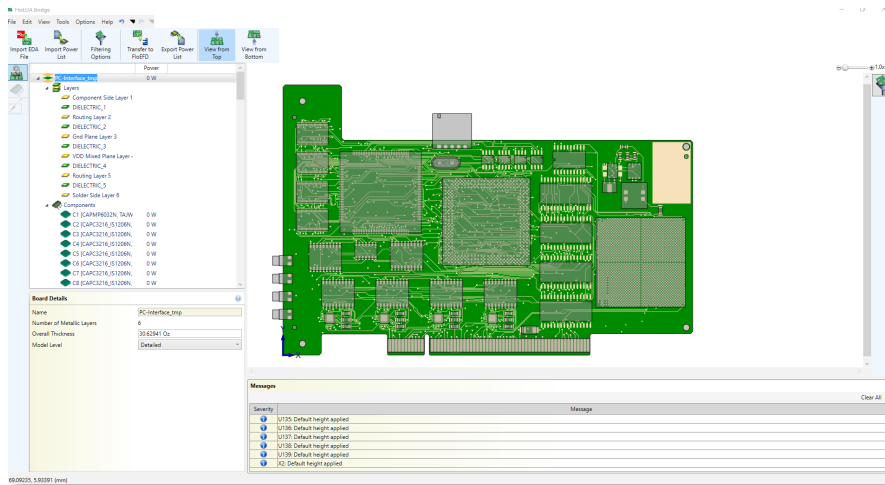


FloEDA Bridge Module



FEATURES AND BENEFITS:

- Import detailed PCB data into MCAD tool quickly
- Save time and effort by using imported material and IC thermal properties immediately for thermal analysis

The new FloEDA Bridge module for FloEFD provides additional capabilities for detailed PCB import into your MCAD tool of choice for thermal analysis. Historically, the best way of getting PCB data into an MCAD tool involved using IDF file pairs which has significant shortcomings especially with regards to the copper geometry within the PCB.

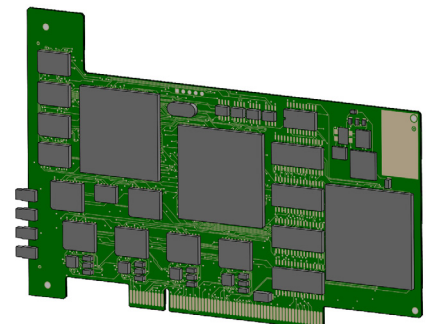
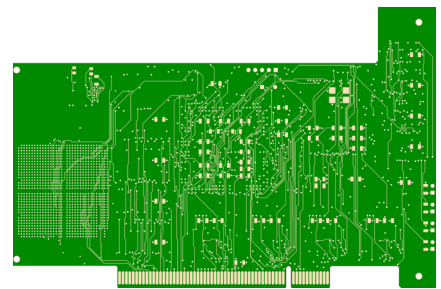
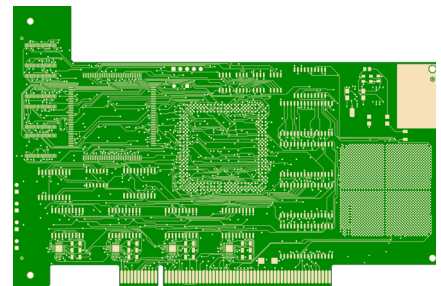
The FloEDA Bridge module allows for detailed PCB import with material and IC thermal properties into FloEFD for subsequent thermal analysis either on its own, or as part of a larger system level assembly.

PCB Import File Formats

The FloEDA Bridge module can use 3 different file formats for import:

- IDF
- CCE (native file format for Mentor Graphics Xpedition or PADS)
- ODB++ (Neutral file format for PCB manufacturing)

The benefit of using either CCE or ODB++ is that the PCB stack-up and copper geometry of the PCB is also read and can be used to create 3D geometry. This is particularly useful when thermal considerations such as thermal vias or copper pours have been designed into the board.



