Simcenter™ Flotherm™ Command Center User Guide

Software Version 2021.1

SIEMENS

Support Center: support.sw.siemens.com
Send Feedback on Documentation: support.sw.siemens.com/doc_feedback_form

Table of Contents

List of Figures

List of Tables

Chapter 1
Introduction

Use the Command Center application of Simcenter™ Flotherm™ software to perform “what if” scenarios on a base project and optimize your design.

Overview of the Command Center

Use the command center to create and solve different scenarios for a Simcenter Flotherm project, compare results and optimize the project.

The general procedure when using the Command Center is:

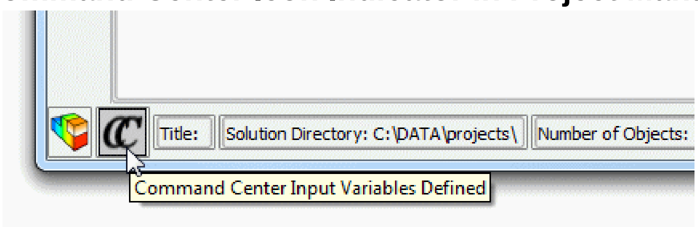
- Set values to be varied (input variables).
- Select which output variables are to be monitored.
- Solve the scenario projects.

The results can be displayed in tables, charts, or profile plots.

Note

The “CC” icon indicator on the Project Manager status bar is highlighted when Command Center input variables have been defined, see Figure 1-1.

Figure 1-1. Command Center Icon Indicator in Project Manager Window



Starting the Command Center

The Command Center application can be opened from any Simcenter Flotherm application window.

Prerequisites

You will need to have valid licenses to run the solver and hence the Command Center. The following license modules are required:

- Standard solver, to enable the Simcenter Flotherm solver.
- Parallel solver, to enable the distribution of multiple solutions among the processors on your machine.

Procedure

1. Start from any open Simcenter Flotherm application window.
2. Either choose **Window > Launch Command Center** or click the **Launch Command Center** icon 

Results

The Command Center window opens with the current Simcenter Flotherm project loaded and **Input Variables** as the default tab.

Note

The file system used for Command Center scenarios changed at Release V12.0. Pre-V12.0 projects loaded into V12.0 or later and then saved will have scenarios saved in an xml-based file system.

If a pre-V12.0 project is loaded into V12.0 or later, *but not saved*, then you can continue using the pre-V12.0 scenarios in a pre-V12.0 release by renaming the file *<project folder>/PDProject/scenario_legacy* to *<project folder>/PDProject/scenario*.

Resetting the Command Center

Use this procedure to reset the Command Center to its default settings. All input variables, output variables, scenarios, and scenario results are removed.

Restrictions and Limitations

- Not allowed while solving a scenario is in progress.

Procedure

1. In the Command Center, choose **Project > Reset**.

   The Reset Command Center dialog box is opened, warning you that the current settings will be lost.

2. Click **OK** to reset the Command Center.

Chapter 2
User Interface

This chapter describes the Command Center application window and general data tree operations.

Introduction to the Command Center User Interface

The Command Center user interface include many of the features found in the Project Manager, such as a data tree, drawing board and property sheets. A major difference is the use of tabbed views to change the window layout and content.

Briefly:

- Use the **Input Variables** view to select which variables are to be varied to create scenarios of the base project.
- Use the **Output Variables** view to select which variables values are to be recorded to compare results between scenarios.
- Use the **Scenario Table** to compare scenario data.
- Use the **Scenario Viewer** to examine the physical layout of each scenario.
- Use the **Solution Monitoring** view as you would use the Profiles application, to monitor solution progress in detail.

The Model Setup/Model/Solver Control pane follows the layout of that in the Project Manager, however, the values are read-only. The values shown in the Input Variables and Output Variables views are those of the Base Project. The values shown in the Scenario Viewer are those of the currently-selected scenario.

The drawing board, when shown, is view-only, objects can be selected but not moved. The drawing board layout shown in the Input Variables and Output Variables views is that of the Base Project. The layout shown in the Scenario Viewer is that of the currently-selected scenario. The layout view can be manipulated in the same way as when using the Project Manager. The drawing board inherits the Drawing Board preferences defined in the Project Manager (**Edit** > **User Preferences**).

The contents of the Project Attributes/Library pane (F7) are read-only. The **Library** tab is not included; library items must be added to the project using the Project Manager before they can become selectable in the Command Center.

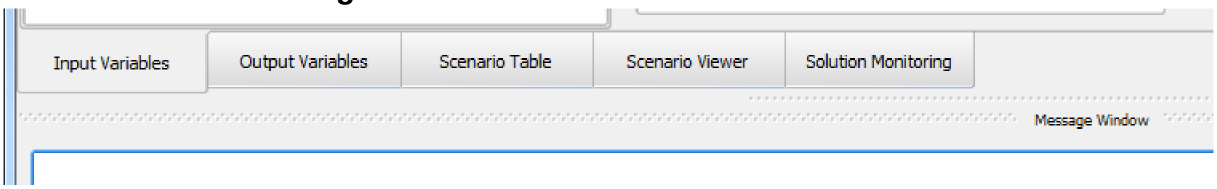
The message window (**Window** > **Message Window**) can be a separate window or docked at the bottom of the Command Center window, where it occupies the full width of the window.

Command Center Window

The Command Center application window has five views. Each view corresponds to a stage of the workflow.

Each view is opened by clicking a tab, for example, see Figure 2-1.

Figure 2-1. Command Center View Tabs



Input Variables View

To access: Click the **Input Variables** tab.

Use this view to select which of the variables of the base project are to be varied, and how they are to be varied.

Description

The Input Variables view is divided into panes, similar to the Project Manager.

Objects

- Data Tree Pane

  This pane has four tabs.

  Use the check boxes located on the left of descriptions to select input variables.

Object	Description
Model Setup tab	Read-only base project values and check boxes for model setup parameters that can be selected as input variables. The following *cannot be selected* as input variables: • Type of Solution • Solar Radiation On/Off If Solar Radiation is switched on in the base project, then Solar Configuration parameters can be selected as input variables by opening the Solar Configuration dialog box (Click to View). • Transient Solution On/Off If Transient Solution is switched on in the base project, then *some* Transient Solution parameters can be selected as input variables by opening the Transient Solution dialog box (Click to View). • Stored Variables

Object	Description
Model tab	A copy of the Project Manager data tree. Object selection is synchronized between the Command Center and the Project Manager. Property sheets show read-only base project values and have check boxes for object parameters that can be selected as input variables. The Command Center always displays positions and sizes of objects using an absolute coordinate system. You can pick from variable lists to access all possible input variables. For example, Define Conducting Layer in PCB object property sheets and Source Type in Source attribute property sheets. You can multiple-select objects then check the check boxes to multiple-select input variables. The following *cannot be selected* as input variables: • Name • Hide. The adjacent check box hides/reveals the object on the Command Center drawing board.
Solver Control tab	Read-only base project values and check boxes for solver control parameters that can be selected as input variables.
User Variables tab	You can de-rate and pro-rate multiple heat sources using the Assigned Heat Source Rating variable, see "Assigning Heat Source Rates Across the Project (AHR)" on page 58.

- Drawing Board

  Enables selection of objects as an alternative to selection from the Model data tree.

- Input Variables Pane

  This pane is divided into three areas: an Input Variables Selection Table, a Type Selector, and an Input Variable Values Definition Table, which is not normally visible.

  Two splitters are used to separate the areas.

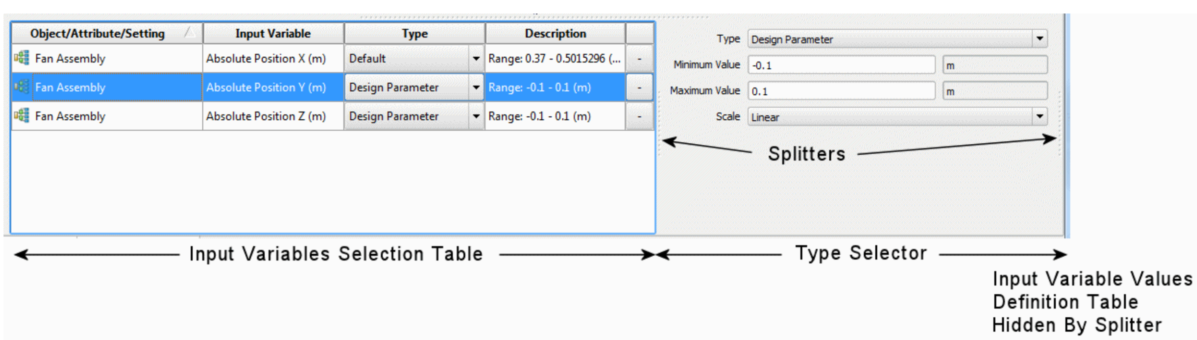


Table 2-1. Input Variables Pane - Input Variables Selection Table Objects

Object	Description
A list of the selected input variables. **Tip:** You can sort the table by any column by clicking on the up/down arrow in the column header, which is made visible by clicking in the header.	
Object/Attribute/ Setting	The object, attribute, or setting to which the input variable is associated.
Input Variable	The name of the input variable, and its units where appropriate.
Type	The type of input variable, refer to Table 2-2. You can select a different type from the dropdown list.
Description	• (Default and Design Parameter types) The range of values. • (Dependent Variable type) The linear function used, as defined in the Dependent Variable dialog box. The linear function is also shown as hover text above the cell.

Table 2-2. Input Variables Pane - Type Selector Objects

Object	Description
Type	The type of input variable: • **Default** − enables you to define values. • **Design Parameter** − numeric, Boolean, or enumeration type values are defined by program as part of a DoE generation. Boolean values do not have a definable range, the possibility of the value being either On or Off is implied. Enumeration type values do not have a definable range, the range includes all supported options and is implied. • **Dependent Variable** − (numeric values only) the value is defined by a dependency on one or more other values.
Minimum Value	(Design Parameter type, numeric type input variable) The lower limit of the range of values to be assigned DoE.
Maximum Value	(Design Parameter type, numeric type input variable) The upper limit of the range of values to be assigned by DoE.

Table 2-2. Input Variables Pane - Type Selector Objects (cont.)

Object	Description
Scale	(Design Parameter type, numeric type input variable) How input variable values are distributed between the lower and upper limits: • **Linear** − A linear distribution of values, for example, see Figure 2-2. • **Logarithmic** − A logarithmic distribution of values, for example, see Figure 2-3. Use to distribute values when the range covers orders of magnitude. Values of contact resistances and network assembly R and C values for a model calibration may benefit from an initial logarithmic distribution. **Note:** The Logarithmic distribution algorithm is best suited for floating point input variables. In cases where the input design parameters are bounded by integers only, then the generated DoE values will be unpredictable.
Calculation box	(Dependent Variable type) Read-only dependency linear equation.
Constant Term	(Dependent Variable type) Read-only value of the constant term required to normalize the dependency linear equation.
Click to Edit	(Dependent Variable type) Click this button to open the Dependent Variable dialog box, see "Dependent Variable Dialog Box" on page 22.

Figure 2-2. Linear Distribution of DoE Values

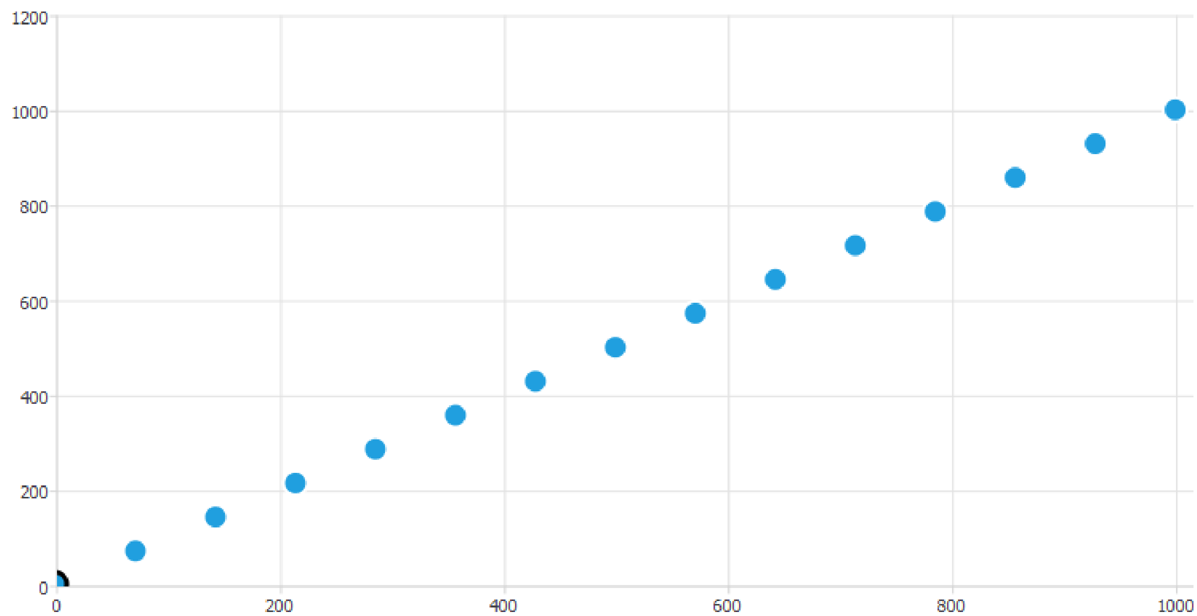


Figure 2-3. Logarithmic Distribution of DoE Values

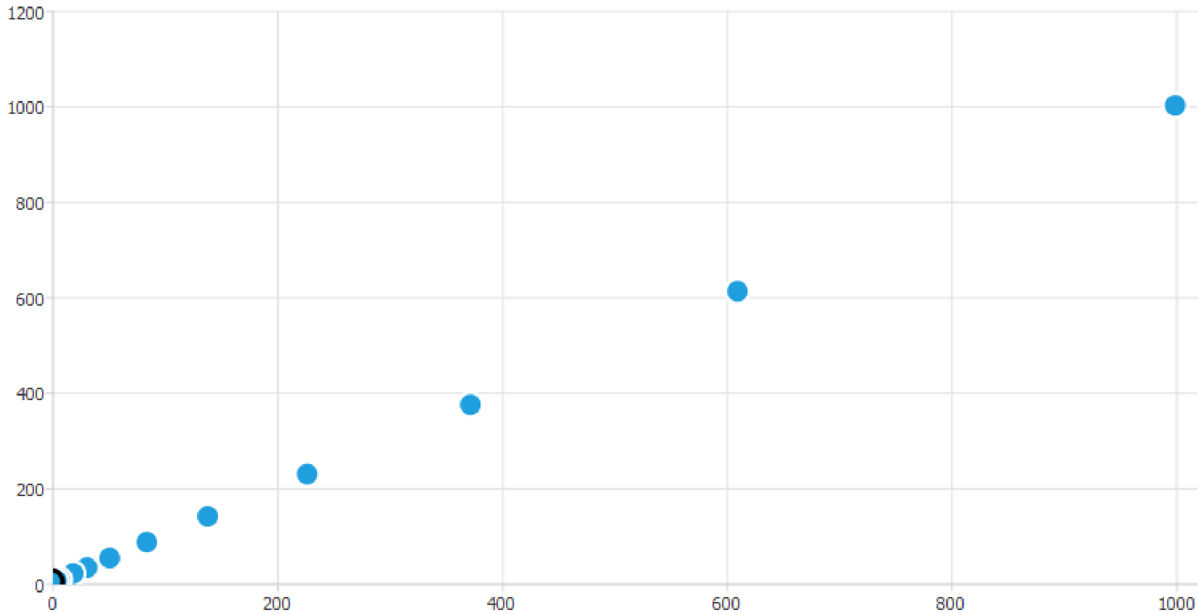


Table 2-3. Input Variables Pane - Input Variable Values Definition Table Objects

Object	Description
Note: Not shown by default. To view this table, click and drag the splitter at the far-right of the Input Variables Pane, see Figure .	
Scenario	The scenario name.
Value	The value used in the scenario for the input variable currently selected in the input variables selection table. Add and remove scenario values using the + and - buttons.

- Project Attributes Pane

  Read-only base project values and check boxes for attribute parameters that can be selected as input variables.

  Tip

  You can multiple-select attributes then check the check boxes to multiple-select attribute parameters as input variables.

  The following *cannot be selected* as input variables:

  - Transient attribute Sub-Function parameters.
  - Transient attribute Transient Time Chart values.

Tip

Variations on transient attributes can be made by changing the transient attribute that is attached to another attribute or object.

Related Topics

Activating Input Variables

Dependent Variable Dialog Box

To access: Click **Click to Edit** from the Input Variables view when the Type of an input variable is Dependent Variable.

Use this dialog box to define the values of an input variable as a linear function of one or more other numeric input variables.

Objects

Object	Description
Linear Function Definition Area	The area used to define the dependency.
Active Input Variables	A list of available numeric input variables. Double-click an input variable to add it to the definition area.
Number Pad	Use this pad to modify input variable values by applying multipliers and/or constants. If the calculation does not maintain the Base Project value, then the program automatically corrects the value by the addition of a constant value, shown in the Constant Term field.

Usage Notes

- Dependent input variables are created from a linear function of the form:

  Input variable = $m_1 x_1 + m_2 x_2 \ldots + c$

  where m is a multiplier, x another input variable and c is a constant.

- The **OK** button is only active for valid definitions.

Related Topics

Input Variables View

Output Variables View

To access: Click the **Output Variables** tab.

Use this view to select the monitor point variables, derived properties and metric information to store for the parametric study under consideration, and for the cost function, if any, used to evaluate the best design.

Objects

- Data Tree Pane

  The Data Tree pane has three tabs.

Object	Description
Model tab	Similar to the Project Manager **Model** tab, except that SmartParts are decomposed into their constituent primitives. Output variables are selected using check boxes in the tabbed property sheet, see "Output Variables Property Sheet" on page 26. **Tip:** You can multiple-select objects then check the check boxes to multiple-select output variables.
Field Summary tab	Base and auxiliary variables that can be selected as output variables. The options available are filtered by icons: • shows common solution variables check boxes: Temperature, Speed, Pressure, and X, Y and Z Velocity. • shows check boxes for any stored variables that have been selected: X, Y and Z Heat Flux, X, Y and Z Grad T, Mag Grad T, Bn, Sc. • shows auxiliary variables check boxes: Flow Angle and Total Pressure. • shows miscellaneous solution variables check box: KE Turb, Diss Turb, Generation, Turb Vis, Solar Viz, Melt Fraction. Click the expand button to open more check boxes which allow selection of Maximum and Minimum values for display as output variables.
Metrics tab	Grid data that can be selected as output variables.

- Drawing Board

  Enables selection of objects as an alternative to selection from the Model data tree.

- Output Variables Pane

  The Output Variables pane is divided into two areas: an output variables selection table and a cost function definition.

Object	Description
Output Variables Selection Table	The currently-selected output variables, for all scenarios. • **Object/Attribute/Setting** − The object, attribute, or setting to which the output variable is associated. • **Output Variable** − the name of the output variable, and its units where appropriate. The units are those defined in the Project Manager Global Units dialog box. • (Include in Cost Function? icon) − a checked check box indicates that the output variable is used in the cost function. You can include an output variable in the cost function directly by checking an empty check box. **Tip:** You can sort the table by any of the columns by clicking in the column header.
Cost Function Definition Applies to the output variable selected in the selection table.	
Include in Cost Function	Check this check box if you want to use the output variable in the Sequential Optimization (SO) techniques.
Cost Type	You have a choice: • Linear - determines the cost function by multiplying the current selected term by the cost weighting. The optimum is the lowest value. • Target - sets the target value for which the optimization techniques try to reach. The optimum is when an acceptable value above or below which the design is considered acceptable.
Cost Weighting	The weighting applied to the output variable: Cost Function = Cost Weighting × Value
Cost Target	(Target) The target of the output variable value such that: Cost Function = Cost Weighting × |Cost Target - Value|

Usage Notes

When a cost function is defined, the function is displayed below the fields.

Cost Function Usage Examples

To make the sequential optimizer search for a specific value: include the relevant output variable in the cost function, set the Cost Type to Target, define the Cost Target value.

To make the sequential optimizer maximize a value instead of minimizing it: include the relevant output variable in the cost function, then set the Cost Weighting to -1.

To make sure that the sequential optimizer finds the global minimum: perform a Design of Experiments (DoE) before starting the sequential optimizer. This can be requested using the Optimize Dialog Box.

Related Topics

Activating Output Variables

Cost Function

Output Variables Property Sheet

To access: Open the Output Variables view, make sure the **Model** tab is selected, then select an object in the data tree or drawing board.

Use this property sheet to select which object-specific output variable values will be recorded for all scenarios.

Description

The property sheet is tabbed. The available tabs and selectable output variables within the output variables property sheet depend on the type of object that is currently selected.

Variables that are output in Project Manager results tables, except string-value variables such as the fan range flag “Fan In Range”/”Fan Out of Range”, can be selected as Command Center output variables.

Filtering of Selectable Output Variables

The Data, Face Temperature, Heat Transfer, Fluid Flow and Region Summary tabbed sheets have selectable icons that filter the options that are displayed.

Expand/Collapse buttons 

are also used to filter the display of check boxes.

Objects

Tab	Description
Data	See “Output Variables Property Sheet Data Tab” on page 27.
Face Temperature	See “Output Variables Property Sheet Face Temperature Tab” on page 28.
Area	See “Output Variables Property Sheet Area Tab” on page 29.
Heat Transfer	See “Output Variables Property Sheet Heat Transfer Tab” on page 30.
Fluid Flow	See “Output Variables Property Sheet Fluid Flow Tab” on page 31.
Region Summary	See “Output Variables Property Sheet Region Summary Tab” on page 32.

Related Topics

Output Variables View

Output Variables Property Sheet Data Tab

To access: Open the Output Variables property sheet, then select the Data tab.

Use check boxes on this tab to select object data as output variables.

Description

The check boxes are dependent on the object selected. Some examples are given below.

Objects

- Monitor Points:

Object	Description
Common	Check boxes for common solution variables.
Stored Variables	Check boxes for stored variables.
Miscellaneous	Check boxes for miscellaneous solution variables.

- Fans:

  Volume Flow (rate at operating point), Static Pressure (at operating point), Swirl Speed, Derated Fan Power, Derated Fan Noise, Fan Efficiency and Dissipated Power.

- Cuboids with Material attachment:

  Mean Solid Temperature and Solid Mass.

