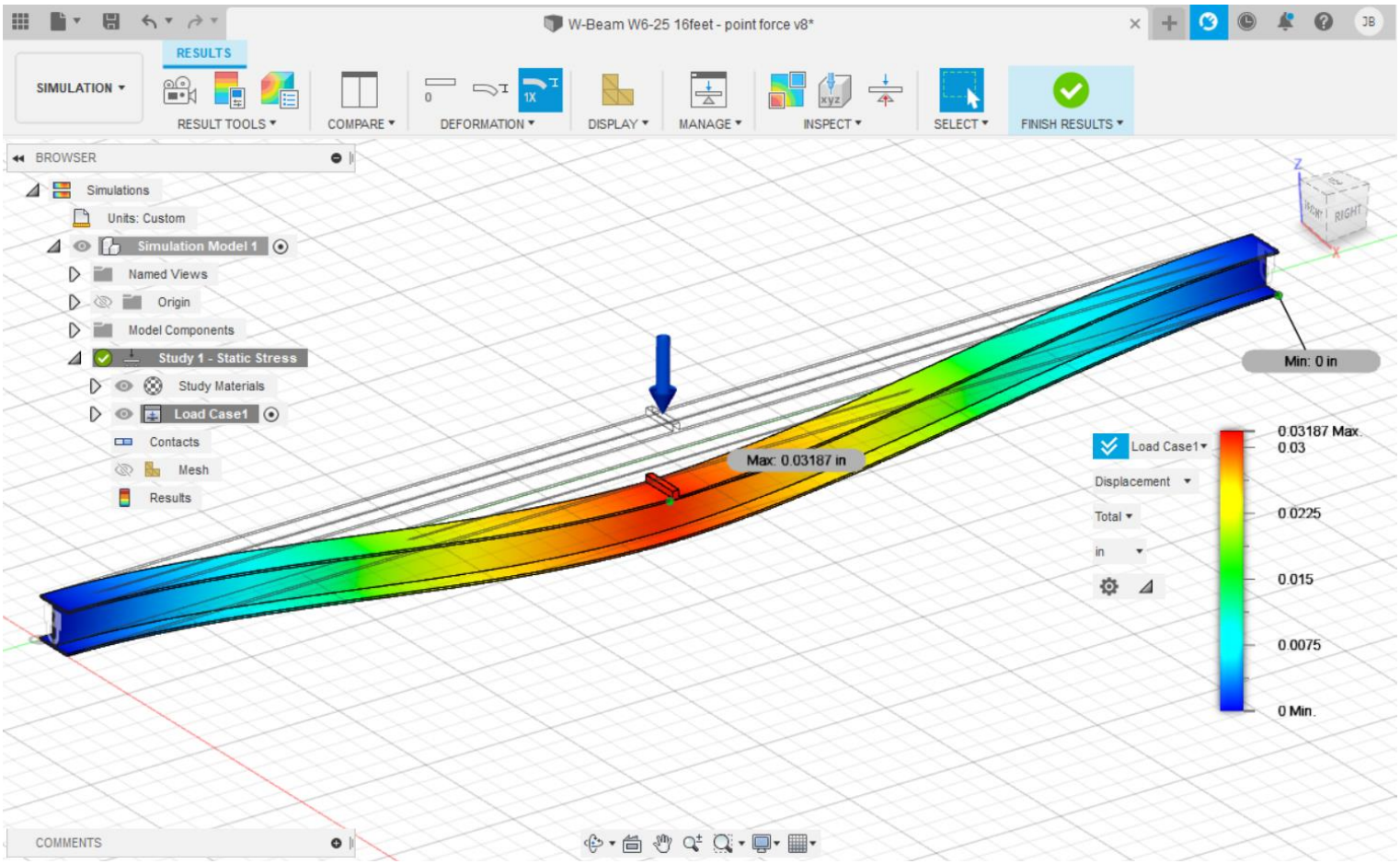
“I’m tired of MatLab. Let’s use Fusion 360 to make pretty beam pictures.”

FEA (Finite Element Analysis) for Static Stress

Fixed-Fixed Beam

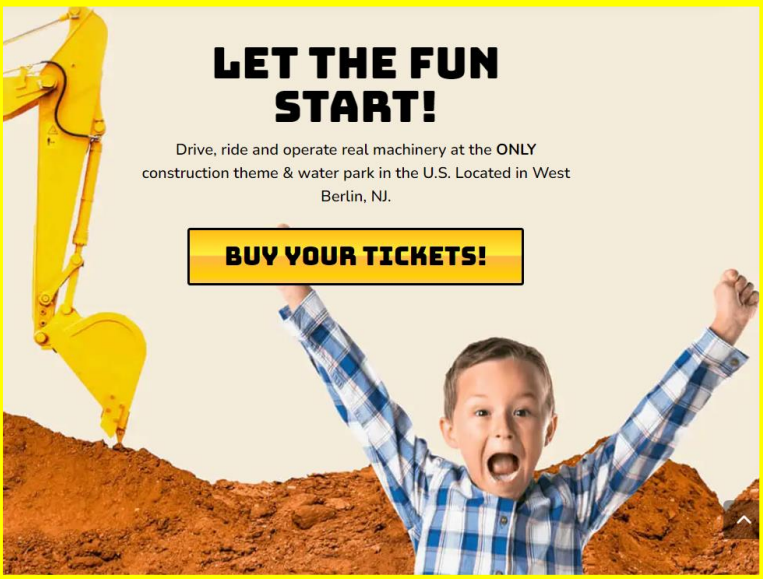


This lesson is sponsored by DiggerLand. **Get ready to build!**

DIGGERLAND
DRIVE RIDE SOAK SLIDE

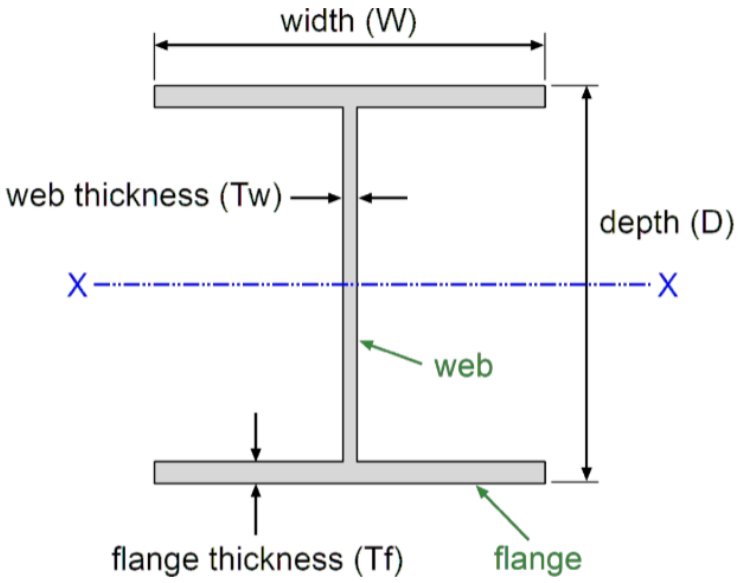
CONSTRUCTION THEME & WATER PARK

(856) 393-5962
100 Pinedge Drive
West Berlin, NJ 08091



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**.



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 **01 to 10** use a Load of **20,000 lbs**, otherwise use **10,000 lbs**. Everyone's **Length is 20 ft**.

	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)
01	W 10 x 112	11.36	10.415	0.755	32.9	112	716	236
02	W 10 x 100	11.1	10.340	0.680	29.4	100	623	207
03	W 10 x 88	10.84	10.265	0.605	25.9	88	534	179
04	W 10 x 77	10.60	10.190	0.530	22.6	77	455	154
05	W 10 x 68	10.40	10.130	0.470	20.0	68	394	134
06	W 10 x 60	10.22	10.080	0.420	17.6	60	341	116
07	W 10 x 54	10.09	10.030	0.370	15.8	54	303	103
08	W 10 x 49	9.98	10	0.340	14.4	49	272	93.4
09	W 10 x 45	10.10	8.020	0.350	13.3	45	248	53.4
10	W 10 x 39	9.92	7.985	0.315	11.5	39	209	45.0
11	W 10 x 33	9.73	7.960	0.290	9.71	33	170	36.6
12	W 10 x 30	10.47	5.81	0.3	8.84	30	170	16.7
13	W 10 x 26	10.33	5.770	0.26	7.6	26	144	14.1
14	W 10 x 22	10.17	5.750	0.240	6.5	22	118	11.4
15	W 10 x 19	10.24	4.020	0.250	5.6	19	96.3	3
16	W 10 x 17	10.11	4.010	0.240	5	17	81.9	3.6
17	W 10 x 15	9.99	4	0.230	4.4	15	68.9	2.9
18	W 10 x 12	9.87	3.960	0.190	3.5	12	53.8	2.2
19	W 8 x 67	9.00	8.280	0.570	19.7	67	272	88.6
20	W 8 x 58	8.75	8.220	0.510	17.1	58	228	75.1
21	W 8 x 48	8.5	8.110	0.400	14.1	48	184	60.9
22	W 8 x 40	8.25	8.070	0.360	11.7	40	146	49.1

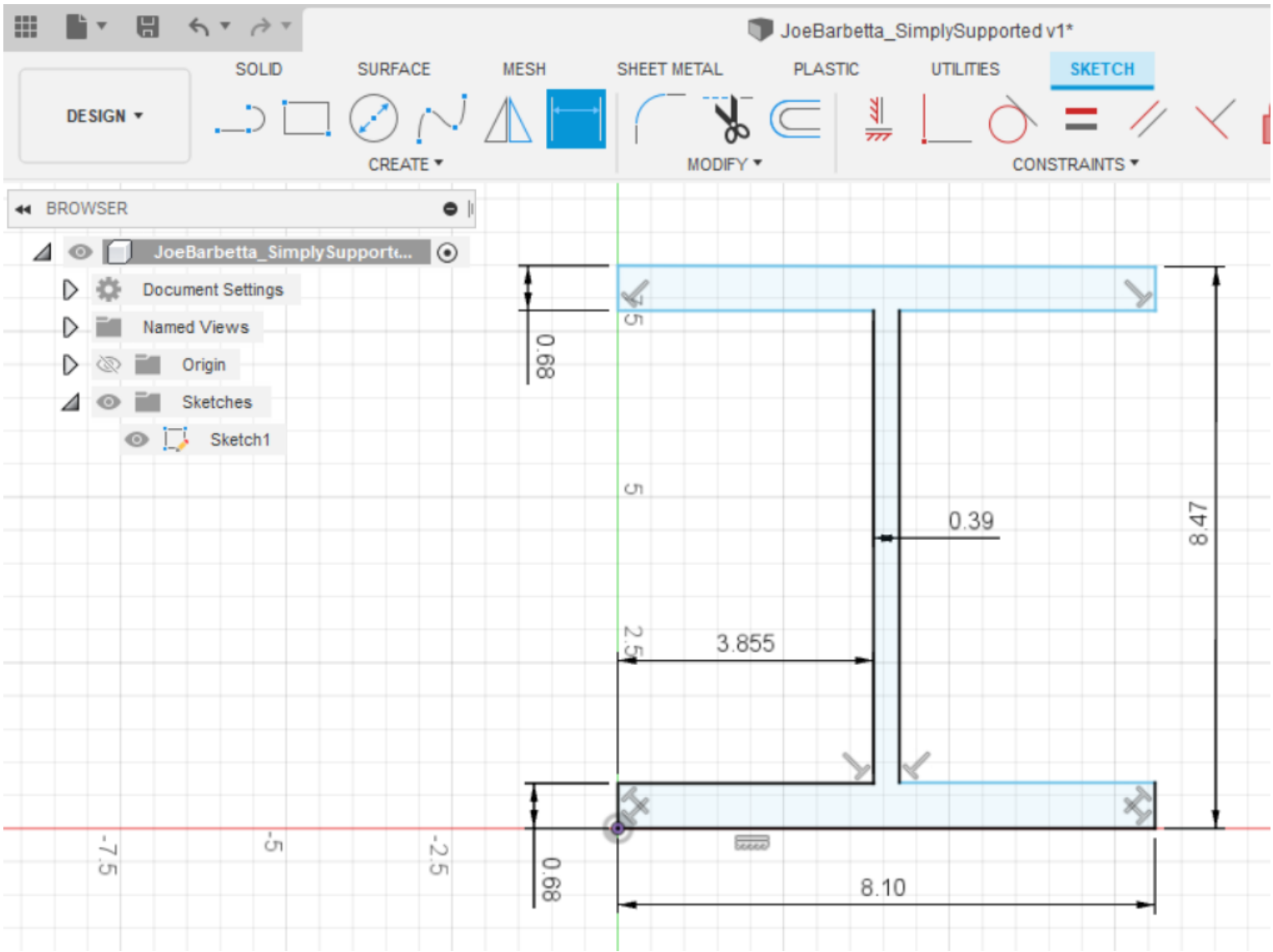
For your **Fixed-Fixed** Beam **calculate the Maximum Deflection** for both a Point Load and Distributed Load using the proper Beam Deflection formulas.

Design your beam in Fusion 360.

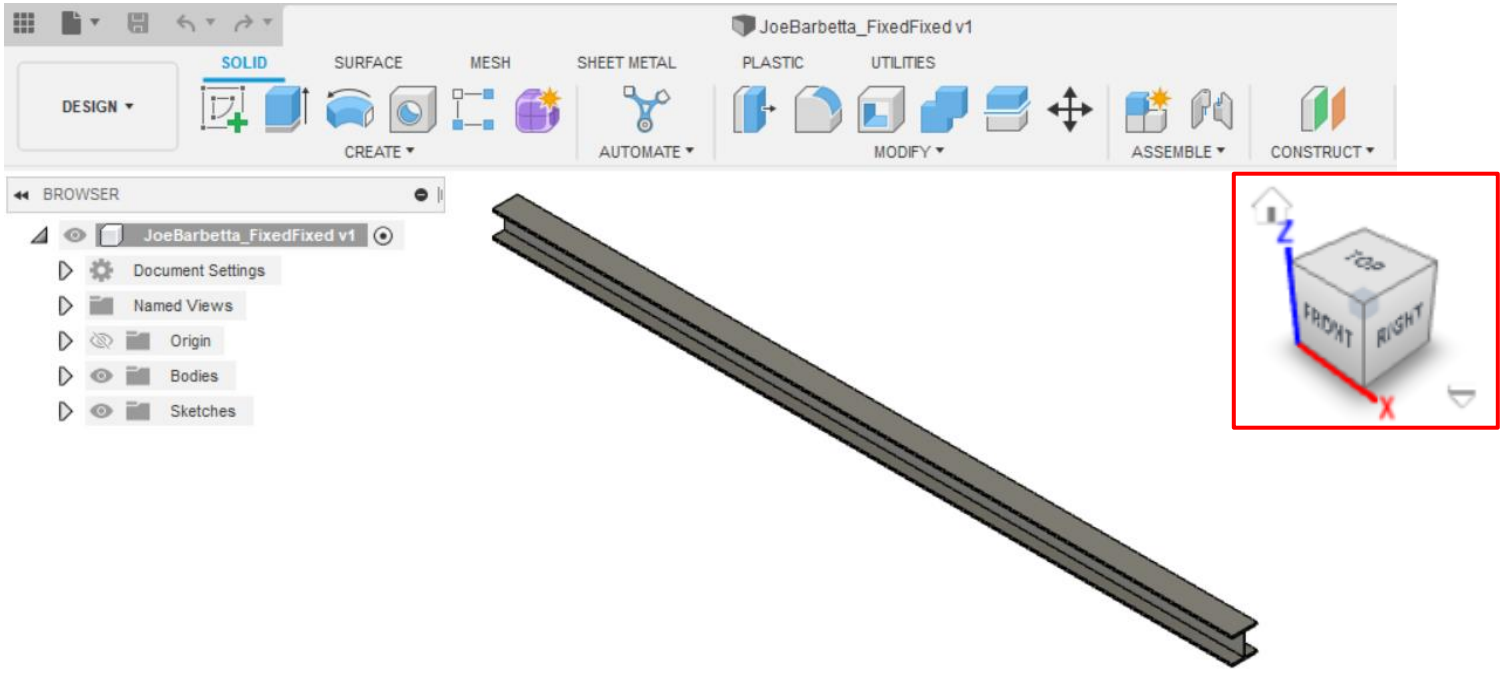
Name the file with your name followed by **_FixedFixed**, e.g. JoeBarbetta_FixedFixed

Take a **screenshot of the *Sketch*** for your beam showing the dimensions for the Depth, Width, WebThickness, and FlangeThickness.

