Cadence® Allegro® Package Designer Interface User Guide

Supports Simcenter™ Flotherm™ EDA Bridge 8.1 or later, Simcenter™ Flotherm™ PCB 5.1 or later, and Allegro Package Designer 17.2 or later

Software Version 2.24
Document Revision 2

SIEMENS

Revision History

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

Table of Contents

Chapter 1
Allegro Package Designer Interface

This document describes the Allegro Package Designer (APD) interface for exporting designs to Simcenter™ Flotherm™ PCB and Simcenter™ Flotherm™ EDA Bridge software for thermal analysis.

Overview of the Allegro Package Designer Interface

The interface enables users to directly send board outline, placement keep out areas, and component and package information from APD 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 APD design.

Installing the APD Interface

The APD interface provides a menu within Allegro Package Designer that enables the export of data.

Prerequisites

- An installation of Cadence APD software for PCB design, version 17.2 or later.

- An installation of Simcenter Flotherm PCB software for thermal analysis of PCBs, version 5.1 or later, 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 Microsoft® Windows® platforms. If they must 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 Allegro.

Procedure

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

   The recommended installation directory is the *allegro_site* location. This can be found using the **Tools > Utilities > Env Variables** option in Allegro and scrolling down to find the entry for *allegro_site*. Usually, this location is:

   <cds_root>\share\local\pcb

   where *<cds_root>* is the installation path for the Cadence software.

   _____ **Note** _____
   The pathname must not contain any space characters.
   _____

   Contact the Cadence software administrator for your location for further information.

2. In the just identified installation directory, check if there is a *skill\apd.ilinit* file because of earlier customizing.

   If so, then rename the file to *apd.ilinit.old*. If the file is different from the one supplied then they should be merged; see the next step for details.

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.

   > *<allegro_site>\forms\flo_settings.form*
   >
   > *<allegro_site>\menus\apd.men*
   >
   > *<allegro_site>\menus\allegro_157_flo.men*
   >
   > *<allegro_site>\menus\flotherm_interface_men_fragment.txt*
   >
   > *<allegro_site>\skill\apd.ilinit*
   >
   > *<allegro_site>\skill\EDAItoFLOEDA2.exe*

<allegro_site>\skill\flotherm_functions.il

<allegro_site>\skill\QtCore4.dll

<allegro_site>\skill\QtGui4.dll

b. Unzip the supplied file:

   ...\eda_interfaces\APD\FloTHERM_APD_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 installation file:

   FloTHERM_APD_Interface_v<version>.zip

   to the *allegro_site* location identified earlier.

   This will add files to the forms, menus and skill directories.

   If an earlier version of Allegro is in use, edit the *apd.mem* in your installation file to include the FloTHERM menu section found in *apd.mem*.

4. If required to start Simcenter Flotherm PCB or EDA Bridge from APD, open the System Properties dialog box and update the environment path variable to include the Simcenter Flotherm PCB *bin* directory.

   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:

a. 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**
>
> To export data for subsequent import into Simcenter Flotherm via EDA Bridge use the **Export .floeda file** menu option. See "Exporting a FLOEDA File" on page 12.

5. Start APD and check that **FloTHERM Interface** is displayed in the top menu bar.

   The interface is now ready to use.

> **Note**
>
> If there are *two* **FloTHERM Interface** entries in the menu bar, then rename file *menus\apd.men* to *apd.men.old*.

If any problems are encountered, consult "Troubleshooting" on page 18.

Using the APD Interface

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

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

- If the design has no board outline data, a warning message will be output. This may also cause a problem when reading into Simcenter Flotherm PCB or EDA Bridge.

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

- If the design has no components, a prompt will ask if the interface should continue. If this occurs:

  o Click **Yes** to transfer just the board outline/package keep out area information, if any.

  o Click **No** to cancel the interface. A warning message will be output, recommending another design should be used.

Procedure

1. Open Allegro Package Designer (APD) 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 Allegro Package Designer (APD) 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.

   > **Note**
   >
   > The full path and filename of the project file must not contain spaces because of an internal Allegro limitation. If this happens, an error message is displayed and the interface will not be invoked.

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. See "FloTHERM Interface Settings Dialog Box" on page 15 for a description of the Use Image calculation method.

Procedure

1. Open Allegro Package Designer (APD) 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 Allegro Package Designer

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

Prerequisites

- A component position file (*.floplc*), exported from Simcenter Flotherm PCB or EDA Bridge.

Procedure

1. Open Allegro Package Designer (APD) 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 "Approve all component placement changes?"

   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*, mirror=*YES/NO*) to (x=*X'*, y=*Y'*, rot=*r'*, mirror=*YES/NO*)

         where the field 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.

   If a component cannot be updated, an Error box will be displayed identifying the component that cannot be moved. The problem may be because the component or an attached test point is fixed or the end position would be outside the board extents.

   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 Allegro 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).

   If any properties have been updated, a message dialog box will be displayed showing how many changes have been made.

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

Results

- Only accepting some component changes, and not others, may result in components overlapping, therefore a DRC check should be carried out after loading the new placement.

