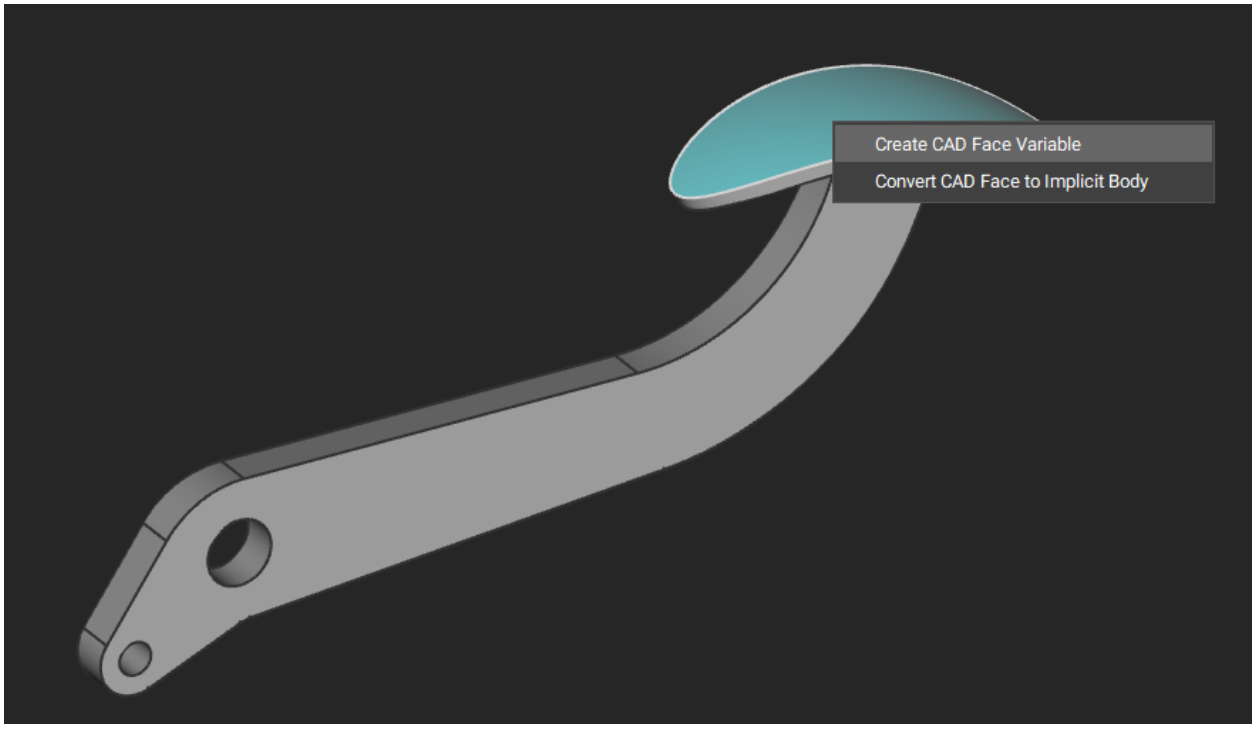
Follow Along: FE Model

In this video, we will walk through creating an FE Model for a brake pedal. To review creating FE Mesh and Volume Meshes, please revisit our Guide to Meshing Course.

