ME 371 – Mechanical Design II
Finite Element Analysis Lab #1

In this lab you will learn the basic workflow of modeling in FEA:
1) Create or import a solid model
2) Assign material properties to the model
3) Assign displacement and force boundary conditions to the mode
4) Mesh the model
5) Solve for the resulting displacements and stresses and interpret results

Part 1: Initial model

Step 1: Import solid model into Fusion 360

The part we'll work with in this lab is a simple 50x50x500 mm solid beam. Follow the steps below:

1) Download the file "Simple cantilever.step" and place in an appropriate directory on your computer.

2) Open Fusion 360, e.g. on Windows, _Start Menu > Autodesk Fusion 360._

3) Open the data panel using the "Show data panel" button in the upper left of your screen. Click on "Upload", and then use the select file button to navigate to your file, or use the drag and drop option. Choose a location to save it in the Fusion 360 cloud file structure in the data panel (e.g. consider making a folder named "Lab 1" in the directory structure in the data panel) and click "Upload".

4) Open the solid model in Fusion 360 by finding it in the data panel and double left-clicking or _right click > Open_

5) Upon completion of these steps, the part should appear as in the figure below. Try clicking on various edges and faces to verify the part dimensions, which will appear in the bottom right of the window.
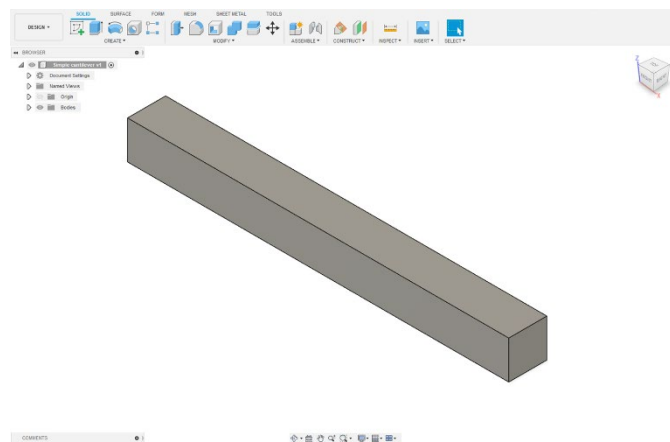
Step 2: Assign material properties to the model

The response of the model to any external load will depend on what material it is made of. In this lab we'll use Fusion 360's material library to assign a material to the model

1) In the upper left click the drop-down menu currently labeled "Design" and choose instead "Simulation". This is where the FEA model settings will be specified.

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 Steel ASTM A36. Before clicking "OK" to confirm, click the "Properties" button to review the material properties associated with this material.
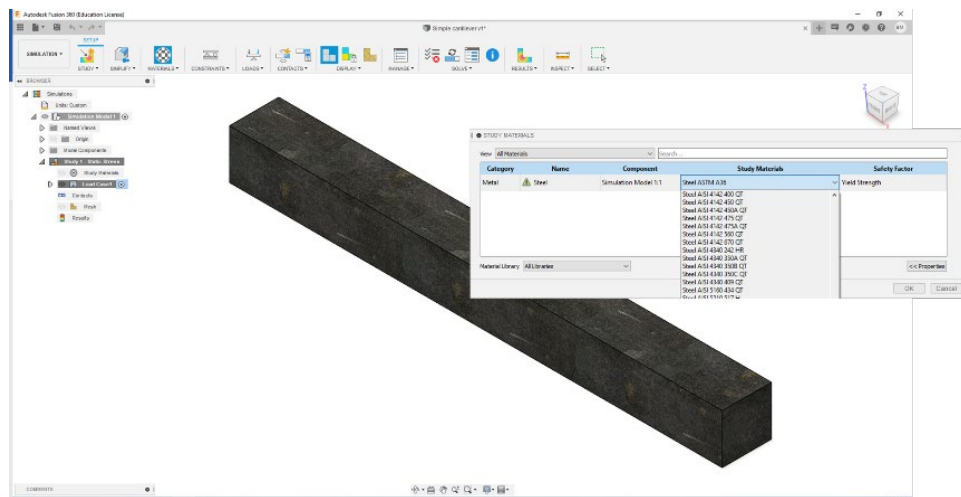
Step 3: Apply boundary conditions to the model

We'll simulate the displacement and stress response of this structure operating as a cantilever beam. We'll apply displacement boundary conditions that model one end of the structure as being fixed to a rigid wall support, and apply a load at the other end.