- Block with Holes:

  Mean Solid Temperature, Solid Mass, Minimum Temperature, Maximum Temperature, Standard Deviation, Volume.

Note

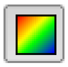
For Block with Holes, holes are ignored, whether they are open space, resistance or solid, when calculating temperature output variables.

Output Variables Property Sheet Face Temperature Tab

To access: Open the Output Variables property sheet, then select the Face Temperature tab.

Use check boxes on this tab to select output variables of surface temperatures and solid/fluid face temperatures.

Objects

Object	Description
 Surface Temperature	Shows Surface Temperature variables check boxes.
 Face Temperature	Shows Face Temperature variables check boxes.

Output Variables Property Sheet Area Tab

To access: Open the Output Variables property sheet, then select the Area tab.

Use check boxes on this tab to select surface areas of geometric objects as output variables.

Objects

- S-S Surface Area

  You have the choice of selecting all solid-solid surface areas or selecting individual X/Y/Z High/Low face solid-solid surface areas.

- S-F Surface Area

  You have the choice of selecting all solid-fluid surface areas or selecting individual X/Y/Z High/Low face solid-fluid surface areas.

Output Variables Property Sheet Heat Transfer Tab

To access: Open the Output Variables property sheet, then select the Heat Transfer tab.

Use check boxes on this tab to select output variables of conduction, convection, or radiation heat flux. Each is expandable to allow selection for individual X/Y/Z High/Low faces.

Objects

Object	Description
Conducted Heat	Shows Conducted Heat output variables check boxes.
Convected Heat	Shows Convected Heat output variables check boxes.
Radiated Heat	Shows Radiated Heat output variables check boxes.
h Conductive Heat Transfer Coefficient	Shows a Conductive Heat Transfer Coefficient output variable check box.
h Convective Heat Transfer Coefficient	Shows a Convective Heat Transfer Coefficient output variable check box.
Heat Flow Rate	Shows Heat Flow output variables check boxes.
Radiation	Shows Radiation output variables check boxes.

Output Variables Property Sheet Fluid Flow Tab

To access: Open the Output Variables property sheet, then select the Fluid Flow tab.

Use check boxes on this tab to select output variables of volume or mass flow, or pressure drop results.

Objects

Object	Description
Volume Flow Rate	Shows Volume Flow output variables check boxes.
Mass Flow Rate	Shows Mass Flow output variables check boxes.
Pressure Drop	Shows a Pressure Drop output variable check box.

Output Variables Property Sheet Region Summary Tab

To access: Open the Output Variables property sheet, then select the Region **Summary** tab.

Use check boxes on this tab to select output variables of results of solution variables within the region. Each is expandable to allow selection of Maximum, Minimum, Mean, and Standard Deviation values.

Objects

Object	Description
Common	Shows common solution variables check boxes: Temperature, Speed, Pressure, and X, Y and Z Velocity.
Stored Variables	Shows stored variables check boxes: X, Y and Z Heat Flux, X, Y and Z Grad T, Mag Grad T, Bn, Sc.
Auxiliary Variables	Shows auxiliary variables check boxes: Flow Angle and Total Pressure.
Miscellaneous	Shows miscellaneous solution variables check boxes: KE Turb, Diss Turb, Generation, Turb Vis, Solar Viz, Melt Fraction.

Scenario Table

To access: Click the **Scenario Table** tab.

Use this table to view the solution status of scenarios. You can also edit input variables values, set the initialization points, choose to store full results for scenarios, and create new scenarios.

Description

The Scenario Table includes a column for each scenario, including the **Base Project**, and an empty column, titled **Append Scenario**.

The table is updated when solving scenarios and when changes are made to the model or the scenarios.

Within limitations, you can paste values into, and copy values from, the table.

Objects

Object	Description
Input Variables (blue background in title cells)	One row for each input variable. Values can be changed by double-clicking in a cell: numeric values can be edited or overtyped, non-numeric (enumeration type) values can be selected from the dropdown list. Dependent variables are read-only.
Solution Status	The current status of each scenario, see "Scenario Solution Status" on page 75.
Store Results?	What results are stored, if any. Double-click in a scenario cell to select an option from the dropdown list: None, Full, or History Only (residual and monitor point data). The Scenarios default value is Full. The Base Project value is Full and is read-only.
Initialize From	The values to initialize from. Double-click in a scenario cell to select a different value. • No Project – Use the initial variables values as defined in the **Solver Control** tab of the Project Manager. • Base Project – Use the Base Project results. • *<scenario name>* – Use the results from a scenario. The default is to use the Base Project results. The Base Project value is No Project and is read-only.

Object	Description
Output Variables (orange background in title cells)	One row for each output variable. The values are read-only and are updated after solving. **Note:** The number of significant figures shown is controlled by a setting in the **Results** tab of the User Preferences dialog box, see "Command Center User Preferences Dialog Box" on page 92. Values in italics indicate Stale results, which may be inconsistent with the current parametric study, however, these results can be used in charts. The units are those defined in the Project Manager Global Units dialog box.
Cost Function (orange background in title cells)	Only shown if a cost function has been defined. The cost function results for any sequential optimization runs. The cost function equation is displayed as hover text over the row. **Note:** The number of significant figures shown is controlled by a setting in the **Results** tab of the User Preferences dialog box, see "Command Center User Preferences Dialog Box" on page 92.

Usage Notes

Selecting Scenarios

- To select a scenario, click the column header.

  The column is highlighted.

  Right-click to open a context-sensitive menu whose operations act on the selected scenario.

Creating Scenarios

- To create a new scenario, add a value to any of the **Append Scenario** column cells.

  The new scenario inherits its other values from the preceding scenario/column, and a new **Append Scenario** column is added to the table.

Related Topics

Pasting Values Directly Into the Scenario Table

Exporting All or Part of the Scenario Table to a Spreadsheet

Scenario Viewer

To access: Click the **Scenario Viewer** tab.

Use the Scenario Viewer to inspect the parameters and physical layout of each scenario before solving, and to view the results for each scenario after solving.

Objects

- Scenario Selection Table

  A three-column table:

  - Column 1 − Results Stored Indicator. An icon indicates that Full results are stored.
  - Column 2 − Scenario Name.
  - Column 3 − Solution Status.

  Select a scenario by clicking in a row, the row is then highlighted.

- Model Pane

  The model parameters for the currently-selected scenario. The parameters are defined within three tabs: **Model Setup**, **Model**, **Solver Control**, and **User Variables**.

- Drawing Board

  The physical layout of the currently-selected scenario. When in Analyze mode, the layout can be overlaid with results.

- Project Attributes Pane (Create Mode)

  Attribute values for the currently-selected scenario.

- Results Tree (Analyze Mode)

  Solver results for the currently-selected scenario.

  The Total Range for scalars is the Maximum and Minimum for all scenarios with full results stored.

  Refer to Results Property Sheets in the *Simcenter Flotherm User Guide* for details on results property sheets.

Usage Notes

Press F10, or click the Create and Analyze icons, to switch between Create and Analyze mode.

Related Topics

Inspecting the Physical Layout of a Scenario

Viewing Scenario Results

Solution Monitoring View

To access: Click the **Solution Monitoring** tab.

Use this view to track the solution of a selected scenario project or to view residual and monitor point plots after a solution.

Objects

- Scenario Selection Table

  A list of scenarios and their solution status. Select a scenario by clicking in a row. The currently-selected scenario is shown by highlighting the row.

- Plot Profiles Area

  A display area for showing plot profile panes for the currently-selected scenario.

  The plot settings cannot be changed directly. To change the settings, change those of the equivalent Base Project plot in Profiles (**Window > Launch Profiles** in the Project Manager).

  Refer to Solution Monitoring and Profile Plots in the *Simcenter Flotherm User Guide* for details on profile plots.

Related Topics

Solving Scenarios

Editing Solution Plots

General Operations

Data Tree and Drawing Board operations.

Filtering the Data Tree

Use the Filter option of the Find tool to unclutter the data tree and drawing board.

Procedure

1. Open one of the views that shows the data tree, that is, the Input Variables view, the Output Variables view or the Scenario Viewer.
2. Press Ctrl+F to open the Find dialog box.
3. Select search criteria to restrict the number of found objects.

   The Find tool has the same functionality as the Find tool in the Project Manager.
4. To remove objects from the data tree and drawing board that do not match the search criteria, check the Filter check box.
5. Click **Find**.

Results

Found objects are highlighted in the data tree and drawing board.

If Filter was checked then other objects are hidden, unless they are children of the found objects.

Objects remain hidden until the Find dialog box is closed.

You can select other views, and select input and output variables, while the Find dialog box is open.

Related Topics

Command Center Find Dialog Box

Saving a Drawing Board Image

Creation of graphics files.

Procedure

1. If the drawing board is divided into frames (Grid View, Split Horizontally or Split Vertically) then make sure that the frame you want to record is selected.
2. Choose **Viewer** > **Output Snapshot**.

The Output Snapshot dialog box is opened.

3. Navigate to a folder, enter a filename, select a file type, and click **Save**.

FloSCRIPT

Advanced users can run a FloSCRIPT file from the Command Center.

XML script files are written for each session in the folder:

<install_dir>\flosuite_v<version>\flotherm\WinXP\bin\LogFiles

Each file has a unique name, using the naming convention *CCLogFile<number>.xml*.

Note

Only the last five log files are retained when a new Command Center session is started. You can retain log files between sessions, however, you must prefix the filename so that it does not begin with the (case-insensitive) “CCLogFile” string.

To run a Command Center FloSCRIPT file, choose **Macro > Play FloSCRIPT** and then select the XML file.

Related Topics

FloSCRIPT [Simcenter Flotherm User Guide]

Command Center General Dialog Boxes

Message window and Find dialog box.

Command Center Message Window

To access: **Window** > **Message Window**

This window reports errors, warnings, and information generated by the Command Center.

Description

This window functions as the Project Manager Message Window but acts in the context of the Command Center. This window is totally independent of the Project Manager Message Window.

Objects

- All objects and fields.

  Refer to Message Window in the *Simcenter Flotherm User Guide*.

Command Center Find Dialog Box

To access: Press Ctrl+F or click the **Find** icon.

Use this dialog box to perform a quick or detailed search for objects in the current topped assembly in the Command Center or for attributes in the Project Attributes tree.

Description

This dialog box functions as the Project Manager Find dialog box but acts on the data tree or Project Attributes tree in the Command Center. This dialog box is independent of the Project Manager Find dialog box.

The Find action will select data tree objects whose results satisfy results table criteria (Common Results Data and Geometry Results Data options in the **Extended Criteria** tab), however, you will have to open the Project Manager in Analysis mode to view the results tables.

Objects

- All objects and fields.

  Refer to Find Dialog Box in the *Simcenter Flotherm User Guide*.

Related Topics

Simple Searches for Objects or Attributes [Simcenter Flotherm User Guide]

Complex Searches for Objects or Attributes [Simcenter Flotherm User Guide]

Filtering the Data Tree

Chapter 3
Overview of Scenario Projects

How the Command Center creates different scenarios based on the project loaded in the Project Manager, and the different phases of command optimization.

Base Project

The base project acts as a starting point from which you define how the changes will occur to create different scenario projects.

Changes could include the position of objects, the size of objects (including parametric settings for SmartParts) and attribute settings.

Although the changes can be set individually, Simcenter Flotherm contains a design optimization feature to automate the operation. To use the design optimization feature, you set the maximum and minimum values for one or more variables, and software, used under license from the Center for Qualitative Methods CQM BV, designs a matrix to best cover the range defined. This is called a Design of Experiments.

Further, once the “best” solution is found, the program will automatically find the optimum solution (Sequential Optimization). To measure the optimum case, you can set cost functions. A simple example of this is “minimize the temperature at point X”, however, complex functions can be created.

Caution

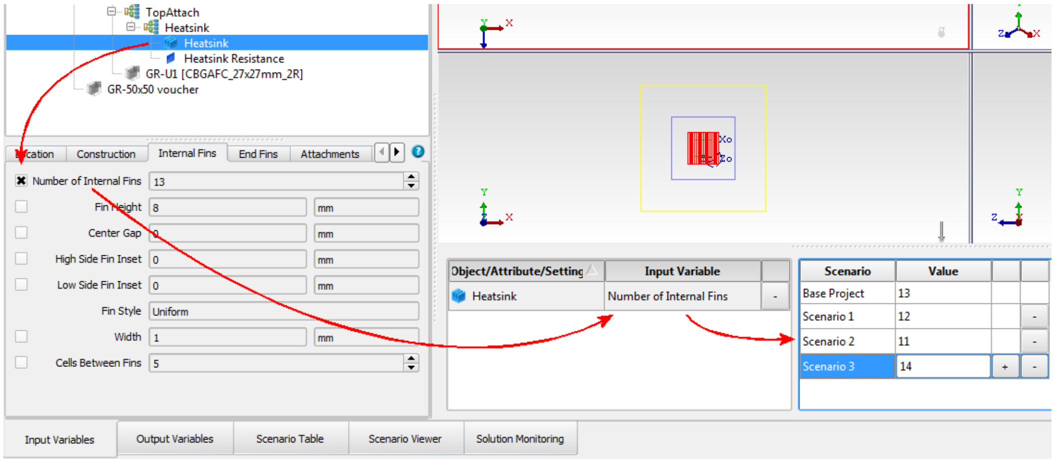
The Command Center is updated with any changes made to the base project in the Project Manager, however, if the base project is re-solved while the Command Center is closed, the output values in the Scenario Table are not updated when the Command Center is re-opened. If you want to re-solve the base project from the Project Manager, and you want the Scenario Table to be updated, make sure that the Command Center is open during the re-solve. Alternatively, reinitialize and solve the base project from the Command Center.

Creating Scenarios

Different scenarios of the base project are created by varying input variables.

Use the Input Variables view to make parametric changes.

Selection is from the data tree.



The changes can be made by ad-hoc, linear, design parameter or linear function methods. The design parameter and linear function methods come into effect when using the design optimization software.

Differing Numbers of Variable Values

The total number of scenario projects is the number of variations *of the most varied* input variable.

Input variables that have fewer variations retain their last variation for the remaining scenario projects. For example, Figure 3-1 shows a Scenario Table where an External Radiant Temperature input variable has six variations (1-to-6), and an External Ambient Temperature input variable has only two variations (1-to-2). The last four External Ambient Temperature values (3-to-6) are held at the same value as the second (2).

Figure 3-1. Values When the Number of Variants Differs

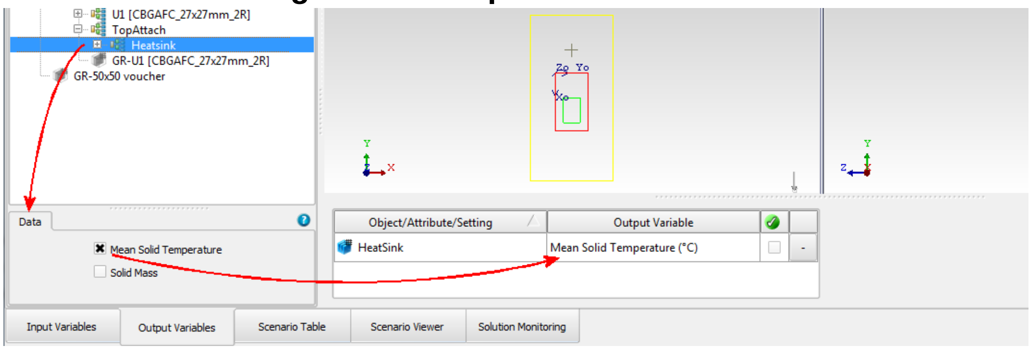
	Base Project	Scenario 1	Scenario 2	Scenario 3	Scenario 4	Scenario 5	Scenario 6
Model Setup : Default Ambient Temperature (°C)	35	30	25	25	25	25	25
Model Setup : Default Radiant Temperature (°C)	35	30	25	20	15	10	5

Choosing Values to be Tracked

Use the Output Variables view to select output variables.

Values such as monitor point variables, temperature of solid blocks and flow through resistances can be selected to be tracked, for example, see Figure 3-2. Any of the selected items can be included in a cost function to control the design optimization software.

Figure 3-2. Output Variables View

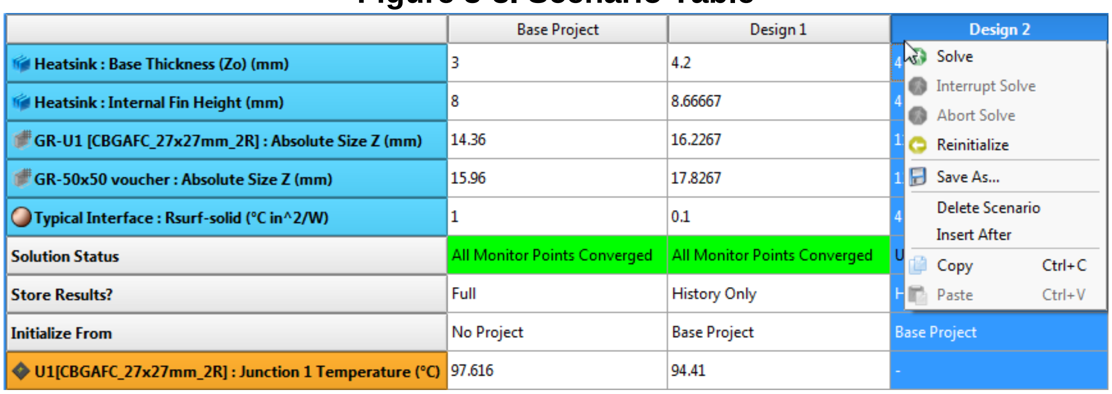


Controlling the Cases

Use the Scenario Table to monitor and control the scenario projects.

Individual scenario solutions can be started and controlled, for example, see Figure 3-3, which shows the scenario context-sensitive menu.

Figure 3-3. Scenario Table



	Base Project	Design 1	Design 2
Heatsink : Base Thickness (Zo) (mm)	3	4.2	
Heatsink : Internal Fin Height (mm)	8	8.66667	
GR-U1 [CBGAFC_27x27mm_2R] : Absolute Size Z (mm)	14.36	16.2267	
GR-50x50 voucher : Absolute Size Z (mm)	15.96	17.8267	
Typical Interface : Rsurf-solid (°C in^2/W)	1	0.1	
Solution Status	All Monitor Points Converged	All Monitor Points Converged	
Store Results?	Full	History Only	
Initialize From	No Project	Base Project	Base Project
U1[CBGAFC_27x27mm_2R] : Junction 1 Temperature (°C)	97.616	94.41	-

The Solution Status is updated during the solution, and output variables values are shown at the end of a successful solution.

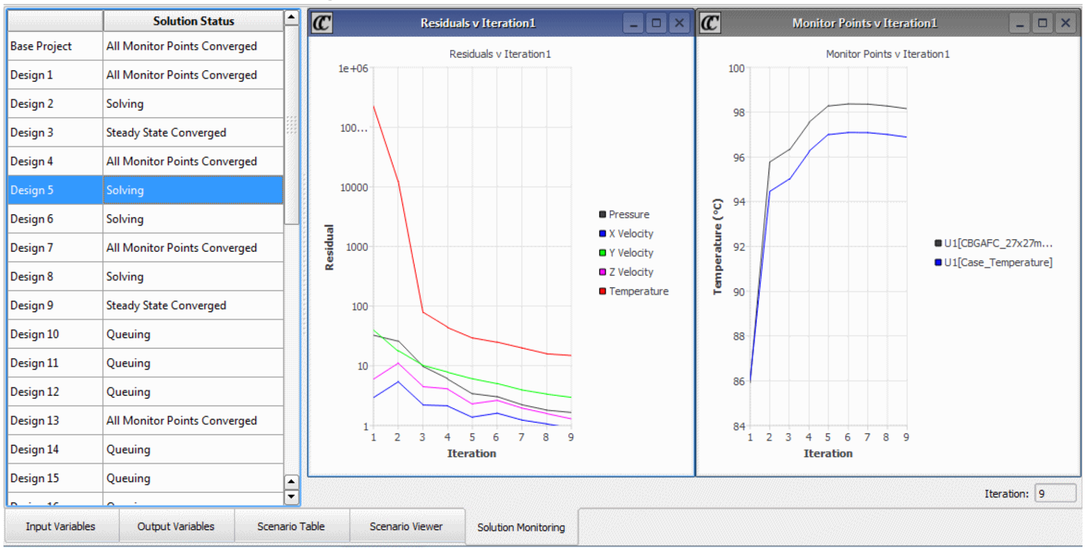
Solution Progress

Solving scenarios can be monitored using the Solution Monitoring view.

Profile plots are displayed for the selected scenario, for example, see Figure 3-4.

The profile plot types that are displayed are those that are displayed in the Profiles window within the Project Manager.

Figure 3-4. Scenario Profile Plots



Automatic Scenario Building

A number of automated features are incorporated into the Command Center.

Automatic Scenario Creation

You can create a sequence of scenario projects that covers every possible combination of activated input variables (Multiply Variables).

You can create scenario projects, changing only one variable at a time (Add Variables).

You can create a sequence of scenario projects where power values only change one at a time (Superposition).

Optimization Techniques

Two design optimization techniques can be used to provide as much information as possible about the influence of the input variables on the output monitored variables:

- Design of Experiments (DoE) creates a number of experiments (scenario projects) where the selected input variables are varied within a defined maximum and minimum range, which can then be solved.
- Sequential Optimization (SO) uses the results of a DoE as input to create and run further scenarios to find the optimum design.

Chapter 4
Building Scenario Projects

How to build scenarios by selecting input variables and defining variations of those variables.

Working With Input Variables

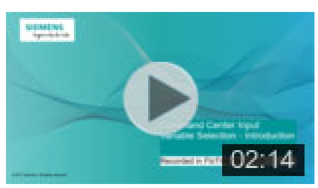
You build scenarios parametrically by activating selected variable types (input variables) and defining sequences of values for those variable types.

Activating Input Variables

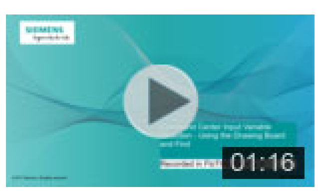
Input variables are activated by selecting them from those available.

Video

Learn how to select geometry, attached attributes, attribute properties, Model Setup parameters, Solver Control settings, and curve points as input variables:



Learn how to select input variables from the Drawing Board and by using the Find tool:



Procedure

1. Open the Input Variables view.

2. This action depends on which type of input variable you want to activate:

Tip

If you want to activate an object parameter or an attribute parameter as an input (or output) variable, you can use the Command Center Find tool to search for objects/attributes in the data tree/Project Attributes tree. The Command Center Find tool has the same functionality as the Project Manager Find tool.