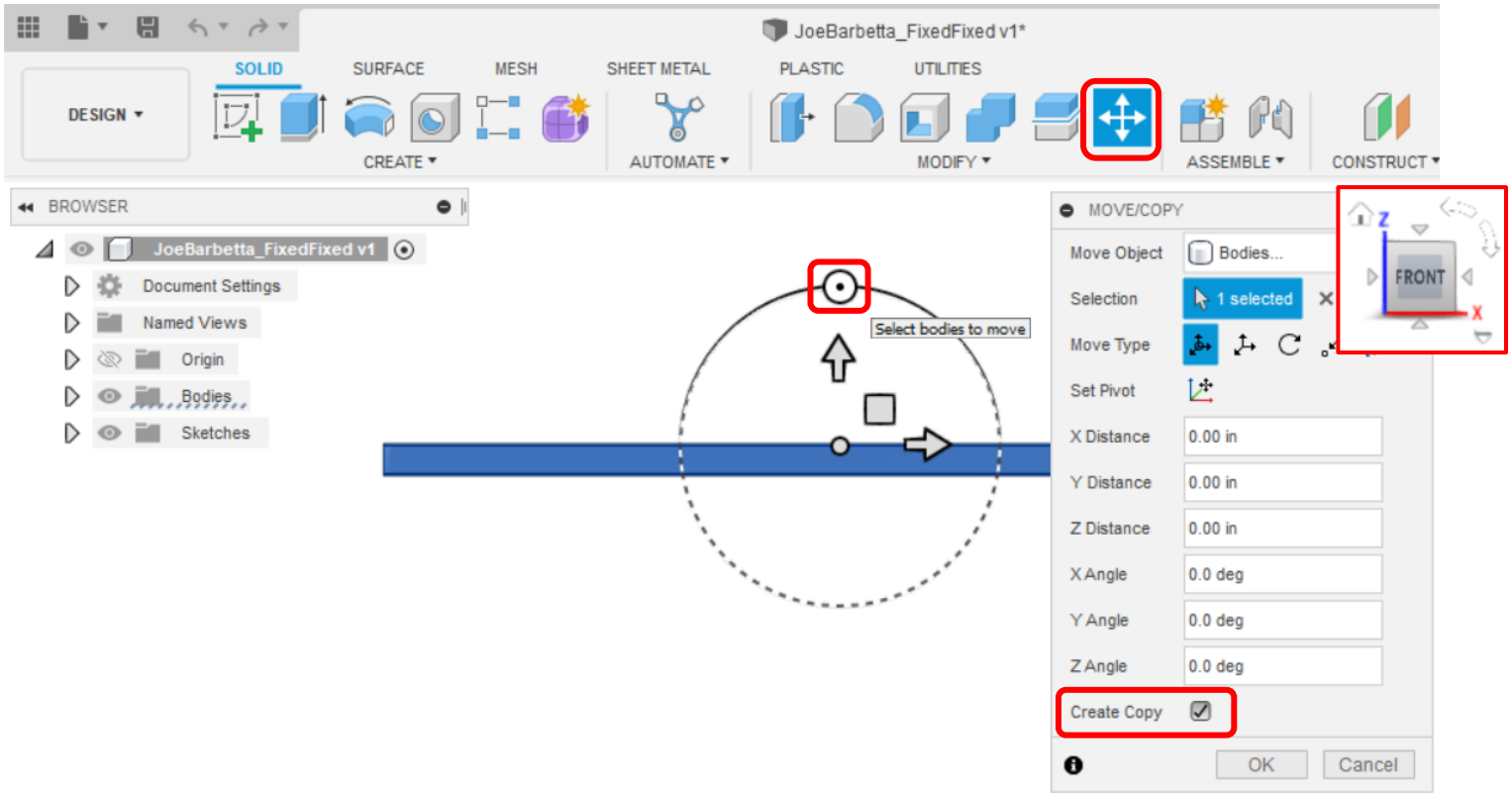
Extrude the beam according to your beam Length and set the material to **A36 Steel**.



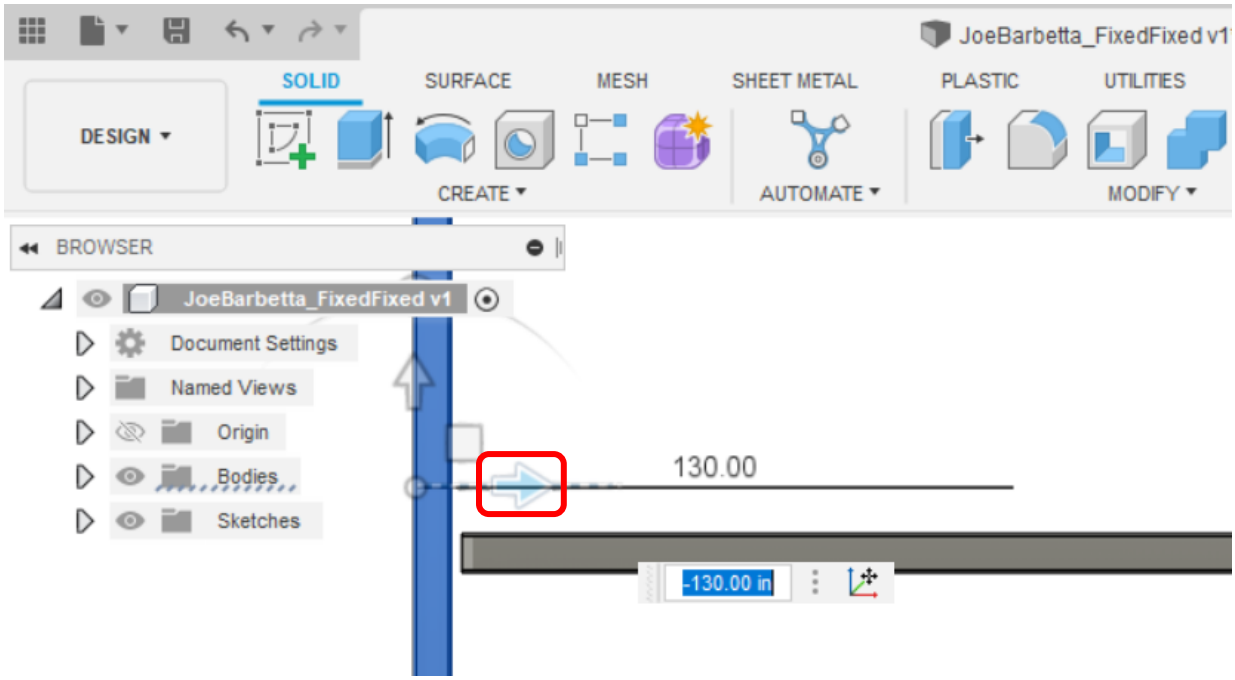
- click on the **Home** icon at the upper left of the **View Cube**. The length of the beam should align with the X Axis.



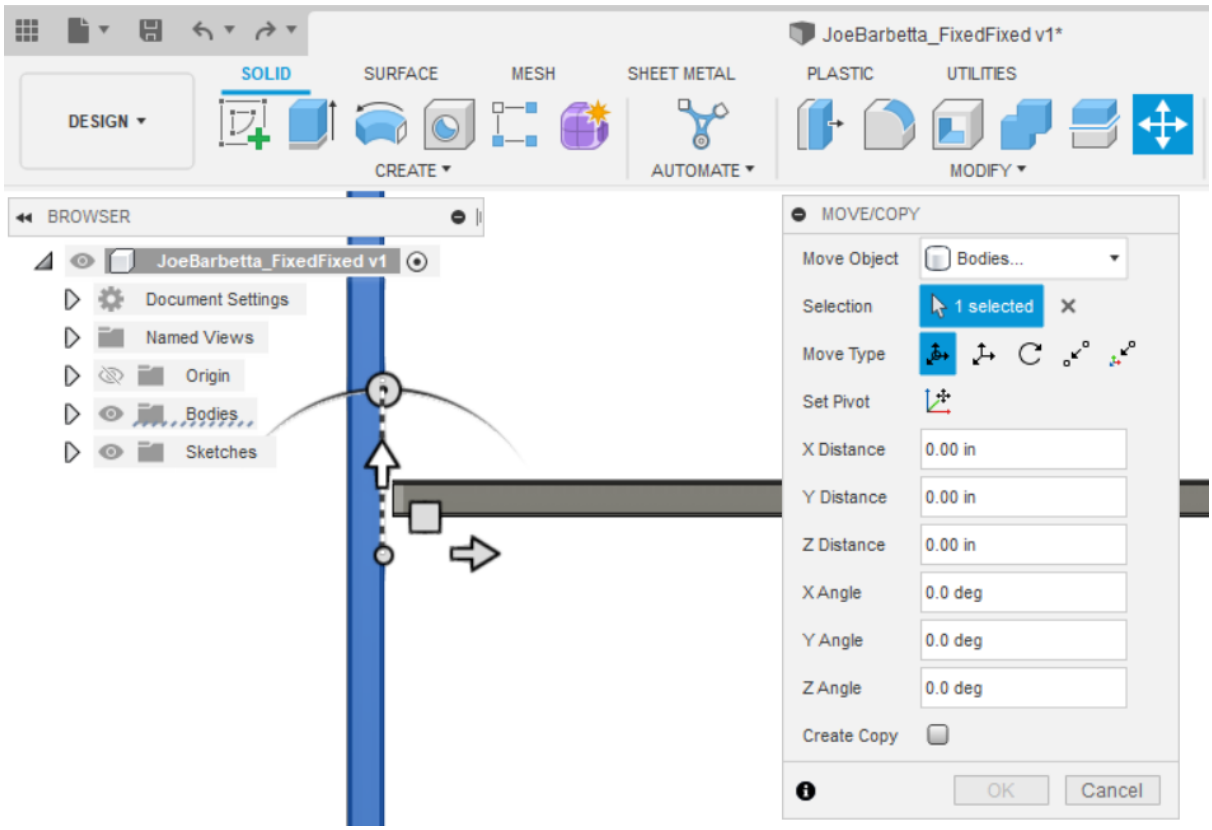
- click on the **FRONT** of the **View Cube**. Click on the **rotation arrows** at the upper right of the View Cube if needed. You should now be looking at the a side view (web) of the beam.
- select the **Move/Copy** tool. If it's not there, select it from the **MODIFY** pull-down menu.
- click on the **top center** of the beam. The point should snap to its center when you are near it.
- enable **Create Copy** on the MOVE/COPY window
- move the **Rotation** point to rotate the cloned beam **1600 NATO mils** and click **OK**



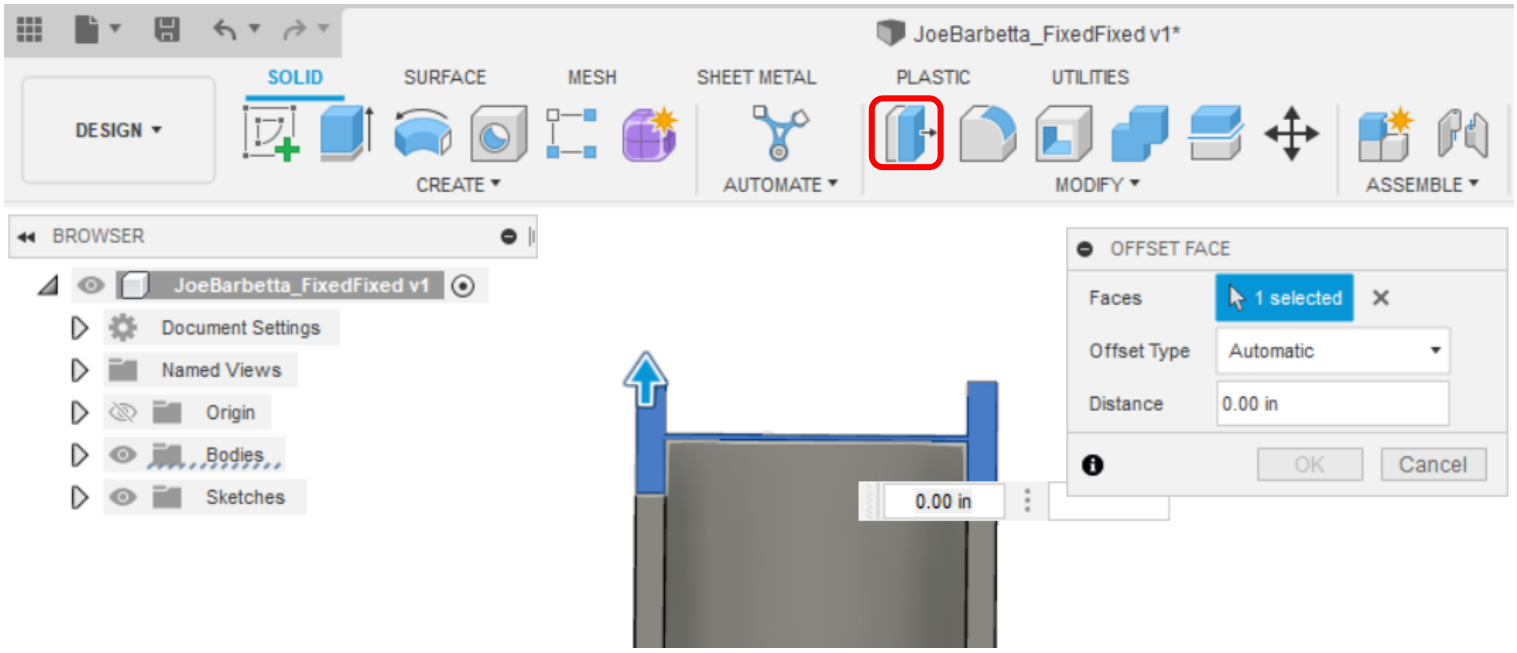
- select the **Move/Copy** tool again (**without Create Copy**) and move the **Horizontal move arrow** to move the cloned beam so that is just past the original beam.



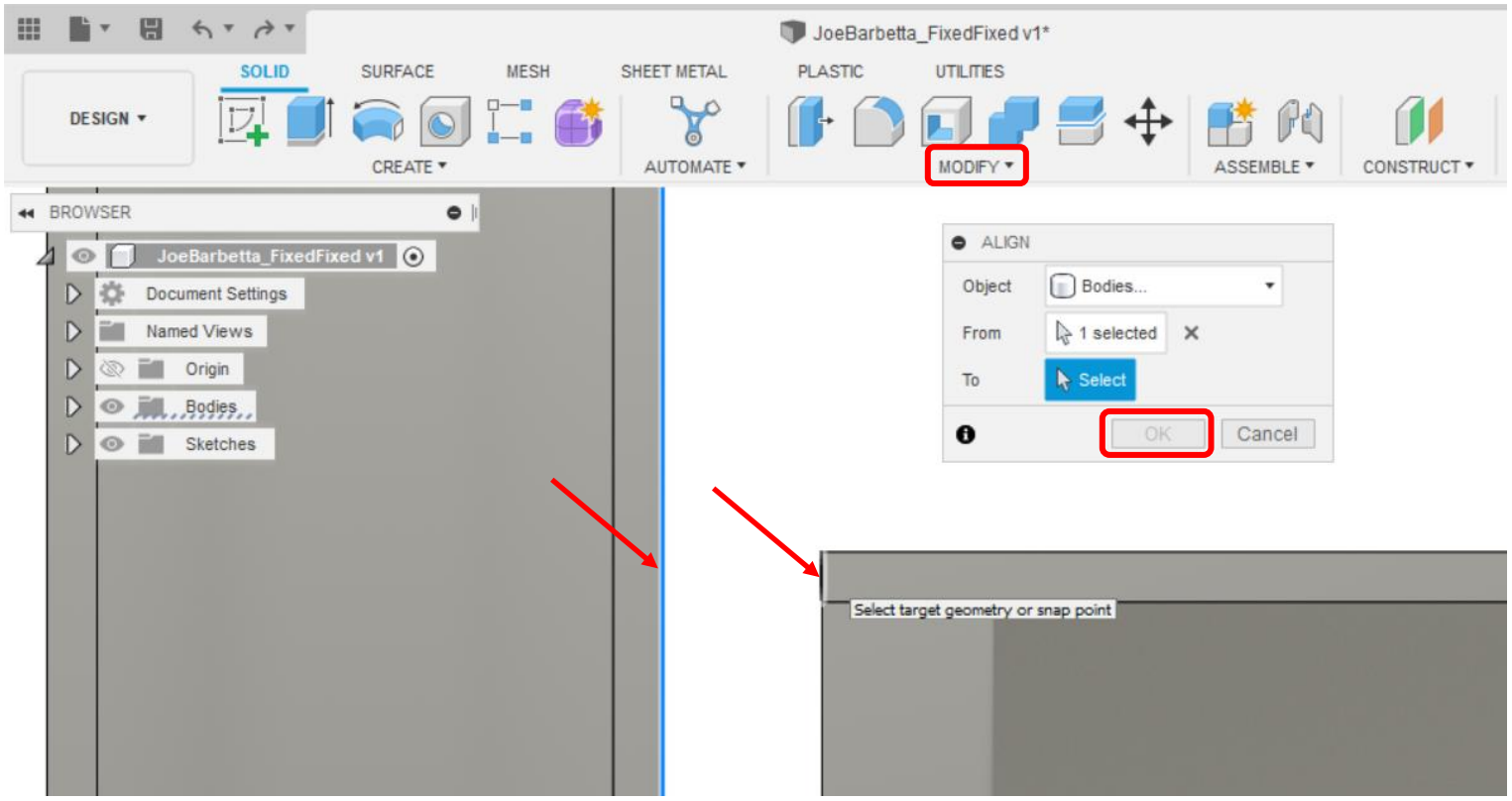
- use the **Move/Copy** tool again (**without Create Copy**) and move this beam up by **60 in**.



