FloTHERM® PCB User Guide

Software Version 6.2

Contractor/manufacturer is:
Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.
Telephone: 503.685.7000
Toll-Free Telephone: 800.592.2210
Website: www.mentor.com
SupportNet: supportnet.mentor.com/
Send Feedback on Documentation: supportnet.mentor.com/user/feedback_form.cfm

Table of Contents

List of Figures

List of Tables

Chapter 1
Working With FloTHERM PCB

Introduction to FloTHERM PCB

Using FloTHERM PCB you can design PCBs and model their thermal behavior. FloTHERM PCB is a speedy PCB design interface for non-thermal experts, allowing the quick construction of comparative studies during the design process.

FloTHERM PCB is a unique, new software program for streamlining concept development of printed-circuit boards (PCBs), whilst ensuring good thermal design and accelerating the PCB design process.

FloTHERM PCB facilitates collaboration between product marketing, electronic engineers, and mechanical engineers on PCB design, particularly during the conceptual phase of the design process.

FloTHERM PCB promotes a conceptual design process that is derived from the functional block diagram. Changes made to the functional block diagram are instantly reflected in the physical layout and thermal representations. This keeps all team members in sync and enables them to contribute to concept development in real time. The result is pre-optimized concepts in less time and drastic reductions in late-cycle rework as product marketing, mechanical, thermal and manufacturing issues are solved before concept commit.

FloTHERM PCB Saves Money

FloTHERM PCB saves your company money by addressing major inefficiencies in the board design process. On top of this, FloTHERM PCB minimizes the risk of board “re-spins” due to thermal problems. In a recent survey, 60% of mechanical engineers in electronics companies stated that thermal issues had forced board layout changes during the previous 12 months. Just one such re-spin costs many times the FloTHERM PCB license fee.

FloTHERM PCB Users

FloTHERM PCB is designed to be used by all those involved in the conceptual design of PCBs, including Product Marketing, Systems Architects, Hardware Designers, and Mechanical/Thermal Engineers.

Product Marketing

Interact with all members of the design team rapidly and effectively, viewing and commenting on design concept changes in real time.

Systems Architects

Stay in control of the conceptual design process with FloTHERM PCB. Create a functional block diagram rapidly, using software menus developed specifically for this task. Communicate instantly with product marketing, hardware design and mechanical/ thermal engineers through graphical outputs and automatically-generated reports. One mouse click flips between functional block diagram, physical layout, and thermal performance views.

If thermal questions arise, you can instantly transfer models to your colleagues for more in-depth analysis. Receive feedback more quickly and accurately than ever before as your colleagues work alongside you within the FloTHERM PCB design environment.

Mechanical and Thermal Engineers

With FloTHERM PCB, you finally have the software you need to provide critical feedback on thermo-mechanical issues in time to influence conceptual board design and layout. You will receive design updates from electrical engineering and product marketing in real time. When concept design is complete, you already have design information in a format that allows you to instantly start your detailed mechanical design in your MCAD software, and your thermal design in FloTHERM

Hardware Design Engineers

With FloTHERM PCB, you can influence concept development and find the right balance between concept and detailed design and manufacturing. At “concept commit” stage, you can export the final design information from FloTHERM PCB directly into your mainstream EDA software.

The Modeling Process

The main stages to creating a FloTHERM PCB study are:

1. Opening a FloTHERM PCB project

   Studies are started by either creating a new project or loading an existing one. “Managing Projects” on page 13

2. Optionally, Defining a Functional Layout

   A logical representation of the functional partitioning and interconnection of the design can be built before defining the physical aspects. See “Defining a Functional Layout” on page 21.

3. Designing the Board Layout

   Boards are designed by adding components to a base MotherBoard. The board and components are manipulated using a Data Tree and Graphics Display Area. “Defining the Board Layout” on page 31

4. Applying the Board Environment

   Board environments are applied either by entering the data manually or copying from an already solved FloTHERM case. “Defining the Environment” on page 65

5. Solving for the thermal effects

   FloTHERM PCB provides trend prediction or accurate temperature prediction solutions. “Solving and Viewing the Results” on page 79

6. Displaying the Solution Results

   Results are displayed in tabular and graphical formats. “Solving and Viewing the Results” on page 79

The Resultant Design

The resultant design of the FloTHERM PCB study can be saved as a FloTHERM PCB project and imported into FloTHERM for a full system analysis. Alternatively, it can be exported as an IDF file for use in EDA application software.

Reports and Screen Dumps

Reports of the project setup can be generated in HTML and screen dumps of the Display Area can be output in PNG format. See “Generating Reports and Screen Dumps” on page 95 Also, board data can be exported in comma separated variable file format for use in spreadsheets. See “Exporting Designs” on page 52.

Chapter 2
Managing Projects

This chapter provides an overview of how to manage FloTHERM PCB projects.

Overview of a FloTHERM PCB Project

Each board design study is encapsulated within a unique FloTHERM PCB project. All data relating to the study is stored in a single **.flopcb* project file created by FloTHERM PCB. This **.flopcb* file may be stored anywhere on your file system.

Working With Project Files

To work on FloTHERM PCB 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 might need to view the results again because if you alter the model after solution, the results are detached from the project. To restore the results, you must use **Edit > Undo Move (Recover Results)** to undo the edit.

Related Topics

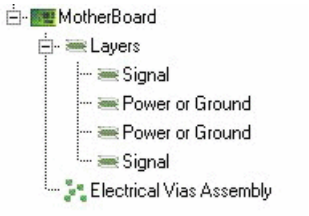
- “Starting a New Project” on page 13
- “Saving a Project” on page 18
- “Loading an Existing Project” on page 14

Starting a New Project

To start a new project, select **File > New** from the menu bar. If FloTHERM PCB detects any unsaved changes, then you will be asked if you want to save them or not before the new session is started.

File > New initializes the board design to a single MotherBoard comprising four layers (Figure 2-1), set in an a Single Card Slot environment with an ambient air temperature of 45˚C and speed of 2 m/s.

Figure 2-1. New Project Data Tree



Related Topics

- "Loading an Existing Project" on page 14

Loading an Existing Project

To load an existing project, select **File > Load** from the menu bar and choose the required project from the file browser. FloTHERM PCB projects have the *.flopcb* file extension.

The *flouser* directory in the *flopcb_v*6.2 directory structure is set aside as the default location for your projects, but **.flopcb* files may be stored and loaded anywhere on your file system.

Related Topics

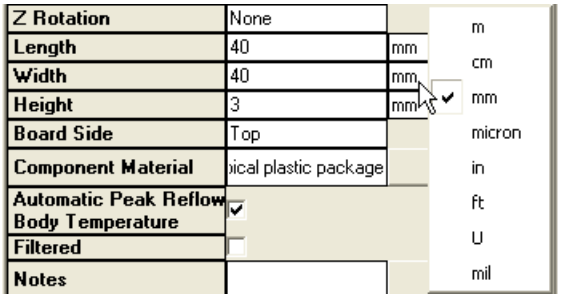
- "Managing Projects" on page 13

Choosing the Project Units

The default unit types used by the FloTHERM PCB and dialogs are chosen using the Global Units Dialog**.** However, these unit type selections can be overridden locally by changing the unit options in the Property Sheets.

For example, Figure 2-2 shows the options available for the width measurement.

Figure 2-2. Choosing Units



Related Topics

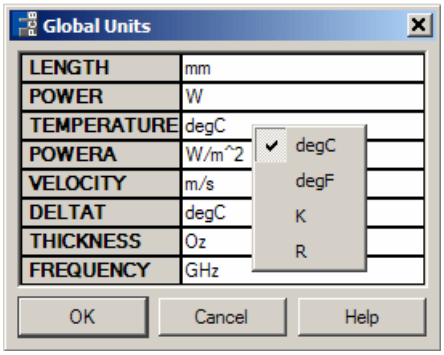
- "Using Property Sheets" on page 42

Global Units Dialog

To access: **Options > Global Units**

Use the **Global Units** dialog to set the default units in the window and dialogs. The dialog box lists the variable types and the default units. To change the default units, click the unit field and choose the unit type from the popup menu, as shown for temperature in Figure 2-3.

Figure 2-3. Global Units Dialog



Click **OK** to apply the setting.

Local Overwrite

The default units can be changed locally using the Property Sheet dimension popups. For example, the global setting for length defaults to millimeters, but you may be entering an object with a specification in inches. Just reset the Properties Sheet input field units by selection from the popup menu prior to entering the data. This unit will be remembered for that object in preference to any unit set in the **Global Units** dialog.

Related Topics

- "Using Property Sheets" on page 42

Setting GUI Preferences

FloTHERM PCB lets you set preferences for the appearance of the GUI using the GUI Preferences Dialog.

GUI Preferences Dialog

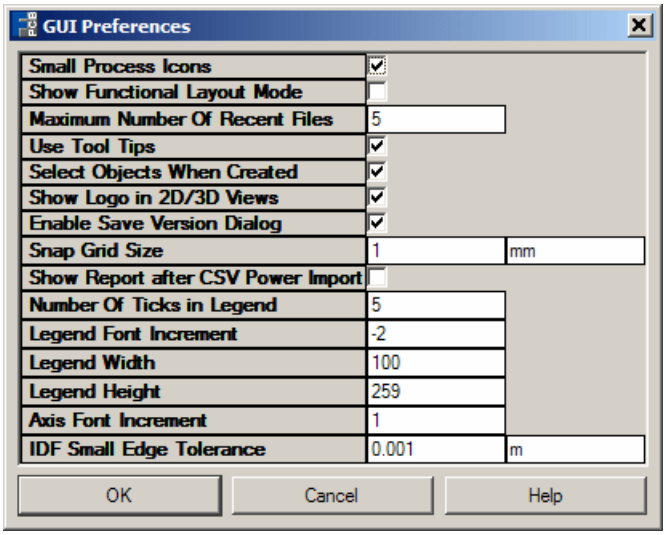
To Access: **Options > GUI Preferences**

Use the **GUI Preferences** dialog **(**Figure 2-4**)** to customize the appearance of the FloTHERM PCB window.

The dialog box consists of a data sheet showing the current preference settings. To change a preference either check/uncheck the option check box or overtype a data field as appropriate, then click **OK**.

The preferences chosen are remembered (and will be retained until you change them) for future sessions of FloTHERM PCB.

Figure 2-4. GUI Preferences Dialog



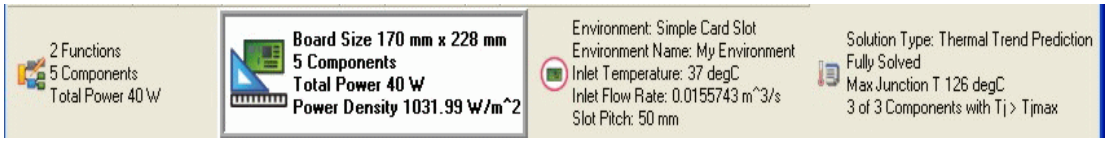
Elements

- **Small Process Icons**

  By default, the S**mall Process Icons** are checked. This reduces the size of the Process Bar towards the top of the window and provides more work space, but you lose the quick summary information as shown in the following figures.

Figure 2-5. Small Process Icons Checked

Figure 2-6. Small Process Icons Unchecked



Clear the **Small Process Icons** check box if you want to display the summary data.

Note

The summary information can also be seen in the ToolTips for the mode status buttons.

- **Show Functional Layout Mode**

  By default this box is unchecked, hiding the Functional Layout button. If you want to create a proposed design using functional connections then check this box to show the Functional Layout button.

- **Maximum Number of Recent Files**

  Sets the number of last opened or saved documents that will be listed in the **File menu** above the **Quit** command.

  Selecting a project name in the File menu will open the project.

- **Use Tool Tips**

  Turn off/on the display of ToolTips.

- **Select Objects When Created**

  Sets new objects selected ready for immediate editing.

- **Show Logo in 2D / 3D Views**

  Show/Hide the FloTHERM PCB logo in the graphics display area.

- **Enable Save Version Dialog**

  With this option selected, selecting **File > Save** launches the "Save Model Dialog" on page 19, so that you can enable the auto-increment of the project file name to prevent the overwriting of the original project. A sequence of model changes can then be easily set up to aid project traceability.

- **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**

  Allows the generation of an Import Power List Report when importing a power list. See"Importing a Power List" on page 50 and "Import Power List Report" on page 50.

- **Number of Ticks in Legend**

  Sets the number of intervals in the results legend displayed in the Results Analysis mode. See "Results Analysis" in *Getting Started with FloTHERM PCB*.

- **Legend Font Increment**

  Sets the scale increase or decrease in size for the legend text displayed in the Results Analysis mode. See "Results Analysis" in *Getting Started with FloTHERM PCB*.

- **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 displayed in the Results Analysis mode. See "Results Analysis" in the *Getting Started with FloTHERM PCB*.

- **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 will be removed.

Setting View Options

The user preferences for mouse buttons controlling the 3D view can be set using the **View 3D Controls** called by choosing **View Controls** from the **Options** menu.

View 3D Controls Dialog

To access: **Options > View Control**

Use the **View 3D Control** dialog to choose which mouse buttons control the 3D view of the board geometry.

The **View 3D Control** dialog consists of three separate property sheets for choosing the **Pan**, **Zoom** and **Rotate** control buttons used when viewing the geometry in the Board Creation, Environment Definition and Results Analysis modes.

To choose a control, first click the function button to display the button check list, then checking the option box to choose the mouse button.

Related Topics

- 3D View
- Display Area

Saving a Project

To save a new project or save an existing project under a new name, select **File > Save As** from the menu bar and enter the file name and select the destination using the file browser. File names of up to 31 characters are allowed.

File > Save for a new project is equivalent to **File > Save As**.

If a FloTHERM PCB project has been saved, then a futher **File > Save** will launch the Save Model Dialog to allow the addition of model notes and provide an option for creating another version of the project by incrementing the filename. This version incrementing allows 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 stops the dialog from appearing and **File > Save** overwrites the current project. The Save Model dialog can be reactivated by checking on the **Enable Save Version Dialog** option in the "GUI Preferences Dialog" on page 15.

Save Model Dialog

To access: **File > Save** when a project has already been saved, provided the dialog has not already been switched off (**Do not show this again** switch) and provided the **Enable Save Version Dialog** is set in the GUI Preferences Dialog.

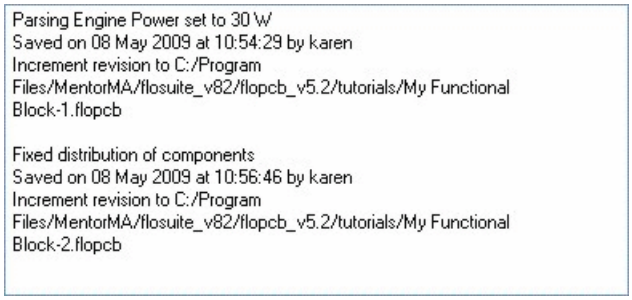
Use this dialog to overwrite or version increment an existing project and add modeling notes.

The dialog consists of an editable text panel containing the project notes and any incremental history.

- **Model Notes**

  The first time the **Model Notes** panel appears for a project, it is displayed blank and acts as a simple text editor, ready to accept your comments for this save (incremental or otherwise). After checking **Incremental Revision** and clicking **Save**, the date of the incremental save, who saved it and the filename the project was saved under, along with any comments added, will be recorded and displayed the next time the is opened. Figure 2-7 shows an example of the history layout.

Figure 2-7. Save History



The 'Parsing Engine Power set to 30W' and 'Fixed distribution of components' are examples of manual text entries added by the user, the rest was generated by the program for project 'My Functional Block Diagram'.

- **File Name**

  A read-only field indicating the full pathname of the project. If you need to change the path or file name, then use **File > Save As** instead.

- **Incremental Revision**

  Increments or appends a revision number to the project file name to create a new project. The new project name will be in the form:

  projectname-n.flopcb

  where:

  projectname is the name of the original loaded project

  -n is the revision number.

  The first time a project is saved, the revision number 1 is appended to the file name. For subsequent **Incremental Revision** saves, the appended number is incremented.

  The timestamp for the incremental revision will appear in the project notes the next time the next time the is opened.

  If **Incremental Revision** is not checked, then clicking **Save** overwrites the current project. Any comments typed in the Model Notes section will be stored with the project.

  Do not show this again stops the dialog from appearing each time the project is saved. The dialog can be reactivated by checking the **Enable Save Version Dialog** option in the GUI Preferences Dialog.

- **Reporting the Model Notes**

  The model notes and project history listed here in the **Save Model** can also be included in the project reports generated using the **Tools > Generate Report** menu. See “Displaying Reports” on page 95.

  To include the model notes and project history, select **Tools > Generate Report > Configure Reports**, then in the **Report Configuration,** select the report required to contain the history and check the **Model Notes** option. The next time the report is generated, the model notes will be included near the top, in the Summary section.

  The report can be saved as an html file and loaded into a web browser from where it can be printed.

Chapter 3
Defining a Functional Layout

A functional block diagram graphically depicts the major functions and interconnections between the functions of an electronic product. They are used to communicate the functional intent of a product to a wide range of people involved in the development such as, marketing, electrical and mechanical engineers, as well as customers.

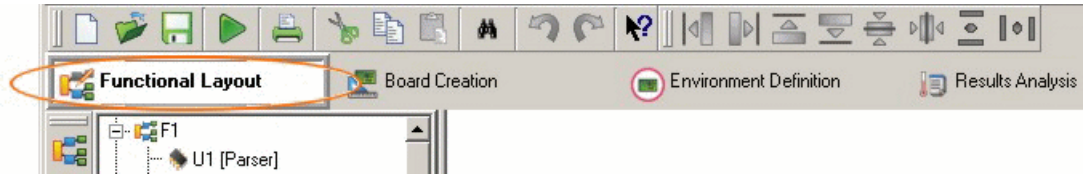
This chapter describes how to define a functional block diagram using the FloTHERM PCB Functional Layout.

Viewing the Functional Layout

The Functional Layout button is hidden from view by default. To show the Functional Layout button, select **Options > GUI Preferences** and check the **Show Functional Layout Mode** box, see "Setting GUI Preferences" on page 15.

To view the functional layout, click the **Functional Layout** button (Figure 3-1).

Figure 3-1. Functional Layout Button

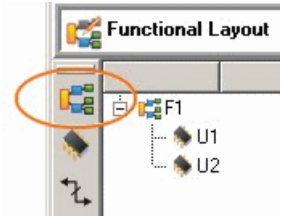


For a description of the window layout, see Functional Layout in *Getting Started with FloTHERM PCB*.

Defining a Functional Group

A functional group can be created by clicking on the functional group icon on the left toolbar:

Figure 3-2. Functional Group Icon

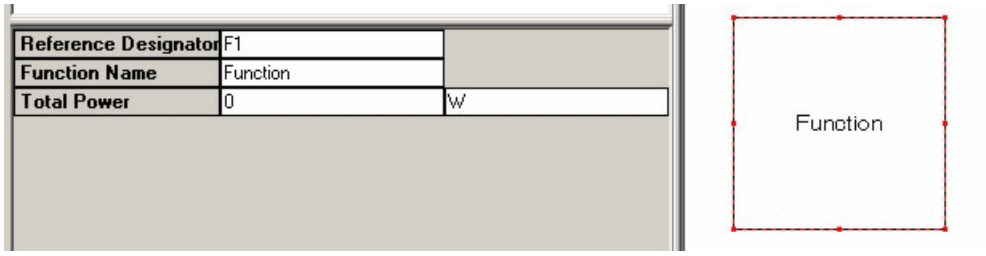


Alternatively, you can create functional groups to enclose existing components, see "Defining a Component" on page 23.

The square wireframe representation of the functional group can be moved and resized in the drawing area.

The functional group has a name, a reference designator and a total power value. These can be changed at any time by first selecting the functional group then editing the information in the property sheet.

Figure 3-3. Functional Group in Property Sheet



By default, functional groups are empty and have no assigned power.

When a **Total Power** value is entered, this value is annotated in the display area along with a summary of how much of this power is currently assigned to the individual components of the functional group, in the form:

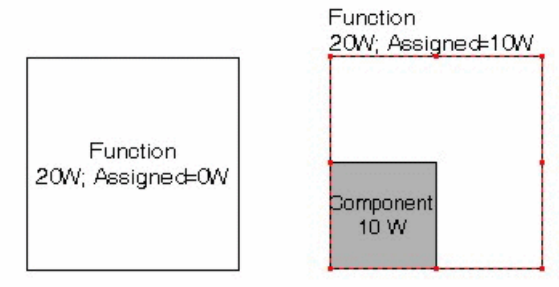
Total Power; Summary of Assigned Power.

If the **Total Power** is greater than the sum of the individual component powers, then this surplus is applied to the board. If the size of the functional group is not specified, that is, the **Size by Bounding Box** is checked on in the property sheet, then the surplus power is smeared over the entire surface of the board. If the functional group **Size by Bounding Box** is checked off and its size is specified, then the surplus power is smeared over the portion of the board occupied by the functional group.

If the **Total Power** is less than the sum of the powers attached to the components, then the **Total Power** setting is ignored.

Figure 3-4 shows the annotations before and after a component is added to the functional group.

Figure 3-4. Before and After Annotations



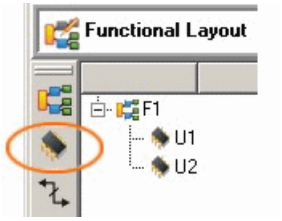
Related Topics

- Functional Block Diagram
- "Converting Into a Physical Layout" on page 29

Defining a Component

A component can be created either as an individual object or as a child of a functional group.

Figure 3-5. Create Component Icon



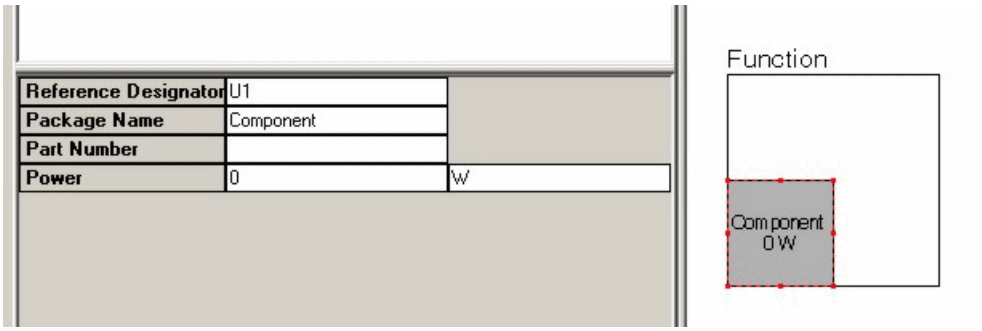
If the component is to be added as a child of an existing functional group, then first select the functional group before clicking the Create new component icon.

If the component is to be added to a new functional group, then after creating the component by clicking the Create new component icon, choose **Add to new Functional Group** from the component popup menu.

The solid square representation of the component can be moved and resized in the drawing area. The component can even be nudged into position using the cursor arrow keys. If the component is a child of a functional group then the functional group will automatically resize so that the component is always drawn within it.

The component has a reference designator, package name, part number and power value. These can be changed at any time by first selecting the component then editing the information in the property sheet.

Figure 3-6. Component in Property Sheet



By default a component has no power assigned.

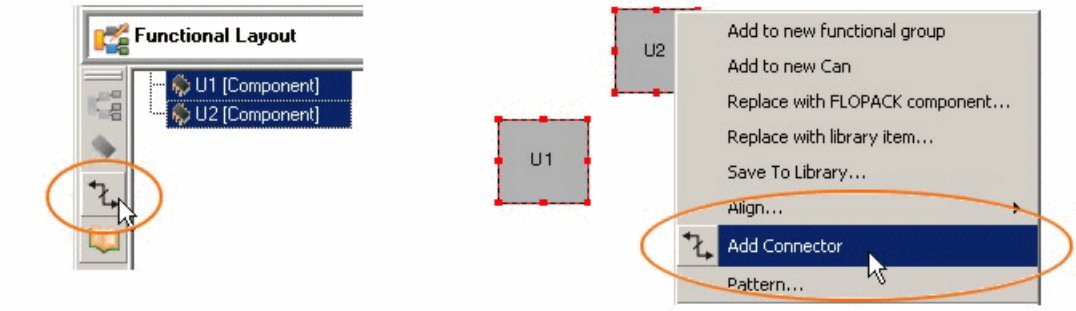
Related Topics

- Functional Block Diagram
- “Converting Into a Physical Layout” on page 29

Defining an Interconnect

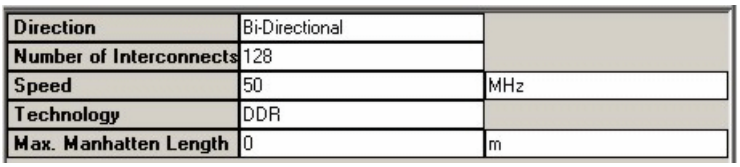
A connection can be drawn between two selected objects. The objects can be any combination of functional group, component or connector. To create an interconnect, select the two objects then either click the Creata new connector icon or select **Add Connector** from the component menu.

Figure 3-7. Use Interconnect Icon or Menu Item



Once the connector is created it should be selected to enable the following information to be edited in the property sheet:

Figure 3-8. Interconnects in Property Sheet



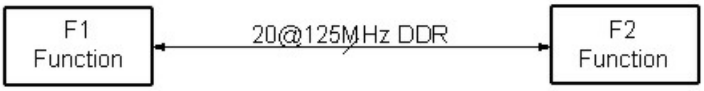
Direction: Bi-Directional / Output / Input — determines which ends of the connector have arrows that define the direction of connectivity between the selected objects.

Number of Interconnects, Speed, Technology — determines the annotation that is automatically associated with the connector in the form:

<Number of interconnects>@<Speed> <Technology>

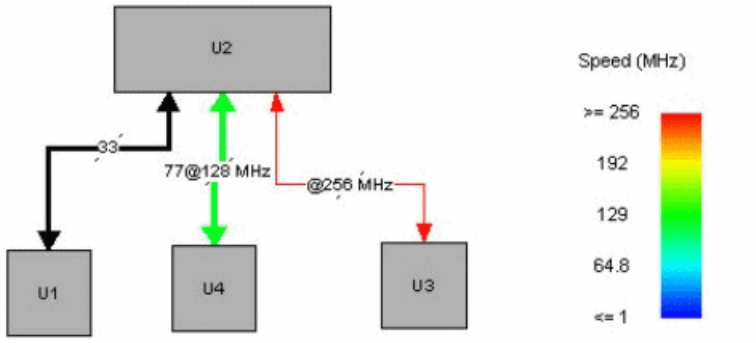
For example:

Figure 3-9. Interconnect With Annotation



The speed and number of interconnects is indicated in the display in Figure 3-10. The connector color indicates the speed, and its thickness shows the number of interconnects.

