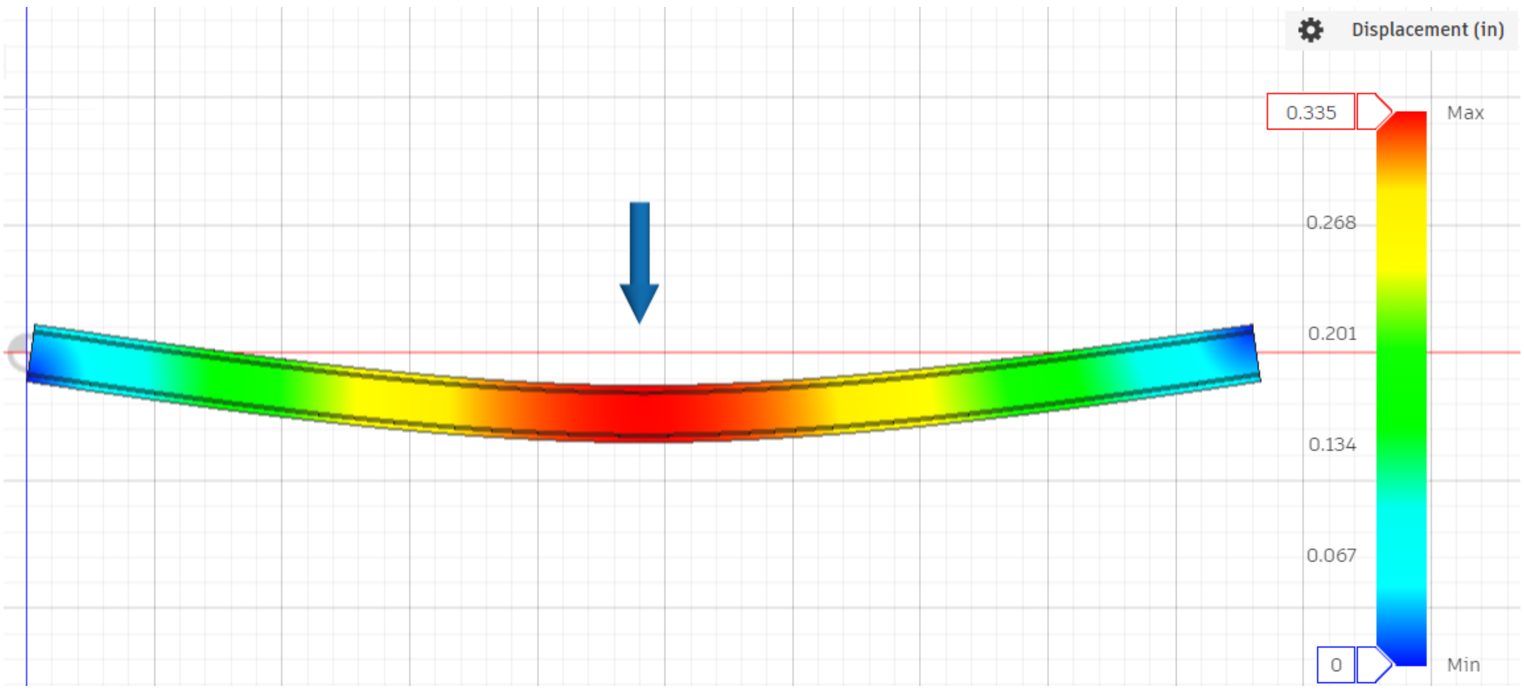
# FEA (Finite Element Analysis) of **Simply Supported Beam** using Fusion

My pretty FEA results should be worth an art credit.



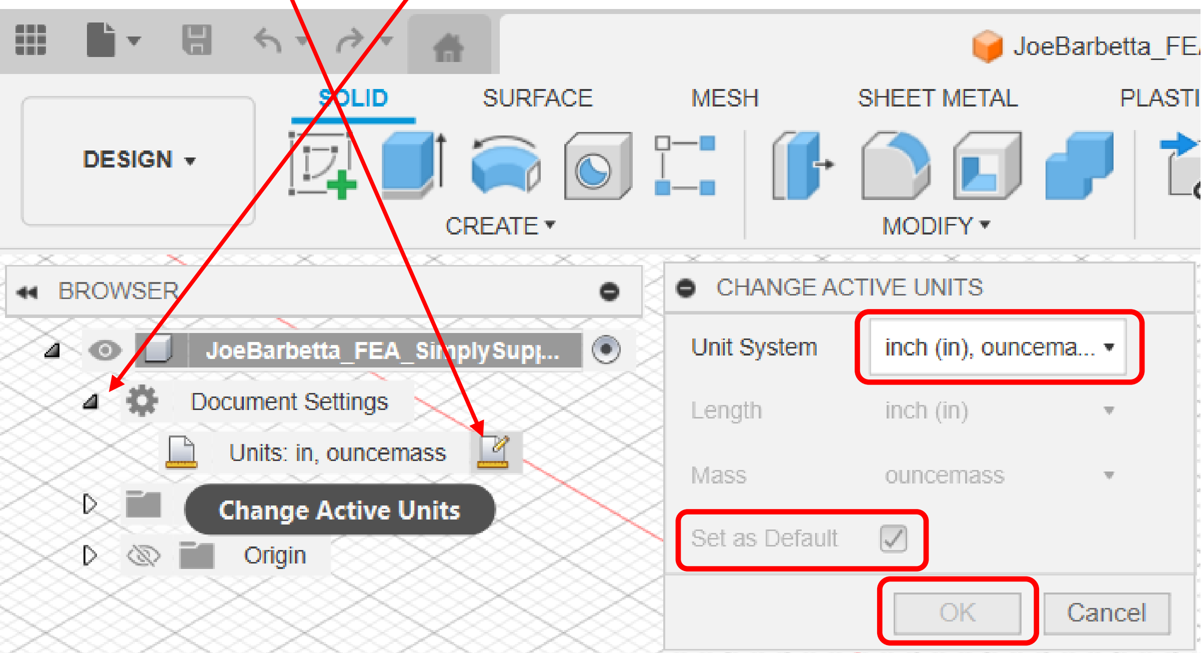
Today’s lesson is sponsored by Diggerland in West Berlin, NJ

# Contents

# Starting a Design in Fusion

- open **Fusion**. If there is no icon on the Desktop, use the Windows search (magnifying glass icon) and type fusion
- from top **File** icon select **Save** and name the file.
  Use your name followed by **_FEA_SimplySupportedBeam** e.g. **JoeBarbetta_FEA_SimplySupportedBeam** (note the use of the underscores)

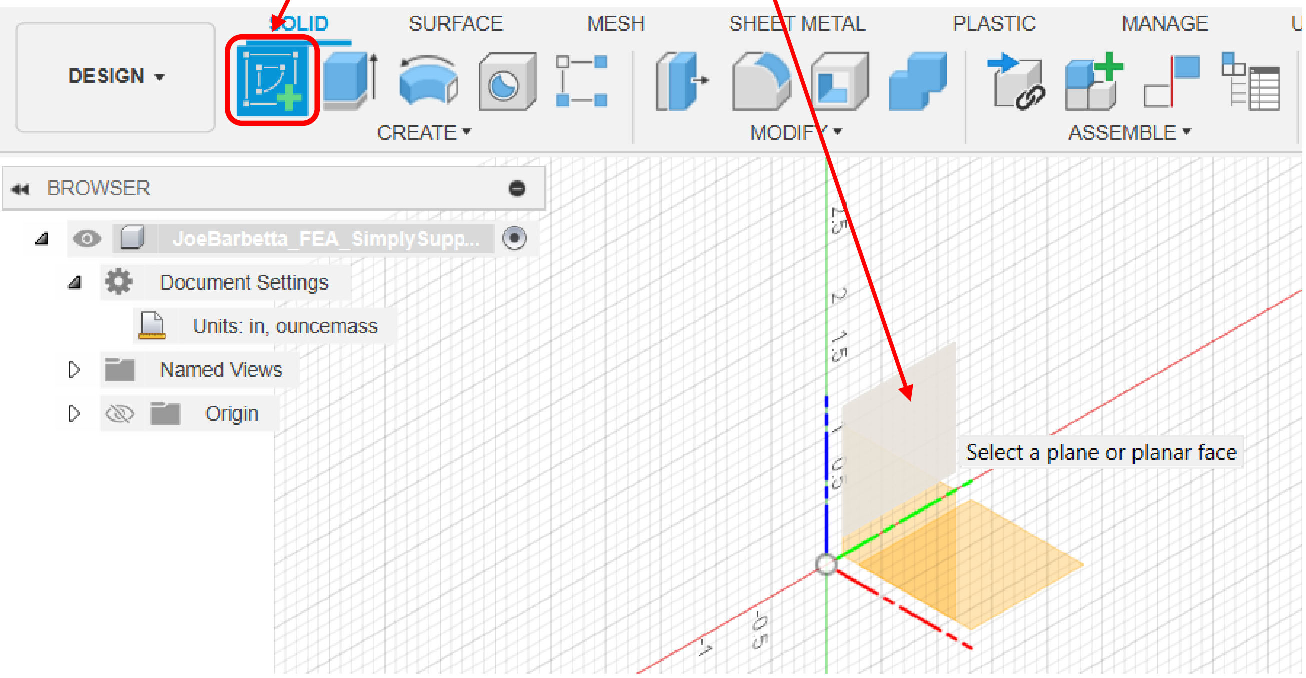
- in the left **"BROWSER"** click the **arrow next to Document Settings**
- click on the **edit icon** that appears to the left when you hover over **Units**
- ensure **Active Units** are set to **Units: in, ouncemass** and click **OK**. You can also enable **Set as Default** if it is not grayed out.



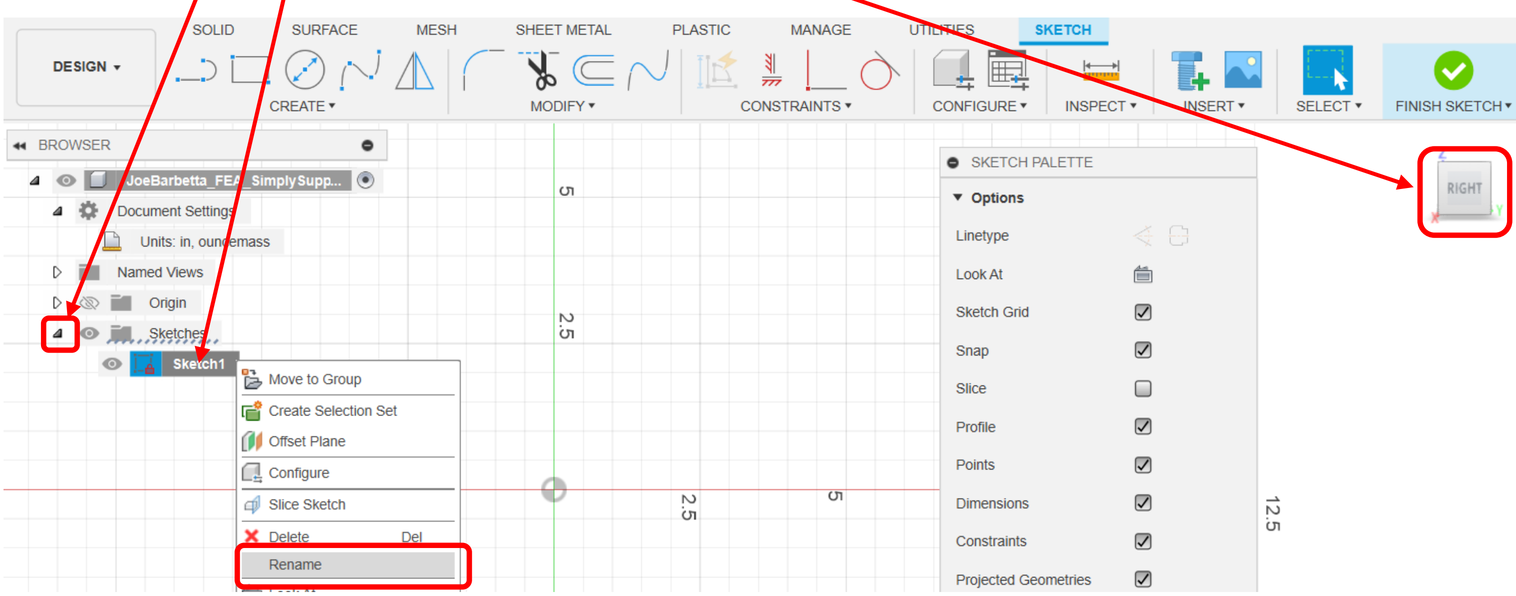
## Creating a Sketch for the Beam Cross-Section

Note that a Fusion expert would suggest starting with the creation of a new Component, but you can say **"Dude, I'm creating a simple beam."**

- select the top **Create Sketch** tool and click on the **top-most rhombus** to select the Y-Z Plane.
If the Create Sketch tool can't be found, find it within the **CREATE** menu.

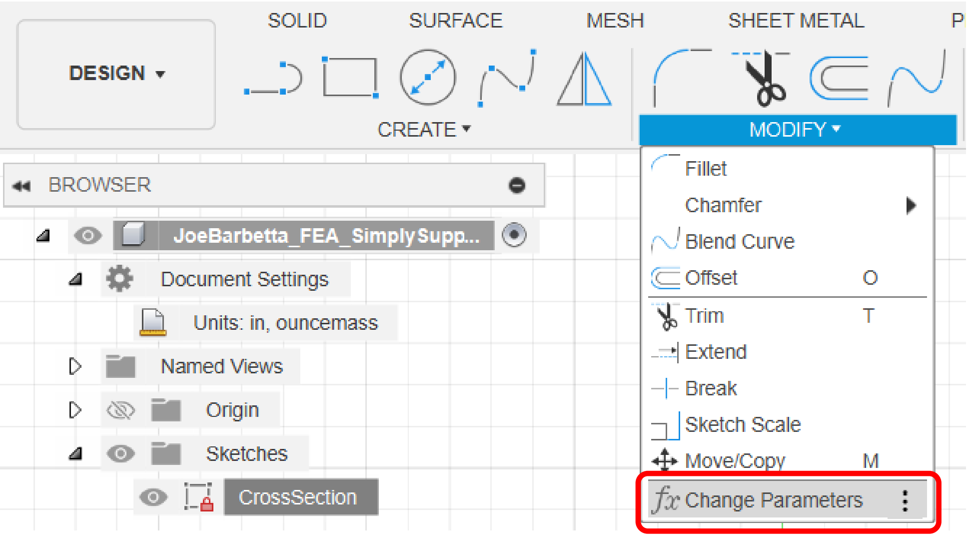



The View Cube text should show "**RIGHT**". If it does not, then redo the Create Sketch and ensure the correct rhombus is selected.
- click on the **arrow** next to the **Sketches** folder to view any sketches
- right-click on the **Sketch** and select **Rename** and enter the text **CrossSection**.

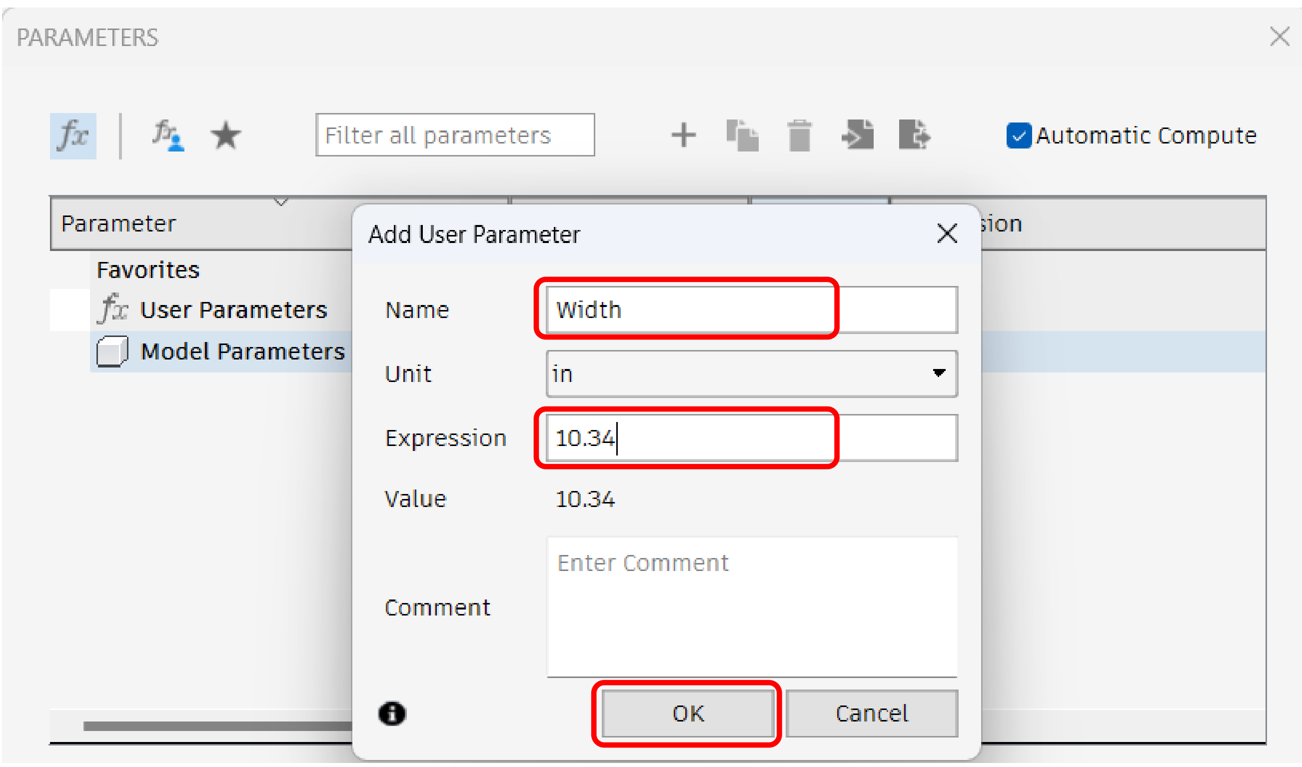


## Setting Parameters

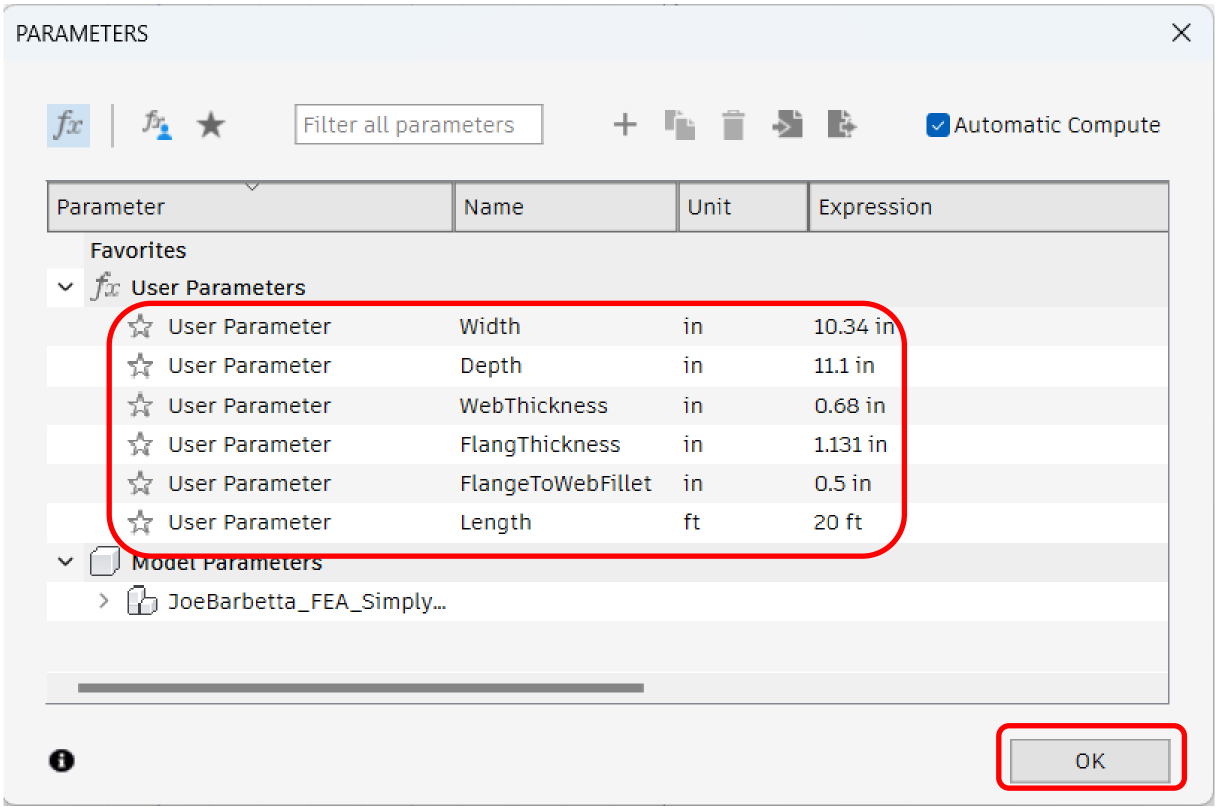
- from the **MODIFY** menu select **Change Parameters**



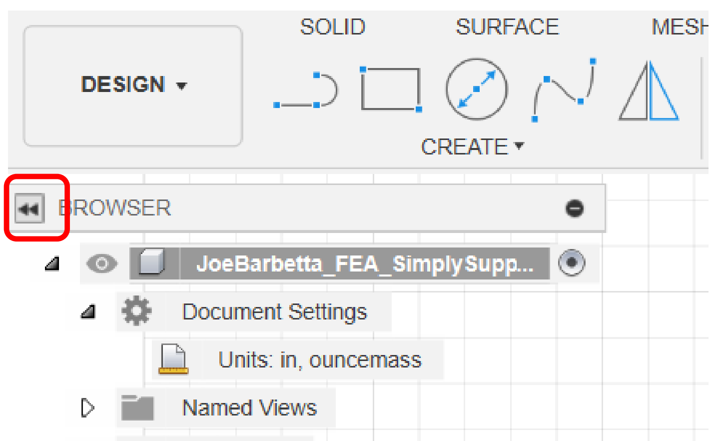

- if a message window about **Parametric Text** shows, click its **OK** button
- click on the **+(plus)** icon to open the Add User Parameter window
- for **Name** enter **Width** and for **Expression** enter **10.34** and click **OK**



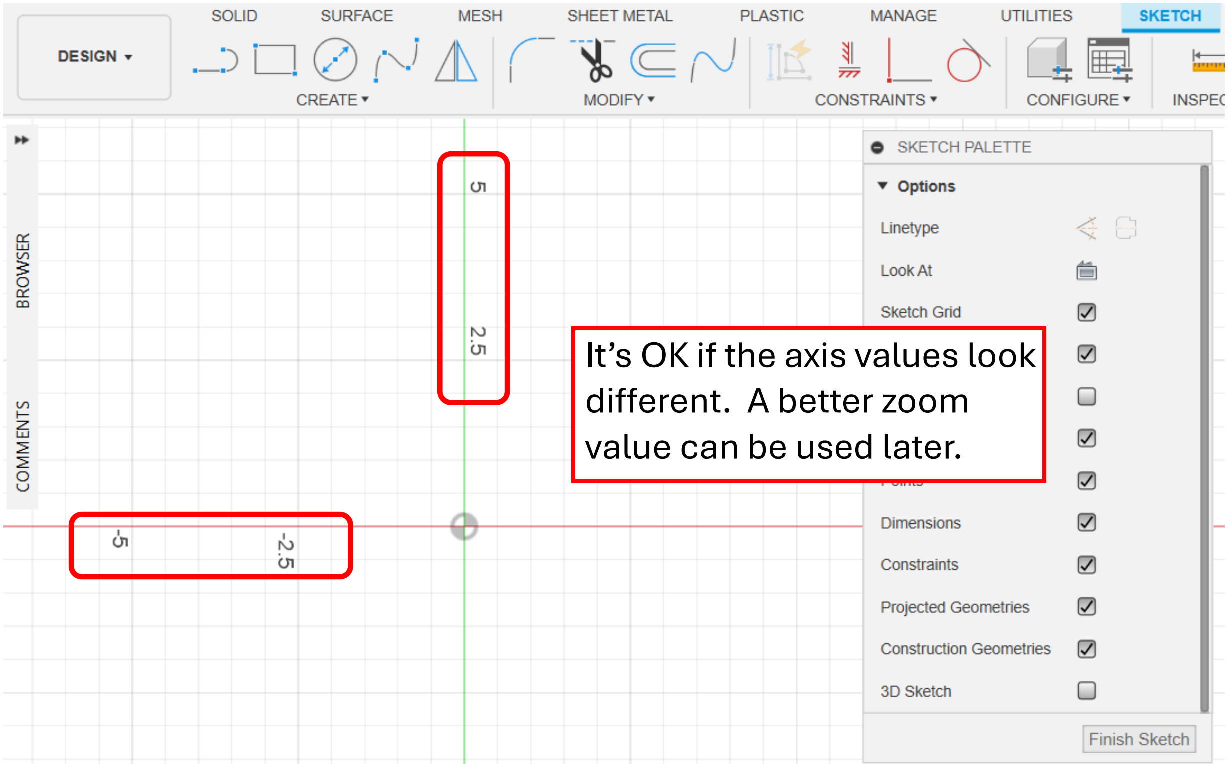
- add additional parameters as shown below and click OK



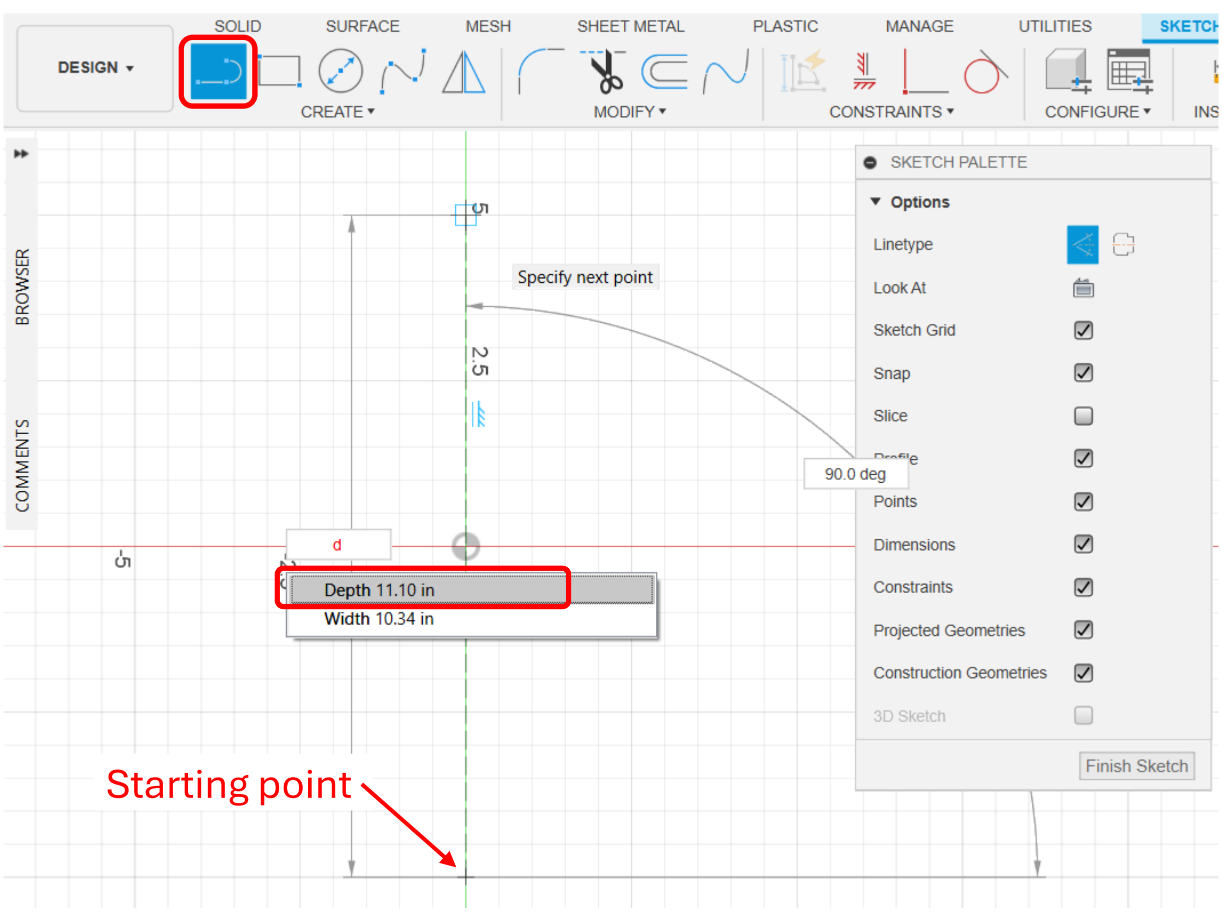

- click on the double arrow to hide the BROWSER. It can later be reopened with the double arrows.



