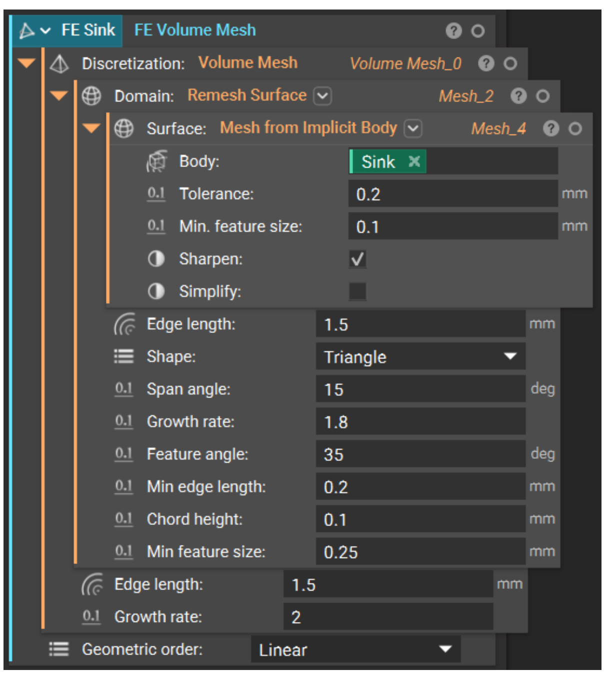
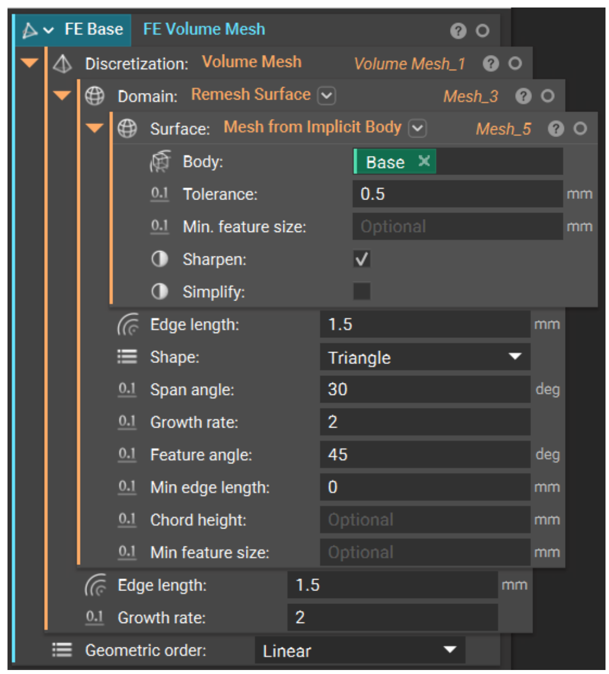
Follow Along: Creating a Thermal FE Model

To prepare for running a **Thermal Analysis** on the heat sink in this lesson, we'll create an FE Model. Download the starter file to begin.

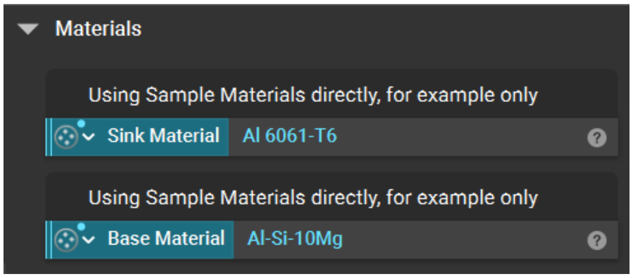
Under the Geometry section, we see the implicit models of both the heat sink and the base plate. We'll use these as the basis of our two FE components for this thermal analysis–they'll both be modeled as FE Solid Components. At the interface between them, we'll add a connector to model the heat transfer through a thermal paste or interface material.

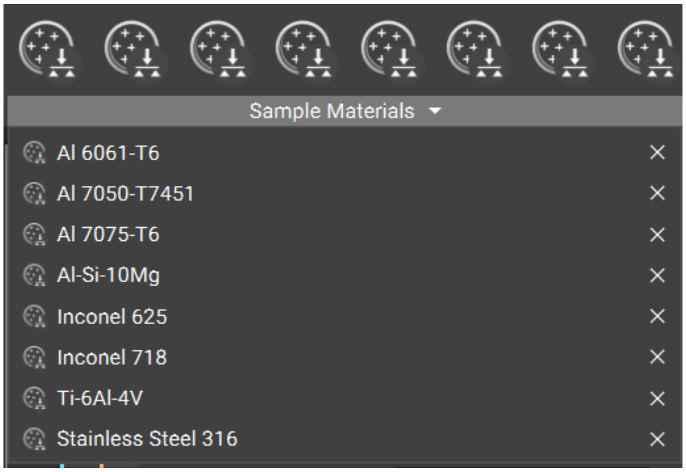
Step 1: FE Volume Mesh−like we've seen in creating **FE Models** for past simulations, we'll need to convert these implicit bodies into **FE Volume Meshes** to run any analyses. In this file, we've already meshed and remeshed the surfaces of both of our bodies. Then, we've converted these surface meshes into **Volume Meshes**, and then to **FE Volume Meshes**.