Figure 3-10. Color and Thickness Indicate Properties

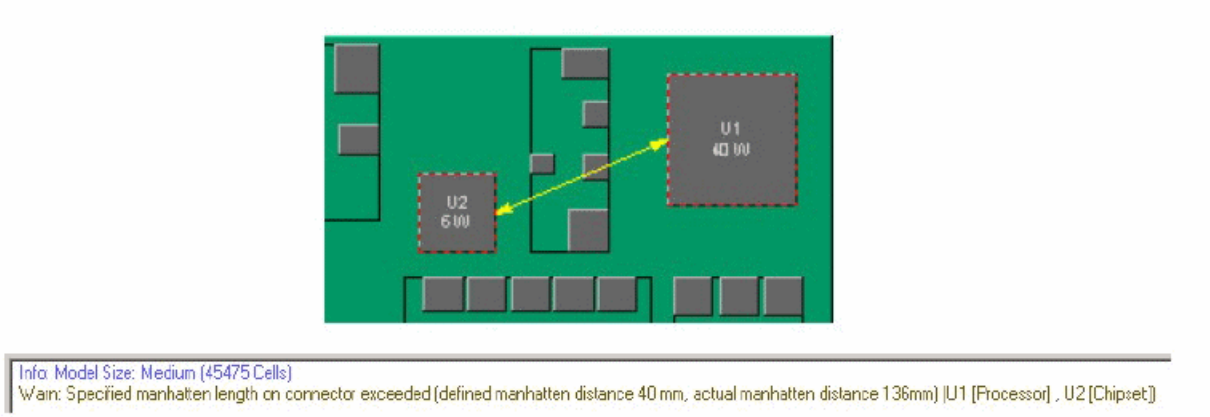


The line thicknesses can be changed using **Options > Connector Thickness**. See "Customizing the Display of Connectors" on page 28.

Maximum Manhattan Length

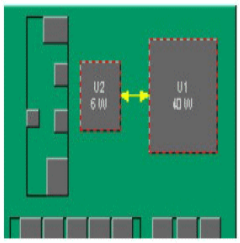
This defines a length constraint for a connector connecting two components. If the distance calculated from the object centers exceeds the **Maximum Manhattan Distance** then a warning will be issued on a **File > Model Check**.

Figure 3-11. Checking Manhattan Length



Clicking on the warning will select the offending components; they can be moved closer together to resolve the constraint. This should be done in the Board Layout mode, where the physical view of the design is displayed. See “Defining the Board Layout” on page 31.

Figure 3-12. Manhattan Distance Fixed



Related Topics

- Functional Block Diagram

Tabulating Connector Information

The connection type, number of interconnections, interconnect speed and technology type for all the connectors can also be displayed and edited using the “Connector Table” in *Getting Started with FloTHERM PCB*.

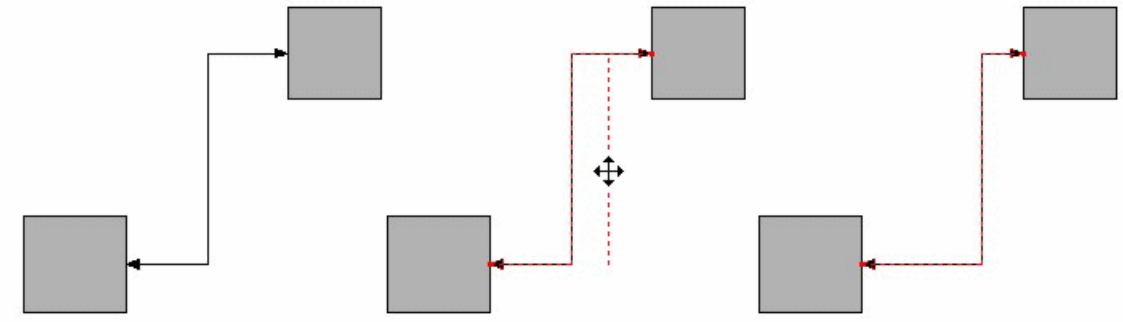
If the Connector Table is not on view in Layout mode, click the **Connector Table** tab in the bottom left panel.

Manipulating the Connectors and Their Annotations In the Display

Moving Connector Lines

Connector lines can be moved by click-dragging from anywhere on the line. There are only grab handles at the end of the lines to enable the ends to be moved to different faces.

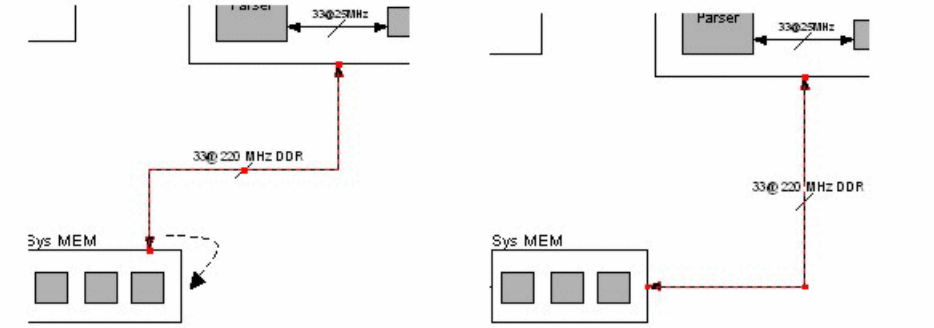
Figure 3-13. Moving Connector Lines



Dog Leg Connector on Side Move

Dog Leg connectors will automatically be created when moving a connector end from one side of an object to another.

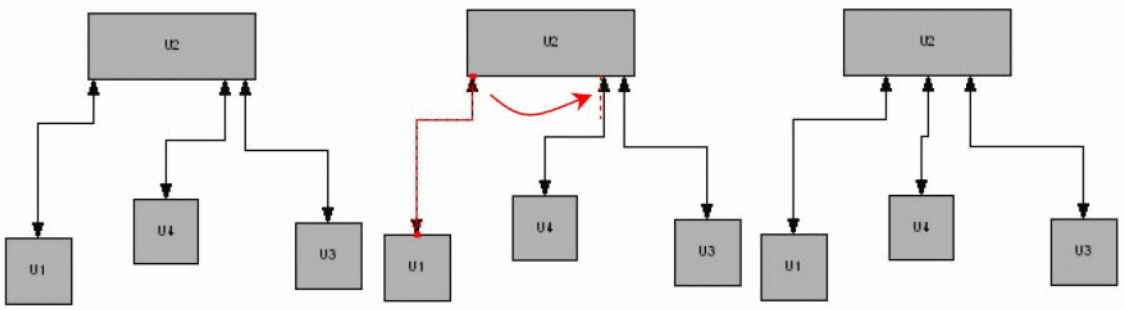
Figure 3-14. Dog Leg Connector Creation



Equispacing of Multiple Edge Connectors

Connectors are equispaced along an edge of an object when they are created. Existing connectors can be equispaced by dragging one connector end on top of another. The connectors will then be automatically repositioned so they are equispaced as in Figure 3-15.

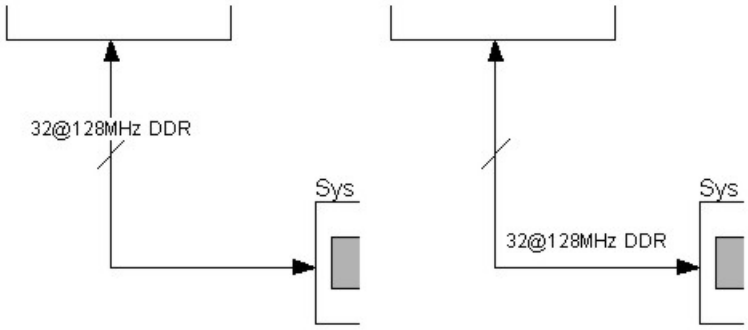
Figure 3-15. Automatically Equispacing Connectors



Moving Connector Annotations

If the connector annotations overlap areas of the functional block diagram, they can be moved to unclutter the display. To move a connector annotation, select and then drag it using the mouse.

Figure 3-16. Connector Annotation Moved



Customizing the Display of Connectors

If an image capture of the functional block diagram is to be imported into a report, then the size of the interconnector lines can be changed to match the documentation standards of the report.

To change the thickness of the connector lines, choose **Options > Connector Thickness** and change the settings in the Connector Line Thickness Dialog.

Connector Line Thickness Dialog

To access: **Options > Connector Thickness**

Use the **Connector Line Thickness** to change the display width of the connector lines.

The connector lines can be represented in four different widths. By default, lines representing:

- less than 8 interconnectors are drawn 1 pixel wide
- less than 33 interconnectors are drawn 2 pixels wide
- less than 73 interconnectors are drawn 3 pixels wide.
- greater than or equal to 73 interconnectors are drawn 4 pixels wide.

To change the width of the connector lines, change the number of interconnects each line width represents.

Related Topics

- "Defining a Component" on page 23
- "Defining a Functional Group" on page 21

Converting Into a Physical Layout

Once the functional layout has been defined the user can then go to the **board layout** mode on the **process bar** to define the physical aspects of the board design.

Any component defined in the functional layout view is represented as a (Default) single physical component. Any functional group defined in the functional layout is represented as a functional group in the physical layout.

Powers that have been defined for Components in the functional layout view are carried over to the physical layout description. Powers defined for Functional Groups in the functional layout are also carried over to the physical layout description. If there are powered components in a powered functional group then the balance of heat (that is, that which is not as yet allocated to discrete physical components) is 'smeared' over the entire board surface. If there are no components in a powered functional group then the heat is smeared over the entire board surface to allow for average board temperature prediction.

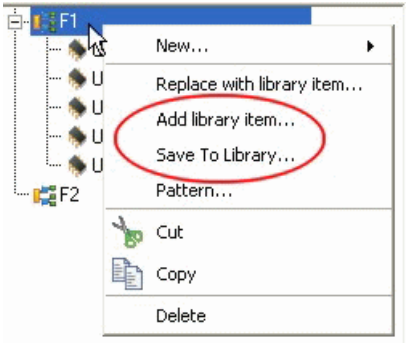
Related Topics

- Functional Block Diagram

Saving and Retrieving a Functional Group From the Library

A functional group can be saved to or retrieved from a library at any stage, using the functional group popup menu as shown below.

Figure 3-17. Functional Group Popup Menu



A functional group can also be added to the current Functional Layout by opening the Library pane then double clicking on the required item (see "Using the Library Pane" on page 37).

Both Functional Layout and **Physical Layout** definitions of the functional group are saved to, and retrieved from, the library. This results in a full integration between the functional and physical definitions of the board design.

Chapter 4
Defining the Board Layout

This chapter describes how to display and build the representation of the physical board layout.

Viewing the Board Layout

The board layout is displayed on start-up. At other times click the **Board Creation** button in the **Process Bar** to display the board layout.

See "Board Creation" in *Getting Started with FloTHERM PCB* for a full description of the window layout.

The board layout is displayed in two main viewing formats: a data view and a graphical view.

Data View

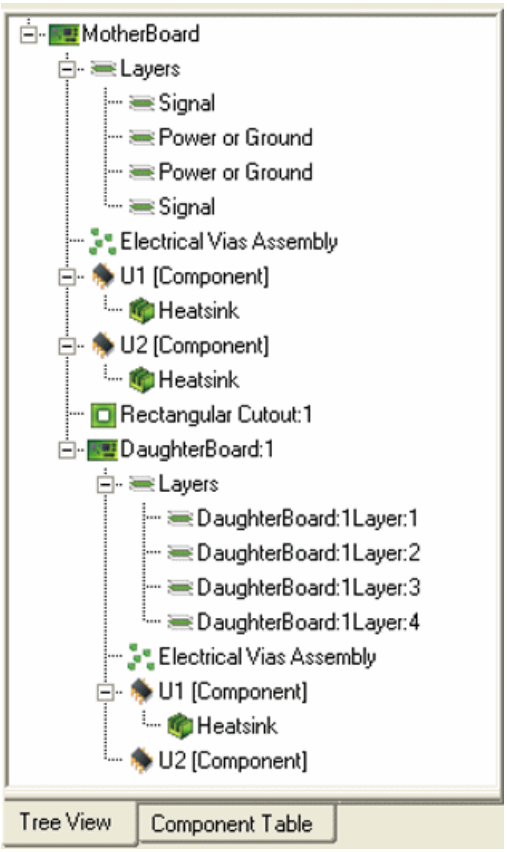
The board layout data view can be swapped between a data tree structure and (providing components are present) a table of component details.

Viewing the Data Tree

On start up, or when the **Tree View** tab is clicked, the project data tree is visible.

To see the full structure of the board, expand the nodes of the data tree (by clicking '+'). For example, Figure 4-1 shows the tree view of a MotherBoard with a DaughterBoard attached. Both boards have four layers, holding node for electrical vias and two components. Three of the components have heatsinks. The MotherBoard also has a rectangular cutout.

Figure 4-1. Tree View of Board Data



Note that there are only two levels of boards. **DaughterBoards** cannot have **DaughterBoards** of their own.

Viewing the Component Table

To display the component details in a table, click the **Component Table** tab. The component table contains one row of data for each component in the project. The data format is shown in Figure 4-2.

Figure 4-2. Component Details Table

Reference Designat	Package Name	Component Type	Power (W)	X Location (cm)	Y Location (cm)	Board Side	Length (cm)	Width (cm)
U1	Component	2 Resistor	0	33.15	2.19	Top	4	4
U1	Component	2 Resistor	0	5.12	0.49	Top	4	4
U2	Component	Simple	0	27.18	2.07	Top	4	4
U2	Component	Simple	0	0.49	0.49	Top	4	4

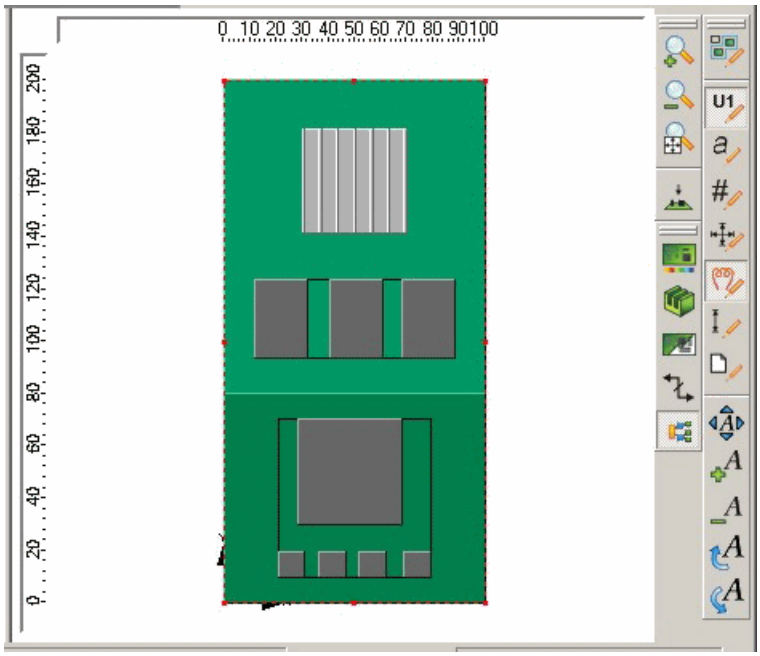
Tree View | Component Table

Clicking a column header sorts the table according to the contents of that column.

Viewing the Board and Components

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

Figure 4-3. 2D Board View



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.

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. To change the view, click the View From icon in the Viewing Options toolbar and choose the view from the drop-down menu.

Note that, when rotating the board, the origin and axes directions rotate with the board.

Coloring the Components

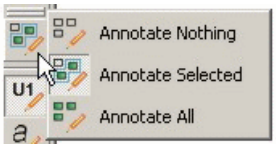
The board components can be colored according to modeling level, power, height or just represented as wireframe. To set the component coloring, click the Color Components icon in the Advanced Display Options toolbar and choose the representation from the popup menu.

Annotating Components

The components can be annotated to show their name, size, power, height and construction notes, using the right side toolbar. To annotate components, first choose the object selection type and then select the information to be displayed as follows:

1. Choose the type of components to be selected for annotation by choosing one of the following options from the Annotate dropdown list:

Figure 4-4. Annotate Dropdown List



2. Choose the information to be displayed by selecting one of the following icons:
   - Annotate With Reference Designator
   - Annotate With Package Name
   - Annotate With Part Number
   - Annotate With Size
   - Annotate With Power
   - Annotate With Height
   - Annotate With Notes

The annotation font size for all the components can be automatically adjusted to match the components they refer to by clicking the Auto Size Font icon. Alternatively, use the Increase Font and Decrease Font icons to increase and decrease the font size for selected components.

Locking Component Movement

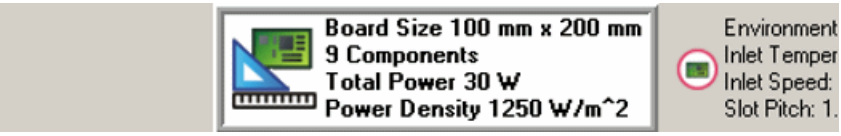
Components can be quickly moved by dragging with the mouse, see “Moving and Resizing Objects by Dragging Boundaries” on page 40.

To prevent accidental movement when selecting components only to display information, click the Lock Component Move icon.

Board Layout Summary

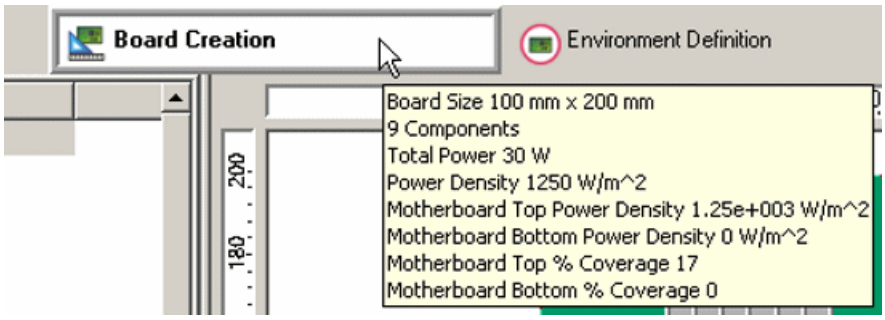
A quick summary of the board layout can be displayed in the process bar or the ToolTip. If you deactivate the Small Process Icons option in the “GUI Preferences Dialog” on page 15 then the Board Creation button will be expanded to provide a brief summary of the board layout.

Figure 4-5. Board Creation Button With Summary



Alternatively, if the **Use Tool Tips** option is activated in the **GUI Preferences**, then the board summary can be displayed in a ToolTip when the mouse pointer hovers over the Board Creation button.

Figure 4-6. Tool Tip for Board Creation Button



Adding Geometry

In FloTHERM PCB you can add geometry to MotherBoards, DaughterBoards, and rectangular components. However, the geometry types that you can add vary according to the object you have selected to receive the object. For example:

- **MotherBoards** can have additional DaughterBoards, rectangular components, FloTHERM PACK components, cylindrical components, functional groups, cutouts, cans, placement keepout areas and potting compounds.
- **DaughterBoards**, as for Motherboard, except additional daughterboards are not available.
- **Rectangular components** can have additional heatsinks and thermal vias.

To add geometry, you can either create new geometry from scratch (see Adding New Geometry Using the Toolbar and Adding Geometry Using the Popup Menus) or load existing geometry from the library (see Adding Existing Geometry Using the Library).

Adding New Geometry Using the Toolbar

To add new geometry using the toolbar, first select the object to receive the new object either in the data tree or the drawing area, then click a geometry icon in the left side toolbar. The new item is then added to the data tree and appears as the top item in the display area. The default properties of the new object can be changed using the drawing area and the properties sheets, see “Editing Geometry” on page 40.

The geometry types available for the selected tree item are indicated by the active items in the new geometry toolbar.

The possible additions are indicated in Table 4-1

Table 4-1. Geometry Options by Tree Type

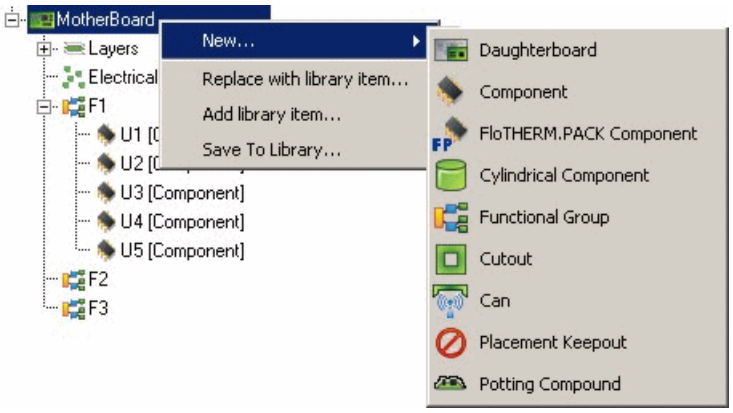
Tree Selection	Geometry Options
MotherBoard	DaughterBoard, rectangular component, FloTHERM PACK component, cylindrical component, functional group, cutout, can, potting compound and board placement keepout region. All new objects are located at the origin of the MotherBoard.
DaughterBoard	Rectangular component, FloTHERM PACK 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, FloTHERM PACK component and cylindrical component.
Layer	Layer patch and layer.
Electrical Vias Assembly	Electrical via

Adding Geometry Using the Popup Menus

New and existing objects can be added using the context menus. To display a context menu, select the parent object in either the data tree or drawing area and right-click.

To add a new object, choose **New** to display the options available for the parent object.

Figure 4-7. New Object Popup Menu



Clicking an object type in the menu is equivalent to using the New Geometry toolbar, the result is in a new object appearing in the data tree and the display area. See “Editing Geometry” on page 40 to change the object default properties.

To add an existing object held in the library, select **Add Library Item** from the popup menu and choose the item from the file browser. Also, see “Adding Existing Geometry Using the Library” on page 37, for information on the library manager.

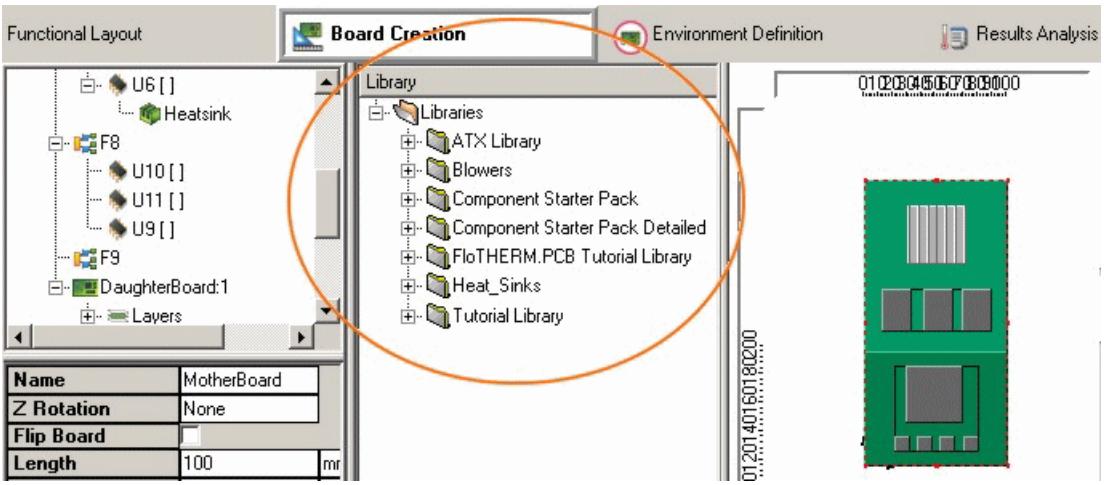
Adding Existing Geometry Using the Library

Geometry held in the library can be added to the FloTHERM PCB project using either the library pane or the data tree popup menu.

Using the Library Pane

To add geometry using the library pane, click the Show Library icon to display the libraries between the data tree and the drawing area.

Figure 4-8. Library Pane



Expand the library tree until the object you require is visible (it must be one of the following: a detailed FloTHERM PACK model, an assembly containing a compact component, or a component created in FloTHERM PCB or FloEDA Bridge and previously saved to a library).

Select the project object (for example, component) that is to receive the library geometry (for example, heatsink) in the FloTHERM PCB 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 then appear below the parent object in the data tree.

For information on populating the libraries from FloTHERM PCB projects, see “Populating the Library from FloTHERM PCB” on page 59.

Alternatively, instead of displaying the library pane, you can access a library item using the popup menu. As above, select the project object that is to receive the library geometry, but right-click and choose **Add Library Item** from the popup menu and the “Library Item Selector Dialog” on page 39 to select and add the library item.

Related Topics

- “Populating the Library from FloTHERM PCB” on page 59

Replacing Geometry

Using the Library

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

First select the geometry in the FloTHERM PCB model, then either:

- Double-click the replacement geometry in the library pane (see "Using the Library Pane" on page 37)

  or

- Right-click and choose **Replace with library item** from the popup menu and selecting the replacement object using the "Library Item Selector Dialog" on page 39.

Note

If a component, the resultant object will retain many settings of the original object. The reference designator, power, location, orientation, board side, functional layout location, size and annotation font increment of the original object will be retained. Objects will be located at the same origin of the object being replaced.

Library Item Selector Dialog

To access: ***Geometry*** **> Add > Replace Library Item**

Use the **Library Item Selector** to browse and select library items suitable for addition to the FloTHERM PCB model.

The **Library Item Selector** consists of a tree structure displaying those libraries with geometry suitable for addition to the current selection in FloTHERM PCB. The tree can be expanded by clicking '+' and collapsed by clicking '-'

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

Alternatively, you can use the Library pane, see "Adding Existing Geometry Using the Library" on page 37.

Copying Components

Copies can be made of geometry either singly or in a pattern.