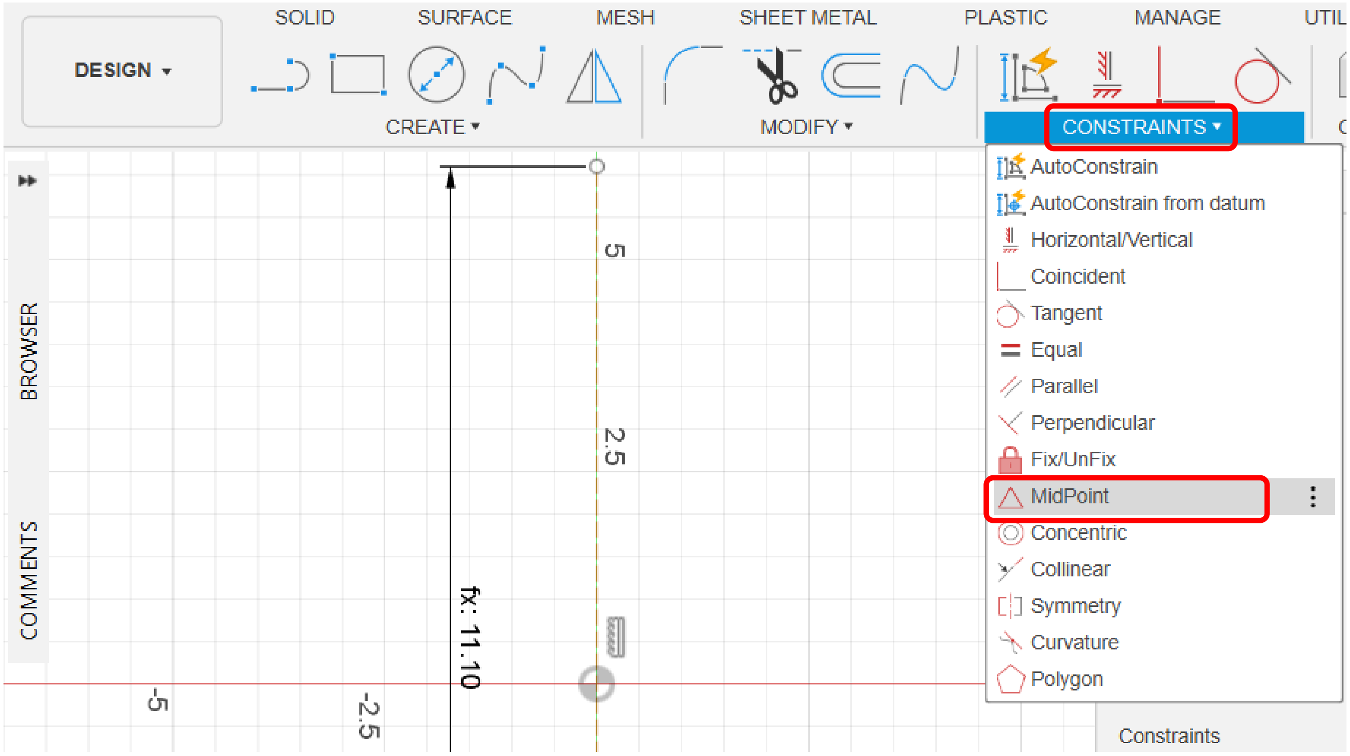
- **zoom** (using the mousewheel) and **pan** (holding the mousewheel down) to achieve a view similar to that below. **Note the axis values of 5.** Our 10 in beam will be centered at the origin so this is a good zoom amount.



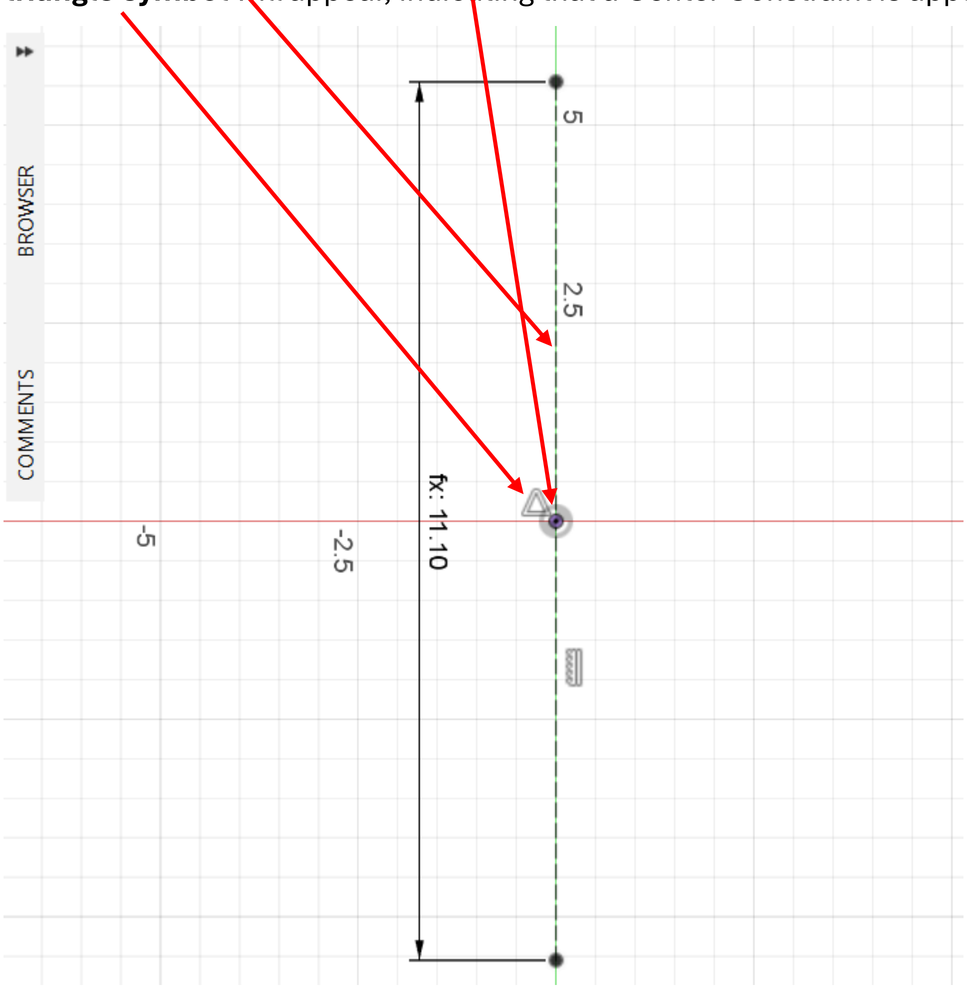

- select the Line tool and click on the **Construction Line** icon for **Linetype** to highlight it blue
- put on your hard hat because we will now create a Construction Line
- click on a point on the **green axis** and about 5 inches below the Origin. The 5 inches is not critical.
- extend the **line upward**, type **d**, select **Depth**, and press the **Enter key**



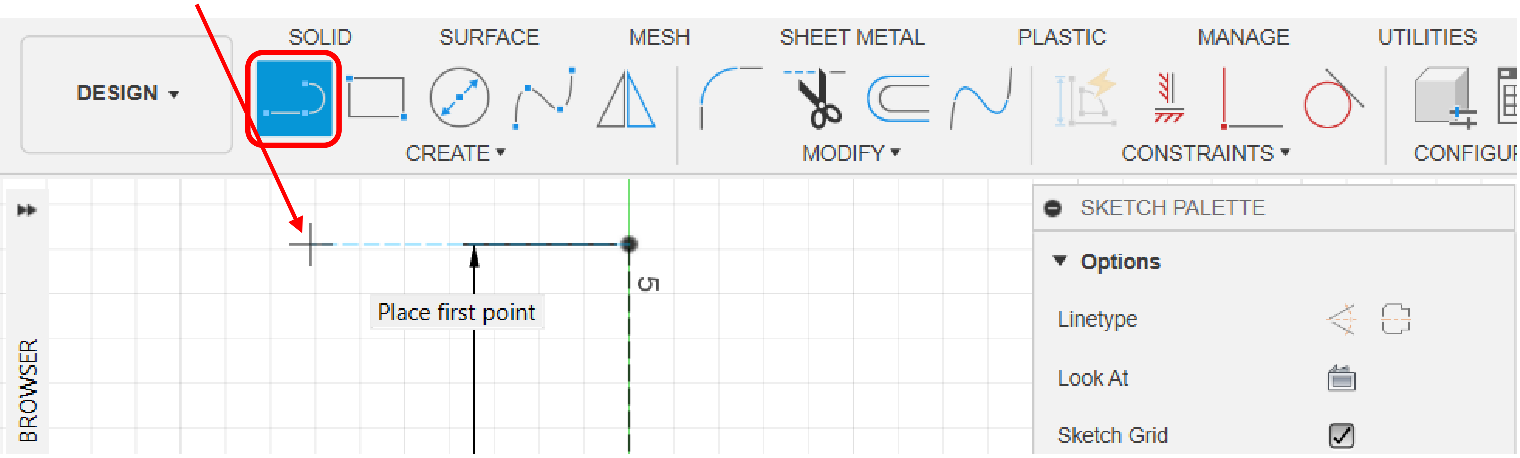
- from the **CONSTRAINTS** menu select **MidPoint**



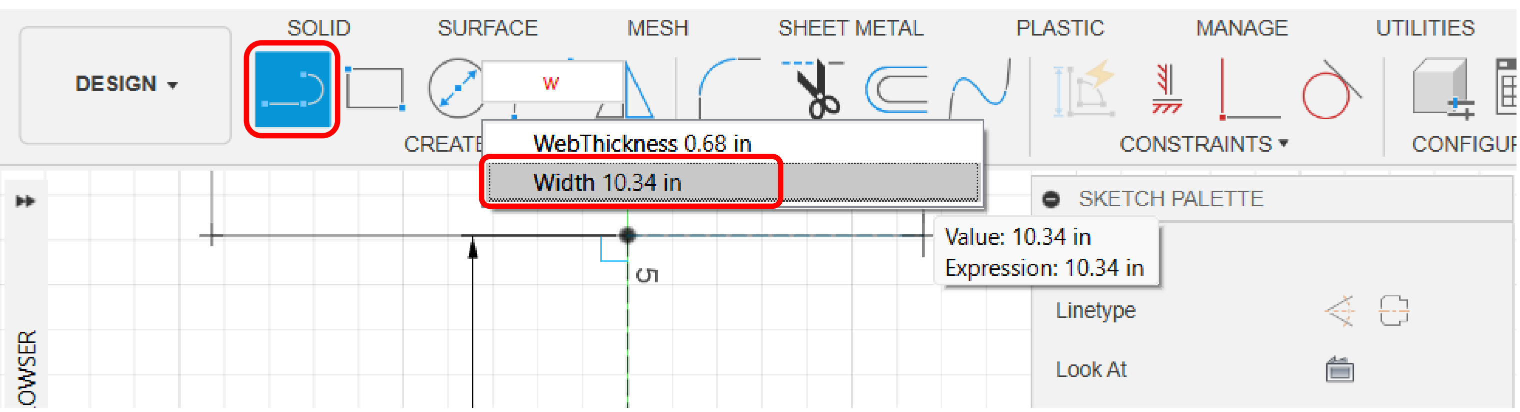

- click on the **line** and then the **Origin**, which should result in the line being centered around the Origin and a **triangle symbol** will appear, indicating that a Center Constraint is applied



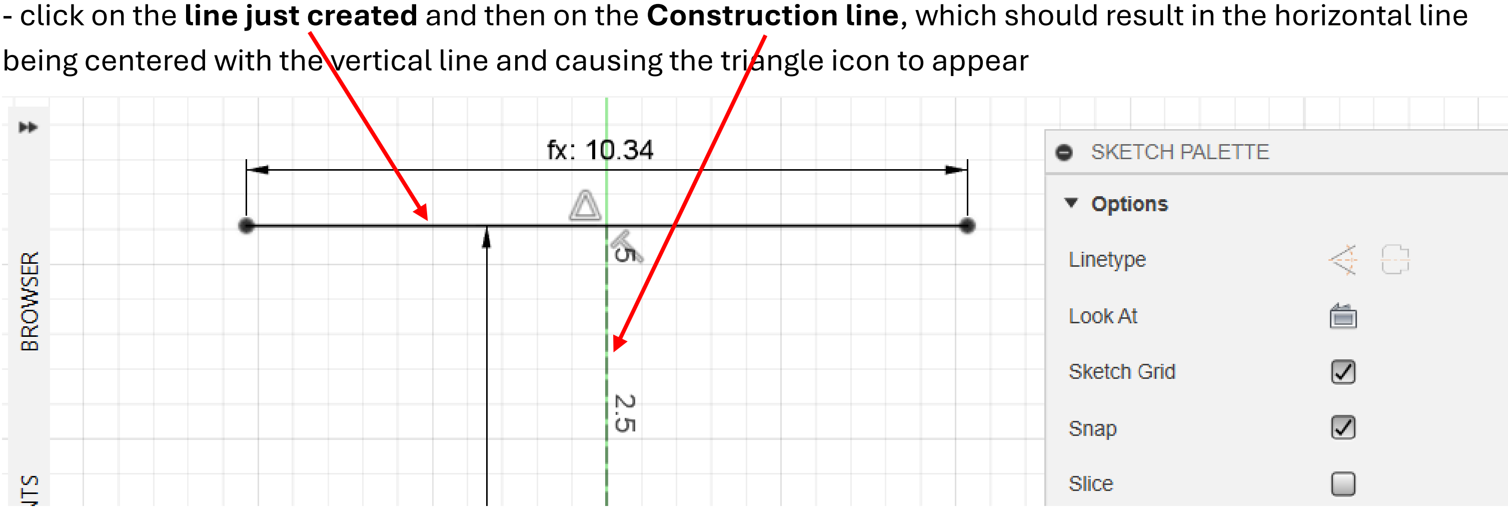
- click on the **Construction Line** icon again to remove the highlighting and remove your hard hat
- select the **Line tool** and before clicking, move the mouse over the **top of the vertical line** and move it to the left, which should show a **dashed blue line**, which indicates that the point is aligned with the top point
- click on a **point about 5 inches to the left**. This distance is not critical.



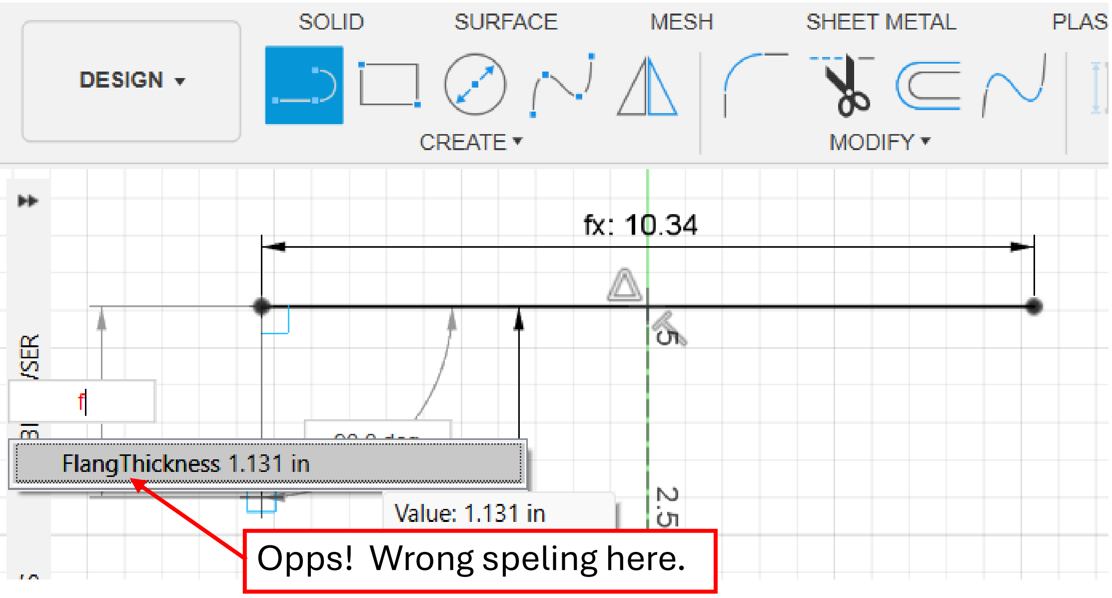

- extend the line **to the right**, type **w**, select **Width**, and press the **Enter key**



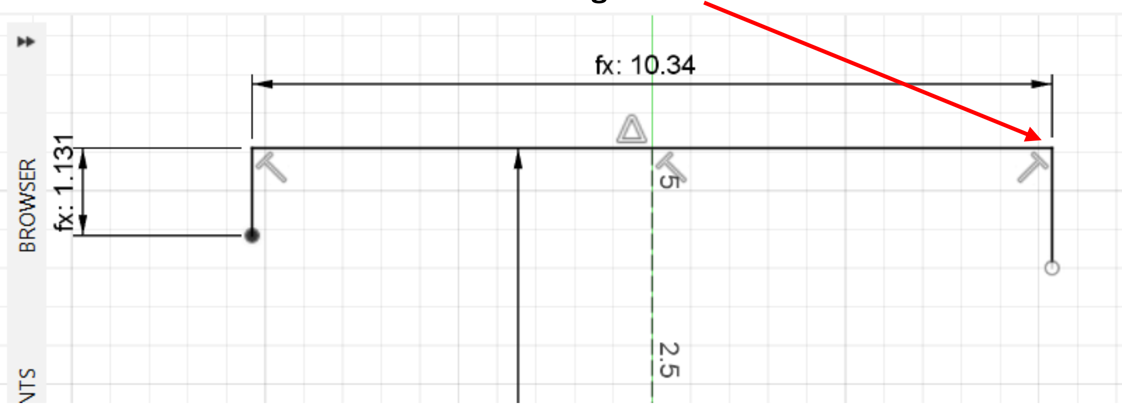

- select **MidPoint** from the **CONTRAINTS** menu again
- click on the **line just created** and then on the **Construction line**, which should result in the horizontal line being centered with the vertical line and causing the triangle icon to appear



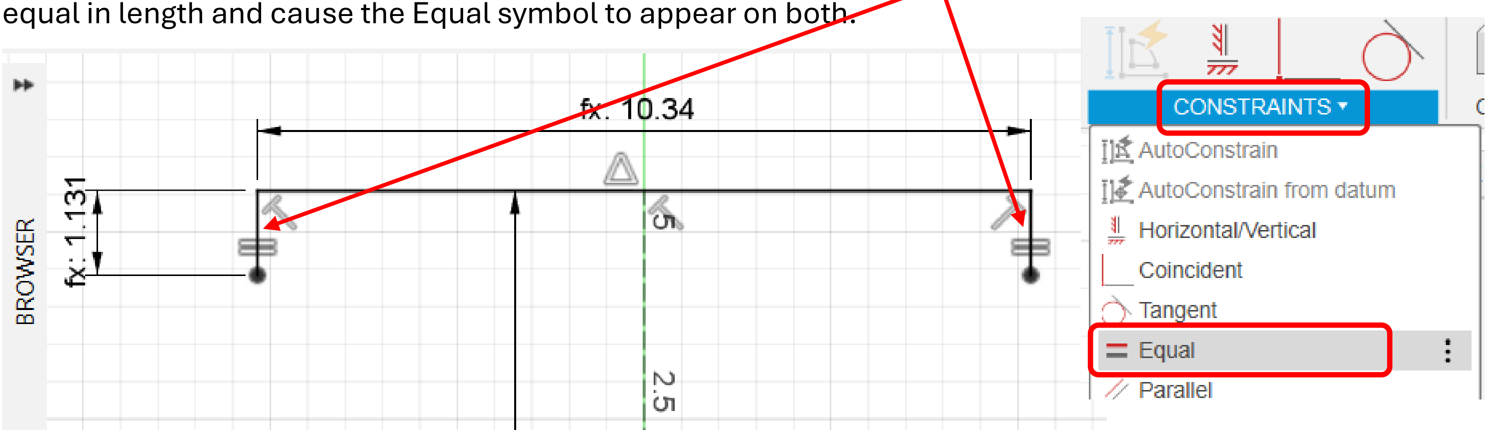
- start a new line at the left end of the top line and extend the line downward
- type **f** and use **FlangeThickness**



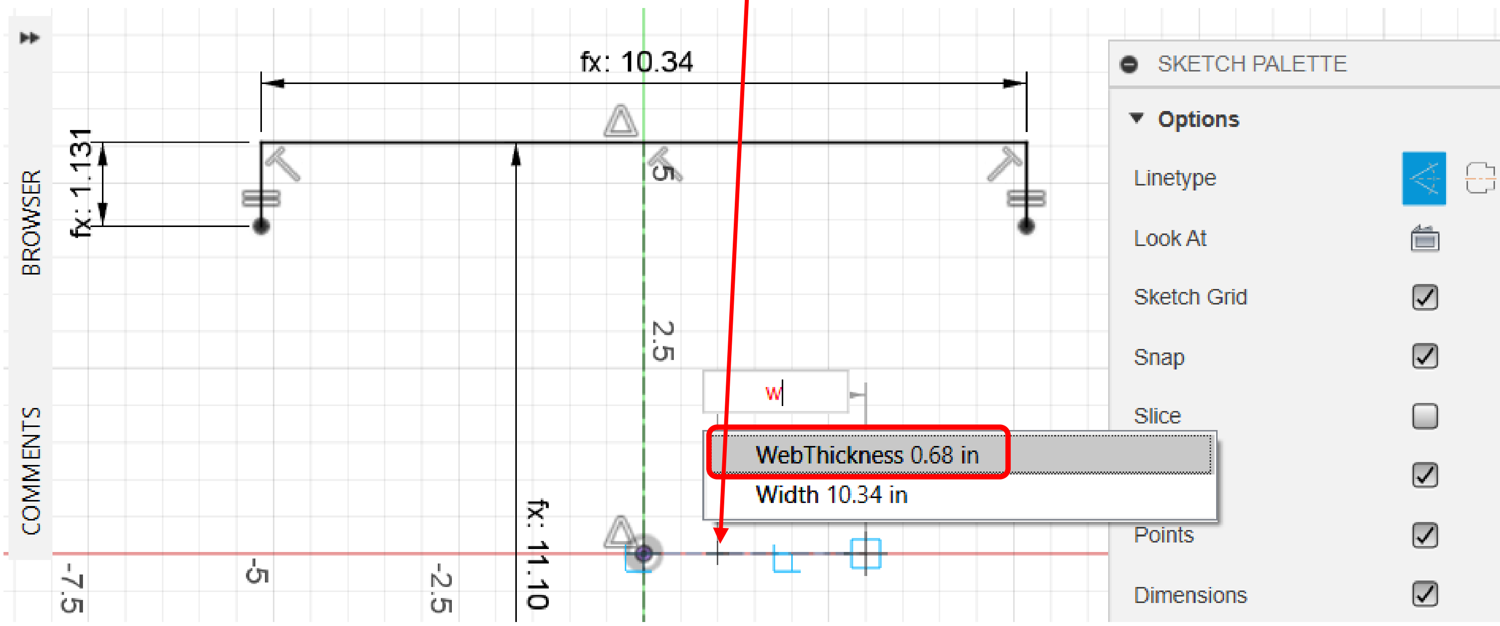

- create another line **downward from the right end of the line** and extend it about an inch downward.