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).
   → This step constrains the displacement of all element nodes located on this face to be zero in all three coordinate directions.
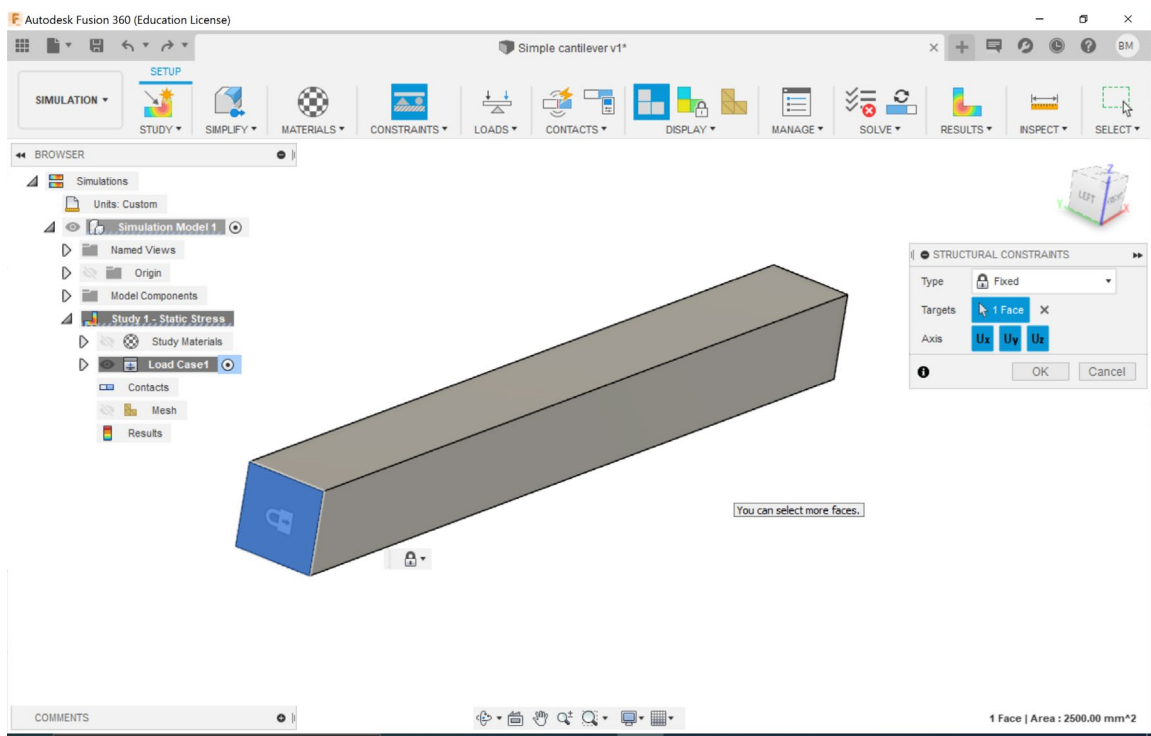
2) Rotate the model to view the other end of the beam. Click the "Loads" button on the upper ribbon. Click the top edge of the face on the other end of the beam. In the "Structural Loads" menu that has appeared, set type "Force", "Direction Type" to "Vectors (x, y, z)", and set the Fz component to be -1000 N while keeping the others at zero. Once you've down this, you should see a blue arrow appear indicating the direction of the force on the edge you have selected. Click "OK" to confirm.