- This option is also able to load the *place.txt* files output by Cadence.

- Allegro will be paused while the FLOEDA file is generated. For large designs, when using the image option, this may take a minute or more.

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 APD.

Objects

Field	Description	Values
Package Outline Layer	The symbol layer (subclass) that is used for the outline of the components in Simcenter Flotherm PCB or EDA Bridge. The options available are dependent on the layers used by the symbols that are referenced by the components in the design. This means that when changing between designs with varied symbol libraries, this layer may be invalid. A warning will be printed if this needs to be changed because the existing setting is not present in a symbol in this design. This warning will also be output when the interface is run if this layer has not been updated. 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 and EDA Bridge. Only the layers used in the symbol definitions are listed, however, components on the bottom of the board will use the component outline for that side.	PLACE_BOUND_TOP layer is used by the IDF interface. Boards with no components loaded will have one entry called NO_SYMBOL_LAYER to indicate this situation
Component Settings		
Default Package Height	The default package height to be used for components that do not have a HEIGHT (or FLOPCB_HEIGHT) property defined.	Default is 20 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 to the left of the board. • When unchecked, any unplaced components are not included in the *.floeda* file.	Default is checked.
Copper Coverage This section describes how to control the representation of the distribution of copper on layers within the board.		
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. If the board is not fully routed, a prompt will be displayed stating what the percentage of nets routed is and whether the interface should continue with the calculation or use the default values. • 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 settings are saved between runs of the Allegro application in a settings file in your home directory. The defaults used in APD are saved in a separate settings file.

Related Topics

Adding and Changing Interface Settings by Command Line

Component Height Properties

Adding and Changing Interface Settings by Command Line

This section describes the use of the **flotherm_add_properties** command.

The **flotherm_add_properties** command can be used to set up the power, height, Simcenter Flotherm PCB package, package material and thermal properties in the dictionary using the names in the Simcenter Flotherm Interface Settings dialog box.

Procedure

1. Open a command input window.

2. Enter the following:

    ```
    >flotherm_add_properties
    ```

Results

If the properties do not already exist in the design dictionary then they will be added, and a confirmation dialog box will be displayed.

If the properties already exist, a message will be displayed. They can be subsequently edited using **Setup > Property definitions** in APD.

The Allegro library can be updated to add these properties to the part (component definition) or symbol to reduce the need to define them within the design and minimize the time required to setup designs for analysis in Simcenter Flotherm PCB.

The precedence level in decreasing order for the all the properties except height is:

- Component

- Symbol

- Part

Component Height Properties

Components, symbols and parts can have a single height property that overrides other height definitions

For components, symbols and parts, the FLOPCB_HEIGHT property overrides:

- The PACKAGE_HEIGHT_MAX attribute assigned to shapes in the symbol from the library if defined,

- The property HEIGHT on parts, or

- The default height set in the FloTHERM Interface Settings Dialog Box.

Simcenter Flotherm PCB supports a single height, so the maximum height is used if only the PACKAGE_HEIGHT_MAX is defined on multiple shapes.

The precedence level for finding the component height is, in decreasing order:

1. Component (FLOPCB_HEIGHT)

2. Symbol (FLOPCB_HEIGHT)

3. Symbol figures (PACKAGE_HEIGHTMAX - largest)

4. Part (FLOPCB_HEIGHT, then HEIGHT)

Troubleshooting

Possible problem symptoms and proposed solutions.

Symptoms	Solution
The **FloTHERM Interface** menu is not displayed in the top bar.	1. Check for error messages on loading. If there is still a problem, menus manually inserted (rather than by program, as is the default from version V2.24 onwards) are available in the *menus* folder. Rename the appropriate version to *apd.men*. 2. Also, check the environment variable MENUPATH and check that the *apd.men* file supplied is the first one found when processing the paths shown in order.
Undefined function errors when the menu options are selected.	Check the environment variable, allegro_site, and check that *apd.ilinit* is in a subdirectory named *skill* of this path.
Error message "Unable to find interface program in path. Check FloTHERM PCB interface installation".	Check that *EDAItoFLOEDA2.exe* is in the same directory as *apd.ilinit*.
Error Message "Error extracting data for output. Check *extract.log* and contact Mentor Graphics Support"	Check the log file for errors and contact Mentor Graphics Support if you are unable to resolve the issue.
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. The command **flopcb_show_paths** will display the current paths in use. This may be overridden for site-specific issues if required, contact Mentor Graphics Support for details.

Symptoms	Solution
Simcenter Flotherm PCB or EDA Bridge gives an error of design has no date.	Check that the board outline is defined and there are no breaks in the outline shape.
The interface fails.	Try running *EDAItoFLOEDA2.exe* from the command line to check the version number, and that all the required libraries are installed. As of version 2.20, Visual Studio 2010 Service Pack 1 redistributable libraries are required to be installed.

If you encounter any other errors, note the circumstances (board used, operation, and so on) and any error messages in the Allegro window and notify Mentor Graphics Support.

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