

Simcenter™ Flotherm™

EDA Bridge User Guide

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Unpublished work.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

Chapter 1	
Introduction to EDA Bridge	11
EDA Bridge or Project Manager SmartParts?.....	11
Simcenter Flotherm SmartParts.....	11
Overview Procedure When Using EDA Bridge	12
Chapter 2	
The User Interface	15
Window Layout	16
EDA Bridge Window	16
Board Data Tree	18
Component Properties Table	18
Display Area	19
Property Sheets	21
Toolbars	22
Context Sensitive Menus	24
Shortcut Keys	25
Chapter 3	
Managing Projects	27
Overview of a Project	27
Working With Project Files.....	27
FloSCRIPT.....	28
Project Operations	30
Starting a New Project	30
Choosing the Project Units	30
Setting GUI Preferences	31
Setting the 3D View Controls (Pan, Zoom, and Rotate)	31
Saving a Project	31
Project Management Dialog Boxes	33
Global Units Dialog Box	34
GUI Preferences Dialog Box	35
View 3D Controls Dialog Box	38
Save Model Dialog Box	39
Chapter 4	
Defining the Board Layout.....	41
Reference Designators	43
Coordinate System.....	43
Viewing the Board and Components.....	44
Changing the View	44
Searching for Components and Functional Groups	44

Coloring Components	45
Annotating Components	46
Displaying Component Notes	47
Adding, Replacing, and Copying Geometry	48
Adding New Geometry Using the Icons	48
Adding Geometry Using Popup Menus	49
Adding Existing Geometry Using the Library Pane	49
Replacing Geometry With Library Items	50
Making a Single Copy of a Component	51
Making a Pattern of a Component	52
Editing Geometry	53
Moving Objects	53
Resizing Objects by Dragging Boundaries	54
Measuring the Distance Between Object Reference Points	55
Aligning and Distributing Components	56
Using Property Sheets	57
Using Component Tables	58
Adding a Material Option From the Library	59
Removing a Material Option	59
Viewing Material Properties	59
Adding Component Notes	60
Importing Designs	61
Importing EDA Board Designs	61
Importing ODB++ Board Designs	62
Importing IDF Board Designs	64
Importing Components CSV Layouts	65
Importing a Power List	66
Generating an Import Power List Report	66
Exporting Designs	68
Transferring an EDA Bridge Model to Simcenter Flotherm	68
Exporting FLOPLC Files	69
Exporting IDF Files	70
Exporting Components CSV Layouts	70
Exporting a Power List	70
Filtered and Deactivated Objects	71
Filtering Components	71
Displaying Filtered-Out Components	72
Deactivating Objects	73
Libraries	74
Viewing Libraries	74
Adding Library Components	75
Manual Swapping With Library Components	75
Automated Swapping With Library Components	75
Importing and Exporting Libraries	76
Importing Library Items	76
Saving Geometry in the Library	78
Creating New Libraries	78
Searching for a Library Item	79
Downloading a Found Library Item	80

Table of Contents

Adding a Found Library Items it to the Project	80
Board Layout File Formats	81
Power List CSV File Format	82
Components CSV Layout File Format	83
Board Layout Dialog Boxes	85
Library Item Selector Dialog Box	86
Pattern Selected Items Dialog Box	87
Move Selected Items Dialog Box	88
Component/Functional Group Search Dialog Box	89
Material Library Selector Dialog Box	90
View Existing Material Data Dialog Box	91
Edit Text Dialog Box	92
Library Selector for Component Import Dialog Box	93
Select Column Types Dialog Box	94
Component Filter Options Dialog Box	96
Library Selector Dialog Box	98
Create New Libraries Dialog Box	99
Library Search Facility Dialog Box	100
Update or Replace? Dialog Box	101

Chapter 5

Modeling Objects

103

Working With Modeling Objects	104
MotherBoards	104
DaughterBoards	104
Layers Properties and Procedures	107
Layers	107
Saving and Retrieving a Layer Stackup to and From a Library	108
Layer Patches Properties and Procedures	109
Layer Patches	109
Defining a Layer Patch	109
Generating Layer Patches From Images or EDA Data	110
Electrical Vias Properties and Procedures	112
Electrical Vias	112
Defining an Area of Electrical Vias	112
Generating Electrical Vias in an Electrical Vias Assembly From an Image	113
Generating Electrical Vias in Dielectric Layers From Images	114
Rectangular Components	116
Cylindrical Components	117
Functional Groups Properties and Procedures	118
Functional Groups	118
Creating Functional Groups	119
Rectangular Cutouts	120
Heatsinks	121
Thermal Vias	124
Cans	125
Potting Compound	126
Placement Keepout Regions	126

Modeling Objects Property Sheets and Dialog Boxes.....	128
MotherBoard Property Sheet	129
DaughterBoard Property Sheet.....	130
Group Layer Property Sheet.....	132
Layer Property Sheet	133
Layer Patch Property Sheet.....	135
Electrical Vias Assembly Property Sheet.....	136
Electrical Via Property Sheet	137
Component Property Sheet.....	139
Cylindrical Component Property Sheet	141
Functional Group Property Sheet	143
Rectangular Cutout Property Sheet.....	144
Heatsink Property Sheet	145
Thermal Via Property Sheet	147
Can Property Sheet	148
Potting Compound Property Sheet	149
Placement Keepout Regions Property Sheet	150
Layer Trace Processing Dialog Box	151
Dielectric Layer Vias Processing Dialog Box	153
Board Vias Processing Dialog Box	155
Chapter 6	
Frequently Asked Questions	157
Board Definition FAQs	157
Component Definition FAQs.....	158
Layer Definition FAQs	161
Solution FAQs	161
Transfer From EDA Bridge to Simcenter Flotherm FAQs	162

Index

List of Figures

Figure 1-1. EDA Bridge Process Flow	13
Figure 2-1. EDA Bridge Window	17
Figure 2-2. Component Properties Table.....	19
Figure 2-3. 2D View of MotherBoard and DaughterBoard.....	20
Figure 2-4. Property Sheet Setting for Multiple Objects.....	21
Figure 2-5. Property Sheet Menu Request Box	22
Figure 2-6. Notes and Notes Button in Property Sheet	22
Figure 4-1. Color Components Dropdown List.....	45
Figure 4-2. Component Coloring Example 1.....	46
Figure 4-3. Component Coloring Example 2.....	46
Figure 4-4. Component Notes	47
Figure 4-5. Resize an Object by Dragging	54
Figure 4-6. Measure Distance Button	55
Figure 4-7. Measured Distance Between Object Corners	56
Figure 4-8. Object Reference Points	56
Figure 4-9. Equispacing and Aligning Components	57
Figure 4-10. Filtering Smaller Components	71
Figure 4-11. Filtered Out Components in Graphics Display Area	73
Figure 4-12. Library Pane	74
Figure 4-13. Components CSV Layout File Displayed in a Spreadsheet	84
Figure 5-1. Parallel DaughterBoard.....	105
Figure 5-2. Perpendicular X-High DaughterBoard	105
Figure 5-3. Perpendicular Y-Low DaughterBoard	106
Figure 5-4. Perpendicular DaughterBoard Offset	106
Figure 5-5. Default MotherBoard Layers	107
Figure 5-6. Icon Representation of Metallic and Dielectric Layers in the Data Tree	107
Figure 5-7. Processed Layer Patches in Data Tree and on a Board.....	111
Figure 5-8. Electrical Vias Generated in a Dielectric Layer	115
Figure 5-9. Component Derived Power.....	117
Figure 5-10. Division of a Board Into Functional Group Areas	118
Figure 5-11. Color by Power or Power Density	119
Figure 5-12. Functional Group Bounding Box	120
Figure 5-13. Heatsink on Component, Front View	121
Figure 5-14. Inline and Staggered Heat Sink Pins.....	121
Figure 5-15. Heatsink Centered on Component	122
Figure 5-16. Default Positioning	122
Figure 5-17. Position After Applying an Offset	122
Figure 5-18. Heatsink on Component Side	122
Figure 5-19. Multiple Component Heatsinks	123
Figure 5-20. Component Heights in Component Table.....	123

Figure 5-21. Thermal Via	124
Figure 5-22. Unstaggered and Staggered Thermal Via Arrays	125
Figure 5-23. A New Can Covering Components.....	125
Figure 5-24. Adding Potting Compound	126
Figure 5-25. Keepout Region at the Origin	127
Figure 6-1. Constructing a Detailed Component Model	159

List of Tables

Table 2-1. Shortcut Keys	25
Table 4-1. Geometry Options by Tree Type	48
Table 4-2. Conversion of True and False Values on Components CSV Data Import	84

Chapter 1

Introduction to EDA Bridge

The EDA Bridge application window of Simcenter™ Flotherm™ software models printed circuit boards in detail, and should be used when a PCB is on the critical heat transfer path.

EDA Bridge or Project Manager SmartParts?	11
Simcenter Flotherm SmartParts	11
Overview Procedure When Using EDA Bridge	12

EDA Bridge or Project Manager SmartParts?

Use EDA Bridge to create detailed representations of Printed Circuit Boards (PCBs) and to interface with Electronic Design Automation (EDA) products.

EDA Bridge can be used both independently or by importing files extracted from EDA software tools.

The representation created in EDA Bridge is more detailed than can be created with the Simcenter Flotherm PCB SmartPart and includes the board outline, layer stack up, via distribution, and component layout.

The copper distribution in each layer is modeled with a tessellated (resolution is user controlled) thermal conductivity map. This ‘filtering’ enables the complex copper distribution in a board to be included without resorting to excessive geometric detail.

Note

- ❑ The higher granularity of the PCB representation from EDA Bridge means that it should be used whenever the PCB is on the critical heat transfer path: natural convection and conduction cooled applications predominantly.
-

Simcenter Flotherm SmartParts

The Simcenter Flotherm SmartParts that can be used to model PCBs and their components are PCB SmartParts, Component SmartParts and Compact Component SmartParts. Descriptions of their applicability are given here.

PCB SmartParts

The PCB SmartPart can be used to create basic models that represent a PCB as an orthotropic block. The PCB SmartPart can be used for applications where the board is not on the critical heat transfer path (forced convection applications) or when the construction details of the board have yet to be determined. PCBs in other situations should be created with EDA Bridge.

Component SmartParts

PCB components can be represented by adding Component SmartParts to PCB SmartParts. Component SmartParts are owned by the parent PCB SmartPart and model the overall thermal effects of a component modeled discretely or smeared over the board. This SmartPart can be used for applications where the board is not on the critical heat transfer path (forced convection applications) or when the construction details of the board have yet to be determined. Components on PCBs in other situations should be created with EDA Bridge.

Compact Components

Compact Component SmartParts are resistor network representations of packages and can be used when the resistance values are known for the component. These resistance values may be available from the component supplier, or can be generated by FloTHERM PACK.

FloTHERM PACK can also be used to generate detailed component models if the highest accuracy is needed, but these require additional mesh. Compact Component SmartParts can be saved to the Library Manager and imported into the EDA Bridge window.

Overview Procedure When Using EDA Bridge

A high-level procedure to give an overview of the process. You create the MotherBoard in EDA Bridge (either manually, or using file import) and then transfer it to Simcenter Flotherm.

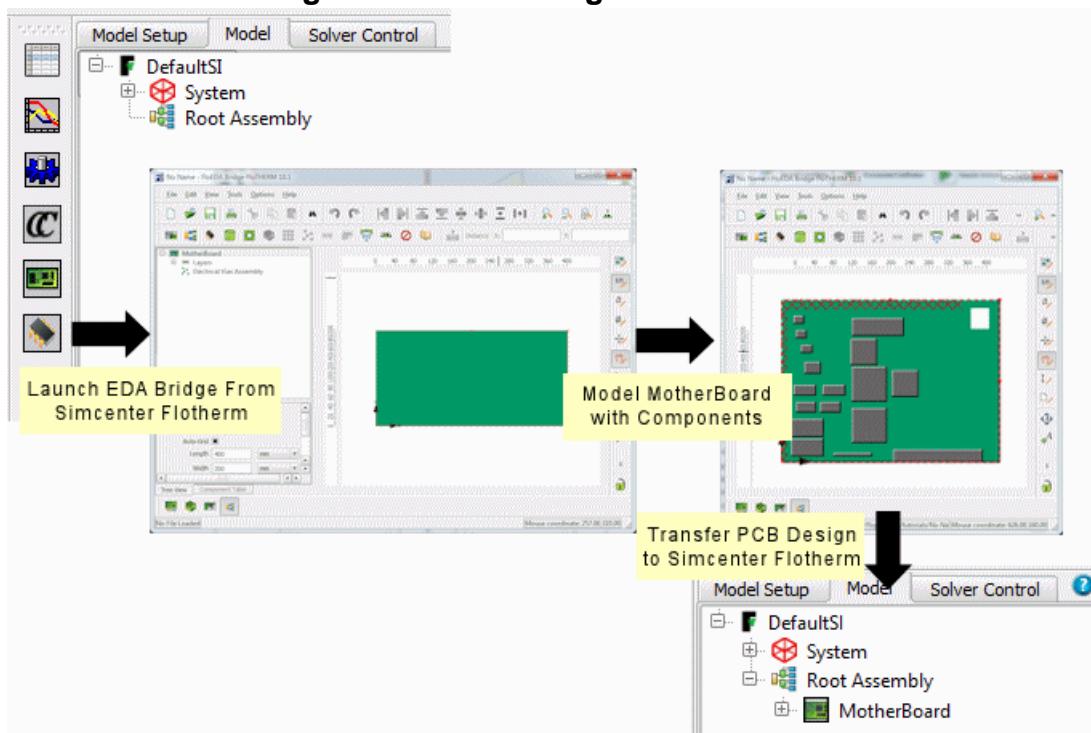
Procedure

1. Start EDA Bridge from Simcenter Flotherm.
2. Model the PCB as a MotherBoard design object.
3. Save the MotherBoard in case further changes are to be made to the design in EDA Bridge.
4. Transfer the MotherBoard from EDA Bridge to Simcenter Flotherm.

Results

The MotherBoard is created as an EDA Board SmartPart in Simcenter Flotherm.

Figure 1-1. EDA Bridge Process Flow



Related Topics

[Transferring an EDA Bridge Model to Simcenter Flotherm](#)

[Defining the Board Layout](#)

[Modeling Objects](#)

Chapter 2

The User Interface

This section describes the main features of the EDA Bridge application window.

Window Layout	16
EDA Bridge Window	16
Board Data Tree	18
Component Properties Table	18
Display Area	19
Property Sheets	21
Toolbars	22
Context Sensitive Menus	24
Shortcut Keys	25

Window Layout

The EDA Bridge window is divided into two main areas: a hierarchical tree and a display area that shows 2D or 3D views.

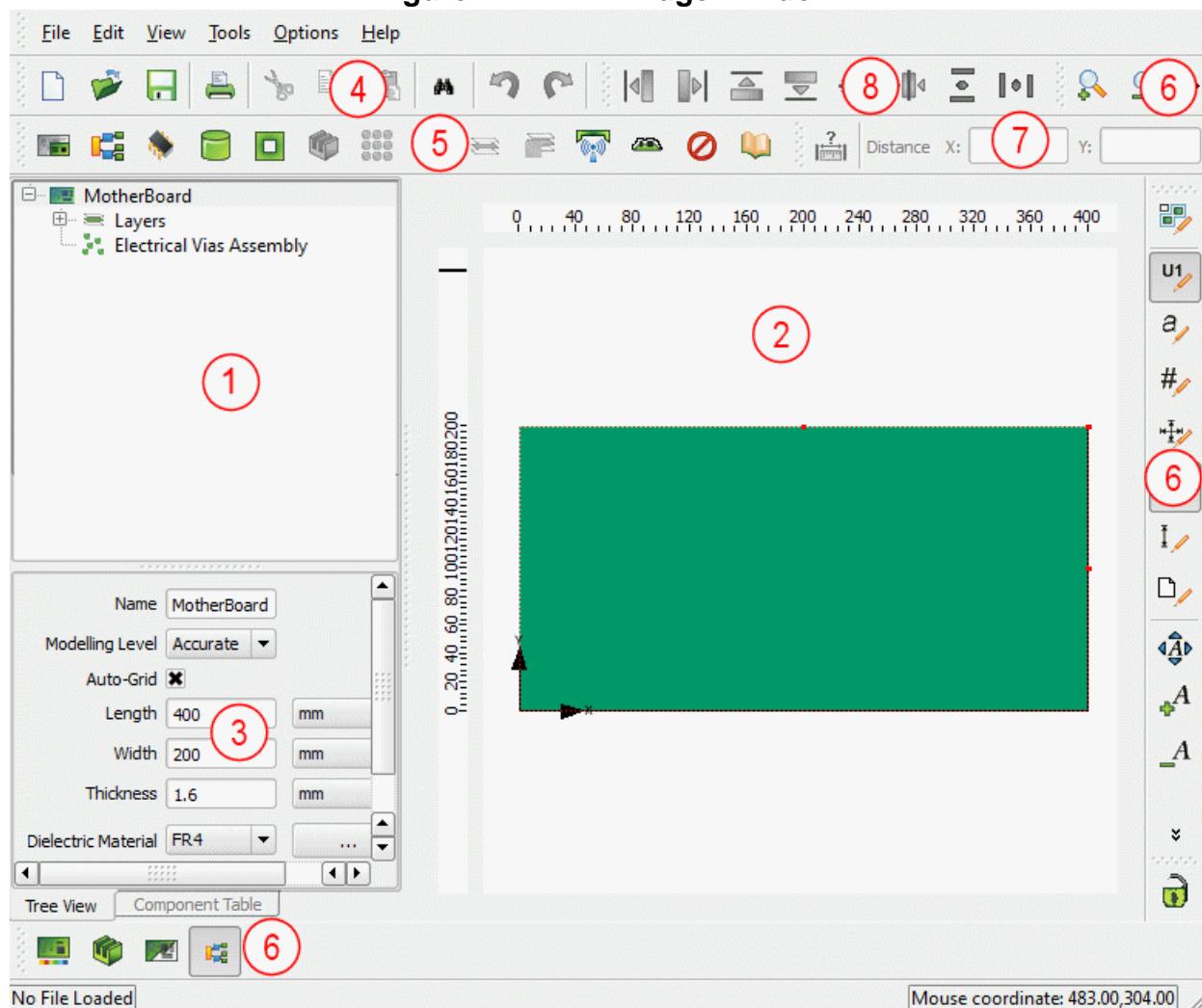
The elements of the window are shown in [Figure 2-1](#).

EDA Bridge Window	16
Board Data Tree	18
Component Properties Table	18
Display Area	19
Property Sheets	21

EDA Bridge Window

The EDA Bridge window displays the board layout and a data tree.

Figure 2-1. EDA Bridge Window



Key to Figure 2-1:

1. Board Data Tree/Component Properties Table depending on tab selection
2. Display Area
3. Property Sheets
4. Standard toolbar.
5. New Geometry toolbar.
6. The viewing toolbars comprising:
 - Viewing Options toolbar.
 - Component Options toolbar.

- Annotation toolbar.
 - Advanced Display Options toolbar.
7. Measurement toolbar.
8. Alignment and Distribution toolbar.

There is also a [Libraries](#), which is not shown in [Figure 2-1](#).

Related Topics

[Toolbars](#)

[Defining the Board Layout](#)

Board Data Tree

The data tree (**Tree View** tab) shows the constituent parts of the board in a hierarchical order of parent, child, and grandchild.

Any components or DaughterBoards on the MotherBoard are her children and any components on the DaughterBoard are her grandchildren.

There are only two levels of boards: MotherBoard and DaughterBoards. DaughterBoards cannot have DaughterBoards of their own.

Objects are selected in the tree by clicking. Multiple selections can be made using Shift+click (contiguous multiple selection) or Ctrl+click (non-contiguous multiple selection).

Related Topics

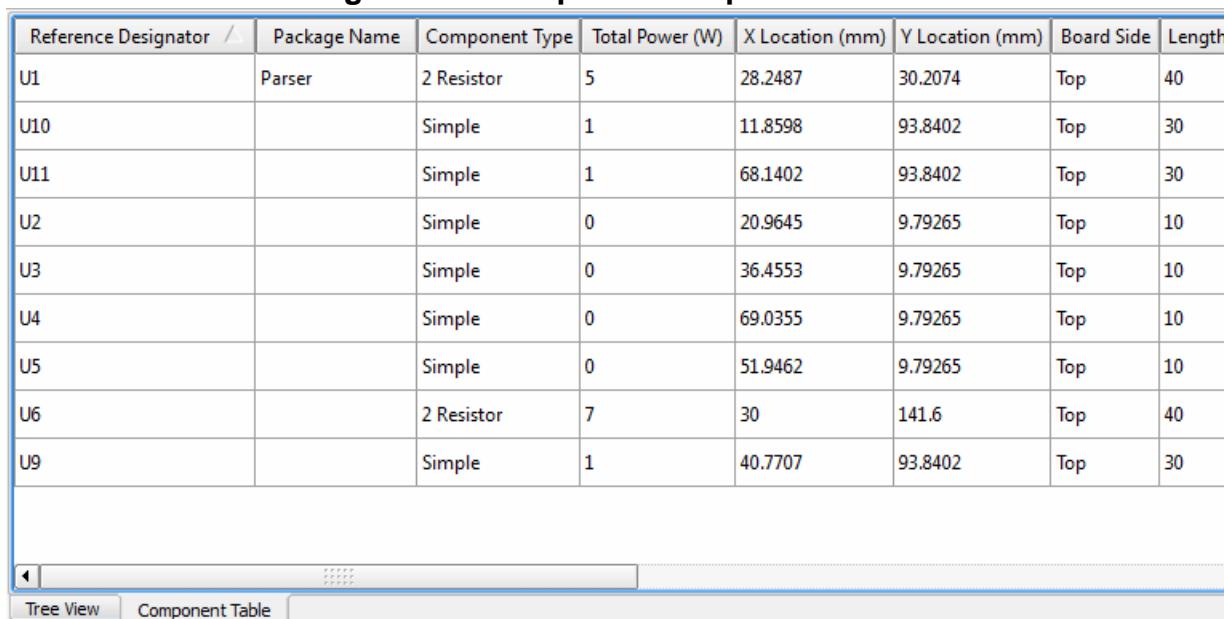
[Component Properties Table](#)

[Defining the Board Layout](#)

Component Properties Table

If components are present, a table of the component properties can be displayed (**Component Table** tab).

Figure 2-2. Component Properties Table

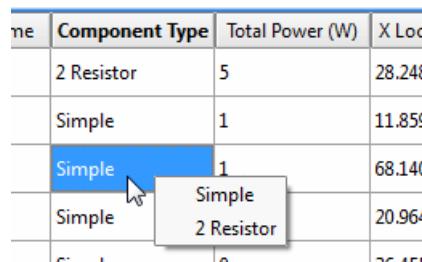


Reference Designator	Package Name	Component Type	Total Power (W)	X Location (mm)	Y Location (mm)	Board Side	Length
U1	Parser	2 Resistor	5	28.2487	30.2074	Top	40
U10		Simple	1	11.8598	93.8402	Top	30
U11		Simple	1	68.1402	93.8402	Top	30
U2		Simple	0	20.9645	9.79265	Top	10
U3		Simple	0	36.4553	9.79265	Top	10
U4		Simple	0	69.0355	9.79265	Top	10
U5		Simple	0	51.9462	9.79265	Top	10
U6		2 Resistor	7	30	141.6	Top	40
U9		Simple	1	40.7707	93.8402	Top	30

Click a header to sort the table by that value, click again to reverse the sort order.

You can edit any component in the model by using this table. There are two types of data entry cells:

- Text entry boxes containing numerical values. Double-click the cell to enter edit mode.
- Menu request boxes for modeling options. Left-click the table cell to open a dropdown menu of options.



Related Topics

[Board Data Tree](#)

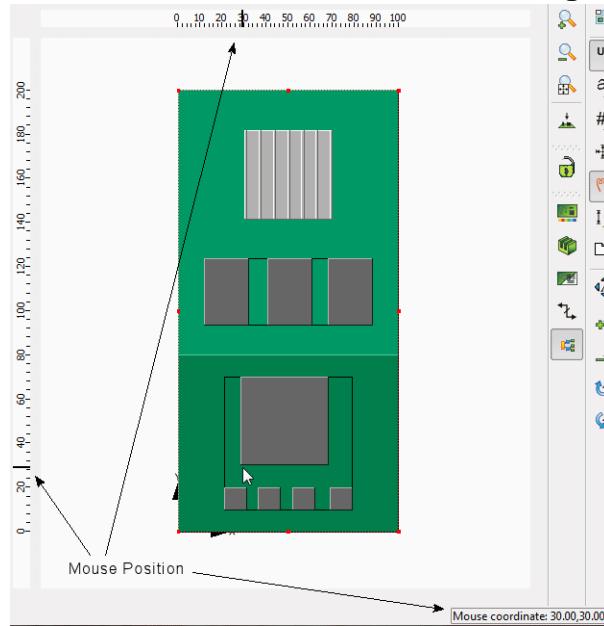
Display Area

The display area provides 2D or 3D views of the physical model.

