Xpedition® Interface User Guide

Supports Simcenter™ Flotherm™ 9.1 or later, Simcenter™ Flotherm™ PCB 6.1 or later, Xpedition® Enterprise EEVX.1.1 or later

Software Version 3.2
Document Revision 4

SIEMENS

Unpublished work. © 2021 Siemens

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

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

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

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

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

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

Revision History

Revision	Changes	Date
1	File and filepath naming changed from Expedition to Xpedition. Supplied *zip* file contains *msi* files for 32-bit and 64-bit installations. See “Installing the Xpedition PCB Interface” on page 7.	Nov 2016
2	Reference to SupportNet changed to Support Center, and web link changed.	Jul 2017
3	Product rebranding.	Mar 2020
4	Document reformatted. Reference to “Mentor, a Siemens Business” changed to “Siemens Digital Industries Software”. No technical changes.	Mar 2021

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

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

Table of Contents

Chapter 1
Xpedition PCB Interface

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

Overview

The interface enables you to export layout information from Xpedition 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.

Installing the Xpedition PCB Interface

The Xpedition interface provides a menu option within Xpedition that enables the export of data.

Prerequisites

The prerequisites for direct interface with Xpedition PCB software are:

- An installation of Xpedition Enterprise Flow, version EEVX.1.1 or later, on a Microsoft® Windows platform.
- An installation of Simcenter Flotherm PCB software for thermal analysis of PCBs, version 6.1 or later OR an installation of Simcenter Flotherm V9.1 or later that contains EDA Bridge.

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

- Knowledge of the use of above tools.

Procedure

1. Double-click the supplied file:

   <install_dir>\flosuite_v<version>\<product>\eda_interfaces\Xpedition\FloTHERM_Xpedition_Addin.zip

   where *<product>* is either *flotherm* or *flopcb_<pcb_version>*.

2. Double-click the file *FloTHERM_Xpedition_Addin_<version>.zip*

   where *<version>* is the version number of the Xpedition addin interface.

   This file contains two Windows Installer Packages: *SetupFloExpAddin.msi* and *SetupFloExpAddin64.msi*.

3. Double-click the appropriate *msi* file:

   - For 32-bit installations, *SetupFloExpAddin.msi*.
   - For 64-bit installations, *SetupFloExpAddin64.msi*.

   and follow the on-screen instructions.

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

Operations

Use the following procedures to export data from Xpedition PCB and load that data to Simcenter Flotherm or Simcenter Flotherm PCB.

Exporting Data from Xpedition PCB

Exported data will be in a floeda file that can be imported into EDA Bridge (then Simcenter Flotherm) or Simcenter Flotherm PCB.

Prerequisites

The interface has been installed, see “Installing the Xpedition PCB Interface” on page 7.

Procedure

1. In Xpedition PCB, choose **Analysis > Export to FloTHERM** to open the Export to FloTHERM dialog box.
2. Decide whether to use the conductor geometry from Xpedition or your (assumed) coverage percentages, then click **OK** to open the Save As dialog box.
3. Use the Save As dialog box to save the data to a file.

   By default the file has an extension of *.floeda*.

   The default location for the saved file is the Xpedition Model's PCB directory.

   The default name of the export file is the same name as the model, but with a *.floeda* extension (that is, replacing *.pcb* or adding the extension if none exists).

   If you have chosen to use the conductor geometry rather than use an assumed coverage, a percentage completed progress bar and the following messages are displayed in the Xpedition PCB status bar (bottom of window):

   - Initializing Images (Step 1 of 8)
   - Processing Traces (Step 2 of 8)
   - Processing Padstacks (Step 3 of 8)
   - Processing Vias (Step 4 of 8)
   - Processing Pins (Step 5 of 8)
   - Processing Conductive Areas (Step 6 of 8)

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

- Writing Image Files (Step 7 of 8)
- Calculating Average Coverage Values (Step 8 of 8)

Errors that may be reported include errors in the processing of curves deemed invalid, pads that have incorrect geometry, or multiple holes found in padstacks.

Results

An information message is displayed on completion:

Export to FloTHERM Completed.

Related Topics

Export to FloTHERM Dialog Box

Loading Exported Data into Simcenter Flotherm

Loading of exported data to Simcenter Flotherm is done via EDA Bridge.

Prerequisites

- A *.floeda* file exported from Xpedition.

Procedure

1. From Simcenter Flotherm, click the Launch EDA Bridge tool icon  to open the EDA Bridge application.
2. From EDA Bridge, choose **File > Import > Import FLOEDA** and select the *.floeda* file using the file selection dialog box.

Loading Exported Data into Simcenter Flotherm PCB

Loading of exported data to Simcenter Flotherm PCB is done directly.

Prerequisites

- A *.floeda* file exported from Xpedition.

