Getting Started with Simcenter™ Flotherm™

Software Version 2021.1

SIEMENS

Unpublished work. © 2021 Siemens

This material contains trade secrets or otherwise confidential information owned by Siemens Industry Software, Inc., its subsidiaries or its affiliates (collectively, "Siemens"), or its licensors. Access to and use of this information is strictly limited as set forth in Customer's applicable agreement with Siemens. This material may not be copied, distributed, or otherwise disclosed outside of Customer's facilities without the express written permission of Siemens, and may not be used in any way not expressly authorized by Siemens.

This document is for information and instruction purposes. Siemens reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Siemens to determine whether any changes have been made. Siemens disclaims all warranties with respect to this document including, without limitation, the implied warranties of merchantability, fitness for a particular purpose, and non-infringement of intellectual property.

The terms and conditions governing the sale and licensing of Siemens products are set forth in written agreements between Siemens and its customers. Siemens' **End User License Agreement** may be viewed at: www.plm.automation.siemens.com/global/en/legal/online-terms/index.html.

No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Siemens whatsoever.

TRADEMARKS: The trademarks, logos, and service marks ("Marks") used herein are the property of Siemens or other parties. No one is permitted to use these Marks without the prior written consent of Siemens or the owner of the Marks, as applicable. The use herein of third party Marks is not an attempt to indicate Siemens as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A list of Siemens' trademarks may be viewed at: www.plm.automation.siemens.com/global/en/legal/trademarks.html. The registered trademark Linux® is used pursuant to a sublicense from LMI, the exclusive licensee of Linus Torvalds, owner of the mark on a world-wide basis.

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

Table of Contents

List of Figures

Chapter 1
A Simple First Project

This tutorial provides step-by-step lessons designed to teach you the fundamentals of Simcenter™ Flotherm™ software.

By completing these lessons, you will learn the basics of starting new projects, creating geometry, defining the solution grid, solving, and displaying results.

Creating a New Project

We will create a new project using the supplied default template.

Procedure

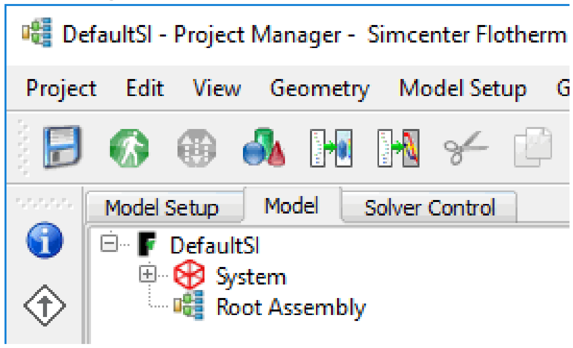
1. Start Simcenter Flotherm from the Start menu:

   MentorMA > Simcenter Flotherm 2021.1

   The default project opens.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Figure 1-1. DefaultSI Project



2. Choose **Project** > **Save As** to open the Save Project dialog, enter the following and click **OK**:

   - Project Name: Basics

Results

The project name, as shown in the title bar and in the root of the data tree, changes from DefaultSI to Basics.

Setting the Size of the Solution Domain

The solution domain in this case is a defined volume of air surrounding the model to be included in the simulation.

Procedure

1. Select the System node of the data tree.

   The System property sheet opens below the data tree.

2. Leave the Position settings at 0 and the default units as meters (m) but change the Sizes to:

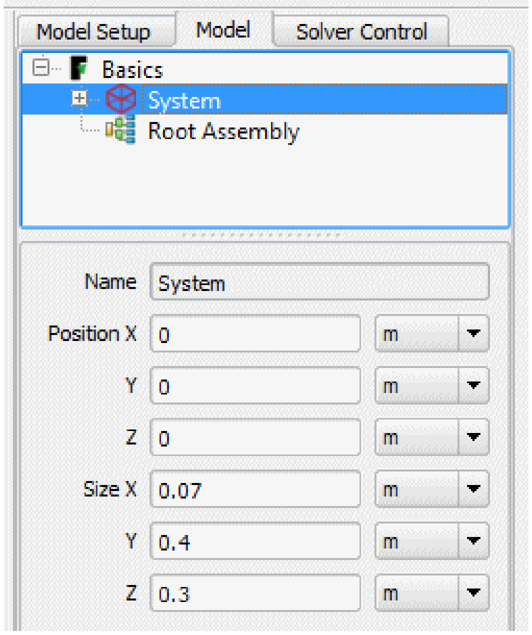
   X = 0.07, Y = 0.40, Z = 0.30

   Tip
   You can use the Tab key to move between property sheet fields.

