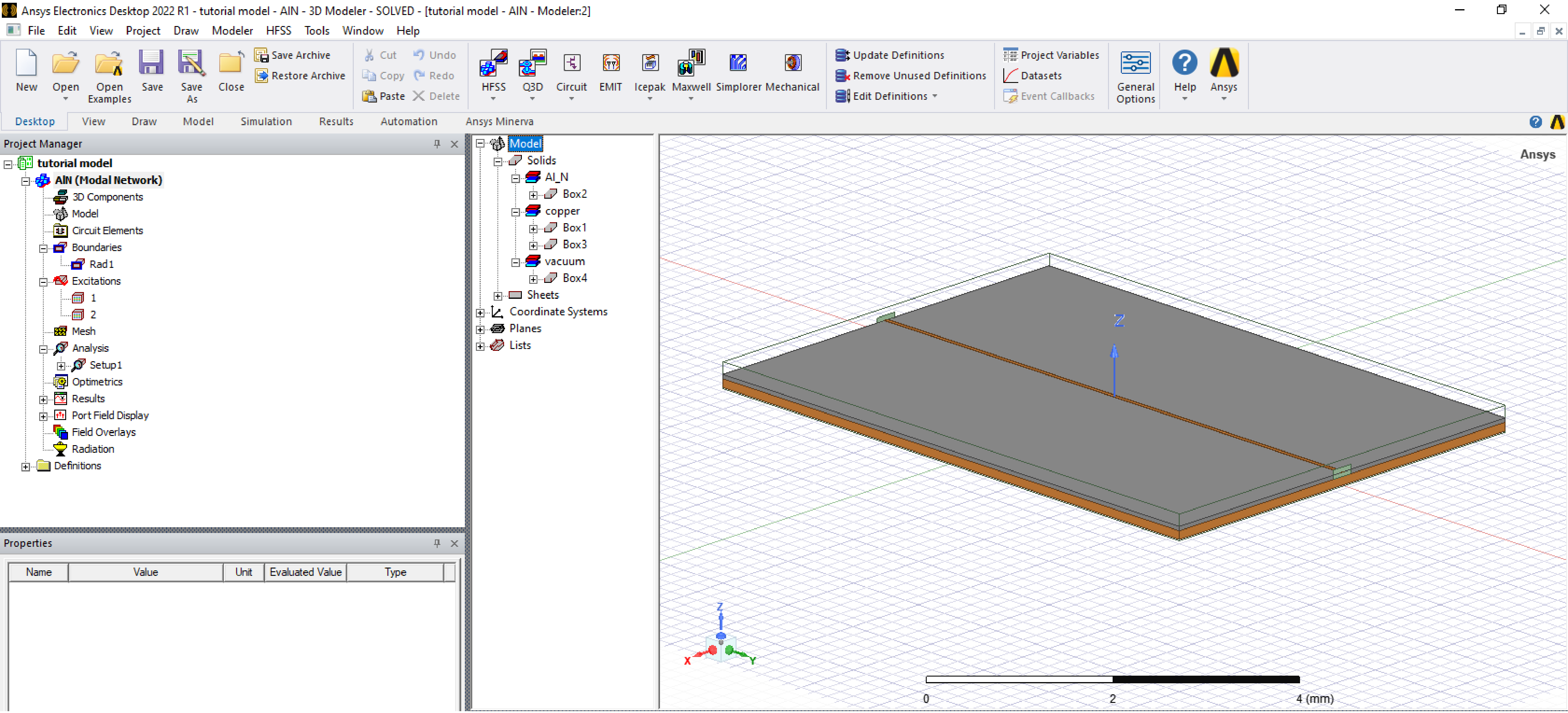
Aubin Lab Guide to Ansys Thermal Simulations

