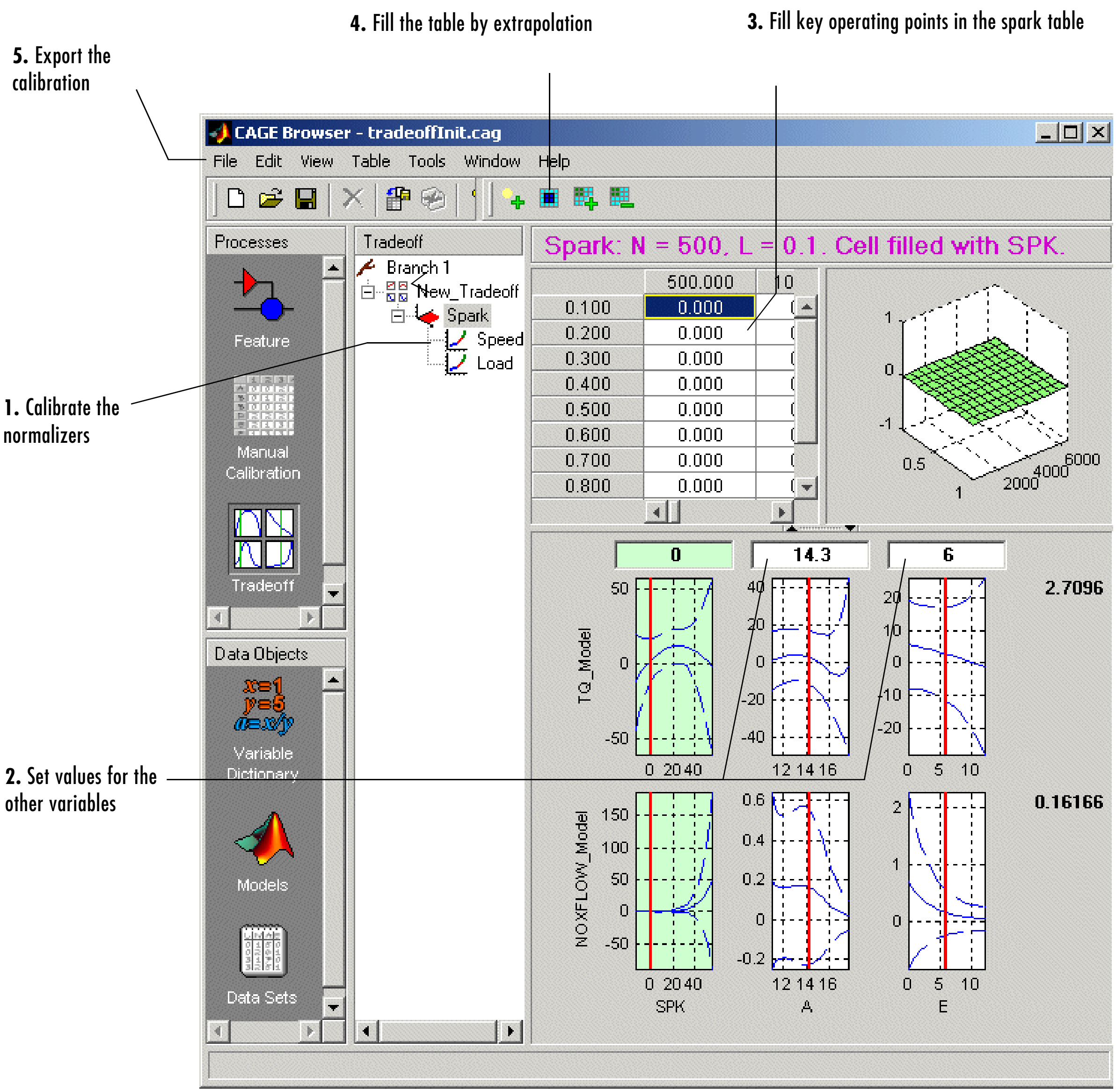
1 Calibrate the normalizers.

2 Set values for the other variables.

3 Fill key operating points with values for spark angle.

4 Fill the table by extrapolation.

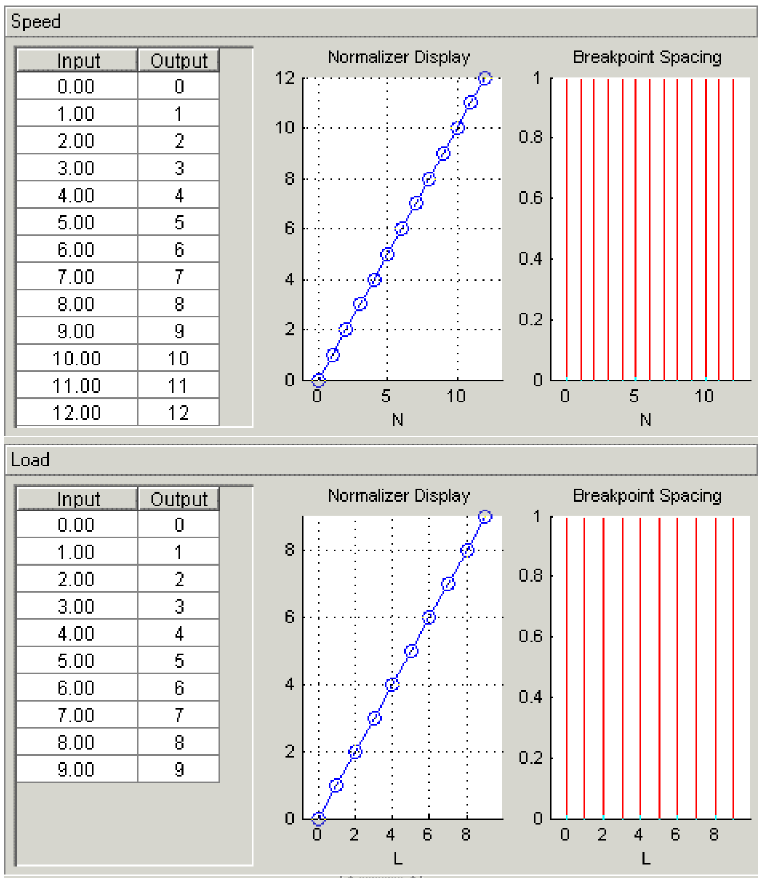
Once you have completed the calibration, you can export the calibration for use in the electronic control unit.


4. Fill the table by extrapolation
3. Fill key operating points in the spark table
5. Export the calibration
1. Calibrate the normalizers
2. Set values for the other variables
CAGE Browser - tradeoffInit.cag
File Edit View Table Tools Window Help
Processes
Feature
Manual Calibration
Tradeoff
Data Objects
Variable Dictionary
Models
Data Sets
Tradeoff
Branch 1
New_Tradeoff
Spark
Speed
Load
Spark: N = 500, L = 0.1. Cell filled with SPK.
TQ_Model
NOXFLOW_Model
SPK
A
E
2.7096
0.16166

Calibrating the Normalizers

A normalizer is the axis of the lookup table (which is the collection of breakpoints). This section describes how to space the breakpoints over the ranges of speed and load.

Expand the tree by clicking in the branch display so you can see the normalizers `Speed` and `Load`. Highlight either normalizer by clicking 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.

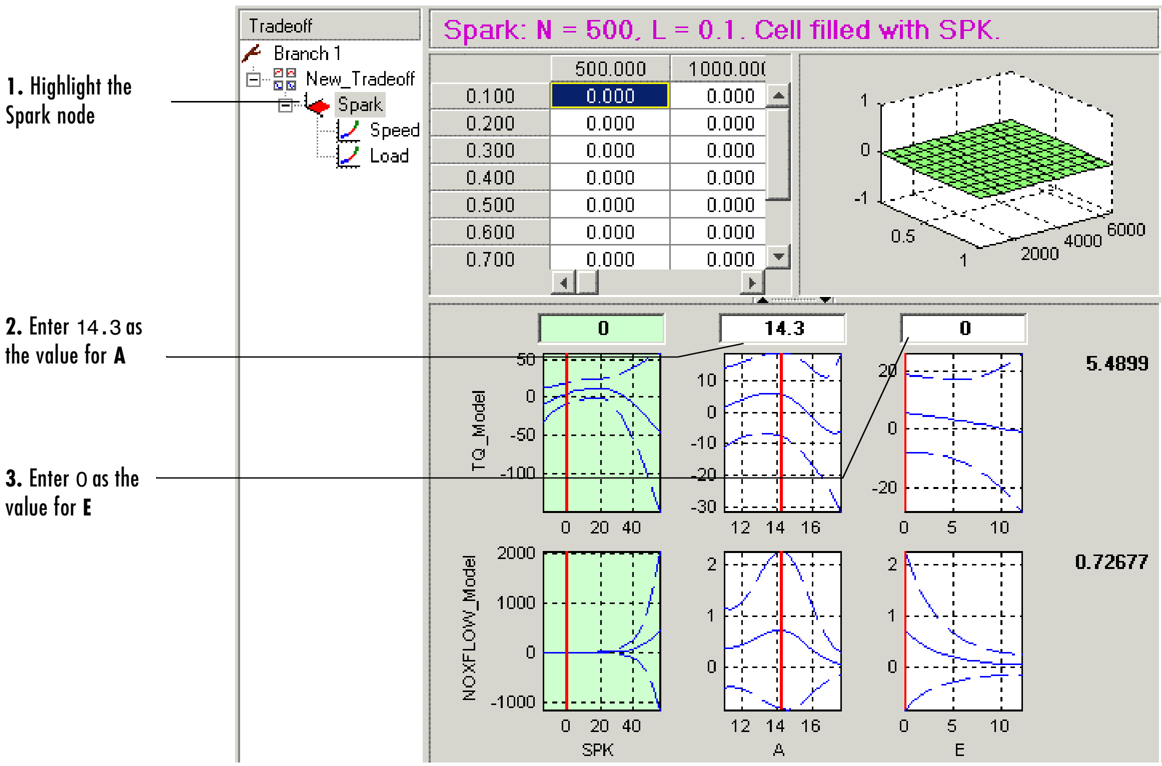
- Click  to space the breakpoints evenly over the ranges of speed and load.
- To accept the default ranges of the variables, click **OK** in the **Breakpoint Initialization Options** dialog box.

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). So you must set values for AFR and EGR for the models.

To set constant values of AFR and EGR for all operating points:

1. Highlight the Spark node in the branch display.
2. In the lower pane, check the value for **A** is 14.3, the stoichiometric constant.
3. Enter 0 as the value for **E**.



You can now fill the spark angle lookup table. The process is described next.

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.

You maximize the torque and restrict NOx emissions to below 250 g/hr.

To fill an operating point:

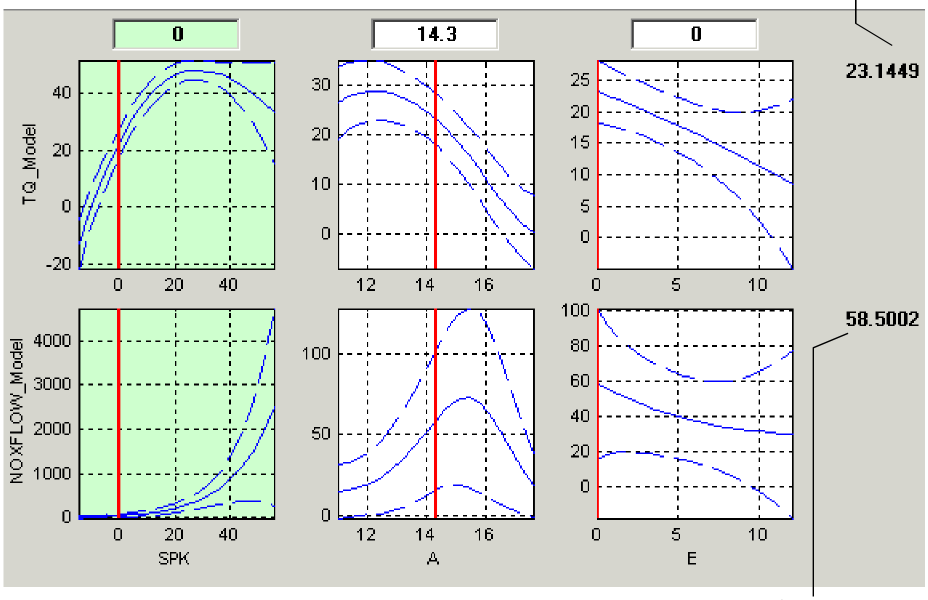
1. Select the operating point `N = 4500` and `L = 0.5` in the lookup table.
2. Find the spark angle that gives the maximum torque and restricts NOx emissions to below 250 g/hr. There are detailed instructions on how to do this tradeoff following.

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.

Graphs of the Models When N = 4500 and L = 0.5

Value of the torque model

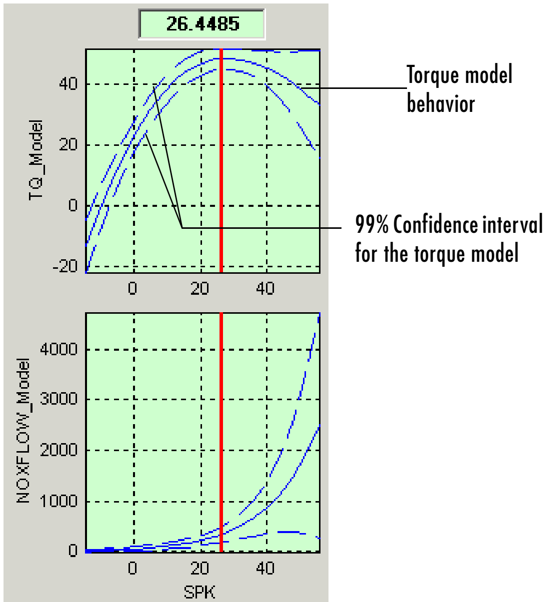


Value of the NOx model

The top three graphs show how the torque model varies with the spark angle, the AFR (labeled `A`), and the EGR (`E`), respectively. The lower panes show how the NOx emissions model varies with these variables, respectively.

Looking at the `Spark` table, the two spark (`SPK`) graphs are green, indicating that these graphs are directly linked to the lookup table.

You can change the value of spark by dragging the red line on the `SPK` graphs or typing values into the edit box. The following graph shows the behavior of the two models when the spark angle is `26.4458`.



3 Select **Find Global Maximum** from the right-click menu of the **SPK - TQ_Model** graph. This calculates the value of spark that gives the maximum value of torque.

At this operating point, the maximum torque that is generated is `48.136` when the spark angle is `26.4485`. However, the value of NOx is `347.7861`, which is greater than the restriction of 250 g/hr. Clearly you have to look at another value of spark angle.

4 Enter `21.5` as the value of **SPK** in the edit box at the top of the **SPK** column.

The value of the NOx emissions model is now `249.1542`. 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.

5 Press **Ctrl+T** or click (Add Point) 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 pink when selected, yellow if you click elsewhere.

6 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 (Add Point) 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	8.3
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

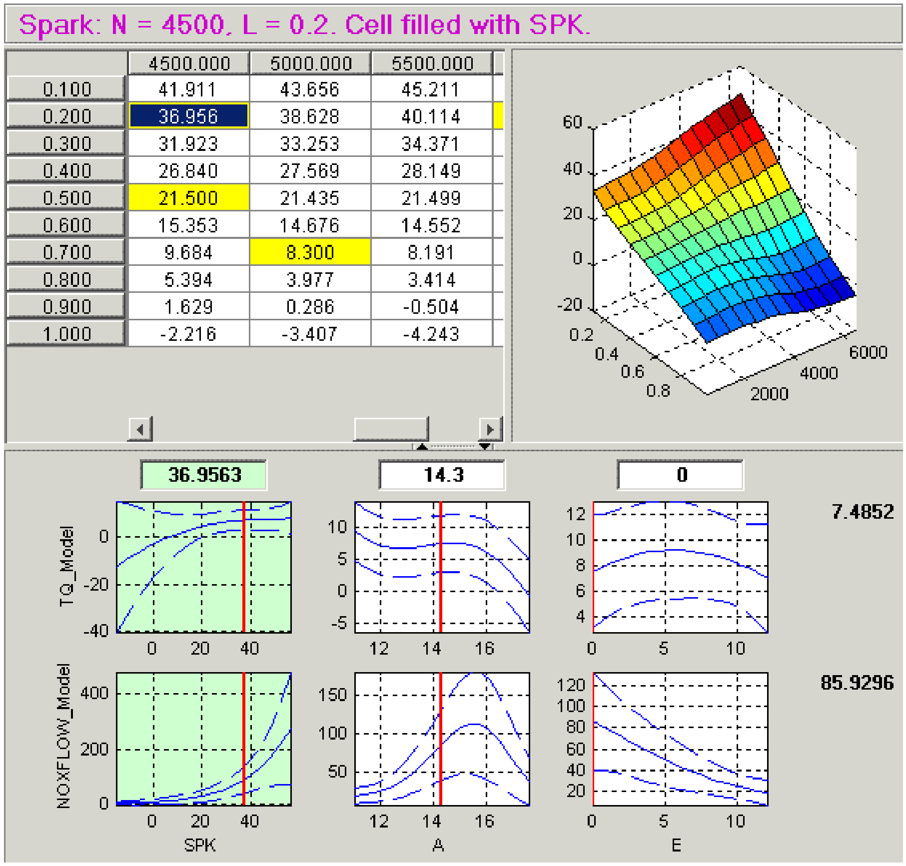
When you have calibrated several key operating points, you can produce a smooth extrapolation of these values across the whole table.

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 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.

The calibrator could now take these techniques to further improve the calibration. That is not the purpose of this tutorial.

For a more detailed description of tradeoff calibrations, see "Tradeoff Calibrations" on page 9-1.

You can now export this calibration for the electronic control unit.

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 type of file you want to save your calibrations as. You can choose from

- **Comma Separated Value (.csv)**
- **MAT-file (.mat)**
- **M-file script**

4 For the purposes of this tutorial, select **Comma Separated Value (.csv)**.

5 Enter `tradeoff.csv` as the file name and click **Save**.

This exports the spark angle table and the normalizers, `Speed` and `Load`, ready for the ECU.

You have now completed the tradeoff calibration tutorial.

Data Sets Tutorial

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:

- "Setting Up the Data Set" on page 4-3 describes the processes required to set up a data set for a comparison.
- "Comparing the Items in a Data Set" on page 4-8 describes how to view different aspects of the data set.
- "Reassigning Variables" on page 4-15 describes how to alter the data set.

For information about calibrating a feature, see "Feature Calibrations" on page 8-1.

You can also use data sets to fill lookup tables from experimental data. For information, see "Filling Tables from Data Tutorial" on page 5-1.

Setting Up the Data Set

Start CAGE by typing

```
cage
```

at the MATLAB prompt.

Note If you have a CAGE session open, select **File –> New –> Project**.

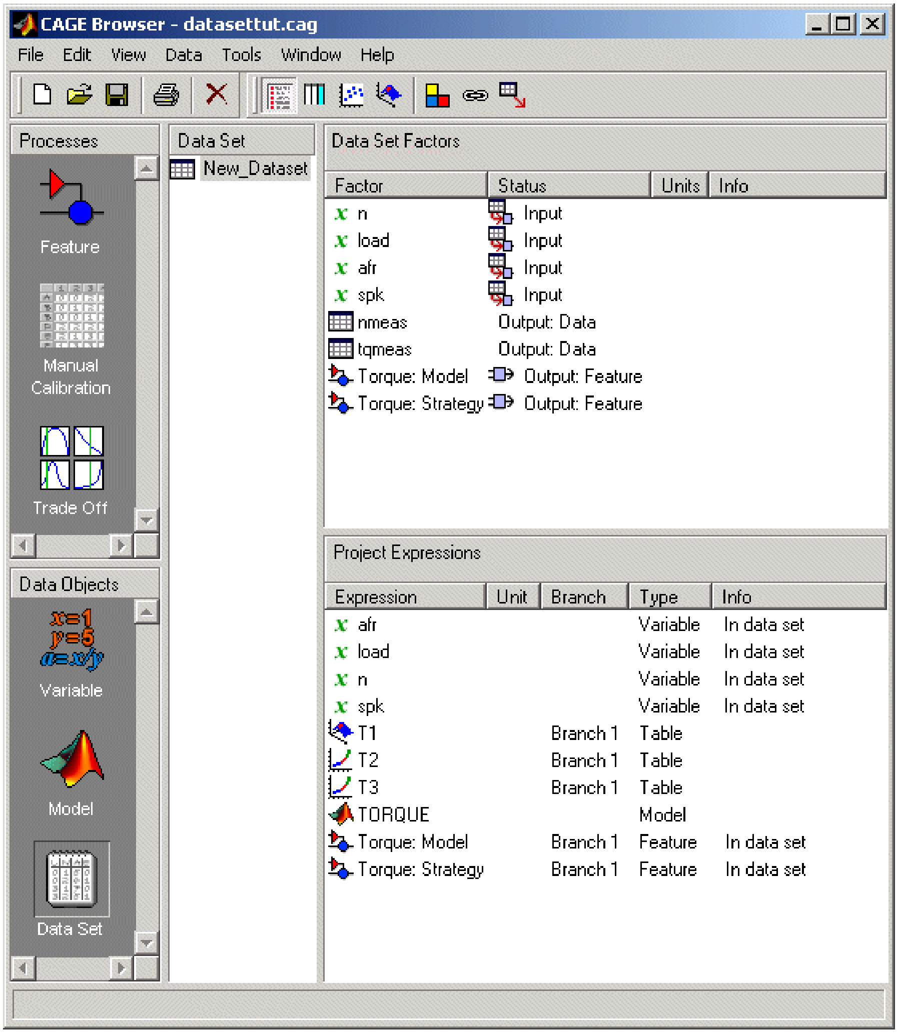
To set up the data set:

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.

When these steps are complete, the list of factors includes four input factors and four output factors, as shown.



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 Calibrations” on page 8-1.

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**.

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`).

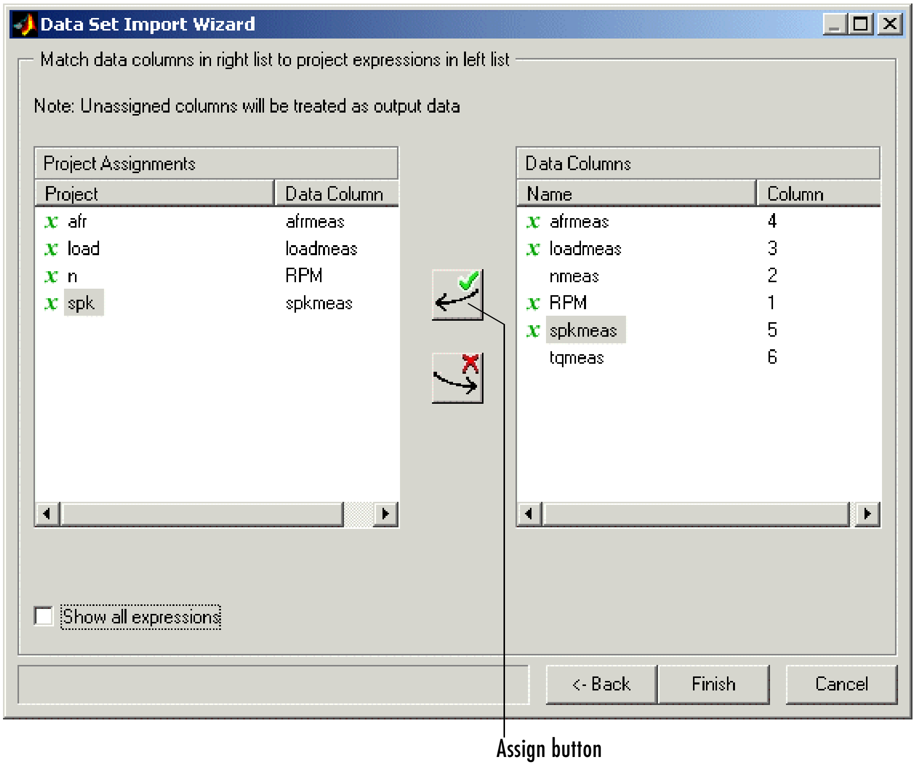
3 The **Data Set Import Wizard** asks which of the columns of data you would like to import. Click **Next** to import them all.

   The following screen asks you to associate variables in your project with data columns in the data.

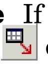
4 Highlight `afr` in the **Project Assignments** column and `afrmeas` in the **Data Column**, then click the assign button, shown.

5 Repeat this to associate `load` with `loadmeas`, `n` with `RPM`, and `spk` with `spkmeas`. The dialog box should be the same as shown.



6 Click **Finish** to close the dialog box.

Note If you need to reassign any inputs after closing this dialog you can click  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.

Comparing the Items in a Data Set

By viewing the data set, you can compare experimental data with calibrations or models in your project.

Viewing the Data Set as a Table

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	2235.000	0.549	9.500	0.100	2247.000	66.700	71.666	66.079
2	3591.000	0.454	13.200	0.100	3613.000	54.100	47.163	46.891
3	4946.000	0.651	12.000	0.100	4974.000	73.700	47.573	79.256
4	881.000	0.648	11.900	5.700	881.000	75.800	99.230	80.211
5	2234.000	0.441	13.300	0.100	2247.000	55.900	51.256	45.152
6	3591.000	0.747	10.900	0.100	3612.000	90.000	92.837	105.586
7	4947.000	0.541	9.700	0.100	4973.000	62.800	57.760	57.587
8	881.000	0.622	9.900	0.100	884.000	72.100	76.198	60.926
9	1219.000	0.333	14.000	0.100	1224.000	41.800	33.226	21.318
10	1558.000	0.382	12.000	0.100	1567.000	49.400	40.487	31.957
11	1896.000	0.209	10.700	3.300	1906.000	28.500	3.492	4.197
12	2234.000	0.284	9.800	3.200	2245.000	36.000	23.063	19.891
13	2574.000	0.407	13.400	3.000	2588.000	49.900	49.629	44.794
14	2914.000	0.595	11.500	3.100	2929.000	70.500	84.680	82.229
15	3251.000	0.781	12.300	3.100	3268.000	90.500	117.424	117.259
16	3589.000	0.668	13.500	3.000	3608.000	77.100	87.987	96.408
17	3930.000	0.452	11.900	3.100	3952.000	52.700	46.511	51.722
18	4268.000	0.235	10.900	3.000	4293.000	27.700	5.253	3.085
19	4606.000	0.194	12.000	3.200	4633.000	21.300	-2.088	-5.771
20	4945.000	0.550	10.100	3.100	4972.000	61.600	55.754	65.743
21	5284.000	0.727	12.600	5.000	5310.000	79.900	78.438	102.468
22	5621.000	0.763	11.300	3.000	5649.000	83.900	75.659	102.957
23	5961.000	0.500	13.200	3.000	5995.000	51.700	35.034	51.304
24	882.000	0.557	12.100	4.400	887.000	64.100	82.455	62.160

In the table, the input cells are white and the output cells are blue.

In addition to viewing the difference between 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`.

	n	I...	afr	spk	nmeas	tqmeas	Torque: Model	Torque: Strategy	tqmeas_minus_Torque
1	2...	...	9....	0.1...	2247.000	66.700	71.666	66.079	-0.621
2	3...	...	1...	0.1...	3613.000	54.100	47.163	46.891	-7.209
3	4...	...	1...	0.1...	4974.000	73.700	47.573	79.256	5.556
4	3...	...	1...	5.7...	881.000	75.800	99.230	80.211	4.411
5	2...	...	1...	0.1...	2247.000	55.900	51.256	45.152	-10.748
6	3...	...	1...	0.1...	3612.000	90.000	92.837	105.586	15.586
7	4...	...	9....	0.1...	4973.000	62.800	57.760	57.587	-5.213
8	3...	...	9....	0.1...	884.000	72.100	76.198	60.926	-11.174
9	1...	...	1...	0.1...	1224.000	41.800	33.226	21.318	-20.482
10	1...	...	1...	0.1...	1567.000	49.400	40.487	31.957	-17.443
11	1...	...	1...	3.3...	1906.000	28.500	3.492	4.197	-24.303
12	2...	...	9....	3.2...	2245.000	36.000	23.063	19.891	-16.109
13	2...	...	1...	3.0...	2588.000	49.900	49.629	44.794	-5.106
14	2...	...	1...	3.1...	2929.000	70.500	84.680	82.229	11.729
15	3...	...	1...	3.1...	3268.000	90.500	117.424	117.259	26.759
16	3...	...	1...	3.0...	3608.000	77.100	87.987	96.408	19.308
17	3...	...	1...	3.1...	3952.000	52.700	46.511	51.722	-0.978
18	4...	...	1...	3.0...	4293.000	27.700	5.253	3.085	-24.615
19	4...	...	1...	3.2...	4633.000	21.300	-2.088	-5.771	-27.071
20	4...	...	1...	3.1...	4972.000	61.600	55.754	65.743	4.143
21	5...	...	1...	5.0...	5310.000	79.900	78.438	102.468	22.568
22	5...	...	1...	3.0...	5649.000	83.900	75.659	102.957	19.057
23	5...	...	1...	3.0...	5995.000	51.700	35.034	51.304	-0.396

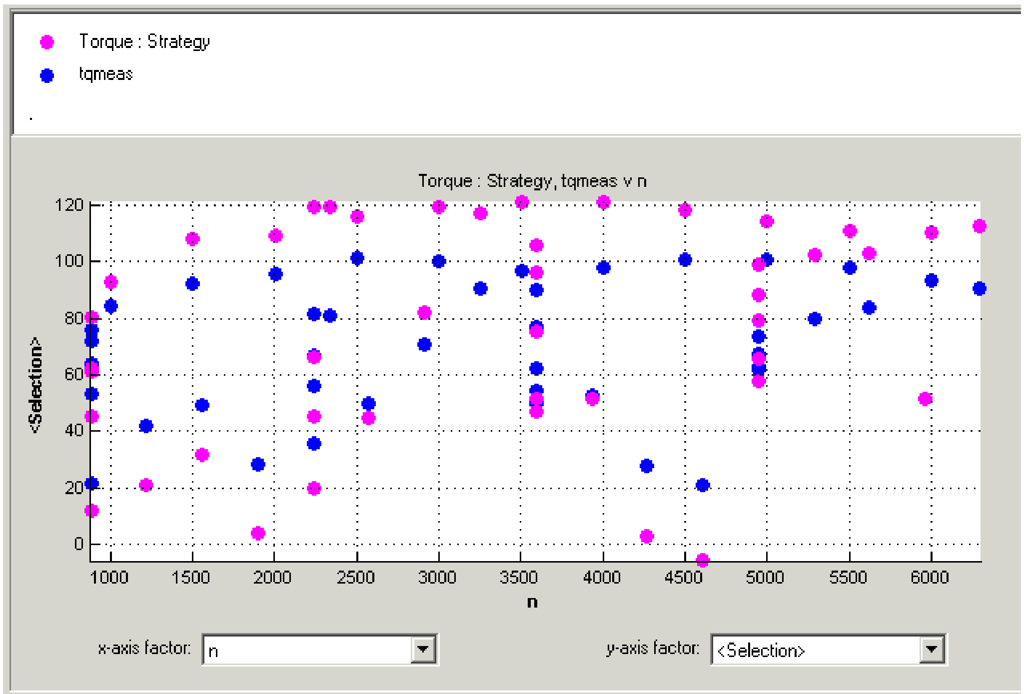
The error column is simply the difference between `tqmeas` and `Torque: Strategy`. This provides a simple way of comparing the feature and the measured data.

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.



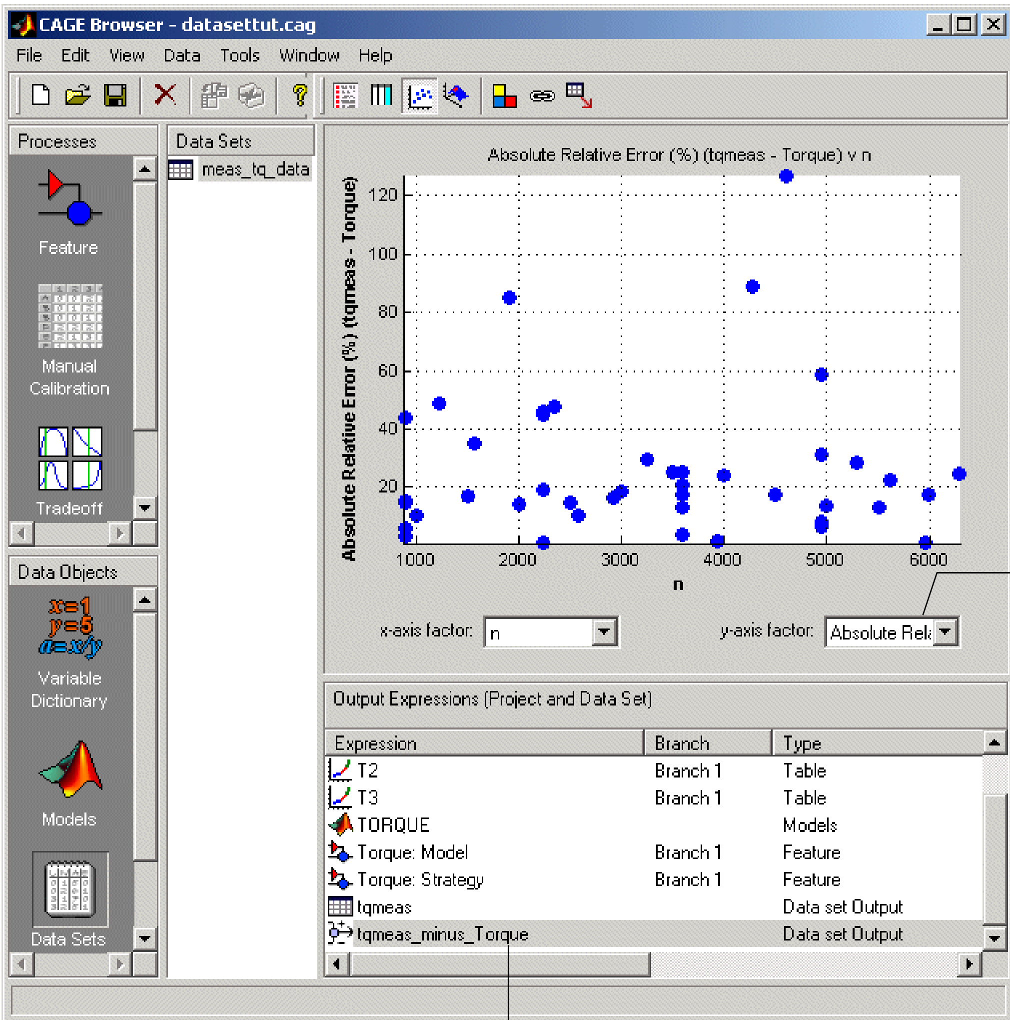
3 Change the ***x*-axis factor** to `n` from the drop-down menu.

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` from the lower list (**Output Expressions**).

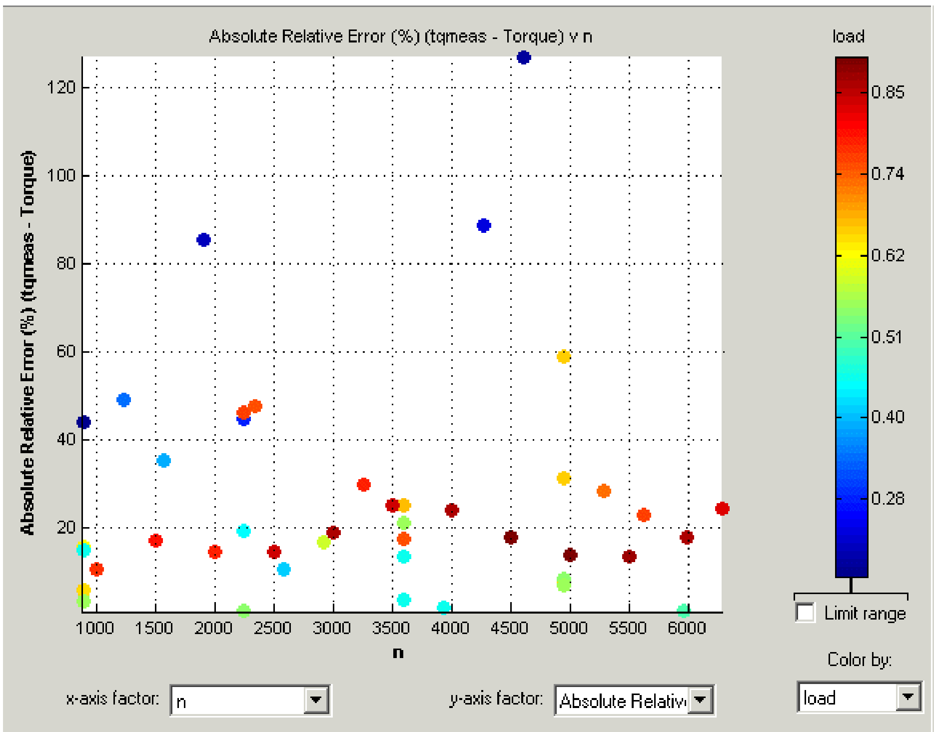
2 For the ***y*-axis factor**, select **Absolute Relative Error (tqmeas - Torque)** from the drop-down menu.

As you can see, there seems to be no particular correlation between engine speed and the error in the calibration.

Coloring the Display

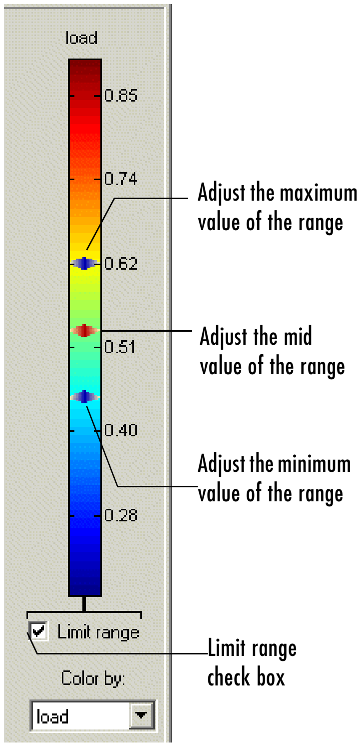
1 Select **Color by Value** from the right-click menu on the graph.