   The System property sheet should now appear as shown in Figure 1-2.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Figure 1-2. Overall Solution Domain Size

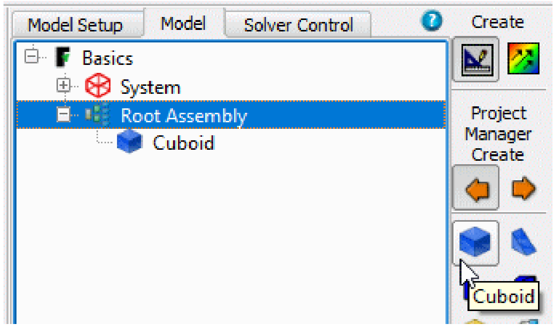


Creating a Large Plate

A Large Plate is created by adding a cuboid of mild steel.

Procedure

1. Select Root Assembly then click the **Cuboid** icon in the Project Manager Create (New Object) palette.



   A new cuboid is added to the data tree.

2. Click the new cuboid. The Cuboid property sheet is opened.

   The new cuboid size is one-tenth that of the solution domain, which is generally the default size of new objects.

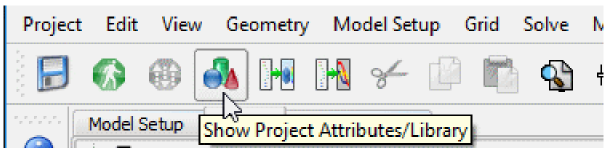
Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

3. In the **Location** tab of the Cuboid property sheet, make the following changes, you can Tab between fields:

    - Change Name to Large Plate.
    - Set Position to (mm) X = 30, Y = 100 and Z = 100
    - Set Size to (mm) X = 5, Y = 100 and Z = 150

    The cuboid is renamed in the data tree.

4. Open the Attributes library by clicking the **Show Project Attributes/Library** icon.

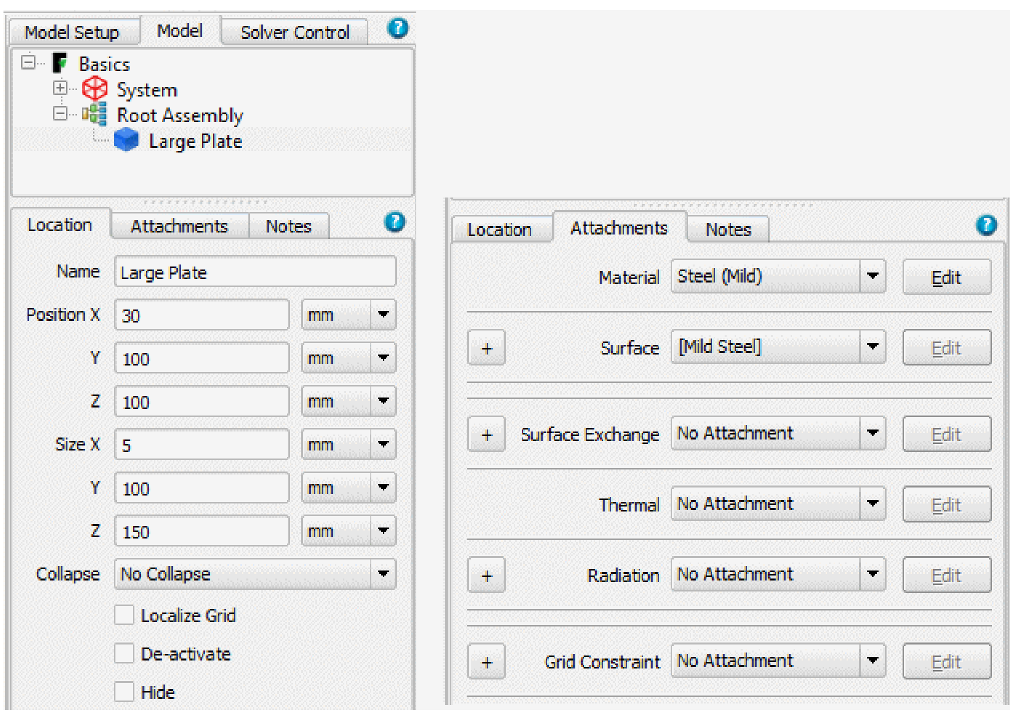


    The Project Attributes/Library is a two-tab pane (**Project** and **Library**) that opens on the right of the window.

5. Open the **Library** tab and expand the folders down to Materials > Alloys
6. Drag and drop the **Steel (Mild)** icon onto the Large Plate in the data tree.

Results

The Steel (Mild) material has been added to the Large Plate cuboid as can be seen in the **Attachments** tab of the property sheet.



Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

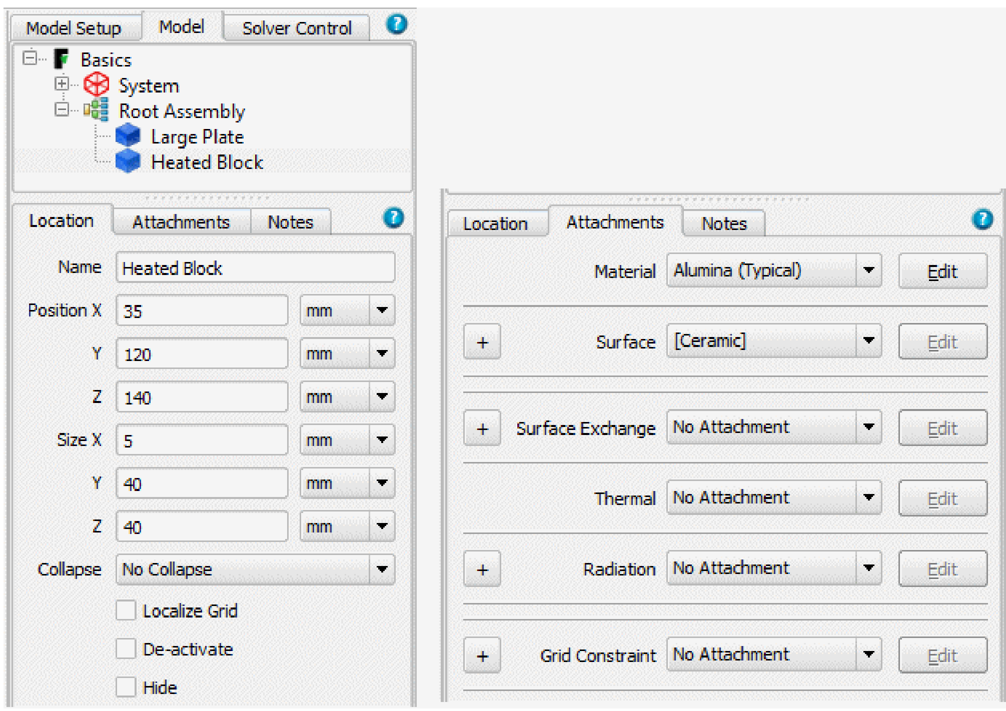
Creating a Heated Block

A cuboid is added with an attached Material attribute to represent a heated block.

Procedure

1. Select Root Assembly and add another cuboid.
2. Make the following changes to the property sheet of the new cuboid:
    - Rename the object to Heated Block
    - Set Position to (mm) X = 35, Y = 120, Z = 140
    - Set Size to (mm) X = 5, Y = 40, Z = 40
3. Expand the Attributes library down to Materials > Ceramics > Alumina (Typical) and drag and drop that material onto the Heated Block cuboid in the data tree.

Results



Creating a New Thermal Attribute and Attaching it to the Heated Block

By attaching a thermal attribute, you are making the Heated Block a source of heat.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Procedure

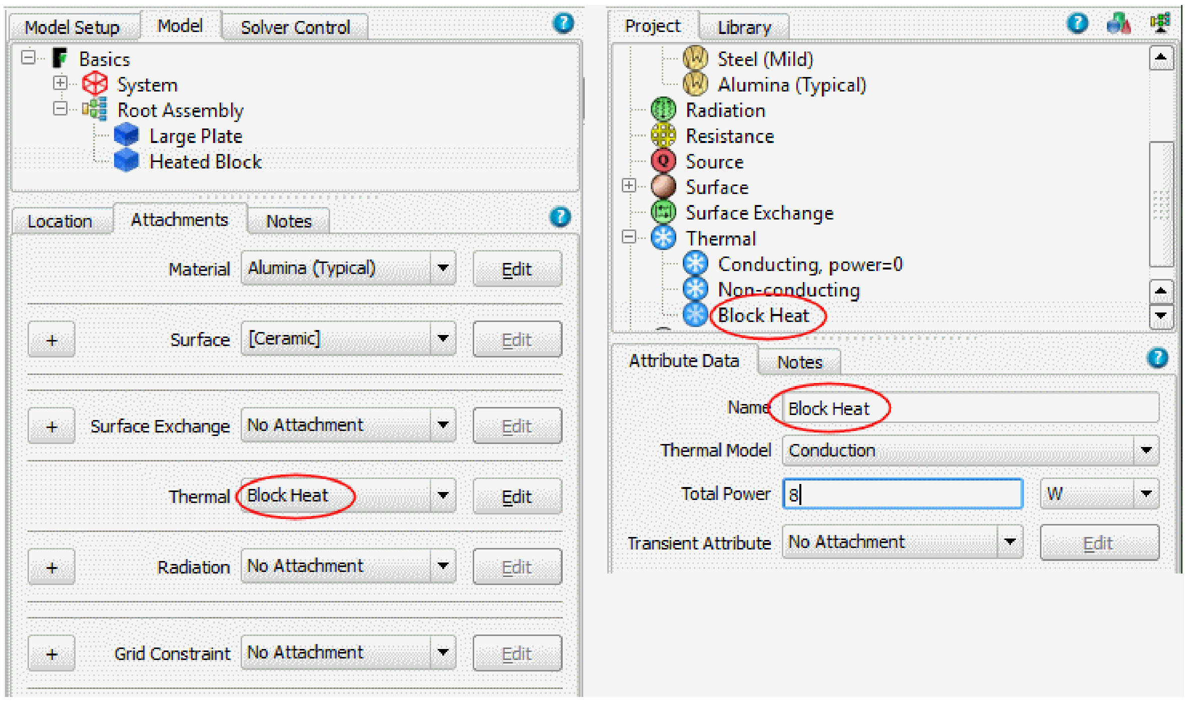
1. Open the **Attachments** tab of the Heated Block property sheet, and select Create New from the Thermal attachment dropdown list.

