Follow Along: Running a Thermal Stress Analysis

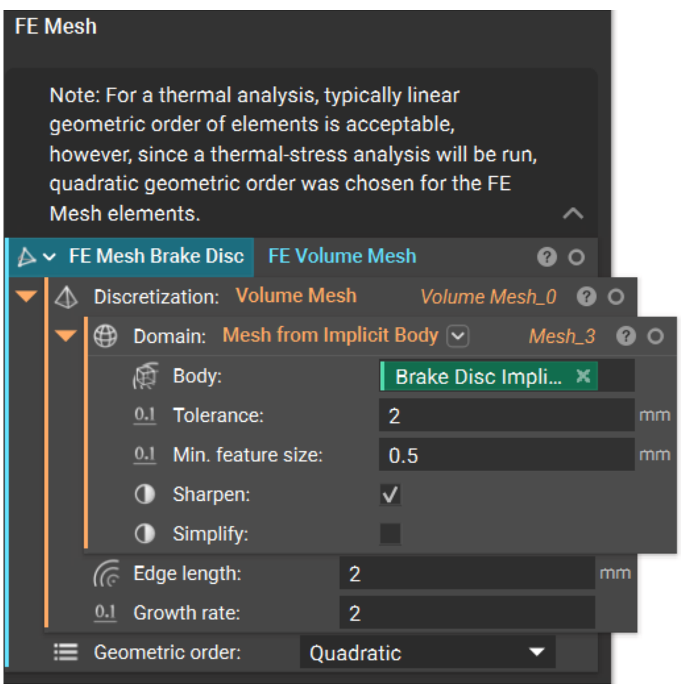
Now that we've covered **Thermal Analysis**, let's walk through the process of setting up and running a Thermal Stress Analysis.

We will begin by walking through the **Thermal Analysis** of the brake disc in the starter file. Then, we will use the resulting Temperature Field to conduct a Thermal Stress Analysis of the same part to see the **Static Analysis** results given the part's loading and temperature conditions.

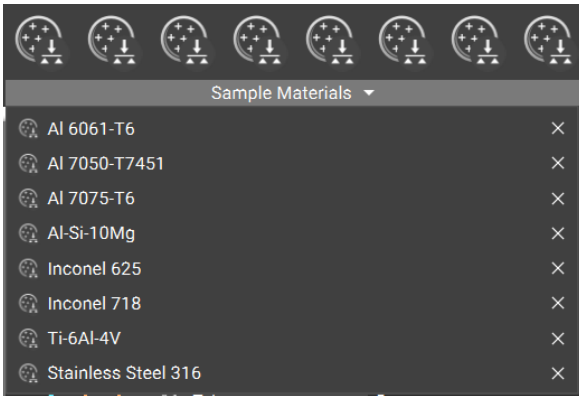
Download the starter file to begin.

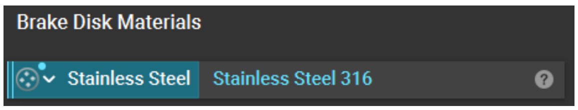
Step 1: Let's review the initial **Thermal Analysis** of the brake disc.

We will begin by creating an **FE Model** based on the *Brake Disc Implicit* body in the Geometry section. We've **Meshed** the body, created a **Volume Mesh**, and converted to an **FE Volume Mesh**.