   ➔ This step spreads out the force of 1000 Newtons across all of the element nodes located along this edge of the solid model.
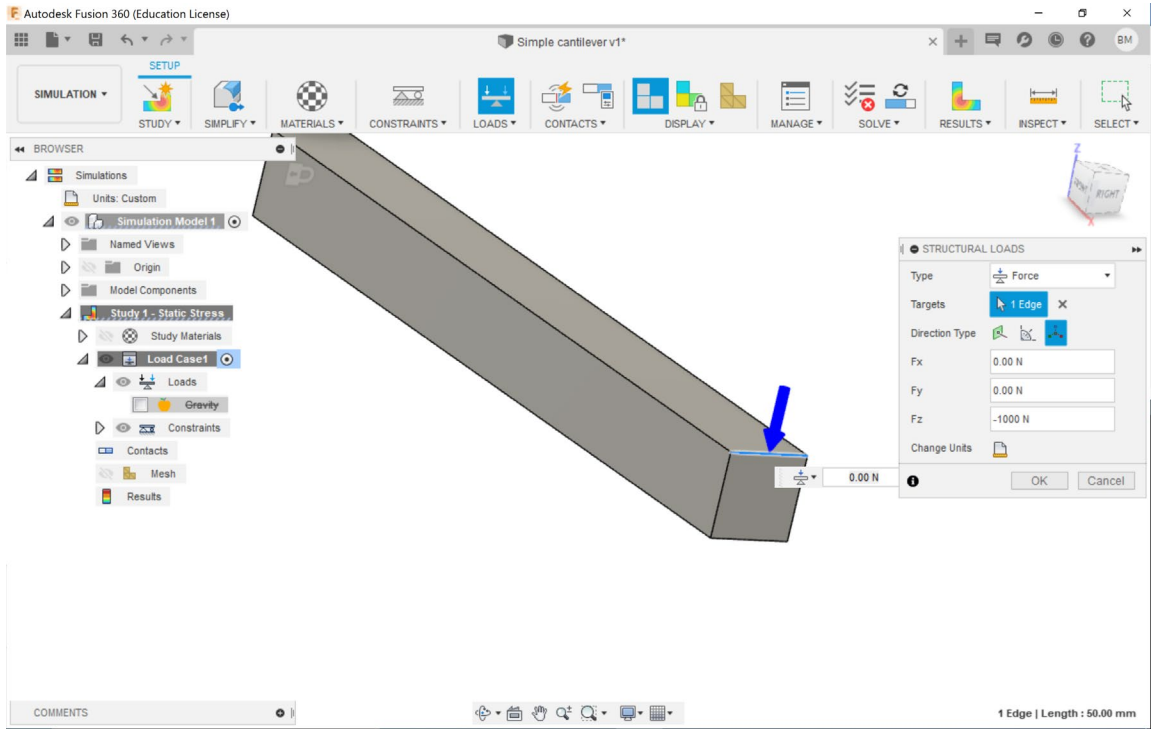
Step 4: Mesh the model

Fusion 360 will take care of meshing the model for you automatically if you do nothing with the mesh settings; however, it is always good practice to create and observe the model mesh to confirm it is reasonable.

1) On the top ribbon, click the *Mange > Settings >Mesh*

2) You'll see settings about the average element size. Click on the "Advanced Settings" menu and note the default settings about the element order (Parabolic), aspect ratio ranges, and minimum element size. We'll look at these settings more closely in a future lab. Click "OK" when you are done reviewing the mesh settings.

3) On the model tree on the left hand side, right click on "Mesh" and choose *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.
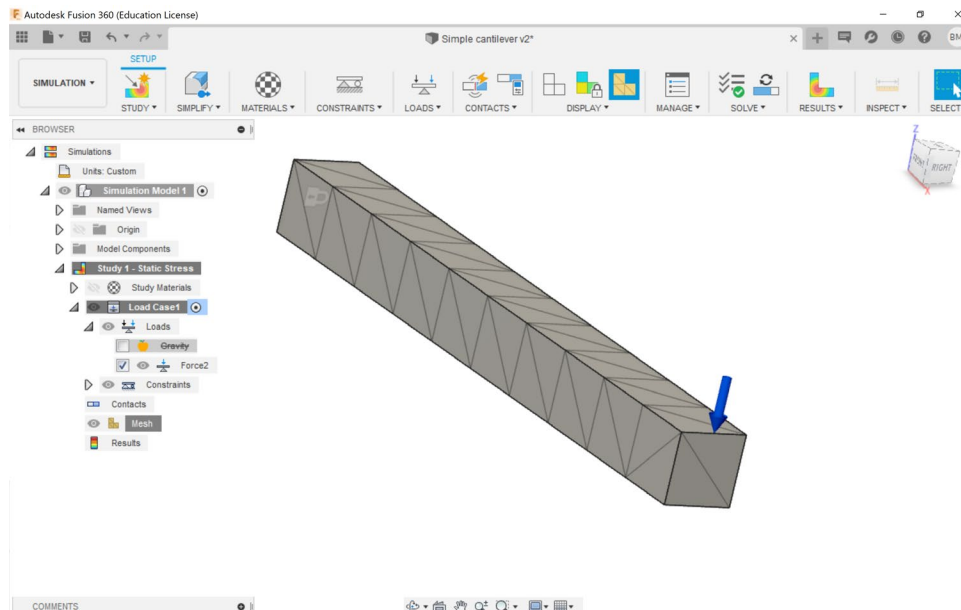
Step 5: Solve the model

You'll now solve the model to obtain the displacements and stresses resulting from the specified boundary conditions.