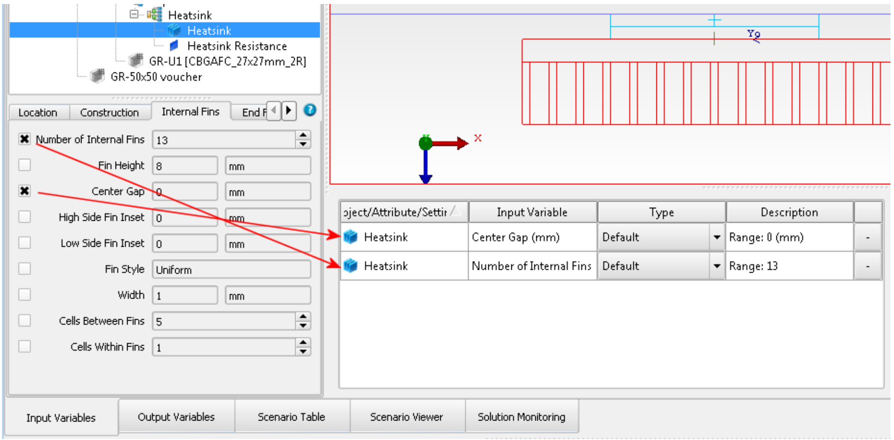
If you want to...	Do the following:
Activate an object parameter as an input variable.	1. Make sure that the **Model** tab is in focus (this is the default tab). 2. Expand (F6) the data tree to find the object in the data tree. 3. Select the object. The object property sheets are opened. **Tip:** You can activate multiple input variables by selecting multiple data tree objects. The variables listed in the property sheets are those that are common to the selected objects. 4. Open the appropriate property sheet tab.
Activate a model setup parameter as an input variable.	1. Click the **Model Setup** tab
Activate a solver parameter as an input variable.	1. Click the **Solver Control** tab.
Activate an attribute parameter as an input variable. **Note:** To vary an attribute attached to a model object, use the object property sheet as in “Activate an object parameter as an input variable.” above.	1. Open the Project Attributes pane. 2. Select the attribute The attribute property sheets are opened. 3. Open the appropriate property sheet tab.

Activation check boxes are on the left of parameters. Values shown are those of the Base Project.

3. Check the appropriate check box(es).

Results

Activated variables are added to the Input Variables Selection table.



Related Topics

Filtering the Data Tree

Assigning Values to an Input Variable Manually

Assigning Values to an Input Variable Manually

You can set up numeric input variables with ad-hoc values.

Prerequisites

- The input variable must have been activated, see "Activating Input Variables" on page 48.

Procedure

1. Select the input variable in the Input Variables Selection table.

   The Type selector defaults to Default.

   The value of the Base Project is shown in the adjacent Input Variable Values Definition table.

2. In the Input Variable Values Definition table, click the + button.

   A new scenario row is added. The value is inherited from the previous row.

3. This step depends on the value type.

   - To change a numeric value, click the value and enter a new value.
   - To change an enumeration type, select one from the dropdown list.

4. Continue adding values.

Defining a Linear Sweep of Input Variable Values

As an alternative to manually defining input variables values, you can set up input variable values as a linear range extending above and/or below the Base Project value.

Restrictions and Limitations

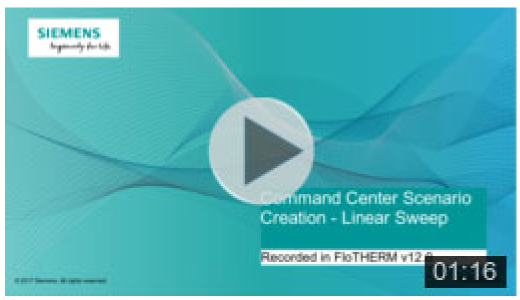
- Applies to numeric input variable types only.

Prerequisites

- The input variable must and have been activated, see "Activating Input Variables" on page 48.

Video

Learn how to create scenarios that have a linear sweep of input variable values:



Procedure

1. Choose **Create Scenarios > Linear Sweep** to open the Linear Sweep dialog box.
2. Select the input variable in the left-hand list (Input Variables) and click >> to move the variable to the right-hand list (Selected Input Variables).
3. In the Linear Variation section, set the Step Size, Number Of Positive Steps and Number Of Negative Steps as appropriate, then click **OK**.

Results

Scenarios are created as necessary and the input variables values are generated based on the linear sweep definition. The values are shown in the Scenario Table.

Negative step values, if there are any, are applied first, followed by positive step values.

If there are existing scenarios and there are more scenarios than steps then the last calculated value is used in any additional scenarios.

The input variables values are updated if the procedure is repeated.

Examples

- Step Size = 0.5, Number Of Positive Steps = 3, Number Of Negative Steps = 2, Base Project value = 5:

Scenario	Input Variable Value	Description
1	4.5	1st Negative Step, 5 − 0.5
2	4	2nd Negative Step, 5 − 1
3	5.5	1st Positive Step, 5 + 0.5
4	6	2nd Positive Step, 5 + 1
5	6.5	3rd Positive Step, 5 + 1.5

Related Topics

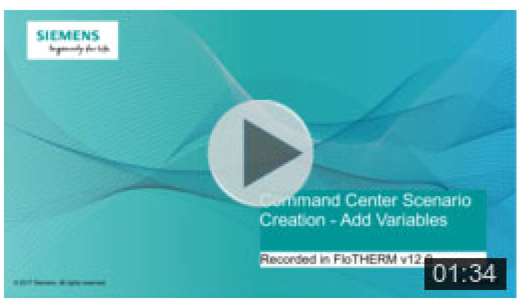
Linear Sweep Dialog Box

Creating Scenarios That Contain Only One Variant of a Variable

Only one of the selected variables changes with each scenario created.

Video

Learn how to create scenarios that contain only one variant of a variable:



Procedure

1. Choose **Create Scenarios** > **Add Variables** to open the Add Variables dialog box.

   A list of available input variables is shown in the Input Variable Selection pane.

2. Select one or more input variables and click >>

   The selected input variables are moved to the Selected Input Variables pane.

3. Click **OK**.

Results

The current values list for all the input variables is updated in the Input Variables panel and the scenarios created are added to the Scenario Table.

Examples

Figure 4-1 shows a Scenario Table before an Add Variables operation.

Figure 4-1. Scenario Table Before an Add Variables

	Base Project	Scenario 1	Scenario 2	*Append Scenario*
Chassis Assembly : Absolute Position X (m)	0	0.25	0.35	
Chassis Assembly : Absolute Position Y (m)	0.00254	0.5	1	
Chassis Assembly : Absolute Position Z (m)	0	0.1	0.2	

Figure 4-2 shows the Scenario Table after the operation. Only one variable value changes with each scenario, scenarios are not created with combinations of variant values.

Figure 4-2. Scenario Table After an Add Variables

	Base Project	Scenario 1	Scenario 2	Scenario 3	Scenario 4	Scenario 5	Scenario 6	*Append*
Chassis Assembly : Absolute Position X (m)	0	0.25	0.35	0	0	0	0	
Chassis Assembly : Absolute Position Y (m)	0.00254	0.00254	0.00254	0.5	1	0.00254	0.00254	
Chassis Assembly : Absolute Position Z (m)	0	0	0	0	0	0.1	0.2	

Related Topics

Add Variables Dialog Box

Creating Scenarios That Cover Every Possible Combination of Input Variables

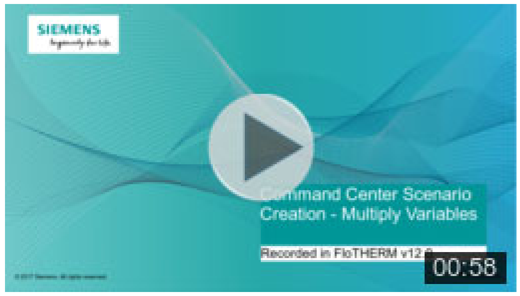
One scenario is created for each unique combination of input variable values.

Restrictions and Limitations

- A maximum of 5000 scenarios can be created using this method.

Video

Learn how to create scenarios that cover all combinations of input variables:



Procedure

1. Choose **Create Scenarios** > **Multiply Variables** to open the Multiply Variables dialog box.

   A list of available input variables is shown in the Input Variable Selection pane.

2. Select one or more input variables and click >>

   The selected input variables are moved to the Selected Input Variables pane.

3. Click **OK**.

Results

The current values list for all the input variables is updated in the Input Variables panel and the scenarios created are added to the Scenario Table.

Examples

Figure 4-3 shows an example where there are three input variables values defined in a scenario.

Figure 4-3. Scenario Table Before a Multiply Variables

	Base Project	Scenario 1	*Append Scenario*
Chassis Assembly : Absolute Position X (m)	0	0.25	
Chassis Assembly : Absolute Position Y (m)	0.00254	0.5	
Chassis Assembly : Absolute Position Z (m)	0	0.75	

Figure 4-4 shows the effect of "multiplying" those three input variables values to produce one scenario for each combination, that is, a total of seven scenarios.

Figure 4-4. Scenario Table After a Multiply Variables

	Base Project	Scenario 1	Scenario 2	Scenario 3	Scenario 4	Scenario 5	Scenario 6	Scenario 7
Chassis Assembly : Absolute Position X (m)	0	0	0	0	0.25	0.25	0.25	0.25
Chassis Assembly : Absolute Position Y (m)	0.00254	0.00254	0.5	0.5	0.00254	0.00254	0.5	0.5
Chassis Assembly : Absolute Position Z (m)	0	0.75	0	0.75	0	0.75	0	0.75

Related Topics

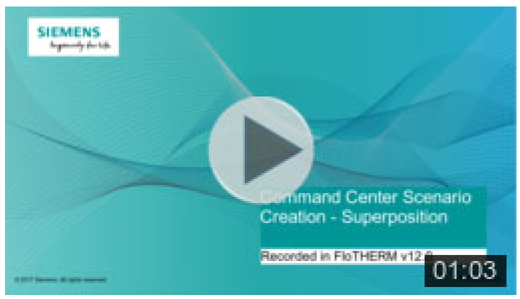
Multiply Variables Dialog Box

Creating Scenarios Such That There is a Unique Non-Zero Power Input Variable Value for Each Scenario

Superposition of power input variables in scenarios.

Video

Learn how to create scenarios that have unique non-zero input variable values:



Procedure

1. Choose **Create Scenarios** > **Superposition** to open the Superposition dialog box.

   A list of available power input variables is shown in the Power Input Variables pane.

2. Select one or more power input variables and click >>

   The selected input variables are moved to the Selected Power Input Variables pane.

3. Select the type of superposition.

   You can choose to inherit base model values or define a default value.

4. Click **OK**.

Results

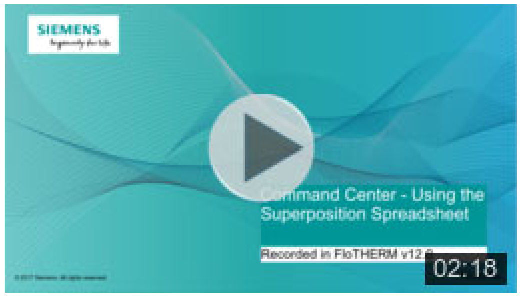
Scenarios are created where a power value is defined for a single power input variable.

The value of the other power input variables in the scenario is either:

- 0 W, if the input variable has been selected for superposition.
- inherited from the previous scenario, if the input variable has not been selected for superposition.

Tip
You can edit the power values directly in the Scenario Table.

Learn how to use the supplied SuperPosition spreadsheet to predict temperatures from superposition scenario results:



Examples

Figure 4-5 shows the result of selecting five power input variables and defining a superposition power of 1 W.

Figure 4-5. Example of User-Defined Power Superposition

	Base Project	Scenario 1	Scenario 2	Scenario 3	Scenario 4	Scenario 5
Component A : Power (W)	5	1	0	0	0	0
Component B : Power (W)	100	0	1	0	0	0
Component C : Power (W)	0	0	0	1	0	0
Component D : Power (W)	5	0	0	0	1	0
Component E : Power (W)	10	0	0	0	0	1

Figure 4-6 shows the result of selecting five power input variables and a Base Model Power superposition.

Figure 4-6. Example of Base Model Power Superposition

	Base Project	Scenario 1	Scenario 2	Scenario 3	Scenario 4	Scenario 5
Component A : Power (W)	5	5	0	0	0	0
Component B : Power (W)	100	0	100	0	0	0
Component C : Power (W)	0	0	0	0	0	0
Component D : Power (W)	5	0	0	0	5	0
Component E : Power (W)	10	0	0	0	0	10

Related Topics

Superposition Dialog Box

Specifying Input Variable Value Limits for a DoE Study

For numeric input variables, the program must know the upper and lower value bounds.

Restrictions and Limitations

- There is a size restriction on the combined number of design parameters and input constraints, see "DoE Size Restriction" on page 102.

Prerequisites

- The input variable must have been activated, see "Activating Input Variables" on page 48.

Procedure

1. Select the input variable in the Input Variables Selection table.

   The Type selector defaults to **Default**.

   The value of the Base Project is shown in the adjacent Input Variable Values Definition table.

2. Change the Type to **Design Parameter**.

3. For numeric input variables, define a range of values by setting a Minimum and Maximum Value.

   Boolean and enumeration type variables do not have user-defined value bounds, the possible values are all those that are valid and are implied.

4. Repeat for all Design Parameter input variables.

Results

When you have set up all Design Parameter input variables, specify any Dependent Variable input parameters as described in "Creating a Dependent Variable" on page 57.

Related Topics

Setting Up and Creating a DoE

Creating a Dependent Variable

Use when you want a variable to vary according to one or two other variables.

Restrictions and Limitations

- Only applies to numeric values.

- Dependencies can only use simple linear, multiplication, or division functions that take one of the following forms:
  - $y = mx_1 + c$
  - $y = x_1 * x_2 * c$
  - $y = c/x_1$
  - $y = c * x_2/x_1$

  where y is the value of the dependent variable, x_1 and x_2 are variables on which the variable is dependent, and m and c are constants.

Procedure

1. Select the input variable in the Input Variables Selection table.

   The Type selector defaults to **Default**.

   The value of the Base Project is shown in the adjacent Input Variable Values Definition table.
2. Change the Type to **Dependent Variable**.
3. Click the **Click to Edit** button to open the Dependent Variable dialog box.

   Available input variables are listed in the right-hand pane of the dialog box.
4. Double-click an input variable to copy it to the left-hand pane where a dependency can be defined.
5. Use the keypad in the Dependent Variable dialog box to create a function.

Results

A Constant Term is added to the function to make sure that the base project value of the dependent variable after calculation corresponds to the actual solved value.

Related Topics

Setting Up and Creating a DoE

Assigning Heat Source Rates Across the Project (AHR)

To save time spent manually changing sources individually, you can de-rate and pro-rate multiple heat sources using the Assigned Heat Source Rating user variable.

Restrictions and Limitations

- When AHR is applied, the following variables are affected:
    - Planar Sources (total or per unit area)
    - Volume Sources (total or per unit volume)
    - Thermal attributes (total or unit area)
    - Assigned power to Die SmartParts
    - Assigned power to PCBs
    - Junction power in the Compact Component SmartPart

  The following objects and settings are *not* affected by AHR:

    - Fixed temperature and Joule Heating thermal attributes
    - Linear and fixed-value heat sources
    - Heat introduced through flow objects: Fans, Recirculation Devices, Fixed Flows, Racks, and Coolers.
- Generally, AHR is not applied to powers that have been explicitly set as input variables. An exception to this is when thermal or source attributes are used as input variables, that is, different attributes are used in different scenarios. In such cases, AHR is applied to the power values of the attributes.
- Care is required when using AHR for *transient* models. Some example behaviors are described:
    - AHR is the only variant:

      Thermal and source attributes will have their base values AHR-applied but their transient attachments will remain attached and unchanged.
    - AHR is varied in combination with other powered objects, except thermal and source attributes, as input variables:

      Powered SmartParts do not have AHR applied while creating the scenarios.
    - AHR is varied along with thermal attributes and source attributes:

      For thermal and source attribute values, AHR is, correctly, not applied to the base values of the attributes. However, the transient attachments will remain attached and will not be AHR-applied.

Procedure

1. Open the Input Variables view.
2. Select the **User Variables** tab.

This is the rightmost tab. If it is not visible, click > to move the tabs to the left.

3. Check the Assigned Heat Source Rating check box.

   The default value is 100 percent.

   Assigned Heat Source Rating is added to the list of input variables.

4. Add input variable values for different scenarios to represent percentage values of the original, for example, a value of 150 is 150% of the original.

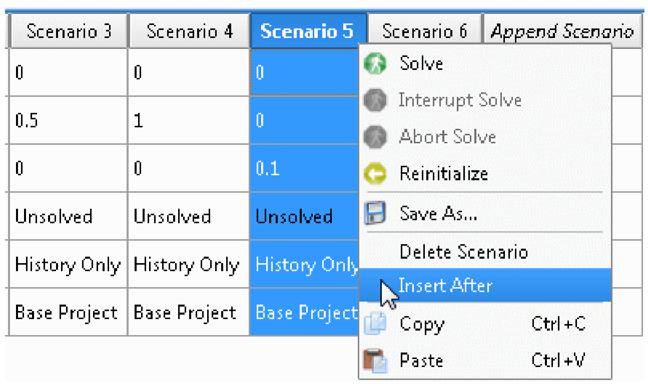
Inserting a Scenario After a Scenario

You can create a new scenario project by inserting a selected scenario into the next scenario project position.

Procedure

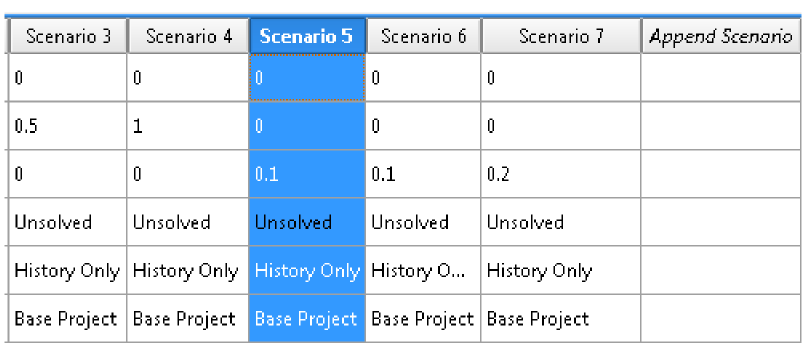
1. Right-click over the scenario header you want to insert after.

2. Choose **Insert After** from the context-sensitive menu.



Results

The selected scenario, including its input variables values is copied to the next scenario. A new scenario is created at the end.



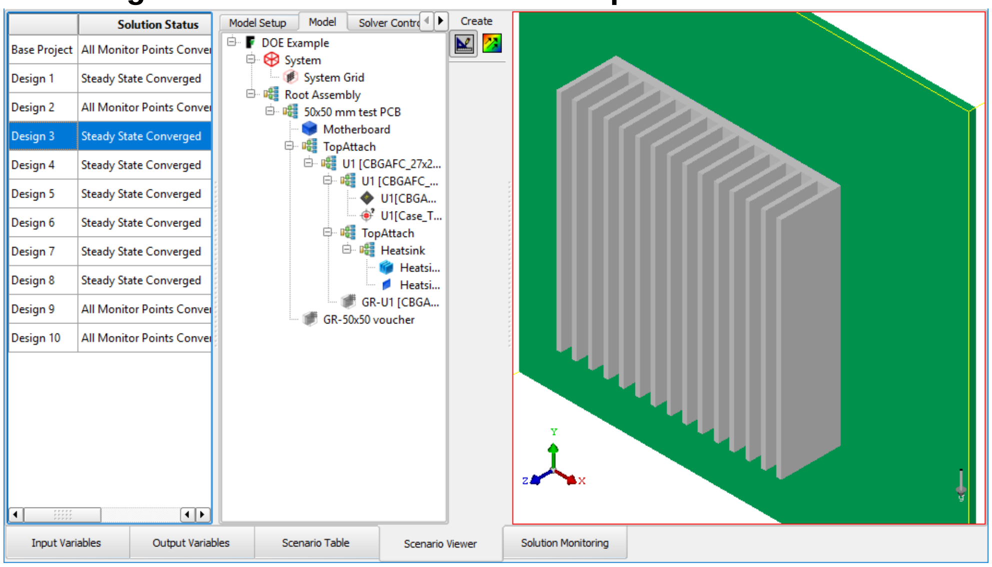
Inspecting the Physical Layout of a Scenario

Visual inspection of scenario layouts is recommended before running the solver.

Procedure

1. Open the Scenario Viewer.
2. Make sure the viewer is in Create mode.
3. Select each scenario in turn and visually inspect the layout in the drawing board.

Figure 4-7. Scenario Viewer Example in Create Mode



4. Check for overlapping objects and the grid distribution (G).

Working With Output Variables

Select the output variables to compare results from different scenarios.

Activating Output Variables

Output variables are activated by selecting them from those available.

Procedure

1. Open the Output Variables view.
2. Depending on which type of output variable you want to select, click the **Model**, **Field Summary** or **Metrics** tab.

   Output variable activation check boxes are on the left of parameters. For objects in the data tree (**Model** tab), you will have to open a property sheet to view the check boxes.

   Tip
   You can activate multiple output variables by selecting multiple data tree objects. The variables listed in the property sheets are those that are common to the selected objects.

3. Check the appropriate check boxes.

Results

Activated variables are added to the Output Variables Selection table.

Related Topics

Output Variables View

Defining a Cost Function

A cost function is a weighted sum of a number of output variables which is used to determine the optimum design.

Restrictions and Limitations

- Defined cost functions *are not scaled* to compensate for any subsequent changes of units. You will have to adjust the cost weightings to allow for such changes.

Procedure

1. In the Output Variables view, select an output variable and check the Include in Cost Function check box. Alternatively, you can check a check box in the Output Variables Selection table, these are in the Include in Cost Function? column .

   Additional fields are displayed for choosing the type and weight of cost functions to be applied to this variable. The current cost function is shown underneath.

