

## ME 371 – Mechanical Design II

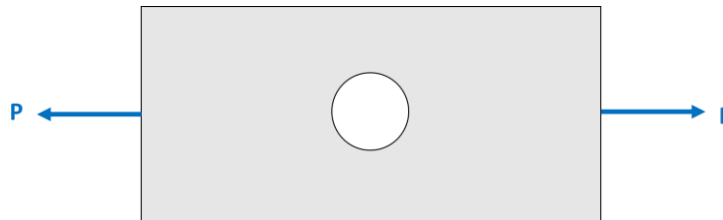
### Finite Element Analysis Lab #2

In this lab you will learn:

- 1) How to specify the size of the elements used to mesh a solid model and refine it globally
- 2) How to use adaptive mesh refinement
- 3) The basics of testing for solution convergence
- 4) How to split faces to specify geometry for application of boundary conditions

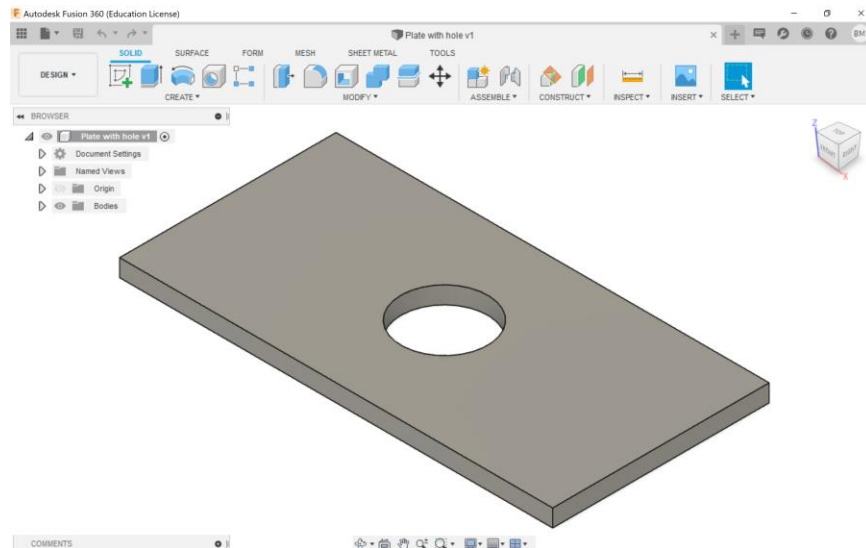
### Analysis of a plate with hole

You will work with a solid model of a 500 x 250 x 20 mm plate with a centrally located 100 mm diameter hole. You'll analyze the model loaded in tension and compare to the theoretical solution from linear elasticity. A schematic of the desired loading situation is shown below.



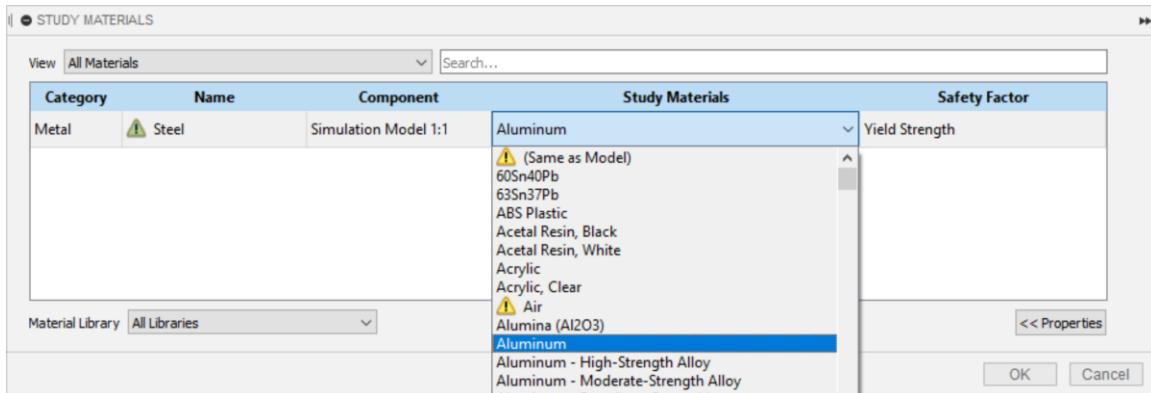
### Step 1: Import solid model into Fusion 360

- 1) Download the file “Plate with hole.step” and place in an appropriate directory on your computer.
- 2) Follow the same set of instructions from Lab 1 to upload the CAD file to the data panel and open the part.