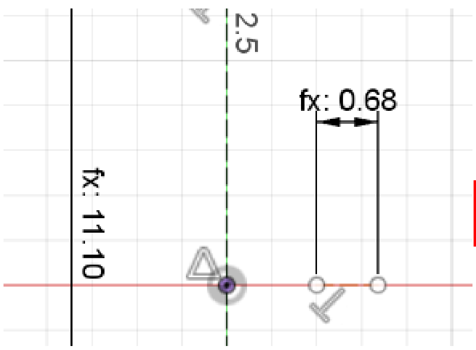



- from the **CONSTRAINTS** menu select **Equal** and click on the **two short lines**, which will make them both equal in length and cause the Equal symbol to appear on both.



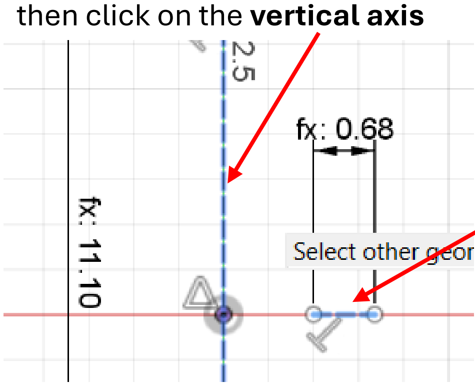
- select the Line tool and click on the **Construction Line** icon for **Linetype** to highlight it blue and put your hard hat back on
- select the **Line** tool and click on a **point on the red axis and about an inch to the right of the Origin**.
- extend the line to the right using **WebThickness**



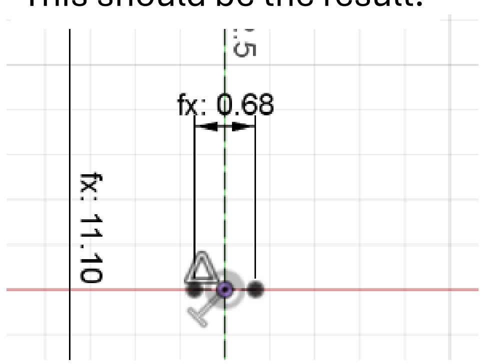




Note that the actual line will barely be visible.

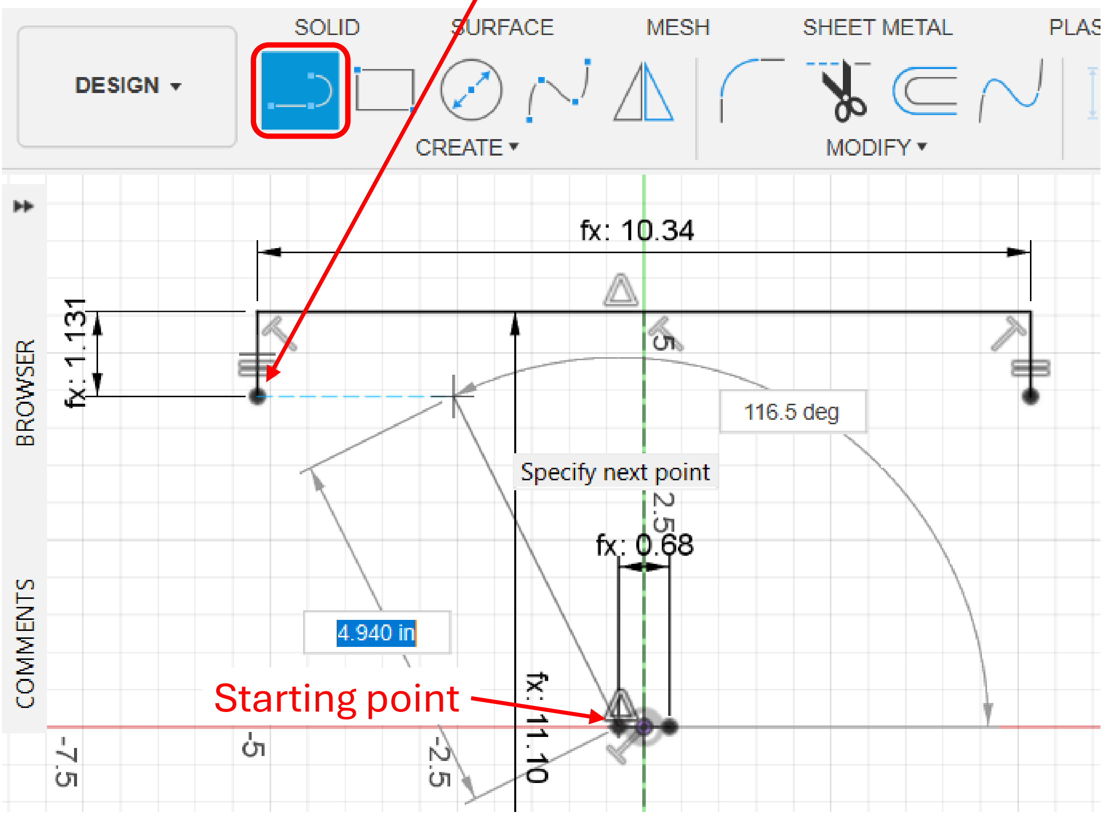
- select **MidPoint** from the **CONTRAINTS** menu and click on the **line just created** and then click on the **vertical axis**



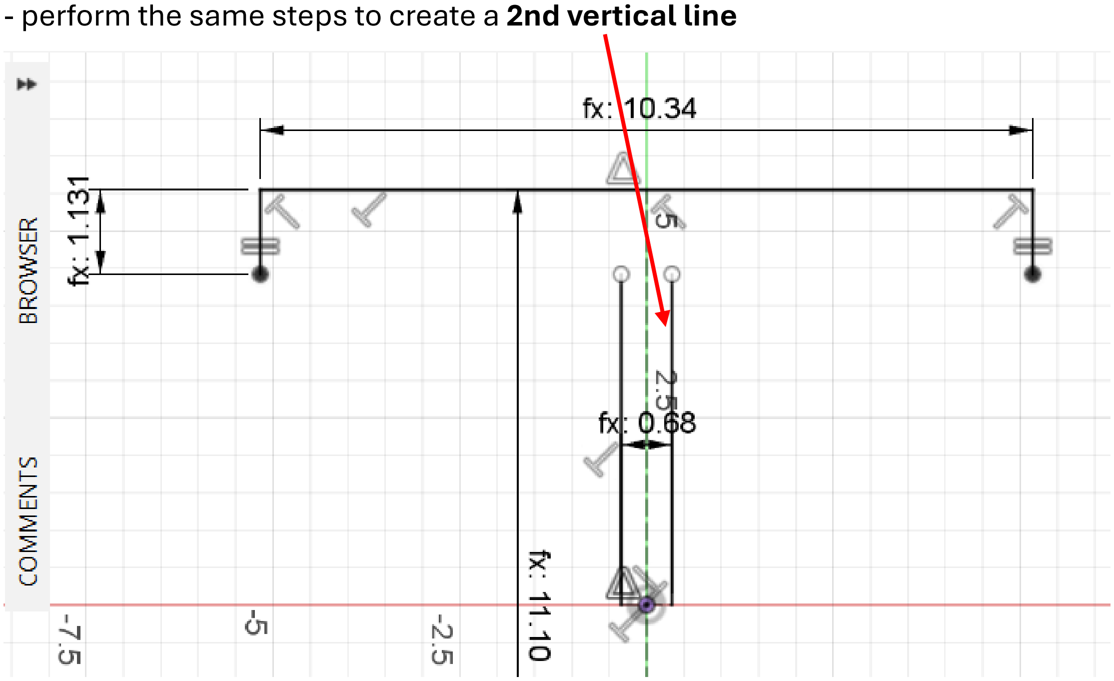

This should be the result.



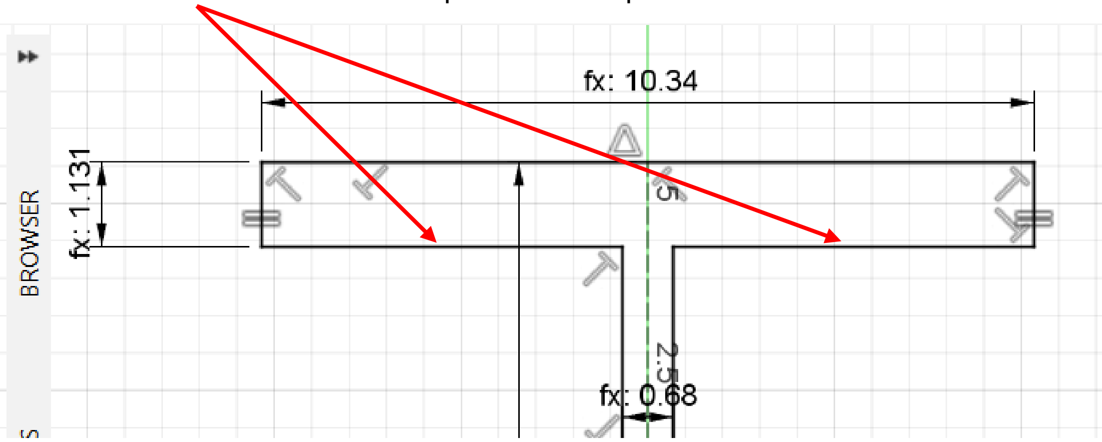
- click on the **Construction Line** icon to remove the highlighting and remove your hard hat
- start a line from the **left end of the Construction line** that was just created
- drag the end upward and over the **end of the left vertical line without clicking** and then drag the line to the right. There should be a dashed blue line extending from that point indicating that the two point should be aligned.
- when the line being drawn is vertical, **click at that point** and **press the Esc key**



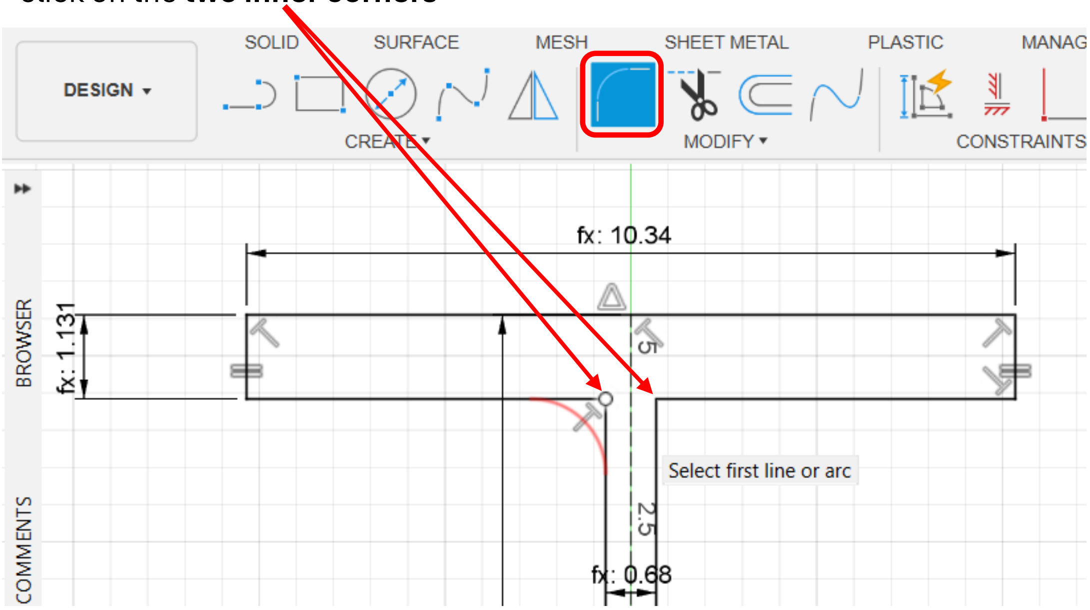

- perform the same steps to create a **2nd vertical line**



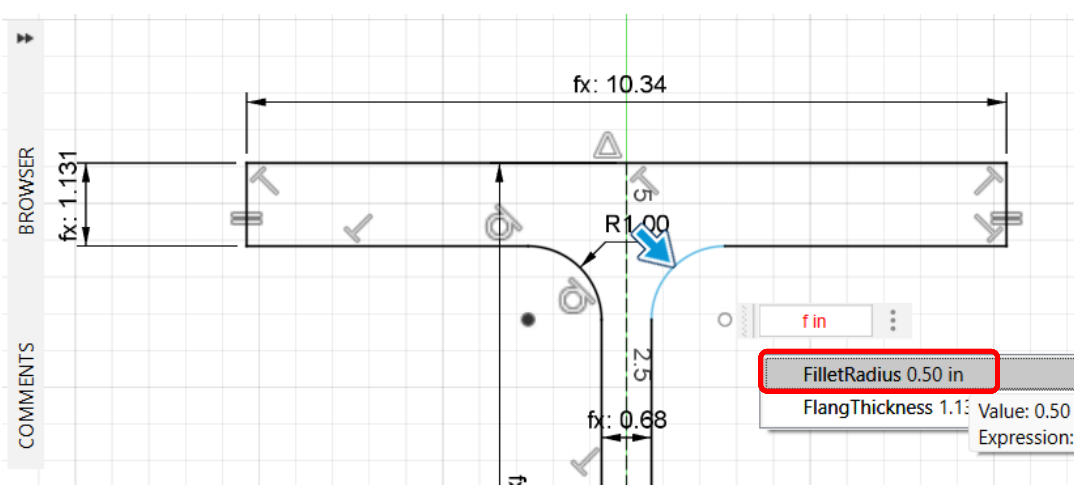
- create **two lines** to close the top of the T shape



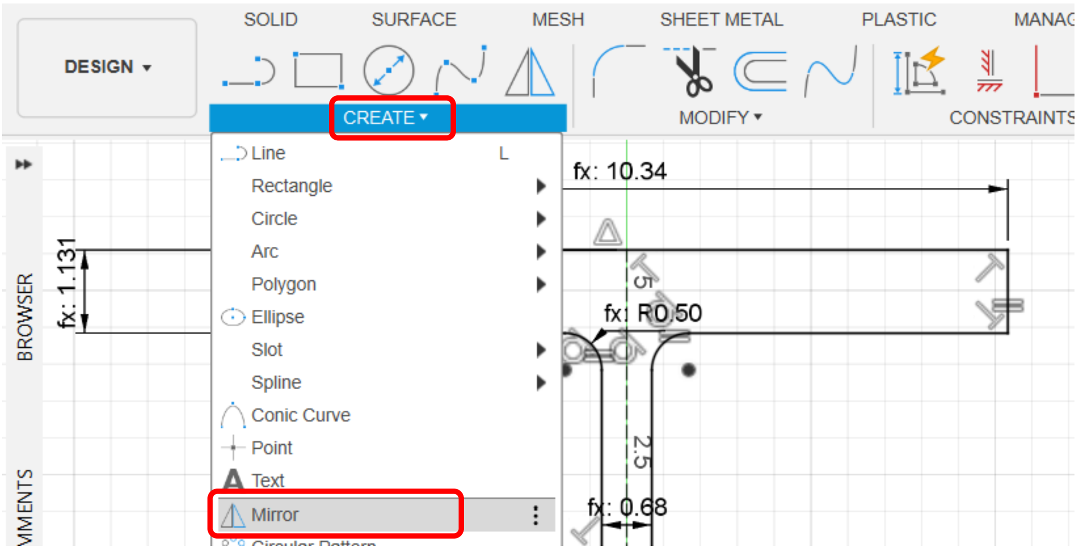

- select the **Fillet** tool.  If it is not visible, find it in the **MODIFY** menu.
- click on the **two inner corners**



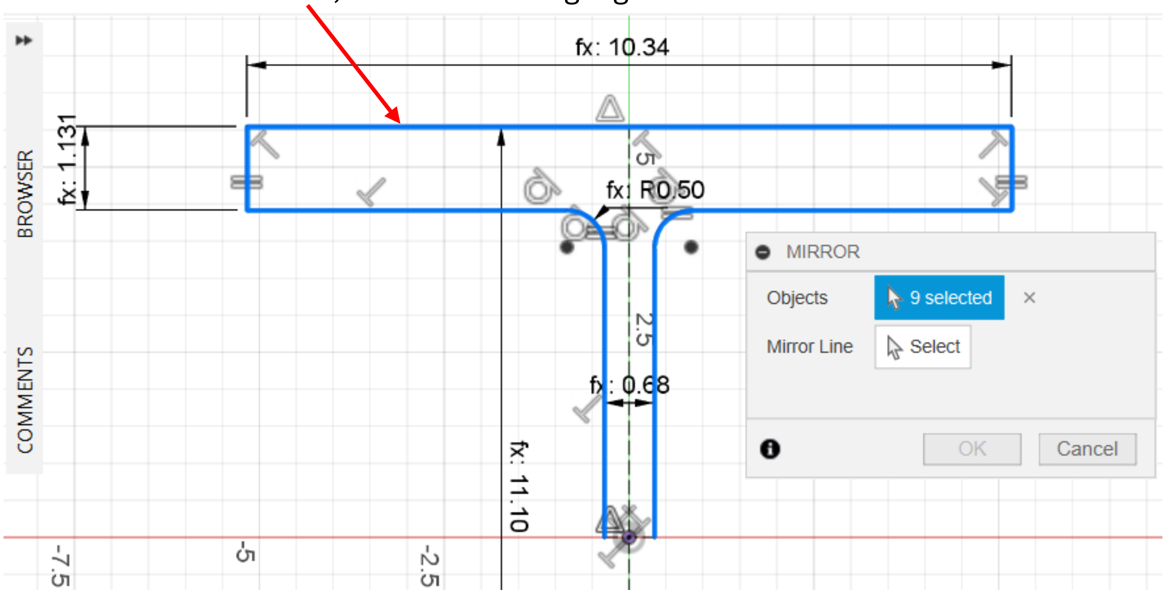

- type **f** in the value box and select **FilletRadius**



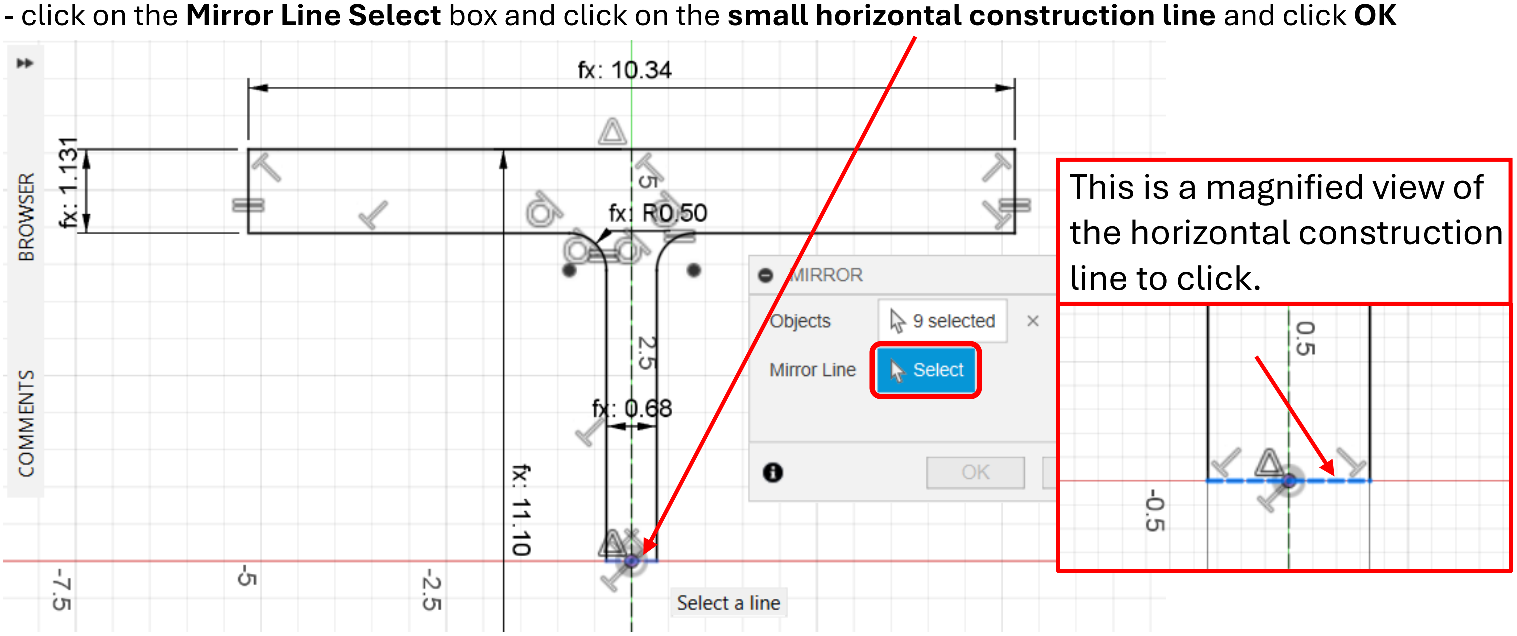
- from the **CREATE** menu select **Mirror**



