Board Station® Interface User Guide

Supports Simcenter™ Flotherm™ EDA Bridge 8.1 or later, Simcenter™ Flotherm™ PCB 5.1 or later, and Board Station PCB 2005BST and 2006BSXE or later

Software Version 2.20
Document Revision 3

SIEMENS

Revision History

Revision	Changes	Date
1	Document reformatted. No technical changes.	Oct 2017
2	Product rebranding and document structural changes.	Mar 2020
3	Document reformatted. No technical changes.	Mar 2021

Table of Contents

This document describes the Mentor Graphics Board Station interface for exporting designs to Simcenter™ Flotherm™ PCB and Simcenter™ Flotherm™ EDA Bridge software for thermal analysis.

Overview of the Board Station Interface

The interface enables users to directly send board outline, placement keep out areas, component and package information from Board Station PCB layout software.

The exported data can be:

- used by Simcenter Flotherm PCB in a new or existing Simcenter Flotherm PCB project, or

- imported into EDA Bridge for subsequent transfer into Simcenter Flotherm.

The interface also enables the user to import modified component placements made by Simcenter Flotherm PCB to improve the thermal performance of the board, back into the Board Station design.

Installing the Board Station Interface

The Board Station interface provides a menu within Board Station that enables the export of data.

Prerequisites

- An installation of Mentor Graphics Board Station PCB software for PCB design, version 2005BST and 2006BSXE or later.

- An installation of Mentor Graphics Simcenter Flotherm PCB software, version 5.1 or later (6.2 recommended) OR an installation of Simcenter Flotherm V8.1 or later (that contains EDA Bridge).

- Knowledge of the use of above tools.

- Both software installations must be on the Microsoft® Windows® platform. If they need to be run on separate machines for operational or licensing reasons, then data files can be transferred over the network, however, you will not be able to invoke Simcenter Flotherm PCB from Board Station.

Procedure

1. Identify the location where the interface should be installed.

   The recommended installation directory is the Board Station customizing directory:

   %MGC_HOME%\etc\cust

   where %MGC_HOME% is the installation path for the Mentor Graphics software. Contact the Mentor Graphics software administrator for your location for further information.

2. Check if the files *startup* and/or *layout.startup* exist in the *startup* sub-folder of the identified installation directory because of previous customization of Board Station. If the files exist, then rename them to have *.old* extensions. If they are different from the ones supplied, then they should be merged by appending the new file to the existing file. For more guidance, contact Mentor Graphics Support.

3. Install the interface in two (or three) stages:

   a. If you have unzipped a previous version of this interface, remove all of the previously unzipped files, see below:

      %MGC_HOME%\etc\cust\flotherm\defaults.txt

      %MGC_HOME%\etc\cust\flotherm\MentorBStoEDAI.exe

      %MGC_HOME%\etc\cust\flotherm\QtCore4.dll

      %MGC_HOME%\etc\cust\flotherm\QtGui4.dll

      %MGC_HOME%\etc\cust\startup\layout.startup

   b. Unzip the supplied file:

      ...\eda_interfaces\Boardstation\FloTHERM_Boardstation_Interface.zip

where ... refers to the installation for either Simcenter Flotherm or Simcenter Flotherm PCB. This is typically *C:\Program Files\MentorMA\ flosuite_<version>\flotherm* or *C:\Program Files\MentorMA\flosuite_<version>\ flopcb_<version>* respectively.

The zip file contains a PDF version of this document and the zip file for the installation.

c. Unzip the following installation file:

FloTHERM_Boardstation_Interface_<version>.zip

into the installation directory where *<version>* is the latest version.

This will add the interface, AMPLE, and default files in the correct structure in a directory called *flotherm*.

The interface and program and defaults files may be moved to a different directory, but the FLO_BS_HOME environment variable must be set up, using the system control panel, to point to this directory.

4. Open the System Properties dialog box and update the environment path variable to include the Simcenter Flotherm PCB *bin* directory if you want to be able to start Simcenter Flotherm PCB from within Board Station.