Step 2: Note that the sample file has specified two Aluminum alloys as materials for the Sink and Base. These were pulled from the Sample Materials dropdown list under the Design Analysis tab.

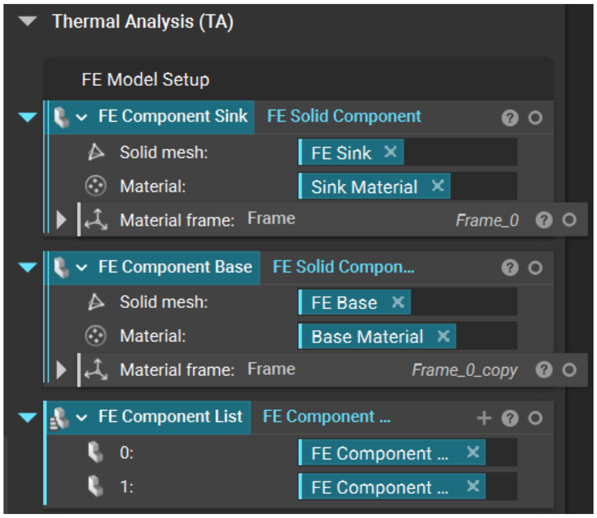




You can also create your own material for thermal analysis using a Material block and adding thermal material property blocks to the Properties list of your Material. If you plan to use a custom material regularly, you can use it as a Custom Block to save some effort down the road.

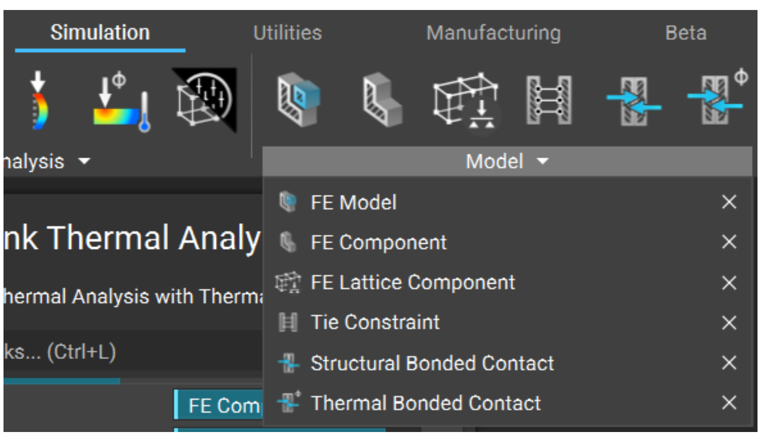
Step 3: FE Solid Components–add an **FE Model** block to the newest section of the Notebook. And I'll add two **FE Solid Component** blocks to the **FE Components List** in the Model.

Notice that the **FE Solid Component** is a Toolkit Block, denoted by the two vertical lines on the left side of the block. This block allows us to avoid having to use an **FE Solid Attribute** nested inside of an **FE Component**.

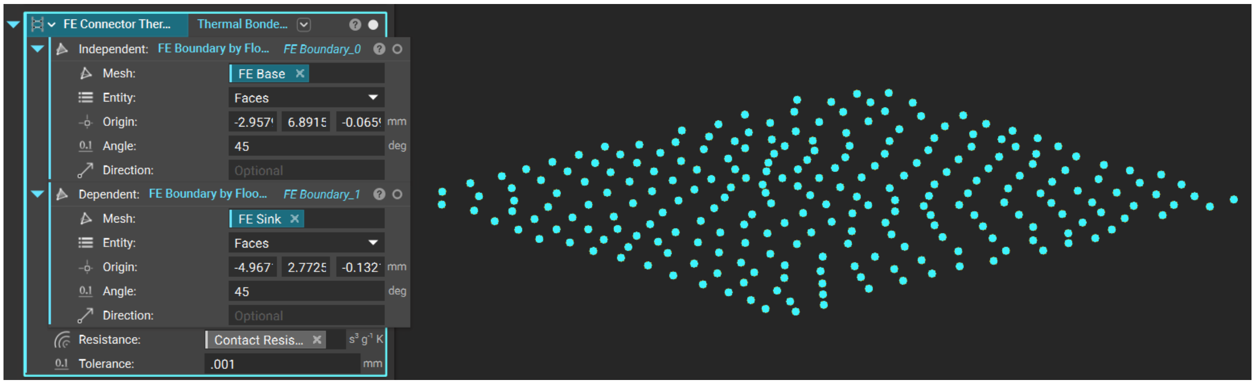


For the Solid meshes and Materials, use the FE Sink and FE Base meshes and materials that have already been defined. For visibility, pull this **Components List** out as a variable.

Step 4: Thermal Bonded Contact–to model the heat transfer between the two bodies, add a **Thermal Bonded Contact** block as the connector. This is found in the Model dropdown under the Simulation tab.



Here, we will select the Base as our Independent surface and the heat sink as our Dependent surface. I'll add an **FE Boundary by Flood Fill** block to both inputs and drag the gimbal to select the two adjacent surfaces. In the Resistance input, we can add the variable called *Contact Resistance* that we have in the Inputs section of the file. Make this a variable called *Thermal Connector*.



Step 5: FE Model–now that we've created our FE Components and Connector, we can add them to our FE Model block, and our model is complete.