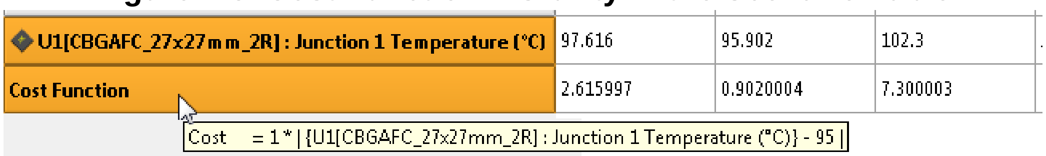
2. Choose a Cost Type of Linear or Target.

3. Enter the Cost Weighting and, if Target has been selected, the Cost Target.

Results

The cost function is visible in the Scenario Table as hover text over the Cost Function row, for example, see Figure 4-8. In this example the cost function compares a compact component junction temperature with a target temperature of 95 degC.

Figure 4-8. Cost Function Visibility in the Scenario Table



Related Topics

Cost Function

Command Center Scenario Panels and Dialog Boxes

The following panels and dialog boxes are used when building and saving scenario projects.

Linear Sweep Dialog Box

To access: **Create Scenarios** > **Linear Sweep**

Use this dialog box to apply a sweep of input variable values through any existing scenarios.

Objects

Field	Description
Input Variable Selection	
Input Variables	A list of available numeric input variables.
Selected Input Variables	Input variables for which the linear sweep function will apply.
Linear Variation	
Step Size	The increment/decrement values applied to each step.
Number Of Positive Steps	The number of scenarios that will have values increasing from the Base Project value. **Note:** If there are more scenarios than steps then the largest value is used in the extra scenarios.
Number Of Negative Steps	The number of scenarios that will have values decreasing from the Base Project value. **Note:** If there are more scenarios than steps then the smallest value is used in the extra scenarios *unless* one or more positive steps have been defined.

Usage Notes

- Negative steps are applied before positive steps.

Related Topics

Defining a Linear Sweep of Input Variable Values

Multiply Variables Dialog Box

To access: **Create Scenarios** > **Multiply Variables**

Use this dialog box to set up a sequence of scenario projects that covers every possible combination of selected varying input variables.

Objects

Field	Description
Input Variable Selection	A list of all the variables previously selected for varying, excluding those that have been moved to the Selected Input Variables list.
Selected Input Variables	A list of variables for which every combination is required, up to a maximum of four entries.

Related Topics

Creating Scenarios That Cover Every Possible Combination of Input Variables

Add Variables Dialog Box

To access: **Create Scenarios** > **Add Variables**

Use this dialog box to create scenario projects that contain only one variant of a variable at a time.

Objects

Field	Description
Input Variable Selection	A list of all the variables previously selected for varying, excluding those that have been moved to the Selected Input Variables list.
Selected Input Variables	A list of variables for which every combination is required, up to a maximum of four entries.

Related Topics

Creating Scenarios That Contain Only One Variant of a Variable

Superposition Dialog Box

To access: **Create Scenarios** > **Superposition**

Use this dialog box to create scenarios where power input variables are uniquely given power values.

Objects

Field	Description
Input Variable Selection	
Power Input Variables	A list of all the active input variables that have a definition of power, excluding those that have been moved to the Selected Power Input Variables list.
Selected Power Input Variables	Power input variables that have been selected for superposition.
Superposition	
Superposition Type	Choose between: • User Defined Power − use a defined power value as the superposition power value. • Base Model Power − use the power value in the base model as the superposition power value.
Power	(User Defined Power) A superposition power value.

Related Topics

Creating Scenarios Such That There is a Unique Non-Zero Power Input Variable Value for Each Scenario

Chapter 5
Scenario Data

This chapter describes how to change scenario data in the Scenario Table.

Renaming a Scenario

You can give scenarios more meaningful names than the defaults.

Procedure

1. Open the Scenario Table and double-click the scenario name in the column header.

   The table cell becomes editable.

2. Over-type with a new name and press the Enter key.

Results

The scenario name is updated in the other Command Center views.

Editing Input Variable Values in the Scenario Table

In addition to editing input variable values in the Input Variables view, you can also edit these values in the Scenario Table.

Restrictions and Limitations

- Dependent Variables cannot be edited from the Scenario Table.

Procedure

1. Open the Scenario Table and double-click in the input variable value cells.

   The table cell becomes editable.

2. Editing depends on the type of value:

   - A numeric value can be over-typed with a new value followed by pressing the Enter key.
   - Non-numeric values are selectable from a dropdown list.

Results

The value is updated in the Input Variables view.

Pasting Values Directly Into the Scenario Table

Values that have been copied to the clipboard can be pasted into the Scenario Table.

Note

This procedure can be used in conjunction with “Exporting All or Part of the Scenario Table to a Spreadsheet” on page 83 to make changes to a Scenario Table.

Restrictions and Limitations

- You cannot paste the following values into the following Scenario Table cells:
  - Dependent Input Variables Values
  - Solution Status
  - Output Variables Values
  - Cost Function Values
- The pasted value must be a valid value for the table cell.

  Enumeration type values are case sensitive. For example, a Store Results? value of “full” is not valid.
- When pasting a group of values from a spreadsheet, the values must be in rows and columns that match those of the scenario table. If you do want to change a value in the group, then either include the same value in the spreadsheet cell or leave the spreadsheet cell empty.

Prerequisites

- Value(s) must have been copied to the clipboard.

Procedure

1. Open the Scenario Table.
2. What you do depends on what data is held in the clipboard:

If you want to...	Do the following:
Paste a single value.	Select the table cell.
Paste a group of values.	Select the top-left cell of the group.
Paste a complete table.	Select the top-left cell of the table.

3. Press Ctrl+V.

Tip

You can create a new scenario by pasting values into the "*Append Scenario*" column. Any undefined values are inherited from the last scenario.

Results

Valid pasted values replace existing values. If a pasted value is invalid or null, then the existing value is retained.

When an input value is replaced with a different value, the solution status changes to Stale.

Deleting Variables From the Scenario Table

You can delete individual or multiple-selected input and output variables from scenarios directly from the scenario table.

Procedure

1. Select or multiple-select input and/or output variables in the scenario table by clicking or Ctrl+clicking in the title cell.

   The respective row(s) become highlighted.
2. Right-click in a selected variable title cell and choose **Delete Variable**.

Deleting a Scenario

Scenarios can be deleted from the Scenario Table, Scenario Viewer or Solution Monitoring view.

Procedure

You have a choice.

If you want to...	Do the following:
Delete a scenario from the Scenario Table, Scenario Viewer or Solution Monitoring view.	1. Select the scenario. 2. Choose **Scenario** > **Delete Scenario**.
Delete a scenario from the Scenario Table (alternative option).	1. Right-click in a scenario title cell. 2. Choose **Delete Scenario**.

Results

The scenario is deleted.

Note
A **Delete Scenario** operation can be undone.

Inserting After a Scenario Project in the Scenario Table

You can insert a new scenario project after the currently selected project which is a copy of the selected project.

Procedure

1. Open the Scenario Table and right-click in the scenario title cell.
2. Choose **Insert After**.

Results

A new scenario with the same values as the selected scenario is inserted after the selected scenario. If subsequent scenarios have default names, for example “Scenario <*n*>” then these are renumbered.

Selecting Results to Initialize From

To ensure that a scenario converges quickly it is advisable to initialize the starting conditions from as similar a project as possible. You can initialize the scenarios either as a group or individually.

Note

If the difference between the scenario output variables is small (for example, in the order of 1%), then you are strongly advised to use either initialize from no project or converge the base project more accurately (for example, reduce the termination criteria for temperature).

Procedure

1. In the Scenario Table, in the Initialize From row, double-click the scenario cell.

   A dropdown list is opened. By default, scenarios are initialized from the results of the **Base Project**.

2. Select the scenario whose results are to be used for initialization.

   You cannot select a scenario that would be solved after the scenario, that is, any scenarios on the right of the scenario in the Scenario Table. This is to prevent a deadlock.

   Tip

   To initialize all scenarios in a single operation, right-click in the Initialize From title cell, and choose one of: **All From Base Project**, **All From No Project** or **All From Previous**.

Storing All Solution Results

By default only residual and monitor point data are stored for each scenario, however, you can save the full results for display in the GDA.

Procedure

1. Open the Scenario Table and, in the Store Results? row, double-click a scenario cell to open a context-sensitive menu.

   History Only is the default and recommended option for scenarios; only residual and monitor point data are stored.

2. Choose **Full** to store all results for the scenario, including history data, derived properties and full field data.

   Tip

   To store *all* results for *all* scenarios, right-click in the Store Results? title cell and choose **All Full**; however, this may require a large amount of storage space.

   During a design optimization, the full results are stored for the best case.

Scenario File Format

The scenario file, *<solution_directory>/<project>/PDProject/scenario_file.xml*, is written to, and read from, by program.

This file contains definitions of all current scenarios in XML format.

Format

The file is divided into subsections:

Subsection	Description
<scenarios>	Contains general data of each scenario, such as the name, state, and initialized_from values.
<input_variables>	Contains definitions of each input variable.
<output_variables>	Contains definitions of each output variable.
<design_of_experiments>	Contains optimization definitions.
<charts>	Contains definitions of each chart.

Parameters

None.

Examples

An example of an input variable definition:

```
-<input_variable>
   -<variable_name property_name="absolutePositionZ" origin="model">
      -<geometry_name>
         -<geometry name="Root Assembly">
            -<geometry name="Heat Sink Assembly" position_in_parent="1">
               <geometry name="Heat Sink:1" position_in_parent="0"/>
            </geometry>
         </geometry>
      </geometry_name>
   </variable_name>
   <new_value property_name="type" new_value="default"/>
   <variable_values/>
</input_variable>
```

Chapter 6
Scenario Solution

The solution of Command Center scenario projects and control of the solvers.

Scenario Solution Status

A list of possible scenario solution statuses and their descriptions.

The solution status is displayed in the scenario table, the scenario viewer and in the scenario monitoring view.

Status	Description
Unsolved	A solution has not been attempted, the scenario has been re-initialized, or the scenario solve has been aborted.

Status	Description
Queuing (<*job_number*>)	The scenario has been nominated for solution, but the solve has not yet begun because of a limit on the number of simultaneous jobs, as specified by the Concurrent Solvers setting in the User Preferences dialog box.
Saving	Creating the model for the scenario.
Translating	A pre-condition to solving.
Pending Initialization	The scenario has nominated for solution and has been set to initialize from another scenario, but that scenario has not yet been solved.
Solving Radiation	Calculation of the Radiation Exchange Factors is in progress.
Solving	Solving is in progress.
Extracting Results	Solving has stopped and results are being made available.
All Monitor Points Converged	The solver has stopped because all monitor points are stable, that is, they are reporting the same temperature within the required accuracy for the specified number of iterations. The model may not have fully converged, but has stopped because of this condition, which has been activated by the Monitor Point Convergence For Temperature setting in the **Solver Control** tab.
Steady State Converged	The solver has stopped and the convergence criteria have been met.
Steady State Not Converged	The solver has stopped but the convergence criteria have not been met.
Transient Converged at each Timestep	The solver has stopped and the convergence criteria have been met at each time step.
Transient Not Converged at each Timestep	The solver has stopped but the convergence criteria have not been met at each time step.
Stale	One or more input variables have been changed for any scenario since this scenario was solved. Output variable values are shown in italics.
Interrupting Solve	A user Interrupt Solve request has been made and the solver is stopping.
Interrupted	The solve has stopped due to a user Interrupt Solve request.
Aborting Solve	A user Abort Solve request has been made and the solver is stopping.
Remote Solve Queued	Solving is queued for a remote machine.

Status	Description
Remotely Solving (<*remote_machine_id*>)	Solving is in progress on a remote machine.
Failed Sanity Error	Translation error.
Failed Radiation	Radiation calculation error.
Abnormal Solver Termination	Refer to the Messages window for more detailed information, for example, failure to get a solver license.
Statuses After Optimization:	
Optimum	Solved optimization reached.
Best So Far	Optimization maximum termination criteria reached.
Good Enough	The optimization Critical Cost Value, as defined in the Optimize dialog box, has been reached.
RSO Optimum	The optimum scenario after performing a Response Surface optimization.
Calibration Optimum	The optimum scenario after performing a Model Calibration.

Solving and Solution Operations

Use the following procedures when solving scenario projects.

Configuring Processor Resources for Solving Scenarios on a Local Machine

You can use solve scenarios in parallel and use multiple processors to solve each scenario.

Procedure

1. Choose **Edit > User Preferences** to open the Command Center User Preferences dialog box.

   The **Solver Configuration** tab is open by default and the selected Run Solvers On default is Local Machine.

2. Enter the number of scenarios that you want to be solved in parallel in the Concurrent Solvers field.

3. Enter the number of processors to use to solve each scenario in the Number Of Processors To Use field.

   Note
   Your local machine should have at least (<*Concurrent Solvers*> * <*Number Of Processors To Use*>) processors to avoid overloading.

4. Click **OK** to save the settings and close the dialog box.

   The configuration will be used the next time you solve.

Related Topics

Command Center User Preferences Dialog Box

Configuring Processor Resources for Solving Scenarios on Remote Machines

Solving Scenarios

You can solve all scenarios in one operation or solve individual scenarios one at a time.

Restrictions and Limitations

- The Input Variables and Output Variables views are unavailable during scenario solving.

Prerequisites

- Make sure that you have set the starting conditions and results to store for the scenario(s) to be solved, refer to "Selecting Results to Initialize From" on page 72 and "Storing All Solution Results" on page 73.
- Optionally, you can re-initialize scenarios:

  Note
  You cannot undo re-initialize.

  - To re-initialize all scenarios, choose **Project** > **Re-Initialize All**.
  - To re-initialize selected scenarios:
    i. In the Scenario Table, press Shift+click or Ctrl+click to select a number of scenarios
    ii. Right-click in the title cell of one of the selected scenarios.
    iii. Choose **Re-Initialize**.

Procedure

1. Perform one of the following:

If you want to...	Do the following:
Solve all scenarios.	Choose **Project** > **Solve All**.
Solve one scenario.	1. In the Scenario Table, right-click in the title cell of the scenario. 2. Choose **Solve**.
Solve a number of scenarios.	1. In the Scenario Table, press Shift+click or Ctrl+click to select a number of scenarios. 2. Right-click in the title cell of one of the selected scenarios. 3. Choose **Solve**.

The Solution Status is updated in the Scenario Table and Scenario Viewer as each scenario is solved.

You can monitor the progress of solutions graphically from the Solution Monitoring view.

2. While a solution is running, you can interrupt the solution and then continue solving, or you can abort the solution.

If you want to...	Do the following:
Interrupt the solving of all scenarios.	Choose **Project** > **Interrupt All**.
Interrupt the solving of selected scenarios.	1. In the Scenario Table, press Shift+click or Ctrl+click to select a number of scenarios. 2. Right-click in the title cell of one of the selected scenarios. 3. Choose **Interrupt Solve**.
Abort the solving of all scenarios.	Choose **Project** > **Abort All**.
Abort the solving of selected scenarios.	1. In the Scenario Table, press Shift+click or Ctrl+click to select a number of scenarios. 2. Right-click in the title cell of one of the selected scenarios. 3. Choose **Abort Solve**.

Results

Output variable values and cost function values are shown in the Scenario Table when each scenario has been solved.

The Scenario Viewer opens in Analyze mode following a solve.

Related Topics

Solution Monitoring View

Selecting the Scenario With the Maximum or Minimum Output Variable or Cost Function Value

Simple selection of scenarios based on solution results.

Procedure

1. Right-click over the output variable or cost function title cell in the scenario table.
2. Choose **Select Min Value** or **Select Max Value**.

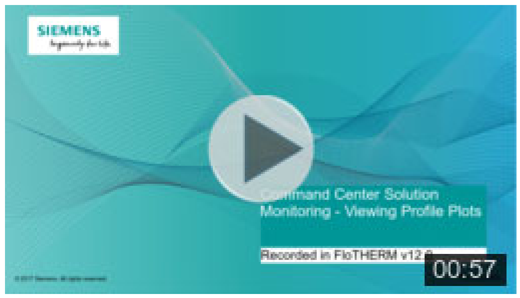
   The relevant scenario becomes the currently-selected scenario.

Editing Solution Plots

Residual and monitor point plots displayed in the Solution Monitoring view are synchronized with selections made in the Project Manager.

Video

Learn how profile plots are made available in the Command Center:



Related Topics

Solution Monitoring View

Running Scenario Solutions in Batch Mode

If the project and scenarios have already been set-up in previous Simcenter Flotherm sessions, the solutions can be run in batch mode.

Procedure

1. Start a command prompt window.
2. Type:

   <*install_dir*>\flosuite_v<*version*>\flotherm\WinXP\bin\flotherm -b <*project*> -c

   where:

   <*install_dir*> is the Simcenter Flotherm installation directory.

   <*version*> is the flosuite version number

   -b initiates batch mode

   -c solves all the unsolved scenarios for this project. If the Base Project is not fully solved, then it is also solved.

   <*project*> is the name of the project to be solved.

   For the full batch command syntax, see Solving in Batch Mode in the *Simcenter Flotherm User Guide*.

For details on canceling solutions in batch mode, see Interrupting a Batch Solution in the *Simcenter Flotherm User Guide*.

Also, see "Distributed Solving of Scenarios" on page 85 for solving across a distributed network to make the most of available resources.

Displaying Scenario Table Row Data in Charts

A row of Scenario Table data can be more easily compared by creating a chart.

Procedure

1. Choose **Chart** > **Create**.
2. Select an **X-Axis Variable** from the dropdown list.

   You can choose Scenario Number, Cost Function (if one is defined), or any input or output variable.
3. Select one or more of the Series options. These are plotted on the y-axis.

   You can change the color of the chart points or you can color by variable value.
4. It is recommended that you give the chart a meaningful title. By default, the title is assigned the name Chart <*n*> where <*n*> is the chart sequence number.
5. Click **Close** to save the chart.

Results

Figure 6-1 is an example of a Results Comparison Chart showing the numbers of heat sink fins for each scenario.

Figure 6-2 shows how the chart is selected to be opened from the Command Center menu.

Figure 6-1. Results Comparison Chart Example

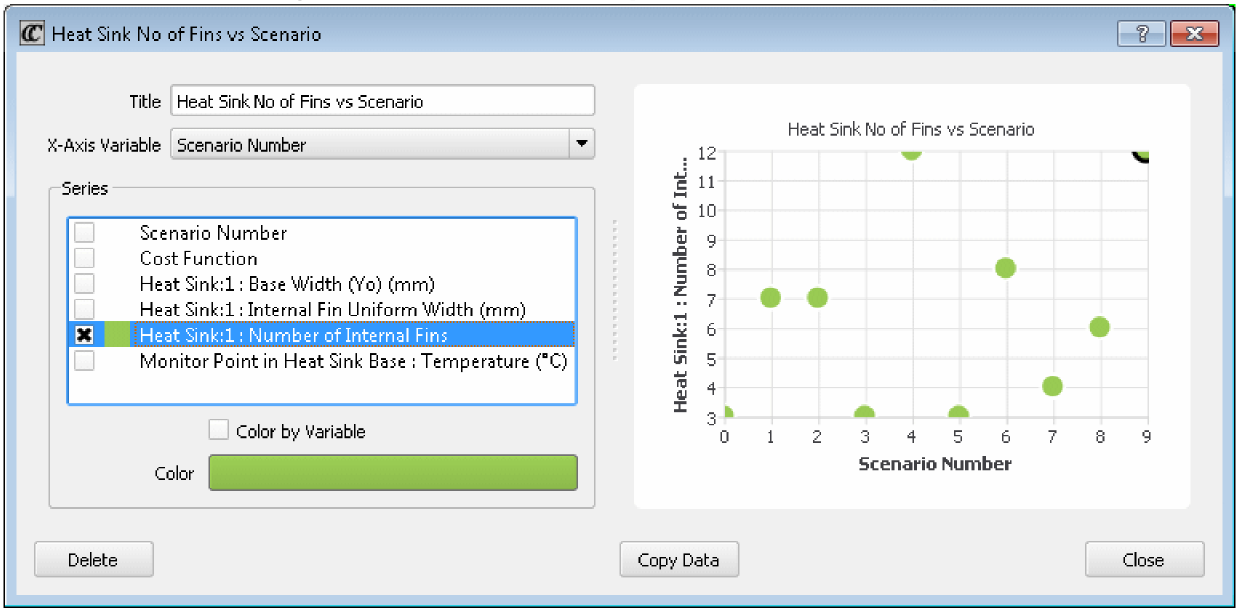
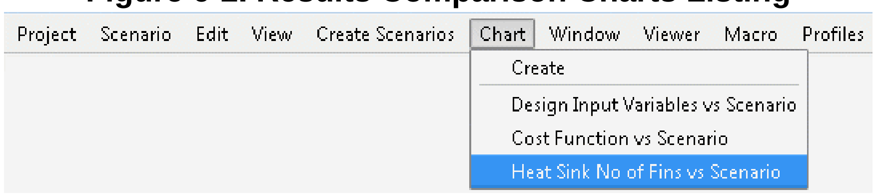


Figure 6-2. Results Comparison Charts Listing



Tip

Click **Copy Data** to copy the chart data for pasting into a spreadsheet.

Related Topics

Results Comparison Chart

Exporting All or Part of the Scenario Table to a Spreadsheet

Scenario Table cells can be copied and pasted into third-party spreadsheet software, such as Microsoft® Excel.

Note

This is the reverse of “Pasting Values Directly Into the Scenario Table” on page 70.

Procedure

1. Open the Scenario Table.

2. What you do depends on which data you want to export:

If you want to...	Do the following:
Copy one or more cells.	Select the cell(s).
Copy a complete row or column, including the header.	Select the row/column header.
Export a complete table.	Select the top-left cell of the table.

3. Press Ctrl+C.

4. Open a spreadsheet application, select a suitable part of a worksheet and paste the contents of the clipboard.

Results

The exported Scenario Table cells are copied in the spreadsheet. Figure 6-3 shows an example where a complete Scenario Table has been exported.

Figure 6-3. Exported Scenario Table in Microsoft Excel

	A	B	C	D	E
1		Base Project	Scenario 1	Scenario 2	Scenario 3
2	Heatsink : Base Thickness (Zo) (mm)	3	4.733	2.067	4.067
3	Heatsink : Internal Fin Height (mm)	8	6.967	17.73	10.93
4	GR-U1 [CBGAFC_27x27mm_2R] : Size Z (m	14.36	15.06	23.16	18.36
5	GR-50x50 voucher : Size Z (mm)	15.96	16.66	24.76	19.96
6	Typical Interface : Rsurf-solid (°C in^2/W)	1	2.060001	0.4267004	4.837002
7	Solution Status	All Monitor Points C	All Monitor Points C	All Monitor Points C	All Monitor Points
8	Store Results?	Full	History Only	History Only	History Only
9	Initialize From	No Project	Base Project	Scenario 1	Scenario 2
10	U1[CBGAFC_27x27mm_2R] : Junction 1 Te	97.598	99.65	88.184	100.21
11	Cost Function	2.598	4.65	-6.816	5.21

Distributed Solving of Scenarios

You can distribute the solving of scenarios between remote machines on a network.