For example, in Windows 10, open Settings and search for "environment". Select "Edit the system environment variables". This opens the **Advanced** tab of the System Properties dialog box. Click **Environment Variables** to display the current user and system environment variables. Update the path variable as follows:

• To the user (or system, depending on requirements) environment variable path, add the following, separating the new value from the existing values and each other with a ";":

For Simcenter Flotherm installations:

<install_dir>\flosuite_<version>\common\WinXP\bin

For Simcenter Flotherm PCB installations:

<install_dir>\flosuite_<version>\flopcb_<version>\WinXP\bin

_____ **Note** _____
If the installation directory contains spaces in the path, there will be a problem due to the Board Station environment not handling spaces in paths very well. The suggested solution is to use the short name, for example, *C:\Program~1\MentorMA* etc.
_____

_____ **Note** _____
To export data for subsequent import into Simcenter Flotherm via EDA Bridge use the **Export .floeda file** menu option.
_____

5. Start Board Station layout and check that **FloTHERM Interface** is displayed in the top menu bar between the **Check** and **Report** items.

   The interface is now ready to use.

   If you encounter a problem, consult "Troubleshooting" on page 19.

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

Using the Board Station Interface

This section describes the operational menu options and customizing dialog boxes.

Editing the Board Stackup

Editing of the board stackup to be used for Simcenter Flotherm PCB or EDA Bridge.

Prerequisites

- If a board cross-section has been set up using **Setup > Layer Materials**, this information will be used to create the initial setup.

Procedure

1. Open Board Station and load your design.

2. Choose **FloTHERM Interface > Board Stackup**.

3. Use the Board Stackup dialog box to set board layer characteristics.

Related Topics

Board Stackup Dialog Box

Loading a Design into a New Simcenter Flotherm PCB Project

How to start Simcenter Flotherm PCB with loaded board data in a new project.

Restrictions and Limitations

- Board outline lines should form a contiguous shape or there may be a problem when reading into Simcenter Flotherm PCB or EDA Bridge.

Procedure

1. Open Board Station and load your design.

2. Choose **FloTHERM Interface > Load into new FloTHERM PCB project**.

Results

No user interaction is required.

A Simcenter Flotherm PCB board data interface file is created.

Simcenter Flotherm PCB starts with loaded board data in a new project.

Loading a Design into an Existing Simcenter Flotherm PCB Project

How to start Simcenter Flotherm PCB with loaded board data in an existing project.

Restrictions and Limitations

- See those listed for "Loading a Design into a New Simcenter Flotherm PCB Project" on page 11.

Procedure

1. Open Board Station and load your design.

2. Choose **FloTHERM Interface > Load into existing FloTHERM PCB project**.

3. Select an existing Simcenter Flotherm PCB project.

   The project may be an empty template or an earlier version of this board.

Results

The interface creates the Simcenter Flotherm PCB board data interface file and invokes Simcenter Flotherm PCB to load the project file then load the board data to update that project.

If this is the second or subsequent time this option has been used, then the path name of the last project used will be displayed in the message window. This is saved within the design so will only be kept if the design is saved after using this option.

Exporting a FLOEDA File

Export a file for subsequent import into Simcenter Flotherm PCB or EDA Bridge.

Restrictions and Limitations

- See those listed for "Loading a Design into a New Simcenter Flotherm PCB Project" on page 11.

- When using the image option, the interface will take longer to run.

Procedure

1. Open Board Station and load your design.

2. Choose **FloTHERM Interface > Export .floeda file**.

3. Use the file browser to set the output path and filename. This file will have an extension of *.floeda*.

   This creates the board data interface file for transfer to another system which runs Simcenter Flotherm PCB or for import into EDA Bridge for subsequent transfer to Simcenter Flotherm.

   If the file already exists, a prompt will be displayed to check that the file should be overwritten.

Loading a Component Position File to Board Station

How to load a *.floplc* file, exported from Simcenter Flotherm PCB or EDA Bridge.

Procedure

1. Open Board Station and load your design.

2. Choose **FloTHERM Interface > Import Placement Changes**.

   Each changed component position can be approved individually or all changes can be approved in one click.

3. To import the placement changes:

   a. Select the *.floplc* file using the file browser. The file is then processed and if any differences in position, rotation or side of any component is detected, a Yes/No dialog box will be displayed asking "Individually approve each placement change?"

   b. Respond to the prompt:

      o Click **No** to allow all component changes to be carried out in a single step.

      o Click **Yes** to selectively approve each component change using a Yes/No/Cancel dialog box. The dialog box will supply the operation information:

        Move *comp_ref* (x=*X* y=*Y* rot=*r* side=*top/bottom*) to (x=*X'* y=*Y'* rot=*r'* side=*top/bottom*)

        where the fields in italics will be replaced with the current values and the new possible values for the component *comp_ref*.

        • Click **Yes** to apply the changes to the component.

        • Click **No** to skip just this change.

- Click **Cancel** to skip all further changes.

