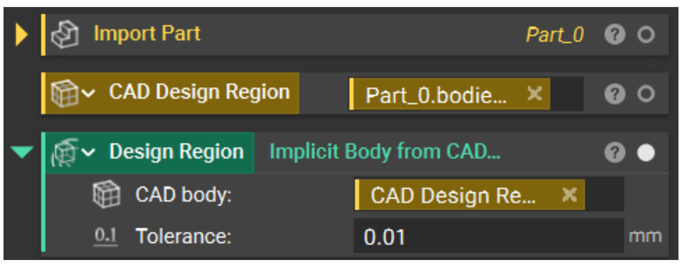
Follow Along: FE Model and Boundary Conditions

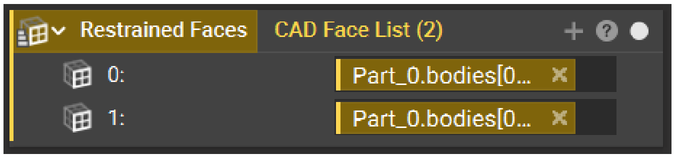
In this video, we will import a CAD part, create an FE Model, and establish boundary conditions for the simple bracket we want to optimize. To review these concepts, please revisit our Simulation course.

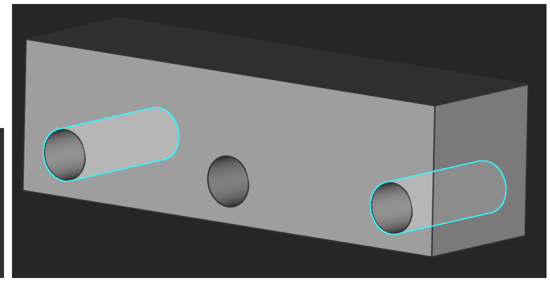
Step 1: Create a section labeled *Geometry*. Import the Design Space CAD file, pull the property chip into the Notebook to create a CAD Variable, and rename the variable as *CAD Design Region*. Add the **Implicit Body from CAD Body** block to convert to an Implicit variable called *Design Region*.



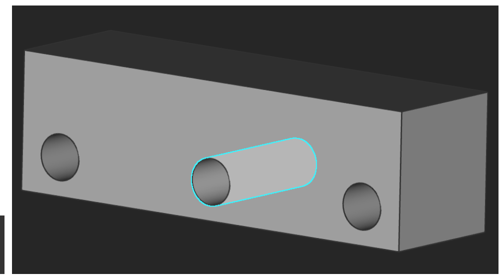
Step 2: Holding CTRL on your keyboard, click on each of the three holes in the CAD Design Region, then right-click and choose “Create CAD Face List Variable” from the dropdown menu. Rename this variable *Interfaces*. These holes will be the interfaces at which we set our boundary conditions later in the workflow–the two outer holes will be restrained, and a load will be applied to the middle hole via a force vector.

Step 3: Now, we’ll specify the CAD faces which will be restrained and that which will be loaded. Using the same process as in Step 2, select the two outer holes and create a variable named *Restrained Faces*.





Then, select the middle hole, select “Create CAD Face Variable” (since this is a single face and not a list), and rename this variable *Loaded Face*.