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

Procedure

Choose one of the following:

If you want to...	Do the following:
Load from the GUI.	1. Choose **File > Import > Import FLOEDA**. 2. Select the *xyz.floeda* file using the file selection dialog box.
Load from a command line.	Enter the command: `runflopcb xyz.floeda` This invokes Simcenter Flotherm PCB with the board data interface loaded, creating a new project.

Updating an Existing Project

An existing layout in a project (which may be an empty template or an earlier version of the board) can be updated with a more up-to-date *.floeda* file.

Procedure

1. Choose one of the following:

If you want to...	Do the following:
Update from the GUI (EDA Bridge or Simcenter Flotherm PCB).	1. Load the existing project file 2. Choose **File > Import > Import FLOEDA** and select the *xyz.floeda* file using the file selection dialog box.
Update using the command line (Simcenter Flotherm PCB only).	Enter the command: `runflopcb xyz_proj.flopcb xyz.floeda` where *xyz_proj.flopcb* is the path name to the existing Simcenter Flotherm PCB project file. The *flopcb* project file must be before the *.floeda* file.

2. When an existing project exists you will have the option to either update or replace a model:

   - Click **Update** if you want to update component placements.

     Updating enables PCB designers to export updates from the EDA tool to thermal engineers, without the need for thermal engineers to re-specify data such as powers on components.

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

- Click **Replace** if you want to totally replace the layout with the one defined in the *.floeda* file.

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

Export to FloTHERM Dialog Box

To access: Xpedition PCB **Analysis > Export to FloTHERM**

Use this dialog box to create a **.floeda* board data interface file for import to Simcenter Flotherm PCB, or import to EDA Bridge for subsequent transfer to Simcenter Flotherm.

Objects

Field	Description
Version	The version of the Xpedition PCB interface you are using.
Layer Processing	
Use conductor geometry	Export the conductor geometry (traces, pads, vias, and so on) as a series of monotone images for each metallic layer and for through-hole vias. These images are pre-processed by the user to form a thermal conductivity map of the PCB prior to solving. For each layer, a percentage coverage figure is exported within the file and this is displayed in the properties sheet of EDA Bridge or Simcenter Flotherm PCB as %Coverage when a layer is selected.
Use assumed coverage	Ignore any defined conductor information and use assumed values to create material properties for each metallic layer. The assumed coverage values are exported and displayed into the properties sheets of EDA Bridge or Simcenter Flotherm PCB.
Signal	(Use assumed coverage) The percentage coverage for signal layers. The default is 20% and can range from 0 to 100%.
Pwr/Gnd	(Use assumed coverage) The percentage coverage for power and ground layers. The default is 90% and can range from 0 to 100%.
Component Export	
Default Component Height	Specifies the exported component height for components whose heights are not specified in Xpedition PCB. The default value is 5 mm. You can specify the units as mm, mils or inches. Entries are rounded down to 3 decimal places. You may use engineering format, for example, 1.25e2, 2.18e-2.

Related Topics

Exporting Data from Xpedition PCB

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

Global Customer Support and Success

A support contract with Siemens Digital Industries Software is a valuable investment in your organization's success. With a support contract, you have 24/7 access to the comprehensive and personalized Support Center portal.

Support Center features an extensive knowledge base to quickly troubleshoot issues by product and version. You can also download the latest releases, access the most up-to-date documentation, and submit a support case through a streamlined process.

https://support.sw.siemens.com

If your site is under a current support contract, but you do not have a Support Center login, register here:

https://support.sw.siemens.com/register

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

Chapter 2
Automation Reference

Xpedition PCB can be automated by scripts which use data objects, methods and so on in the data models.

The Simcenter Flotherm Xpedition Interface (provided by an add-in) also makes its data model available to users, and this is described in this chapter.

For information about Xpedition PCB add-ins, see Using Add-in Controls in the *PCB Automation Reference* manual supplied with Xpedition PCB.

Automation Start Script

A VBScript file can be used to run the Xpedition PCB Simcenter Flotherm add-in in automation mode.

You must have Xpedition installed with the correct environment set, the Xpedition install process will do this for you.

As an example, if your script file is named *my_script_file.vbs*, then type the following at the windows command line:

```
:>mgcsript my_script_file.vbs
```

Xpedition can then be made to run with the Simcenter Flotherm add-in and the settings provided within the script.

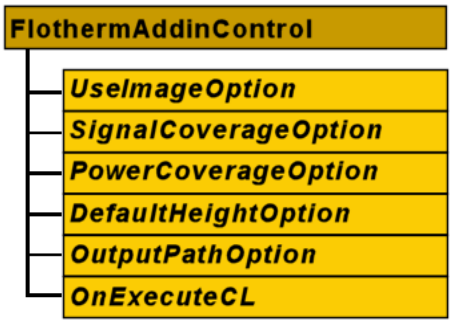
Note - Viewing PDF files within a web browser causes some links not to function. Use HTML for full navigation.

Xpedition Interface Data Model

The add-in object, FlothermAddinControl, and its methods.

Figure 2-1 shows the data model for the Xpedition interface.

