Engineering 1182

Fundamentals of Engineering

SolidWorks Simulations:
Structural Analysis
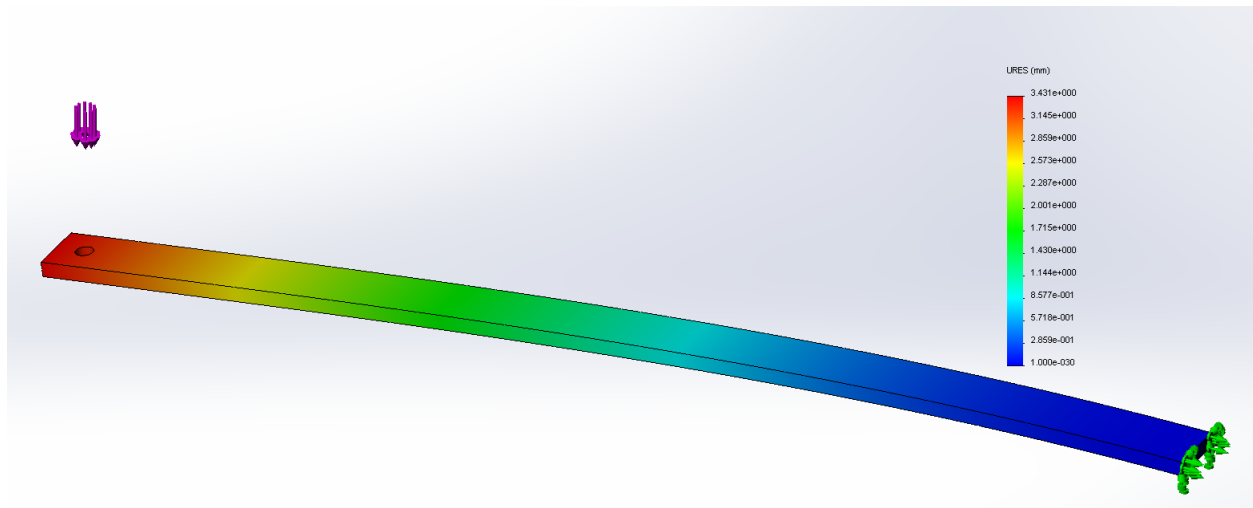
Objectives

1. Simulate in SolidWorks the deflection of a cantilever beam from a fixed load.
2. Understand the benefits and limitations of physical experiments and computer simulations.

Background

Numerical Analysis

Numerical analysis is a problem-solving approach for many practical engineering problems. This technique uses numerical approximations as opposed to symbolic manipulation to solve complex equations or systems of equations. Here, we will use numerical analysis to simulate the material responses of a cantilevered beam. Specifically, we will perform a structural analysis using SolidWorks Simulations to calculate displacements, strains, and stresses of a cantilever beam under a given load.

Discretization & Mesh

This calculation is performed using a finite element method (FEM) wherein the part being studied (in our case, a beam) is discretized into smaller parts called finite elements. These elements are described by their nodes, or the bounding points of the elements. Figure 1 below shows various element shapes and the nodes (red dots) that define them. These elements make up what is known as a mesh, shown below in Figure 2. When you "mesh" a part, the elements and nodes are produced for the entire volume (the inside too!) of the model.
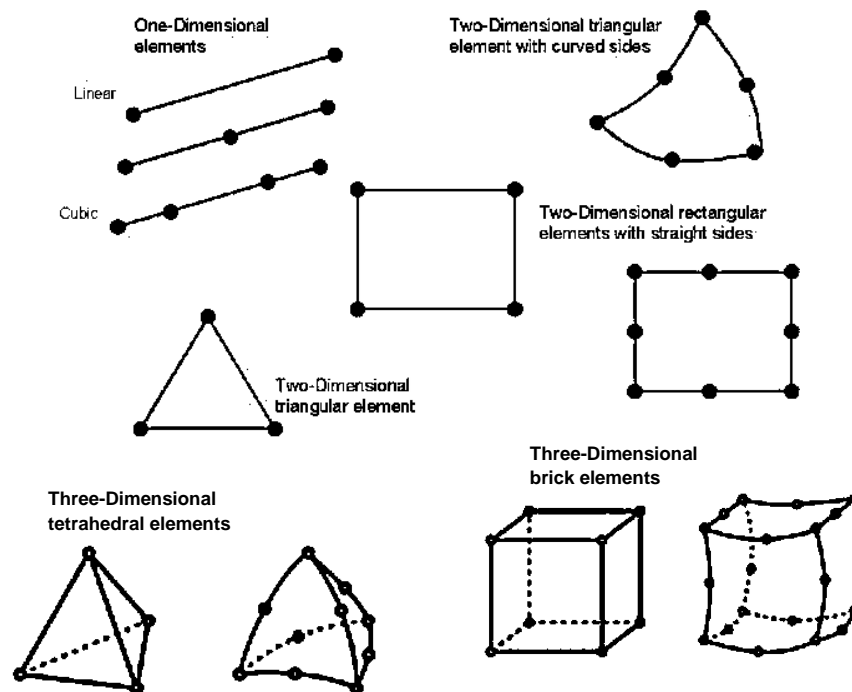
Figure 1: Various element types made up of nodes for part discretization.
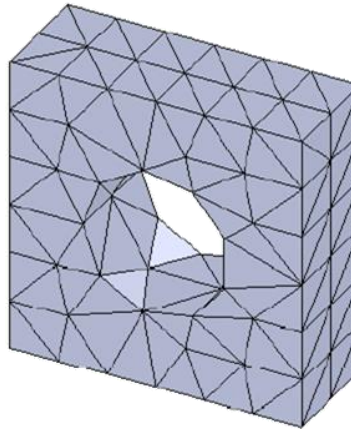
Figure 2: Meshed model of a block with a hole in the middle showing three dimensional tetrahedral elements.

By breaking up a part into small, finite elements, a computer can apply governing equations of the behavior which are known, understood, and much simpler to solve than an equation that would describe the response of the entire part. The computer sums up the responses of all these finite elements to approximate the more complex equation that describes the response over a much larger domain.

Boundary Conditions

In order to accurately represent the response of the material, it is necessary to constrain the movement of the part using boundary conditions. For instance, with the cantilevered beam, one boundary condition necessary to achieve a correct deflection behavior would be to fix one end of the beam such that it does not move.

Additionally, we need to apply the boundary condition that causes the beam to deflect. In our case, we are applying a load at the unfixed end of the beam. This force is quantified and then applied in the simulation to the correct position on the beam. Figure 3 below shows the force boundary condition applied (purple arrows) at one end of the beam, the fixed boundary condition at the other end of the beam (green arrows), and the resulting deflection throughout the length of the beam.
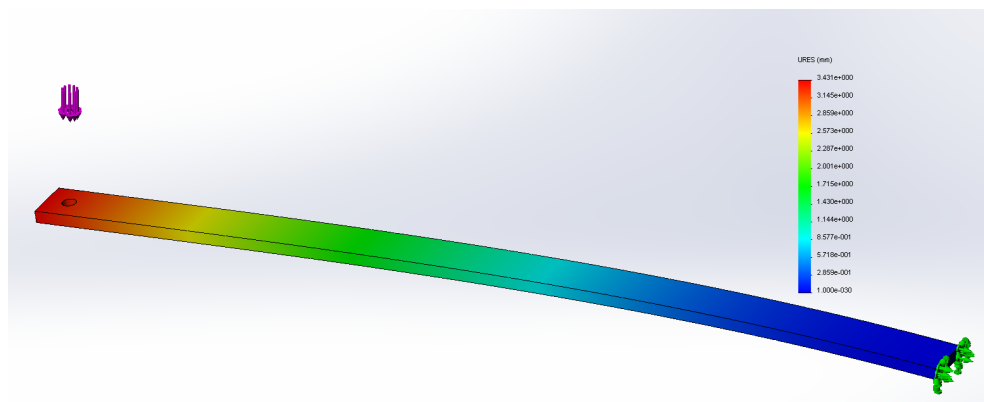
Figure 3: Example of FEA on a cantilevered beam depicting boundary conditions and resulting deflection.