2D View

By default, the MotherBoard is drawn in 2D over the X-Y plane with the coordinate origin in the left bottom corner.

Figure 2-3. 2D View of MotherBoard and DaughterBoard



Horizontal and vertical rulers indicate the location of the mouse with respect to the bottom left corner of the MotherBoard. The exact location is also indicated on the far right-hand side of the bottom status bar. The units are those set in the Global Units dialog box (**Options > Global Units**).

The board can be viewed in 2D from the top, bottom, back, and side by choosing the respective option after clicking the **View From** icon.

Objects can be moved in all 2D views by dragging with the mouse. However, the resizing of objects by dragging boundaries can only be performed in the top or bottom 2D views. See “[Moving Objects](#)” on page 53.

Multiple components can be selected by a left-mouse click-drag. Any component inside or bisected by the sketched rectangle will be selected when the mouse button is released. The click-drag should not start over a component, otherwise that component will be moved.

3D View

To change to 3D view, click the **View From** icon and choose **3D**. Unlike the 2D view, no rulers are shown and objects cannot be resized or re-located using the mouse.

Alternatively, use the mouse controls to change the view:

- Use the left button to rotate.
- Use the middle button to pan.
- Use the right button to zoom (up for zoom-out, down for zoom-in).

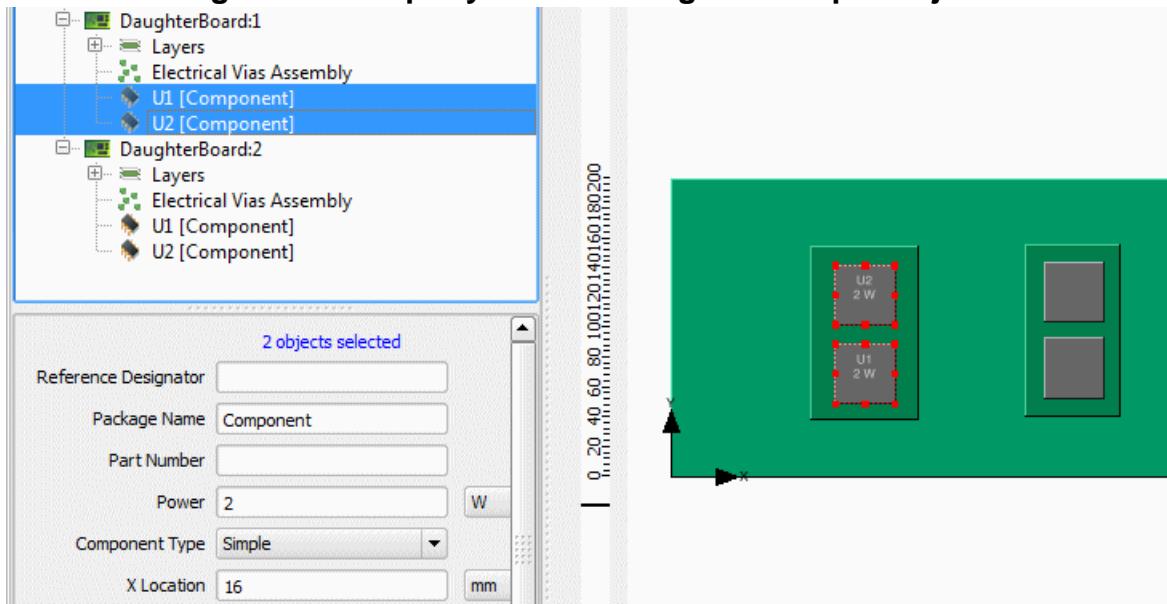
These default mappings can be changed using the [View 3D Controls Dialog Box](#).

Property Sheets

When you select an object when the tree view is shown, then the property sheet for the object is displayed beneath the data tree.

When two or more objects of the same type are selected, data can be entered once and applied to all the selected items. Settings that are different are left blank and have to be changed individually.

Figure 2-4. Property Sheet Setting for Multiple Objects



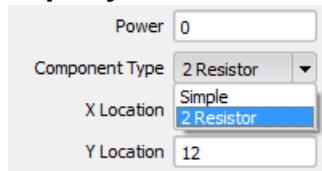
There are two types of data entry fields in the property sheets:

- Text entry boxes for numerical values
- Menu request boxes for modeling options

In both cases, left-click to activate the entry box.

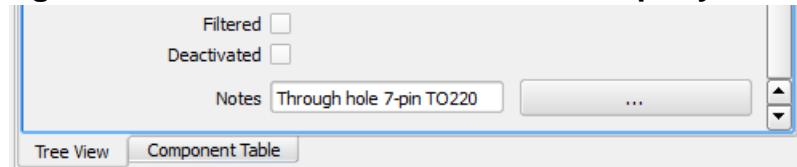
- For text boxes, the text insertion mark appears ready to change the name or value.
- For menu request boxes, a choice of modeling options is displayed, see [Figure 2-5](#).

Figure 2-5. Property Sheet Menu Request Box



Construction notes can be added for all objects except layer patches and cutouts. Clicking the [...] button (see [Figure 2-6](#)) opens the [Edit Text Dialog Box](#) for adding notes.

Figure 2-6. Notes and Notes Button in Property Sheet



Toolbars

The toolbars can be moved and either left floating or docked along the top, bottom, left side or right side of the window. Hover text describes each toolbar icon.

Standard Toolbar



This toolbar provides shortcut icons for commonly used **File > ...** and **Edit > ...** menu options.

New Geometry Toolbar

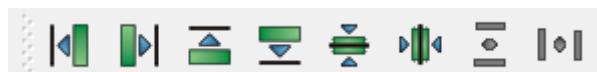


Use this toolbar to add new geometry and open the library.

By default, new objects are selected on creation, ready for editing using their property sheets.

If you do not want objects selected on creation, then deactivate “Select Objects When Created” in the [GUI Preferences Dialog Box](#).

Alignment and Distribution Toolbar



This toolbar provides shortcut icons for **Edit > Align > ...** menu options.

Use this toolbar to align and distribute components.

Viewing Options Toolbar



Use this toolbar to change the view in the display area.

- **Refit** changes the location and size of the geometry display to fit the window.
 - Click the **View From** icon to display a list of options that change the orientation of the displayed board:
 - **3D** changes the view to 3D. All the other options are 2D views.
 - **Through To Bottom** displays the board from the top. The geometry is represented in wireframe so that you can see through the board and view the components on the bottom of the board.

Component Options Toolbar



Use this toolbar to prevent accidental dragging of components. The icons switch the locking of graphical movement on or off. The locked/unlocked status is stored and persists between sessions.

Annotation Toolbar



Use this toolbar to annotate your design in the 2D view. The size and rotation of annotations may be modified.

- The **Annotate** icon enables you to choose to annotate nothing, only the selected components or all components.
 - The annotated information is selectable using the **Annotate with ...** icons.
 - **Auto Size Font** automatically resets all visible annotations so that they fit within the objects to which they refer.

If more annotations are added then you must click **Auto Size Font** again to refit the new annotation.

- The **Rotate Font** icons rotate the annotation in increments of 90 degrees.

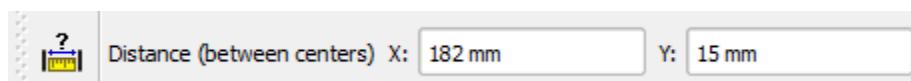
The units displayed in the annotations are those selected in the [Global Units Dialog Box](#).

Advanced Display Options Toolbar



Use this toolbar to color the board and display heatsinks, and functional groups.

Measurement Toolbar



When two objects are selected, this toolbar shows the center-to-center distances in the X and Y directions between the objects.

Context Sensitive Menus

Context-sensitive (right-click) menus are available for geometry editing functions and for editing the library and loading library items.

Geometry Editing Context-Sensitive Menus

To display a context sensitive menu, select the geometry to be changed in either the data tree or drawing area and right-click.

The option list varies according to the geometry selected.

- **Add to new functional group** creates a new functional group which will include the selected object(s). This might be commonly used after importing IDF files where component groupings need to be redefined. This then enables easy restructuring of the layout at the functional group level to reflect a new design or the ability to save functional groups of components to a library for subsequent retrieval.
For more information, see “[Functional Groups](#)” on page 118.
- **Add to new can** creates a can enclosing the selected objects.
- **Paste** copies the contents of the system paste buffer into the 2D view. If the pasted geometry is the result of a copy, it will be placed offset from the original, otherwise it is located at the same position from which it was cut.

Library Context-Sensitive Menus

The option list varies according to whether a library or library item is selected.

- **Create new Library** calls the [Create New Libraries Dialog Box](#) to add a directory to the library.
- **Refresh** refreshes the library tree to include any directories or items that have been added to the library using external software.
- **Load** copies the selected library item into the project tree below the selected parent. For example, heatsinks and thermal vias can be added to a component selected in the project tree.
- **Export** saves the selected library item as a **.library* file.
- **Import** calls a file browser to choose a **.library* or **.pdml* file to import into the Simcenter Flotherm library.
 - A **.library* file is listed as a new library folder in the data tree as a child of the selected library.
 - A **.pdml* file, normally representing a detailed or network component, is copied into the selected library folder.

Shortcut Keys

Key combinations for menu options, also known as hotkeys.

[Table 2-1](#) lists the EDA Bridge shortcut keys.

Table 2-1. Shortcut Keys

Key	Function
Ctrl+N	Start a new project
Ctrl+O	Load a project
Ctrl+S	Save the project
Ctrl+P	Print the drawing area
Ctrl+Q	Quit EDA Bridge
Ctrl+Z	Undo
Ctrl+Y	Redo
Ctrl+X	Cut
Ctrl+C	Copy
Ctrl+V	Paste
Ctrl+F	Find
Del	Delete the selected geometry

Chapter 3

Managing Projects

How to manage EDA Bridge projects.

Overview of a Project	27
Working With Project Files	27
FloSCRIPT	28
Project Operations	30
Starting a New Project	30
Choosing the Project Units	30
Setting GUI Preferences	31
Setting the 3D View Controls (Pan, Zoom, and Rotate)	31
Saving a Project	31
Project Management Dialog Boxes	33
Global Units Dialog Box	34
GUI Preferences Dialog Box	35
View 3D Controls Dialog Box	38
Save Model Dialog Box	39

Overview of a Project

Each board design study is encapsulated within a unique EDA Bridge project.

All data relating to the study is stored in a single *.flopccb project file created by EDA Bridge. This *.flopccb file may be stored anywhere on your file system.

Working With Project Files

To work on a EDA Bridge project, you either create a new project or load an existing one.

After a project is saved, it can be reloaded later, and if required, exported for archive as well as for use in other projects.

It is *important to save your project* after thermal solution in case you need to view the results again. If you change the model after solution, the results are detached from the project. To restore the results, you must undo the edit.

Related Topics

[Starting a New Project](#)

Saving a Project

FloSCRIPT

Advanced users can run a FloSCRIPT file from EDA Bridge.

The following EDA Bridge operations are supported by FloSCRIPT:

- File menu actions: Load, Save, Save As, Transfer, Quit, Import > Import ODB, Import > Import CSV Layout, Import > Import FLOEDA
- Tools menu actions: Filter Components, Component Library Swap
- Modifying the [MotherBoard Property Sheet](#) values
- Adding new geometry
- Editing object property sheets, with the following exceptions, which are not supported:
 - Importing layer or via images
 - Adding or deleting materials using the [Material Library Selector Dialog Box](#)

All other EDA Bridge operations, including the following, are not supported by FloSCRIPT:

- Edit menu actions
- View menu actions
- [Display Area](#) operations, such as moving objects
- Right-click context menu actions
- Modifying component details using Component Tables

Note

 The Filter Components and Component Library Swap actions are recorded to FloSCRIPT files when accessed from the Tools menu but are not recorded during model import.

In recorded FloSCRIPT files the parameters of the [Component Filter Options Dialog Box](#) are specified in m, W or W/m², regardless of the units displayed in the dialog box. When writing a FloSCRIPT file specifying these parameters the values must also be expressed in m, W or W/m².

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 *EDALogFile<number>.xml*.

Note

 Only the last five log files are retained when a new EDA Bridge 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) “EDALogFile” string.

To run an EDA Bridge FloSCRIPT file, choose **Macro > Play FloSCRIPT** and then select the XML file.

Related Topics

[FloSCRIPT \[Simcenter Flotherm User Guide\]](#)

Project Operations

This section describes general project tasks including setting user preferences.

Starting a New Project	30
Choosing the Project Units	30
Setting GUI Preferences	31
Setting the 3D View Controls (Pan, Zoom, and Rotate).....	31
Saving a Project	31

Starting a New Project

A new project clears the current project from the window and includes a new MotherBoard.

Procedure

Choose **File > New**.

If EDA Bridge detects any unsaved changes, then you will be given the option to save them before the new session is started.

Results

The initialized board design is a single MotherBoard comprising four layers, set in a Single Card Slot environment with an ambient air temperature of 45°C and speed of 2 m/s.

Choosing the Project Units

You can set the default units used by the EDA Bridge window and dialog boxes.

Procedure

1. Choose **Options > Global Units** to open the Global Units dialog box.
2. For each parameter, select the units from the dropdown list.
3. Click **OK** to save any changes.

The default units are overridden locally by changing the unit options in property sheets.

Related Topics

[Global Units Dialog Box](#)

[Using Property Sheets](#)

Setting GUI Preferences

You can set your preferences for the appearance of the EDA Bridge window.

Procedure

1. Choose **Options > GUI Preferences** to open the GUI Preferences dialog box.
2. Use the fields and check boxes to set your GUI preferences.
3. Click **OK** to save any changes.

Related Topics

[GUI Preferences Dialog Box](#)

Setting the 3D View Controls (Pan, Zoom, and Rotate)

Changing the default mouse buttons that control pan, zoom, and rotate when viewing 3D images.

Procedure

1. Choose **Options > View Controls** to open the View 3D Controls dialog box.
The dialog box has separate sets of check boxes for Pan, Zoom, and Rotate.
2. Use the check boxes to reassign the mouse buttons if required.
3. Click **Close** to save any changes.

Related Topics

[View 3D Controls Dialog Box](#)

Saving a Project

It is recommended that you save the project regularly.

Restrictions and Limitations

Filenames of up to 31 characters are allowed.

Procedure

You have a choice:

If you want to...	Do the following:
Save a new project or save an existing project under a new name.	<ul style="list-style-type: none">Choose File > Save As, enter a filename and select the destination using the file browser.
Save the existing project under the same name.	<ol style="list-style-type: none">Choose File > Save. If the project has been saved, then the Save Model dialog box may be opened. It will not be opened if the Do not show this again option in the Save Model dialog box has been checked.You can use the Save Model dialog box to add model notes and provide an option for creating another version of the project by incrementing the filename. This version incrementing enables you to easily keep a record of your sequence of design changes.

Checking the Do not show this again option in the Save Model dialog box stops the dialog box from appearing and **File > Save** overwrites the current project.

Results

EDA Bridge projects have the *.flopccb* file extension and can be re-loaded by choosing **File > Load** and selecting the project file from the file browser.

Related Topics

[Save Model Dialog Box](#)

Project Management Dialog Boxes

This section describes the dialog boxes used when managing projects.

Global Units Dialog Box	34
GUI Preferences Dialog Box.....	35
View 3D Controls Dialog Box.....	38
Save Model Dialog Box	39

Global Units Dialog Box

To access: **Options > Global Units**

Use this dialog box to set the default units in the window and dialog boxes.

Objects

- A list of the variable types.
 - The default units for each variable type are shown.

Usage Notes

To change the default units, click the unit field and select the unit type from the popup menu. Click **OK** to apply the setting.

Tip  Units can be changed locally in property sheets. For example, the global setting for length may be millimeters (mm), but you may prefer to enter a length in inches (in). You can set the property sheet input field units using the local popup menu before entering the data. This “local” unit will be remembered for that object in preference to the unit set in this “global” dialog box.

Related Topics

[Choosing the Project Units](#)

[Using Property Sheets](#)

GUI Preferences Dialog Box

To access: **Options > GUI Preferences**

Use this dialog box to set preferences and to customize the appearance of the EDA Bridge window. The settings are remembered (and will be retained until you change them) for future sessions of EDA Bridge.

Objects

Field	Description
Use Tool Tips	Turns hover text on or off.
Select Objects When Created	Sets new objects selected ready for immediate editing.
Snap Grid Size	Sets the gap size between the snap grid lines used for aligning objects. When dragging objects in the display area, they snap up, down, left, or right to the nearest snap grid line, that is, by the snap grid size. The default snap grid size is 1 mm.
Show Report After CSV Power Import	Enables the generation of an Import Power List Report when importing a power list.
ODB++ Component Outline	Specifies how components defined in ODB++ files are imported. Choose from: <ul style="list-style-type: none">• Body – Include the package body only. Excludes pins. The default setting.• Envelope – Include the package body and any pins.

Field	Description
ODB++ Image Scaling Factor	<p>Sets the factor by which the width and height of extracted layer images are increased during ODB++ import. The image resolution will increase by the square of the scaling factor. Default is 1.</p> <p>The default value is generally suitable for most board designs. If the extracted images are not sufficiently accurate representations of the board layers, a higher scaling factor can be used.</p> <p> Note:</p> <ul style="list-style-type: none"> Depending on complexity of the board design, specifying a large scaling factor may lead to a significant increase in the time taken to generate layer images and import the board design. As image resolution increases the accuracy of the representation of copper features increases, meaning that the % Coverage may change. With the scaling factor set to 1 a board design imported in EDA Bridge 2021.1 or later will show slightly different % Coverage values than the same board imported in EDA Bridge 2020.2 or earlier. This is due to improvements made to the image extraction process.
Number Of Ticks in Legend	Sets the number of intervals in the results legend.
Legend Font Increment	Sets the scale increase or decrease in size for the legend text.
Legend Width	Sets the legend width.
Legend Height	Sets the legend height.
Axis Font Increment	Sets the scale increase or decrease in size for the axis text.
IDF Small Edge Tolerance	Determines how accurately the bounding outlines of keepout regions and the board shape are represented on IDF and .floeda import. Features with edges below the entered size are removed.
Use Unrestricted Reference Designator Naming	Select to stop automatic naming of reference designators.
Retain Part Number on Library Swap	Check this check box to keep the part number of a component after the component is swapped for a library component, see “ Replacing Geometry With Library Items ” on page 50. Unchecked by default.

Field	Description
Retain filtered setting on Library Swap	<p>Check this box to keep the filtered status of a component after the component is swapped for a library component, see “Replacing Geometry With Library Items” on page 50.</p> <p>Unchecked by default.</p> <p> Note: If a component is transferred to Simcenter Flotherm in a filtered state, the grid constraints are not calculated and applied to the local grid region. They must be manually applied if the component becomes ‘unfiltered’ in a subsequent analyses.</p>
Retain deactivated setting on Library Swap	<p>Check this box to keep the deactivated status of a component after the component is swapped for a library component, see “Replacing Geometry With Library Items” on page 50.</p> <p>Unchecked by default.</p>
Maximum Resolution of Longest Side	Places an upper limit on the selectable values of Resolution of Longest Side in layer and via processing dialog boxes.

Related Topics

[Generating an Import Power List Report](#)

[Importing a Power List](#)

[Reference Designators](#)

View 3D Controls Dialog Box

To access: **Options > View Control**

Use this dialog box to assign which mouse buttons control the 3D view.

Description

The selected mouse button, or key+mouse button, controls pan, zoom, and rotate when viewing the geometry in 3D.

Select one of the Mouse Buttons (Left, Middle, or Right) and, optionally, the Control Button or the Shift Button.

Objects

Field	Description
Pan	
Button Selection	The default setting for pan control is the <i>Middle</i> Mouse Button.
Zoom	
Button Selection	The default setting for zoom control is the <i>Right</i> Mouse Button.
Rotate	
Button Selection	The default setting for rotate control is the <i>Left</i> Mouse Button.

Related Topics

[Display Area](#)

[Setting the 3D View Controls \(Pan, Zoom, and Rotate\)](#)

Save Model Dialog Box

To access: **File > Save** when the project has already been saved.

Use this dialog box to overwrite or version increment an existing project and add modeling notes. The dialog box consists of an editable text panel containing the project notes and any incremental history.

Objects

Field	Description
Model Notes	A panel for user comments for this save (incremental or otherwise).
File Name	A read-only field indicating the full pathname of the project. If you need to change the path or filename, then use File > Save As .
Increment Revision	If checked, the date of the incremental save, who saved it and the filename the project was saved under, along with any comments added, are recorded and displayed the next time the dialog box is opened. In addition, appends a revision number to the project filename to create a new project: <i><projectname>-<n>.floppcb</i> where <i><projectname></i> is the name of the original loaded project and <i><n></i> is the revision number. The timestamp for the incremental revision appears in the project notes the next time the dialog box is opened. If not checked, then clicking Save overwrites the current project, however, any comments typed in the Model Notes section are stored with the project.
Do not show this again	If checked, stops the dialog box from being displayed.

Related Topics

[Saving a Project](#)

Chapter 4

Defining the Board Layout

How to display and build the representation of the physical board layout.

The board layout is displayed in two main viewing formats: a data tree view and a graphical view. See “[Window Layout](#)” on page 16 for a full description of the window layout.

Reference Designators	43
Coordinate System	43
Viewing the Board and Components	44
Changing the View	44
Searching for Components and Functional Groups	44
Coloring Components	45
Annotating Components	46
Displaying Component Notes	47
Adding, Replacing, and Copying Geometry	48
Adding New Geometry Using the Icons	48
Adding Geometry Using Popup Menus	49
Adding Existing Geometry Using the Library Pane	49
Replacing Geometry With Library Items	50
Making a Single Copy of a Component	51
Making a Pattern of a Component	52
Editing Geometry	53
Moving Objects	53
Resizing Objects by Dragging Boundaries	54
Measuring the Distance Between Object Reference Points	55
Aligning and Distributing Components	56
Using Property Sheets	57
Using Component Tables	58
Adding a Material Option From the Library	59
Removing a Material Option	59
Viewing Material Properties	59
Adding Component Notes	60
Importing Designs	61
Importing EDA Board Designs	61
Importing ODB++ Board Designs	62
Importing IDF Board Designs	64
Importing Components CSV Layouts	65
Importing a Power List	66
Generating an Import Power List Report	66

Exporting Designs	68
Transferring an EDA Bridge Model to Simcenter Flotherm	68
Exporting FLOPLC Files	69
Exporting IDF Files	70
Exporting Components CSV Layouts.....	70
Exporting a Power List	70
Filtered and Deactivated Objects.....	71
Filtering Components	71
Displaying Filtered-Out Components.....	72
Deactivating Objects.....	73
Libraries	74
Viewing Libraries	74
Adding Library Components	75
Manual Swapping With Library Components	75
Automated Swapping With Library Components	75
Importing and Exporting Libraries	76
Importing Library Items	76
Saving Geometry in the Library	78
Creating New Libraries.....	78
Searching for a Library Item.....	79
Downloading a Found Library Item	80
Adding a Found Library Items it to the Project	80
Board Layout File Formats.....	81
Power List CSV File Format	82
Components CSV Layout File Format	83
Board Layout Dialog Boxes	85
Library Item Selector Dialog Box	86
Pattern Selected Items Dialog Box	87
Move Selected Items Dialog Box	88
Component/Functional Group Search Dialog Box	89
Material Library Selector Dialog Box	90
View Existing Material Data Dialog Box	91
Edit Text Dialog Box	92
Library Selector for Component Import Dialog Box	93
Select Column Types Dialog Box	94
Component Filter Options Dialog Box	96
Library Selector Dialog Box.....	98
Create New Libraries Dialog Box	99
Library Search Facility Dialog Box	100
Update or Replace? Dialog Box	101

Reference Designators

Reference Designators are used to identify components.

Reference Designators comprise at least two characters; by default single-character reference designators are appended with the “1” character, multiple occurrences of the same single-character reference designator are appended thereafter with “2”, “3”, and so on. Therefore, A, B, C become A1, B1, C1, and U, U, U will become U1, U2, U3 in the project design.

You can switch off this naming convention using the Use unrestricted Reference Designator naming field in the [GUI Preferences Dialog Box](#).

Caution

- When using unrestricted reference designator naming it is possible to have components with non-unique reference designators.
-

Coordinate System

By default, the MotherBoard is drawn over the X-Y plane with the origin in the left bottom corner as indicated above.

All components are located relative to the origin of the MotherBoard.

Mouse Coordinates

The coordinates of the mouse as it passes over the 2D display area are displayed in the bottom right-hand corner of the window.

These coordinates are used as a constant reference while using the mouse to reposition geometry.

Viewing the Board and Components

The graphics display area shows 2D or 3D views of the physical shape of the board.

Changing the View.....	44
Searching for Components and Functional Groups.....	44
Coloring Components	45
Annotating Components	46
Displaying Component Notes	47

Changing the View

The default view of the board is from the top, but there are also options to view from the bottom, front, or side.

Procedure

Click the **View From** icon in the Viewing Options toolbar and choose the view from the drop-down menu.

