

Module 3: Labs - Thermal Simulation of Packages Using Ansys

28 April 2025 12:38 AM

L1: Intro & Getting Started with Ansys Electronics Desktop

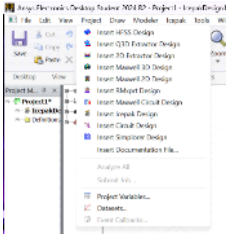
Tool: Ansys Electronics Desktop Student

For labs, we would be importing a flip chip geometry & will be performing analysis.

EDT offers a suite of tool for testing & working.

HFSS: For RF & EMI simulations
Q3D: For circuits
Icepack: One which will be using to carry out thermal analysis.

How to select Icepack: Go to "Projects" -> "Insert Icepack Design" or my Selecting "Icepack" in the tool ribbon.



Panel Description:

Left Panel:

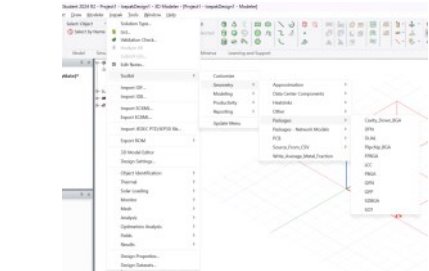
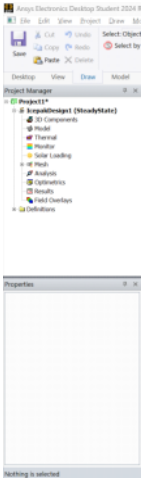
- 1. Option to add 3D models, thermal boundaries & edges to be monitored.
- 2. Meshing: Uses FEM analysis which forms a mesh.
- 3. Analysis: Solutions for the thermal equations.
- 4. Results: Thermal sim results.

In "Draw": we can construct our own models

For this training, we would be importing an in-built package.
Can import PCBs too.

To Import:

- 1. Select the "Icepack" button from the top ribbon
- 2. In the drop down, select "Toolkit" -> "Geometry" -> "Packages" -> Select the package that you'd like to check. (Flipchip BGA) for tutorial.



L2: Setting Up a Flip Chip BGA Package

When we select the package type as flip chip BGA, a prompt is show.
This allows us to define the physical specifications of the packages, model the internal die, substrate & solder.

Dimensions: Allows us to set the external packages dimensions & specify it's position



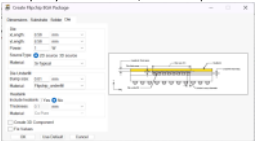
Substrate: Can specify the number of connection layers, substrate thickness, trace signals, material for traces & vias (thermal & electrical).



Solder: Define as a peripheral array or full array, row count, ball pitch & ball material.



Die: Specify die dimensions, material & power dissipation along with underfill material, bump pitch & heatsinks.



By generating the chip, we can observe its layers from the model pane & by selecting the individual layers

L3: Material Definitions & Thermal Power Sources

In Model: We can get a design list for all materials.

Next display to explore: Thermal



We then define the power boundary condition. In this case, we could analyse the total power generated



next display to explore: thermal



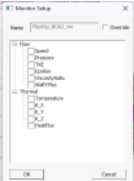
We then define the power boundary condition. In this case, we could analyse the total power generated across the die.

Now, it is also necessary to put a substrate boundary condition. Here, we define that the substrate is at a fixed ambient temperature.

Procedure: Right click layer -> Assign Thermal -> Source & then in the drop down, select Fixed temperature

In case of any errors, we also need to check for any messages.
We cannot have multiple overlapping boundary conditions

Following boundary conditions, we move to assigning monitors for temperature.
Procedure: Right-click on layer -> Assign Monitor -> Point & select the type of monitor.



Do the same for die, underfill.

L4: Meshing & Running Thermal Analysis

In our case, we can avoid addition of any solar loading & move directly to adding meshes.

Procedure:
Select "Simulation" from the top ribbon bar -> Select "Generate Mesh".
Name the project & save it.

Once the mesh has been generated, we get a window which states the mesh & check properties of the mesh from the "Quality tab of the display"

Can do meshing for the entire system or re-do meshing for specific layers.

We close the popup & then check for the mesh under the "Mesh" tab of the Project Manager.

Once done, we shift towards analysis.

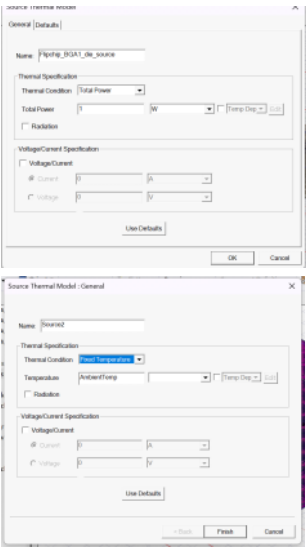
For Analysis -> Right click "Analysis" from the Project Manager and select "Add Solution Step".

In terms of settings, we keep the default values & check all settings.

One done, we select "OK".

Once everything is setup, we validate the settings by selecting the "Validate" option from the top ribbon in the simulation tab.

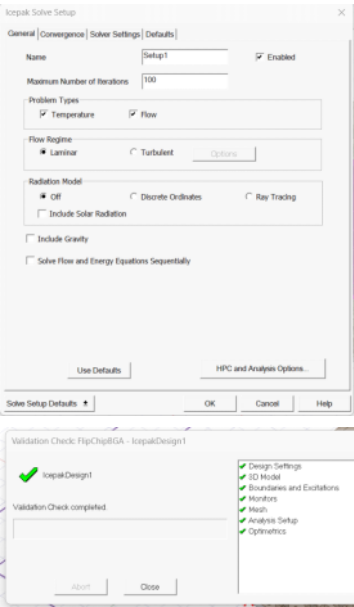
Once done, we get a panel with check marks against the inputs provided.
Does not show if the conditions provided are correct, but will show that they have been set up.



Skewness: Gives us an estimate of the quality of points generated on the mesh.

Higher the number, better are the simulation results.

Meshing is important to have a more accurate simulation.



L5: Viewing Results & Exploring Other Package Types

Upon normal completion, we know that the simulation is now complete.
Now, we can plot the results over the package by selecting the package, right clicking it and selecting "Plot Fields"

We Specify the plot names, folder. tick the "Surface Only" option, enable gaussian smoothing and generating the plot.

Post this, we can analyse the package based on the temperature given on display.