## Step 2: Assign material properties to the model

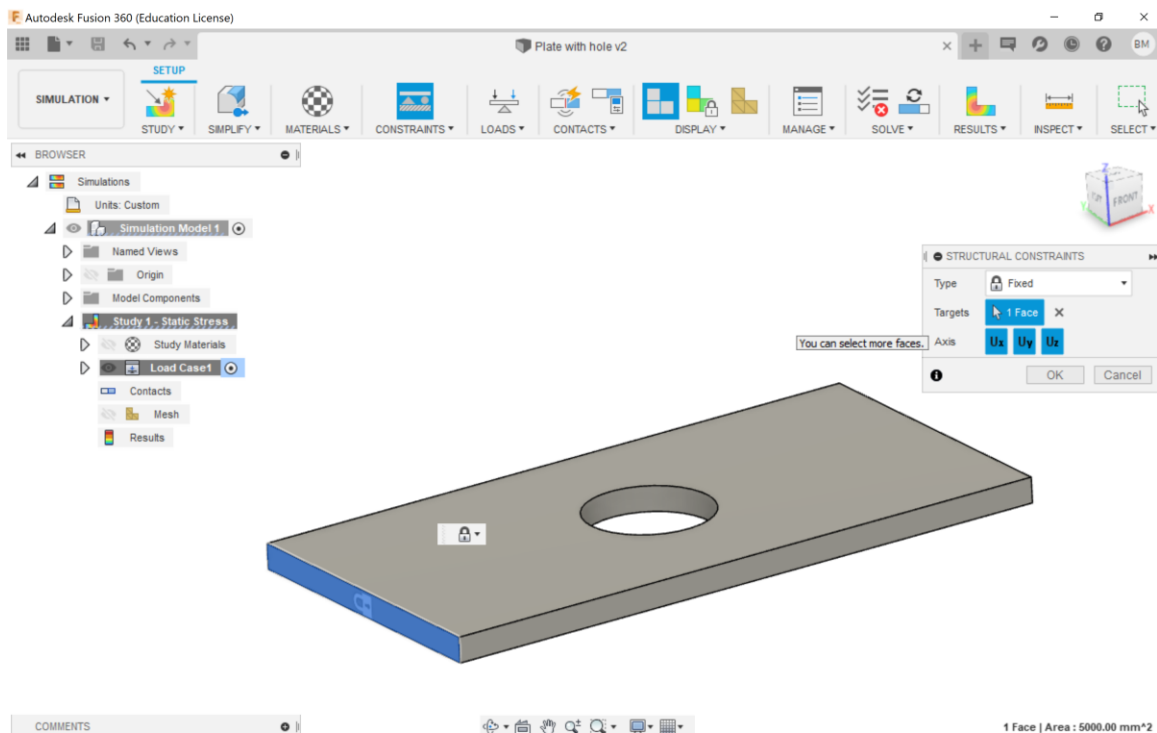
- 1) In the upper left click the drop-down menu currently labeled “Design” and choose instead “Simulation”.
- 2) Click on “Static Stress” and then “Create Study”
- 3) Along the top ribbon, click “Materials”. In the new menu that appears, set the “Study Material” to be Aluminum. Before clicking “OK” to confirm, click the “Properties” button to review the material properties associated with this material.