When rotating the board, the origin and axes directions rotate with the board.

Searching for Components and Functional Groups

Components or functional groups can be highlighted in the data tree and display area.

Restrictions and Limitations

- The search is limited to Components and Functional Groups within the project. Other objects are ignored.

Procedure

1. Choose **Edit > Find** or press Ctrl+F to open the Component/Functional Group Search dialog box.
2. Enter a text string in the Find field and check Reference Designator or Package Name depending on whether you are searching by Reference Designator or by Package Name. For Functional Groups, Package Name is equivalent to Function Name.
3. Check Match Case if the search is to be case-sensitive.
4. Check Partial match if only a partial match of the search string is required.
5. Click **Find**.

Results

The Components and Functional Groups that match the search criteria are highlighted in the data tree and display area.

Related Topics

[Component/Functional Group Search Dialog Box](#)

Coloring Components

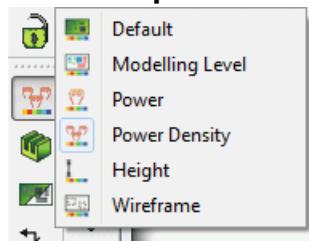
By default, the components on a board are colored gray, however, they can be shown in outline (that is, as wireframe) or colored.

by: Modeling Level, Power, Power Density, or Height.

Procedure

Choose **View > Color Components** or click the **Color Components** icon in the Advanced Display Options toolbar, then select an option, see [Figure 4-1](#).

Figure 4-1. Color Components Dropdown List



Note

 When coloring components based on Modeling Level, the actual representation of detailed components is shown, not the bounding box. This is useful when viewing components such as the supplied TO packages which have a specific side for heatsink attachment.

Results

A legend is shown when components are colored. The range is determined by the maximum and minimum values.

[Figure 4-2](#) and [Figure 4-3](#) show examples of different component coloring.

Figure 4-2. Component Coloring Example 1

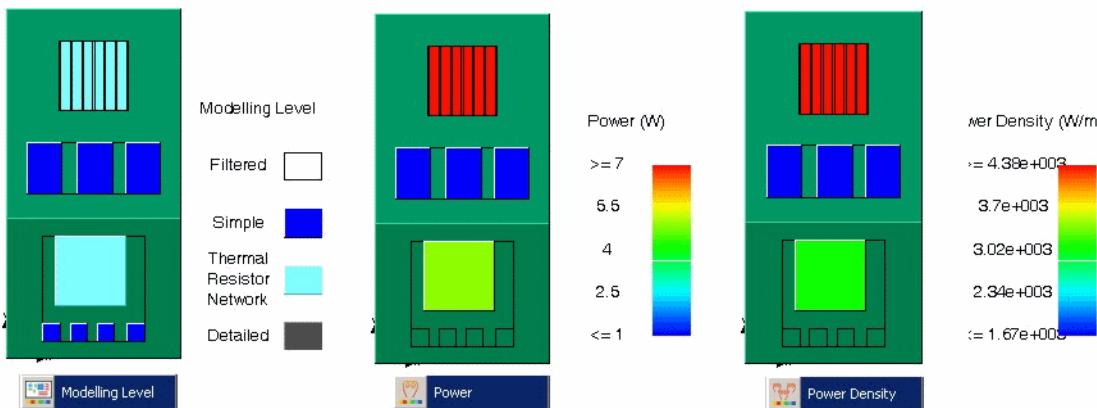
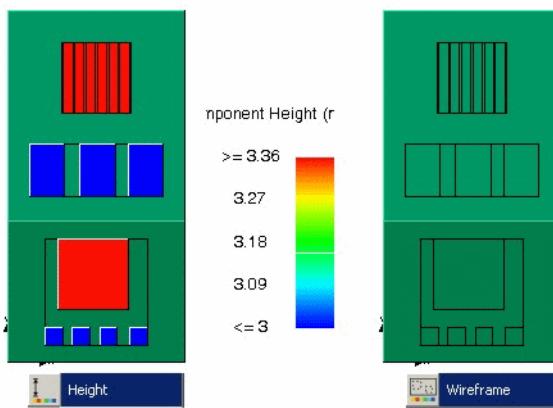


Figure 4-3. Component Coloring Example 2



Annotating Components

The components can be annotated in the display area to show their name, size, power, height, and construction notes.

Procedure

1. Choose the information to be displayed by selecting the relevant **Annotate With** icons in the Annotate toolbar.
 - To show annotations on all components, click the **Annotate** icon and select **Annotate All**.
 - To show annotations on only the currently-selected objects, click the **Annotate** icon and select **Annotate Selected**.
 - To hide all annotations, click the **Annotate** icon and select **Annotate Nothing**.

2. The font sizes of the annotations can be adjusted as follows:
 - To automatically adjust the annotation font size for all the components, click the **Auto Size Font** icon.
 - To adjust the font size manually for selected components only, click the **Increase Font** and **Decrease Font** icons.

Displaying Component Notes

Component notes can be displayed over components in the display area.

Procedure

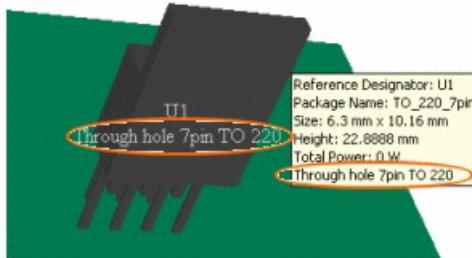
1. Click the **Annotate** icon and select either **Annotate Selected** or **Annotate All**.
2. Click the **Annotate With Notes** icon.

Results

The notes are displayed in white on top of the component, see [Figure 4-4](#).

The notes are also shown as part of the component hover text.

Figure 4-4. Component Notes



Related Topics

[Adding Component Notes](#)

Adding, Replacing, and Copying Geometry

You can add geometry to MotherBoards, DaughterBoards, and rectangular components. The geometry types that you can add depends on the parent object selected.

Adding New Geometry Using the Icons	48
Adding Geometry Using Popup Menus	49
Adding Existing Geometry Using the Library Pane.....	49
Replacing Geometry With Library Items	50
Making a Single Copy of a Component	51
Making a Pattern of a Component.....	52

Adding New Geometry Using the Icons

The New Geometry toolbar contains icons for adding new geometry to the model.

Procedure

1. Select the object to receive the new object either in the data tree or the drawing area.
2. Click a geometry icon in the left side toolbar.

The geometry types available for the selected tree item are indicated by active icons.
The possible additions are indicated in [Table 4-1](#)

Table 4-1. Geometry Options by Tree Type

Tree Selection	Geometry Options
MotherBoard	DaughterBoard, rectangular component, cylindrical component, functional group, cutout, can, potting compound and board placement keepout region. All new objects are located at the origin of the MotherBoard.
DaughterBoard	Rectangular component, cylindrical component, functional group, cutout, can, potting compound and board placement keepout region. All new objects are located at the origin of the DaughterBoard.
Rectangular Component	Heatsink and Thermal via. The heatsink and thermal via are spread across the component.
Functional Group	Rectangular component, and cylindrical component.
Layer	Layer patch and layer.
Electrical Vias Assembly	Electrical via

Results

The new item is then added to the data tree and appears as the top item in the display area.

Related Topics

[Editing Geometry](#)

Adding Geometry Using Popup Menus

New objects can be added to a parent object by using the context-sensitive menus.

Procedure

1. Select one or more parent objects in either the data tree or drawing area and right-click to display the context-sensitive menu. You can add the same object to more than one parent.
2. You have a choice:

If you want to...	Do the following:
Add a new object.	<ol style="list-style-type: none">1. Choose New from the context-sensitive menu. The menu options available depend on the parent object selected.2. Choose an object type.
Add an existing object from a library.	<ol style="list-style-type: none">1. Choose Add Library Item from the context-sensitive menu. The Library Item Selector dialog box opens2. Navigate to the relevant library, select the object to be added and click OK.

Results

The added object appears in the data tree as a child of the parent object(s). Only valid objects can be added from a library.

Related Topics

[Adding Existing Geometry Using the Library Pane](#)

[Editing Geometry](#)

[Library Item Selector Dialog Box](#)

Adding Existing Geometry Using the Library Pane

The library is a resource of components that can be added to a PCB.

Restrictions and Limitations

- Library items that can be added must be one of the following:
 - A detailed FloTHERM PACK model.
 - An assembly containing a compact component or network assembly, for example downloaded from FloTHERM PACK.
 - A component created in Simcenter Flotherm PCB or EDA Bridge and previously saved to a library.

Procedure

1. Click the **Show Library** icon to display the library pane between the data tree and the drawing area.
2. Expand the library tree until the object you require is visible.
3. Select the project object (for example, component) that is to receive the library geometry (for example, heatsink) in the data tree and either double-click the library item, or right-click the library item and choose **Load** from the popup menu.

The library item will appear below the parent object in the data tree.

Related Topics

[Saving Geometry in the Library](#)

Replacing Geometry With Library Items

A EDA Bridge object can be swapped for an object of the same type in the library using the library pane or popup menu.

Procedure

Select the geometry in the EDA Bridge model, then either:

- Double-click the replacement geometry in the library pane
 - or
- Right-click, choose **Replace with library item** and select the replacement object using the Library Item Selector dialog box.

Results

For components, the following settings of the original object are retained:

- Reference Designator
- Part Number, if the Retain Part Number on Library Swap check box in the GUI Preferences dialog box has been checked.

- Power
- Center, that is, the center of the replacement is positioned at the center of the original component.
- Orientation
- Board Side
- Functional Layout Location
- Size
- Annotation Font Increment
- Filtered status – if the ‘Retain filtered setting on Library Swap’ option has been selected in the GUI Preferences dialog box
- Deactivated status – If the ‘Retain deactivated setting on Library Swap’ option has been selected in the GUI Preferences dialog box

Other objects are located at the same origin of the object being replaced.

Related Topics

[Adding Existing Geometry Using the Library Pane](#)

[Library Item Selector Dialog Box](#)

Making a Single Copy of a Component

Copies of existing components on the board can be made using copy and paste or by Ctrl+dragging.

Procedure

1. Select the object to be copied.
2. Click Ctrl+C (copy) then click Ctrl+V (paste).

Alternatively, hold the Ctrl key and drag the copied object to a different location.

Results

When using copy and paste, the new geometry is offset from the original.

Related Topics

[Making a Pattern of a Component](#)

Making a Pattern of a Component

Cloning components across the board in two dimensions.

Procedure

1. Select the object(s) to be patterned.
2. Choose **Edit > Pattern** to open the Pattern Selected Items dialog box.
3. Define the number of copies and their separation in one or both directions.

Related Topics

[Making a Single Copy of a Component](#)

[Pattern Selected Items Dialog Box](#)

Editing Geometry

Basic object location changes can be made in the 2D display area by dragging, nudging, and aligning, but for more detailed construction changes use the component property sheets and component tables.

Moving Objects	53
Resizing Objects by Dragging Boundaries	54
Measuring the Distance Between Object Reference Points	55
Aligning and Distributing Components.....	56
Using Property Sheets	57
Using Component Tables	58
Adding a Material Option From the Library	59
Removing a Material Option	59
Viewing Material Properties.....	59
Adding Component Notes	60

Moving Objects

You can use a mouse, arrow keys or dialog box.

Restrictions and Limitations

Movement snaps to the snap grid size.

Prerequisites

Make sure that the **Lock Component Move** icon is showing that lock is off,

Procedure

You have a choice:

If you want to...	Do the following:
Drag an object to a new location.	<ol style="list-style-type: none"> 1. Left-click select the object and keep the mouse button pressed then move the mouse. To constrain the movement in the horizontal and vertical directions, hold down the SHIFT key while dragging the mouse. The outline boundary of the object follows the mouse pointer. 2. Release the mouse button when the object has been moved to its new location.

If you want to...	Do the following:
Move an object incrementally (nudging).	<ol style="list-style-type: none">1. Left-click select the object.2. Click a cursor left, right, up, or down arrow movement key. <p>The object moves by snapping to the next snap grid line in the direction of the cursor key.</p>
Move an object a specified distance in the X and/or Y direction.	<ol style="list-style-type: none">1. Select the object and choose Edit > Move By to open the Move Selected Items dialog box.2. Enter a distance(s) in the required direction(s) and click OK.

Related Topics

[Move Selected Items Dialog Box](#)

[GUI Preferences Dialog Box](#)

Resizing Objects by Dragging Boundaries

Using the mouse to resize an object.

Restrictions and Limitations

- Resizing is in increments of the Snap Grid Size.
- Only applies to boards and rectangular components, other objects must be resized using the property sheets.

Prerequisites

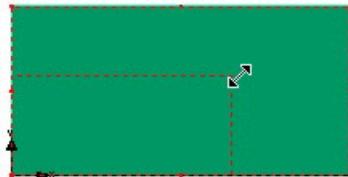
Make sure that the **Lock Component Move** icon is showing that lock is off,

Procedure

1. Select and drag the object boundary box.

To maintain the length and breadth ratio, select and drag a vertex of the bounding box.

Figure 4-5. Resize an Object by Dragging



2. Release the mouse button when the object has been resized.

Related Topics

[GUI Preferences Dialog Box](#)

Measuring the Distance Between Object Reference Points

Use the distance measurement tool to display the distance between two selected objects.

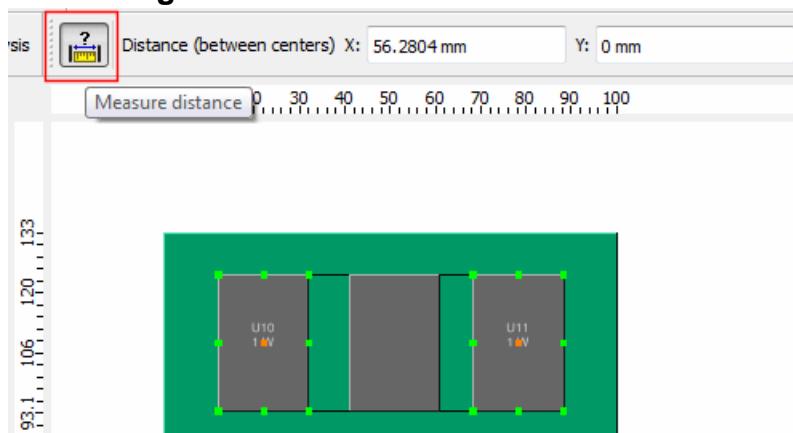
Prerequisites

- Two objects selected.
- View From: Top or Bottom. Distances in the Z direction (that is, the board depth) are not measured.

Procedure

1. Click the **Measure Distance** button, see [Figure 4-6](#).

Figure 4-6. Measure Distance Button



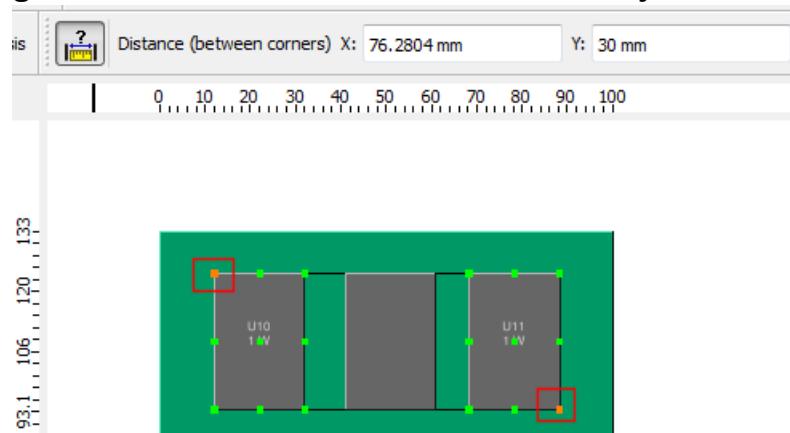
In [Figure 4-6](#) the centers are selected, the other reference points (green) are selectable. The description text indicates that the distance is between centers.

Note

As an alternative you can select **Edit > Measure** or right-click one of the selected objects in the data tree and select **Measure**, however, these operations only change the measurement tool from center selection to point selection mode. The **Measure Distance** button toggles between these two modes.

2. Click on any of the other points, for example, see [Figure 4-7](#).

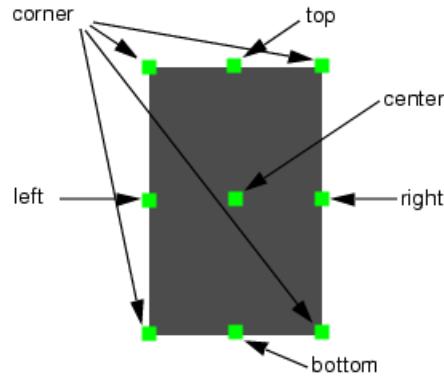
Figure 4-7. Measured Distance Between Object Corners



The distance figures are updated and the description shows that the distance is between corners.

The names of the reference points are shown in [Figure 4-8](#).

Figure 4-8. Object Reference Points



Aligning and Distributing Components

Components can be repositioned to align with other components or have equal distances between them.

Restrictions and Limitations

The selection order is important:

- When aligning objects, the objects align with the *first* of the objects selected.
- When equispacing objects, the *two most extreme components remain in the same locations*, all components between are repositioned so as to equalize the horizontal or vertical gap spacing between them.

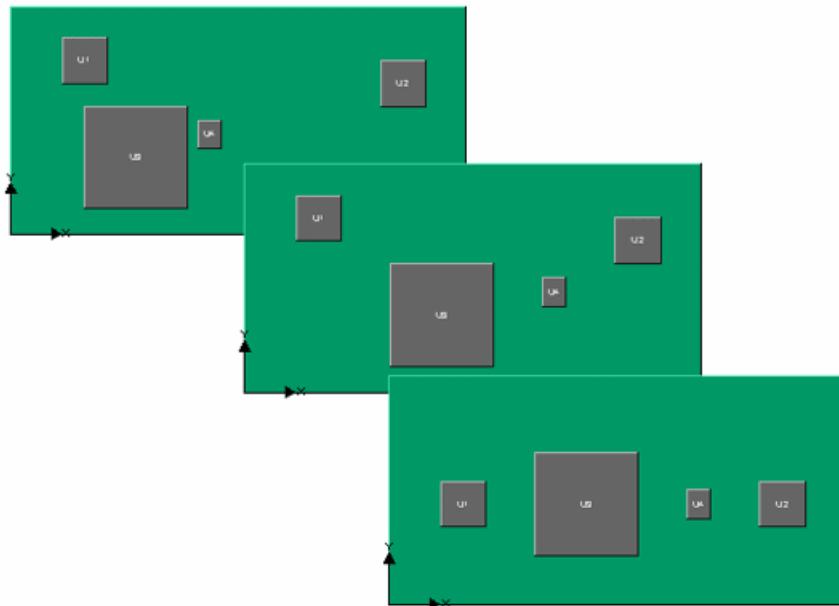
Procedure

1. To align objects:
 - a. Select the objects in either the data tree or the drawing area (Ctrl+click for multiple selections).
 - b. Click one of the alignment icons, or choose **Edit > Align** and then an **Align** option.
2. Three or more components may be equispaced either horizontally or vertically. To equispace objects:
 - a. Select the components.
 - b. Click one of the distribution icons, or choose **Edit > Align** and then an **Equispace** option.

Results

Figure 4-9 shows the use of equispacing followed by an aligning about the horizontal centers.

Figure 4-9. Equispacing and Aligning Components



Using Property Sheets

General usage advice applicable to all property sheets.

Procedure

1. To change geometry properties, select the geometry in the data tree or drawing area and complete the property sheet that appears below the data tree.

2. To change entries in the property sheets, select the data fields with a left-click. Some data entry boxes require text insertion, others a selection from a popup menu.
 - For text entry, the mouse pointer changes to a text insertion marker. Only the first 32 characters are recognized.
 - If the mouse pointer remains unchanged when passing over the data field, a left-click will open an option menu.

Choose an option from the menu to complete the data field. When changing a material property, if the material property you require is not included in the popup option menu, but it is in the Simcenter Flotherm material library, then you add it to the option list.

- You can also add construction information to the components which can be displayed in the property sheet and the display area.

For details on the individual modeling objects, refer to their property sheet descriptions under “[Modeling Objects](#)” on page 103.

Using Component Tables

Component details can be displayed and edited in the Component Tables.

Procedure

1. Click the Component Tables tab to display the details of all the components in the model.
2. To sort a table by one of the columns, do the following:
 - a. Click on the heading of a column to sort the entire table by that value.
 - b. Click again to reverse the sort order.Press Shift+click in the Reference Designator cells to select a range of components based on a sorted parameter, for example, selecting all components on the bottom of the board after having sorted on the board side.
3. To edit a component, select the table cell to be changed then:
 - If you are changing a text field, double-click to obtain a text entry marker,
or
 - Choose a modeling option from the popup menu.

Adding a Material Option From the Library

In property sheets, the dropdown list of materials can be expanded to include materials in the Simcenter Flotherm library.

Procedure

1. Click the [...] button next to the dropdown list of materials to open the Material Library Selector dialog box.
2. Expand the tree in the Material Library pane.
3. Select the material and click the '+' button.
The material name is copied to the Selected Items pane.
4. Optionally, repeat Step 3 to add more materials.
5. Click **Save to Config**, then click **OK** to close the dialog box.
The new material(s) are listed in the dropdown selector.

Related Topics

[Material Library Selector Dialog Box](#)

Removing a Material Option

If the materials selection list needs tidying, you can remove options no longer required.

Procedure

1. Click the [...] button next to the dropdown list of materials to open the Material Library Selector dialog box.
2. Select the material to be removed in the Selected Items pane, then click the '-' button.
The item is then deleted from the Selected Items list.
3. Click **Save to Config** to remove the item from the materials selection list and click **OK** to close the dialog box.

Related Topics

[Material Library Selector Dialog Box](#)

Viewing Material Properties

The properties of a selected material can be displayed in the View Existing Material Data dialog box.

Procedure

1. Click the [...] button next to the dropdown list of materials to open the Material Library Selector dialog box.
2. Select the material in either the Selected Items or Material Library panes, then click **View** to open the View Existing Material Data dialog box.
3. After viewing the material properties, click **OK** to close the View Existing Material Data dialog box.
4. Click **OK** to close the Material Library Selector dialog box.

Related Topics

[Material Library Selector Dialog Box](#)

[View Existing Material Data Dialog Box](#)

Adding Component Notes

Notes can be attached to individual components for reference or identification purposes.

To add notes to the overall project, use the Save Model dialog box.

Procedure

1. Open the Edit Text dialog box.
2. Enter text and click **OK**.

Related Topics

[Save Model Dialog Box](#)

[Edit Text Dialog Box](#)

[Displaying Component Notes](#)

Importing Designs

This section describes how to import board designs that are in other formats.

Importing EDA Board Designs.....	61
Importing ODB++ Board Designs	62
Importing IDF Board Designs	64
Importing Components CSV Layouts	65
Importing a Power List	66
Generating an Import Power List Report.....	66

Importing EDA Board Designs

Import of designs exported from any of the supplied direct EDA interfaces.

Prerequisites

A board design exported as a *floeda* file from one of the following direct EDA interfaces:

- Cadence Allegro PCB Designer, refer to the *Cadence Allegro PCB Designer Interface User Guide*.
- Cadence Allegro Package Designer (APD), refer to the *Cadence Allegro Package Designer Interface User Guide*.
- Mentor Graphics Board Station, refer to the *Board Station Interface User Guide*.
- Mentor Graphics Expedition Enterprise Flow, refer to the *Xpedition Interface User Guide*.
- Zuken CR-5000 Board Designer, refer to the *Zuken CR-5000 Board Designer Interface User Guide*.

Procedure

1. Press Ctrl+E (**File > Import > Import FLOEDA**).

If a model is already loaded, the Update or Replace? dialog box is opened. Click **Replace** or **Update**. If you click **Update** then the Library Selector for Component Import and Component Filter Options dialog boxes are not opened.

A file browser is opened with a *.floeda filter.

2. Select the file to be imported and click **Open**.

The Library Selector for Component Import dialog box is opened.

3. Check the libraries (if any) to be searched for replacement components during import and click **OK**, otherwise click **Cancel**.

The Component Filter Options dialog box is opened.

4. Use the fields in the dialog box to define components to be filtered or deleted during import, then click **Filter**, **Delete**, or **Cancel**.

Results

The design is imported into EDA Bridge. Filtered out components are dimmed in the design tree.

Components cannot be moved, to prevent accidental dragging during selection. To move components, click the **Lock Component Move** icon.

Designs imported from the direct interfaces to Cadence Allegro PCB Designer, Cadence Allegro Package Designer (APD), Mentor Graphics Board Station or Zuken CR-5000 Board Designer that contain monotone images of the copper distribution for the layers will have these images saved as separate *.png files in <solution directory>|PDProject.