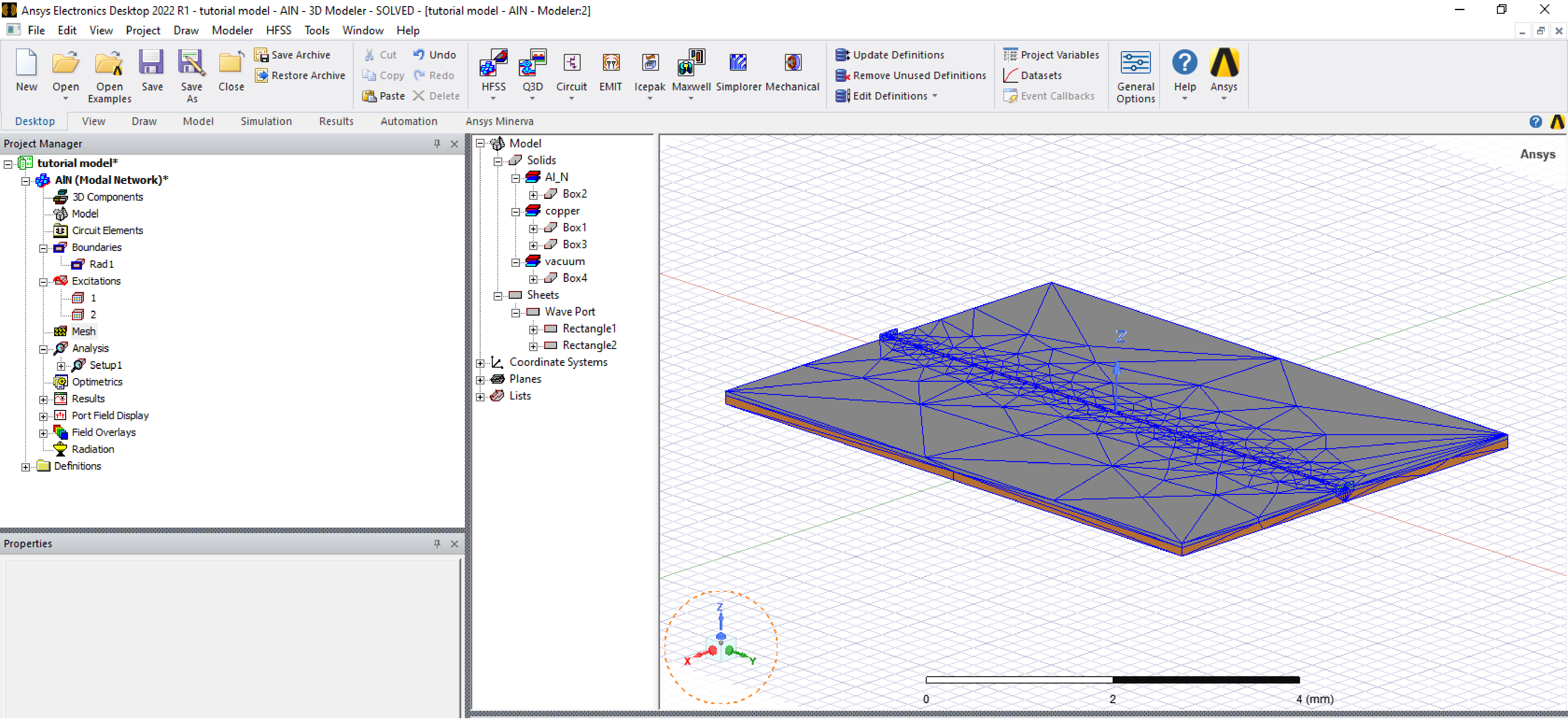
Paul Kelemen

May 2025



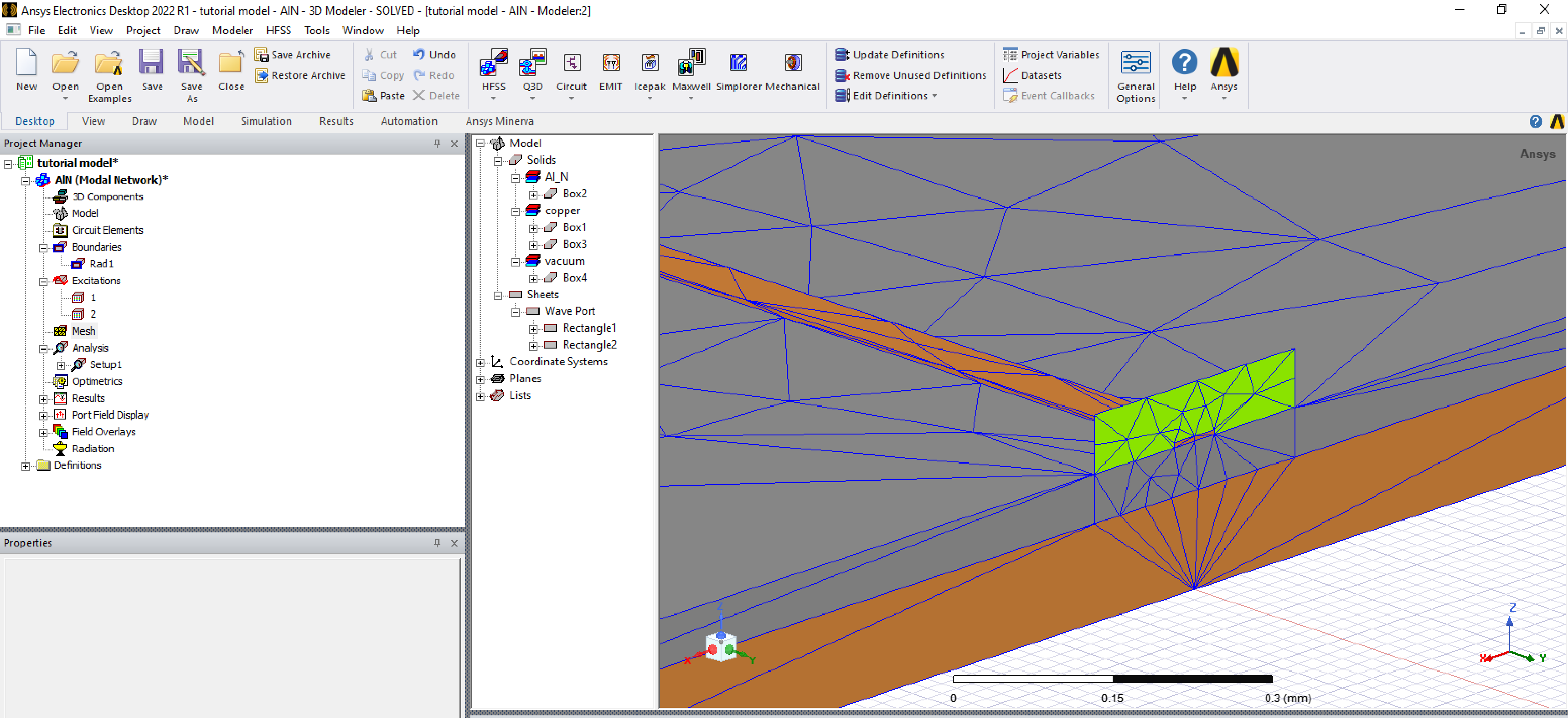
This tutorial presents information and tips on how to create effective thermal simulations for Ansys HFSS models.
Pictured above is a simple HFSS model, consisting of a copper ground plate, AlN heatsink thick-film, copper microstrip, and dual wave ports (20 watts).
For the majority of this tutorial, the “vacuum” box will be hidden from active view.

Intro



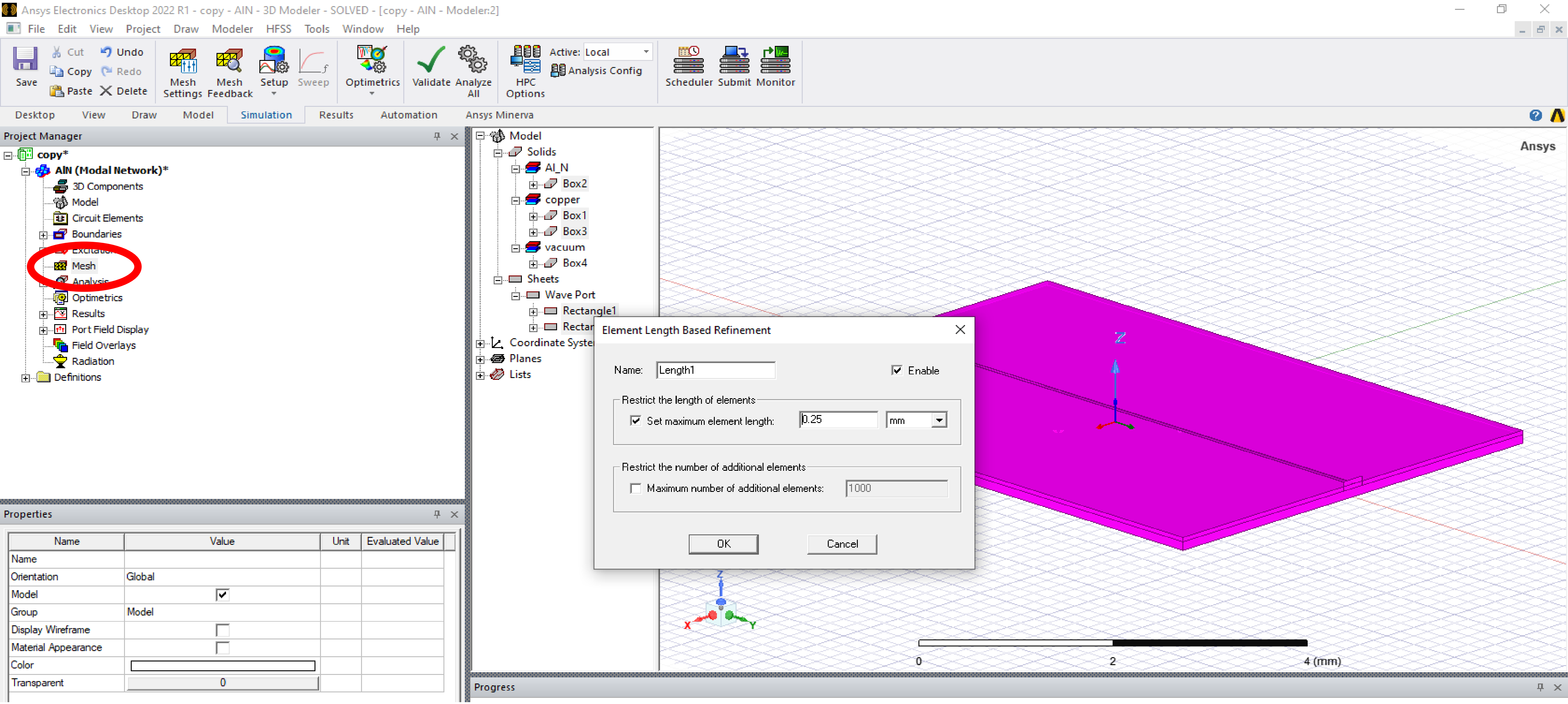
Ansys thermal simulations require small meshing to accurately calculate surface temperatures in your model.
For this reason, the large meshes autogenerated by HFSS for E&M simulation (pictured above) are generally insufficient and must be adjusted.

Remeshing



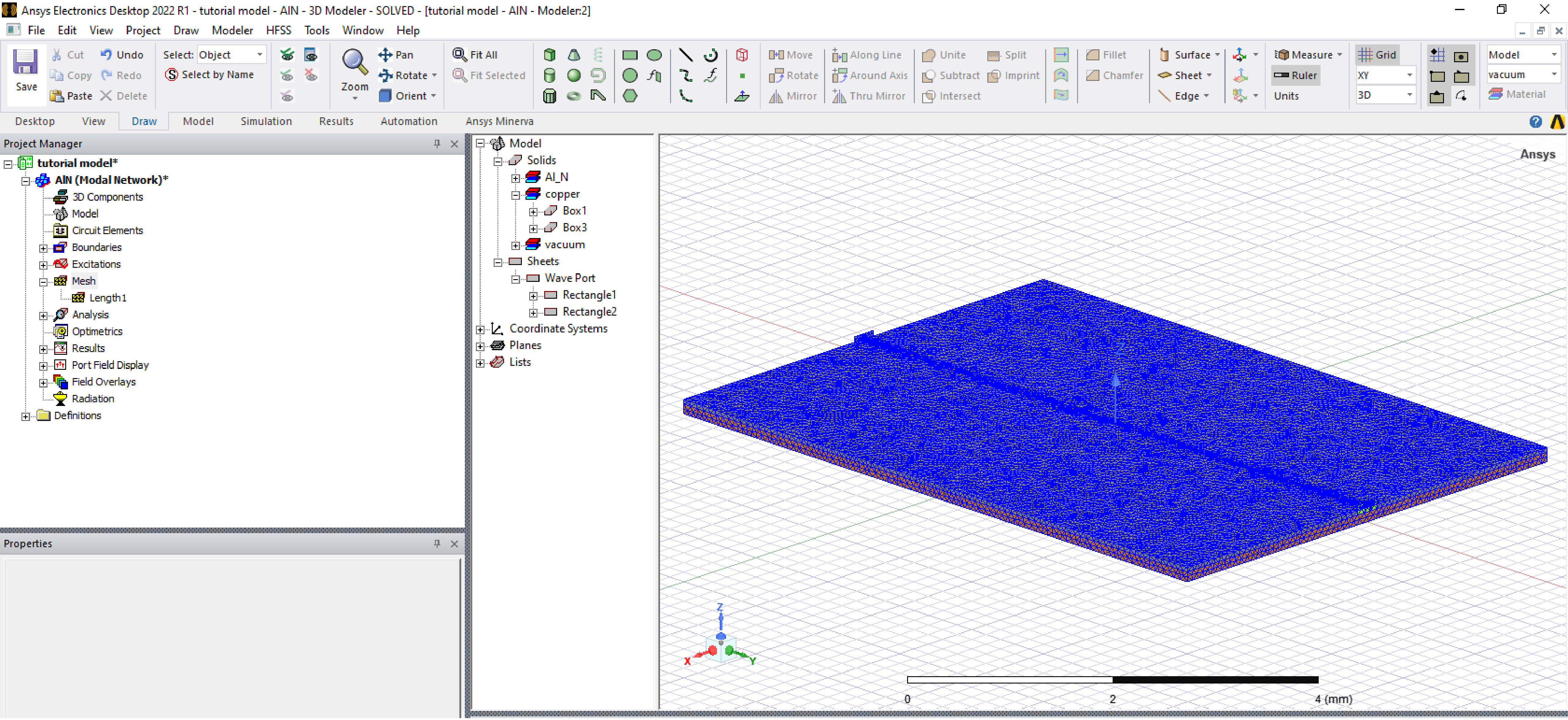
Pictured above is a zoomed-in image of the previous slide, further demonstrating the large mesh over the microstrip and surrounding heatsink. Shrinking this mesh for our model will result in higher-definition thermal simulations.