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



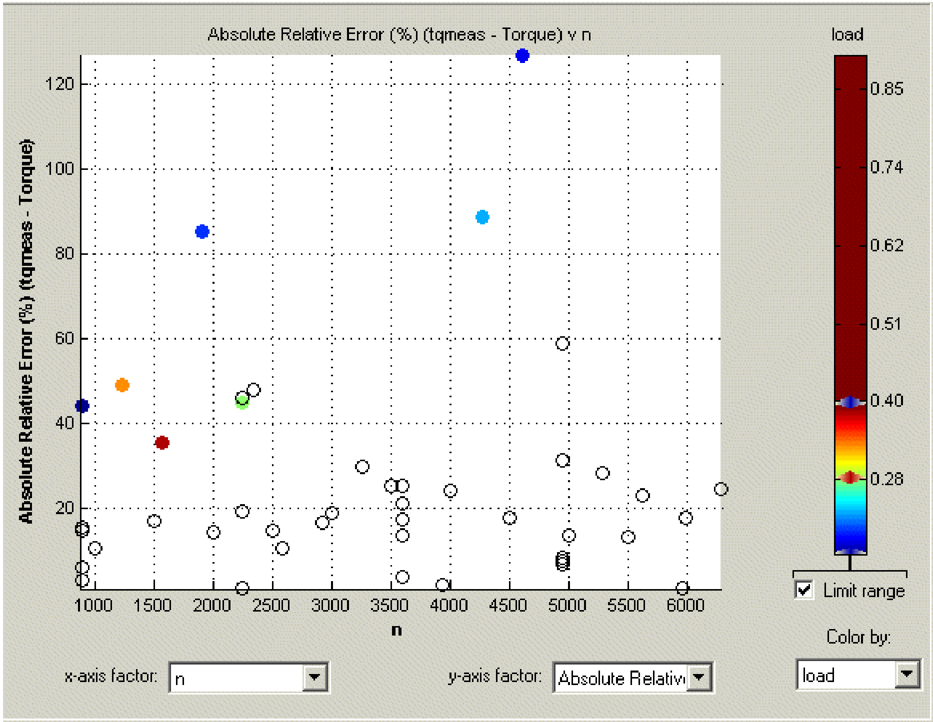
To view the colors in more detail, you can limit the range of the colors:

1 Select the **Limit range** box.

2 Right-click on the graph and select **Restrict Color to Limits**.

3 Set the minimum value of the color range to be as low as possible by dragging the minimum value down.

4 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

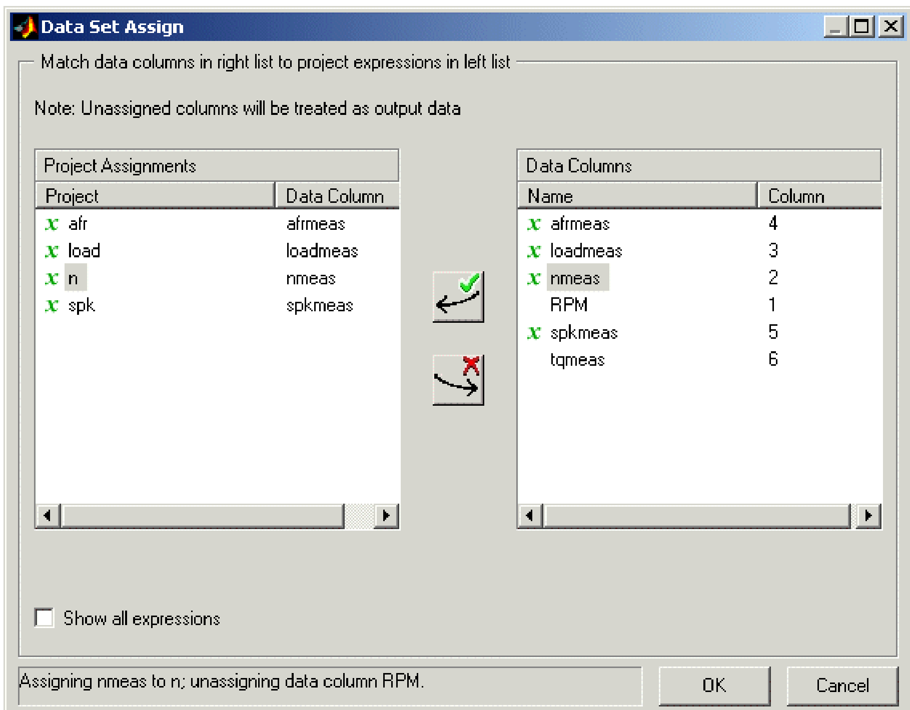
Instead of using the test cell settings for the engine speed (`RPM`), you might want to use the measured values of engine speed (`nmeas`). So you have to reassign the variable `n` to `nmeas`.

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 complete functionality of data sets, see "Data Sets" on page 10-1.

You have now completed the data sets tutorial.

5

Filling Tables from Data Tutorial

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.

This tutorial takes you through the steps of calibrating a lookup table for torque, based on experimental data.

- "Setting Up a Table and Experimental Data" on page 5-3 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 5-10 describes the process of filling the lookup table.
- "Selecting Regions of the Data" on page 5-14 describes how you can select some of the data for inclusion when you fill the table.
- "Exporting the Calibration" on page 5-16 describes how to export your completed calibration.

Setting Up a Table and Experimental Data

1 Start CAGE by typing

```
cage
```

at the MATLAB prompt.

2 Select **File –> New Project**.

This section describes how to set up a blank table ready for filling using experimental data.

The steps that you need to follow to set up the CAGE session are

1 Add the variables for speed and load.

2 Add a new table to your session.

3 Set up your normalizers.

4 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:

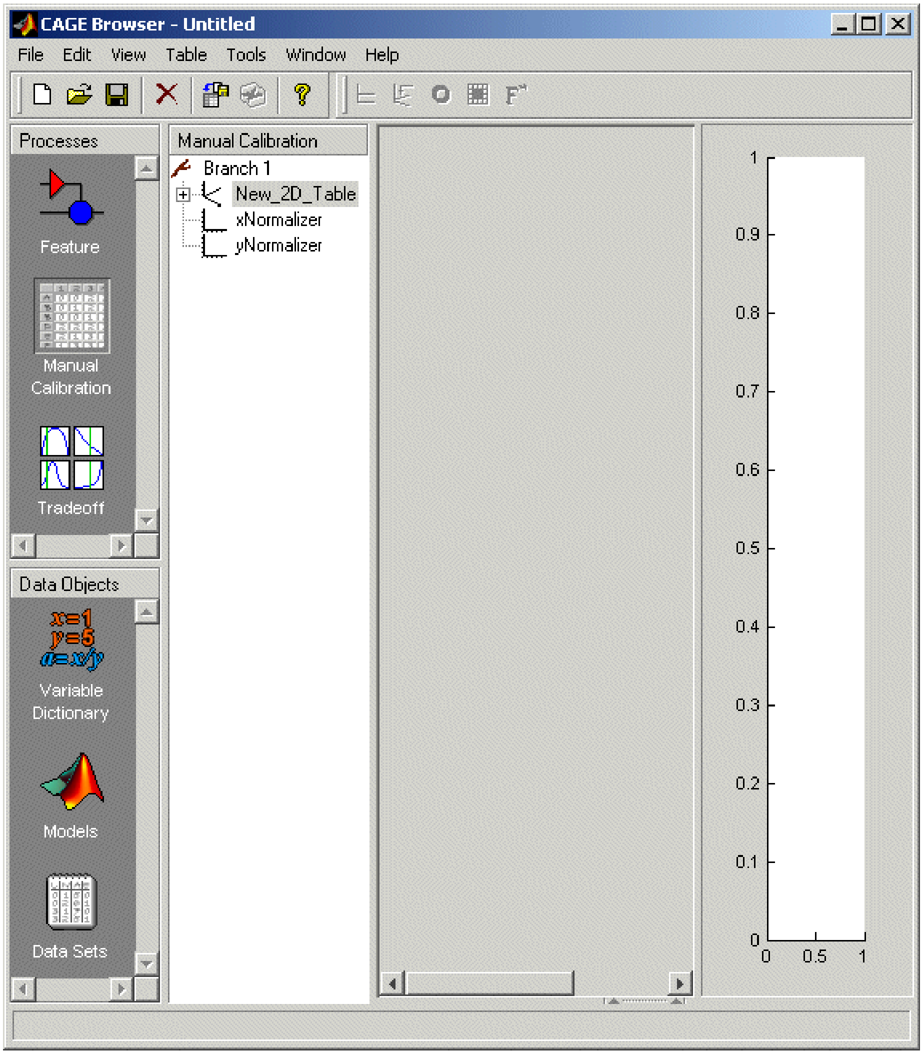
1 Select **File –> New –> 2D table**.

   This opens a dialog box that asks you to specify the variable names for the normalizers.

2 To accept `N` as the variable for normalizer **X** and `L` as the variable for normalizer **Y**, click **OK**.

Note In CAGE, a 2-D table is defined as a table with two inputs.

CAGE takes you to the **Manual Calibration** view, where you can see the following.



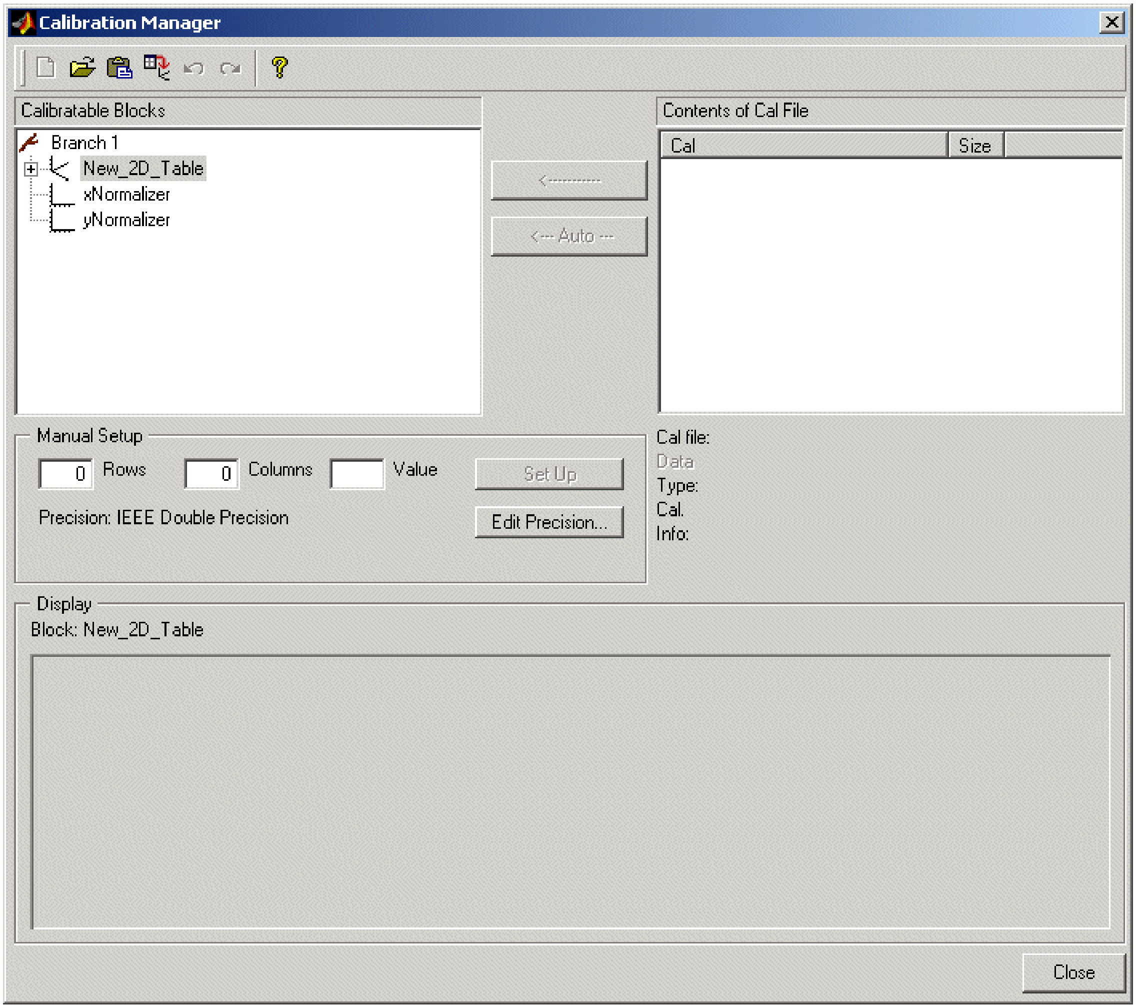
You must now set up and initialize the normalizers with suitable values for the engine speed and load.

Setting Up the Normalizers

Currently, the lookup table has neither rows nor columns, so you must set up the table.

To set up the table:

1 Open the **Calibration Manager** dialog box, shown, by clicking .



2 Highlight the `New_2D_Table` in the **Calibratable Blocks** display.

3 To determine the size of each normalizer, enter `10` as the number of rows and `7` as the number of columns.

4 Set the value in each cell to be `0`.

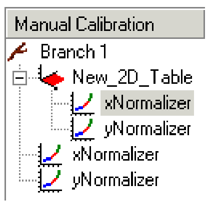
5 Click **Set Up** to confirm your selection.

6 Click **Close** to close the **Calibration Manager** and return to the table view.

Setting the Values of the Normalizers

You can now set values of the normalizers so that they cover the range of the variables `N` and `L`:

1 Expand the table branch by clicking ⊞, and select `xNormalizer` as shown.



This displays the two normalizers for the table.

2 To space the breakpoints evenly over the range of the variables `N` and `L`, click .

This opens a dialog box that suggests the range of values for each normalizer.

3 To accept the suggested ranges of `N` and `L`, click **OK**. These suggested ranges are determined by the variable dictionary.

You now 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.

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.

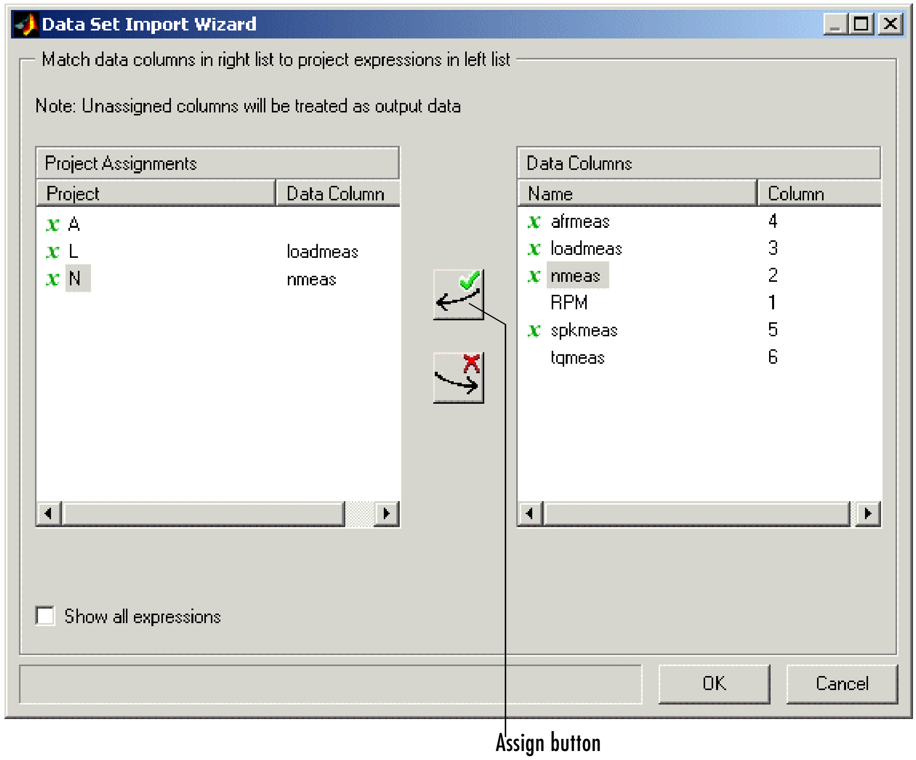
   This set of data includes six columns of data: the test cell settings for engine speed (`RPM`), and the measured values of torque (`tqmeas`), engine speed (`nmeas`), air-fuel ratio (`afrmeas`), spark angle (`spkmeas`), and load (`loadmeas`).

3 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.



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 revs 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:

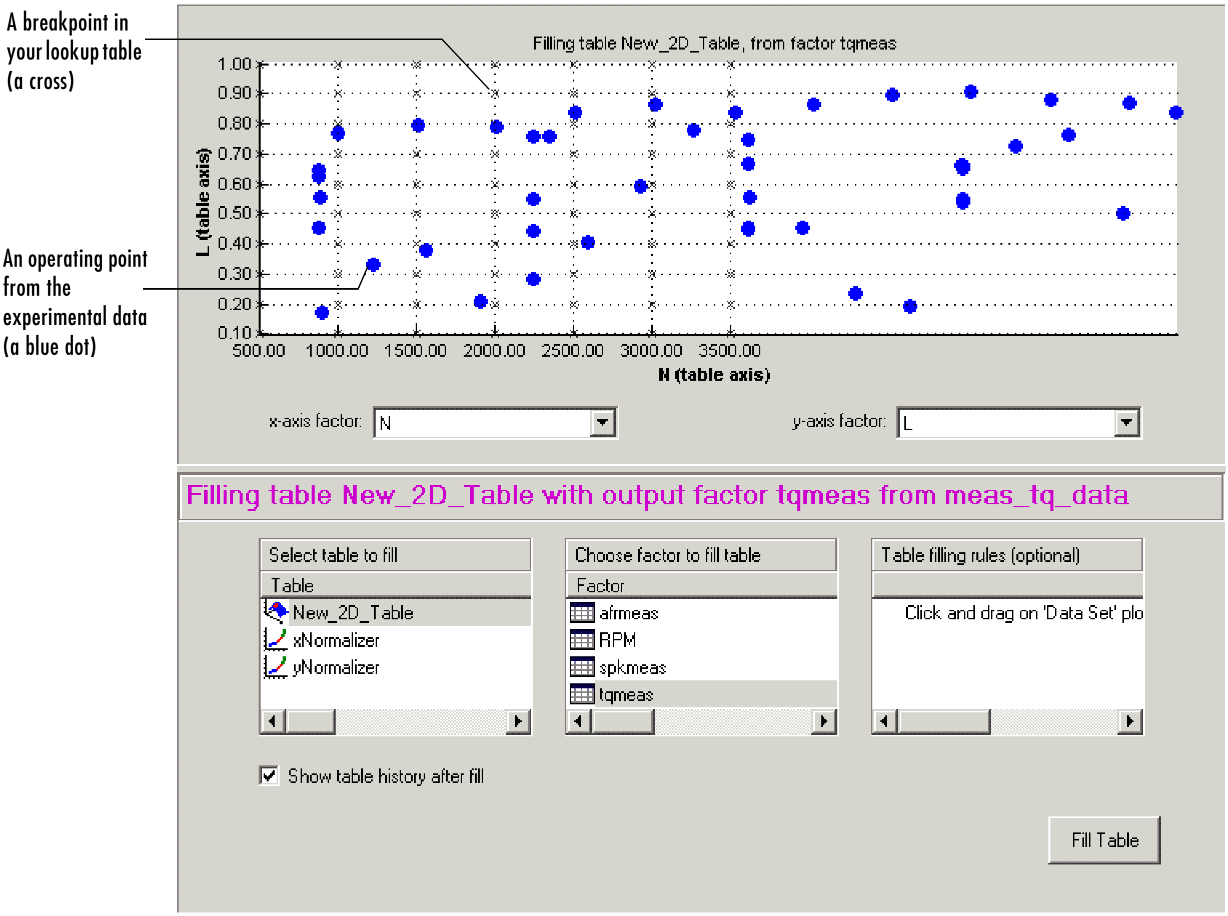
1 To view the **Table Filler** display, click  in the toolbar in the **Data Sets** view.

  This display asks you 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** list. This is the table that you want to fill.

3 Select `tqmeas` from the **Factor** 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.



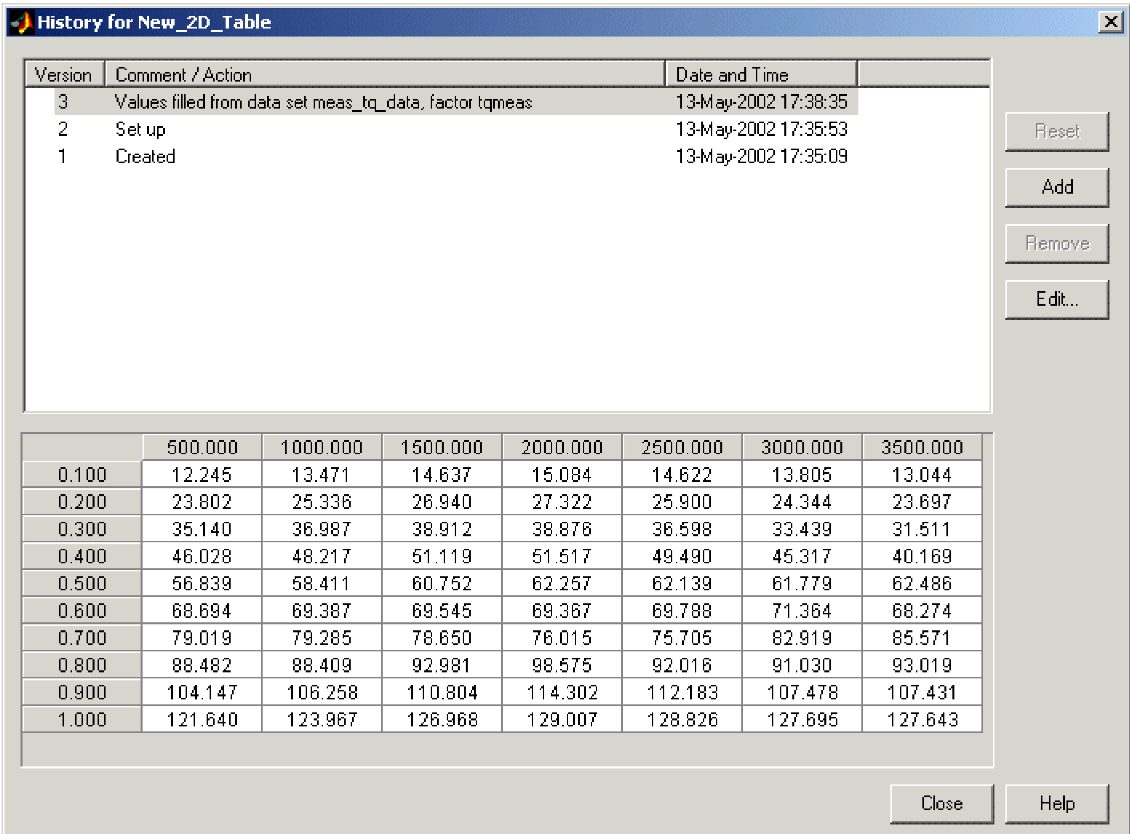
The upper pane displays the breakpoints of your table as crosses and the operating points where there is data as blue dots.

5 To view the table after it is filled, ensure that the **Show table history after fill** box, at the bottom left, is selected.

Data sets display the points in the experimental data, not the values at the breakpoints.

6 To fill the table with values of `tqmeas` extrapolated over the range of the normalizers, click **Fill Table**.

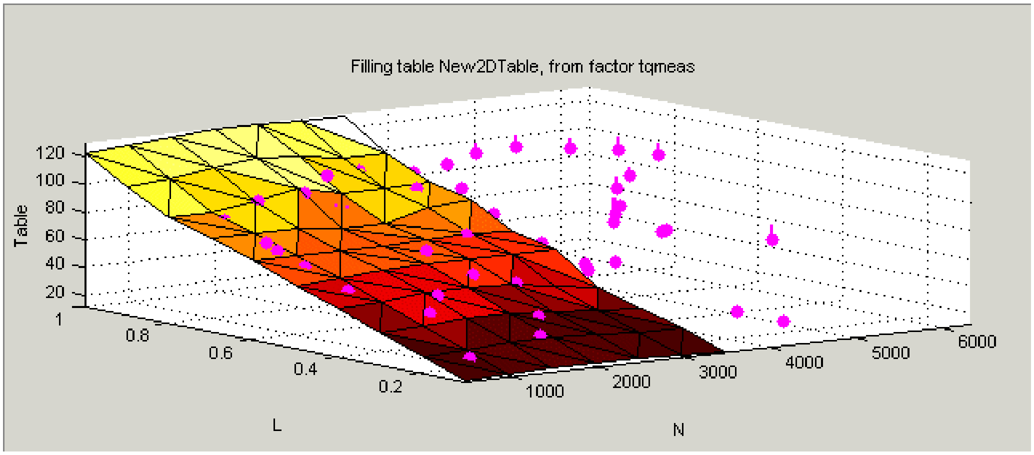
This opens the **History** dialog box, shown.



	500.000	1000.000	1500.000	2000.000	2500.000	3000.000	3500.000
0.100	12.245	13.471	14.637	15.084	14.622	13.805	13.044
0.200	23.802	25.336	26.940	27.322	25.900	24.344	23.697
0.300	35.140	36.987	38.912	38.876	36.598	33.439	31.511
0.400	46.028	48.217	51.119	51.517	49.490	45.317	40.169
0.500	56.839	58.411	60.752	62.257	62.139	61.779	62.486
0.600	68.694	69.387	69.545	69.367	69.788	71.364	68.274
0.700	79.019	79.285	78.650	76.015	75.705	82.919	85.571
0.800	88.482	88.409	92.981	98.575	92.016	91.030	93.019
0.900	104.147	106.258	110.804	114.302	112.183	107.478	107.431
1.000	121.640	123.967	126.968	129.007	128.826	127.695	127.643

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

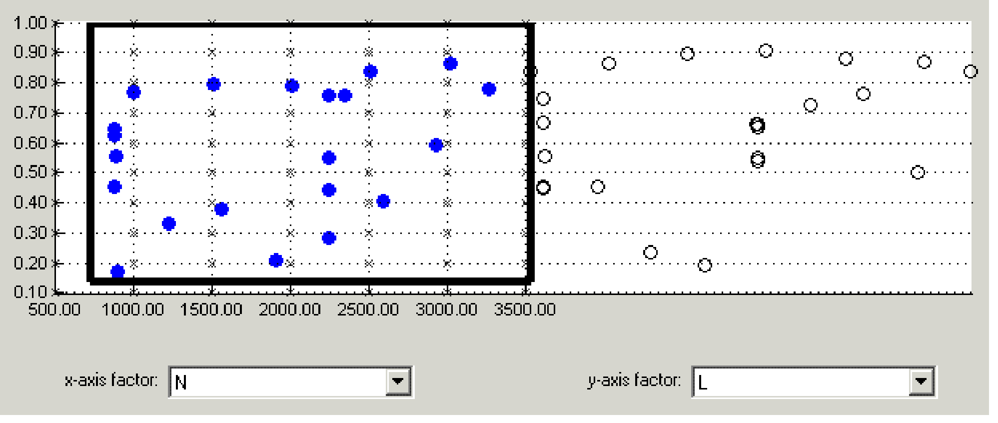
You can ignore points in the data set when you fill your lookup table.

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:

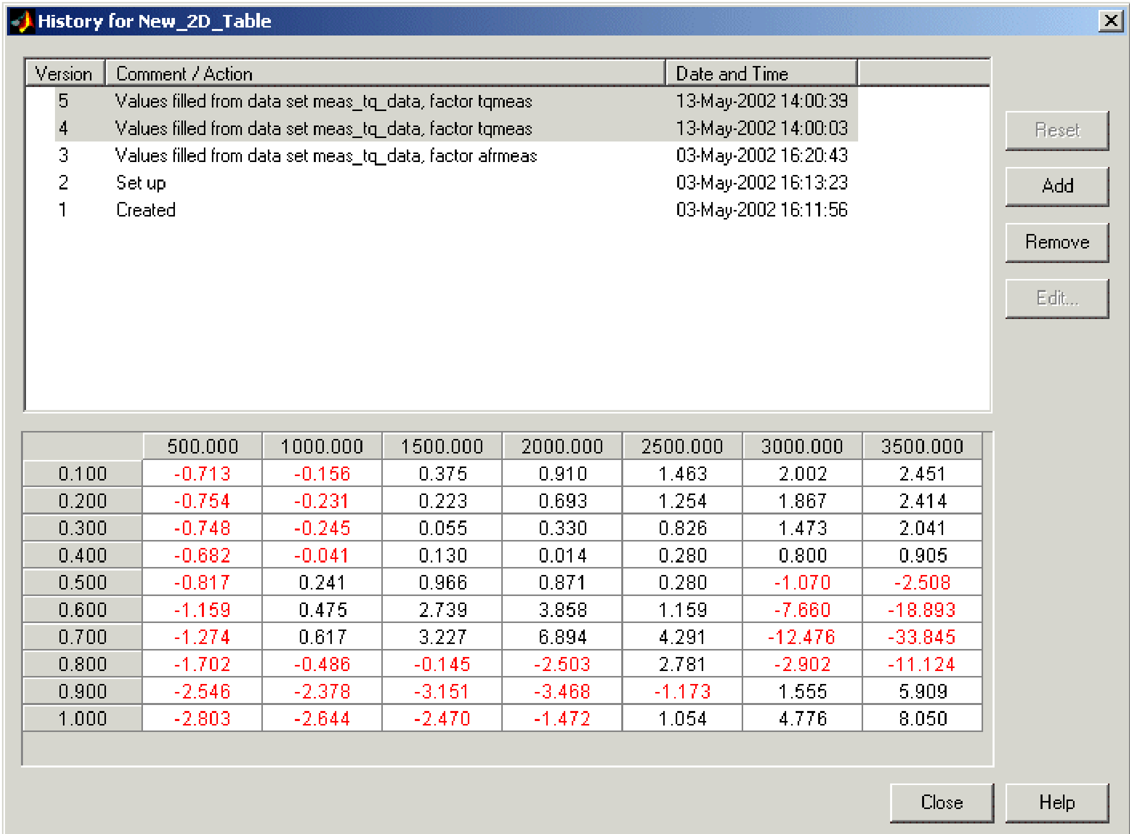
1 Select **Data –> Plot –> Data Set**. This returns you to the view of where the breakpoints lie in relation to the experimental data.

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 3 and 4, 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.

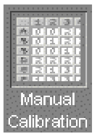
You have filled a lookup table with values taken from experimental data.

Exporting the Calibration

You can export the calibration for use in an electronic control unit (ECU).

To export the calibration:

1 To highlight the table that you want to export, you must first click **Manual Calibration**, shown.



2 Highlight the `New_2D_Table`.

3 Select **File –> Export –> Calibration**.

4 Choose the type of file you want to save your calibrations as. You can choose from

 a **Comma Separated Value (.csv)**

 b **MAT-file (.mat)**

 c **M-file script**

5 For the purposes of this tutorial, select **Comma Separated Value (.csv)**.

6 Enter `table_filling_tutorial.csv` as the file name and click **Save**.

This exports the successful calibration, ready for the ECU.

You have now completed this tutorial.

6

Using CAGE

The following reference sections describe how to use CAGE to perform calibrations:

- “How to Use CAGE” on page 6-3 is an overview of CAGE processes: where to find the functionality for different processes and how to set up variables and models before performing calibrations.
- “Normalizers” on page 7-1 describes what normalizers are, and how to space breakpoints in a normalizer.
- “Feature Calibrations” on page 8-1 describes how to calibrate lookup tables by reference to models built using the model browser.
- “Tradeoff Calibrations” on page 9-1 describes how to calibrate lookup tables by adjusting one value to fulfill different objectives.
- “Data Sets” on page 10-1 describes how to use CAGE to compare calibrations to experimental data, and how to use experimental data to fill lookup tables.
- “Calibration Manager” on page 11-1 describes how to use the **Calibration Manager**.
- “Surface Viewer” on page 12-1 describes how to use the **Surface Viewer**.
- “Manual Calibration and the History Display” on page 13-1 describes how to add and delete tables and manually calibrate tables; and how to use the **History** viewer.

How to Use CAGE

Before you can perform a calibration using CAGE, you need to set up the variables, constants, and the models you want to use. If you import a model, it has variables associated with it, in which case you might not have to import a variable dictionary.

The following sections describe how to set up variables and models before performing calibrations:

- "Setting Up Your Variable Items" on page 6-7
- "Setting Up Your Models" on page 6-14

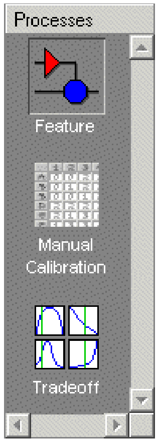
You can also use CAGE to calibrate tables directly from experimental data by interpolation, without using models. See "Filling Tables from Data Tutorial" on page 5-1 for an example.

The view and functionality available in CAGE depend on two things:

- Which of the six large buttons you select in the **Processes** and **Data Objects** panes
- The item you highlight in the branch display (tree)

See the next section, "CAGE Views and Processes" on page 6-4, for a summary of the functionality you can reach in each view and links to in-depth help for each process.

CAGE Views and Processes



The **Processes** pane has three buttons:

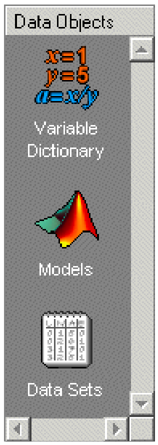
- **Feature** shows the **Feature** view, with the tables and strategies that are associated with that feature. See “Feature View” on page 8-45.

  A feature is a strategy (or collection of tables) and a model used to calibrate those tables. In the **Feature** view, you can fill tables by comparing a strategy to a model. See “Feature Calibrations” on page 8-1. You can import existing strategies or construct new ones using Simulink from the feature view.

  From the feature node in the branch display, you can access the **Surface Viewer** to examine the strategy or model or both. See “Surface Viewer” on page 12-1.
- **Manual Calibration** enables you to calibrate tables manually. It also acts as a store for all the tables and normalizers in your session. Here you can add and delete tables from the project. From any table display (here, or in other views) you can access the **History Display** to manage changes in your tables and normalizers. You can use the **History Display** to reverse changes. See “Using the History Display” on page 13-5.
- **Tradeoff** shows the **Tradeoff** view, with a list of the tables and models to display. Here you can see graphically the effects of manually altering variables to trade off different objectives (such as maximizing torque while minimizing emissions). At the tradeoff node, you can calibrate table values to achieve the best compromise between competing objectives. You can

calibrate using single or multimodel tradeoffs. See “Tradeoff Calibrations” on page 9-1.

You can reach the **Calibration Manager** from any of the three **Process** views. Here you can set up the size of tables (manually or using existing calibration files) and edit the precision used for values (to match the kind of electronic control unit you are going to use). See “Calibration Manager” on page 11-1.



The **Data Objects** pane has three buttons:

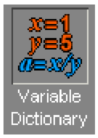
- **Variable Dictionary** stores all the variables, constants, and formulas in your session. Here you can view, add, and edit any variables in any part of your session. See “Setting Up Your Variable Items” on page 6-7.
- **Models** stores all the models in your session. Here you can view a graphical display of these models, including a diagram of the model’s input structure. This is useful because a model can have other models as inputs. You can change the inputs here. For example, you can change your model’s input Spark to be connected to a model for Spark rather than to the variable `Spark`. You can also access the surface viewer here to examine models. See “Setting Up Your Models” on page 6-14 and “Surface Viewer” on page 12-1.
- **Data Sets** enables you to evaluate your models and features over a custom set of input values. Here you can create and edit a set of input values and view several models or features evaluated at these points. You can compare your tables and models with experimental data to validate your calibrations.

You can also fill tables directly from experimental data by loading the experimental data as a new data set. See "Data Sets" on page 10-1.

Setting Up Your Variable Items

The **Variable Dictionary** is a store for all the variables, constants, and formulas in your session.

To view or edit the items in the **Variable Dictionary,** click the button, shown, in the **Data Objects** pane.



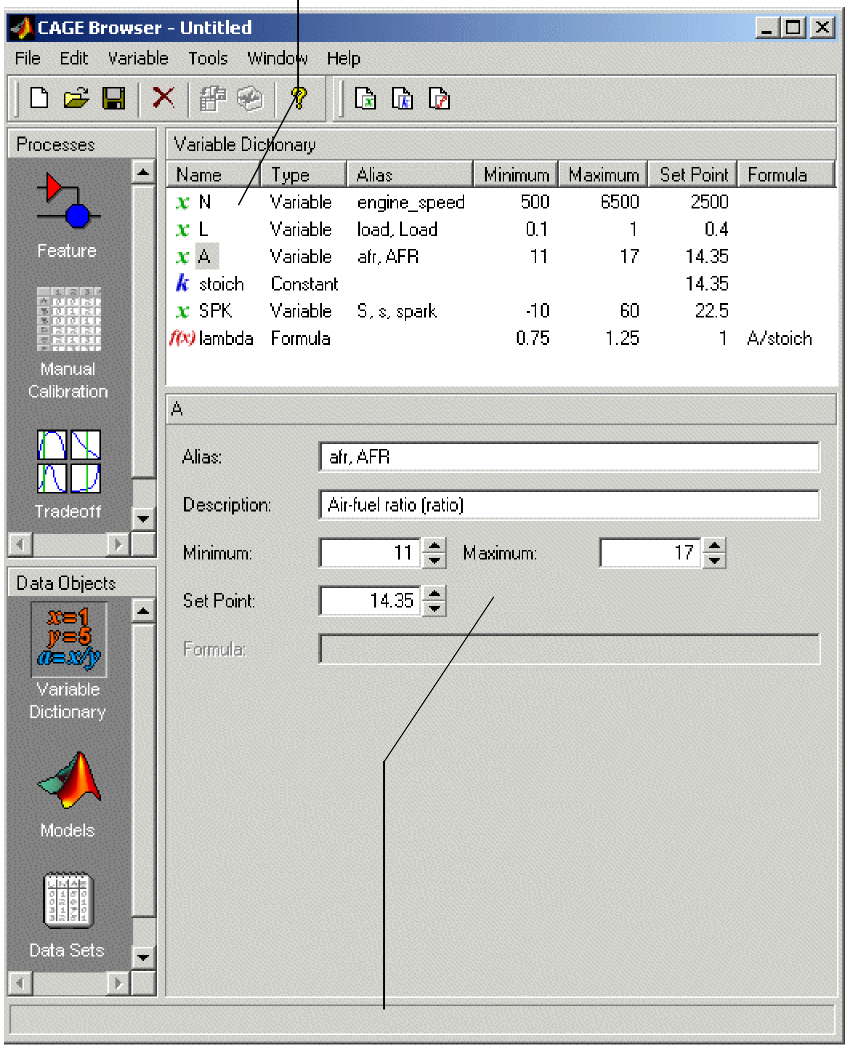
Selecting the **Variable Dictionary** view displays the variables, constants, and formulas in the current project.

This section describes the following:

- “Importing and Exporting a Variable Dictionary” on page 6-9
- “Adding Variable Items” on page 6-10
- “Using the Variable Menu” on page 6-12
- “Using Aliases” on page 6-13

Following is an example of the **Variable Dictionary** view.

List of all the constants, variables, and formulas in the project



Edit boxes to change the settings of the selected constant, variable, or formula

The upper pane shows a list of all the current variables, constants, and formulas. The lower pane displays edit boxes so you can specify the settings of the selected variable, constant, or formula.

Importing and Exporting a Variable Dictionary

A variable dictionary contains all the variable items for your calibrations. You can set up your variable dictionary once, and use it in many calibrations.

Importing a Variable Dictionary

To import a dictionary of variables from an `.xml` file:

1 Select **File –> Import –> Variable Dictionary**.

2 Select the correct dictionary.

Exporting a Variable Dictionary

After setting up a variable dictionary, you can save the dictionary for use in many different calibrations.

To export a dictionary of variables to an `.xml` file:

1 Select **File –> Export –> Variable Dictionary**.

2 Select a suitable name for the dictionary.

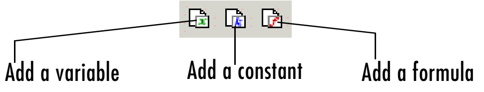
See Also

• “Setting Up Your Variable Items” on page 6-7
• “Adding Variable Items” on page 6-10

Adding Variable Items

To add variable items you can use the **Variable Dictionary** toolbar, shown, or you can select items from the **File –> New –> Variable Items** menu.

Variable Dictionary Toolbar



Adding a Variable

To add a variable:

1 Select **File –> New –> Variable Item –> Variable**.

   A new variable is added to the variable dictionary.

2 Select **Edit –> Rename** to alter the name of the variable.

3 Specify the **Minimum** and **Maximum** values of the variable in the edit boxes in the lower pane.

4 Specify the value of the **Set Point** in the edit box.

Using Set Points in the Variable Dictionary. The set point of a variable is a point that is of particular interest in the range of the variable.

For example, for the air-fuel ratio variable, AFR, the range of values is typically 11 to 17. However, whenever only one value of AFR is required, it is preferable to choose 14.3, the stoichiometric constant, over any other value. So enter `14.3` as the **Set Point**.

CAGE uses the set point as the default value of the variable wherever one value from the variable range is required. For instance, CAGE uses the set point when evaluating a model over the range of a different variable.

For example, a simple model for torque depends on AFR, engine speed, and relative air charge. CAGE uses the set point of AFR when it calculates the values of the model over the ranges of the engine speed and relative air charge.

Adding a Constant

To add a constant:

1 Select **File –> New –> Variable Item –> Constant**.

   A new constant is added to the variable dictionary.

2 Select **Edit –> Rename** to alter the name of the constant.

3 Specify the value of the constant in the **Set Point** edit box, in the lower pane.

Adding Formulas

You might want to add a formula to your session. For example, the formula

$$\lambda = \frac{\mathtt{afr}}{\mathtt{stoich}}$$

where `afr` is the air-fuel ratio and `stoich` is the stoichiometric constant.

