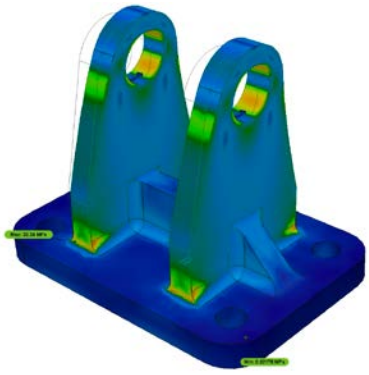
Lesson: Principles of model meshing for simulation

In this lesson, you explore model meshing, element types, reasons for locally refining a mesh and stress resolution and its dependency on mesh density.

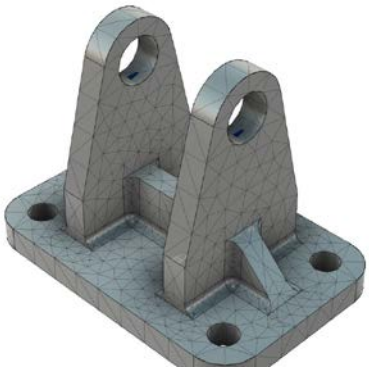


The completed exercise

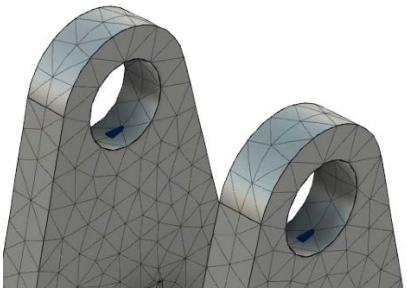
Learning Objectives:

- Set mesh properties
- Use adaptive mesh refinement.
- Apply local mesh control.

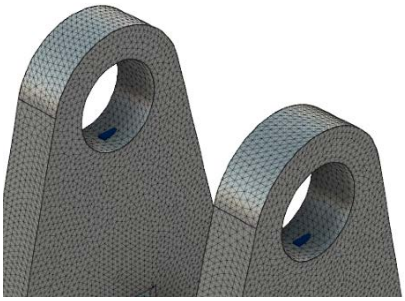
1. Open the file Lift Point – Meshing.f3d and save it to your current project. Click Simulation>Manage>Settings and choose the Mesh tab. Click OK to accept the default settings. Once a study has been created, a mesh folder will be added to the browser. Right click on Load Case1's Mesh and choose Generate Mesh.



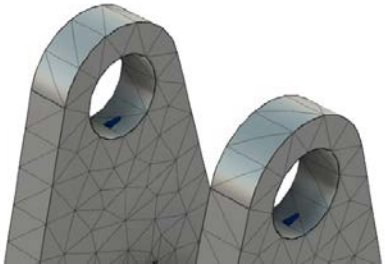
2. In the Browser, click the pencil icon next to the Mesh feature to edit the mesh. Move the Model-based Size slider to the middle and click OK. Right click on Load Case1's Mesh and choose Generate Mesh. Notice the number of elements has doubled.



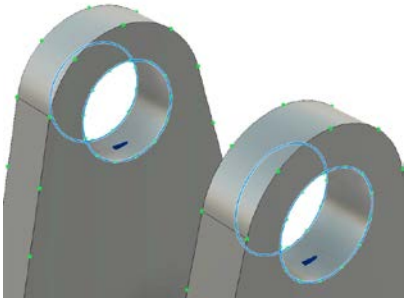
3. Repeat step two except move the slider to the 1% end. A warning will pop up but click Yes. Right click on Load Case1's Mesh and choose Generate Mesh. Notice the number of elements has significantly increased.



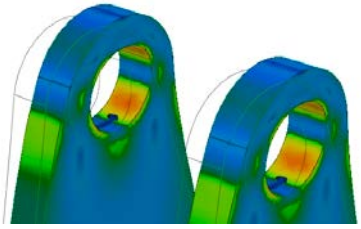
4. Edit the mesh again but this time activate Absolute Size of 8 and click OK. Generate the mesh and view the results. Note the similarity to the first mesh you generated.



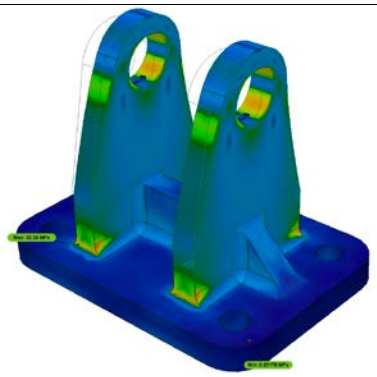
5. Click Simulation>Manage>Local Mesh Control and select the four perimeters for the collars' holes. In the Local Mesh Control control panel, change the Length to 1 and click OK. Then regenerate the mesh and note the differences. This is a way to get better resolution in the important areas.



6. Click Simulation>Solve>Solve and in the control panel click Solve. It will take a moment to display the results. Investigate the results.



7. To further refine the results, click Simulation>Manage>Adaptive Mesh Refinement. Investigate the options, choose High, and click OK. Solve this configuration. Investigate the results.



8. Switch to a mesh view to note how Fusion 360 has refined the mesh in the highly stressed areas.