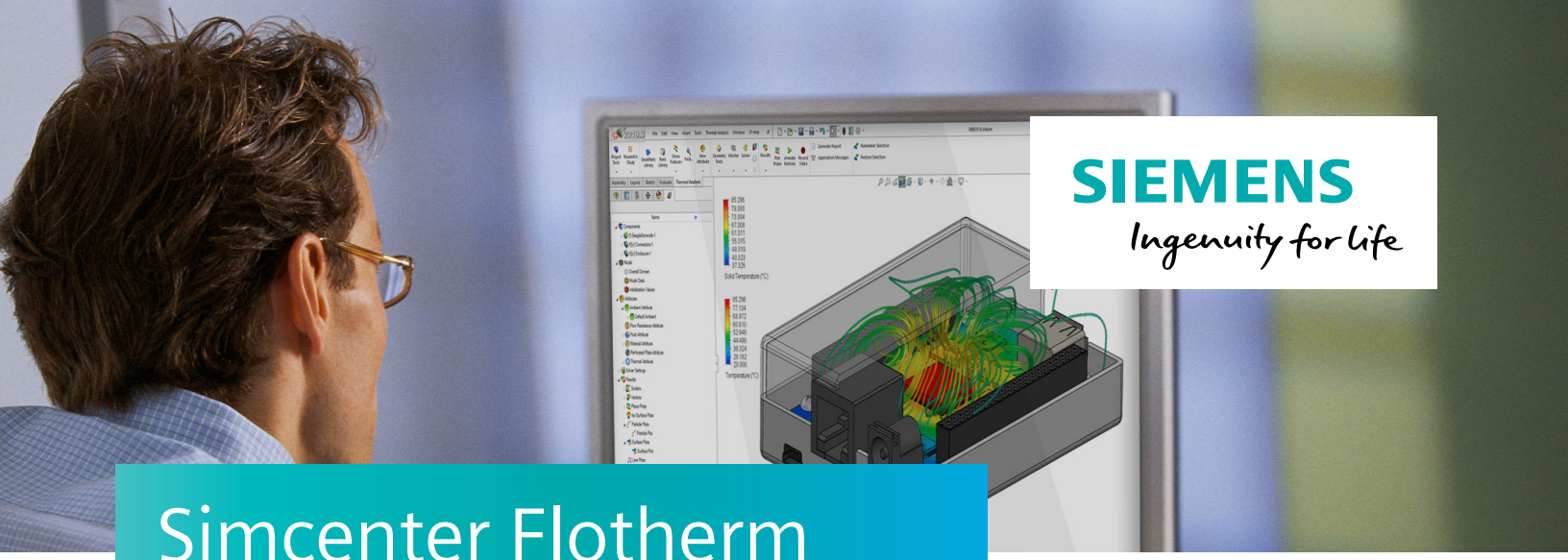


SIEMENS
Ingenuity for life

Simcenter Flotherm XT EDA Bridge

Import data from the main EDA systems for accurate thermal design in-sync with the EDA design flow

Benefits

- Import detailed PCB data into Simcenter Flotherm XT quickly and simply
- Minimize time and effort by using imported material and IC thermal properties
- Imported board model is 100% ready for thermal analysis
- Multiple modeling levels to match speed and accuracy of the simulation with detail available from the EDA design flow
- Includes powerful 'Edit EDA Data' function to import a new EDA file while retaining the component powers and model types making it easy to keep the thermal model in sync with the EDA design flow.
- Full undo/redo capability, with ability to save work in-progress without losing any existing modifications to an imported EDA file.
- Includes a board outline editor; a power mode editor to switch between multiple powered states associated with the use case for the product being designed

Summary

The Simcenter™ Flotherm™ XT EDA Bridge module provides additional capabilities for detailed PCB import into Simcenter Flotherm XT, which is part of Xcelerator, a comprehensive and integrated portfolio of software and services form Siemens Digital Industries Software.

The Simcenter Flotherm XT EDA Bridge module allows for detailed PCB import with material and IC thermal properties into Simcenter Flotherm XT for subsequent thermal analysis either on its own, or as part of a larger system-level assembly. The FLOEDA Bridge module can import several different file formats:

- IDF
- IDX (currently does not read detailed copper)
- CCE (native file format for Mentor's Xpedition or PADS software)
- ODB++ (a neutral file format for PCB manufacturing)
- IPC-2581B

The benefit of using CCE, ODB++ or IPC-2581B 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 modeling levels:
The PCB can be modelled in one of three levels depending on the granularity required from the thermal simulation:

Compact: 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.

Detailed: Each layer will have its own material property based on the layer copper coverage, this includes dielectric layers with vias.

Material map: A unique approach where the copper and dielectric within the routed board are represented using a network assembly. The fidelity of the representation can be adjusted by switching between fine, which ensures two network assemblies across the width of the smallest trace, and averaging, which allows for full control to either coarsen or refine.

Material representation:
Analytical: A well-known legacy approach where effective properties are determined based on volume averaging of the copper and dielectric of either the individual board layers or entire board.