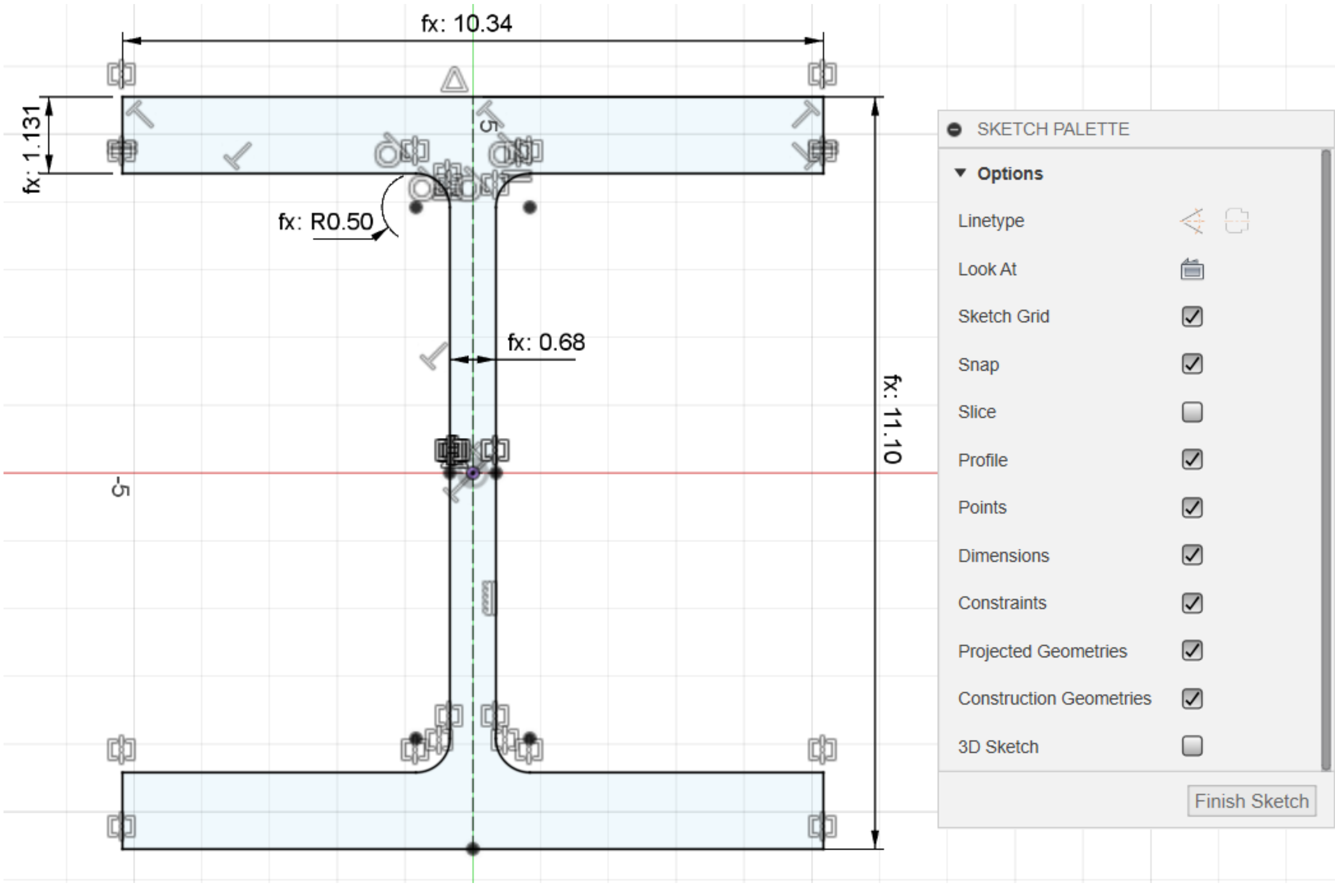

- **double-click on a line**, which should highlight them all blue



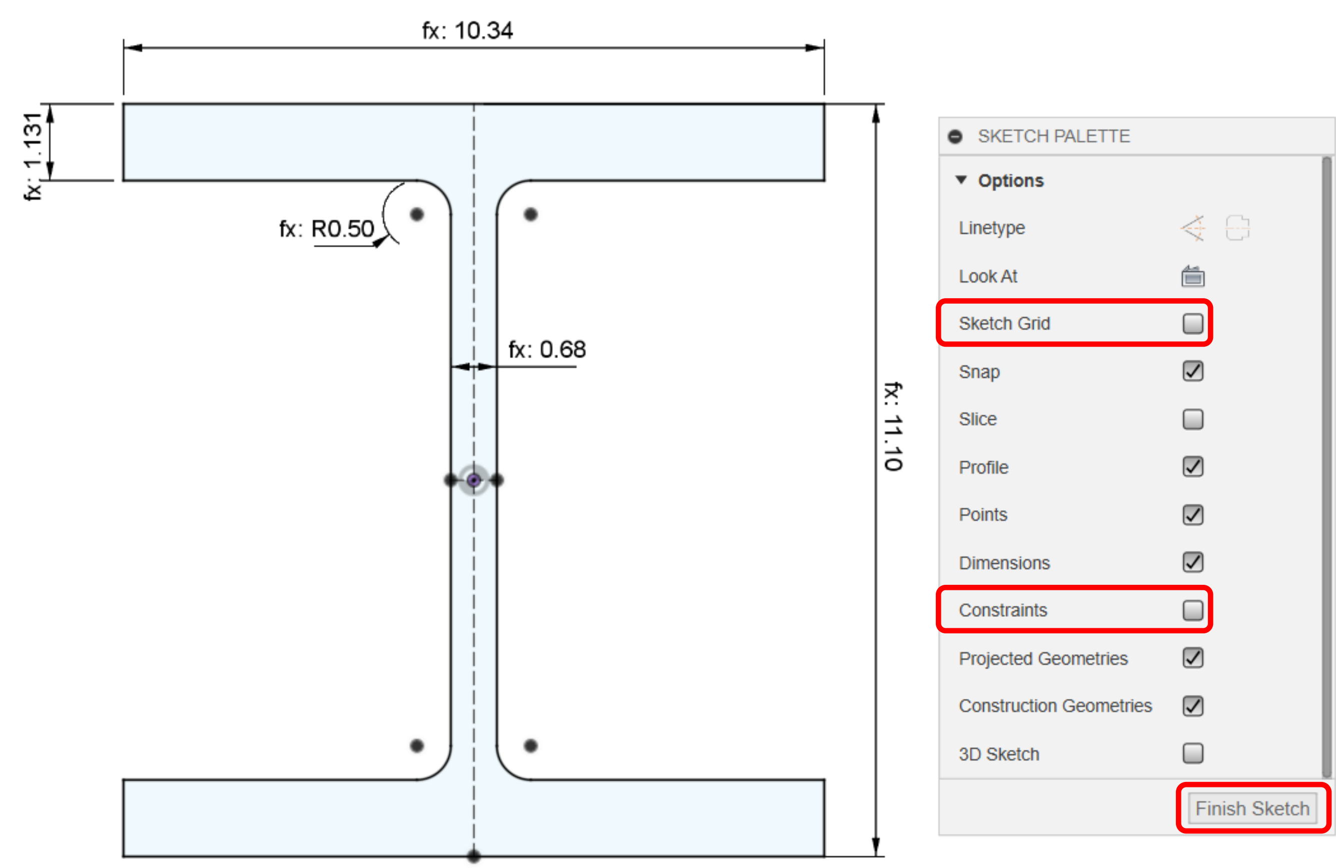

- click on the **Mirror Line Select** box and click on the **small horizontal construction line** and click **OK**



- drag the various dimension values to better positions as shown.

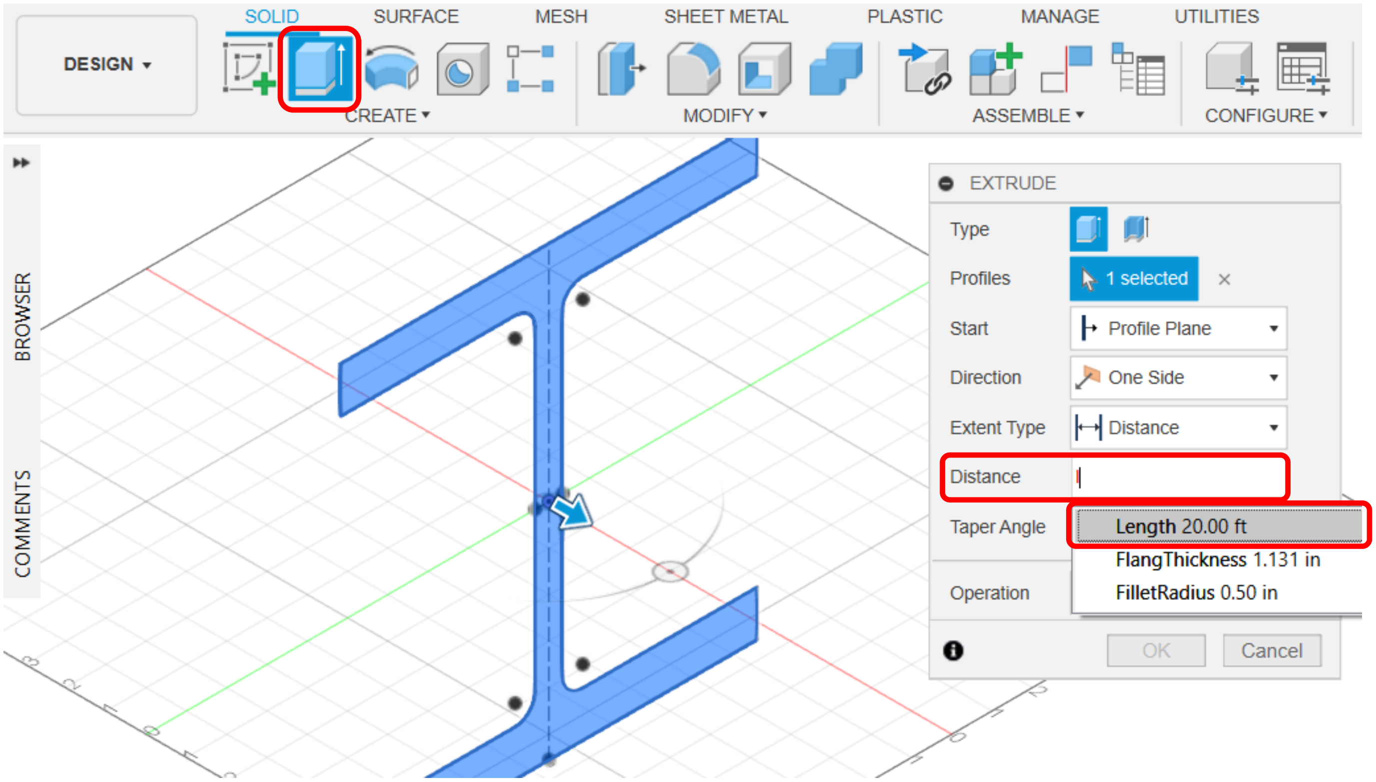



- click on the checkboxs for **Sketch Grid** and **Constraints** to hide them and take a screenshot of the beam without the SKETCH PALETTE. Then turn the Sketch Grid and Constraints back on.
- click **Finish Sketch**

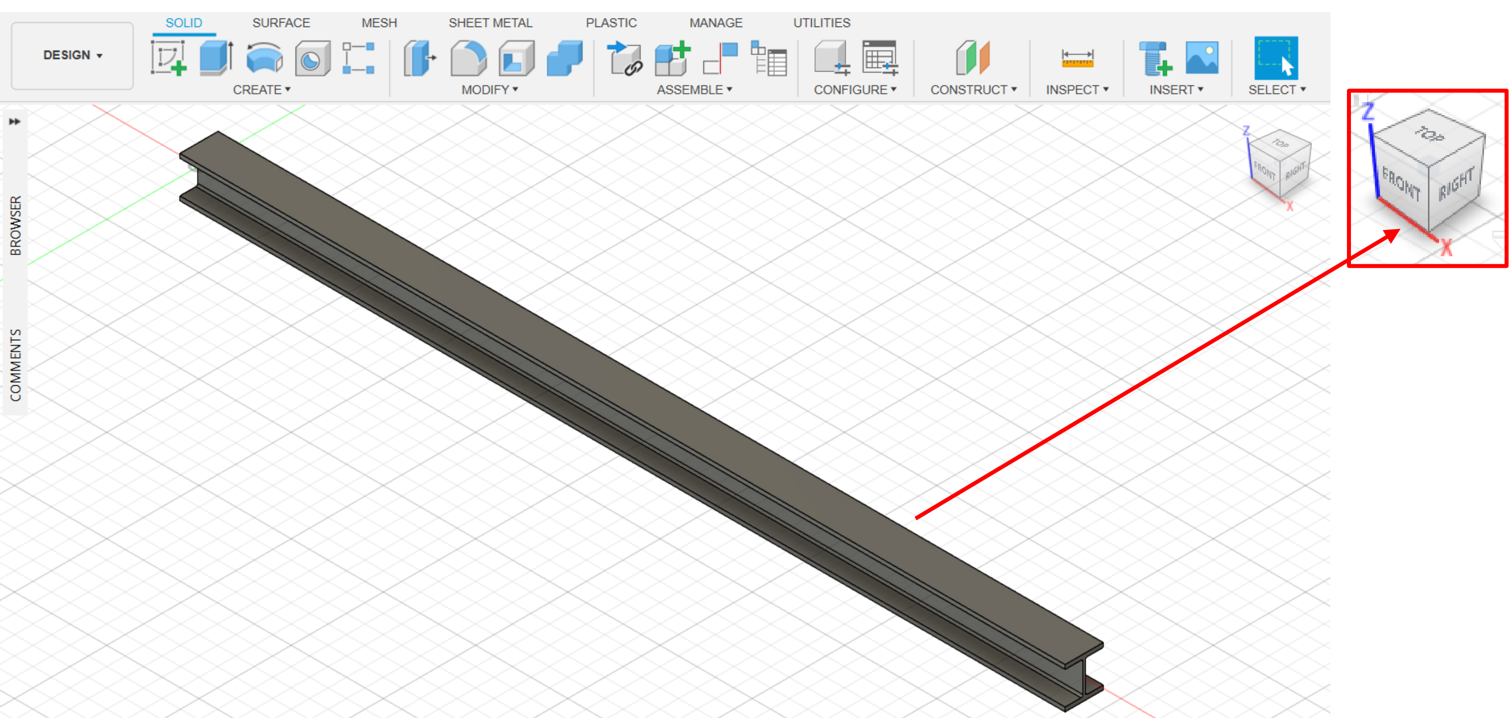


## Extruding the Cross Section Profile

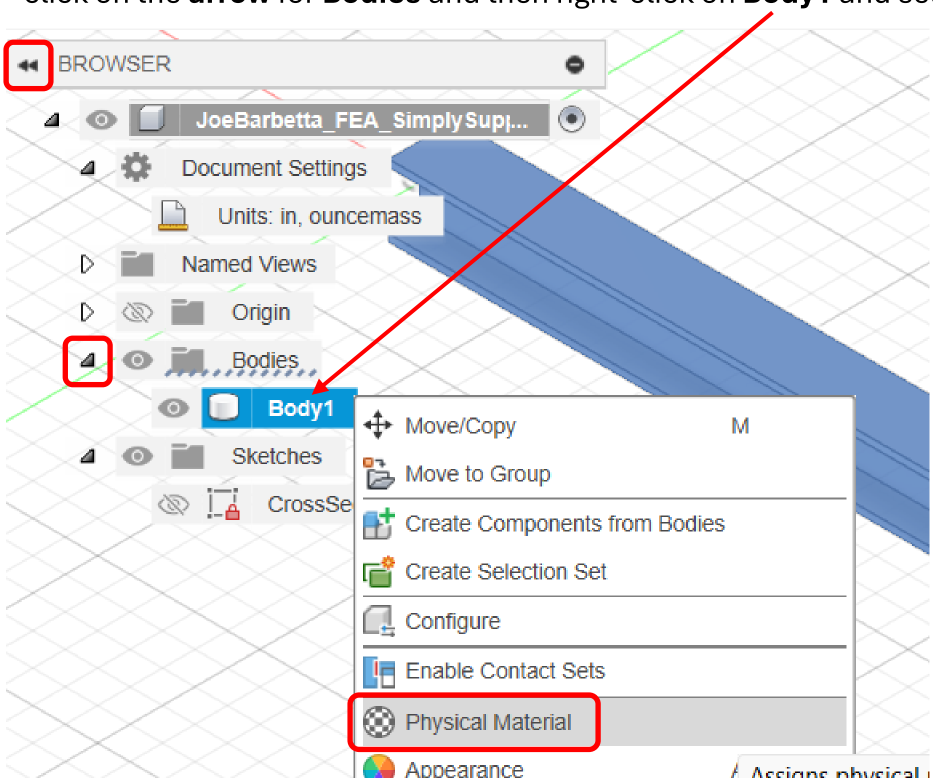
- click on the **Home** icon at the **View Cube**
- select the **Extrude** tool and click on the Profile if it is not highlighted yet. If the Extrude tool is not visible, find it in the CREATE menu.
- type **L** in the **Distance** box and select **Length**. Because the units were set for feet it will be automatically converted to inches.




- click on the **Home** icon at the **View Cube** and the result should look like this. The beam length should be parallel to the X-Axis. If it is not, the wrong plane was selected when starting the sketch.

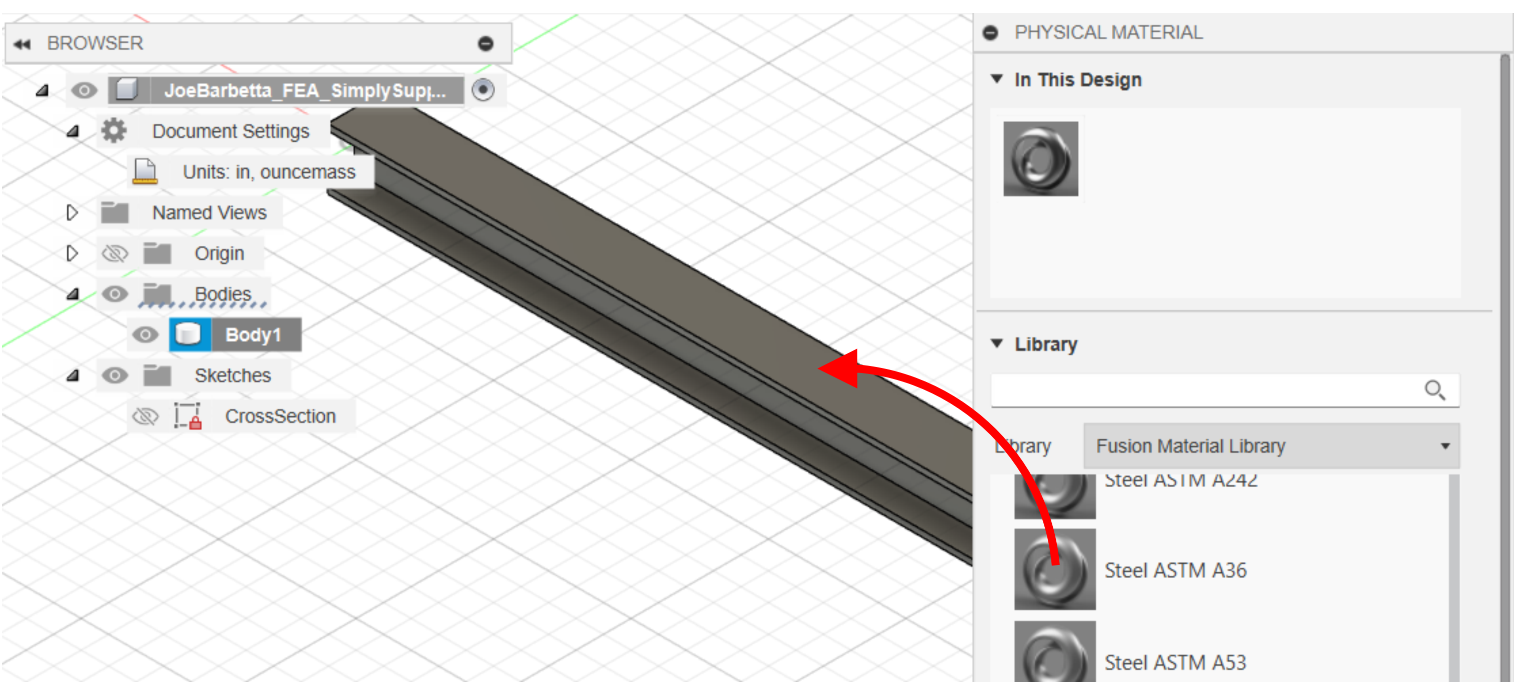


- click on the double arrow to reopen the BROWSER
- click on the **arrow** for **Bodies** and then right-click on **Body1** and select **Physical Material**



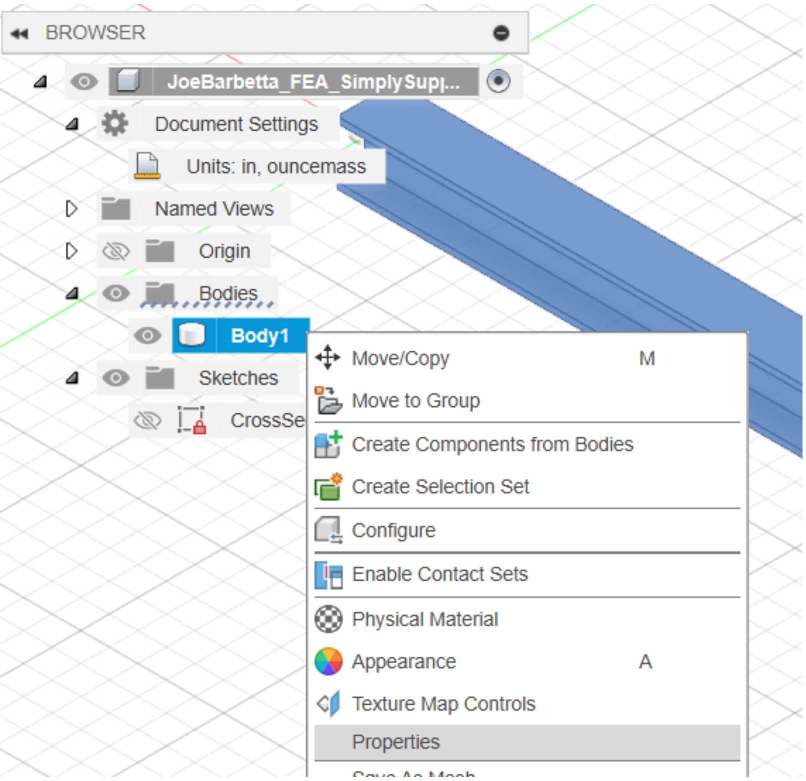

- in the PHYSICAL MATERIAL window scroll down and click on the **Metals folder** to open it
- scroll down through the metals to fins **Steel ASTM A36** and **drag the icon onto the beam**

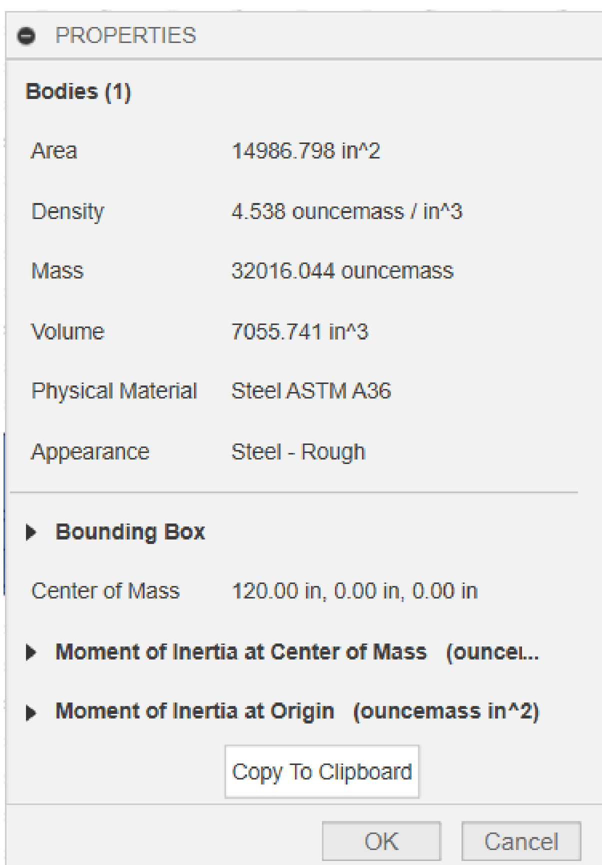
**ASTM** stands for **American Society for Testing and Materials**, which is an organization that sets standards for various industries. A36 steel has been the traditional steel alloy used for structural members. However, A992 and A572 are more common in recent years. It is possible to add custom materials to the Fusion library, but for simplicity, we will use the A36, which is presently in the library.