Accuracy of Results

An important thing to note with any type of modeling or simulation is that the results are only as good as the accuracy of the governing equations and the conditions you set and will generally not account for any other applicable factors. The largest source of error typically results from a mesh which is too coarse to properly capture the physics of the problem. As a general rule, the smaller the elements used to discretize the part, the better the simulation will be at the cost of computational time. Another assumption required for this problem considers a constant value of Young's modulus for every element of the mesh and therefore does not account for any defects or variability of strength within the material along the beam.

Depending on the specific assumptions and conditions, there can be varying degrees of difference between the predicted and real-world behavior. Therefore, it is often important to validate a design through research and experimentation.

Procedure

In this study, we will use SolidWorks Simulation to perform a structural analysis and calculate maximum deflection of rectangular beams of three different materials. This simulation will mimic the hands-on experiment from the ENGR 1181 Beam Bending lab and will serve as an introduction to SolidWorks Simulations, which will be explored further this semester.

Download & Open the Model

Find the SolidWorks part file named "Rectangular_Beam.SLDPRT" on Carmen and save it to your computer. When you open the file, you will see a beam just like those used in the ENGR 1181 Beam Bending lab, as shown in Figure 4.
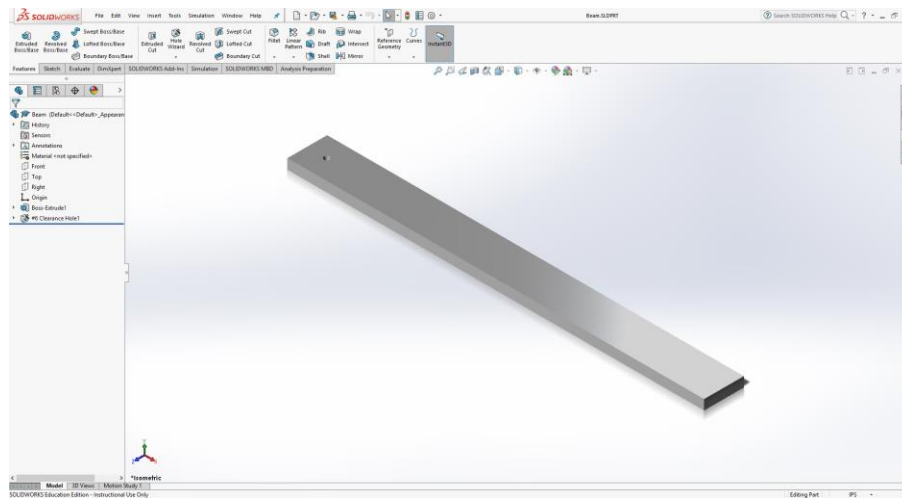

Figure 4: The beam model.

Opening Simulation

By default, SolidWorks Simulation may not open automatically when SolidWorks does. To open SolidWorks Simulation, hover over the arrow next to the SolidWorks logo at the top-left portion of the screen, then go to Tools > Add-Ins, as shown in Figure 5. You may need to scroll down to see "Add-Ins" using the arrow at the bottom of the Tools menu.
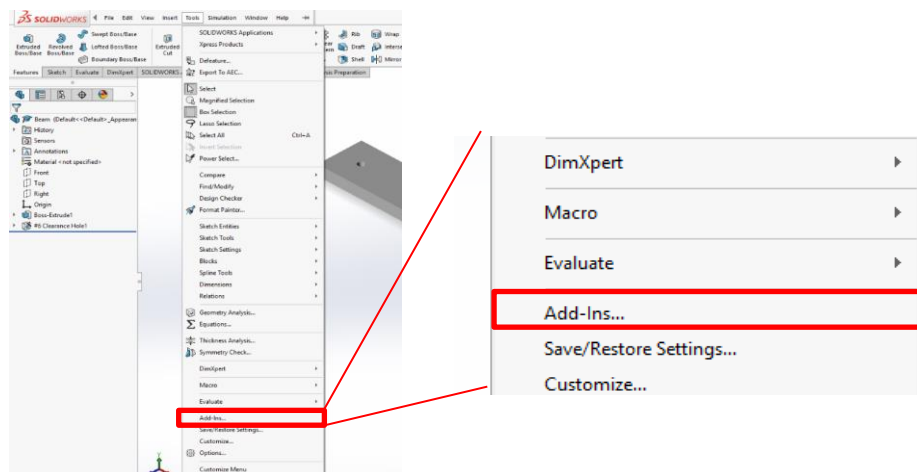

Figure 5: The SolidWorks "Tools" menu.

The "Add-Ins" window, shown in Figure 6, will appear. Check the box to the left of "SolidWorks Simulation" and click "OK" to enable Simulation.
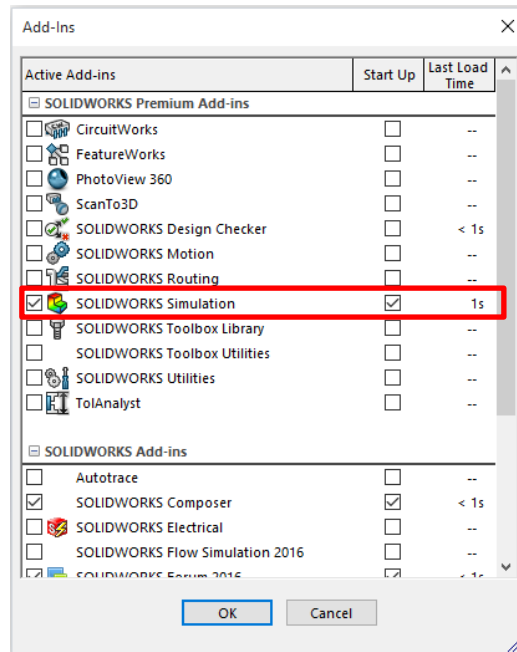

Figure 6: The "Add-Ins" window.

There should now be a new tab named "Simulation" in the upper-left, next to "Features," "Sketch," "Evaluate," etc. (see Figure 7, below).
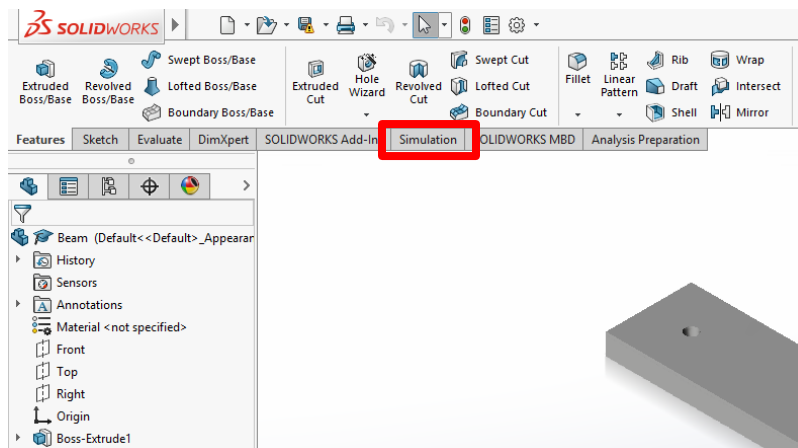

Figure 7: Simulation tab now present.