1) Along the top ribbon, click on *Solve > Pre-check*. This will confirm that your settings are such that the model can be solved. If you get any errors at this stage, review the procedure here to make sure you did not skip any steps.

2) Click on the "Solve" button in the top ribbon to solve the model. The default and recommended option is to solve on the cloud. Make sure your simulation model is selected with a checkbox and then click "Solve 1 Study" button.

3) After a few moments, the simulation results will be available. A "Results Detail"s box will appear; close this for now as we'll explore the results in a bit more detail on our own. Close the "Job Status" box as well if it is still open

4) The default view is a plot of safety factor over the domain of the part. The safety factor definition is specified during the material assignment process; default is safety factor based on material yield strength. Next to the color legend, click the drop down menu currently labeled as "Safety Factor" and change it to stress to see a plot of any stress component throughout the body.
   a. The default view is "Von Mises". The von Mises stress can be considered an effective stress (1 value rather than 6 components) and is commonly used in failure criteria evaluation; we'll learn more about the von Mises stress later in the course.
   b. Try looking at the various stress plot options, e.g. "1st Principal", "Normal XX", etc, by clicking on the drop down menu currently labeled "von Mises". You can change "Stress" to "Displacement" in the drop down menu to view the displacement magnitude (labeled "Total") or any individual x/y/z component. You can also view any strain component throughout the body. Take a moment to cycle through these other display options and view the results.
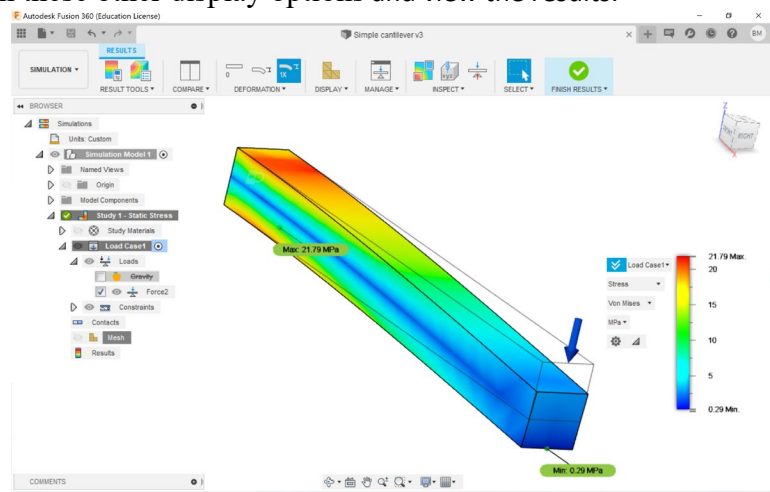
5) By default, Fusion 360 will display the deformed shape of the model, with displacements amplified so that they are easily observed even if they are small – Fusion 360 calls these

amplified displacements "Adjusted" displacements. Use the "Deformation" tab along the top ribbon to change to viewing the model using "Undeformed", "Actual" or "Adjusted" displacements.

➔ "Adjusted" displacements are helpful in confirming that the deformations make sense to use given the loads we have applied; just keep in mind they are not necessarily representative of the actual displacement magnitudes experienced by the model.

6) When finished viewing results, click on the "Finish Results" button on the top ribbon. Save (ctrl+S) the model for future reference. The simulation results can be accessed in the future by going to the "Simulation" module and clicking the "Results" button on the top ribbon.

Part 2: Change the boundary conditions

Instead of applying a load at one end, we'll now prescribe a specified deflection at the end of the beam. Follow the instructions below:

1) Copy the current load case to a new one, as a starting point for changing the boundary conditions: Under the model tree on the left, *right click "Load Case 1" > Clone load case.*

2) Right click the new load case (likely named "Load Case2") and click "Activate load case" to make sure it is the active case under consideration. This allows us to work on and adjust the boundary conditions in this load case.