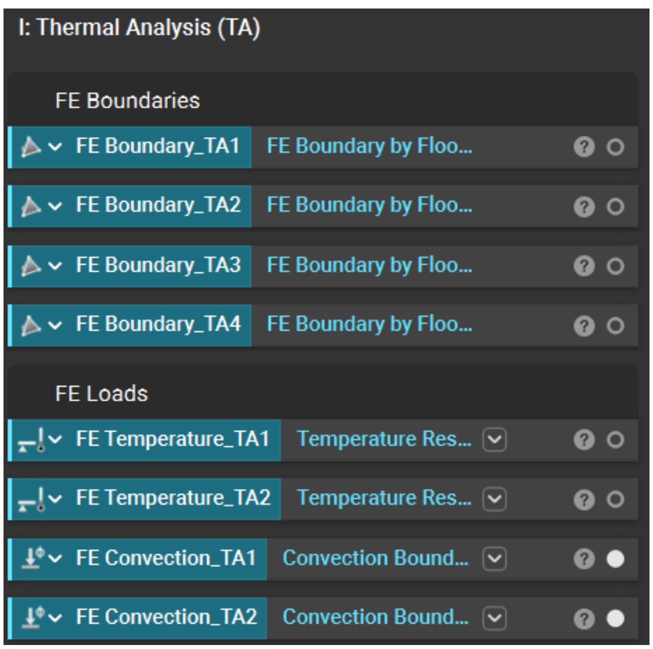


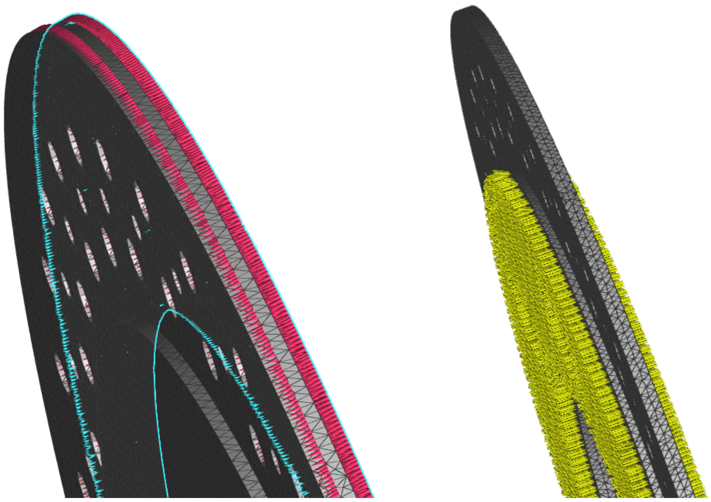
We've then created a variable from the Sample Materials in the Design Analysis tab.