### Step 3: Apply boundary conditions to the model

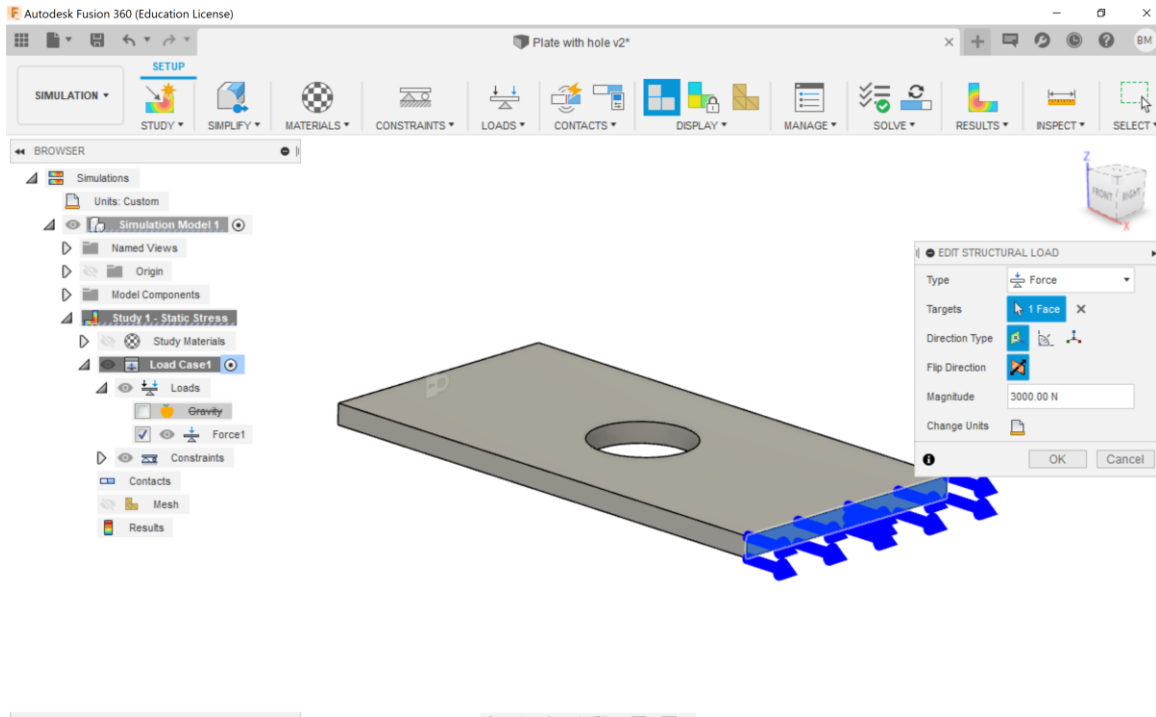
**Important note:** FEA models always (with rare exceptions) require displacement boundary conditions that restrict rigid body translations and rotations. Otherwise, the solution for the displacement field is not unique and the linear system  $\mathbf{Ku} = \mathbf{F}$  does not have a solution. Even though we wish to analyze a plate subjected to tensile loading on either end, we must fix (i.e. specify zero displacement) one face while specifying an applied load on the opposite end in order to restrict the solid from being able to freely translate in space.

- 1) Click the “Constraints” button along the upper ribbon. Rotate the model to view one of the square faces at either end. Click this face. Under the “Structural Constraints” menu that has popped up, set the type to “Fixed” and make sure that all three of the  $U_x$ ,  $U_y$ , and  $U_z$  buttons are selected (highlighted in blue).



- 2) Rotate the model to view the other end of the beam. Click the “Loads” button on the upper ribbon. Click the face on the end of the plate that is not fixed. Choose Direction Type “Normal”. Click the “Flip Direction” button if the arrows shown in the model viewer indicate compressive rather than tensile loading. Specify a magnitude of 3000 N. Click “OK” to confirm.

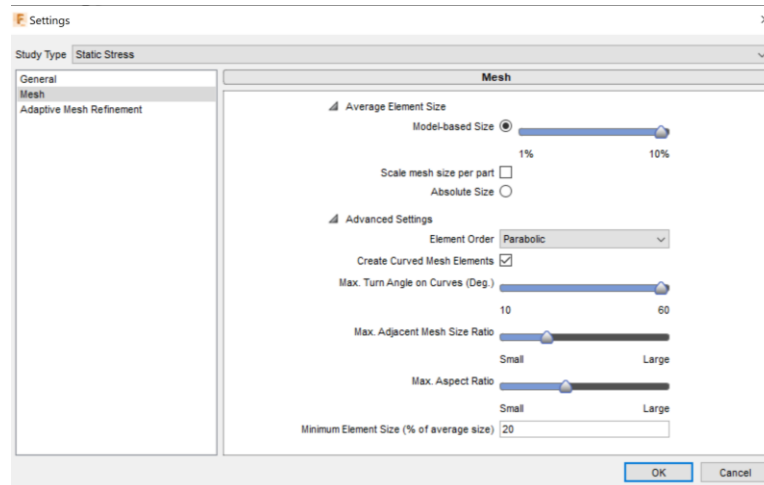
➔ This will result in a 1 MPa nominal tensile stress at the edge of the hole.