Remeshing (cont.)



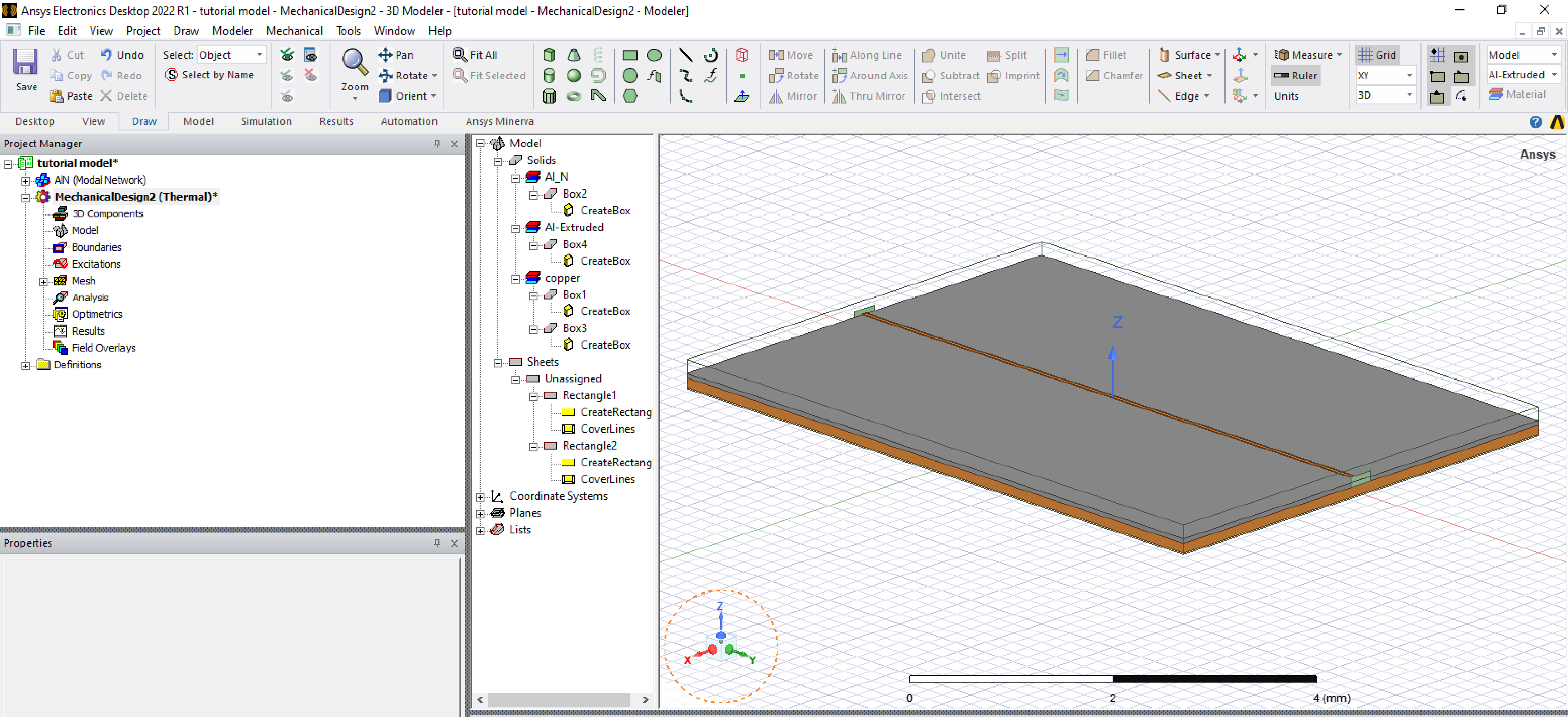
Use “Ctrl+A” to select the entirety of the HFSS model, and right-click on “Mesh” in your project manager (circled in red).
Navigate as follows: “Assign Mesh Operation” -> “On Selection” -> “Length Based”.
From here, you can set a “maximum element length” across the model body.
Select an initial “guess” length, rerun your E&M simulation, and generate a plot of your mesh. Adjust your “guess” until satisfied with mesh element density.

Remeshing (cont.)



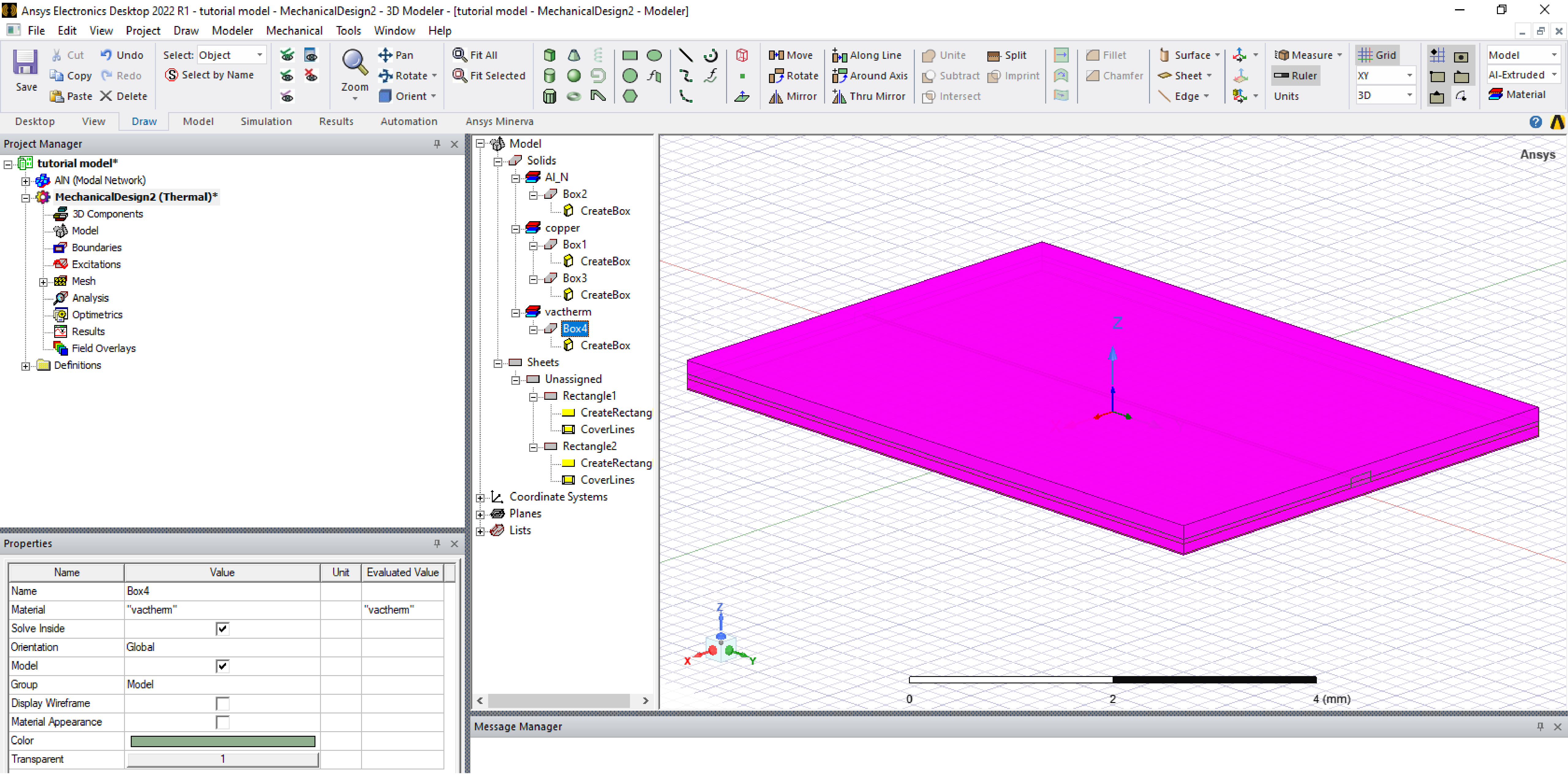
Pictured above is a new plot of our surface mesh after applying mesh operations on slide 5