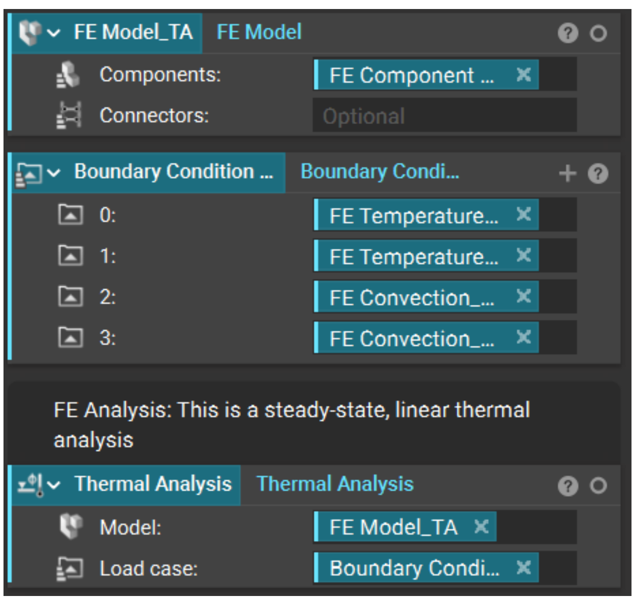


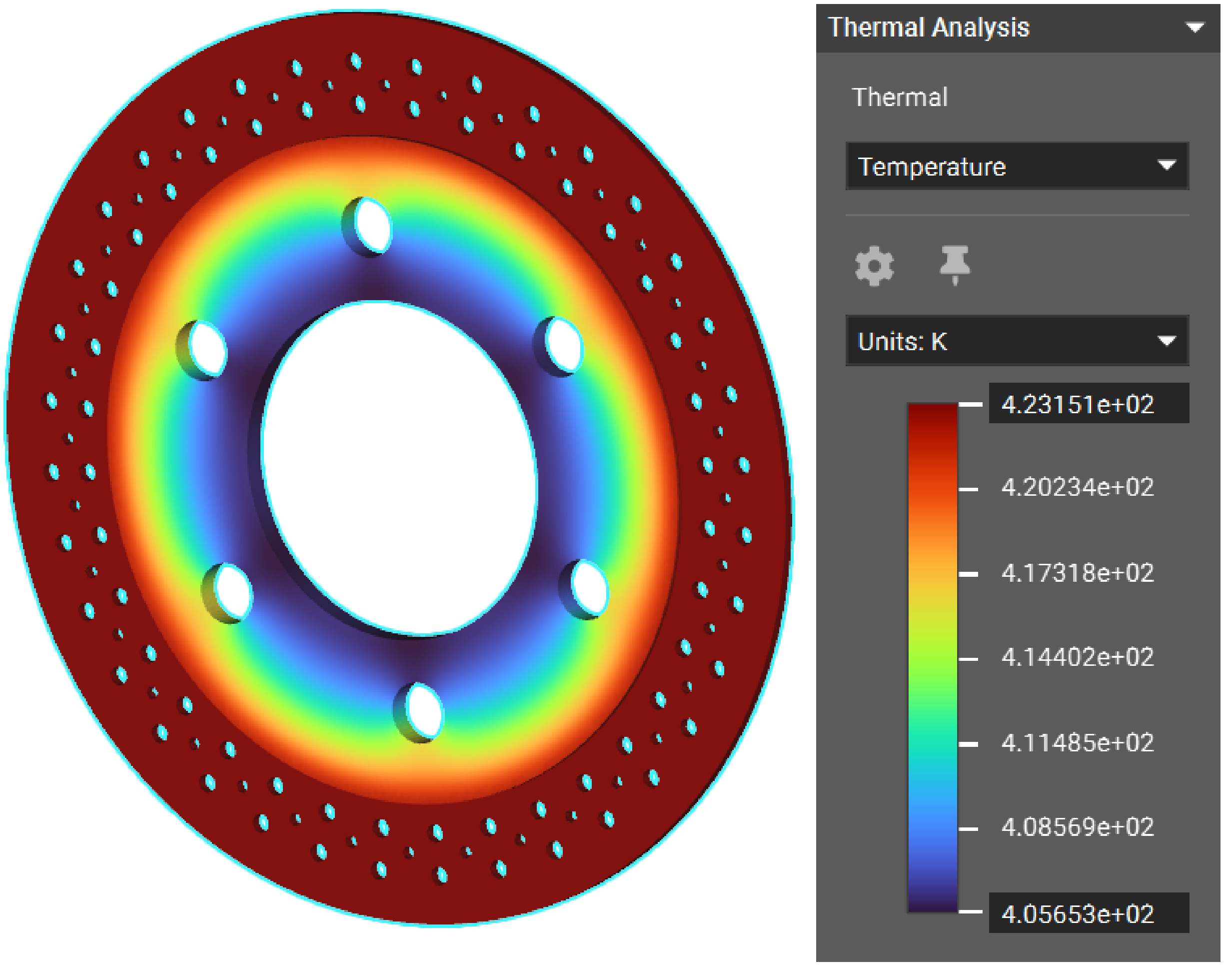
Then, we defined our boundaries and set boundary conditions for the model.
We start with **Temperature Restraints** (*Braking Temp* of 150C) on the outer faces of the disc and **Convection Boundary Loads** (at *Ambient Temp*) on the inner faces.