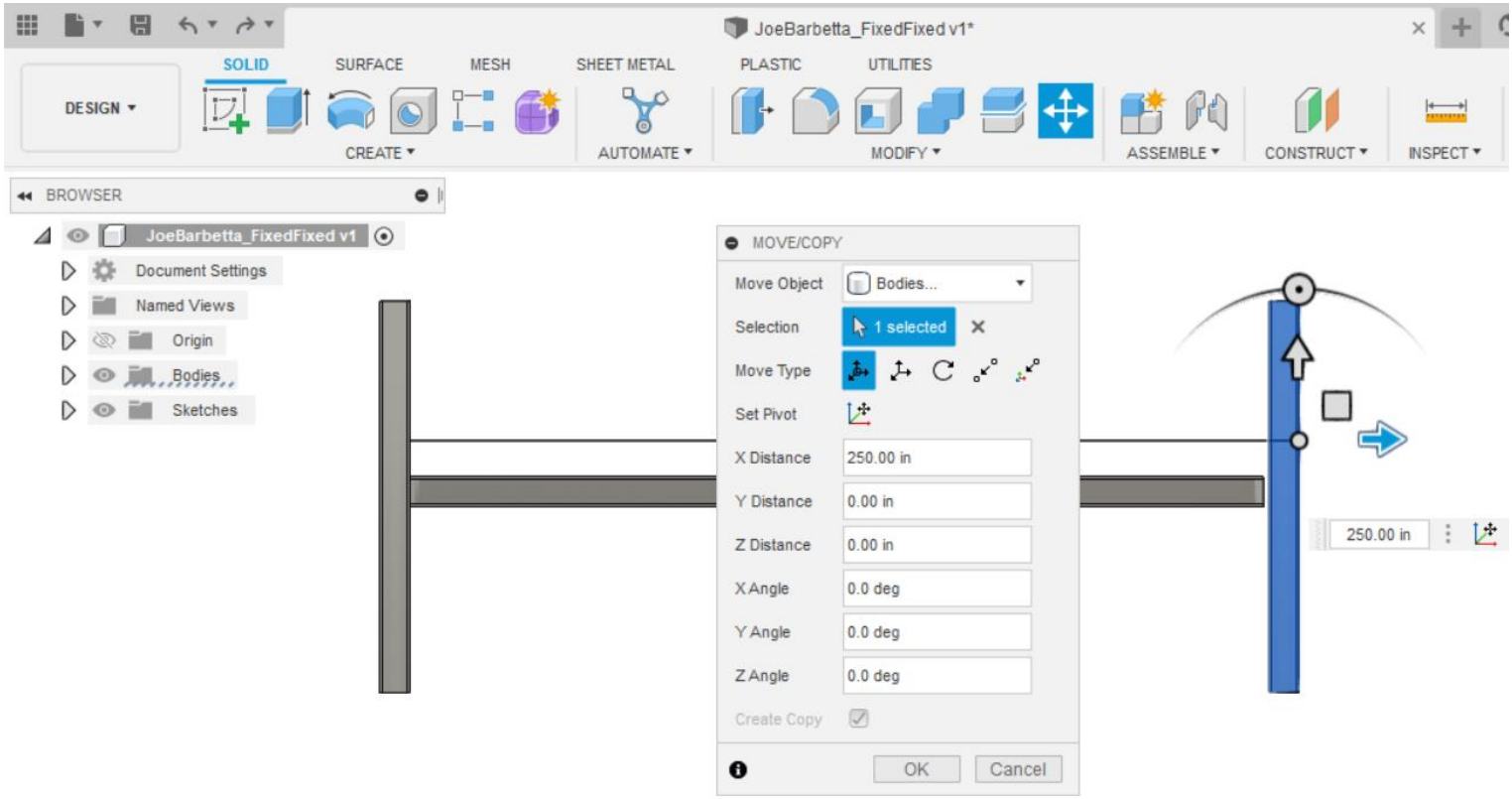
- zoom into the end face of the vertical beam
- select the **Press Pull** tool, click on the **end face**, and enter **-130** (note the minus sign)



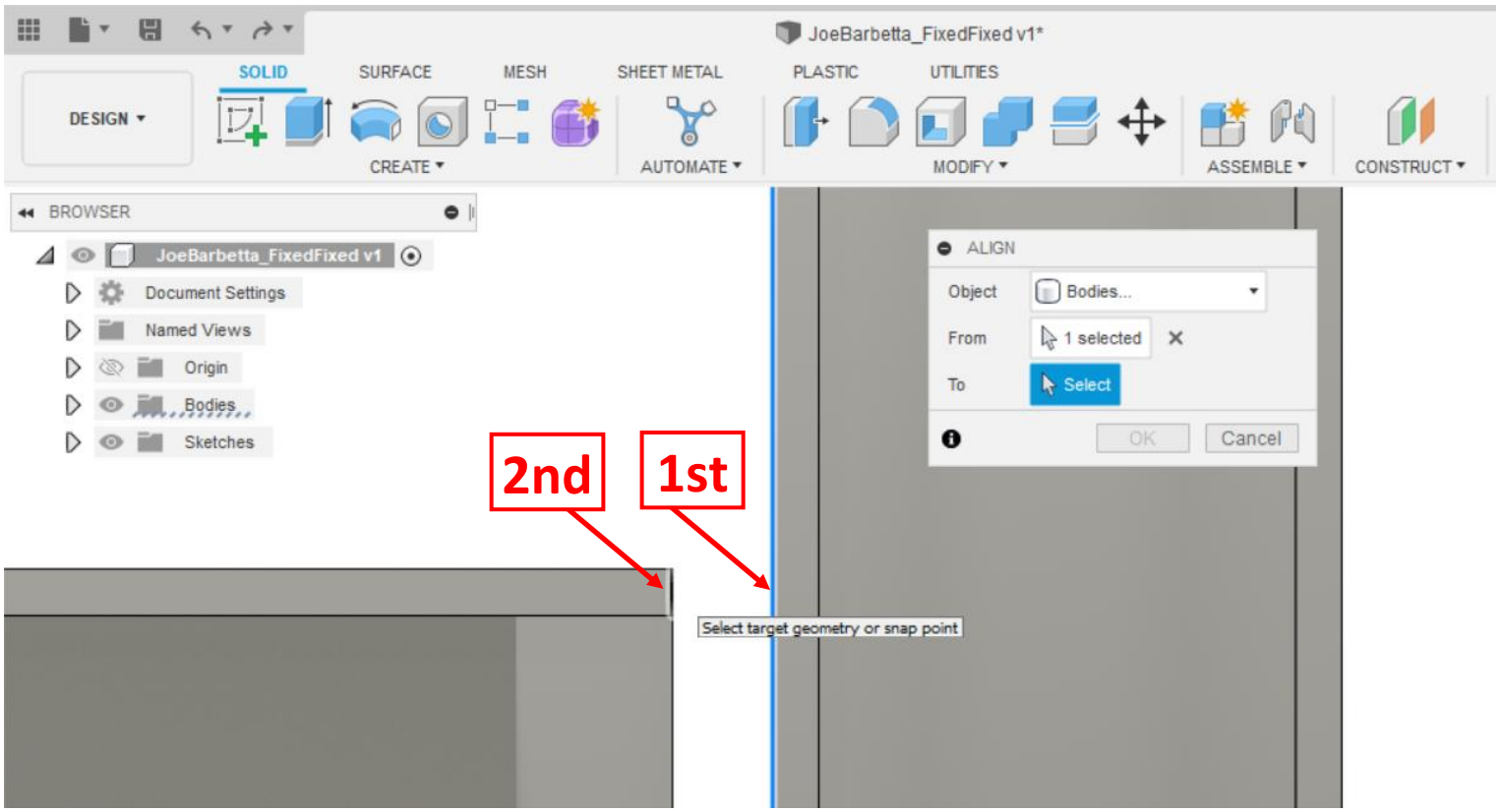
- zoom into the space between the beams and from the **MODIFY** pull-down menu select the **Align** tool
- click on the right edge of the vertical beam and the edge of the end of the top flange as shown to join the beams
- click **OK**



- zoom out, move a clone of the vertical beam just past the other end of the horizontal beam



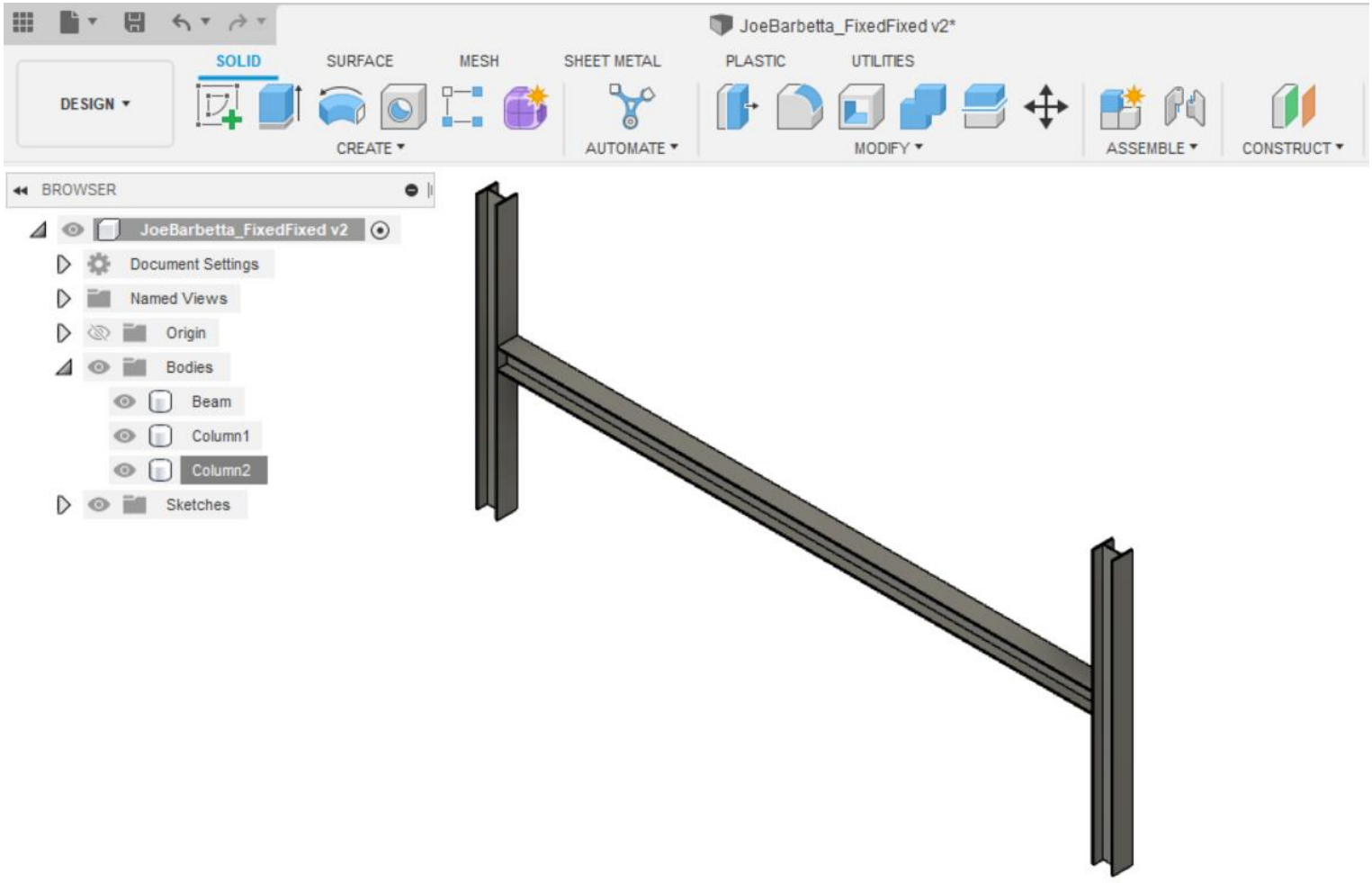
- zoom into the space between the beams and from the **MODIFY** pull-down menu select the **Align** tool
- click on the right edge of the vertical beam and the edge of the end of the top flange as shown to join the beams. It's important to **click on the edge of the vertical beam first**, otherwise the horizontal beam will be pulled away from the other beam.



A home view of the beam structure should look like that below. Note that your proportions may vary slightly due to beam size differences.

- click on the **Expand Arrow** in the BROWSER to open the **Bodies** folder
- right-click on each name, select Rename, and change the names to Beam, Column1 and Column2. When you hover over the names, the corresponding beam will lighten.

- save your project in case Fusion 360 freezes or crashes during simulation

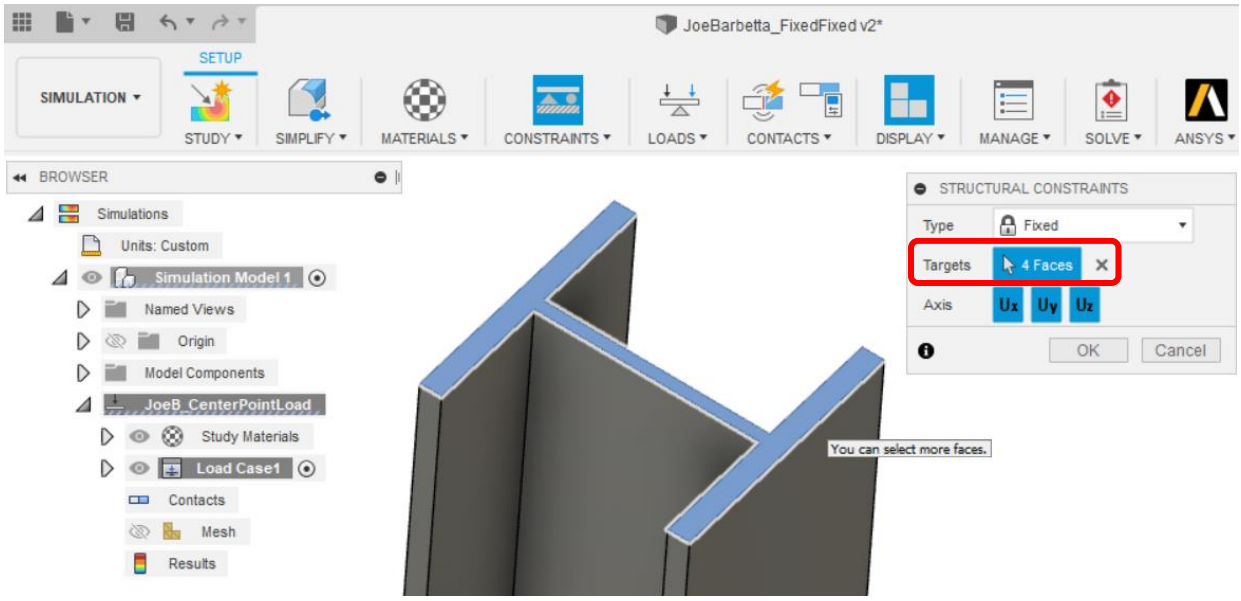


From the top left Workspace menu switch from **DESIGN** to **SIMULATION**

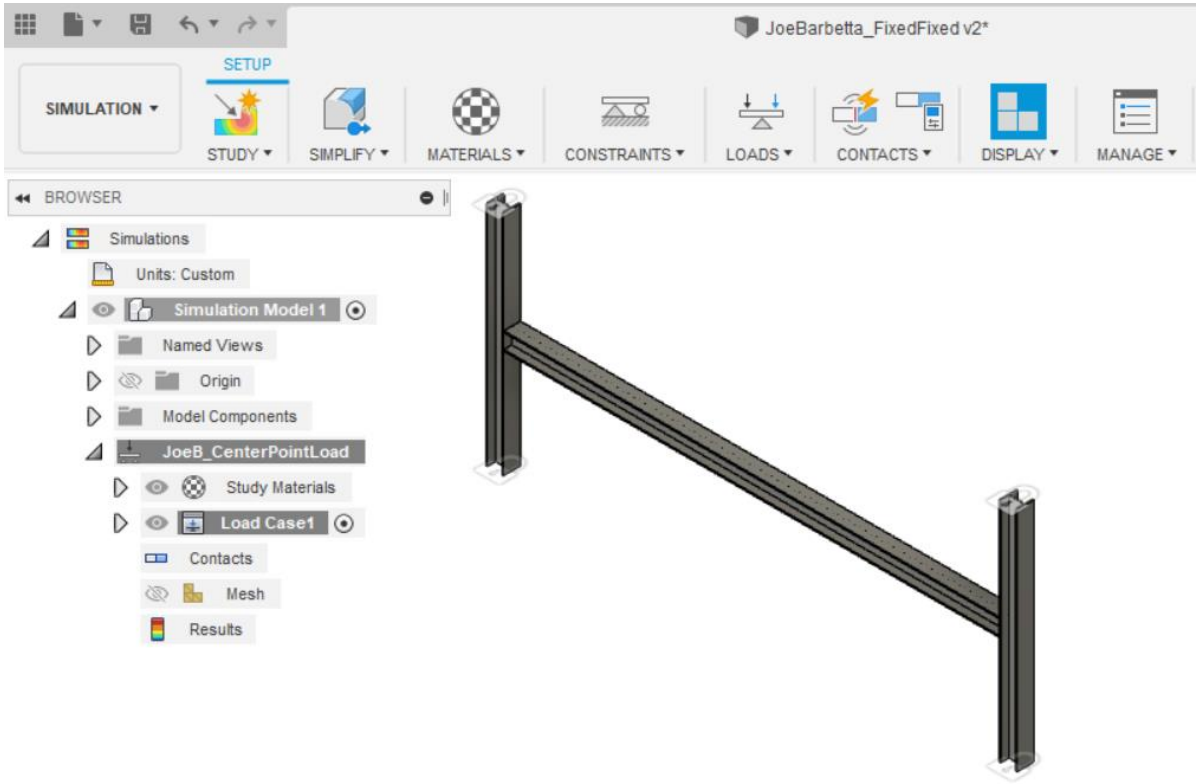
You will be presented with Simulation Study options. Spend a few seconds admiring the vast array of simulation options and then select **Static Stress** and click **Create Study**. Change the name to your first name and last name initial followed by _CenterPointLoad, e.g. **JoeB_CenterPointLoad** and click **OK**.

From the top **CONTACTS** pull-down menu select **Automatic Contacts**. Keep the default **Contact Detection Tolerance** of 0.10 mm and click **Generate**. Because the Align tool was used in the Design Workspace, there should be 0 space between the beam and columns and thus 0.10 mm is fine. Note that nothing visible will happen after the contact generation, but for simulation the beam is now connected (and constrained) by the columns. This would be similar to welded beam junctions.

