Zuken CR-5000 Board Designer Interface User Guide

Supports Simcenter™ Flotherm™ EDA Bridge 8.1 or later, Simcenter™ Flotherm™ PCB 5.1 or later, and CR-5000 Board Designer 9.0 or later

Software Version 2.42
Document Revision 2

SIEMENS

Revision History

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

Table of Contents

Chapter 1
CR-5000 Board Designer Interface

This document describes the Zuken CR-5000 Board Designer interface to Simcenter™ Flotherm™ PCB and Simcenter™ Flotherm™ EDA Bridge software.

Overview

The CR-5000 Board Designer interface enables users to transfer board outline, placement keep out areas, component and package information from CR-5000 Board Designer PCB layout software to be used by Simcenter Flotherm PCB in a Simcenter Flotherm PCB project, or to be imported into EDA Bridge for subsequent transfer into Simcenter Flotherm.

Certain of these items can be customized using command line parameters.

The interface works by using the pcf (board data), mrf (manufacturing data) and ftf (footprint data) ASCII output files produced by the Zuken utilities **pcout**, **mrout** and **ftout**. The license feature ZX0409 (ASCII input/output) must be available on the Zuken license server.

Installing the CR-5000 Board Designer Interface

An installation requires that the supplied interface files are located in a specified folder.

Prerequisites

The prerequisites for direct interface with the Zuken CR-5000 Board Designer software are:

- An installation of Zuken CR-5000 Board Designer PCB software for PCB design, version 9.0 or later.

- An installation of Simcenter Flotherm PCB software for thermal analysis of PCBs, 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 the above tools.

Procedure

1. Open the Environment Variables dialog box to display the current user and system environment variables.

   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.

2. 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

   If any problems are encountered, refer to "Troubleshooting" on page 14.

   If neither Simcenter Flotherm PCB nor EDA Bridge are installed then create a new directory (into which the interface will be unzipped) and add it to the path environment variable.

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. Refer to Table 1-1.

Table 1-1. Old Unzipped Installation Files

Path	Filename
directory defined in step 2 above	QtCore4.dll
	QtGui4.dll
	ZukenBrdDesToEDAI.exe

b. Unzip the supplied file:

...\eda_interfaces\CR5000\FloTHERM_CR5000_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 for the installation.

c. Unzip the installation file:

FloTHERM_CR5000Interface_v<version>.zip

into the directory defined in step 2 above.

Operations

Generate interface files before running the interface.

Generating ASCII Input Files

Generates *.pcf*, *.ftf* and *.mrf* ASCII input files required for the interface.

Restrictions and Limitations

The CR-5000 software installation can be run on Microsoft® Windows® or UNIX® platforms, but then the data files must be transferred to Windows for the interface to be run. If they are run on separate machines for operational or licensing reasons then the resulting data files can be transferred over the network.

Prerequisites

This procedure assumes:

- The name of the pcb is *xyz.pcb* and

- The name of the manufacturing file is *xyz.mrl*.

The procedure will then create input files with the following names: *xyz.pcf*, *xyz.ftf* and *xyz.mrf*.

Procedure

1. Open a command prompt and use the **cd** command to change to the directory where the CR-5000 pcb data file is located.

2. To generate the *pcf* file, enter:

   ```
   pcout xyz
   ```

   The following option could also be used:

   ```
   -p:unit <unit>
   ```

   where <unit> is micron, mm, inch or mil.

3. To generate the *ftf* file, enter:

   ```
   ftout –m pcb xyz
   ```

The *–m pcb* switch is required to read the footprint data from the pcb file rather than the footprint data. The following option could also be used:

```
-p:unit <unit>
```

where <unit> is micron, mm, inch or mil.

4. To generate the *mrf* file, enter:

```
mrout –m panelspec xyz
```

The following options could also be used:

```
-p:unit <unit>
```

where <unit> is micron, mm, inch or mil.

And, if only the *mrdb* file is available:

```
-m master
```

The *mrl* or *mrdb* file may not be available or may have a different name.

If the file has a different name, then use this name instead of xyz in the **mrout** command above and add the following option to the command line:

```
-o xyz.mrf
```

Results

- If the *mrl* or *mrdb* file is not available, then, when the interface is run, a default stackup will be used based on a 62 mil thick board, 1.4 mil copper layers with FR4 dielectric evenly distributed between the copper layers.

- If the *mrl* or *mrdb* file is available, but the board stackup has zero thicknesses, then the copper thickness will be set to 1.4 mils and 8 mil dielectric layers added between the copper layers. It should also be checked to ensure it has the same number of layers as the design, otherwise only those whose names match the layers in the design will be output.

- If the *mrl* or *mrdb* file is available and the complete layer stack up is defined, then the total board thickness value is ignored in favour of the summation of all dielectric and copper layer thicknesses. If any of the dielectric layer thicknesses is absent, then the total board thickness value is used to determine an average dielectric layer thickness.