Remeshing (cont.)



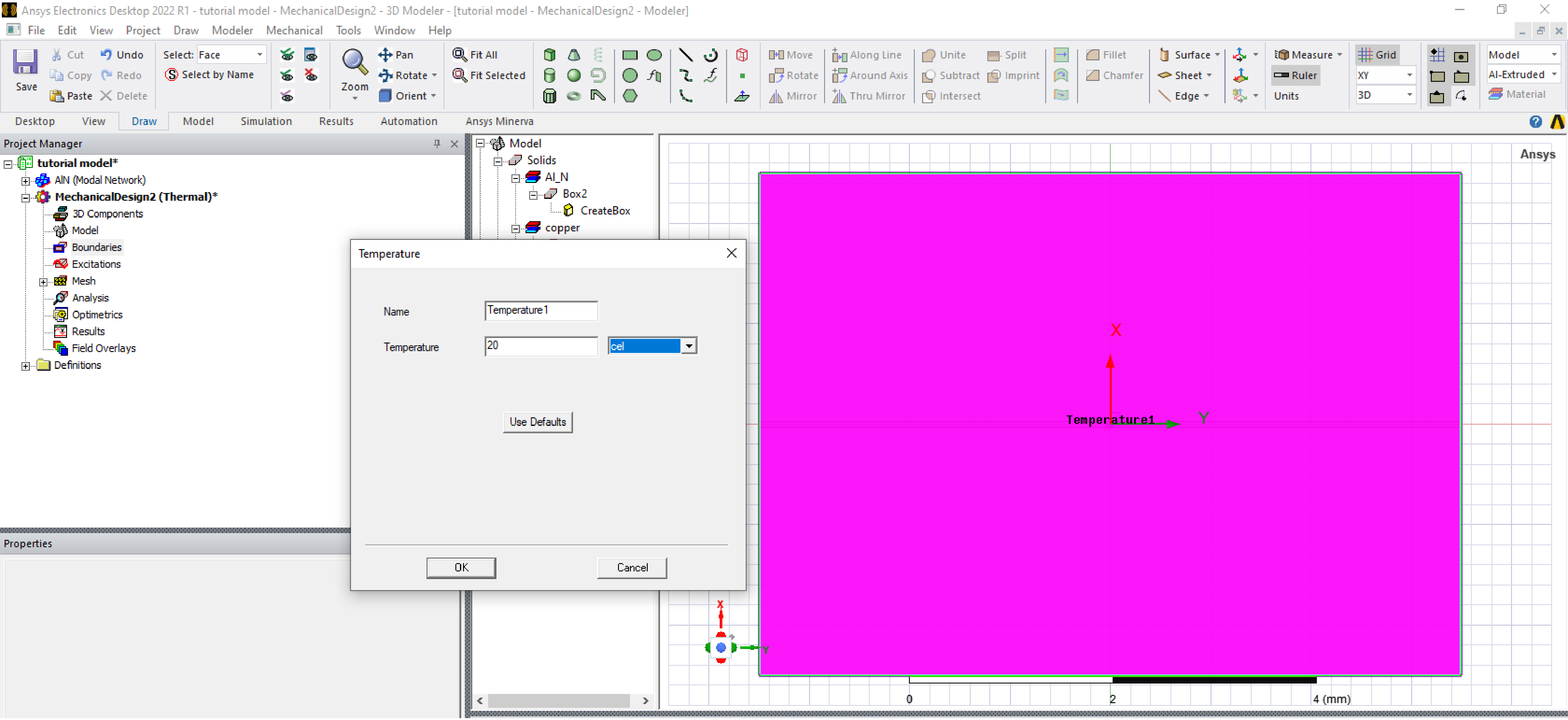
Navigate to your Ansys desktop and insert a “MechanicalDesign” workspace. Copy and paste your model from HFSS workspace to your “MechanicalDesign” workspace.
Note that when using the “vacuum” material in HFSS, the “MechanicalDesign” workspace will automatically change the material to an extruded variant of another material in your model.
This happens because the “vacuum” material has a value of zero for its thermal conductivity. We will fix this error on the next slide.

Initiating a Mechanical Workspace



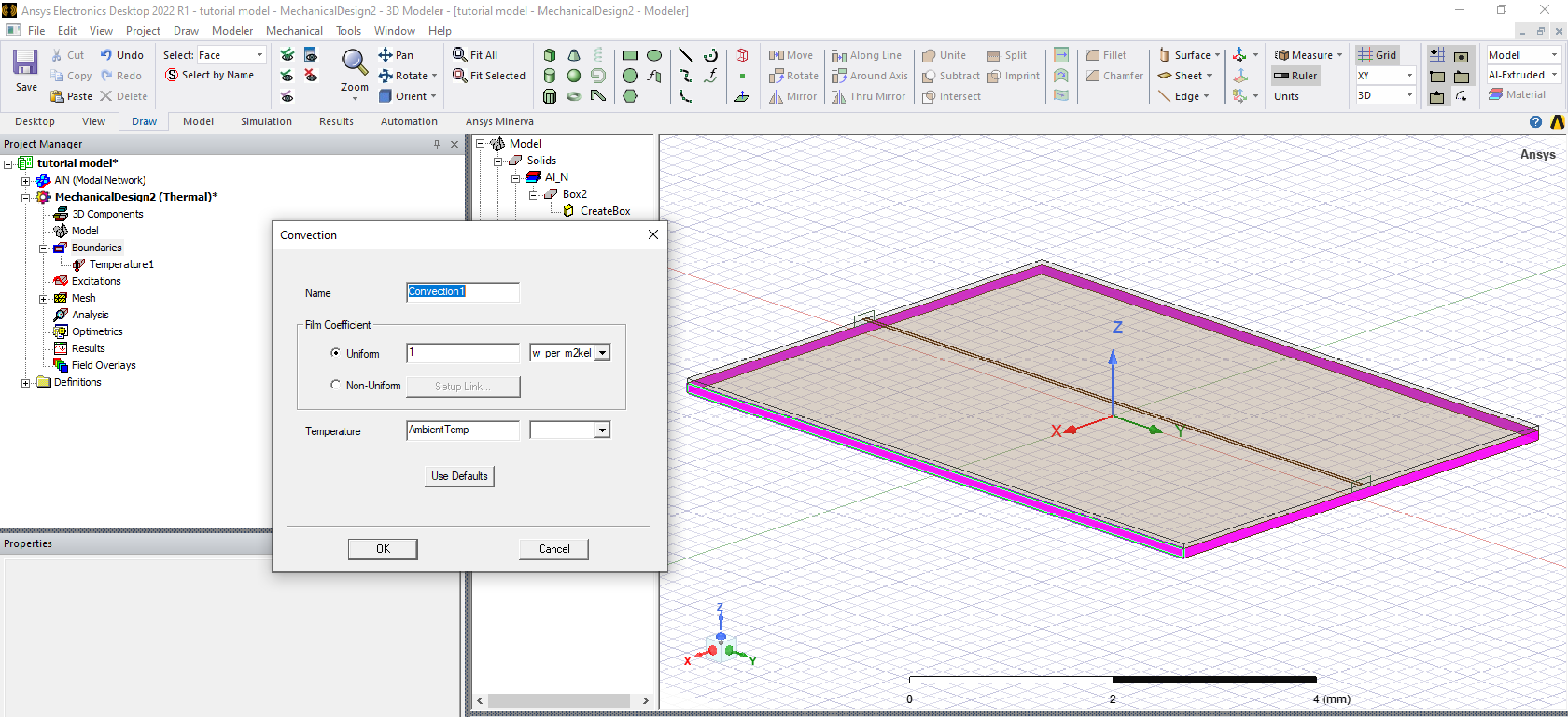
Define a new material called “vactherm” that has properties otherwise identical to the “vacuum” material, save its thermal conductivity. For this value, input a very small number x<10^-4. This will allow you to effectively simulate the “vacuum” material.