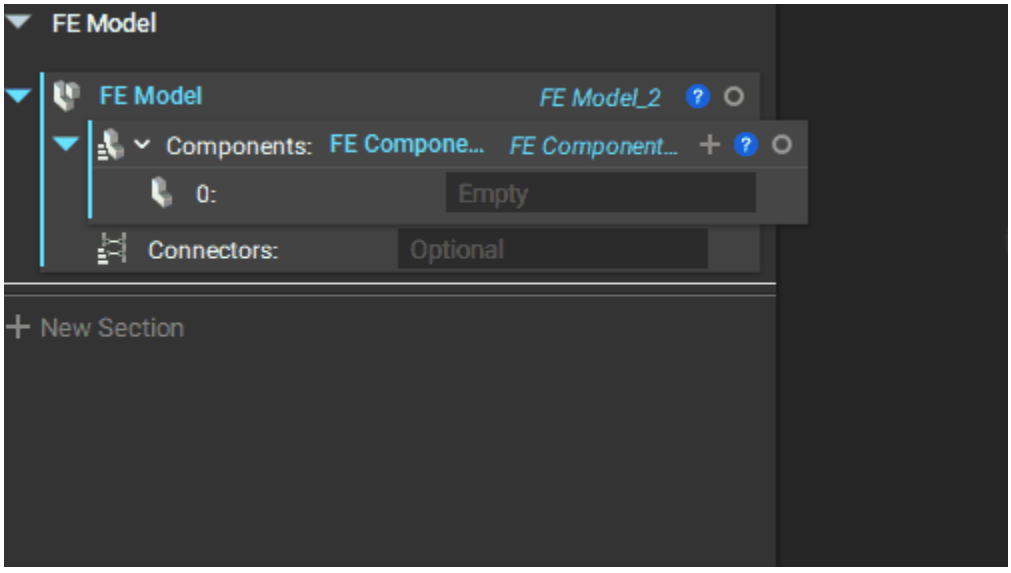
Please download the CAD brake pedal file below to follow along with the tutorial.

Step 1: First, rename the section in your notebook to Geometry by double clicking on Section 1. Import the CAD brake pedal file into the notebook by dragging and dropping that file from your folder. Next double click on the brake pedal to select the entire body, right click and choose Create CAD Body Variable. Label this CAD Body.

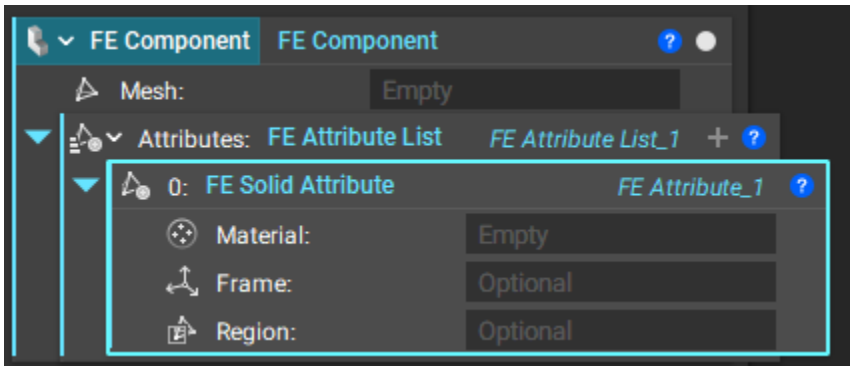
Step 2: We will also define the loaded face when we later create boundary conditions. Single click on the face shown below, right click and choose Create CAD Face Variable. Label this variable Loaded Face.



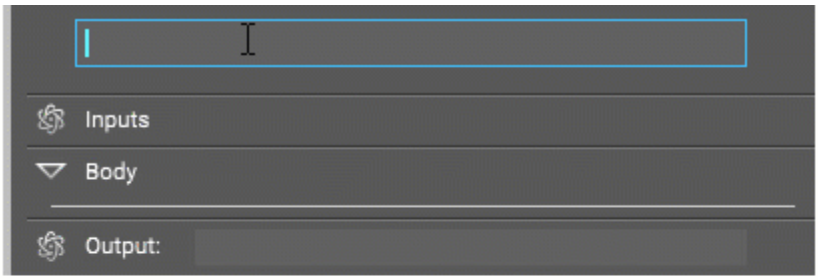
Step 3: Now that we defined the geometry, we can create the FE Model. Click on New Section and rename this section FE Model. Use the “stepping backwards” technique to create the FE Model. First, add the **FE Model** block into the notebook. The FE Model needs an FE Component List and Connector. We will begin with the FE Component List. Double click on that input and choose FE Component. Then right click on the **FE Component** block and make it a variable labeled FE Component.