Generating a FLOEDA File

Run the interface command with previously-generated ASCII input files.

Procedure

Assuming the file names generated by "Generating ASCII Input Files" on page 10, enter the following ZukenBrdDesToEDAI command:

```
ZukenBrdDesToEDAI –job xyz -cov image –out xyz.floeda
```

Results

The following output will be generated as the interface runs:

```
Zuken CR5000 Board Designer to EDAI Version <version_no> <date>
Reading file : ./xyz.pcf
Reading file : ./xyz.mrf
Reading file : ./xyz.ftf
Processing design data ...
... done
Coverage for 1 = %
Coverage for die1_1 = %
Coverage for 2 = %
Coverage for die1_2 = %
<additional rows for each metallic and dielectric layer>
Saving output file: ./xyz.floeda
Output complete.
```

Related Topics

ZukenBrdDesToEDAI

Loading Data into Simcenter Flotherm PCB

Use one of the following, depending on whether you are creating a new Simcenter Flotherm PCB project or updating one that already exists.

Loading Data into a New Simcenter Flotherm PCB Project

Use this procedure when loading a layout for the first time.

Prerequisites

- A Simcenter Flotherm PCB board data interface file, see "Generating a FLOEDA File" on page 11.

Procedure

You have a choice:

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

Updating an Existing Simcenter Flotherm PCB Project

An existing layout in a Simcenter Flotherm PCB or EDA Bridge project can be updated with a newer *.floeda* file, which may be an empty template or an earlier version of this board.

Prerequisites

- A Simcenter Flotherm PCB board data interface file, see "Generating a FLOEDA File" on page 11.

Procedure

You have a choice:

If you want to...	Do the following:
Load data from the command line.	Enter the following 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.
Load data from the GUI.	1. Start Simcenter Flotherm PCB. 2. Choose **File > Import > Import FLOEDA** and select the *xyz.floeda* file.

If an earlier version of the board already exists, then you have the option to either:

- **Update** the existing layout with that defined in the *.floeda* file.

- Totally **Replace** the layout with the one defined in the *.floeda* file.

Loading Data into Simcenter Flotherm

Data can be imported into Simcenter Flotherm by use of the EDA Bridge application window.

Procedure

1. Start EDA Bridge from Simcenter Flotherm.

2. Choose **File > Import > Import FLOEDA** and select the *.floeda* file created by the CR-5000 interface.

3. If an earlier version of the board already exists, then you have the option to either:

   - **Update** the existing layout with that defined in the *.floeda* file.

   - Totally **Replace** the layout with the one defined in the *.floeda* file.

4. When the data has been loaded, then the board can then be transferred from EDA Bridge to Simcenter Flotherm as an assembly by choosing **File > Transfer**.

Troubleshooting

Advice when using the Zuken CR-5000 Board Designer interface.

If any of the following problems continue, contact Mentor Graphics Support.

Symptoms	Solution
The interface program or Simcenter Flotherm PCB does not start.	Check that files *runflopcb.bat* file and *ZukenBrdDesToEDAI.exe* 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.
Any other errors.	Note the circumstances (board used, operation, and so on) and any error messages in the command prompt window and notify Mentor Graphics Support. Supply the command line used, the ASCII files and the *.pcb* file if possible.

Commands

This section contains a description of the CR-5000 Board Designer interface command.

Command Line Syntax Conventions

This manual uses command usage line syntax conventions.

Table 1-2. Conventions for Command Line Syntax

Convention	Example	Usage
Boldface	SET COMmand Editing -**Off** | -**Vi** | -**Emacs** | -**Gmacs**	A boldface font indicates a required argument.
[ ]	EXIt [-Discard]	Square brackets enclose optional arguments. Do not enter the brackets.
Italic	DOFile *filename*	An italic font indicates a user-supplied argument.
{ }	ADD LFsrs *lfsr_name* {**Prpg** | **Misr**} *length seed* [-Out | -In]	Braces enclose arguments to show grouping. Do not enter the braces.
|	ADD LFsrs *lfsr_name* {**Prpg** | **Misr**} *length seed* [-Out | -In]	The vertical bar indicates an either/or choice between items. Do not include the bar in the command.
Underline	SET DOfile Abort **ON**| **OFf**	An underlined item indicates either the default argument or the default value of an argument.
…	ADD LFsr Connections *primary_pin lfsr_name position*…	An ellipsis follows an argument that may appear more than once. Do not include the ellipsis when entering commands.

ZukenBrdDesToEDAI

Creates a *.floeda* file from data read from *.pcf*, *.ftf* and *.mrf* ASCII input files.