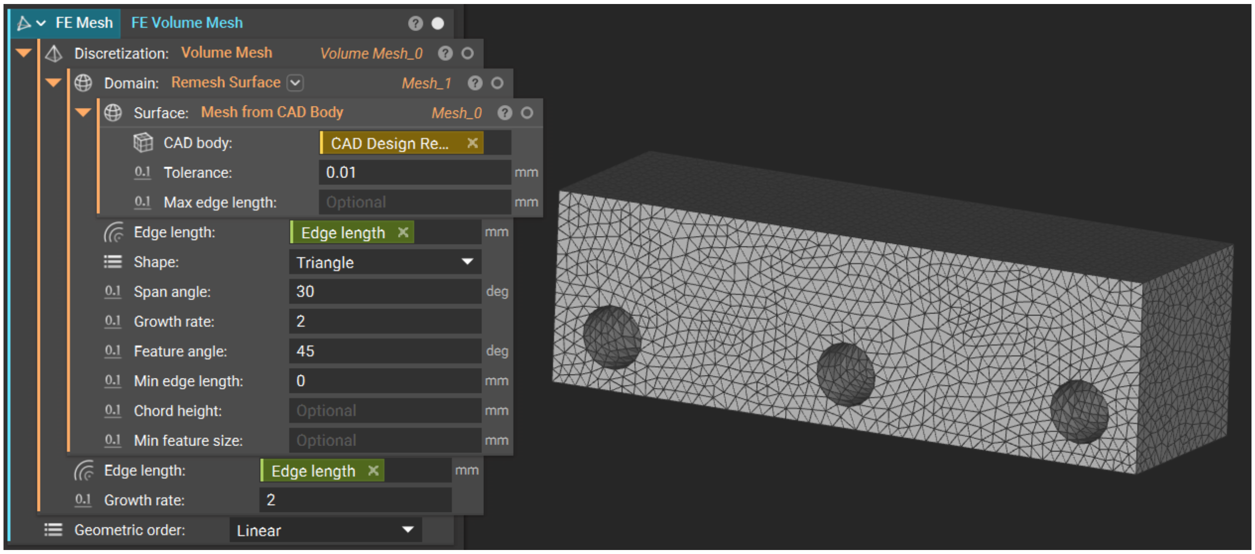


Step 4: Create another section labeled *FE Model*. Here, we will create our FE Volume Mesh and FE Model to be used in our Topology Optimization. Begin with a **Mesh from CAD Body** block, and add the *CAD Design Region* to the CAD body input. Note that the output mesh consists of irregularly shaped and sized triangles. For more complicated part geometries, it is good practice to input the optional Max edge length to produce a nicer mesh to implement into the FE Mesh faster and for better results. Because this is such a simple part, we can leave this input blank.

After meshing, add a **Remesh Surface** block to your Notebook. This will help to regularize the mesh and produce a better FE Mesh. Use the mesh we created above as the Domain input. Make the Edge length input into a variable–we will use this edge length again as an input for our FE Volume Mesh, and it is best practice to keep these lengths consistent. As your Shape input, select Triangle, as FE Volume Meshes are tetrahedral.

Now, use this Remesh Surface as the Domain input in a **Volume Mesh** block. Add the Edge length variable we created earlier. The mesh should now convert from a surface mesh to a filled-in volume mesh.

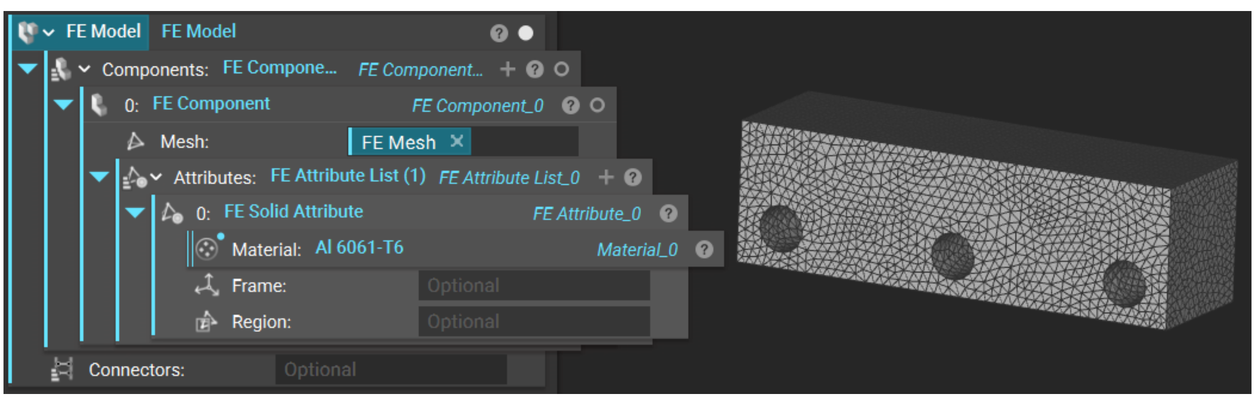
Finally, use this Volume Mesh as the Discretization in an **FE Volume Mesh** block, and use a Linear geometric order. You’ve now successfully created an FE Volume Mesh. Right-click on the block, make it a variable, and name it *FE Mesh*.



Step 5: Now, we'll use our FE Mesh as a component in an FE Model. Add an **FE Model** block to your Notebook. In the Components List, add an **FE Component** block. Use the *FE Mesh* variable as the Mesh input.

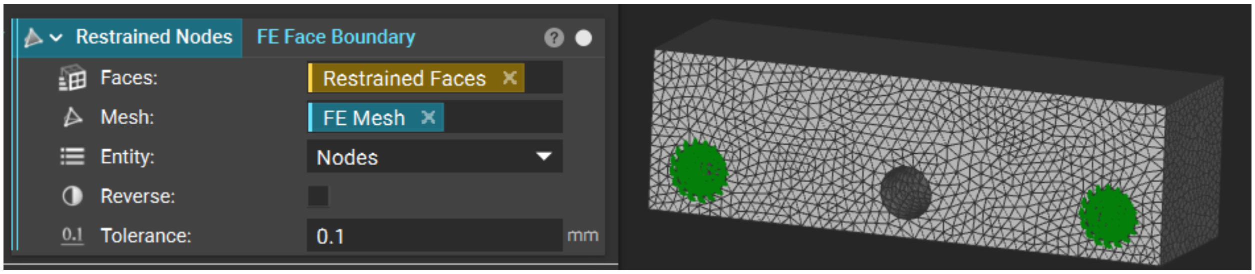
To assign material properties to the part, add an **FE Solid Attribute** to the Attributes list in the FE Component block. nTopology offers a pre-loaded library of commonly used materials which can be found by double-clicking on the Material input. Here, we'll choose Al6061-T6.

