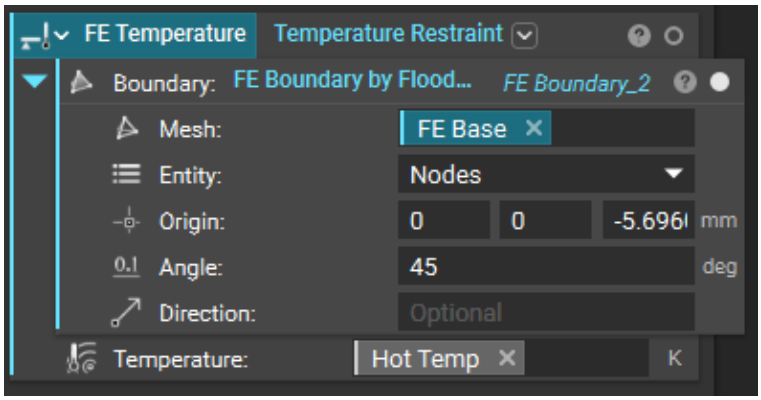
Follow Along: Creating Thermal Boundary Conditions

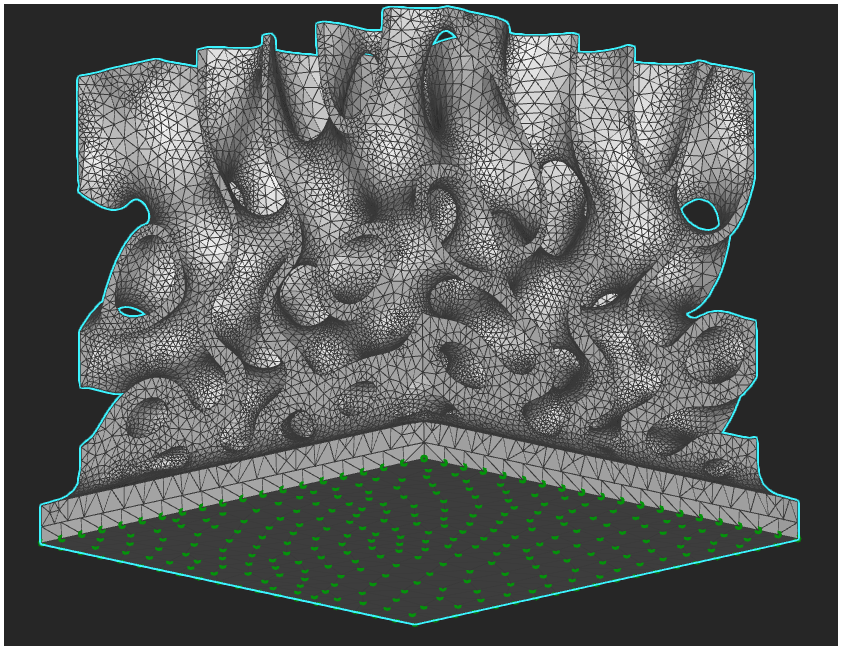
Now that we've created our **FE Model**, we need to create the other input for our **Thermal Analysis**–boundary conditions. You can continue working in the .ntop file from the previous lesson.

To this *FE Sink* model, we'll apply three boundary conditions. We will include a **Convection Boundary Load** to the surface of the heat sink where heat transfers to the surrounding fluid and a **Radiation Boundary Load** where heat is transferred from the heat sink to its surroundings via radiation. Finally, we'll add a **Temperature Restraint** at the base where the temperature remains constant.

Step 1: We will begin by creating a **Temperature Restraint**. Add this block to the Notebook from the Boundary Conditions dropdown list under the Simulation tab. To select a Boundary input, add an **FE Boundary by Flood Fill** block and drag the gembal to select the Nodes along the bottom surface of the *FE Base* mesh.

For the Temperature input, use the variable *Hot Temp* from the Inputs section of the Notebook.