Static Analysis

For this simulation, we must specify the location and magnitude of each load, as well as specify where and how the model is supported. Identifying areas where stresses are highest/lowest quickly shows a designer where a model can be improved by adding support or by removing excess material.

We are going to investigate the deflection of a beam due to loading much as we did in the hands-on Beam Bending lab. Therefore, we will assume a load equal to 4.90 Newtons (a force equivalent to that experienced

by the beam in the physical experiment when loaded with all 10 weights) is applied at the hole in the beam by which the weight holder would rest in the physical experiment. To accomplish this, we will be running a static study.
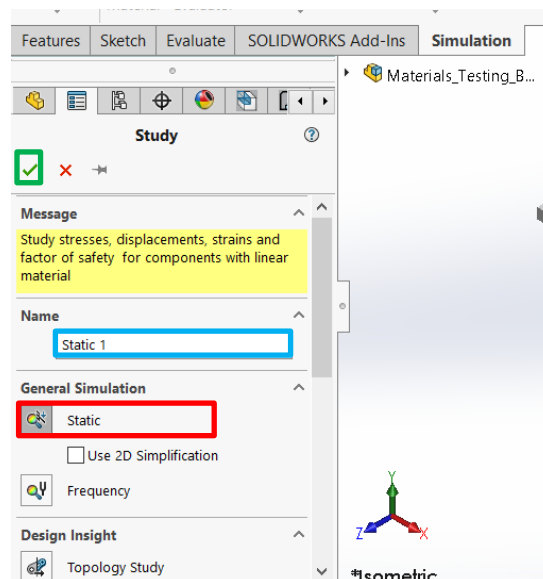
Creating a New Study

To create a static study, click the Simulation tab in the upper-left. There should be a button labeled "New Study." Click the arrow just beneath it and choose "New Study," as seen in Figure 8.
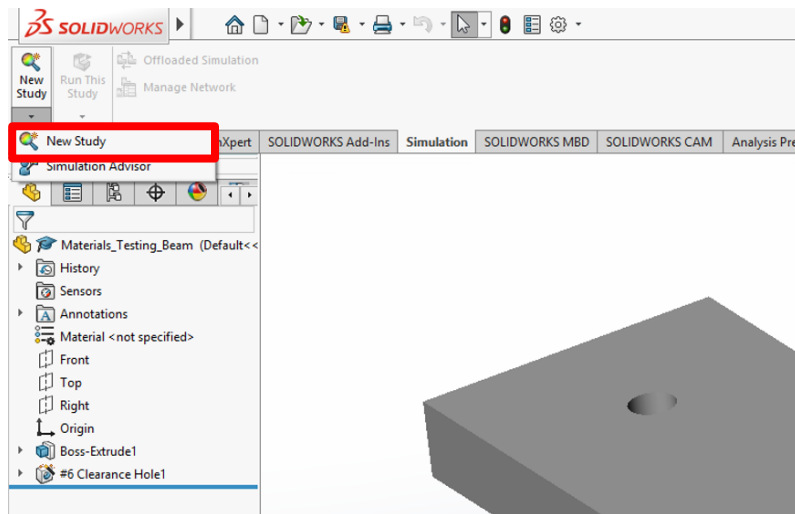
Figure 8: Under Study Advisor, click "New Study."

In Figure 9 below, you can see all the types of studies available in SolidWorks Simulation. Click "Static," name the study something memorable, and click the green check mark.


Figure 9: Starting a new static study.

Below the normal display pane on the left side of the SolidWorks window, a static study pane should open. We can use this to specify our fixtures and loads (boundary conditions).

Setting Up Fixed Geometries

To set up the fixtures on the model, right-click "Fixtures" in the static study pane. Choose "Fixed Geometry" as the fixture type for this study. This is shown in Figure 10. Note that there are MANY ways to constrain your part such as pins, rollers, or hinges, if your model requires it. For now, though, the fixed geometry suffices.
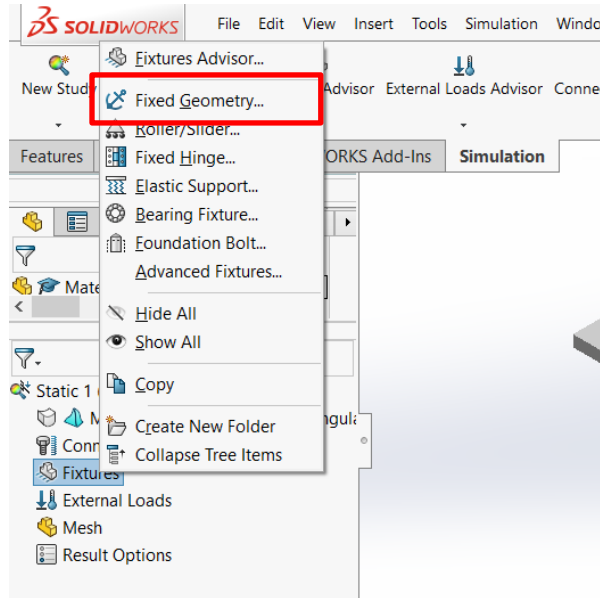
Figure 10: Adding a fixture.

When you click "Fixed Geometry," the Fixture pane opens on the left. Select the right vertical face of the beam (furthest from the hole) and press the green check mark. This is seen in Figure 11. Setting this face of the beam as fixed simulates what we have done in the hands-on portion of this lab where this end of the beam was clamped down and held in place on the beam bending apparatus. You can select multiple faces at a time, if you wish, but for this example only one face is fixed.
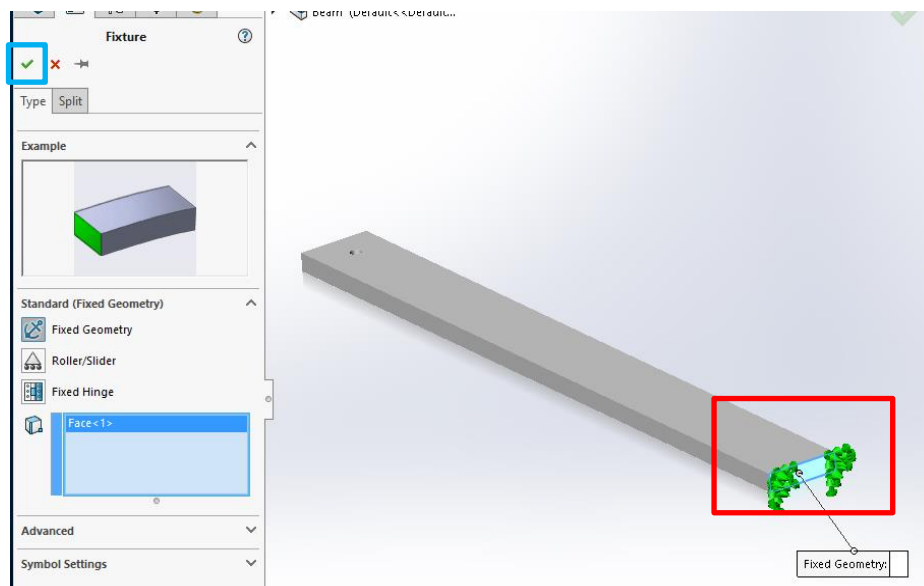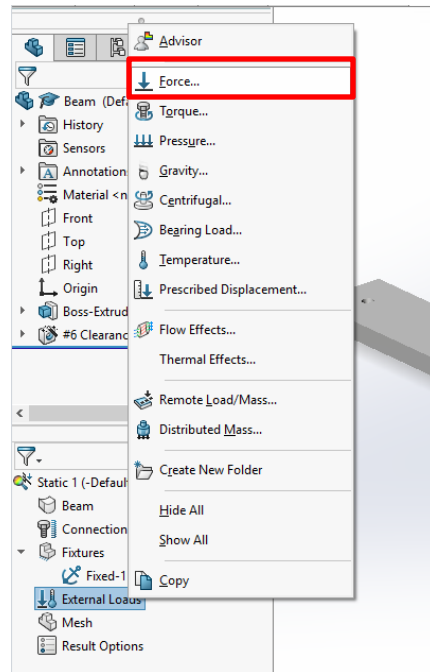