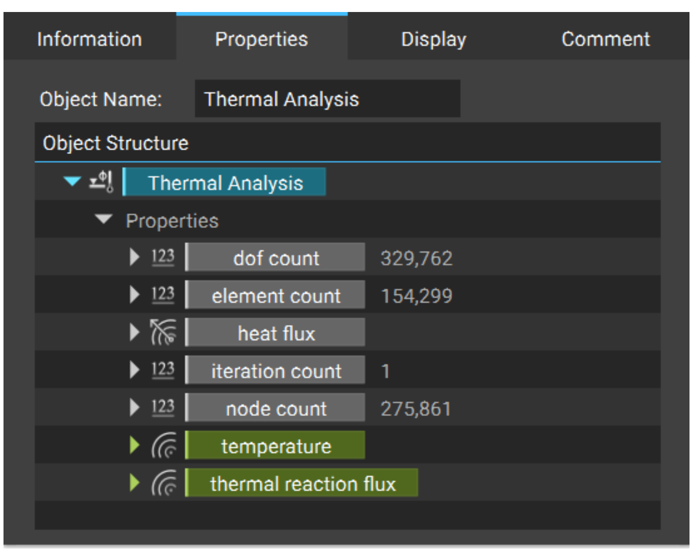


We’ll create an **FE Solid Component** for the FE Model–this consists of the brake disc **FE Mesh** and *Stainless Steel*. We’ll then add this and our Boundary Conditions to a **Thermal Analysis** block. The analysis will run automatically.



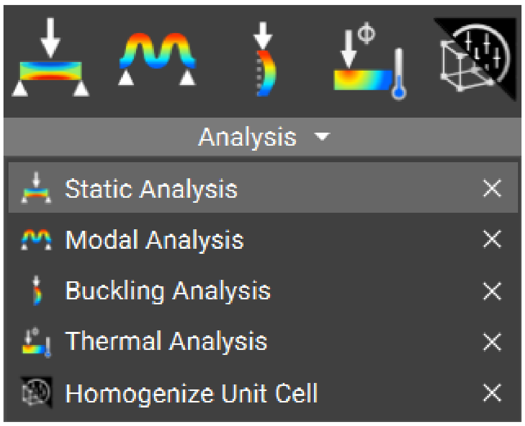


We can then pull the resulting Temperature Field from the Properties list in the Block Details.



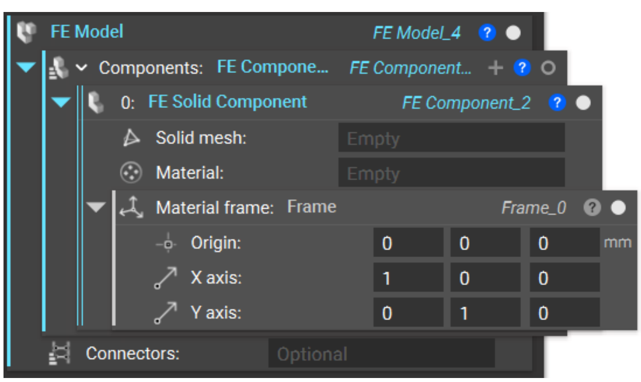
Step 2: Now, let's use these results as an **Applied Temperature Load** in a Thermal Stress Analysis.

Under the Thermal Stress Analysis section, drop a **Static Analysis** block from the Simulation tab. Make this a variable called *Thermal Stress Analysis*.



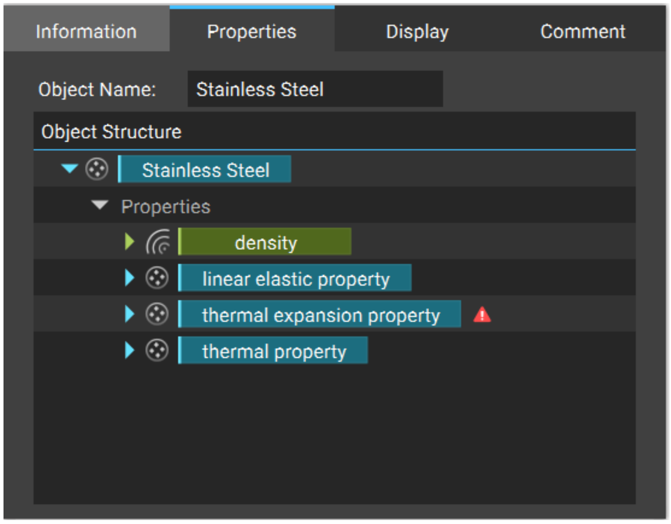
Step 3: Note that we cannot use the same **FE Model** as we did in the **Thermal Analysis**, because we have not yet defined the thermal properties of our material.