Related Topics

[Update or Replace? Dialog Box](#)

[Library Selector for Component Import Dialog Box](#)

[Component Filter Options Dialog Box](#)

Importing ODB++ Board Designs

Board designs available in the data exchange format, ODB++, can be imported

Restrictions and Limitations

- Missing dielectric layers are handled as follows:
 - Some dielectric layers missing:
The summation of the thicknesses of all defined layers is subtracted from board thickness. The remainder is divided by the number of missing dielectric layers.
 - No dielectric layers defined:
The conductor layers thicknesses is subtracted from the board thickness and then divided by (*<number of conductor layers>* – 1). The result is the thickness for each dielectric layer created.

Prerequisites

- An ODB++ fileset, version 7 or 8.

The full definition is a set of files bundled into a tar file (*.tar.gz, *.tgz or *.tar). You can either import the tar file or specify the parent directory of an untarred fileset.

Procedure

1. Make sure that the appropriate ODB++ Component Outline option in the GUI Preferences dialog box has been chosen.
Refer to “[GUI Preferences Dialog Box](#)” on page 35.
2. Choose **File > Import > Import ODB**, then:

If you want to...	Do the following:
Import from a *.tar.gz, *.tgz, or *.tar archive file.	1. Choose Import ODB++ Archive , 2. Select the file and click Open .
Import from a parent folder of subfolders <i>fonts</i> , <i>matrix</i> , <i>misc</i> , and so on.	1. Choose Import ODB++ Directory . 2. Select the parent folder and click Select Folder .

The Library Selector for Component Import dialog box is opened.

3. Check the libraries (if any) to be searched for replacement components during import and click **OK**, otherwise click **Cancel**.

The Component Filter Options dialog box is opened.

4. Use the fields in the dialog box to define components to be filtered or deleted during import, then click **Filter**, **Delete**, or **Cancel**.
5. If errors are detected, then the Possible Issues on Import dialog box will open, showing possible issues with the imported board. Take note of the issues and click **OK** to close the dialog box.

Results

The design is imported as a MotherBoard object. The name of the board is prepended with the filename or folder name of the imported design, where spaces and full-stops in the original name are replaced with underscores.

Filtered out components are grayed out in the design tree.

Components cannot be moved, to prevent accidental dragging during selection. To move components, click the **Lock Component Move** icon.

Conductor and Dielectric layers are imported.

If present, monochrome images for each layer are imported and attached to the relevant layer, allowing Electrical Via modeling objects to be created from the images, and % Coverage values to be estimated. The default % Coverage value of Dielectric layers is 0 (zero) if no images are imported.

Related Topics

[Library Selector for Component Import Dialog Box](#)

[Component Filter Options Dialog Box](#)

[Generating Electrical Vias in Dielectric Layers From Images](#)

Importing IDF Board Designs

Board designs available in the IDF format can be imported into EDA Bridge.

Prerequisites

The full definition of an IDF board design is contained in two files:

- A board definition file (*.emn, *.bdf, or *.brd)
- A component definition file (*.emp, *.ldf, *.lib, or *.pro)

You can import both these files. However, when starting a design, often only the board definition is required, therefore, you have the option of canceling the import of the component definition file.

Procedure

1. Press Ctrl+I.

If a model is already loaded, the Update or Replace? dialog box is opened. Click **Replace** or **Update**. If you click **Update** then the Library Selector for Component Import and Component Filter Options dialog boxes are not opened.

The Select board file browser is opened with a *.emn, *.bdf, *.brd filter.

2. Select the file to be imported and click **Open**.

The Open library file browser is opened with a *.emp, *.ldf, *.lib, *.pro filter.

3. Select the file to be imported and click **Open** to import the library file.

If you clicked **Cancel**, then only the board file is imported. No further dialog boxes are opened.

If you clicked **Open**, then the Library Selector for Component Import dialog box is opened.

4. Check the libraries (if any) to be searched for replacement components during import and click **OK**, otherwise click **Cancel**.

The Component Filter Options dialog box is opened.

5. Use the fields in the dialog box to define components to be filtered or deleted during import, then click **Filter**, **Delete**, or **Cancel**.

Results

The design is imported. Filtered out components are dimmed in the design tree.

Components cannot be moved, to prevent accidental dragging during selection. To move components, click the **Lock Component Move** icon.

A new Placement Keepout Region is created on IDF import if such regions are in the IDF file pair that defines regions on a board where components cannot be placed or can be placed but must not exceed a maximum component height.

If a non-zero Max Height value is entered, then components can be placed in the region but must not exceed the Max Height. The keepout region is annotated on selection with the maximum components height, if non-zero. A 3D view of the board will show the keepout regions at their defined maximum component height.

Placement keepout regions can be manually created by clicking the **Create new Placement Keepout Region** icon.

Related Topics

[Update or Replace? Dialog Box](#)

[Library Selector for Component Import Dialog Box](#)

[Component Filter Options Dialog Box](#)

[Placement Keepout Regions](#)

Importing Components CSV Layouts

The Components CSV file offers an alternative method of integration between EDA tools and EDA Bridge if IDF export is not available in the EDA tool.

Prerequisites

- The CSV file containing one or more column headings in [Components CSV Layout File Format](#).
- Reference designators should comprise at least two characters; single-character reference designators will be appended with the “1” character.
- For Simple component types, a Material can be included in the Components CSV file.

Procedure

1. Choose **File > Import > Import CSV Layout**.

The Update or Add? dialog box is opened.

2. You have the option of either updating the current loaded layout or changing it by adding components based on information in the CSV file.

If you select **Update**, then the components to be updated are those with matching reference designators.

Results

If a Material is specified then it should exist in the library. The library is searched (case insensitively) for the first instance of a material that fully matches the name. If there is more than one material with the same name in the library then the first one found is used.

If the specified Material is not found in the library, or is not specified, then the value of Attribute Material1 specified in the *pcbdata.cfg* file is imported. This file is located in the following folder:

<install_dir>\flosuite_<version>\flotherm\config

If Material is specified for a non-applicable object, then that information is ignored.

Related Topics

[Components CSV Layout File Format](#)

[Exporting Components CSV Layouts](#)

Importing a Power List

Often, power information changes frequently throughout the board design process. Importing a power list in CSV format enables you to update a number of component power values at the same time.

Restrictions and Limitations

The import power list only enables you to modify power values, it does not allow you to add or remove components.

Prerequisites

A CSV file in Power List CSV file format.

Procedure

Choose **File > Import > Import Power List** and select the file.

Related Topics

[Power List CSV File Format](#)

[Exporting a Power List](#)

Generating an Import Power List Report

When importing new power values you can generate an Import Power List Report.

Prerequisites

- Switch on Import Power List Report generation (off by default) by checking the Show Report after CSV Power Import check box in the GUI Preferences dialog box.

Procedure

Import a power list as described in “[Importing a Power List](#)” on page 66.

Results

The report shows discrepancies between what you are importing and what is already in the design, and components that do not have power values assigned, that is, components with power values = 0.

The following are the headings of the report:

- Components existing in thermal model and missing in power list.
These may be missing entries from the power list.
- Components existing in power list and missing in thermal model.
These may be incorrectly named entries in the power list.
- Components in thermal model without power values.

You can print the report, and save the report in HTML format or as a TXT file.

Note

 When saving a report in HTML, in addition to the *.html* file there will also be a folder generated. For example, a report file named *my_report.html* will have a folder named *my_report_files*. Both the file and the folder are required to view the report correctly.

Related Topics

[Power List CSV File Format](#)

Exporting Designs

The primary purpose of EDA Bridge is to design then transfer a board design to Simcenter Flotherm for solving. Board designs can also be exported in FLOPLC and IDF file formats. During the design process, component data including power, or only component power, can be exported in CSV file format.

Transferring an EDA Bridge Model to Simcenter Flotherm	68
Exporting FLOPLC Files	69
Exporting IDF Files	70
Exporting Components CSV Layouts	70
Exporting a Power List	70

Transferring an EDA Bridge Model to Simcenter Flotherm

To solve the model, it must be transferred to EDA Bridge.

Restrictions and Limitations

- Board materials created in EDA Bridge have zero Electrical Resistivity, therefore they are considered as a dielectric material in Joule Heating calculations in Simcenter Flotherm.
- Notes text does not get transferred from EDA Bridge property sheets to **Notes** tabs in Simcenter Flotherm.

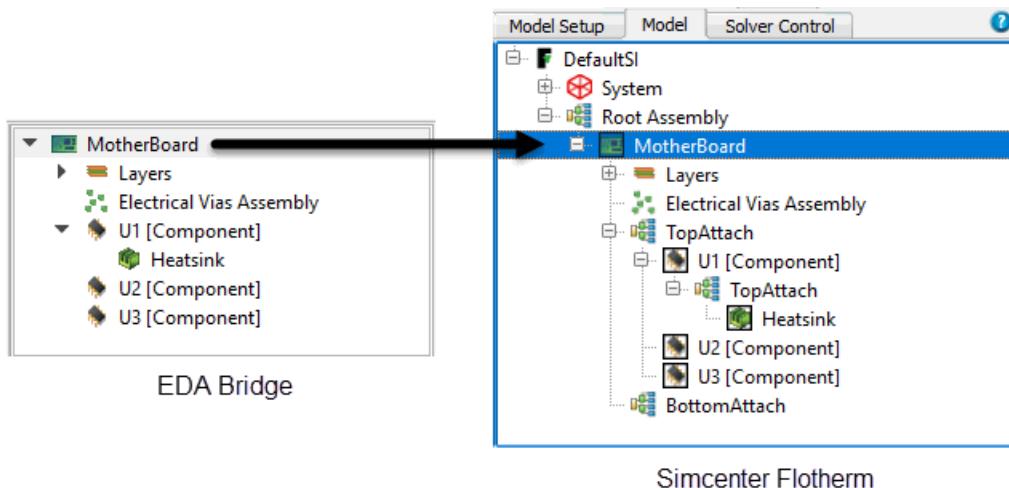
Procedure

You have a choice:

If you want to...	Do the following:
Transfer the model to Simcenter Flotherm and keep EDA Bridge open.	<ul style="list-style-type: none">• Choose File > Transfer. Repeated transfers from EDA Bridge create new MotherBoard assemblies in the Simcenter Flotherm model.
Transfer the model to Simcenter Flotherm and close EDA Bridge.	<ul style="list-style-type: none">• Choose File > Transfer and Quit.

Results

Objects are transferred as EDA SmartParts, see [EDA SmartParts](#) in the *Simcenter Flotherm SmartParts Reference Guide*.



Exporting FLOPLC Files

You can export a **.floplc* file containing the placement, component power dissipation and thermal data that can be read by supplied direct interface tools.

Restrictions and Limitations

Placement update into CR-5000 is not supported.

Procedure

Choose **File > Export > Export FLOPLC** and enter the filename and destination from the browser.

Note

As Mentor Graphics Mechanical Analysis products identify placement files by the *.floplc* extension, when naming the file ensure you keep the default *.floplc* extension.

Results

The exported placement file can now be imported into Allegro or Board Station using the **FloTHERM Interface > Import Placement Changes** menu option in the interfaces supplied with EDA Bridge.

Exporting IDF Files

You can export the board design in the IDF format files (*.emn and *.emp) that can be read by EDA software.

Procedure

Choose **File > Export > Export IDF** to call a file browser to choose the destination of first the *.emn file, then the *.emp file.

Exporting Components CSV Layouts

You can export board design information in comma separated variable files.

Restrictions and Limitations

- Material names will be exported automatically into Components CSV files, where applicable.
- When exporting a 2R component, if the simple variation of the component has non-default material referenced, this will not be preserved.

Procedure

1. Choose **File > Export > Export CSV** to call the [Select Column Types Dialog Box](#) and choose the information to be stored.
2. Use the file browser to choose the filename and destination.

Related Topics

[Components CSV Layout File Format](#)

Exporting a Power List

You can export the list of power values. Typically you will export a power list to modify a number of power values and then import those new values back into the design.

Procedure

Choose **File > Export > Export Power List** and name the CSV file.

Related Topics

[Power List CSV File Format](#)

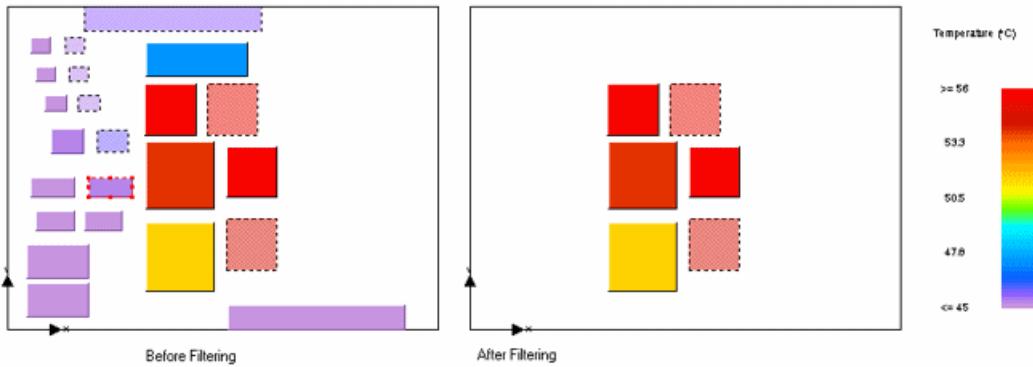
[Importing a Power List](#)

Filtered and Deactivated Objects

Heat dissipated from components is modeled either discretely, restricting heat output to the component location, or smeared over the board.

[Figure 4-10](#) shows the results display before and after filtering out the smaller components from discrete modeling.

Figure 4-10. Filtering Smaller Components



Deactivated objects take no part in the solution.

Filtering Components	71
Displaying Filtered-Out Components	72
Deactivating Objects	73

Filtering Components

Filtering of components is useful when there are many low powered components on the board.

Restrictions and Limitations

- Filtering is a ‘one-off’ operation, that is, any components that are created after a filter operation are not subject to the settings in the Component Filter Options dialog box.
- When the heat values are smeared across a board with cutouts, the cutout areas will still have a proportion of the heat source applied. That is, the heat values are smeared over both the board and cutouts.

Procedure

1. Choose **Tools > Filter Components** to open the Component Filter Options dialog box.
2. Select how the filter parameters are to be combined:
 - Select **Or** to filter components that satisfy only one of the parameters.
 - Select **And** to filter components that satisfy all of the parameters.

For example, filtering components that are small *and* have negligible thermal impact ensures that large unpowered components, for example, junctions, remain part of the thermal solution.

3. Enter the filter parameters in the remaining fields.
4. Click **Filter**.

Results

Filtered components are dimmed in the data tree and their property sheets have the Filtered check box checked.

Power from filtered components is smeared across the board.

Related Topics

[Component Filter Options Dialog Box](#)

[Filtered and Deactivated Objects](#)

[Displaying Filtered-Out Components](#)

Displaying Filtered-Out Components

Filtered-out components can be made visible by changing the default method of coloring components.

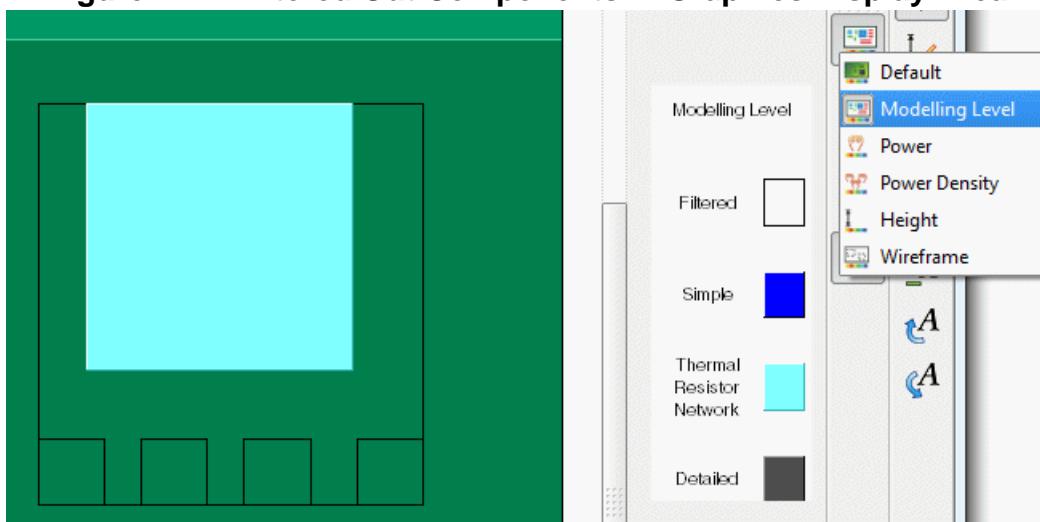
Procedure

1. Click the Color Components icon.
2. Select **Modeling Level**.

Results

Filtered-out components are displayed in wireframe, for example, see [Figure 4-11](#).

Figure 4-11. Filtered Out Components in Graphics Display Area



Related Topics

[Filtering Components](#)

Deactivating Objects

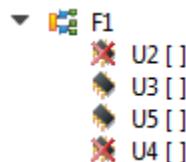
Components and Cylinders can be temporarily removed from the model by deactivating them.

Procedure

Check the Deactivate check box on the object's property sheet.

Results

The deactivated object is removed from view and takes no part in the solution. Deactivated objects are marked with a red cross over the object icon in the data tree.



If a deactivated object belongs to a functional group, then the bounding box of the functional group continues to extend around the object although the object itself is invisible.

Libraries

Simcenter Flotherm libraries contain items that can be copied into EDA Bridge, allowing more detailed component thermal models to be used. In addition, objects created in EDA Bridge can be saved to a library.

Viewing Libraries	74
Adding Library Components	75
Manual Swapping With Library Components	75
Automated Swapping With Library Components	75
Importing and Exporting Libraries	76
Importing Library Items	76
Saving Geometry in the Library	78
Creating New Libraries	78
Searching for a Library Item	79
Downloading a Found Library Item	80
Adding a Found Library Items it to the Project	80

Viewing Libraries

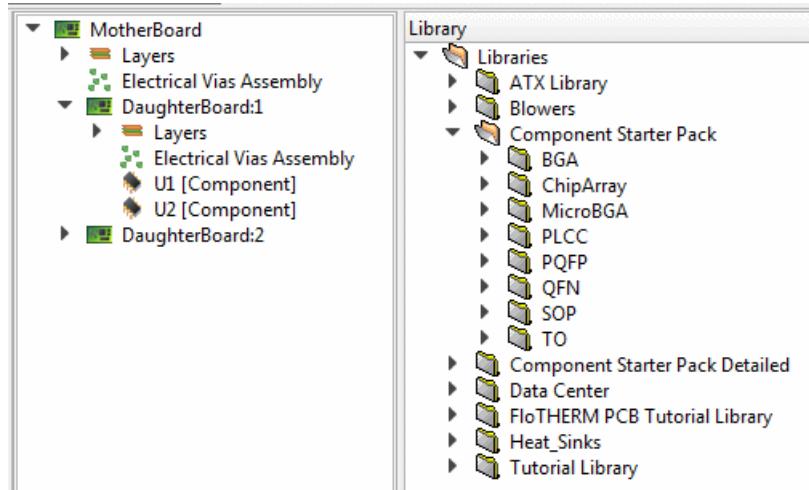
Libraries are listed hierarchically in the Library pane.

Procedure

If the Library pane is not already displayed, click the **Show Library** icon in the New Geometry toolbar.

The layout configuration changes to display the available libraries in a separate pane.

Figure 4-12. Library Pane



Adding Library Components

A simple method for adding components from a library to a motherboard.

Procedure

1. Select the motherboard (or a daughterboard) to which the component is to be added.
2. Double-click a valid library item.

See [Rectangular Components](#) for a list of the component types supported.

Manual Swapping With Library Components

One or more components can be swapped for a component in the library by selection.

Procedure

You have a choice:

If you want to...	Do the following:
Swap with library items by double-clicking.	<ol style="list-style-type: none">1. Select one or more components in the model.2. Open the library and double-click the component in the library.
Swap with library items using the Library context menu.	<ol style="list-style-type: none">1. Select one or more components in the model.2. Open the library and right-click the component in the library.3. Choose Load from the popup context menu.
Using the component's context menu.	<ol style="list-style-type: none">1. Select one or more components in the model.2. Right-click and choose Replace with library item.3. Select the replacement object using the Library Item Selector dialog box and click OK.

Automated Swapping With Library Components

You can replace components with library items, based on the Package Names of the components. This process is built into the importing of IDF or FLOEDA files.

Procedure

1. Choose **Tools > Component Library Swap** to open the Library Selector for Component Import dialog box.

2. Expand the Libraries tree, if required, and select the libraries to be searched by checking the associated check boxes, then click **OK**.

Results

Any component in the selected library(ies) that has the same, or has a partially matched Package Name as that found in the model will be swapped into the model and located such that the component center and orientation are maintained.

For a partial match, the name of the component should match the start of the library file package name. For example, a library component model named “SOT223” replaces components SOT223, SOT223:1, SOT223:1:X and so on that are found in the board design file.

If a match is not found, then a simple component representation is created instead.

Related Topics

[Importing EDA Board Designs](#)

[Importing IDF Board Designs](#)

[Library Selector for Component Import Dialog Box](#)

Importing and Exporting Libraries

An entire library can be exported to the file system for subsequent distribution to other EDA Bridge installations.

Library items are stored in PDML format, allowing full data integration to Simcenter Flotherm and FloTHERM PACK.

Procedure

1. To export a library, right-click the library and choose Export to open a file browser.
2. Name the file, which must have a *.library* extension, browse to a destination folder and click **Save**.
3. The file can be e-mailed, for example, to another Simcenter Flotherm installation.
4. To import a library, right-click an existing library and choose Import.

Importing Library Items

Library items in Mentor Graphics Mechanical Analysis PDML can be added to a library.

Restrictions and Limitations

Apart from objects created within EDA Bridge, only the following items are supported for import into EDA Bridge from external sources:

- Simcenter Flotherm Heatsink SmartPart
- Simcenter Flotherm PCB SmartPart
- Simcenter Flotherm Compact Component SmartPart
- Simcenter Flotherm Network Assembly SmartPart
- Simcenter Flotherm Surface Attributes (for Heatsink Interfacial Resistance)
- FloTHERM PACK 2R, DELPHI, and Detailed Components

To preserve FloTHERM PACK model integrity, these components cannot be modified except for position, orientation, max temperatures, and power. You are prevented from changing the other fields in the GUI. If you build a 2R model directly in EDA Bridge (by switching the Component Type to 2 Resistor from the default of Simple) then you can change those fields, and the CSV layout import will update the 2R values as expected.

- T3Ster *.xCTM files. A *.xCTM file is a file representing a compact thermal resistor/capacitor model of an IC package, generated by Mentor Graphics T3Ster-Master software.

Attempts to import objects not of the above type results in a ‘Failed to import Library Item into selected part(s)’ message.

Procedure

You have a choice:

To import...	Do the following:
Directly via the GUI.	<ul style="list-style-type: none">• Open the Library and right-click the library folder into which you want to import.• Choose Import to open the ‘Select Library file to import’ dialog box.• The dialog box has a filter that defaults to *.pdml files. To import a *.xCTM file, then change the filter.• Navigate to the file folder, select the file, and click Open.

To import...	Do the following:
Indirectly via the file system.	<ol style="list-style-type: none">1. Use File Explorer to copy or move the file to the directory that corresponds to the library as seen from EDA Bridge.2. In EDA Bridge, right-click the library and choose Refresh. <p>This searches the directory and updates the library.</p>

Related Topics

[Importing and Exporting Libraries](#)

Saving Geometry in the Library

EDA Bridge board geometry can be copied to the Simcenter Flotherm library,

Procedure

1. Right-click the geometry and choose **Save to Library** to open the Library Selector dialog box.
2. Select the destination library folder and click **OK**.
3. Open the Library pane to check that the geometry has been added correctly.

Results

After exporting the design to the library, it can be quickly replicated in other EDA Bridge or Simcenter Flotherm projects.

The board geometry appears as an assembly, not a PCB SmartPart, in the Simcenter Flotherm library.

Related Topics

[Creating New Libraries](#)

[Library Selector Dialog Box](#)

Creating New Libraries

In addition to the delivered libraries, you can create new libraries.

Restrictions and Limitations

You cannot create a new library if the selected library is read-only. Read-Only library folders can only be set to writable within Simcenter Flotherm.

Procedure

1. Right-click a library folder and choose **Create new Library** to open the Create New Libraries dialog box.
2. Use the Create New Libraries dialog to add a folder to the Simcenter Flotherm library. Geometry may then be added to or copied from this directory using the library manager.
3. Enter the name to be displayed in the library tree in the Library Name field.
4. Enter the full path of the directory to be used as a library folder in the Directory Name field. Alternatively, click Browse to find the directory.

Results

If the library is created as a child of an existing library then a new directory will be created on the file system as a subfolder.

If a new library is created as a child of the ‘Libraries’ root node then its location can be manually changed to anywhere on the local file system.

Related Topics

[Create New Libraries Dialog Box](#)

Searching for a Library Item

Search options include partial-matching and case-sensitivity.