Single Copy

To make a single copy, use either the paste buffer or Ctrl+drag:

- Using the paste buffer, select the object(s) in either the data tree or the Drawing Area and then use **Edit > Copy** to copy it into the paste buffer and **Edit > Paste** to paste it onto the selected board or component . The new geometry will be offset from the original. Ctrl-C and Ctrl-V are the keyboard shortcuts for **Edit > Copy** and **Edit > Paste** respectively.

- Using Ctrl+drag the new geometry is created at the location of the mouse when the mouse button is released.

Patterning

Patterns of objects can be quickly created by cloning in two dimensions by defining the number of copies and their separation in either direction using the Pattern Selected Items Dialog.

Pattern Selected Items Dialog

To access: **Edit > Pattern**

Use the **Pattern Selected Items** dialog to create a two dimensional matrix array pattern of a single geometry object.

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

To create a pattern of the selected object, enter the number of copies and spacing (pitch) along your first and second selected directions.

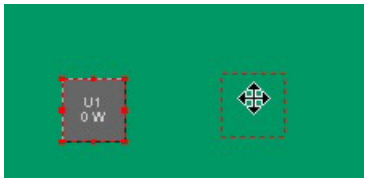
Editing Geometry

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 and Resizing Objects by Dragging Boundaries

Providing the Lock Component Move icon is off, you can edit components by dragging. **To move** an object, left-click select it and keep the mouse button pressed then move the mouse. The outline boundary of the object follows the mouse pointer.

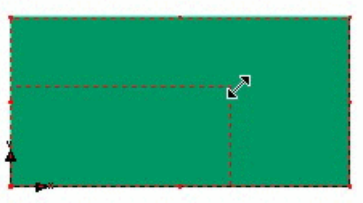
Figure 4-9. Move an Object by Dragging



On release of the mouse button the object moves to the mouse position. To constrain the movement in the horizontal and vertical directions, hold down the shift key while dragging the mouse.

To change the size of the boards and rectangular components, select and drag the object boundary box. Other objects must be resized using the property sheets (see Using Property Sheets). To maintain the length and breadth ratio, select and drag a vertex of the bounding box.

Figure 4-10. Resize an Object by Dragging



When dragging objects, the movement snaps to the nearest snap grid line. See “Snap Grid Size” on page 17.

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

Nudging Geometry

To nudge an object, left-click select it, then click a cursor left, right, up or down arrow movement key. The object will then move by snapping to the next snap grid line in the direction of the cursor key, see “Snap Grid Size” on page 17. This is useful for precise manipulation.

Aligning and Distributing Components

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

Aligning Components

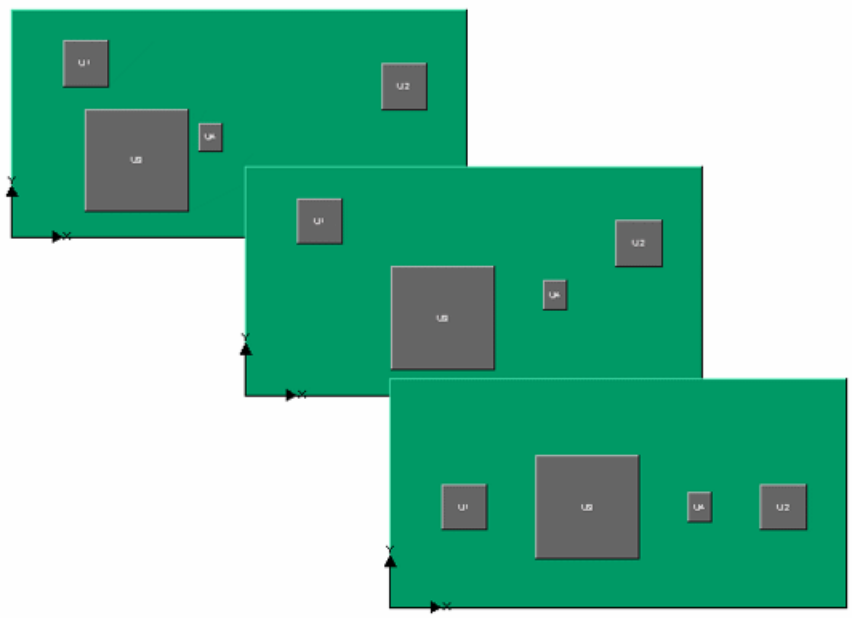
To align objects, first select them in either the data tree or the drawing area (Ctrl-click for multiple selections) and then click one of the alignment icons. The objects will align with the *first* of the objects selected.

Equispacing Components

Three or more components may be equispaced either horizontally or vertically. To equispace objects, first select them all, then click one of the distribution icons. 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.

Figure 4-11 demonstrates the use of equispacing followed by an aligning about the horizontal centers.

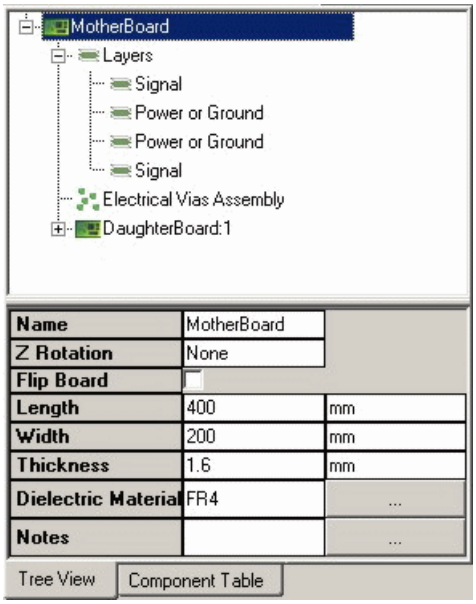
Figure 4-11. Equispacing and Aligning Components



Using Property Sheets

Any item of geometry can be edited using the property sheets. 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.

Figure 4-12. Geometry Data Tree Property Sheet

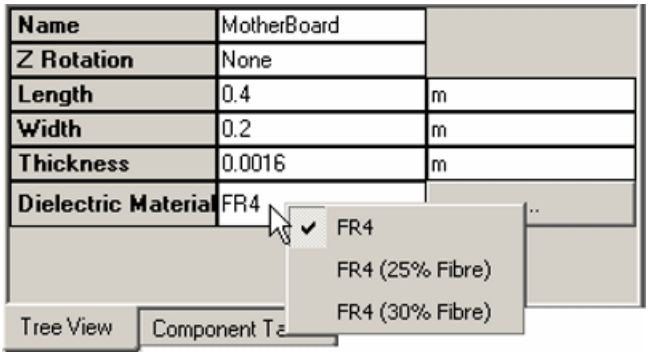


To change entries in the property sheets, select the data fields with a left-click. Some data entry boxes require text insertion, others a selection from a popup menu.

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 popup an option menu.

Figure 4-13. Property Sheet Popup 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 FloTHERM material library, then you add it to the option list. See “Choosing Material Types” on page 45.

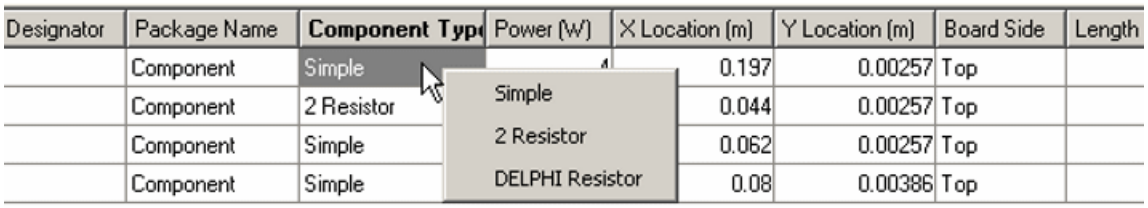
Note that you can also add construction information to the components which can be displayed in the properties sheet and the display area.

For details on the individual modeling objects, refer to their property sheet descriptions in the *FloTHERM PCB Modeling Objects Reference Manual*.

Using Component Tables

Component details can be displayed and edited in the Component Tables. Click Component Tables tab to display the details of all the components in the model. 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, otherwise choose a modeling option from the popup menu.

Figure 4-14. Component Table Popup Option Menu



For a view of the complete table see "Viewing the Component Table" on page 32.

Sorting Component Tables

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 underside of the board after having sorted on board side.

Finding Objects

To find components or functional groups quickly, use the search launched by choosing **Find** in the **Edit Menu**.

Component/Functional Group Search Dialog

To access: **Edit > Find**

Use the **Component/Functional Group Search** dialog to find and select either components or functional groups of components. When found, the objects become highlighted in the display.

To find a component or functional group, enter the search string into the **Find** field and choose the search criteria:

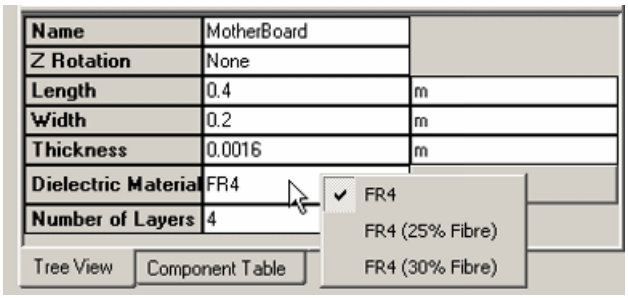
- **Reference designator** for specific objects

- **Package Name** for packages
- **Match Case** for objects with same name and capitalization as the search string
- **Partial Match** for objects with names containing the search string

Choosing Material Types

Material types are chosen for components using the Component Property Sheet. To set the material type, right-click the material data entry field and choose an option from the material list.

Figure 4-15. Choosing From the Material List



Material Options List

The material options list can be reconfigured to contain material types chosen from the FloTHERM library. To reconfigure the list, click the [**…**] button to the right of the material data choice field and use the Material Library Selector Dialog. If the material type required is not available, using the Material Library Selector, you can call the Get New Material Data Dialog to create it.

Material Library Selector Dialog

To access: **Property Sheet > (Material)**

Use the **Material Library Selector** dialog to add or delete material types from the component material options list.

The left-side of the indicates the options to appear in the list.

The right-side of the displays a data tree of the material types available. Click ‘+’ to expand the tree and ‘-’ to collapse the tree.

The **Material Library Selector** is used to select both materials and surface finishes. When selecting materials, you have additional options to create a new material or view the details of an existing material. However, surface finishes (for example, Heat Sink Interface Resistance) can only be selected from existing definitions and cannot be created or viewed.

To Add a Material Option

To add an option, expand the data tree until the material type you require is on display. Highlight the material and click '+'. The material name then appears in the **Selected Items** list. Click **Save to Config** to add the selection to the option list. This new addition will now appear when the material option list displayed for this object.

Removing a Material Option

If the material option list need tidying, you can remove those options no longer required. To remove a material from the options list, select it in the **Selected Items** list and click '-'. The item is then deleted from the **Selected Items** list. Click **Save to Config** to remove the item from the option list. The material type will no longer appear in the material option list for this object.

Create New Materials

To create new materials, select an existing library in the library tree, then click **New** to launch the Get New Material Data and define the new material type.

Viewing Material Details

To view the material definition, select it in the **Material Library Selector**, then click **View** to launch the "View Existing Material" on page 46.

Get New Material Data Dialog

To access: **Property Sheet > (Material) > New**

Use the **Get New Material Data** dialog to define a new material type and make it available for selection in the property sheet.

The new material is defined by setting its name, thermal conductivity and emissivity.

Clicking **OK** adds the new material to the both the selected items list and the library in the Material Library Selector Dialog, from where you can **Save to Config** and add the new material as a property sheet choice option.

View Existing Material

To access: **Property Sheet > (Material) > View**

Use the **View Existing Material** dialog to display the settings for the library material. The dialog fields are the same as those in the Material Library Selector Dialog, but read-only.

Adding Construction Notes

Notes can be attached to individual components for reference or identification purposes using the **Edit Text Dialog** launched from the object property sheet **Notes** button.

If you want to add notes to the overall project, use the Save Model Dialog.

Edit Text Dialog

To access: **Property Sheet > Notes > […]**

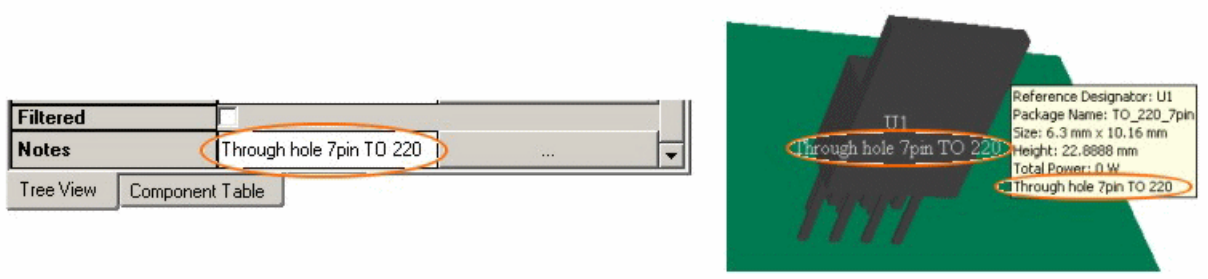
Use the **Edit Text** dialog to create component construction notes that can be displayed in the properties sheets and the display area.

The **Edit Text** dialog is a very basic text editor with no wrap around. Click **OK** to attach the entered notes to the component.

Displaying the Component Notes

The notes created are displayed at the bottom of the component properties sheet and in the display area, as well as appearing in the component ToolTips.

Figure 4-16. Component Notes



To display the properties sheet, select the component in the tree view.

To display the notes in the display area, click the Annotate icon and select either Anotate All or Anotate All, then click the Annotate With Notes icon.

ToolTips appear when the mouse pointer pauses over the component.

Importing Designs

Four types of design formats may be imported into FloTHERM PCB:

- **EDA Board Designs** — for designs exported from any of the supplied direct EDA interfaces.

- **IDF Board Designs** — for IDF board and component definition files.
- **Components CSV Layout** — a CSV file containing board layout information.
- **Importing a Power List** — a CSV file containing component powers.

Note

When importing data from a CSV file, reference designators should comprise at least two characters; single-character reference designators will be appended with the "1" character, multiple occurrences of the same single-character reference designator will be appended thereafter with "2", "3", and so on. Thus, A, B, C will become A1, B1, C1, and U, U, U will become U1, U2, U3 in the project design.

EDA Board Designs

FloTHERM PCB is supplied with direct interfaces to:

- Cadence Allegro PCB Design Editor, see Allegro Interface in the *Cadence Allegro Interface User Guide*.
- Mentor Graphics BoardStation, see BoardStation Interface in the *BoardStation Interface User Guide*.
- Mentor Graphics Expedition Enterprise Flow, see Expedition PCB Interface in the *Expedition Interface User Guide*.
- Zuken CR-5000 Board Designer, see "CR-5000 Board Designer Interface in the *Zuken CR-5000 Board Designer Interface User Guide*.

Board designs can be exported from one of the direct EDA interfaces and imported into FloTHERM PCB using the following methods:

1. Launching FloTHERM PCB with a design already loaded from within the direct interface.
2. Exporting *.*floeda* files from the interface to be later loaded into FloTHERM PCB using **File > Import > Import FLOEDA**.

Note that when using **File > Import > Import FLOEDA**, if a model is already loaded, you will be prompted to replace or update the board before choosing the *.*floeda* file from a browser. For an explanation of the replace or update options, see "Note on Update of Replace?" on page 49.

The libraries to be searched for replacement components during import can be chosen using the Library Selector for Component Import Dialog.

IDF Board Designs

Board designs available in the IDF format can be imported into FloTHERM PCB. The full definition of an IDF board design is contained in two files:

1. A board definition file (**.emn*, **.bdf*, or **.brd*)
2. A component definition file (**.emp*, **.idf*, **.lib*, or **.pro*)

In FloTHERM PCB you can import both these files. However, when starting a FloTHERM PCB design, often only the board definition is required, so you will have the option of cancelling the import of the component definition file.

To import an IDF board design use **File > Import > Import IDF**.

After opting to replace or update the board, choose the IDF board and library files from the consecutive prompting file browsers. For an explanation of the replace or update options, see "Note on Update of Replace?" on page 49.

If you choose to import both files, then the library to be searched for replacement components may optionally be selected from the Library Selector for Component Import Dialog and the Component Filter Options Dialog appears allowing small thermally irrelevant components to be either filtered or deleted completely.

If however you **Cancel** the second file browser, then only the board file is imported.

Note on Update of Replace?

Often the component placement definition will change in the EDA layout tool throughout the design process. To allow for this, when there are one or more components in the currently loaded project, the **Update or Replace?** query appears to let you choose to update an existing FloTHERM PCB model on design import.

If you choose to update, all existing definitions of component type, heatsinks, and so on will be retained, only the component location, board side and board shape will then be updated based on the Reference Designator matching. Note that 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.

Placement Keepout Region

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 mom-zero. A 3D view of the board will show the keepout regions at their defined maximum component height.

Placement keepout regions can also be manually created by clicking the Create new Placement Keepout Region icon. See “Placement Keepout Regions”.

Components CSV Layout

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

Use **File > Import > Import CSV Layout** to launch the **Update or Add?** to give you the option of either updating the current loaded layout or change it by adding components based on information in the CSV file.

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

Note: The CSV file should contain one or more column headings of the format created when **File > Export > Export CSV** is used. For a description of the format see “CSV File Format” on page 54. Also, see the **Note** about importing reference designators under “Importing Designs” on page 47.

Importing a Power List

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

<reference_designator>,<power_in_Watts>

Note that the import power list only allows you to modify power values, it does not allow you to add or remove components.

To import a power list, select **File > Import > Import Power List** and select the file.

Such a file can be created from an existing design by selecting **File > Export > Export Power List**. See “Exporting a Power List” on page 53 for more information. Also, see the **Note** about importing reference designators under “Importing Designs” on page 47.

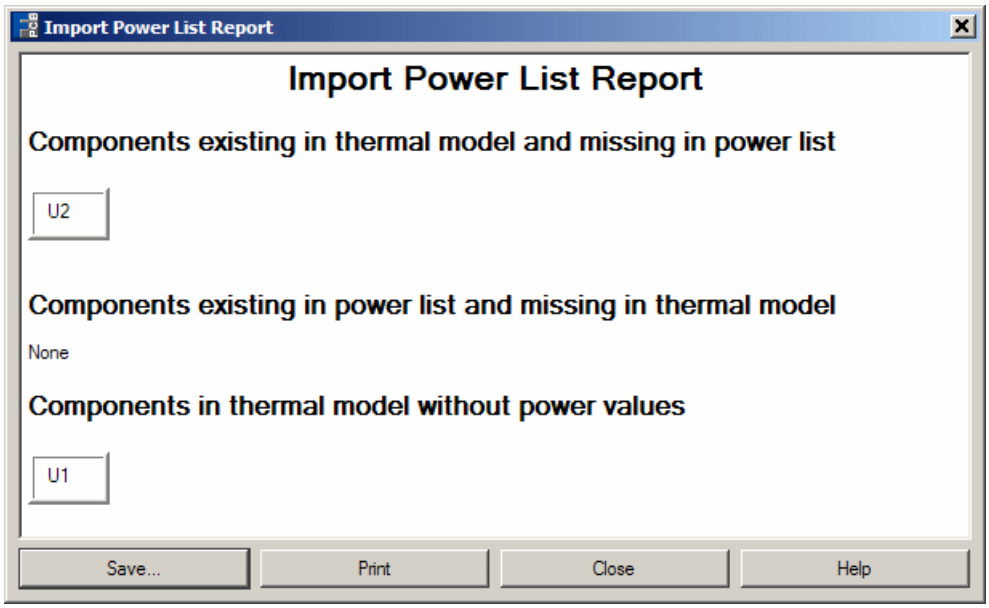
Import Power List Report

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

Import Power List Report generation is turned off by default but can be turned on by ticking the **Show Report after CSV Power Import** box in the GUI Preferences dialog.

The report (an example is shown in Figure 4-17) 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.

Figure 4-17. Import Power List Report



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

- "IDF Board Designs" on page 49

Library Selector for Component Import Dialog

To access: **Tools > Component Library Swap**

Use the **Library Selector for Component Import** dialog to choose the libraries to be searched for cross-reference links when importing components.

The dialog box contains a tree structure which can be expanded by clicking '+' and collapsed by clicking '-'.

Select the library(ies) and then click **OK**.

Any component in the selected library(ies) with the same (or partially matched) Package Name as that encountered in the IDF or FLOEDA file will then be swapped into theFloTHERM PCB model and located such that the component center and orientation are maintained. This allows for more detailed component thermal models to be imported in conjunction with the library data than would otherwise be created.

For a partial match, the name of the component in the selected library should match the beginning of the library file package name. For example, a component model in a library called SOT223 will replace the components SOT223, SOT223:1, SOT223:1:X defined in the board design file.

Note that, if a match is not found, then a simple component representation is created instead.

Exporting Designs

FloTHERM PCB board designs can be exported in six different file formats:

- **.floplc* — to save the board placement, component power dissipation, and thermal data for use in the Allegro PCB Design Editor.
- **.pack* — to save the board in a compressed format of both model and solution data ready for archiving, transfer to Mentor Graphics Mechanical Analysis Support, or unpacking into FloTHERM for detailed results analysis.
- **.pdml* — to save the board set-up information to and from the external file system. Useful for sending FloTHERM PCB projects to customer support in a compact form.
- **.emn* and **.emp* — files that can be read by EDA software, see IDF Files.
- **.csv* — comma separated file formats for loading board designs (see CSV Files) and power lists (see Exporting a Power List) into spreadsheets.

FLOPLC Files

FloTHERM PCB is supplied with direct interfaces to:

- Cadence Allegro PCB Design Editor, see Allegro Interface in the *Cadence Allegro Interface User Guide*.
- Mentor Graphics BoardStation, see BoardStation Interface in the *BoardStation Interface User Guide*.
- Zuken CR-5000 Board Designer, see CR-5000 Board Designer Interface in the *Zuken CR-5000 Board Designer Interface User Guide*.

To export placement files to Allegro or BoardStation, use **File > Export > Export FLOPLC** and choose the file name and destination from the browser. Placement update into CR-5000 is not supported.

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.

PACK Files

To export packed board designs, use **File > Export > Export PACK** and choose the file name and destination from the file browser.

Note

As Mentor Graphics Mechanical Analysis products identify packed designs by the .pack filename extension, when naming the packed file ensure you keep the default .pack extension.

PDML Files

To export PDML board designs use **File > Export > Export PDML** and choose the file name and destination from the file browser.

Note

As Mentor Graphics Mechanical Analysis products identify PDML designs by the .pdml filename extension, when naming the PDML file ensure you keep the default .pdml extension.

IDF Files

To export the board design in the IDF format files (**.emn* and **.emp*), use **File > Export > Export IDF** to call a file browser to choose the destination of first the **.emn* file, then the **.emp* file.

CSV Files

To export board design information in comma separated variable files use **File > Export > Export CSV** to call the Select Column Types Dialog and choose the information to be stored. A file browser then appears for you to choose the file name and destination. For an example of a CSV file, see "CSV File Format" on page 54.

Exporting a Power List

To export the list of power values, select **File > Export > Export Power List** and name the CSV file. The resultant CSV file will contain two text columns:

<reference_designator>,<power_in_Watts>

and can be read into a spreadsheet. The example below shows three components, U1, U2 and U3, powered by 2, 4, and 4 Watts respectively.

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

Typically you will export a power list to modify a number of power values and then import those new values back into the design, see "Importing a Power List" on page 50.

Select Column Types Dialog

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

Use the **Set Column Types** dialog to choose the project information to be output to a comma separated file. The resultant file can be loaded easily into a spreadsheet.

This dialog box lists all the possible column types. To include a column, check it in the list.

CSV File Format

The comma separated text file output from the **File > Export > Export CSV** command contains a header row followed by a row for each component in the project, for example:

```
Package Name,Reference,Designator,Total Power (W),Component Type,X
Location (mm),Y Location (mm),X Size (mm),Y Size (mm),Z Size (mm),Board
Side,Filtered
Component,U1,5,Simple,0,0,40,40,3,Top,False
Component,U2,5,Simple,0,0,40,40,3,Top,False
Component,U3,0,Simple,0,0,40,40,3,Top,False
Component,U4,0,Simple,0,0,40,40,3,Top,False
Component,U5,0,Simple,0,0,40,40,3,Top,False
```

The equivalent spreadsheet display is shown in Figure 4-18.

Figure 4-18. Spreadsheet Based on CSV Output File

	A	B	C	D	E	F	G
1	Package Name	Reference Designator	Total Power (W)	Component Type	X Location (mm)	Y Location (mm)	X Size (mm)
2	Component	U1	5	Simple	0	0	40
3	Component	U2	5	Simple	0	0	40
4	Component	U3	0	Simple	0	0	40
5	Component	U4	0	Simple	0	0	40
6	Component	U5	0	Simple	0	0	40

Related Topics

- "Exporting Designs" on page 52

Filtering Objects for Solution Purposes

FloTHERM PCB models the heat dissipated from components in one of two ways:

- Discrete, restricting heat output to the component location

  or

- Smeared over the board

Either method can be globally applied using the **Component Filter Options**.

Component Filter Options Dialog

To access: **Tools > Filter Components**

Use the **Component Filter Options** dialog to determine the heat dissipation model for the components or just to simplify the model.

Note: To use this dialog, you must be in Board Creation mode.

The **Component Filter Options** dialog consists of a property sheet for defining the component filtering or deletion controls. The chosen options are combined using the logical **AND/OR** selected in the **Combine Parameter By** field.

Filtering by Value

Components with property values lower than those entered in the dialog property sheet, 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.

Filtering by Name

Components can also be filtered or deleted by their reference designators. Enter the character, or part of the character string, of the reference designator required, separating multiple designators with a comma. For example, enter into the **Ref. Des. Contains** field:

- **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 **Name Contains** field should be left empty or set to a unique string which does not occur in any of the design reference designators.

Filtering by Multiple Parameters

By default, if multiple non-zero filter parameters are set, the components will either be filtered or deleted if either one **OR** the other parameters are satisfied. Alternatively, by toggling **AND** you can choose to filter only if all the non-zero parameters are satisfied.

In the settings example above, components that are small AND have negligible thermal impact will be filtered or deleted. This ensures that large unpowered components, for example, junctions, remain part of the thermal solution.

Resulting Filtered Representation

Filter filters 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. It is therefore more efficient, in solution time, to filter out small components.

Note

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.