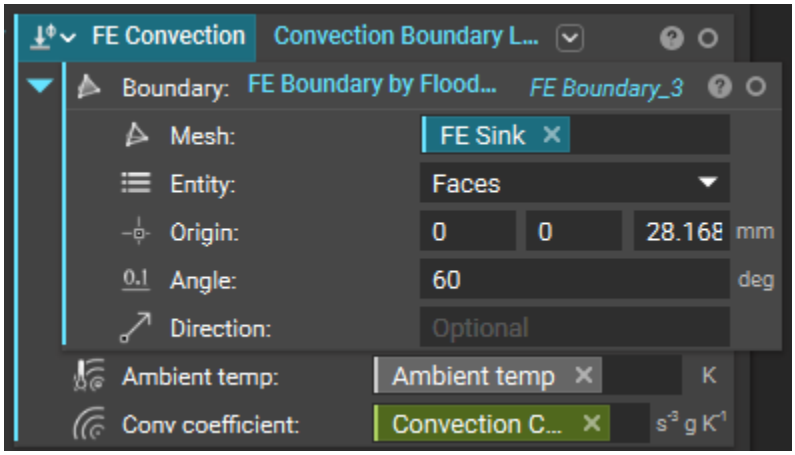


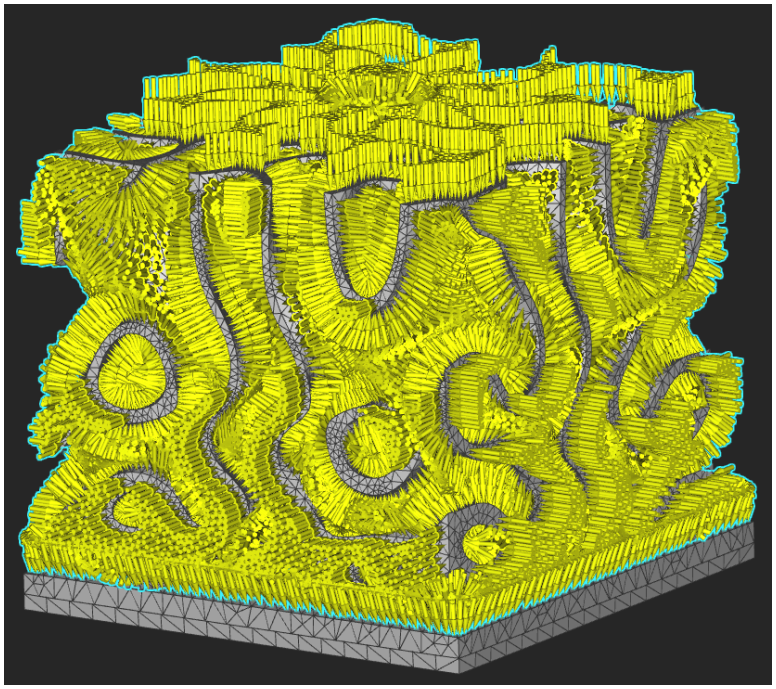
Step 2: Next, create a **Convection Boundary Load** block, found in the Boundary Conditions dropdown list under the Simulation Tab, to model convection from the heat sink to the surroundings.

Add an **FE Boundary by Flood Fill** block to define the Boundary, and drag the gembal to select as much of the outer surface of the heat sink as possible. Modify the Angle input to select the optimal boundary.

Use the *Ambient temp* and *Convection Coefficient* variables from the Input section of the Notebook as the remaining two inputs.

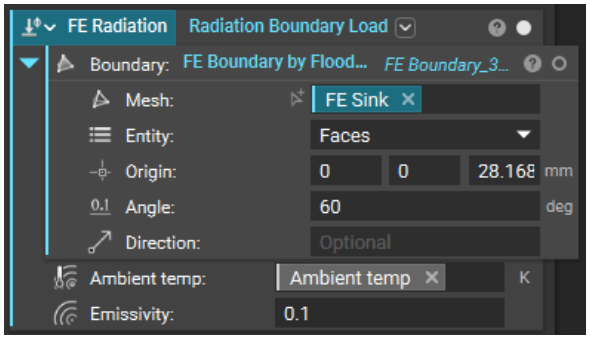
Make this into a variable called *FE Convection*.

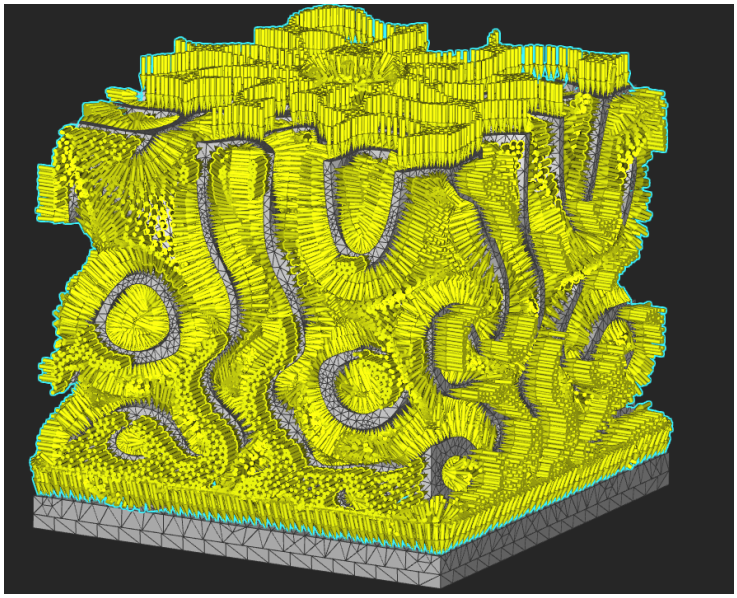




Step 3: Next, create a **Radiation Boundary Load** block, found in the Boundary Conditions dropdown list under the Simulation Tab, to model radiation from the heat sink to the surroundings.

Add an **FE Boundary by Flood Fill** block to define the Boundary, and drag the gembal to select as much of the outer surface of the heat sink as possible. Modify the Angle input to select the optimal boundary (or, copy and paste the Boundary from the *FE Convection* variable in Step 2).





Step 4: Add these three boundary conditions to a **Boundary Conditions List** block, and make it into a variable called *Boundary Conditions*.