   The **Project** tab opens with a new Thermal attribute (Thermal:0) selected.

2. Set the following values:

   - Name = Block Heat
   - Thermal Model = Conduction
   - Total Power = 8 W

   When you change the Name field in the attribute property sheet, the name is also changed in the **Project** tab and in the Heated Block property sheet.



3. Choose **Project** > **Save**.

Note

It is recommended that you save a project at regular intervals as you are working.

Setting the Grid

The grid defines the sizes of the solved cells and, therefore, the accuracy of the model.

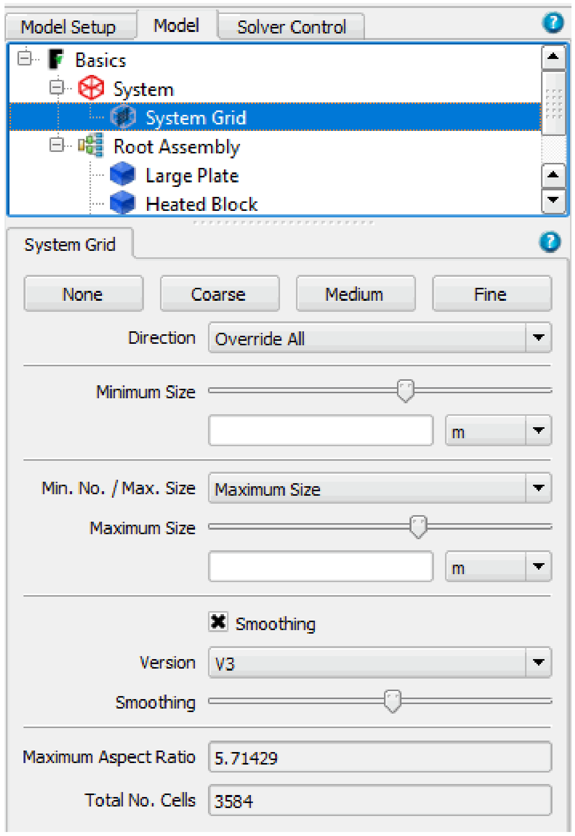
Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Procedure

1. Press R to refit the project on the drawing board, press W to display the cuboids as wireframes and then press G to display the grid.

   The grid is created by the geometry boundaries alone (that is, the keypoint grid). This will not be sufficient to achieve an accurate solution, so more grid must be added. There are a number of methods available, but for this example we will use a pre-set system grid.

2. Select System > System Grid from the data tree to display the System Grid property sheet, then click **Fine**.

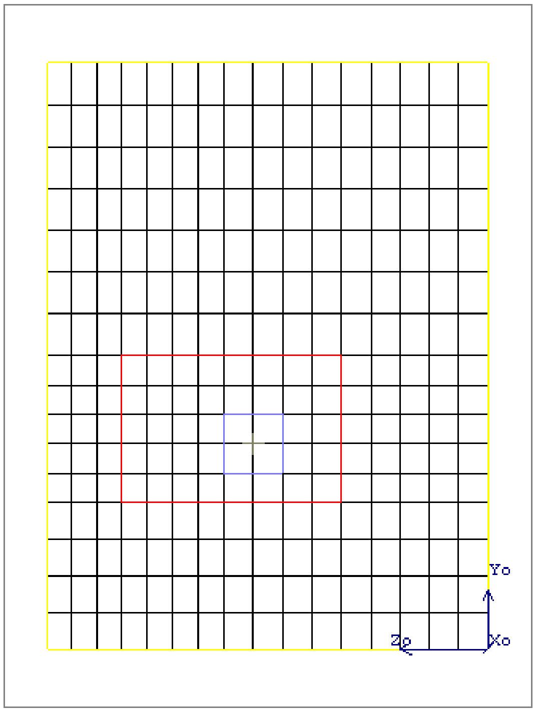


Results

The grid display updates to that shown in Figure 1-3. The program has defined positions for the minimum and maximum cell sizes using a smoothing algorithm.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Figure 1-3. Result of Applying a “Fine” Grid



Solving the Project

If you have followed the earlier sections, your model should now be ready for a solver run.