Figure 11: Selecting a fixture.

Specifying External Loads

To specify the weight load, right-click "External Loads" in the static study pane. Click the option for Force, as seen in Figure 12.

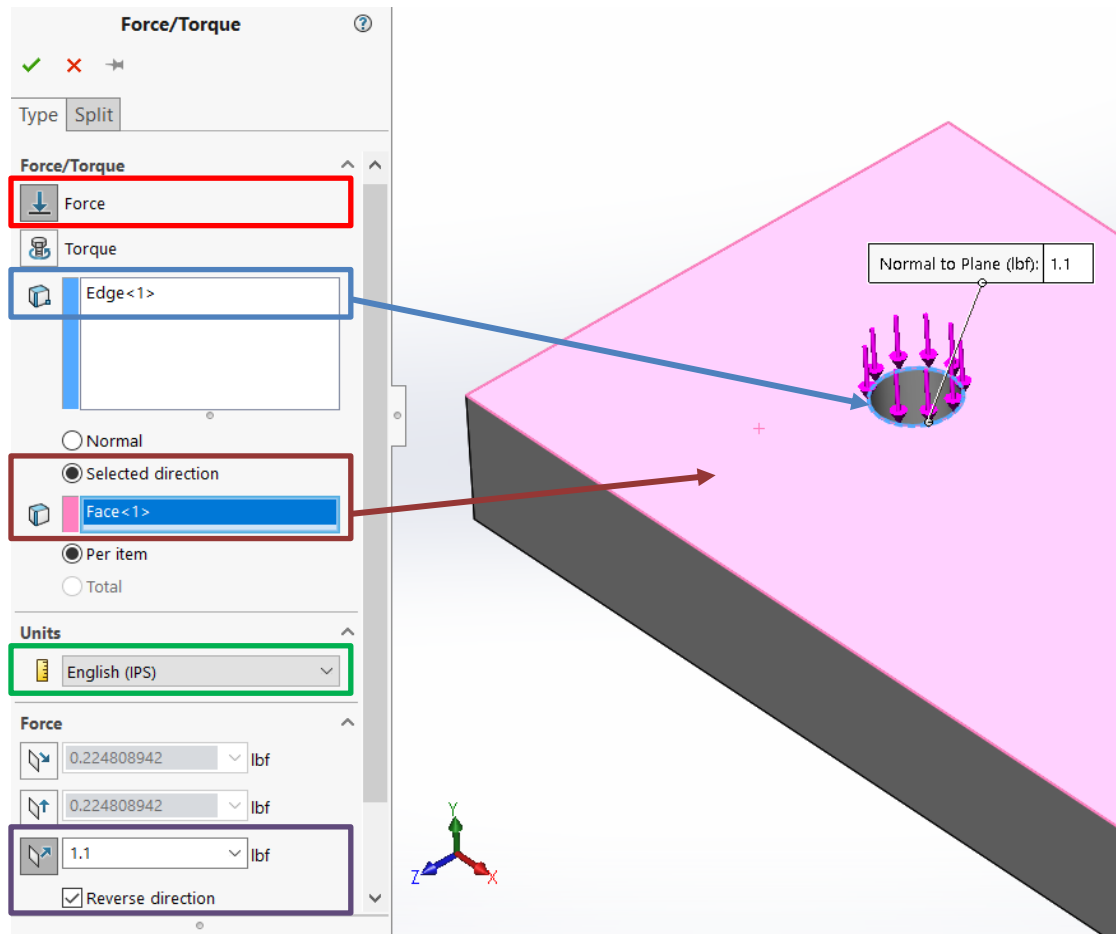**Figure 12:** Opening the Force/Torque Pane.

Once you click on "Force," the Force/Torque pane opens, as seen in Figure 13 on the next page. There are several settings needed to properly define the applied force on the beam.

First, click the top edge of the hole in the beam (you may have to zoom in using the mouse wheel to make sure you are selecting only the top edge rather than the 'face' of the hole) to select it as the location on the beam being loaded. Your Force/Torque pane should show 'Edge <1>' as the selection and you should see purple arrows around the hole, as shown in Figure 13.

Next, choose the 'Selected Direction' option. We need a reference face/edge/plane to establish the direction we want to apply the force. Select the top face of the beam as the reference by clicking on the top surface of the beam. The selection 'Face <1>' should appear to the right of the pink box as shown in Figure 13.

Once the reference frame is selected, we must define the magnitude and direction of our force. Under "Units", use the drop down menu to select 'English (IPS)'. Then, select the **third option** shown in the Force pane, labelled "Normal to Plane", shown below in Figure 13. Notice that the purple arrows that appear are pointing upwards. To simulate the experiments performed in the hands-on portion of the lab, we want the force to be applied downwards. To accomplish this, check the 'Reverse Direction' box in the Force pane.

Change the value of the force to 1.1lbf. This is equivalent to that experienced by the beam in the physical experiment when loaded with all 10 weights. Verify that your window looks the same as Figure 13 below, and then click the green check mark to add this force to the study.

Figure 13: Force/Torque Pane with all settings properly defined

Creating a Mesh

With the forces and fixtures specified, we are nearly ready to run the simulation. SolidWorks first needs to break the model into many small tetrahedral units. The boundaries of these units are created by a network of nodes created around the object which together are called a *mesh*. Smaller meshes (as in meshes with smaller individual units and more units overall) produce more precise results but require additional computing time. Large element meshes run quickly but may produce inaccurate results, especially around sharp edges and small features. It is common to use a mesh with varying element sizes: smaller units around the areas of interest in a model, such as potential failure points, and larger units where precise results are less valuable.