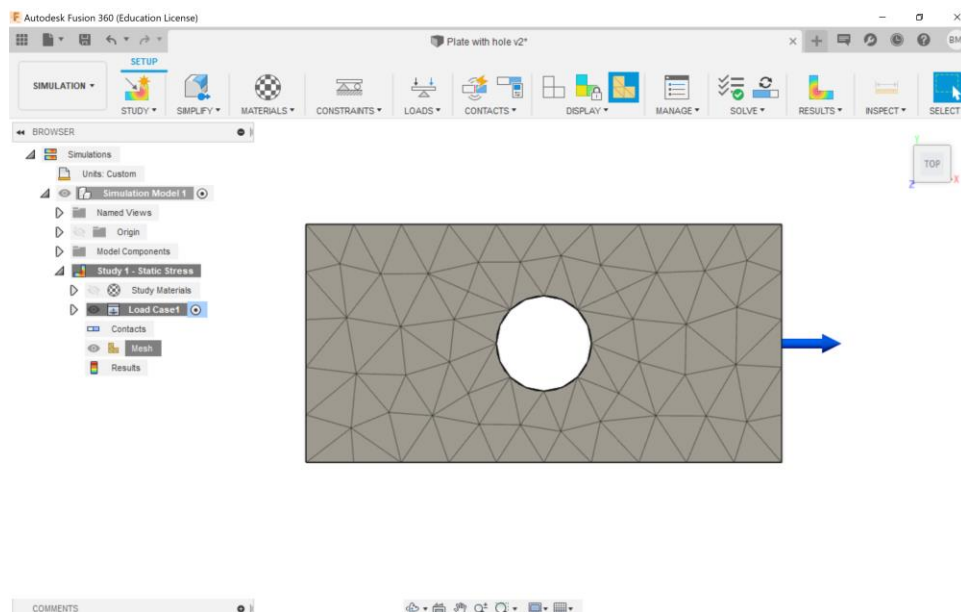
#### Step 4: Solve the model with a coarse mesh

We'll first solve the model using the default mesh settings in Fusion 360. *You might wish to skip ahead to Lab Question 1 to find the theoretical maximum stress based on the theoretical stress concentration factor for this geometry, so you know how close to theory the FEA solutions are.*

- 1) On the top ribbon, click *Mange > Settings > Mesh*. Review the default settings, and accept by clicking “OK”.



- 2) On the model tree on the left hand side, right click on *Mesh > Generate mesh*. After a moment, the mesh will be displayed on the part. You can use the “Toggle mesh visibility” button along the top ribbon to turn the mesh display on and off.

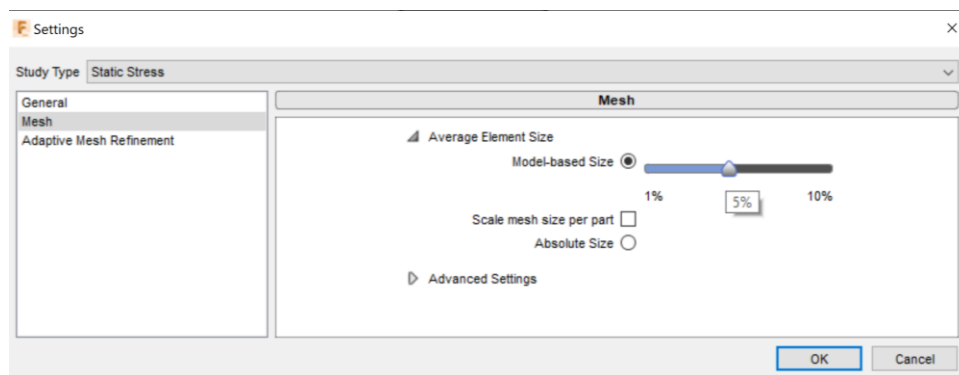


- 3) Pre-check and then solve the model as in Lab 1. View the results of the “Normal XX” stress component and note its maximum value and location