## Checking Properties

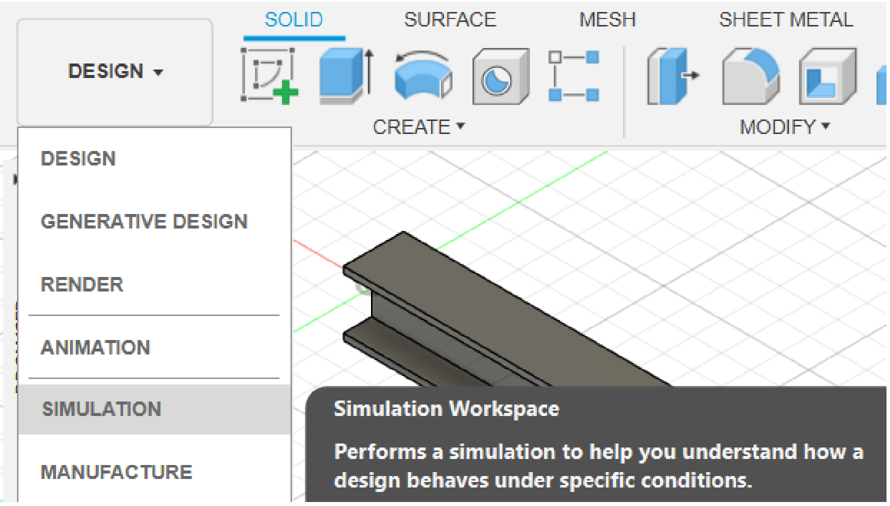
- click on the arrow for the **Bodies** folder, then right-click on **Body1** and select **Properties**



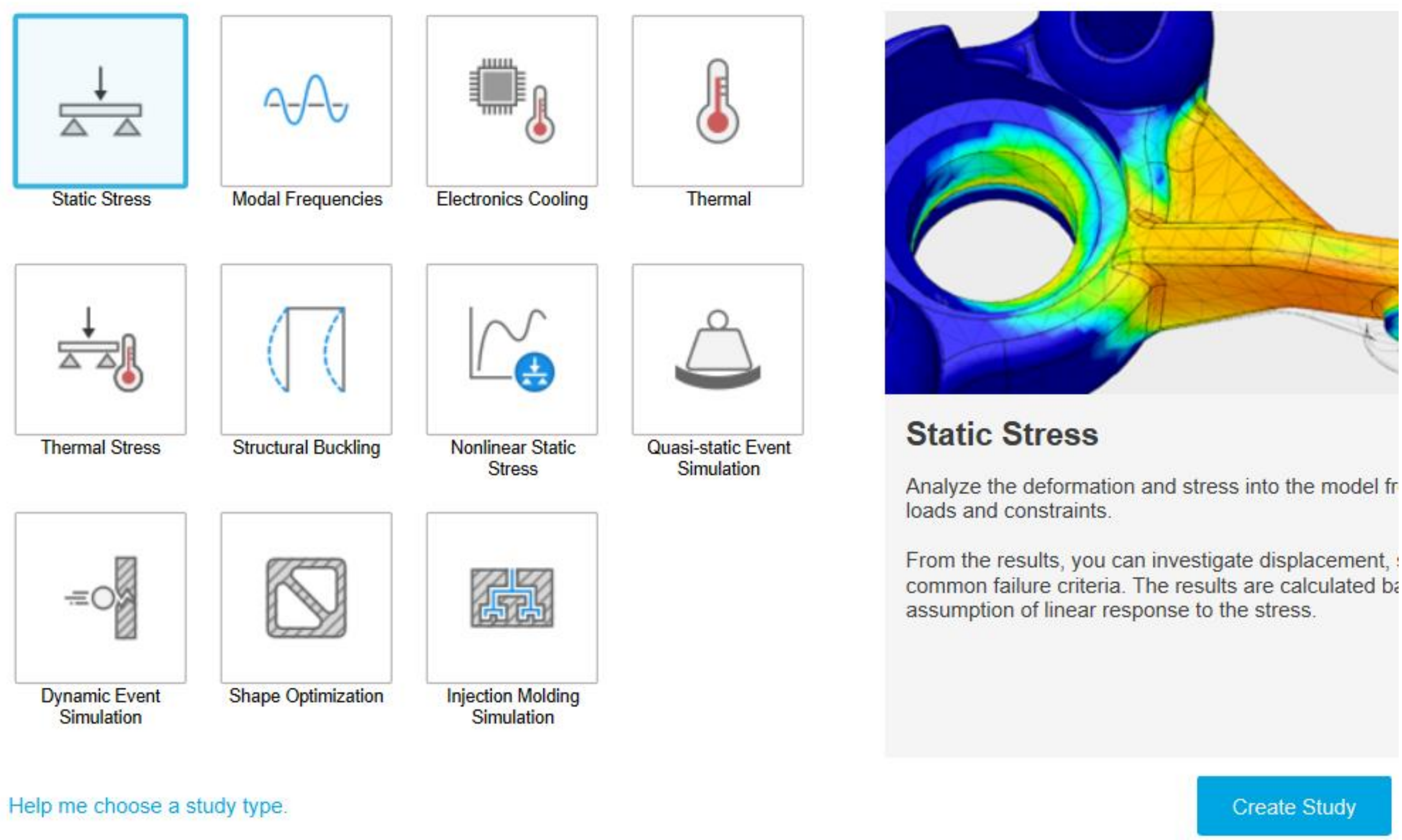



# Using the Fusion Simulation Workspace

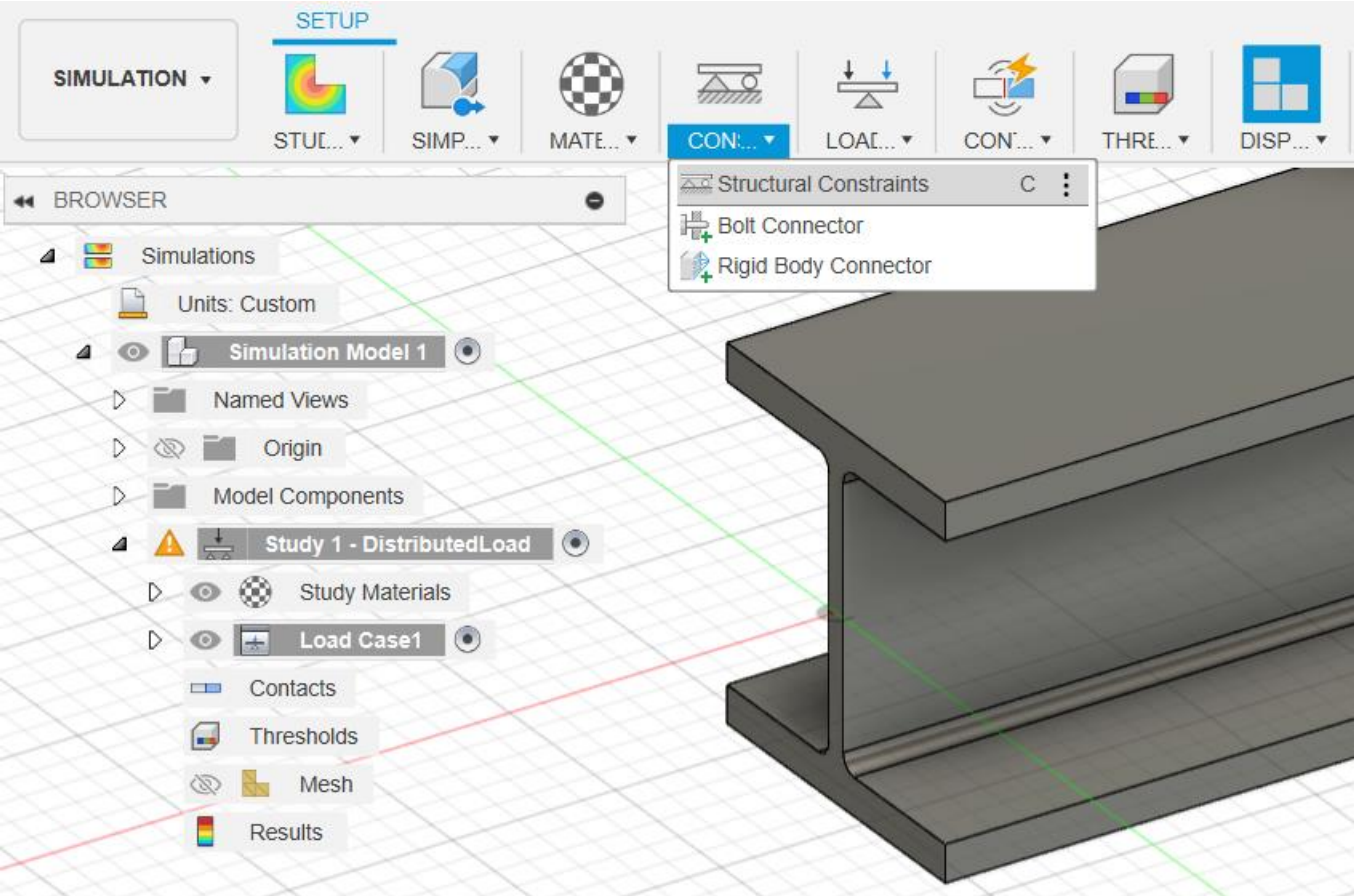
- from the **Workspace** menu select **SIMULATION**



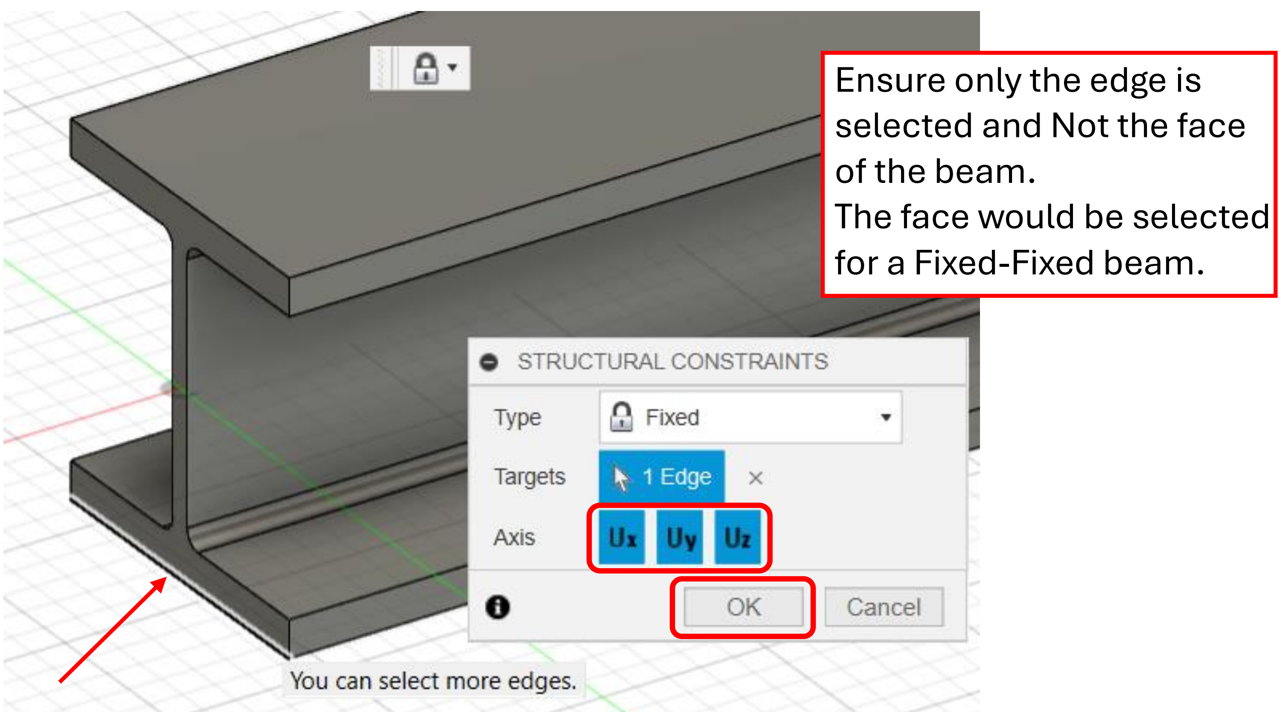

- Yell. “That’s a lot of simulations!”
- select Static Stress and then click on the lower right Create Study button



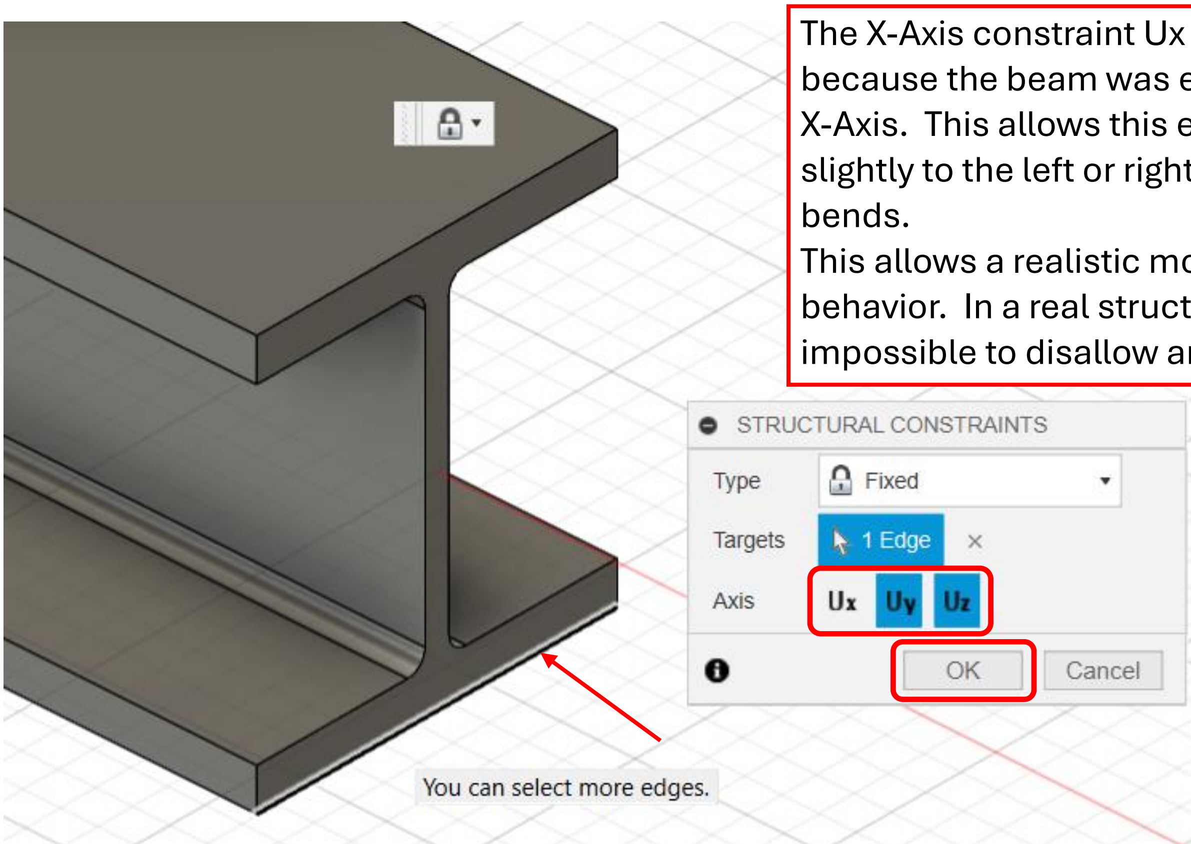
- double-click on **Study Title** and change Static Stress to **Distributed Load**
- zoom into the end of the beam as shown
- from the **CONTACTS** menu select **Structural Constraints**




- click on the **bottom edge** of the beam as indicated by the red arrow
- ensure that **all 3 Axis icons are highlighted** and click **OK**

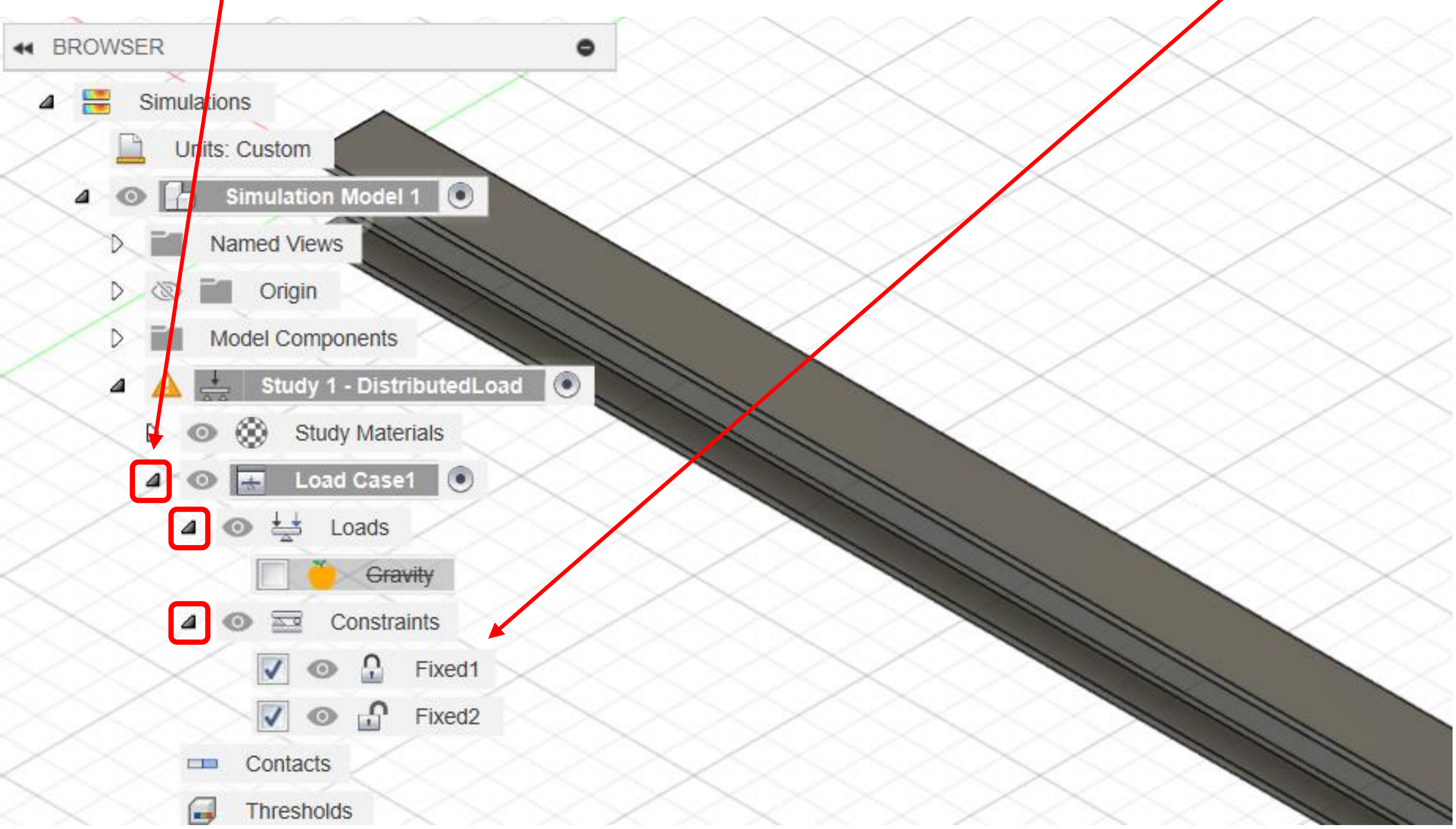


- zoom into the other end of the beam and click on its **bottom edge** as indicated by the red arrow
- click on the **Ux** icon to disable the X-Axis constraint

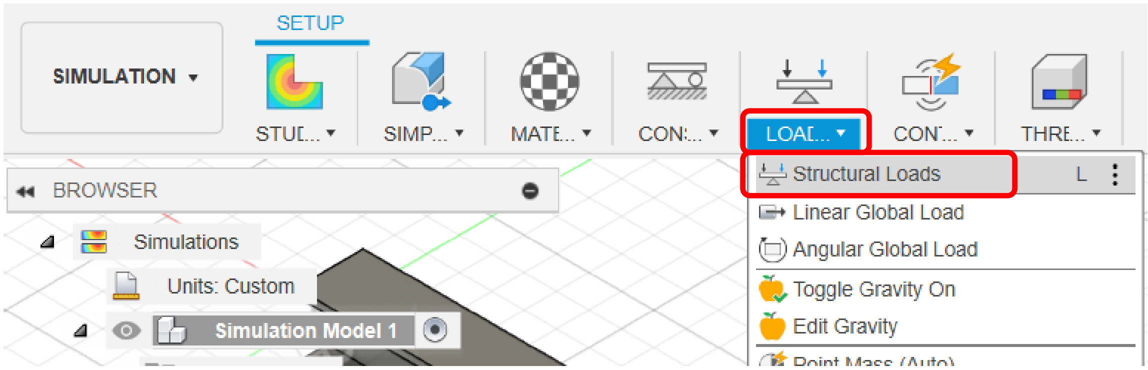



The X-Axis constraint Ux is disabled because the beam was extruded along the X-Axis. This allows this edge to move slightly to the left or right as the beam bends.
This allows a realistic model of the beam behavior. In a real structure it would be impossible to disallow any movement.

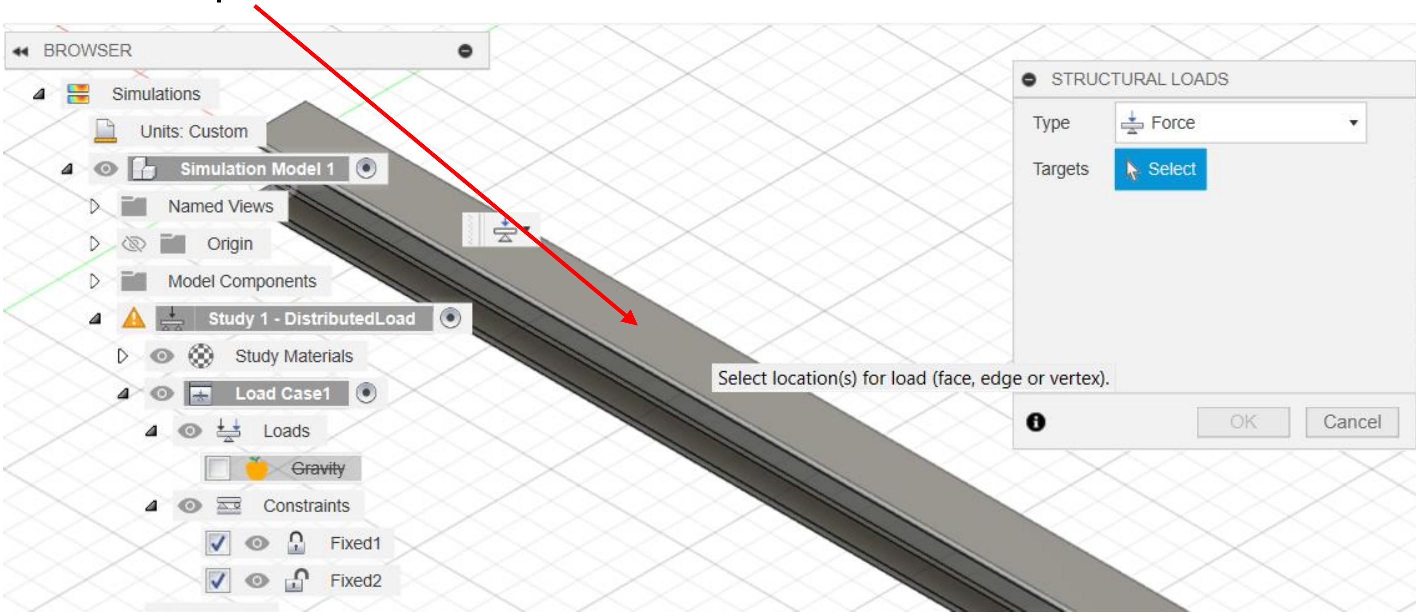
- click on the **arrows** to open the **Load Case**, **Loads**, and **Constraints**. One can see the two **Fixed** constraints. One has an unlocked icon because the X-Axis constraint Ux was disabled.



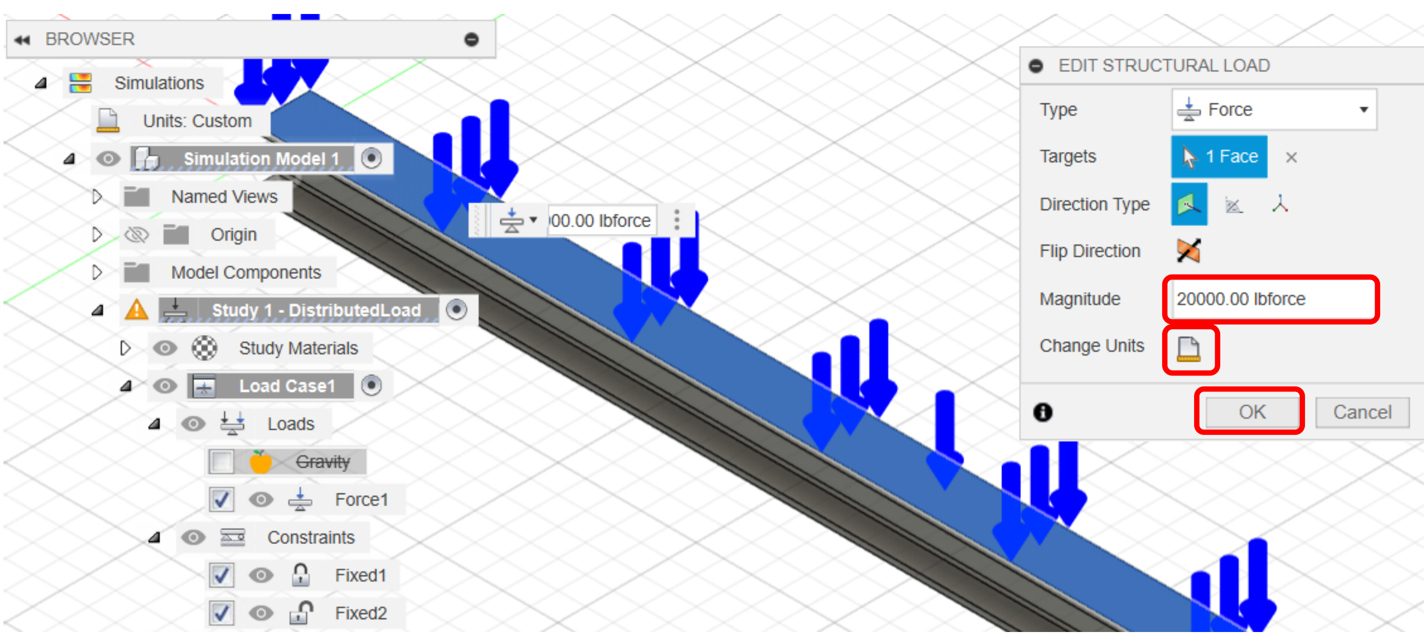
- from the **LOADS** menu select **Structural Loads**




