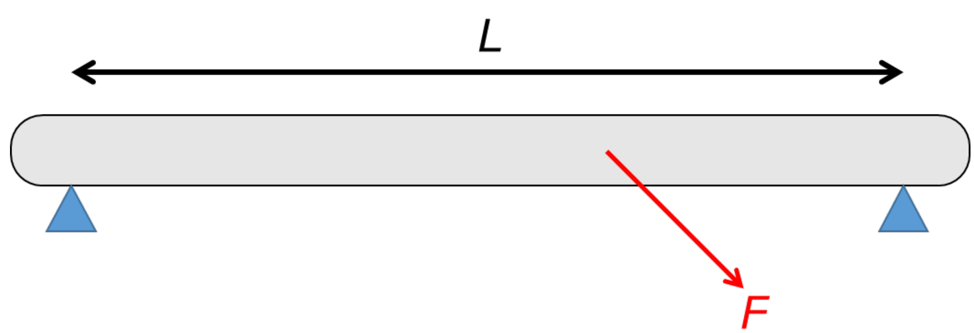
ME 371 – Mechanical Design II
Finite Element Analysis Lab #3

In this lab you will learn:

1) How to use pin supports and bearing load boundary conditions in Fusion 360
2) How to perform some simple calculations to confirm the FEA solution is reasonable

Analysis of pin-supported beam

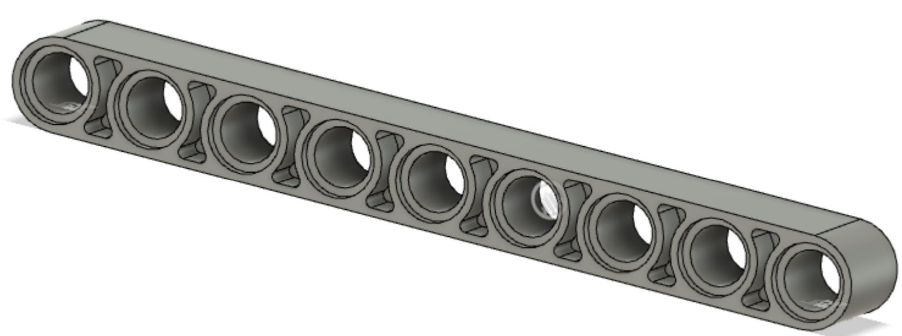
In this lab we'll work with a solid geometry and loading situation for which there is no theoretical solution available for direct comparison. We'll analyze a beam with a periodic pattern of holes and cutouts along its length, pin supported at each end, subjected to an applied bearing load (supplied by, for example, a pin or shaft through one of its holes). The figure below gives a rough schematic of the loading situation we will model (periodic holes/cutouts are omitted here).



Step 1: Import solid model into Fusion 360

1) Download the file "beam_9.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.

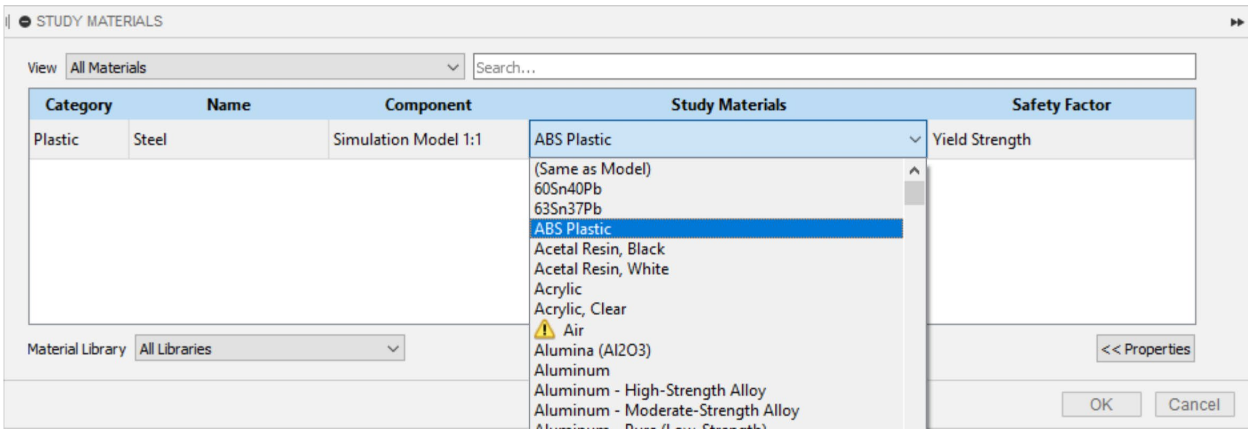


Step 2: Assign material properties to the model

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"