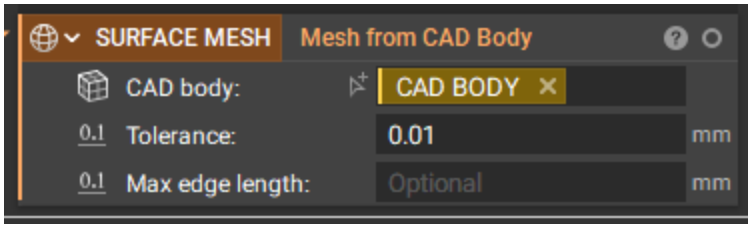
Step 4: The FE Component block requires an FE Mesh and FE Attribute list. Let's begin with an FE Attribute by double clicking on that input and selecting the **FE Solid Attribute** block. The only input we will use in this Follow Along is the **Material** Block.



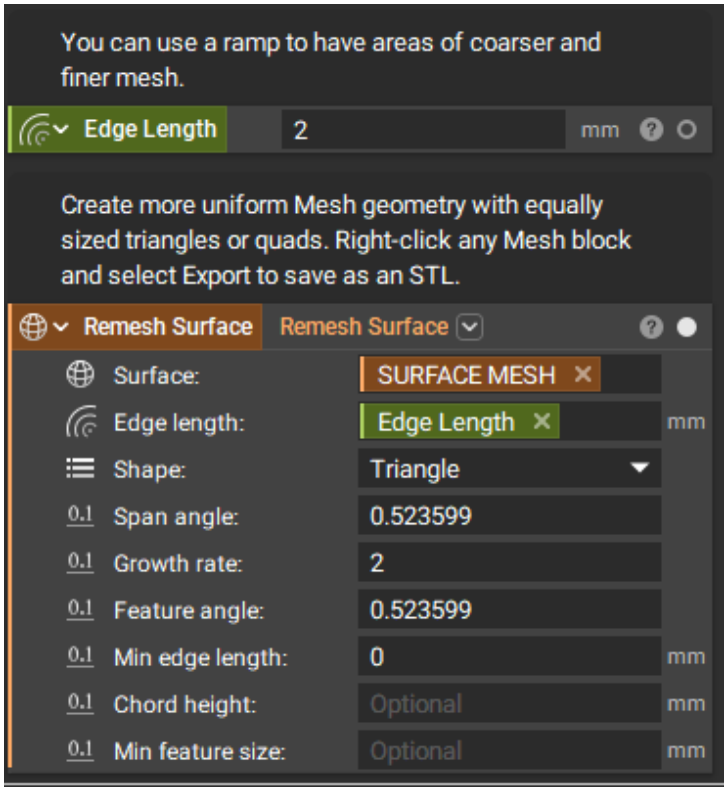
Step 5: Make the material input a variable and add the **Isotropic Material** block to the input. Add an **Isotropic Material Property** and the **Isotropic Linear Elastic Property** by double-clicking in the Property input. Enter the following values: Young's modulus: 2.21e+13 Pa, Poisson's ratio: 0.3.

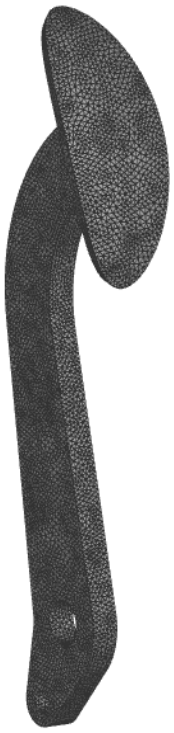


Step 6: The FE Component Block also requires an FE Mesh, and we will begin by converting the CAD brake pedal into a mesh with the **Mesh from Cad Body** block.