In the static study pane, right-click "Mesh" and choose "Create Mesh" (see Figure 14).
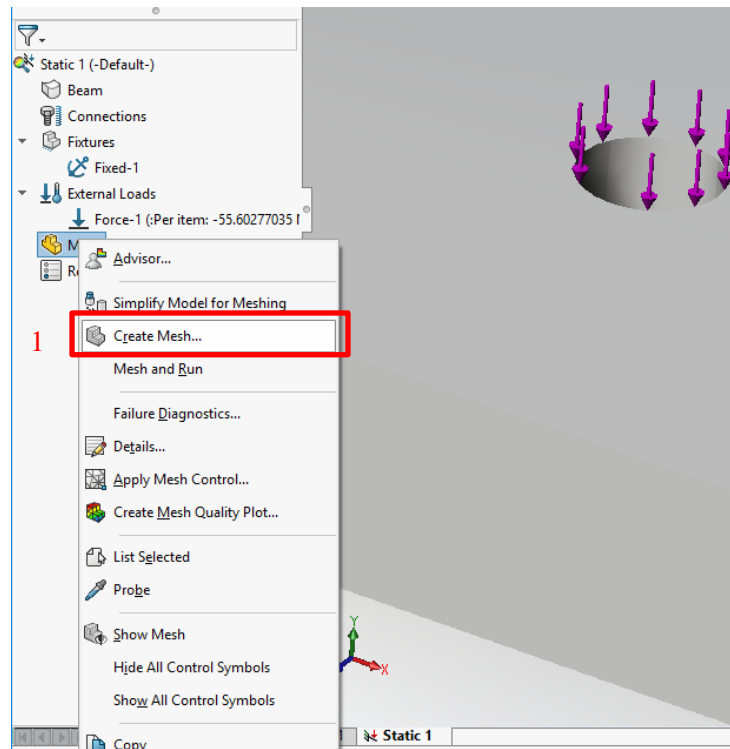
Figure 14: Create a mesh.

In the Mesh pane that appears, there are a number of settings available to adjust the characteristics of the mesh. For this study, we will accept the default mesh settings by clicking the green check mark.

This will create a uniformly sized mesh over your entire model, which should look like Figure 15, below. **Capture a screenshot of your simulation setup similar to that shown in Figure 15 and paste it in the corresponding location in the Lab Worksheet.**
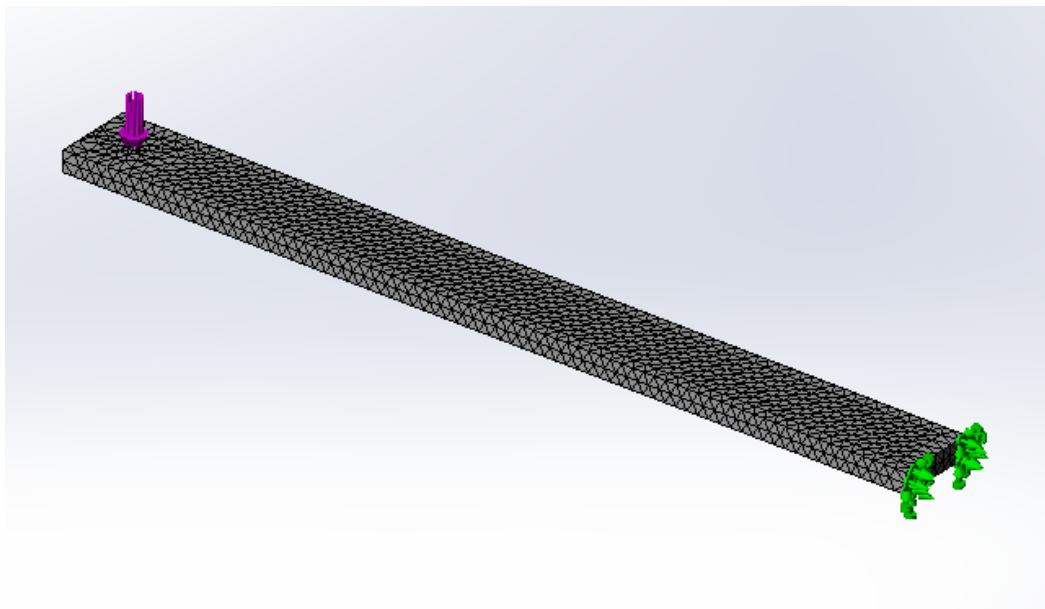
Figure 15: Meshed beam.

Choosing a Material

Lastly, we need to define what material the beam is made of so that SolidWorks will be able to perform calculations using the correct material properties. To apply a material to the beam, right click on "Rectangular_Beam" and select "Apply/Edit Material" as shown in Figure 16.
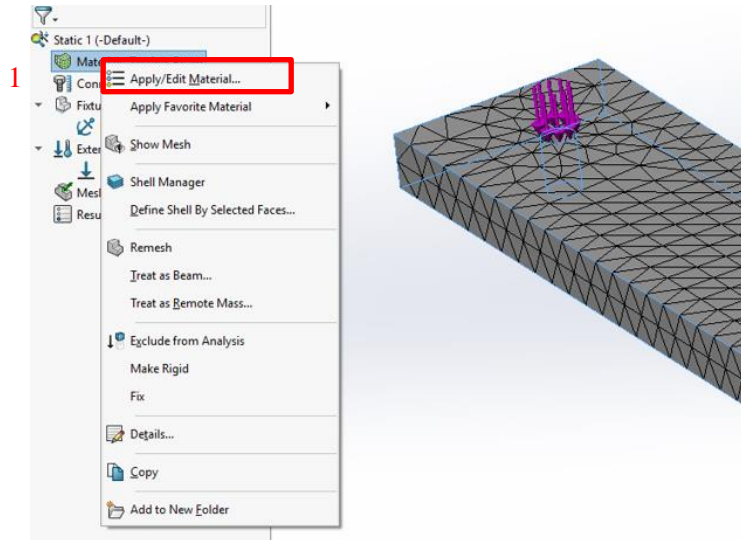

Figure 16: Apply material to beam.

The "Material" window will open, which contains many different materials for you to choose from. For this first simulation, we will choose our beam to made of Cast Alloy Steel, which is found in the 'Steel' folder under 'SolidWorks Materials'. After selecting Cast Alloy Steel, be sure you change the units to 'English (IPS)' so the units match those calculated during the lab. Click on 'Apply' to apply these material properties to our beam as shown in Figure 17. **Note the "Elastic Modulus" is provided in the list of properties in this window.** Close this window once you have applied the material.
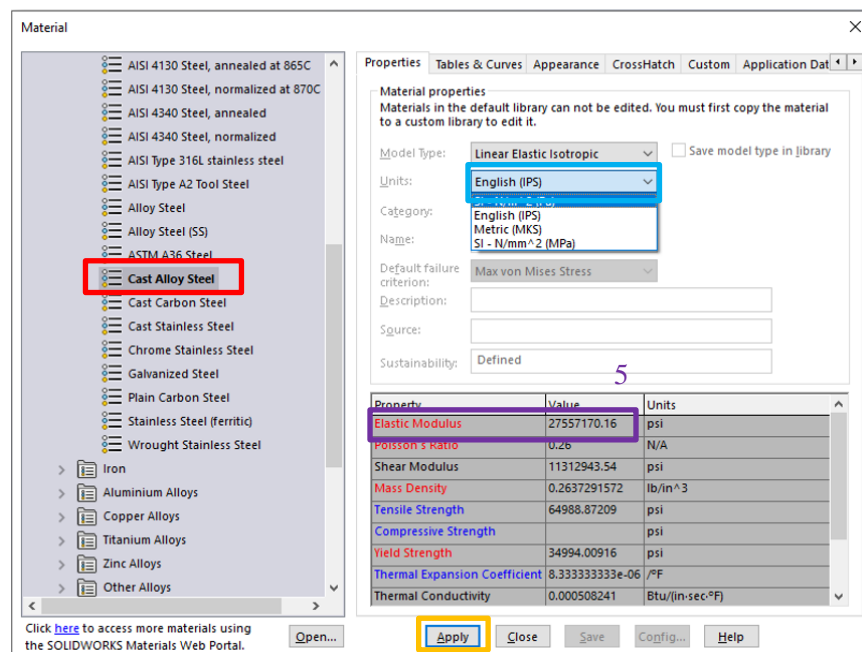

Figure 17: Select 'Cast Alloy Steel' from Material selection window.