To add a formula:

1 Select **File –> New –>Variable Item –> Formula**.

   The **Add Formula** dialog box appears.

2 In the dialog, enter the right side of the formula, `afr/stoich`, and click **OK**.

   A new formula is added to the variable dictionary.

3 Select **Edit –> Rename** to alter the name of the formula.

Note Formulas can only have one variable.

See Also

- “Setting Up Your Variable Items” on page 6-7
- “Adding Variable Items” on page 6-10

Using the Variable Menu

The **Variable** menu in the variable dictionary enables you to alter variable items.

Change item to

- **Alias**

  Changes the selected item to be an alias of another item in the current project. For example, if you have two variables, `engine_speed` and `n`, you can change `n` to be an alias of `engine_speed`, with its maximum and minimum values. For more information, see the next section, "Using Aliases" on page 6-13.
- **Formula**

  Changes a variable or constant into a formula. You have to define the right side of the formula, and use the edit boxes to change the ranges.
- **Constant**

  Changes a variable or formula into a constant. The value of the constant is by default the midpoint of the variable's range.
- **Variable**

  Changes a constant or formula into a variable. The minimum value of the variable is, by default, the value of the constant, and its maximum is, by default, twice the minimum value.

See Also

- "Setting Up Your Variable Items" on page 6-7
- "Using Aliases" on page 6-13

Using Aliases

The variable dictionary enables you to use the same set of variables, constants, and formulas with many different models and calibrations.

Why Use Aliases?

It is possible that in one model the engine speed has been defined as `N`, and in another it has been defined as `rpm`. The alias function enables you to refer to the same quantity by a variety of different names.

Creating an Alias

For example, in a variable dictionary there are two variables:

- `N` with a range of `500` to `6500`
- `rpm` with a range of `2500` to `3500`

To set rpm to be an alias of `N`:

1. Highlight the variable `rpm`.
2. Select **Variable –> Change item to –> Alias**.
3. In the dialog, choose `N` from the list.

This eliminates the variable `rpm` from your variable dictionary, and every model and calibration that refers to `rpm` now refers to `N` instead.

Note If `N` is made an alias of `rpm` in the preceding example, the range of `N` is restricted to the range of `rpm`, `2500` to `3500`.

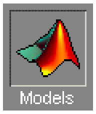
See Also

- "Setting Up Your Variable Items" on page 6-7

Setting Up Your Models

CAGE generally calibrates lookup tables by reference to models. The **Models** view is a storage place for all the models in your session.

To view and edit the models in your session, select **Models** by clicking the button shown in the **Data Objects** pane.



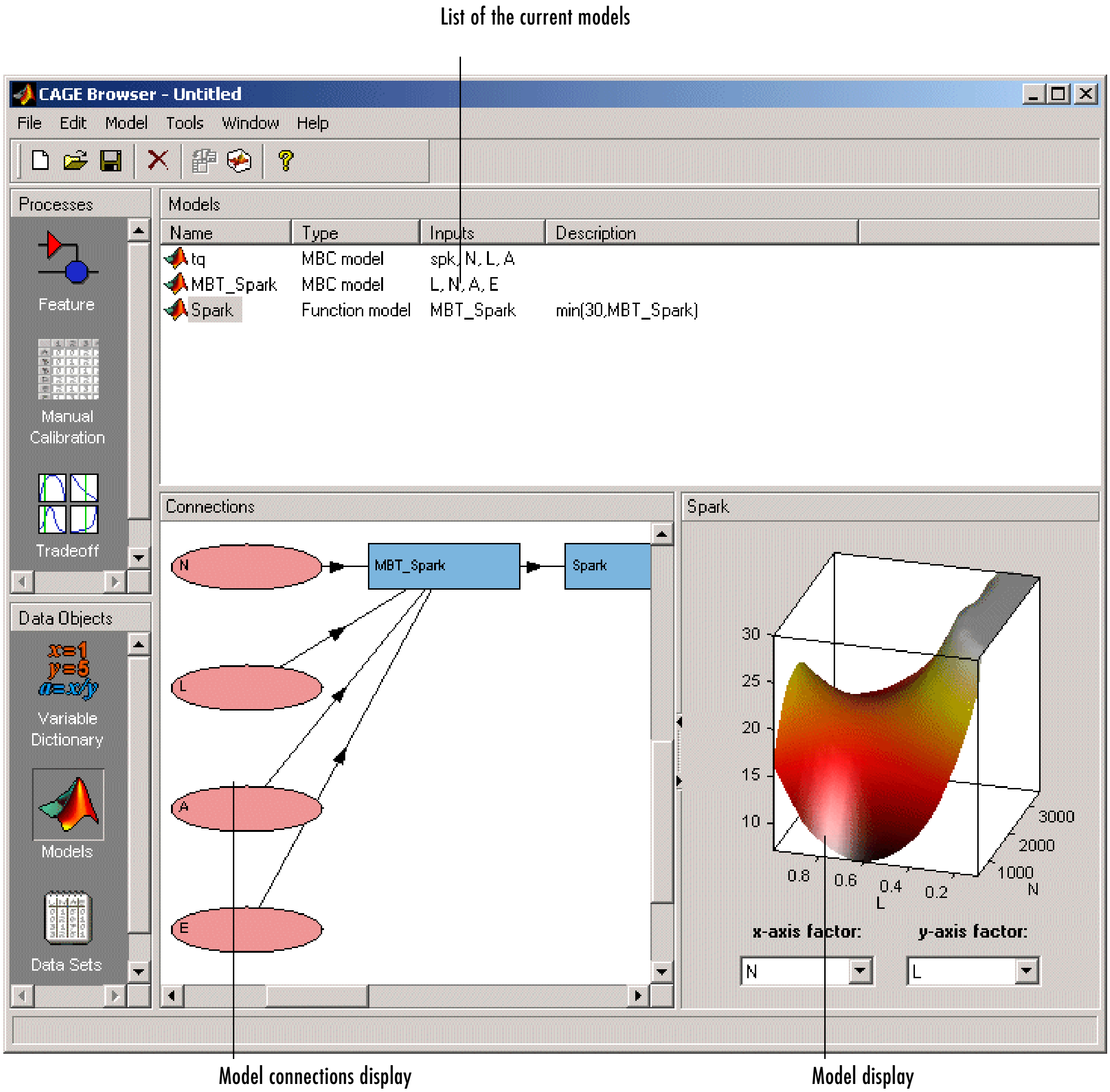
This section describes the following:

- "Importing Models" on page 6-16
- "Adding New Function Models" on page 6-18
- "Renaming and Editing Models" on page 6-19

The **Models** view displays the following:

- A list of all the models in the current project.
- The model connections. That is, which constants, variables, and models are linked to the selected model.
- An image of the response surface of the selected model.

Following is an example of the **Models** display.


List of the current models
CAGE Browser - Untitled
File Edit Model Tools Window Help
Processes
Feature
Manual Calibration
Tradeoff
Data Objects
Variable Dictionary
Models
Data Sets
Models
Name Type Inputs Description
tq MBC model spk, N, L, A
MBT_Spark MBC model L, N, A, E
Spark Function model MBT_Spark min(30,MBT_Spark)
Connections
N
L
A
E
MBT_Spark
Spark
x-axis factor:
y-axis factor:
Model connections display
Model display

Importing Models

CAGE enables you to calibrate lookup tables by referring to models constructed in the Model Browser.

To import a Model Browser model:

1 Select **File –> Import –> Model**.

   This opens the **Model Import Wizard**. The following steps take you through the three screens of the wizard.

2 Select the correct file by clicking  to browse for the correct file.

3 CAGE can only open Model Browser files.

   a If the model is saved as an `.exm` file, select **Automatic** from the **Open As** drop-down menu.

   b If the model is not saved as an `.exm` file, select **MBC Model** from the **Open As** drop-down menu. For example, the file extension might be accidentally changed.

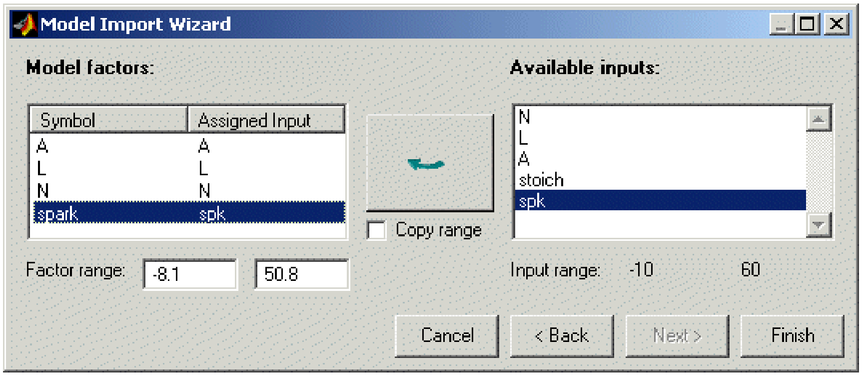
4 Click **Next >** to select the model file.

5 Select the models that you want to import by highlighting the models from the list.

6 Click **Next >** to select the models.

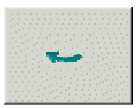
7 Associate the model factors with the available inputs in your session.

   For example, to associate the model factor `spark` with the variable `spk` in your session:



a Highlight a model factor, `spark`, in the list on the left and the corresponding variable, `spk`, in the list on the right.

b Click the select input button, shown.



c Repeat 7a and 7b for all the model factors.

8 Click **Finish** to close the wizard and return you to the **Models** view.

Note You can skip steps 6 and 7 by selecting the **Automatically assign/ create inputs** box at step 5.

You can now see a display of the model surface and the model connections.

See Also

- "Setting Up Your Models" on page 6-14
- "Adding New Function Models" on page 6-18
- "Renaming and Editing Models" on page 6-19

Adding New Function Models

A function model is a model that is expressed algebraically.

For example, you might want to view the behavior of torque efficiency. So you create a function model of torque efficiency = torque/peak torque.

To add a function model to your session:

1. Select **File –> New –> Function Model**.

   This opens the **Function Model Wizard**.

2. In the dialog box, enter the formula for your function model. For example, enter `torque_efficiency=torque/peak_torque`.

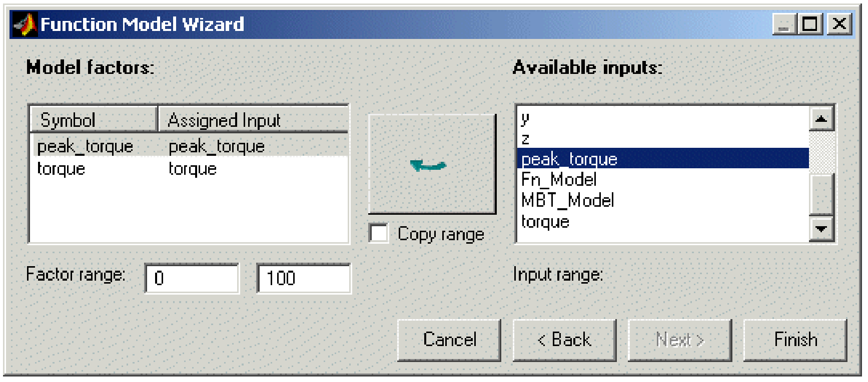
3. Press **Enter**. CAGE checks that the function is recognized; if so, you can click **Next >**. If the function is incorrectly entered, you cannot click **Next**.

4. Select the models that you want to import by highlighting the models from the list.

5. Click **Next >** to select the models.

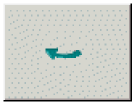
6. Associate the model factors with the available inputs in your session.

   For example, to associate the model factor `peak_torque` with the `peak_torque` model in your session:



a Highlight a model factor, `peak_torque`, in the list on the left and the corresponding model, `peak_torque`, in the list on the right.

b Click the select input button, shown.



c Repeat 7a and 7b for all the model factors.

7 Click **Finish** to close the wizard and return you to the **Models** view.

Note You can skip steps 6 and 7 by selecting the **Automatically assign/create inputs** box at step 5.

You can now see a display of the model and its connections.

See Also

- "Setting Up Your Models" on page 6-14
- "Importing Models" on page 6-16
- "Renaming and Editing Models" on page 6-19

Renaming and Editing Models

Renaming Models

To rename a model:

1. Highlight the model that you want to rename.
2. Select **Edit –> Rename**.
3. Enter the new name for the model and press **Return**.

You can also rename the model by selecting a model and clicking the name.

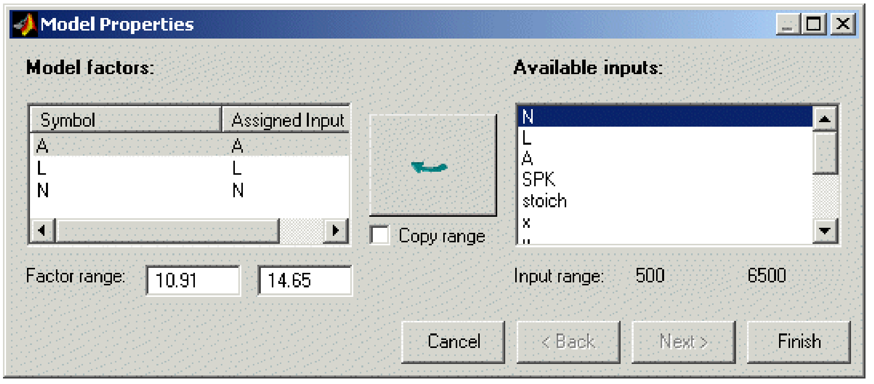
Editing Ranges of Model Factors

To edit the ranges of the model factors in the currently selected model:

1 Highlight the model that you want to edit.

2 To change the ranges of the model factors, select **Model –> Properties**.

   This opens the **Model Properties** dialog box, shown.



3 Highlight the model factor that you want to alter.

4 Enter the new range in the **Factor range** boxes.

Note This does not change the range of the variable over the entire project; rather, it changes the range of the variable in the selected model. If you want to change the range of a variable in the entire session, change the range in the variable dictionary. For more information, see "Using the Variable Menu" on page 6-12.

Editing Model Connections

You can adjust a model so that variables, formulas, or other models are the factors of the model. For example, a model of torque depends on the spark angle. In place of the spark angle, you can have a model of the maximum brake torque (MBT).

To edit the connections of a model:

1 Highlight the model.

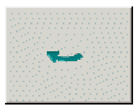
2 Select **Model –> Properties**.

   This opens the **Model Properties** dialog box.

3 Highlight the model factor that you want to adjust, in the list on the left.

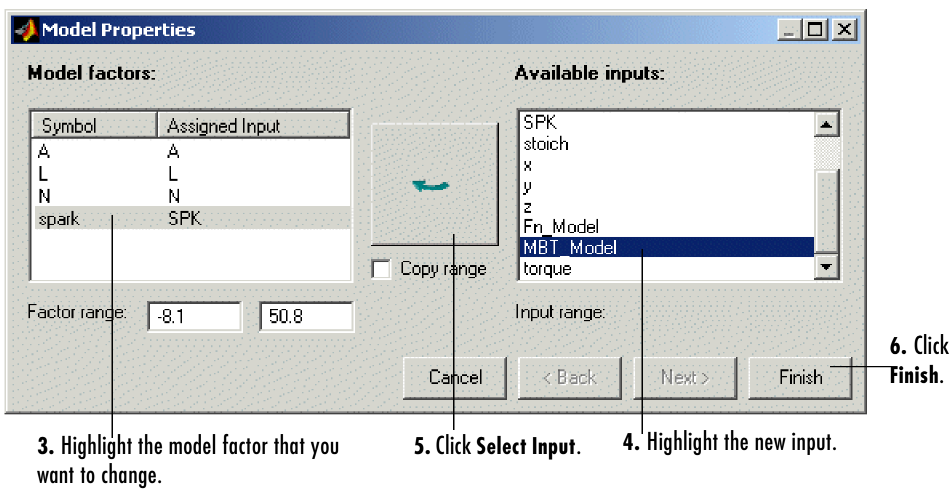
4 Highlight the new input for that factor, in the list on the right.

5 Click the **Select Input** button, shown.



6 To close the dialog box, click **Finish**.

Example of Editing the Connections of the Model



Exporting Calibrations

When you have filled some tables using any of the CAGE processes, you can export the tables.

1 Select **File –> Export –> Calibration**

2 Choose the type of file you want to save your calibrations as. You can choose from

   a **Comma Separated Value (.csv)**

   b **MATLAB-file (.mat)**

   c **M-file script**

3 Enter the file name and click **Save**.

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** or **Tradeoff** node outputs the whole feature or tradeoff (all tables and nodes).

Selecting a branch node outputs all the **Features** or **Tradeoffs** under the branch.

Specifying Locations of Files

You can specify preferred locations of project and data files, using **File –> Preferences**.

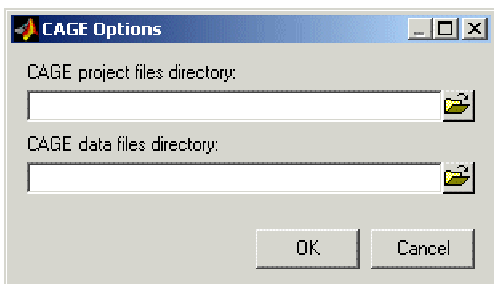
Project files have the file extension `.cag` and store entire CAGE sessions.

Data files are the files that form part of the CAGE session. For example, the following is a list of some of the data files used in CAGE:

- Simulink diagrams (`.mdl`)
- Experimental data (`.xls`, `.csv`, or `.mat`)
- Variable dictionaries (`.xml`)
- Models (`.exm`)

To specify preferred locations for project and data files:

1 Select **File –> Preferences**. This opens the dialog box shown.



2 Enter the directory where your CAGE project files are stored. Alternatively, click  to browse for the directory.

3 Enter or browse for the directory where your data files are stored.

4 Click **OK**.

7

Normalizers

A normalizer is the axis of your lookup table. It is the same as the collection of the breakpoints in your table.

This section includes the following:

- “About Normalizers” on page 7-3 describes what normalizers and breakpoints are.
- “Calibrating the Normalizers” on page 7-4 describes how to calibrate normalizers by spacing the breakpoints.
- “Normalizer View” on page 7-14 describes what you can see when you highlight a normalizer in the branch display.

About Normalizers

CAGE distinguishes between the normalizers and the tables that they belong to.

Using models to calibrate lookup tables enables you to perform analysis of the models to determine where to place the breakpoints in a normalizer. This is a very powerful analytical process.

It is important to stress that in CAGE a lookup table can either be one dimensional or two dimensional. One-dimensional tables are sometimes known as characteristic lines or functions. Two-dimensional tables are also known as characteristic maps or tables. This is important to stress, as normalizers are very similar to characteristic lines.

For example, a simple strategy to calibrate the behavior of torque in an engine might have a two-dimensional table in speed and relative air-charge (a measure of the load). Additionally, this strategy might take into account the factors of air-fuel ratio (AFR) and spark angle. Each of these compensating factors is accounted for by the use of a simple characteristic line. In CAGE, these characteristic lines are one-dimensional tables. In the example strategy, there are the following tables and normalizers:

- One characteristic map: the torque table
- Six characteristic lines:
  - Two tables: one for AFR and one for spark angle
  - Four normalizer functions: speed, load, AFR, and spark angle

Notice also that a breakpoint is a point on the normalizer where you set values for the lookup table.

Thus, when you *calibrate a normalizer* you place the individual breakpoints over the range of the table's axis.

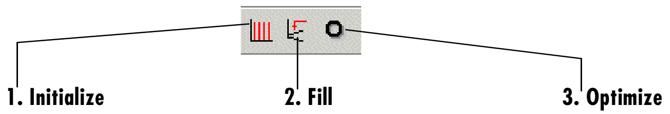
Calibrating the Normalizers

Select a normalizer in the branch display. This enables you to calibrate the normalizer, and it displays the **Normalizer** view.

For more information about the **Normalizer** view, see "Normalizer View" on page 7-14.

This section describes how you can use CAGE to space the breakpoints over the range of the normalizers.

Normalizer Toolbar



To space the breakpoints, either click the buttons on the toolbar or select from the following options on the **Normalizer** menu:

- **Initialize**

  This spaces the breakpoints evenly along the normalizer. For more information, see "Initializing Breakpoints" on page 7-5.
- **Fill**

  This spaces the breakpoints by reference to the model. For example, you can place more breakpoints where the model curvature is greatest. For more information, see "Filling Breakpoints" on page 7-6.
- **Optimize**

  This moves the breakpoints to minimize the least square error over the range of the axis. For more information, see "Optimizing Breakpoints" on page 7-10.

The next sections describe each of these in detail.

Initializing Breakpoints

Initializing the breakpoints places the breakpoints at even intervals along the range of the variable defined for the normalizer.

For example, a torque table has two normalizers, engine speed and relative air charge, or load. You can evenly space the breakpoints of both normalizers over the range 500 rpm to 6500 rpm for speed and 0.1 to 1 for the relative air charge.

To space the breakpoints evenly:

1 Click  on the toolbar or select **Normalizer –> Initialize**.

2 In the dialog box, enter the range of values for the normalizer.

   In the preceding example, for the speed normalizer, `N`, enter `500 6500`, and for the load normalizer, `L`, enter `0.1 1`.

3 Click **OK**.

Note If the selected table has two normalizers, both are evenly spaced automatically.

Filling Breakpoints

Filling breakpoints spaces the breakpoints in such a way as to place the breakpoints by reference to the model. For example, one method places the majority of the breakpoints where the curvature of the model is greatest.

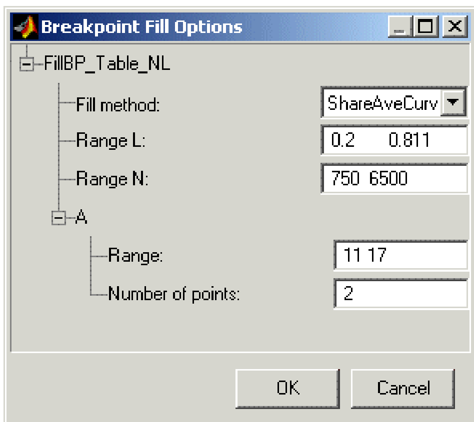
This option is only available when you are performing **Feature** calibrations.

For example, a model of the spark angle that produces the maximum brake torque (MBT) has the following inputs: engine speed, *N*, relative air charge, *L*, and air-fuel ratio, *A*. You can space the breakpoints for engine speed and relative air charge over the range of these variables by referring to the model.

To space the breakpoints based on model curvature:

1 Click  or select **Normalizer –> Fill**.

The **Breakpoint Fill Options** dialog box opens.



2 Choose the appropriate method to space your breakpoints, from the drop-down menu in the dialog box.

For the preceding example, select **ShareAveCurv**. For more information about the methods for spacing the breakpoints, see "Filling Methods" on page 7-7.

3 Enter the ranges of the values for the normalizers.

In the preceding example, for **Range N**, enter `500 6500`, and for **Range L**, enter `0.1 1`.

4 Enter the ranges of the other model variables.

CAGE spaces the breakpoints by reference to the model. It does this at selected points of the other model variables. In the preceding example, enter `11 17` for the **Range** of **A** and enter `2` for the **Number of points**. This takes two slices through the model at $A = 11$ and $A = 17$. Each slice is a surface in N and L. That is, $MBT(N, L, 11)$ and $MBT(N, L, 17)$.

CAGE computes the average value of these two surfaces to give an average model, $MBT_{AV}(N, L)$.

5 Click **OK**.

Note If any of the breakpoints are locked, each group of unlocked breakpoints is independently spaced according to the selected algorithm.

If you increase the number of slices through the model, you increase the computing time required to calculate where to place the breakpoints.

Filling Methods

This section describes in detail the methods for spacing the breakpoints of your normalizers in CAGE.

- For one-dimensional tables, the two fill methods are
  - `ReduceError`
  - `ShareAveCurv`
- For two-dimensional tables, the two fill methods are
  - `ShareAveCurv`
  - `ShareCurvThenAve`

ReduceError

Spacing breakpoints using `ReduceError` uses a greedy algorithm:

1 CAGE locks two breakpoints at the extremities of the range of values.

2 Then CAGE interpolates the function between these two breakpoints.

3 CAGE calculates the maximum error between the model and the interpolated function.

4 CAGE places a breakpoint where the error is maximum.

5 Steps 2, 3, and 4 repeat.

6 The algorithm ends when CAGE locates all the breakpoints.

ShareAveCurv and ShareCurvThenAve

Consider calibrating the normalizers for speed, N, and relative air-charge, L, in the preceding MBT model.

In both cases, CAGE approximates the $MBT_{\mathrm{AV}}(N, L)$ model, in this case, using a fine mesh.

The breakpoints of each normalizer are calibrated in turn. In this example, these routines calibrate the normalizer in N first.

Spacing breakpoints using `ShareAveCurv` or `ShareCurvThenAve` calculates the curvature, K, of the model $MBT_{\mathrm{AV}}(N, L)$

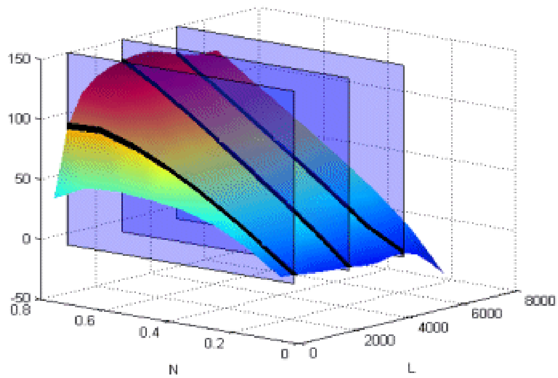
$$K = \sum_{i=1}^{\text{fine mesh}} (MBT_{AV}''(N, L))^{1/2}$$

as an approximation for

$$K = \int_{750}^{6000} \left|MBT_{AV}''(N, L)\right|^{1/2} dN$$

Both routines calculate the curvature for a number of slices of the model at various values of L. For example, the figure shown has a number of slices of a model at various values of L.

Model Slices at Various Values of L



Then

- `ShareAveCurv` averages the curvature over the range of N, then spaces the breakpoints by placing the i^{th} breakpoint according to the following rule.
- `ShareCurvThenAve` places the i^{th} breakpoint according to the rule, then finds the average position of each breakpoint.

Rule for Placing Breakpoints. If j breakpoints need to be placed, the i^{th} breakpoint, N_{i}, is placed where the average curvature so far is

$$\int_{750}^{N_i} \left|MBT_{AV}''(N,L)\right|^{1/2} dN = \frac{i-1}{j-1} \times K$$

Reference. de Boor, C., *A Practical Guide to Splines*, New York, Springer–Verlag, 1978.

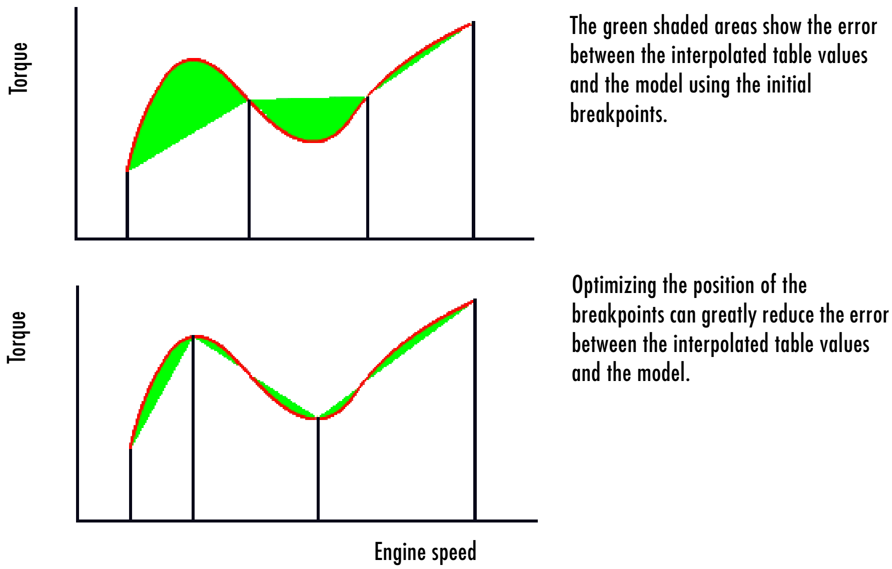
See Also

- "Calibrating the Normalizers" on page 7-4

Optimizing Breakpoints

Optimizing breakpoints alters the position of the table normalizers so that the total square error between the model and the table is reduced.

This routine improves the fit between your strategy and your model. The following illustration shows how the optimization of breakpoint positions can reduce the difference between the model and the table. The breakpoints are moved to reduce the peak error between breakpoints. In CAGE this happens in two dimensions across a table.



To see the difference between optimizing breakpoints and optimizing table values, compare with the illustration in “Optimizing Table Values” on page 8-17.

For an example of breakpoint optimization, say you have a model of the spark angle that produces the MBT (maximum brake torque). The model has the following inputs: engine speed, *N*, relative air charge, *L*, and air-fuel ratio, *A*. You can optimize the breakpoints for *N* and *L* over the ranges of these variables.

To optimize the breakpoints:

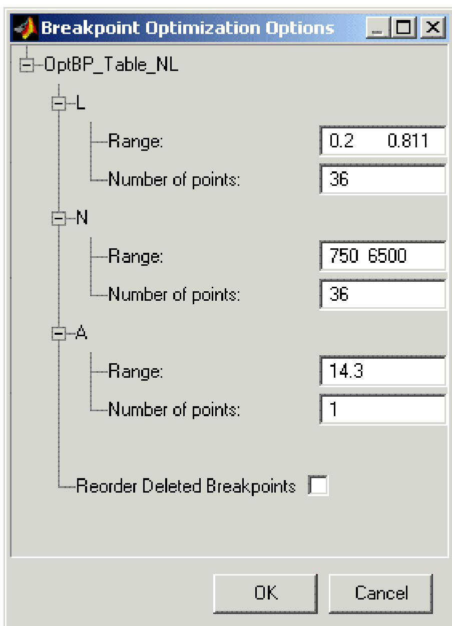
1. Ensure that the optimization routine works over reasonable values for the table by choosing one of these methods:

   a. Select **Normalizer –> Initialize**.

   b. Select **Normalizer –> Fill**.

2. Click  on the toolbar or select **Normalizer –> Optimize**.

   This opens the following dialog box.



3 Enter the ranges for the normalizers.

For the preceding example, enter `0.2 0.811` for the **Range** of **L**, and enter `750 6500` for **N**.

4 Enter the appropriate number of grid points for the optimization.

This defines a grid over which the optimization works. In the preceding example, the number of grid points is `36` for both *L* and *N*. This mesh is combined using cubic splines to approximate the model.

5 Enter ranges and numbers of points for the other model variables.

In the preceding example, the **Range** of **A** is `14.3` and the **Number of points** is `1`.

6 Decide whether or not to reorder deleted breakpoints, by clicking the radio button.

If you choose to reorder deleted breakpoints, the optimization process might redistribute them between other nondeleted breakpoints (if they are more useful in a different position).

For information about deleting breakpoints, see "Deleting Breakpoints" on page 7-20.

7 Click **OK**.

CAGE calculates the table filled with the mesh at the current breakpoints. Then CAGE calculates the total square error between the table values and the mesh model.

The breakpoints are adjusted until this error is minimized, using nonlinear least squares optimization (`lsqnonlin`).

When optimizing the breakpoints, it is worth noting the following:

- The default range for the normalizer variable is the range of the variable.
- The default value for all other model variables is the set point of the variable.
- The default number of grid points is three times the number of breakpoints.

See Also

- See the `lsqnonlin` reference page.
- "Calibrating the Normalizers" on page 7-4

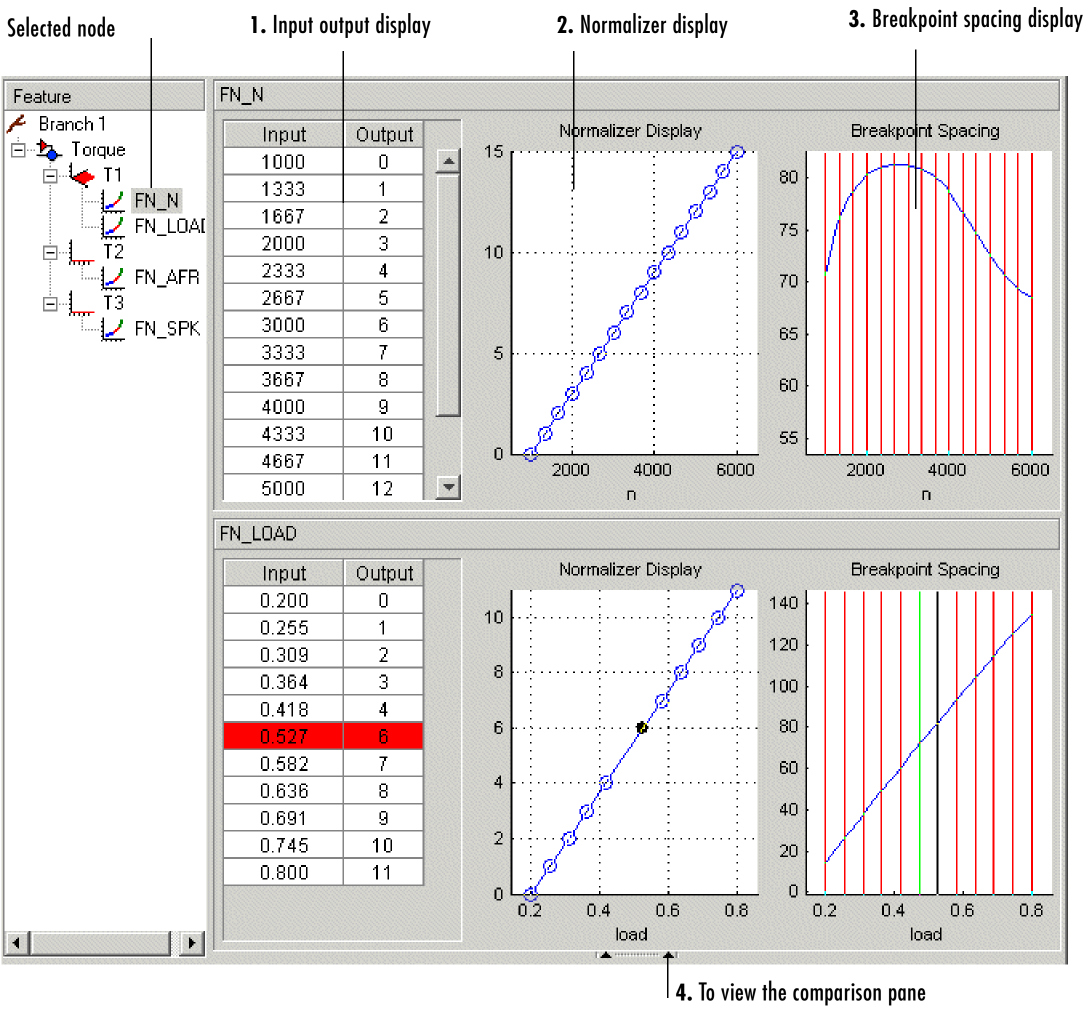
Normalizer View

The normalizer node shows the **Normalizer** view, which displays

- One normalizer if the table selected is one dimensional
- Both normalizers if the table is two dimensional

The table in the following example is two dimensional.

Normalizer View



Note If the table has two normalizers, both are displayed. The normalizer for the table columns at the top, the normalizer for the table rows below. This is true whichever normalizer on the tree is highlighted.

The parts of the display are

1 The **Input Output** display shows the breakpoints of the normalizer. For information, see “Input/Output Display” on page 7-16.

2 The **Normalizer Display** is a graphical representation of the **Input Output** display. For information, see “Normalizer Display” on page 7-18.

3 The **Breakpoint Spacing** display shows a slice of the model over the range of the breakpoints. For information, see “Breakpoint Spacing Display and Deleting Breakpoints” on page 7-19.

4 The comparison pane. For information, see “Viewing the Comparison Pane” on page 7-22.

The following sections describe in detail each part of the **Normalizer** view.

Input/Output Display

Input	Output
500	0
1055	1
1609	2
2164	3
2718	4
3273	5
3828	6
4332	7
4836	8
5391	9
5895	10
6500	11

The table consists of the breakpoints of the normalizer function.

The table has inputs and outputs:

- The inputs are the values of the breakpoints.
- The outputs refer to the row/column indices of the attached table.

To change values of the normalizers using the **Input Output** display, double-click a cell in the **Input** column and change its value.

Viewing the History of a Normalizer

To view the history of the normalizer function:

1. Right-click the table.
2. Select **Show History** from the menu.

This opens the **History** dialog box. For a more detailed description of the **History** dialog box, see "Using the History Display" on page 13-5.

Locking and Unlocking Breakpoints in the Input/Output Display

Locking breakpoints ensures that the locked breakpoint does not alter its position. You might want to lock a breakpoint when you are satisfied that it has the correct value.

To lock a breakpoint in the **Input/Output** display:

1. Right-click the selected breakpoint.
2. Select **Lock/Unlock** from the menu.

Locked breakpoints have red cells.

To unlock cells follow the same procedure.

See Also

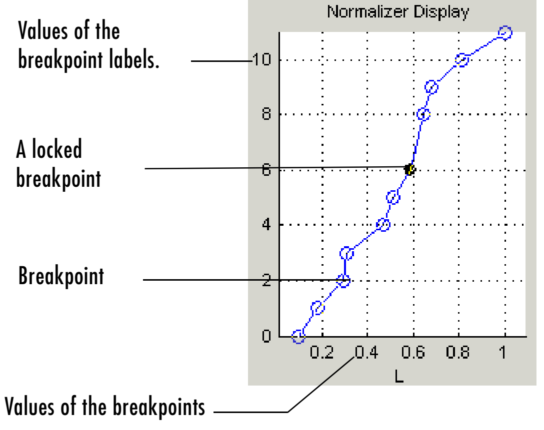
- "Normalizer View" on page 7-14

Normalizer Display

This displays the values of the breakpoints plotted against the marker numbers of the table (that is, the inputs against the outputs).

Click and drag the breakpoints to move them.

Example of the Normalizer Display



Locking and Unlocking Breakpoints in the Normalizer Display

To lock a breakpoint in the **Normalizer Display**, right-click the selected breakpoint and select **Lock Breakpoint**. You might want to lock a breakpoint when you are satisfied that it has the correct value.

Locked breakpoints are colored black.

See Also

- "Normalizer View" on page 7-14

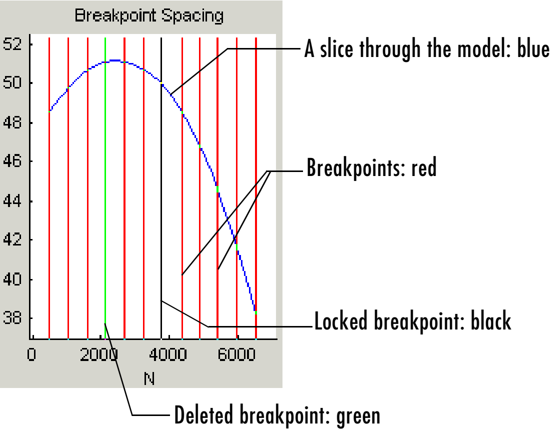
Breakpoint Spacing Display and Deleting Breakpoints

The **Breakpoint Spacing** display shows

- A slice through the model in blue
- The breakpoints in red

To move breakpoints, click and drag.

Example of the Breakpoint Spacing Display



Locking Breakpoints in the Breakpoint Spacing Display

You might want to lock a breakpoint when you are satisfied with its value.

To lock a breakpoint in the **Breakpoint Spacing** display, right-click a breakpoint and select **Lock Breakpoint** from the menu.

Locked breakpoints are colored black.

Deleting Breakpoints

Deleting breakpoints removes them from the normalizer table. There are still table values for the deleted breakpoints: CAGE determines the positions of the deleted breakpoints by spacing them linearly by interpolation between the nondeleted breakpoints.

