# Follow Along: Stress Driven Mesh Refinement

We can use simulation data to further refine FE Meshes and attain more accurate results at areas of high stress. Let's apply this idea to the same bracket we used in the last Follow Along lesson.

Step 1: Begin by creating a new Section and labeling it *Geometry*. Import the CAD file of the bracket, and pull the CAD body into your Notebook. Add an **Implicit Body from CAD Body** block to the Notebook, and convert the CAD body to an Implicit, making it a variable called *Implicit CAD Body*.
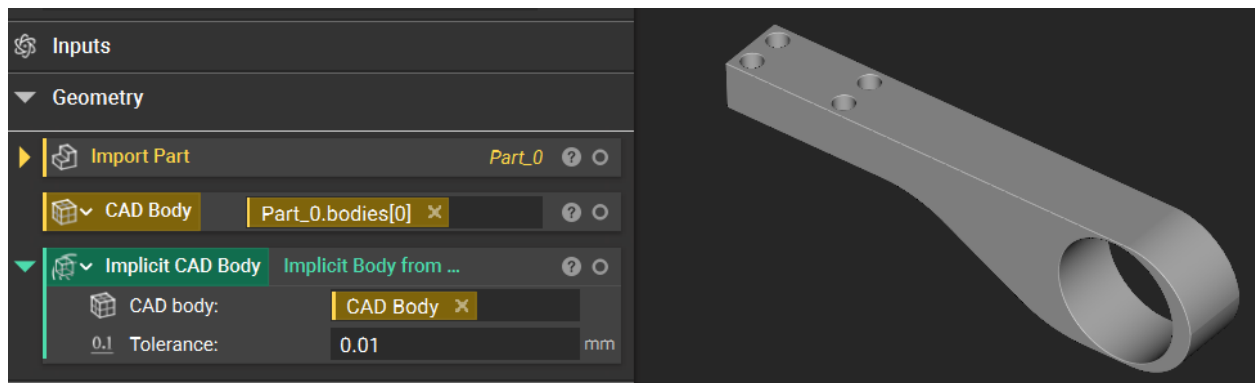

Step 2: Create a new Section labeled *Field From Stress Data*.

From the Fields tab in the Ribbon, add an **Import Scalar Point Map block** to the Notebook, set the file path to the .csv file of the stress data, set the Units to mm, and set the Scale to 1Pa. Your Point Map should look like the one below. Make this block a variable called *Stress Data*.
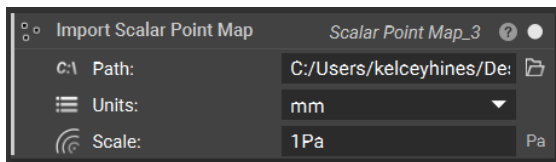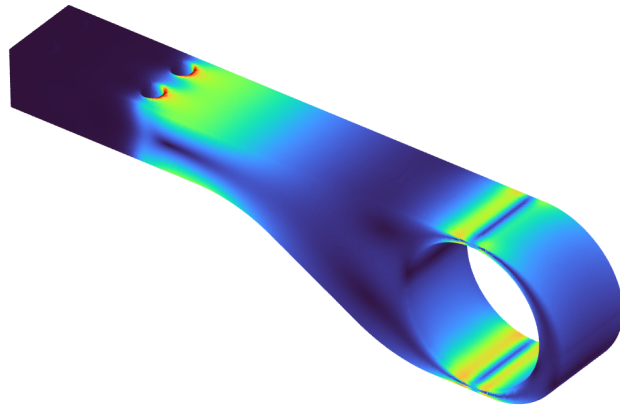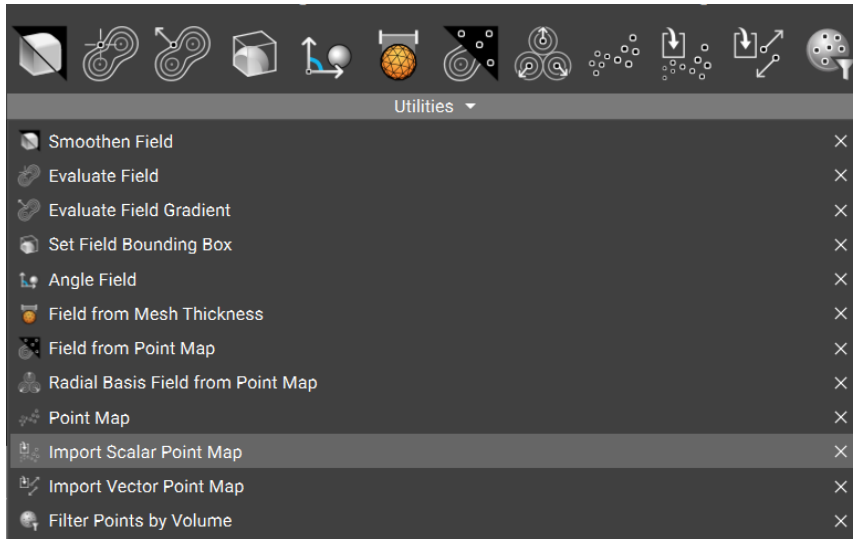
We need to convert this Point Map to a Field before using it to drive geometry. Add a **Field from Point Map** block to your Notebook, and enter the inputs shown below. Make this Field into a variable called *Stress Field*.
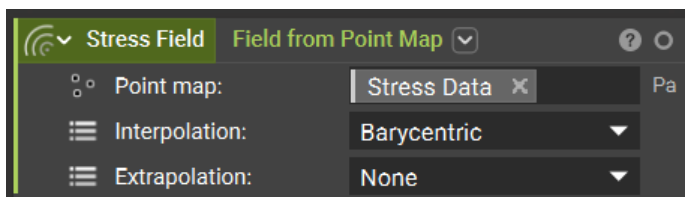
Step 3: Create a third Section called *Refine Mesh Based on Stress Results*.