Running the Simulation

We are now ready to run the simulation of our beam. Begin the static study by clicking "Run This Study" in the Simulation tab (Figure 18). Wait until the simulation runs to completion – for more complex models & boundary conditions, it will take longer for the simulation to make all the necessary calculations.
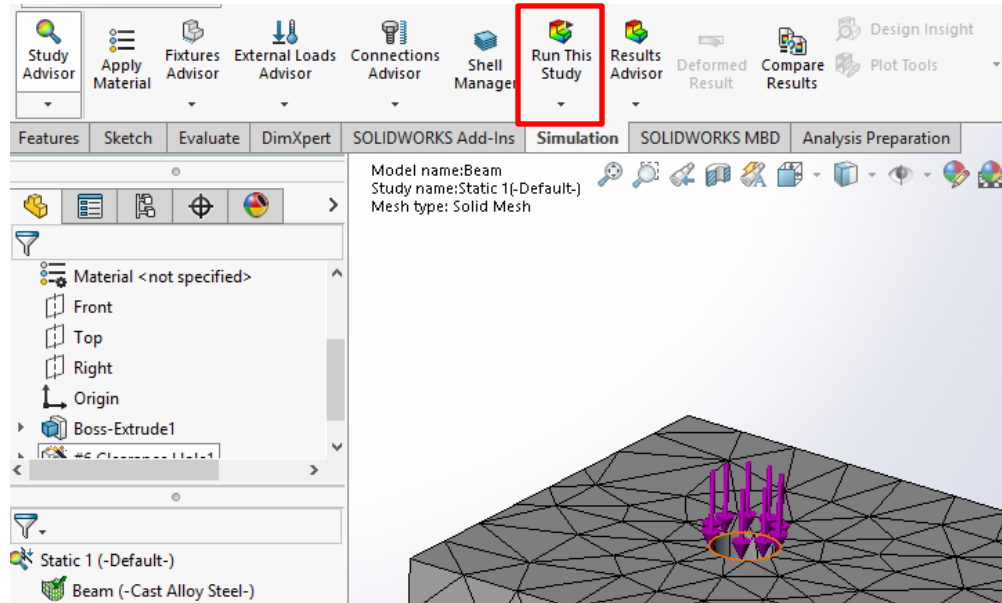

Figure 18: Run the Simulation.

The Results

When the simulation is complete, a new folder will appear in your Study pane named "Results." Under this pane, double-click "Displacement1" (show in Figure 19) to view the displacement experienced by your beam as calculated by our simulation.
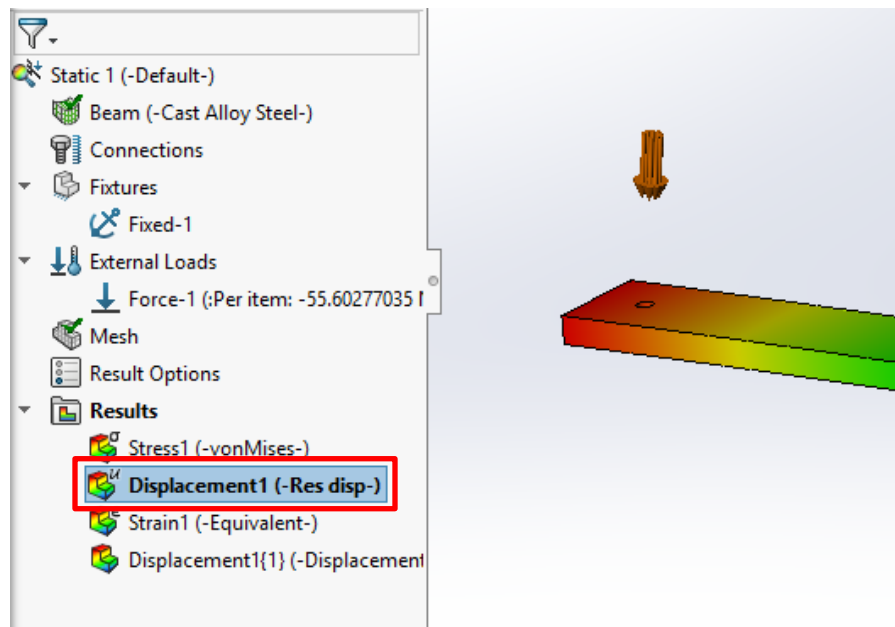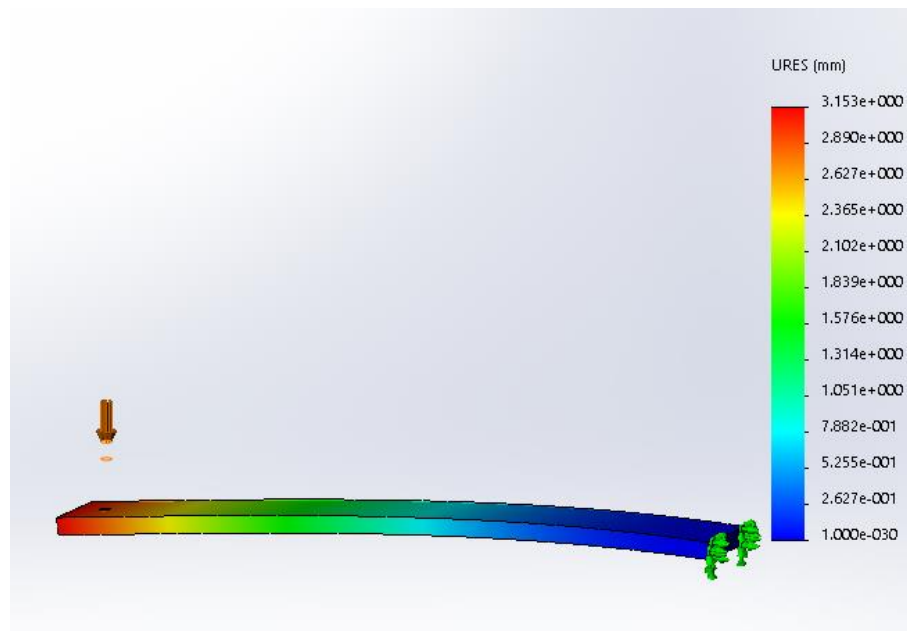

Figure 19: Double-click "Displacement1" under the "Results" folder to view beam displacement.

Your results should look something like Figure 20, below.

**Figure 20:** Results from Simulation.

NOTE: The actual displacement caused by the 1.1 lbf load is smaller than what appears visually on this model. This is because the deformation has been exaggerated in this model to help visualize the mode of deformation, which may be too small to view on a true scale.