Deleting Components From the Model

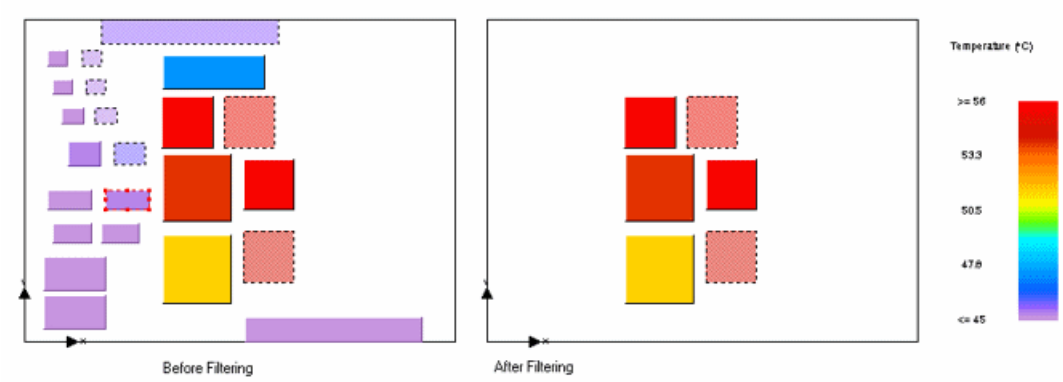
Delete deletes components with lower property values or matching reference designators from the model.

Temporary Operation

Filtering using the Component Filter Options is a ‘one-off’ operation, that is, it is not always ‘on’. So, components created after a Component Filter Options filter will not be subject to the settings in this.

The figure below shows the results display before and after filtering out the smaller components from discrete modeling.

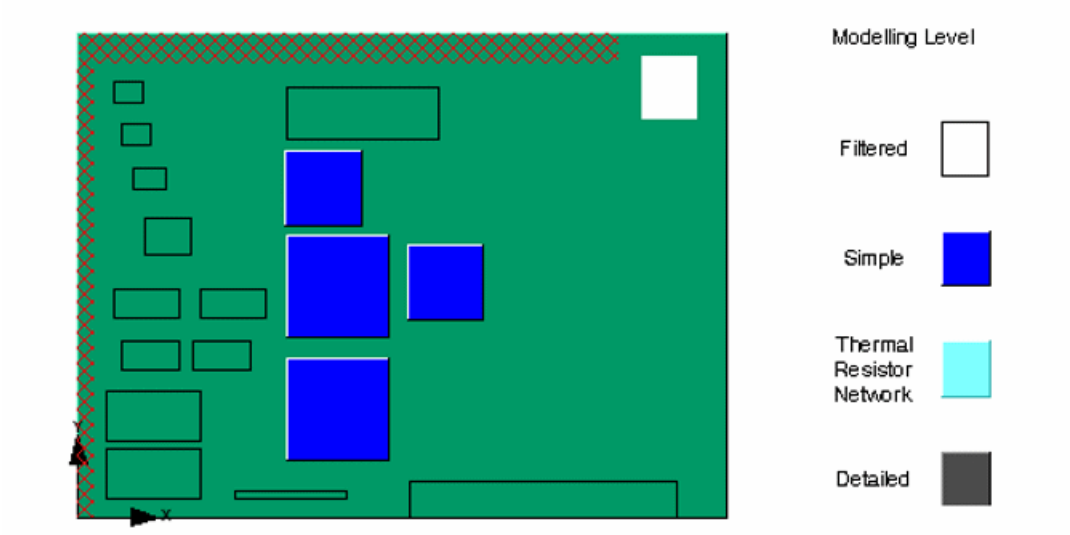
Figure 4-19. Filtering Smaller Components



Displaying Filtered-Out Components

To display filtered-out components, go to Board Creation mode and click the Color Components icon. Any filtered-out components will appear in wireframe over the board as indicated in Figure 4-20.

Figure 4-20. Filtered Out Components



Using the Libraries

Library Configuration

Any object that can be created in FloTHERM PCB can be saved to, and retrieved from, a library, see “Adding Existing Geometry Using the Library” on page 37, “Replacing Geometry” on page 38, and “Populating the Library from FloTHERM PCB” on page 59.

Libraries are located on the file system and located from within FloTHERM PCB via the FLOCENTRALDIR environment variable, normally set in the *runflopcb.bat* file. The *runflopcb.bat* file is found in:

<install_dir>/flosuite_v92/flopcb_v6.2/WinXP/bin

If FloTHERM PCB is installed along side FloTHERM then, by default, FLOCENTRALDIR would be:

<install_dir>\flosuite_v92\flotherm\flocentral\Libraries

Library items are stored in Mentor Graphics Mechanical Analysis PDML format thus offering full data integration to FloTHERM and FloTHERM PACK (see www.mentor.com/products/mechanical/products/flotherm-pack).

Additional User Libraries

In addition to the delivered libraries, you can create new libraries. See "Creating New Libraries" on page 60, for a description of how a new library is created. 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 sub-folder. 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.

Importing and Exporting Libraries and Library Items

An entire library can be exported to the file system for subsequent distribution to other FloTHERM PCB installations by accessing the library's popup menu and selecting **Export**. A filename is supplied and a .library file is created. This can then be e-mailed for example to another remote FloTHERM PCB installation. The library can then be imported by accessing an existing library's popup menu and selecting **Import**.

Individual library items in Mentor Graphics Mechanical Analysis PDML format can be added to a library via the file system only. This will be the case for individual items downloaded from www.mentor.com/products/mechanical/products/flotherm-pack. Place the pdml file in the directory on the file system that corresponds to the library as seen from within FloTHERM PCB. From within FloTHERM PCB access the popup menu of that library and select **Refresh**. This will search the directory and update the displayed list of available library items in FloTHERM PCB.

Note that apart from objects created within FloTHERM PCB, only the following items are supported for import into FloTHERM PCB from external sources (for example, FloTHERM and FloTHERM PACK):

- FloTHERM Heatsink SmartPart
- FloTHERM PCB SmartPart

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

  Note that to preserve FloTHERM PACK model integrity, these components cannot be modified except for position, orientation, max temperatures, and power. You are provented from changing the other fields in the GUI. If you build a 2R model directly in FloTHERM PCB (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.

Attempts to import objects not of the above type will result in a 'Failed to import Library Item into selected part(s)'.

Creating FloTHERM Environment Libraries

Libraries of FloTHERM PCB environments can be created using the **Tools > Get FloTHERM Environments** functionality. These are stored in the directory:

%FLOCENTRALDIR%\FLOPCBEnvironments

These can also be distributed to other FloTHERM PCB installations using the **Export** and **Import** popup menu items.

Creating a Central Library Configuration

A group of people using FloTHERM PCB can most effectively share library data by ensuring that each installation has FLOCENTRALDIR set to a common directory on the local file system. This environment variable can be set either in the individual *runflopcb.bat* file or as a system environment variable.

Populating the Library from FloTHERM PCB

Saving Geometry in the Library

To copy FloTHERM PCB board geometry to the FloTHERM library, select the geometry, right-click, then choose **Save to Library** from the popup menu and set the file destination in the Library Selector Dialog. After exporting the design to the library, it can be quickly replicated in other FloTHERM PCB or FloTHERM projects.

The board geometry will appear as an assembly, not a PCB SmartPart, in the FloTHERM library. The detail of the board, once added to a FloTHERM model, will be dependent on the current solution type in the "Thermal Solution Docker" on page 80 (launched by **File > Solve**). If a detailed description of the board is required (for example, layers modeled explicitly) then the **Accurate Thermal Prediction** option in the solution, must be checked on.

New library folders can be created using the C**reate New Libraries**, see “Creating New Libraries” on page 60.

Related Topics

- “Setting the Solution Type” on page 80

Library Selector Dialog

To access: ***Geometry selection*** **> Save to Library**

Use the **Library Selector** dialog to choose the library to hold the exported geometry (a MotherBoard, DaughterBoard, component, or functional group selected in FloTHERM PCB) to the FloTHERM library.

This geometry can then be accessed from FloTHERM and used for system analysis.

The dialog consists of a tree structure which can be expanded by clicking ‘+’ and collapsed by clicking ‘-’.

To copy the design into the FloTHERM library, highlight the library name in the tree and click **OK**.

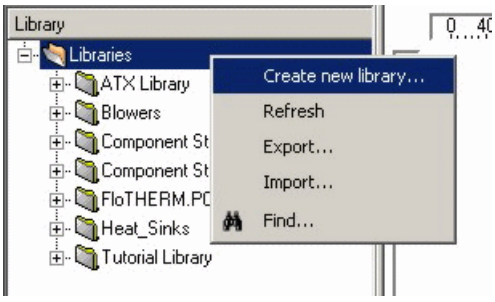
Related Topics

- “Populating the Library from FloTHERM PCB” on page 59
- “Adding Existing Geometry Using the Library” on page 37

Creating New Libraries

New libraries can be created using the **Create New Libraries** dialog.

Figure 4-21. Creating a New Library



Create New Libraries Dialog

To access: Click the Show Library icon, right-click a library, and select **Create new library**

Use the **Create New Libraries** dialog to add a folder to the FloTHERM library. Geometry may then be added to or copied from this directory using the library manager.

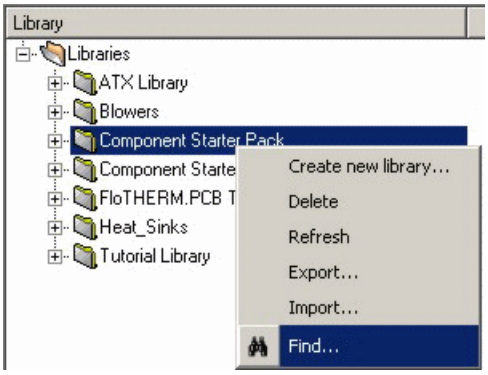
Enter the name to be displayed in the library tree in the **Library Name** field.

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.

Searching Libraries

Libraries can be searched using the **Library Search Facility** dialog.

Figure 4-22. Searching a Library



Library Search Facility Dialog

To access: Click the Show Library icon, right-click a library, and select **Find**

Use the **Library Search Facility** dialog to quickly find and select items in a library according to an entered search criteria. The dialog will list out any found matches. It can also be used to select the found items in the library tree ready for deletion or downloading into the FloTHERM PCB project.

To Search for a Library Item

1. Enter all or part of the item name you want to find in the **Find** field.
2. Click **Partial**, if searching for a substring match.
3. Click **Match Case** if searching for a capitalization match.
4. 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.

Quickly Downloading a Found Item

Found library items can be quickly downloaded to valid selected project parts by double-clicking in the **Found List**. Select the part to receive the download before launching the Library Search Facility dialog.

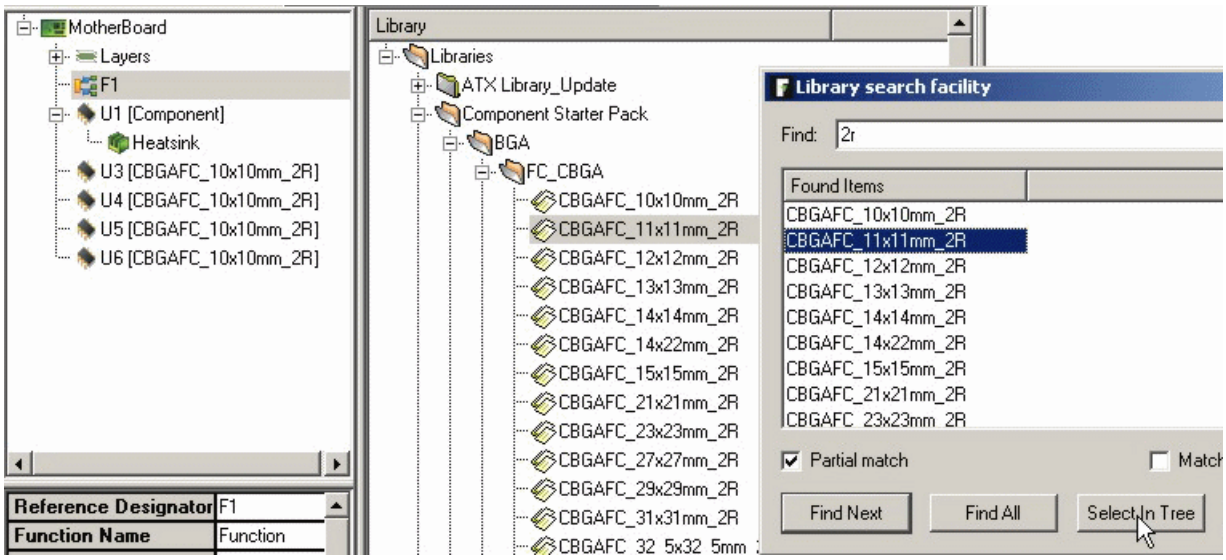
Selecting Items in the Library Tree

To select a found item in the library tree:

1. Search for the library item using the **Library Search** dialog.
2. Select the item in the **Found Items** list,
3. Click **Select in Tree**.

The item will appear selected in the Library tree. The Library tree will expand, if necessary, to display the selected item.

Figure 4-23. Selecting Library Tree Items



After closing the dialog, the selected item can be deleted from the library or downloaded to the project using the right-click popup menu.

Figure 4-24. Library Tree Item Popup Menu

Note

In the example shown in Figure 4-23, double-clicking CBGAFC_11x11mm_2R in the dialog will download it into F1.

Chapter 5
Defining the Environment

This chapter describes how to define the environment surrounding the board. It describes the environment data and explains how to view and change the set-up.

Viewing the Environment Data

To view the environment data and the setup fields, click the **Environment Definition** button.

Figure 5-1. Environment Definition Button

The environment is displayed in four areas:

- Environment Mode Button for a summary
- Properties Sheet allowing manual change
- Environment library tree containing environments that can be copied into FloTHERM PCB.
- 2D or 3D view of the solution domain and air flow direction.

See "Environment Definition" in *Getting Started with FloTHERM PCB* for a description of the window layout.

Default Environment

On starting a new project, the environment is set by default to model a card slot environment with an ambient temperature of 45˚ and speed of 2 m/s. A summary of the data can be seen in the Environment Definition button if Small Process Icons is set to False in the "GUI Preferences Dialog" on page 15.

2D and 3D Views

The display area shows the solution domain outline and the air flow direction. By clicking the View From icon and then the 3D icon, the display changes to 3D, and you can rotate the view.

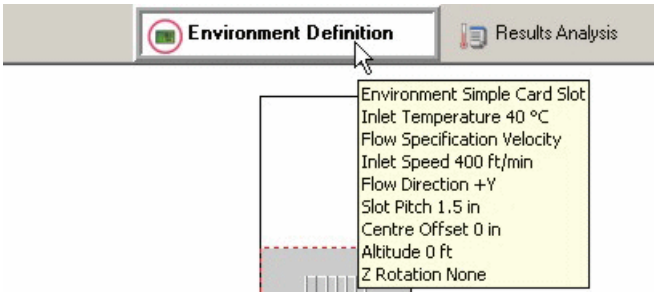
Environment Summary

A quick summary of the environment can be displayed in the process bar or the ToolTip. If you deactivate the Small Process Icons option in the "GUI Preferences Dialog" on page 15 then the Environment button will be expanded to provide a brief summary of the environment data.

Figure 5-2. Environment Button Summary Data

Alternatively, if the **Use Tool Tips** option is activated in the **GUI Preferences**, then the environment summary can be displayed in a TooltTip when the mouse pointer hovers over the Environment Definition button.

Figure 5-3. ToolTip for Environment Button



Changing the Environment Using the Properties Sheet

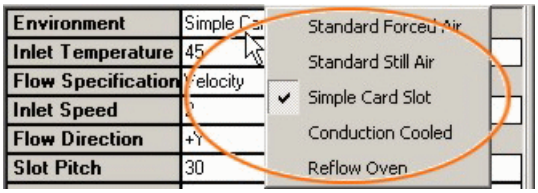
To change the environment using the Properties Sheet, ensure that the **Environment Mode Button** is active, then change the Property Data Sheet fields. The options available differ according to the **Environment** chosen.

The environment that you define in FloTHERM PCB should be a reasonable representation of the air flow and thermal conditions that you expect the board to be subjected to in operation. For example, you can define the altitude above sea level. Heights of between 0 and 40,000 ft are accepted. The properties of the air are changed based upon the defined height. Non-zero altitude values should be set for high altitude applications only, for example, not when above 50ft.

Environment Types

There are five options for defining the environment: Simple Card Slot (the default), Standard Forced Air, Standard Still Air, Conduction Cooled and Reflow Oven.

Figure 5-4. Choosing Environment Type



Standard Forced Air

This option will place the board in the JEDEC (www.jedec.org) standardized forced air test. This option is suitable if you do not know the exact operating environment but want to compare the performance of the board in a standardized forced convection cooling environment.

The size of the environment is derived from the size of the board under consideration.

Information Required:

- **Ambient Temperature** of the surrounding air
- **Speed** of the air
- **Flow Direction** setting the expected upstream flow direction
- **Altitude** for high altitude modeling
- **Z Rotation** of board
- **Flip Board**, if required to turn the board over

Standard Still Air

This option places the board in the JEDEC (www.jedec.org) still air test. This option is suitable if you do not know the exact operating environment but want to compare the performance of the board in standardized natural convection conditions.

The size of the environment is derived from the size of the board under consideration.

Information Required:

- **Ambient Temperature** of the surrounding air
- **Gravity Direction** to model the effects of gravity
- **Altitude** for high altitude modeling
- **Z Rotation** of board
- **Flip Board**, if required to turn the board over

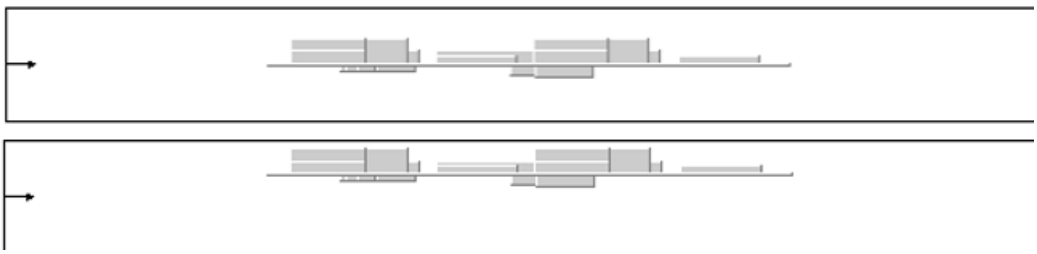
Simple Card Slot

This option will place the board design in a card slot. Choose this option if the actual board is intended to be placed in a card cage. This option requires that you know approximately the inlet conditions to the card slot.

Information required:

- **Inlet Temperature** for the air temperature entering the slot
- **Flow Specification** for the volume flow rate or an inlet velocity
- **Inlet Speed** of the air entering the slot
- **Flow Direction** of the inlet air
- **Slot Pitch** for the distance between the cards in the card cage
- **Center Offset** for an offset distance to position the simple card slot with a defined offset from the central location. This is useful when representing the effects of empty neighboring slots.

Figure 5-5. Center Offset



- **Altitude** for high altitude modeling

FloTHERM PCB will assume a uniform velocity and temperature distribution entering the card slot from the direction that is specified. The upstream and downstream distance cannot be changed.

Conduction Cooled

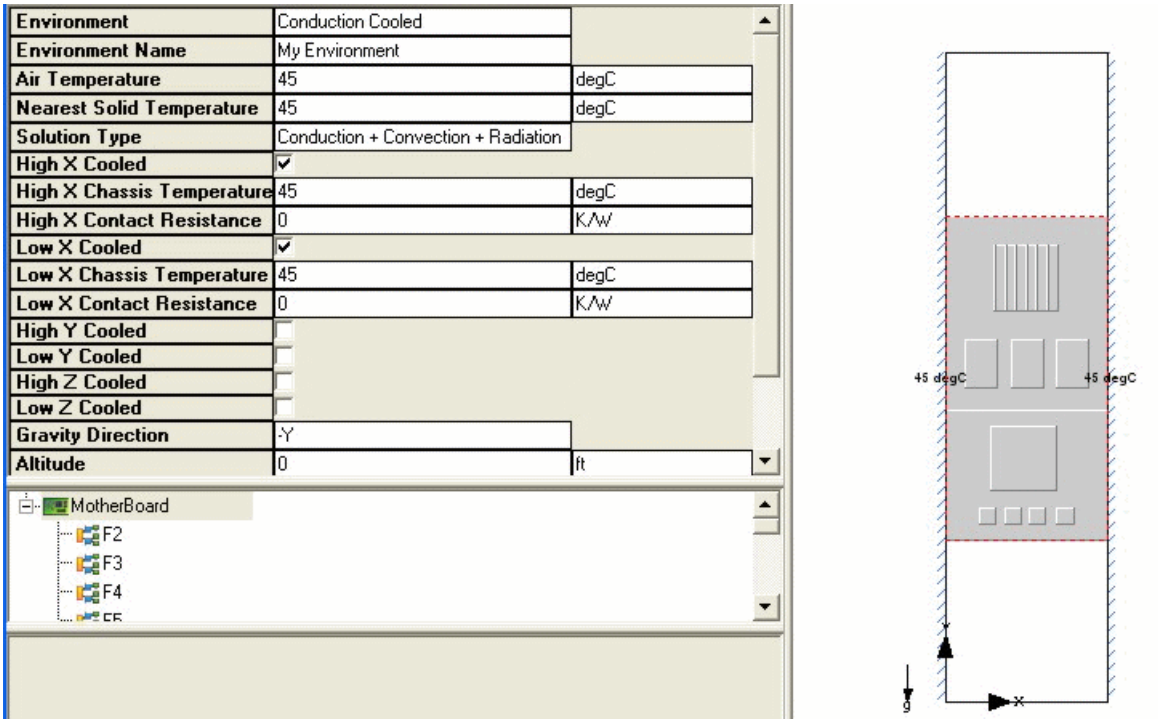
This option models the conduction cooling of boards. Conduction cooling can be represented:

- Along any edge(s) or side(s) of the board
- For individual components
- Selected areas of the board
- In a vacuum

Board Cooling

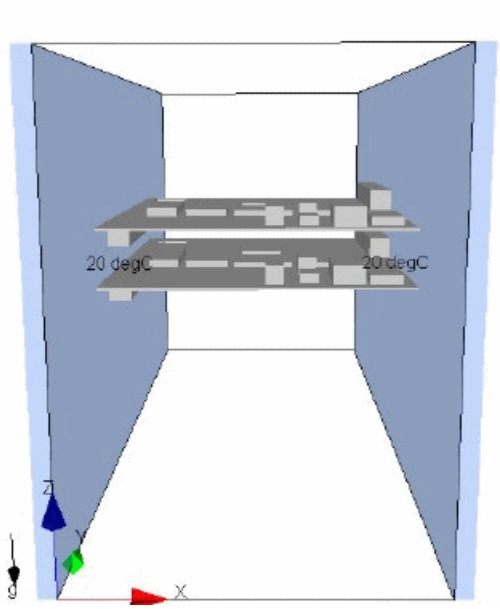
To model conduction cooling, select the **High/Low X**,**Y,** or **Z Cooled** property sheet option for the side or sides you are interested in, then set the edge temperature, together with an additional thermal contact resistance, in the fields that appear under the cooled check box as shown in Figure 5-6, to model the effects of wedgelocks and other methods.

Figure 5-6. Setting Conduction Cooling Options



A side of the motherboard selected for conduction cooling will impose a fixed temperature on the entire side of the environment, not just the motherboard edge. Any object that abuts that side of the environment, for example daughterboards and heatsinks, will be conduction cooled at the specified temperature.

Figure 5-7. Conduction Cooling of Abutting Objects



Note

If the top or bottom face of the board is set as conduction cooled, a solution will only be allowed if there are no components on the cooled side of the board.

The physical environment that the PCB resides in is not explicitly represented. Instead, the local **Air Temperature** is defined as is the **Nearest Solid Temperature**; the latter is used to calculate the radiation exchange with the PCB being modelled.

The direction of gravity and the PCBs Z rotation can be altered so as to locate the board in any orientation.

Information Required:

- **Air Temperature** surrounding the board
- **Nearest Solid Temperature** from the board
- **Solution Type** set to one of three options:
  - **Conduction + Convection + Radiation** (heat removed by conduction, convection and radiation)
  - **Conduction Only** (ignores the heat removal by air flow and radiation)
  - **Vacuum** (heat removed by conduction and radiation only, ignores convection)
- **Side/edge** to be cooled and its thermal contact resistance
- **Gravity Direction** to model the effects of gravity

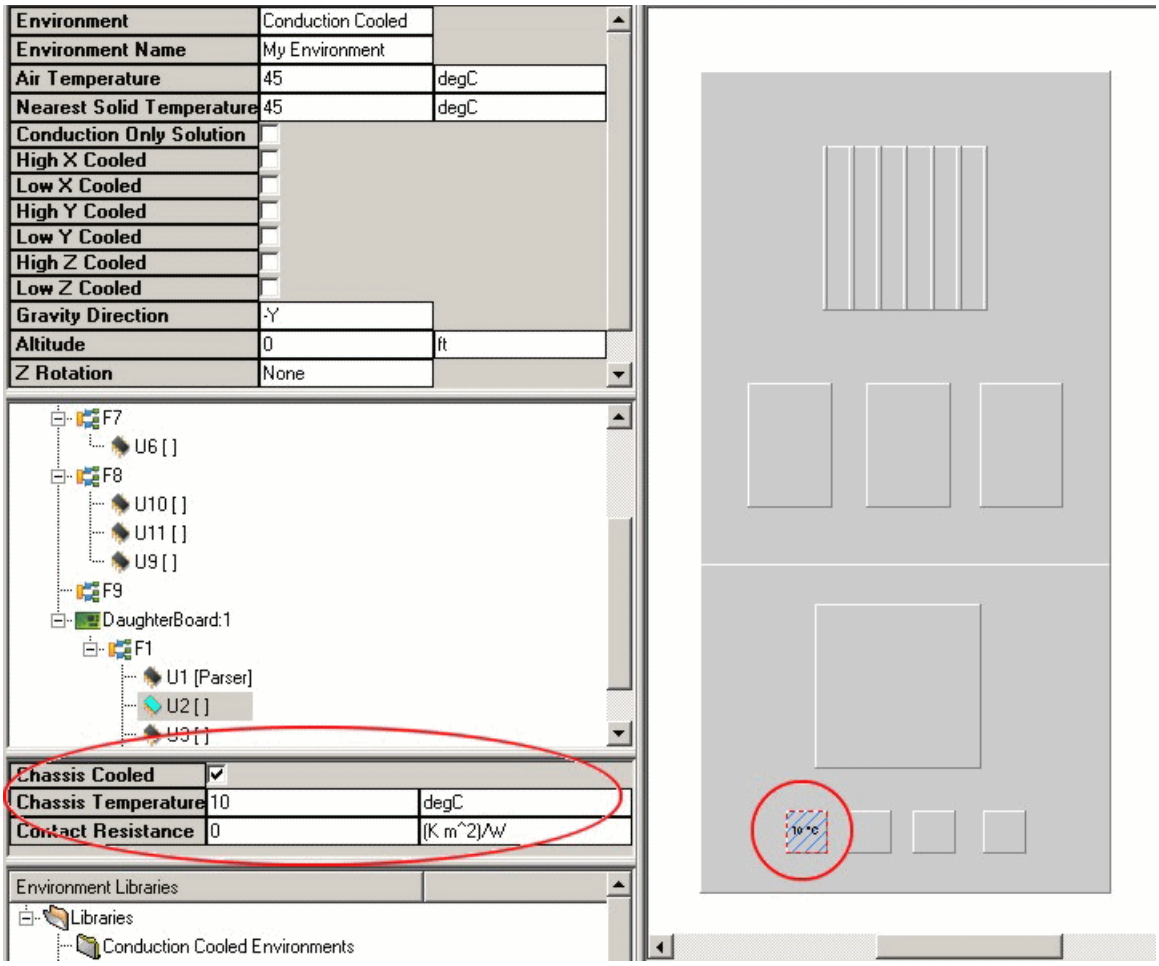
- **Altitude** for high altitude modeling
- **Z Rotation** of board
- **Flip Board**, if required to turn the board over

Component Cooling

In addition to, or instead of, the board side(s) being conduction cooled, the individual components can also be conduction cooled. This represents the cooling of critical components by moulding the chassis so as to be in conduction contact with the component top.