FloVolunteer

You can use FloVolunteer to manage the project solution across the network.

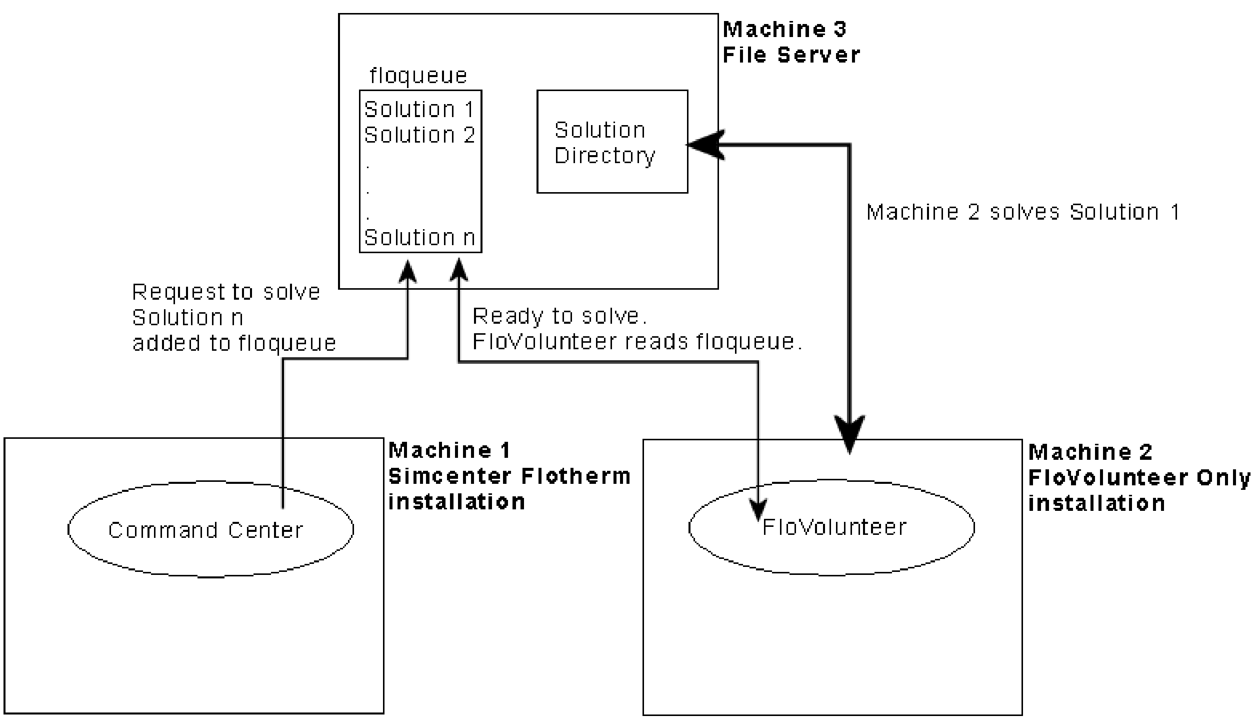
FloVolunteer requires:

- A network of machines all capable of running the Simcenter Flotherm exchange factor generator and solver
- Access to a common solution data directory (which can be a multiple level directory) containing all the projects to be solved.

Any Command Center requesting a solve submits jobs to a queue on the server machine. All designated FloVolunteer systems must be configured as a background service. These systems are then available for processing the jobs listed in the queue.

Figure 6-4 shows a simple process example with three machines connected in a network.

Figure 6-4. Network Example Using FloVolunteer



Machine 3 is a file server, holding the solution data directory and the job queue, floqueue.

1. The Command Center running on Machine 1 requests a solution.
2. This request is added to floqueue on Machine 3.
3. In parallel, FloVolunteer running on Machine 2 requests solver jobs from floqueue.
4. floqueue submits a solver job to Machine 2, which runs the solver and writes the results to the solution data directory.
5. When FloVolunteer finishes solving the job, it notifies floqueue.
6. The Command Center reads the results from the solution data directory on Machine 3.

Configuring Processor Resources for Solving Scenarios on Remote Machines

On remote machines running FloVolunteer as a background service, you can use solve scenarios in parallel and use multiple processors to solve each scenario.

Restrictions and Limitations

- To avoid overloading the machine, make sure that the number of concurrent scenarios multiplied by the number of processors is less than or equal to the total number of processors available.

Procedure

1. Define the number of concurrent scenarios to solve by using the FLOVOLUNTEERMAXJOBS environment variable.

   For more details, refer to Environment Variables in the *Simcenter Flotherm User Guide*.
2. Define the number of processors to use to solve each scenario by using the FLOVOLUNTEERPARALLELTHREADS environment variable.
3. Restart the machine so that any changes to the environment variables take effect.

Related Topics

Configuring Processor Resources for Solving Scenarios on a Local Machine

Solving Scenarios Across a Network

When configured, remote machines running FloVolunteer will solve scenarios.

Prerequisites

- The Simcenter Flotherm solution data directory and the job queue directory must be located on a shared drive accessible by all the users of the Command Center and FloVolunteer.
- The appropriate FloVolunteer environment variables must be defined on the remote machines. For details, refer to the FloVolunteer chapter of the *Detailed Installation Instructions*.
- FloVolunteer must be running on the remote machines as a background service. For details, refer to the FloVolunteer chapter of the *Detailed Installation Instructions*.
- The FLOHOME and FLOQUEUELOCATION environment variables must be defined in file *<install_dir>\flosuite_v<version>\flotherm\WinXP\bin\flotherm.bat* file on the Command Center machine. The file has commented-out lines for the definition of these variables.

Note

For each instance of FloVolunteer, you can control the number of concurrent scenarios that are solved and the number of processors used to solve each scenario, see “Configuring Processor Resources for Solving Scenarios on Remote Machines” on page 86.

Procedure

1. Start up Simcenter Flotherm, load the project from which you want to solve scenario projects and start the Command Center.
2. Choose **Edit > User Preferences** to open the Command Center User Preferences dialog box and check that the **Remote Solver On** selection is **Remote Machine(s)**.

   If it is not then FLOQUEUELOCATION has not been correctly defined on the Command Center machine.
3. Start solving the scenarios.

Results

Solution requests are submitted to the queue file.

As these jobs are picked up by the remote machine and solved, the solution status in the Scenario Table changes to:

Queuing (*<job_number>*)

and then:

Solving (*<remote_machine_id>*)

Tip

If the solution status remains at Queuing, then check that FloVolunteer is running and that FLOQUEUELOCATION and FLOHOME are set up correctly on remote machines.

Tip
If the solution status changes to Failed Solver Error (<*remote_machine_id*>), then check that the remote machine has access permissions to the solution directory. You may need to move the solution directory.

Related Topics

Command Center User Preferences Dialog Box

Viewing Scenario Results

Use the Scenario Viewer in Analyze mode to view results for each solved scenario.

Restrictions and Limitations

- Results tables are not available. To view results tables for a scenario, save the scenario as a project then load the project into the Project Manager.

Prerequisites

- Scenarios must have been solved with Full results.

Procedure

1. Open the Scenario Viewer.
2. Press F10 or click the Analyze icon to enter Analyze mode.

   The Results tree is opened.
3. Select a solved scenario.
4. Use the post-processing functions such as plots, annotations, and animations as when analyzing results in the Project Manager.

Results

The Results tree is synchronized with the Project Manager Results tree, that is, each tree is updated when changes are made to the other tree.

Related Topics

Viewing Results [Simcenter Flotherm User Guide]

Saving Scenarios

Scenarios can be saved for import into Simcenter Flotherm.

Saving Scenarios as Part of the Current Project

Saving the current scenarios so they will be reloaded to the Command Center window when the project is reloaded and the Command Center is opened. It is recommended that you do this regularly to protect your work.

Procedure

1. Make sure that the Command Center window is in focus.
2. Press Ctrl+S, or choose **Project > Save**, or click the **Save** icon.

Exporting Scenarios in PDML Files

You can export all scenarios to a folder where each scenario is captured in a separate PDML file.

Procedure

1. Choose **Project > Export All Scenarios** to open the PDML Save Directory dialog box.
2. Select the directory to hold the PDML files and click **OK**.

Results

A PDML file is created for each scenario, including the base project.

The name for the base project file is:

<base project name>_Base Project.pdml

The names for the other scenarios follows the convention:

<base project name>_<scenario name>.pdml

If any of the PDML scenario files already exist, they are overwritten.

Saving Scenarios as Projects

You can save scenarios as projects, which can be loaded into Simcenter Flotherm independently. Full results and post-processing features, should they exist, are saved with the project.

Procedure

1. Select a scenario.
2. Choose **Scenario > Save As**.

   The Scenario Save Project As dialog box is opened with the default project name of *<base project name>_<scenario name>*.
3. If required, change the default name by overtyping. Click **OK**.

Results

The scenario is saved as a project in the Simcenter Flotherm solution directory and can be loaded into Simcenter Flotherm.

If full results were stored, then these are saved with the project. If post-processing features, such as plots and annotations, had been created for the scenario, then these are also saved with the project.

Related Topics

Scenario Save Project As Dialog Box

Scenario Dialog Boxes

The following dialog boxes are used when running and monitoring the solver.

Command Center User Preferences Dialog Box

To access: **Edit > User Preferences**

Use this dialog box to control the solution strategy for multiple scenario projects.

Objects

Object	Description
Solver Configuration tab	
Run Solvers On	• Local Machine − Runs the solver(s) on the machine that the Command Center is running on. The Local Machine settings are saved and reused in new Command Center sessions. • Remote Machine(s) − Runs the solvers on remote machines connected to the local network using FloVolunteer. This option is unavailable if a queue server (FLOQUEUELOCATION environment variable) for the FloVolunteer is not configured on the network.
Concurrent Solvers	(Local Machine) The number of scenarios to be solved concurrently. If there are N processors on your machine, the Command Center can solve up to N scenario projects at the same time on it. If you request to solve more scenario projects than there are processors, they will still all be solved at the same time, but it may take longer as they will be competing for resources. By default, one scenario project per processor is solved on your local machine.
Number Of Processors To Use	(Local Machine) The number of processors to use to solve each scenario. **Note:** Make sure you do not overload the machine. Ensure that the number of solvers (specified by the Concurrent Solvers field) multiplied by the number of processors specified here is less than or equal to the number of processors on your machine.
Results tab	
Number of Significant Figures Displayed	The number of significant figures shown in output variable and cost function values in the Scenario Table. Default and maximum = 7.

Related Topics

Configuring Processor Resources for Solving Scenarios on a Local Machine

Solving Scenarios Across a Network

FloVolunteer

Results Comparison Chart

To access: **Chart > Create or <*chart_title*>**

Use these charts for quick interpretation of the scenario project data. The results and input variables for different scenario projects can be compared on the same chart.

Note

A Correlation Chart (**Chart > Correlation Chart**) is created when you click a Correlation Matrix cell, see "Correlation Matrix" on page 125. The variables plotted are those that define the selected matrix cell.

Description

Objects

Field	Description
Title	Sets the title for the chart. By default, each new chart is named Chart <*n*> where <*n*> is sequence number.
X-Axis Variable	A dropdown list includes Scenario Number, Cost Function (if defined) and all input and output variables.
Series	
Series Selection List	Check boxes for Scenario Number, Cost Function (if defined) and all input and output variables. When checked, the values are plotted on the Y-axis of the chart. Color boxes act as a legend for the chart.
Color by Variable check box	Check this check box if you want chart points to appear in colors determined by values of the same or a different variable.
Color	(Color by Variable unchecked) Click the color bar to open the Modify Color dialog box.
Color by Variable dropdown list	(Color by Variable checked) The variable used to color chart points. A legend on the chart shows the range of values in a color spectrum from blue (lowest) to red (highest).
Copy Data	Click to copy values to the clipboard. The clipboard contents can then be pasted into a spreadsheet.

Usage Notes

- Chart points are highlighted for the currently-selected scenario. When a chart point is selected, the associated scenario becomes the currently-selected scenario.
- You can zoom in and out of a profile plot using the mouse scroll bar.

  To define a zoom-in area, left-click, drag then release. To return to full view, right-click.

- Chart point values are shown as hover text: X for the X-axis variable value, Y for the Y-axis variable (series) value and, if used, C for the value of the variable used to color a chart point.

  Double-click a point to lock the hover text in place. To release the hover text, double-click the point or text box.

Related Topics

Displaying Scenario Table Row Data in Charts

Scenario Save Project As Dialog Box

To access: Select a scenario, then choose **Scenario** > **Save As**

Use this dialog box to save the selected scenario project as a new Simcenter Flotherm project.

Objects

Field	Description
Project Name	The name of the project that will be created from the currently-selected scenario. The default name assigned is: *<base project name>_<scenario name>* but this can be edited by over-typing.

Related Topics

Saving Scenarios as Projects

Chapter 7
Design Optimization

The Command Center uses three design optimization techniques (DoE, SO, and RSO) to provide as much information as possible about the influence of input variables on output variables.

Experimental design methods are widely used in research as well as in industrial settings, however, sometimes for very different purposes.

The primary goal in scientific research is usually to show the statistical significance of an effect that a particular factor exerts on the dependent variable of interest.

In industrial settings, the primary goal is usually to extract the maximum amount of information regarding the factors affecting a product or process from as few (costly) observations as possible.

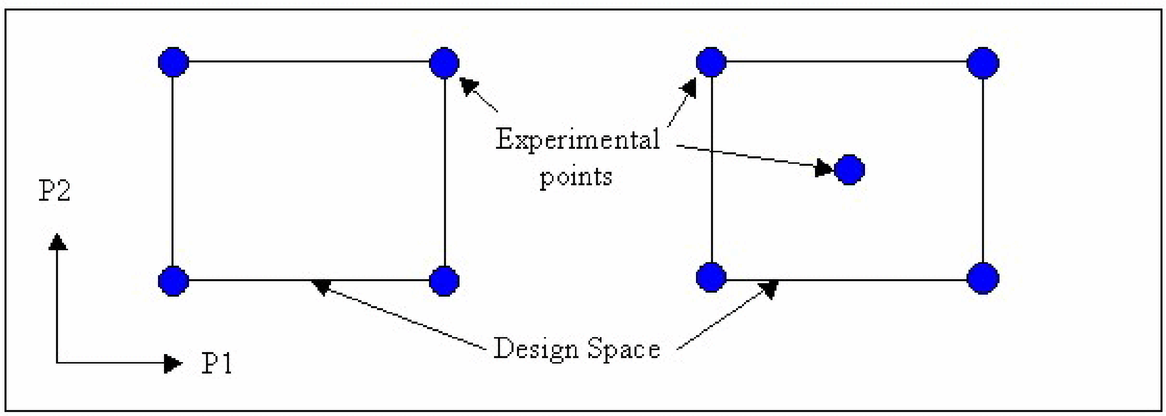
Design Space

The Command Center optimization techniques are used to explore a Design Space, that is, a conceptual space covering a range of values for the design variables.

Traditional DoE techniques used in the design of physical experiments generally explore the extremities and, occasionally, the center of a design space. Maximizing the difference in the input variables in the experiment makes it easier to determine the influence on the output variables, given that the output variables are subject to random and systematic experimental error, requiring the experiments to be repeated many times before a statistically valid result is obtained. It also makes it easier to determine any variation in the systematic error over the design space.

If two parameters, denoted by P1 and P2 are varied, the design space can be visualized as a box.

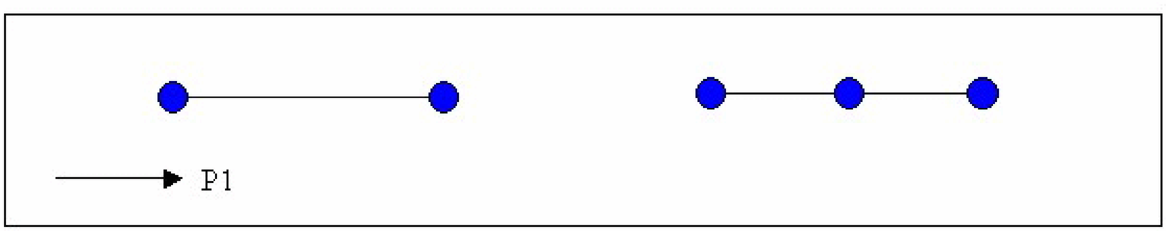
Figure 7-1. Design Space as a Box



Although the experimental designs above contain only 4 or 5 experimental conditions, perhaps 40 or 50 will be performed to gain knowledge about the experimental error.

One disadvantage of this approach is that if the output variable of interest has very little or no dependence on one of the input variables, say P2, then the results collapse in one direction.

Figure 7-2. Collapsed Design Space



Therefore, the four experiments to explore the extremities of the design space above yield only two pieces of useful information.

Numerical experiments do not suffer from random experimental error, since repeating exactly the same computer run leads to exactly the same result, see “Numerical Issues” on page 101. Consequently, each experiment need only be performed once. For the same number of experiments (40 or 50) a much more elaborate experimental design is possible.

Experimental designs for numerical experiments should be both space filling and provide maximal information. Space filling means that the experiments should be spread through the design space. Maximal information means that the data do not collapse if the output variable is insensitive to one of the input variables.

Design of Experiments (DoE)

A Design of Experiments (DoE) creates a number of experiments (scenario projects) where the selected input variables are varied within a defined range.

DoE Strategy

Before forming a DoE, you should have a strategy. The following subsections have been written for your guidance.

Consider Output Variables

It is worth thinking about what combination of output variable values are considered to be the optimum design, because, in general, trade-offs will need to be made.

The Cost Function used during sequential optimization can only contain output variables, so make sure you output all the quantities you require.

“Minimizing the temperature rise of all components” can be expressed as:

$$\textit{Optimum Design} = \text{minimum sum of } \Delta T_{C1} + \Delta T_{C2} + \ldots + \Delta T_{CN}$$

However, this may be too simple a view. The design that satisfies this statement may give a relatively low temperature rise for all components except one, which may be very hot.

Choose a Small Number of Input Variables

Choosing a small number of input variables to vary in one experimental design is desirable for the following reasons:

- If the output variable is insensitive to one of the input variables this information is found in the minimum number of runs.
- It is easier to deduce this from the runs performed.
- Two input variables is perhaps ideal, as this enables their effect on the output variables to be visualized graphically as a response surface.

Investigating the influence of other input variables can then be undertaken as a separate experimental design, based on the best design obtained.

Build Up the Picture

The DoE software in the Command Center automatically detects the presence of existing experiments (scenarios) and fits new experiments in around these, to provide a more thorough coverage of the design space.

It is often useful to perform relatively few experiments and look at the results before deciding whether to do more experiments, since:

- The optimum design may lie outside the input variable range being investigated.
- The output variables may vary smoothly with changes in the input variables, so the shape of the response surface can be determined using relatively few experiments.

Zoom In on the Range of Interest

Hopefully, the optimum combination of the output variable(s) lies within the chosen design space. Again, performing relatively few experiments and looking at the results can indicate that further effort should be concentrated on one part of the design space.

A subsequent experimental design can be performed over a more restricted range of the input variables. The DoE software automatically detects the presence of the existing experiments and fits new experiments around these.

Make Effective Use of Computing Resources

The Command Center can use other machines on a network to solve DoE scenarios concurrently, see "Distributed Solving of Scenarios" on page 85.

Remember, it is generally better to restrict the number of input variables varied in a single experimental design to make it easier to understand the results, so use the computing resources to run experimental designs in which other input variables of interest are varied.

Numerical Issues

One point to note is that although numerical experiments are free from random experimental errors (that is, doing the same run twice gives exactly the same result) they can be prone to systematic errors, resulting from the sources identified below.

To ensure that the variation in the output variables is because of physical effects, care should be taken to minimize these sources of error.

Incomplete Convergence

One source of systematic error is incomplete convergence. The default Termination Residuals are designed to provide acceptably well converged solutions. Generally, the numerical effort required to achieve a perfectly converged solution is not warranted, since:

- The error in the calculated values resulting from the incomplete convergence is relatively small given the uncertainties in the model, and
- The design may need to be changed.

Incomplete convergence means that the solution obtained depends, albeit weakly, on the initial conditions. Consequently, the value of an output variable can be slightly higher or lower than the value that which would be achieved if the solution were perfectly converged.

The most effective way to assess the influence this is having on the results obtained is to use monitor points to record the iteration history of the output variables. In a steady-state case, these should become invariant with iteration before the solution stops. If the solution is judged not to be adequately converged, the termination residuals can be reduced and the solution continued.

Another source of systematic error is grid dependence. Often it is impractical to achieve full grid independence, where the result of interest does not change if the grid is made finer.

Grid dependence has the greatest effect on the output variables if as part of the DoE one of the input variables being considered changes the grid, for example by changing the size or position of an object.

DoE Size Restriction

There is a size restriction on a DoE scenario such that:

$$number\_of\_input\_constraints + (2 \times number\_of\_design\_parameters) < 100$$

where:

number_of_input_constraints is the number of constraints added using the "Design Of Experiments Dialog Box" on page 119.

number_of_design_parameters is the number of activated variables that have VariationType set to Design Parameter using the Input Variables view.

Designed Experiments

There are two special features that distinguish a designed experiment.

1. A predetermined series of tests is carried out before any analysis is done.

This is in contrast to the usual ad-hoc approach in which the results of the first test will determine the settings used in the second test, and so on.

2. Changes are usually made to more than one design parameter before each new test.

   This may sound like a bad idea because it will be impossible to tell which specific change made the difference, but, if the sequence of changes is carefully chosen, it is possible to identify patterns in the output that can be linked to specific parameters or interactions between parameters. The possibility of looking for patterns in a linked series of tests makes designed experimentation a very efficient way of generating knowledge.

We will refer to the product that is the subject of the testing as a system and the parameters that are changed will be called *factors*. The *experiment* is the complete set of tests and the pre-determined sequence of changes is called the *experimental design*. Generally, each test is called a *run*. Any performance measure for the system is called a *response variable*.