Usage

ZukenBrdDesToEDAI **–job** *file_name_stem* [-dir *dir_name*] [-out *file_name*]
    [-olayer *lyr_name*] [-bolayer *lyr_name*] [-bolayer2 *lyr_name2*] [-ht *ht_value*]
    [-pwra *pwra_name*] [-pkga *pkga_name*] [-pkgm *pkgm_name*]
    [-cov {calc | image | {def [-scov *sig_cov*] [-pcov *pwr_cov*]}}]
    [-nounpl]

Arguments

- **-job** *file_name_stem*

  A required switch and string pair that specifies the job filename stem name of the input file, without the extension. An extension of ".pcf," ".ftf," and ".mrf" will be appended when the interface starts to read the files.

- -dir *dir_name*

  An optional switch and string pair that specifies the name of the directory where the input files are located. This parameter is only needed if you are invoking the interface from a different directory.

- -out *file_name*

  An optional switch and string pair that specifies the name of the output file to be used. A *floeda* extension should be used, otherwise the file cannot be read by the Simcenter Flotherm PCB import FLOEDA function. Existing files will be overwritten without warning. The default is *zukenbd.floeda*.

- -olayer *lyr_name*

  An optional switch and string pair that specifies the symbol layer that is used for the package outline of the components in Simcenter Flotherm PCB or EDA Bridge. The name used must be that of a layer used by the footprints that are referenced by the components in the design. If this is not specified or there is no data on that layer for a component symbol in a particular design, then the bounding box of the other graphics in that symbol (or the pins if no other graphics) will be used by Simcenter Flotherm PCB or EDA Bridge. To replicate the same board and layout definition that would be described by the IDF output available from the CR-5000 Board Designer, use "-olayer COMP_AREA_A" or "-olayer CompArea-A".

- -bolayer *lyr_name*

  Specifies the board layer that is used for the board outline in Simcenter Flotherm PCB or EDA Bridge. The name used must be a system layer. If this parameter is not specified, the layer BOARD_FIGURE will be used by Simcenter Flotherm PCB or EDA Bridge. If the board outline cutouts are defined on a different layer to the BOARD_FIGURE layer, or the layer specified by the -bolayer switch, then the -bolayer2 switch should be used to identify this layer.

Individual board outline figures will be joined if they have coincident end points and any figures that are completely contained in the main board outline will be converted to cutouts.

- -bolayer *lyr_name2*

  See above.

- -ht *ht_value*

  An optional switch and number pair that specifies the default package height to be used for components that do not have a HEIGHT property defined. *ht_value* must be zero or greater and must be in the same units as was used for the ASCII file output (-p:unit switch) or in microns if no switch was used (data is converted from dbunits to microns). Default is 150 mils (or equivalent in design units). Note the Zuken IDF default is 100 microns.

- -pwra *pwra_name*

  An optional switch and string pair that specifies the property sought for components to extract their operational power rating. If the thermal analysis setup has been used to assign power values to components, then use "-pwra powerDiss" on the command line. The default name is POWER_OPR (operating power) and its value is expected to be in milliwatts (mW).

- -pkga *pkga_name*

  An optional switch and string pair that specifies the property sought for components with a known set Simcenter Flotherm PCB package. The default name is FLOPCB_PKG.

- -pkgm *pkgm_name*

  An optional switch and string pair that specifies the property sought for components with a known set Simcenter Flotherm PCB package material. The default name is FLOPCB_PKG_MATL.

- -cov {calc | image | {def [-scov *sig_cov*] [-pcov *pwr_cov*]}}

  An optional switch and literal pair that specifies the copper coverage calculation method based on whether the board is routed or unrouted. Copper coverage is the percentage copper on a conductive layer. The valid method literals are:

  calc — For a routed board, the copper coverage is calculated from the area of the tracks, areas and pin/via pads currently routed on each conductive layer. No allowance is made for tracks overlapping pads. If the board is unrouted use the def option.

  image — For an unrouted board, specifies that the conductive layers will use a picture of the board layer as the basis for a more accurate coverage pattern. This option can only be used with Simcenter Flotherm PCB version 5.2 or later.

  def — For an unrouted board, specifies that the conductive layers will be set to a value of 20% and power layers (conductive layers with a positive-negative mixed layer type) will be set to 90%, unless the optional switch and number pairs are specified:

  –scov *sig_cov* — an optional switch and number pair that specifies that the conductive layers will be set to the value of *sig_cov*. Values must be between 0 and 100 inclusive.

–pcov *pwr_cov* — an optional switch and number pair that specifies that the power layers will be set to the value of *pwr_cov*. Values must be between 0 and 100 inclusive.

The values used for each layer can also be overridden in Simcenter Flotherm PCB or EDA Bridge, if required.

- -nounpl

  An optional switch that omits all unplaced, or out-of-board, components from the output file.

  Do not use this option if unplaced components are intended to be placed in Simcenter Flotherm PCB or EDA Bridge.

Related Topics

Generating a FLOEDA File