- click on the **top face** of the beam



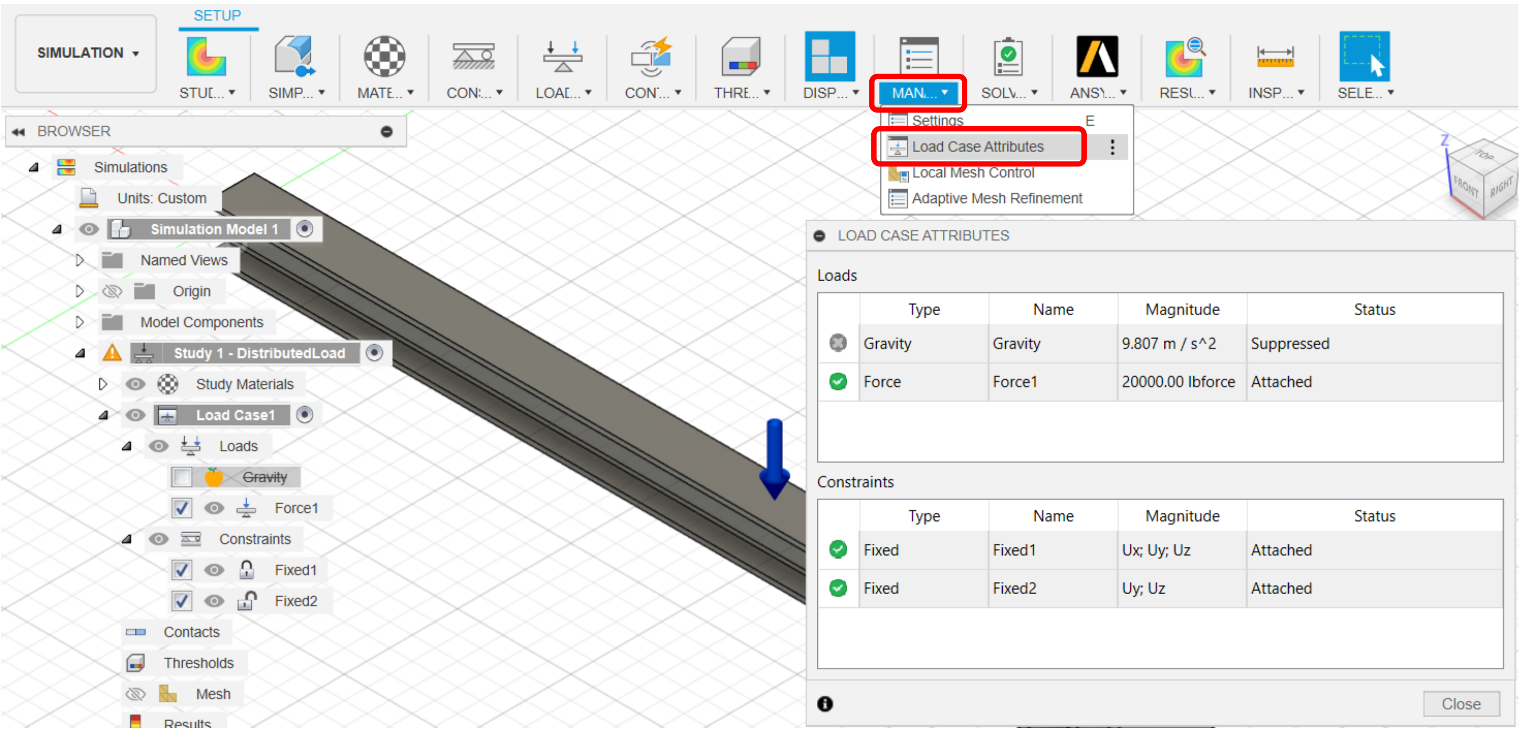

- enter the load value. Here it is 20000 lbs, but your assigned load may differ. Note that one can click on Change Units to change the units if needed. Click **OK**



# Viewing Load Case Attributes

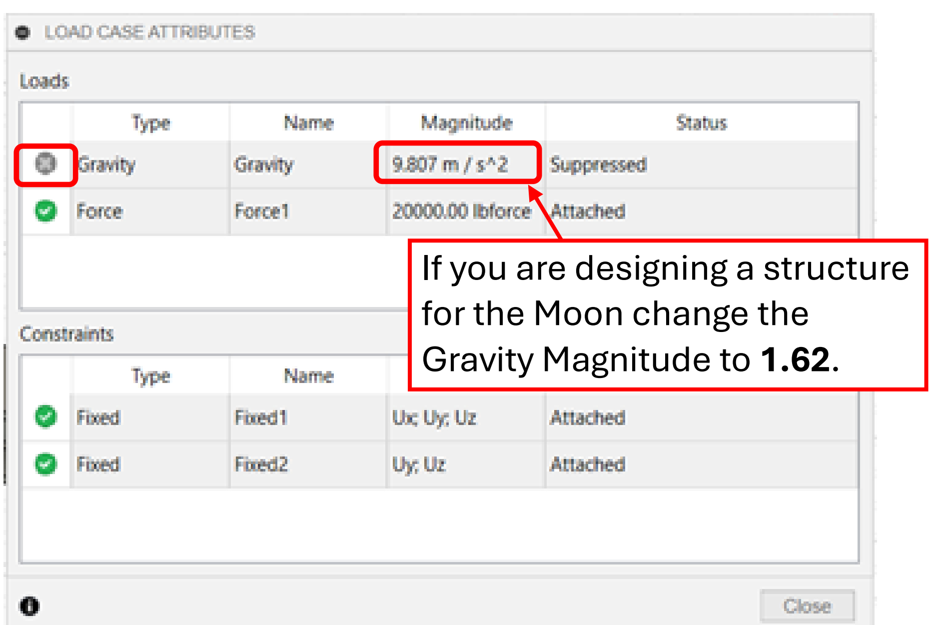
- Note that the load shows as a single arrow now. Hover over it and it will expand into may arrows. It changes to a single arrow to reduce clutter.

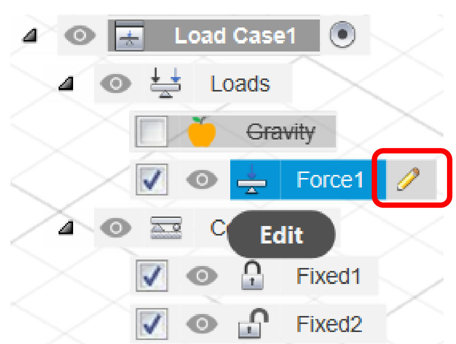
- from the **MANAGE** menu select **Load Case Attributes**




- Note that Gravity is disabled by default. The load on the beam will likely be much hgiher than the weight of the beam itself. Fusion does give you the option to use it.

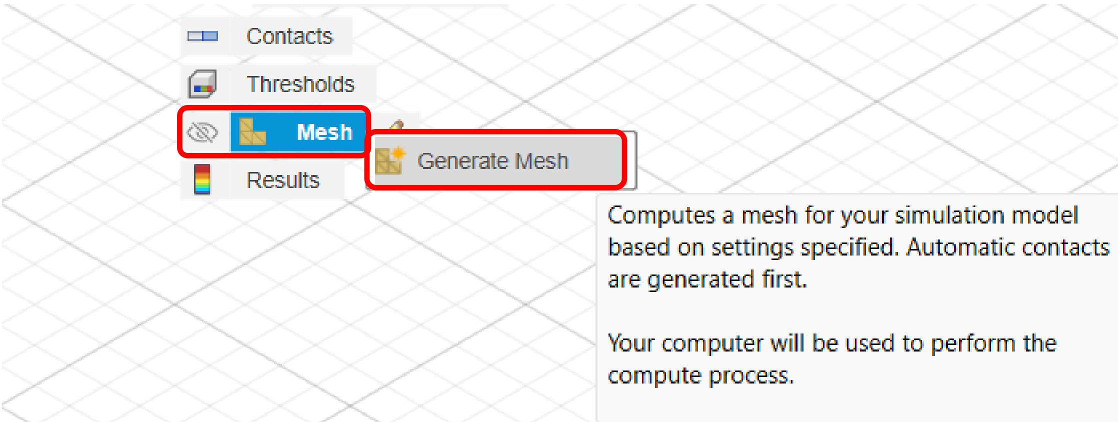
- As shown on the right, any Load or Constraint can be changed by hovering over the item and clicking on the pencil icon



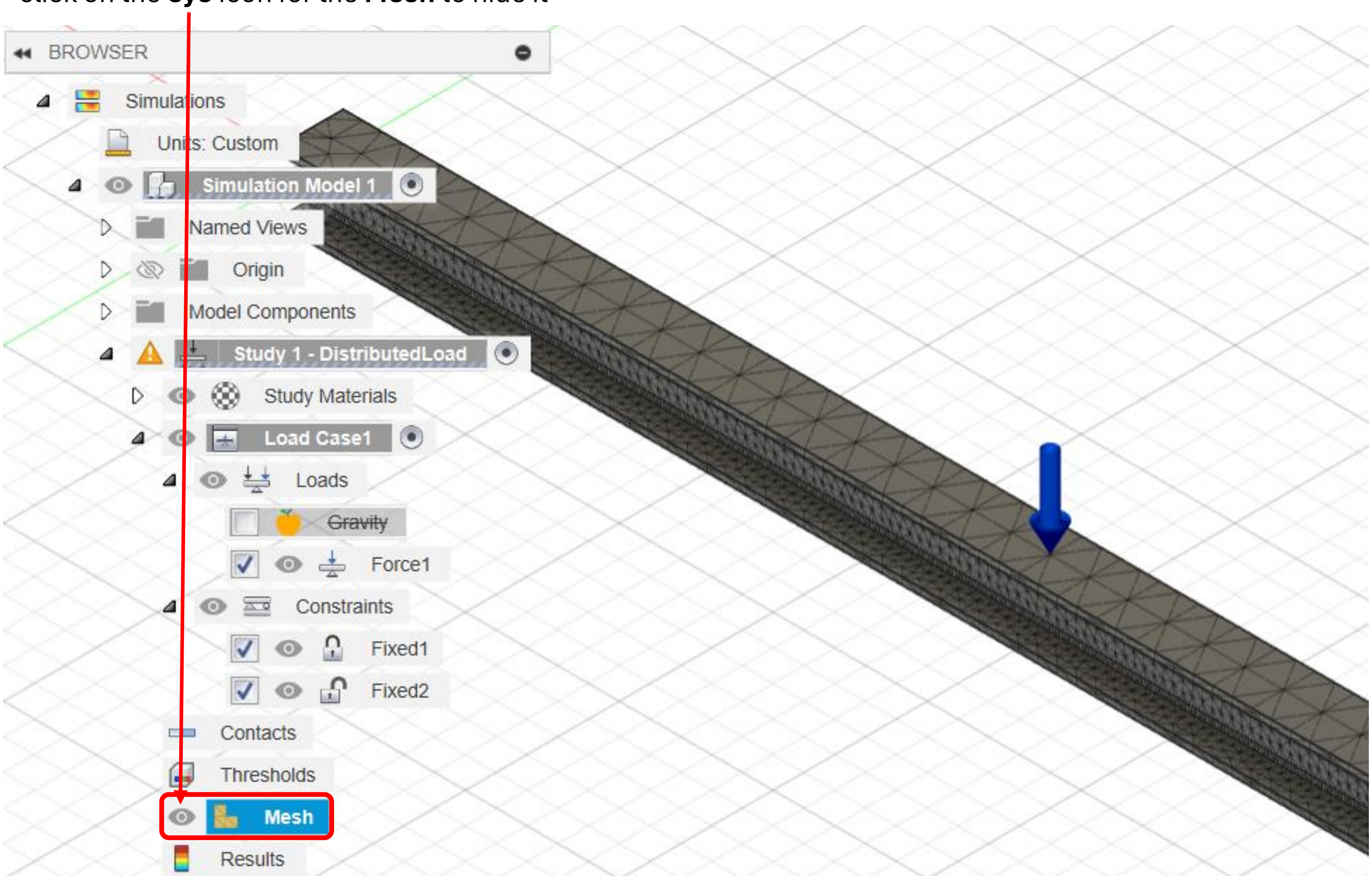



## Generating the Mesh

- right-click on the **Mesh** item near the bottom of the study list and select **Generate Mesh**

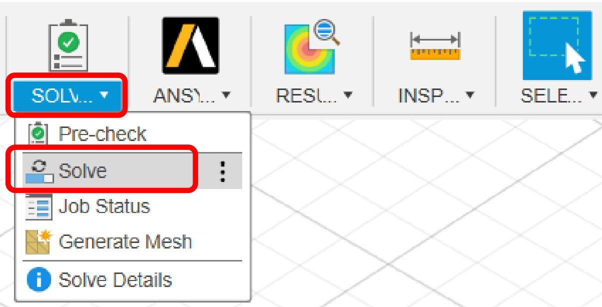



- click on the **eye** icon for the **Mesh** to hide it

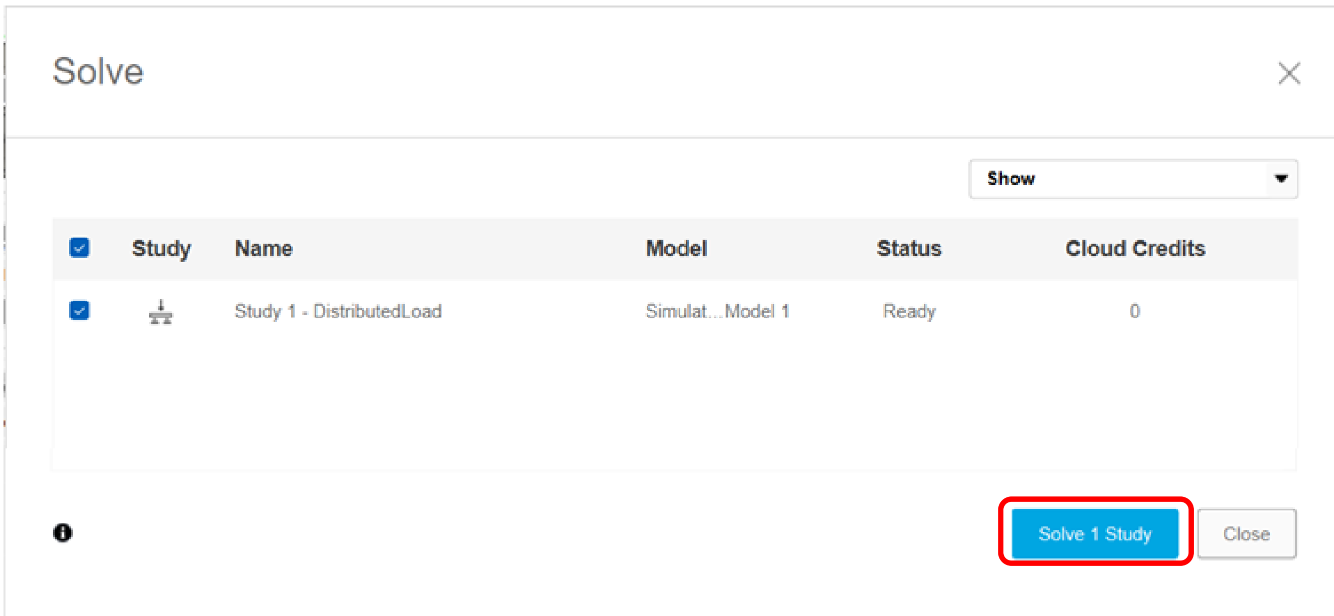


# Solving the Simulation

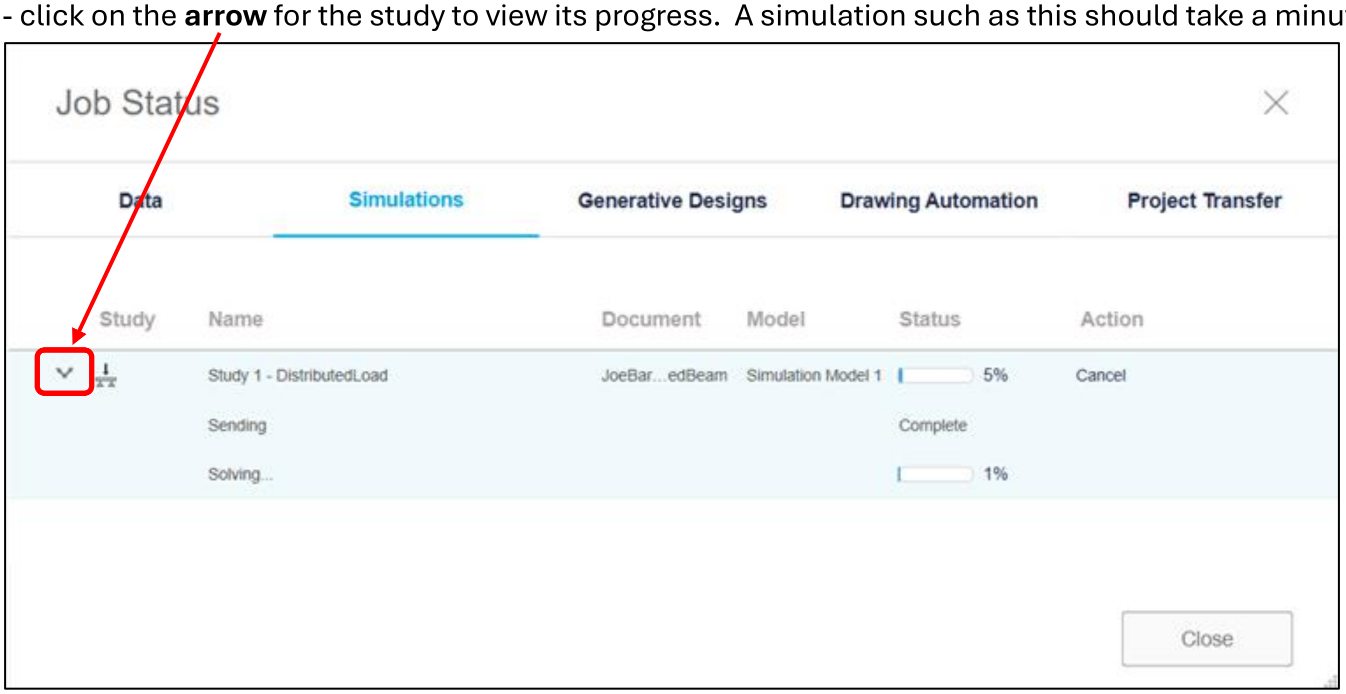
- from the top right **SOLVE** menu select **Solve**



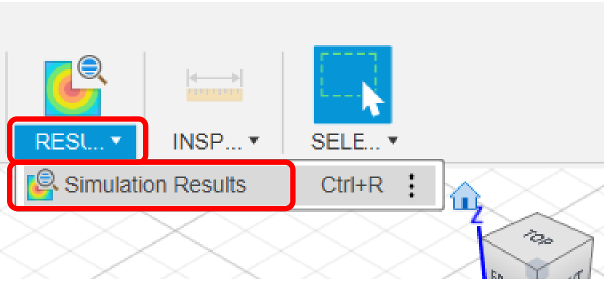

- click the **Solve 1 Study** button




- click on the **arrow** for the study to view its progress. A simulation such as this should take a minute or two.



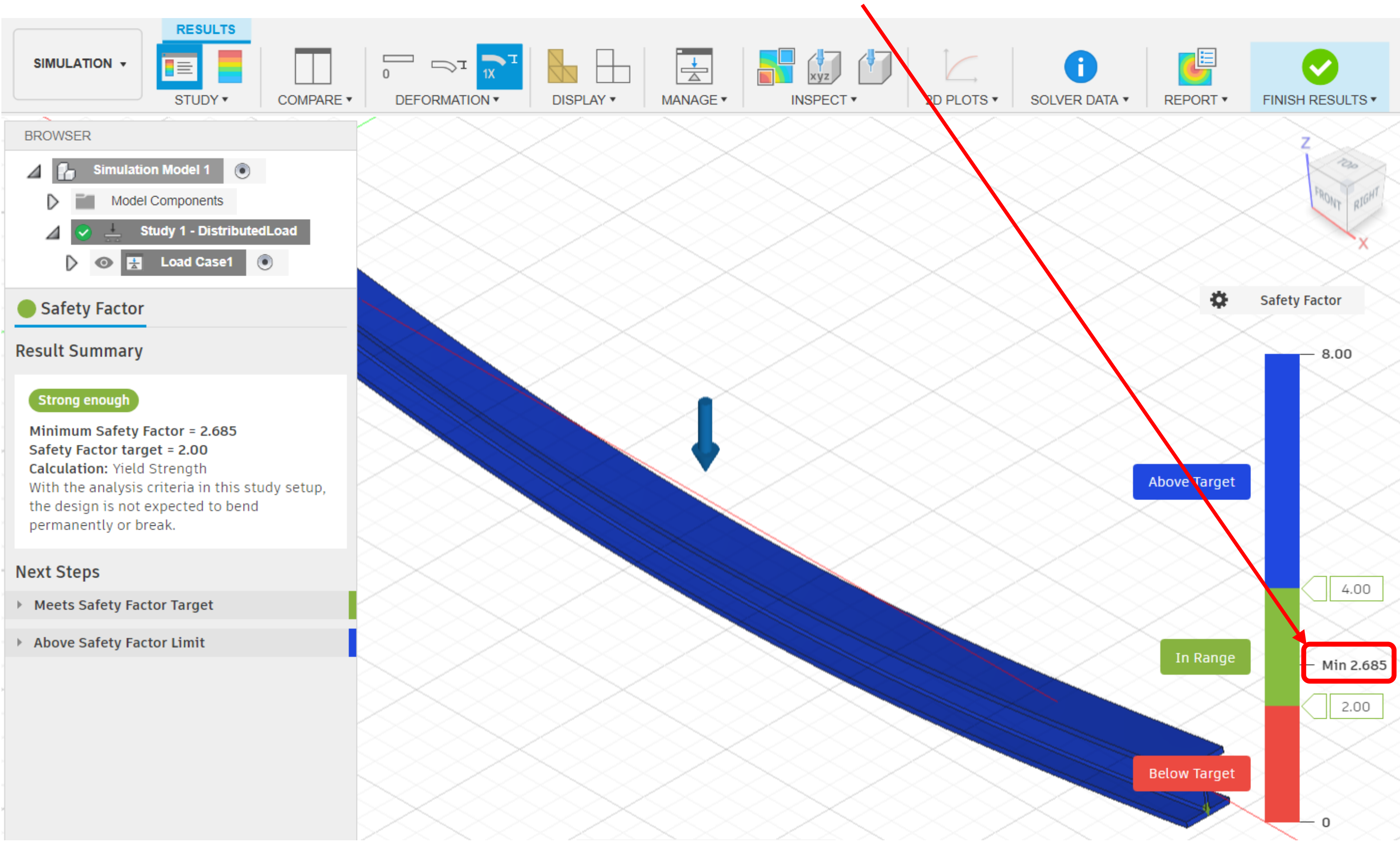
- when the simulation is solved close the progress screen and from the **RESULTS** menu select **Simulation Results**



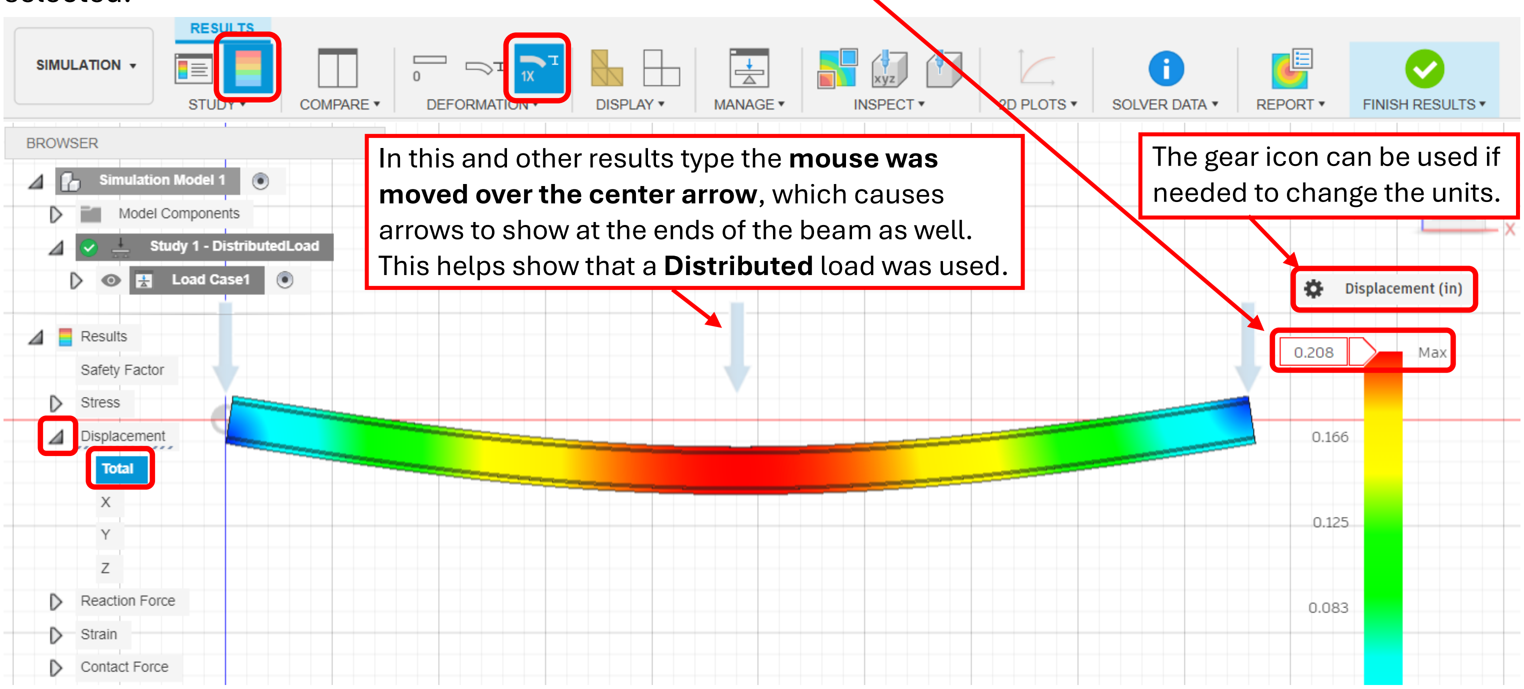

- enjoy the neat “data from the cloud” animation


Autodesk uses Amazon Web Services, which has a major data center in Virginia.  Your simulation likely ran on a computer in this building and the results are sent in pulses of light over fiber optics to our school.