Deleting breakpoints frees ECU memory. For example, a speed normalizer runs from 500 to 5500 rpm. Six breakpoints are spaced evenly over the range of speed, that is, at 500, 1500, 2500, 3500, 4500, and 5500 rpm. If you delete all the breakpoints except the endpoints, 500 and 5500 rpm, you reduce the amount stored in the ECU memory. The ECU calculates where to place the breakpoints by linearly spacing the breakpoints between the 500 rpm breakpoint and the 5500 rpm breakpoint.

To delete a breakpoint, right-click the breakpoint and select **Delete Breakpoint**.

Deleted breakpoints are green in the **Breakpoint Spacing** display.

Show the Model's Curvature

You might want to view the curvature of the model to manually move breakpoints to where the model's curvature is greatest.

To display the model slice as its second-order derivative, the curvature of the model:

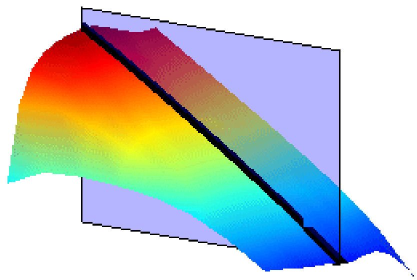
1. Right-click the model in the **Breakpoint Spacing** display.
2. Select **Display –> Model Curvature**.

You can revert to displaying the model by selecting **Display –> Model** from the right-click menu.

Multiple Slice View

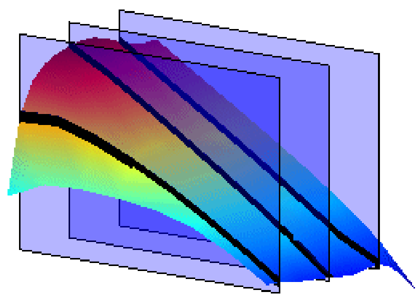
By default the **Breakpoint Spacing** display shows one slice through the model.

Slice Through a Model Surface



Viewing many slices of the model gives a better impression of the curvature of the model. For example, see the following figure.

Many Slices Through a Model Surface



To view multiple slices through the model:

1 Right-click the model slice in the **Breakpoint Spacing** display.

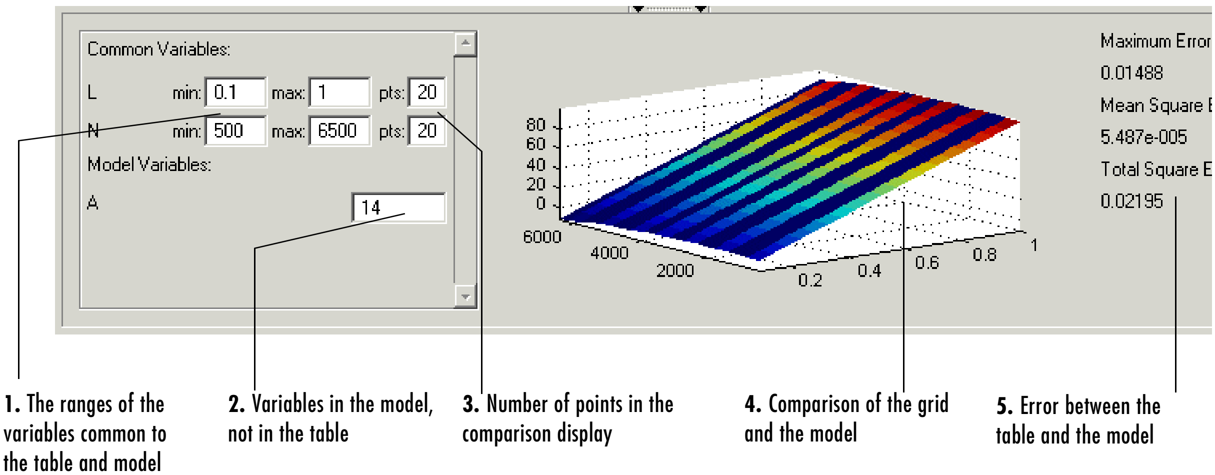
2 From the menu, select **Number of Lines** and choose the number of slices that you want to view from the list.

See Also

- “Normalizer View” on page 7-14

Viewing the Comparison Pane

To view the comparison pane, select **View –> Comparison**. Alternatively, click , the “snapper point” at the bottom of the normalizer display panes. This is labeled in the diagram of the “Normalizer View” on page 7-14.



The comparison pane displays a comparison between the following:

- A full factorial grid filled using these breakpoints
- The model

Note This is not a comparison between the current table values and the model. To compare the current table values and the model, see “Calibrating the Tables” on page 8-12.

To make full use of the comparison pane:

1 Adjust the ranges of the variables that are common to the model and table.

2 Adjust the values selected for any variables in the model that are not in the selected table.

   The default for this is the set point of the variable, as specified in the variable dictionary. For more information, see “Using Set Points in the Variable Dictionary” on page 6-10.

3 Check the number of points at which the display is calculated.

4 Check the comparison between the table and the model.

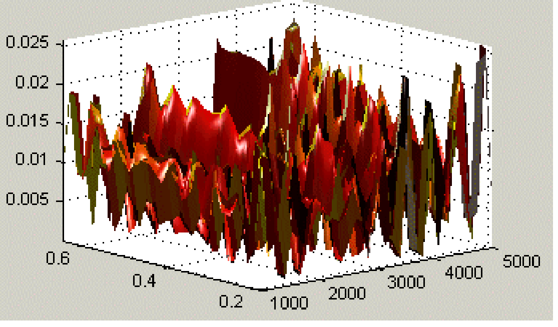
   Right-click the comparison graph to view the error display.

5 Check some of the error statistics for the comparison, and use the comparison to locate where improvements can be made.

Error Display

The comparison pane can also be used to display the error between the model and the strategy.

Error Display in the Comparison Pane



To display the error:

1 Right-click the axes of the comparison display.

2 Select **Error** from the menu.

This changes the graph to display the error between the model and the strategy.

You can display the error data in one of the following ways:

- **Error**. This is the difference between the feature and the model.
- **Squared Error**. This is the error squared.
- **Absolute Error**. This is the absolute value of the error.
- **Relative Error (%)**. This is the error as a percentage of the value of the model.
- **Absolute Relative Error (%)**. This the absolute value of the relative error.

To select one of these displays of the error data:

1 Right-click the display.

2 Select **Error Display** and select the appropriate display of the error from the context menu.

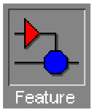
See Also

- “Normalizer View” on page 7-14
- “Comparing the Strategy and the Model” on page 8-35

  This describes the comparison made when a table node is selected in the branch display.

8

Feature Calibrations



A **Feature** calibration is the process of calibrating lookup tables and their normalizers by comparing a Simulink strategy to a model.

The Simulink strategy is an algebraic collection of lookup tables. It is used to estimate signals in the engine that cannot be measured and that are important for engine control.

CAGE calibrates an electronic control unit (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.

This discussion includes the following major topics:

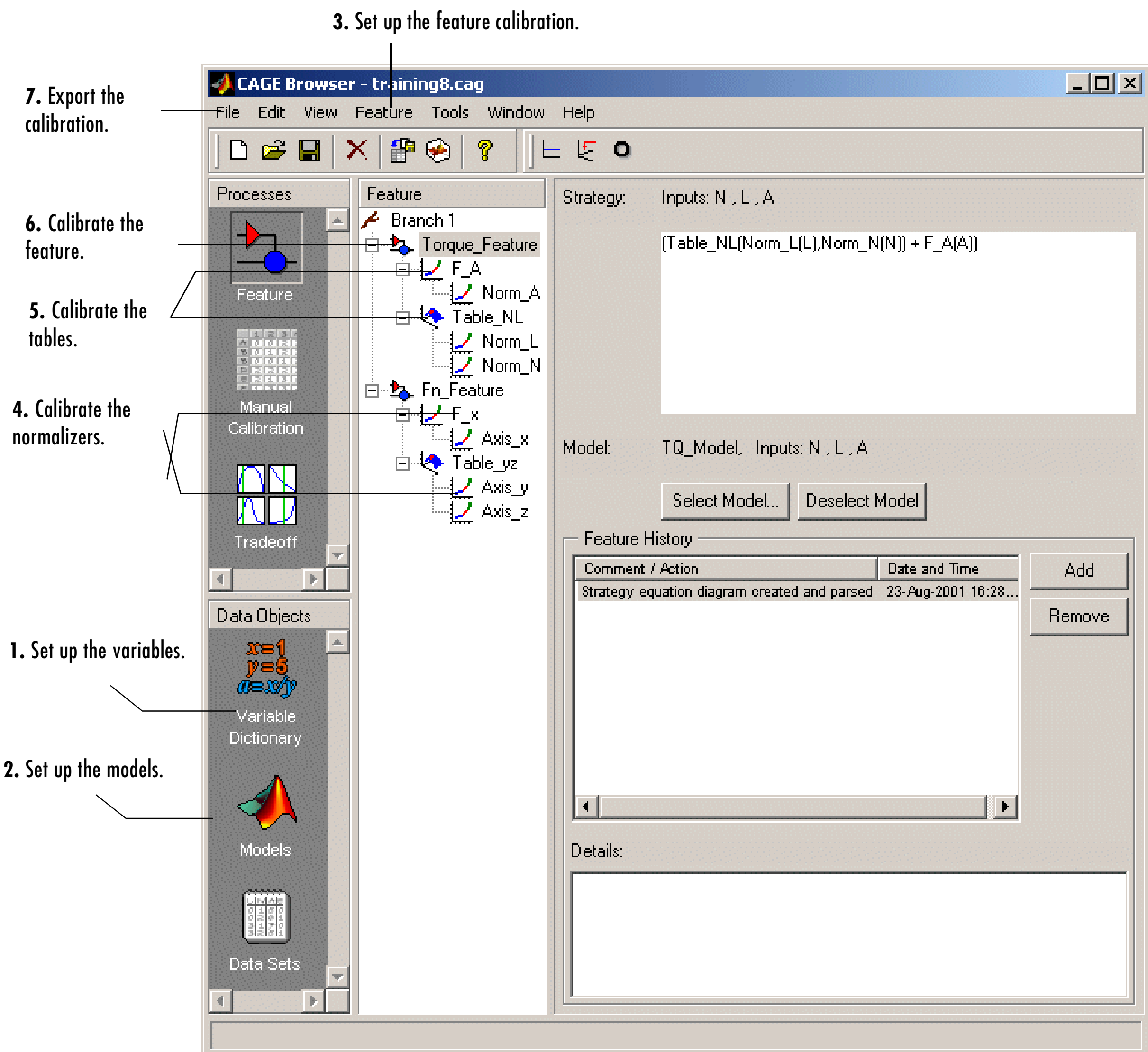
- “Performing Feature Calibrations” on page 8-3
- “Setting Up a Feature Calibration” on page 8-5
- “Calibrating the Tables” on page 8-12
- “Table View” on page 8-30
- “Calibrating the Feature Node” on page 8-38
- “Feature View” on page 8-45

See “Feature Calibration Tutorial” on page 2-1 for a tutorial showing how to perform a simple feature calibration.

Performing Feature Calibrations

The basic procedure for performing feature calibrations is as follows:

1. Set up the variables and constants. (See “Setting Up Your Variable Items” on page 6-7.)
2. Set up the model or models. (See “Setting Up Your Models” on page 6-14.)
3. Set up the feature calibration. (See “Setting Up a Feature Calibration” on page 8-5.)
4. Calibrate the normalizers. (See “Calibrating the Normalizers” on page 7-4.)
5. Calibrate the tables. (See “Calibrating the Tables” on page 8-12.)
6. Calibrate and view the entire feature. (See “Calibrating the Feature Node” on page 8-38.)
7. Export the normalizers, tables, and features. (See “Exporting Calibrations” on page 6-22.)



The normalizers, tables, and features form a hierarchy of nodes, each with its own view and toolbar.

Setting Up a Feature Calibration

A feature calibration is the process of calibrating lookup tables and their normalizers by comparing a collection of lookup tables to a model. The collection of lookup tables is determined by a strategy.

A feature refers to the object that contains the model and the collection of lookup tables. For example, a simple feature for calibrating the lookup tables for the maximum brake torque (MBT) consists of

- A model of MBT
- A strategy that adds the two following tables:
  - A speed (*N*), load (*L*) table
  - A table to account for the behavior of the air-fuel ratio (*A*)

Having already set up your variable items and models, you can follow the procedure below to set up your feature calibration:

1. Add a feature. This is described in the next section, “Adding a Feature” on page 8-5.
2. Assign a model. This is described in “Assigning a Model” on page 8-6.
3. Set up your strategy. This is described in “Setting Up Your Strategy” on page 8-6.
4. Set up the tables. This is described later, in “Setting Up Tables” on page 11-3.

This section describes steps 1, 2, and 3 in turn.

When you have completed these four steps, you are ready to calibrate the normalizers, tables, and features.

Adding a Feature

A feature consists of a model and a collection of lookup tables, organized in a strategy.

To add a feature to your session, select **File –> New –> Feature**. This automatically switches you to the **Feature** view and adds an empty feature to your session.

An incomplete feature is a feature that does not contain both an assigned model and a strategy. If a feature is incomplete, it is displayed as in the branch display. If a feature is complete, it is displayed as in the branch display.

Assigning a Model

Having already added a feature and a model to your session, you can assign a model to your feature.

To assign a model to your feature:

1 Highlight the **Feature** node in the branch display.

2 Click **Select Model** to select the model you want to work with.

   If there is only one model in your project, it is selected automatically.

   If there is more than one model in your project, a dialog box appears. Highlight the correct model to assign to your feature and click **OK**.

Setting Up Your Strategy

A strategy is an algebraic collection of tables, and forms the structure of the feature.

For example, a simple strategy to calibrate a feature for MBT adds two tables:

- A table ranging over the variables speed and relative air charge
- A table to account for the behavior of the model as the AFR varies

To evaluate the feature side by side with the model, you need to have a strategy that takes some or all of the same variables as the model.

The strategy is expressed using Simulink diagrams.

You can either import a strategy or you can construct a strategy.

The following topics are described next:

- "Importing a Strategy" on page 8-7
- "Constructing a Strategy" on page 8-8
- "Exporting Strategies" on page 8-10
- "Finding Blocks in Simulink" on page 8-11

Importing a Strategy

To import a Simulink strategy:

1. Highlight the **Feature** in the branch display.

2. Select **File –> Import –> Strategy**.

3. Select the appropriate `.mdl` file. CAGE checks the strategy for more than one outport.

4. Select the outport that you want to use.

   If there is more than one outport to your strategy, a Simulink window opens. Double-click the correct blue outport to parse (or import) the strategy to your feature.

   If there is only one outport to your strategy, a dialog box opens:

   a. Select **Automatic** to parse the strategy without viewing it.

   Or

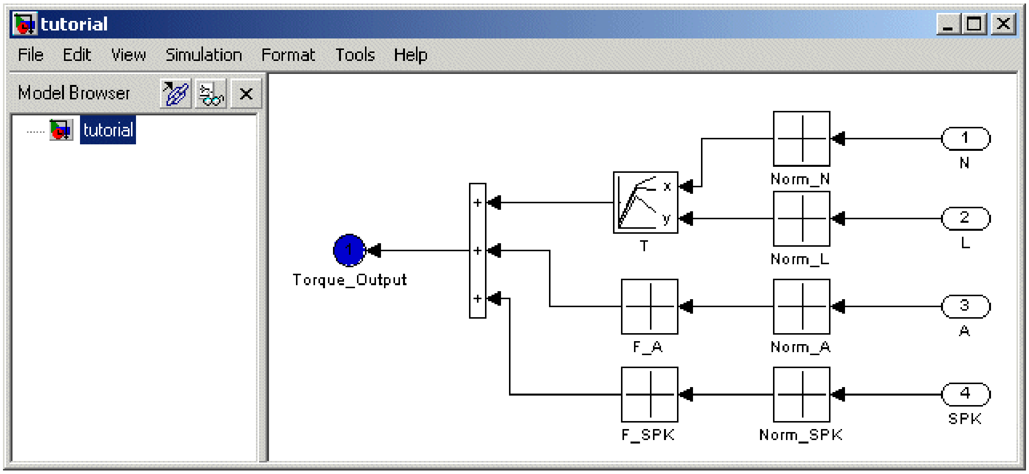
   b. Select **Manual** to edit the strategy. Double-click the blue outport circle to parse the strategy to your feature.

Note When you double-click the blue outport, the Simulink windows shut and parse this strategy to your feature.

To view a representation of your strategy, select the **Feature** node. Your strategy is represented in the **Strategy** pane.

For information about using Simulink to amend strategies, see “Constructing a Strategy” on page 8-8.

Example. In the `matlab\toolbox\mbc\mbctraining` directory, there is a Simulink diagram called `tutorial.mdl`. If you import this and select **Manual** in the dialog box, you see the following diagram.



Double-click the `Torque-Output` outport to parse the strategy into the **Strategy** pane.

Constructing a Strategy

For you to perform a feature calibration, the strategy and the model must have some variables in common.

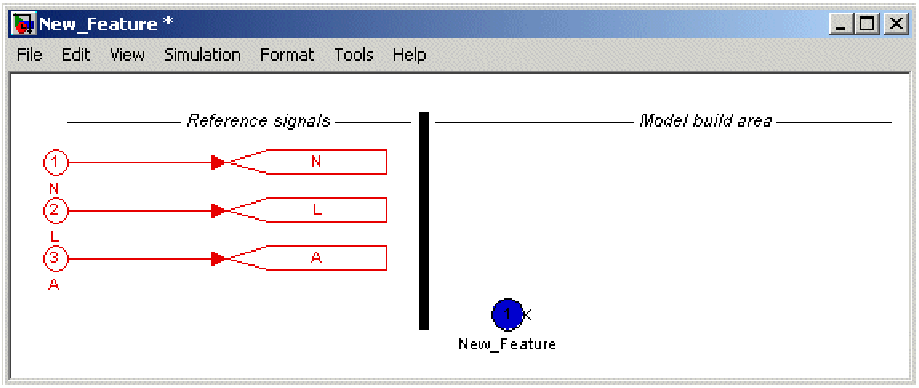
To construct a strategy using Simulink:

1. Highlight the correct feature by clicking the **Feature** node.
2. Select **Feature –> Graphical Strategy Editor** or press **Ctrl+E**.

   Three Simulink windows open:

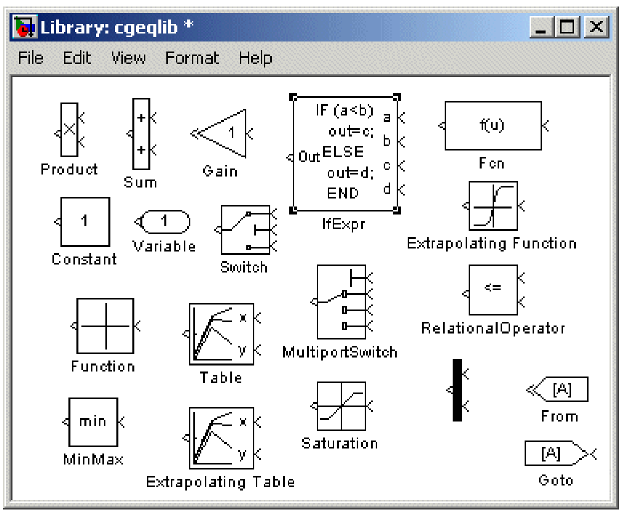
   - The strategy window for editing your strategy

Example of a Strategy Window



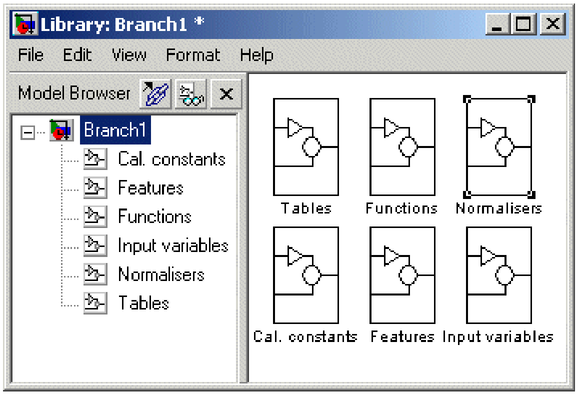
- A library window with all the blocks available for building a strategy

Library of All Available Blocks



- A library window with all the existing blocks in your session, organized in libraries

Library of All Existing Blocks



3 In the strategy window, build your strategy using the blocks available in the library windows.

4 Double-click the blue outport circle to parse the strategy into the CAGE session.

Note This closes all three Simulink windows and parses your strategy into the feature.

For more information about using Simulink to build your strategy, see Simulink Help.

Exporting Strategies

Simulink strategies can be exported. For example, you might want to

- Include a strategy in a Simulink vehicle model
- Evaluate the strategy using Real-Time Workshop® to produce C code
- Evaluate the strategy using Simulink

To export a strategy from CAGE:

1 Highlight the **Feature** node that contains the strategy that you want to save.

2 Select **File –> Export –> Strategy**.

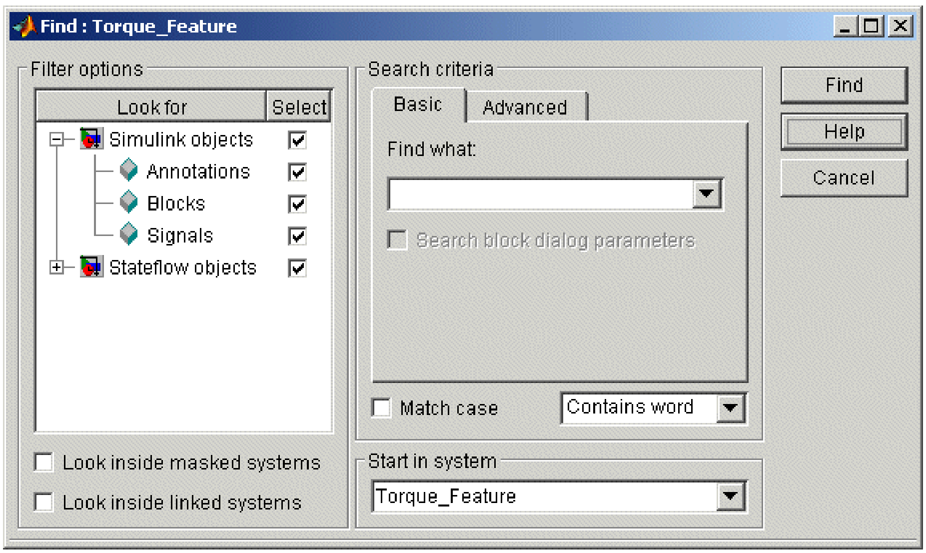
3 Assign a name for your strategy.

The strategy is saved as a Simulink model (`.mdl`) file.

Finding Blocks in Simulink

To find blocks in Simulink, highlight a Simulink window and press **Ctrl+F** or select **Edit –> Find**.

This opens the **Find** dialog box, which helps find the blocks associated with your feature.



For more information about finding blocks, see Simulink Finder Help.

Calibrating the Tables

After you set up your session and your tables, you can calibrate your tables.

Highlight a table in the branch display to view the **Table** view. For more information about the **Table** view, see “Table View” on page 8-30.

In CAGE, a table is defined to be either a one-dimensional or a two-dimensional lookup table. One-dimensional tables are sometimes known as characteristic lines or functions. Two-dimensional tables are also known as characteristic maps or tables.

Each lookup table has either one or two axes associated with it. These axes are normalizers. See “Normalizers” on page 7-1 for more information.

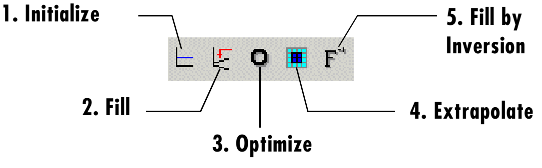
For example, a simple MBT feature has two tables:

- A two-dimensional table with speed and relative air charge as its normalizer inputs
- A one-dimensional table with AFR as its normalizer input

Before you can calibrate your tables, you must calibrate your normalizers. For information, see “Calibrating the Normalizers” on page 7-4.

This section describes how you can use CAGE to fill your lookup tables by reference to a model.

Table Node Toolbar



To fill the table values, either click the buttons in the toolbar or select from the following options in the **Table** menu:

- **Initialize**

  Sets each cell in the lookup table to a specified value. For information, see “Initializing Table Values” on page 8-13.

- **Fill**

  Fills the table values by reference to the model. For information, see “Filling Table Values” on page 8-14.
- **Optimize**

  Fills the table values by minimizing the total square error between the table values and the model. For information, see “Optimizing Table Values” on page 8-17.
- **Extrapolate**

  Fills the table values based on the cells specified in the extrapolation mask. You can choose values in cells that you trust to define the extrapolation mask and fill the rest of the table using only those cells for extrapolation. For information, see “Filling the Table by Extrapolation” on page 8-20.
- **Fill by Inversion**

  Fills the table by creating an inversion of another table. For information, see “Inverting a Table” on page 8-22.

The next sections describe each of these in detail.

Initializing Table Values

Initializing table values sets the value of every cell in the selected table to a constant.

To initialize the values of the table:

1 Click  or select **Table –> Initialize**.

2 In the dialog box that appears, select the constant value that you want to insert into each cell.

When initializing tables, you should think about your strategy. Filling with zeros can cause a problem for some strategies using “modifier” tables. For example, your strategy might use several speed-load tables for different values of AFR, or you might use an AFR table as a “modifier” to add to a single speed-load table to adjust for the effects of different AFR levels on your torque output.

If your table is a modifier that is added to other tables, you should initially fill it with 0s; if it is a modifier that multiplies other tables, you should fill it with

1s. If you do not, when CAGE tries to fill the table by rearranging the strategy equation (model = table * modifier table), there is a problem, as you cannot divide by zero. This operation will fail.

See “How CAGE Fills Tables” on page 8-14.

Filling Table Values

This tool fills the table with the values of the model at the operating points specified in your normalizers.

To fill the table values by reference to the model:

- Click  or select **Table –> Fill**.

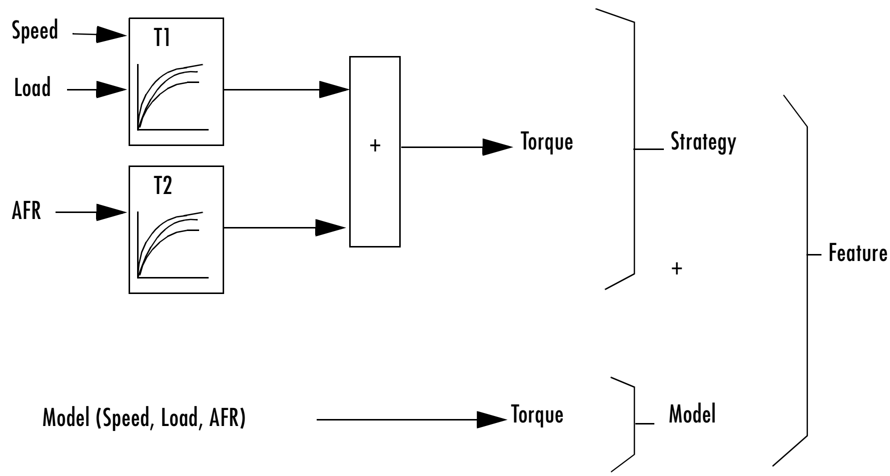
How CAGE Fills Tables

CAGE fills tables in a feature calibration by rearranging the equation model = strategy.

Example

A very simple example strategy for torque might consist of two tables:

- A speed-load (or relative air-charge) table filled with values of torque
- An air-fuel ratio (AFR) modifier table to account for the behavior of AFR



This example is a strategy with a base speed-load (N, L) table for torque and a modifier table to account for the behavior of AFR (A). Your strategy might use several speed-load tables for different values of AFR, or as in this case you might use an AFR table as a modifier to add to a single speed-load table to adjust for the effects of different AFR levels on your torque output.

$$T1(N,L) + T2(A) \approx Model(N, L, AFR) -$$

With the tables arranged in the following manner, this is the feature equation:

$$Model \approx T1 + T2$$

To fill the speed-load table, the equation is rearranged to

$$T1 = Model - T2$$

If the AFR modifier table (T2) is initialized with 0s, this becomes

$$T1 = Model - 0 \text{ or}$$

$$T1 = Model$$

Each cell in the table is therefore filled with the corresponding values of the model at the operating point specified by the breakpoints.

For example, to fill the T1 cell (Speed = 2500, Load = 0.5), CAGE evaluates the model at Speed = 2500, Load = 0.5, and uses the value of AFR that you choose in the dialog that appears.

You can choose one value of AFR for the whole table (for example, 14.3), or you can choose a range of values and fill using the average model value at each cell. For example, if you choose AFR = 11, 13, 15, the model is evaluated at all three values for each cell and the average model value is used. The default AFR value is the set point (which you can set in the variable dictionary).

Note To fill using model values averaged over a range of an input variable, you type the minimum and maximum (separated by a space) in the **Range** edit box, and the number of desired points in the **Points** edit box.

When the base table (T1) is filled, CAGE rearranges the equation again to fill the modifier table (T2):

$$T2(A) = Model(N,L,A) - T1(N,L)$$

For example, to fill the T2 cell at AFR = 12.5, you choose values of speed and load (such as 3000,0.4) and CAGE evaluates the following:

$$T2(12.5) = Model(3000, 0.4, 12.5) - T1(3000, 0.4)$$

As before, you can choose a range of values for speed and load and use the average to fill the table.

Note Be careful not to initialize modifier tables with 0 if they are multipliers in your strategy. In this case, solving $Model \approx T1 \times T2$ for T1 gives $T1 \approx Model/T2$, and you cannot divide by zero. This operation will fail.

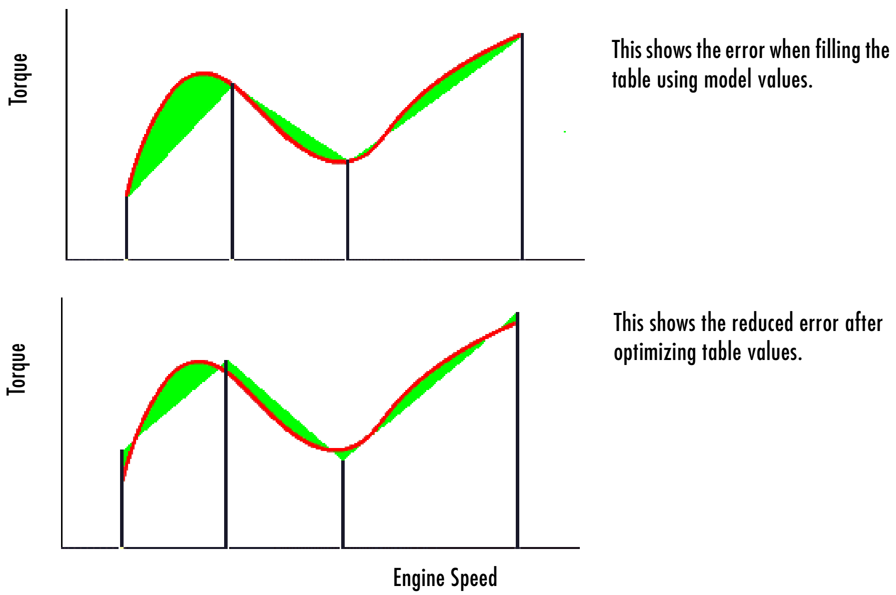
Solving $Model \approx Strategy$ algebraically for a table in the strategy is not always possible. In these cases you must use optimization.

Optimizing Table Values

Optimizing the table values minimizes the current total square error between the table values and the model.

This routine improves the fit between your strategy and your model. Using **Fill** places model values directly into your table; whereas the optimization process can shift those values up and down to give the least overall error between the interpolation between table values and the model surface. You should use **Fill** first to place model values into your table — this gives the optimization routine a good starting point.

This process is illustrated by the following example; the green shaded areas show the error between the mesh model (evaluated at the number of grid points you choose) and the table values.



To see the difference between optimizing table values and optimizing the positions of breakpoints, compare with the illustration in "Optimizing Breakpoints" on page 7-10.

For an example of optimizing table values, say you have a model of the spark angle that produces the maximum brake torque (MBT). The model has inputs engine speed, *N*, relative air charge, *L*, and air-fuel ratio, *A*. The strategy that you are using to calibrate the MBT feature has an *N*-*L* table and a table to account for the variation of MBT over the range of *A*.

To optimize the table values:

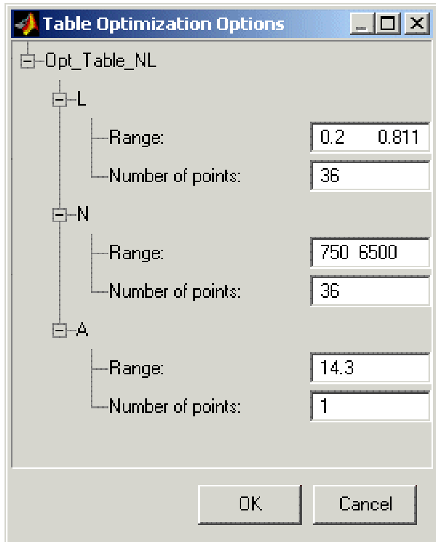
1 Ensure that the optimization routine works over a reasonable range of values by selecting **Table –> Fill**.

   Optimization works best if you start with a sensible range of values. Using **Fill** places model values at the appropriate operating points into each cell of the table. From this point, the optimization routine has the best chance of finding a good solution quickly.

   For example, a model for MBT might have a range of values from 20 degrees to 35 degrees. Running the optimization routine when most of the cell values are outside this range (say if your table is filled with 0s), is very time consuming.

2 Click  or select **Table –> Optimize**.

   This opens the following dialog box.



3 Enter the ranges for the normalizers.

For the preceding example, enter `0.2 0.811` for the range of **L** and enter `750 6500` for **N**.

4 Enter the number of grid points for the optimization.

This defines a grid over which the optimization works. Above, the number of grid points is 36 for both L and N. This mesh is used to approximate the model. The default number of grid points is three times the number of breakpoints in the table.

5 Enter the ranges and numbers of points for the other model variables.

In the preceding example, the range of **A** is `14.3` and the **Number of points** is `1`. The mesh approximates the value of the model at only one value of *A*.

6 Click **OK**.

CAGE evaluates the model over the number of grid points specified, then calculates the total square error between this mesh model and the table values.

CAGE adjusts the table values until this error is minimized, using `lsqnonlin`.

When optimizing the table values, it is worth noting the following:

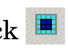
- The default range for a normalizer variable is the range of the variable.
- The default value for all other model variables is the set point of the variable's range.
- The default number of grid points is three times the number of breakpoints.
- Increasing the number of grid points increases the quality of the approximation, but also the computation time.

See Also

- See the `lsqnonlin` reference page.
- "Calibrating the Tables" on page 8-12

Filling the Table by Extrapolation

Filling a table by extrapolation fills the table with values based on the values already placed in the extrapolation mask. The extrapolation mask is described below.

To fill a table by extrapolating over a preselected mask, click  or select **Table –> Extrapolate**.

This extrapolation does one of the following:

- If the extrapolation mask has only one value, all the cell values change to the value of the cell in the mask.
- If the extrapolation mask has two or more collinear values, the cell values change to create a plane parallel to the line of values in the mask.
- If the extrapolation mask has three or more coplanar values, the cell values change to create that plane.
- If the extrapolation mask has four or more ordered cells (in a grid), the extrapolation routine fills the cells by a grid extrapolation.
- If the extrapolation mask has four or more unordered (scattered) cells, the extrapolation routine fills the cell values using a thin plate spline interpolant (a type of radial basis function).

Using the Extrapolation Mask

The extrapolation mask defines a set of cells that form the basis of any extrapolation.

For example, a speed-load (or relative air-charge) table has values in the following ranges that you consider to be accurate:

- Speed 3000 to 5000 rpm
- Load 0.4 to 0.6

You can define an extrapolation mask to include all the cells in these ranges. You can then fill the rest of your table based on these values.

To add or remove a cell from the extrapolation mask:

1 Right-click the table

2 Select **Add to/Remove from Mask** from the menu.

Cells included in the extrapolation mask are colored yellow.

Cells that are locked and in the extrapolation mask are colored green.

Generating the Extrapolation Mask from the Predicted Error

Predicted error (PE) is the standard deviation of the error between the model and the data used to create the model. You can automatically generate an extrapolation mask based on the predicted error.

To generate a mask automatically:

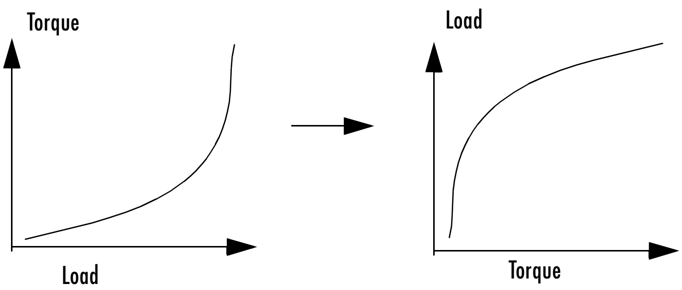
1 Right-click the **Menu** button of the table and select **Generate Extrapolation Mask from Predicted Error**.

2 In the dialog box, set the PE threshold. Click **OK**.

The cells in the table where the predicted error is within the threshold now form the extrapolation mask, and thus are colored yellow.

Inverting a Table

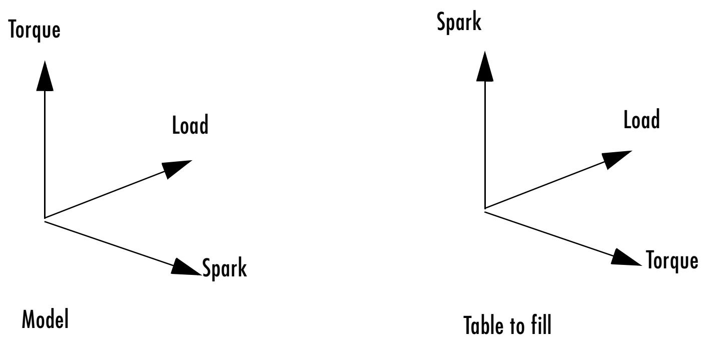
You can use CAGE to produce a table that is the inverse of another table. This involves swapping a table input with a table output, and you can invert 1-D or 2-D tables.



Inverting a table allows you to link a *forward strategy* to a *backward strategy*; that is, swapping inputs and outputs. This process is desirable when you have a “forward” strategy, for example predicting torque as a function of speed and load, and you want to reverse this relationship in a “backward strategy” to find out what value of load would give a particular torque at a certain speed.

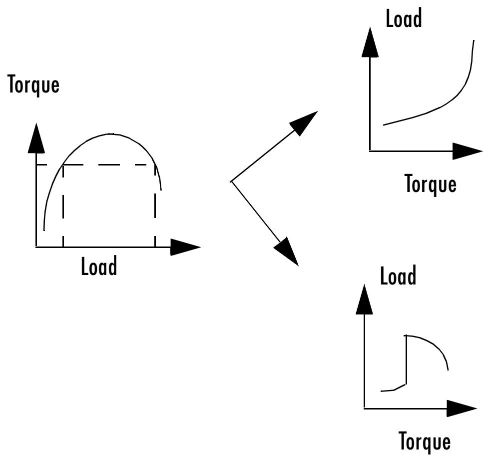
Normally you fill tables in CAGE by comparing with data or models. Ideally you want to fill using the correct strategy, but that might not be possible to find or measure. If you only have a forward strategy but want a backward one, you can fill using the forward strategy (tables or model) and then invert the table.

For example, in order to fill a table normally from a model, you need the model response to be the table output, and the model inputs to be a function of the table inputs (or it should be possible to derive the input — for example, air mass from manifold pressure). If the available model is “inverted” (the model response is a table input and the table output is a model input) and you cannot change the model, you can invert the table in CAGE.



