

# Simcenter FLOEFD EDA Bridge Module

## Using imported detailed PCB designs and IC thermal properties to streamline thermal analysis

### Benefits

- Save time and effort by using imported detailed PCB designs and IC thermal properties for analysis
- Rapidly import detailed PCB data into Simcenter FLOEFD
- Improve analysis accuracy with more detailed thermal modeling of electronics

### Summary

Simcenter™ FLOEFD™ software, EDA Bridge module provides capabilities for detailed import of printed circuit boards (PCBs) into your mechanical computer-aided design (MCAD) tool of choice in preparation for thermal analysis. Historically, the best way to access PCB data was to use Intermediate Data Format (IDF) file pairs, which have significant shortcomings, especially regarding the copper geometry in the PCB.

The Simcenter FLOEFD EDA Bridge enables detailed PCB import with

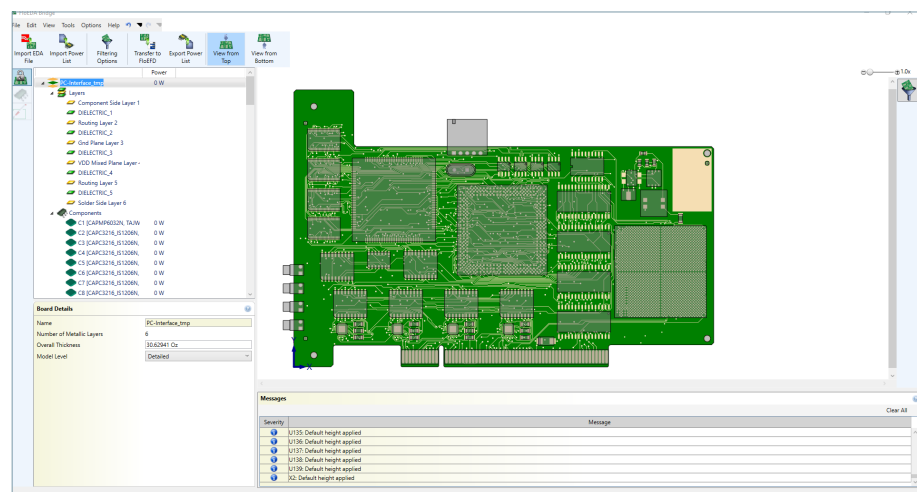
material and integrated circuit (IC) thermal properties into Simcenter FLOEFD for thermal analysis either on its own, or as part of a larger system-level assembly.

### PCB import file formats

The Simcenter FLOEFD EDA Bridge can use four file formats for import:

- IDF
- CC and CCE (native file format for Siemens Digital Industries Software's Xpedition and PADS)
- ODB++ (neutral file format for PCB manufacturing)
- IPC2581B (neutral file format of the IPC Digital Product Model Exchange consortium)

The benefit of using either CCE, ODB++ or IPC2581B is the PCB stack up and copper geometry can be read and used to create 3D geometry. This is particularly useful when thermal considerations,



# Simcenter FLOEFD EDA Bridge Module

such as thermal vertical connect access (vias) or copper pours, have been designed into the board.

## PCB modeling levels

The PCB can be modeled in one of three detailing levels depending on the granularity required by the thermal simulation:

1. Compact – Provide volume average of copper in the entire PCB. A biaxial material property is created to account for in-plane and through-plane thermal conductivity based on the copper content within the board. This option creates the lowest mesh overhead.
2. Detailed – Each layer is individually modeled and has its own material property based on the layer copper coverage. In addition, selected copper traces can be explicitly imported and therefore modeled in every detail including through connects. This option creates a medium to highest level mesh overhead if many traces are modeled explicitly.
3. Material map (smart PCB) – This model imports a pixelated representation of the PCB in which the detail of the representation depends on the resolution of the pixelation. This option creates the lowest mesh overhead but increased central processing unit (CPU) time depending on the resolution level.

## Explicit copper

When using CCE, ODB++ or IPC2581B, the netlist is also read by Simcenter FLOEFD EDA Bridge and can be selected from the PCB tree. Explicit copper geometry can be created in one of two ways:

1. Specific nets can be explicitly selected and modeled. The software will then be used to 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.

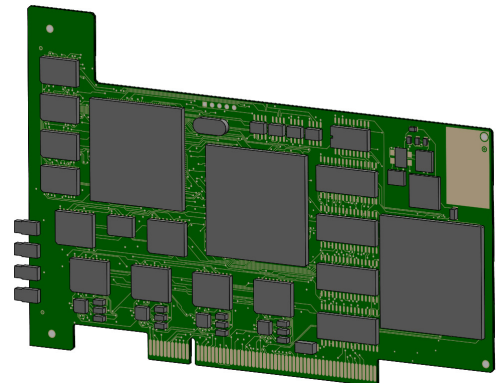
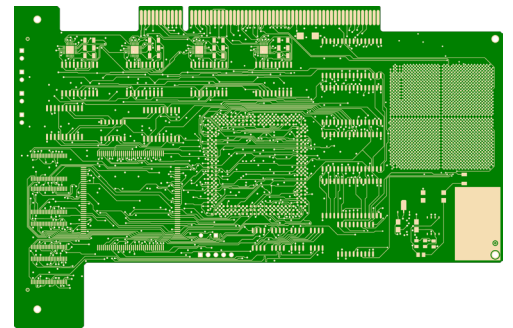
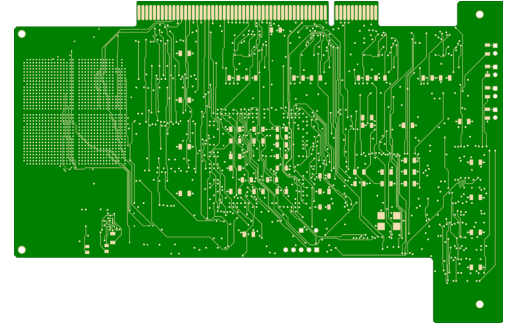
## IC modeling