Add an **FE Model** block with an **FE Solid Component** into the Notebook. Input the Brake Disc mesh.



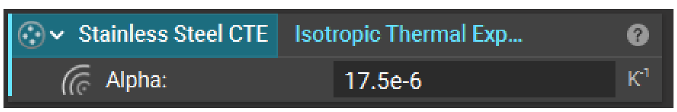
Now, we need to create a material with defined thermal properties. Add an **Isotropic Material** to the Material input, and make it a variable called *Stainless Steel TSA*. We will need to apply both the linear elastic and thermal properties of Stainless Steel 316, as well as its coefficient of thermal expansion.

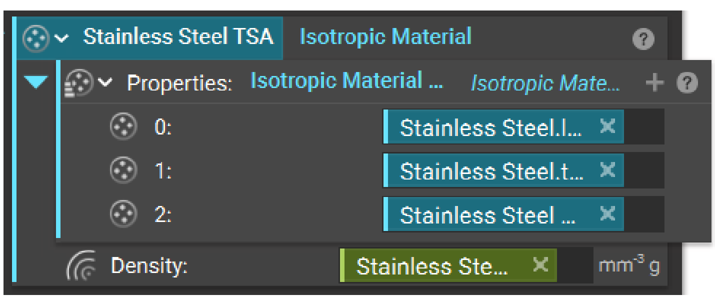
Add two inputs to the **Isotropic Material Properties**, and pull the first two inputs from the pre-defined stainless steel Properties list in the Block Details.



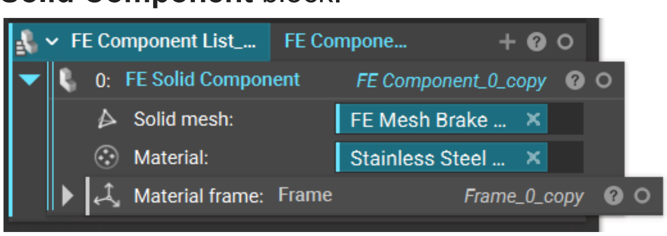
When running a Thermal Stress Analysis, we also have to include the material's Density into the Component. Add this to the **Isotropic Material** block from the Properties list as well.

Use an **Isotropic Thermal Expansion** block to define the coefficient of thermal expansion, and use it as the third input in the **Isotropic Material** block.

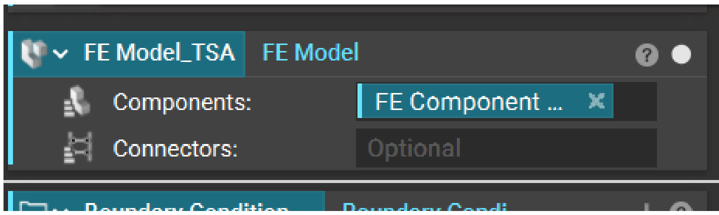


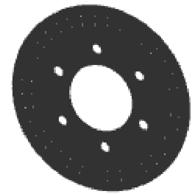


Now, we have finished defining our material and its properties. Add this material to the **FE Solid Component** block.



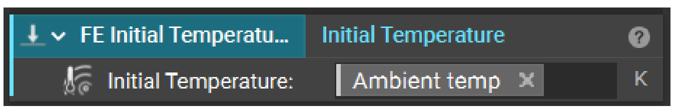
Add the Component to the **FE Model** block and make it into a variable called *FE Model_TSA*.