Simcenter Flotherm XT EDA Bridge

Benefits *continued*

- Power, maximum case temperature and maximum junction temperature for each component can be imported where supported by the imported file format.
- Joule heating is supported by allowing the user to transfer individual nets to Simcenter Flotherm XT. The user will be able to apply the appropriate boundary conditions to perform the analysis.
- Block, 2-Resistor, DELPHI or 'detailed' component modeling levels can be selected, including an automated ability to 'swap-in' higher fidelity models from an existing thermal model library.
- RC ladder compact thermal models created by Simcenter T3STER and Simcenter POWERTESTER can be read in, as can Network Assemblies from Simcenter Flotherm PACK.

Empirical: A unique, patented approach where effective properties are based on a percent coverage correlation to explicit copper representation. Multiple validation examples have shown the results based on Empirical effective thermal conductivities more accurately predict component temperatures than the Analytical method.

Explicit copper: After the board has been routed Simcenter Flotherm XT EDA Bridge module can import CCE, ODB++ or IPC-2581B which contains the board netlist and associated copper layout. Unique approaches to modeling explicit copper geometry can be created in one of two ways.

- *Explicit net:* Specific nets can be selected and modelled as explicit. The software will then create 3D geometry to resemble that entire net, including vias, in Simcenter Flotherm XT. This is particularly useful when performing a Joule heating Thermal analysis on specific individual nets.
- *Thermal territories:* Specific IC packages can be selected to model the neighboring copper explicitly. This can encompass many nets but will not exceed the physical extent around the IC specified by the user. The depth of the thermal territory through the board can also be set.

Chip package modelling:
Chip packages can be thermally represented in a variety of ways, listed here in order of increasing thermal accuracy.

If component heights are not set in the EDA tool, then a default can be specified during import.

Simple block: a simple thermally-conducting block can be used to representation components that are thermally benign, or have low power densities. The block size can be based on the placement outline or assembly outline with the material properties defined. During import of the CCE file, the user is prompted to select either Placement or Assembly Component Outline. Currently, there are 17 selectable material properties for simple components depending on the package family.

2-Resistor: JEDEC guideline compliant thermal resistances θJB and θJC available from datasheets specified in EDA Bridge it will be imported into Simcenter Flotherm XT as a 2-Resistor Network Assembly.

DELPHI multi resistor: JEDEC guideline compliant advanced thermal resistance network with additional network nodes imported as a Network Assembly.

Detailed model: a 3D CAD model of the chip package containing all the thermally-important internal features.

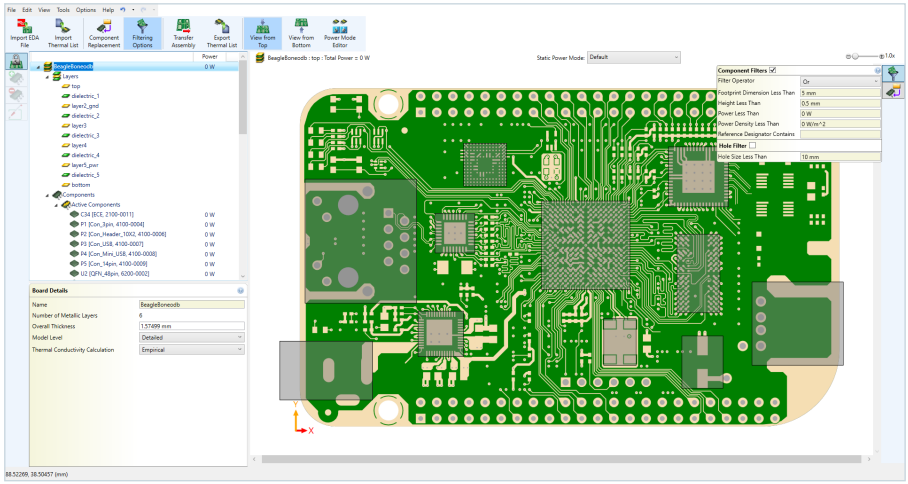
Existing component models can be replaced by component models from the user library. The component is replaced based on the origin of the part. The library component can be simple block, 2-Resistor, DELPHI network or fully detailed component.

Component filtering:
Components on the board can be filtered based on one or more criteria. Users can filter parts based on: footprint dimension, height, power, power density or reference designator, thereby allowing users to remove thermally-insignificant components from the analysis to speed up computational time. Mounting holes can also be filtered.