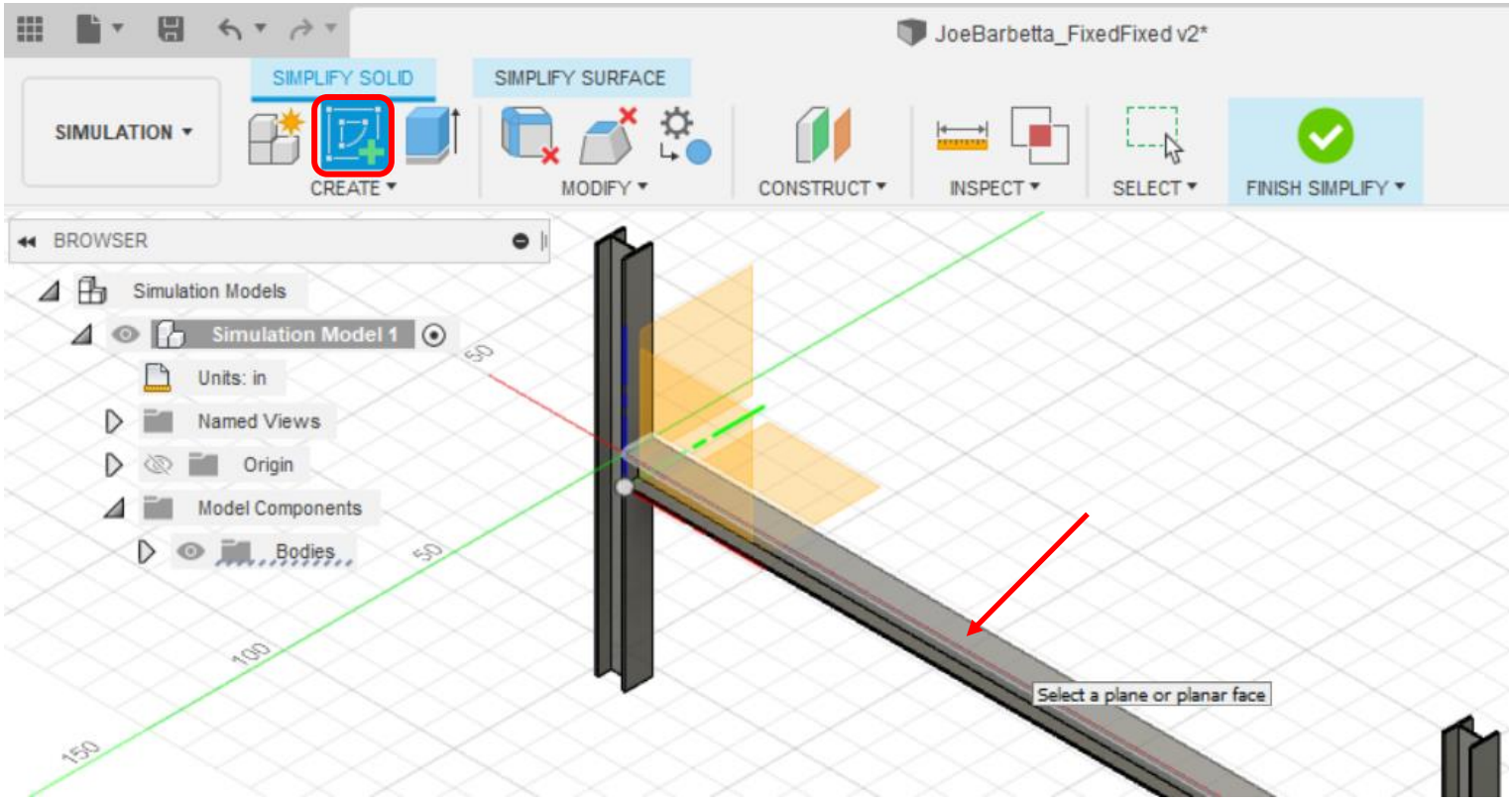
From the **CONSTRAINTS** pull-down menu select **Structural Constraints**. Zoom into the **end faces of each Column** to select them, being careful to **select the Faces and Not the Edges**. Note that **Targets** will then show **4 Faces** after each has been selected. The Axis setting can remain with the default of all enabled (highlighted).



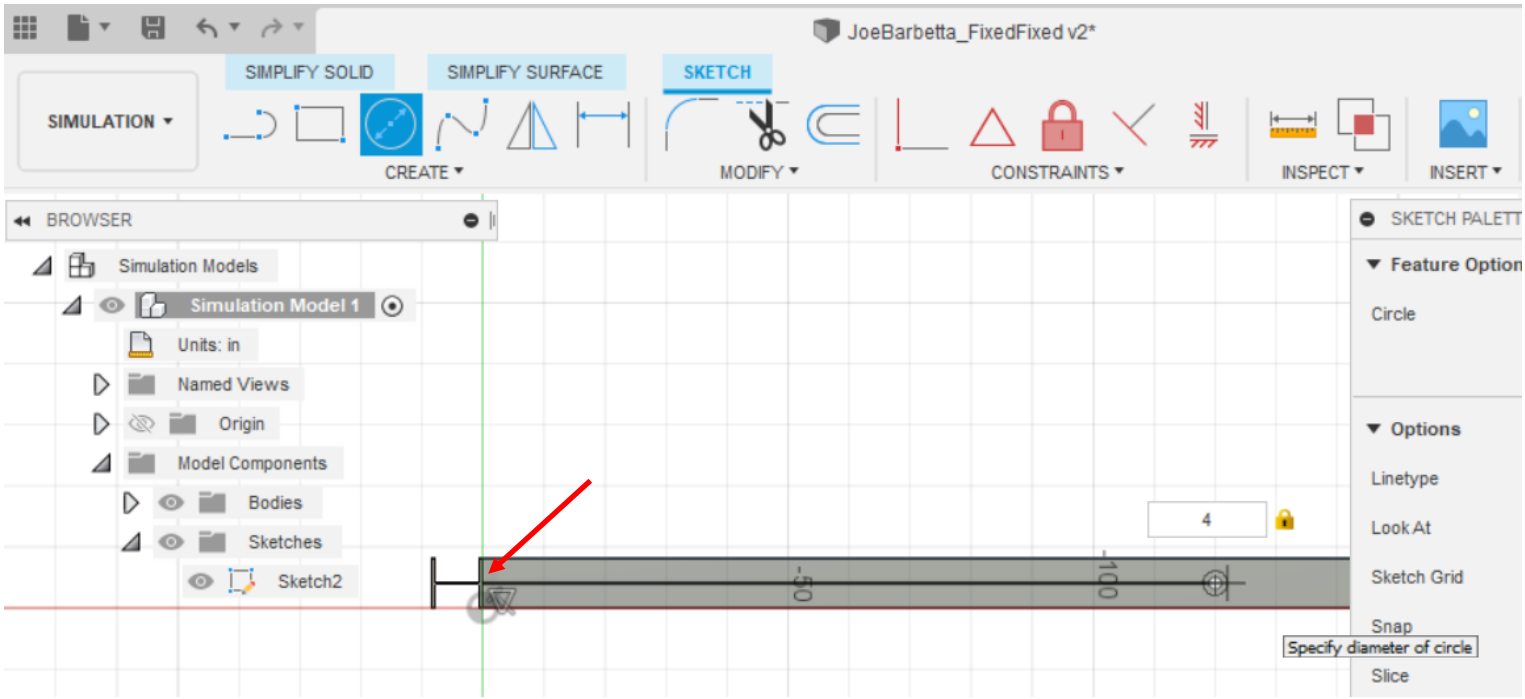
- click **OK** and return to the **Home** view. There should be a lock icon at each end of the columns.



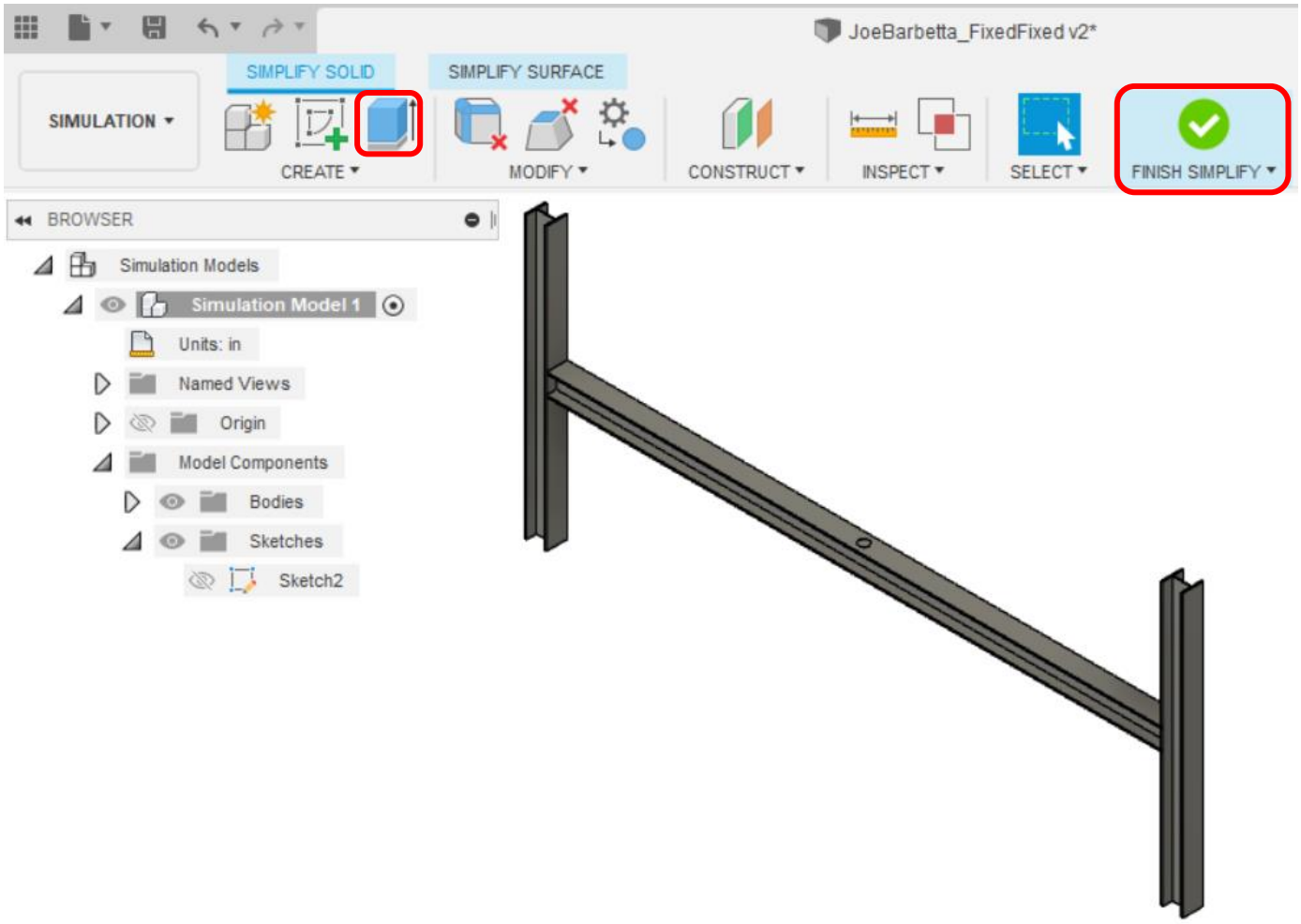
- select the **Simplify** tool and then the **Create Sketch** tool and then click on the **top surface** of the beam.



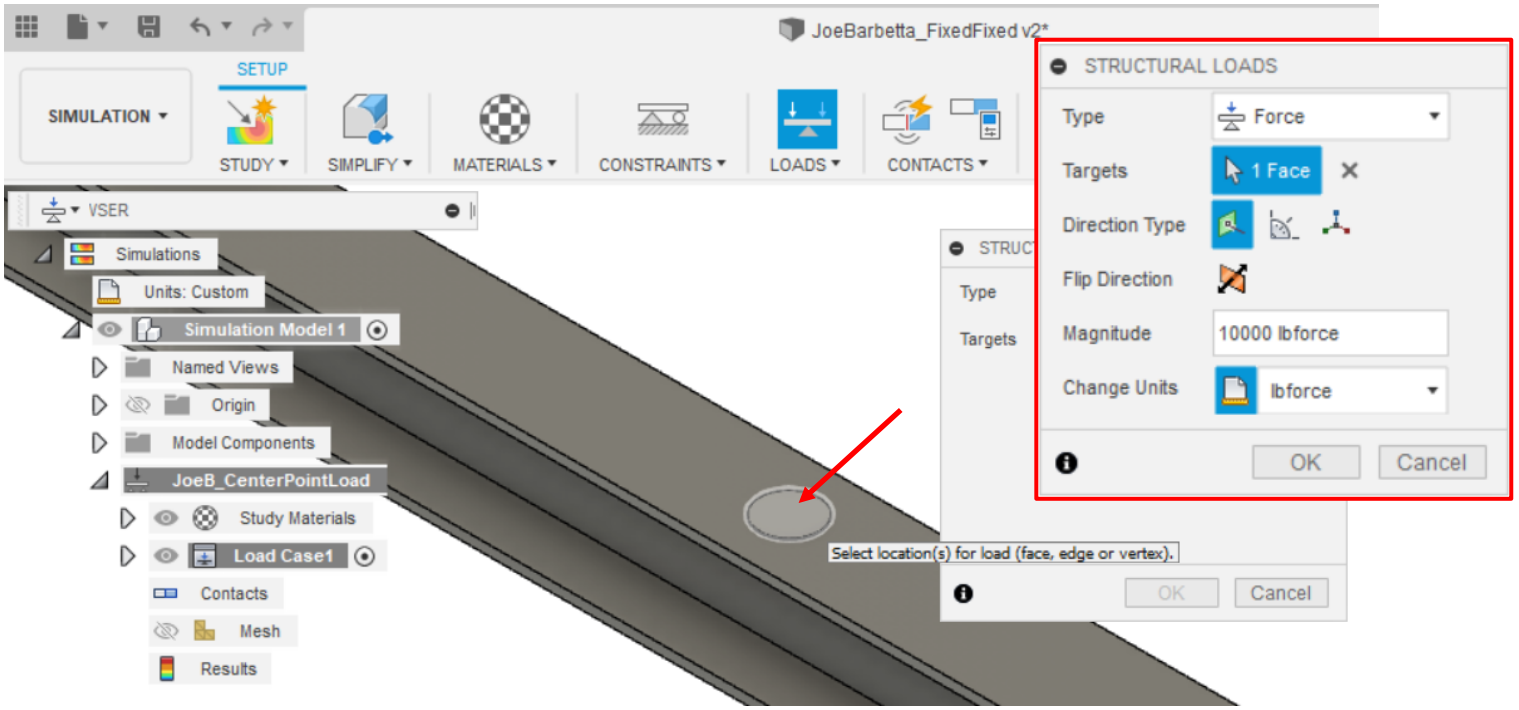
- create a **line** along the center of the beam from its end to **1/2 the beam length**. Don't click Finish Sketch.
- create a **circle** with its center at the end of the line with a diameter of **4**. Click **Finish Sketch**.



- select the **Extrude** tool and use a value of **0.1**. This will create a slightly raised face to apply a load to.
- click **FINISH SIMPLIFY**



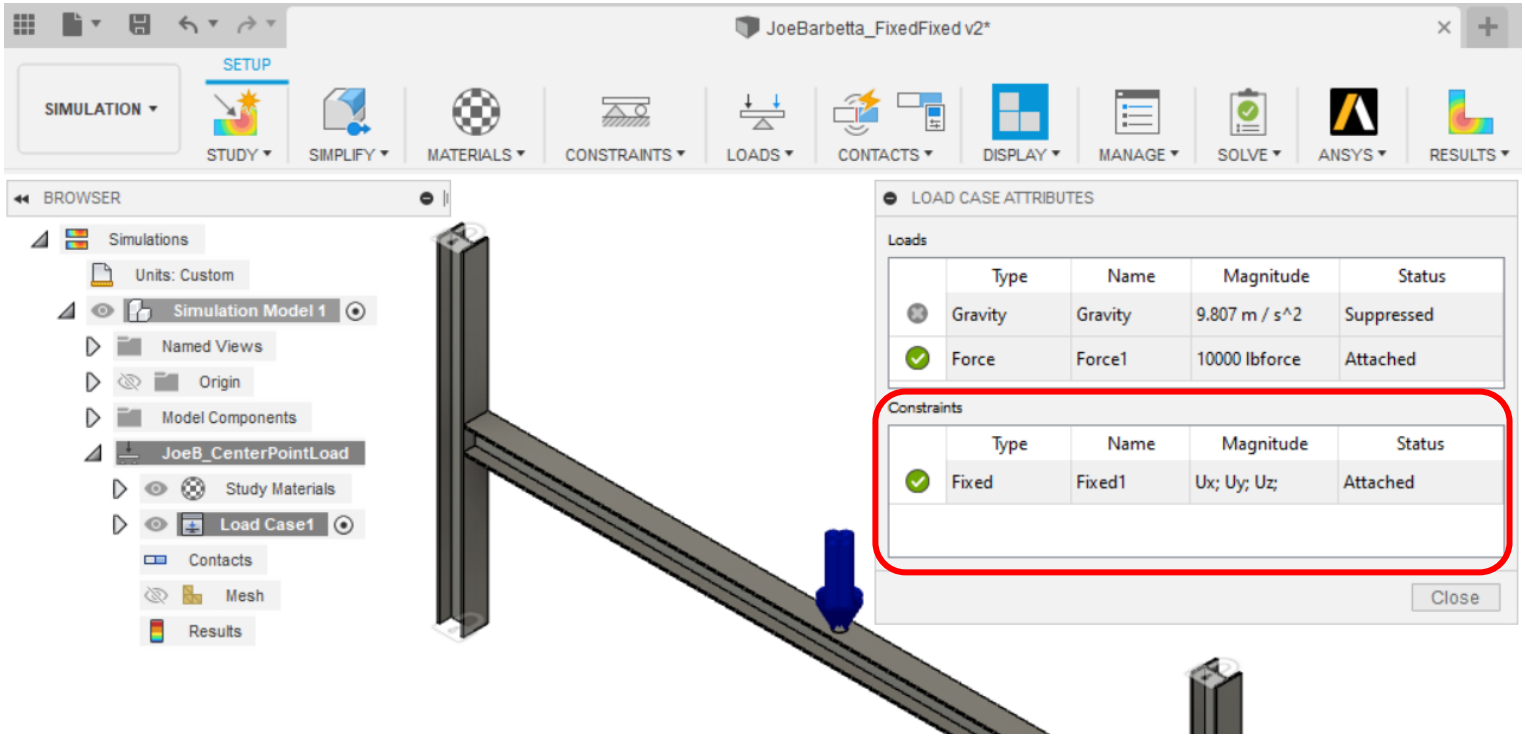
- from the **LOADS** pull-down menu select **Structural Loads**
- zoom into the center face and click on the inside of the circle to select it.
- change the units to **lbforce**, enter your **assigned load**, and click **OK**



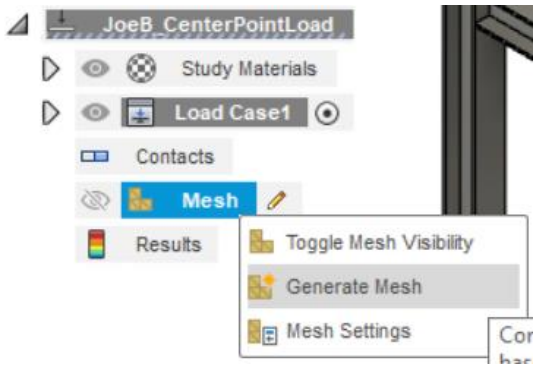
- return to the Home view and from the **MANAGE** pull-down menu select **Load Case Attributes**. Note that there is a single item in the lower **Constraints** section, which represents all four constraints applied earlier. If there is a sperate one for each column end, this is fine.