Procedure

1. Click the **Show Library** icon to open the Library pane, then right-click a library folder and choose **Find** to open the Library Search Facility dialog box.
2. Enter all or part of the item name you want to find in the Find field.
3. Click **Partial**, if searching for a substring match.
4. Click **Match Case** if searching for a capitalization match.
5. Click **Find Next** to search for a single instance or **Find All** for all instances, in the selected library. If there are multiple matches you can use **Find Next** repeatedly to step from one match to the next.

Related Topics

[Library Search Facility Dialog Box](#)

[Downloading a Found Library Item](#)

[Adding a Found Library Items it to the Project](#)

Downloading a Found Library Item

A quick way of downloading a library item to a selected project part, without the need to expand the library.

Prerequisites

- In the project tree, select a valid part to receive the library item

Procedure

1. Find items by [Searching for a Library Item](#)
2. Double-click the item in the Found List.

Related Topics

[Library Search Facility Dialog Box](#)

[Searching for a Library Item](#)

[Adding a Found Library Items it to the Project](#)

Adding a Found Library Items it to the Project

When a library item has been found by a library search, the item can be selected in the library hierarchy tree then added to the project.

Prerequisites

- Items have been found by [Searching for a Library Item](#).

Procedure

1. Select the item in the Found Items list of the Library Search Facility dialog box.
2. Click **Select in Tree** then close the dialog box.
If necessary, the Library tree expands to display the selected item.
3. Close the Library Search Facility dialog box
4. To include the selected item to the project, right-click the item and choose **Load**.
Alternatively, double-click the item.

Related Topics

[Library Search Facility Dialog Box](#)

[Searching for a Library Item](#)

Board Layout File Formats

Power lists and components defined by CSV files.

Power List CSV File Format..... **82**

Components CSV Layout File Format **83**

Power List CSV File Format

A comma-separated text file.

Use this file to add power values to components by file import as an alternative to manually adding data directly into property sheets.

Format

A Power List CSV file must conform to the following formatting and syntax rules:

- The list can be any length.
- The file format is two columns of data separated by a comma.

```
<reference_designator>,<power_in_Watts>
<reference_designator>,<power_in_Watts>
...
...
```

Such a file can be created from an existing design by exporting a power list.

Parameters

- reference_designator

The reference designator of the component, preferably of at least two characters; single-character reference designators will be appended with the “1” character when imported.

- power_in_Watts

A value in floating point format of the component’s power in Watts.

Examples

The example below shows three components, U1, U2, and U3, powered by 2, 4, and 4 Watts respectively.

```
U1,2
U2,4
U3,4
```

Related Topics

[Reference Designators](#)

[Importing a Power List](#)

[Exporting a Power List](#)

Components CSV Layout File Format

A comma-separated text file generated by choosing **File > Export > Export CSV**.

Use this file to view or use component data in spreadsheet format, or to import component data to a model.

Format

A header row followed by a row for each component in the project.

Parameters

The row content is defined by the columns selected in the Select Column Types dialog box.

Examples

The following is an example file viewed in a text editor:

```
Package Name,Reference Designator,Part Number,Total Power (W),Component Type,X Location  
(mm),Y Location (mm),X Size (mm),Y Size (mm),Z Size (mm),Board Side,Z  
Rotation,Filtered,Deactivated,Tj_max (degC),Tc_max (degC),Auto Peak Temp (degC),Junction To  
Case Resistance (K/W),Junction To Board Resistance (K/W),Average Temperature,Material  
QFN_32pin,U14,6200-  
0083,0,Simple,21.3233,27.3558,6.2484,6.2484,0.899998,Top,None,False,True,,,Auto,,,Lumped  
component  
SOIC_8pin,U4,6200-0004,0,2 Resistor,9.271,13.0175,7.493,5.715,1.3,Bottom,270  
Degrees,False,False,90,75,Auto,15,10,,  
SOP_8pin,U6,6200-0005,0,Simple,41.9481,9.01954,5.0038,3.55092,1.3,Bottom,270  
Degrees,False,False,,,Auto,,,Lumped component  
SOT23-5,U7,6200-0084,0,Simple,27.0891,19.812,3.6068,2.921,1.15001,Bottom,90  
Degrees,True,False,,,Auto,,,Filtered,Lumped component  
DCU_8pin,U15,6200-0001,0,Simple,49.8348,9.4107,4.2164,2.7686,0.750011,Bottom,270  
Degrees,False,False,,,Auto,,,Lumped component  
FC_CBG_A_Detailed,U5,ThermAtt,0,Detailed,21.335,4.825,47.5,47.5,2.85,Top,180  
Degrees,False,False,95,60,Auto,,,
```

This data is displayed in Microsoft Excel as shown in [Figure 4-13](#).

Figure 4-13. Components CSV Layout File Displayed in a Spreadsheet

	A	B	C	D	E	F	G	H	I	J	K
1	Package Name	Reference Designator	Part Number	Total Power (W)	Component Type	X Location (mm)	Y Location (mm)	X Size (mm)	Y Size (mm)	Z Size (mm)	Board Side
2	QFN_32pin	U14	6200-0083		0 Simple	21.3233	27.3558	6.2484	6.2484	0.9	Top
3	SOIC_8pin	U4	6200-0004		0 2 Resistor	9.271	13.0175	7.493	5.715	1.3	Bottom
4	SOP_8pin	U6	6200-0005		0 Simple	41.9481	9.01954	5.0038	3.55092	1.3	Bottom
5	SOT23-5	U7	6200-0084		0 Simple	27.0891	19.812	3.6068	2.921	1.15001	Bottom
6	DCU_8pin	U15	6200-0001		0 Simple	49.8348	9.4107	4.2164	2.7686	0.75001	Bottom
7	FC_CBGa_Detailed	U5	ThermAtt		0 Detailed	21.335	4.825	47.5	47.5	2.85	Top

L	M	N	O	P	Q	R	S	T	U
Z Rotation	Filtered	Deactivated	Tj_max (degC)	Tc_max (degC)	Auto Peak Temp (degC)	Junction To Case Resistance (K/W)	Junction To Board Resistance (K/W)	Average Temperature	Material
None	FALSE	TRUE			Auto				Lumped component
270 Degrees	FALSE	FALSE	90	75	Auto	15	10		Lumped component
270 Degrees	FALSE	FALSE			Auto				Lumped component
90 Degrees	TRUE	FALSE			Auto			Filtered	Lumped component
270 Degrees	FALSE	FALSE			Auto				Lumped component
180 Degrees	FALSE	FALSE	95	60	Auto				

When importing Components CSV data to a model, Simcenter Flotherm will make the following substitutions to data in the Filtered and Deactivated columns:

Table 4-2. Conversion of True and False Values on Components CSV Data Import

CSV File Values	Imported as:
True	True
TRUE	
true	
1	
False	False
FALSE	
false	
0	

Related Topics

[Importing Components CSV Layouts](#)

[Exporting Components CSV Layouts](#)

[Select Column Types Dialog Box](#)

Board Layout Dialog Boxes

Dialog boxes available when laying out a board design.

Library Item Selector Dialog Box	86
Pattern Selected Items Dialog Box.....	87
Move Selected Items Dialog Box.....	88
Component/Functional Group Search Dialog Box.....	89
Material Library Selector Dialog Box	90
View Existing Material Data Dialog Box.....	91
Edit Text Dialog Box	92
Library Selector for Component Import Dialog Box.....	93
Select Column Types Dialog Box	94
Component Filter Options Dialog Box.....	96
Library Selector Dialog Box	98
Create New Libraries Dialog Box.....	99
Library Search Facility Dialog Box	100
Update or Replace? Dialog Box	101

Library Item Selector Dialog Box

To access: Right-click geometry and choose **Replace with library item**.

Use this dialog box to browse and select library items suitable for addition to the EDA Bridge model.

Objects

- Libraries
 - A tree structure displaying those libraries with geometry suitable for addition to the current selection in EDA Bridge.

Usage Notes

To add an item, select it in the tree and click **OK**. The item will then appear in the EDA Bridge tree as a child of the current selection.

Related Topics

[Replacing Geometry With Library Items](#)

[Adding Existing Geometry Using the Library Pane](#)

Pattern Selected Items Dialog Box

To access: **Edit > Pattern**

Use this dialog box to create a two dimensional matrix array pattern of a single geometry object.

Objects

Field	Description
Number in X	The number of copies in the X direction.
Pitch in X	The pitch (spacing) between copies in the X direction.
Number in Y	The number of copies in the Y direction.
Pitch in Y	The pitch (spacing) between copies in the Y direction.

Usage Notes

The pattern is directed along any two coordinate axis in either the positive or negative directions.

Related Topics

[Making a Pattern of a Component](#)

Move Selected Items Dialog Box

To access: Select an object, then **Edit > Move By**

Use this dialog box to specify precise movement of the selected object(s).

Objects

Field	Description
Distance in X	The distance the selected object is to be moved in the X direction.
Distance in Y	The distance the selected object is to be moved in the Y direction.

Usage Notes

- Child objects are moved with their parent, and can only be moved with their parent.
- For child assemblies and components, the move is in the directions of the parent's axes.
- Objects that belong to the same functional group can be moved independently but if the functional group is selected, then all objects belonging to that group are moved.
- Multiple-selected objects are moved together.
- The MotherBoard and layers cannot be moved.
- When changing units, distances are converted to the new units.

Related Topics

[Moving Objects](#)

Component/Functional Group Search Dialog Box

To access: **Edit > Find** or **Ctrl+F**

Use this dialog box to find and select either components or functional groups of components.
When found, the objects become highlighted in the display.

Objects

Field	Description
Find	Search string.
Reference designator	Find specific objects.
Package Name	Find packages.
Match Case	Find objects with same name and capitalization as the search string.
Partial match	Find objects with names containing the search string.

Related Topics

[Searching for Components and Functional Groups](#)

Material Library Selector Dialog Box

To access: From a property sheet, click the [...] button next to the material name (hover text: Import a material from a library).

Use this dialog box to add or delete materials from the materials selection list.

Description

The left-side lists the materials that are directly selectable from the property sheet. The right-side shows the library of available materials.

Use this dialog box to select both materials and surface finishes. When selecting materials, you can view the details of an existing material. However, surface finishes (for example, Heatsink Interface Resistance) can only be selected from existing definitions and cannot be viewed.

Objects

Field	Description
+	Click to add the selected Material Library item to the Selected Items pane.
-	Click to remove the selected item from the Selected Items pane.
View	Click to open the View Existing Material Data dialog box.
Save to Config	Click to save any changes to the material selection dropdown list in the property sheet.

Related Topics

[Adding a Material Option From the Library](#)

[View Existing Material Data Dialog Box](#)

View Existing Material Data Dialog Box

To access: From the Material Library Selector dialog box, select a material type (either from the Selected items list or from the library), then click **View**.

Use this dialog box to view the properties of a selected material.

Objects

- Read-only values for properties of the material.

Orthotropic materials have thermal conductivity defined along each axis: x, y and z.

Related Topics

[Viewing Material Properties](#)

[Material Library Selector Dialog Box](#)

Edit Text Dialog Box

To access: From an object's property sheet, click the [...] button next to Notes (Edit notes associated with this object).

Use this dialog box to create component construction notes that can be displayed in the property sheets and the display area.

Objects

- A text editor.
 - Simple editing functionality with no wrap around.

Related Topics

[Displaying Component Notes](#)

Library Selector for Component Import Dialog Box

To access: Opens during a IDF or FLOEDA file import after file selection, or

Tools > Component Library Swap

Use this dialog box to select libraries to be searched for cross-reference links when importing components, allowing more detailed component thermal models to be imported in conjunction with the library data.

Objects

- Libraries
 - A tree of libraries with check boxes which allow individual library selection.

Related Topics

[Automated Swapping With Library Components](#)

[Importing EDA Board Designs](#)

[Importing IDF Board Designs](#)

Select Column Types Dialog Box

To access: **File > Export > Export CSV**

Use this dialog box to select which project information is to be included in an exported CSV file.

Objects

Field	Description
Reference Designator	Values as in the component property sheets.
Part Number	
Power	
Package Name	
Component Type	
Location	
Size	
Board Side	
Z Rotation	None, 90 Degrees, 180 Degrees or 270 Degrees. Any values other than these are imported as None.
Filtered	TRUE or FALSE.
Include Deactivated Components	By default, deactivated components are not exported.
Deactivated	TRUE or FALSE.
Temperature Limits	Only applicable to 2 Resistor, Delphi, or Detailed models. Maximum Junction and Maximum Case Temperatures, Tj_max and Tc_max.
2R Resistances	Junction-to-Case and Junction-to-Board resistances for 2 Resistor components.
Average Temperature	As in the results analysis.
Junction/Case Temperatures	As in the results analysis.
Board Temperature	As in the results analysis.
Reflow Data	Applicable to solder reflow predictions. Includes largest maximum, peak body, over peak, largest minimum, solder melt and under melt temperatures and over melt times as in the results analysis.
Material	As in the property sheets.

Usage Notes

This dialog box lists all the possible column types. Unchecked boxes exclude those columns from the CSV file.

Related Topics

[Components CSV Layout File Format](#)

Component Filter Options Dialog Box

To access: The dialog box opens during a IDF or FLOEDA file import after file selection, or
Tools > Filter Components

Use this dialog box to determine the heat dissipation model for the components or just to simplify the model.

Description

This dialog box consists of a property sheet for defining the component filtering or deletion controls.

Objects

Field	Description
Combine Parameters By	The chosen options are combined using the logical AND/OR selected.
Side Length Less Than	Components with property values lower than those entered here, and/or with the corresponding reference designators, can be filtered or deleted. If the value is equal to zero, then that filtering/deletion option is effectively Off. For example, Length = 0 mm will filter or delete nothing.
Height Less Than	Components with property values lower than those entered here, and/or with the corresponding reference designators, can be filtered or deleted. If the value is equal to zero, then that filtering/deletion option is effectively Off. For example, Length = 0 mm will filter or delete nothing.
Power Less Than	
Power Density Less Than	
Ref. Des. Contains	Enter the character, or part of the character string, of the reference designator required, separating multiple designators with a comma. For example: <ul style="list-style-type: none">• R – for all resistors to be filtered or deleted• R, C – for all resistor and capacitor type components to be filtered or deleted, regardless of whether AND has been selected to combine the other filter parameters If not filtering by reference designator, the field should be left empty or set to a unique string which does not occur in any of the design reference designators.
Filter	Click to filter components with lower property values or matching reference designators. Filtered components are not seen as discrete entities by the solver; their heat values are not lost, but are smeared over the board. When the component property values are above or equal to the values entered, then the component is modeled as a solid object, otherwise the heat dissipation of the component is smeared over the entire board surface (see Usage Notes). It is therefore more efficient, in solution time, to filter out small components.

Field	Description
Delete	Click to delete components with lower property values or matching reference designators from the model.

Related Topics

[Filtered and Deactivated Objects](#)

[Filtering Components](#)

[Importing EDA Board Designs](#)

[Importing IDF Board Designs](#)

Library Selector Dialog Box

To access: Right-click geometry then choose **Save to Library**

Use this dialog box to select a library folder to hold the exported geometry to the Simcenter Flotherm library. This geometry can then be accessed from Simcenter Flotherm and used for system analysis.

Objects

- Libraries
 - A tree of libraries.

Usage Notes

To copy the design into the Simcenter Flotherm library, select the library folder in the tree and click **OK**.

Read-Only library folders can only be set to writable within Simcenter Flotherm.

Related Topics

[Saving Geometry in the Library](#)

[Adding Existing Geometry Using the Library Pane](#)

Create New Libraries Dialog Box

To access: Click the **Show Library** icon to open the Library pane, then right-click a library folder and choose **Create new library**.

Use this dialog box to add a folder to the Simcenter Flotherm library. Geometry may then be added to or copied from this directory using the library manager.

Objects

Field	Description
Library Name	The name to be displayed in the library tree
Directory Name	The pathname of the directory to be used as a library folder.

Related Topics

[Creating New Libraries](#)

Library Search Facility Dialog Box

To access: Click the **Show Library** icon to open the Library pane, then right-click a library folder and choose **Find**.

Use this dialog box to quickly find and select items in a library according to an entered search criteria.

Objects

Object	Description
Find	Search string.
Found Items	Lists all found items.
Partial match	Check to make the search a partial match of the search string.
Match Case	Check to make the search case-sensitive.
Find Next	Click to view the next found item.
Find All	Click to view a list of all found items.
Select In Tree	Click to highlight the selected found item in the Library pane tree.

Related Topics

[Searching for a Library Item](#)

[Downloading a Found Library Item](#)

[Adding a Found Library Items it to the Project](#)

Update or Replace? Dialog Box

To access: Opens during an IDF or FLOEDA file import if a model already exists,
Use this dialog box to choose to update or replace an existing EDA Bridge model.

Description

Often the component placement definition changes in the EDA layout tool throughout the design process. This dialog box is opened when there are one or more components in the currently loaded project.

Objects

- Update

All existing definitions of component type, heatsinks, and so on are retained.

Only the component location, board side and board shape are updated, based on the Reference Designator matching.

Note

 Updating a component placement in this way is designed to work when the imported model has the same number of components with the same Reference Designator assignments as the original model.

- Replace

Replaces the existing model with the imported model.

Related Topics

[Importing EDA Board Designs](#)

[Importing IDF Board Designs](#)

Chapter 5

Modeling Objects

A set of modeling objects is provided from which you can add instances of objects to your board.

Working With Modeling Objects	104
MotherBoards	104
DaughterBoards	104
Layers Properties and Procedures	107
Layers	107
Saving and Retrieving a Layer Stackup to and From a Library	108
Layer Patches Properties and Procedures	109
Layer Patches	109
Defining a Layer Patch	109
Generating Layer Patches From Images or EDA Data	110
Electrical Vias Properties and Procedures	112
Electrical Vias	112
Defining an Area of Electrical Vias	112
Generating Electrical Vias in an Electrical Vias Assembly From an Image	113
Generating Electrical Vias in Dielectric Layers From Images	114
Rectangular Components	116
Cylindrical Components	117
Functional Groups Properties and Procedures	118
Functional Groups	118
Creating Functional Groups	119
Rectangular Cutouts	120
Heatsinks	121
Thermal Vias	124
Cans	125
Potting Compound	126
Placement Keepout Regions	126
Modeling Objects Property Sheets and Dialog Boxes	128
MotherBoard Property Sheet	129
DaughterBoard Property Sheet	130
Group Layer Property Sheet	132
Layer Property Sheet	133
Layer Patch Property Sheet	135

Electrical Vias Assembly Property Sheet	136
Electrical Via Property Sheet	137
Component Property Sheet	139
Cylindrical Component Property Sheet	141
Functional Group Property Sheet	143
Rectangular Cutout Property Sheet	144
Heatsink Property Sheet	145
Thermal Via Property Sheet	147
Can Property Sheet	148
Potting Compound Property Sheet	149
Placement Keepout Regions Property Sheet	150
Layer Trace Processing Dialog Box	151
Dielectric Layer Vias Processing Dialog Box	153
Board Vias Processing Dialog Box	155

Working With Modeling Objects

Selecting an item in the node tree or the graphical 2D view results in valid geometry icons in the New Geometry toolbar becoming active.

When you click an active icon, the new object is added to the data tree, under the currently selected object.

By default, new objects become selected on creation, with their property sheet open. If you do not want objects selected on creation, then deactivate Select Objects when Created in the GUI Preferences dialog box.

MotherBoards

All EDA Bridge projects have a MotherBoard, which is the root object onto which you add objects.

By default, when a new project is opened, a MotherBoard with a group of four layers is created.

The following geometry can be added to a MotherBoard: [Layers](#), [DaughterBoards](#), [Rectangular Components](#), [Functional Groups](#), [Rectangular Cutouts](#), [Cans](#) and [Placement Keepout Regions](#).

Related Topics

[MotherBoard Property Sheet](#)

DaughterBoards

DaughterBoards represent mezzanine boards, mounted in parallel with, or perpendicular to, the MotherBoard.

A default DaughterBoard has four layers.

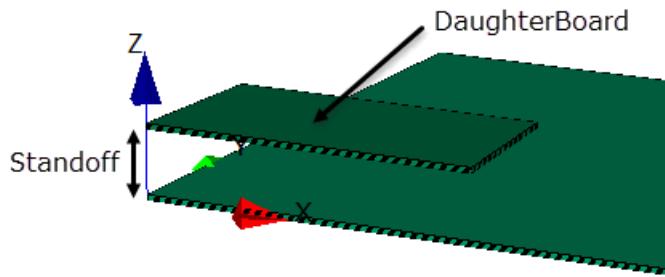
A DaughterBoard can be parallel or perpendicular to the MotherBoard, depending on the Mount Type setting in the DaughterBoard property sheet.

The following geometry can be added to a DaughterBoard: [Layers](#), [Rectangular Components](#), [Cylindrical Components](#), [Functional Groups](#), [Rectangular Cutouts](#), [Cans](#), and [Placement Keepout Regions](#).

Parallel DaughterBoards

By default, a new parallel DaughterBoard is created at the origin of the MotherBoard with a standoff, see [Figure 5-1](#). The area of the board is one-quarter that of the MotherBoard, and has the same thickness and FR4 material type.

Figure 5-1. Parallel DaughterBoard



Perpendicular DaughterBoards

There are four variants of perpendicular DaughterBoard: X-High, X-Low, Y-High, and Y-Low. The X and Y variants are located along the x- or y-axis of the MotherBoard. The High and Low variants have their top surfaces on the high or low side of the board.

Figure 5-2. Perpendicular X-High DaughterBoard

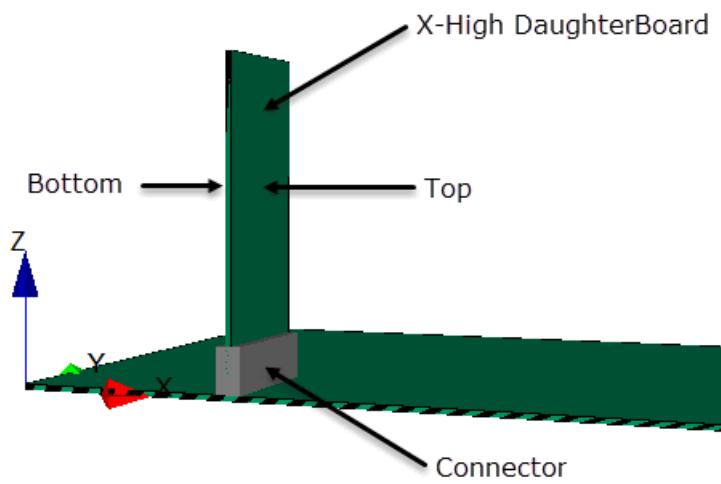
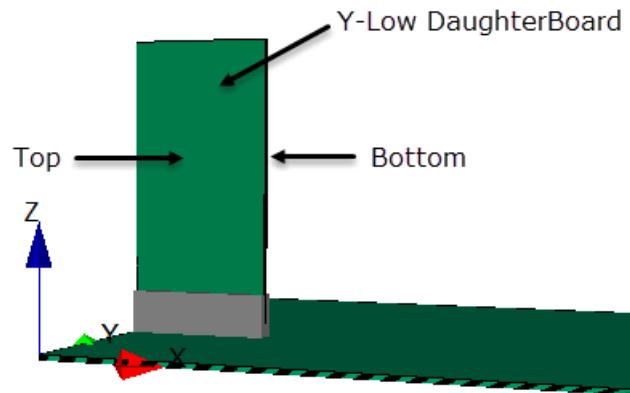


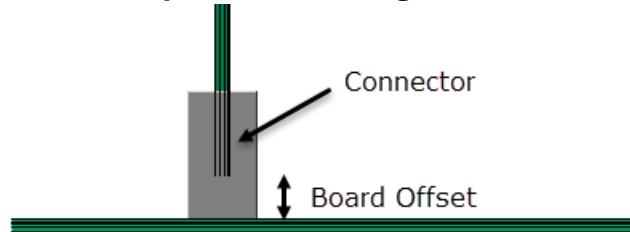
Figure 5-3. Perpendicular Y-Low DaughterBoard



A perpendicular DaughterBoard is mounted on the MotherBoard by a connector. The connector block length is the same as the DaughterBoard width.

The board offset is the distance from the edge of the board to the surface of the MotherBoard.

Figure 5-4. Perpendicular DaughterBoard Offset



Related Topics

[DaughterBoard Property Sheet](#)

Layers Properties and Procedures

Use to define layer properties of a board.

Layers	107
Saving and Retrieving a Layer Stackup to and From a Library	108

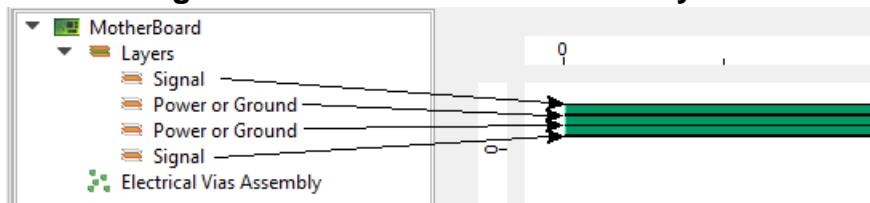
Layers

A group of four layers are provided by default with the Motherboard or a new Daughterboard. More layers can be added.