Conduction cooled components are set by first selecting the component(s) either in the display area or in the node tree located below the conduction cooled environment property sheet, clicking **Chassis Cooled** and entering the chassis temperature and contact resistance.

Figure 5-8. Component Cooling Setup



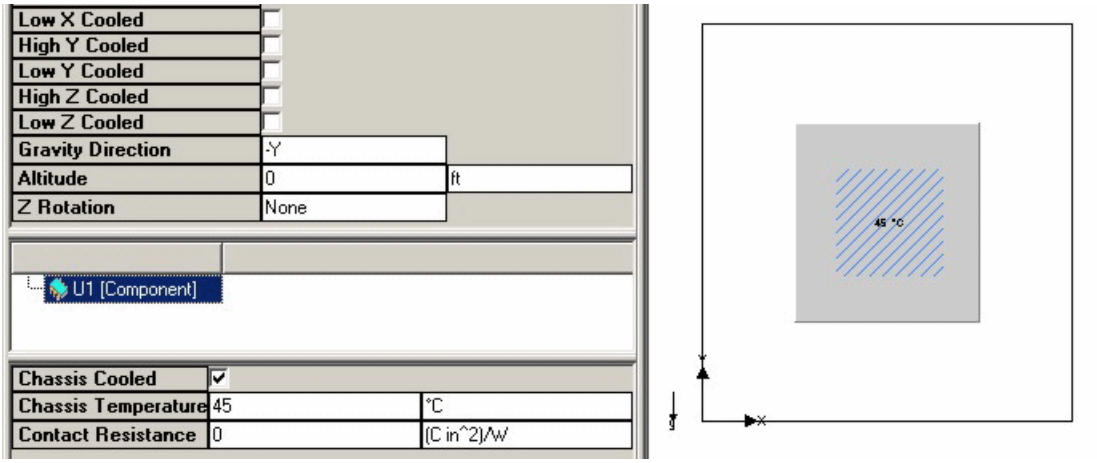
Area Conduction Cooling

Conduction cooling of a specified area, or areas, of the board can also be performed. To model area conduction cooling:

1. In **Board Creation**, define an unpowered component of the required size and position on the board
2. Set it to be filtered using its construction property sheet
3. In the **Environment** node tree, select it and set it as conduction cooled

The component will not be represented, but the area of the board it occupies will be conduction cooled

Figure 5-9. Conduction Cooling an Area



Conduction Only Solution

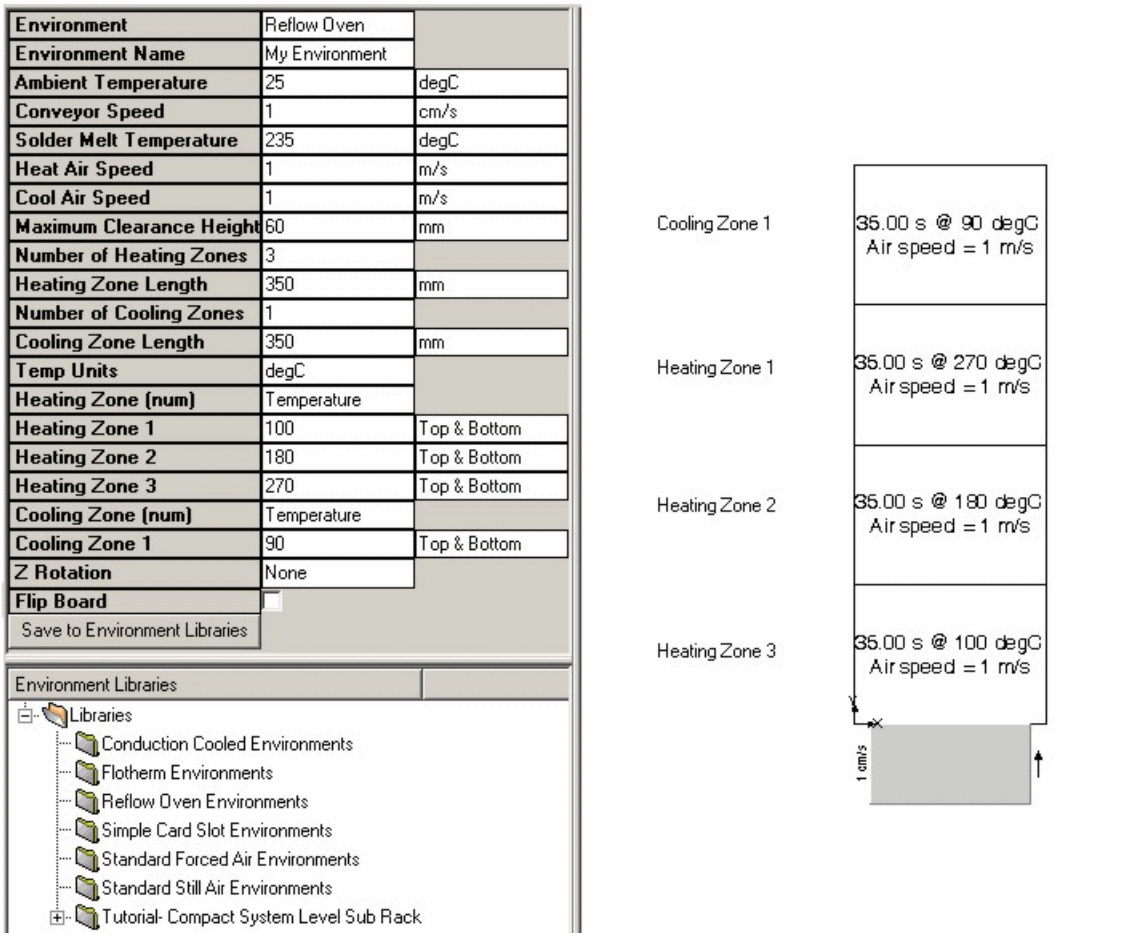
The **Conduction Only Solution** option will provide a conservative estimate of component and board temperatures by assuming the heat removed by the air flow and radiation exchange is negligible to the extent that it is justified to be ignored.

Reflow Oven

This option will place the board inside a reflow oven.

To simulate the solder reflow process, set-up a number of heating and cooling zones in the oven, specifying their length and temperature. The time the board spends in each zone is calculated from the **Conveyor Speed** and the zone length. The zone details are indicated in the display area.

Figure 5-10. Simulating the Solder Reflow Process

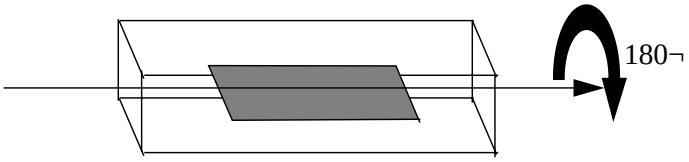


Information Required:

- **Ambient Temperature** for the surrounding air temperature
- **Conveyor Speed** for the speed the board travels through the oven
- **Solder Melt Temperature** for the temperature at which the solder will melt. This value defaults to 235˚C and is the same for all components.
- **Heat Air Speed** for the speed across the heating zone
- **Cool Air Speed** for the speed across the cooling zone
- **Maximum Clearance Height** between the board and surface and the perforated plate through which air is supplied
- **Number of Heating Zones** in the oven
- **Heating Zone Length** of each heating zone
- **Number of Cooling Zones** the number of cooling zones

- **Cooling Zone Length** of each cooling zone
- **Temp Units** for the units of temperature used for calculation and display
- **Heating Zone (num)** read-only table heading
- **Heating Zone *n*** temperature applied to top and bottom or top only of the board
- **Cooling Zone (num)** read-only table heading
- **Cooling Zone *n*** temperature applied to top and bottom or top only of the board
- **Flip Board**, if reflow solder is required on the other side of the board. Toggling **Flip Board** on rotates the board 180˚ about the axis through the oven.

Figure 5-11. Board Flip for Reflow on Other Side



Alternative FloTHERM Environment

If the board's operating environment does not fit well into the above choices, or a more detailed analysis is desired, then consider using a FloTHERM environment. See "Copying a FloTHERM Environment" on page 74.

Copying a FloTHERM Environment

Using the Tools Menu

The environment of regions surrounding a PCB in a FloTHERM library project can be copied into FloTHERM PCB. Use the **Tools > Get FloTHERM Environment** command to launch the **Browse for Folder**. Choose the project containing the required model, then the region containing the required environment from the Choose Region Dialog. It is envisaged that this operation is done once, because the **Tools > Get FloTHERM Environment** operation creates an Environment Library item, which appears in the Environment Library panel. Selection of the environment from FloTHERM Environment Library panel subsequently, will be done often.

Using the Environment Library

The Environment library provides a shortcut to the environments that have already been accessed by FloTHERM PCB. To copy an environment from a project listed in the using the Environment library, expand the project node and double-click the region containing the required environment. The environment library is held in the *FLOCENTRALDIR/FLOPCBEnvironment* directory

Restrictions on FloTHERM Environments

FloTHERM PCB sets certain constraints on the types of environments that can be copied from FloTHERM. The types of projects and regions *NOT* allowed to provide environments for FloTHERM PCB are listed in "FloTHERM Projects Disallowed" on page 75 and "FloTHERM Regions Disallowed" on page 76.

FloTHERM Projects Disallowed

The following types of FloTHERM project are *NOT* allowed to provide environments for a FloTHERM PCB solution:

Table 5-1. FloTHERM Project Restrictions

Project Restriction	Message issued on Project directory selection
Not 2D	2D projects cannot be used to create environment
Not transient	Transient projects cannot be used to create environment

The FloTHERM project has to be in a 'just solved' state, that is, a solution has terminated and no subsequent modifications have been made to the project

Table 5-2. Projects That Must Be Re-Solved in FloTHERM

Project Restriction	Message issued on Project directory selection
Grid has not changed	Project not consistent with solution: backup directory exists. Re-solve in FloTHERM
Residual history exists	Project not consistent with solution: residual file does not exist. Re-solve in FloTHERM
Project has been translated	Project not consistent with solution: Tag file does not exist. Re-solve in FloTHERM

To enable the environment to be correctly defined, the FloTHERM project must provide a minimum number of solved variables.

Table 5-3. Project Variables Required

Variables Restriction	Message issued on Project directory selection
Temperature, Pressure, Velocities and Heat Flux[1] required	Project must have Temperature, Pressure, Velocities and Heat Flux

1. Storage of this variable is activated in the **Model** > **Modeling** and should be set so prior to the solve.

Related Topics

- "Copying a FloTHERM Environment" on page 74

FloTHERM Regions Disallowed

A Region object is used to define the extents of an environment that is to be used in FloTHERM PCB. The following region configurations are disallowed:

Table 5-4. FloTHERM Project Region Restrictions

Region Restriction	Message issued on Region selection
Region must not be collapsed (i.e., must be 3D)	Region is not valid if it is collapsed
Must not be on edge of a grid space	Region is not valid if it is at a localized grid boundary or if it is at the solution domain boundary
Upper limit on number of environment cell 'faces'	More than 4000 cells exist on region boundary

A single PCB SmartPart MUST exist in the region. This PCB SmartPart must be wholly inside the region. It is allowed to internally abut the region. This region is optionally converted into the FloTHERM PCB motherboard when the environment is selected in FloTHERM PCB.

Table 5-5. PCB SmartPart Restrictions in Region

Restriction	Message issued on Region selection
PCB must be wholly inside the region	PCB smartparts must not intersect the region boundary

Only a limited number of FloTHERM objects are allowed to exist in a region that is to be selected to define an environment in FloTHERM PCB. This is to ensure the automated gridding and solution control capabilities in FloTHERM are never compromised.

Table 5-6. Restrictions on Objects in Regions

Object NOT allowed in Region	Message issued on Region selection
Fans	Invalid object(s) in region: Fan
Recirculation Devices	Invalid object(s) in region: Recirculation Device
Fixed Flows	Invalid object(s) in region: Fixed Flow
Square Diffusers	Invalid object(s) in region: Fixed Flow
Heat Sinks	Invalid object(s) in region: Heat Sink

Table 5-6. Restrictions on Objects in Regions (cont.)

Object NOT allowed in Region	Message issued on Region selection
Compact Component	Invalid object(s) in region: Compact Component
Source	Invalid object(s) in region: Source
Dissipating Cuboid	Invalid object(s) in region: Dissipating Cuboid
Sloping Block	Invalid object(s) in region: Sloping Block

Due to technicalities in the definition of the boundary conditions used to define the extents of the environment the following are further restrictions

Table 5-7. Restrictions on External Objects Abutting a Region

Object not allowed to externally abut the Region	Message issued on Region selection
Resistances	Resistances must not abut the region boundary
Dissipating Cuboids	Dissipating Cuboids must not abut the region boundary

Related Topics

- "Copying a FloTHERM Environment" on page 74

Choose Region Dialog

To access: **Tools > Get FloTHERM Environments > Project selection**

Use this dialog to choose the region containing the environment to be copied into FloTHERM PCB.

The dialog box lists the regions present in the FloTHERM project selected for the data transfer in the **Tools > Get FloTHERM Environment** procedure. Highlight the required region and click **OK**.

For information on the types of FloTHERM regions you can't copy into FloTHERM PCB, see "Restrictions on FloTHERM Environments" on page 75.

Related Topics

- "Copying a FloTHERM Environment" on page 74

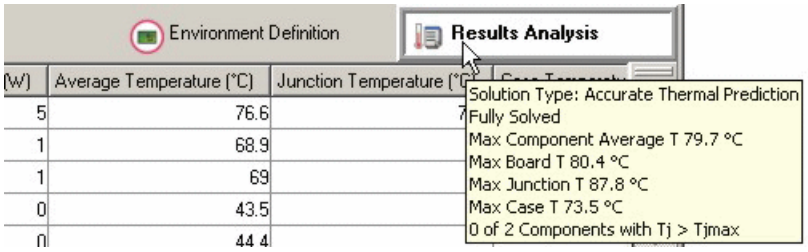
Chapter 6
Solving and Viewing the Results

This chapter describes how to solve FloTHERM PCB projects. It describes the solution status and how to the choose the solution type, start the solution as well as view the results.

Displaying the Solution Status

The solution status of the FloTHERM PCB project is displayed in the ToolTip when the mouse lingers over the **Solution and Results Mode** button

Figure 6-1. ToolTip for Results Analysis Button



Alternatively, the solution status can be displayed in the process bar if you deactivate the **Small Process Icons** option in the "GUI Preferences Dialog" on page 15.

Figure 6-2. Results Analysis Button With Summary

Related Topics

- "Checking the Model and Starting the Solution" on page 79
- "Viewing the Results" on page 89

Checking the Model and Starting the Solution

To check the model and run a solution in FloTHERM PCB, first choose **Check Model** from the **File** menu. FloTHERM PCB checks the model and displays any warnings in a "Thermal Solution Docker" on page 80, which appears at the bottom of the window. To start the solution, in the docker, select the solution type and click the green arrow button.

Alternatively, if you click the **Results Analysis** button when results are not present, the solution docker will appear to invite you to start the solution.

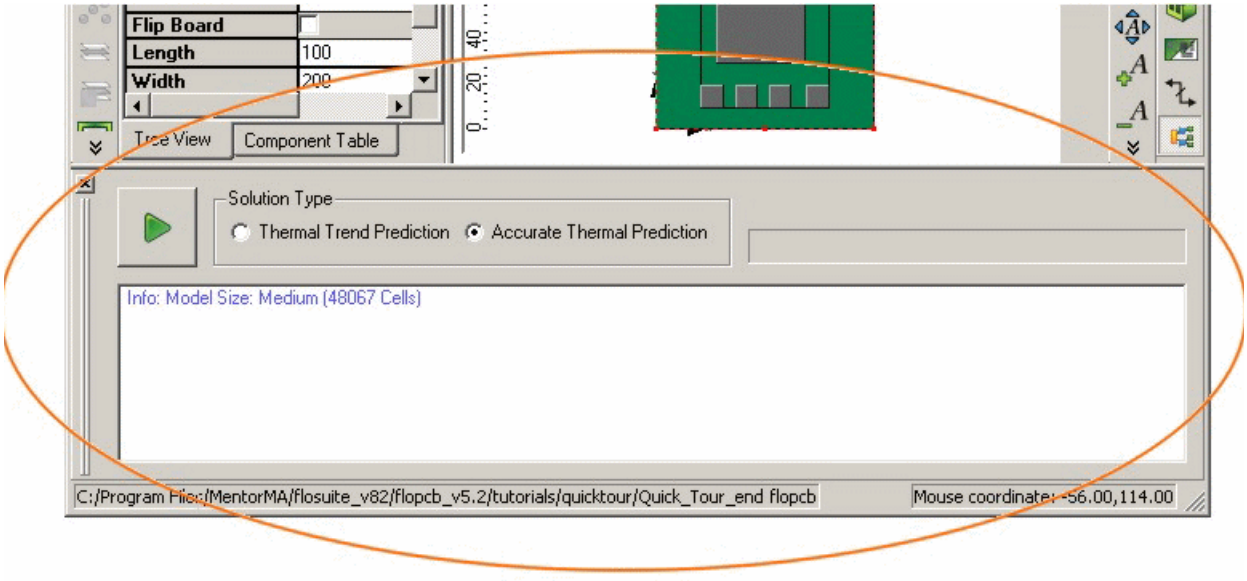
Related Topics

- "Viewing the Results" on page 89

Thermal Solution Docker

Use the thermal solution docker to re-check the model and start the solution.

Figure 6-3. Thermal Solution Docker



Starting and Interrupting the Solution

Click the green arrow button to start the solution. If no modeling errors are detected, the solution will continue, unless interrupted, until the temperature of all the components does not change with successive iterations and the residuals reach a default termination criteria value.

Click the orange square button to interrupt the solution. Click the green arrow button to continue the solution.

If you interrupt the solution and save the project, this partially converged project can be later re-loaded into FloTHERM PCB to continue the solution.

Setting the Solution Type

There are two solution types, choose:

- **Thermal Trend Prediction** — for fast solutions. But, the results are only accurate enough to resolve trends when comparing different layouts. The board layers are averaged into a single block and heatsinks are modelled as compact.
- **Accurate Thermal Prediction** — for longer and more accurate solutions. But, the results provide a grid independent temperature prediction. The board layers and heatsinks are modelled in detail.

The difference between the two solutions is the grid density and also the representation of the board. For trend prediction, the board is treated as a single orthotropic board. For accurate prediction, each layer is modeled explicitly.

Note

The **Reflow Oven** environment is restricted to **Accurate Thermal Prediction** solutions only.

Model Size

The size of the model is indicated, in blue text, in the message panel, prior to solution. The reported model size is described as small (model size < 30k grid cells), medium (30k grid cells < model size < 200k grid cells) large (model size > 200k grid cells). The total number of cells is also reported.

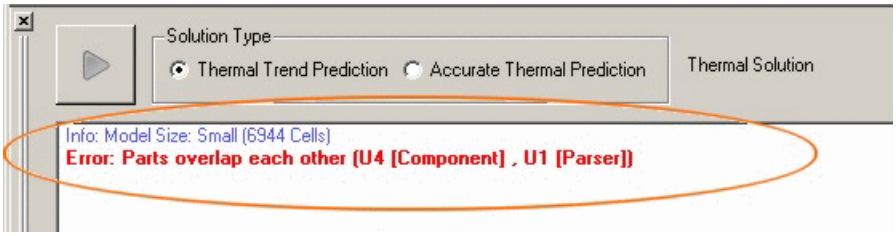
Re-Checking the Model

The ***Re-Check Model*** button (in red) appears when changes have been detected. Click the **Re-Check Model** button before solving to perform a sanity check. If errors are detected, information will be displayed in the message panel.

Message Panel

After clicking the green arrow button to start the solution, FloTHERM PCB performs a sanity check. If any problems are detected, error or warning messages are displayed. Fatal errors that prevent solution are displayed in red text.

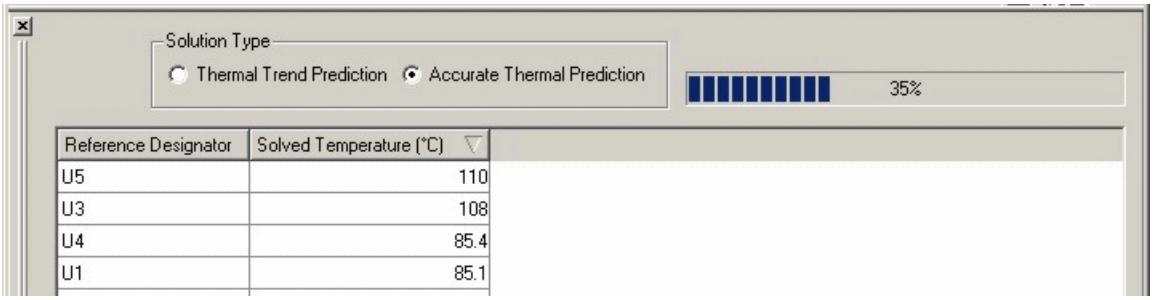
Figure 6-4. Detected Fatal Errors Shown in Red



In Figure 6-4, clicking on the error message selects the offending object(s) ready for correction. One of the objects can then be moved to solve the overlap. The model must then be re-checked prior to solution by clicking **Re-Check Model**.

If no fatal errors are detected, the solution continues. The solution progress is indicated in the progress bar and the component temperatures are displayed as they are calculated.

Figure 6-5. Temperature Display and Progress Bar



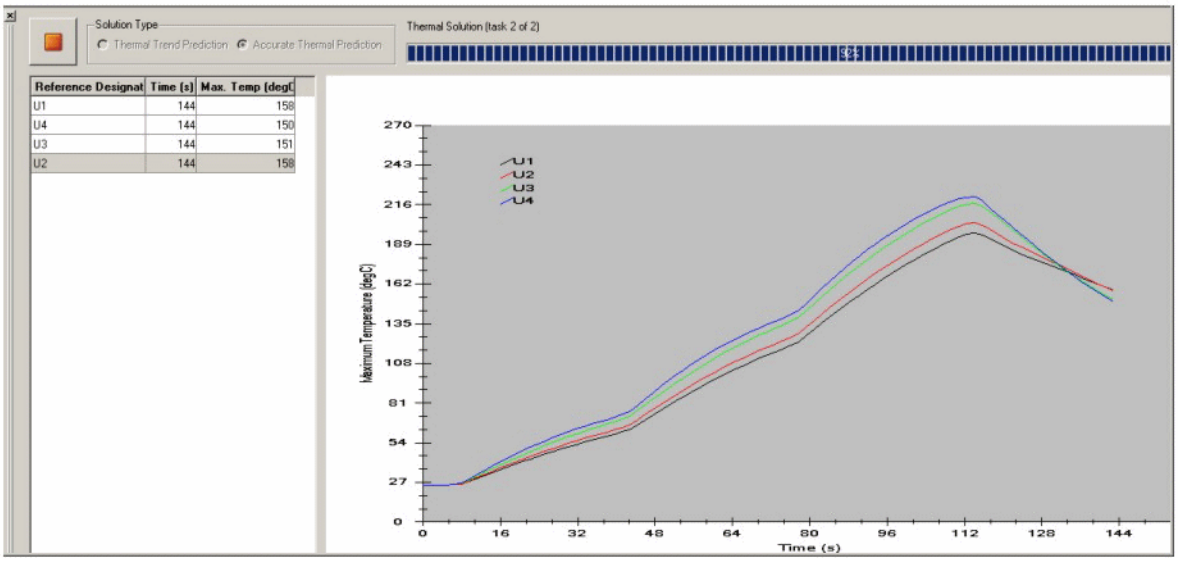
The display of the temperatures provides a quick check on the validity of the model without having to wait until the full solution completes.

Note that, for reflow oven environments, the maximum temperature of each component at each time step is displayed.

Special Display for Reflow Oven Environments

Note that, when solving reflow oven environments, the component Maximum Temperature versus Time profile is a plotted.

Figure 6-6. Reflow Oven Profile



Display of Results

After solution completes, the solution docker is removed and the results configuration is displayed to show a table of component temperatures. See “Viewing the Results” on page 89.

Solution Bound to Model

After solution, the results remain until changes in the model cause an inconsistency between the set-up and results, then the results become inaccessible. Selecting **Edit** > **Undo** undoes the change and restores the results.