In the diagram of a table shown, the x- and y-axes represent the normalizers (which you want to be spark and load) and the z-axis is the output at each breakpoint (torque). To fill this table correctly from the model is a two-step process. First you need to fill a table that has the same input and output as the model, and then fill a second table by inversion.

For the inversion to be deterministic and accurate, the table to be inverted must be monotonic; that is, always increasing or decreasing. This requirement is explained by the following one-dimensional example. Every point on the y-axis must correspond to a unique point on the x-axis. The same problem applies also to two-dimensional tables: for any given output in the first table there must be a unique input condition; that is, every point on the z-axis should correspond to a unique point in the x-y plane. Some table inversions have multiple values and so do not meet this requirement, just as the square root function can take either positive or negative values. You can use the inversion wizard in CAGE to handle this problem; you can control the inversion process and determine what to do in these cases.



The preceding example illustrates a table with multiple values. There are two solutions for a single value of torque. CAGE has a table inversion wizard that can help overcome this problem. You can specify whether you want to use the upper or lower values for filling certain parts of the the table; this allows you to successfully invert a multiple-valued function. See the inversion instructions for 1-D and 2-D tables in the next sections.

The process of inverting a one-dimensional table is different from the process of inverting a two-dimensional table.

Inverting One-Dimensional Tables

To invert a one-dimensional table:

1 Ensure that your session contains two tables:

 a The first table from your forward strategy, filled

 b The second table from your backward strategy, which you want to fill

2 Highlight the second table.

3 Click  or select **Table –> Inversion**.

 The lower pane now acts as a wizard.

4 In the lower pane, highlight the table that you want to invert.

5 Click **Next**. The next page asks what CAGE should do if it encounters multiple values. The options are

 - **Maximum** selects the uppermost range if a given number has two possible inverses (like selecting the positive square root of a number).
 - **Minimum** selects the lower of the two if a given number has two possible inverses (like selecting the negative square root of a number).
 - **Intermediate** selects the middle range if a given number has more than two possible inverses.
 - **Automatic** selects the range that produces the least error (see below; the last page of the wizard plots the error metric).

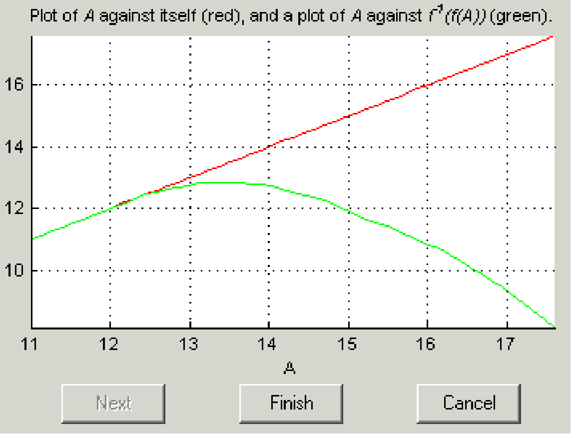
 For example, the function $y = x^2$ is impossible to invert over the range -1 to 1. You can specify to invert the range from 0 to 1, sacrificing the inversion in the lower range, or the reverse. To select the range from 0 to 1, highlight **Maximum**.

 The display shows a comparison between the table (green) and the function $x = f^{-1}(f(x))$.

6 Highlight the part of the table to invert, then click **Next**.

 The last page of the wizard has a comparison plot that shows how successful the inversion has been. If your forward function is y = f(x), and your inverse function is x = g(y), then, combining these, in an ideal world, you should have x = g(f(x)). The plot then displays a red line showing x against x and a green line showing x against g(f(x)). The closeness of these two lines indicates how good the inversion has been: a perfect inverse would show the lines exactly on top of each other. In the following example, the lines are together and then diverge; this plot can show you which part of your table has not successfully inverted and where you should try a different routine.

Inverting a One-Dimensional Table



Note The automatic inversion routine tries to minimize the total distance between these lines. This can sometimes lead to unexpected results. For example, given the function f(x) = x^2 between –1 and 1, if you select either positive or negative square root as the inverse, this induces a large error in the combined inverse. If you choose g(y) = sqrt(y), then g(f(-1)) = 1, an error of 2. To minimize this, the automatic routine might choose to send everything to zero and accept a medium error over the whole range rather than a large error over half the range. The more knowledge you have of the form of the “forward” table will help you make an informed choice about which routine to select.

---

7 Click **Finish** to accept the inversion or **Cancel** to ignore the result and return to the original table.

Inverting Two-Dimensional Tables

To invert a two-dimensional table:

1 Ensure that your session contains two tables:

   a The first table from your forward strategy, filled

   b The second table from your backward strategy, which you want to fill

2 Highlight the second table.

3 Click  or select **Table –> Inversion**.

   The lower pane now acts as a wizard.

4 In the lower pane, highlight the table that you want to invert.

5 Click **Next**.

6 Identify the corresponding signals.

   The forward table and backward table share a common input. This page of the wizard lists all possible combinations of inputs into the forward and backward tables and asks you to highlight the combination that gives the two common inputs. To illustrate this, if the forward table gives torque in terms of the variables engine speed and load, whereas you want the backward table to give load in terms of RPM and Tq, then the list would read

- RPM and engine speed
- RPM and load
- Tq and engine speed
- Tq and load

   In this case, you would select the first option.

7 Highlight the part of the table to invert, then click **Next**.

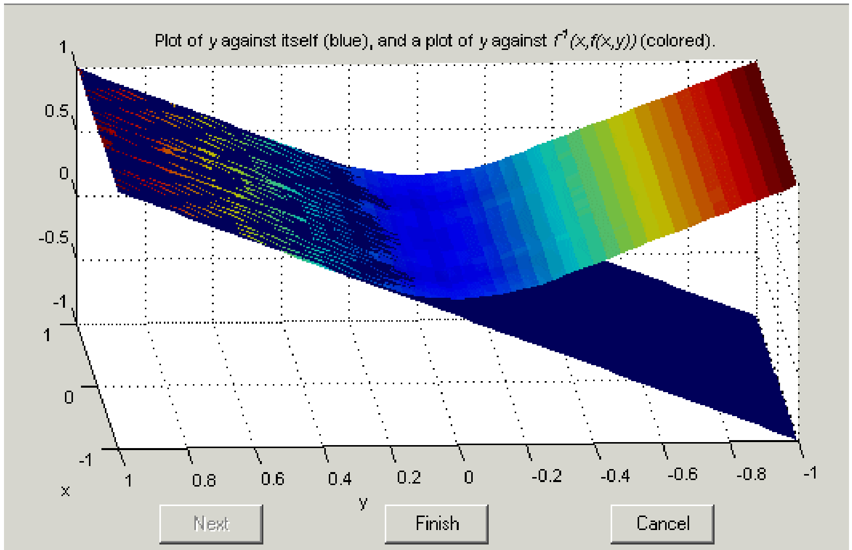
   CAGE asks what to do if it encounters multiple values. The choices are

   - **Maximum** selects the uppermost range (like choosing a positive square root of a number).

- **Minimum** selects the lower value if there are two choices (like choosing a negative square root of a number).
- **Intermediate** selects the middle range when there are more than two choices.
- **Automatic** selects the range that produces the least error. CAGE tries to choose values to put in the inverse table that minimize an error metric similar to the error metric for 1-D tables (see "Inverting One-Dimensional Tables" on page 8-25).

8 Choose one of these options and click **Next**.

The last page of the wizard has a comparison plot that shows how successful the inversion has been. If the forward function is z = f(x,y), and the inverse function is x = g(y,z), then, combining these, in an ideal world you should have x = g(y,f(x,y)). The plot then displays a plane showing x plotted against x and y, and a colored surface showing g(y,f(x,y)) plotted against x and y. The closeness of these two planes indicates how good the inversion is. Following is an example. In this case, the forward table is a quadratic (z = y^2); the backward table is inverted using the positive square root of z (maximum range). As you can see, this leads to large errors at negative values of y, but good inversion for positive values of y.



9 Click **Finish** to accept the result or **Cancel** to ignore the result and return to the original table.

Table View

When you select a table in the branch tree (in feature or manual calibration), you see the **Table** view. In CAGE, a table is defined to be either a one-dimensional or a two-dimensional lookup table. One-dimensional tables are sometimes known as characteristic lines or functions. Two-dimensional tables are also known as characteristic maps or tables. CAGE regards them both as similar objects.

Each lookup table has either one or two axes associated with it. These axes are normalizers.

For example, a simple MBT feature has two tables:

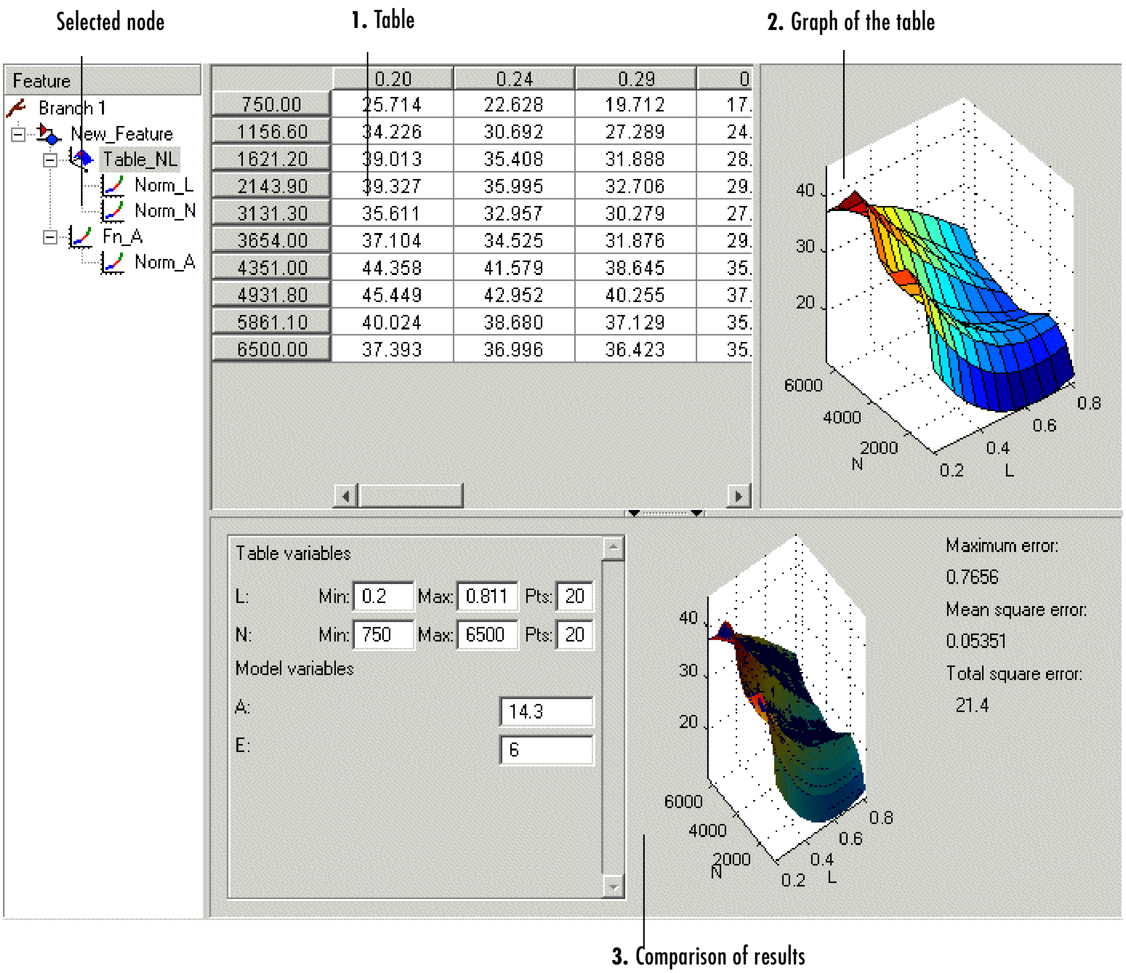
- A two-dimensional table with speed and relative air charge as its normalizers
- A one-dimensional table with AFR as its normalizer

For feature calibration (filling the tables by comparing a strategy and a model), see “Calibrating the Tables” on page 8-12.

For an example of manual calibration (filling tables using experimental data), see “Filling Tables from Data Tutorial” on page 5-1.

The example following is a feature view. In manual calibration, you do not see the comparison pane because you are not comparing tables with a model.

Table Display in Feature Calibration



	0.20	0.24	0.29	0
750.00	25.714	22.628	19.712	17.
1156.60	34.226	30.692	27.289	24.
1621.20	39.013	35.408	31.888	28.
2143.90	39.327	35.995	32.706	29.
3131.30	35.611	32.957	30.279	27.
3654.00	37.104	34.525	31.876	29.
4351.00	44.358	41.579	38.645	35.
4931.80	45.449	42.952	40.255	37.
5861.10	40.024	38.680	37.129	35.
6500.00	37.393	36.996	36.423	35.

The parts of the display are numbered and labeled as follows:

1 The table pane displays the breakpoints of the normalizer and the values of the table. (See “Viewing a Table” on page 8-32.)

2 The graph of the table pane displays the table values graphically. (See “Using the Graph of the Table” on page 8-34.)

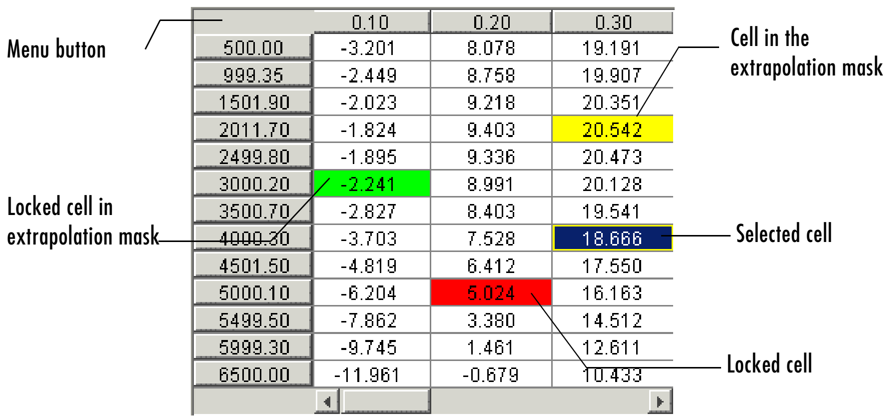
3 The comparison-of-results pane displays a comparison between the current output of the strategy and the feature model. (See “Comparing the Strategy and the Model” on page 8-35.)

Note You can view the **History** display by selecting **View –> History**. For information, see “Using the History Display” on page 13-5.

This section describes each of these parts in detail.

Viewing a Table

The table displays the values of your lookup table and displays the breakpoints of the normalizers. For example, the following table shows a lookup table with speed and relative air-charge (load) as its normalizers.



	0.10	0.20	0.30
500.00	-3.201	8.078	19.191
999.35	-2.449	8.758	19.907
1501.90	-2.023	9.218	20.351
2011.70	-1.824	9.403	20.542
2499.80	-1.895	9.336	20.473
3000.20	-2.241	8.991	20.128
3500.70	-2.827	8.403	19.541
4000.30	-3.703	7.528	18.666
4501.50	-4.819	6.412	17.550
5000.10	-6.204	5.024	16.163
5499.50	-7.862	3.380	14.512
5999.30	-9.745	1.461	12.611
6500.00	-11.961	-0.679	10.433

To edit a value in the table, double-click the cell. Selected cells are light blue (or pink if they are locked or in the extrapolation mask).

Locking and Unlocking Cell Values

When you are satisfied with a region of the table, you might want to lock the cell values in that region, to ensure that those values do not change.

To lock or unlock a cell value, right-click the cell and select **Lock/Unlock** from the menu.

Locked cells are colored red in the display (they turn pink when selected).

Menu Button Options

Right-clicking the menu button in the top left corner of the table gives you the following options:

- **Lock/unlock entire table**. This toggles between the table's being locked or unlocked.
- **Generate Extrapolation mask from predicted error**. See "Generating the Extrapolation Mask from the Predicted Error" on page 8-21.
- **Clear extrapolation mask**. This ensures that none of the cells are in the extrapolation mask.

Properties

The table properties enable you to specify the precision type of the table data.

You can choose from

- Floating-point precision
- Polynomial ratio, fixed-point precision
- Lookup table, fixed-point precision

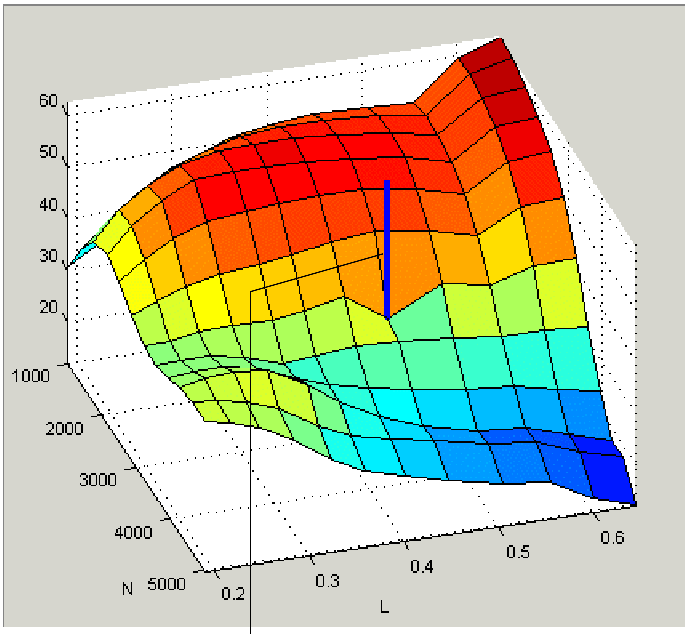
To display the properties of the table, select **Table –> Properties**.

This opens the **Table Properties** dialog box.

Table properties are discussed in detail in "Table Properties" on page 11-7.

Using the Graph of the Table

The table view displays both the table values and a graph of the table. This gives a useful display of the table's behavior. Shown is an example of a graph.

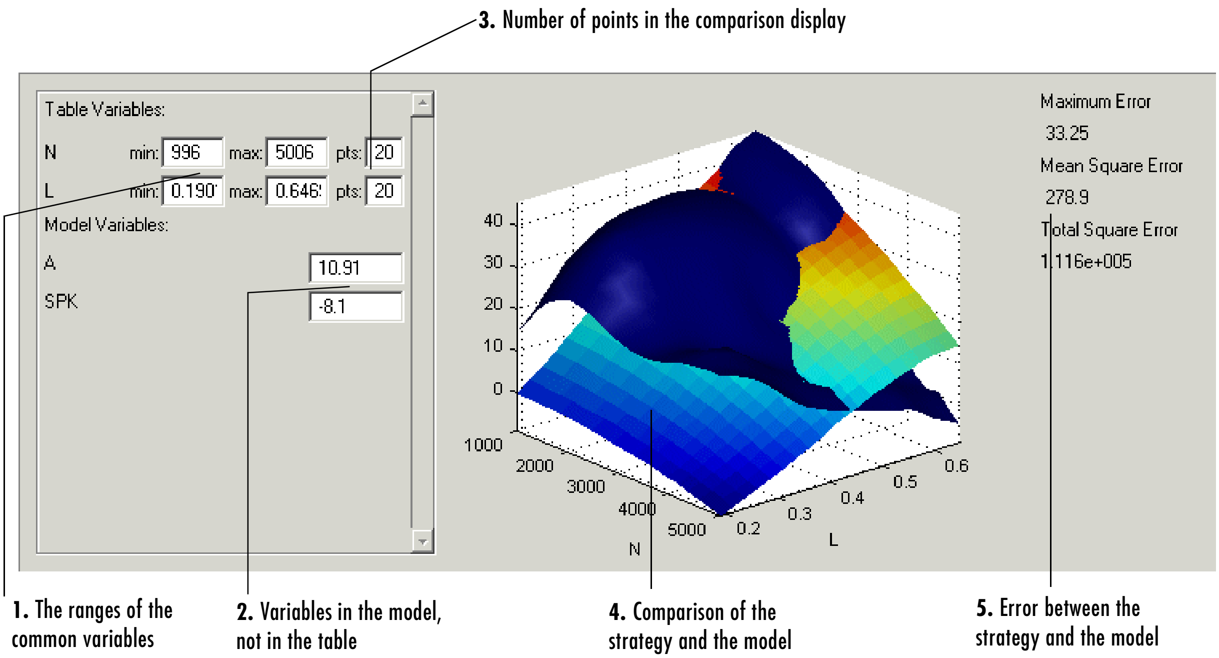


Line indicates which value in the table you are editing.

- You can rotate the graph of the table by clicking and dragging the axes.
- You can alter values in the table by clicking and dragging vertically any point.
- When you click a point, a blue line indicates the selected point in the table.

Comparing the Strategy and the Model

When you calibrate a strategy, or collection of tables, by reference to a model, it is useful to compare the strategy and the model. The following pane illustrates a comparison.



Note This is a comparison between the current strategy values and the model, unlike the comparison pane from the normalizer node, which compares the model and a full factorial grid filled using the breakpoints.

To make full use of the comparison of results pane:

1 Check the ranges of the variables that are common to the model and table.

2 Check the values selected for any variables in the model that are not in the selected table. The default for this is the set point of the variable's range.

3 Check the number of points at which the display is calculated.

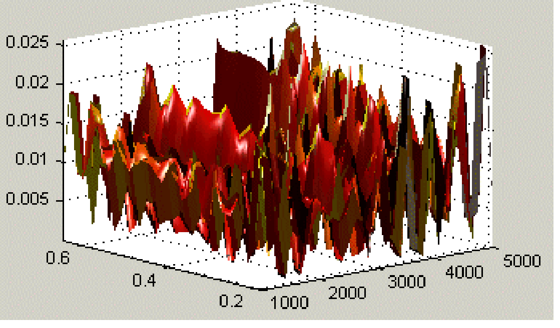
4 Check the comparison between the table and the model. You can rotate this comparison by clicking and dragging, so that you can view all parts of the comparison easily.

5 Check some of the error statistics for the comparison.

You can also view the error over the range of the feature.

Error Display

The comparison-of-results pane can also be used to display the error between the model and the strategy.



To display the error:

1 Right-click the axes of the comparison display.

2 Select **Error** from the menu.

This changes the graph to display the error between the model and the strategy.

You can display the error data in one of the following ways:

- **Error**. This is the difference between the feature and the model.
- **Squared Error**. This is the error squared.

- **Absolute Error**. This is the absolute value of the error.
- **Relative Error (%)**. This is the error as a percentage of the value of the model.
- **Absolute Relative Error (%)**. This the absolute value of the relative error.

To select one of these displays of the error data:

1 Right-click the display.

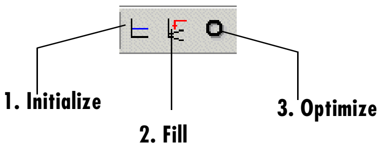
Select **Error Display** and select the appropriate display of the error from the context menu.

Calibrating the Feature Node

Selecting a **Feature** node displays the **Feature** view. For more information about the **Feature** view, see “Feature View” on page 8-45.

The **Feature** view enables you to calibrate the entire feature. Calibrating the feature means that you fill the breakpoints in the normalizer, and the table values, by referring to a model.

Feature Node Toolbar



To calibrate the feature, either click the buttons on the toolbar or select from the following options on the **Feature** menu described in these sections:

1 “Initializing the Feature” on page 8-38

2 “Filling the Feature” on page 8-40

3 “Optimizing the Feature” on page 8-42

Initializing the Feature

For example, a simple feature for maximum brake torque (MBT) consists of the following tables:

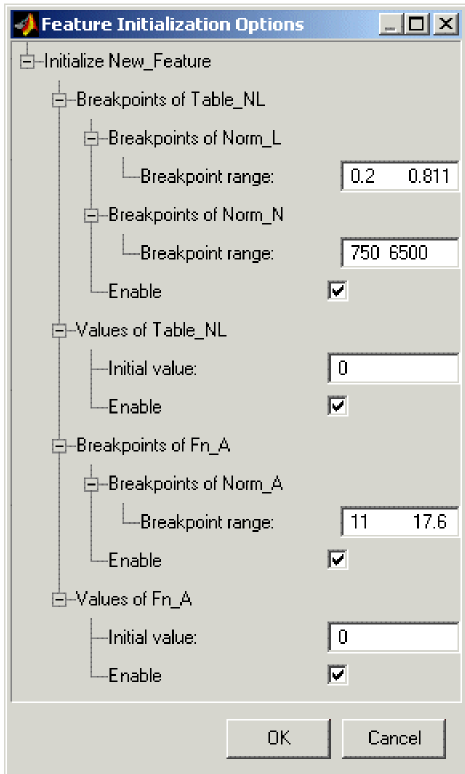
- A speed (N), load (L) table
- A table to account for the behavior of air-fuel ratio (A)

Initializing this feature sets the values of the normalizers for speed, load, and AFR over the range of each variable and put specified values into each cell of the two tables.

A table that is already initialized provides a useful starting point for a more detailed calibration.

To initialize the feature:

1 Click . This opens the **Feature Initialization Options** dialog box, as shown.



2 Enter the ranges for the breakpoints in your normalizers. In the preceding example, enter the following breakpoint ranges:

- **L** has range `0.2 0.811`.
- **N** has range `750 6500`.
- **A** has range `11 17.6`.

3 Enter the initial table value for each cell in each table. Above, enter the cell values as

- **Table_NL** has initial value `0`.
- **Fn_A** has initial value `0`.

4 Click **OK**.

Note The default values in this dialog box are taken from the variable dictionary. If you clear any **Enable** box, the associated table or normalizer is left unchanged.

Filling the Feature

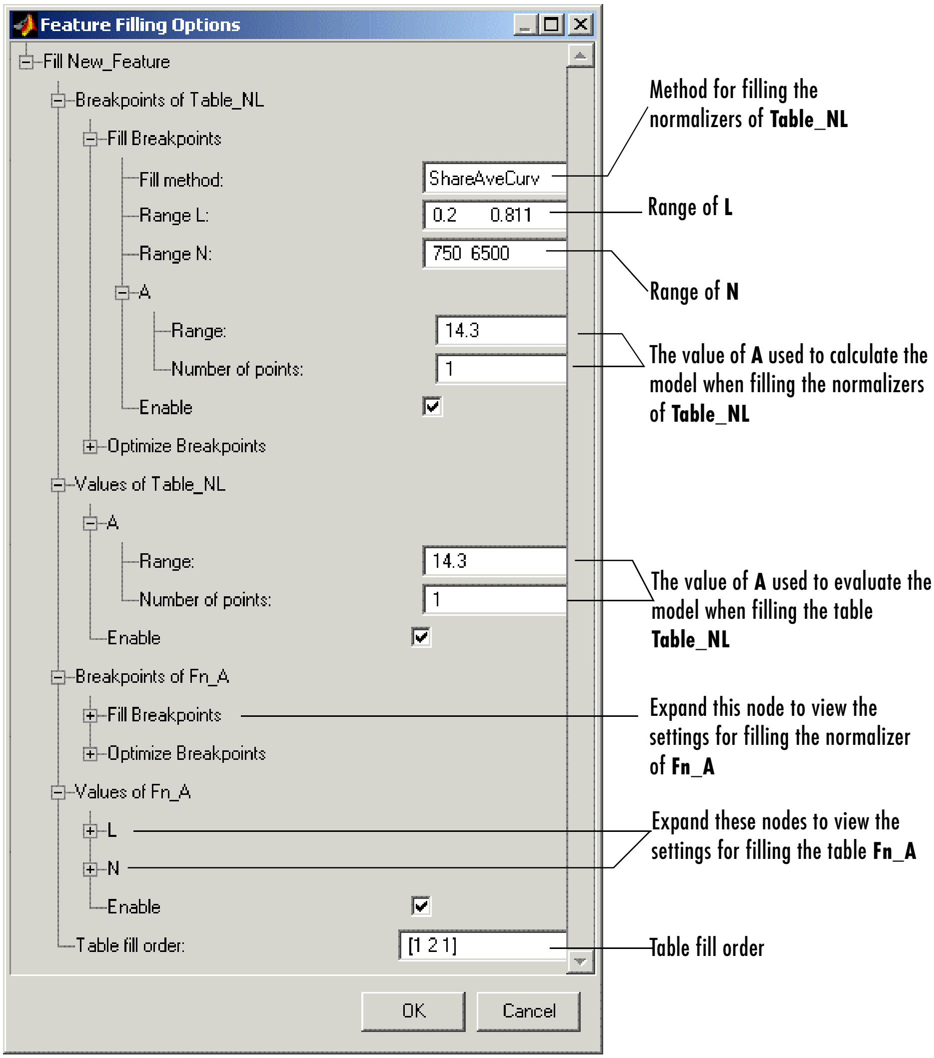
A very quick way to calibrate a feature is to fill it. This does two things:

- CAGE spaces the breakpoints of the normalizers by reference to the model. For example, the breakpoints can be spaced to place most breakpoints where the curvature of the model is greatest. This process is described in detail in “Filling Breakpoints” on page 7-6.
- Then CAGE fills the tables by reference to the model. This process is described in detail in “Filling Table Values” on page 8-14.

This section describes the procedure to fill a feature. For a detailed description about the filling processes, see the sections listed above.

To fill a feature:

1 Click . This opens the **Feature Filling Options** dialog box, shown.



2 Select the correct method for filling your normalizers.

3 Enter the ranges for the normalizers of the tables.

4 Enter the ranges of the other variables when filling the normalizers.

5 Enter the ranges of the other variables when filling the tables.

6 Enter the table fill order.

   In this example, enter `[1 2 1]`. This fills the normalizers and then fills the table values of **Table_NL**. Then CAGE fills the normalizers, and then the table values of table **Fn_A**. After which CAGE then fills the normalizers and then the table values for **Table_NL** again.

7 Click **OK**.

You can iterate this process using the table fill order edit box, as in the example. This further improves the fit of the strategy and the model.

Note Feature fill gives you the choice of optimizing each normalizer before filling the table values. To do this, expand the **Optimize Breakpoints** node and select the **Enable** box. If you clear any **Enable** box, that table is not filled.

When you have completed a calibration, you can export your feature. For information, see “Exporting Calibrations” on page 6-22.

Optimizing the Feature

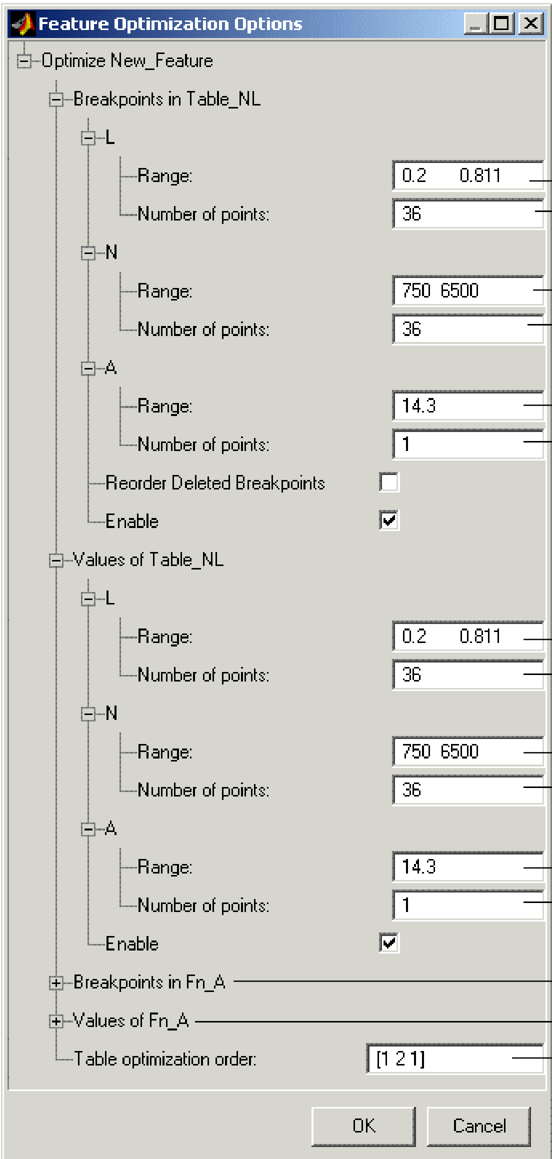
After filling the feature, you can improve the fit of the strategy to the model by optimizing the feature. The optimization routine does the following:

- First CAGE optimizes the breakpoints for the normalizers. (See “Optimizing Breakpoints” on page 7-10.)
- Next CAGE optimizes the values of the tables. (See “Optimizing Table Values” on page 8-17.)

This section gives the procedure for optimizing a feature. For further details of how optimization works, see the references given above.

To optimize a feature:

1 Click . This opens the **Feature Optimizing Options** dialog box, as shown.



Options for Optimizing Normalizers of Table_NL

Range of normalizer **L**

Number of grid points for normalizer **L**

Range of normalizer **N**

Number of grid points for normalizer **N**

The value of **A** used to evaluate the model when optimizing the normalizers of **Table_NL**

Options for Optimizing Table Values of Table_NL

Range and number of grid points for normalizer **L**, used for optimizing **Table_NL**

Range and number of grid points for normalizer **N**, used for optimizing **Table_NL**

The value of **A** used to evaluate the model when optimizing **Table_NL**

Options for Optimizing the Normalizers and Table Fn_A

Expand these nodes to view the settings for optimizing the normalizers and table **Fn_A**.

Table fill order

2 Enter the ranges of the normalizers for the normalizer optimization.

3 Enter the numbers of grid points for the normalizers.

4 Enter the values for the other variables.

5 Enter the ranges of the normalizers for the table optimization.

6 Enter the numbers of grid points for the normalizers.

7 Enter the values of the other variables.

8 Enter the table fill order.

   In this example, enter `[1 2 1]`. This optimizes the normalizers and then the table values of **Table_NL**. Next CAGE optimizes the normalizers, and then the table values of table **Fn_A**. After which CAGE optimizes the normalizers and then the table values for **Table_NL** again.

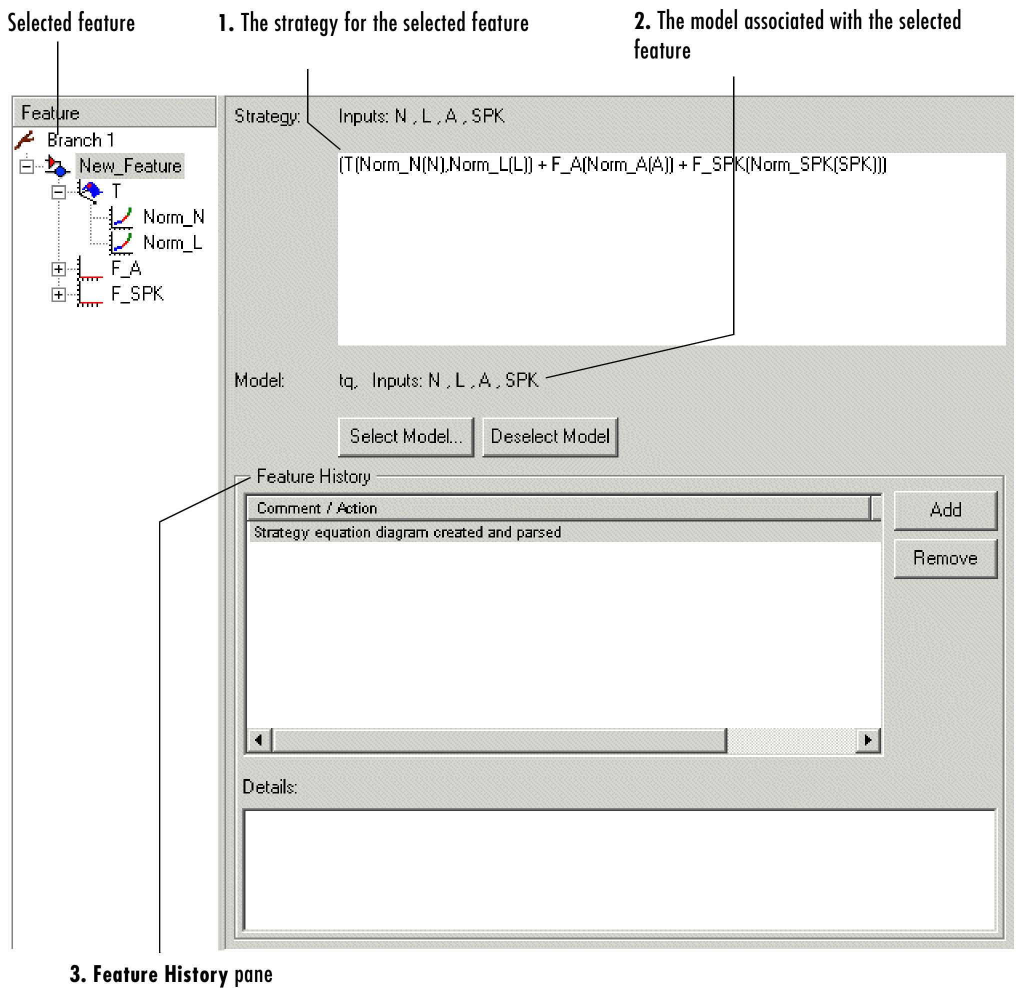
9 Click **OK**.

Note You can iterate this process using the table fill order edit box, as in the example. This further improves the fit of the strategy and the model.

When you have completed a calibration, you can export your feature. For information, see “Exporting Calibrations” on page 6-22.

Feature View

As you select a **Feature** node you see the **Feature** view, shown. This section describes the **Feature** view and the **Feature** menu options.



The parts of the **Feature** view include

1 The strategy for the selected feature. This is the algebraic collection of the tables that you are using to calibrate the selected feature.

2 The model associated with the selected feature.

3 The **Feature History** pane, which displays the history of the feature.

Feature Menu

The **Feature** menu has the following options:

- **Select Model**

  Use this to select the correct model for your feature.
- **Deselect Model**

  Use this to clear the current model from your feature.
- **Convert Feature to a Model**

  Takes the current feature and converts it to a model, which you can view by clicking the **Model** button.
- **Clear Current Strategy**

  Clears the current strategy from your feature.
- **Graphical Strategy Editor**

  Opens your current strategy for editing. For more information, see "Setting Up Your Strategy" on page 8-6.
- **Parse Strategy Diagram**

  Performs the same function as double-clicking the blue outport of your strategy diagram. For more information, see "Setting Up Your Strategy" on page 8-6.

  Enables you to view the feature and the model using the surface viewer. For information, see "Surface Viewer" on page 12-1.
- **Initialize**

  Initializes the feature; also in toolbar. See "Initializing the Feature" on page 8-38 for details.

- **Fill**

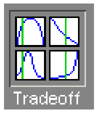
  Fills the feature; also in toolbar. See “Filling the Feature” on page 8-40 for details.

- **Optimize**

  Optimizes the feature; also in toolbar. See “Optimizing the Feature” on page 8-42 for details.

9

Tradeoff Calibrations



A tradeoff calibration is the process of calibrating lookup tables by adjusting the control variables to result in table values that achieve some desired aim.

For example, you might want to set the spark angle and the air-fuel ratio (AFR) to achieve the following objectives:

- Maximize torque
- Restricting CO emissions

The data in tradeoff is presented in such a way as to aid the calibrator in making the correct choices. For example, sometimes the model is such that only a slight reduction in torque results in a dramatic reduction in CO emissions.

This section on tradeoff calibrations includes the following:

- “Performing a Tradeoff Calibration” on page 9-3 for an overview of the tradeoff process
- “Setting Up a Tradeoff Calibration” on page 9-5
- “Calibrating Tables in a Tradeoff Calibration” on page 9-9
- “Using Regions” on page 9-15
- “Multimodel Tradeoffs” on page 9-17

See Also

- “What Is a Tradeoff Calibration?” on page 3-3

  This is a tutorial giving an example of how to complete a tradeoff calibration.