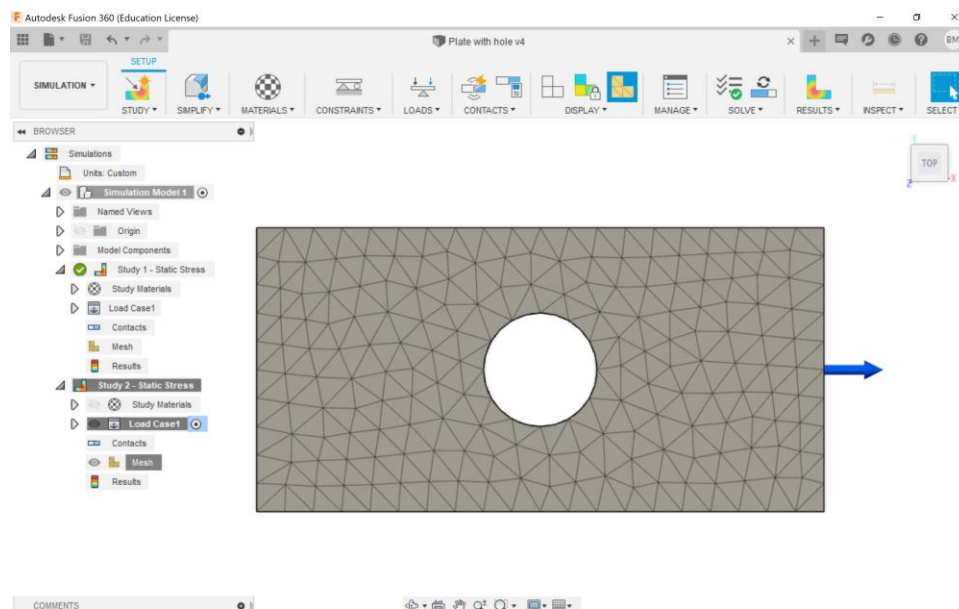
## Step 5: Solve the model using a finer mesh

We'll now refine the mesh by asking Fusion 360 to use smaller element sizes everywhere in the model.

- 1) In the model tree, right-click *Study 1 – Static Stress* > *Clone Study*.  
→ Make sure all subsequent work in this step occurs on *Study 2 – Static Stress*. This will allow us to make changes to the mesh without overwriting previous results.
- 2) On the top ribbon, click *Manage > Settings > Mesh*. Change the *Model-based Size* to 5% by moving the slider. You can hover your mouse over the slider to confirm the percentage value you are selecting. Click “OK” to accept.



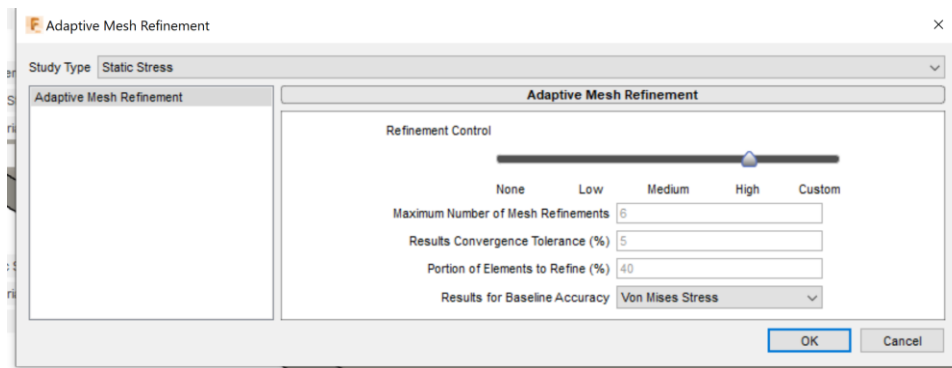
- 3) Generate the new mesh by right clicking *Mesh* > *Generate Mesh* in the model tree. Pre-check and solve the model and note the maximum normal stress value at the edge of the hole.