- hover over the blue **Load Arrow**. They should multiply only over the center circle as shown.

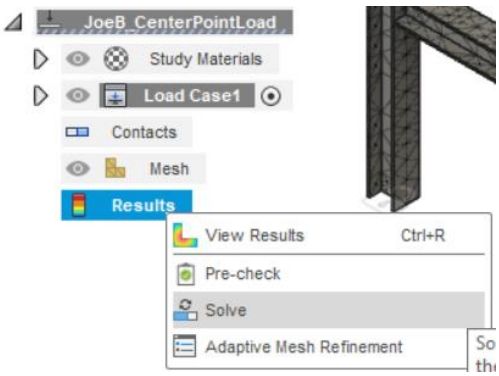
Note that one can always double-click on any field of a **Load** or **Constraint** to change an attribute. To delete a Load or Constraint, one must delete its icon at its point on the design.



- right-click on **Mesh** and select **Generate Mesh**

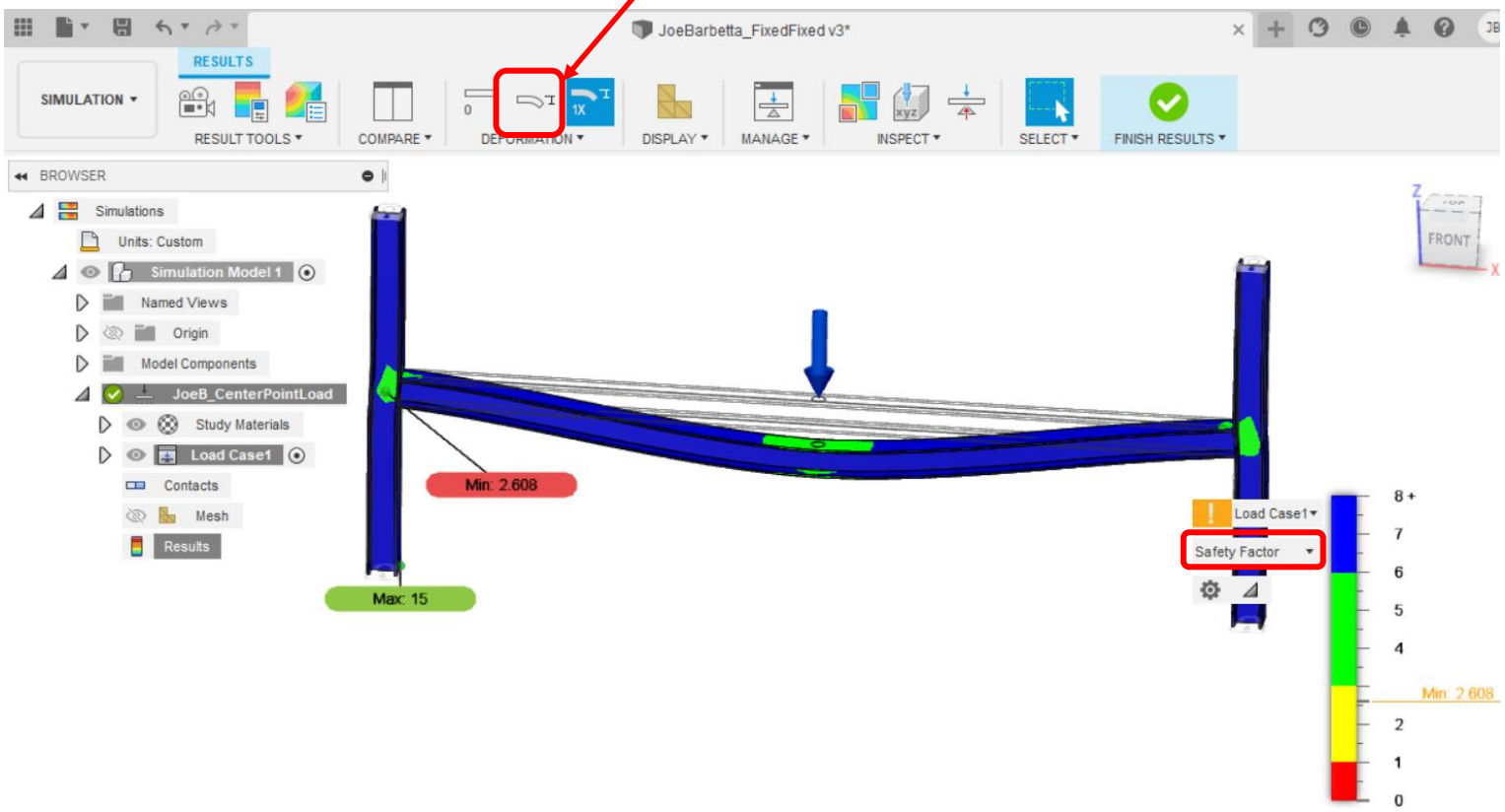


- right-click on **Results** and select **Solve**

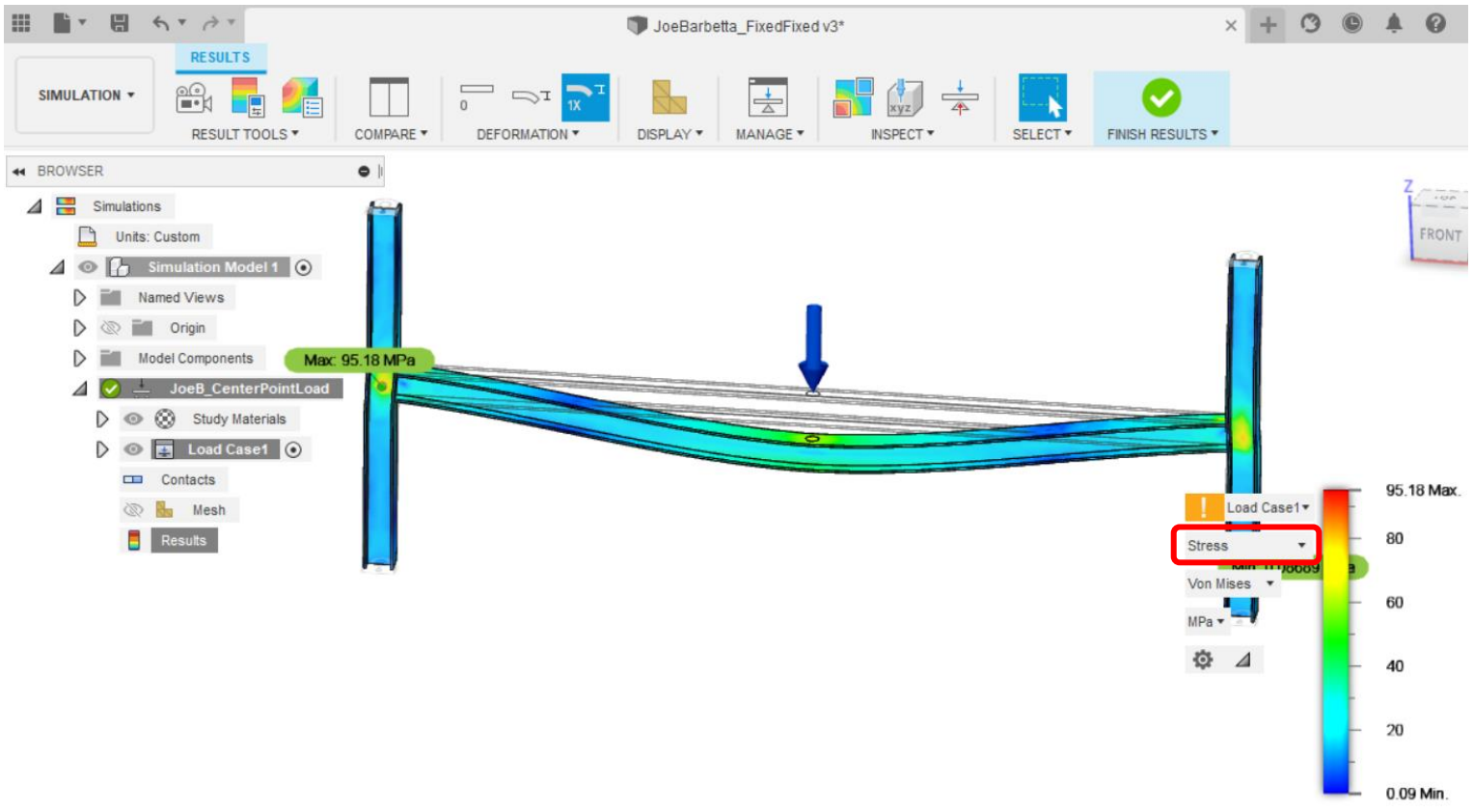


- if someone is looking over your shoulder and criticizes your for creating saggy structural designs, you must quickly point out that this is an exaggerated deformation. You can click the **Actual** icon to view the true deformation.

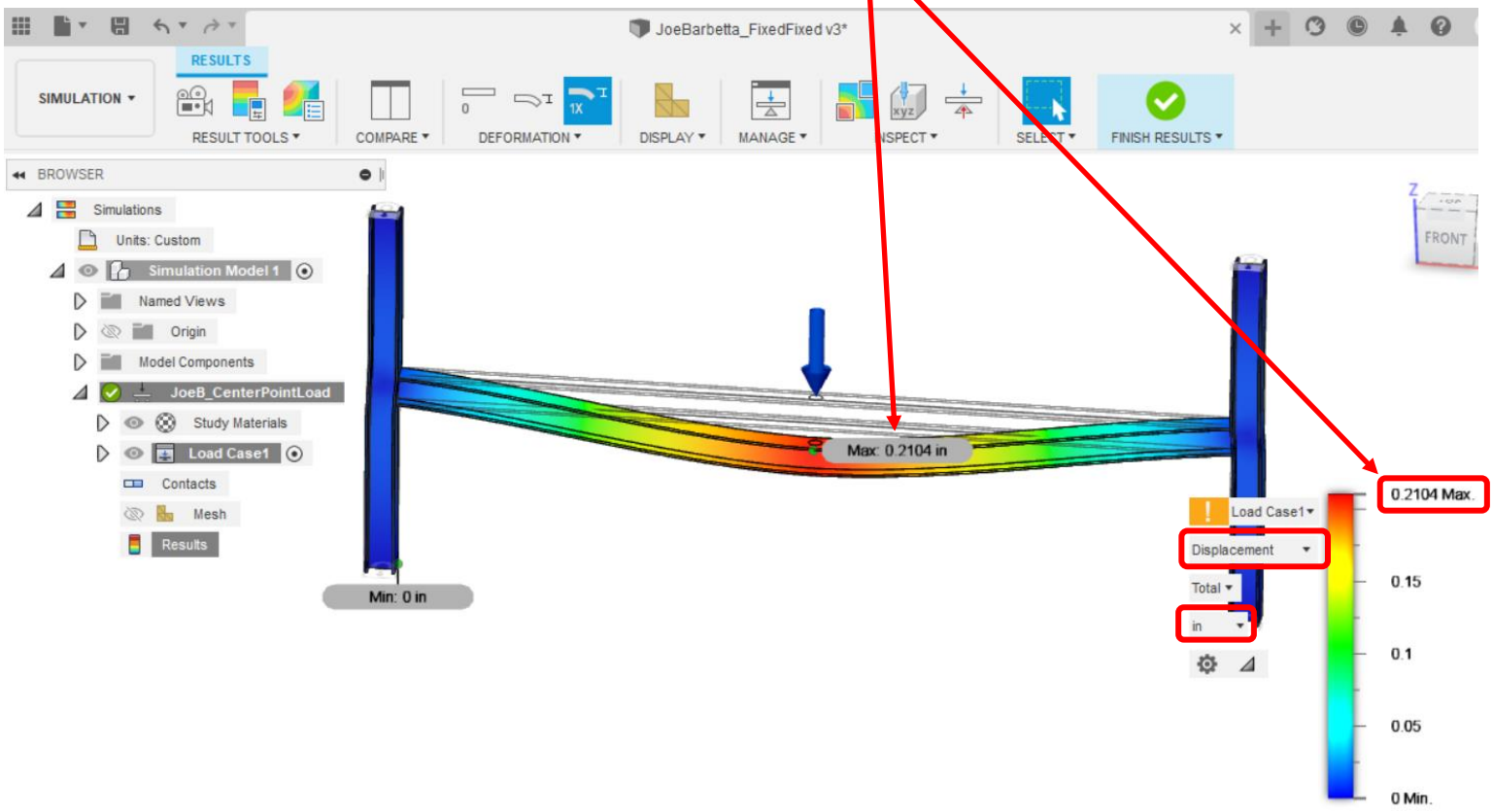
By default, Fusion 360 first shows **Safety Factor**. It is common to specify a Safety Factor of 3 and that is the value between yellow and green on the color bar. A Safety Factor of 3 means that the design is 3 times stronger than it needs to be.



- select **Stress** at the color scale to see where the greatest stresses are



- select **Displacement** at the color scale and select **in** to see the **Maximum Deformation**, which is displayed with a label, but is also shown as the maximum range of the color scale.



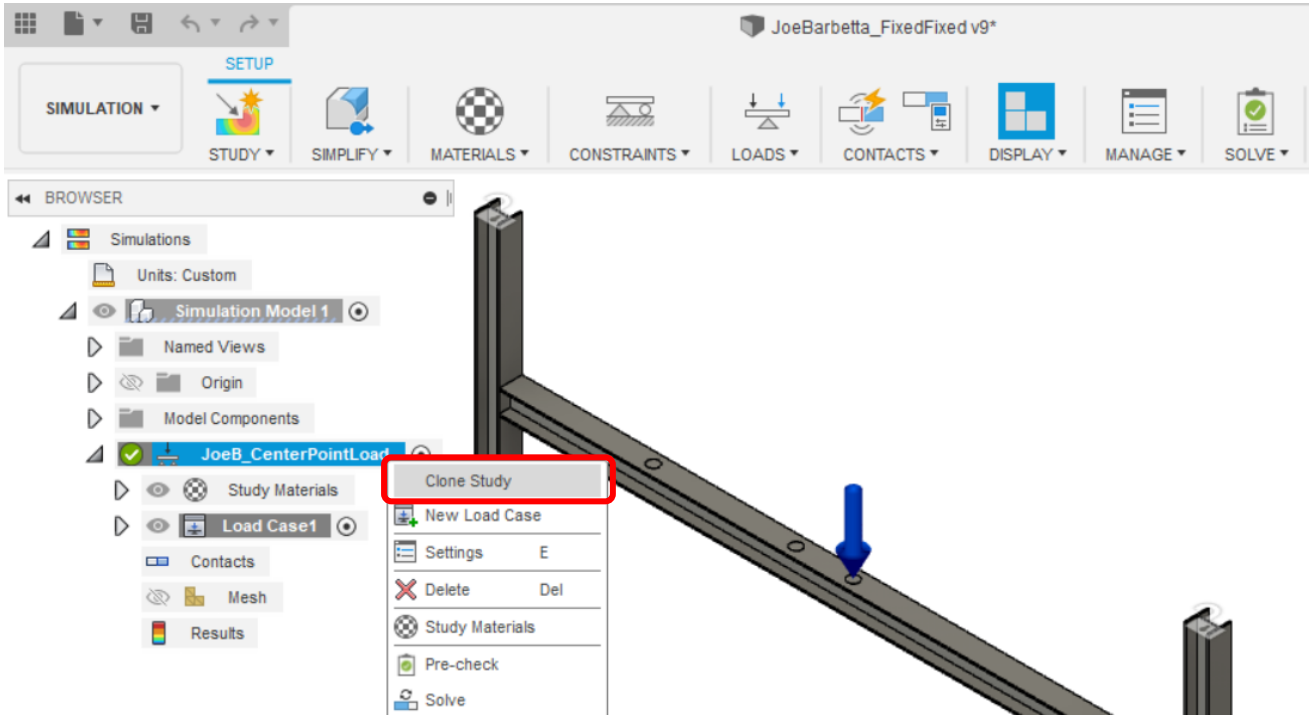
Create the **Dual Point Load** Study

- click the top right **FINISH RESULTS**

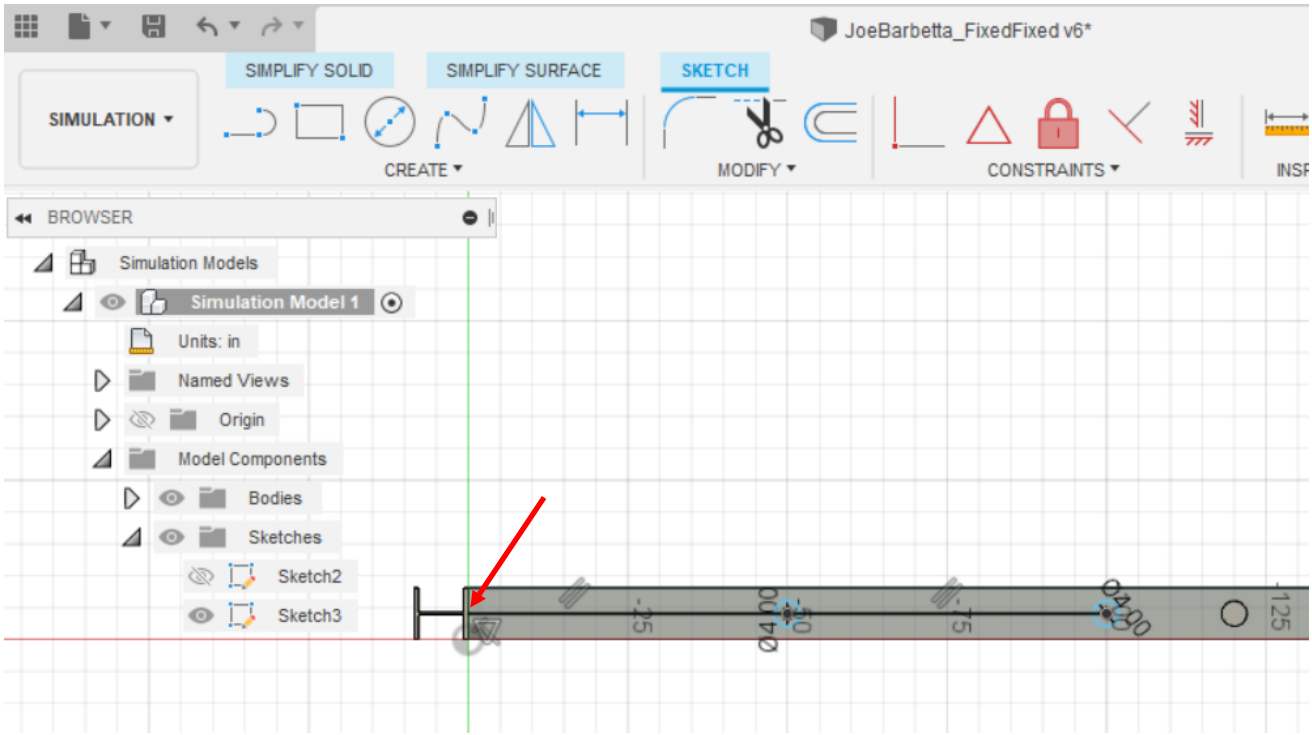
- create a new study by right-clicking on the 1st study name and selecting **Clone Study**. This is a short cut to creating a new study because we will be using the same Constraints.