PCB Modelling Levels

The PCB can be modelled in one of three levels depending on the granularity required from the thermal simulation:

1. Compact: Volume average of copper in the entire PCB. An orthotropic material property is created to account for the in-plane and through-plane thermal conductivities based on the copper content within the board. Least mesh overhead.
2. Detailed: Volume average of copper within each layer in the board. Each layer will have its own material property based on the layer copper coverage, this includes dielectric layers with vias. Medium mesh overhead.
3. Explicit: Copper geometry will be extracted and modelled explicitly based on the thermal territory or net selected. The remainder of the PCB will be modelled at the "detailed" level. High mesh overhead.

Explicit Copper

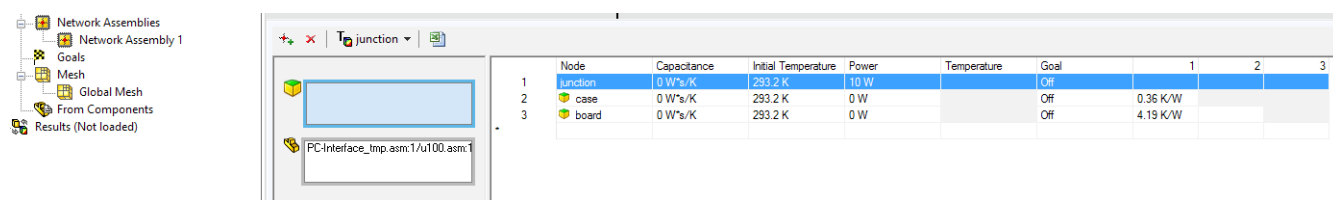
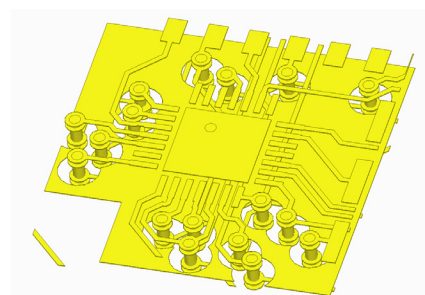
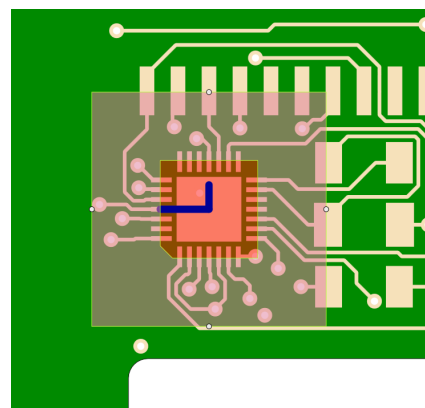
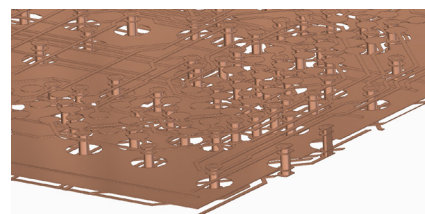
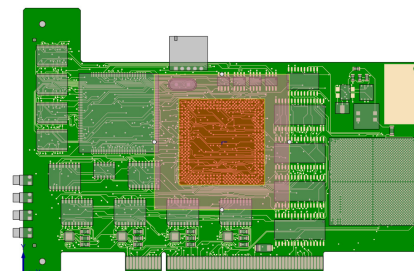
When using ODB++ or CCE, the netlist is also read by the FloEDA Bridge module and can be selected in the PCB tree. Explicit copper geometry can be created in one of two ways:

1. Specific nets can be selected and modelled as explicit. The software will then create 3D geometry to resemble that net, including vias, in MCAD.
2. Thermal Territories can be created around an IC. This can encompass many nets but will not exceed the extent around the IC specified by the user. The depth of the thermal territory can also be set.