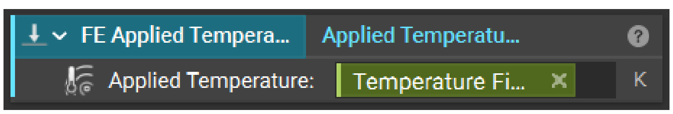


We have now completed our **FE Model** and can move on to creating our Boundary Conditions.

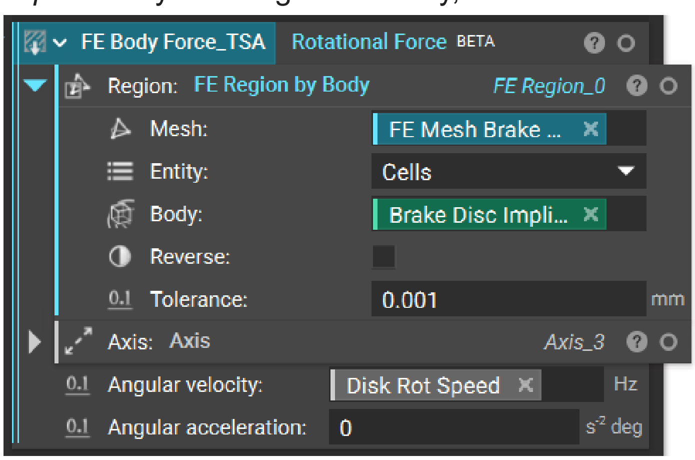
Step 4: Let's begin by defining the **Initial Temperature Load**. Add this block to the Notebook, and set the Initial Temperature as the *Ambient temp* variable in the Inputs section. Call this variable *FE Initial Temperature*.



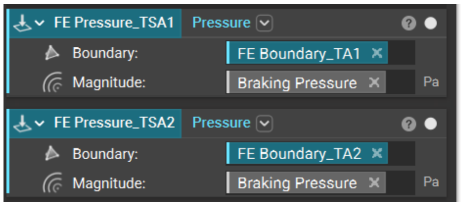
Next, add an **Applied Temperature Load** block and use the Temperature Field from the Thermal Analysis as your input. Make this a variable called *FE Applied Temperature*.



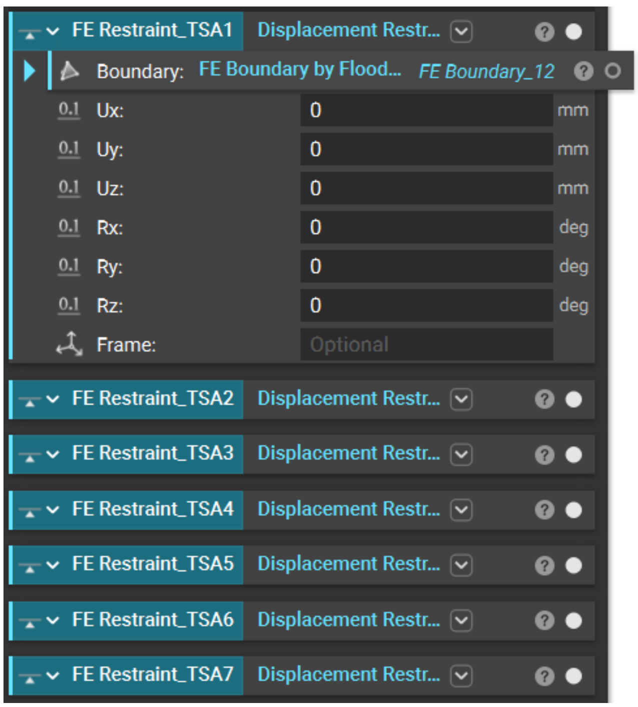
Then, add a **Rotational Force** block to model the expected rotation of the brake disc. For the Region input, use an **FE Boundary by Body** block to select the entire *Brake Disc Implicit* body. For Angular velocity, use the variable *Disk Rot Speed*.