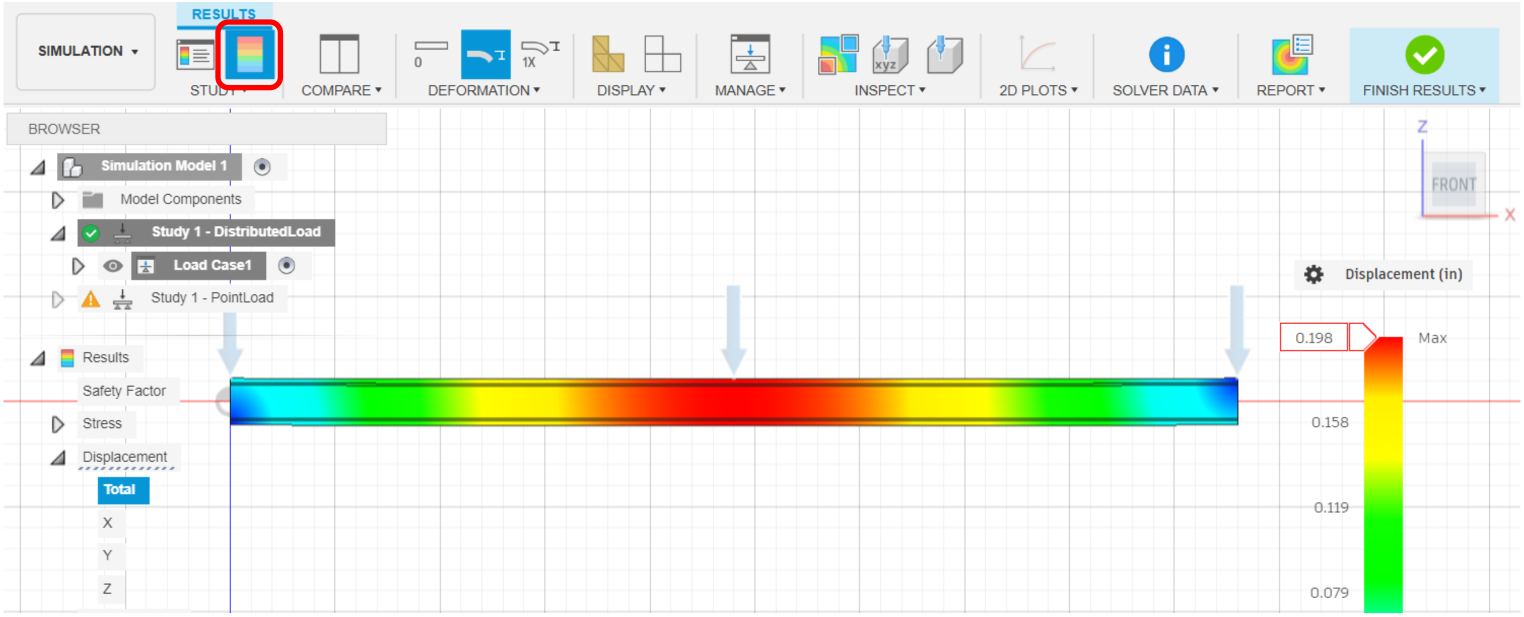
This is how the results is first presented.  Note that the **Minimum Safety Factor** is in the "good" range.



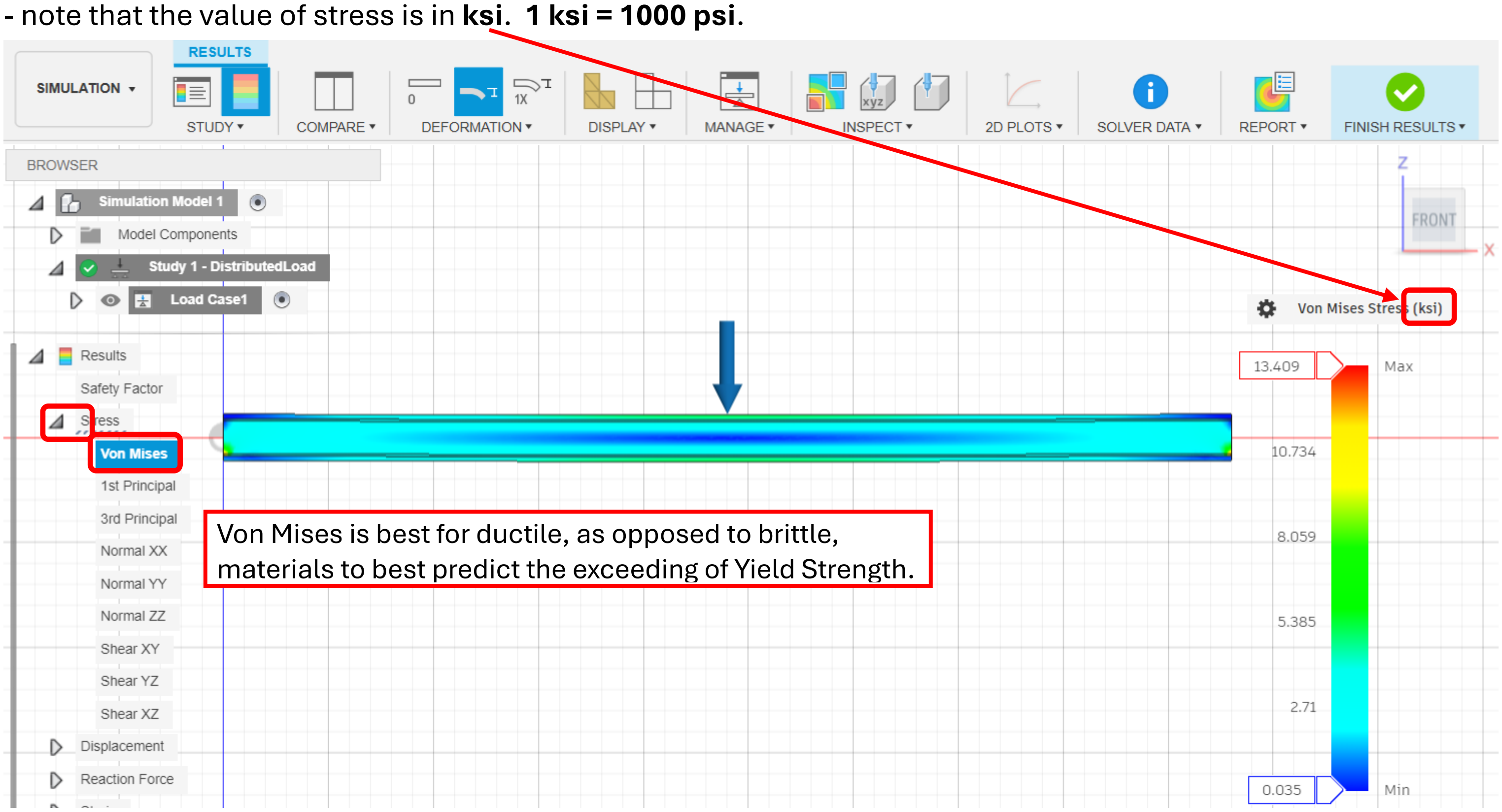

- click on the **Results** icon
- click on the **arrow** next to **Displacement** and select **Total**
- note that the **Max** Displacement shows as the **top value**.  Here it is 0.**208 in**.  The term used in structrual engineering is ***deflection***.
- note that the image of the beam shows a **greatly exagerated deflection** because **Adjusted Deformation** is selected.



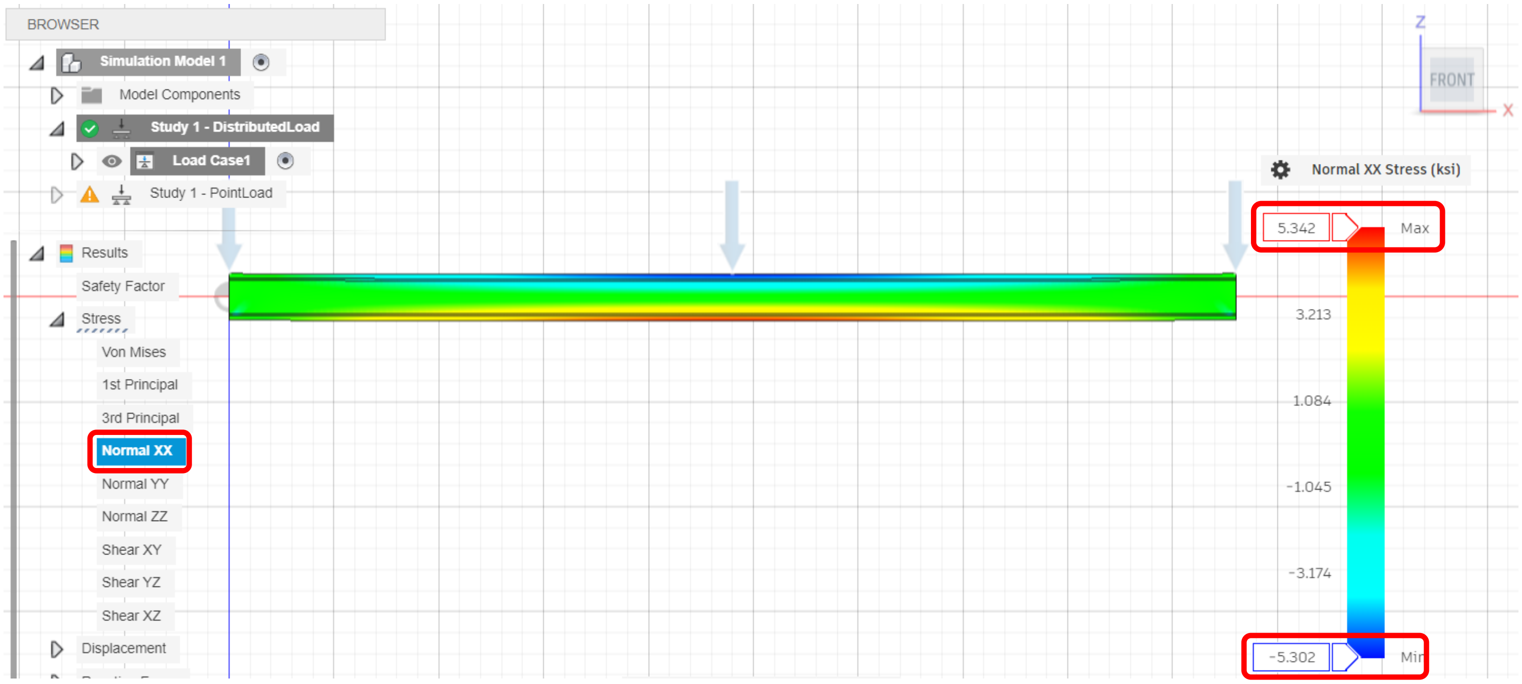
- click the **Actual Deformation** icon to see that the actual deflection is barely visible. The reason that the default is Adjusted Deformation is to allow the deflection shape to be determined. The shape of the deflection is that expected for a Simply-Supported beam. It would look quite different for a Fixed-Fixed beam.




- click on the **arrow** next to **Stress** and select **Von Mises**. One can see that the stresses are low on the *neutral axis* in the center of the beam.
- note that the value of stress is in **ksi**. **1 ksi = 1000 psi**.



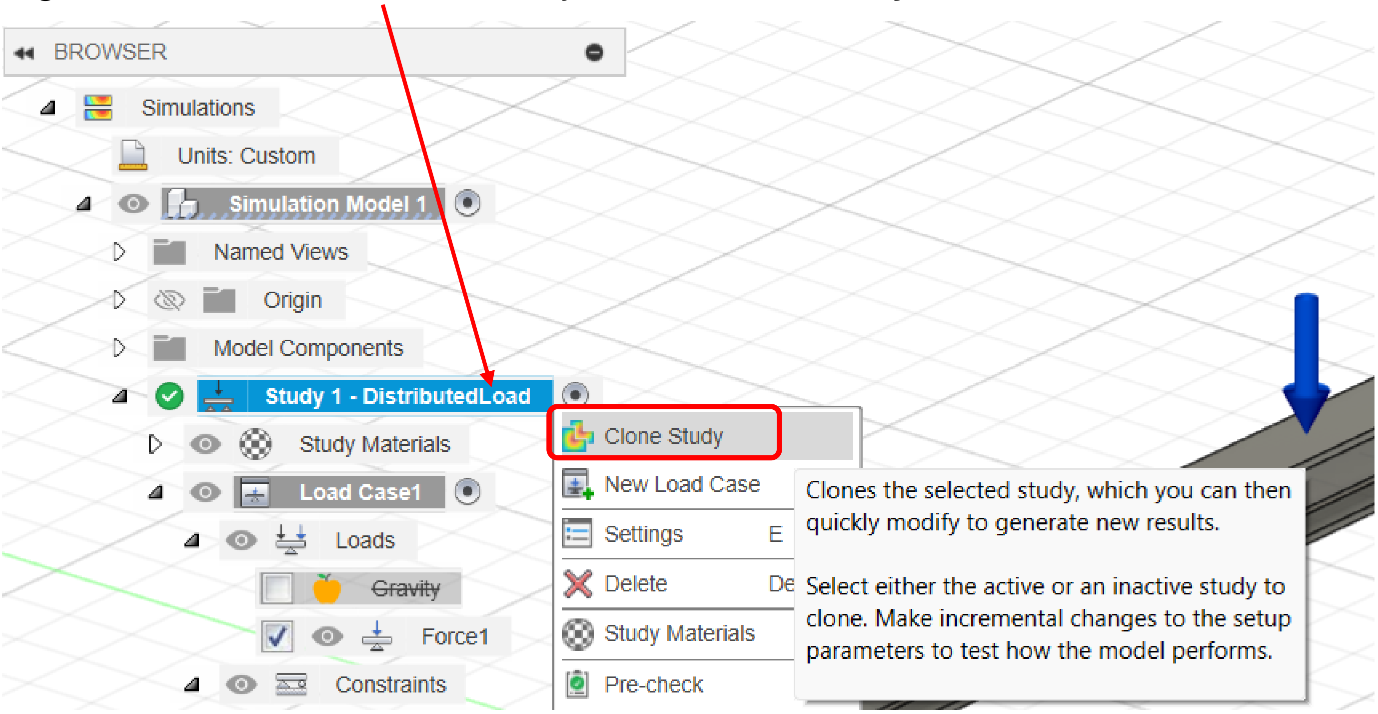
- select **Normal XX**.  One can see that stresses now range from a **negative to a positive** value.  This shows how the **top of the beam is in *compression*** and the **bottom of the beam is in *tension***.



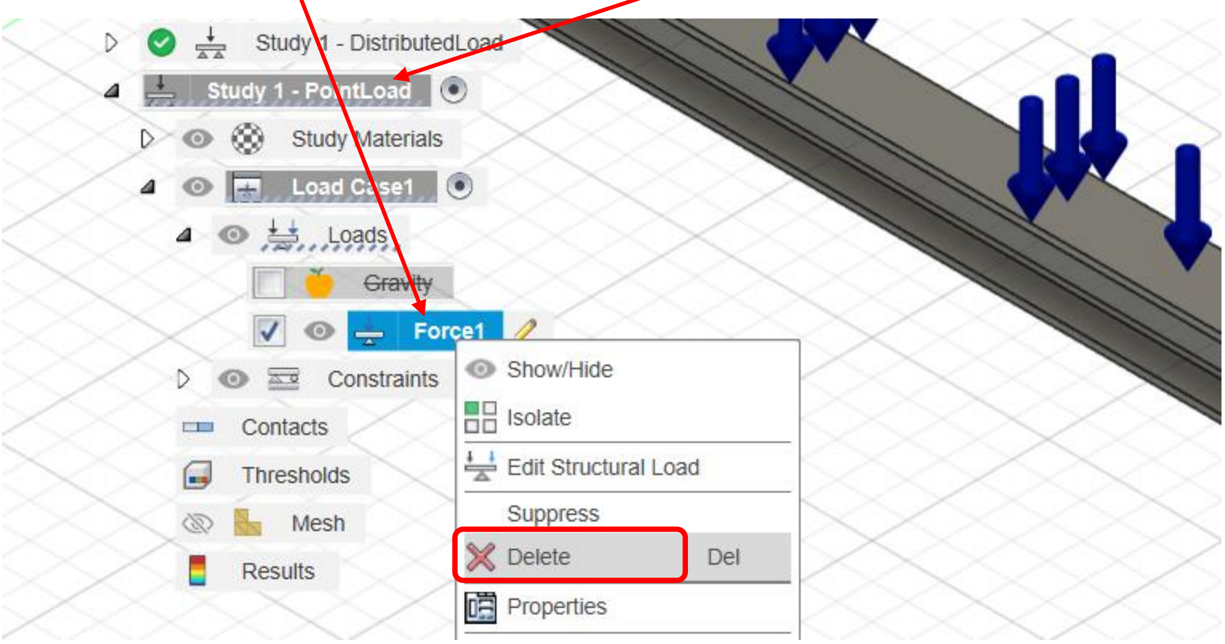

Continued on next page.

## Peforming a Point Load Analysis

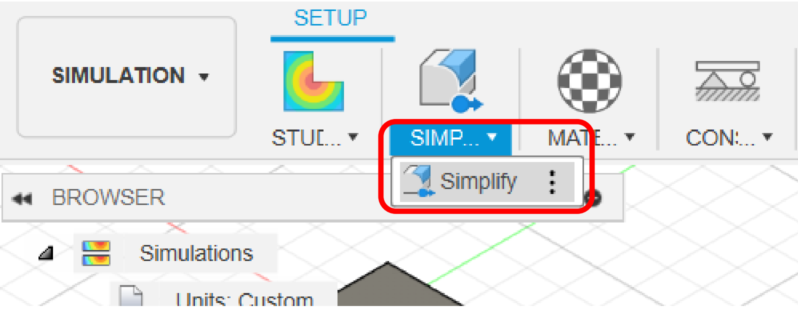
- right-click on the **DistributedLoad** study and select **Clone Study**



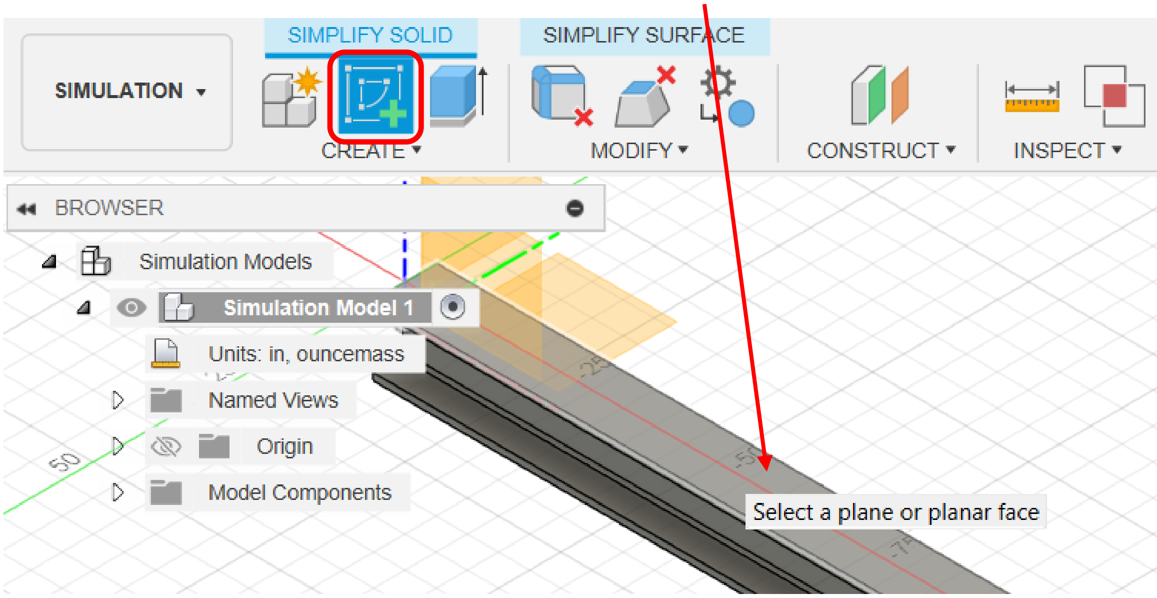

- double-click on the **Study Name** and change it to **PointLoad**
- right-click on the **Force1** load and select **Delete**



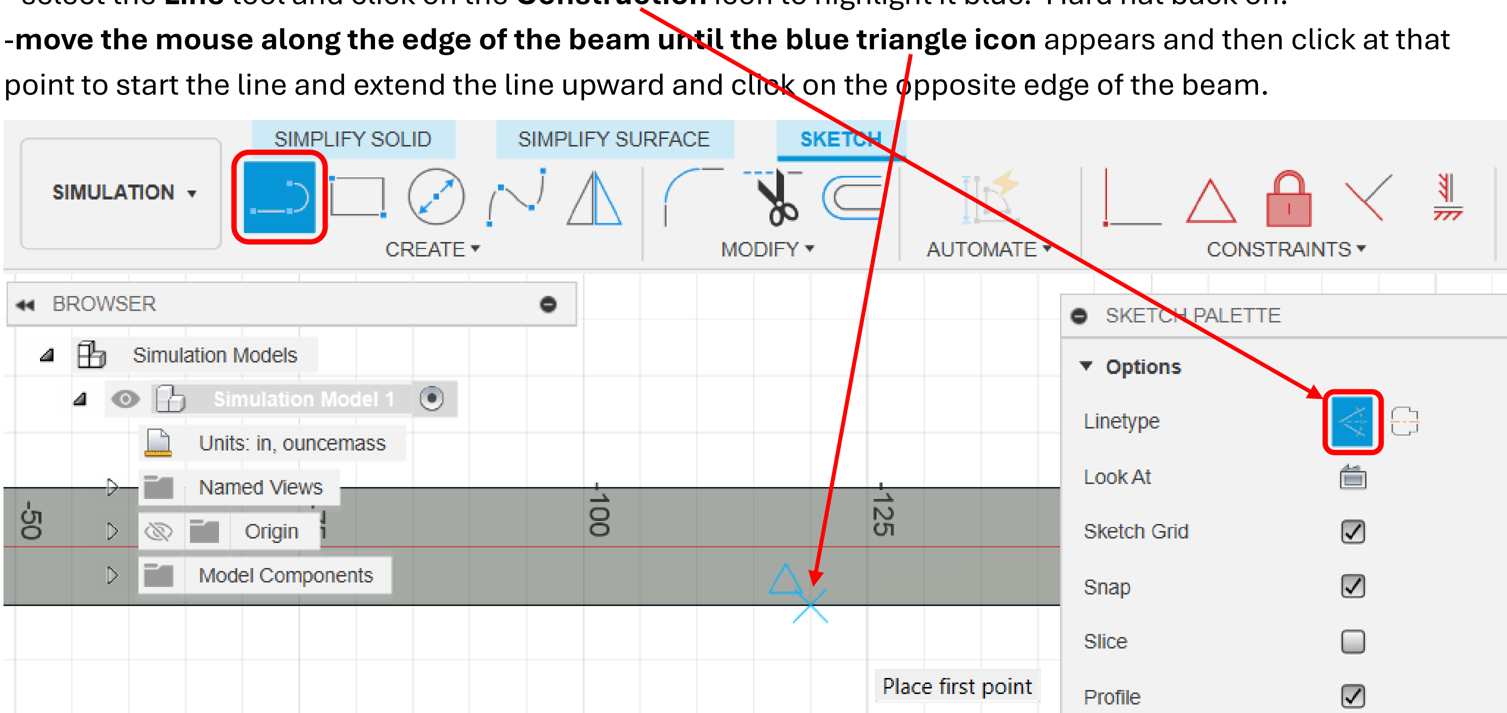
- from the **SIMPLIFY** menu select **Simplify**