Design of Computer Experiments

Complex computer simulations are often time-consuming. This drives the need for proper designed computer experiments.

The first step is to identify the factors and their ranges that specify the design space to be explored. After that a suitable experimental design should be chosen to obtain as much information as possible about the systems behavior using as few simulations as possible.

In choosing the experimental design we have to be careful, since most classical Design of Experiments (DoE) approaches have been developed for *physical experimentation* in which experiments and measurements of response parameters are subject to noise. Such experimental designs have certain characteristics that, although advisable for experimentation under uncertainty, are undesirable when doing deterministic computer simulations. For instance, no additional information is gained from the repeated simulation of the same run, which is often proposed in classical DoE. Applying such approaches may result in excessive simulation time and partial exploration of the design space.

For computer simulation purposes, experimental design should be *space-filling* and *non-collapsing*:

- *Space-filling designs* – To explore the design space and to capture as much information as possible, the runs should be spread throughout the design space as evenly as possible, that is, the experimental design should be space-filling. We assume that no information about the function underlying the simulation model is available.
- *Non-collapsing designs* – It is usually not known beforehand which factors are important and which are not. An experimental design is called non-collapsing if, in case one or more factors appear to be unimportant, every run in the design still gives

information about the influence of the other factors on the response parameters. In this way, none of the time consuming computer simulations may become useless.

The DoE method developed and implemented for the Command center generates space-filling Latin Hypercube Designs (LHD). Such designs are non-collapsing and space-filling. As a measure for the space-filling of a design, the *minimal Euclidean distance* between two of its runs is used. The larger this minimal distance the better the experimental design.

Further Reading

For an introduction to numerical experimental design techniques, refer to the following documents.

- Sachs, J., W.J. Welch, T.J. Mitchel, and H.P. Wynn (1989), *Design and Analysis of Computer Experiments*, Statistical Science 4, 409-435.
- Montgomery D.C. (1984), *Design and Analysis of Experiments (Second Edition)*, John Wiley & Sons, New York.

Setting Up and Creating a DoE

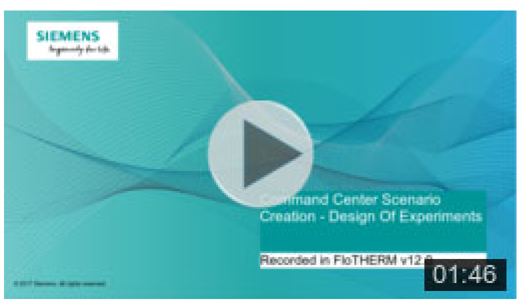
A DoE creates scenarios whose input variables are restricted within user-specified ranges.

Restrictions and Limitations

- Collision Detection is not supported if either of these restrictions is not met:
  - Dependent variable expressions must be solely a function of Design Parameter type input variables, otherwise Warning W/20102 is generated.
  - Dependent variable expressions must be linear, otherwise Warning W/20103 is generated.

Video

Learn how to create a design of experiments set of scenarios.



Procedure

1. Choose the input variable types and set their ranges using the Design Parameter variable type option of the Input Variables panel.
2. Choose **Create Scenarios** > **Design Of Experiments** to open the Design Of Experiments dialog box.
3. By default, DoE projects are named Design <*n*>, where <*n*> is numbered from 0, but you can change this default by editing the Design Name field.
4. Optionally, check the Enable Collision Detection check box.

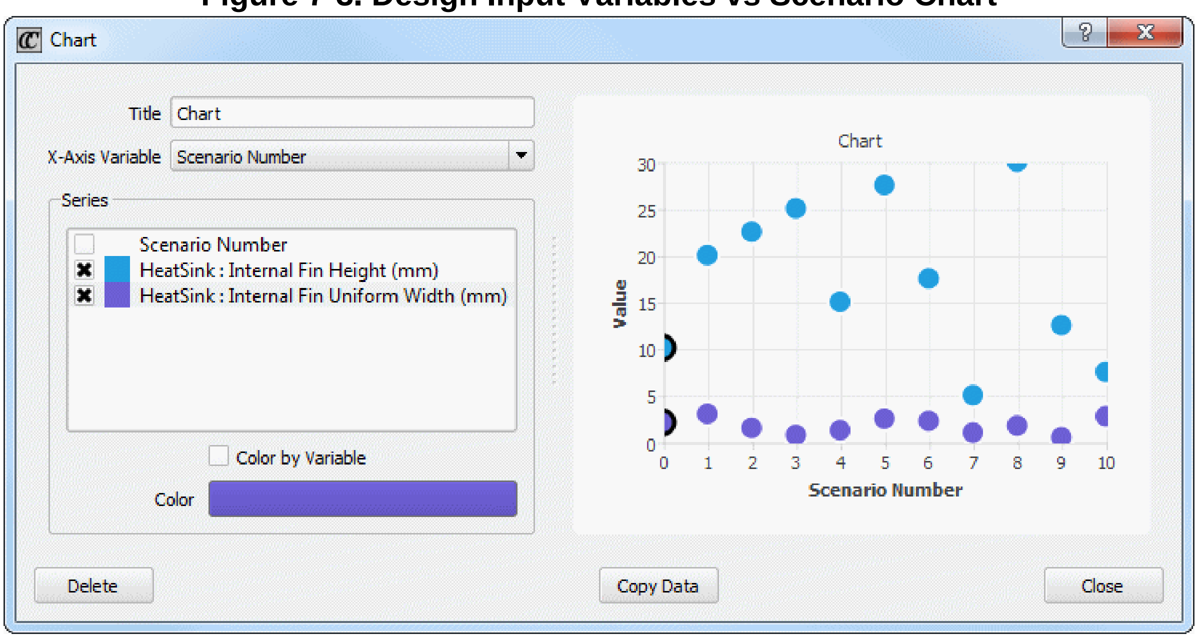
   This stops objects that are moving positions between scenarios from colliding.
5. Optionally, create input constraints.
6. Enter the number of experiments that you want to generate in the Number Of Experiments To Design field.
7. Click **OK** to create a set of automatically generated scenario projects.

Results

It is advisable to check the projects that have been created before attempting a solve:

- Open the Scenario Viewer and select each scenario to view any dimensional changes.
- Open the Scenario Table and check the input variables of the scenarios.
- In addition, you can see how input variables vary graphically by choosing **Chart** > **Create** and selecting input variables to be plotted against Scenario Number, for example, see Figure 7-3.

Figure 7-3. Design Input Variables vs Scenario Chart



Related Topics

Design Of Experiments Dialog Box

Creating an Input Constraint

Results Comparison Chart

Creating an Input Constraint

Input constraints can be added to help guide optimization.

Restrictions and Limitations

- There is a size restriction on the combined number of design parameters and input constraints, see "DoE Size Restriction" on page 102.

Prerequisites

- Use this procedure, optionally, as part of running a DOE or an optimization, see "Setting Up and Creating a DoE" on page 104 and "Running a Sequential Optimization" on page 110.

  You can define constraints in either of the Design Of Experiments or Optimize dialog boxes. Changes made in one dialog box are maintained in the other.

Procedure

1. In the Constraints area of the Design Of Experiments or Optimize dialog box, click **Add Constraint**.

   If using the Optimize dialog box, make sure that the **Input** tab is selected.

2. Enter a constraint definition.

   Each constraint definition has the format:

   > Sum or multiplier of *terms* <= or >= *constant value*

   Where a single *term* is a *multiplier* and a selected *input variable*.

   - The *multiplier* and *constant value* are entered directly from the keyboard.
   - The *input variable* is selectable from a dropdown list of Design Parameter type input variables.
   - The relationship <= or >= is selectable from a dropdown list.

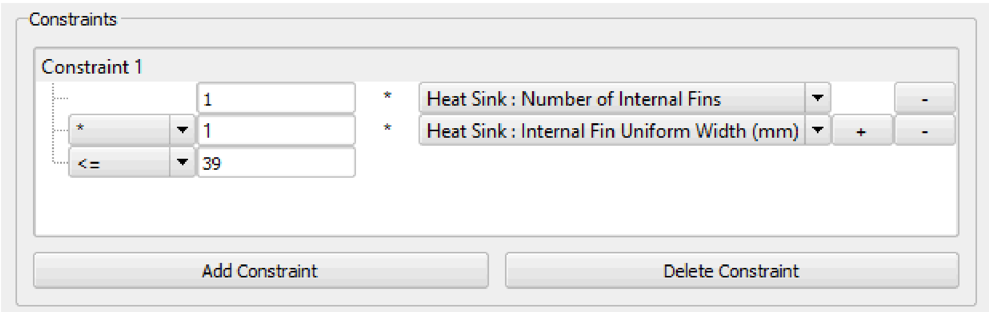
3. When adding additional *terms*, you have a choice of combining these by using either the add (+) or multiply (*) operator.

Examples

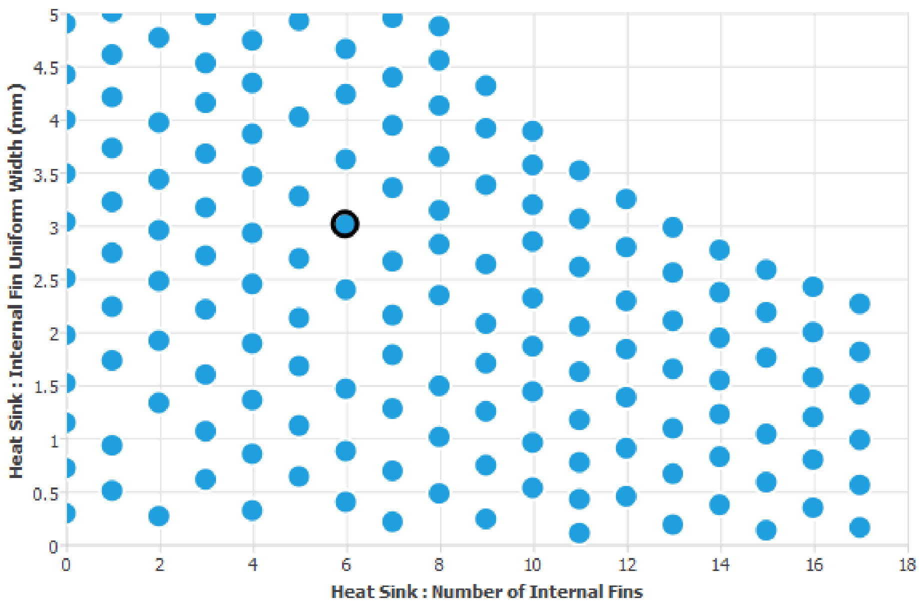
Maintaining a Heat Sink Fin Gap When Varying the Number of Fins and the Fin Thickness

In this example, the number of internal fins (N) and fin thickness (T), are design variables. There is a constraint that the total fin width must fit within 40 mm. Such a constraint can be expressed as:

$$N \times T \leq 39 \text{ mm}$$



The effect of using this constraint can be seen by creating a chart that plots fin width against number of fins. In the following chart, no scenarios are created that occupy the top right of the chart as these would exceed the constraint.



Related Topics

Design Of Experiments Dialog Box

Optimize Dialog Box

Sequential Optimization (SO)

A Sequential Optimization (SO) uses the results of a DoE as input to create and run further scenarios to find the optimum design. Like a DoE, an SO is performed within a design space defined by input variables and their ranges.

Cost Function

A cost function is the value that can be minimized, maximized, or made a specified number by the sequential optimizer.

A cost function can use a single output variable, for example, solid cuboid temperature, or a combination of output variables.

The cost function is the weighted sum of output variable values:

$$\text{Cost Function} = W_1R_1 + W_2R_2 + ... + W_nR_n$$

Where W_n are the weighting factors, and R_n are the values of the output variables.

The optimum design is that with the optimum value of a set cost function. The optimum value of a cost function is the lowest or highest value of that function, depending on the way the function is expressed.

A cost target can also be defined, such that:

$$\text{Cost Function} = W \times (R - Cost\_Target)$$

Related Topics

Defining a Cost Function

Sequential Optimization Strategy

Before running an SO, it is advisable to consider the following aspects.

The Number of Input Variables to Investigate

Like DoE, the Command Center enables any number of input variables to be considered in a SO run.

The Cost Function

As it is the cost function that the optimization uses to obtain the optimum design, you must consider carefully what output variables to include in the cost function, the relative weightings applied to each output variable, and whether there is an acceptable value below which the design is considered acceptable.

Starting From a DoE

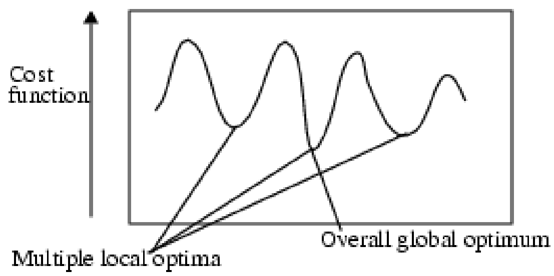
The SO software built into the Command Center makes use of the results of any scenarios that have been run within the design space considered for the optimization, to obtain information about the shape of the response surface. Therefore, the more experiments performed within the design space, the more information is available to guide the optimizer's decision regarding the next design to run, which can reduce the number of steps taken to get to the optimum.

To use the results of a DoE, all the output variables that appear in the cost function need to have been defined as output variables.

Starting From Several Designs

The SO software finds an optimum value for the cost function. However, this could be a local optimum close to the starting design, rather than the global optimum, that is, the lowest value for the cost function over the entire design space.

Figure 7-4. Multiple Local Optima



Performing several SO runs starting from different designs within the design space, which converge to the same design, provides confidence that the global optimum has been found.

Effective Use of Computing Resources

As the Command Center can use other machines on a network to perform concurrent SO runs, this can make performing several SO runs to build confidence that the global optimum has been found. See “Distributed Solving of Scenarios” on page 85.

Numerical Issues

The comments provided under the DoE above are also valid for the SO. The SO uses knowledge of the response parameters from DoE runs to guide it towards an optimum, so any numerical error in those results affects the optimization process.

Another point to note is that unlike the DoE, which always starts each new design from the results for the Base Project, SO starts from the fields for the current best design, so the level of convergence can change from one step to the other.

As with DoE, to ensure that the variation in the output variables is because of physical effects, care should be taken to minimize these sources of error.

Running a Sequential Optimization

Additional scenario projects are created before selecting the optimum project.

Restrictions and Limitations

- Restrictions apply to collision detection as described under “Setting Up and Creating a DoE” on page 104.

Prerequisites

- A cost function has been defined.

Procedure

1. Set up a DoE and, optionally, create and solve the DoE scenarios.
2. Click the Optimize icon or choose **Project** > **Optimize** to open the Optimize dialog box.
3. Choose one of the **Sequential From** Optimization Types.
4. Set the Stopping Criteria.

   Maximum Number of Of Steps limits to number of additional scenarios that may be created. This number may not be reached if optimization occurs first.

   You can enter a Critical Cost Value; the optimization run stops if the cost function reaches this value.
5. Optionally, check the Enable Collision Detection check box.

   This stops objects that are moving positions between scenarios from colliding.
6. Optionally, create Input Constraints.
7. Optionally, create Output Constraints.

8. If you have not previously created and solved DoE scenarios then you can choose to do this by checking the Design Experiments Before Optimization check box and specifying a scenario stem name (Design Name) and the number of scenarios to be created (Number of Experiments To Design).

   If you have checked this check box then DoE scenarios will be created and solved before the Optimization.

9. Click **OK** to start the optimization.

Results

Before optimization, any unsolved scenarios (except those with a status of Steady State Not Converged, Failed, or Interrupted) are solved.

The Design Input Variables vs Scenario and Cost Function vs Scenario results comparison charts are opened during optimization. The Design Input Variables vs Scenario chart is updated when input variable values have been assigned. The Cost Function vs Scenario chart is updated when the cost function has been calculated. You can close these charts and reopen them later.

New scenarios, named SO Step-*n*, are created until an optimum scenario is identified.

If an optimum is not found before the last SO scenario has been solved, then the Best So Far scenario is highlighted using the Solution Status cell.

You can re-run the optimization if you want to try for a better scenario. For any re-runs, the scenario name sequence restarts from SO Step-1.

When an optimum has been found, that is, the program has decided that a sufficiently distinct scenario cannot be created, an INFO message is output and the optimization is stopped.

Related Topics

Optimize Dialog Box

Creating an Input Constraint

Creating an Output Constraint

Creating an Output Constraint

Define constraints on output variables so that results that are unacceptable are not flagged optimal.

Prerequisites

- Use this procedure, optionally, as part of running an optimization, see "Running a Sequential Optimization" on page 110.

Procedure

1. In the Constraints area of the Optimize dialog box, open the **Output** tab then click **Add Constraint**.

2. Enter a constraint definition.

   As described in Step 2 of "Creating an Input Constraint" on page 106, except that the constraint is defined by one or more output variables.

3. Continue adding terms and constraints as required.

Examples

- If a particular junction temperature of a component exceeds 90 degC then the scenario cannot be considered as an Optimum:

  1 * Compact Component 1:Junction 2 Temperature <= 90

Related Topics

Optimize Dialog Box

Response Surface Optimization (RSO)

RSO is a faster method of optimization than performing an SO from the best design available.

RSO Description and Strategy

Response Surface Optimization (RSO) fits a response surface to the results of a DoE and recommends the optimum design.

The Command Center estimates the true response surface from a small finite number of experiments (for which the input and output are known).

The accuracy of the response surface in providing a good representation of the true response surface is highly dependent on:

1. The number of DoE scenarios.
2. The number of input and output variables (for example, dimensionality of the response surface).
3. The smoothness or complexity of the response surface.

The more solved DoE scenarios there are, the lower the resulting error in the RSO surface fit. Although dependent on the dimensionality of the design space and the smoothness of the true response surface, a rule of thumb is that 10 to 15 experiments per input variable should result in a good first cut response surface. Further DoEs can be done to see how much the reported RSO error decreases. If the error does not decrease then the response surface fit is as good as possible.

The RSO error is an 'RMS' type measure of the average error of the surface in how good it is globally in fitting to the DoE points. Although case dependent, errors of 10% or less usually indicate the response surface is good form fit, that is, the shape is accurate and the location of the global minima will be accurate, although the actual value might not be. Confirmation of the global optima as recommended by the response surface should be confirmed by solving the recommended optimum.

If the design space (that is, the difference between minimum and maximum design parameter values) is large and therefore encompasses many local minima and maxima, many more DoE points will be required to capture all the peaks and troughs of the response surface. If a smaller input parameter range was used, fewer points would be required to achieve a low error fit.

The RSO will attempt to determine correlations between all input and all output variables. The more input and output variables there are, the more DoE points will be required to resolve correlations between them.

In summary:

- Create a DoE with at least 10 to 15 DoE points per input variable
- Minimize the number of input and output variables to just those critical ones
- If the RSO error is > 10% then perform an additional DoE and redo the RSO
- If the response surface is very bumpy, consider reducing the range between input parameter lower and upper bounds, that is, try zooming in to one region.

The response surface error in providing a good representation of the true response surface is shown in the recommended optimum solution status name.

The error can be decreased by performing more experiments (using the DoE capability) and re-running the RSO.

Related Topics

Running an RSO

Running an RSO

Includes post-run operations. You can run an RSO several times to try and improve on the optimum design.

Restrictions and Limitations

- Ensure the DoEs have been optimized without variable constraints.
- When using Simcenter Flotherm remotely, the Response Surface Viewer has limited support with some graphics cards that support improved OpenGL over RDP. As a workaround, you are advised to use VNC instead of RDP.

Procedure

1. Define the input and output variables, the cost function and build a set of experiments.
2. Solve all the scenarios.
3. Click the **Optimize** icon  or choose **Project > Optimize** to open the Optimize dialog box.
4. Set the Optimization Type to Response Surface From All and remove any Constraints.
5. Click **OK**.

   When the optimization has completed, a new design is added to the Scenario Table, named RSO Design-<*n*>. The Solution Status is RSO Optimum (+/- <%>), where <%> is a measure of difference from the recommended optimum.

The Scenario Table is populated with the expected values, italicized, of the output parameters and the cost function.

6. To view the results, click the **Response Surface Viewer** icon 
7. You can perform any of the following post-run operations:

If you want to...	Do the following:
Compare RSO results.	1. Copy the RSO-generated design, creating two identical scenarios, one of which has response parameter values and cost function predicted by the RSO. 2. Solve the other scenario to get calculated values for the response parameters and cost function. 3. Comparing the results from the two scenarios will confirm whether the predicted design is close to the true optimum.
Re-run an RSO.	1. Delete the RSO-generated optimum design. 2. Re-run the RSO to see how the optimum design changes when the results for the previously predicted optimum designs are added to the set of designs being fitted.
Populate the design space.	1. Generate further scenarios by re-running the DoE software. 2. Solve all the new scenarios. 3. Re-run the RSO.
Zoom in on a region of the design space.	1. Refine the input variable range as desired, for example, by reducing the range, centered on the predicted optimum design. 2. Generate further designs by re-running the DoE software. 3. Re-run the RSO.

Results

If there is more than one input variable, then you can define 2.5D and 3D plots in the Response Surface Viewer.

Use the mouse scroll wheel to zoom in and out of plots.

Left-click and drag to create a zoom area in 2D and 2.5D plots; right-click to return to full view.

Right-click and drag in a 3D plot to rotate the plot.