We're only working with this one simple part, so no connectors are needed. Right-click to make a variable called *FE Model*.

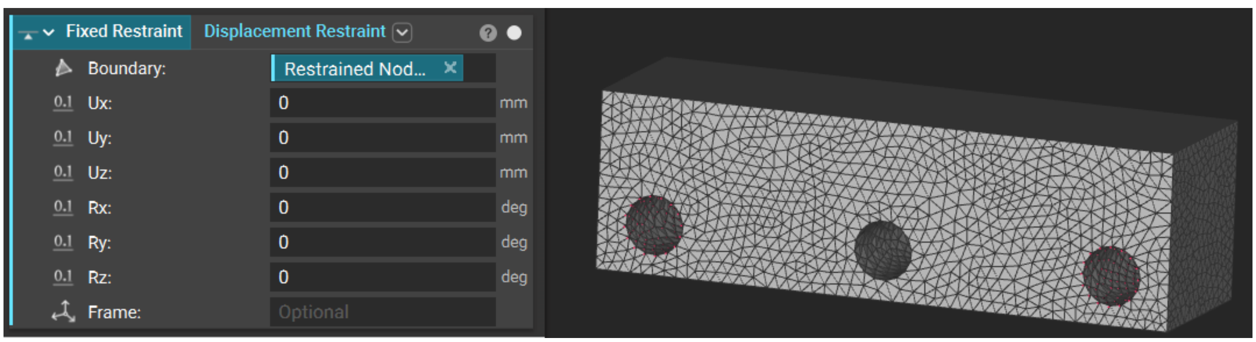


Step 6: Next, we'll establish our Boundary Conditions at the restrained faces. To get started, create a new section called *Boundary Conditions*.

We'll start by selecting all of the nodes in the FE Model that we wish to restrain. Add an **FE Face Boundary** toolkit block to your Notebook and use the CAD Face List variable, *Restrained Faces*, as the Faces input. Add the variable *FE Mesh*. For Entity, select Nodes. This process will select all the nodes within the FE Mesh that intersect the restrained CAD faces. Make this block a variable called *Restrained Nodes*.

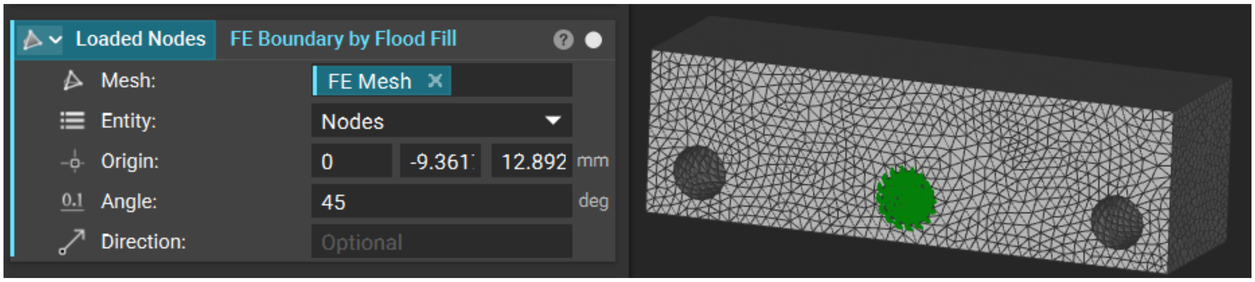


Once these nodes are selected, add a **Displacement Restraint** block to the Notebook. Use the *Restrained Nodes* as your input, and leave all other inputs as 0. This indicates zero translation or rotation of all selected nodes. Make this block a variable called *Fixed Restraint*. You've now created your first of two boundary conditions.



Step 7: Finally, let's establish the boundary condition at the loaded hole. To select the center hole, we could use the **FE Face Boundary** block we used in Step 6, but let's explore another option for selecting our nodes.

Add an **FE Boundary by Flood Fill** block to your Notebook. Again, input your *FE Mesh*. Select Nodes as the Entity we wish to capture. Type an origin location or drag the gimbal to the inside of the hole. The closest node to the origin will be included in the FE Boundary, and the 45-degree angle will drive the remainder of the nodes included in the Boundary. Make this block into a variable called *Loaded Nodes*.



Finally, to apply a load to this feature, add a **Force** block to the Notebook. Use *Loaded Nodes* as the Boundary and input a vector of [0,-1000,0] N, indicating a force of 1000 Newtons in the negative y-direction. This load will be distributed among all the nodes in our *Loaded Nodes* boundary. Make this into a variable called **Force**. You've now successfully created your second boundary condition.