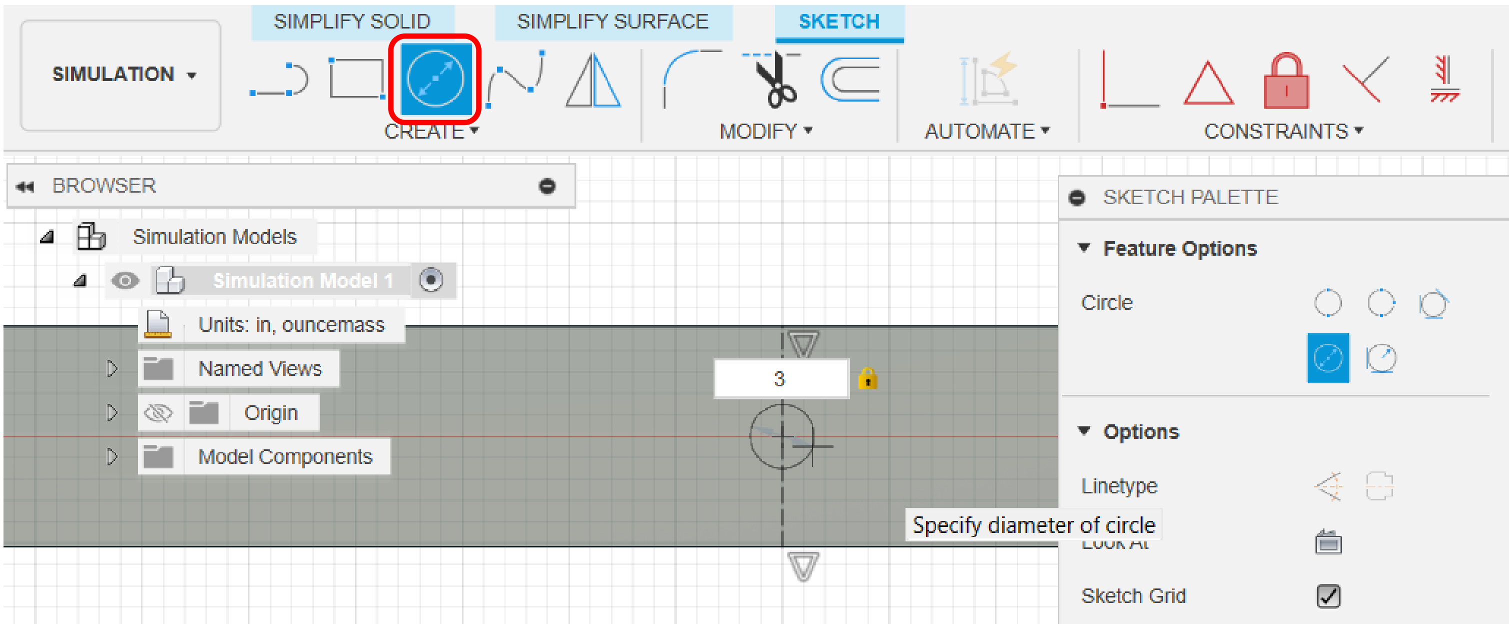

- select the **Create Sketch** tool and click on the **top face** of the beam



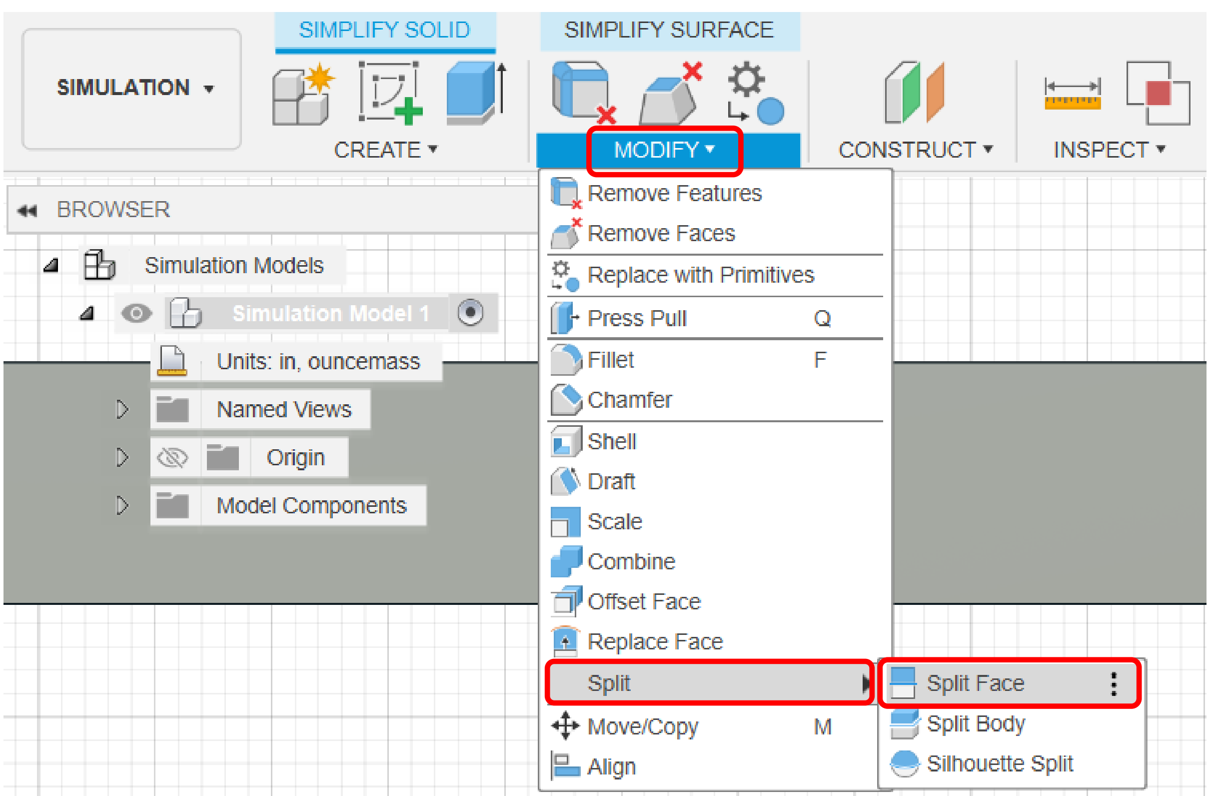

- zoom into the **center area of the beam**
- select the **Line** tool and click on the **Construction** icon to highlight it blue. Hard hat back on.
-**move the mouse along the edge of the beam until the blue triangle icon** appears and then click at that point to start the line and extend the line upward and click on the opposite edge of the beam.



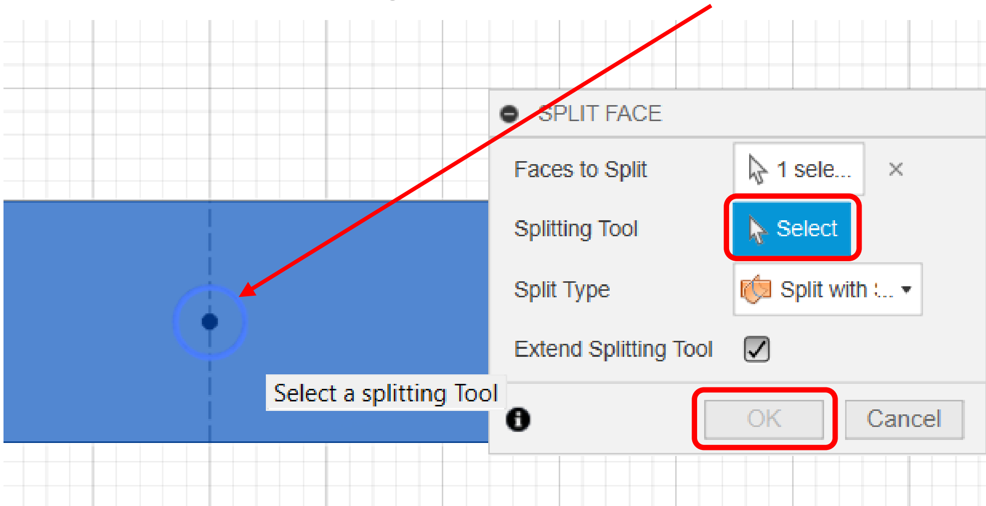
- click on the **Contruction** icon again to turn it off.  You can keep the hardhat on if you like.
- select the **Center Diameter Circle** tool
- move the mouse over the line just drawn and click when the **blue triangle icon** appears click
- extend the circle outward and enter a value of **3**




- from the **MODIFY** menu select **Split** and **Split Face**

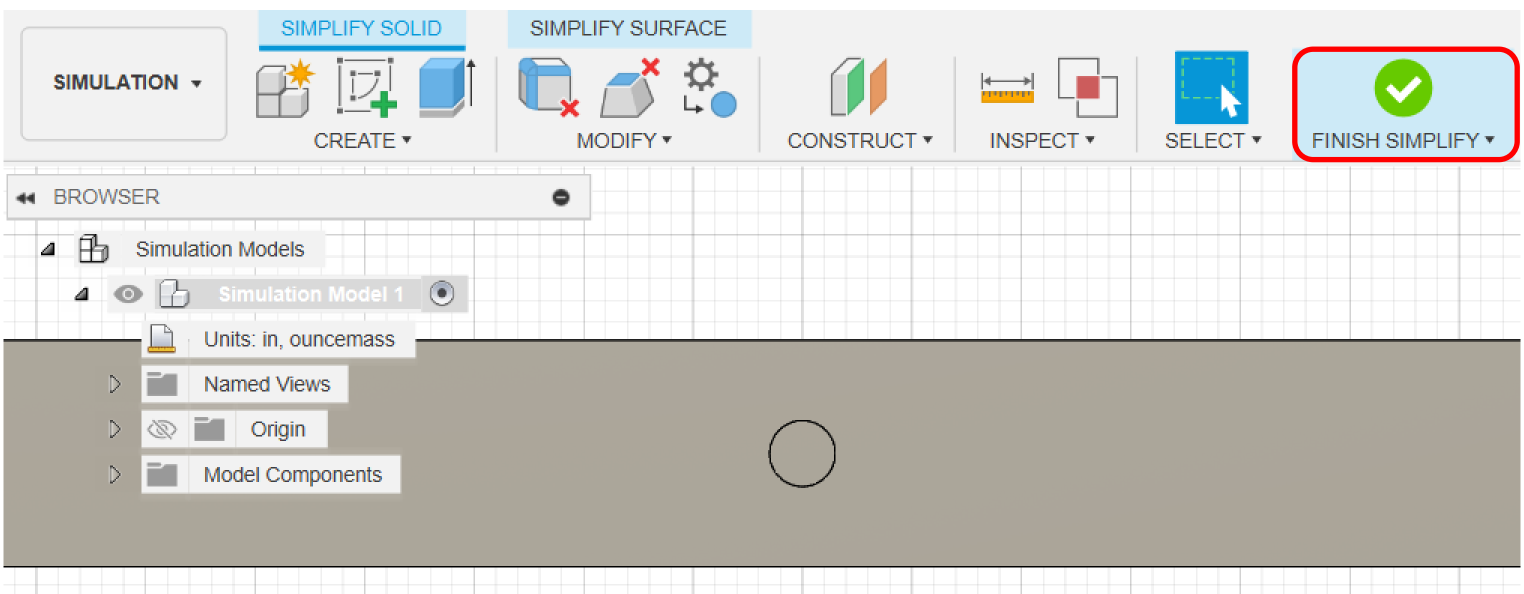


- click on the **top face** of the beam
- click on **Select** for **Splitting Tool** and click on the **circle** and click **OK**

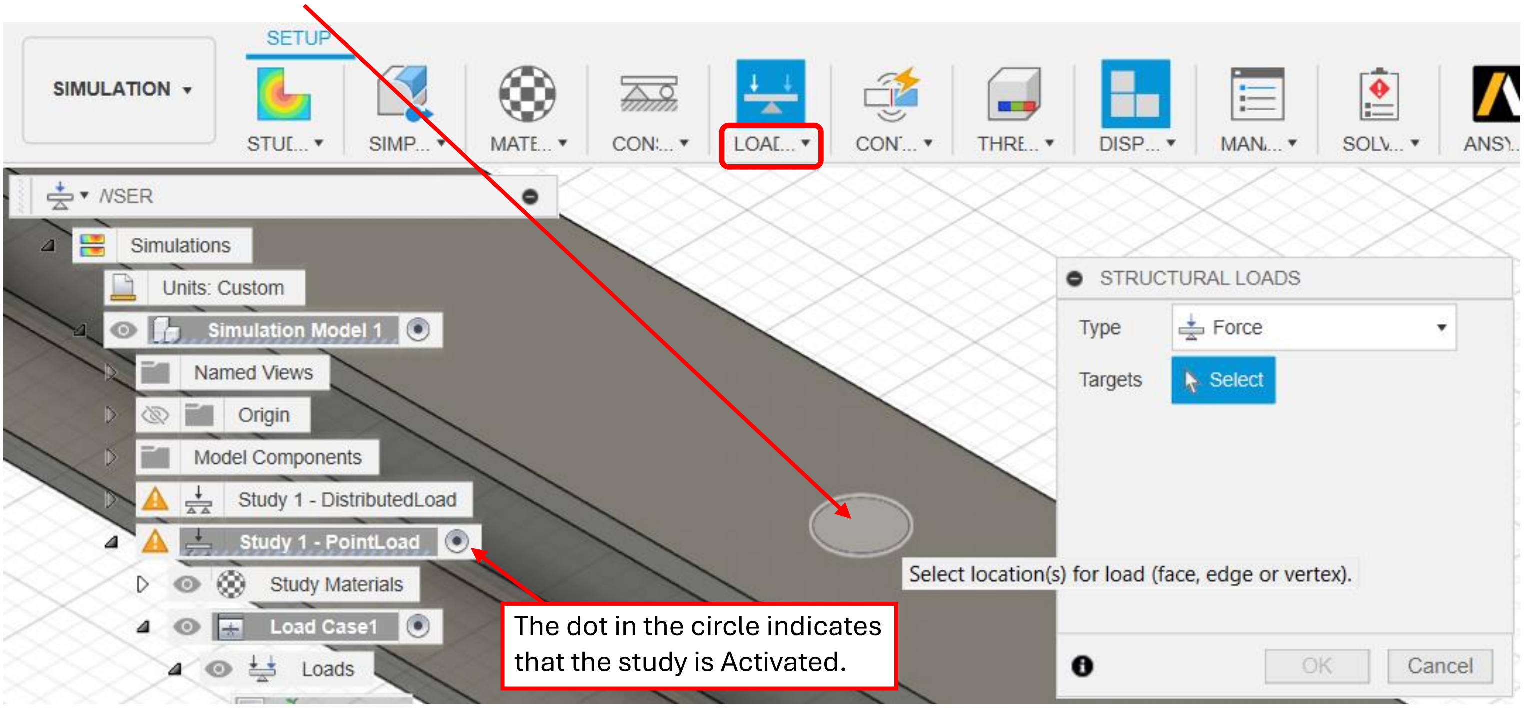


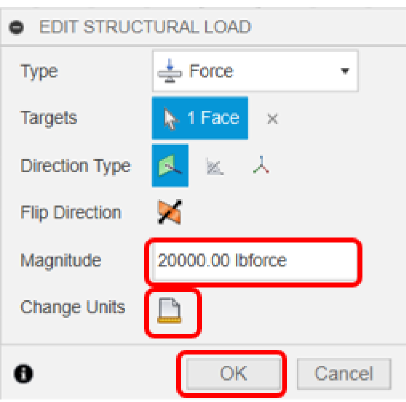

- click on **FINISH SIMPLIFY**
- yell **"What the Sigma!  Why was this called simplify?"**

In many cases the Simplify operations can be used to delete features to focus the simulation on the important part of the structure and thus simplifying the body.  Here we are using it to create a small surface to create a Point Load.



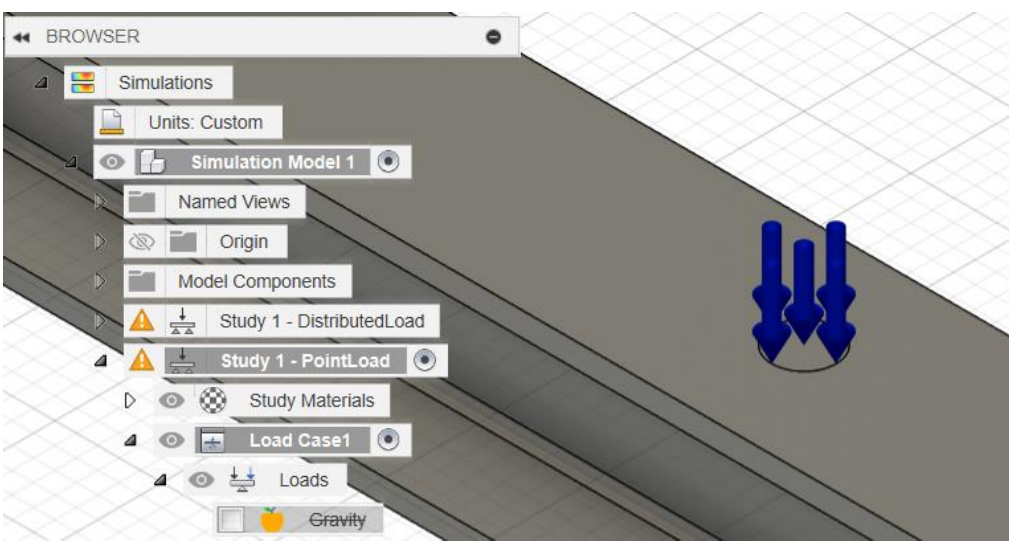
- enesure that the **PointLoad** study is **Activated**.  It should be because it was just created.
- note that the **arrow** next to the **Distributed Load study** was clicked to hide the study details to reduce clutter.
- from the **LOADS** menu select **Structural Loads**
- click on the **circular area** in the center of the beam



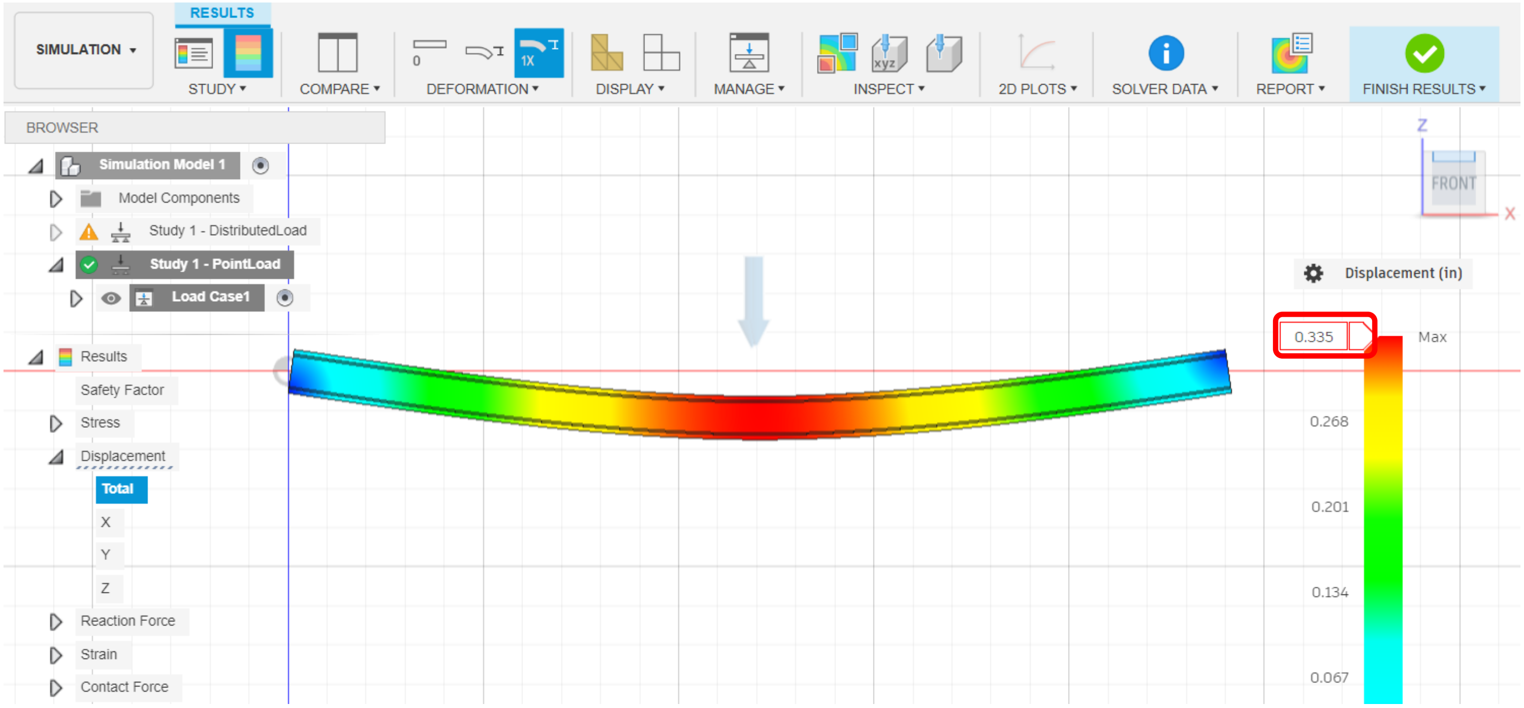

- enter the load value and click OK

- when hovering over the force arrow multiple arrows will show, but they should all be in the circular area.



- run the simulation as was done with the Distributed load and view the results

- note that the displacement should be higher than that of the Distributed Load. Here the displacement is **0.335** campared to **0.208 for the Distributed Load**.




## Deliverables

A single pdf file including the following:

**All of the folowing is based on your assigned beam. See the following page.**

1) A photo of your derivation for FlangeThickness and calculations for FlangeThickness
2) A screenshot of your Fusion Sketch of the beam cross-section showing dimensions
3) A photo of your beam formula calculation for **Simply Supported Maximum Deflection** for a **Distributed Load** and **Point Load**

4) A screenshot of your Fusion Stress simulation for a **Distributed Load** showing **Safety Factor**
5) A screenshot of your Fusion Stress simulation for a **Distributed Load** showing **Displacement** in **inches**
6) A screenshot of your Fusion Stress simulation for a **Distributed Load** showing **Von Mises Stress** in **ksi**

7) A screenshot of your Fusion Stress simulation for a **Point Load** showing **Safety Factor**
8) A screenshot of your Fusion Stress simulation for a **Point Load** showing **Displacement** in **inches**
9) A screenshot of your Fusion Stress simulation for a **Point Load** showing **Von Mises Stress** in **ksi**

10) A simple table showing the **difference between the Calculated Displacement(Deflection) and that from the FEA Displacement result**. Include a column for the difference in percent.

## Beam and Load Assignments

Your beam will match the ending number on the sticker on the base of your PC, e.g. HK-166-**14**

If your number is **1 to 10** use a Load of **20,000 lbs**, otherwise use **10,000 lbs**. Everyone's **Length is 20 ft**.

**Note that Flange Thickness is not specified. You have to figure that out from the information given. See the following page.**

| | Designation | Dimensions | | | | | Static Parameters: Moment of Inertia | |
|---|---|---|---|---|---|---|---|---|
| | Imperial (in x lb/ft) | Depth h (in) | Width w (in) | Web Thickness s (in) | Sectional Area ($in^2$) | Weight (lb/ft) | Ix ($in^4$) | Iy ($in^4$) |
| 1 | W 12 x 58 | 12.19 | 10.01 | 0.36 | 17.0 | 58 | 475 | 107 |
| 2 | W 12 x 53 | 12.06 | 9.995 | 0.345 | 15.6 | 53 | 425 | 95.8 |
| 3 | W 12 x 50 | 12.19 | 8.08 | 0.37 | 14.7 | 50 | 394 | 56.3 |
| 4 | W 12 x 45 | 12.06 | 8.045 | 0.335 | 13.2 | 45 | 350 | 50.0 |
| 5 | W 12 x 40 | 11.94 | 8.005 | 0.295 | 11.8 | 40 | 310 | 44.1 |
| 6 | W 12 x 35 | 12.50 | 6.56 | 0.3 | 10.3 | 35 | 285 | 24.5 |
| 7 | W 12 x 30 | 12.34 | 6.52 | 0.26 | 8.8 | 30 | 238 | 20.3 |
| 8 | W 10 x 77 | 10.60 | 10.190 | 0.530 | 22.6 | 77 | 455 | 154 |
| 9 | W 10 x 68 | 10.40 | 10.130 | 0.470 | 20.0 | 68 | 394 | 134 |
| 10 | W 10 x 60 | 10.22 | 10.080 | 0.420 | 17.6 | 60 | 341 | 116 |
| 11 | W 10 x 54 | 10.09 | 10.030 | 0.370 | 15.8 | 54 | 303 | 103 |
| 12 | W 10 x 49 | 9.98 | 10 | 0.340 | 14.4 | 49 | 272 | 93.4 |

| | Imperial (in x lb/ft) | Depth h (in) | Width w (in) | Web Thickness s (in) | Sectional Area ($in^2$) | Weight (lb/ft) | Ix ($in^4$) | Iy ($in^4$) |
|---|---|---|---|---|---|---|---|---|
| 13 | W 10 x 45 | 10.10 | 8.020 | 0.350 | 13.3 | 45 | 248 | 53.4 |
| 14 | W 10 x 45 | 10.10 | 8.020 | 0.350 | 13.3 | 45 | 248 | 53.4 |
| 15 | W 10 x 39 | 9.92 | 7.985 | 0.315 | 11.5 | 39 | 209 | 45.0 |
| 16 | W 8 x 67 | 9.00 | 8.280 | 0.570 | 19.7 | 67 | 272 | 88.6 |
| 17 | W 8 x 58 | 8.75 | 8.220 | 0.510 | 17.1 | 58 | 228 | 75.1 |
| 18 | W 8 x 48 | 8.5 | 8.110 | 0.400 | 14.1 | 48 | 184 | 60.9 |
| 19 | W 8 x 40 | 8.25 | 8.070 | 0.360 | 11.7 | 40 | 146 | 49.1 |
| 20 | W 10 x 33 | 9.73 | 7.960 | 0.290 | 9.71 | 33 | 170 | 36.6 |
| 21 | W 10 x 30 | 10.47 | 5.81 | 0.3 | 8.84 | 30 | 170 | 16.7 |
| 22 | W 10 x 26 | 10.33 | 5.770 | 0.26 | 7.6 | 26 | 144 | 14.1 |
| 23 | W 12 x 26 | 12.22 | 6.490 | 0.23 | 7.7 | 26 | 204 | 17.3 |
| 24 | W 12 x 22 | 12.31 | 4.03 | 0.26 | 6.5 | 22 | 156 | 4.7 |

## Determining the Flange Thickness (Tf)

The W-Beam catalog lists the dimensions of the beam cross-sections except for the Flange Thickness. However, it does list the Sectional Area. Calculate the Flange Thickness using the Depth, Width, Web Thickness and Sectional Area for your beam. Take a photo of your derivation of the Flange Thickness and the result using your beam dimensions. You can use the variable names as shown: **W, D, Tw, Tf,** and **A**.