Simulating a Vacuum



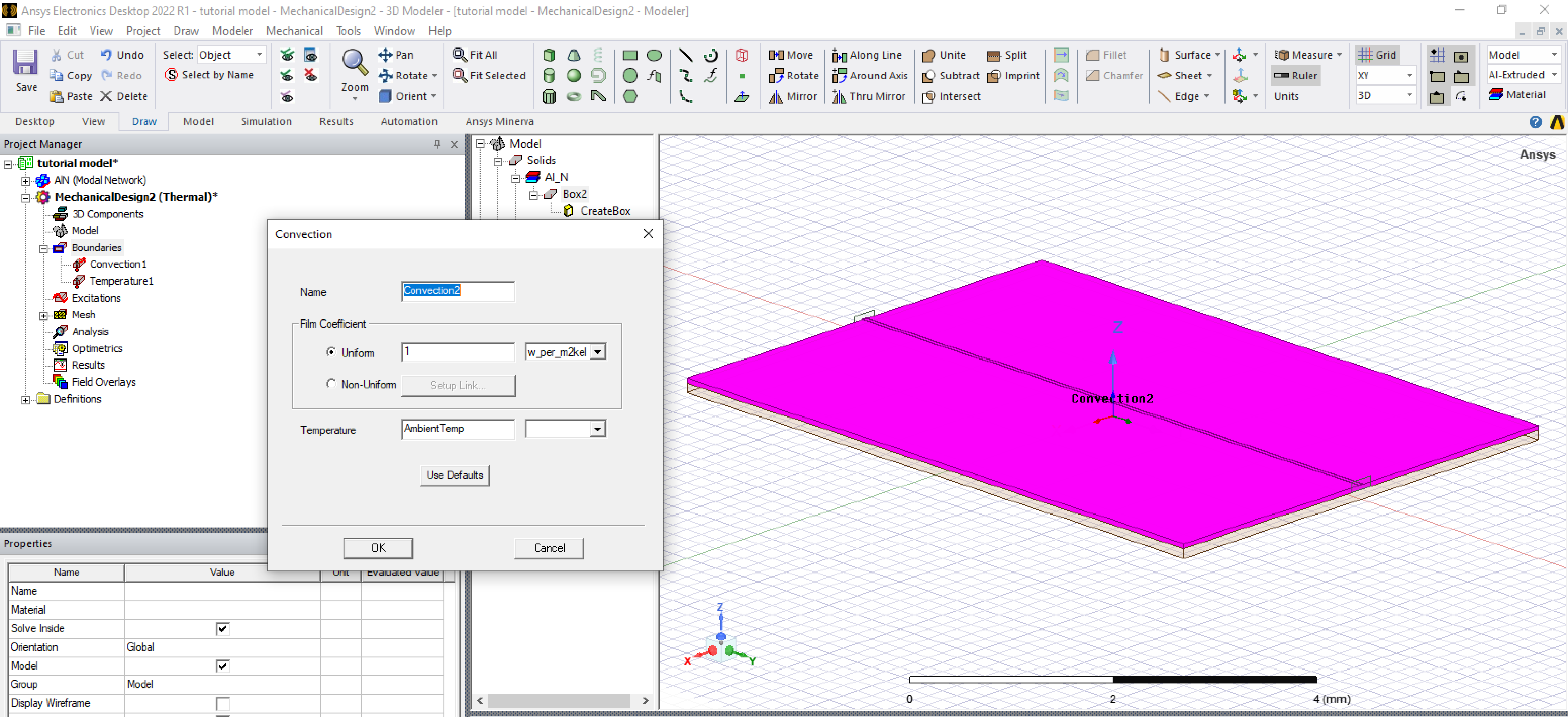
At the bottom of our copper ground plate, we will set a temperature boundary condition. Set your selection type to “Face” and select the bottom plane. Right click the plane. Set a temperature boundary and input your desired condition. For this example, we will use 20C.

Setting Temperature Boundary Conditions



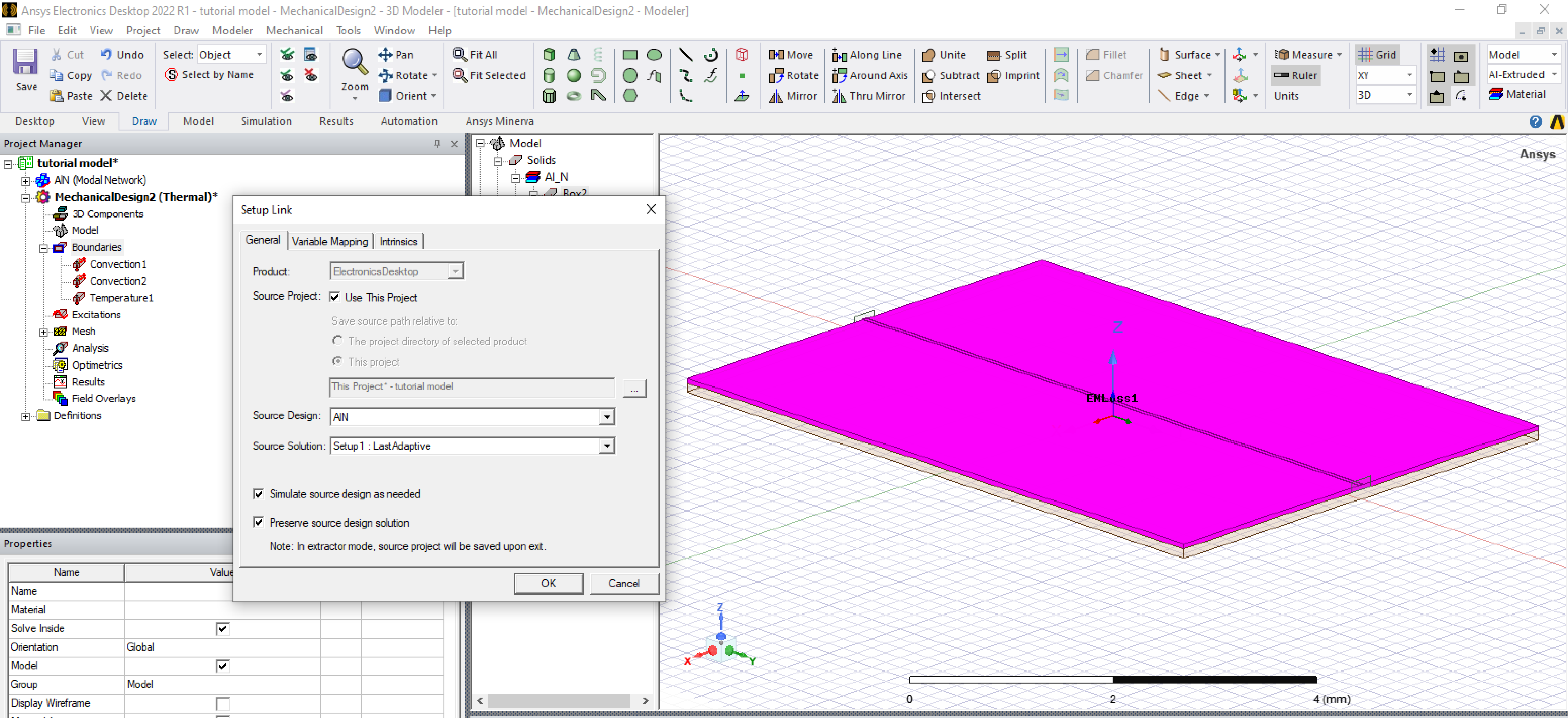
The “MechanicalDesign” workspace will not allow other boundaries or excitations to overlap with your temperature boundary condition. For this reason, we will set convection boundary conditions in two steps. First, select the four vertical faces of our ground plane. Right click on them and set a convection boundary as depicted above.