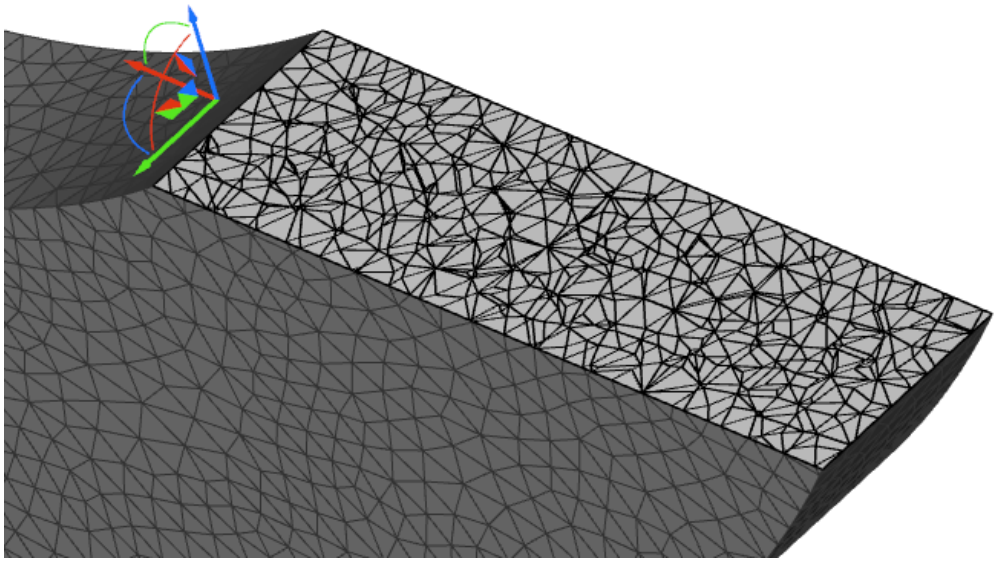


Step 7: Remeshing the surface mesh creates equally sized polygons in the mesh, allowing for an easier transition to a volume mesh. Do this by using the **Remesh Surface** block. Place the **Mesh from CAD body** block into the surface option in the Remesh Surface block, use the edge length of 2mm and triangle for the edge length.

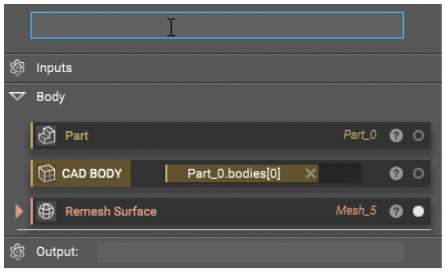


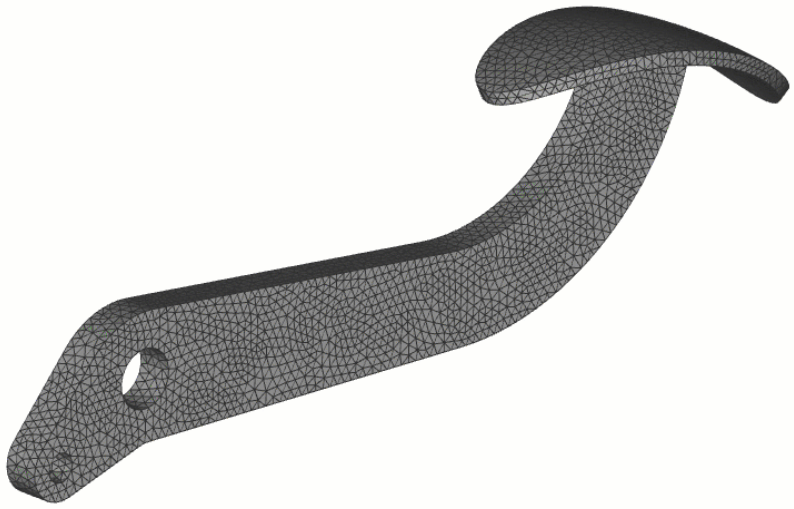


Step 8: Convert the Surface mesh into a Solid Mesh by using the **Volume Mesh** block. It is best practice to keep the edge length the same for both the surface mesh and the volume mesh, so it is easier to form tetrahedrons. The image below shows a section cut of the volume mesh, where the edge length was the same value.



Step 9: Convert the Solid Mesh into a Finite Element Mesh by using the **FE Volume Mesh block**. This conversion adds integration points within the mesh. The linear option inserts nodes at the vertices of elements, while the quadratic option inserts mid-side nodes along the straight edges of elements. We recommend using the quadratic option if you are going to be using the Von Mises Stress results because they are better at predicting stress values. Curved elements are not supported.





Step 10: Now that the FE Components and the FE Model is complete, we have a completed FE Model.