The solution results remain consistent, and therefore accessible, for changes in:

- Object name
- Reference designator
- Board name
- Object notes
- Creation or deletion of a connector
- All data associated with connectors
- Creation or deletion of a keepout region
- All data associated with a keepout region
- Tj_max, Tc_max
- Creation of a new functional group via the toolbar or component popup menu
- Functional group reference designator, function name, size by bounding box check box

Solution Processing Messages

When a thermal solution is started, FloTHERM PCB checks for restricted object overlapping conditions, out of physical bounds data entries, and so on. These **messages** are issued either as warnings, allowing a solve to proceed, or errors that deny a solution.

This section contains the FloTHERM PCB messages that may appear during solution and offers instructions about what to do next. The names of the objects causing the problems are indicated in brackets.

Note

The error or warning messages are environment specific. That is, they will only appear when relevant to the current solution. You may find swapping environment types causes different messages to be displayed when solving.

Warnings

There are no components in the model

No components have been added to the layout. Add components to the board layout, using the Functional and Layout modes. See “Defining a Functional Layout” on page 21, and “Defining the Board Layout” on page 31.

If there are functional groups defined with a specified power then a solution would provide average board temperatures.

There is no power specified

No power has been supplied to the model, so there is nothing to be gained by solving.

Power is added to components or functional groups using the properties sheets, see “Defining the Board Layout” on page 31.

All powers will be derived

Only detailed or resistor network components are used in the model and their powers have been set to ‘?’ in the properties sheet. Their power dissipation will be derived during solution. See “Derive Power Dissipation for a Component at Fixed Tj”.

There are more than 100 non-filtered components in the model

This may result in long thermal solution times. To reduce the number of non-filtered components, increase the tolerance values in the “Component Filter Options Dialog” on page 55.

The board is greater than 1 metre in size (Board Name)

The dimensions of the named board is outside the expected size range. Change the board size using the properties sheet. See “Using Property Sheets” on page 42.

The board is greater than 1 cm thick (Board Name)

The dimensions of the named board is outside the expected size range. Change the board size using the properties sheet. See “Using Property Sheets” on page 42.

The environment inlet speed is greater than 20m/s

The inlet speed is higher than the normal operating range. Correct the inlet speed using the environment properties sheet. See “Changing the Environment Using the Properties Sheet” on page 66.

The environment inlet temperature is greater than 100˚C

The inlet temperature is higher than the normal operating range. Correct the inlet temperature using the environment properties sheet. See "Changing the Environment Using the Properties Sheet" on page 66.

Plate Fin Heatsink not aligned with Air Flow (Heatsink Name)

The air flow direction, which can be in either the + or -Y direction, is displayed in the environment definition mode and set using the environment properties sheet. See "Changing the Environment Using the Properties Sheet" on page 66.

However, to ensure effective cooling, it is advisable to rotate the heatsink so as to align with the flow direction. To rotate the heatsink, select it in the board creation mode and change the Z Rotation in the heat sink properties sheet. See "Heatsinks" for a description of the property sheet.

Thermal Via extends outside component (via name, component name)

The size of the thermal via array is larger than the footprint of the component. As thermal vias are designed to enhance heat transfer from under a package, it is not normally the case that they extend beyond the package.

Heatsink has fin width less than 0.5 mm (Heatsink name)

The default heatsink fin width is calculated to be one-tenth of the height of the fin. Extruded heatsink fin widths are usually limited to more than 0.5 mm due to manufacturing constraints. If the heatsink height is such that the resulting fin width is less than 0.5 mm, then this message indicates that the fin width might need to be manually modified in the Heatsink property sheet. See Heatsinks for a description of the property sheet.

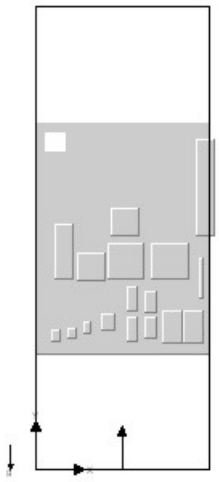
You have requested an accurate solution but there are only 'Simple' components. You are advised to add thermal resistor network, or FloTHERM PACK Detailed, components that warrant an Accurate solution

Simple components are the best representation that can be constructed when little or no detailed component information is available. They are best suited to performing comparative 'Trend' studies. An 'Accurate' solution is best performed when there are component representations capable of predicting accurate information, i.e thermal resistor network or detailed.

Unpowered component extends outside defined environment

Often IDF components representing connectors overlap the edge of the board. A solution is allowed if a component is **Simple**, has zero power and spans the edge of the solution domain, but a warning is issued in case this overlap is not caused by a connector.

Figure 6-7. Connector Overlap of Board Edge



However, if the warning is caused by bad positioning, then reset the location of the named part using the property sheets, see “Using Property Sheets” on page 42, before resolving. The location of the part in respect of the solution domain, can be viewed in the environment mode. See “Defining the Environment” on page 65.

Filtered Component extends outside defined environment (Part Name)

Although it will be ignored by the thermal exclusion, the component does extend beyond the bounds of the environment. Check this is intended.

Top Layer is not on the top surface of the board (Layer Name, Board Name)

Generally, this is not a commonly encountered board stackup configuration. The top layer should align with the top of the board.

Bottom Layer is not on the bottom surface of the board (Layer Name, Board Name)

Generally, this is not a commonly encountered board stackup configuration. The bottom layer should align with the bottom of the board.

Functional Group is larger than board (Functionalgroup name)

Board size is not sufficiently large to accommodate the functional group. Either decrease the size of the functional group or increase the size of the board.

Air Temperature less than Nearest Solid Temperature in Conduction Cooled Environment

Normally the air temperature is greater than the nearest solid temperature. Check this is intended.

Temperature of chassis is greater than air temperature (Chassis Name)

Normally the chassis temperature is less than the air temperature. Check this is intended.

There is no Conduction Cooling defined

The board is modelled as suspended in free air. To achieve conduction cooling, define at least one board edge, surface or component surface as conduction cooled.

Cutout extends outside defined environment (Part Name)

Although this has no effect on the results, a consistent definition of the cutouts would be where they are wholly within or internally abutting the board perimeter.

Component not fully inside user defined Functional Group Extents (Component Name, Functionalgroup Name)

A user-sized functional group defines an area of the board surface with which all components of the functional group should reside.

Errors

Cannot solve with more than 4 components in DEMO MODE

Reduce the number of components, or upgrade to a full license.

Board needs at least one layer (Board Name)

Layers represent the structure of a board, so there must be at least one layer per board.

To add a layer, select the board and either click the Create new Layer icon or change the number of layers in the board properties sheet. New layers are added to the center of the board. See “Layers”.

Layer is thicker than board (Layer Name, Board Name)

The layer cannot be thicker than the board, so change the thickness of either the layer or the board using the property sheets. See “Using Property Sheets” on page 42.

Sum of Layer thicknesses is thicker than board (Board Name)

The sum of the layers cannot be thicker than the board, so change the thickness of either the layers or the board using the properties sheets. See “Using Property Sheets” on page 42.

Part is not on the board (Part Name, Board Name)

The named part has been moved off the board. Relocate the part back onto the board either by dragging or using the property sheet in Board layout mode. See “Using Property Sheets” on page 42.

Layer Patch is not completely on the board (Patch Name, Board Name)

Reset the location of the patch using the property sheet. See “Layer Patches”.

Layer is not completely on the board (Layer Name, Board Name)

Reset the location of the layer using the property sheet. See “Layers”.

Part is not on the board - it is over a Cutout (Part Name, Cutout Name)

Either remove the cutout or move the part until it is part of the board. See “Rectangular Cutouts”.

Parts overlap each other (Part Name, Part Name)

FloTHERM PCB cannot solve for overlapping parts. Relocate the parts using the property sheets. “Using Property Sheets” on page 42.

Part is larger than board (Part Name)

The part is larger than its parent board. Resize the part using the property sheets. See “Using Property Sheets” on page 42.

Component has inappropriate Junction To Case Resistance (Component Name)

Check the settings in the component property sheet. See “Rectangular Components”.

Component has inappropriate Junction To Board Resistance (Component Name)

Check the settings in the component property sheet. See Rectangular Components.

Part extends outside defined environment (Part Name)

Reset the location of the named part using the property sheets, see “Using Property Sheets” on page 42, before resolving. The location of the part in respect of the solution domain, can be viewed in the environment mode. See “Defining the Environment” on page 65.

Heatsink is not on the component (Heatsink Name, Component Name)

The named heatsink has been moved off the component. Relocate the heatsink back onto the component either by dragging or using the property sheet in Board layout mode. See “Using Property Sheets” on page 42.

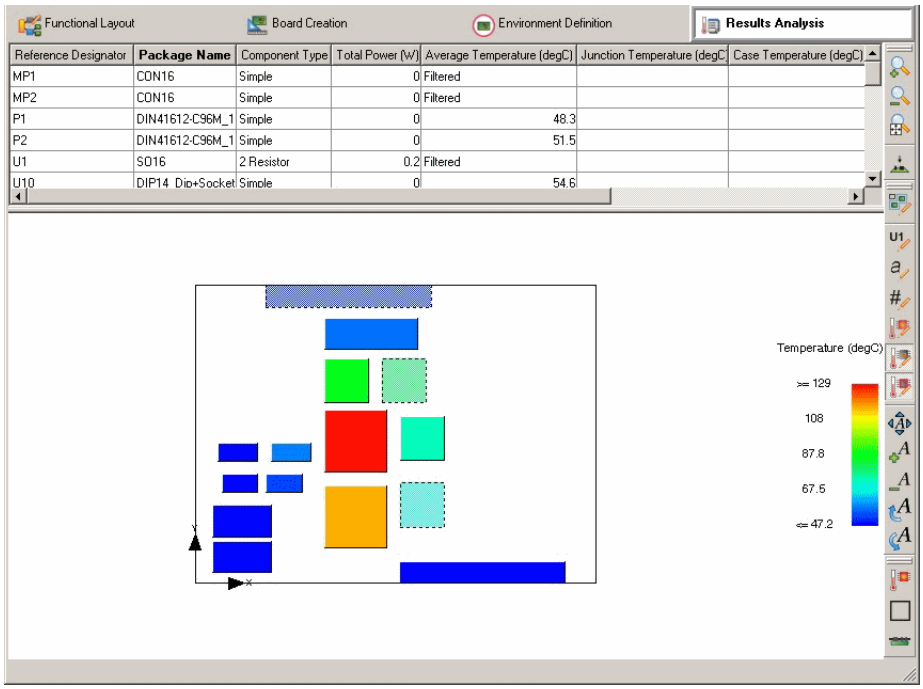
Related Topics

- "Viewing the Results" on page 89

Viewing the Results

If the results are not on display, then click the **Results Analysis** button. The results configuration is similar to that shown below.

Figure 6-8. Tabular and Graphical Results Display



The results are displayed in tabular and graphical formats.

Tabular Results

A table of results appears above the Display Area. The tables can be sorted by clicking the column headings. Click once to sort the table in descending order of the value selected, click again to sort in ascending order.

For a description of the table, see Results Data Table in *Getting Started with FloTHERM PCB*.

Graphical Results

The graphical view of the results appears below the results table. The results are represented by a color coding over the geometry. The color key is also displayed. The default legend scaling

option can be changed by choosing **Options > Legend** and using the "Legend Options Dialog" on page 92.

The view can be modified using the icons in the right toolbar, see Results View Controls in *Getting Started with FloTHERM PCB*.

Related Topics

- "Comparing Results" on page 90

Comparing Results

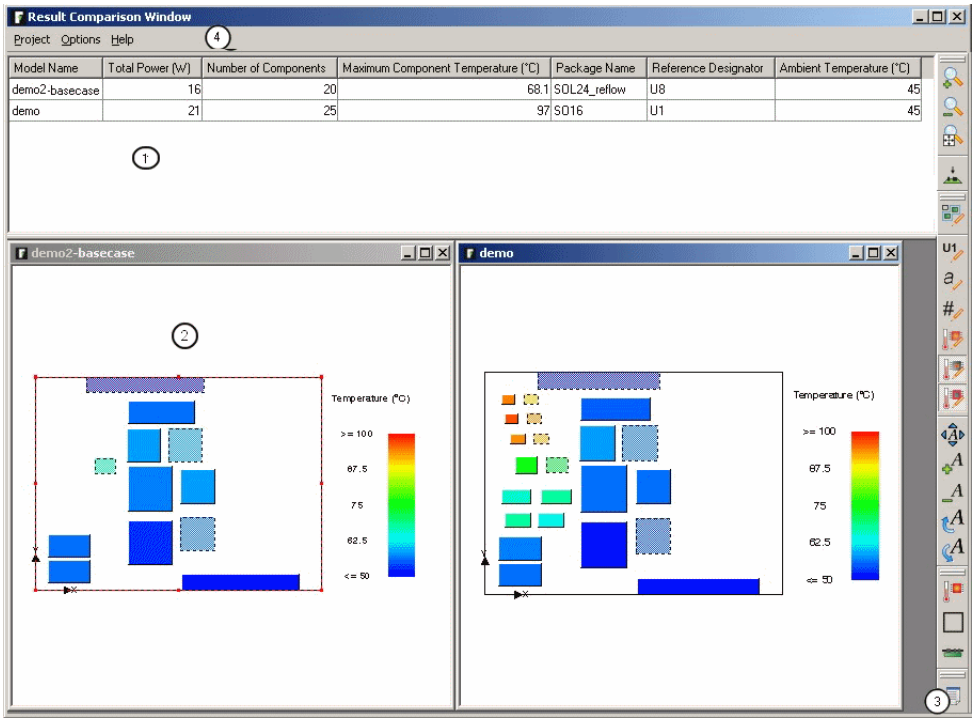
The results of this project can be compared with those from previous saved FloTHERM PCB project solutions. To compare the results, select **Tools > Compare Results** to launch the Results Comparison window.

Result Comparison Window

The Results Comparison window is capable of displaying multiple plots of solution results so that you can compare the graphical results with those of previous saved projects.

The window consists of a table of results, as well as the graphic output, for each project selected for comparison by selecting **Project > Open New**. The main window features are shown in Figure 6-9. The current project has **-basecase** appended for identification.

Figure 6-9. Results Comparison Window

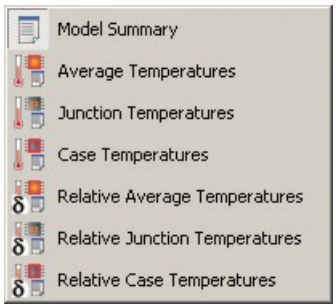


1. See “Results Table” on page 91
2. See “Results Plotting Area” on page 91
3. See “Plot Viewing Controls” on page 92
4. See menu bar for “Project Menu” on page 92 and “Options Menu” on page 94

Results Table

The results table contains entries for all the projects opened for comparison. There is one row for each project, and the type of information displayed is controlled by the Table Type icon. Clicking the Table Type icon opens a menu, see Figure 6-10.

Figure 6-10. Table Type Popup Menu



Model Summary is the total power and maximum component temperature.

The Table Type icon changes to show the table type selected.

Note that, like the main results table, the tables can be sorted by clicking the column headings. Click once to sort the table in descending order of the value selected, click again to sort in ascending order. Also, clicking a component in the table highlights it in the plot below, and vice versa.

Related Topics

- “Comparing Results” on page 90

Results Plotting Area

The results plotting area displays 2D color coded plots for each of the projects opened. Each plot is drawn in a separate panel (or sub window) with standard window controls. By default, the plot panels are tiled within the display area and their size is adjusted, the Project menu allows you to tile or cascade the panels.

Related Topics

- “Comparing Results” on page 90

Plot Viewing Controls

The plot views can be modified using the right toolbar which contains movable button panels for changing the view direction, zooming, annotating and coloring the components as well as displaying plane plots.

These plot controls are the same as those for the main solution configuration. See Results View Controls in *Getting Started with FloTHERM PCB*. The changes made in the viewing controls are applied to all the plots in the Results Comparison Window.

Related Topics

- "Comparing Results" on page 90

Legend Options Dialog

To access: **Options > Legend**

Use this dialog to choose the range of temperatures to be plotted in the **Results Analysis** and **Result Comparison** windows. The is useful for increasing the visibility of differences over a narrower range.

To change the value range, overtype the data entry fields and check **Use User-defined Options**. The units are changed by clicking the units box and choosing an option from the popup menu.

Uncheck **Use User-defined Options** to use the default range, which is determined from the coldest and hottest objects on display.

Result Comparison Window Menus

Project Menu

Table 6-1. Result Comparison Window Project Menu Options

Option	Description	Shortcut
Open New	Launches a file selection to choose the *.flopcb file for comparison. After selection, the project data appears in the Results Comparison Window.	
Print	Launches the system Print dialog to obtain a postscript representation of the drawing area and either output it to a printer or save it as a disk file.	Ctrl-P
Save Image(s)	Launches a file selection to save the display area screen image as *.gif file.	

Table 6-1. Result Comparison Window Project Menu Options

Option	Description	Shortcut
Tile	Displays all plot panels in a uniformly tiled arrangement, filling the whole of the plot window, see Figure 6-11.	
Cascade	Displays all plots uniformly sized and on top of each other slightly displaced relative to each other, see Figure 6-12.	

Figure 6-11. Tiled Plot Panels

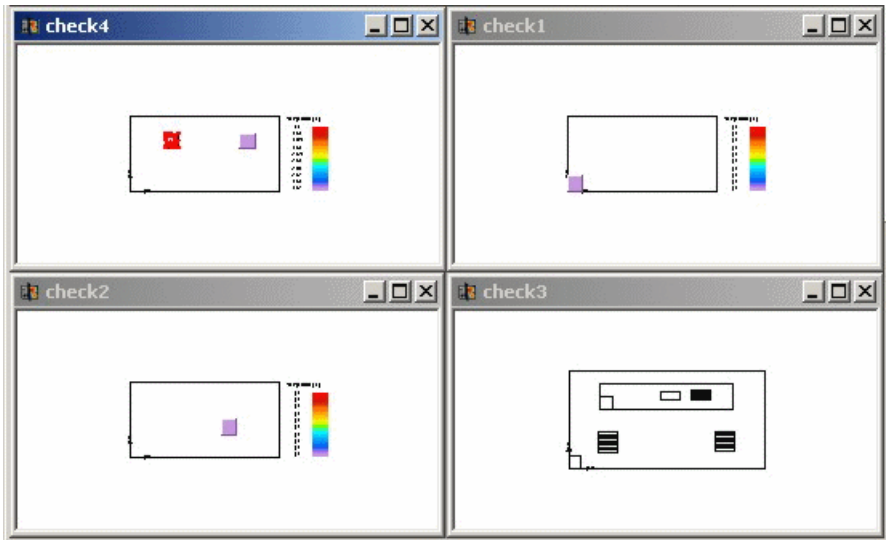
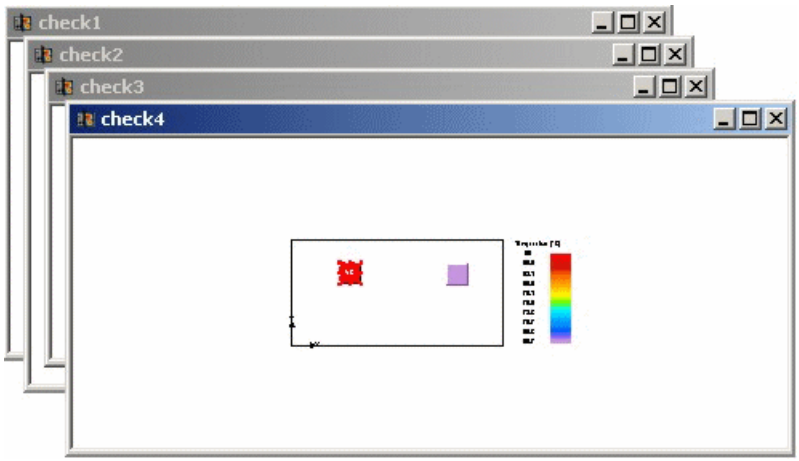


Figure 6-12. Cascaded Plot Panels



Options Menu

Table 6-2. Result Comparison Window Options Menu Options

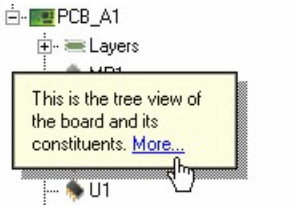
Option	Description
Legend	Launches the Legend Options Dialog.

Help Menu

Table 6-3. Result Comparison Window Help Menu Options

Option	Description
Contents	Launches the online help system from the top.
Help On Window	Launches the on-line help system from the description of the Results Comparison Window.
Help on	Launches the GUI context-sensitive links for the on-line help. Click the mouse pointer over the area of interest to display help text, see Figure 6-13. If this is not enough help then click More to go to the relevant section of the on-line help.

Figure 6-13. Context-Sensitive Help Popup



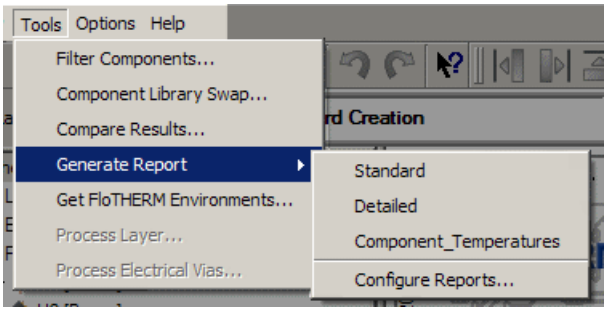
Chapter 7
Generating Reports and Screen Dumps

This chapter describes how to generate HTML reports and screen dumps for a FloTHERM PCB project.

Displaying Reports

To display reports select **Tools** > **Generate Report** and choose the type of report from the submenu.

Figure 7-1. Report Generation Menu



There are three supplied reports:

- **Standard** - a report containing general details of the project including the environment, board summaries, functional groups and plots. Plots include a top view of the board layout, a functional block diagram, average temperature results, and junction temperature results.
- **Detailed** - as the Standard report but containing more detailed information and the following extra plots: a bottom view of the board layout, components coloured by power, the environment, failed junction temperature results, the temperature plane in the top layer, and the velocity plane 3mm above the board.
- **Component_Temperatures** - a report containing a table of components and their temperatures, see Component_Temperatures Configuration.

You can also configure your own reports using the Report Configuration Dialog. When new reports have been configured they can be selected from the drop-down menu (Figure 7-1).

All reports can be saved as HTML files. The Component_Temperatures report can also be saved as a CSV file.

In addition, any HTML report containing a single table may be output as a CSV file, for example, a component section, or the table for a single layer stack.

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.

Report Viewer

Access: **Tools > Generate Report** > *Report*

Use the **Report Viewer** to display a report on the current FloTHERM PCB project. The reports are displayed in a scrollable, re-sizeable window.

To save the contents of the window in an *.html* file, click **Save** and use the file browser. Alternatively, click **Print** and output the report to a printer.

Report Layout

The report layout is split into six main sections for the following data types:

- **Summary** — general project information such as project name, solution status and history notes.
- **Environment** — environmental conditions such as air flow, gravity, altitude
- **Board Summary** — board data such as size, number of components, power output
- **Functional Groups** — functional group data such as name, size, power output
- **Components** — component details
- **Plots** — pictorial representations of the project data and results plots. The results plots available depend on the environment type.

  For all but reflow oven environments, the results plots can include: 2D views of the board layout, functional block diagram, environment, plots of average temperature results, junction temperature results, failed junction temperature results and plots of temperature in the top board layer as well as a vector plot of velocity in a plane above the board.

  For reflow oven environments, the results plots can include: a map of the heating and cooling zones, as well as plots of over/under temperature, largest maximum temperature, largest minimum temperature and a temperature profile.

But the actual information displayed within these sections is defined using the “Report Configuration Dialog” on page 97.

Standard and Detailed Configurations

The default **Detailed** report configuration contains all the above information, but the **Standard** report does not include the full component details or plots of components colored by power, environment, failed Junction temperature results and plots of temperature in the top board layer or the vector plot of velocity in a plane above the board.

For further description the data available for reporting, see “Report Configuration Dialog” on page 97.

Component_Temperatures Configuration

The default **Component_Temperatures** report configuration contains component reference designators and part numbers, dissipated power, average/junction/case temperatures, and temperature limits.

The report is available in Batch mode by implementing the **-savecomponentresults** option, see “Starting and Running FloTHERM PCB in Batch Mode” in *Getting Started with FloTHERM PCB*.

Report Configuration Dialog

To Access: Tools > Generate Reoprt > Configure Reports

Use the **Report Configuration** dialog to define the configuration of reports that are generated using **Tools > Generate Report**. Existing report configurations can be edited and new ones created.

The dialog consists of two main sections:

- A list containing all the report configurations known to the project
- Data selection options for the report chosen in the list

Report Types

The configurations listed under **Reports** indicate the types of reports that can be generated using the **Tools > Generate Report** command. This list is replicated in **Tools > Generate Report** selection menu.

By default, the **Standard** and **Detailed** report configurations are listed and can be edited. Selecting a report in the list displays the data selection options on the right-hand side of the so you can choose the type of information to be included when the report is generated.

The buttons controlling the **Reports** are:

- **New** — calls the **New Report** to name and create a new configuration. The new configuration name appears in the list highlighted ready for you to set the data options. The default configuration for a new report is completely blank.
- **Copy** — calls the **New Report** to create another copy of the configuration selected in the **Reports** list. This new configuration is highlighted ready for you to change the data options. This is useful when you want to set up a separate report template that has only small changes from the supplied configurations.
- **Delete** — deletes the selected configuration from the list.

Data Selection

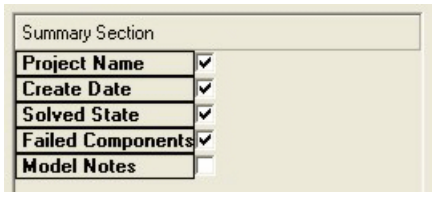
When a configuration is selected in the **Reports** list, the right side of the dialog updates to display the selection data options.

The selection data is accessed via six mode buttons mirroring the report sections. Click the mode button to display the data options for that report section. To include the listed information type in a report, ensure the check box is activated.

Summary Section

The **Summary** section is displayed when a configuration is first selected and can contain the following project status information.

Figure 7-2. Summary Section



Environment Section

The **Environment** section can contain the following information.