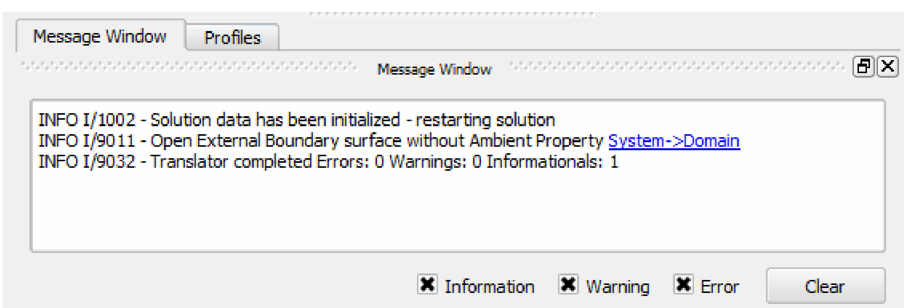
Prerequisites

- Save the project.

Procedure

Either choose **Solve** > **Solve** or click the **Solve** icon.

A sanity check is performed first and the Message Window reports that an open external boundary does not have an ambient attached.



For now, ignore this since the default ambient will be sufficient for our purposes.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

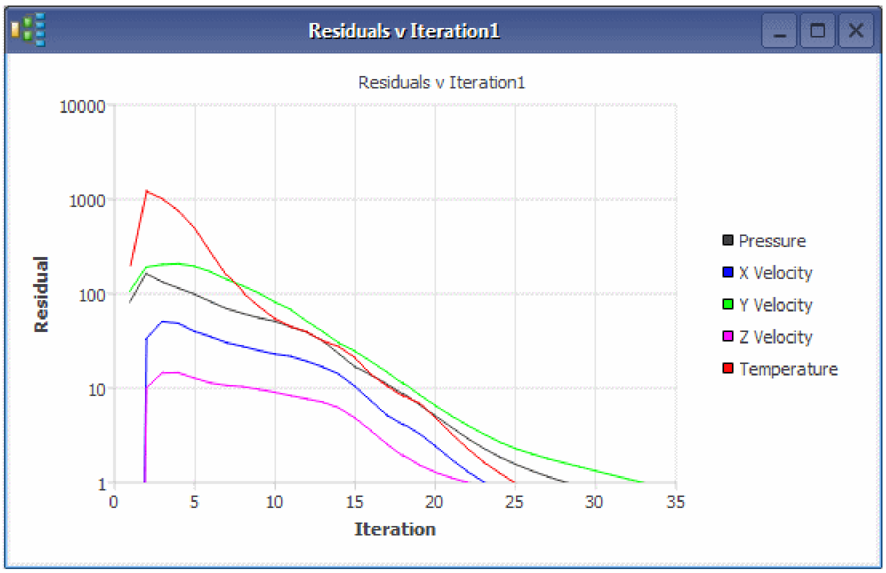
After the sanity check has been performed, the solution continues and the Profiles window opens.

Note

Generally, it is good practice, before running a solve, to perform a sanity check by choosing **Solve** > **Sanity Check**. Message descriptions and troubleshooting information can be found in the *Simcenter Flotherm User Guide*.

Results

The solution completes to show converged plots.



Visualizing the Results

Use the GDA to view plots of results superimposed over the model.

Procedure

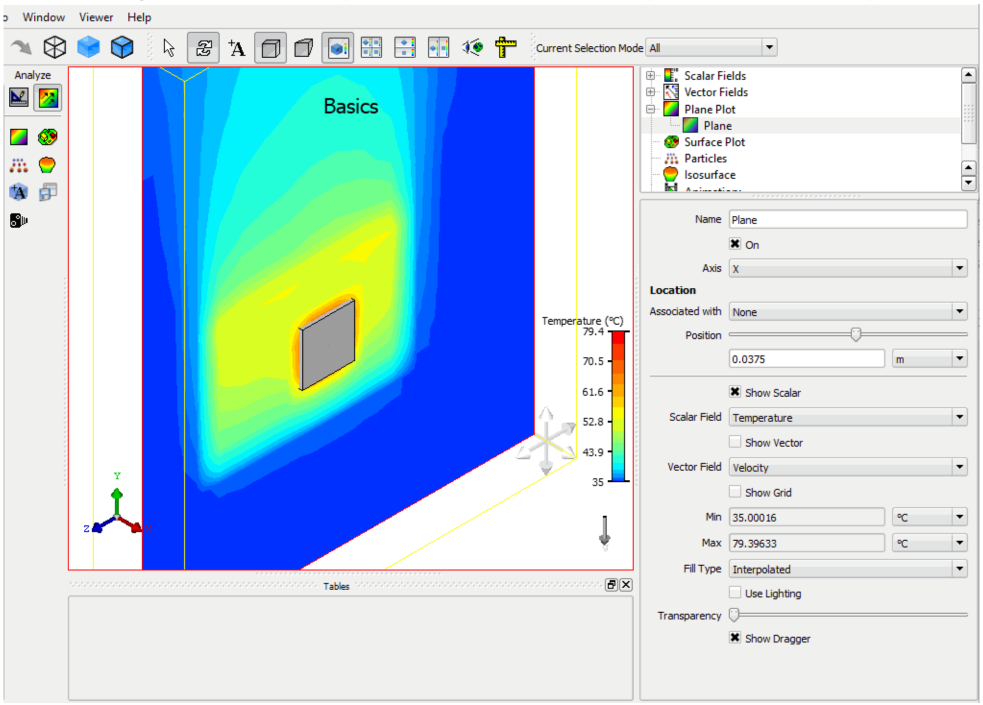
1. Click the **Analyze** icon and then press SHIFT+I to change to an isometric view.
2. In the Results Tree, right-click the Plane Plot node and choose **Create Plane Plot**.