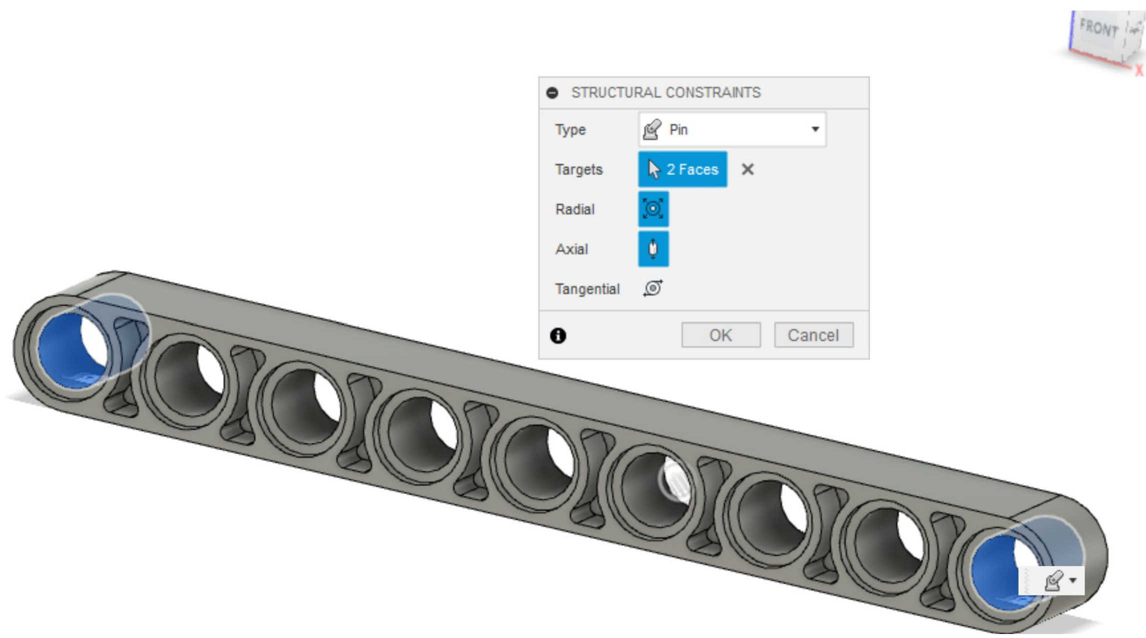
3) Along the top ribbon, click "Materials". In the new menu that appears, set the "Study Material" to be ABS Plastic. Before clicking "OK" to confirm, click the "Properties" button to review the material properties associated with this material.



Step 3: Apply pin constraints to the model

A pin constraint can be applied over a cylindrical surface, e.g. a hole. Using default settings, it will prevent radial and axial motion of this surface, while allowing it to freely rotate, thus simulating connection to a pinned support.

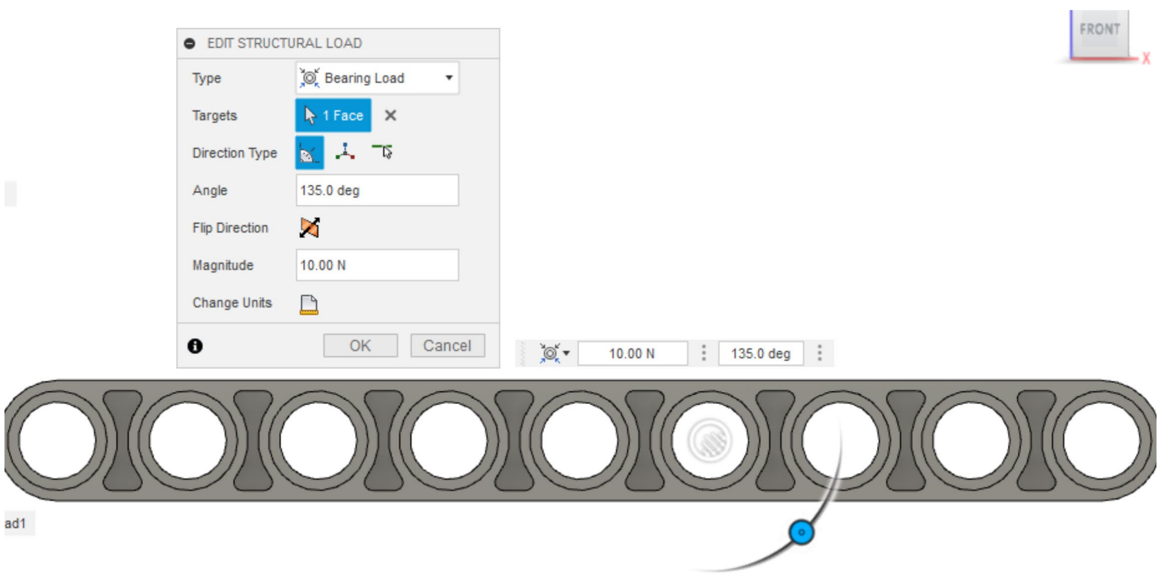
1) Click the *Constraints* > *Structural Constraints* button along the upper ribbon. Set Type to Pin. Select the interior of the holes at either end of the beam, and confirm that the Radial and Axial constraints are active (highlighted in blue). Click OK to accept.