On the Motherboard, the default layers are named Signal, Power or Ground, Power or Ground, and Signal. On the DaughterBoard, the default layers inherit the name of the DaughterBoard.

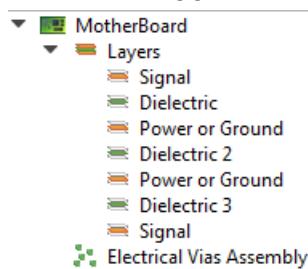
The first and last layers are aligned with the top and bottom of the board respectively, and the two intermediate layers are spaced evenly across the thickness of the board, as shown in [Figure 5-5](#).

Figure 5-5. Default MotherBoard Layers



A new layer is added below the selected layer. By default, a new layer is a Metallic layer, but this can be changed, using the Layer property sheet, to a Dielectric layer. Dielectric layers are represented by green icons as shown in [Figure 5-6](#).

Figure 5-6. Icon Representation of Metallic and Dielectric Layers in the Data Tree



To control the spacing of the layers, uncheck the Equispaced check box off in the Layer Group property sheet and enter offset values for each layer.

[Layer Patches](#) can be added to metallic layers.

[Electrical Vias](#) can be added to dielectric layers.

Modeling Layers

- Typical signal layer parameters are 20% coverage, 0.034 mm (1 oz) thickness.
- Typical power or ground layer parameters are 90% coverage, 0.017 mm (0.5 oz) thickness.

Related Topics

[Saving and Retrieving a Layer Stackup to and From a Library](#)

[Layer Property Sheet](#)

[Group Layer Property Sheet](#)

Saving and Retrieving a Layer Stackup to and From a Library

The layer definition for a board can be saved to, and retrieved from, a library.

Procedure

1. Right-click the layer stackup node in the data tree.

2. Choose **Save To Library**.

The Library Selector dialog box is opened.

3. Select a library and click **OK**.

Provided the library has write-access, the layer stackup is written to the library.

4. To retrieve the layer stackup, select an existing layer stackup node in the data tree, right-click and choose **Replace with library item**.

The Library Selector dialog box is opened.

5. Open the relevant library, select the layer stackup and click **OK**.

The existing layer stackup will be over-written by the library layer stackup definition.

Related Topics

[Layers](#)

Layer Patches Properties and Procedures

Use Layer Patches to define regions within a layer that have a different coverage of copper compared with the rest of the layer.

Layer Patches	109
Defining a Layer Patch	109
Generating Layer Patches From Images or EDA Data	110

Layer Patches

An example of Layer Patch usage is close to a high pin count package where there is a high density of signal tracks.

The default coverage of 100% is appropriate to the representation of copper pads, often placed underneath a component to enhance heat flow into the board.

For thermal prediction under natural convection or low air speed environments, it is important to better represent the copper variation close to thermally important packages.

Layer Patches can be individually specified, but if you have a monochrome image of the PCB, a series of patches can be automatically generated to represent the coverage.

Related Topics

- [Defining a Layer Patch](#)
- [Generating Layer Patches From Images or EDA Data](#)
- [Layer Patch Property Sheet](#)
- [Layers](#)

Defining a Layer Patch

Use the Layer Patch property sheet to define a layer patch on a metallic layer.

Procedure

1. Right-click a Metallic layer node in the data tree and choose **New > Layer Patch**.
A layer patch is created and the property sheet is opened.
New layer patches are added to the data tree below the selected layer and located at the origin of the layer.
2. Use the property sheet to define the layer patch. If the board is unlocked then you can use the mouse in the drawing board to move and resize the layer patch.

Results

To view patches on a single layer, select the layer node in the data tree. Layer patches belonging to other unselected layers, as well as the MotherBoard, are hidden from view; all other objects remain in view. This is useful for viewing patches on internal board layers.

Related Topics

[Layer Patch Property Sheet](#)

[Layer Patches](#)

Generating Layer Patches From Images or EDA Data

Monochrome images of PCBs, detailing copper and insulators in black and white respectively, can be used to quickly set-up representative layer patches. Using one of the supplied direct EDA interfaces will enable import of all board data, including image representation for each layer.

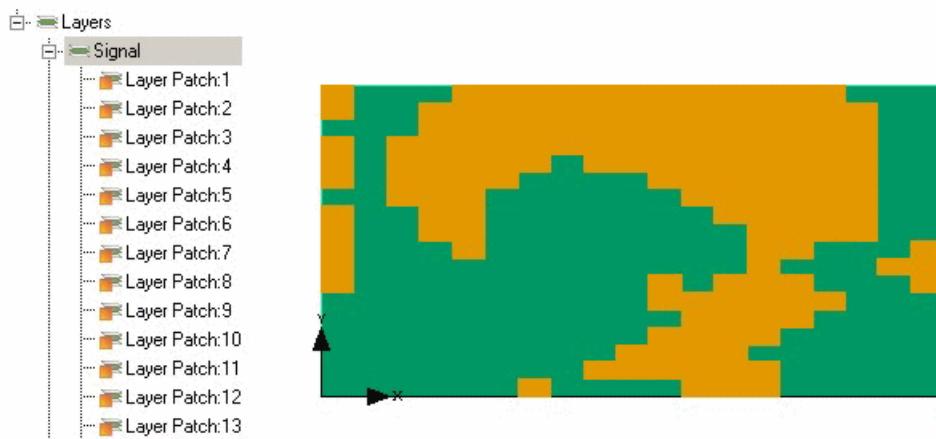
Procedure

1. Right-click a Metallic layer in the tree and choose **Process Layer** from the popup menu.
If File Name in the Layer property sheet has been set to the name of an image file, the Layer Trace Processing dialog box is opened, otherwise a file browser is opened to select the image before the dialog box is opened.
2. Use the dialog box to adjust the representation.
Image colors can be optionally inverted when the range is processed into layer patches.
3. When finished, click **Create Patches**.

Results

[Figure 5-7](#) shows an example of the complexity of patches that can be created.

Figure 5-7. Processed Layer Patches in Data Tree and on a Board



The Layer Type field in the Layer property sheet is not editable when layer patches are defined in the layer.

Related Topics

[Layer Patches](#)

[Layer Trace Processing Dialog Box](#)

Electrical Vias Properties and Procedures

Use Electrical Vias to model the thermal effects of heat transfer down through the board when through-hole vias are present.

Electrical Vias	112
Defining an Area of Electrical Vias	112
Generating Electrical Vias in an Electrical Vias Assembly From an Image	113
Generating Electrical Vias in Dielectric Layers From Images	114

Electrical Vias

Areas containing electrical vias can be individually specified in terms of number and average diameter, and so on, or according to percentage coverage in that area.

After initially creating the electrical vias, they are all visible when either the Electrical Vias Assembly node, or individual electrical vias are selected in the tree.

The location of individual electrical vias over the board can be seen by selecting Electrical Vias in the tree.

If you have monochrome images of the PCB layers, then vias can be automatically generated on individual dielectric layers, thereby modeling blind and buried vias. Vias created in this way are children of dielectric layers in the tree.

Related Topics

- [Electrical Via Property Sheet](#)
- [Electrical Vias Assembly Property Sheet](#)
- [Defining an Area of Electrical Vias](#)
- [Generating Electrical Vias in an Electrical Vias Assembly From an Image](#)
- [Generating Electrical Vias in Dielectric Layers From Images](#)

Defining an Area of Electrical Vias

Use the Electrical Via property sheet to define vias manually or automatically.

Procedure

1. You have a choice:

If you want to...	Do the following:
Define an area of vias through all layers.	Right-click the Electrical Vias Assembly node in the data tree and choose New > Electrical Vias .
Define an area of vias through a single layer.	Right-click a Dielectric layer node in the data tree and choose New > Electrical Vias .

An Electrical Via is created and the property sheet is opened.

2. Use the property sheet to define the vias. If the board is unlocked then you can use the mouse in the drawing board to move and resize the vias area.
3. You have a choice:

If you want to...	Do the following:
Manually define vias.	<ol style="list-style-type: none"> 1. Select a Definition Type of Manual. 2. Define the number, average diameter, average plating thickness, and plating and fill materials.
Allow automatic representation of vias.	<ol style="list-style-type: none"> 1. Select a Definition Type of Automatic. 2. Specify the percentage coverage.

Related Topics

[Electrical Via Property Sheet](#)

[Electrical Vias](#)

Generating Electrical Vias in an Electrical Vias Assembly From an Image

An image file can be used to create a complex distribution of electrical vias.

Prerequisites

- The model has been created from an imported *.floeda* file.

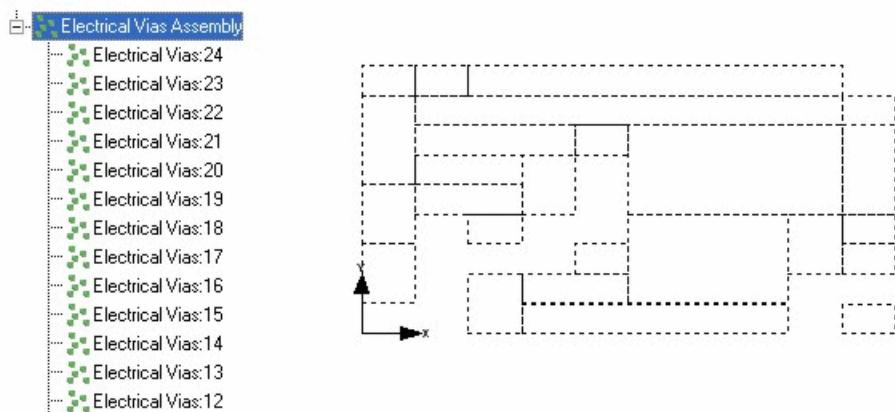
Images representing the distribution of metal on layers and the distribution of electrical vias will have been automatically attached to the appropriate layers and the electrical via holding node.

Procedure

1. Right-click the Electrical Vias Assembly node and choose **Process Electrical Vias**.
If File Name in the Electrical Vias Assembly property sheet has been set to the name of an image file, the Board Vias Processing dialog box is opened, otherwise a file browser is opened to select the image before the dialog box is opened.
2. Use the dialog box to adjust the representation.
Image colors can be optionally inverted.
3. When finished, click **Create Vias**.

Results

Electrical Vias are created in the data tree under the Electrical Vias Assembly node.



Related Topics

[Board Vias Processing Dialog Box](#)

[Electrical Vias](#)

Generating Electrical Vias in Dielectric Layers From Images

Image files of PCB dielectric layers can be used to create a complex distribution of electrical vias.

Restrictions and Limitations

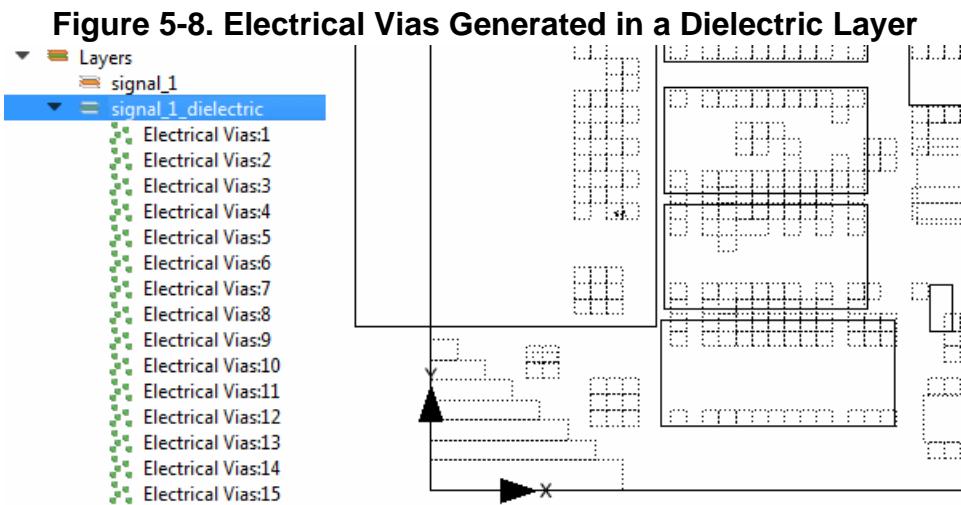
- You must have monochrome images of the dielectric layers, showing the vias. These may be embedded in an ODB++ file or may be separate image files.

Procedure

1. This step depends on how the board design has been loaded:
 - If the board design has been loaded by importing an ODB++ file, images representing the distribution of metal on metallic layers and the distribution of electrical vias on dielectric layers will be automatically attached to the appropriate layers in the data tree.
 - If the board design has not originated from an ODB++ file, then add the appropriate Dielectric layers manually, see “[Layers](#)” on page 107.
2. Right-click a Dielectric layer in the data tree and choose **Process Layer** from the popup menu.
 If File Name in the Layer property sheet has been set to the name of an image file, the Dielectric Layer Vias Processing dialog box is opened, otherwise a file browser is opened to select the image before the dialog box is opened.
3. Use the dialog box to adjust the representation.
 Image colors can be optionally inverted when the range is processed into vias.
4. When finished, click **Create Vias**.

Results

Electrical Vias are created in the data tree under the respective dielectric layer.



The Layer Type field in the Layer property sheet is not editable when vias are defined in the layer.

Related Topics

[Dielectric Layer Vias Processing Dialog Box](#)

[Electrical Vias](#)

Rectangular Components

The definition requirements of a rectangular component depends on the component type and how it was produced.

New components can be created using the **Create new Component** icon, or they can be created by copying from the Simcenter Flotherm library. New components are always created as Simple models, but library components can be of any modeling level.

The following are the types of component model:

- **Simple** — a single block representation requiring material definition.
- **2-Resistor** — a thermal resistor network.
- **DELPHI Resistor** — a Project DELPHI compact model.
- **Detailed** — a detailed FloTHERM PACK model imported from the library.
- **T3Ster** — a compact thermal resistor/capacitor model of an IC package generated by Mentor Graphics T3Ster-Master software and imported from the library.

Only Simple and 2-Resistor component types can be created from within EDA Bridge. These can be saved to, and restored from, a library, whereupon they are fully editable.

Components that have been created by FloTHERM PACK (2-Resistor, DELPHI Resistor and Detailed) can be imported into EDA Bridge, but are not fully editable. Only their location, board side, filtered, power, rotation, T_j _max, T_c _max can be changed. Any single Compact Component or Network Assembly SmartPart created in FloTHERM PACK (2-Resistor or DELPHI Resistor) can also be imported, but are not fully editable.

[Heatsinks](#) and [Thermal Vias](#) can be added to rectangular components.

Modeling Rectangular Components

If little or no information is known about the package, the Simple modeling level is appropriate for comparative Trend type thermal modeling. The only input required is the component location, overall size and material, either Typical plastic package (default) or Typical ceramic package.

If thermal resistor network information, in terms of θ_{jc} and θ_{jb} becomes available, then model the component as a 2-Resistor and define these two values. EDA Bridge does not accept θ_{ja} thermal resistances because of the inherent inaccuracies of employing such data.

The best advice is to use FloTHERM PACK to create component representations for use in EDA Bridge or Simcenter Flotherm. Such representations provide the best, most accurate, thermal predictions.

Details about FloTHERM PACK can be found at:

www.mentor.com/products/mechanical/products/flotherm-pack

Derived Power Dissipation for a Component at Fixed T_j

For a Detailed or resistor network component, the power dissipation can be derived during solution.

If you enter a “?” into the Power field of the property sheet, the solution will fix the T_j for the component equal to T_{j_max} and report the power dissipation required to achieve that T_j.

After solution, the derived powers are shown in the results table (Figure 5-9).

Figure 5-9. Component Derived Power

Reference Designator	Component Type	Total Power (W)
U1	2 Resistor	?=6.71
U2	Detailed	?=4.31



Related Topics

[Adding Existing Geometry Using the Library Pane](#)

[Component Property Sheet](#)

Cylindrical Components

Cylindrical Components are typically used to represent capacitors.

Select the board to contain the component, then click the Create new cylindrical component icon.

The component is added to the data tree. In the 2D view, the component is located at the origin of the parent board.

Related Topics

[Cylindrical Component Property Sheet](#)

Functional Groups Properties and Procedures

Use Functional Groups to manipulate related groups of components.

Functional Groups..... 118

Creating Functional Groups..... 119

Functional Groups

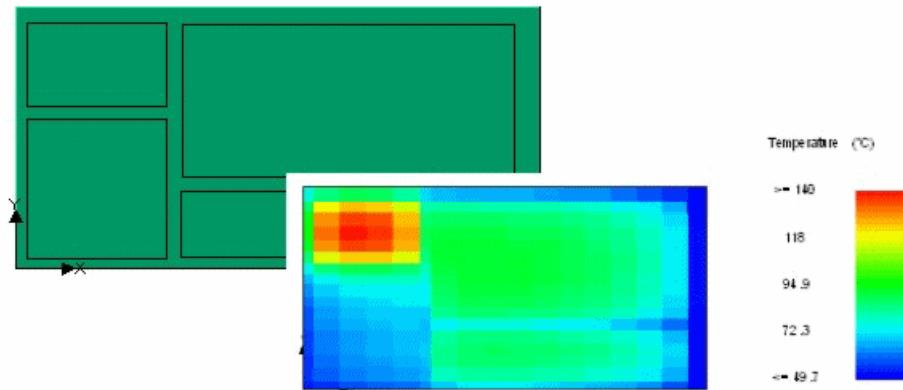
Functional Groups are useful for moving and rotating groups of components.

[Rectangular Components](#) and [Cylindrical Components](#) can be added to functional groups.

Functional Group Areas

Conceptual board layout is often initiated by partitioning the board into areas that are allocated for specific functions. If you define the area of a functional group, by unchecking the Size by Bounding Box check box in the Functional Group property sheet, heat defined for that functional group is smeared over that area of the board, see [Figure 5-10](#), therefore providing a method of thermal prediction prior to component selection for determination of minimum board temperatures.

Figure 5-10. Division of a Board Into Functional Group Areas



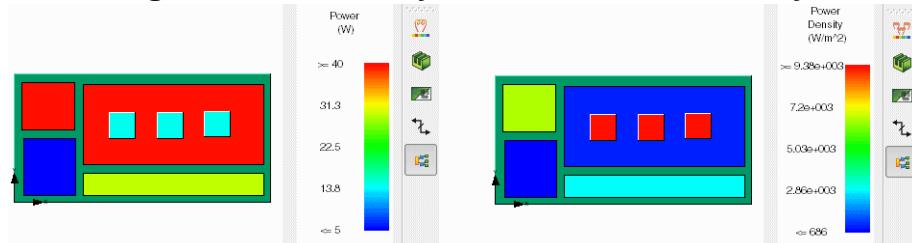
By unchecking the Size by Bounding Box check box, it is possible for a functional group component to be situated outside of the functional group area. If this occurs, then a warning is issued prior to thermal solution.

If Size By Bounding Box is checked (the default setting), then the displayed size of the functional group will always be equal to the bounding box of all the components within it (or the board size if there are no components).

Functional Groups Power and Power Density

Sized functional groups that have a power defined can be colored by Power or Power Density (by clicking the **Color Components** icon). If the functional group contains powered components, then the functional group area is colored by any unassigned power.

Figure 5-11. Color by Power or Power Density



Related Topics

[Creating Functional Groups](#)

[Functional Group Property Sheet](#)

Creating Functional Groups

You can create an empty functional group, or a functional group of existing components.

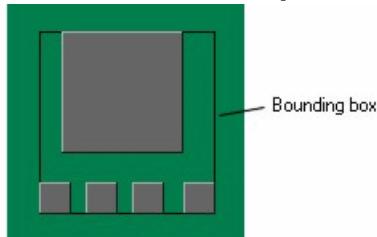
Procedure

1. You have a choice:

If you want to...	Do the following:
Create a new empty functional group ready to hold new components.	<ol style="list-style-type: none"> 1. Select the board to contain the functional group. 2. Click the Create new Functional Group icon. <p>The functional group is added to the data tree, ready to accept member components. An empty functional group does not appear in the 2D view because it is a reference origin to which components are added later.</p>

If you want to...	Do the following:
Create a new functional group from existing components.	<ol style="list-style-type: none">1. Select the components to be placed in the functional group2. Right-click over the components and choose Add to new Functional Group. The bounding box of a populated functional group is a black outline (see Figure 5-12) if the Display Functional Groups icon (Advanced Display Options toolbar) is active.

Figure 5-12. Functional Group Bounding Box



2. Move the location of the origin of a functional group to move all the components.
3. Use the functional group property sheet to define the start X-Y location and the combined power output of the functional group.
4. To swap a functional group with one held in the library, right-click the functional group in the data tree and choose **Replace with library item**, then use the [Library Item Selector Dialog Box](#) to find and load the required functional group.

Related Topics

[Functional Groups](#)

[Functional Group Property Sheet](#)

Rectangular Cutouts

Use Rectangular Cutouts to create rectangular holes in boards, thereby creating irregular shaped boards.

By default, a cutout is located at the origin of the parent board.

The default size is larger than the default size of a DaughterBoard, therefore, when adding a cutout to a DaughterBoard, the DaughterBoard may disappear from view. If this happens, select the new cutout in the tree, and make the cutout smaller using the re-sizing handles in the 2D view. The board will come into view as the cutout gets smaller.

When viewing through the board (by clicking the **View From** icon in the Viewing Options toolbar, then choosing **Through to Bottom**), only the dotted outline of the hole is visible.

Related Topics

[Rectangular Cutout Property Sheet](#)

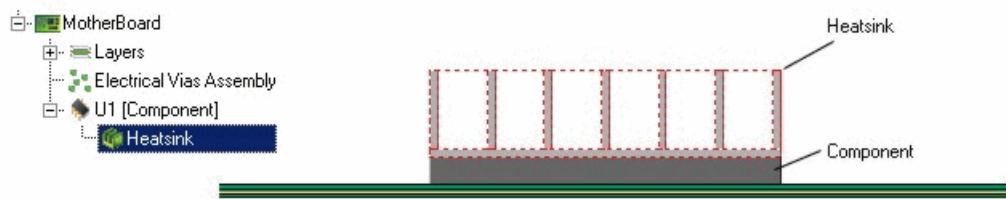
Heatsinks

Heatsinks can have plates or pins. Pins can be inline or staggered.

Providing that the Display Heatsinks icon is not set to Invisible (that is, it is either Rendered or Wireframe), heatsinks will appear in the display area, covering the components to which they have been added. use the Invisible option to more easily select underlying components in the graphical display area for subsequent moving/editing.

In the 2D front view, the heatsink covers the top of the component with the fins aligned with the flow direction.

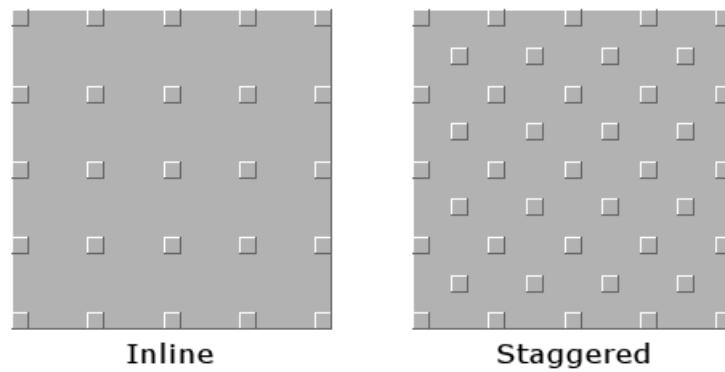
Figure 5-13. Heatsink on Component, Front View



Changes made to the size of the board or component do not affect the heatsink; the heatsink remains its original size until edited in the property sheet.

[Figure 5-14](#) is a Top view, showing the difference in distribution between Inline and Staggered heat pins.

Figure 5-14. Inline and Staggered Heat Sink Pins



By default, heatsinks are positioned centrally to cover the top of components. Use the property sheet to resize and/or place on a different face of the component, as well as change the offset from the default central position.

Figure 5-15. Heatsink Centered on Component

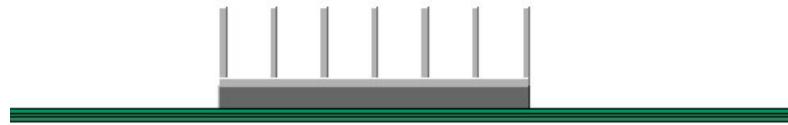


Figure 5-16 and Figure 5-17 show two views of a heatsink before and after applying an offset.