- right-click on the new simulation name and select Settings and change the name to your first name and last name initial followed by _DualPointLoad, e.g. **JoeB_DualPointLoad** and click **OK**.

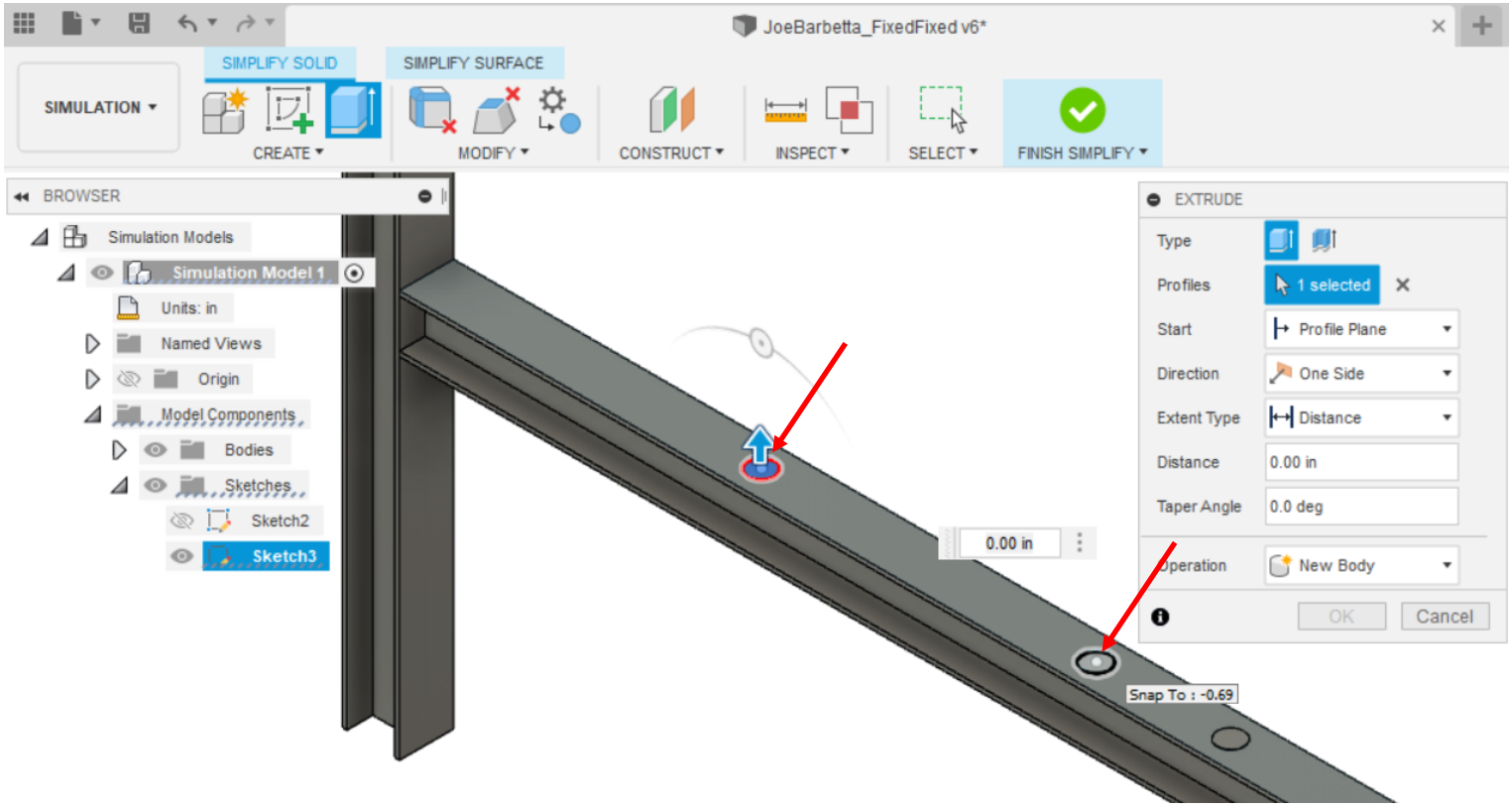
- as done earlier, select the **Simplify** tool and then the **Create Sketch** tool and then click on the **top surface** of the beam.



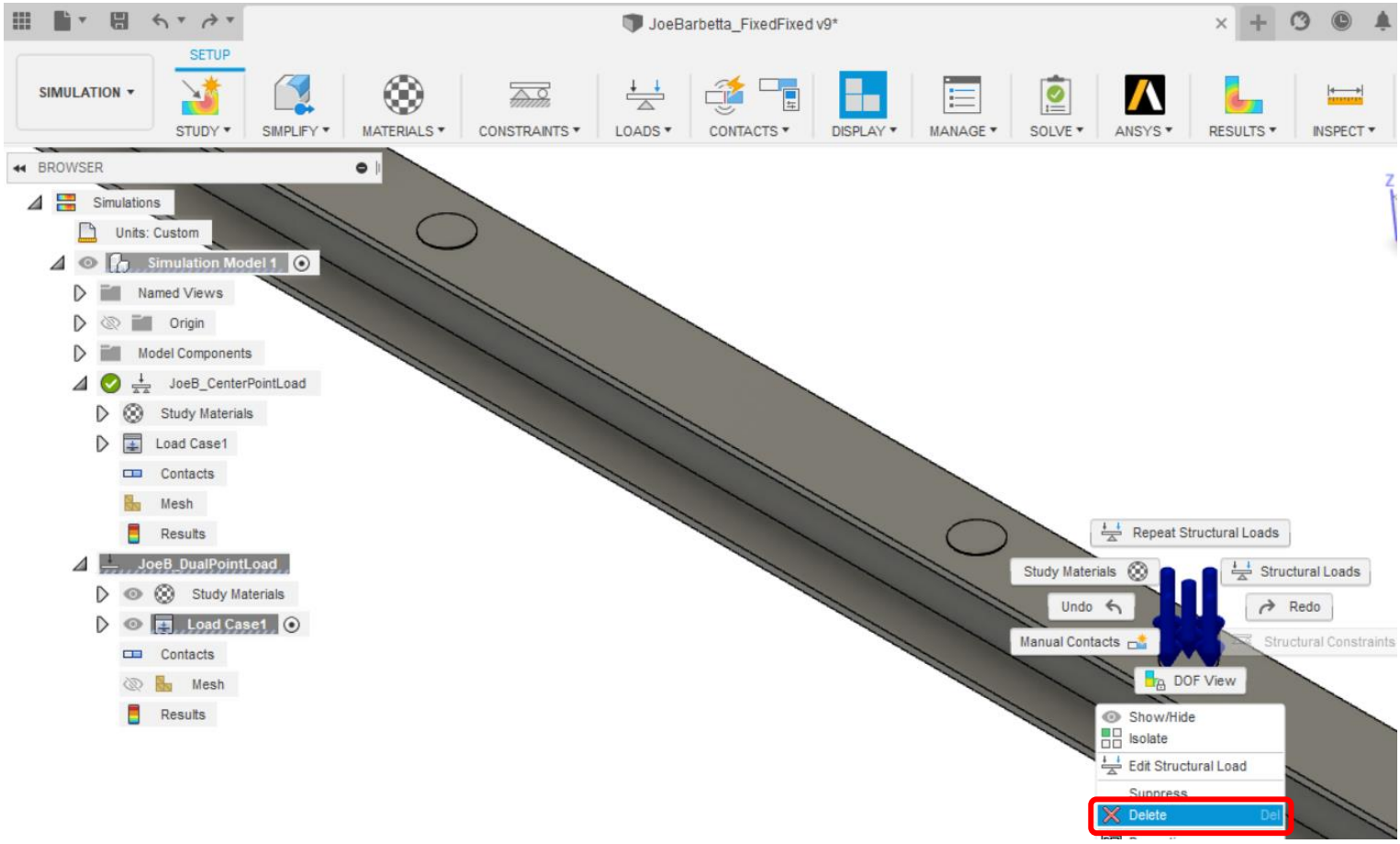
- create a **50 in** line from the end of the beam and then create a **4 in** diameter circle at its end
- from that point create a 2nd **50 in** line and a **4 in** diameter circle at its end
- delete both lines and click **Finish Sketch**



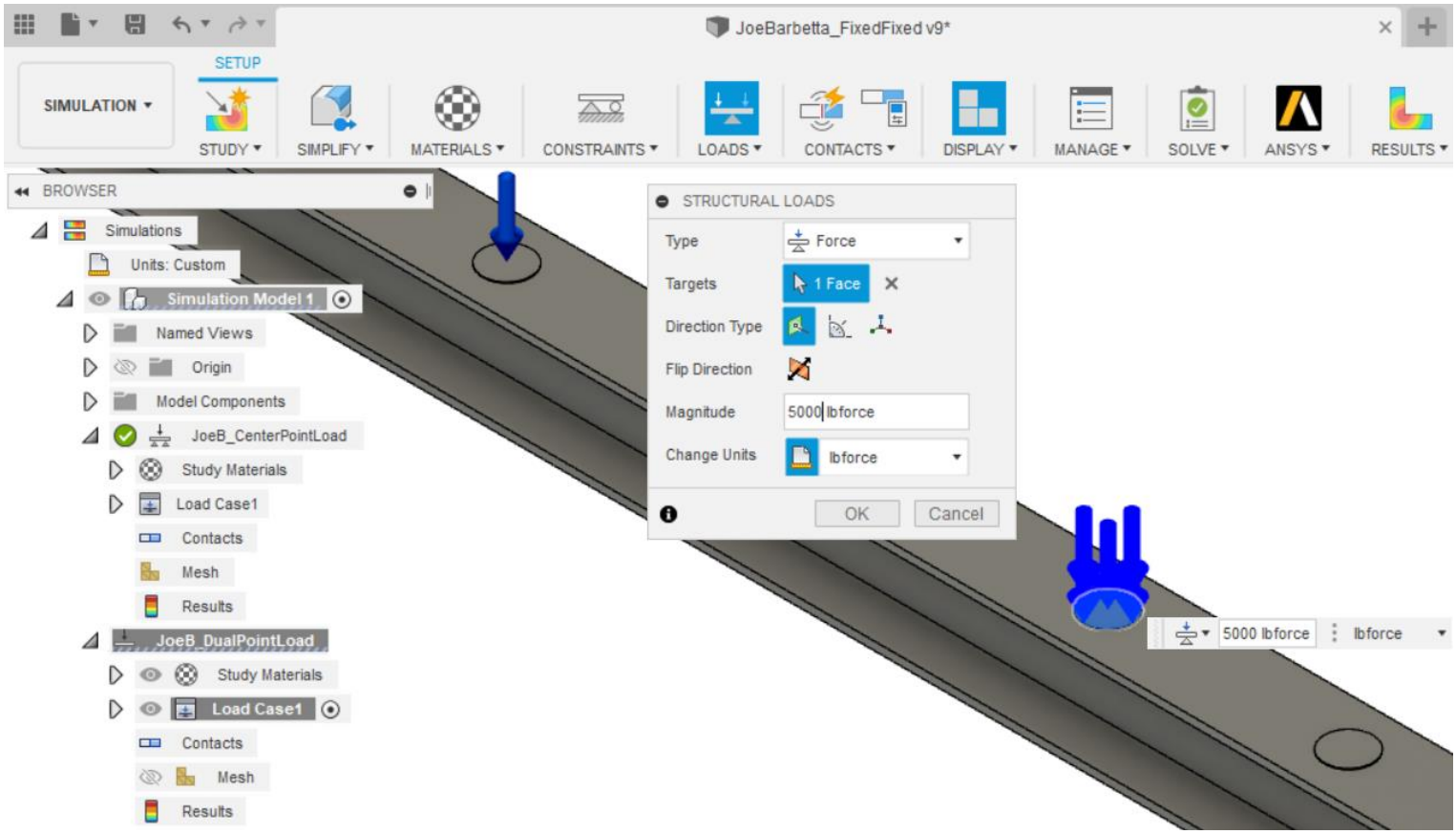
- extrude both inner regions up **0.1 in**, click **OK**, and click **FINISH SIMPLIFY**



- zoom into the 3 circles on the top of the beam. Right-click on the **Load Arrow** and select **Delete**

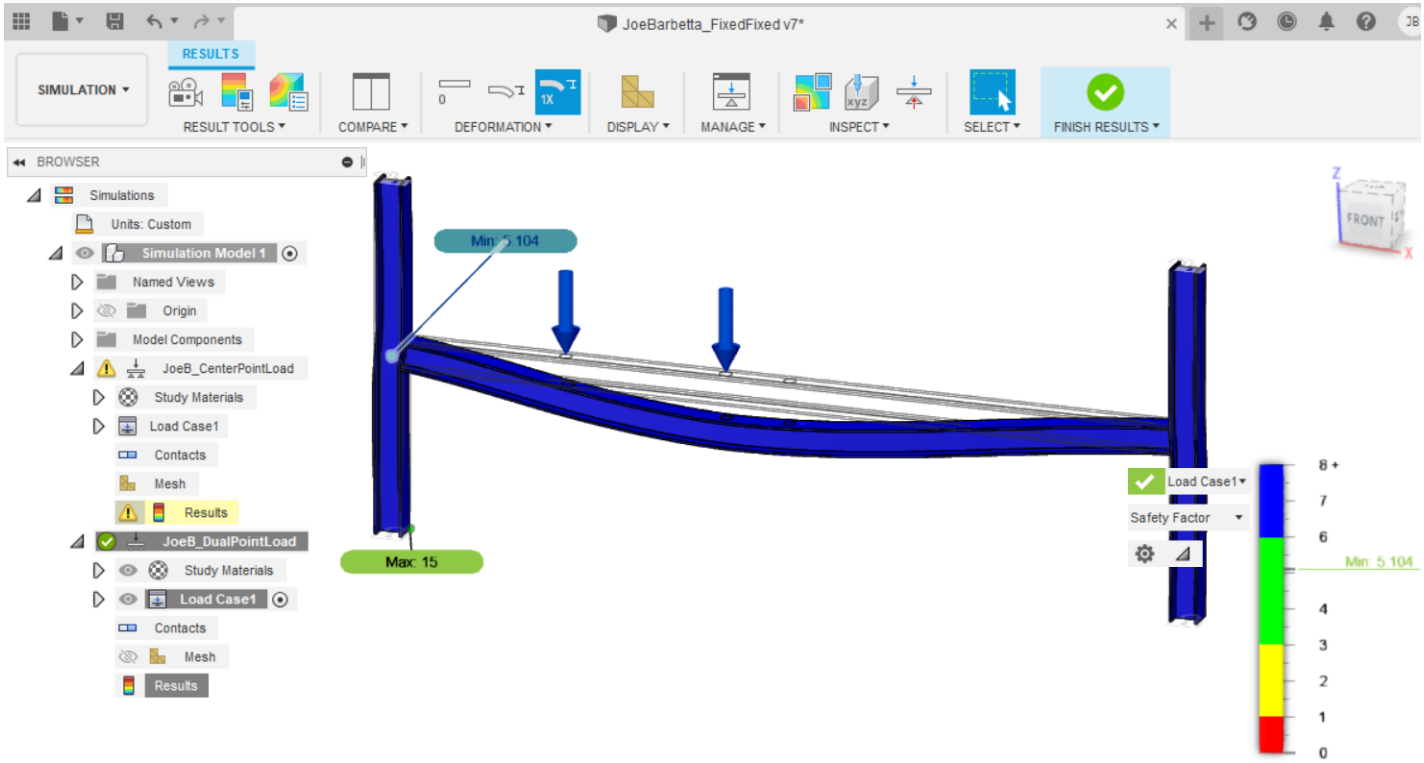


- from the **LOADS** pull-down menu select **Structural Loads**
- select the **1st circle** (if you have trouble selecting the inner region, zoom in) and set the load to **1/2 of your assigned load**.
- do so again for the **2nd circle**, but Not for the 3rd.