   A new plot (Plane) is added as a child of the Plane Plot node and the Temperature scalar plot is displayed.
3. Change the plot position.

   Change 0.035 m to 0.0375 m, see Figure 1-4.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

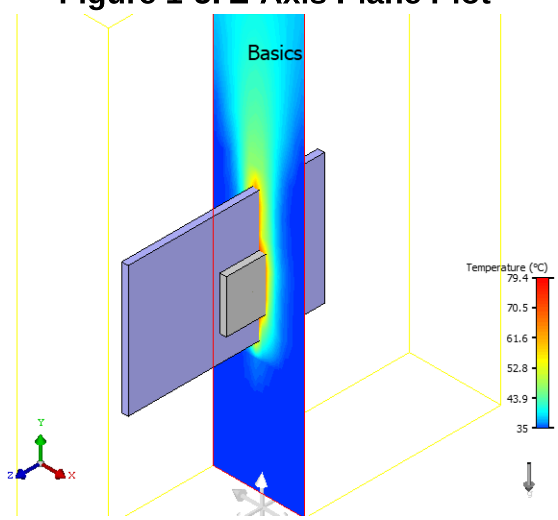
Figure 1-4. Temperature Plane Plot (Default X-Axis)



4. With the new plane plot still selected, change the Axis to Z.

   The result is shown in Figure 1-5.

Figure 1-5. Z-Axis Plane Plot



Viewing Results in Tables

Tables of results data are provided by the Project Manager in Analyze mode.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

For example, you can investigate:

- The amount of heat conducted from the heated block.
- The amount of heat convected from the surface of the heated block to the air.

Procedure

1. Click the **Analyze** icon.
2. Select the Heated Block in the data tree.

   Three tables are made available in the Tables pane: Solid Conductor Fluxes, Solid Conductor Temperatures, and Solid Conductors Summary.

Figure 1-6. Example Solid Conductors Summary Table

Tables — Solid Conductor Fluxes | Solid Conductor Temperatures | Solid Conductors Summary

	Type	Min S-F Surface Temperature (°C)	Max S-F Surface Temperature (°C)	Min S-S Surface Temperature (°C)	Max S-S Surface Temperature (°C)	Conducted Heat In (W)	Conducted Heat Out (W)	Net Conducted Heat (W)	Convected Heat In (W)	Convected Heat Out (W)	Net Convected He (W)
Heated Block	Cuboid	78.8	79.3	78.1	78.7	0	7.43	-7.43	0	0.573	-0.573

Figure 1-7. Example Solid Conductor Fluxes Table

Tables — Solid Conductor Fluxes | Solid Conductor Temperatures | Solid Conductors Summary

	Face	face Area n^2)	S-S Surface Area (m^2)	Conducted Heat In (W)	Conducted Heat Out (W)	Net Conducted Heat (W)	Convected Heat In (W)	Convected Heat Out (W)	Net Convected Heat (W)	Convective Heat Transfer Coefficient (W/(m^2 K))	Conductive Heat Transfer Coefficient (W/(m^2 K))
Heated Block	X High		0	0	0	0	0	0.505	-0.505	7.16	-
Heated Block	X Low		0.0016	0	7.43	-7.43	0	0	0	-	107
Heated Block	Y High		0	0	0	0	0	0.015	-0.015	1.71	-
Heated Block	Y Low		0	0	0	0	0	0.0167	-0.0167	1.89	-
Heated Block	Z High		0	0	0	0	0	0.0189	-0.0189	2.16	-
Heated Block	Z Low		0	0	0	0	0	0.017	-0.017	1.92	-

Loading a Supplied Application Example

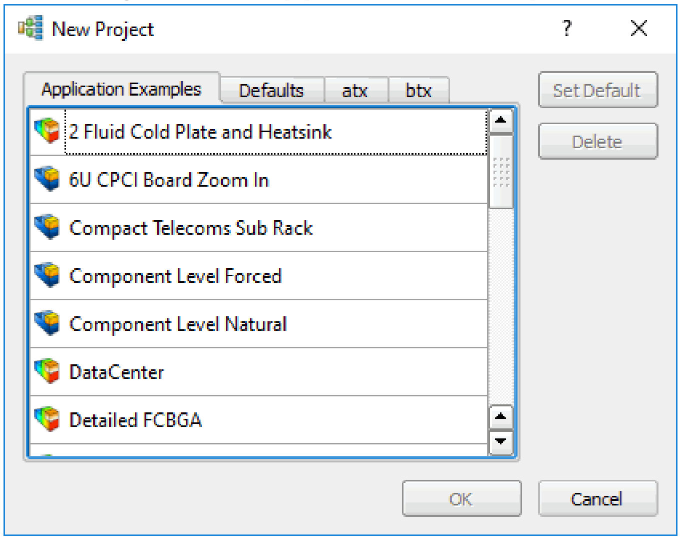
Practical application examples are supplied with the installation, some of which include results.