To make deformations look more realistic and change the units of our results, we need to change the displacement settings. Double-click on the text box in the upper left corner (Figure 21) to open the "Displacement Plot" pane.
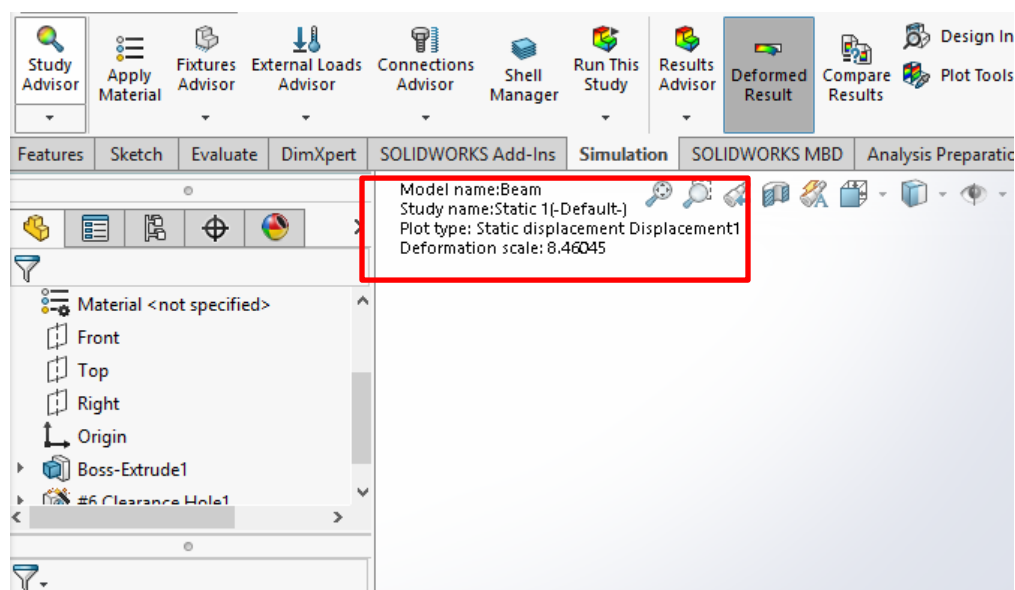

Figure 21: Open displacement plot settings by double-clicking in the text box.

In the "Displacement Plot" pane, under "Display", set the component to "UY: Y Displacement" (as we are only interested in the displacement of the beam in the Y direction) and change the units to **inches**. Under "Deformed Shape", set the scale to True Scale. Double-check that your changes match those shown in Figure 22, then click the green checkmark to apply these changes to your displacement plot.
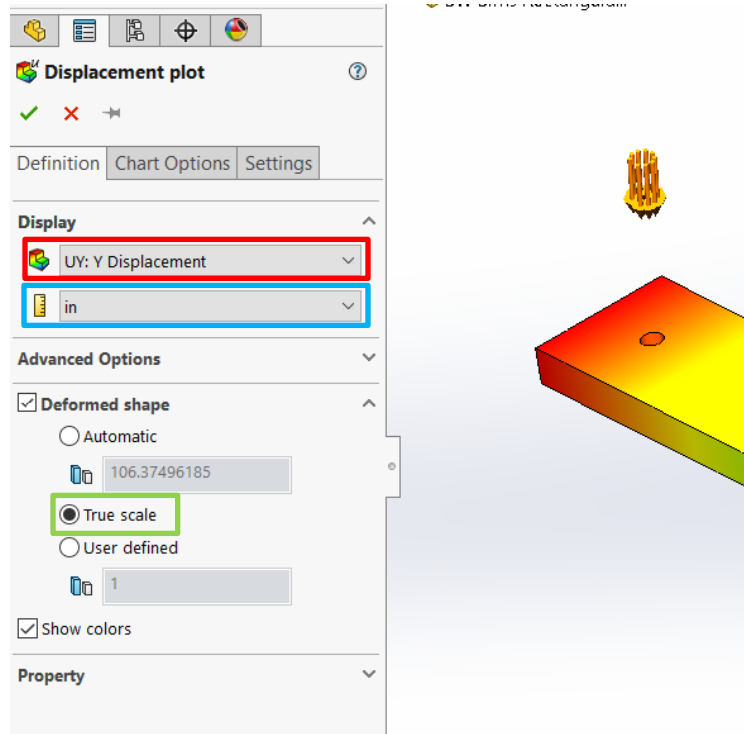
Figure 22: Set Deformed Shape to True Scale and Units to Inches in Displacement Plot Pane.

Your results should now look like those shown in Figure 23. The color legend indicates the displacement (abbreviated as "U") of the beam in the Y direction. The maximum displacement in this scale, once "UY" is set as described above, is now displayed at the bottom of your color legend, as the displacement is in the negative y direction. **Capture a screenshot of your simulation like that shown in Figure 23 and paste it in the corresponding location in the Lab Worksheet.**
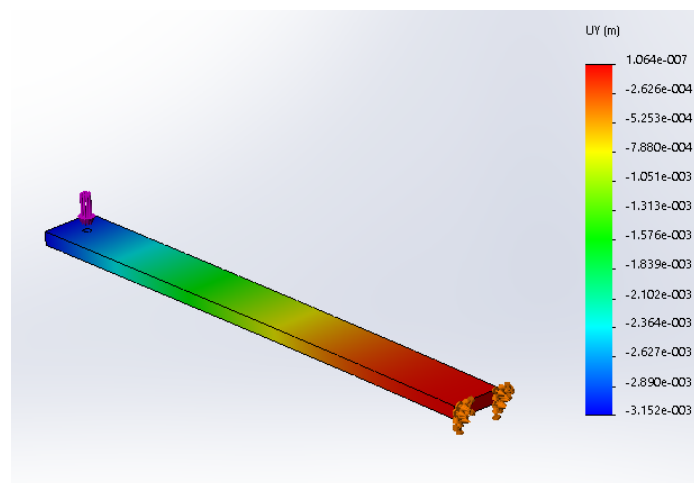


Figure 23: Color legend of resultant displacement.

Determining Beam Displacement

From the legend shown in Figure 23 above, we could easily get the maximum displacement of the beam, which occurs at the very tip of the unfixed end. However, in your hands-on experiments, the measurement for deflection was actually taken at the location of the dial caliper shown in Figure 24 below. For better comparison, we want to find the value of deflection in our simulation at that same position.

**Figure 24:** Physical Beam Bending Apparatus

To find the value at the matching position along the beam, we are going to need to extract and examine the data from the SolidWorks Simulation. Click on the down arrow under "Results Advisor" and select "List Stress, Displacement and Strain" as shown in Figure 25.
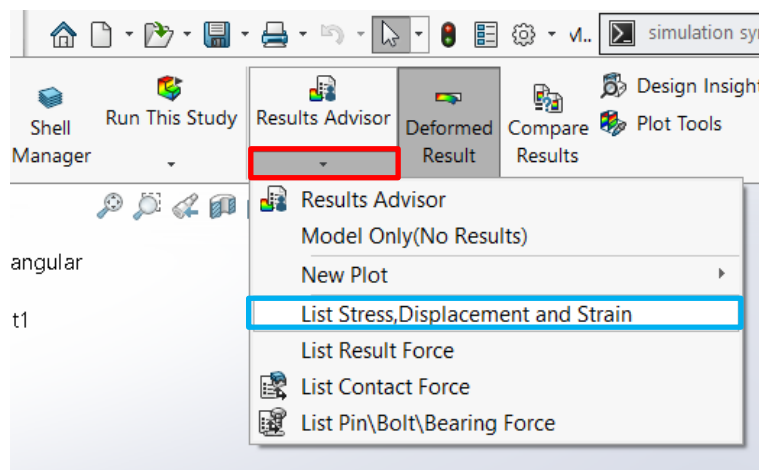
Figure 25: Select "List Stress, Displacement and Strain" under "Results Advisor".