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

1. Simple: Use block representations of the components. The size is based on the assembly or placement outline with the material properties defined.
2. Two-resistor: Use Joint Electron Device Engineering Council (JEDEC)  $\theta_{JB}$  and  $\theta_{JC}$  thermal resistances.
3. DELPHI multi-resistor: Use advanced thermal resistance network with additional nodes.

## PDML import

The PDML was originally a Simcenter Flotherm™ software format that is often used by vendors to supply users with IC package simulation model. This IC package definition in \*.pdml format can be imported into Simcenter FLOEFD and contains information about the geometry, power load, material properties or the thermal compact model definition and surface radiative properties.



Node	Capacitance	Initial Temperature	Power	Temperature	Goal	1	2	3
1 Junction	0 W/m <sup>2</sup> /K	293.2 K	10 W		OFF			
2 case	0 W/m <sup>2</sup> /K	293.2 K	0 W		OFF	0.36 K/W		
3 board	0 W/m <sup>2</sup> /K	293.2 K	0 W		OFF	4.19 K/W		

Component Filters	
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 <sup>2</sup>
Reference Designator Contains	R C TP
<b>Hole Filter</b> <input checked="" type="checkbox"/>	
Hole Size Less Than	10 mm

### Electronic component filtering

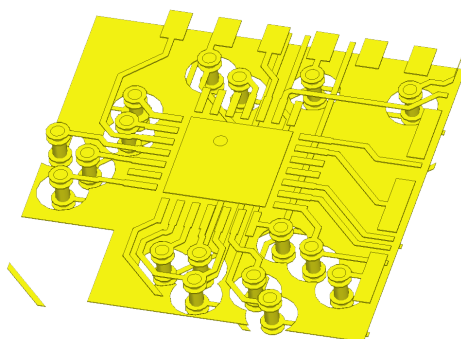
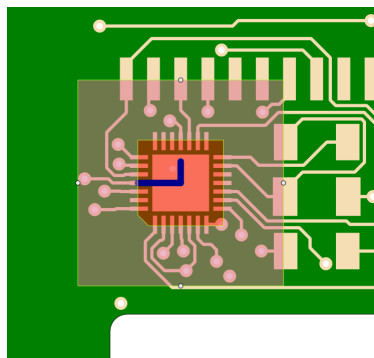
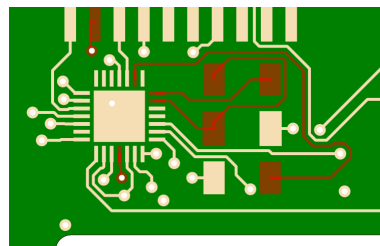
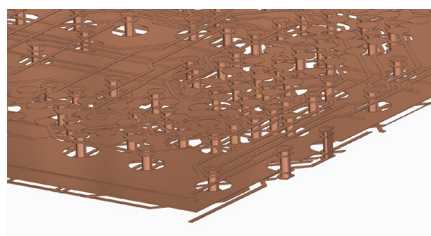
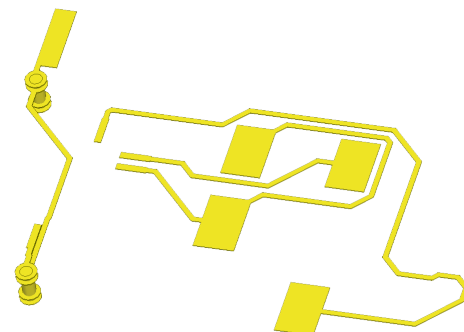
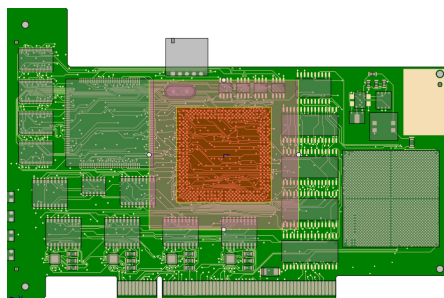
ICs, resistors and other components can be filtered based on one or more criteria. This is designed to enable users to remove thermally insignificant components from the analysis to speed up computational time. Mounting holes can also be filtered.

Users can filter parts based on: footprint dimension, height, power, power density or reference designator.

### Import power list

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

Possible imported boundary conditions range from IC modeling type and properties to their dissipated power.



Siemens Digital Industries Software  
siemens.com/software

Americas +1 314 264 8499  
Europe +44 (0) 1276 413200  
Asia-Pacific +852 2230 3333