Add a **Ramp** block to your Notebook, using the *Stress Field* as the Scalar field input. Set the inputs as shown below. This will drive our meshes' edge lengths to be longer in areas of low stress, and much shorter as stress increases. Make this a variable called *Edge Length Ramp*.
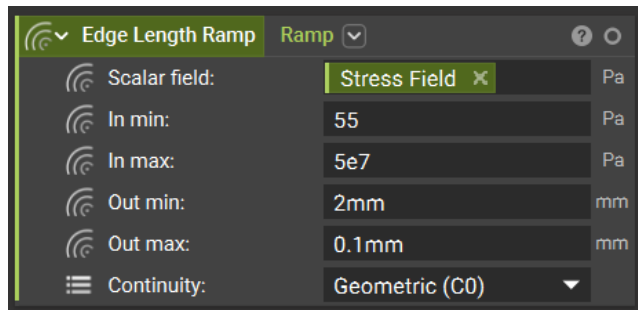

Add a **Mesh from CAD Body** block to mesh the initial CAD part. Note the irregular triangle shapes and sizes.
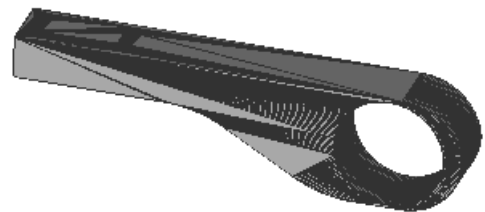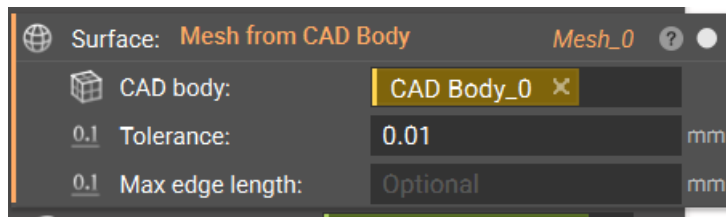