This will bring up the "List Results" pane. Here, change the "Quantity" to Displacement, the "Component" to UY: Y Displacement, and the units to **inches**. Once your selections match those shown in Figure 26, click the green checkmark.
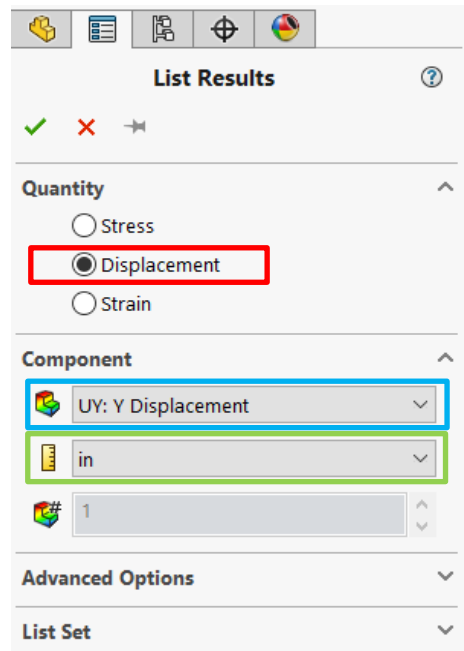
Figure 26: In the "List Results" pane, change the selections as shown above.

You will now see a window titled "List Results" as shown in Figure 27 below. The two columns we are concerned with, highlighted with red and blue boxes below, are **"X (in)"** and **"UY (in)"**. The "X (in)" column will give us the position along the beam in the x direction in inches, and "UY (in)" will show the displacement in the y direction at the node specified by the (X, Y, Z) position.

NOTE: You will see multiple entries for each X position. This is due to the mesh that we made in order to discretize our beam into many finite elements and run the simulation. The beam has been broken up into many tetrahedral units that are bounded by nodes, and the mesh created for our model beam has more than one node with the same initial X position. The SolidWorks simulation keeps track of the position of these nodes before and after the force is applied to determine the displacement.

Figure 27: The "List Results" pane with columns of interest highlighted

Scroll down through the "X (in)" column to find a node near -7.5 inches. It is helpful to sort the "X (in)" column by clicking on the column header. We are interested in the displacement at this position along our beam as this is the distance to where our dial indicator measured displacement in the hands-on lab (see Figure 28). Determining displacement at this position provides us with a direct comparison to those results.
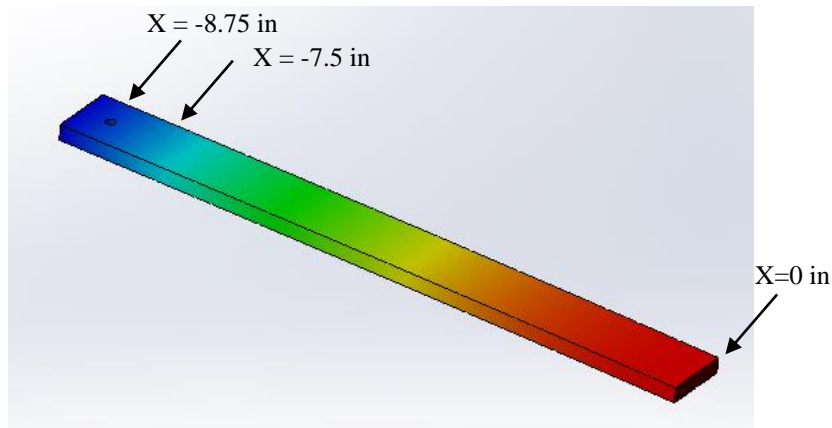
Figure 28: Depiction of our beam's orientation in the SolidWorks workspace demonstrating why we are looking for nodes at x = -7.5 inches.

Select a node at this position ("X (in)" ≈ -7.5 in) as showing in Figure 29 below and **record your displacement ("UY (in)") in Table 1 of the Lab Worksheet**. NOTE: "UY (in)" values will appear negative in the List Results Pane, since the beam is being deflected in the downward direction. We are only interested in the magnitude, so you can disregard the negative sign in your Lab Worksheet.

You should also take note of the node number; this can be used to quickly find your displacement value at the exact same position of the beam in additional simulations for the same solid model and mesh.

Figure 29: Node at "X (in)" ≈ -7.5 in the List Results pane

Once you have recorded your displacement result for the Cast Alloy Steel beam, you will repeat this simulation for two other materials – **Copper** and **6061 Alloy Aluminum**.

Change the material of the beam as we did previously (Right click on 'Rectangular_Beam' → 'Apply/Edit Material', shown again in Figure 30). Select the material you wish to test next, click 'Apply', and then close the window. You can now re-run the simulation for the new material by simply clicking "Run" on the top menu.
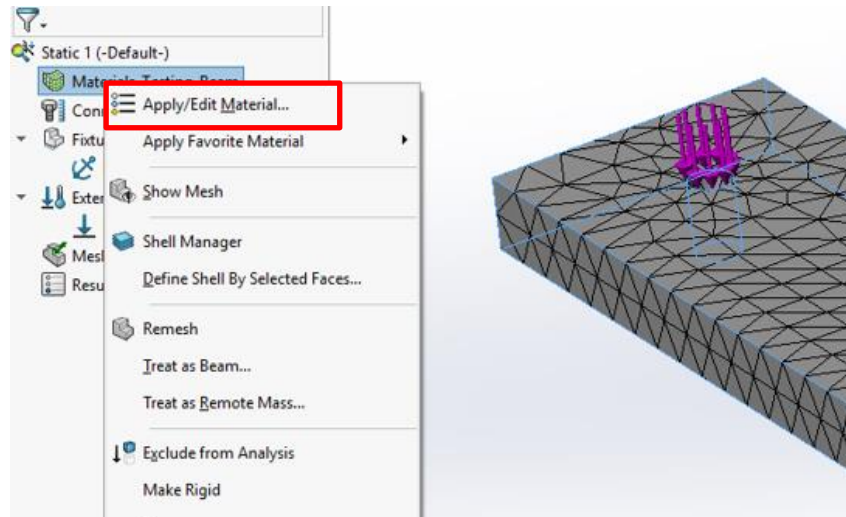
Figure 30: Edit your beam material to run a new simulation.

For copper and aluminum, please select the following material from the SolidWorks materials list:
- Copper (found under "Copper Alloys")
- 6061 Alloy (found under "Aluminum Alloys")
  - NOTE: The simulation may present a warning asking to solve with the "Large displacement flag activated." Simply press "Yes" to utilize this feature and continue the simulation.
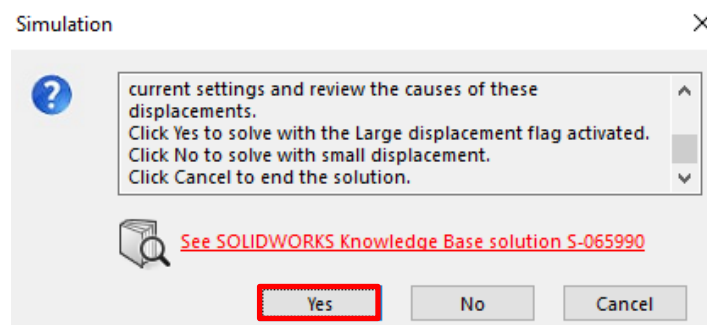
Figure 29: Select the "Large displacement flag activated" feature for the Aluminum alloy.

For each simulation, follow the applicable steps above to find and record the UY displacement value at "X (in)" ≈ -7.5 for each material in your Lab worksheet. You will be using the data from these simulations to answer the remaining questions in the worksheet.