Procedure

1. Choose **Project > New** to open the New Project dialog, and then click the **Application Examples** tab, see Figure 1-8.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Figure 1-8. Application Examples



Examples that have included solved results have icons with plot color shading.

2. Select an example and click **OK**.

   The project is opened.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Chapter 2
A Quick Tour

A brief look at some of the main features of Simcenter Flotherm, with links to the relevant user documentation.

If you have completed "A Simple First Project" on page 7 then you will already be familiar with some of the functionality of Simcenter Flotherm.

Simcenter Flotherm uses a multi-window environment to present different views of the project data and provide an interface to other software packages.

The Project Manager

The Project Manager is the first application window to be displayed and is the main controlling window.

From here you can create new or load existing projects. Loaded project data is displayed in a tree hierarchy.

If Simcenter Flotherm does not start with a loaded project, initially the Project Manager displays a project stub terminating at the start of the geometry tree (Root Assembly) to which you add objects parametrically to create your own flow model. Alternatively you can load standard project templates or tutorial examples.

If the Project Manager is hidden behind another application or minimized, click the **Launch Project Manager** icon to re-display it.

For further information, see Project Manager Window in the *Simcenter Flotherm User Guide*.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Monitoring the Solution in Profiles

After a project has been constructed, you solve it to produce the results of the flow calculations.

To start the solution process, click the **Solve** icon. The Profiles window displays the solution convergence progress.

When called by the activation of the solver, the Profiles window displays a convergence plot showing the numerical errors in the calculated flow and temperature distributions as the solution proceeds. This is in the form of a plot of residual error against iteration number for steady solutions. This is repeated for each time-step for transient (time-dependent) solutions. If monitor points (virtual probes for all the solved variables) are present, then a monitor point plot is also displayed.

After solution, the Profiles window can be called at any time to display the solution plots by clicking the **Launch Profiles** icon in any application window.

For further information, see Solution Monitoring and Profile Plots in the *Simcenter Flotherm User Guide*.

Visualizing Results in Analyze Mode

The GDA displays the Simcenter Flotherm results in a 3D real-world view when in Analyze mode.

The view may be restricted to a particular section of the model, and rotated, panned, and zoomed to enable intricate structures to be displayed clearly.

Initially, the display is a 3D perspective view drawn viewing from +Z. To get a more general 3D view, click and drag the mouse across the screen.

Results from the simulations can be viewed as vectors (for flow patterns in an arbitrary plane), contour fills (for distribution in an arbitrary plane) surface temperature plots (for all the points of a given value in the 3D view) object surface values or animations of the calculated flow.

For further information, see Viewing Results in the *Simcenter Flotherm User Guide*.

Project Reporting by Tables in Analyze Mode

Tables of results data are available after solving.

Results tables can be exported in CSV files for further analysis.

For further information, see Reporting Project Data and Results in Tables in the *Simcenter Flotherm User Guide*.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Interfacing With MCAD Tools in MCAD Bridge

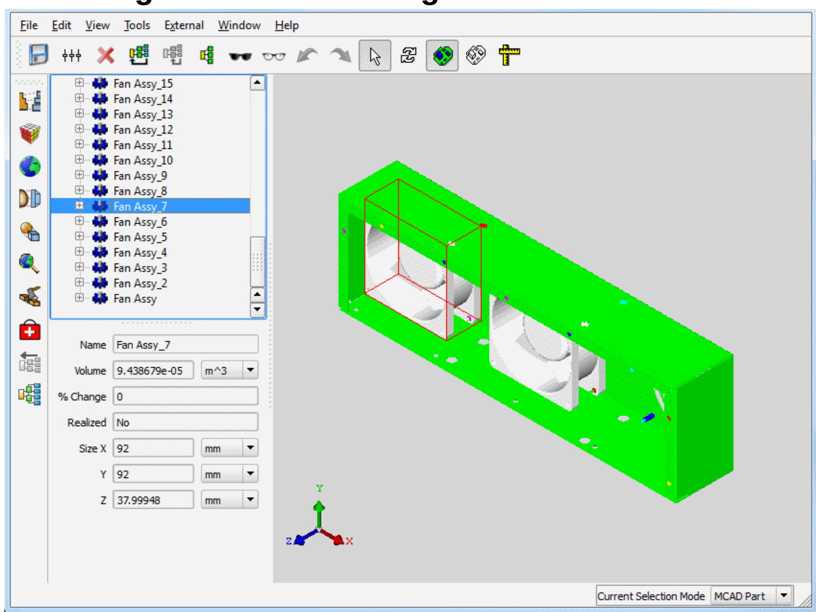
The MCAD Bridge provides an interface with external MCAD programs.