3) Expand the new load case, expand "Loads", right-click the load shown (likely named "Force2") and click "Delete" – this removes the applied load from the model.

4) Click the "Constraints" ribbon along the top. Click the face on the end that the load was originally applied to. Set type to "Prescribed Displacement". Make sure only the Uz button is selected, and type "-0.25 mm" into the Uz box.
   ➔ This will force all element nodes on this face to displace 0.25 mm in the negative z direction, but will not constrain their displacement in the x and y directions.
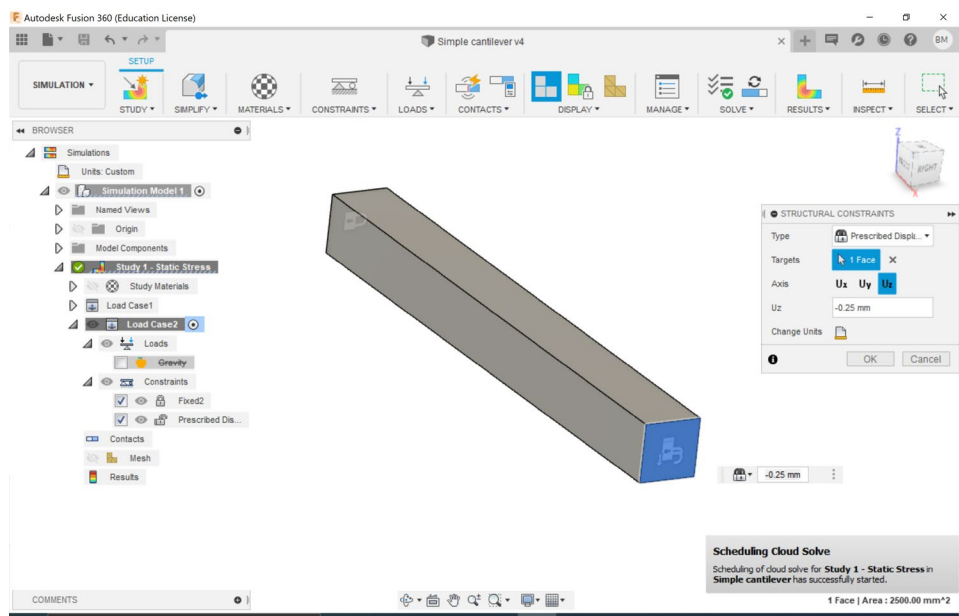
5) Solve the model as before (the mesh has already been created, no need to regenerate). When viewing the model, you can use the Load Case drop-down menu to switch between reviewing results for the previous and current load cases.
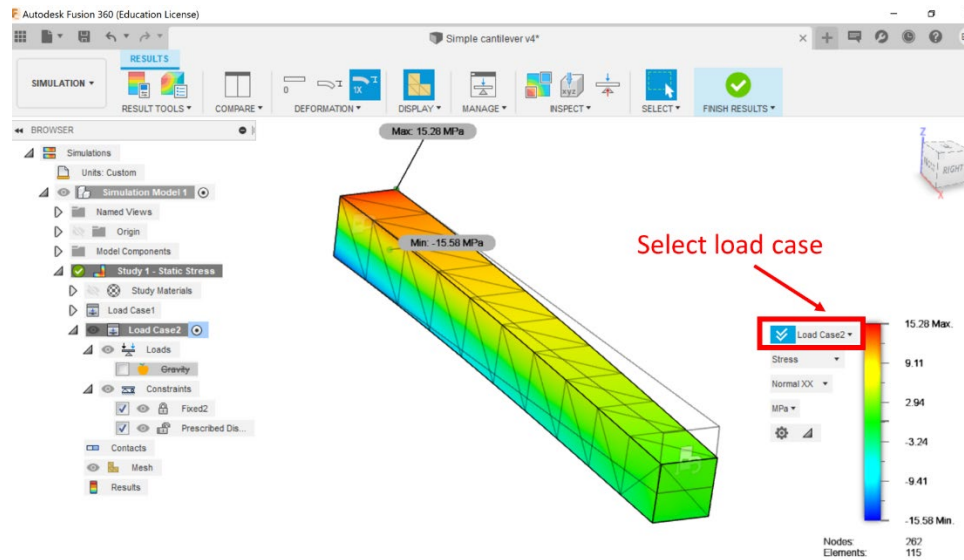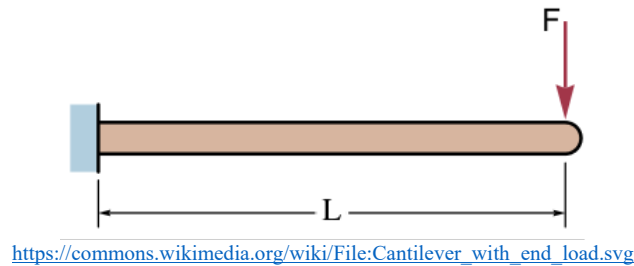
Lab 1 Questions