Performing a Tradeoff Calibration

The basic procedure for performing tradeoff calibrations is as follows:

1 Set up the variables and constants. (See “Setting Up Your Variable Items” on page 6-7.)

2 Set up the model or models. (See “Setting Up Your Models” on page 6-14.)

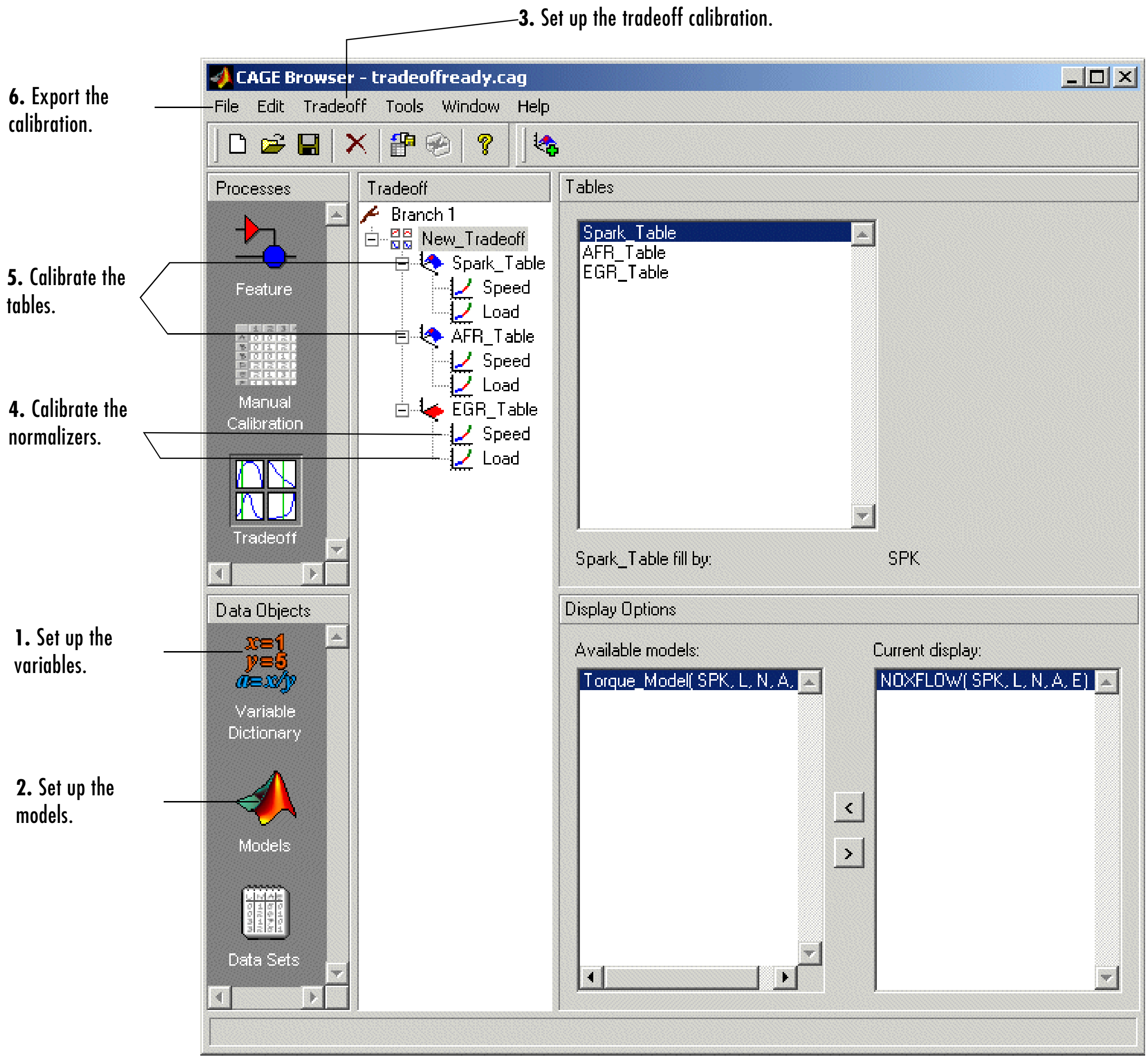
3 Set up the tradeoff calibration. (See “Setting Up a Tradeoff Calibration” on page 9-5.)

4 Calibrate the normalizers. (See “Calibrating the Normalizers” on page 7-4.)

5 Calibrate the tables. (See “Calibrating Tables in a Tradeoff Calibration” on page 9-9.)

6 Export the normalizers, tables, and tradeoffs. (See “Exporting Calibrations” on page 6-22.)

You can also use regions to enhance your calibration. (See “Using Regions” on page 9-15.)



The normalizers, tables, and tradeoff form a hierarchy of nodes, each with its own view and toolbar.

Setting Up 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 instead of the values that produce the maximum value of torque.

A tradeoff also refers to the object that contains the models and tables. Thus, a simple tradeoff can involve balancing the torque output while restricting NOx emissions.

After you set up your variable items and models, you can follow the procedure below to set up your tradeoff calibration:

1. Add a tradeoff. This is described in the next section, "Adding a Tradeoff" on page 9-5.

2. Add tables to the tradeoff. This is described in "Adding Tables to a Tradeoff" on page 9-6.

3. Display the models. This is described in "Displaying Models in Tradeoff" on page 9-7.

This section describes steps 1, 2, and 3 in turn.

When you finish these steps, you are ready to calibrate the normalizers and tables.

Adding a Tradeoff

To add a tradeoff to your session, select **File –> New –> Tradeoff**. This automatically switches you to the **Tradeoff** view and adds an empty tradeoff to your session.

An incomplete tradeoff is a tradeoff that does not contain any tables. If a tradeoff is incomplete, it is displayed as in the branch display. If a tradeoff is complete, it is displayed as in the branch display.

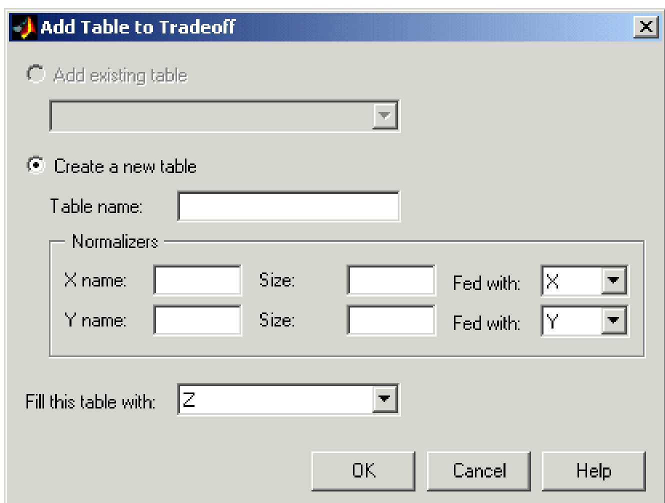
After you add a tradeoff you must add tables to your tradeoff.

Adding Tables to a Tradeoff

1 Add a table by selecting **Tradeoff –> Add Table**.

   Note that you must select the top tradeoff branch in the tree display to use the **Tradeoff** menu. This is automatically selected if your tradeoff has no tables yet (it is the only branch). You must also add at least three variables (in the variable dictionary) to your project before you can add a table, because CAGE needs a variable to fill the table and two more variables to define each of the two normalizers.

   A dialog box opens.



2 Either select a current table from your CAGE session to calibrate, using the top list box, or create a new table. You can add existing tables in the session;

you can choose them from the drop-down list as long as they are not empty (that is, they must be initialized tables that have values and defined sizes).

To create a new table:

a Enter the name for the table.

b Enter names for each of the normalizers.

c Enter sizes for each of the normalizers.

d Select the input for each of the normalizers.

e Select the variable or model you want to fill the table with.

3 Click **OK**.

CAGE adds a table node to the tree.

4 Repeat this procedure for each new table you want to add.

Note Each additional table must have the same normalizers as the first table, so you do not have to perform steps **b**, **c**, and **d** repeatedly.

Displaying Models in Tradeoff

To display models when viewing tables in the tradeoff display:

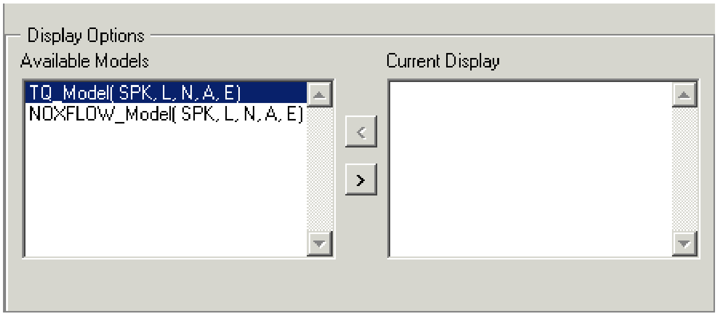
1 Highlight the tradeoff node.

2 From the **Available Models**, select the one you want to display.

Models that are referenced by tables are automatically displayed.

3 Click  to move the selected model into the **Current Display** pane.

4 Repeat steps 2 and 3 until you have displayed all the models that you want to work with.



Deselecting a Model.

1 In the **Current Display**, select the model that you want to remove.

2 Click  to move the selected model into the **Available Models** pane.

3 Repeat until you have cleared all the appropriate models.

Once you have displayed all the models that you want to work with, you are ready to calibrate your normalizers and tables.

Calibrating Tables in a Tradeoff Calibration

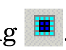
Selecting a table node in the branch tree display enables you to view the models that you have displayed and calibrate that table.

To calibrate the tables:

1. Select the table that you want to calibrate.
2. Highlight one operating point from the table.
3. Set the values for other input variables.

   For information, see “Setting Values of Other Variables” on page 9-11.
4. Determine the value of the desired operating point.

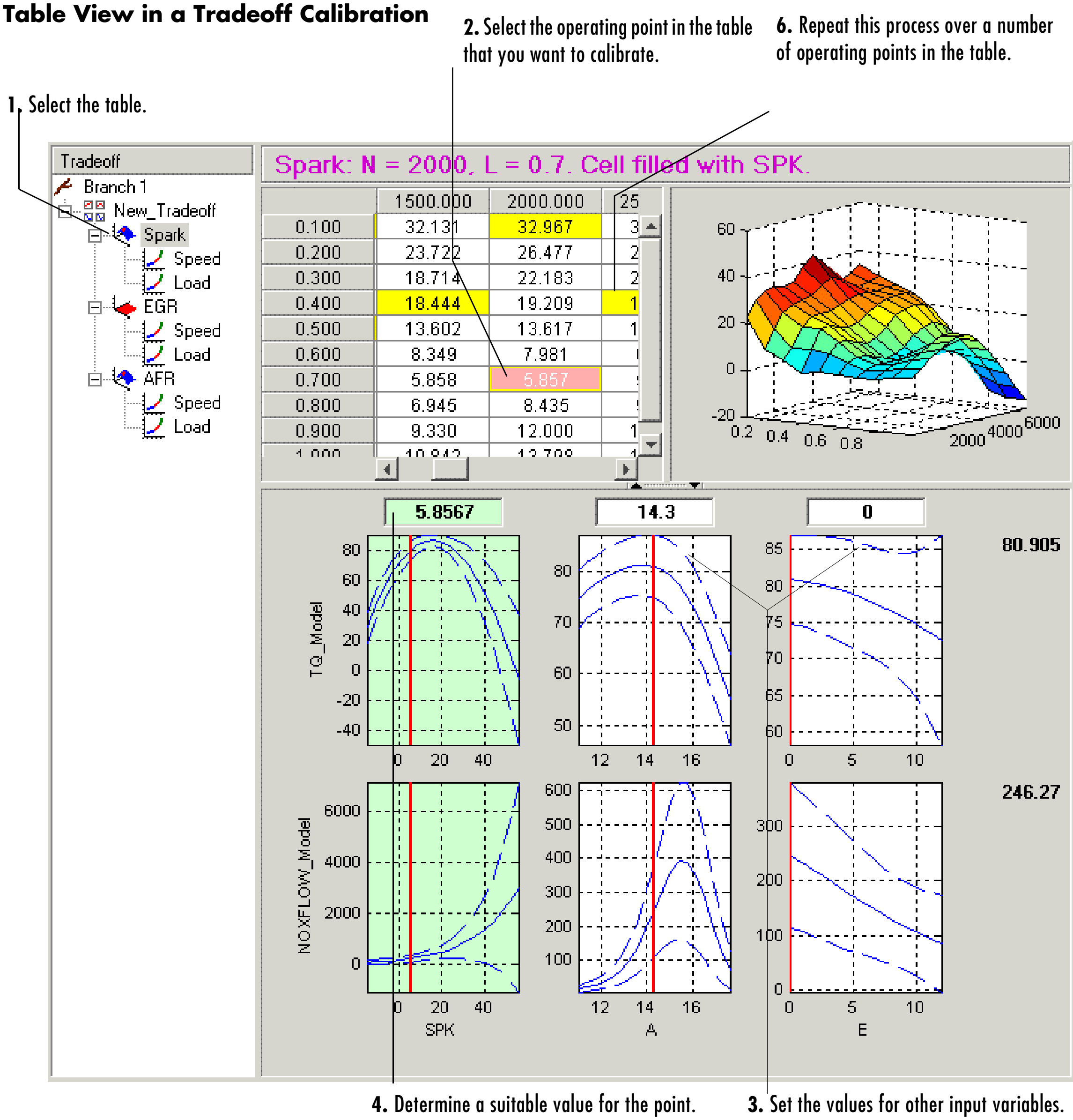
   For information, see “Determining a Value at a Specific Operating Point” on page 9-13.
5. Click  to apply this value to the lookup table.

   This automatically adds the point to the extrapolation mask.
6. Repeat steps 2, 3, 4, and 5 at various operating points.
7. Extrapolate to fill the table by clicking .

   For information, see “Filling the Table by Extrapolation” on page 8-20.

After you complete all these steps you can export your calibration. For information, see “Exporting Calibrations” on page 6-22.

Table View in a Tradeoff Calibration



Notice that the graphs colored green indicate how the highlighted table will be filled:

- If a row of graphs is highlighted, the table is being filled by the indicated model evaluation (the value shown at the right of the row).
- If the column of graphs is green, the table is being filled by the indicated input variable (shown in the edit box above the column).

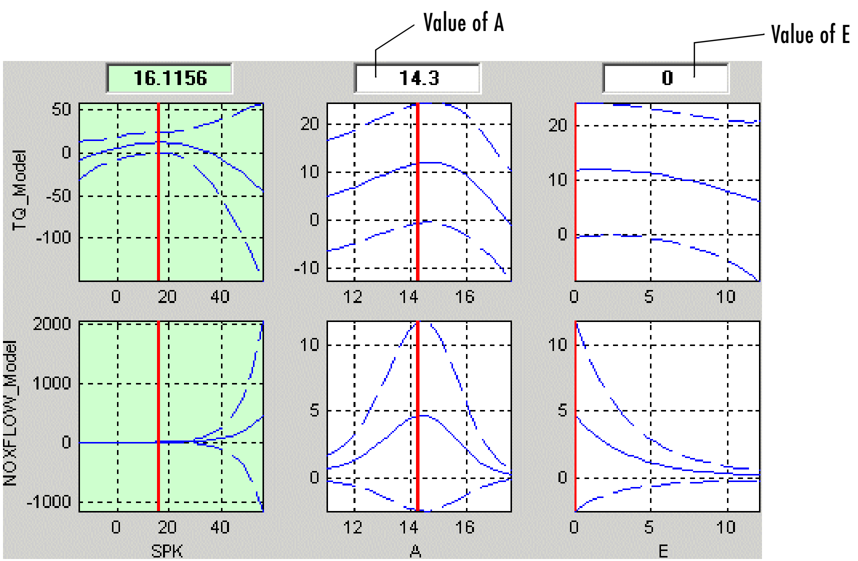
The next sections describe the following in detail:

- "Setting Values of Other Variables" on page 9-11
- "Determining a Value at a Specific Operating Point" on page 9-13

Setting Values of Other Variables

Typically the models that you use to perform a tradeoff calibration have many inputs. When calibrating a table of just one input, you need to set values for the other inputs.

Graphs in Table View



Setting Values for Individual Operating Points

To set values for inputs at individual operating points:

1. Highlight the operating point in the lookup table.

2. Use the edit boxes to specify the values of the other variables.

In the preceding example, the spark table is selected (the SPK graph is colored green). You have to specify the values of AFR (A) and EGR (E) to be used:

1. Select the spark table from the branch menu.

2. Click in the edit box for A and set its value to 14.3.

3. Click in the edit box for E and set its value to 0.

Setting Values for All Operating Points

For example, if you are using a tradeoff to calibrate a table for spark angle, you might want to set the initial values for tables of air-fuel ratio (AFR) and exhaust gas recycling (EGR).

To set constant values for all the operating points of one table:

1. Highlight the table in the branch display.

2. Select one operating point in the table.

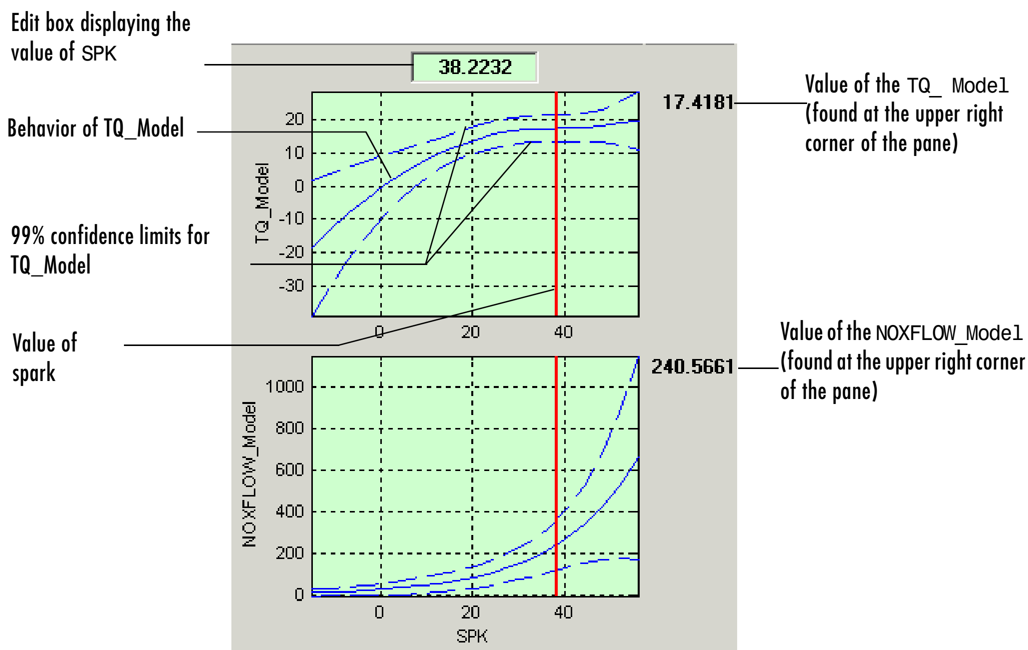
3. Enter the value of the cell.

   This automatically adds this cell to the extrapolation mask.

4. Click  to extrapolate over the entire table.

This fills the table with the value of the one cell.

Determining a Value at a Specific Operating Point



Performing a tradeoff calibration necessarily involves the comparison of two or more models.

For example, in this case, the calibrator uses his or her judgment to select a value of spark that gives acceptable values for both the torque and the NOx flow models.

To select a value of spark, do one of the following:

- Click the edit box as shown above and enter the required value.
- Drag the red line.

Once you determine the value of your variable at this operating point, you apply this value to the table by pressing **Ctrl+T**, by selecting **Table –> Apply Point**, or by clicking  (Add Point) in the toolbar.

Right-Click Menu

Right-clicking a graph enables you to

- Find Nearest Turning Point
- Find Global Maximum
- Find Global Minimum

Using Zoom Controls on the Graphs

To zoom in on a particular region, click with both mouse buttons simultaneously and define the region as a rectangle.

To zoom out to the original graph, double-click the selected graph.

Note Zooming on one graph adjusts other graphs to the same scale.

View Options

Selecting the **View** menu offers you the following options:

- **History**

  This opens the **History** display. For information, see "Using the History Display" on page 13-5.
- **Show/Hide Factors**

  This opens a dialog box that allows you to show or hide factors. This is particularly useful if you are trading off models that have a large number of factors.
- **Show/Hide Models**

  This opens a dialog box that allows you to show or hide models. This is particularly useful if you are trading off a large number of models.
- **Show Predicted Error**

  When you select this, the graphs display the 99% confidence limits for the models.
- **Same Y Limits On Graphs**

  When you select this, all the graphs share a common *y*-axis.

Using Regions

A region is an area that defines locally where to extrapolate before globally extrapolating over the entire table.

For example, consider filling a large table that has twenty breakpoints for each normalizer, by extrapolation. Two problems arise:

- To have meaningful results, you need to set values at a large number of operating points.
- To set values at a large number of operating points takes a long time.

To overcome this problem, you can

1. Define regions within the lookup table.
2. In each region, set the values of some operating points.
3. Click  to fill the table by extrapolation.

Each region is filled by extrapolation in turn. Then the rest of the table is filled by extrapolation. The advantage of using regions is that you can have more meaningful results by setting values for a smaller number of operating points.

Tradeoff Table

	500.000	1000.000	1500.000	2000.000	2500.000
0.100	22.028	29.094	32.197	32.967	32.383
0.200	10.040	18.056	24.116	26.781	27.388
0.300	-0.000	10.517	19.067	22.467	22.763
0.400	0.640	10.256	18.444	19.268	17.475
0.500	1.731	8.761	13.162	13.511	11.641
0.600	0.891	5.396	7.895	7.963	6.423
0.700	0.027	3.368	5.342	5.857	5.548
0.800	0.128	3.337	5.809	7.606	8.958
0.900	0.964	4.447	7.609	10.517	13.283
1.000	1.929	5.630	9.156	12.488	15.546

Cells are colored

- Yellow if they form part of the extrapolation mask
- Blue if they are part of a region

- Gold if they are part of the extrapolation mask and part of a region

Defining a Region

1 Highlight the rectangle of cells in your table.

2 Click  to define the region.

The cells in the region are colored blue.

Clearing a Region

1 Highlight the rectangle of cells in your table.

2 Click  to clear the region.

Multimodel Tradeoffs

There are two types of tradeoff that you can add to your session, a tradeoff of independent models, as described earlier (see “Performing a Tradeoff Calibration” on page 9-3), or a tradeoff of interconnected models (a multimodel tradeoff).

A multimodel tradeoff is a specially built collection of models from the Model Browser.

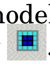
You can build a series of models so that each operating point has a model associated with it. In the Model Browser, you can export models for a multimodel tradeoff from the test plan node. The models must be two-stage and must have exactly two global inputs.

The procedure for calibrating by using a multimodel tradeoff follows:

1 Add the multimodel tradeoff. (See the following section, “Adding a Multimodel Tradeoff” on page 9-18.)

2 Calibrate the tables. (See “Calibrating Using a Multimodel Tradeoff” on page 9-20.)

3 Export your calibration. (See “Exporting Calibrations” on page 6-22.)

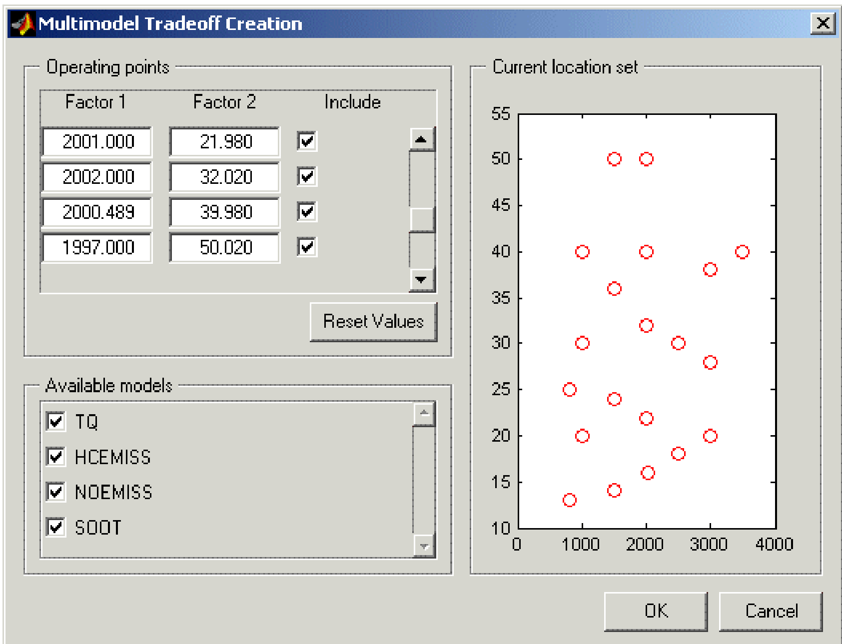
When you calibrate the tables in a multimodel tradeoff, you can only adjust a value in the tables if there is a model defined at this operating point. These cells are colored purple in the table. At each of these operating points, you have a model that you use to trade off, and by doing this you can adjust the value in the table. All other cells in the table do not have models associated with them and you cannot edit them (but they can be filled by extrapolation). You trade off values at each of the model operating points in exactly the same way as when using independent models, as described in “Determining a Value at a Specific Operating Point” on page 9-13. When you have determined table values at each of the model operating points, you can fill the whole table by extrapolation by clicking . See “Filling the Table by Extrapolation” on page 8-20.

Adding a Multimodel Tradeoff

To add a multimodel tradeoff to your session:

1 Select **File –> New –> Multimodel Tradeoff**. The file must have been exported from the MBC Model Browser using the **Tradeoff** button (only enabled for two-stage models with exactly two global inputs).

2 Select the correct file to import and click **Open**. This opens a dialog box.



3 In the **Operating points** pane, you can clear the check boxes for any operating points that you do not want to import.

Notice that the operating points are displayed graphically in the **Current location set** pane. If an operating point is deselected, it is displayed as gray here, rather than red.

CAGE creates tables for all the models and input variables, with breakpoints at all the operating points.

4 You can adjust any of the operating points to reduce the number of breakpoints.

For example, in the session pictured, there are operating points at values of `2001`, `2002`, `2000.489`, and `1997`. This results in breakpoints in the table at each of these four values. However, all four operating values are very close to `2000` and might all have been intended to run at exactly `2000`. You can choose to adjust all these to `2000` by typing in the edit boxes. The table then has a single breakpoint at 2000 instead of the four closely spaced breakpoints. You can click **Reset Values** to return to the original operating points.

5 In the **Available models** pane, clear any model that you do not want to import.

For example, you might want to perform a tradeoff of soot (`SOOT`) and torque (`TQ`) in the preceding example. Clear the check boxes for `HCEMISS` and `NOEMISS`.

6 Click **OK**.

When you import the multimodel tradeoff, you import the tables and normalizers, so you do not have to calibrate the normalizers.

Note When you calibrate the tables, you can only adjust the values of the tables at the operating points defined for the models. These are colored purple in the table.

You can now calibrate your tables. See the next section, "Calibrating Using a Multimodel Tradeoff" on page 9-20.

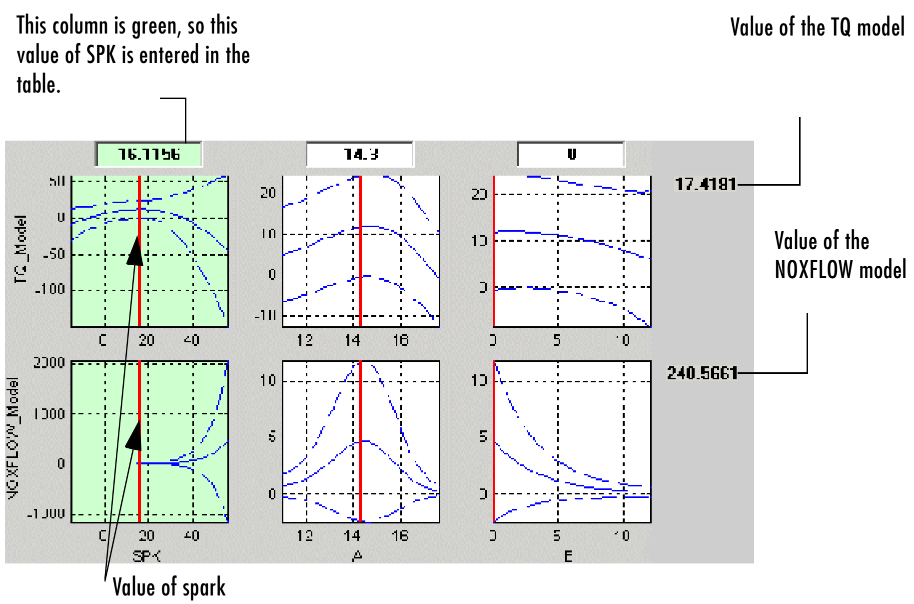
Calibrating Using a Multimodel Tradeoff

Each purple (editable) operating point in your tables has a model associated with it. You use the display of these models to help you trade off values at these points to fulfill your aims in exactly the same way as when using independent models in “ordinary” tradeoff mode, as described in “Determining a Value at a Specific Operating Point” on page 9-13.

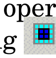
1 Change input values by dragging the red lines on the graphs or by typing directly into the edit boxes above the graphs.

2 Look at the model evaluation values (to the right of each row of graphs) and the input variable values (in the edit boxes above the graphs) to see if they meet your requirements.

Remember that the green highlighted graphs indicate how the selected table is filled: if a row is green, the model evaluation value (to the right) fills the table at that operating point; if a column is green, the input variable value (in the edit box above) fills the table. See the example following; the SPK column of graphs is green, so the value of SPK in the edit box is entered in the table when you click the Add Point button ( ).



3 When you are satisfied with the tradeoff given by the value of your variable at this operating point, you apply this value to the table by pressing **Ctrl+T**, selecting **Table –> Apply Point**, or clicking  in the toolbar.

4 When you have determined table values at each of the model operating points, you can fill the whole table by extrapolation by clicking . See "Filling the Table by Extrapolation" on page 8-20.

You can then export your calibration; see "Exporting Calibrations" on page 6-22.

10

Data Sets

The **Data Set** view has two main functions:

- Validating calibrations with experimental data
- Filling tables by reference to a set of experimental data

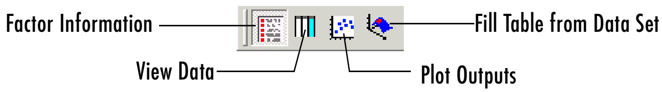
For worked examples about data sets, see

- “Data Sets Tutorial” on page 4-1
  This shows the process of validating a calibration.
- “Filling Tables from Data Tutorial” on page 5-1
  This shows the process of filling a table from experimental data.

This section describes in detail

Data Sets Views

Data Sets consists of four views. These views display different aspects of the data set. Each view is accessible from the **View** menu or by clicking the appropriate button on the toolbar.



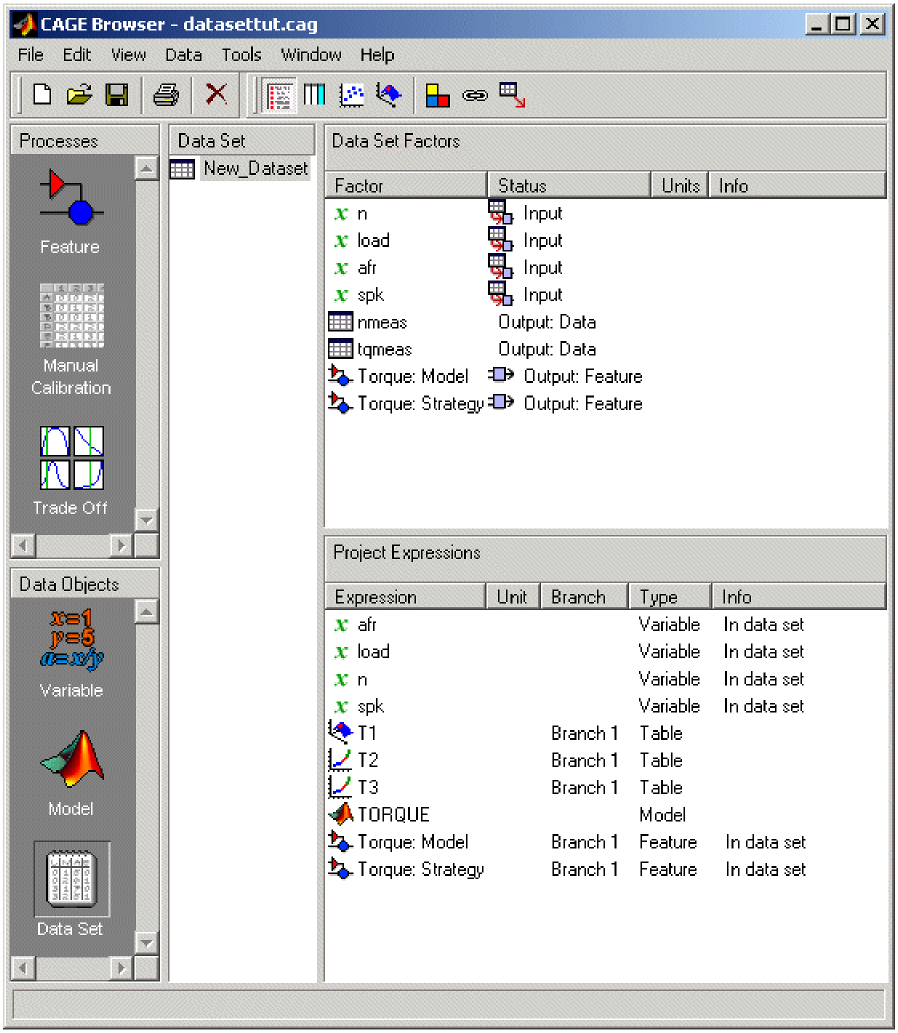
- **Factor Information**

  List of all available project expressions, which can be added to the data set for display and evaluation.
- **View Data**

  Displays the data in a table. Individual entries can be altered. Columns of data can be assigned to CAGE expressions.
- **Plot Outputs**

  Displays models and features evaluated at the data points (of the data set).
- **Fill Table from Data Set**

  This mode allows you to fill tables by reference to experimental data.


CAGE Browser - datasettut.cag
File Edit View Data Tools Window Help
Processes
Feature
Manual Calibration
Trade Off
Data Objects
Variable
Model
Data Set
Data Set
New_Dataset
Data Set Factors
Factor Status Units Info
n Input
load Input
afr Input
spk Input
nmeas Output: Data
tqmeas Output: Data
Torque: Model Output: Feature
Torque: Strategy Output: Feature
Project Expressions
Expression Unit Branch Type Info
afr Variable In data set
load Variable In data set
n Variable In data set
spk Variable In data set
T1 Branch 1 Table
T2 Branch 1 Table
T3 Branch 1 Table
TORQUE Model
Torque: Model Branch 1 Feature In data set
Torque: Strategy Branch 1 Feature In data set

Setting Up Data Sets

The **Data Sets** view displays the strategies, tables, and models, etc., by default as a list of factors in the **Factor Information** view. You can also display the same factors as columns in a grid, with all factors displayed as columns in the list, by selecting the **View Data** toolbar button ( ). The data set works over a grid of values, which is not necessarily the same as the normalizers of any included tables in the data set.

You have to set the input factors and their values to define the grid in the data set. You can do this in one of three ways:

- Import experimental data. (See “Importing Experimental Data” on page 10-5.)
- Import the values from a table in your CAGE session. (See “Importing Data from a Table in Your Session” on page 10-7.)
- Specify the factors and their values manually. (See “Specifying the Factors Manually” on page 10-8.)

The next sections describe each of these in detail.

Importing Experimental Data

You can import experimental data to a data set, either to validate a calibration or to use it as the basis for a calibration.

You can import data that is stored in the following formats:

- Microsoft Excel spreadsheets
- Comma-separated value files
- MAT files

Importing from Excel or Comma-Separated Value. When you import data from either a Microsoft Excel spreadsheet or from a comma-separated value file, you must ensure that the data is organized in the following manner:

- The first column can either be row markers (text) or entries (numbers).
- The first row can either be column headers (text) or entries (numbers).
- All the other row and column entries must be numbers.

Importing From MAT-files. When you import from a MAT-file, you must ensure that the file contains numbers only, that is, a double-array.

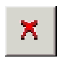
To import experimental data:

1 Select **File –> Import –> Data**.

2 In the file browser, select the correct file to import.

   This opens the **Data Set Import Wizard**.

3 Discard any columns of data you do not want to import by selecting the column and clicking the button shown.

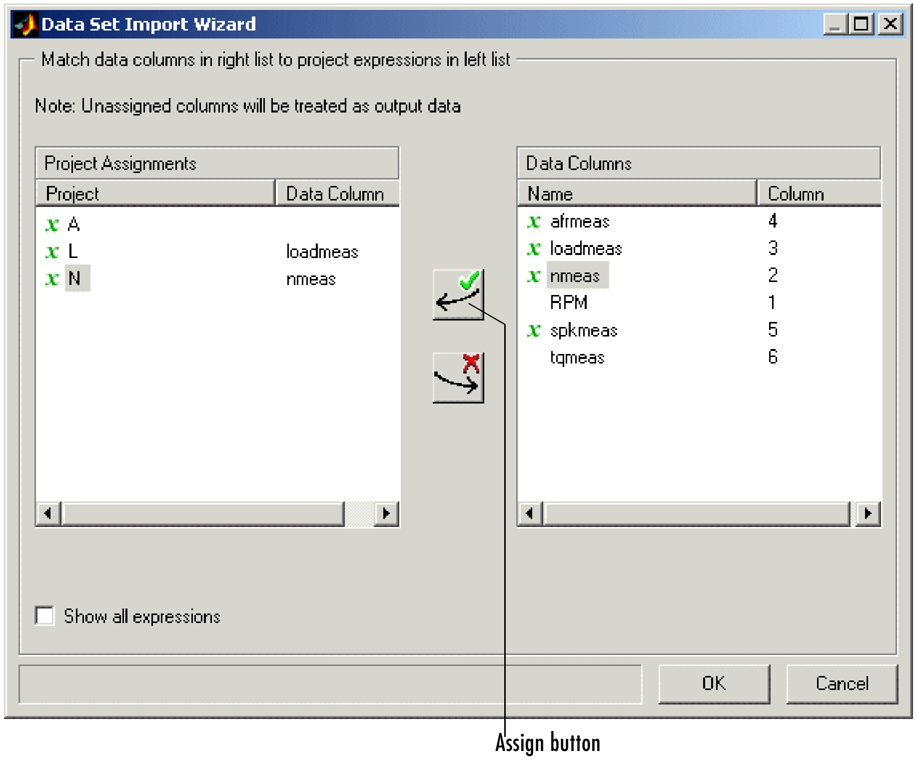
4 Click **Next**.

   The following screen asks you to associate variables in your project with data columns in the data.

5 Highlight the variable in the **Project Assignments** column and the corresponding data column in the **Data Column**, then click the assign button, shown.

6 Repeat step 5 until you are satisfied that you have associated all the variables and data columns. Any unassigned data columns are treated as output factors.



7 Click **Finish** to close the dialog box.

This imports your data into the data set. When you have imported your data, you can view your data set.

Importing Data from a Table in Your Session

To import data from a table:

1 Select **Data –> Import –> Import from Table**.

If your data set is not empty, a dialog box asks whether you want to **Fill** the data set from the table or **Overwrite** the data set from the table. Select **Fill** to use the table values to fill the factors in your data set. Select **Overwrite** to disregard all factors in your data set and fill the data set with the input and output factors from the table. A dialog box opens.

2 Select the correct table from your session to import and click **OK**.

When you have imported your data, you are ready to view the data set.

Specifying the Factors Manually

1 Select the **Data Set** view by clicking the large **Data Sets** button in the **Data Objects** pane.

2 Add a data set to the project by selecting **File –> New –> Data Set**.

3 Select the factors. (See “Selecting the Factors” on page 10-8.)

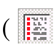
4 Build the grid. (See “Manually Setting Values of the Input Variables” on page 10-10.)

Once you have completed these steps you can view the data set.