The MCAD Bridge is started by clicking the **Launch MCAD Bridge** icon

Initially, the window is blank, you have to import geometry from either the current Simcenter Flotherm project or an external MCAD file (Figure 2-1).

For further information, see the Simcenter Flotherm MCAD Bridge User Guide.

Figure 2-1. Interfacing With MCAD Tools



Building PCBs Using EDA Bridge

Use EDA Bridge to create detailed representations of PCBs.

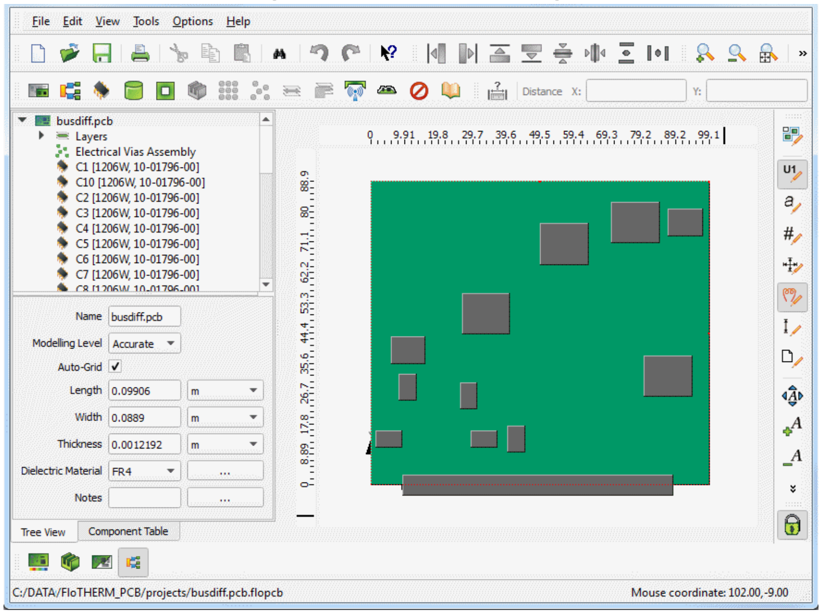
The EDA Bridge (Figure 2-2) is started by clicking the **Launch EDA Bridge** icon

Initially, the EDA Bridge window is blank. The PCB can be constructed from EDA Bridge geometry parts, or loaded from the library. Alternatively, designs can be imported from EDA interface tools using either imported IDF, or *.*floeda* files created by one of the supplied direct EDA interfaces.

For further information, see the Simcenter Flotherm EDA Bridge User Guide.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Figure 2-2. EDA Bridge



Design Optimization Using the Command Center

Use the Command Center to create scenarios for a project.

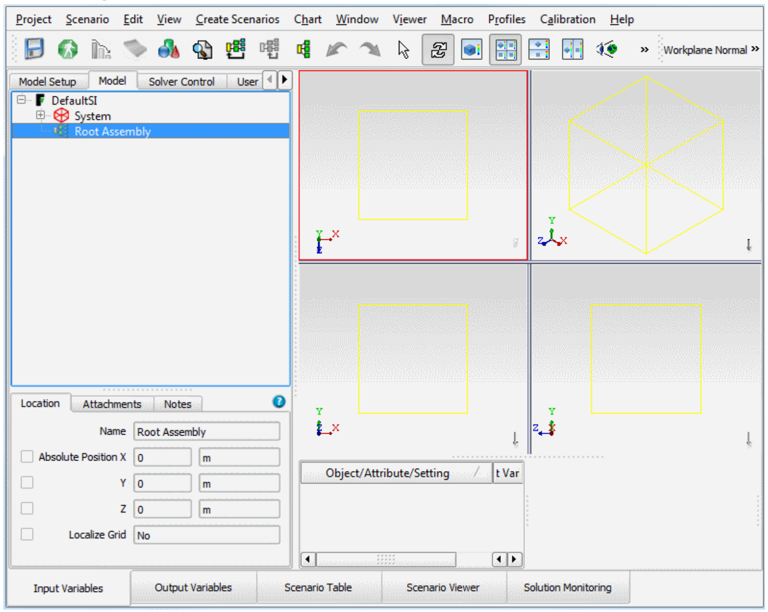
The Command Center (Figure 2-3) is opened by clicking the **Launch Command Center** icon 

Scenarios allow you to observe the effects of varying project parameters.

For further information, see the Simcenter Flotherm Command Center User Guide.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Figure 2-3. The Command Center Window



Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.