Examples

Figure 7-5. Response Surface Viewer 2D Plot

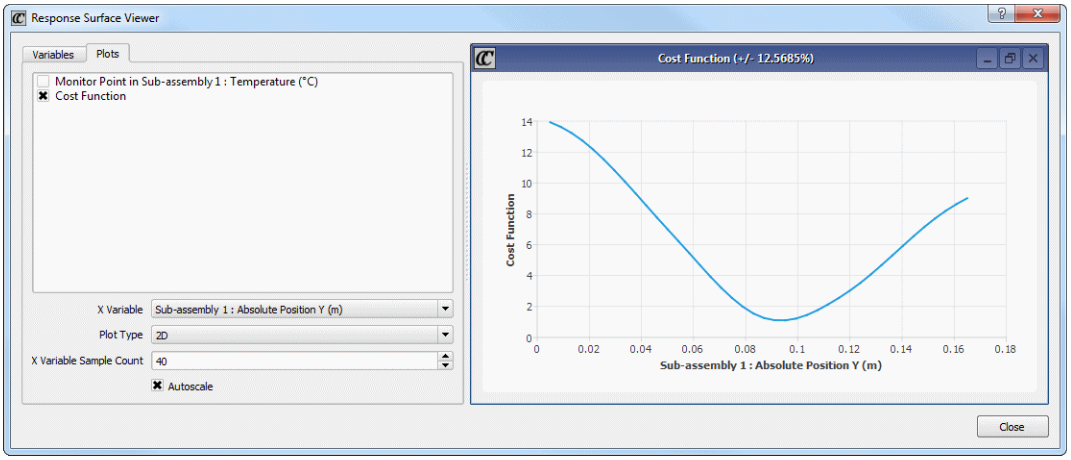
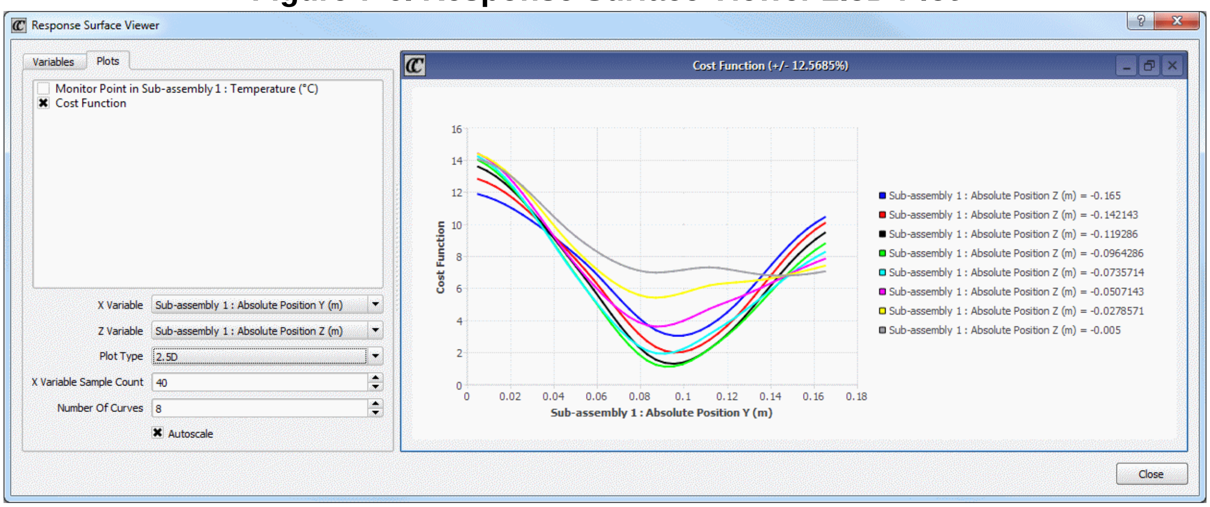
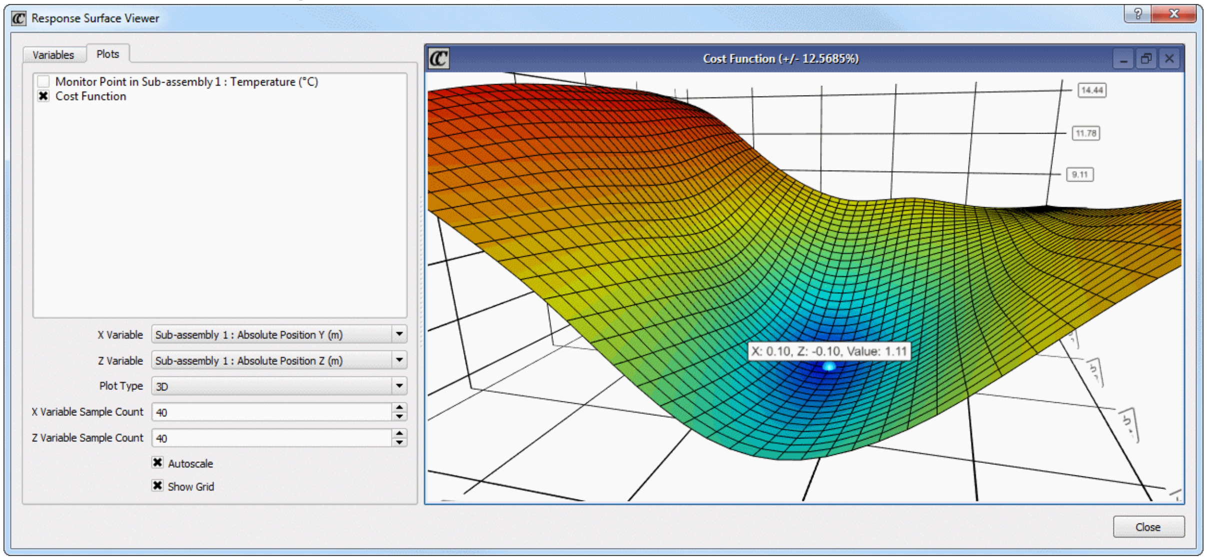


Figure 7-6. Response Surface Viewer 2.5D Plot



In Figure 7-7, a point on the plot has been clicked to show its estimated value. The calculated accuracy of values on the plot is shown in the title bar.

Figure 7-7. Response Surface Viewer 3D Plot



Tip

Having selected a point on a 3D plot, you can create a new scenario directly from the Response Surface Viewer by opening the **Variables** tab (noting that the variables values have changed to those at the selected point on the plot) and clicking **Create Scenario**.

Related Topics

Optimize Dialog Box

Response Surface Viewer

Optimization Viewers and Dialog Boxes

The following windows and dialog boxes are used during optimization.

Design Of Experiments Dialog Box

To access: **Create Scenarios** > **Design Of Experiments**

Use this dialog box to create a sequence of scenarios using experimental design methods and to define constraints (or limits) on the variable changes.

Objects

Field	Description
Design Name	Sets the stem-name for the series of scenario projects such that scenarios are named *stem-name:n*, where *n* is a sequence number.
Enable Collision Detection	Stops objects colliding when they are moved or resized.
Design Limits	
Input Variable Table	Read-only list of all active numeric input variables and their ranges.
Constraints	
Constraint Definition Area	Use this area to define linear constraints on input variables to define the limits to the variable changes used in DoE runs.
Number Of Experiments to Design	The number of scenarios that will to be created. The default is five times the number of variables listed under Design Limits.

Related Topics

Setting Up and Creating a DoE

Creating an Input Constraint

Optimize Dialog Box

To access: **Project** > **Optimize** or by clicking the **Optimize** icon when DoE sequence set and Output Variable with cost factor are chosen.

Use this dialog box to set up the sequential optimization controls.

Objects

Object	Description
Optimization Type	Defines where optimization begins. • Sequential From Best — Start optimization from the best design from the DoE. The more DoE runs that are performed, the higher the probability that this choice will find the global optimum. • Sequential From User Specified — Start optimization from a specified scenario project. • Sequential From All — Perform an SO run from each design in the DoE. This is the most rigorous start point, but is computationally expensive. • Response Surface From All — Fit a response surface to the results of a DoE and calculate the optimum design. This is the quickest method.
Start From	(Sequential From User Specified) The dropdown list contains the Base Project and all the scenarios.
Stopping Criteria (all optimization types except Response Surface From All)	
Maximum Number of Steps	Limits the number of "steps" (sequential optimization scenarios) for this optimization run. **Note:** The sequential optimization scenarios are named SO Step-<*n*> where <*n*> = 1, 2, 3, and so on. Each optimization run starts from <*n*> =1.
Critical Cost Value	Stops the optimization run if the cost function is reduced to this value. The scenario whose cost function meets this value has its status set to Good Enough.
Cost Function	
Cost Function Panel	A read-only definition of the cost function defined in the Output Variables panel.
Design Limits	
Input Variable Table	Read-only list of all active numeric input variables and their ranges.

Object	Description
Constraints	
Enable Collision Detection	Use to stops objects colliding when they are moved or resized.
Input tab	Use this tab area to define linear constraints on input variables to define the limits to the variable changes used in DoE runs.
Output tab	Use this tab area to define linear constraints on output variables to bound what an acceptable design result can be.
Initial Designs (Sequential From Best and Sequential From All)	
Design of Experiments Before Optimization	Perform a DoE prior to the SO run. This is an additional DoE to any that have been run previously.
Design Name	(Design of Experiments Before Optimization) Sets the stem-name for the series of scenario projects such that scenarios are named *stem-name:n*, where *n* is a sequence number.
Number of Experiments to Design	(Design of Experiments Before Optimization) Sets the number of scenarios to be created.

Related Topics

Running a Sequential Optimization

Running an RSO

Response Surface Viewer

To access: **Chart > Response Surface Viewer**, or click the **Response Surface Viewer** icon , or right-click over an output variable or cost function title cell in the Scenario Table and choose **Show Response Surface**.

Use this viewer to see how input variables affect output variables and the cost function.

Description

The viewer plots should be available after all scenarios have been solved, signified by the icon changing from gray to colored. When the icon is gray, self-explanatory hover text advises why the response surface plots are not available, for example, input or output variables have not been defined, there are an insufficient number of solved scenarios (there must be more solved scenarios than input variables), or the calculation of the surface plots is in progress.

You can view variations of input variables in 2D, 2.5D or 3D plots. Examples are shown in Figure 7-5, Figure 7-6 and Figure 7-7.

The expected accuracy of a value anywhere within the range of a plot is shown in the title bar of the plot as a percentage (+/− <*percentage_accuracy*>).

If there have been changes since the plots were last calculated, for example, new scenarios have been added, then the message “Response Surfaces are stale” is output at the bottom of the viewer. Re-solve the scenarios to update the plots.

Objects

Object	Description
Variables tab	
Input Variables variation	Use the slider bar or data entry field to set the value of each input variable. If the input variable is specified in the plot, then the slider and data entry field are unavailable. **Note:** When you select a point on a 3D plot, the values of input variables shown are updated to those of the selected point.
Create Scenario	Click to create a new scenario using the values of input variables shown. Predicted values, shown in italics, of all output variables populate the Scenario Table.

Object	Description
Plots tab	
Output Variables selection	Defines the Y-axis values (output variables and the cost function). A separate plot is created for each check box that is checked, see "Display of Multiple Plots" on page 124.
X Variable	Defines the X-axis values (input variables).
Z Variable	For 2.5D plots, defines multiple plots on a 2D grid (input variables). For 3D plots, defines the Z-axis values (input variables).
Plot Type	Choose from: • 2D – A plot of one output variable or the cost function vs one input variable. • 2.5D – shows the effects of another input variable on a 2D plot. Each 2.5D plot contains color-coded lines, corresponding to values of the additional input variable. • 3D – uses three axes to plot one output variable or the cost function vs two input variables.
X Variable Scale	Choose from: • Linear • Logarithmic
Z Variable Scale	(3D plots) Choose from: • Linear • Logarithmic
X Variable Sample Count	Defines the smoothness of the plot along the X-axis.
Z Variable Sample Count	(3D plots) Defines the smoothness of the plot along the Z-axis.
Number of Curves	(2.5D plots) The number of second input variable curves.
Autoscale	Scale the plots to not extend beyond the ranges defined on the axes and, therefore, always remain visible.
Show Grid	(3D plots) Show the grid on the surface of the plot. The grid cell size is determined by the sample count values.

Usage Notes

Zooming

Use the mouse scroll wheel to zoom in and out of plots.

Left-click and drag to create a zoom area in 2D and 2.5D plots; right-click to return to full view.

3D Plot-Specific Usage Notes

Right-click and drag in a 3D plot to rotate the plot.

Use the X, Y and Z keys to view a 3D plot from the +X, +Y and +Z direction, respectively.

Select a point on the plot to display the axes values at that point, see Figure 7-7 for an example.

Display of Multiple Plots

A separate plot is created for each output variable that is selected. These plots are tiled in the plots area of the viewer as the output variables are selected. You can resize plots.

To always display a plot on top of other plots:

1. Open the title bar context-sensitive menu by right-clicking in the title bar of the plot, and
2. Select **Stay on Top**.

To re-tile plots that have not been minimized:

1. Open the plot area context-sensitive menu by right-clicking anywhere in the plot area that is unoccupied by a plot.
2. Select **Tile Plots**.

Related Topics

Running an RSO

Correlation Matrix

To access: **Chart** > **Correlation Matrix**

Use this matrix to visually identify design parameters that correlate well with output variables, including the calibration cost function. The matrix also highlights less relevant design parameters, which you can then exclude from the analysis, thereby saving time.

Description

Figure 7-8. Full Correlation Matrix Showing R Values

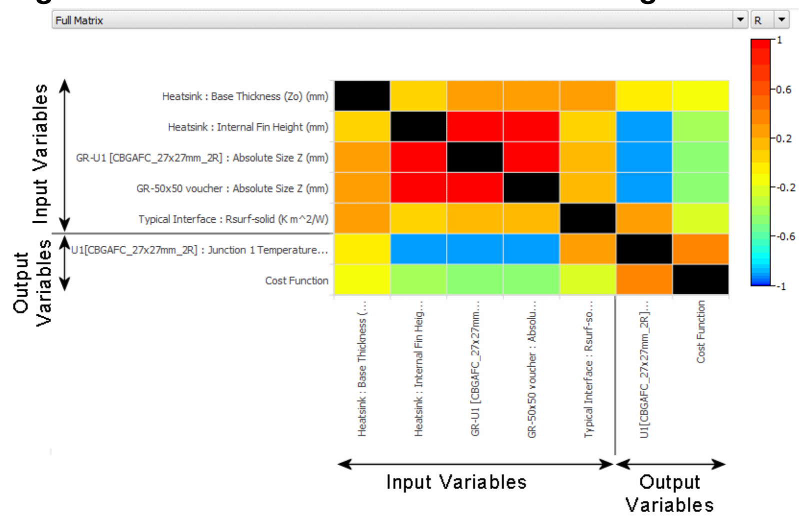
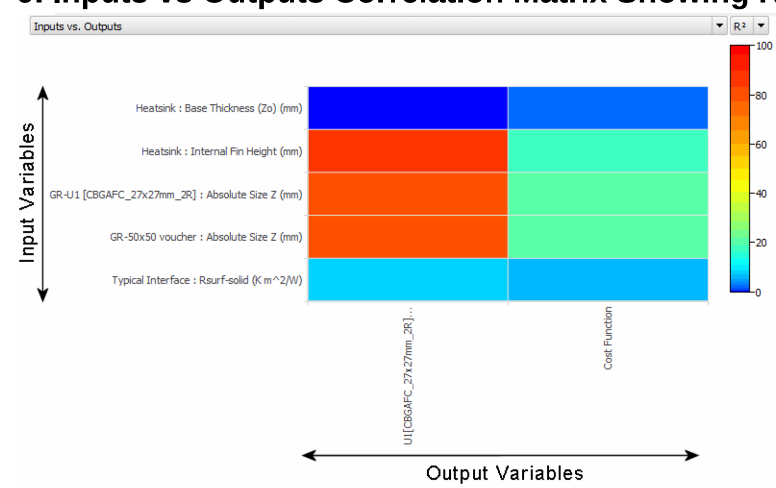


Figure 7-9. Inputs vs Outputs Correlation Matrix Showing R^2 Values



Objects

Object	Description
Matrix Selector	The matrix is made up of colored cells, where each cell indicates the correlation between two variables. A cell is colored black where the two variables are the same as these cells can be ignored. • Full Matrix − the x-axis and y-axis each list all input and output variables. A line separates the input variables from output variables and the cost function, see Figure 7-8. • Inputs vs. Outputs − input variables are listed along the y-axis, output variables along the x-axis. • Outputs vs. Inputs − output variables are listed along the y-axis, input variables along the x-axis. • Outputs vs. Outputs − the x-axis and y-axis each list all output variables. • Inputs vs. Inputs − the x-axis and y-axis each list all input variables.
Coefficient Selector	• R − The correlation coefficient, as a measure of the relationship between the input variable and the output variable. Values lie between −1 and +1. Positive values indicate a relationship between the input and output variables such that as the value of the input variable increases, the value of the output variable also increases. A value close to +1 indicates a strong positive linear correlation. A value close to 0 indicates that there is a poor linear or near-random correlation between the input and output variables. Negative values indicate a relationship between the input and output variables such that as the value of the input variable increases, the value of the output variable decreases. A value close to −1 indicates a strong negative linear correlation. • R^2 − The coefficient of determination, as a percentage. Values lie between 0 and 100. This figure does not discern between positive and negative correlations, but gives a clearer indication between strong and poor correlations.

Usage Notes

Click on a cell to display a correlation chart of the two variables

Figure 7-10 shows a correlation chart when R = −0.38, a random correlation.

Figure 7-10. Random Correlation

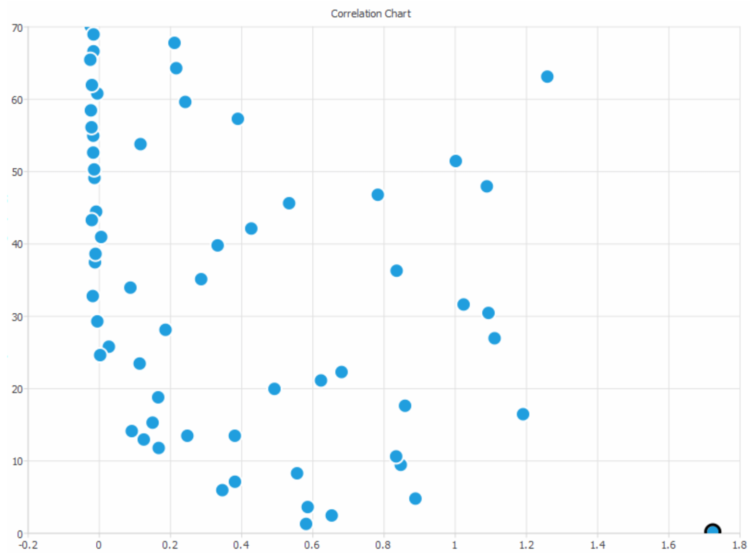
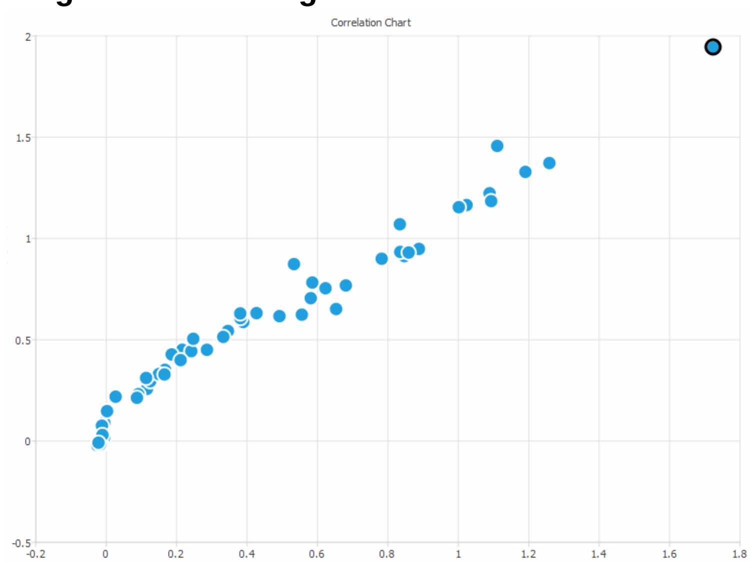


Figure 7-11 shows a correlation chart when R = 0.983, a strong positive linear correlation.

Figure 7-11. Strong Positive Linear Correlation



Related Topics

Results Comparison Chart

Chapter 8
Model Calibration

Experimental data acquired from T3Ster can be used to calibrate a Simcenter Flotherm model to a high degree of accuracy.

Note

Access to Model Calibration is controlled by license.

Model Calibration Workflow Overview

The Command Center can import data from T3Ster Master v2.4 and higher and optimize a Simcenter Flotherm model to produce a simulated structure function that closely matches a T3Ster structure function.

For a detailed description of the procedure within Simcenter Flotherm, see "Calibrating a Model Using Data From T3Ster" on page 132:

1. The thermal response of the device is measured using T3Ster.

   T3Ster uses an electrical test method, defined by JESD 51-1, to measure the junction temperature.

2. A "Best-Guess" model is created in Simcenter Flotherm and the Base Project solved for a transient analysis.

3. A T3Ster calibration file is imported and is compared with the Simcenter Flotherm model.

4. Command Center input variables (typically dimensions and materials) are selected and Design of Experiments scenarios created.

5. The scenarios are solved and the model settings optimized to match the T3Ster-measured structure functions.

When successful, the result is a validated 3D detailed device model, suitable for use in any transient or steady state analysis.

Model Calibration Curve Comparison

The Calibration Error is used as an indicator of how close the Simcenter Flotherm model structure function curves are to those of the experimental model.

The Calibration Error is based on the sum of the shortest distances between the curves. This enables all areas of the curve to have appropriate weighting. The calibration error is computed for all imported measurements, and the weighted sum forms the calibration cost function.

The curve comparison is from 0 to the Calibration Extent.

R^2 is only calculated for the optimal scenario, and is reported, when the calibration task completes, in the Model Calibration dialog box as an Accuracy figure equal to $100 \times R^2$. Table 8-1 advises how to interpret this figure.

Note

The accuracy figure may not correspond initially with a visual comparison of the two curves when included sections of the structure functions contain near-horizontal or near-vertical lines.

Table 8-1. Correlation Between Scenario and Experimental Data

Accuracy	Correlation
100	The ideal result, the model is fully calibrated and perfectly describes the experimental data.
Greater than 90.	A very good result, all major model parameters have been identified.
Between 90 and 50.	Several important parameters are probably missing, or a parameter range is inappropriate.
Less than 50.	The chosen design parameters or ranges are not capable of describing the experimental data well.

Model Calibration Extent

It is recommended that you limit the calibration extent to the value of Theta_{JC}, if you, typically, only want to calibrate the package model, not the model of the package and testing environment together.

T3Ster TDIM projects will contain the value of Theta$_{JC}$, and the calibration extent will be set to that value when the *flocalibration* file is imported.

Alternatively, the Theta$_{JC}$ value may be known, for example, from a specification sheet.

Another way of obtaining the Theta$_{JC}$ value would be to solve the baseline model and determine the value from the results. However, this relies on the baseline model being reasonably close to the T3Ster data, if it is not, then you would need to scale the value observed in the baseline simulation to the scale of the T3Ster data.

Advanced users, who are confident interpreting structure function curves, may only want to calibrate the early section of the heat flow path, for example, in the vicinity of the die. In such cases the calibration extent could be set to a lower value than Theta$_{JC}$.

Theta$_{JC}$ from the *flocalibration* file is also used to set the upper value of the calibration extent. The value must be greater than 0 (zero).