Step 4: Apply a bearing load to the model

A bearing load is appropriate when applying a directional load to a cylindrical face. Fusion 360 distributes the load over the face as if the face were in contact with a solid cylindrical rod imposing the load.

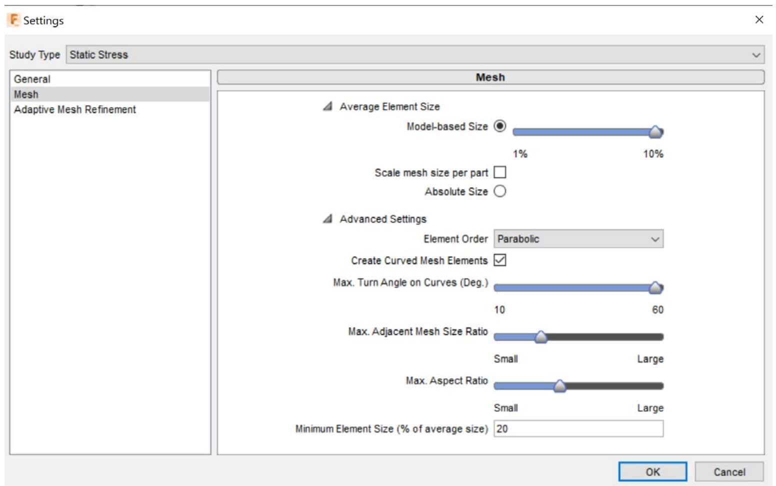
2) Click the *Loads* > *Structural Loads* button along the upper ribbon. Set Type to Bearing Load. Apply it to the interior cylindrical surface of the 6th hold from one end of the beam (as shown below). Set Direction Type to Angle, and input an angle of 135 degrees and a force magnitude of 10 N. Click OK to accept.



Step 5: Solve the model with a coarse mesh

We'll first solve the model using the default mesh settings in Fusion 360.

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



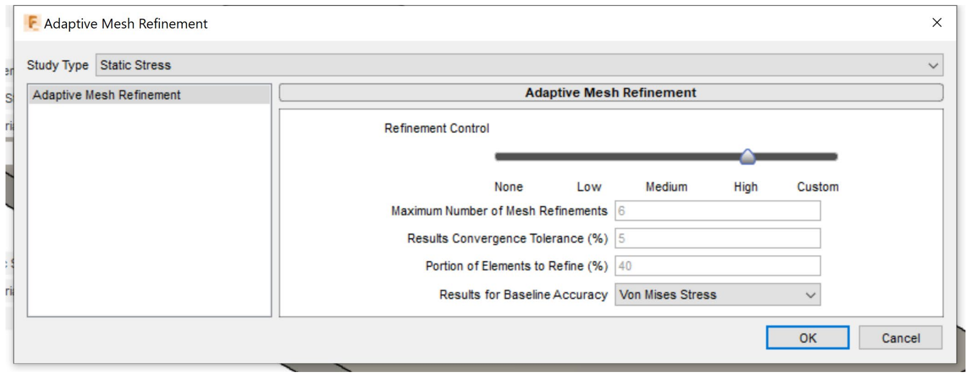
2) On the model tree on the left hand side, right click on *Mesh > Generate mesh.* After a moment, the mesh will be displayed on the part. You can use the "Toggle mesh visibility" button along the top ribbon to turn the mesh display on and off.

3) Pre-check and then solve the model as in Lab 1.

Step 6: Solve the model using adaptive mesh refinement

1) In the model tree, right-click *Study 1 – Static Stress* > *Clone Study.*

2) In the upper ribbon, click *Manage* > *Adaptive Mesh Refinement*. Set the Refinement Control slider to "High".