Figure 5-16. Default Positioning

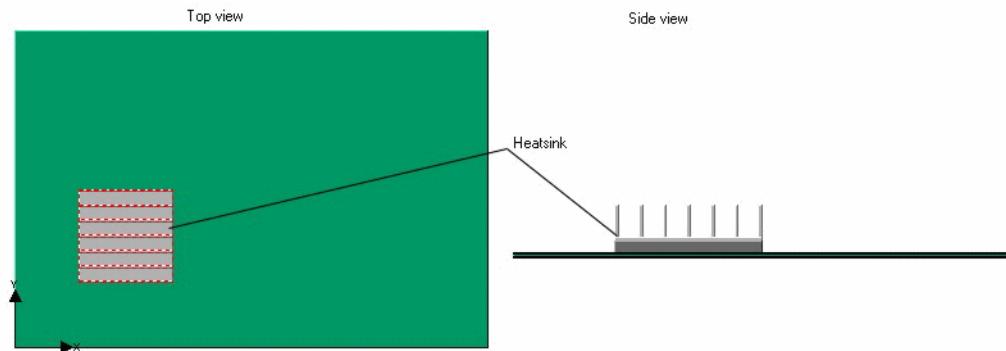
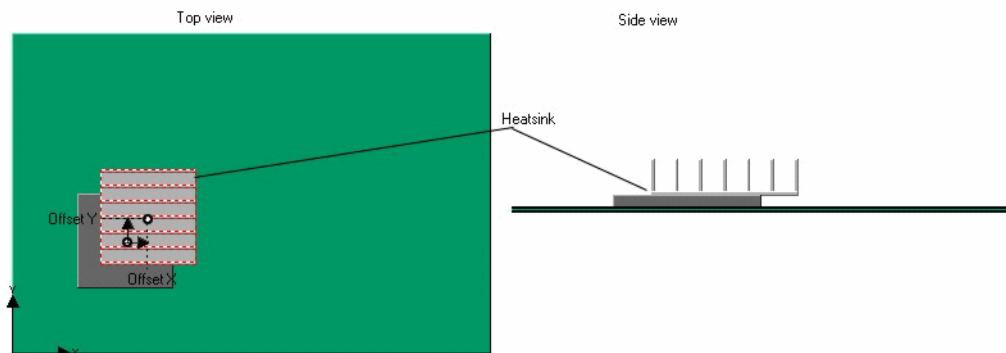


Figure 5-17. Position After Applying an Offset



Use the **Component Side** popup menu to position the heatsink on any face of the component. This is especially relevant when heatsinking through-hole type packages, for example, TO220.

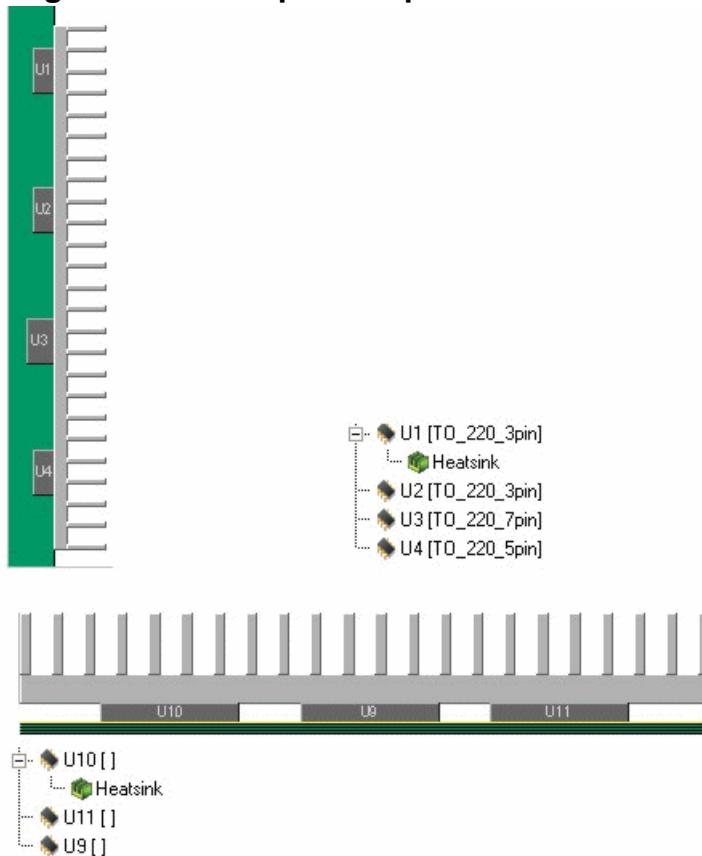
Figure 5-18. Heatsink on Component Side



Heatsink on Multiple Components

If a heatsink is positioned so that it spans multiple components, those components will be cooled by the heatsink provided that all the components are the same height or they have been aligned to the side that abuts the heatsink.

Figure 5-19. Multiple Component Heatsinks



Ensure that all components abut the heatsink base by:

- Aligning side mounted heatsinks left or right
- Keeping the height of the components identical on top mounted heatsinks. This can be done either by patterning a component, or by inspection in the component table displayed by clicking the **Component Table** tab at the bottom of the data tree.

For more information, see [Component Properties Table](#).

Figure 5-20. Component Heights in Component Table

Y Location (mm)	Board Side	Length (mm)	Width (mm)	Height (mm)	Filtered
0	Top	30	20	3	False
0	Top	30	20	3	False
0	Top	30	20	3	False

Component heights can also be checked by coloring components by Height.

Heatsink Library Save and Retrieval

Heats sinks can be saved to and swapped from a library by using the context menu **Save to library** and **Replace with Library Item** options.

Heat Sinks in Solder Reflow

Heat sinks details are ignored in solver reflow environments and are replaced by a simple block with a typical plastic package component material type.

If the heat sinks in solder reflow are not well modeled, then swap them for a simple or detailed appropriate component.

Related Topics

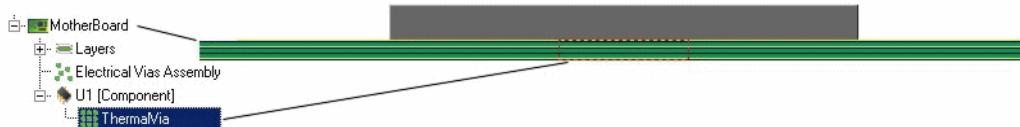
[Heatsink Property Sheet](#)

Thermal Vias

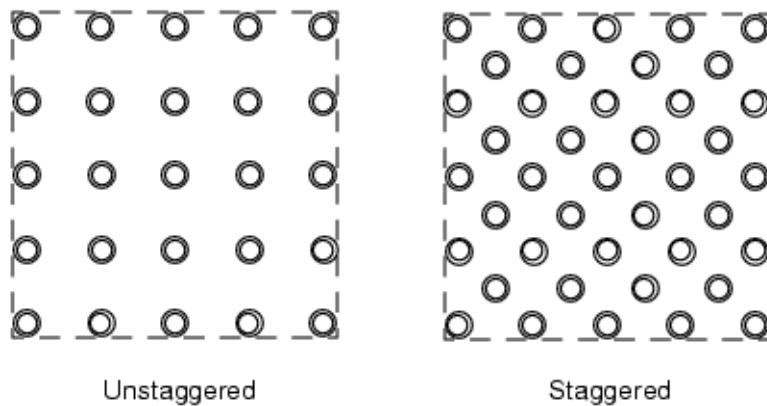
Although a thermal via represents an array of thermal vias, it is modeled as a single block with an averaged conductivity.

The thermal via is added to the data tree below the selected component. In the 2D view, the thermal via is located centrally underneath the component.

Figure 5-21. Thermal Via



[Figure 5-22](#) shows the difference in distribution between Unstaggered and Staggered thermal vias.

Figure 5-22. Unstaggered and Staggered Thermal Via Arrays

Related Topics

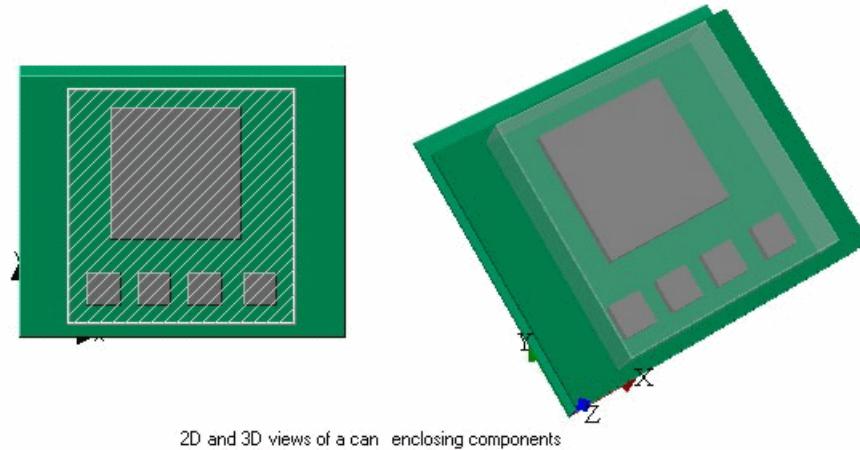
[Thermal Via Property Sheet](#)

Cans

Use the Can object to model an EM can (RF shield).

To create an empty can, select the MotherBoard or DaughterBoard to hold the can then click the **Create new Can** icon. The new can is located at the origin of the board.

To create a can that envelopes existing objects, select the components on a board to be included in the can, right-click and choose **Add to new Can** from the popup menu. The new can is created so that it covers the selected components.

Figure 5-23. A New Can Covering Components

By default, the can is 0.5 mm thick and constructed from mild steel.

Related Topics

[Can Property Sheet](#)

Potting Compound

Potting compounds represent the material applied to seal components on a board. They can be placed over all or part of either side of the board. Multiple non-overlapping potting compounds are allowed.

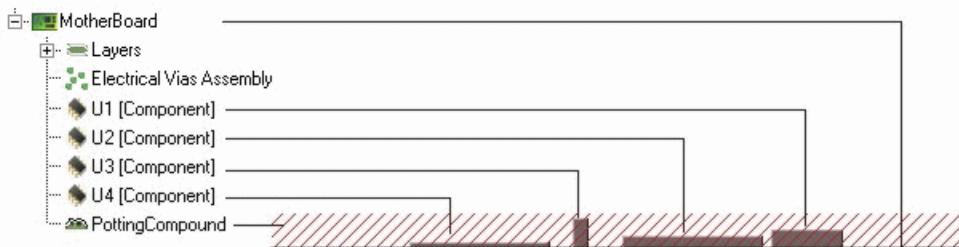
The potting compound is always located as the last entry in the parent board node to prevent any overwriting by the further addition of components.

By default, a typical epoxy overmold, with a thermal conductivity of 0.68 W/mK, is applied to cover the top surface of the selected MotherBoard or DaughterBoard up to a height of 10 mm.

The material can be replaced with one defined by the user or loaded from the supplied Resins material library.

To create a potting compound, select the board to be sealed and click the **Create Potting Compound** icon. The potting compound appears as a red mesh in the display area.

Figure 5-24. Adding Potting Compound



Related Topics

[Potting Compound Property Sheet](#)

Placement Keepout Regions

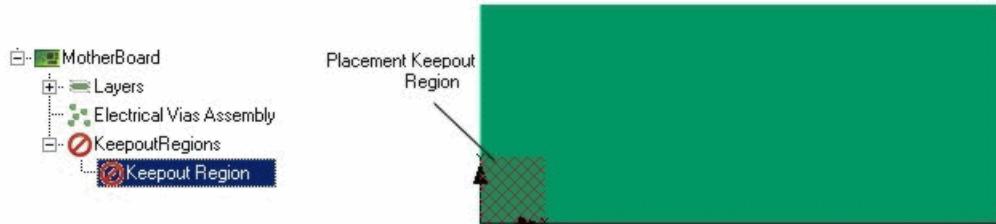
Placement keepout regions are location tools only, used to warn when components are positioned within specific locations on the MotherBoard or DaughterBoards.

Within a placement keepout region you can either be warned when all components or only components above a defined maximum height occupy the defined keepout region.

The annotation for a placement keepout region indicates the maximum component height allowed.

By default, a keepout region is located on top of, and at the origin of, the selected board. In the data tree, placement keepout regions are stored together below the Keepout Regions node.

Figure 5-25. Keepout Region at the Origin



Related Topics

[Placement Keepout Regions Property Sheet](#)

Modeling Objects Property Sheets and Dialog Boxes

The following property sheets and dialog boxes are associated with the modeling objects.

MotherBoard Property Sheet	129
DaughterBoard Property Sheet	130
Group Layer Property Sheet	132
Layer Property Sheet	133
Layer Patch Property Sheet	135
Electrical Vias Assembly Property Sheet	136
Electrical Via Property Sheet	137
Component Property Sheet	139
Cylindrical Component Property Sheet	141
Functional Group Property Sheet	143
Rectangular Cutout Property Sheet	144
Heatsink Property Sheet	145
Thermal Via Property Sheet	147
Can Property Sheet	148
Potting Compound Property Sheet	149
Placement Keepout Regions Property Sheet	150
Layer Trace Processing Dialog Box	151
Dielectric Layer Vias Processing Dialog Box	153
Board Vias Processing Dialog Box	155

MotherBoard Property Sheet

To access: Select a MotherBoard.

Use this property sheet to define the size, rotation, and material of the motherboard.

Objects

Field	Description	Defaults
Name	A text entry field identifying the MotherBoard.	MotherBoard
Modeling Level	<p>There is a choice of two modeling levels:</p> <ul style="list-style-type: none"> • Trend, where the board is treated as a single orthotropic board and heatsinks are modeled as compact. • Accurate, where each layer is modeled explicitly and heatsinks are modeled as compact. 	Accurate
Auto-Grid	Add regions to the components and to the board, attach grid constraints and localize the grid on the regions. This is to ensure that the simulation in Simcenter Flotherm will have enough grid cells for the components and the board to resolve the physics of heat transfer and flow correctly, and in detail. The regions are hidden (grayed-out in Project Manager) for clarity. The regions can be deleted in Simcenter Flotherm if desired, or Auto-Grid switched off.	On
Length	X-direction dimension.	0.4 m
Width	Y-direction dimension.	0.2 m
Thickness	Z-direction dimension.	0.0016 m
Conductor Material	The default conductor material for any layers of the MotherBoard and any DaughterBoards.	Copper (Pure)
Dielectric Material	The material of the MotherBoard, and the default dielectric material for any layers of the MotherBoard and any DaughterBoards.	FR4
Notes	Construction notes.	

Related Topics

[MotherBoards](#)

DaughterBoard Property Sheet

To access: Select a DaughterBoard.

Use this property sheet to define the size, rotation, standoff, and material of a daughterboard.

Objects

Field	Description	Default
Name	A text entry field identifying the DaughterBoard.	DaughterBoard:<n>
X Location	DaughterBoard location with respect to the origin of the MotherBoard.	0 m for Parallel and Perpendicular Y. 0.0492 m for Perpendicular X.
Y Location	DaughterBoard location with respect to the origin of the MotherBoard.	0 m for Parallel and Perpendicular X. 0.0242 m for Perpendicular Y.
Mount Type	Choose from: <ul style="list-style-type: none">• Parallel – The DaughterBoard and MotherBoard are parallel with one other.• Perpendicular X/Y Low/High – The DaughterBoard is perpendicular to the MotherBoard, and perpendicular to the x- or y-axis of the MotherBoard. The top of the DaughterBoard is the surface in the Low or High direction.	Parallel
Standoff	(Parallel) The distance between the DaughterBoard and MotherBoard.	0.02 m
Board Offset	(Perpendicular) The distance between the edge of the DaughterBoard and MotherBoard.	0.005 m
Connector Height	(Perpendicular) The height of the connector used to mount the DaughterBoard onto the MotherBoard. The length of the connector is always equal to the width of the DaughterBoard.	0.015 m
Connector Width	(Perpendicular) The width of the connector used to mount the DaughterBoard onto the MotherBoard.	0.008 m

Field	Description	Default
Connector Material	(Perpendicular) The material of the connector used to mount the DaughterBoard onto the MotherBoard.	Typical Connector
Board Side	The DaughterBoard location: over the Top or under the Bottom of the MotherBoard.	Top
Z Rotation	Choice of a 90, 180 or 270 degree rotation of the DaughterBoard about the z-axis.	None
Length	X-direction dimension.	0.1 m
Width	Y-direction dimension.	0.05 m
Thickness	Z-direction dimension.	0.0016 m
Conductor Material	The default conductor material for any layers of the DaughterBoard.	Inherited from the MotherBoard.
Dielectric Material	The material of the DaughterBoard, and the default dielectric material for any layers of the DaughterBoard.	Inherited from the MotherBoard.
Notes	Construction notes.	

Related Topics

[DaughterBoards](#)

Group Layer Property Sheet

To access: Select the parent of a group of Layers, that is, the Layers child of a MotherBoard or DaughterBoard.

Use this property sheet to specify an equispaced group of layers or to define the layer offset distances from the top of the board.

Objects

Object	Description	Default
Name	A text entry field identifying the layers.	Layers
Equispaced	When checked (on) the layers are equispaced over the thickness (Z-direction) of the board.	On
Layer0 Offset	(Equispaced off) The distance from the top of the board to the first layer.	0 (top of the board)
Layer< n > Offset	(Equispaced off) The distance from the top of the board to the < n >th layer.	Equidistant location

Usage Notes

When the model is checked (**File > Check Model**), then an error message will be issued and the solution will not be allowed if a layer is offset such that it is located beyond the boundary of the board.

Related Topics

[Layers](#)

Layer Property Sheet

To access: Select a Layer.

Use this property sheet to define the properties of a PCB layer.

Objects

Field	Description	Default
Name	Identifies the layer.	signal:<n> or DaughterBoard:Layer:<n>
Layer Type	Metallic or Dielectric. Note: Not selectable if a layer patch has been added to a Metallic layer or a via has been added to a Dielectric layer.	Metallic
Thickness	Default thickness in Oz reflects the standard definition units for conductor layers within a PCB	Signal = 1 Oz Power or Ground = 0.5 Oz
Conductor Material	The conductor material of the layer.	Inherited from the parent MotherBoard or DaughterBoard, for example, [Copper (Pure)].
Dielectric Material	The dielectric material of the layer.	Inherited from the parent MotherBoard or DaughterBoard, for example, [FR4].
% Coverage	The area-weighted averaged proportion of conductor material in the layer. A layer with a very low density of traces would have a low percent coverage. Solid copper layers have a very high percent coverage. For most applications, signal layers can be assumed to have 20% coverage while power or ground layers can be assumed to have approximately 90% coverage.	Signal = 20% Power or Ground = 90%

Field	Description	Default
File Name	<p>The name of a file containing a black and white image representing the distribution of metal on that layer.</p> <p>For metallic layers, this identifies the file used to generate an approximate representation of patches, see “Generating Layer Patches From Images or EDA Data” on page 110.</p> <p>For dielectric layers, this identifies the file used to generate an approximate representation of vias, see “Generating Electrical Vias in Dielectric Layers From Images” on page 114.</p>	
Notes	Construction notes.	

Usage Notes

If you need to define areas of varying % Coverage, then add layer patches.

Related Topics

[Layer Patches](#)

[Layers](#)

Layer Patch Property Sheet

To access: Select a Layer Patch.

Use this property sheet to define the size, position, and percentage copper coverage of a layer patch.

Objects

Field	Description	Default
Name	Identifies the layer patch.	Layer Patch:<n>
X Location	Location in the X-direction, relative to parent board origin	0
Y Location	Location in the Y-direction, relative to parent board origin	0
Length	X-direction dimension if board has not been rotated.	0.1 m
Width	Y-direction dimension if board has not been rotated.	0.1 m
% Coverage	Data entry field referring to the area weighted averaged proportion of copper in that layer. The default % Coverage of 100% is suited for representing copper pads. The % Coverage is represented in the display area by varying the patch color in graded steps from green (0%) to orange (100%).	100
Patch Material	Choice of Copper (Pure) or Copper (Aluminized).	Copper (Pure)

Related Topics

[Defining a Layer Patch](#)

[Generating Layer Patches From Images or EDA Data](#)

[Layer Patches](#)

Electrical Vias Assembly Property Sheet

To access: Select an Electrical Via Assembly.

Use this property sheet to define the plating and fill materials of an electrical via assembly, and to reference an image file.

Objects

Field	Description	Default
Name	Identifies the electrical via assembly.	Electrical Vias Assembly
Elec. Vias Plating Mat.	Popup selection menu for the plating material. Click [...] to open the Material Library Selector dialog box.	Copper (Pure)
Elec. Vias Fill Mat.	Popup selection menu for the fill material. Click [...] to open the Material Library Selector dialog box.	None (that is, air)
File Name	The name of the file containing the electrical via image to be used to generate an approximate representation. The image file is used to represent the vias. It will be loaded into the Board Vias Processing dialog box when Process Electrical Vias is chosen from the Electrical Vias Assembly context menu.	
Notes	Construction notes.	

Related Topics

[Electrical Vias](#)

[Board Vias Processing Dialog Box](#)

[Material Library Selector Dialog Box](#)

Electrical Via Property Sheet

To access: Select an Electrical Via.

Use this property sheet to define the area and location of a group of electrical vias, with the option to specify the number and average diameter (manual) or a percentage coverage (automatic).

Objects

Data Fields	Description	Defaults
Name	Identifies the electrical via.	Electrical Vias
X, Y Location	Start location of the electrical via with respect to the Motherboard.	0 mm
Length	Dimension of electrical via in the X-direction.	50 mm
Width	Dimension of electrical via in the Y-direction.	50 mm
Definition Type	<ul style="list-style-type: none"> Manual — requires the number, average diameter, plating thickness and material defined. These values are checked prior to solution. If the derived percentage coverage is greater than or equal to 90.69% an error message is reported. Automatic — requires the Percentage Coverage. The materials are inherited from the parent Electrical Vias Assembly property sheet. 	Manual
Number	(Manual) The number of vias.	10
Average Diameter	(Manual) The average diameter.	0.3 mm
Average Plating Thickness	(Manual) The average plating thickness.	0.025 mm
Plating Material	(Manual) Material choice menu. Click [...] to open the Material Library Selector dialog box.	Copper (Pure)
Percentage Coverage	(Automatic) The percentage coverage of the Electrical Vias object. Accepts values up to the maximum theoretical packing density for uniform diameter vias of 90.69%.	5%
Fill Material	(Manual) Material choice. Click [...] to open the Material Library Selector dialog box.	None
Notes	Construction notes.	

Related Topics

[Defining an Area of Electrical Vias](#)

[Electrical Vias](#)

[Material Library Selector Dialog Box](#)

Component Property Sheet

To access: Select a Component.

Use this property sheet to define a component and its position on the board.

Objects

Field	Description	Default
Reference Designator	Identifies the component.	U< n >
Package Name	A text entry field identifying the package to which the component belongs.	Component
Part Number	Text entry field identifying the component part number.	Blank
Power	<p>Data entry field. Note that power units for Detailed models <i>cannot</i> be changed; only units for Simple and 2 Resistor models can be changed.</p> <p>Power source types that EDA Bridge does not handle, but which may be present in the component, are named here (for example, (Source/Area), Fixed, Linear, Non-Linear), but you cannot see any further details. You may change such a field to a Total Source value, but if you do so you cannot revert back to the original power source type.</p>	0 W
Component Type	Choice of model type between a Simple or 2 Resistor model). For imported components, this field is blank or read-only and may be DELPHI Resistor or T3Ster.	Simple
X Location	Data entry field, relative to parent board origin	0 m
Y Location	Data entry field, relative to parent board origin	0 m
Z Rotation	90/180/270 degrees	None
Length	(Simple) Dimension in X direction.	0.04 m
Width	(Simple) Dimension in Y direction.	0.04 m
Height	(Simple) Dimension in Z direction.	0.003 m
Board Side	Top/Bottom of MotherBoard	Top
Junction To Case Resistance	(2 Resistor) Data entry field.	1e+010 K/W
Junction To Board Resistance	(2 Resistor) Data entry field.	1e+010 K/W
Tc_max	(2 Resistor, DELPHI, or Detailed types) Data entry field.	75°C

Field	Description	Default
Tj_max	(2 Resistor, DELPHI, or Detailed types) Data entry field.	90°C
Component Material	(Simple) Material choice of Typical Ceramic Package or Typical Plastic Package.	Typical Plastic Package
Automatic Peak Reflow Body Temperature	Toggles on/off the automatic calculation of the maximum temperature the component can suffer during reflow based on the IPC-J-STD-020C standard. When set off, you can manually enter the peak temperature.	On
Peak Reflow Body Temperature	(Automatic Peak Reflow Body Temperature off) Sets the value of the maximum temperature the component can suffer during reflow. The default value is derived from the IPC-J-STD-020-C standard but could also be supplied by the component supplier.	Calculated value
Filtered	When filtered, the component power is smeared across the board.	Off
Deactivated	When deactivated, the component is not visible and takes no part in the solution.	Off
Notes	Construction notes.	Blank text

Related Topics

[Rectangular Components](#)

[Deactivating Objects](#)

[Filtered and Deactivated Objects](#)

Cylindrical Component Property Sheet

To access: Select a Cylindrical Component.

Use this property sheet to define the size, location, and material of a cylindrical component or capacitor.