Add a **Remesh Surface** block to make triangles more uniform, and add the *Edge Length Ramp* into the Edge length field input. Notice that the triangles in the remeshed surface are more refined at the high-stress areas.
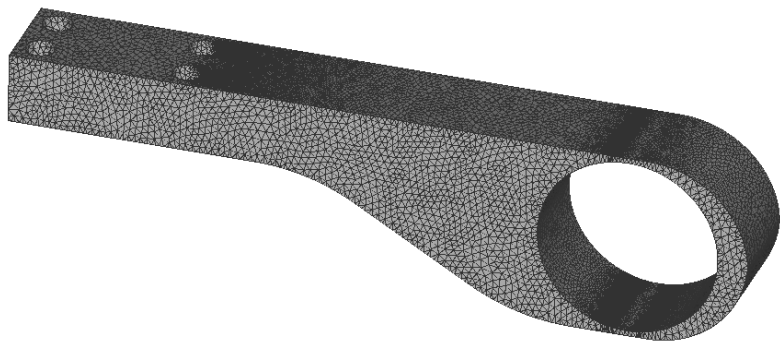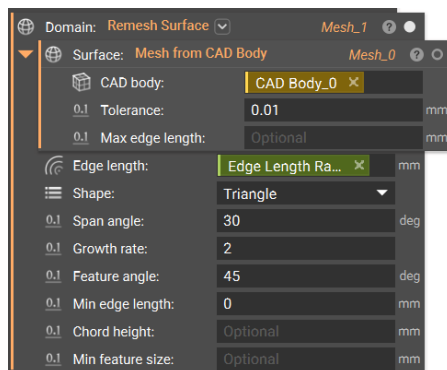
Then, add a **Volume Mesh** block since we're refining this mesh for simulation and will require tetrahedral elements. Again, use the *Edge Length Ramp* in this block.
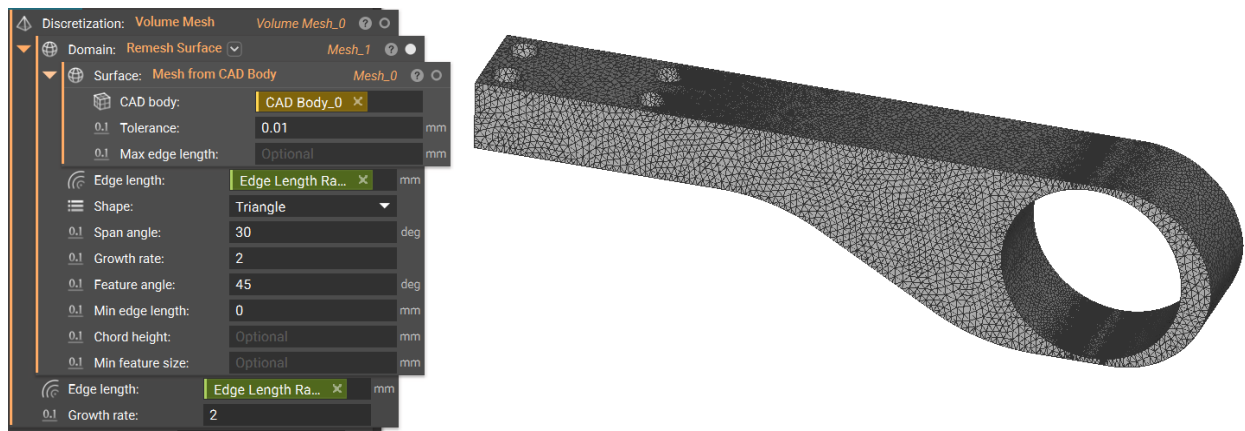
Finally, add an **FE Volume Mesh** block to convert to an FE Mesh. Make this into a variable called *FE Mesh*.
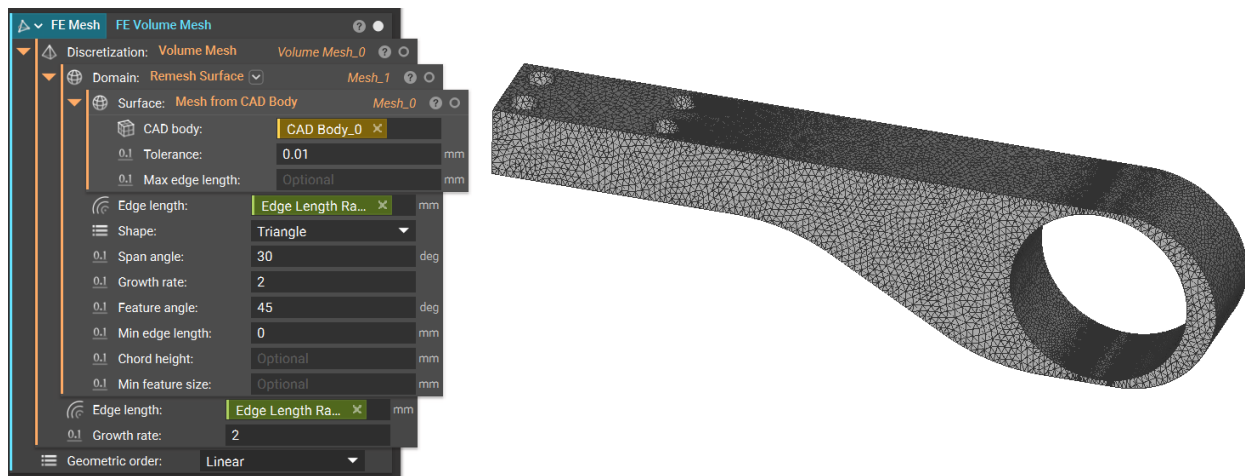
Running a simulation with this mesh, versus one with less refinement at areas of high stress, will yield more accurate FEA results.