The file also contains property updates as a result of any changes to the power, height, and thermal properties made in Simcenter Flotherm PCB or EDA Bridge. These changes are backannotated into the Board Station file and will be used for future *.floeda* files produced from this design to reduce the need to change the values every time. Where these changes are to an existing property or overriding a value on the symbol or part, then a warning message is output to the message window.

At the end of processing the file, a message dialog box will be displayed showing how many components were moved (if any) and if any properties have been updated.

4. Acknowledge any messages by clicking **OK**.

Results

If any component has been moved then a placement check will be run and any errors resulting from the new thermal placement will be reported. Placement error checking is turned off during the reading of the FLOPLC file. This problem may be because the component interfering with a drill hole or the end position resulting in the component being outside or crossing the board outline.

Note

Moved components that were routed may need their connecting tracks re-routed.

If the new placement from Simcenter Flotherm PCB is not on the placement grid, then the component will be placed on the grid. If the Simcenter Flotherm PCB file is then read again, it will show up as needing to be moved again.

Board Stackup Dialog Box

To access: **FloTHERM Interface > Board Stackup**

Use this dialog box to set board layer characteristics.

Objects

Object	Description
Read Stackup from BoardStation Data	Loads or updates the data displayed in the dialog box with data from Board Station.
	Use this button if the name of a physical layer is changed in Board Station.
Board Thickness	Set this to a value representing the depth from the top of the top-most signal layer to the bottom of the lowest layers. The inner electrical layers will be spaced equidistantly within this board thickness in Simcenter Flotherm PCB or EDA Bridge.
Layers and their properties	As multiple layers make up a split plane, these layers should be grouped together using the **Move up** and **Move down** buttons, then the layers should be set to have the same split name. The second and other layers in the split plane are indicated with a " symbol in the Type column.
Add Above	(Advanced mode) Adds an insulation layer, with default settings for air.
	If no layer is selected, one will be added to the top of the stackup.
	The Change Layer dialog box is opened.
Add Below	(Advanced mode) Adds an insulation layer, with default settings for air.
	If no layer is selected, one will be added to the bottom of the stackup.
	The Change Layer dialog box is opened.
Move up	Moves the selected layer(s) up.
	If the layer above is a split layer with a different split name, the layers selected will be moved above it.
Move down	Moves the selected layer(s) down.
	If the layer above is a split layer with a different split name, the layers selected will be moved below it.
Change	Opens the Change Layer dialog box to change the properties of the selected layer(s).
Delete	(Advanced mode) Deletes a layer. Only electrical layers with no logical layers can be deleted.

Usage Notes

The board stackup will be loaded into Simcenter Flotherm PCB or EDA Bridge as a board with equispaced electrical layers, that is, the same dielectric thickness is used between each layer (or merged layer in the case of split planes).

The interface cannot use the defined Mentor stackup as-is because the actual manufactured physical stackup is needed, so the facility to merge multiple layers into a single split plane is provided.

The stackup is saved in the job directory and is automatically read when layout is started. If the stackup does not exist, then the stackup is read from the existing physical layer stackup in the design using the layer thicknesses and any other properties that have been setup using the **Setup > Layer Materials** menu option.

It is recommended that as much information is set up using the physical stackup layer definition, such that, when the stackup is read from Board Station data, the amount of subsequent editing is minimized.

Use Advanced mode if greater control of the stackup is required.

Advanced Mode

Advanced mode is controlled by the UI_MODE setting in */etc/cust/flotherm/defaults.txt*. This file also controls the default internal and external copper thicknesses, material and conductivity, and the dielectric thickness and constant.

Uncomment the # UI_MODE ADVANCED line, that is, remove the leading # character.

In advanced mode, the dielectric (core/pre-preg) layers are viewable and can be edited.

A stackup can be modified to have thicker core and thinner pre-preg layers.

_____ **Caution** _____
Switching from Advanced to normal mode results in the loss of the dielectric information, as the layers will be equispaced again.
_____

Related Topics

Editing the Board Stackup

Change Layer Dialog Box

Change Layer Dialog Box

To access: Click **Change** from the Board Stackup dialog box.

To access in Advanced mode: Click either **Add Above** or **Add Below** from the Board Stackup dialog box in Advanced mode.

Use this dialog box to change the split layer name of layer(s) selected in the Board Stackup dialog box. If multiple layers of different types have been selected, only the first layer is displayed and only changes made to layers of the matching electrical or non-electrical type will be applied.