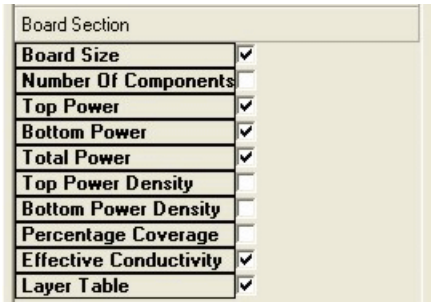
Figure 7-3. Environment Section

The **Environment Definition** includes the settings for the environment type (that is, Standard still Air, Standard Forced Air, Simple Card Slot, or Conduction Cooled).

Board Section

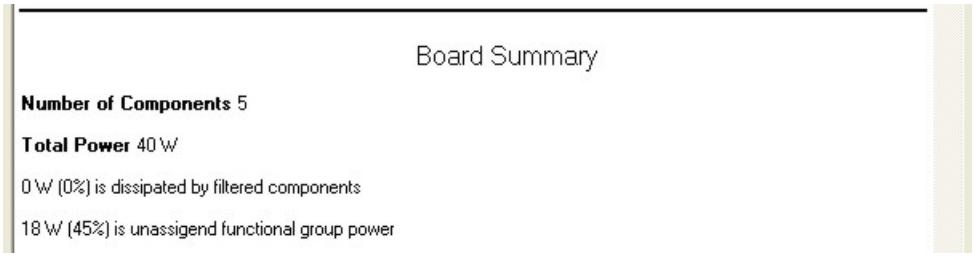
The **Board** section can contain the following information.

Figure 7-4. Board Section



Total Power requests a summary of the total power, that is the total power, how much power is dissipated by filtered components and how much power defined from a functional group is not assigned to components.

Figure 7-5. Total Power Summary



Percentage Coverage is the percentage coverage of components compared to the available area for placement. It is calculated as the total area of components on a board side divided by the total board area.

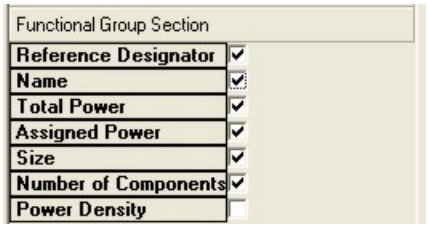
Percentage Coverage is reported for both sides, and if the value is greater than 100, then it is physically impossible to place the components on the defined board side.

Effective Conductivity reports the in-plane and through plane effective thermal conductivities of each board. These are the values used when a **Trend** solution is performed.

Functional Group Section

The **Functional Group** section can contain the following information.

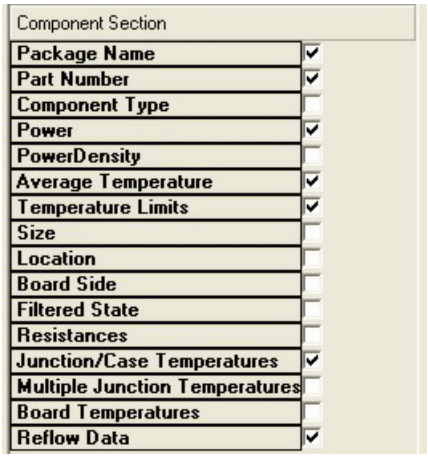
Figure 7-6. Functional Group Section



Component Section

The **Component** section can contain the following information.

Figure 7-7. Component Section



Plot Section

The Plot section contains the list of plots to be added to the report. **Standard** reports include plots of:

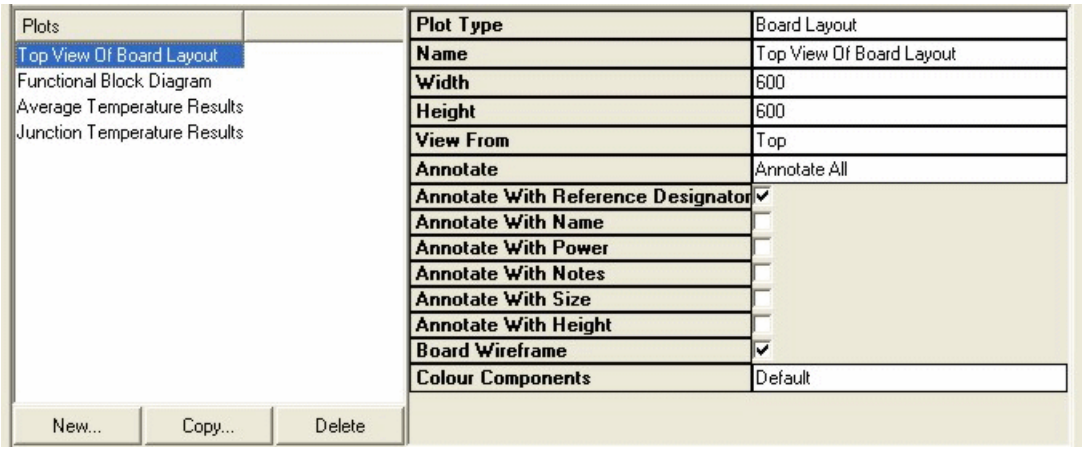
- Top View of Board Layout
- Functional Block Diagram
- Average Temperature Results
- Junction Temperature Results

Detailed reports contain the additional plots:

- Bottom View of Board Layout
- Components Colored by Power
- Environment
- Failed Junction Temperature Results
- Temperature Plane in Top Layer
- Velocity Plane 3mm Above Board

Selecting a plot in the list displays the data options for that plot type. The figure below shows the options for the **Top View Of Board Layout**.

Figure 7-8. Plot Section



Note

If the data options are not visible, then increase the width of the Report Configuration.

Choose the data to be reported by editing the data options in the plot property sheet.

To add a new plot choose **New** and use the property sheet to define the report. Alternatively, use **Copy** to duplicate a plot definition. **Delete** deletes the plot from the report.

The plots will be added to the bottom of the report in the order listed.

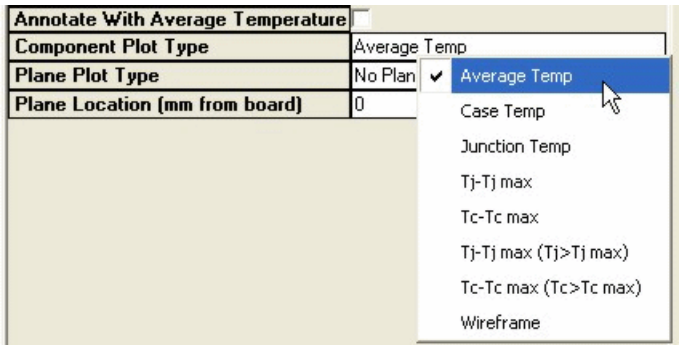
Plot Types

The types of plot available are:

- **Functional Block Diagram** — a block diagram showing the functional connections between the electronics
- **Board Layout** — a 2D or 3D view of the board physical layout
- **Environment Plot** — one 2D environment view showing the gravity and air flow directions
- **Results Plot** — for displaying the results in a variety of formats

The results can be displayed over a temperature or velocity plane for the following plot types.

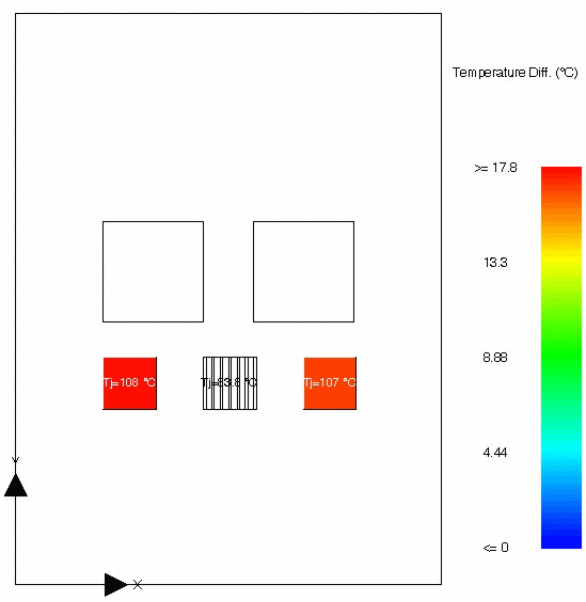
Figure 7-9. Component Plot Types



Examples of Plot Types

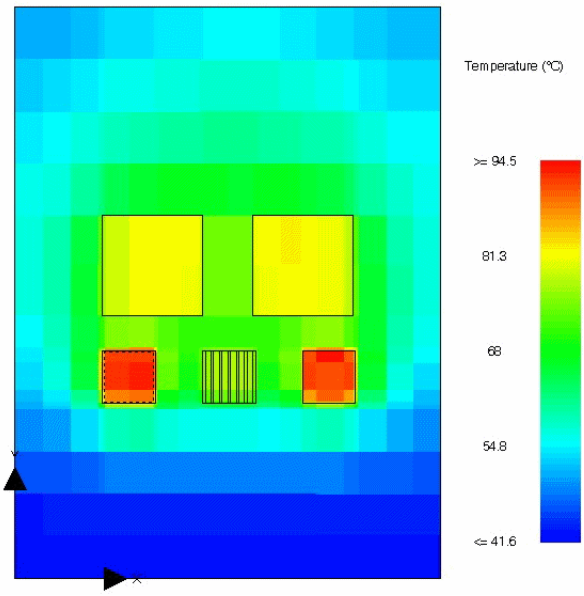
Example of plot of failed junction temperatures.

Figure 7-10. Failed Junction Temperature Plot



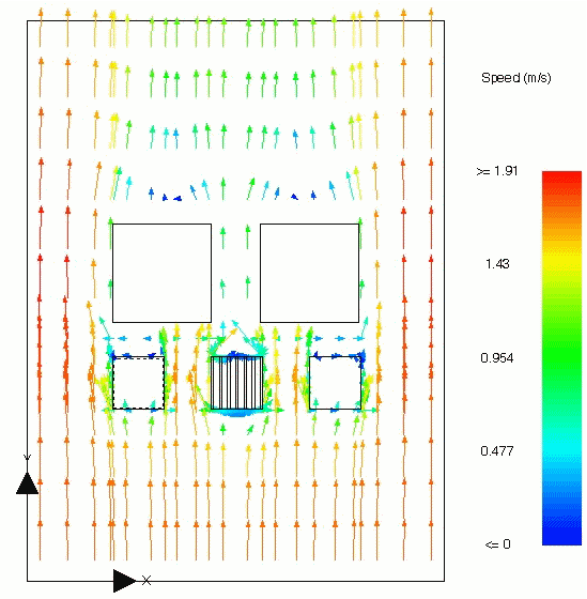
Example of a plot of temperature in the top board layer.

Figure 7-11. Top Board Layer Temperature Plot



Example output of velocities in a plane over the board

Figure 7-12. Velocities Over a Board



Generating Reports

After configuring the report, generate the report by selecting **Tools > Generate Reports**.

Related Topics

- “Report Viewer” on page 96

Creating Screen Dumps

A screen dump (or capture) of the display area can be obtained in GIF format by selecting **File > Save Image** and choosing the file name and destination in the file browser.

Appendix A
Frequently Asked Questions

This chapter lists questions and answers of common queries that you may encounter when using FloTHERM PCB. The questions are numbered and grouped below. Click on a question to see the answer.

Functional Block Diagramming

1. What does the functional layout tab do?
2. What is the benefit of the integration between the functional block diagram and the physical layout?

Board Definition

3. Can FloTHERM PCB model DaughterBoards mounted perpendicularly to a MotherBoard (for example, Memory SIMMS/DIMMS)?
4. What does the percent coverage represent in the property sheet for a board layer?
5. How do I know what values to input for the percent coverage?
6. Can I define areas of varying percent coverage in a board layer?
7. How are the layers positioned in the board?
8. How do I set the modelling level of the board prior to saving to a library for import into FloTHERM?

Component Definition

9. Is there another method to import board layout information other than using File > Import > Import IDF?
10. How do I change the material of my heatsink?
11. How do I determine the total power of all filtered components?
12. How do I replace multiple components with a component representation from the library?
13. How do I assign the same power to multiple components?
14. What type of package thermal models does FloTHERM PCB support?
15. How can I construct a detailed component model in FloTHERM for subsequent import into FloTHERM PCB?
16. Why can I only resize some of my components?

17. How do I import a component generated by FloTHERM PACK into FloTHERM PCB?
18. How do I sort my components based on size, power, and so on?
19. What different methods are there for specifying component power dissipation for a layout?
20. How can I import two resistor component information from a spreadsheet into FloTHERM PCB?
21. Why can't I move or resize components using the mouse?

Layer Definition

22. How can I see patches on internal layers?

Board Environment

23. How should I define the board's environment?
24. Using Tools > Get Flotherm Environments, why can't I select a region containing a detailed description (for example, assembly of cuboids, and so on) of a PCB?
25. I wish to use lfm instead of m/s for velocities in all my projects. Can I change the user setting to be lfm by default?

Solution

26. What is the difference between thermal trend prediction and accurate thermal prediction?
27. How do I model a board with the heat assumed spread over the board surface?
28. How can I change the solution type once results are available?
29. Can I interrupt a solution and continue it in a later FloTHERM PCB session?
30. Does FloTHERM PCB automatically handle radiation?
31. What temperature will the object in FloTHERM PCB radiate to?
32. How do I accelerate my solution without compromising the accuracy of my results?
33. What should I send to Mentor Graphics in case of a support problem?
34. Does FloTHERM PCB perform any model integrity or error checking?
35. Why am I unable to initiate a thermal solution within FloTHERM PCB?

Results Analysis

36. Can't the thermal board analysis be done in FloTHERM?
37. What is the difference between a Standard and Detailed report?
38. Why do my results often become unavailable?
39. Can I export the thermal results?

List of FAQs

Functional Block Diagramming

1. **What does the functional layout tab do?**

   The functional layout capability allows you to generate a logical representation of functional partitions and interconnections for the board, prior to defining the physical layout. All of the critical aspects of a functional block diagram are available including the ability to create functional groups, discrete components, and connectors that are assigned a directionality, number of interconnects, speed, and technology.

   Functional layouts are used to communicate the functional intent of a product to a wide range of people involved in product development such as marketing, electrical and mechanical engineers and customers.

   See Functional Layout in *Getting Started with FloTHERM PCB* for a description of the layout tab.

2. **What is the benefit of the integration between the functional block diagram and the physical layout?**

   Any functional group defined has both a representation in the functional block diagram and also in the physical layout. When a functional group is saved to a library both functional and physical aspects of its definition are retained. When the functional group is retrieved from the library both the functional description, including interconnects, and the physical description, in terms of component definition and placement, is imported. This enables new designs, or modifications to existing designs, to be rapidly defined both functionally and physically.

Board Definition

3. **Can FloTHERM PCB model DaughterBoards mounted perpendicularly to a MotherBoard (for example, Memory SIMMS/DIMMS)?**

   Yes, FloTHERM PCB **DaughterBoards** can represent mezzanine, parallel mounted boards or perpendicularly mounted boards.

   See DaughterBoards in the *FloTHERM PCB Modeling Objects Reference Manual*.

4. **What does the percent coverage represent in the property sheet for a board layer?**

   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.

5. **How do I know what values to input for the percent coverage?**

   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.

6. **Can I define areas of varying percent coverage in a board layer?**

   Yes, DaughterBoards can be defined attached to a selected board layer. These allow for rectangular non-overlapping regions of differing %coverage or even a different material. The default % coverage of 100% is suited for representing copper pads. See Layer Patches in the *FloTHERM PCB Modeling Objects Reference Manual*.

7. **How are the layers positioned in the board?**

   By default, layers are evenly spaced through the width of the board. This can be overridden by first checking off **Equispaced** in the layers 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 in the *FloTHERM PCB Modeling Objects Reference Manual*.

8. **How do I set the modelling level of the board prior to saving to a library for import into FloTHERM?**

   Use **File > Solve** to open the Thermal Solution Docker and set either **Thermal Trend Solution** or **Accurate Thermal Solution**, then close the dialog by pressing the  button. Now save the board to a library by right-clicking the **MotherBoard** and selecting **Save To Library** from the popup menu.

   If **Thermal Trend Solution** is set, then the board will be represented by a single object with orthotropic thermal conductivity. If **Accurate Thermal Solution** is set then each layer in the board is individually represented and the grid definition will be much finer.

Component Definition

9. **Is there another method to import board layout information other than using File > Import > Import IDF?**

   Yes, a CSV (comma separated variable) file can be imported using the **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.

10. **How do I change the material of my heatsink?**

    By default, heatsinks 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 heatsink property sheet, to bring up the Material Library Selector Dialog. Navigate to the new material and click "**+**" to make it available. Then **OK** the Material Selector dialog.

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

If the required material does not already exist in a library a new material can be created from within the Material Library Selector Dialog by first selecting a library then pressing the **New** button.

11. **How do I determine the total power of all filtered components?**

    This information is included in the board section of a report created by **Tools** > **Generate Report**.

12. **How do I replace multiple components with a component representation from the library?**

    Use **Ctrl+click** to multiple select the components you want to replace. Then access the popup menu and select **Replace with library item** Navigate to the required library item then double click. Also see FloTHERM PCB Context Sensitive Menus in *Getting Started with FloTHERM PCB* for a reference to **Replace with library item**.

13. **How do I assign the same power to multiple components?**

Use **Ctrl+click** to multiple select the components you want to assign the same power to. Note that the **property sheet** indicates that multiple components are selected. See Property Sheets in *Getting Started with FloTHERM PCB*. 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.

14. **What type of package thermal models does FloTHERM PCB support?**

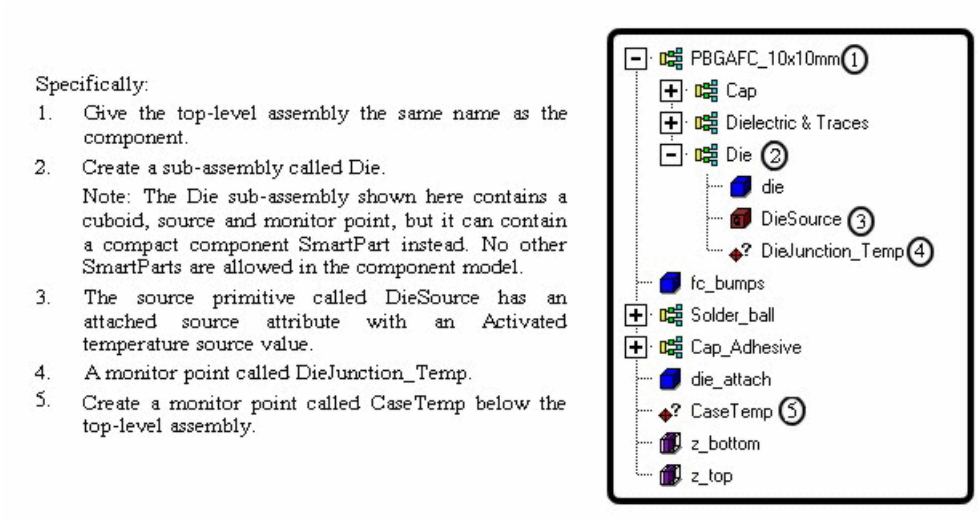
    FloTHERM PCB supports the following package models:

    - 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

15. **How can I construct a detailed component model in FloTHERM for subsequent import into FloTHERM PCB?**

    The assembly structure and naming convention should be the same as that used by FloTHERM PACK for detailed components.

Figure A-1. Constructing a Detailed Component Model



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.

Note that this approach can be used to create a detailed representation of any type of component, but in FloTHERM PCB it will be presented as if there exists a Junction and a Case Temperature.

16. **Why can I only resize some of my components?**

    Components of Simple **component type** can be resized either graphically or by using the property sheet as shown in Rectangular Components in the *FloTHERM PCB Modeling Objects Reference Guide*. 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.

17. **How do I import a component generated by FloTHERM PACK into FloTHERM PCB?**

    If a single PDML (**.pdml*) file is downloaded then this file should first be placed on the file system. Then, within FloTHERM PCB, 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 FloTHERM PCB as explained in “Adding Existing Geometry Using the Library” on page 37.

    Libraries of multiple objects downloaded in .library format can be imported using the import function, see “Using the Libraries” on page 57.

18. **How do I sort my components based on size, power, and so on?**

A tabular view of the component layout can be viewed by clicking on the **Component Table** tab in Board Creation mode (see Component Tables in *Getting Started with FloTHERM PCB*). 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 underside of the board after having sorted on board side.

19. What different methods are there for specifying component power dissipation for a layout?

    A power list can be imported at any time to reset power dissipations, see "Importing a Power List" on page 50. 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 FloTHERM PCB. Note that this power is for each library item and not for the unique instance of that component on the board.

20. **How can I import two resistor component information from a spreadsheet into FloTHERM PCB?**

    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.

21. **Why can't I move or resize components using the mouse?**

    Components can only be moved or resized using the mouse if the Lock Component Move icon is off. The Lock Component Move 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

22. **How can I see patches on internal layers?**

    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.

Board Environment

23. **How should I define the board's environment?**

    The environment that you define in FloTHERM PCB should be a reasonable representation of the air flow and thermal conditions that you expect the board to be subjected to in operation.

    There are six options for defining the environment in FloTHERM PCB: **Simple Card Slot**, **Standard Forced Air**, **Standard Still Air**, **Conduction Cooled**, **Reflow Oven** and **FloTHERM Environment**. Choose the setting that best represents your design.

    Simple Card Slot

    This option will place the board design in a card slot. Choose this option if the actual board is intended to be placed in a card cage. This option requires that you know approximately the inlet conditions to the card slot.

Information required:

- Flow Specification: enter a volume flow rate or an inlet velocity
- Inlet Temperature: Enter the air temperature entering the slot
- Pitch distance: Enter the distance between the cards in the card cage

FloTHERM PCB will assume a uniform velocity and temperature distribution entering the card slot from the direction that is specified.

Standard Forced Air

This option will place the board in the JEDEC (www.jedec.org) standardized forced air test. This option is suitable if you do not know the exact operating environment but want to compare the performance of the board in a standardized forced convection cooling environment.

Information Required:

- Flow Specification: Enter the expected upstream air velocity
- Ambient Temperature: Enter the surrounding air temperature

Standard Still Air

This option places the board in the JEDEC (www.jedec.org) still air test. This option is suitable if you do not know the exact operating environment but want to compare the performance of the board in standardized natural convection conditions.

Information Required:

- Ambient air temperature
- Direction of gravity
- Altitude: For applications operating at high altitude.

Conduction Cooled

This option places the board in a conduction cooled environment. The board can be cooled by conduction from the motherboard edges, the tops of components or areas on the board surface.

Information Required:

- Ambient air temperature: Enter the typical air temperature around the board (or within the sealed system)
- Nearest solid temperature: Enter a representative temperature of the surrounding chassis
- Side/edge to be cooled and its thermal contact resistance: Specify which side and/or edges of the board are to be conduction cooled
- Direction of gravity
- Altitude: For aerospace application enter the height above sea level.

Reflow Oven

This option will place the board inside a reflow oven.

To simulate the solder reflow process, you set up a number of heating and cooling zones in the oven, specifying their length, temperature and conveyer speed. The time the board spends in each zone is calculated from the conveyer speed and the zone length.

Information Required:

- Ambient air temperature
- Speed the board travels through the oven
- Temperature at which the solder will melt. This value defaults to 235˚C and is the same for all components.
- Speed across the heating zone
- Speed for the speed across the cooling zone
- Maximum clearance height between the board and surface and the perforated plate through which air is supplied
- Number of heating zones in the oven
- Length of each heating zone
- Number of cooling zones
- Length of each cooling zone
- Units of temperature used for calculation and display

- Temperature applied to top and bottom or top only of the board for each heating zone
- Temperature applied to top and bottom or top only of the board for each cooling zone

FloTHERM Environment

This option places the board in a detailed velocity and temperature field that is extracted from a user-specified FloTHERM solution. This option is suitable if:

- The board's operating environment doesn't fit well into the above choices, or
- A more detailed analysis is desired

Information Required:

An existing FloTHERM solution with a FloTHERM Region object completely enveloping a FloTHERM PCB SmartPart object. There is a set of rules that the desired FloTHERM solution must conform with, see "FloTHERM Regions Disallowed" on page 76.

Further Information:

For further information on the environments available, see "Changing the Environment Using the Properties Sheet" on page 66.

24. **Using Tools > Get Flotherm Environments, why can't I select a region containing a detailed description (for example, assembly of cuboids, and so on) of a PCB?**

    When a region is selected from a FloTHERM system level model in FloTHERM PCB (see Tutorial 5 - Selection of an Environment Calculated in FloTHERM in *FloTHERM PCB Tutorials*) there is the option of retaining the current FloTHERM PCB motherboard or to use the PCB defined in the system level model as a starting point for the FloTHERM PCB study. This latter option requires the PCB description used in the FloTHERM model to be converted into a form that FloTHERM PCB can represent. This can only be achieved if the FloTHERM PCB is defined using the PCB SmartPart. The former option requires there to be a PCB SmartPart in the FloTHERM model to enable the existing FloTHERM PCB motherboard to be correctly positioned in the selected environment.

25. **I wish to use lfm instead of m/s for velocities in all my projects. Can I change the user setting to be lfm by default?**

    Yes. The default installation is in SI units but you can change this by using **Option > Global Units** to launch the Global Units Dialog. Left click on the unit associated with velocity and choose the unit that you want, for example, ft/min. Click **OK** to close the dialog. From now on, any velocity defined will be in lfm.

Solution

26. **What is the difference between thermal trend prediction and accurate thermal prediction?**

    The thermal trend prediction results in fast solutions accurate enough to resolve trends when comparing different layouts. An accurate thermal prediction results in longer, more detailed solutions that are capable of producing accurate junction and case temperature predictions.

27. **How do I model a board with the heat assumed spread over the board surface?**

    Set all components to be **Filtered**. This is most effectively done using the Component Filter Options Dialog. Alternatively, if no components are specified, creating one or more Functional Groups, in the *FloTHERM PCB Modeling Objects Reference Manual* with powers defined results in the total amount of heat being smeared over the board surface. 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.

28. **How can I change the solution type once results are available?**

    The Thermal Solution Docker can be accessed by selecting **File > Solve**.

29. **Can I interrupt a solution and continue it in a later FloTHERM PCB session?**

    Yes, simply load a partially converged project and continue the solution using **File > Solve**. Change the solution type then start the new solution.

30. Does FloTHERM PCB automatically handle radiation?

    Yes, FloTHERM PCB will account for radiation to the surrounding environment in all the following cases.