Setting Convection Boundary Conditions



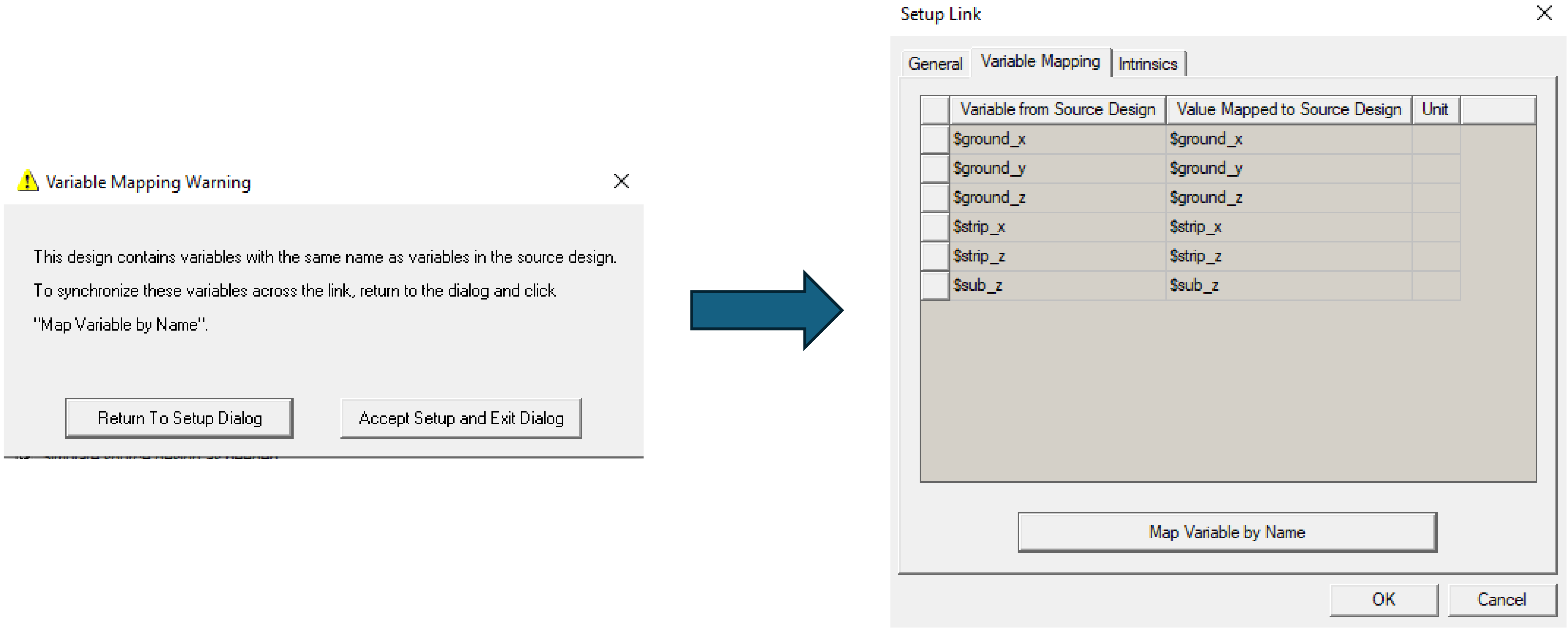
Next, reset your selection type to “Object” and select the AlN heatsink plate and microstrip.
Right click on these objects and set a convection boundary as depicted above.

Convection Boundary Conditions (cont.)

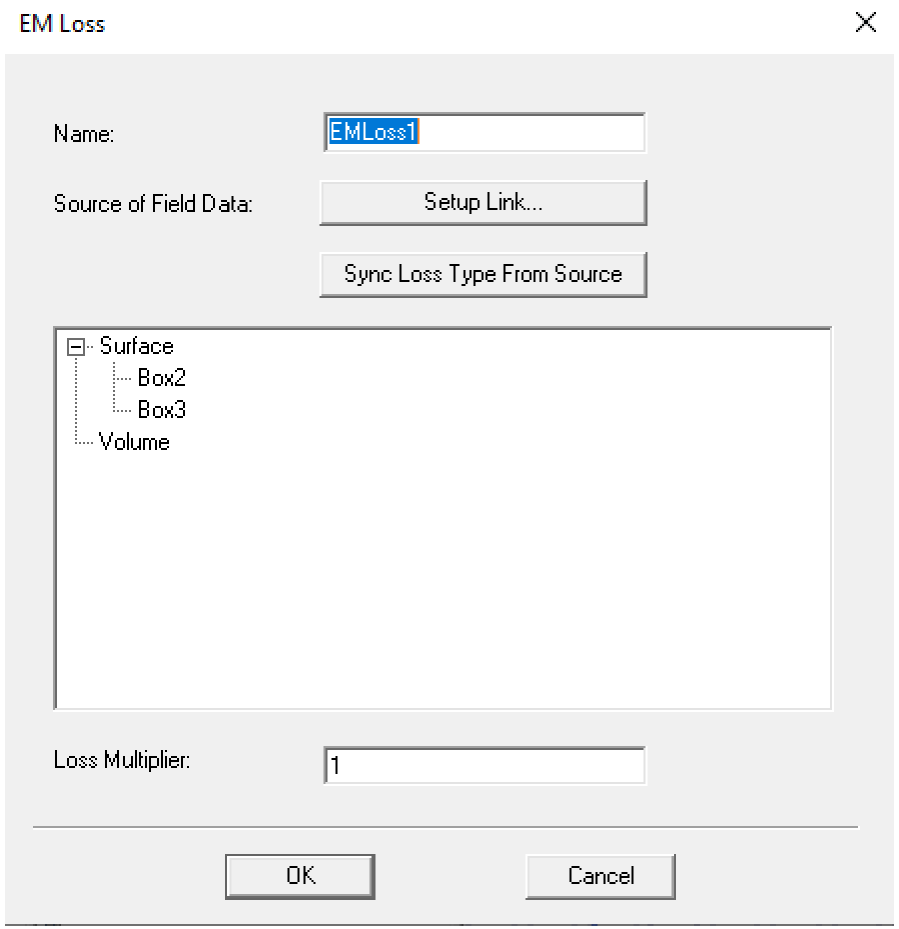


Next, reselect the AlN heatsink plate and microstrip. Assign an "EMLoss" excitation.
A pop-up menu should appear. Select "Use this project", "Simulate source design as needed", and "Preserve source design solution".