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

Question 1

The FEA model that you have created models the classic example of an end-loaded cantilever beam:

According to Euler beam theory, the vertical deflection δ of the end of such a beam is given by $\delta = \frac{FL^3}{3EI}$ where F is the applied load, L is the length of the beam, E is the elastic modulus and I is the area moment of inertia of the beam's cross-section about its neutral axis.

a) Find the theoretical deflection of the beam analyzed in the solid model, and compare with the maximum deflection predicted by the FEA model.
    i. Hint: remember the material properties can be viewed by clicking on *Materials > Properties* while in the Simulation module (and not in "Results" mode)
b) How does the theoretical deflection compared to the deflection predicted in the model? Does this give you confidence that you set up the model correctly?

Question 2

The bending stress in an elastic beam is given by $\sigma = \frac{MC}{I}$.

a) What is the theoretical maximum stress in the beam from the solid model? Where <u>exactly</u> does it occur?
b) What is the FEA-predicted maximum bending stress in the model? Where <u>exactly</u> does it occur? How does it compare to the theoretical prediction? **Include a screen shot of your results to confirm your maximum stress value.**
    i. Hint: select the appropriate stress component when viewing results to correctly report this value.
c) FEA has a tendency to over-predict stresses at sharp corners (perfect 90 degree sharp corners never exist in real world geometry) and at unrealistically stiff supports. While in "Results" mode, use the "Create point probe" button on the top ribbon to hover your mouse around the model and view the stresses at any point. Is the maximum bending stress closer to the theoretical value away from sharp corners? Comment on your findings. **Include a screen shot of your results and probe location(s) to confirm your findings.**

Question 3

View the model results for the first load case (Part 1) and note the 6 strain components. What is the magnitude of the largest strain component? Do the same for the largest stress component (i.e. "1st Principal" stress) – how does it compare to the material's yield stress? Recall that we are performing this FEA analysis assuming that the material behaves in a linear elastic manner – are the stresses and strains small enough for this assumption, and therefore the FEA results, to be valid? You do **not** need to include a screenshot as part of your answer.

Question 4

In Part 2 of the lab, you prescribed a vertical displacement of 0.25 mm on one end of the beam while keeping the other end fixed

    a) While in "Results" mode for the 2nd load case analyzed in Part 2, click on the "Reactions" button on the top ribbon and left-click the face with fixed displacement conditions to select it. A window should appear displaying the three reaction forces and moments acting on that face. These represent the FEA-calculated reaction forces and moments at the wall support for the cantilever beam. Based on the theoretical load-deflection relation for the cantilever beam, how does the reaction force in the z direction compare to theory? Comment on the result and whether this gives you confidence that the model has been set up correctly. You do **not** need to include a screenshot as part of your answer.

> **Takeaways from this lab:**
> 1) The basic requirements to analyze a solid model using FEA are 1) material property specification; 2) Application of constraints and loads (i.e., displacement and force boundary conditions), and 3) discretization of the solid body with a finite element mesh.
> 2) Displacement and force boundary conditions can be applied to points, edges, and faces in a solid model.
> 3) When possible, it is good to verify FEA results by comparing to theory. Of course, we don't usually have an exact solution available to compare against. In this case we can often find approximations. For example, if we had a beam with a complex, non-prismatic geometry, we might be able to use Euler beam theory to estimate an upper and lower bound on expected displacements or stresses, and compare our FEA results to this.