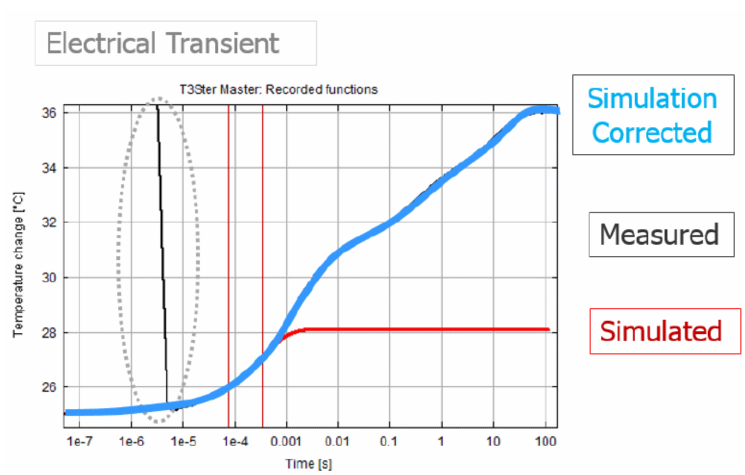
T3Ster Master Simulation Correction

T3Ster Master provides options (modes) for transient correction.

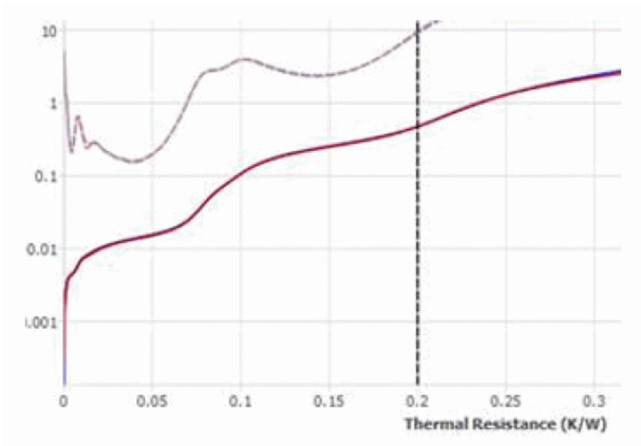
'Square Root' Correction

If the "Use T3M Correction Settings on FloTHERM data" check box (Model Calibration dialog box) is unchecked, then, for imported *flocalibration* files generated with the Square Root correction option, the electrical transient section is replaced with Simcenter Flotherm simulation data for each scenario, after the scenario has been solved.

The replacement ensures that the calibration result is independent of the T3Ster Master operator and that the curves are always compared in a consistent manner.



This has the effect of ensuring the early portion of the structure function must match, and guarantees the unmeasured electrical transient deltaT is not a factor.



If the “Use T3M Correction Settings on FloTHERM data” check box is checked, then the Square Root correction settings found in imported *flocalibration* files are used to modify the Simcenter Flotherm simulation data.

‘Simulated’ Correction

For imported *flocalibration* files generated with the T3Ster Master v3.0 (and higher) Simulated option, the data present is used without modification during the calibration process.

‘No correction’ Correction

For imported *flocalibration* files generated with the No correction option, the data present is used without modification during the calibration process.

‘Minimum seek’ Correction

For imported *flocalibration* files generated with the Minimum seek option, the data present is used without modification during the calibration process.

Calibrating a Model Using Data From T3Ster

Use the Model Calibration dialog box to work through this procedure.

Prerequisites

- One or more *flocalibration* files containing experimental data exported from T3Ster Master v2.4 or later.

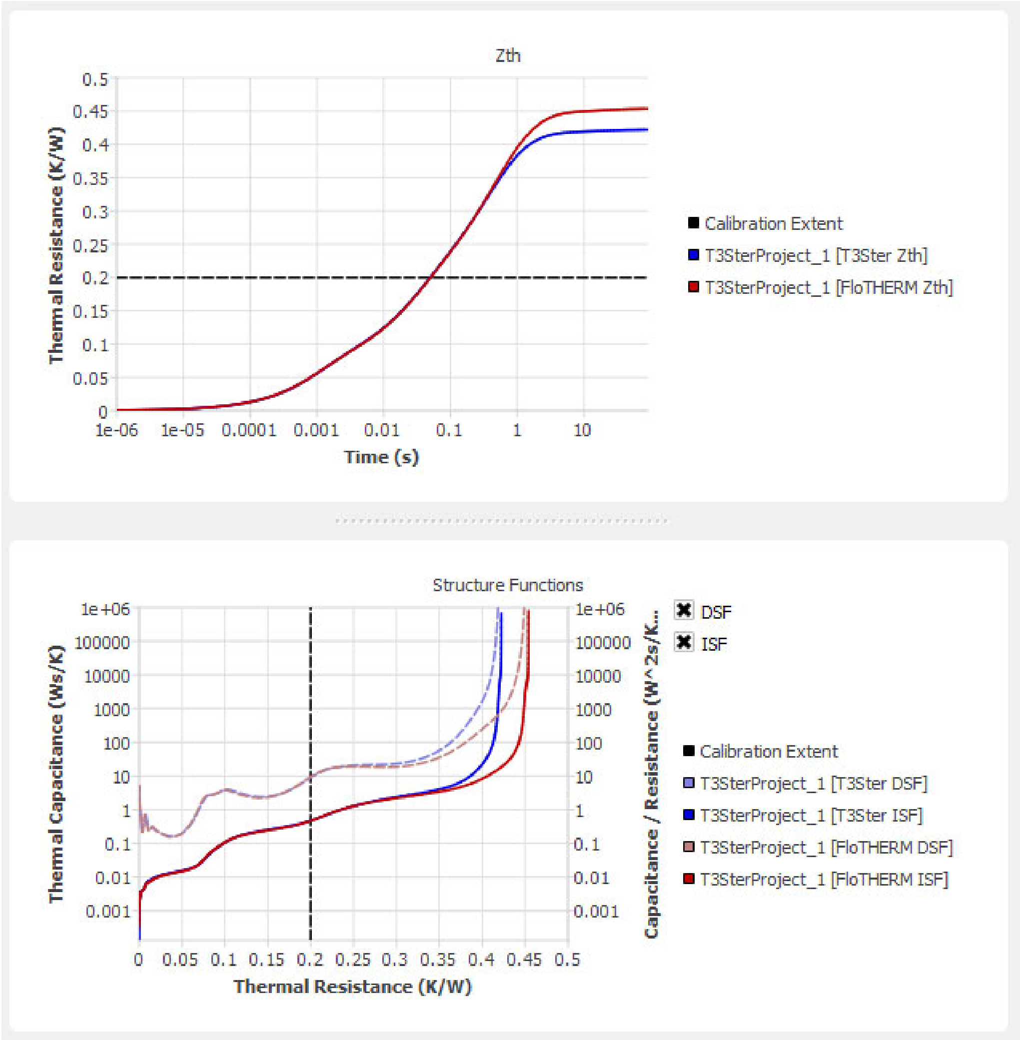
- A Simcenter Flotherm model for comparison which must:
    - Be a Transient analysis, and the transient duration must be greater than the experimental transient duration.
    - Not be a Flow Only type of solution.
    - Include a temperature response to be used for comparison with the experimental temperature response. This can be by either from a volume region or monitor point:
        - In the majority of cases, T3Ster senses the average temperature in the active 'surface', typically a few microns thick. In these cases, use a volume region coincident with the heat source object. Applicable to power transistors, LEDs, and simple ICs.
        - In some cases, T3Ster senses temperature at I/O pins of an IC. In these cases, use a monitor point at the correct location at the external band of the chip.

Procedure

1. From Simcenter Flotherm, start the Command Center and choose **Calibration** > **Model Calibration** to open the Model Calibration dialog box.
2. Click Import T3Ster Measurement, select one or more *flocalibration* files then click **OK** to import experimental data from T3Ster.

   The T3Ster Zth and Structure Functions curves are displayed for comparison with the Simcenter Flotherm model.

Figure 8-1. Comparison Curves



Tip

You can zoom in and out of these curves, refer to the Usage Notes in the dialog box description, "Model Calibration Dialog Box" on page 140.

You can compare the effect of using T3Ster Master Square Root correction on the Simcenter Flotherm curves by checking and unchecking the "Use T3M Correction Settings on FloTHERM data" check box in the Calibration section of the dialog box, see "'Square Root' Correction" on page 131.

3. Specify the Simcenter Flotherm temperature response that is to be compared with the experimental data by selecting a monitor point or volume region from the Model Temperature combo box.

   The Zth and Structure Functions curves are displayed.

Note

The *T3Ster power value* is used when calculating Zth and Structure Functions curves for the Simcenter Flotherm base case and scenarios.

4. Click **Setup Check** to open the Model Calibration Setup Check dialog box and make sure the baseline model matches the experimental conditions before calibrating.

   The Power[1], Duration, and initial temperature (Tinitial) values are displayed for the imported experimental data (T3Ster) and the base Simcenter Flotherm model.

   a. The transient duration must be at least as long as experimental. If it needs to be adjusted then make sure that the Regenerate Time Patches check box is checked.

      The Simcenter Flotherm time grid patches are re-created, based on one time patch per decade, a minimum of ten time steps and with an increasing index of 1.5.

      In addition, the Keypoint Tolerance value in the Project Manager Transient Solution dialog box is changed to 1e-09 s.

   b. Click **Adjust** to modify the Simcenter Flotherm model to ensure the power, duration, and initial temperature match the imported T3Ster measurement settings.

      The Adjust command causes all scenarios to be re-built, therefore, it is recommended that it be used *before* rather than *after* solving scenarios.

      If there is only one T3Ster measurement loaded, then all total power settings (as described above) are scaled in the base case to match the T3Ster power. If there is more than one T3Ster measurement loaded, then the Simcenter Flotherm model power is not modified.

   c. Click **Close** to close the Model Calibration Setup Check dialog box.

5. Specify the parameters of the model you would like to calibrate in the calibration run. Typical input variables chosen are those whose precise values are uncertain, dimensions that are difficult to measure, or material properties, for example, TIM thicknesses, active area dimensions, and so on.

   a. Open the Input Variables view of the Command Center and select one or more input variables.

   b. For each variable, select the Design Parameter variance type.

   c. Specify the range but setting the Minimum and Maximum Values for the variable.

6. Calibration errors are automatically added as output variables when T3Ster measurements are loaded, and these are included in the cost function when a Model Temperature has been selected. Do not add any other output variables. If there are

1. The Simcenter Flotherm power reported is the sum of any Total Power (that is, not Power/Volume or Power/Area) settings in the Simcenter Flotherm model in Source attributes, Thermal attributes and Die SmartParts.

multiple calibration error output variables, you can change the Cost Weighting, which defaults to 1, to adjust the cost function contribution from each calibration error.

Note

If you set the cost weighting of a calibration error to 0, then the value of the calibration error is not calculated. Also, any measurements with a cost weighting of 0 are not included in R^2 calculations for the optimal scenario.

7. Return to the Model Calibration dialog box.

   The input variables and their ranges are displayed in the Design Limits section of the dialog box. Linear Function type input variables are fully considered in the optimization, but are not displayed in the dialog box.

8. Use the dialog box to create a Design of Experiments set of scenarios. In the Create Scenarios section, enter a Number of Experiments to Design (typically five times the number of input variables) and click **Design Experiments**.

   The Command Center Scenario Table shows the scenarios that have been created.

9. In the Calibration section of the dialog box, use Calibration Extent to control the range of R_{th} that will be considered for calibration.

   Structure Functions represent the entire heat flow path, from die, through the package, and into the environment. Typically, you will be interested in calibrating the package, but not the environment. A value that represents $Theta_{JC}$ should be used here. If the imported *flocalibration* file was created from a T3Ster TDIM project then the value is set to $Theta_{JC}$ upon import. Refer to "Model Calibration Extent" on page 130.

   The Calibration Extent value is shown as a black dashed line on the Zth and Structure Function charts.

10. In the Calibration section of the dialog box, choose the Optimization Type.

    The Command Center supports both Response Surface and Sequential Optimization techniques and either can be utilized for the model calibration.

11. Click **Calibrate Model** and wait and for all solves to complete.

    After unsolved scenarios have been solved, the calibration cost function (the difference between the simulation and experimental ISFs) is calculated, and optimization is started. When using Response Surface Optimization, the optimal result is first predicted with Response Surface manipulation, and then fully solved.

Results

As scenarios are solved, the Scenario Table will contain the following output variables:

- Calibration: <*T3Ster measurement*> - Calibration Error

  See "Model Calibration Curve Comparison" on page 130.

- Cost Function

  The weighted sums of the T3ster project calibration errors. The optimization process drives this value towards zero, and drives the Accuracy figure towards 100.

Selection from the Scenario dropdown list at the top of the chart section of the calibration dialog box displays the Z_{th} and Structure Function curves for each solved scenario.

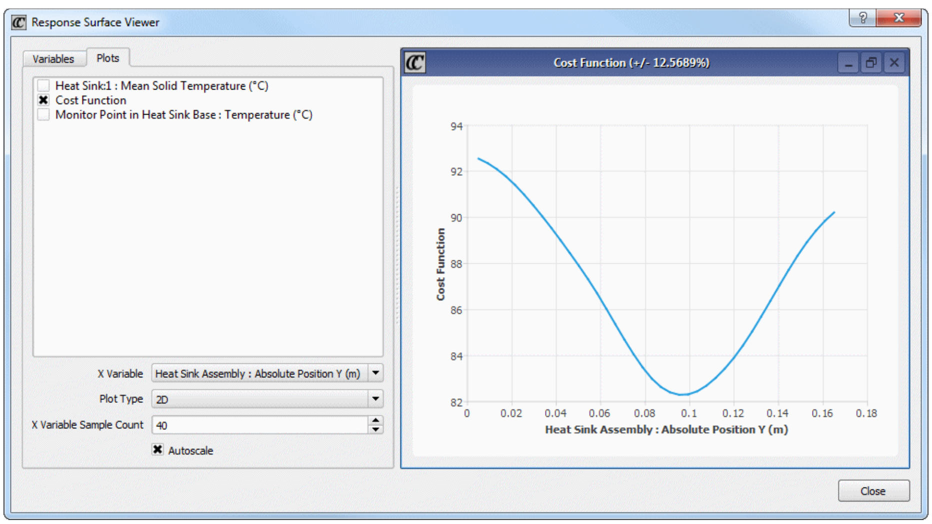
When all solves are finished:

- The calibration Accuracy is reported in the Calibration section of the dialog.
- The chart area displays Z_{th} and Structure Functions for the optimal scenario.

When the Response Surface method is utilized for optimization, examination of the response surfaces (**Chart** > **Response Surface Viewer**) enables a better understanding of the design sensitivity.

The Cost Function response surface plot should have a well-defined minimum value within the range of input variable values, for example, see Figure 8-2. Such a plot confirms that the range of input variable values was well chosen. If the minimum lies on the edge of the range, then there may be a better value beyond that range.

Figure 8-2. Response Surface Viewer 2D Plot Showing a Well-Defined Cost Function Minimum



Also note the indicated accuracy of the Response Surface in the title of the chart. A large number here can be reduced by creating and solving additional scenarios, and re-running the Calibrate command.

Examples

A spreadsheet, *IGBT_Creator.xlsm*, and calibration file, *IGBT_experiment.flocalibration*, are supplied in the following folder:

> *<install_dir>/flosuite_<version>/flotherm/examples/Demonstration Models/Detailed Model Calibration*

The spreadsheet contains a macro that creates a typical IGBT model. Inputs for the IGBT stack geometry, materials, power, and so on are used to created a transient model for use with the calibration feature.

The calibration file is a typical measurement associated with this type of module.

Related Topics

Model Calibration Dialog Box

Model Calibration Setup Check Dialog Box

Exporting Transient Data to T3Ster

You can save Simcenter Flotherm transient data of all scenarios to CSV files for import into T3Ster Master.

Prerequisites

- The CALIBRATION_EXPORT_DATA environment variable must be set to a valid directory path.

Procedure

If the environment variable has been set, then export occurs when you open the Model Calibration dialog box.

Results

The directory specified by the environment variable is populated with CSV files containing Time vs Temperature data for all measurements and scenarios.

The filenames take the form *<T3Ster flocalibration filename>_<scenario number>.csv*.

These files can be imported into T3Ster Master by choosing **File > Import**.

Examples

The following is an extract from a CSV file:

```
Time(s), Temperature(degC)
0,25
3.1623e-008,25.001
8.9443e-008,25.003
1.6432e-007,25.004
2.5298e-007,25.0055
3.5355e-007,25.007
4.6476e-007,25.008
5.8566e-007,25.0095
7.1554e-007,25.011
```

Model Calibration Dialog Box

To access: **Calibration** > **Model Calibration**. The Load Calibration Data Progress dialog box may be displayed before this dialog box is opened.

Use this dialog box to manage the Model Calibration process and to view comparison curves between model and experimental data.

Note

Access to this dialog box is controlled by license.

Description

The dialog box is divided into panels that can be resized using vertical and horizontal splitters. The right-hand side of the dialog box is used to display plots.

Objects

Object	Description
Import T3Ster Measurement	Click to open a file browser for selection of one or more Simcenter Flotherm Calibration files exported from T3Ster.
Setup Check	Click to open the Model Calibration Setup dialog box. See "Model Calibration Setup Check Dialog Box" on page 144. The button incorporates a setup check status icon. The check status icon only indicates that the T3Ster and Simcenter Flotherm Duration and Tinitial values match. **Caution:** The Check Status icon does not indicate that the T3Ster and Simcenter Flotherm Power values match. There may be instances where a power mismatch is valid, particularly for a multi-calibration setup with thermocouples or passive components present in the Simcenter Flotherm model. Therefore, you are advised to check that all the parameters are set up as per your expectation before proceeding with the calibration.
T3Ster Measurements A table containing a row for each imported calibration file. Maximum of 20 imported measurements.	
T3Ster Measurement	The root filename of the imported file.
Power	The power value read from the imported file.
Duration	The transient duration value read from the imported file.

Object	Description
Model Temperature	A dropdown list of available monitor points and volume regions in the Simcenter Flotherm project. Items in the list are in the same order as the data tree. Multiple volume regions can be selected from the list. When multiple volume regions are selected, the volume-weighted average temperature is used. Multiple-selection enables power devices with distributed loading to be calibrated. **Caution:** If selected multiple volume regions overlap, then, depending on the size of the overlap, the volume-weighted average temperature will be higher than expected because of double-counting in the overlapped volume.
Design Limits	
Design Limits Table	Read-only Minimum and Maximum values of Design Parameter input variables.
Create Scenarios	
Number of Experiments to Design	The number of scenarios that will be created.
Design Experiments	Click to create Design Experiments scenarios. This is equivalent to the **OK** button on the Design Of Experiments dialog box, see "Design Of Experiments Dialog Box" on page 119.
Calibration	
Use T3M Correction Settings on FloTHERM data	When checked, the Square Root correction made in T3Ster Master, and stored in the *flocalibration* file, is applied to the Simcenter Flotherm curves. When unchecked, no correction is made. See "'Square Root' Correction" on page 131.
Calibration Extent	Use to limit the extent of the calibration to exclude the environment, see "Model Calibration Extent" on page 130.
Optimization Type	The choice is either Sequential From Best or Response Surface. Refer to "Optimize Dialog Box" on page 120.
Calibrate Model	Click to solve the scenarios and create an optimal scenario based on the calibration cost function.
Accuracy	The weighted average R^2 value of all measurements for the optimum scenario, expressed as a percentage. See "Model Calibration Curve Comparison" on page 130.

Object	Description
Plot Area	
Scenario	Use this dropdown list to select a scenario. The optimum scenario is identified in the dropdown list by the scenario name being appended with "[Lowest Cost]". **Note:** Scenario selection is synchronized with the Command Center application window.
Zth	The Z_{th} curves for the selected T3Ster Measurement(s) and Scenario.
Structure Functions	The Structure Functions curves for the selected T3Ster Measurement(s) and Scenario. DSF (Differential Structure Functions) are displayed as dashed lines. ISF (Integral Structure Functions) are displayed as solid lines.

Usage Notes

Multi-Select of T3Ster Measurements

You can select more than one row of the T3Ster Measurements table to display Z_{th} and Structure Functions curves for the respective measurements in the same plot area.

Size

To maximize/restore the size of the dialog box, double-click anywhere within the title bar.

Declutter

Individual plots can be hidden or shown by clicking the plot name in the legends. When a plot is hidden, the plot name is dimmed.

Zoom Controls

Make sure that focus is on the panel by mouse-clicking on the panel.

- To zoom in to a plot, rubber-band an area using the mouse, use the mouse scroll wheel, or press the + (plus) key.
- To zoom out of a plot, use the mouse scroll wheel or press the - (minus) key.
- To reset a plot to its default extent, right-click the mouse or press the R key.

Pan Controls

To pan around a plot that has been zoomed in, click on the panel then use the arrow keys.

Related Topics

Calibrating a Model Using Data From T3Ster

Load Calibration Data Progress Dialog Box

Model Calibration Setup Check Dialog Box

To access: **Calibration** > **Model Calibration** then click **Setup Check**.

Use this dialog box to compare the parameter values of power, transient duration and initial temperature read from the T3Ster calibration file with those defined in the Simcenter Flotherm model, and make adjustments to the Simcenter Flotherm model.

Objects

Object	Description
Calibration Setup Check Table	Values of power, transient duration and initial temperature from the T3Ster calibration file and from the Simcenter Flotherm model. The third column indicates: • OK: the values do not need to be adjusted. • Warning: the model values may need to be adjusted to obtain good calibration results. It is up to the user to decide whether to adjust the model values, small differences may be tolerable. **Note:** The transient duration of the model should be the same as, or longer than, the duration of the experiment (T3Ster).
Regenerate Time Patches	On by default. Time patches are regenerated based on a new transient duration.
Adjust	Click if you want the Simcenter Flotherm model values to match the T3Ster values. **Note:** Any existing scenarios are re-built and, therefore, solved results are invalidated.

Related Topics

Model Calibration Dialog Box

Load Calibration Data Progress Dialog Box

To access: Opens before the Model Calibration dialog box is opened if results are available, and when the Model Temperature to Calibrate selection is changed.

This dialog box shows the progress of loading data for the Model Calibration dialog box plots.

Objects

- Progress Bar and Times

  Shows the progress of loading Temperature vs Time data for each scenario, and then calculating and plotting the structure functions.

Related Topics

Model Calibration Dialog Box

Index