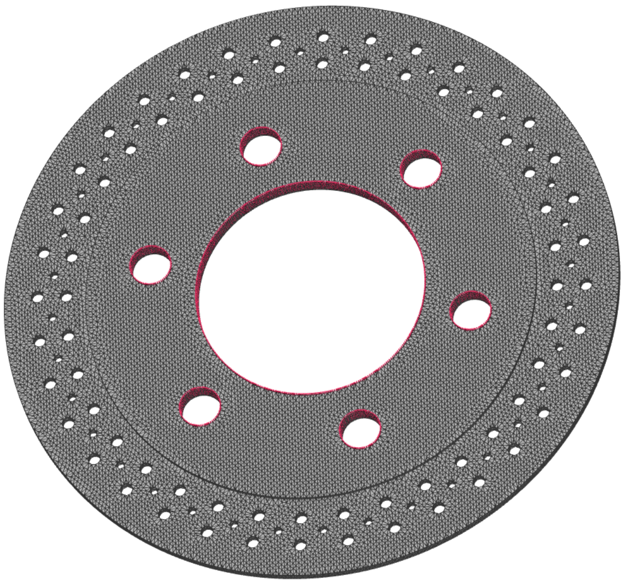


To model the brake pressure, add **Pressure** blocks for both outer faces of the disc, using the *Braking Pressure* variable as the Magnitude input. Make them variables, and call these two Boundary Conditions *FE Pressure_TSA1* and *FE Pressure_TSA2*.

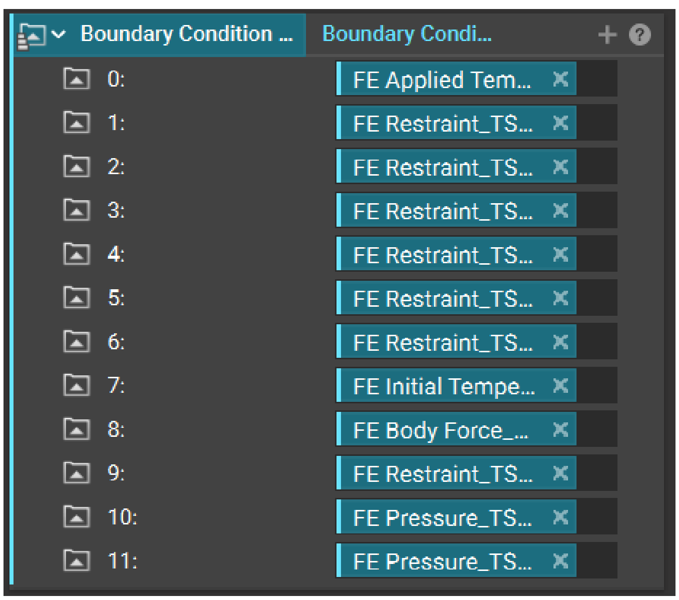


Next, restrain the six small holes and large center hole using **Displacement Restraint** blocks. Use **FE Boundary by Flood Fill** blocks for Boundary inputs, and drag the gembal to select desired surfaces. Leave all other inputs as 0. Make these seven blocks into variables, naming them *FE Restraint_TSA1* through *FE Restraint_TSA7*.

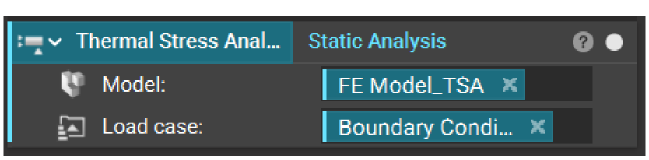




Add all of these Boundary Conditions to a **Boundary Conditions List** block. Make this into a variable.



Step 5: Add the **Boundary Conditions List** and **FE Model** to the **Static Analysis** block, and it automatically runs.



View the results in the window. We can see the Displacement, Strain, Stress, and Reaction Forces from the thermal loading and other Boundary Conditions.