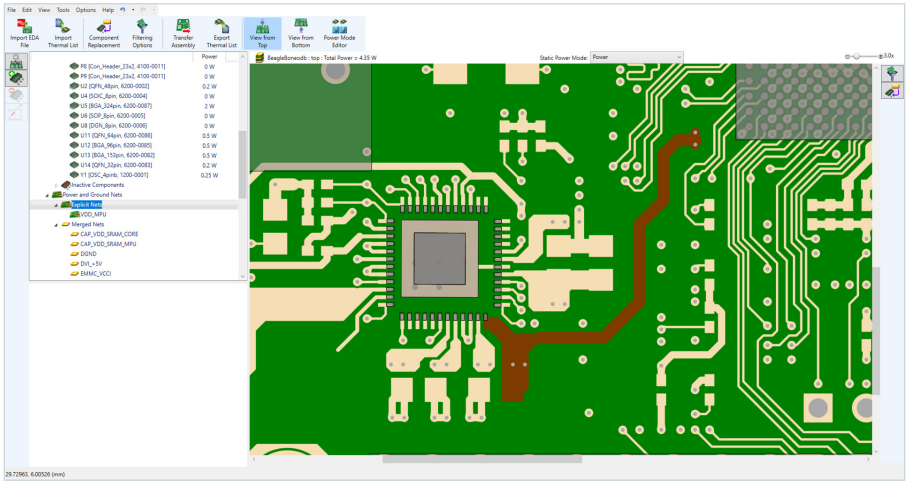
Power modes:
The use of power modes allows the user to create different scenarios for the power consumption of the PCB depending on how the system is being used, with modern thermal design focused on understanding and managing the thermal performance for a number of use cases. Each use case corresponds to a power mode, which is defined as a set of powers for each component that are then transferred into Simcenter Flotherm XT.

The different power modes can be used to run a parametric study comparing maximum IC temperature from each power mode, and the response of switching from one power mode to another.

Power modes can be created using two different methods. In the EDA Bridge graphics window the user can select Power Mode Editor and create both static and dynamic power modes. Alternatively, Static Power Modes can be created using the Import Thermal List option. The user can Export the Thermal List and save as an excel spreadsheet. By adding power mode columns to the right of the default column, the user can add as many static power modes as they want to analyze.



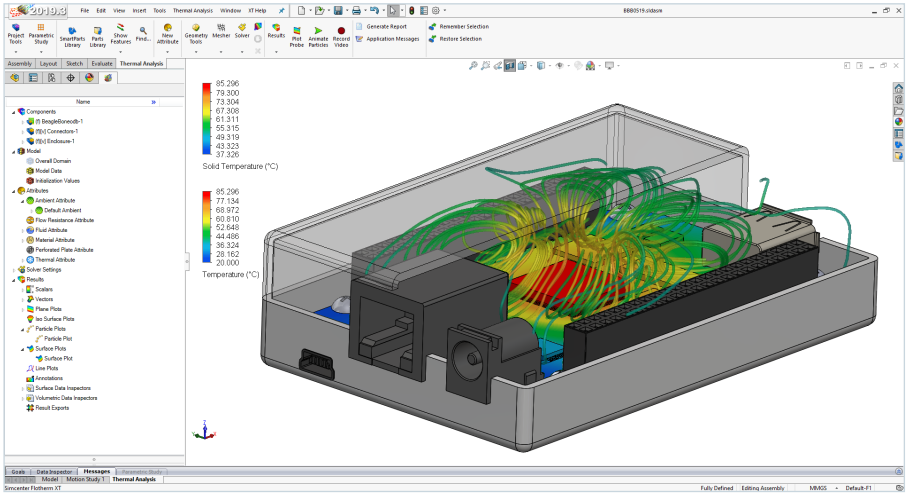
Board with thermal territories defined, and component models swapped in from library



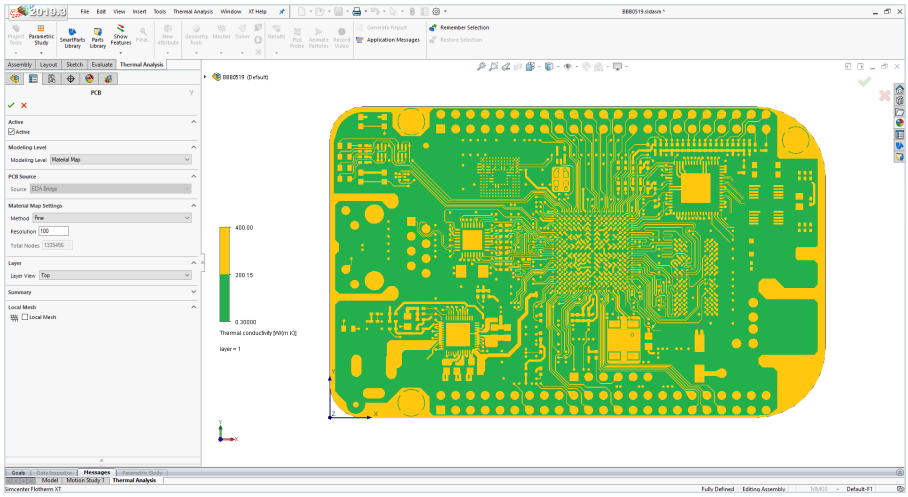
Single net selected for transfer for Joule heating analysis

Import thermal list:
An excel or 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 component-by-component basis. This functionality is extremely useful when a high number of components are present. A Microsoft Excel or CSV file can also exported for later use or to create a power list for checking or editing.

The power list allows the user to set the component to either Filtered/Unfiltered, and to define a max case temp, a max junction temp, a junction-to-case resistance, a junction-to-board resistance and different power modes. User can change existing power modes or create new power modes.



Simulation after board is imported into Simcenter Flotherm XT



High fidelity material map capturing local copper detail

Siemens Digital Industries Software
siemens.com/software

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

82123-C3 6/20 A