Objects

Field	Description	Values
Layer Name	The layer name. Read-only.	
Layer Type	Select the type of layer: • Signal • Full Plane • Split Plane • (Advanced mode) Insulator Layers cannot be changed from electrical to non-electrical or vice-versa.	
Material	(Advanced mode) Enables the material to be changed.	
Split Layer Name	(Split Plane layers) As multiple layers make up a split plane, these layers should be grouped together using the **Move up** and **Move down** buttons on the Board Stackup dialog box, then the layers should be set to have the same split name. The second and other layers in the split plane are indicated with a " symbol in the Type column in the Board Stackup dialog box.	
Thickness	Updated as layer properties are changed. Read-only.	In board units.
Conductivity	(Electrical layers) The conductivity of the layer(s).	In Siemens/m
Dielectric	(Insulating layers) The dielectric constant of the layer(s).	

Related Topics

Board Stackup Dialog Box

FloTHERM Interface Settings Dialog Box

To access: **FloTHERM Interface > Settings**

Use this dialog box to customize items transferred to Simcenter Flotherm PCB or EDA Bridge from Board Station. The settings are saved between runs of the Board Station application in a settings file in the user's home directory.

Objects

Field	Description	Values
Package Outline Layer	The symbol layer that is used for the outline of the components in Simcenter Flotherm PCB or EDA Bridge. The name used must be that of a layer used by the symbols that are referenced by the components in the design. If there is no data on that layer for a component symbol in a particular design then the bounding box of the pins will be used by Simcenter Flotherm PCB or EDA Bridge. Normally this would be: • COMPONENT_THERMAL_OUTLINE (if defined) • COMPONENT_PLACEMENT_OUTLINE or • COMPONENT_BODY_OUTLINE Placement outlines normally include space for the jaws of placement machines so the thermal or body outlines are much closer to reality. Other layers in symbols may be used but these are the only attribute values which are processed.	
Default Package Height	The default package height to be used for components that do not have a HEIGHT property defined.	Default is 150 mils and must be zero or greater.
Power Property	The property to be looked for components for their operational power rating.	Default is POWER_OPR (operating power) and is expected to be in milliwatts (mW).
Package Name Property	The property to be looked for on components, symbols and parts for which the Simcenter Flotherm PCB package is known.	Default is FLOPCB_PKG.
Package Material Property	The property to be looked for on components for which the package material is known.	Default is FLOPCB_PKG_MATL.

Field	Description	Values
Include Unplaced Components	• When checked, all unplaced components are included in the *.floeda* file for placement within Simcenter Flotherm PCB or EDA Bridge. Unplaced components are placed at 0,0. • When unchecked, any unplaced components will be ignored on output (that is, they will not be included in the *.floeda* file).	Default is checked.
Copper Coverage The percentage copper on a conductive layer. The values used for each layer can be overridden in Simcenter Flotherm PCB or EDA Bridge if required.		
Calculation Method	• Calculate — calculate the coverage based on the current state of routing on the board. This adds up areas of track copper, copper shapes and lines, and pin/via pad shapes. No allowance is made for tracks overlapping pads. • Use default — for an unrouted board. • Use Image — uses a picture of the board layer as coverage.	
Default Signal Coverage	The default coverage in percentage for signal layers.	From 0 to 100%. The default is 20%
Default Plane Coverage	The default coverage in percentage for reference plane layers.	From 0 to 100%. The default is 90%

Usage Notes

The site defaults are stored in *%MGC_HOME%/etc/cust/flotherm/defaults.txt*

Troubleshooting

Possible problem symptoms and proposed solutions.

Symptoms	Solution
The **FloTHERM Interface** menu is not displayed in the top bar.	Check the environment variable AMPLE_PATH and check that the *layout.startup* supplied is the first one found when processing the paths shown in order.
Error message "Error generating floeda file. Check *<job dir>/flo_run.log* for error messages".	Check the log file that is referenced for errors, and contact Mentor Graphics Support if you are unable to resolve the issue.

Symptoms	Solution
Simcenter Flotherm PCB does not start	Check that the *runflopcb.bat* file can be found in one of the directories on the environment variable PATH as described in the installation instructions. This may be overridden for site specific issues if required, contact Mentor Graphics Support for details.

If you encounter any other errors, note the circumstances (board used, the operation, and so on) and any error messages in the Board Station transcript and background window, and notify Mentor Graphics Support. Send all files starting with "flo" that are in the job directory that was being used.

If any of these problems continue, contact Mentor Graphics Support for advice.