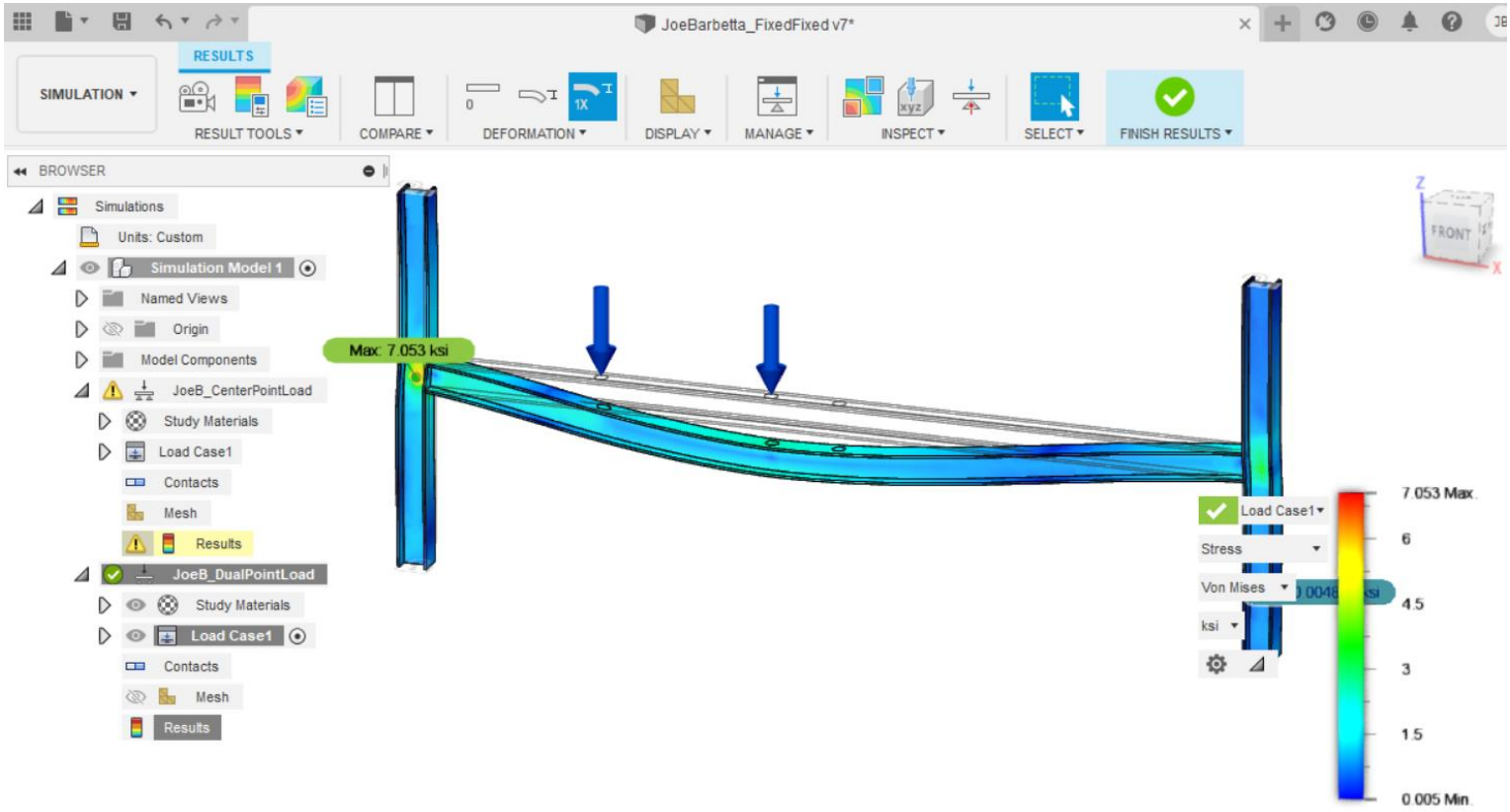


- generate a mesh for the **Dual Point Load** and **Solve**

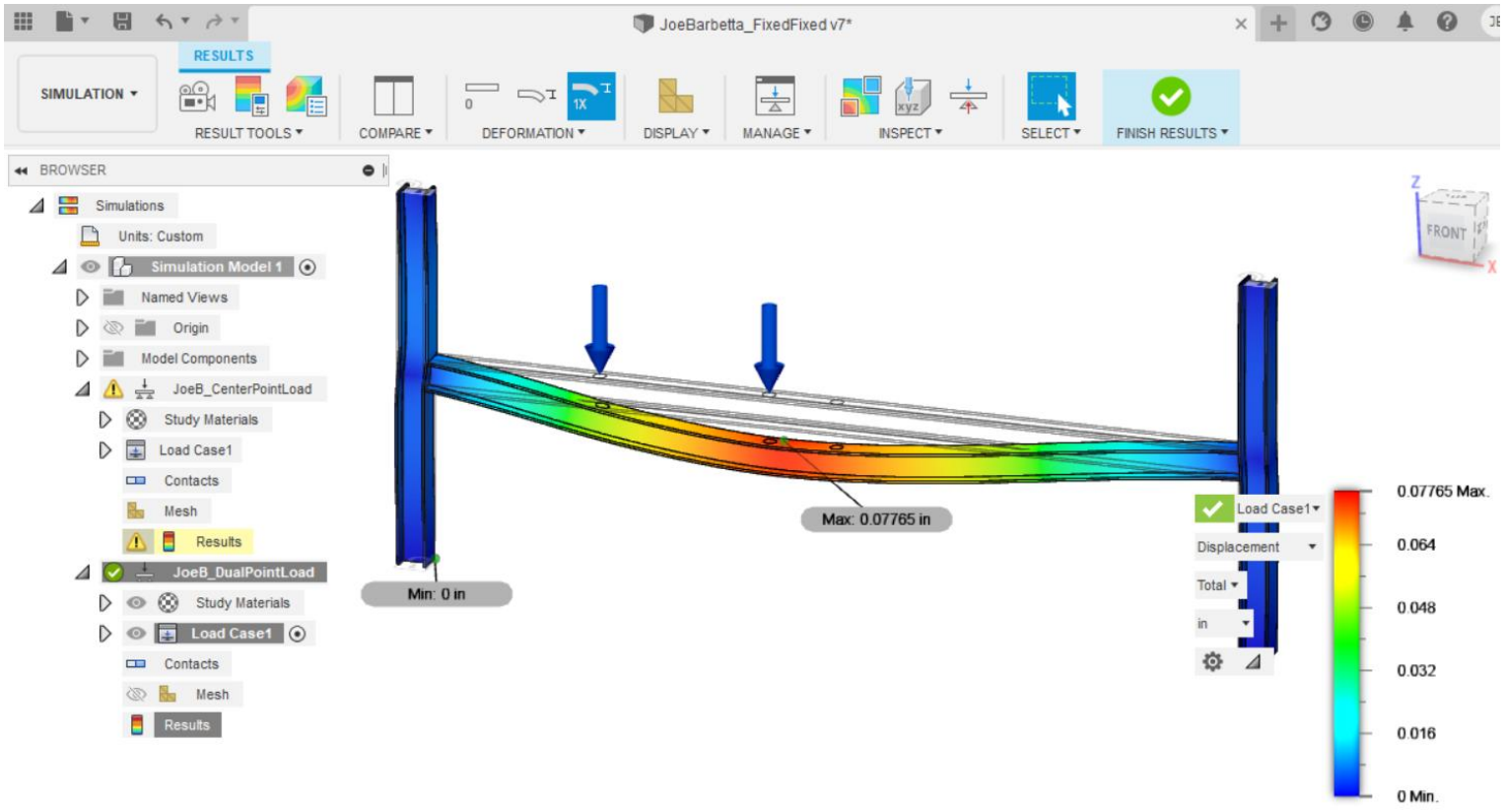
Note that you are now simulating a scenario that your typical beam formulas can't solve.



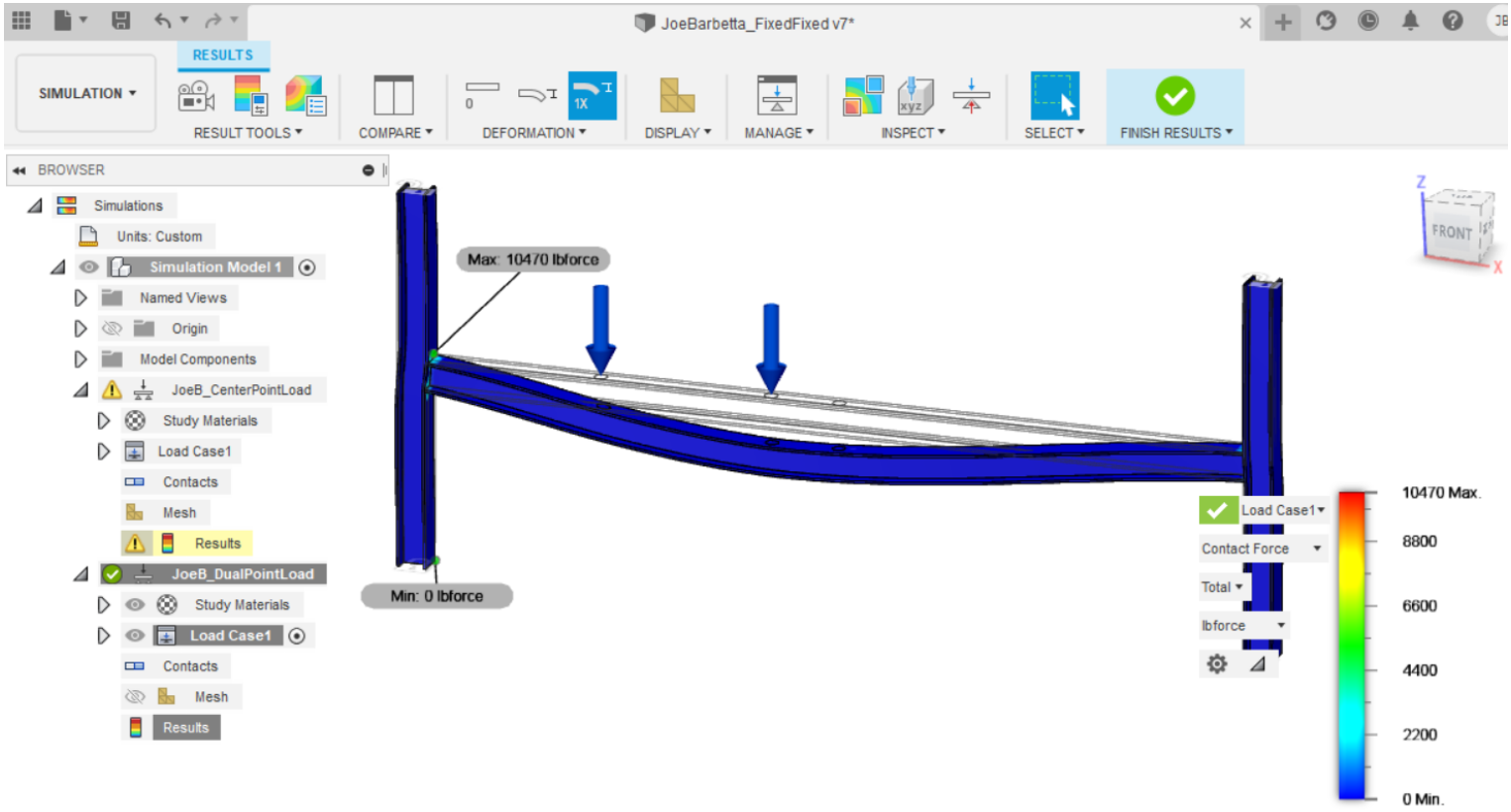
Here is an example of the Stress results.



Here is an example of the Displacement results.

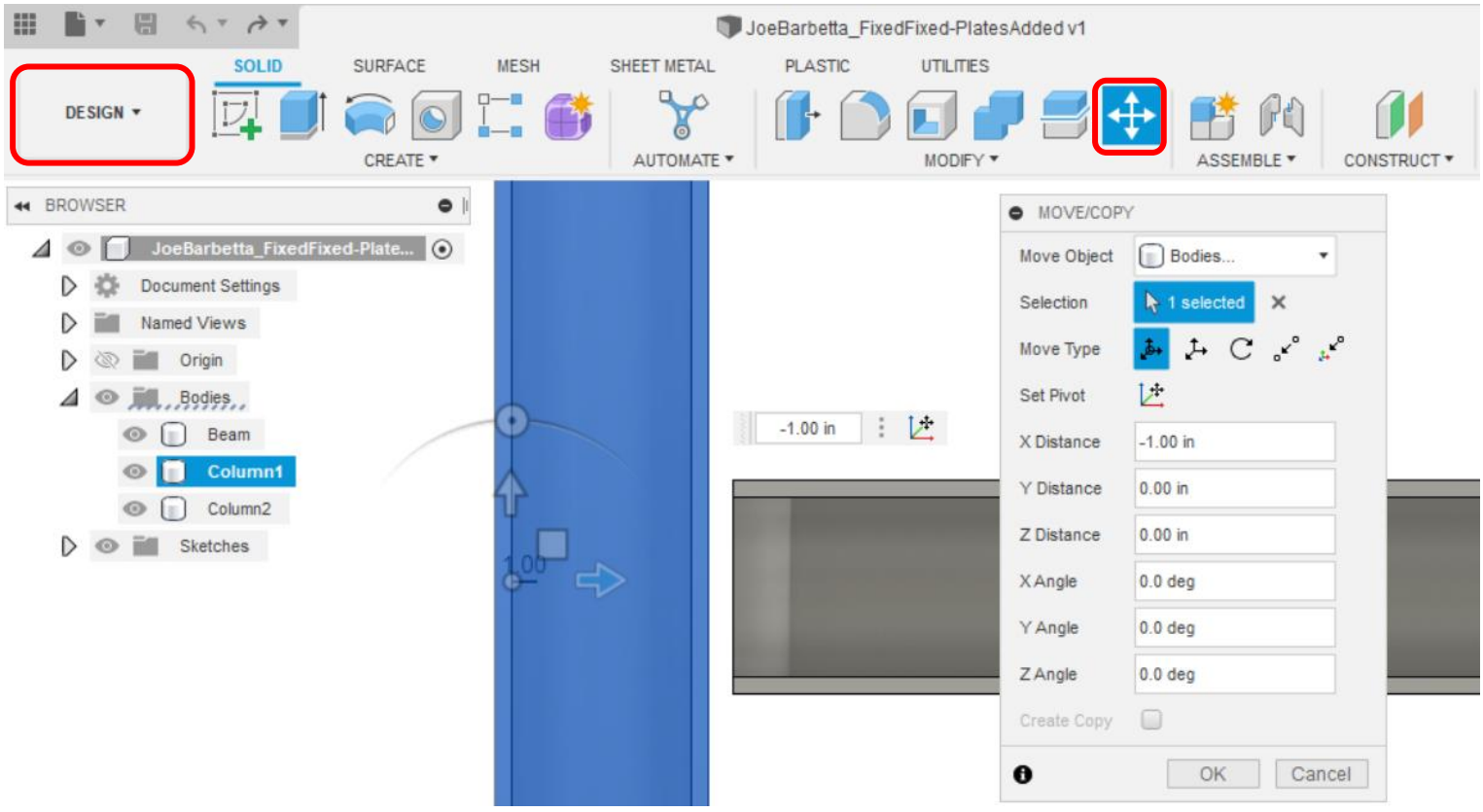


- select **Contact Force** to see the forces that your welds may be subjected to

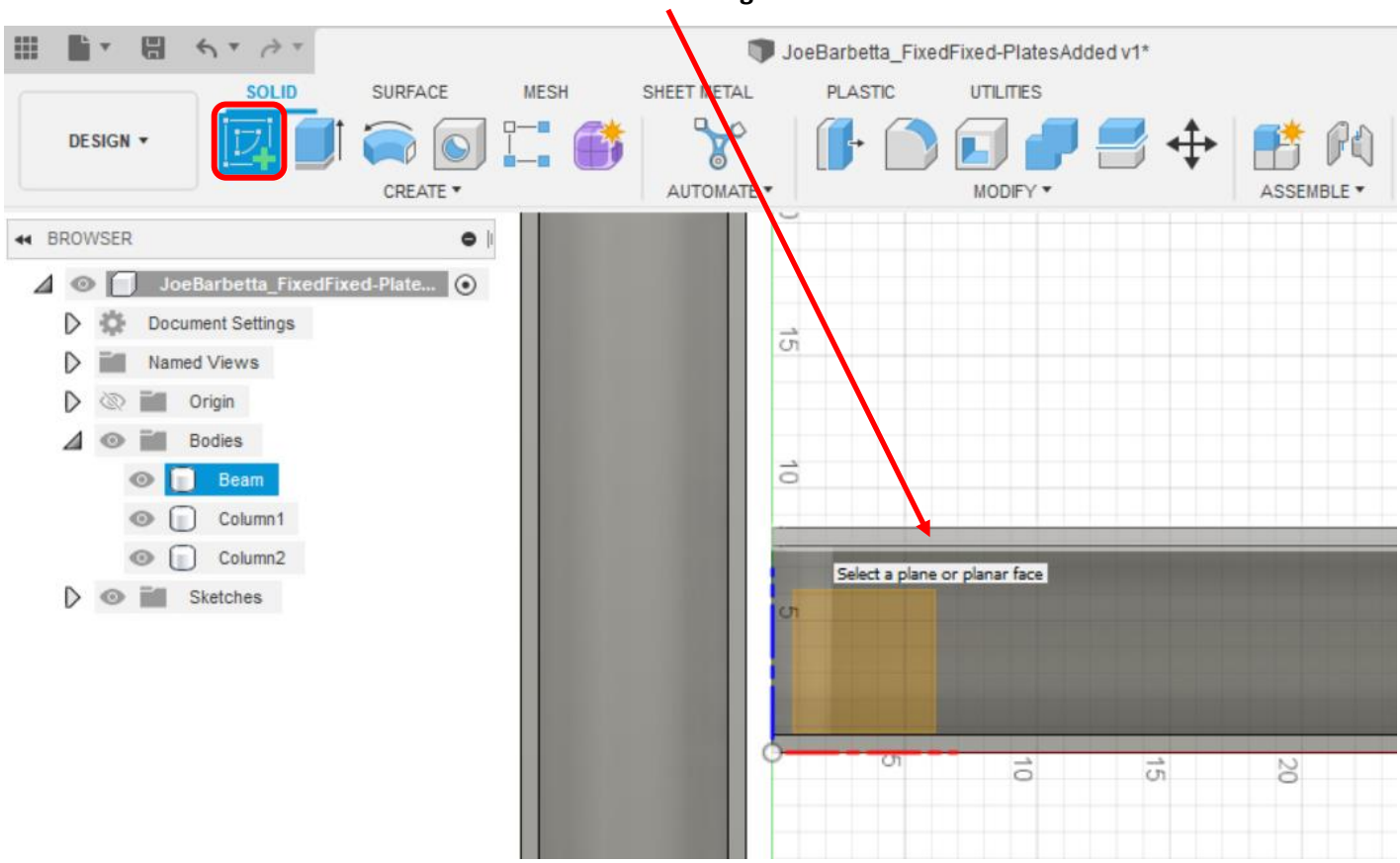


Save your project using **Save** and then use **Save As** with the original name followed by **-PlatesAdded**, e.g. JoeBarbetta_FixedFixed-PlatesAdded.