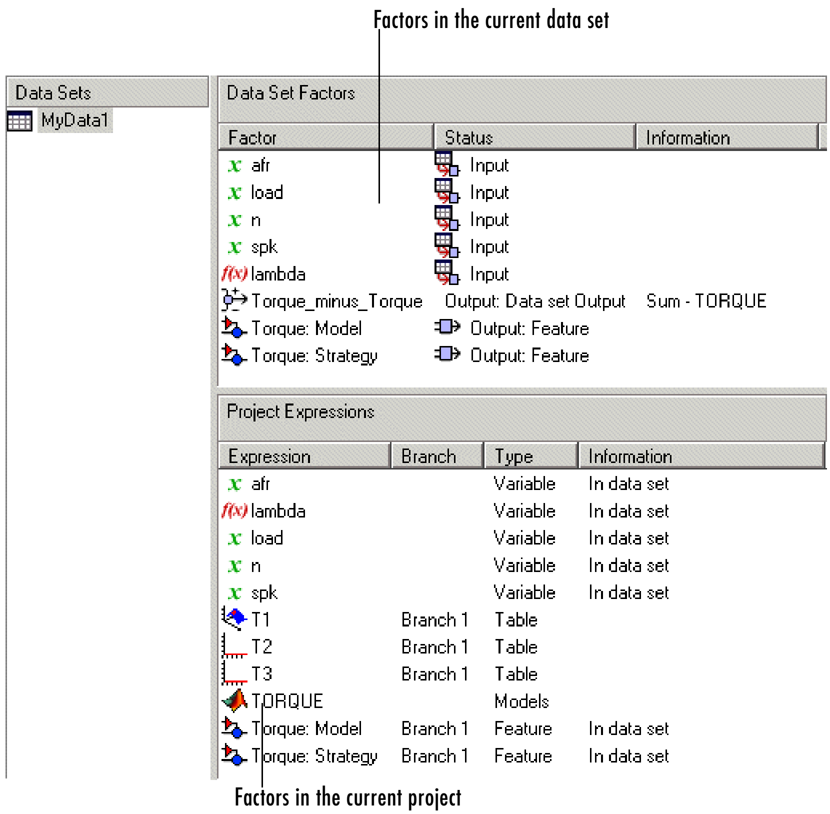
This section describes

- “Selecting the Factors” on page 10-8
- “Manually Setting Values of the Input Variables” on page 10-10

Selecting the Factors

Clicking the **Factors View** button in the toolbar (). This displays two list boxes.

- The upper list shows all factors within the data set. You can sort factors by clicking the column headings.
- The lower list shows CAGE project expressions.



You can use this view to add factors to or remove factors from the data set.

To add a factor to a data set:

1 Select the factor or factors that you want to add to the data set from the list in the lower **Project Expressions** pane.

To make multiple selections, use the standard **Shift+click** or **Ctrl+click**.

2 Select **Data –> Factors –> Add to Data Set**. Alternatively, you can right-click the factor and select **Add to Data Set** from the context menu.

To remove a factor from a data set:

1 Select the factor or factors that you want to remove from the data set.

2 Select **Remove from Data Set** from the right-click menu.

Note Links between the two lists are always preserved, so clicking load in the upper list also selects load in the lower list. In other words, you can copy or remove from either list and the relevant results appear in both. Multiply selecting in one list therefore deselects everything in the other list.

Manually Setting Values of the Input Variables

Clicking the **Build Grid** toolbar button () or selecting **Data –> Build Grid** enables you to set the values of the input variables for the data set.

To build a full factorial grid:

1 Select **Data –> Build Grid**.

2 Select the factor that you want to define a grid for.

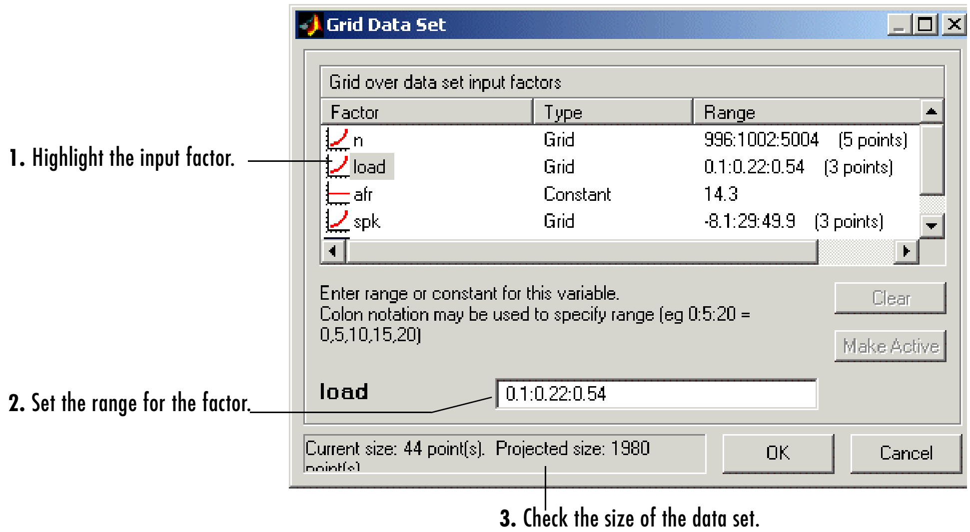
3 Set the grid for the factor.

   To set a grid of 5, 10, 15, 20, 25, 30, input the following: `5:5:30`, where the first number is the minimum, the second is the step size, and the last number is the maximum value.

4 Check the size of the data set in the pane. The current size reported at the bottom of the dialog is the size if you click **Cancel** to leave the data set unchanged. The projected size is created if you click **OK**. In the following example, the projected size of `45` you can see is obtained by multiplying the number of points for each factor with a grid (in this case, 3 * 5 * 3).

5 Select the next factor that you want to define a grid for.

6 When you have set the grids for all the factors, click **OK**.



Creating a Factor from the Error Between Factors

To create a factor that is the difference between two other factors:

1 Highlight the two factors, using **Ctrl+click** or **Shift+click**.

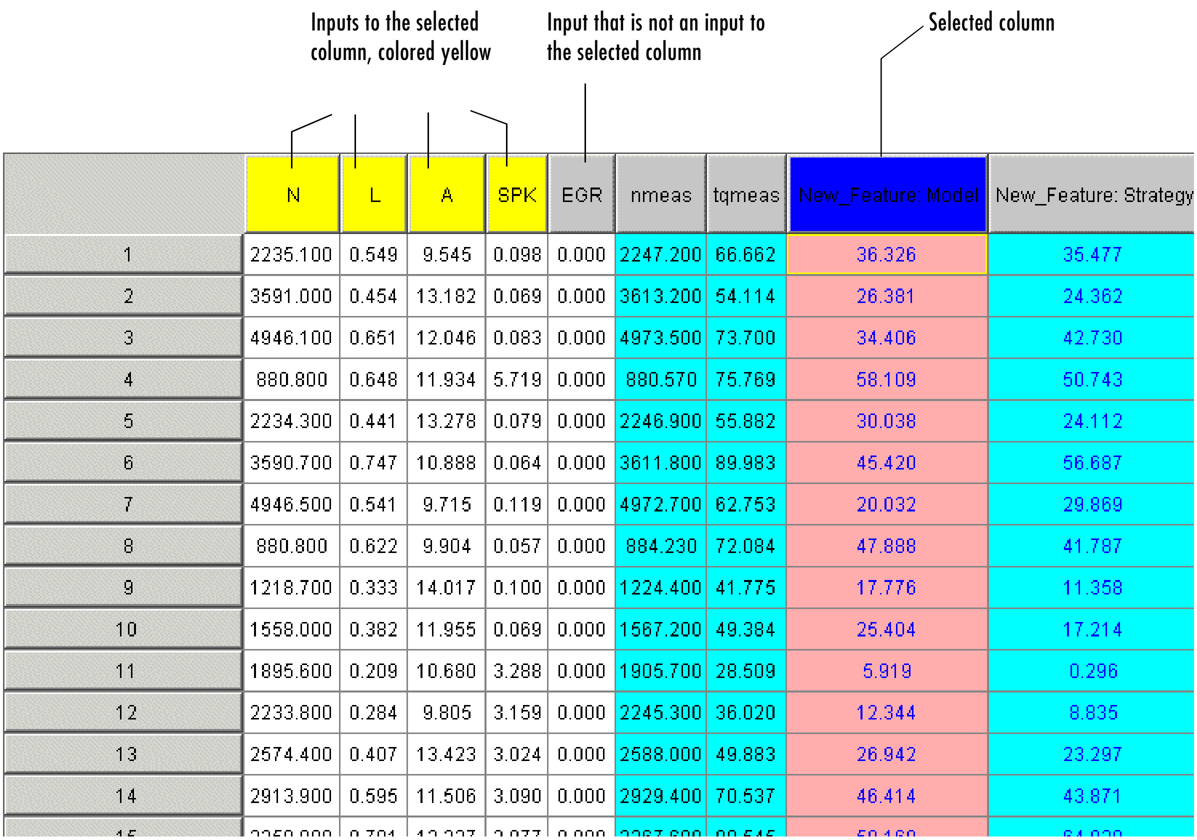
2 Select **Create Error** from the right-click menu on either column head.

This creates a new factor that is the difference between the two other factors.

Viewing Data in a Table

Click the **View Data** button ( ) in the toolbar or select **View –> Data** to display the data in tabular form and a list of the current items in the project.

Note that this view is only enabled if you have a grid of points at which to evaluate and display the models and variables. This grid is not necessarily derived from the normalizers of any tables included in the data set. You can set the grid by importing experimental or table data, or by using the **Build Grid** toolbar button ( ). See “Setting Up Data Sets” on page 10-5.



	N	L	A	SPK	EGR	nmeas	tqmeas	New_Feature: Model	New_Feature: Strategy
1	2235.100	0.549	9.545	0.098	0.000	2247.200	66.662	36.326	35.477
2	3591.000	0.454	13.182	0.069	0.000	3613.200	54.114	26.381	24.362
3	4946.100	0.651	12.046	0.083	0.000	4973.500	73.700	34.406	42.730
4	880.800	0.648	11.934	5.719	0.000	880.570	75.769	58.109	50.743
5	2234.300	0.441	13.278	0.079	0.000	2246.900	55.882	30.038	24.112
6	3590.700	0.747	10.888	0.064	0.000	3611.800	89.983	45.420	56.687
7	4946.500	0.541	9.715	0.119	0.000	4972.700	62.753	20.032	29.869
8	880.800	0.622	9.904	0.057	0.000	884.230	72.084	47.888	41.787
9	1218.700	0.333	14.017	0.100	0.000	1224.400	41.775	17.776	11.358
10	1558.000	0.382	11.955	0.069	0.000	1567.200	49.384	25.404	17.214
11	1895.600	0.209	10.680	3.288	0.000	1905.700	28.509	5.919	0.296
12	2233.800	0.284	9.805	3.159	0.000	2245.300	36.020	12.344	8.835
13	2574.400	0.407	13.423	3.024	0.000	2588.000	49.883	26.942	23.297
14	2913.900	0.595	11.506	3.090	0.000	2929.400	70.537	46.414	43.871
15	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]

Columns are color coded by factor type:

- Input factors are colored white.
- Output factors are colored light blue.

Selecting an output column highlights the input columns associated with it by turning the header cells yellow.

Standard editing facilities are available.

Double-click an input cell to edit the value.

Cut and paste use the desktop clipboard. Cells, columns, and rows can be copied directly to and from other applications (for example, Excel).

Note You can only edit input values, not output values.

Plotting Outputs

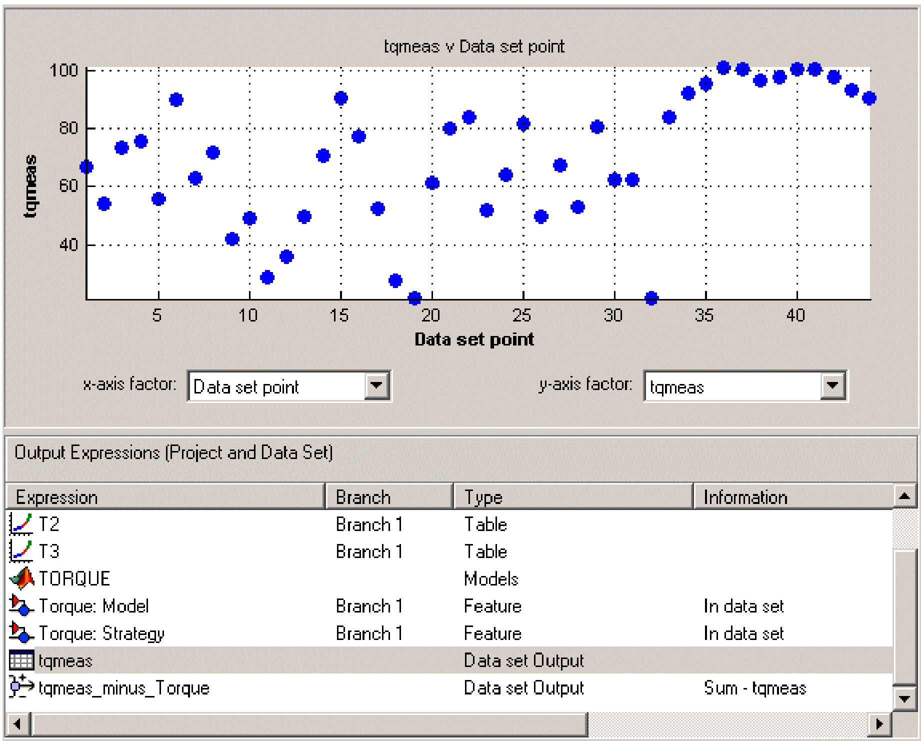
Use this to plot the outputs of your data sets.

To view a plot:

1 Select **View –> Plot** or click the toolbar button.

2 Select an expression from the list to view.

A plot of the selected output factor appears in the top pane.



3 Use the pop-up menus below the plot to change the factors displayed.

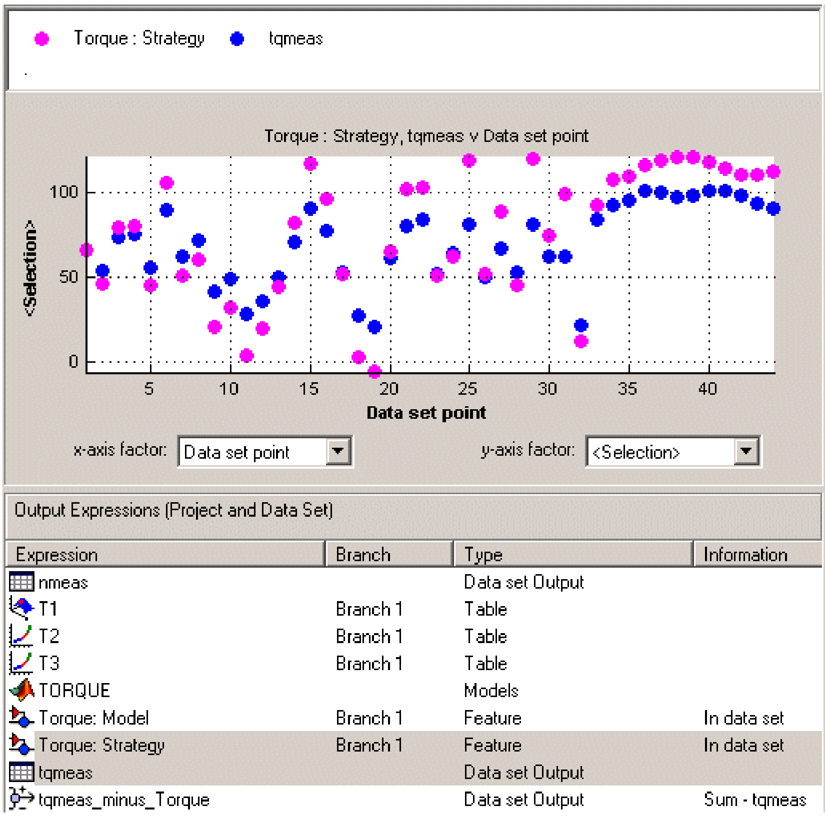
To zoom in on an area of interest:

- Press both mouse buttons simultaneously and drag a rectangle; double-click the graph to return to full size.

Plotting Multiple Selections

You can plot a multiple selection by using standard **Ctrl+click** and **Shift+click** operations.

A legend at the top of the screen displays the key to the graph.



Multiple Plot Outputs

When exactly two items are displayed, further plot options are available:

- Plot the first item against the second item (**X-Y Selection**).
- Display the error using one of the following options:
  - Error
  - Absolute error
  - Relative error (%)
  - Absolute relative error (%)

Using Color to Display Information

You can use the plot view to display more information by coloring the plots.

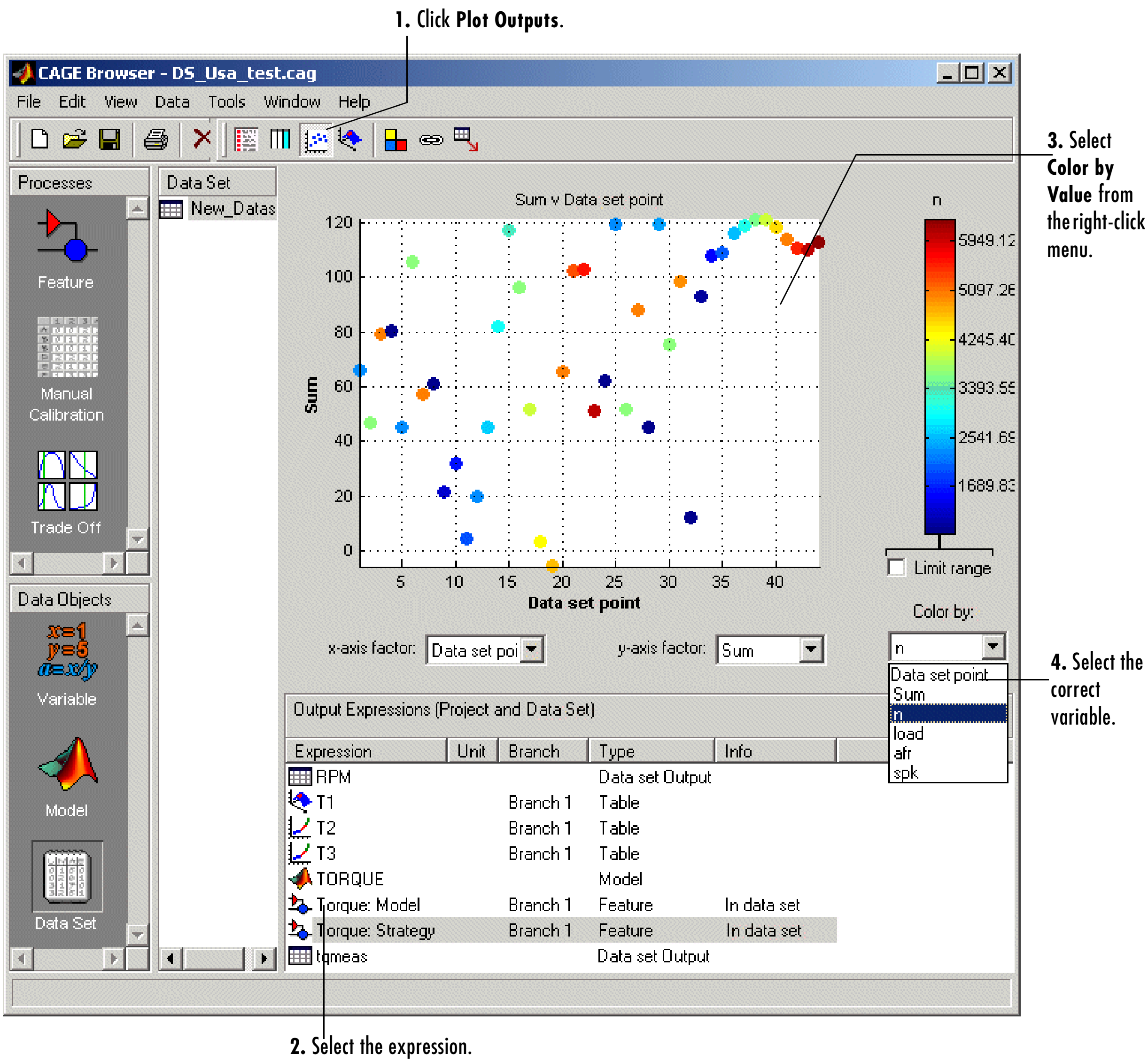
Coloring a Plot

1 Select **View –> Plot** or click .

2 Highlight the correct expression in the **Output Expressions (Project and Data Set)** pane.

3 Select **Color by Value** from the right-click menu of the plot.

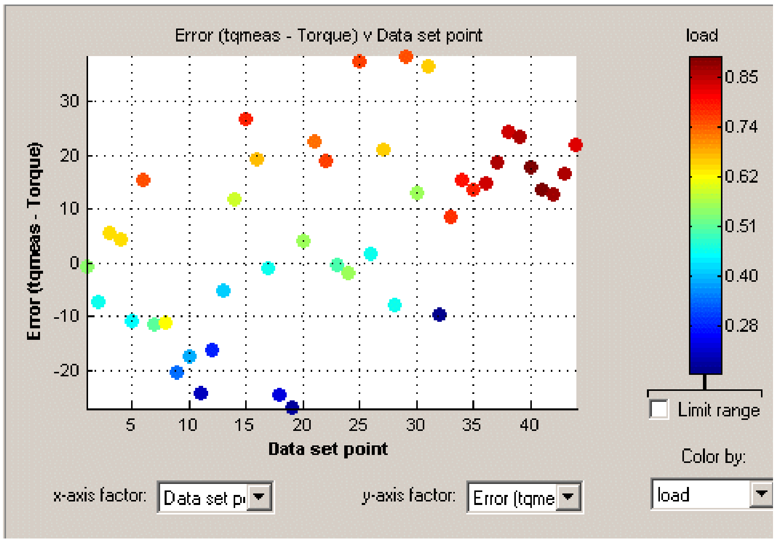
4 Select from the pop-up menu the variable you want to use to color the plot.



1. Click **Plot Outputs**.
CAGE Browser - DS_Usa_test.cag
File Edit View Data Tools Window Help
Processes
Feature
Manual Calibration
Trade Off
Data Objects
Variable
Model
Data Set
Data Set
New_Datas
Sum v Data set point
Sum
Data set point
n
5949.12
5097.26
4245.40
3393.55
2541.69
1689.83
Limit range
Color by:
x-axis factor: Data set poi
y-axis factor: Sum
Data set point
Sum
n
load
afr
spk
Output Expressions (Project and Data Set)
Expression Unit Branch Type Info
RPM Data set Output
T1 Branch 1 Table
T2 Branch 1 Table
T3 Branch 1 Table
TORQUE Model
Torque: Model Branch 1 Feature In data set
Torque: Strategy Branch 1 Feature In data set
tqmeas Data set Output
3. Select **Color by Value** from the right-click menu.
4. Select the correct variable.
2. Select the expression.

In the following figure, you can see

- A plot of the `Sum` vs `Data Set Point` (this is the strategy from a torque feature calibration).
- The points are colored by load.
- For this example it can be seen that in general, the higher the load, the higher the value of torque.



Restricting the Color

You might be interested in only part of the display; for example, you might only be interested in points with a low engine speed. The various display options enable you to color only the points that you are interested in.

To restrict the color:

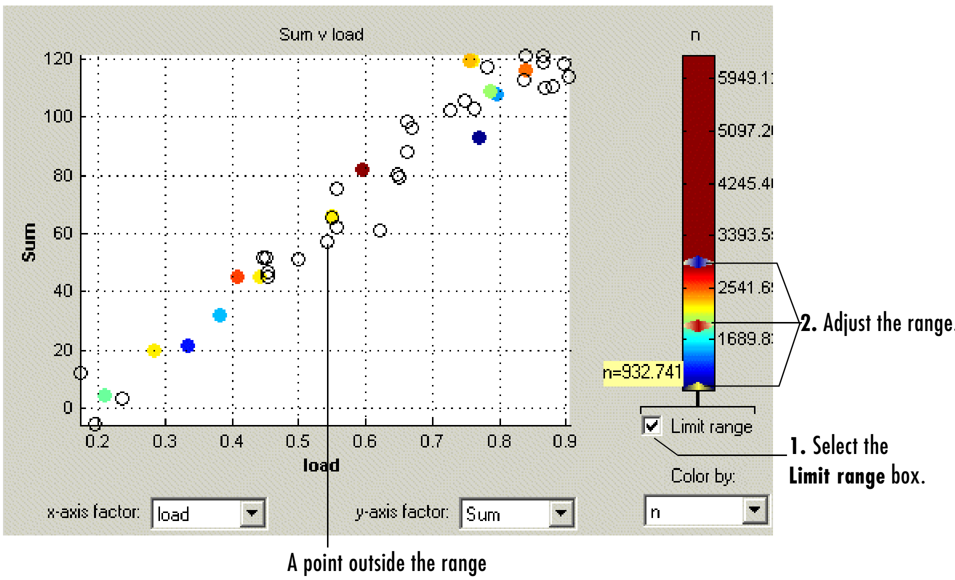
1. Select the **Limit range** box.

2. Adjust the maximum, midpoint, and minimum of the range by dragging the icons on the color bar.

3. Examine the data points and those that are outside the range.

Use the right-click menu to alter the view of the points outside the range:

- Select **Restrict Color to Limits** to compress the colours in the colour bar within the limit markers, as in the example shown. Points outside the limits are still coloured, but only dark red or dark blue, depending on which end of the range they are. This increases the range of colours over the range you are interested in (between the limits), making it easier to see the distribution of points.
- Select **Exclude** to remove all points outside the limits from the display.
- Select **No Color Outside Limits** to display the points as in the example shown. Points outside the limits are plotted as empty circles.
- Select **Color Outside Limits** to display all points in colour, including those outside the limits.



Linking Factors in a Data Set

A factor can be linked to another. The factor then takes on the values of that other factor, overwriting the original values.

For example, you might want to link a variable spark with a model for maximum brake torque (MBT) to evaluate a torque model.

To link two factors:

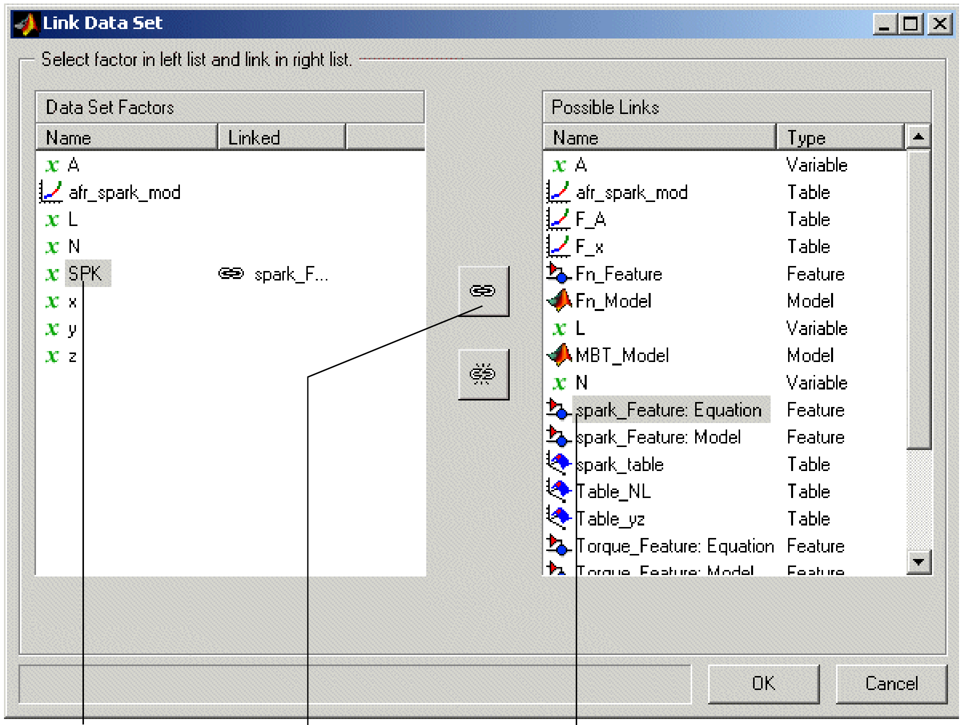
1 Select **Data –> Links**. This opens a dialog box.

2 Select the data set factor that you want to overwrite.

   CAGE generates a list of factors that you could possibly link to the selected factor. (For example, you cannot link to a factor that depends on the selected factor.)

3 Select the factor that you want to link the selected factor with.

4 Click  to link the two factors.



2. Select the factor that you want to overwrite.

4. Click here to link the factors.

3. Select the factor that you want to link it with.

CAGE then overwrites the data set factor with the link.

To break a link and return to normal evaluation, click .

Once all the links have been created or broken as you want, click **OK** to exit the dialog.

See Also

- "Setting Up Data Sets" on page 10-5

Assigning Columns of Data

To analyze imported data, you need to assign columns of data to input factors in the CAGE data set.

Data can be imported into a data set from outside CAGE, for example from an engine test cell. In many cases, this data contains a set of input points (or operating points) and the values of important measurable variables at those points. To compare data like this with models (and/or tables) in a CAGE data set, you have to assign columns of the data to the corresponding input factors in the data set.

To assign data:

1 Select **Data –> Assign**.

2 In the dialog box, highlight the column that you want to assign and the variable that you want to assign it to.

3 Click  to assign.

To unassign data:

1 Select **Data –> Assign**.

2 In the dialog box, highlight the variable that you want to unassign.

3 Click  to unassign.

Note Assigning data to a CAGE expression overwrites that expression in the data set. This does not affect the expression in the other parts of the CAGE project.

Manipulating Models in Data Set View

A model in a data set can be treated as either an input or an output. This is particularly useful when a model is used as an input to another model and you want to view specific values of the input model. For example, linking a model of MBT Spark to a Spark model allows the evaluation of a TQ model at MBT.

To change a model to an input:

1. Highlight the desired model in either the factor view or the table view.
2. Select **Treat as Input** from the right-click menu.

To revert a model to an output:

1. Highlight the desired model in either the factor view or the table view.
2. Select **Treat as Output** from the right-click menu.

Filling Tables from Experimental Data

Any table in the project whose axes (normalizers) exist as factors in the data set can be filled from imported experimental data.

CAGE extrapolates the values of the experimental data over the range of your table. Then it fills the table by selecting the values of the extrapolation at your breakpoints.

To fill the table with values based on the experimental data:

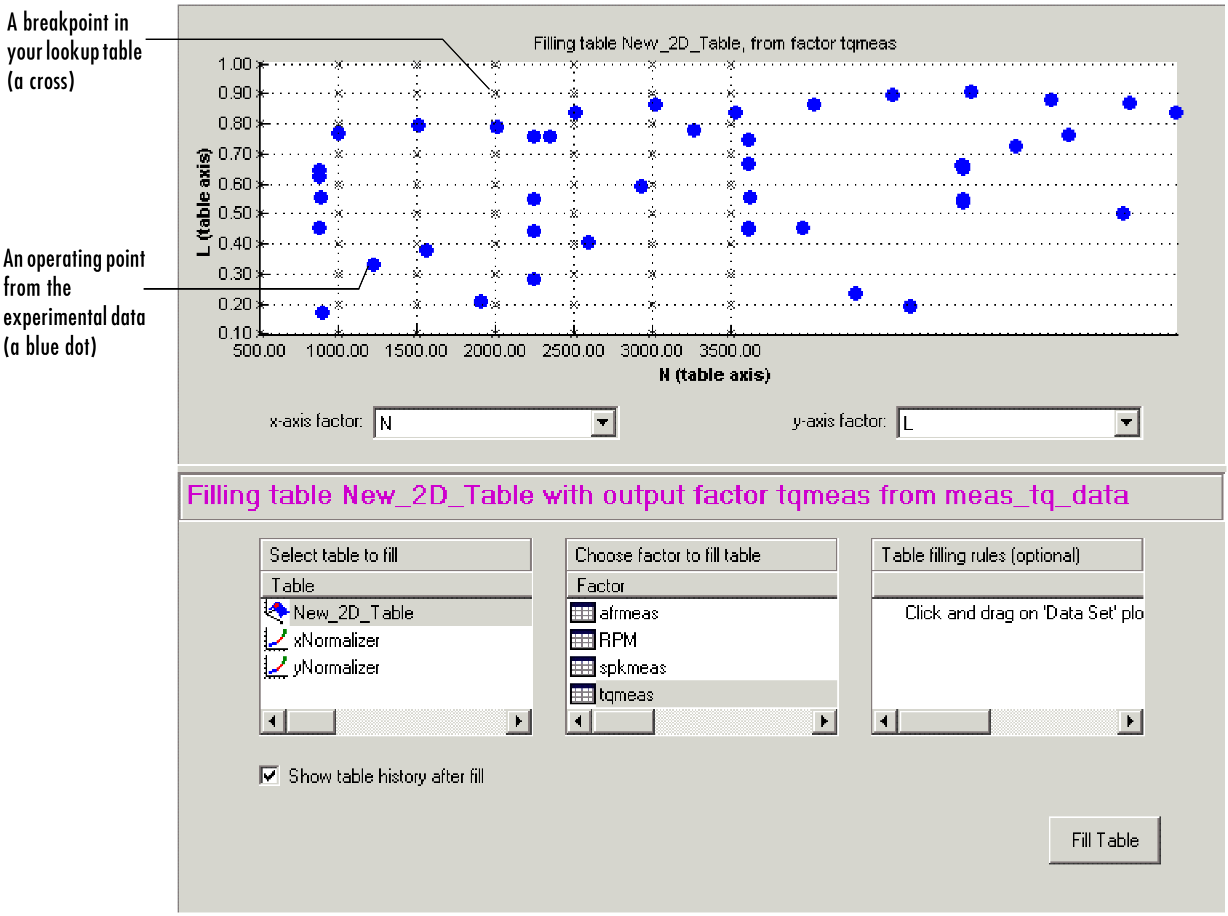
1 To view the **Table Filler** display, click  in the toolbar.

This display asks you to specify the table you want to fill and the factor you want to use to fill it.

2 In the lower pane, select the table from the **Table** list. This is the table that you want to fill.

3 Select the experimental data from the **Factor** list. This is the data that you want to use to fill the table.

For example, see the following display.



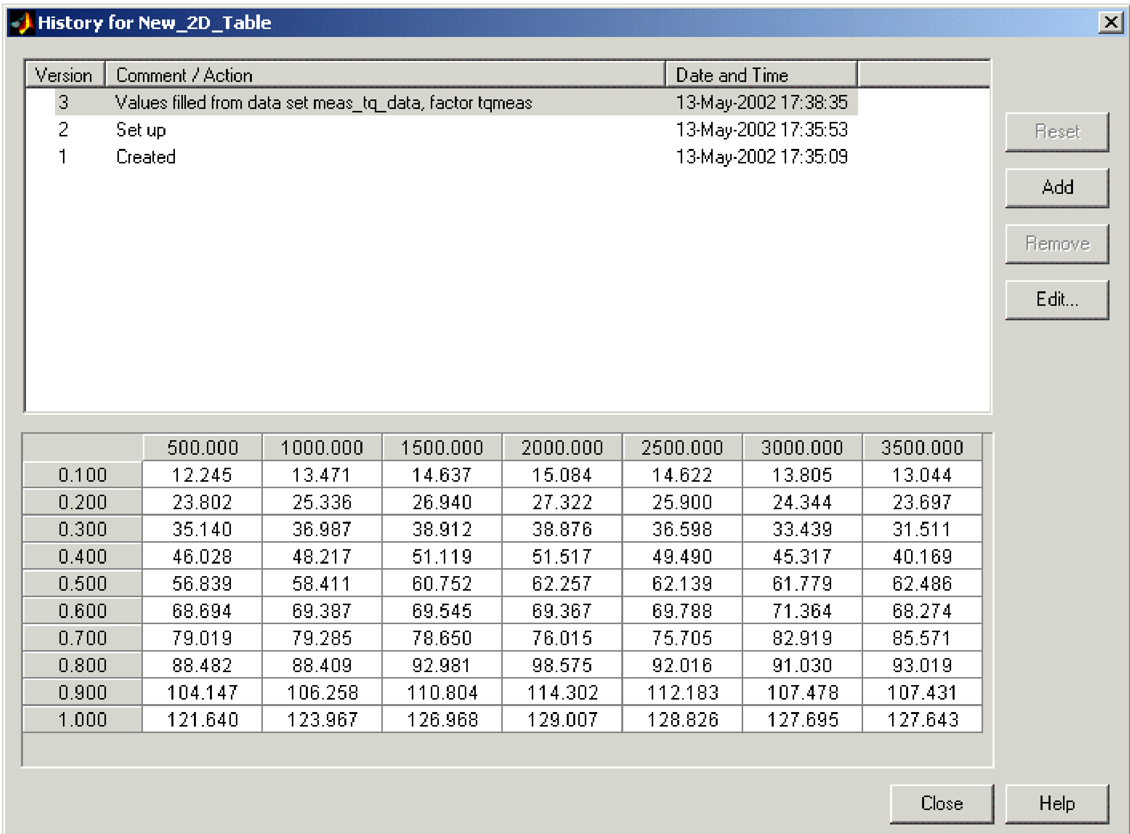
The upper pane displays the breakpoints of your table as crosses, and the operating points where there is data as blue dots.

4 To view the table after it is filled, make sure the **Show table history after fill** box at the bottom left is selected. This is selected by default.

Data sets always display the points in the experimental data, not the values at the breakpoints.

5 To fill the table, click **Fill Table**.

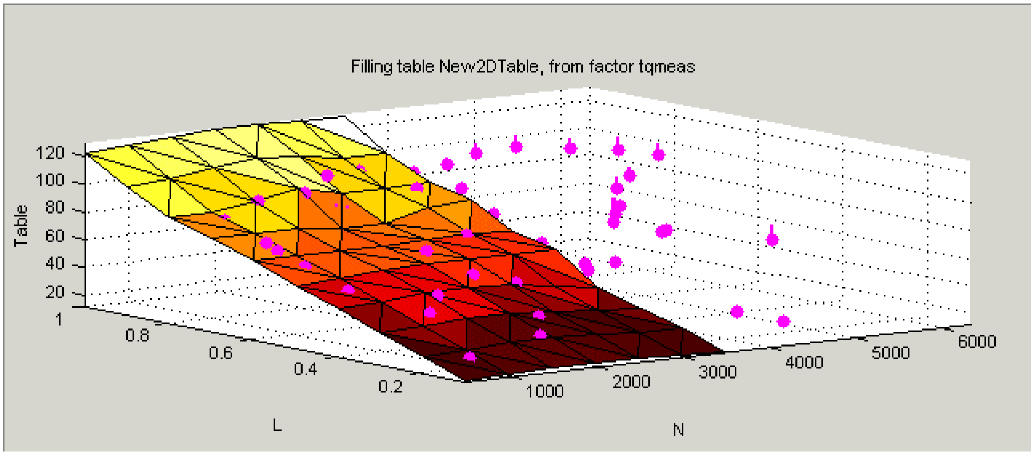
If the **Show table history after fill** box is selected, the **History** dialog box opens, similar to the one shown.



	500.000	1000.000	1500.000	2000.000	2500.000	3000.000	3500.000
0.100	12.245	13.471	14.637	15.084	14.622	13.805	13.044
0.200	23.802	25.336	26.940	27.322	25.900	24.344	23.697
0.300	35.140	36.987	38.912	38.876	36.598	33.439	31.511
0.400	46.028	48.217	51.119	51.517	49.490	45.317	40.169
0.500	56.839	58.411	60.752	62.257	62.139	61.779	62.486
0.600	68.694	69.387	69.545	69.367	69.788	71.364	68.274
0.700	79.019	79.285	78.650	76.015	75.705	82.919	85.571
0.800	88.482	88.409	92.981	98.575	92.016	91.030	93.019
0.900	104.147	106.258	110.804	114.302	112.183	107.478	107.431
1.000	121.640	123.967	126.968	129.007	128.826	127.695	127.643

6 Click **Close** to close the **History** dialog box and return you to the **Table Filler** display.

7 To view the graph of your table, select **Data –> Plot –> Surface**.



This display shows the table filled with the experimental points overlaid as purple dots.