3) Solve the model. Note the refinement of the mesh near the application of the bearing load compared to the initial coarse mesh solution.

Lab 3 Questions

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

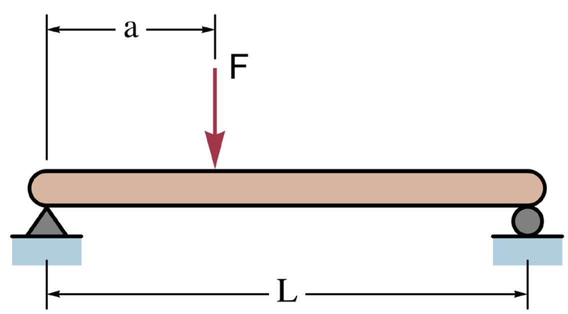
Question 1

Submit screenshots of the FEA solution contour plots of the Total Displacement and von Mises stress contours for both the initial solution using default mesh settings and the AMR solution. *Consider using the "Compare" tool along the upper ribbon when viewing the results. You can set each panel to display the results of one of the two studies using the drop down menu in the bottom left of the panels displayed in the "Compare" view.*

a) How does the maximum displacement compare in each model?
b) How does the maximum von Mises stress compare in each model?
c) How is the AMR mesh different than the mesh using default settings? Does this change make sense to you based on the loading conditions and resulting deformation?

Question 2

The loading situation modeled in this lab is similar to the one shown below



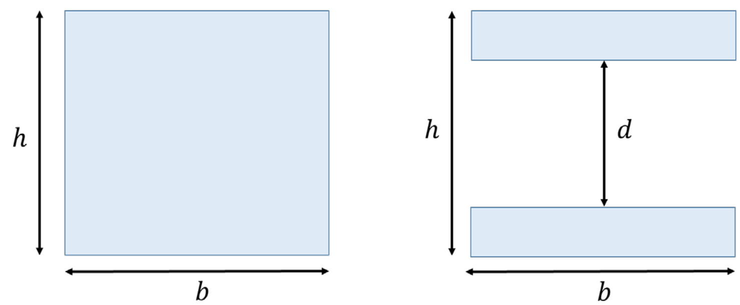
The equation for the maximum vertical deflection for this beam according to Euler beam theory is

$$\delta_{max} = \begin{cases} \dfrac{Fa(L^2 - a^2)^{\frac{3}{2}}}{9\sqrt{3}LEI}, & \text{if } a \le \dfrac{L}{2} \\[2ex] \dfrac{Fb(L^2 - b^2)^{\frac{3}{2}}}{9\sqrt{3}LEI}, & \text{if } a > \dfrac{L}{2} \end{cases} \quad \text{where } b = L - a$$

a) How are the boundary conditions (supports and load) different in the Euler beam model in the figure above compared to the FEA simulation?

b) Obtain a lower and upper bound on the maximum deflection of the solid beam in this lab with the following procedure.
   i. Find the values of $E, a, L,$ and F based on the solid model geometry, load conditions, and material properties. *You can use the Measure or Inspect tools in the upper ribbon to click on part features and get sizes/distances/etc. You can view material properties in the material property assignment window.*

ii. The moment of inertia I varies along the section because of the periodic pattern of holes and cutouts along its length. Find an upper bound on the 2nd moment of inertia I by assuming the part is a solid rectangle through its entire length. Find a lower bound on I by assuming that the part has a hollow rectangular cutout through its entire length (as if the middle section was missing everywhere instead of just periodically along the beam length). These situations are represented by the figures below, where h is the height, b the width, and d is length of missing material. When choosing a value of d, use the largest present in the model, for example the diameter of the outer rims at the holes, or the height of the periodic cutouts.



$$I = \frac{1}{12}bh^3 \qquad\qquad I = \frac{1}{12}bh^3 - \frac{1}{12}bd^3$$

iii. Does the FEA prediction of maximum deflection fall between your upper and lower bound estimates on the deflection of the beam? Does this calculation give you confidence that the FEA results are accurate?

Takeaways from this lab:

1) Pin constraints and bearing loads can model common connection and load points in a solid component that is supported by pin connections or has loads applied to cylindrical surfaces.
2) It's a good idea to perform a back of the envelope calculation to confirm that your model results make sense. Some sort of geometry simplification and comparison to theoretical solutions, as suggested here, is often a good starting point. Keep in mind the validity of your comparison given the differences in assumptions, boundary conditions, etc. between your chosen theoretical solution for comparison vs your FEA model.