    For FloTHERM PCB set environments:

    - Trend solution — Standard Still Air
    - Trend solution — Simple Card Slot and Standard Forced Air below 0.2 m/s
    - Accurate solution — All cases

    For Imported FloTHERM Environments:

    - Any FloTHERM environment in which radiation was active in the original FloTHERM model, regardless of the solution type.

31. **What temperature will the object in FloTHERM PCB radiate to?**

    In the case of a standard still air environment, all objects will radiate to the surroundings at the specified ambient temperature. In a simple slot or wind tunnel environment, all objects will radiate to the surroundings at the specified inlet temperature. In the case of a

FloTHERM environment, all objects will radiate to the temperature (air or solid) present at the selected region boundary.

32. **How do I accelerate my solution without compromising the accuracy of my results?**

    Thermally insignificant components can be **Filtered**. Filtered components are not deleted from the model. Their power dissipation is summed and modelled as smeared over the entire board surface, their physical obstructions are not resolved. Components that are small (for example, <2mm in length) AND have a very low power dissipation (for example, <0.1W) 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. Alternatively, component Power Density can be used to filter components in the Filter Component dialog.

33. **What should I send to Mentor Graphics in case of a support problem?**

    Use **File > Save** to save a *.flopcb* file that can be submitted to your local support office from http://supportnet.mentor.com/.

    If the project contains thermal results, the *.flopcb* file might be too large to submit. If this is the case then use **File > Export > Export PDML** to produce a .pdml file. This file contains a description of the model but contains no results and so is much smaller.

34. **Does FloTHERM PCB perform any model integrity or error checking?**

    Yes. When a thermal solution is started, FloTHERM PCB checks for restricted object overlapping conditions, out of physical bounds data entries, and so on. These **messages** are issued either as warnings, allowing a solve to proceed, or errors that deny a solution.

35. **Why am I unable to initiate a thermal solution within FloTHERM PCB?**

    FloTHERM PCB has been started in its basic mode of operation. This is because a FLOPCBLAYOUT license line was found, not a FLOPCBANALYSER line. This mode of operation is appropriate for users of FloTHERM PCB who do not perform thermal solution, but who only use FloTHERM PCB for the functional block diagramming, layout definition and thermal results inspection. FloTHERM PCB will have been licensed accordingly to achieve this.

Results Analysis

36. **Can’t the thermal board analysis be done in FloTHERM?**

    Yes, FloTHERM PCB is based on exactly the same technology. The advantage of FloTHERM PCB is that it is dedicated enough to be able to be used 'live' in design review meetings. It can be used by anyone at those meetings to define and communicate information and to investigate thermal performance of many board proposals

37. **What is the difference between a Standard and Detailed report?**

    Project reports are created by choosing **Tools > Generate Report > Detailed or Standard**. A **Detailed** report contains all input and results information available for the

loaded FloTHERM PCB project. A **Standard** report is a simplified version of a **Detailed** report. Users can configure their own reports to include any combination of information included in the full detailed report.

See "Displaying Reports" on page 95.

38. **Why do my results often become unavailable?**

    A project that has been solved will have results 'attached' to it, viewable in the Results mode. Any edit to the model will 'detach' the results. This is because the results are no longer valid when compared to the current model definition. **Edit > Undo** can be used to rewind the model to a state corresponding to the previous results. Results will become available again once **Edit > Undo** has been applied enough times.

39. **Can I export the thermal results?**

    Yes, this can be done in a number of ways:

    - **Tools > Generate Report** to view or save an HTML summary of the project

      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.

    - **File > Export > Export CSV** to produce a CSV file for subsequent import into a spreadsheet program

    - **File > Export > Export PACK** for unpacking into FloTHERM for more detailed results post processing. This is a one way transfer in that the model can not be loaded into FloTHERM PCB once saved from FloTHERM.

Index

— T —

— U —

— V —

— W —

End-User License Agreement

The latest version of the End-User License Agreement is available on-line at:
www.mentor.com/eula

IMPORTANT INFORMATION

USE OF ALL SOFTWARE IS SUBJECT TO LICENSE RESTRICTIONS. CAREFULLY READ THIS LICENSE AGREEMENT BEFORE USING THE PRODUCTS. USE OF SOFTWARE INDICATES CUSTOMER'S COMPLETE AND UNCONDITIONAL ACCEPTANCE OF THE TERMS AND CONDITIONS SET FORTH IN THIS AGREEMENT. ANY ADDITIONAL OR DIFFERENT PURCHASE ORDER TERMS AND CONDITIONS SHALL NOT APPLY.

END-USER LICENSE AGREEMENT ("Agreement")

This is a legal agreement concerning the use of Software (as defined in Section 2) and hardware (collectively "Products") between the company acquiring the Products ("Customer"), and the Mentor Graphics entity that issued the corresponding quotation or, if no quotation was issued, the applicable local Mentor Graphics entity ("Mentor Graphics"). Except for license agreements related to the subject matter of this license agreement which are physically signed by Customer and an authorized representative of Mentor Graphics, this Agreement and the applicable quotation contain the parties' entire understanding relating to the subject matter and supersede all prior or contemporaneous agreements. If Customer does not agree to these terms and conditions, promptly return or, in the case of Software received electronically, certify destruction of Software and all accompanying items within five days after receipt of Software and receive a full refund of any license fee paid.

1. **ORDERS, FEES AND PAYMENT.**

   1.1. To the extent Customer (or if agreed by Mentor Graphics, Customer's appointed third party buying agent) places and Mentor Graphics accepts purchase orders pursuant to this Agreement ("Order(s)"), each Order will constitute a contract between Customer and Mentor Graphics, which shall be governed solely and exclusively by the terms and conditions of this Agreement, any applicable addenda and the applicable quotation, whether or not these documents are referenced on the Order. Any additional or conflicting terms and conditions appearing on an Order will not be effective unless agreed in writing by an authorized representative of Customer and Mentor Graphics.

   1.2. Amounts invoiced will be paid, in the currency specified on the applicable invoice, within 30 days from the date of such invoice. Any past due invoices will be subject to the imposition of interest charges in the amount of one and one-half percent per month or the applicable legal rate currently in effect, whichever is lower. Prices do not include freight, insurance, customs duties, taxes or other similar charges, which Mentor Graphics will state separately in the applicable invoice(s). Unless timely provided with a valid certificate of exemption or other evidence that items are not taxable, Mentor Graphics will invoice Customer for all applicable taxes including, but not limited to, VAT, GST, sales tax and service tax. Customer will make all payments free and clear of, and without reduction for, any withholding or other taxes; any such taxes imposed on payments by Customer hereunder will be Customer's sole responsibility. If Customer appoints a third party to place purchase orders and/or make payments on Customer's behalf, Customer shall be liable for payment under Orders placed by such third party in the event of default.

   1.3. All Products are delivered FCA factory (Incoterms 2000), freight prepaid and invoiced to Customer, except Software delivered electronically, which shall be deemed delivered when made available to Customer for download. Mentor Graphics retains a security interest in all Products delivered under this Agreement, to secure payment of the purchase price of such Products, and Customer agrees to sign any documents that Mentor Graphics determines to be necessary or convenient for use in filing or perfecting such security interest. Mentor Graphics' delivery of Software by electronic means is subject to Customer's provision of both a primary and an alternate e-mail address.

2. **GRANT OF LICENSE.** The software installed, downloaded, or otherwise acquired by Customer under this Agreement, including any updates, modifications, revisions, copies, documentation and design data ("Software") are copyrighted, trade secret and confidential information of Mentor Graphics or its licensors, who maintain exclusive title to all Software and retain all rights not expressly granted by this Agreement. Mentor Graphics grants to Customer, subject to payment of applicable license fees, a nontransferable, nonexclusive license to use Software solely: (a) in machine-readable, object-code form (except as provided in Subsection 5.2); (b) for Customer's internal business purposes; (c) for the term of the license; and (d) on the computer hardware and at the site authorized by Mentor Graphics. A site is restricted to a one-half mile (800 meter) radius. Customer may have Software temporarily used by an employee for telecommuting purposes from locations other than a Customer office, such as the employee's residence, an airport or hotel, provided that such employee's primary place of employment is the site where the Software is authorized for use. Mentor Graphics' standard policies and programs, which vary depending on Software, license fees paid or services purchased, apply to the following: (a) relocation of Software; (b) use of Software, which may be limited, for example, to execution of a single session by a single user on the authorized hardware or for a restricted period of time (such limitations may be technically implemented through the use of authorization codes or similar devices); and (c) support services provided, including eligibility to receive telephone support, updates, modifications, and revisions. For the avoidance of doubt, if Customer requests any change or enhancement to Software, whether in the course of

receiving support or consulting services, evaluating Software, performing beta testing or otherwise, any inventions, product improvements, modifications or developments made by Mentor Graphics (at Mentor Graphics' sole discretion) will be the exclusive property of Mentor Graphics.

3. **ESC SOFTWARE.** If Customer purchases a license to use development or prototyping tools of Mentor Graphics' Embedded Software Channel ("ESC"), Mentor Graphics grants to Customer a nontransferable, nonexclusive license to reproduce and distribute executable files created using ESC compilers, including the ESC run-time libraries distributed with ESC C and C++ compiler Software that are linked into a composite program as an integral part of Customer's compiled computer program, provided that Customer distributes these files only in conjunction with Customer's compiled computer program. Mentor Graphics does NOT grant Customer any right to duplicate, incorporate or embed copies of Mentor Graphics' real-time operating systems or other embedded software products into Customer's products or applications without first signing or otherwise agreeing to a separate agreement with Mentor Graphics for such purpose.

4. **BETA CODE.**

4.1. Portions or all of certain Software may contain code for experimental testing and evaluation ("Beta Code"), which may not be used without Mentor Graphics' explicit authorization. Upon Mentor Graphics' authorization, Mentor Graphics grants to Customer a temporary, nontransferable, nonexclusive license for experimental use to test and evaluate the Beta Code without charge for a limited period of time specified by Mentor Graphics. This grant and Customer's use of the Beta Code shall not be construed as marketing or offering to sell a license to the Beta Code, which Mentor Graphics may choose not to release commercially in any form.

4.2. If Mentor Graphics authorizes Customer to use the Beta Code, Customer agrees to evaluate and test the Beta Code under normal conditions as directed by Mentor Graphics. Customer will contact Mentor Graphics periodically during Customer's use of the Beta Code to discuss any malfunctions or suggested improvements. Upon completion of Customer's evaluation and testing, Customer will send to Mentor Graphics a written evaluation of the Beta Code, including its strengths, weaknesses and recommended improvements.

4.3. Customer agrees to maintain Beta Code in confidence and shall restrict access to the Beta Code, including the methods and concepts utilized therein, solely to those employees and Customer location(s) authorized by Mentor Graphics to perform beta testing. Customer agrees that any written evaluations and all inventions, product improvements, modifications or developments that Mentor Graphics conceived or made during or subsequent to this Agreement, including those based partly or wholly on Customer's feedback, will be the exclusive property of Mentor Graphics. Mentor Graphics will have exclusive rights, title and interest in all such property. The provisions of this Subsection 4.3 shall survive termination of this Agreement.

5. **RESTRICTIONS ON USE.**

5.1. Customer may copy Software only as reasonably necessary to support the authorized use. Each copy must include all notices and legends embedded in Software and affixed to its medium and container as received from Mentor Graphics. All copies shall remain the property of Mentor Graphics or its licensors. Customer shall maintain a record of the number and primary location of all copies of Software, including copies merged with other software, and shall make those records available to Mentor Graphics upon request. Customer shall not make Products available in any form to any person other than Customer's employees and on-site contractors, excluding Mentor Graphics competitors, whose job performance requires access and who are under obligations of confidentiality. Customer shall take appropriate action to protect the confidentiality of Products and ensure that any person permitted access does not disclose or use it except as permitted by this Agreement. Customer shall give Mentor Graphics written notice of any unauthorized disclosure or use of the Products as soon as Customer learns or becomes aware of such unauthorized disclosure or use. Except as otherwise permitted for purposes of interoperability as specified by applicable and mandatory local law, Customer shall not reverse-assemble, reverse-compile, reverse-engineer or in any way derive any source code from Software. Log files, data files, rule files and script files generated by or for the Software (collectively "Files"), including without limitation files containing Standard Verification Rule Format ("SVRF") and Tcl Verification Format ("TVF") which are Mentor Graphics' proprietary syntaxes for expressing process rules, constitute or include confidential information of Mentor Graphics. Customer may share Files with third parties, excluding Mentor Graphics competitors, provided that the confidentiality of such Files is protected by written agreement at least as well as Customer protects other information of a similar nature or importance, but in any case with at least reasonable care. Customer may use Files containing SVRF or TVF only with Mentor Graphics products. Under no circumstances shall Customer use Software or Files or allow their use for the purpose of developing, enhancing or marketing any product that is in any way competitive with Software, or disclose to any third party the results of, or information pertaining to, any benchmark.

5.2. If any Software or portions thereof are provided in source code form, Customer will use the source code only to correct software errors and enhance or modify the Software for the authorized use. Customer shall not disclose or permit disclosure of source code, in whole or in part, including any of its methods or concepts, to anyone except Customer's employees or contractors, excluding Mentor Graphics competitors, with a need to know. Customer shall not copy or compile source code in any manner except to support this authorized use.

5.3. Customer may not assign this Agreement or the rights and duties under it, or relocate, sublicense or otherwise transfer the Products, whether by operation of law or otherwise ("Attempted Transfer"), without Mentor Graphics' prior written consent and payment of Mentor Graphics' then-current applicable relocation and/or transfer fees. Any Attempted Transfer without Mentor Graphics' prior written consent shall be a material breach of this Agreement and may, at Mentor Graphics' option, result in the immediate termination of the Agreement and/or the licenses granted under this Agreement. The terms

of this Agreement, including without limitation the licensing and assignment provisions, shall be binding upon Customer's permitted successors in interest and assigns.

5.4. The provisions of this Section 5 shall survive the termination of this Agreement.

6. **SUPPORT SERVICES.** To the extent Customer purchases support services, Mentor Graphics will provide Customer updates and technical support for the Products, at the Customer site(s) for which support is purchased, in accordance with Mentor Graphics' then current End-User Support Terms located at http://supportnet.mentor.com/about/legal/.

7. **AUTOMATIC CHECK FOR UPDATES; PRIVACY.** Technological measures in Software may communicate with servers of Mentor Graphics or its contractors for the purpose of checking for and notifying the user of updates and to ensure that the Software in use is licensed in compliance with this Agreement. Mentor Graphics will not collect any personally identifiable data in this process and will not disclose any data collected to any third party without the prior written consent of Customer, except to Mentor Graphics' outside attorneys or as may be required by a court of competent jurisdiction.

8. **LIMITED WARRANTY.**

8.1. Mentor Graphics warrants that during the warranty period its standard, generally supported Products, when properly installed, will substantially conform to the functional specifications set forth in the applicable user manual. Mentor Graphics does not warrant that Products will meet Customer's requirements or that operation of Products will be uninterrupted or error free. The warranty period is 90 days starting on the 15th day after delivery or upon installation, whichever first occurs. Customer must notify Mentor Graphics in writing of any nonconformity within the warranty period. For the avoidance of doubt, this warranty applies only to the initial shipment of Software under an Order and does not renew or reset, for example, with the delivery of (a) Software updates or (b) authorization codes or alternate Software under a transaction involving Software re-mix. This warranty shall not be valid if Products have been subject to misuse, unauthorized modification or improper installation. MENTOR GRAPHICS' ENTIRE LIABILITY AND CUSTOMER'S EXCLUSIVE REMEDY SHALL BE, AT MENTOR GRAPHICS' OPTION, EITHER (A) REFUND OF THE PRICE PAID UPON RETURN OF THE PRODUCTS TO MENTOR GRAPHICS OR (B) MODIFICATION OR REPLACEMENT OF THE PRODUCTS THAT DO NOT MEET THIS LIMITED WARRANTY, PROVIDED CUSTOMER HAS OTHERWISE COMPLIED WITH THIS AGREEMENT. MENTOR GRAPHICS MAKES NO WARRANTIES WITH RESPECT TO: (A) SERVICES; (B) PRODUCTS PROVIDED AT NO CHARGE; OR (C) BETA CODE; ALL OF WHICH ARE PROVIDED "AS IS."

8.2. THE WARRANTIES SET FORTH IN THIS SECTION 8 ARE EXCLUSIVE. NEITHER MENTOR GRAPHICS NOR ITS LICENSORS MAKE ANY OTHER WARRANTIES EXPRESS, IMPLIED OR STATUTORY, WITH RESPECT TO PRODUCTS PROVIDED UNDER THIS AGREEMENT. MENTOR GRAPHICS AND ITS LICENSORS SPECIFICALLY DISCLAIM ALL IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NON-INFRINGEMENT OF INTELLECTUAL PROPERTY.

9. **LIMITATION OF LIABILITY.** EXCEPT WHERE THIS EXCLUSION OR RESTRICTION OF LIABILITY WOULD BE VOID OR INEFFECTIVE UNDER APPLICABLE LAW, IN NO EVENT SHALL MENTOR GRAPHICS OR ITS LICENSORS BE LIABLE FOR INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES (INCLUDING LOST PROFITS OR SAVINGS) WHETHER BASED ON CONTRACT, TORT OR ANY OTHER LEGAL THEORY, EVEN IF MENTOR GRAPHICS OR ITS LICENSORS HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN NO EVENT SHALL MENTOR GRAPHICS' OR ITS LICENSORS' LIABILITY UNDER THIS AGREEMENT EXCEED THE AMOUNT RECEIVED FROM CUSTOMER FOR THE HARDWARE, SOFTWARE LICENSE OR SERVICE GIVING RISE TO THE CLAIM. IN THE CASE WHERE NO AMOUNT WAS PAID, MENTOR GRAPHICS AND ITS LICENSORS SHALL HAVE NO LIABILITY FOR ANY DAMAGES WHATSOEVER. THE PROVISIONS OF THIS SECTION 9 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

10. **HAZARDOUS APPLICATIONS.** CUSTOMER ACKNOWLEDGES IT IS SOLELY RESPONSIBLE FOR TESTING ITS PRODUCTS USED IN APPLICATIONS WHERE THE FAILURE OR INACCURACY OF ITS PRODUCTS MIGHT RESULT IN DEATH OR PERSONAL INJURY ("HAZARDOUS APPLICATIONS"). NEITHER MENTOR GRAPHICS NOR ITS LICENSORS SHALL BE LIABLE FOR ANY DAMAGES RESULTING FROM OR IN CONNECTION WITH THE USE OF MENTOR GRAPHICS PRODUCTS IN OR FOR HAZARDOUS APPLICATIONS. THE PROVISIONS OF THIS SECTION 10 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

11. **INDEMNIFICATION.** CUSTOMER AGREES TO INDEMNIFY AND HOLD HARMLESS MENTOR GRAPHICS AND ITS LICENSORS FROM ANY CLAIMS, LOSS, COST, DAMAGE, EXPENSE OR LIABILITY, INCLUDING ATTORNEYS' FEES, ARISING OUT OF OR IN CONNECTION WITH THE USE OF PRODUCTS AS DESCRIBED IN SECTION 10. THE PROVISIONS OF THIS SECTION 11 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

12. **INFRINGEMENT.**

12.1. Mentor Graphics will defend or settle, at its option and expense, any action brought against Customer in the United States, Canada, Japan, or member state of the European Union which alleges that any standard, generally supported Product acquired by Customer hereunder infringes a patent or copyright or misappropriates a trade secret in such jurisdiction. Mentor Graphics will pay costs and damages finally awarded against Customer that are attributable to the action. Customer understands and agrees that as conditions to Mentor Graphics' obligations under this section Customer must: (a) notify Mentor Graphics promptly in writing of the action; (b) provide Mentor Graphics all reasonable information and assistance

to settle or defend the action; and (c) grant Mentor Graphics sole authority and control of the defense or settlement of the action.

12.2. If a claim is made under Subsection 12.1 Mentor Graphics may, at its option and expense, (a) replace or modify the Product so that it becomes noninfringing; (b) procure for Customer the right to continue using the Product; or (c) require the return of the Product and refund to Customer any purchase price or license fee paid, less a reasonable allowance for use.

12.3. Mentor Graphics has no liability to Customer if the action is based upon: (a) the combination of Software or hardware with any product not furnished by Mentor Graphics; (b) the modification of the Product other than by Mentor Graphics; (c) the use of other than a current unaltered release of Software; (d) the use of the Product as part of an infringing process; (e) a product that Customer makes, uses, or sells; (f) any Beta Code or Product provided at no charge; (g) any software provided by Mentor Graphics' licensors who do not provide such indemnification to Mentor Graphics' customers; or (h) infringement by Customer that is deemed willful. In the case of (h), Customer shall reimburse Mentor Graphics for its reasonable attorney fees and other costs related to the action.

12.4. THIS SECTION 12 IS SUBJECT TO SECTION 9 ABOVE AND STATES THE ENTIRE LIABILITY OF MENTOR GRAPHICS AND ITS LICENSORS FOR DEFENSE, SETTLEMENT AND DAMAGES, AND CUSTOMER'S SOLE AND EXCLUSIVE REMEDY, WITH RESPECT TO ANY ALLEGED PATENT OR COPYRIGHT INFRINGEMENT OR TRADE SECRET MISAPPROPRIATION BY ANY PRODUCT PROVIDED UNDER THIS AGREEMENT.

13. **TERMINATION AND EFFECT OF TERMINATION.** If a Software license was provided for limited term use, such license will automatically terminate at the end of the authorized term.

13.1. Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement immediately upon written notice if Customer: (a) exceeds the scope of the license or otherwise fails to comply with the licensing or confidentiality provisions of this Agreement, or (b) becomes insolvent, files a bankruptcy petition, institutes proceedings for liquidation or winding up or enters into an agreement to assign its assets for the benefit of creditors. For any other material breach of any provision of this Agreement, Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement upon 30 days written notice if Customer fails to cure the breach within the 30 day notice period. Termination of this Agreement or any license granted hereunder will not affect Customer's obligation to pay for Products shipped or licenses granted prior to the termination, which amounts shall be payable immediately upon the date of termination.

13.2. Upon termination of this Agreement, the rights and obligations of the parties shall cease except as expressly set forth in this Agreement. Upon termination, Customer shall ensure that all use of the affected Products ceases, and shall return hardware and either return to Mentor Graphics or destroy Software in Customer's possession, including all copies and documentation, and certify in writing to Mentor Graphics within ten business days of the termination date that Customer no longer possesses any of the affected Products or copies of Software in any form.

14. **EXPORT.** The Products provided hereunder are subject to regulation by local laws and United States government agencies, which prohibit export or diversion of certain products and information about the products to certain countries and certain persons. Customer agrees that it will not export Products in any manner without first obtaining all necessary approval from appropriate local and United States government agencies.

15. **U.S. GOVERNMENT LICENSE RIGHTS.** Software was developed entirely at private expense. All Software is commercial computer software within the meaning of the applicable acquisition regulations. Accordingly, pursuant to US FAR 48 CFR 12.212 and DFAR 48 CFR 227.7202, use, duplication and disclosure of the Software by or for the U.S. Government or a U.S. Government subcontractor is subject solely to the terms and conditions set forth in this Agreement, except for provisions which are contrary to applicable mandatory federal laws.

16. **THIRD PARTY BENEFICIARY.** Mentor Graphics Corporation, Mentor Graphics (Ireland) Limited, Microsoft Corporation and other licensors may be third party beneficiaries of this Agreement with the right to enforce the obligations set forth herein.

17. **REVIEW OF LICENSE USAGE.** Customer will monitor the access to and use of Software. With prior written notice and during Customer's normal business hours, Mentor Graphics may engage an internationally recognized accounting firm to review Customer's software monitoring system and records deemed relevant by the internationally recognized accounting firm to confirm Customer's compliance with the terms of this Agreement or U.S. or other local export laws. Such review may include FLEXlm or FLEXnet (or successor product) report log files that Customer shall capture and provide at Mentor Graphics' request. Customer shall make records available in electronic format and shall fully cooperate with data gathering to support the license review. Mentor Graphics shall bear the expense of any such review unless a material non-compliance is revealed. Mentor Graphics shall treat as confidential information all information gained as a result of any request or review and shall only use or disclose such information as required by law or to enforce its rights under this Agreement. The provisions of this Section 17 shall survive the termination of this Agreement.

18. **CONTROLLING LAW, JURISDICTION AND DISPUTE RESOLUTION.** The owners of certain Mentor Graphics intellectual property licensed under this Agreement are located in Ireland and the United States. To promote consistency around the world, disputes shall be resolved as follows: excluding conflict of laws rules, this Agreement shall be governed by and construed under the laws of the State of Oregon, USA, if Customer is located in North or South America, and the laws of Ireland if Customer is located outside of North or South America. All disputes arising out of or in relation to this Agreement shall be submitted to the exclusive jurisdiction of the courts of Portland, Oregon when the laws of Oregon apply, or Dublin, Ireland when the laws of Ireland apply. Notwithstanding the foregoing, all disputes in Asia arising out of or in relation to this Agreement shall be resolved by arbitration in Singapore before a single arbitrator to be appointed by the chairman of the Singapore International

Arbitration Centre ("SIAC") to be conducted in the English language, in accordance with the Arbitration Rules of the SIAC in effect at the time of the dispute, which rules are deemed to be incorporated by reference in this section. This section shall not restrict Mentor Graphics' right to bring an action against Customer in the jurisdiction where Customer's place of business is located. The United Nations Convention on Contracts for the International Sale of Goods does not apply to this Agreement.

19. **SEVERABILITY.** If any provision of this Agreement is held by a court of competent jurisdiction to be void, invalid, unenforceable or illegal, such provision shall be severed from this Agreement and the remaining provisions will remain in full force and effect.

20. **MISCELLANEOUS.** This Agreement contains the parties' entire understanding relating to its subject matter and supersedes all prior or contemporaneous agreements, including but not limited to any purchase order terms and conditions. Some Software may contain code distributed under a third party license agreement that may provide additional rights to Customer. Please see the applicable Software documentation for details. This Agreement may only be modified in writing by authorized representatives of the parties. Waiver of terms or excuse of breach must be in writing and shall not constitute subsequent consent, waiver or excuse.

Rev. 100615, Part No. 246066