Creating Rules

You can ignore points in the data set when you fill your lookup table.

By defining a region to include or exclude such points, you create a rule for the table filling.

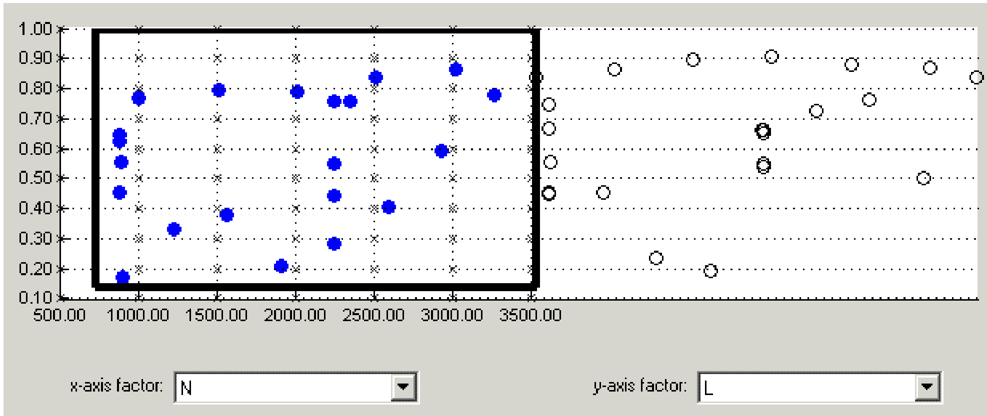
For example, you might want to fill a lookup table that has a range of operating points that is smaller than the range of the experimental data.

To ignore points in the data set:

1 Select **Data –> Plot –> Data Set**. This displays 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. For example, see the following display.

   This region defines a rule in the **Table filling rules** pane.



3 To fill the table based on an extrapolation over these data points only, click **Fill Table**.

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 can now review your data set using the options in the **View** and **Plot** panes of **Data Sets**.

You can add any number of rules to follow when filling tables. For example, you might be aware that a particular test run included in the chosen area is not good data. You can click and drag to enclose any chosen point, then right-click that rule (in the **Table filling rules** pane) and select **Exclude Points**. You can set any number of rules to make sure you fill the table by using just the points you are interested in.

Right-Click Options

Right-clicking the **Table filling rules** pane gives you the following options:

- **Enable Rule**: Apply the rule to the data.
- **Disable Rule**: Do not apply the rule, but also do not delete it.
- **Exclude Points**: Do not include these points in table filling.
- **Include Points**: Include points in table filling.

- **Promote Rule**: Change order of rules.
- **Demote Rule**: Change order of rules.
- **Clear Rule**: Delete this rule.

You can use these options to enable an iterative process. You can fine-tune the selection of data points: try different selections of data to fill your tables, check the results, then reuse the same rules for the same or different tables.

11

Calibration Manager

The **Calibration Manager** dialog box enables you to manage the sizes, values, and precision of all items that can be calibrated. You can set these properties manually or from a calibration file.

This discussion on the Calibration Manager includes these sections:

- “Setting Up Tables” on page 11-3
- “Table Properties” on page 11-7

Setting Up Tables

To set up tables in CAGE, you first open the **Calibration Manager** dialog box. Do this by selecting **Tools –> Calibration Manager** or by clicking  in the toolbar.

You can either set up your tables manually or from a calibration file. You can also copy table data from other sources.

Setting Up Tables Manually

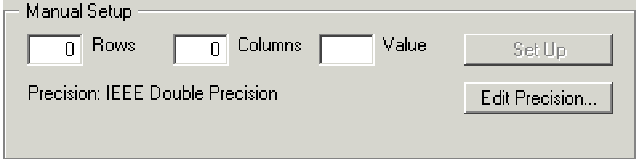
1 Select the normalizer or table to set up from the list on the left.

2 Enter the number of rows and columns in the edit boxes in the **Manual Setup** pane and select initial values for each cell in the table.

Note When initializing tables for a feature calibration (comparing a model to a strategy) you should think about your strategy. CAGE cannot fill those tables if you try to divide by zero. Modifier tables should be initialized with a value of 1 for all cells if they are multipliers, and a value of 0 if they are to be added to other tables. See “How CAGE Fills Tables” on page 8-14 and “Initializing Table Values” on page 8-13.

---

3 Check the display of your table, then click **Close**.



Setting Up Tables Using an Existing Calibration File

1 Open the file by clicking .

This opens the **Import Calibration** dialog box.

2 Select the type of file you want to open (M or Mat file) or select **Automatic**. Click **OK** to open the file browser.

3 Browse to the calibration file, select it and click **Open**.

Note `tutorialcal.mat` is an example calibration file in the mbctraining folder.

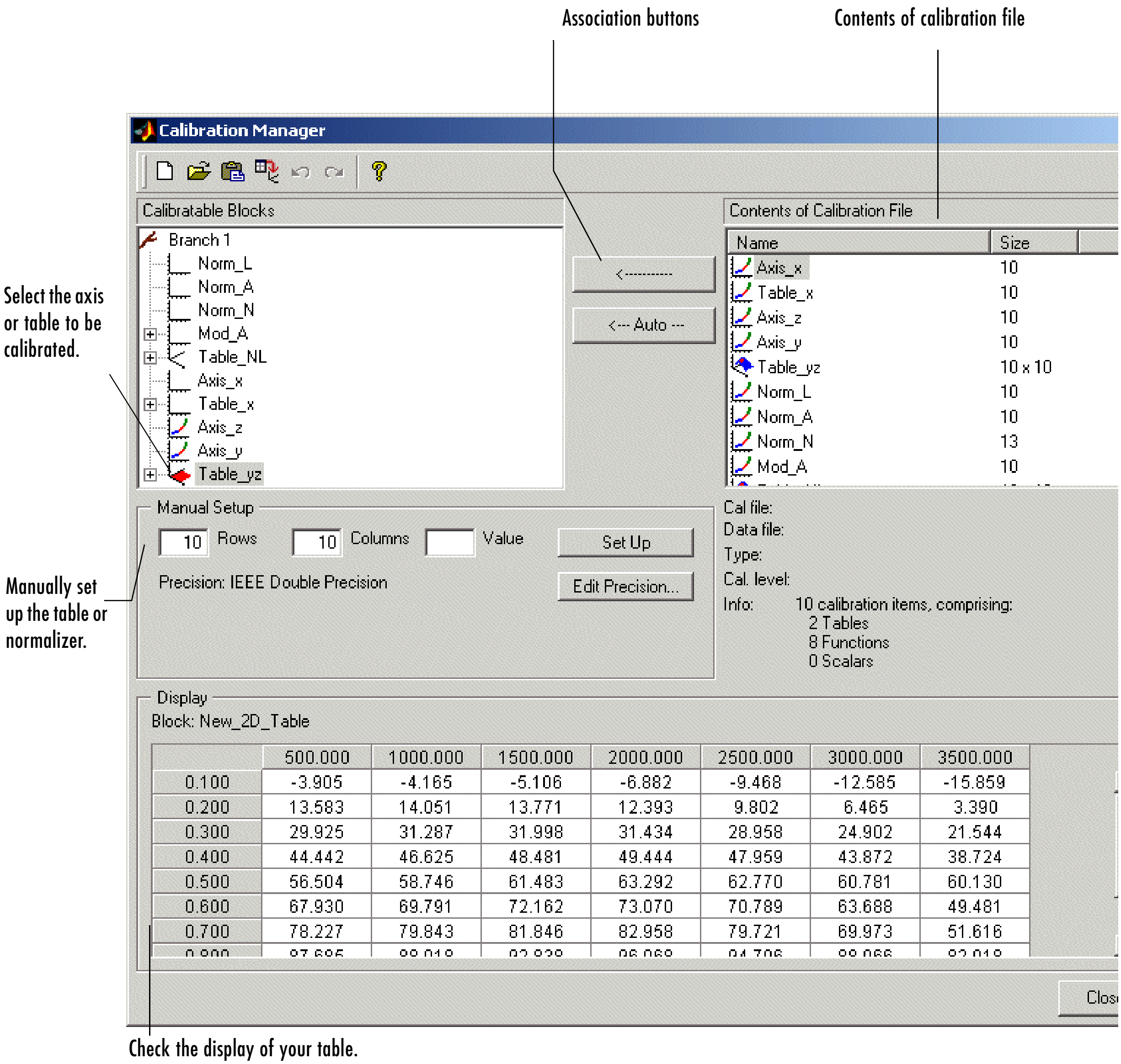
---

4 Highlight both the table in the **Contents of Calibration File** pane and the table in the **Calibratable Blocks** pane that you want to associate with it.

5 Associate these two files by clicking <----------.

   To associate all the items listed in the **Calibratable Blocks** pane with items having the same names listed in the **Contents of Calibration File** pane, click <--- Auto ---.

6 Check the display of your table, then click **Close**.


Association buttons
Contents of calibration file
Select the axis or table to be calibrated.
Manually set up the table or normalizer.
Check the display of your table.
Calibration Manager
Calibratable Blocks
Branch 1
Norm_L
Norm_A
Norm_N
Mod_A
Table_NL
Axis_x
Table_x
Axis_z
Axis_y
Table_yz
<-----------
<--- Auto ---
Contents of Calibration File
Name
Size
Axis_x 10
Table_x 10
Axis_z 10
Axis_y 10
Table_yz 10 x 10
Norm_L 10
Norm_A 10
Norm_N 13
Mod_A 10
Manual Setup
10 Rows
10 Columns
Value
Set Up
Precision: IEEE Double Precision
Edit Precision...
Cal file:
Data file:
Type:
Cal. level:
Info: 10 calibration items, comprising:
2 Tables
8 Functions
0 Scalars
Display
Block: New_2D_Table
500.000 1000.000 1500.000 2000.000 2500.000 3000.000 3500.000
0.100 -3.905 -4.165 -5.106 -6.882 -9.468 -12.585 -15.859
0.200 13.583 14.051 13.771 12.393 9.802 6.465 3.390
0.300 29.925 31.287 31.998 31.434 28.958 24.902 21.544
0.400 44.442 46.625 48.481 49.444 47.959 43.872 38.724
0.500 56.504 58.746 61.483 63.292 62.770 60.781 60.130
0.600 67.930 69.791 72.162 73.070 70.789 63.688 49.481
0.700 78.227 79.843 81.846 82.958 79.721 69.973 51.616
Clos

Note You can add additional file formats to configure CAGE to work with your processes.

Contact The MathWorks for details about adding file formats at `www.mathworks.com/products/mbc/`.

Finding Items in the Calibration File

In a large calibration file, you might want to search for an item by name. To search the **Contents of Calibration File** pane:

1. Click the **Contents of Calibration File** list.
2. Type the first few letters of the item that you are searching for.
3. The cursor moves to the letters specified as you type.

Copying Table Data from Other Sources

You can paste table values from Excel, for example, by copying the array in Excel and clicking **Paste** :

1. Open the desired Excel file and copy the array that you want to import.
2. In the **Calibration Manager** dialog box, click **Paste** .

You can also set up a table from a text file:

1. Click  in the toolbar.
2. Select the desired file, then click **Open**.

Note If the size of the table is different from the file that you are copying, CAGE changes the size of the table in the session.

Table Properties

Table properties allow you to edit the precision of selected tables and normalizers according to the way tables are implemented in the electronic control unit (ECU). The ECU designer chooses the type of precision for each element to make best use of available memory or processor power.

To edit the precision of a table or normalizer:

1. Select **Edit Precision** in the **Calibration Manager** dialog box.

   Alternatively, if you highlight a table in a feature calibration, display the table properties by selecting **Table –> Properties**.

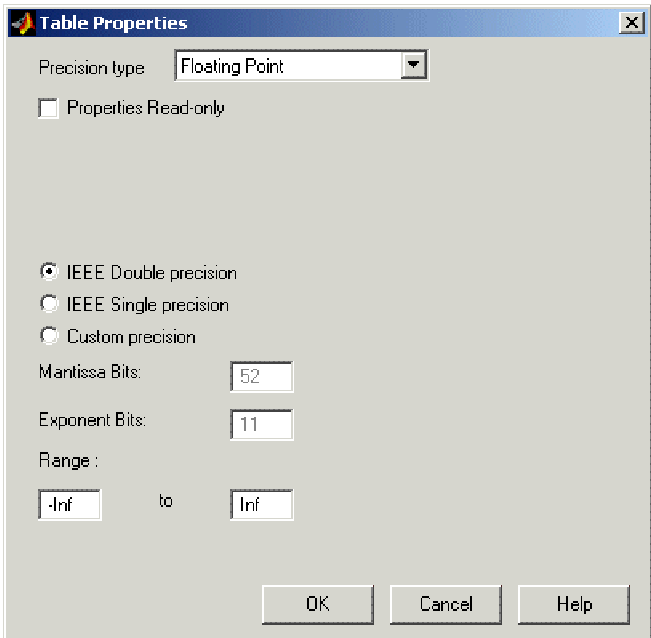
2. Decide whether you want the precision to be writable, then either select or clear the **Properties Read-only** check box.

3. Decide the type of precision you require for the table:
   - **Floating Point** (See “Floating-Point Precision” on page 11-7.)
   - **Polynomial Ratio, Fixed Point** (See “Polynomial Ratio, Fixed Point” on page 11-9.)
   - **Lookup Table, Fixed Point** (See “Lookup Table, Fixed Point” on page 11-11.)

The following sections describe these types of precision in detail.

Floating-Point Precision

The advantage of using floating-point precision is the large range of numbers that you can use, but that makes the computation harder.



There are three types of floating-point precision that you can choose from:

- **IEEE double precision (64 bit)**
- **IEEE single precision (32 bit)**
- **Custom precision**

If you choose **Custom precision**, you must specify the following:

- Number of mantissa bits
- Number of exponent bits
- Range of values restricting the values in the table

When you are done, click **OK**.

See Also

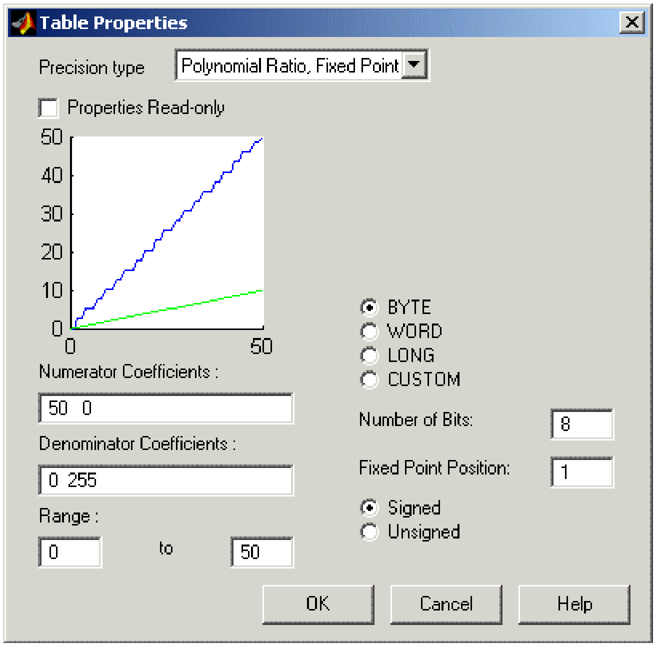
- For more information on IEEE double precision in MATLAB®, see Moler, C., "Floating points," *The Mathworks Company Newsletter*, 1996.

Polynomial Ratio, Fixed Point

The advantage of using fixed-point precision is the reduction in computation needed for such numbers. However, it restricts the numbers available to the user.

For example, the polynomial ratio is of the form (see the ratio shown)

$$y = \frac{50x + 0}{0 + 255}$$



To edit the polynomial ratio:

1 Select the **Numerator Coefficients** edit box and enter the coefficients. In the preceding example, enter `50 0`.

The number of coefficients determines the order of the polynomial, and the coefficients are ordered from greatest to least.

2 Select the **Denominator Coefficients** edit box and enter the coefficients. In the preceding example, enter `0 255`.

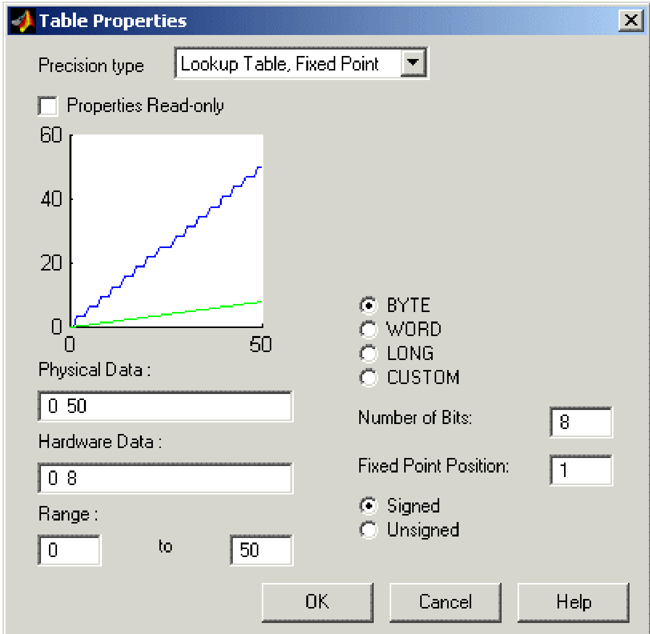
3 Determine the range of values that you want to have in the table. In the preceding example, enter `0 50`.

To edit the size of the precision, choose from

- **Byte (8 bits)**
- **Word (16 bits)**
- **Long (32 bits)**
- **Custom**

Next, determine whether you want the numbers to be signed (negative and positive) or unsigned (nonnegative).

Lookup Table, Fixed Point



The advantage of using fixed-point precision is the reduction in computation needed for such numbers. However, it restricts the numbers available to the user.

For example, consider using a lookup table for the physical quantity *spark advance for maximum brake torque (MBT spark)*. Typically, the range of values of MBT spark is 0 to 50 degrees. This is the physical data. The ECU can only store bytes of information and you want to restrict the hardware store to a range of 0 to 8, with at most one decimal place stored.

To adjust the fixed-point precision of the lookup table:

1 Select the **Physical Data** edit box and enter the range of the physical data. In the preceding example, enter `0 50`.

2 Select the **Hardware Data** and enter the range to store. In the preceding example, enter `0 8`.

3 Determine the range of values that you want to have in the table. In the preceding example, enter `0 50`.

To edit the size of the precision, choose from

- **Byte (8 bits)**
- **Word (16 bits)**
- **Long (32 bits)**
- **Custom**

In the preceding example, the hardware is restricted to 8 bytes and to one decimal place.

4 Choose whether you want the numbers to be signed (negative and positive) or unsigned (nonnegative) by clicking the radio buttons.

12

Surface Viewer

The **Surface Viewer** enables you to view the model or the feature as it varies over the ranges of its variables. You can automatically step through values of a variable, to make a movie of the behavior of the feature or model.

You can view the model or feature using a variety of plot types.

Note The **Surface Viewer** is only available when you are viewing models or the feature node of a feature calibration.

This section includes the following topics:

- "Viewing a Model" on page 12-3
- "Setting Variable Ranges" on page 12-5
- "Displaying the Model or Feature" on page 12-7
- "Displaying Errors" on page 12-10
- "Making Movies" on page 12-12
- "Printing and Exporting the Display" on page 12-13

Viewing a Model

To access the surface viewer, select **Tools –> Surface Viewer** or click on the toolbar.

As the surface viewer opens, select the correct model or feature from the project, by highlighting it in the lower list.

These are the main steps to view the model or feature using the **Surface Viewer** dialog box:

1. Select the range for the variables. (See “Setting Variable Ranges” on page 12-5.)

2. Display the model or feature in the correct format. (See “Displaying the Model or Feature” on page 12-7.)

   For example, as you view a feature, you can view either the strategy or the model associated with that feature, or both. Alternatively, you can display the error between the model and the strategy.

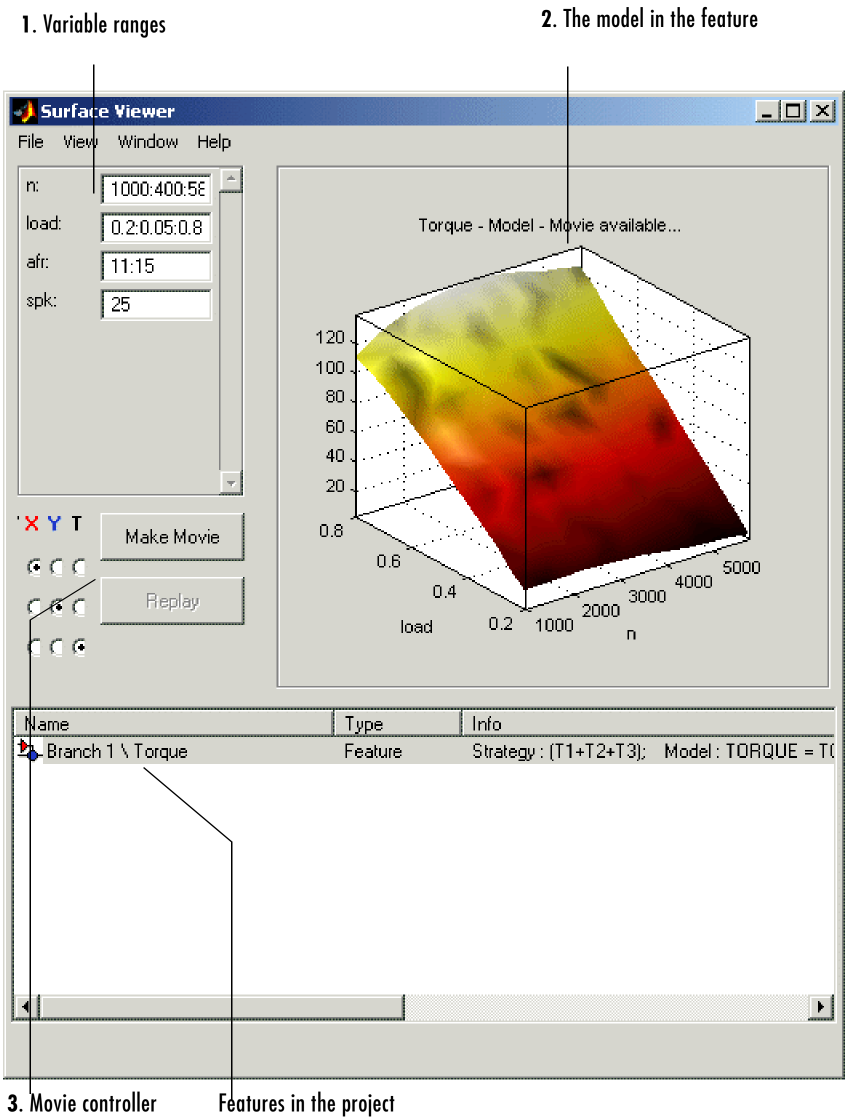
3. Make the movie. (See “Making Movies” on page 12-12.)

   This enables you to view the model or feature as it steps through several values of a variable. For example, if you want to view a feature calibrated for maximum brake torque (MBT) as it varies over exhaust gas recycling (EGR), you can make a movie of the feature.

4. Print or export the display. (See “Printing and Exporting the Display” on page 12-13.)

The following sections describe these steps in more detail.

Note You can use the surface viewer to display your model or feature without making a movie.


1. Variable ranges
2. The model in the feature
Surface Viewer
File View Window Help
n: 1000:400:58
load: 0.2:0.05:0.8
afr: 11:15
spk: 25
X Y T
Make Movie
Replay
Torque - Model - Movie available...
120
100
80
60
40
20
0.8
0.6
0.4
0.2
load
1000
2000
3000
4000
5000
n
Name
Type
Info
Branch 1 \ Torque
Feature
Strategy : (T1+T2+T3); Model : TORQUE = T(
3. Movie controller
Features in the project

Setting Variable Ranges

The **Surface Viewer** does not work over continuous ranges, only at discrete points. You must specify, for the model or feature, the discrete points you want to include in the display. You can display models or features over a range of points.

There are three ways you can indicate the points to include in the display. You can specify

- A single value
- Each discrete point at which you want to evaluate the model (or feature)

  For example, if you want to display the variable *x* at 0, 1, 7, 30, and 50, enter the following in the appropriate edit box:

  ```
  0 1 7 30 50
  ```

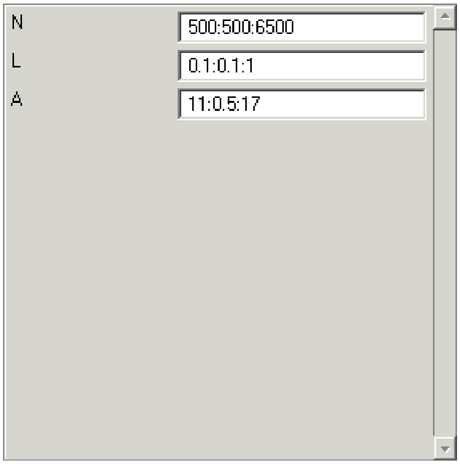
- The values, in the following manner:

  `minimum value``:`*`step increment`*`:`*`maximum value`*

  For instance, if you want to display the variable *x* from 0 to 50 in increments of 5, enter the following in the appropriate edit box:

  ```
  0:5:50
  ```

When you alter the variables, you can select whether you want the display to update automatically or not. You can toggle the automatic update on and off by selecting **View –> Automatic Evaluation**. When you want to update the display, select **View –> Evaluate Now**.

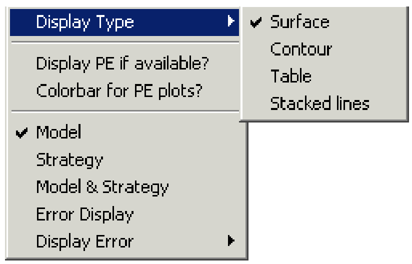


Before you can make a movie, ensure that all the variable values are set up in this manner. You need to specify ranges for at least three variables to make a movie. See “Making Movies” on page 12-12.

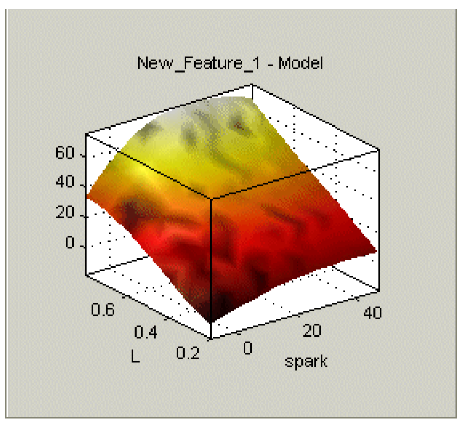
Displaying the Model or Feature

There are several aspects of the display to consider before making the movie.

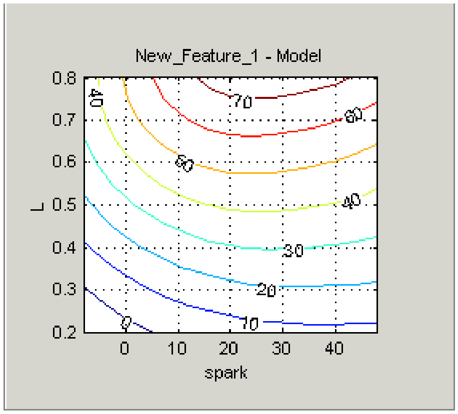
- You can rotate the model by left-clicking and dragging.
- Right-clicking the model or feature display brings up a menu (see below). This gives options on how to display the model or feature.



- Use the right-click menu to display the model or feature as
  - A surface

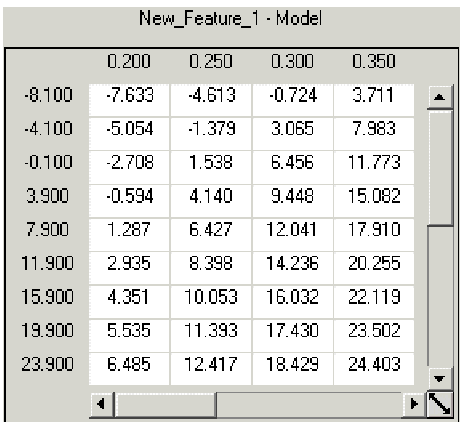


- Contour lines



- A table

  You can only display the model or feature as a table if no more than two inputs are variables. For more information, see “Setting Variable Ranges” on page 12-5.

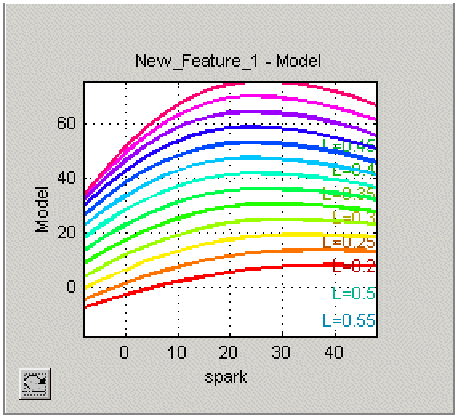


New_Feature_1 - Model

	0.200	0.250	0.300	0.350
-8.100	-7.633	-4.613	-0.724	3.711
-4.100	-5.054	-1.379	3.065	7.983
-0.100	-2.708	1.538	6.456	11.773
3.900	-0.594	4.140	9.448	15.082
7.900	1.287	6.427	12.041	17.910
11.900	2.935	8.398	14.236	20.255
15.900	4.351	10.053	16.032	22.119
19.900	5.535	11.393	17.430	23.502
23.900	6.485	12.417	18.429	24.403

- Stacked lines

  You can swap the variable that is displayed on each axis by clicking .



Displaying Other Views

Note that if you are using the surface viewer to view a feature, you can use the right-click menu to view the following:

- Model
- Strategy
- Model and strategy

Viewing More Than One Model or Feature

To view more than one model or feature, select **View –> Multiple Display Mode**.

Note For information on the two different error displays available using the right-click menu, see the next section, "Displaying Errors" on page 12-10.

Displaying Errors

There are two different error displays available using the right-click menu:

- Error between the model and the feature (See “Feature Error Data” following.)
- Predicted error of the model (See “Predicted Error Data” on page 12-10.)

Feature Error Data

When you are viewing a feature, this displays the error between the strategy and the model.

To display the error, select **Error** from the right-click menu.

You can choose the style that you want to use to display the error. You can choose from one of the following styles by selecting **Display Error** from the right-click menu:

- **Error**. This is the difference between the feature and the model.
- **Absolute Error**. This is the absolute value of the error.
- **Relative Error (%)**. This is the error as a percentage of the value of the model.
- **Absolute Relative Error (%)**. This is the absolute value of the relative error.

To view the error statistics, select **View –> Error Statistics**. This opens a dialog box with a list of the summary statistics for that model or feature.

Predicted Error Data

If the model is imported from the Model Browser, it is possible to display the *predicted error* (PE) data.

The PE is the absolute error between the model and the data used to construct the model. The Model Browser compares the model and the original data, and stores the error between the model and the original data as the PE of the model.

Viewing the PE

Select **Display PE** from the right-click menu.

You can only view the PE if you are displaying one of the following:

- Surface
- Table

Surface Display of PE

- Where the error is greatest, the surface is red.
- Where the error is least, the surface is blue.

It is possible to see a scale for the PE by clicking **PE Colorbar** from the right-click menu.

Table Display of PE

The table shows all points that are beyond the PE threshold as being purple.

Setting the PE Threshold

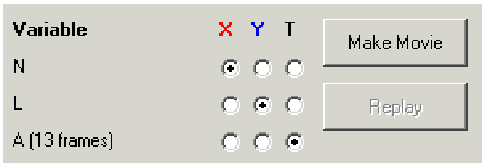
To set the threshold of the PE display:

- Select **Display Options –> Set PE Threshold**.

  The **Predicted Error Threshold** dialog box appears.
- Select an appropriate value for the PE threshold and click **OK**.

Making Movies

The movie option allows you to see an evaluation over two variables at successive values of a third variable. For example, a model of torque might have speed (N), load (L), and air-fuel ratio (A) as inputs.



The movie option allows you to view how the torque model behaves over the ranges of speed and load for successive values of air-fuel ratio.

You can only access the movie controls when three variables have a range defined (rather than a single value). You can view the model surface plotted across the range of two variables, and define the third variable as "time" to see the model surface change across the third variable's range.

Controlling the Movie

You have to indicate which variable you want to display on which axis:

1. Select the variable you want to vary over time (`T`) by clicking the correct variable in the `T` column.

   In the preceding torque model example, you can see that `A` is selected to vary over `T`.

2. Select the variable you want to display on the *x* and *y* axes.

   In the preceding torque model example, you can see that `N` is selected for the *x* axis and `L` is selected for the *y* axis.

3. Click **Make Movie**.

Once the movie is made, you can replay the movie by clicking **Replay**.

Printing and Exporting the Display

To print the display, select **File –> Print**.

You can export the display to a comma-separated variable file.

To export the display, select **File –> Export to CSV**, then select one of the following options:

- **Model** to export the model
- **Strategy** to export the strategy
- **Error** to export the error between the feature and model

13

Manual Calibration and the History Display

Using the Manual Calibration View

Select this view by clicking the **Manual Calibration** button. It opens automatically if you add a table using the **File –> New** menu.

The **Manual Calibration** view lists all the tables and normalizers in the current CAGE session.

Here you can add or delete tables and normalizers, and you can calibrate them manually. Once you have added new tables here you can also fill them using experimental data by going to the **Data Sets** view.

You can use the **History** display from here (and from any other table or normalizer view in CAGE) to view and manage changes in your tables. You can use the **History** display to reverse changes and revert to previous versions of your tables.

See

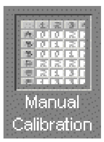
- "Adding and Deleting Tables" on page 13-3
- "Filling Tables from Data Tutorial" on page 5-1
- "Using the History Display" on page 13-5

Adding and Deleting Tables

When you are calibrating a collection of tables using either **Feature** or **Tradeoff** calibrations, you cannot easily add or delete tables without affecting the entire calibration.

You might want to add a table (for example, to fill by reference to experimental data). Or you might want to delete a table (for example, after adjusting a strategy for a feature calibration).

To add or delete tables, you can first select the **Manual Calibration** view. CAGE automatically switches to this view if you add a table using the **File –> New** menu items.



The **Manual Calibration** view lists all the tables and normalizers in the current CAGE session.

Adding Tables

To add a table to a session:

1 Decide whether you want to add a one- or a two-dimensional table.

   For example if you want to add a modifier table to account for the variation in exhaust gas recirculation, add a one-dimensional table (which has one input). If, however, you want to add a table with speed and load as its normalizer inputs, then add a two-dimensional table.

2 Select **File –> New –> 1D Table** or **File –> New –> 2D Table** as appropriate.

   This automatically switches you to the **Manual Calibration** view.

3 Select **Tools –> Calibration Manager** to determine the size of your new table. For information, see “Setting Up Tables” on page 11-3.

You can manually calibrate by entering values in any table.

Deleting Tables

To delete a table or a normalizer from a session:

1. Select **Manual Calibration** view.
2. Highlight the required table or normalizer.
3. Click .

Note When deleting items, you must delete from the highest level down. For example, you cannot delete a table that is part of a feature; you must delete the feature first.

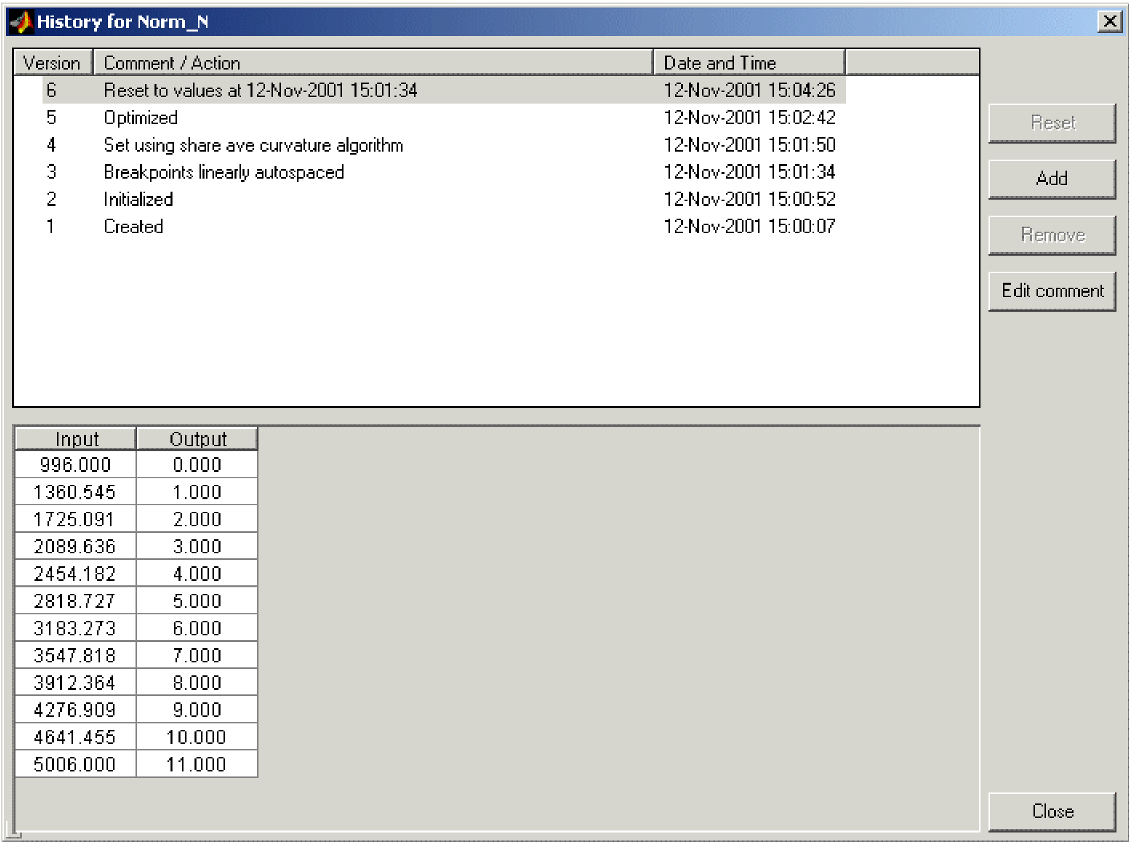
Using the History Display

The **History** display enables you to view the history of any table or normalizer in a CAGE session.

The **History** display lets you

- Revert to previous versions of tables and normalizers (See “Resetting to Previous Versions” on page 13-7.)
- Compare different versions of tables and normalizers (See “Comparing Versions” on page 13-8.)

You can view the History display of a table or normalizer by selecting **View –> History**.



The upper pane of the **History** display lists all the versions of the highlighted object.

The lower pane displays the normalizer or table of the highlighted version.

Resetting to Previous Versions

To reset the normalizer or table to a previous version:

1. Highlight the previous version that you want to revert to.
2. Click **Reset**.

Note Tables are independent of normalizers, so if you reset a table to a previous version you must also reset the normalizers to that version (if they have changed).

To remove previous versions of the object or comments:

1. Highlight the version that you want to remove.
2. Click **Remove**.

Adding and Editing Comments About Versions

To add comments:

1. Click **Add**.
2. In the dialog box enter your comment.
3. Click **OK**.

To edit comments:

1. Select the comment that you want to edit.
2. Click **Edit comment**.
3. In the dialog box, edit the comment.
4. Click **OK**.

Comparing Versions

To compare two different versions of a normalizer or table, highlight the two versions using **Ctrl+click**. Note the following:

- The lower pane shows the difference between the later and the earlier versions.
- Cells that have no entries have no difference.
- Cells that have red entries have a lower value in the later version.
- Cells that have black entries have a higher value in the earlier version.

Comparison Table in the History Display

	1468.000	1772.000	2153.000	2616.000	3117.000	3549.000	3963.000
0.200							
0.259							
0.323		-15.347					
0.392						-15.947	
0.462			-13.149	-9.119			
0.530		13.703				9.764	
0.594				19.264			
0.646							
0.694							
0.731							
0.766							
0.800							

Index

L

M

N

P

R

S

T

V