Figure 2-1. Simcenter Flotherm Xpedition Interface Data Model



The relevant parts of the Xpedition data model are shown in Figure 2-2 and Figure 2-3.

In Figure 2-2, Item is an Xpedition addin (green signifying the property Read). The Simcenter Flotherm add-in is “Flotherm Addin”, which must be checked for in a script.

Figure 2-2. Add-ins in the Data Model

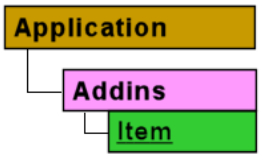
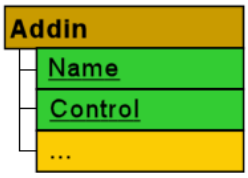


Figure 2-3 shows the two properties of the Addin objects that are of interest to us:

- The Name property returns the add-in control instance name.
- The Control property returns the control interface of the add-in.

The *PCB Automation Reference* manual lists and describes all of the properties.

Figure 2-3. Add-in Controls in the Data Model



The following example shows how to set the option for exporting image data in the floeda file.

```
If addin.Name = "Flotherm Addin" Then
  Call addin.Control.UseImageOption(True)
```


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

"Example of Using Methods in a Script" on page 21 shows examples for all of the FlothermAddinControl object methods.

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

FlothermAddinControl Object

The FlothermAddinControl object has several methods available that you can use to define settings and operations.

UseImageOption Method

Use to export image data.

Prerequisites: None.

Usage

FlothermAddinControl.UseImageOption([True | False])

Description

Sets the option for exporting image data in the floeda file.

Arguments

If True, exports image data.

If False, does not export image data, but exports the *assumed* coverage signal and power figures instead. The default assumed coverage figures are given in “Export to FloTHERM Dialog Box” on page 13, and can be overridden using the SignalCoverageOption Method and PowerCoverageOption Method.

Example

See “Example of Using Methods in a Script” on page 21.

SignalCoverageOption Method

Use to set the conductor coverage of signal layers.

Prerequisites: None.

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

Usage

FlothermAddinControl.SignalCoverageOption(***percent***)

Description

Sets the percentage conductor coverage of signal layers.

Only used when the assumed coverage option is active, see UseImageOption Method.

Arguments

- ***percent***

  A Real number between 0 and 100.

Example

See "Example of Using Methods in a Script" on page 21.

PowerCoverageOption Method

Use to set the conductor coverage of power layers.

Prerequisites: None.

Usage

FlothermAddinControl.PowerCoverageOption(***percent***)

Description

Sets the percentage conductor coverage of power layers.

Only used when the assumed coverage option is active, see UseImageOption Method.

Arguments

- ***percent***

  A Real number between 0 and 100.

Example

See "Example of Using Methods in a Script" on page 21.

DefaultHeightOption Method

Use to set a default height for components.

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

Prerequisites: None.

Usage

FlothermAddinControl.DefaultHeightOption(***height***,***units***)

Description

Sets the default height to be used for components and the units to be used.

Arguments

- ***height***

  A Real number.

- ***units***

  0=mm, 1=mils, 2=inches.

Example

See “Example of Using Methods in a Script” on page 21.

OutputPathOption Method

Use to set the output file path.

Prerequisites: None.

Usage

FlothermAddinControl.OutputPathOption(***path***)

Description

Sets the absolute path to the output floeda file.

Arguments

- ***path***

  An absolute path name of the output file, enclosed in double quotation marks.

Example

See “Example of Using Methods in a Script” on page 21.

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

OnExecuteCL Method

Use to execute the add-in.

Prerequisites: None.

Usage

FlothermAddinControl.OnExecuteCL

Description

Executes the add-in and creates the export file.

Example

See "Example of Using Methods in a Script" on page 21.

Example of Using Methods in a Script

The following code examples show how you to use the FlothermAddinControl object methods to set options. These examples are not complete but are for illustration purposes only.

- To open your design, in this case, file *busdiff.pcb*:

```
design ="C:\Documents and Settings\john_smith\Expedition Models\
busdiff_design_d03\PCB\busdiff.pcb"

Set app = CreateObject("MGCPCB.ExpeditionPCBApplication")

If app.IsValid Then
  Set docObj = app.OpenDocument(design)
.
.
```

- To get the addins collection and set it to the addins variable, then loop through the addins looking for the Simcenter Flotherm export addin.

```
Set addins = app.Addins

For Each addin In addins
  If addin.Name = "Flotherm Addin" Then
    Call addin.Control.UseImageOption(True)
    Call addin.Control.SignalCoverageOption(12.8)
    Call addin.Control.PowerCoverageOption(26)
    Call addin.Control.DefaultHeightOption(196.85,1)
    Call addin.Control.OutputPathOption("C:\Documents and Settings\
john_smith\Expedition Models\MyDesign.floeda")
    Call addin.Control.OnExecuteCL
.
.
```


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

The output is file *MyDesign.floeda*.

Also see “Xpedition Interface Data Model” on page 16.

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