Assigning "EMLoss" Excitation

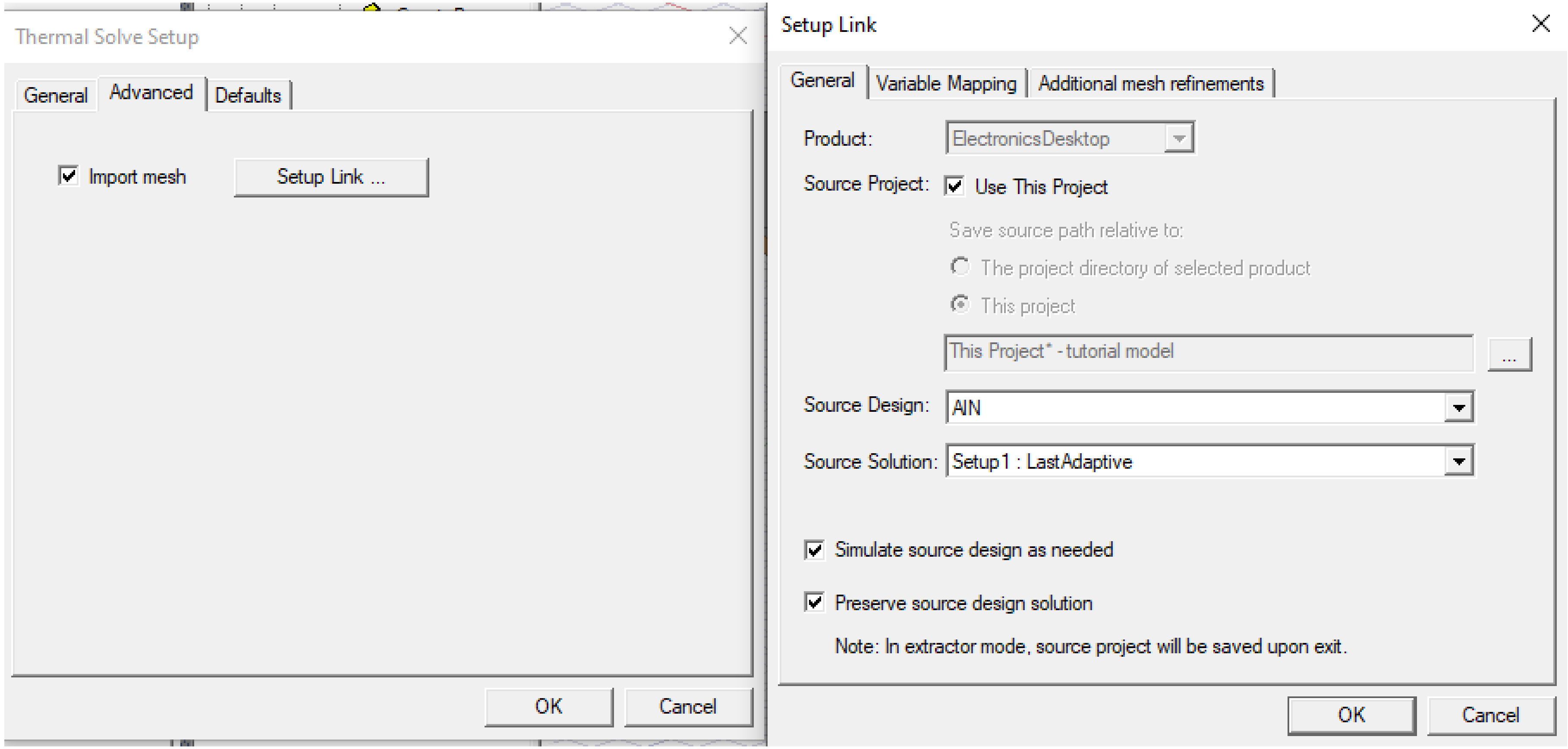


The following warning message may pop up. Select “Return to Setup Dialog”, and on the next menu, click “Map Variable by Name”.



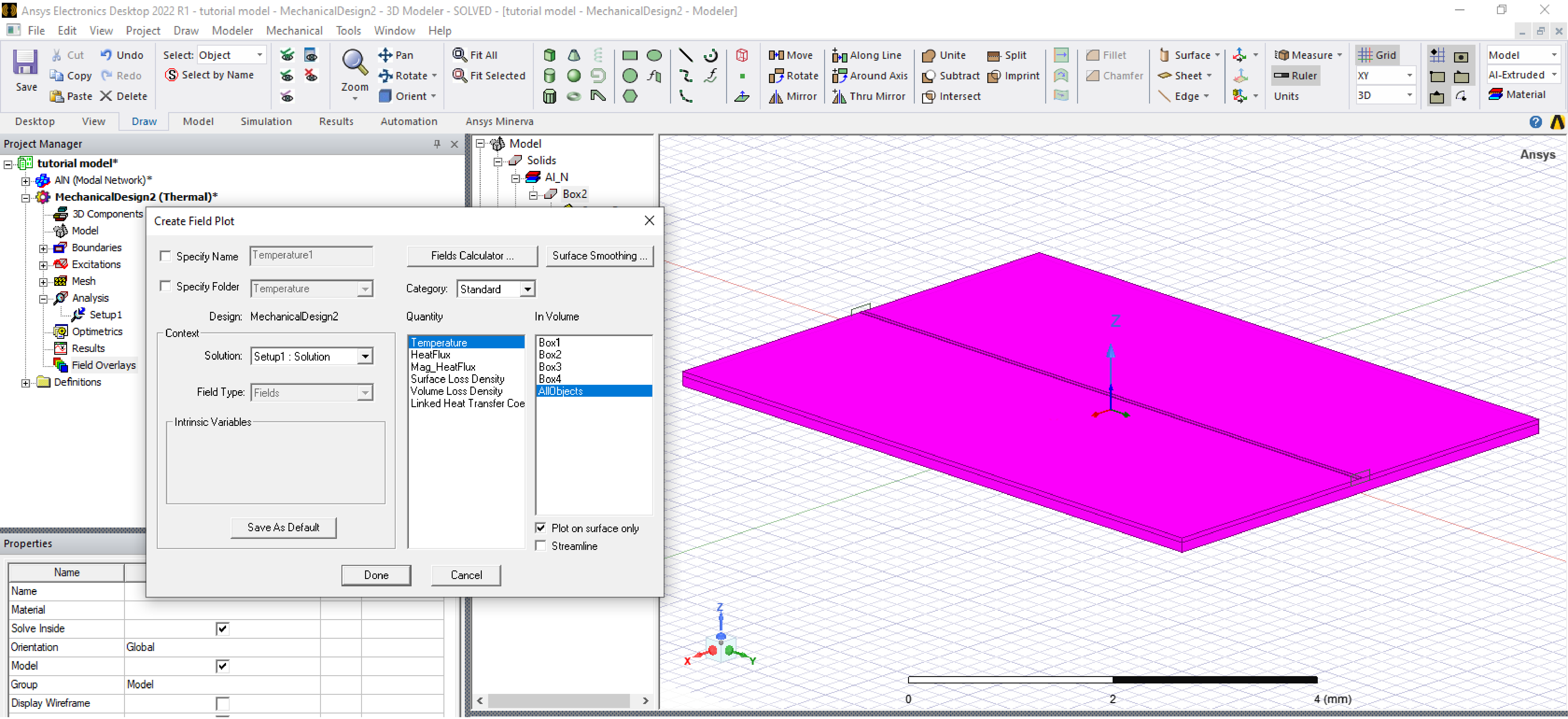
Once the last two steps have been completed. The following menu should pop up. Select “Ok”.

“EMLoss” Excitation (cont.)



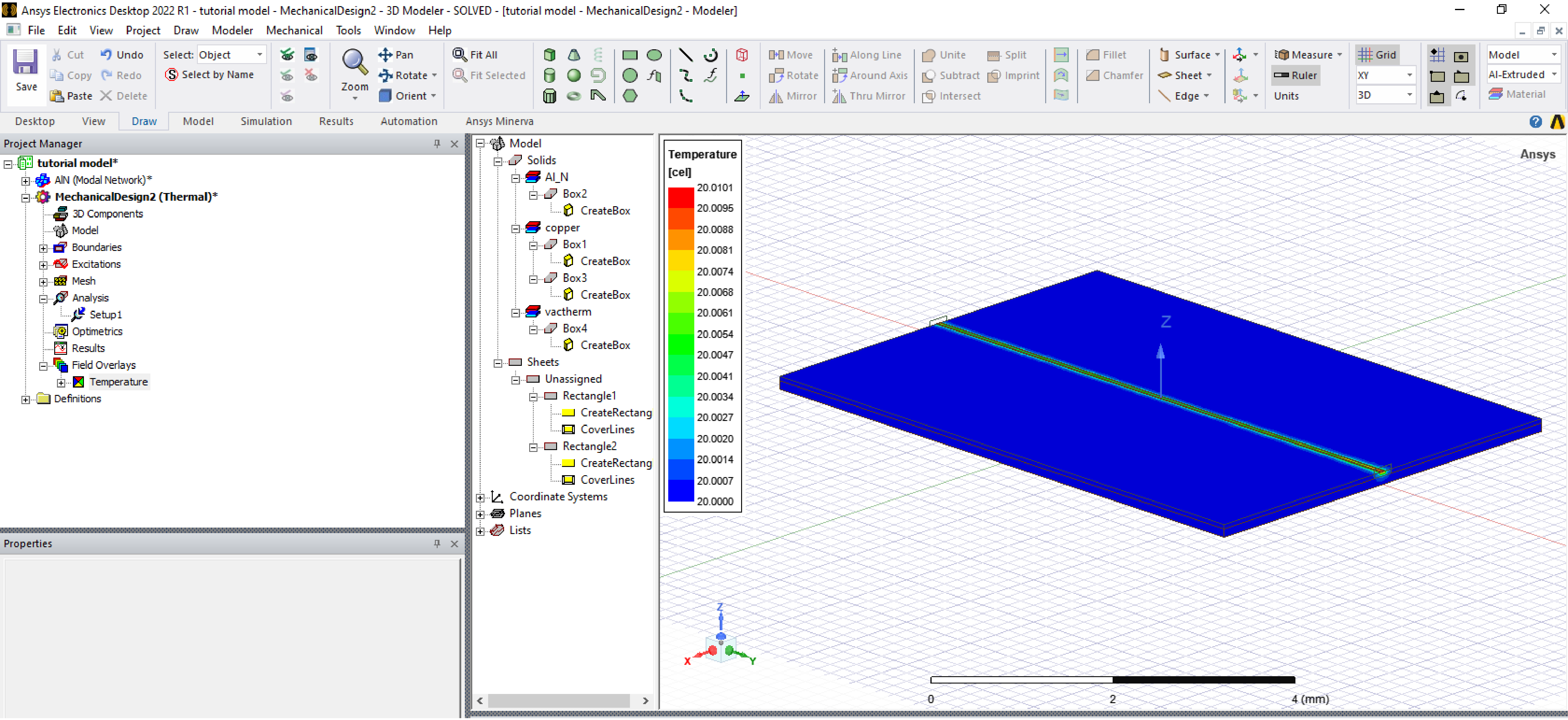
Next, navigate to “Analysis” under your Project Manager. Right click on it and select “Add Solution Setup”.
Select “Advanced” and then “Import Mesh”. A pop-up menu will appear.
Select “Use this project”, “Simulate source design as needed”, and “Preserve source design solution”.