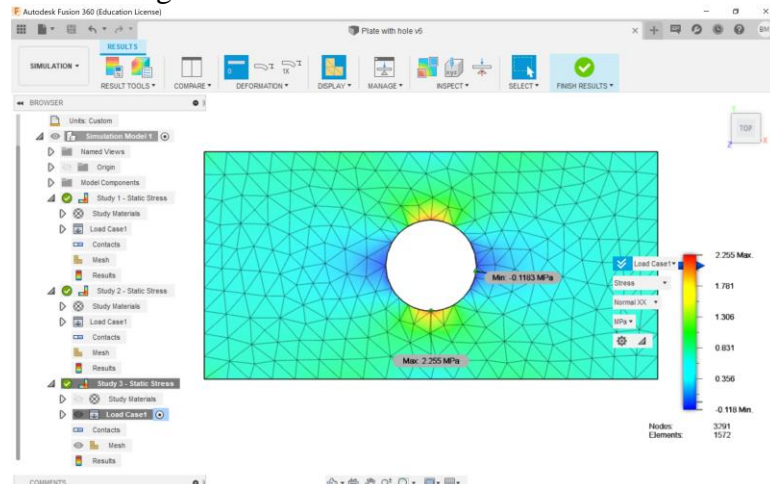
## Step 6: Solve the model using adaptive mesh refinement

**Adaptive mesh refinement (AMR)** is an algorithm that will automatically refine the mesh in parts of the model where the displacement or stress changes rapidly over the spatial domain. Refinement stops when a certain parameter, often the maximum stress or displacement in the model, experiences a relative change smaller than a specified tolerance between successive refinements.

- 1) In the model tree, right-click *Study 2 – Static Stress* > *Clone Study*. *\* Note: you can change the names of the studies you are creating by right-clicking the study name > Settings. You can change the name in the first window that appears. Consider changing the study names to help keep track of the type of mesh used in each.*
- 2) In the upper ribbon, click *Manage* > *Adaptive Mesh Refinement*. Set the Refinement Control slider to “High”. Change the “Results for Baseline Accuracy” from “Von Mises Stress” to “1<sup>st</sup> Principal Stress”.  
→ The default settings will stop refining the mesh when the specified stress value changes by less than 5% between successive refinements.



- 3) Solve the model. Note the refinement of the mesh around the central hole compared to the rest of the model – this is the AMR algorithm at work. Note the maximum normal stress component at the edge of the hole.



## **Lab 2 Questions**

Answer the following questions in a separate document and upload to the appropriate assignment link on the course page.

### **Question 1**

Use the [stress concentration factor finder on mechanicalc.com](https://mechanicalc.com) to find the theoretical value of the maximum stress at the edge of the hole for the geometry under consideration in this lab.

➔ Hint: Recall  $\sigma_{max} = K_t \sigma_{nom}$ . The online calculator linked above will give you the value of  $K_t$  for a specified geometry.

### **Question 2**

While viewing the simulation results from the solution using AMR for mesh refinement, click *Results Tools > Convergence Plot* along the top ribbon. Submit a screenshot showing the first principal stress convergence rate. How many mesh refinements were necessary to achieve the target relative solution error of 5%?

### **Question 3**

Submit screenshots of plots of the “Normal XX” stress component in the results of the three simulations you ran as part of this lab. Make sure the screenshots include the mesh. Find the percent error of each FEA solution relative to the theoretical solution, i.e. % error =  $\frac{|\sigma_{theory} - \sigma_{FEA}|}{\sigma_{theory}}$ .

Which solution (coarse mesh, finer mesh, finer mesh with AMR) appears to be the least accurate? Which is the most accurate? Are these trends in line with your expectations?

#### **Takeaways from this lab:**

- 1) Testing for solution convergence via mesh refinement is critical to obtaining accurate results in your model.
- 2) In general, we can have confidence that an FEA solution has converged if key metrics like stress and displacement undergo relatively small changes in value upon subsequent mesh refinements
- 3) To test for convergence, you can either manually specify smaller element sizes in the model and monitor changes in stress or displacement fields at critical locations in the model with each mesh refinement, or you can use adaptive mesh refinement (AMR) to help automate this process. Depending on the complexity of the model, a combination of these things can also be helpful.