IC Modelling

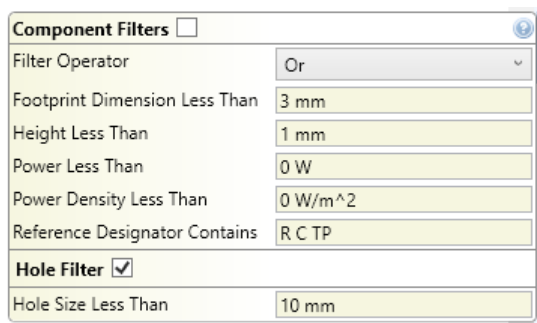
ICs can be thermally represented in 3 ways. There is increasing accuracy with each version of the thermal resistor network. If component heights are not set in the EDA tool, then a default can be specified.

1. Simple: Block representations of the components. Size based on assembly or placement outline with the material properties defined.
2. 2 Resistor: JEDEC θ_{JB} and θ_{JC} thermal resistances from datasheets if manually specified or if specified in the FloEDA Bridge it will be imported as a DELPHI model with 2 resistor components only.
3. DELPHI Multi resistor: Advanced thermal resistance network with additional nodes.



IC Filtering

ICs can be filtered based on one or more criteria. Designed to allow users to remove thermally insignificant components from the analysis to speed up computational time. Mounting holes can also be filtered.

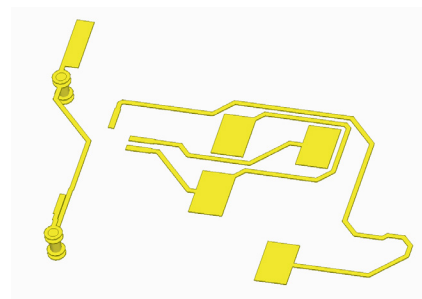
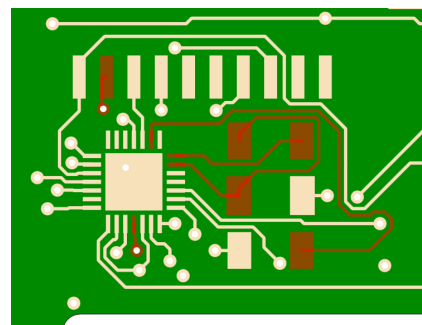


Component Filters <input type="checkbox"/>	
Filter Operator	Or
Footprint Dimension Less Than	3 mm
Height Less Than	1 mm
Power Less Than	0 W
Power Density Less Than	0 W/m ²
Reference Designator Contains	R C TP
Hole Filter	<input checked="" type="checkbox"/>
Hole Size Less Than	10 mm

Users can filter parts based on: Footprint Dimension, Height, Power, Power Density or Reference Designator.

Import Power List

A CSV file containing the reference designator and a number can be used to apply multiple power boundary conditions in one operation rather than on a part-by-part basis. This functionality is useful when a high number of components are present. A CSV file can be exported for later use or editing if needed.



For the latest product information, call us or visit: www.mentor.com/mechanical

©2017 Mentor Graphics Corporation, all rights reserved. This document contains information that is proprietary to Mentor Graphics Corporation and may be duplicated in whole or in part by the original recipient for internal business purposes only, provided that this entire notice appears in all copies. In accepting this document, the recipient agrees to make every reasonable effort to prevent unauthorized use of this information. All trademarks mentioned in this document are the trademarks of their respective owners.

Corporate Headquarters

Mentor Graphics Corporation

8005 SW Boeckman Road
Wilsonville, OR 97070-7777
Phone: 503.685.7000
Fax: 503.685.1204

Sales and Product Information

Phone: 800.547.3000
sales_info@mentor.com

Visit www.mentor.com/company/office_locations/ for the list of Mechanical Analysis Division Offices

Mentor[®]
A Siemens Business

MGC 08-17

1034740