Objects

Field	Description	Default
Reference Designator	Identifies the cylindrical component.	U< n >
Package Name	The name of the package to which the component belongs.	Component
Part Number	The component part number.	Blank
Power	Use to model powered electrolytic capacitors.	0 W
X Location	Bounding box footprint of component located relative to parent board origin	0 m
Y Location	Bounding box footprint of component located relative to parent board origin	0 m
Radius	Data entry field	0.01 m
Height	Data entry field	0.02 m
Board Side	Located either on the Top or Bottom of the MotherBoard.	Top
Cylindrical Component Material	Material choice of Typical Electrolytic Capacitor or Typical Plastic Package.	Typical Electrolytic Capacitor
Automatic Peak Reflow Body Temperature	Automatic calculation of the maximum temperature the component can suffer during reflow based on the IPC-J-STD-020C standard. When unchecked, you can manually enter the peak temperature.	On
Peak Reflow Body Temperature	(Automatic Peak Reflow Body Temperature off) It sets the value of the maximum temperature the component can suffer during reflow. The default value is derived from the IPC-J-STD-020-C standard but could also be supplied by the component supplier.	Calculated value
Filtered	When checked, the cylindrical component power is smeared across the board.	Off
Deactivated	When checked, the cylindrical component is not visible and takes no part in the solution.	Off
Notes	Construction notes.	Blank text

Related Topics

- [Cylindrical Components](#)
- [Filtered and Deactivated Objects](#)
- [Deactivating Objects](#)

Functional Group Property Sheet

To access: Select a Functional Group.

Use this property sheet to define the identifier, total power, location, and size of a functional group.

Objects

Field	Description	Default
Reference Designator	Identifies the functional group.	F< n >
Function Name	A text entry field identifying the function.	Function
Total Power	Data entry field	0 W
X Location	Data entry field, relative to MotherBoard origin	0 m
Y Location	Data entry field, relative to MotherBoard origin	0 m
Size By Bounding Box	A check box for setting the size of the functional group to always match that of the bounding box of all the components it contains (or the board size if there are no components).	On
X Size	Data Entry field for size in the x-direction. Only available when Size By Bounding Box is not checked.	Board X Size
Y Size	Data Entry field for size in the y-direction. Only available when Size By Bounding Box is not checked.	Board Y Size
Z Rotation	Rotate about the z-axis by 90, 180 or 270 degrees.	None
Notes	Construction notes.	Blank Text

Related Topics

[Functional Groups](#)

[Creating Functional Groups](#)

Rectangular Cutout Property Sheet

To access: Select a Rectangular Cutout.

Use this property sheet to define the location and size of a rectangular cutout in the board.

Objects

Object	Description	Default
Name	Identifies the cutout.	Rectangular Cutout:< n >
X Location	Data entry field.	0 m
Y Location	Data entry field.	0 m
Length	Data entry field.	0.1 m
Width	Data entry field.	0.1 m

Related Topics

[Rectangular Cutouts](#)

Heatsink Property Sheet

To access: Select a Heatsink.

Use this property sheet to define the dimensions and configuration of a heatsink.

Objects

Field	Description	Default
Name	Identifies the heatsink.	Heatsink
Z Rotation	90/180/270 degrees	None
Base Width	Data entry field	0.04 m
Base Length	Data entry field	0.04 m
Base Thickness	Data entry field	0.001 m
X Offset	Horizontal offset from the heatsink center position.	0 m
Y Offset	Vertical offset from the heatsink center position.	0 m
Type	Connection choice of Plate Fin, Pin Fin or No Fin. The No Fin option is to represent a simple heat spreader.	Plate Fin
Fin Height	(Plate Fin) Data entry field.	0.009 m
Fin Width	(Plate Fin) Data entry field.	0.0009 m
Number of Fins	(Plate Fin) Data entry field.	5
Center Gap	(Plate Fin) Data entry field.	0 m
Pin Height	(Pin Fin) Data entry field.	0.009 m
Pin Width	(Pin Fin) Data entry field.	0.0009 m
Pin Length	(Pin Fin) Data entry field.	0.009 m
Pin Arrangement	(Pin Fin) Arrangement choice of Inline or Staggered. When Staggered, more pin fins are added but the total size of the heatsink remains the same.	Inline
Pins in X Direction	Data entry field, pin fin only	3
Pins in Y Direction	Data entry field, pin fin only	5
Heatsink Material	Material choice	Aluminum (Pure)

Field	Description	Default
Specify Interface Material	Interface specification choice of By Material or By Value. Enables the interface material to be specified by material type or the interface resistance value.	By Material
Heatsink Interface Resistance	(By Material) A choice box for the material type.	Typical Interface ($1 \text{ (C in}^2\text{)/W})$
Interface Resistance	(By Value) A data entry field for the interface resistance value.	$1 \text{ (C in}^2\text{)/W}$
Component Side	The side of the component abutting the heatsink. A choice of Top, Front, Back, Left, or Right.	Top
Notes	Construction notes.	Blank Text

Related Topics

[Heatsinks](#)

Thermal Via Property Sheet

To access: Select a Thermal Via.

Use this property sheet to define a staggered or unstaggered array of PCB thermal vias.

Objects

Field	Description	Default
Name	Identifies the thermal via.	ThemalVia
Diameter	Data entry field setting the outer drilled diameter of a via hole.	0.003 m
Plating Thickness	Data entry field.	2.5e-005 m
Pitch in X	Data entry field.	0.0012 m
Pitch in Y	Data entry field.	0.0012 m
Number in X	Data entry field.	10
Number in Y	Data entry field.	10
Staggered	Defines a staggered array. When staggered, more vias are added but the total occupancy (bounded area) remains the same.	Checked off
Plating Material	Material choice of Copper (Pure) or Copper (Aluminized).	Copper (Pure)
Fill Material	Material choice of None or Solder.	None
Notes	Construction notes.	Blank Text

Related Topics

[Thermal Vias](#)

Can Property Sheet

To access: Select a Can.

Use this property sheet to define the location, size, and material of a Can.

Objects

Field	Description	Default
Name	Identifies the can.	Can
X Location	Data entry field	0 mm
Y Location	Data entry field	0 mm
Length	Data entry field	40 mm
Width	Data entry field	20 mm
Height	Data entry field	15 mm
Thickness	Data entry field	0.5
Board Side	Choice of Top or Bottom	Top
Can Material	Material choice	Steel (Mild)
Notes	Construction notes.	

Related Topics

[Cans](#)

Potting Compound Property Sheet

To access: Select a Potting Compound.

Use this property sheet to define the location, size, and material of potting compound.

Objects

Field	Description	Default
Name	Identifies the potting compound.	Potting Compound
X Location	Data entry field	0 mm
Y Location	Data entry field	0 mm
Z Rotation	90/180/270 degrees	None
Length	Data entry field	Length of the board
Width	Data entry field	Width of the board
Height	Data entry field defining the height of the top of the potting compound from the board surface.	10 mm
Board Side	Choice of Top or Bottom	Top
Material	Material choice for potting compound	Epoxy Overmold (Typical)
Notes	Construction notes.	

Related Topics

[Potting Compound](#)

Placement Keepout Regions Property Sheet

To access: Select a Placement Keepout Region.

Use this property sheet to define the size and location of an area on the board where components should not be located (placement keepout region).

Objects

Field	Description	Default
Name	Identifies the Placement Keepout region.	Keepout Region
X Location	Data entry field	0 mm
Y Location	Data entry field	0 mm
Length	Data entry field	1/8th the board length
Width	Data entry field	1/8th the board length
Max Height	Data entry field defining the maximum height of objects allowed in the Placement Keepout Region. A value of 0 prohibits the placement of any object.	0
Board Side	Choice of Top or Bottom	Top
Notes	Construction notes.	Blank Text

Related Topics

[Placement Keepout Regions](#)

Layer Trace Processing Dialog Box

To access: Right-click a Metallic layer in the Layers node of the data tree and choose **Process Layer**. If there is no associated image file, then a file browser is opened so that you can browse to, and select, an image file.

Use this dialog box to adjust the representation of the layer patches to be generated.

Objects

Field	Description
Resolution of Longest Side	The slider bar adjusts the resolution of the image. Move the slider to the right to reduce the accuracy of the image and hence the number of patches required. Setting <i>the resolution to be the same</i> for all processed layers aligns the grid lines associated with the layer objects, resulting in a higher quality, and more easily solved grid. The maximum resolution you can select is either the resolution of the longest side of the original image or the Maximum Resolution of Longest Side setting in the GUI Preferences Dialog Box , whichever is lower.
Number of % Conductor Bands	The slider bar adjusts the maximum number of shades of gray in the image. This has the effect of reducing the increments of conductivity between low (white) and high (black) conductivities. Move the slider to the right to reduce the accuracy of the image and hence the number of patches required.
Show Original Board	Hold down to display the original image in the viewer. Release to redisplay the approximated representation.
Refresh Patch Number	Calculates the number of layer patches to be created. To ensure speed of operation, it is recommended that this is less than 500. A warning is displayed if a large number of vias is about to be created.
Invert image colors	Check to invert the image. By default, the imported image is expected to display the conductors and insulators in black and white, respectively, however, some EDA tools generate inverted images which need to be inverted again to be processed correctly.

Usage Notes

The last used slider settings are saved and used as the default values for the next process layer generation.

If the file and board have different aspect ratios, you will be able to defer the creation of the patches.

Click **Create Patches** to create patches.

Related Topics

[Layer Patches](#)

[Generating Layer Patches From Images or EDA Data](#)

Dielectric Layer Vias Processing Dialog Box

To access: Applies only to imported ODB++ files. Right-click a Dielectric layer in the Layers node of the data tree and choose **Process Layer**. If there is no associated image file, then a file browser is opened so that you can browse to, and select, an image file.

Use this dialog box to adjust the number of vias to be generated.

Objects

Field	Description
Resolution of Longest Side	The slider bar adjusts the resolution of the image. Move the slider to the right to reduce the accuracy of the image and hence the number of patches required. Setting <i>the resolution to be the same</i> for all processed layers aligns the grid lines associated with the layer objects, resulting in a higher quality, and more easily solved grid. The maximum resolution you can select is either the resolution of the longest side of the original image or the Maximum Resolution of Longest Side setting in the GUI Preferences Dialog Box , whichever is lower.
Number of % Conductor Bands	The slider bar adjusts the maximum number of shades of gray in the image. This has the effect of reducing the increments of conductivity between low (white) and high (black) conductivities. Move the slider to the right to reduce the accuracy of the image and hence the number of patches required.
Show Original Board	Hold down to display the original image in the viewer. Release to redisplay the approximated representation.
Refresh Vias Number	Calculates the number of vias to be created. To ensure speed of operation, it is recommended that this is less than 500. A warning is displayed if a large number of vias is about to be created.
Invert image colors	Check to invert the image. By default, the imported image is expected to display the conductors and insulators in black and white, respectively, however, some EDA tools generate inverted images which need to be inverted again to be processed correctly.

Usage Notes

The last used slider settings are saved and used as the default values for the next process layer generation.

Click **Create Vias** to create vias.

Related Topics

[Generating Electrical Vias in Dielectric Layers From Images](#)

[Electrical Vias](#)

Board Vias Processing Dialog Box

To access: Right-click the Electrical Vias Assembly node of the data tree and choose **Process Electrical Vias**. If there is no associated image file, then a file browser is opened so that you can browse to, and select, an image file.

Use this dialog box to adjust the representation of the electrical vias.

Objects

Field	Description
Resolution of Longest Side	To adjust the resolution of the image and the number of vias required, move the slider bar: <ul style="list-style-type: none">• to the right to reduce the accuracy of the image and hence the number of vias• to the left to increase the refinement of the image and increase the number of vias. The maximum resolution you can select is either the resolution of the longest side of the original image or the Maximum Resolution of Longest Side setting in the GUI Preferences Dialog Box , whichever is lower.
Number of % Conductor Bands	The slider bar adjusts the maximum number of shades of gray in the image and therefore the number of resulting materials used to capture the variation in thermal conductivity. This has the effect of reducing the increments of conductivity between low (white) and high (black) conductivities. Move the slider to the right to reduce the accuracy of the image and hence the number of vias required.
Show	Hold down to display the original image in the viewer. Release to redisplay the approximated representation.
Refresh Vias Number	Calculates the number of electrical vias that will be generated. To ensure speed of operation, it is recommended this is less than 200. A warning is displayed if a large number is about to be created.
Invert image colors	Check to invert the image. By default, the imported image is expected to display the copper and insulators in black and white, respectively, however, some EDA tools generate inverted images which need to be inverted again to be processed correctly.

Usage Notes

The last used slider bar settings are saved and used as the default values for the next generation of vias.

If the file and board have different aspect ratios, you will be able to defer the creation of the vias.

Click **Create Vias** to create a set of electrical vias. These appear in the tree and over the board.

Related Topics

[Generating Electrical Vias in an Electrical Vias Assembly From an Image](#)

[Electrical Vias](#)

Chapter 6

Frequently Asked Questions

Answers for frequently asked questions (FAQs) about EDA Bridge.

Board Definition FAQs	157
Component Definition FAQs	158
Layer Definition FAQs	161
Solution FAQs	161
Transfer From EDA Bridge to Simcenter Flotherm FAQs	162

Board Definition FAQs

Answers to questions about board layout and definition.

1. Q: Can EDA Bridge model DaughterBoards mounted perpendicularly to a MotherBoard (for example, Memory SIMMS/DIMMS)?
A: Yes, EDA Bridge DaughterBoards can represent mezzanine, parallel mounted boards or perpendicularly mounted boards.
See “[DaughterBoards](#)” on page 104.
2. Q: What does the percent coverage represent in the property sheet for a board layer?
A: This refers to the area weighted averaged proportion of copper in that layer. A layer with a very low density of traces would have a low percent coverage. Solid copper layers would of course have a very high percent coverage.
3. Q: How do I know what values to input for the percent coverage?
A: For most applications, signal layers can be assumed to have 20% coverage while power or ground layers will have approximately 90% coverage. By default, an added layer will be a signal layer with 20% coverage.
4. Q: Can I define areas of varying percent coverage in a board layer?
A: Yes, [DaughterBoards](#) can be defined attached to a selected board layer. These allow for rectangular non-overlapping regions of differing percent coverage or even a different material. The default percent coverage of 100% is suited for representing copper pads.
See “[Layer Patches](#)” on page 109.
5. Q: How are the layers positioned in the board?

A: By default, layers are evenly spaced through the width of the board. This can be overridden by first checking off Equispaced in the Group Layer property sheet, then define the location of any individual layer with respect to the top surface of the board.

For more information, see “[Group Layer Property Sheet](#)” on page 132.

Component Definition FAQs

Answers to questions about components.

1. Q: Is there another method to import board layout information other than using **File > Import > ImportIDF**?

A: Yes, a CSV (comma separated variable) file can be imported using

File > Import > Import CSV Layout. The format of the CSV file is best described by examining the contents of an exported board layout using **File > Export > Export CSV**. This exported file can be edited and re-imported. On import of the CSV file an option is given of either adding components to an existing layout or updating an existing layout.

2. Q: How do I change the material of my heatsink?

A: By default, heat sinks are assumed to be constructed from pure aluminum (thermal conductivity = 201 W/mK). To change this material click the material library button, found next to the material in the heat sink property sheet, to bring up the [Material Library Selector Dialog Box](#). Navigate to the new material and click “+” to make it available. Then **OK** the Material Library Selector dialog box.

The same method is used to change the material of the PCB dielectric, a layer and a layer patch.

3. Q: How do I replace multiple components with a component representation from the library?

A: Press Ctrl+click to multiple select the components you want to replace. Then right-click one of the components to access the popup menu and select Replace with library item. Navigate to the required library item then double-click.

4. Q: How do I assign the same power to multiple components?

A: Press Ctrl+click to multiple select the components you want to assign the same power to. The property sheet header indicates that a number of objects are selected, see “[Property Sheets](#)” on page 21. Data that is common between the selections is shown, data that varies across the selections is left blank. Editing the power field will assign that power to all the selected components. This method can also be used to assign the same size, position, board side, and so on. to many components in one operation.

5. Q: What type of package thermal models does EDA Bridge support?

A: EDA Bridge supports the following package models:

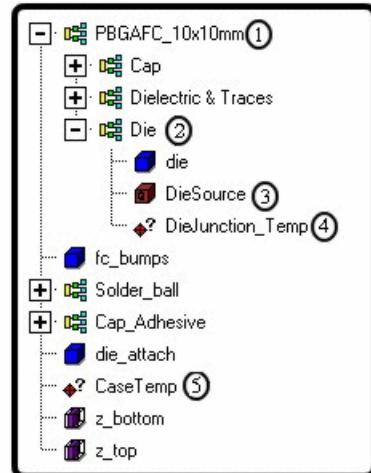
- Simple lumped models (for example, single object with a representative thermal conductivity)
 - Two-resistor compact models (for example, Theta_jc and Theta_jb)
 - Multi-resistor Delphi compact models
 - Detailed models generated by FloTHERM PACK
6. Q: How can I construct a detailed component model in Simcenter Flotherm for subsequent import into EDA Bridge?

A: Ideally, the assembly structure and naming convention should be the same as that used by FloTHERM PACK for detailed components, however, only the first rule in [Figure 6-1](#) is mandatory.

Figure 6-1. Constructing a Detailed Component Model

Specifically:

1. Give the top-level assembly the same name as the component.
2. Create a sub-assembly called Die.
Note: The Die sub-assembly shown here contains a cuboid, source and monitor point, but it can contain a compact component SmartPart instead. No other SmartParts are allowed in the component model.
3. The source primitive called DieSource has an attached source attribute with an Activated temperature source value.
4. A monitor point called DieJunction_Temp.
5. Create a monitor point called CaseTemp below the top-level assembly.



All name matches are case sensitive. Monitor points are used to extract junction and case temperatures. Multiple Die assemblies can be present to represent MCM or stacked die packages.

This approach can be used to create a detailed representation of any type of component, but in EDA Bridge it will be presented as if there exists a Junction and a Case Temperature.

7. Q: Why can I only resize some of my components?

A: Components of Simple component type can be resized either graphically or by using the property sheet as shown in [“Rectangular Components”](#) on page 116. Components that are defined as thermal resistor network (2 resistor or DELPHI) or detailed FloTHERM PACK types cannot be resized. This is because the parameters defining their construction are valid only for their specific size and so their size should not be altered.

8. Q: How do I import a component generated by FloTHERM PACK into EDA Bridge?

A: If a single PDML (*.pdml) file is downloaded then this file should first be placed on the file system. Then, within EDA Bridge, open the Library pane, select a library, right-click popup, select import then select the .pdml file. This item will then be available for loading into EDA Bridge as explained in “[Adding Existing Geometry Using the Library Pane](#)” on page 49.

Libraries of multiple objects downloaded in .library format can be imported using the import function, see “[Importing and Exporting Libraries](#)” on page 76.

9. Q: How do I sort my components based on size, power, and so on?

A: A tabular view of the component layout can be viewed by clicking on the Component Table tab (see “[Component Properties Table](#)” on page 18). Clicking on the heading of a column will sort the entire table by that value. Clicking again will reverse the sort order. This is extremely useful when used in conjunction with click, Shift+click in the Reference Designator cells to select a range of components based on a sorted parameter, for example, selecting all components on the bottom of the board after having sorted on board side.

10. Q: What different methods are there for specifying component power dissipation for a layout?

A: A power list can be imported at any time to reset power dissipations, see “[Importing a Power List](#)” on page 66. Entries in the imported CSV file take the form <Reference Designator>,<Power (W)>.

IDF V3 files have the ability to store a power dissipation for each component in the IDF library file. This power value is read and set by EDA Bridge. This power is for each library item, not for the unique instance of that component on the board.

11. Q: How can I import two resistor component information from a spreadsheet into EDA Bridge?

A: A CSV file can be imported using **File > Import > Import CSV Layout**.

To determine the format of the file to import, first create a single component and define it as a two resistor component type. Then, use **File > Export > Export CSV** to export it as a CSV file and view the output in a text editor, noting the column heading format. Populate the CSV file to be imported using the two resistor information you have available then re-import the file.

12. Q: Why can't I move or resize components using the mouse?

A: Components can only be moved or resized using the mouse if the **Lock Component Move** icon is off. This icon is designed to prevent accidental dragging of components.

Lock is indicated by a closed lock, 

Unlocked is indicated by an open lock, 

Click the icon to toggle states.

Layer Definition FAQs

Answers to questions about layers.

1. Q: How can I see patches on internal layers?

A: To view patches on internal layers, not visible from the top or bottom views of the board, select the layer node in the data tree. Layer patches belonging to the other unselected layers, as well as the Motherboard, are hidden from view; all other objects remain in view.

Solution FAQs

Answers to questions about solving.

1. Q: How do I model a board with the heat assumed spread over the board surface?

A: Set all components to be Filtered. This is most effectively done using the [Component Filter Options Dialog Box](#). Alternatively, if no components are specified, by creating one or more Functional Groups with powers defined results in the total amount of heat being smeared over the board surface (see “[Functional Groups](#)” on page 118). Both of these methods will result in average board temperature predictions.

If you specify a size of a functional group any power defined for that functional group will be smeared locally over that portion of the board surface.

2. Q: How do I accelerate my solution without compromising the accuracy of my results?

A: Thermally insignificant components can be Filtered. Filtered components are not deleted from the model. Their power dissipation is summed and modeled as smeared over the entire board surface, their physical obstructions are not resolved. Components that are small (for example, <2 mm in length) AND have a very low power dissipation (for example, <0.1 W) should be considered insignificant. This filtering can be done either individually in the component property sheet or globally by using the [Component Filter Options Dialog Box](#). Alternatively, component Power Density can be used to filter components in the Filter Component dialog box.

Related Topics

[Component Filter Options Dialog Box](#)

Transfer From EDA Bridge to Simcenter Flotherm FAQs

Answers to questions about transfers to Simcenter Flotherm.

1. Q: When I transfer a board to Simcenter Flotherm I see dimmed regions with localized grid. What are they?

A: These are regions added when Auto-Grid is switched on (default behavior), see “[MotherBoard Property Sheet](#)” on page 129.

Index

— Numerics —

2D or 3D Board Views, [20](#)

— A —

Accurate Thermal Solution, [101](#)

Adding Geometry, [48](#)

Aligning Geometry, [56](#)

Axis, [43](#)

 font increase, [36](#)

— C —

Components

 Component Filter Options dialog box, [96](#)

 Component/Functional Group Search
 dialog box, [89](#)

 filtering, [71](#)

 importing, [61](#)

 moving precisely, [88](#)

 tables of, [19](#), [58](#)

Context sensitive menus, [24](#)

Coordinate System, [20](#), [43](#)

Create New Libraries dialog box, [99](#)

CSV Files, [82](#), [83](#)

— D —

Data Tree, [18](#)

Dragging, [20](#), [53](#)

 horizontal and vertical constraint, [53](#)

— E —

Editing

 Edit Text dialog box, [92](#)

 Geometry Details, [53](#)

Exporting Boards, [69](#)

— F —

Filtering Components, [71](#)

Find

 Library Search Facility, [100](#)

FLOEDA Files

 exporting placement, [69](#)

FloSCRIPT

 GUI usage, [28](#)

Font Increment

 in legend, [36](#)

Font Increment

 axis in results display, [36](#)

— G —

Geometry

 adding, [48](#)

Global Units

 Global Units dialog box, [34](#)

GUI Preferences

 GUI Preferences dialog box, [35](#)

— H —

History

 incremental versioning of projects, [39](#)

Horizontal Movement Constraint, [53](#)

Hover text

 turn on/off, [35](#)

— I —

IDF

 small edge tolerance, [36](#)

Import Power List Report

 generation after CSV import, [35](#)

Importing Designs, [61](#)

Incremental Revision, [39](#)

— L —

Layers

 layer patches, [109](#)

Legend

 font increment, [36](#)

 height, [36](#)

 number of ticks, [36](#)

 width, [36](#)

Libraries

 library file, [25](#)

 Library Search Facility, [100](#)

-
- Library Selector dialog box, 98
Library Selector for Component Import dialog box, 93
menus, 24
populating, 78
Load Library Item, 25
- M**
- Match Case (finding objects), 89
Materials
 Material Library Selector, 90
Menus
 context sensitive, 24
Mouse Controls for 3D Viewing
 defaults, 20
 preferences, 38
Move
 geometry, 53
Move Selected Items dialog box, 88
- N**
- New Geometry, 48
Notes, 61
Nudging, 53
- P**
- Pattern
 Pattern Selected Items dialog box, 87
PDML File
 importing, 25
Power
 functional groups, 119
 power list import, 66
Projects, 27
 version increment, 31
Property Sheets, 21
- R**
- Reference Designators
 definition, 43
 importing designs, 65
 unrestricted names, 36
Refresh Libraries, 25
- S**
- Saving
 project, 31
- Save Model dialog box, 39
Search
 for components/functional groups, 89
 Library Search Facility, 100
Select Columns Types dialog box, 94
Small Edge Tolerance
 IDF, 36
Snap Grid
 size, 35
Solution Configuration
 Thermal Solution dialog box, 101
Solving
 thermal solution dialog box, 101
Sorting Component Tables, 19, 58
- T**
- T3Ster files
 importing, 77
Thermal Trend Solution, 101
- U**
- Unlock Component Movement, 160
- V**
- Vertical Movement Constraint, 53
View Board in 2D or 3D, 20
View Existing Material dialog box, 91
Viewing Tools
 View 3D Controls dialog box, 38