Setting up Analysis



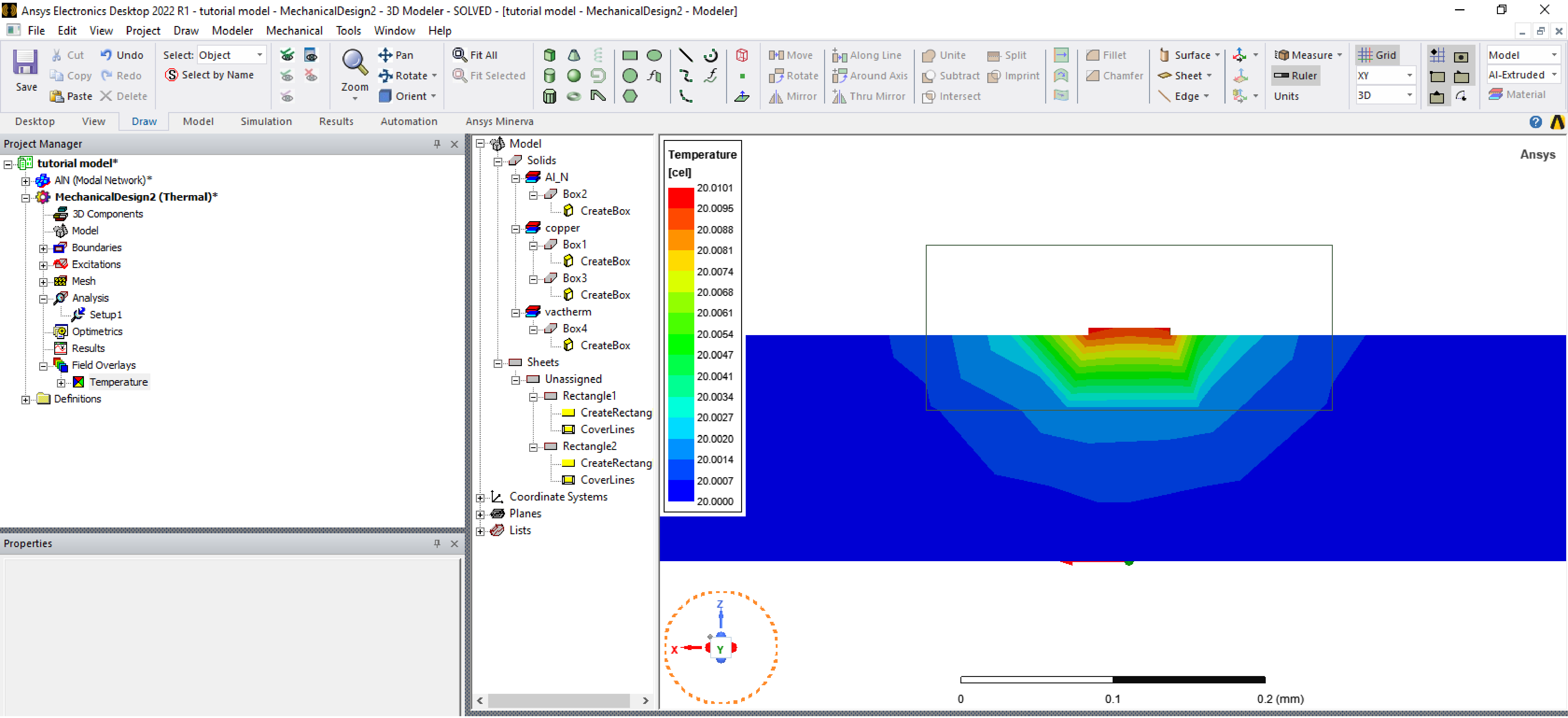
Your setup is complete! Run the analysis by right-clicking on “Analysis” under your Project Manager and selecting “Run” or “Analyze”. Wait for the simulation to complete, and then select the copper grounding plate, heatsink, and microstrip, and right click on them. Plot the temperature as shown above.

Running the Simulation and Plotting



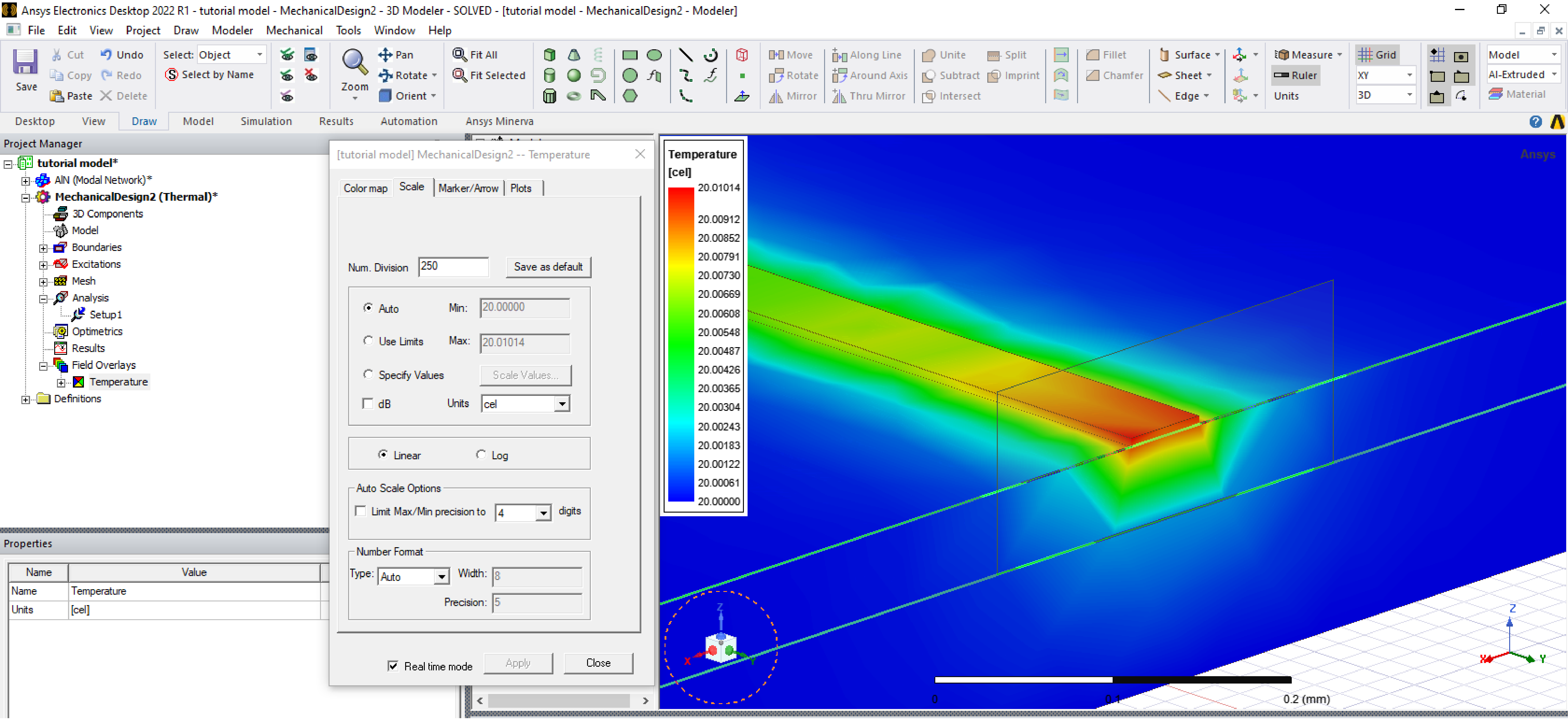
You should now have a temperature plot!

Plotting (cont.)



The temperature plot may look sectioned and "blocky" as depicted above.
To fix this, right click on your temperature plot under your Project Manager's "Field Overlays" section.

Plotting (cont.)



Select “Modify Attributes”, click “Scale”, and increase the number of divisions until you achieve your desired plot smoothness.
A word of caution: If you raise your number of divisions too high, the computer will begin to lag.

Plotting (cont.)