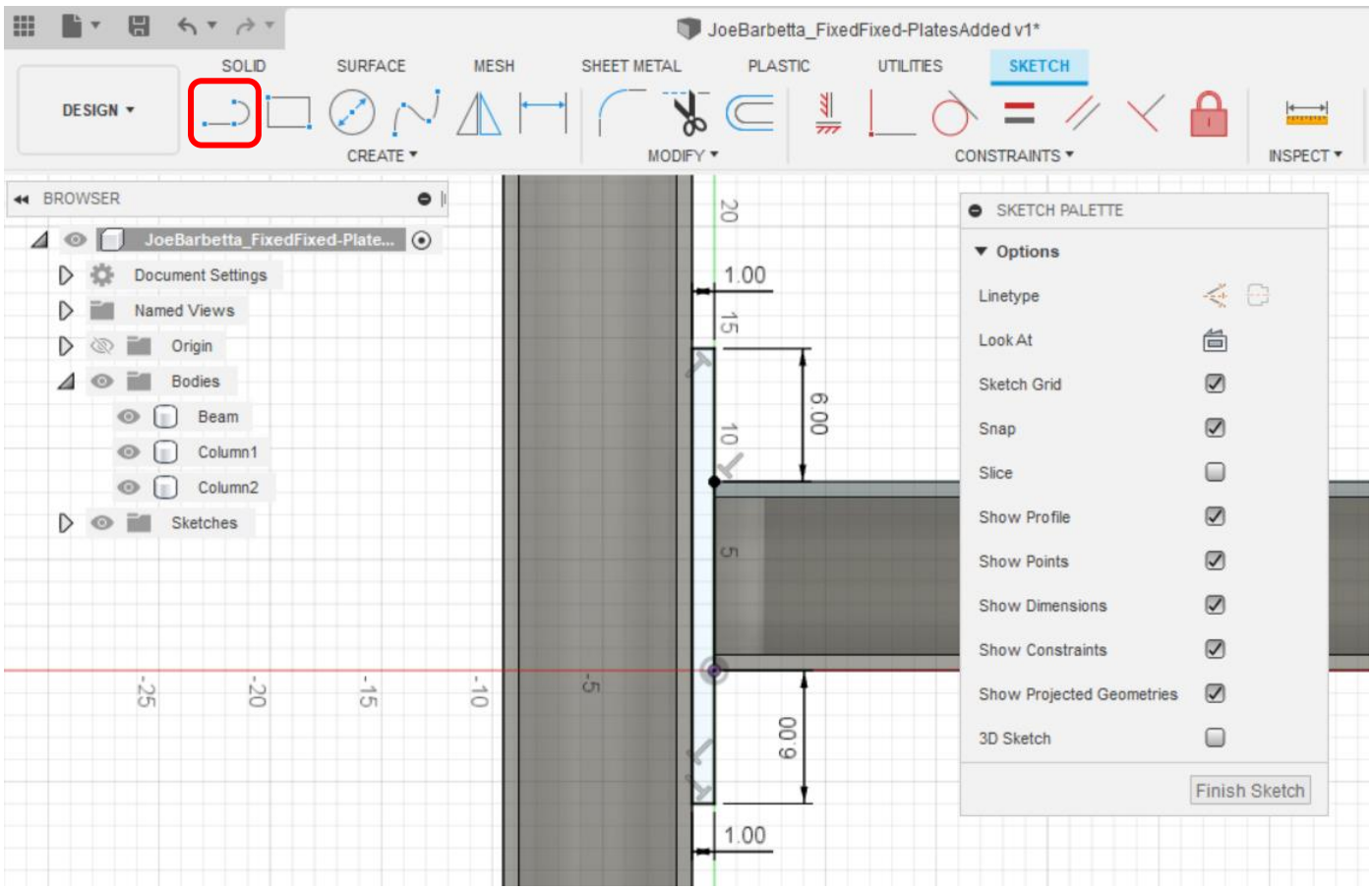
- enter the **DESIGN Workspace** and zoom into the left column and beam.
- select the **Move/Copy** tool and move the column away from the beam by **1 in**, and click **OK**.



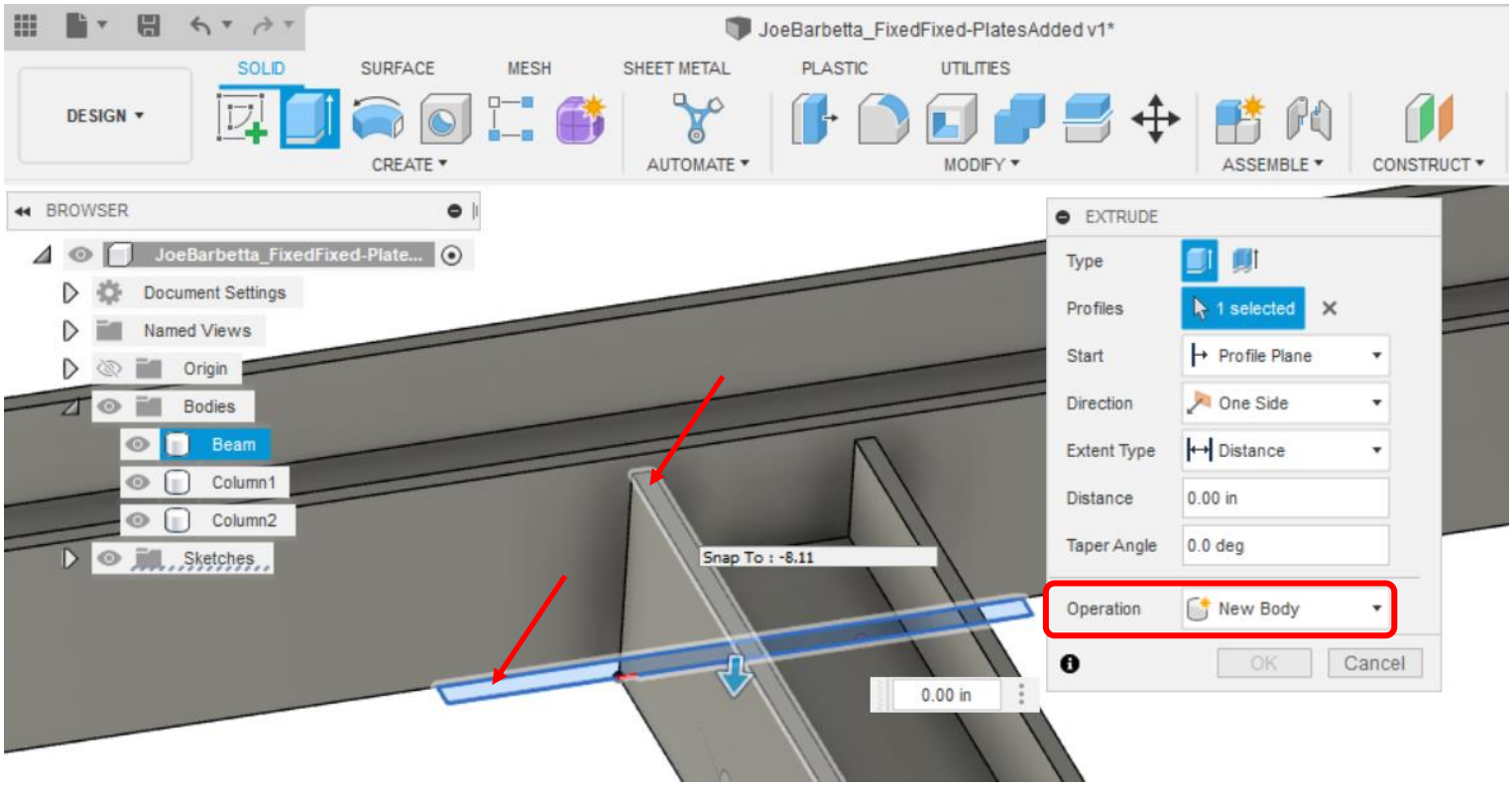
- select the **Create Sketch** tool and click on the **face of the flange** as shown



- use the **Line** tool to create the end profile of a **1 in** thick plate that extends **6 in** above and below the beam. The interior of the profile should turn light blue when it is complete. Click **Finish Sketch**.

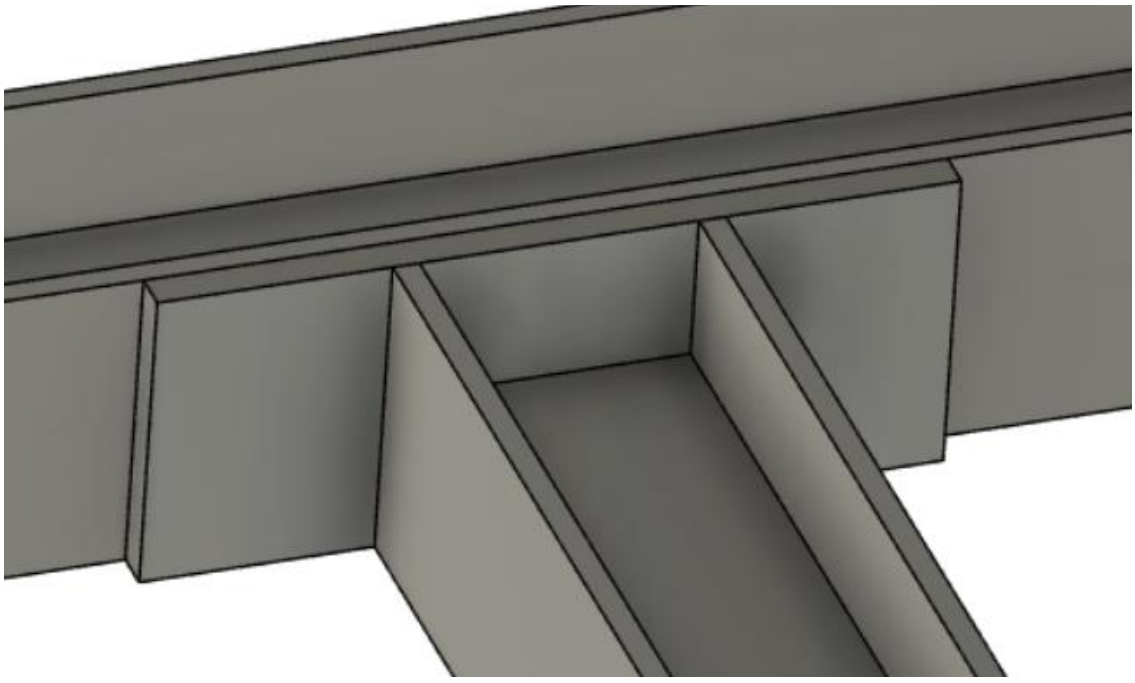


- change the view to allow access to the profile just created and the opposite face of the flange.
- select the **Extrude** tool, click on the **profile**, and then on the **face of the beam flange**, as shown.
- change the **Operation** back to **New Body** as it will automatically switch to Join on the 2nd click and click **OK**.

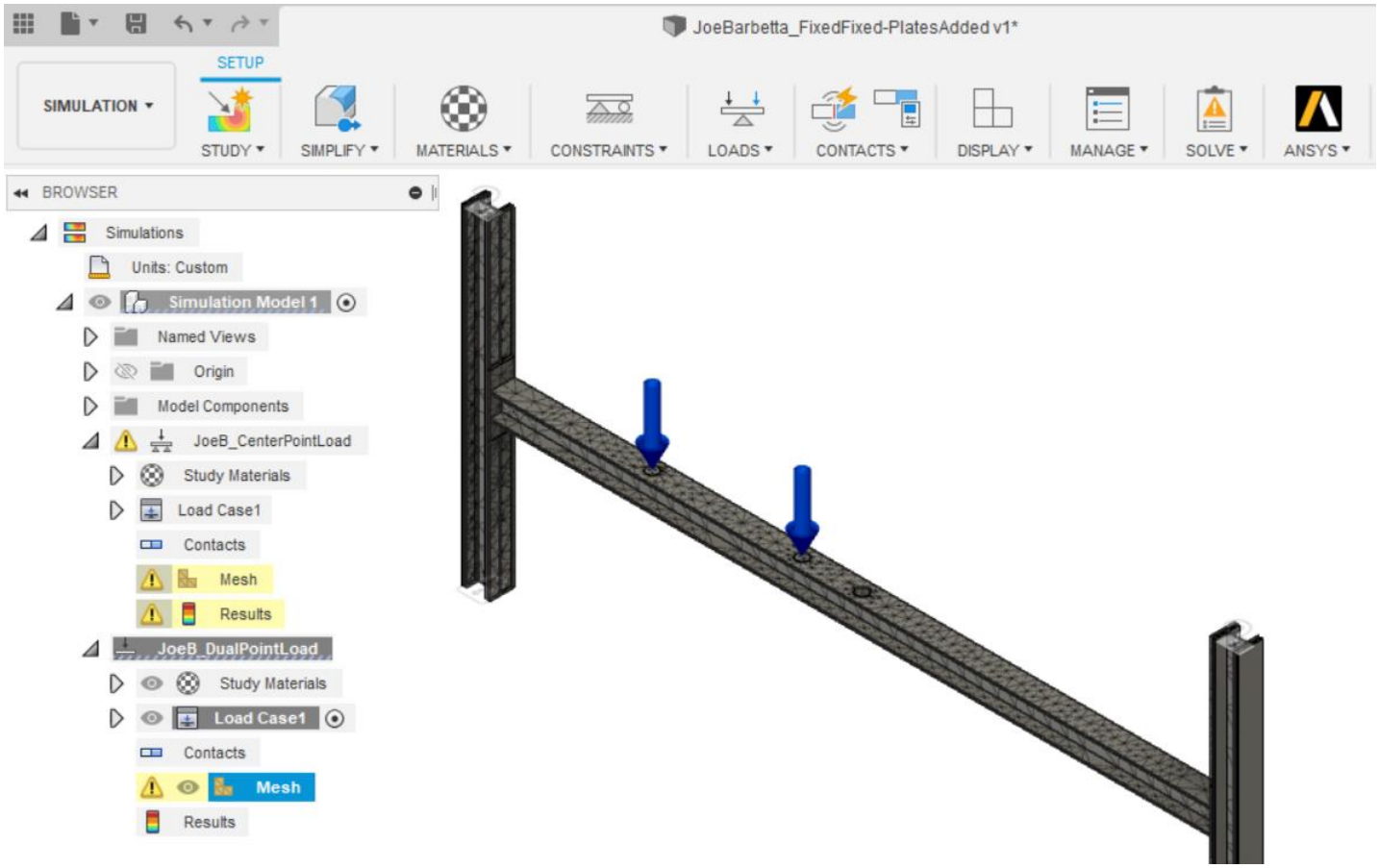


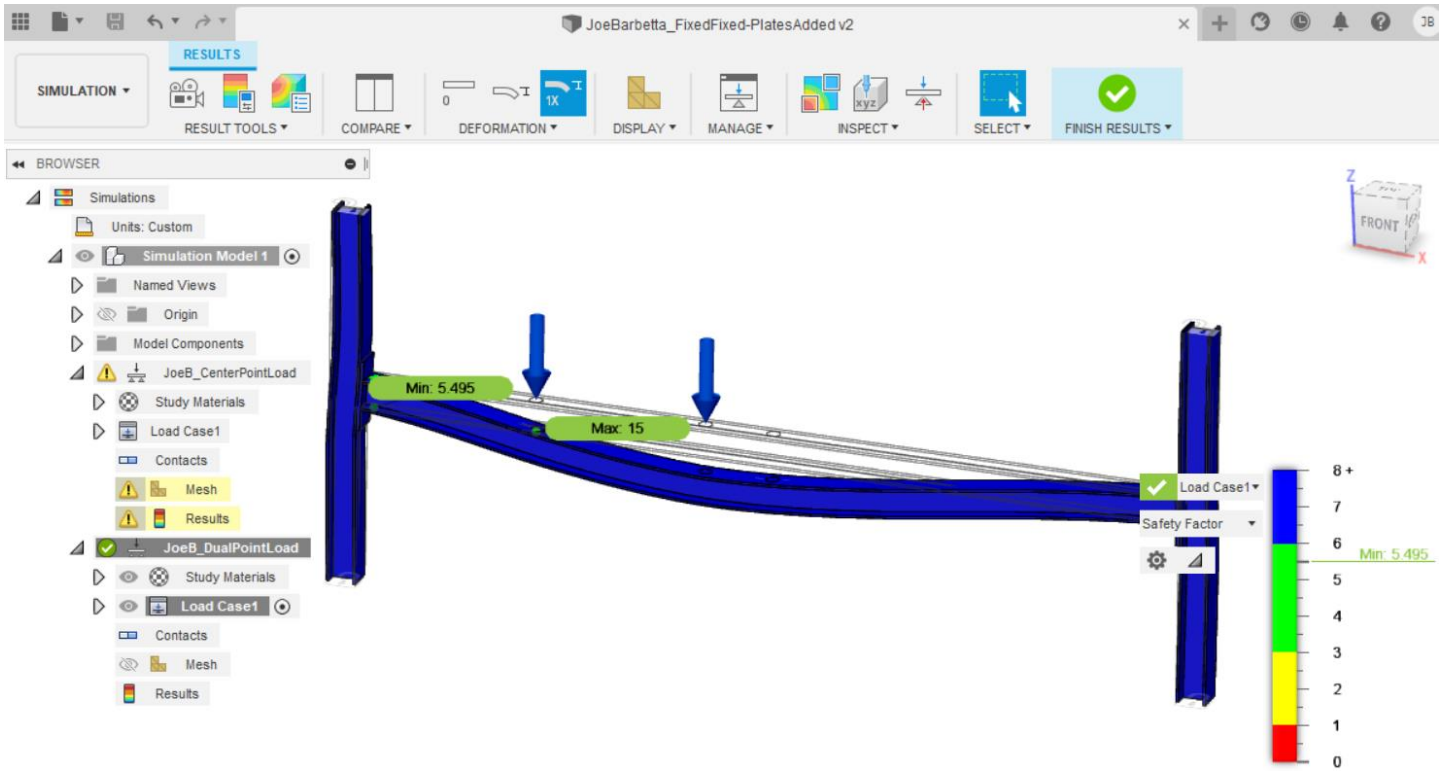
The plate should look as below.

- repeat these steps to create the same size plate at the other end of the beam.

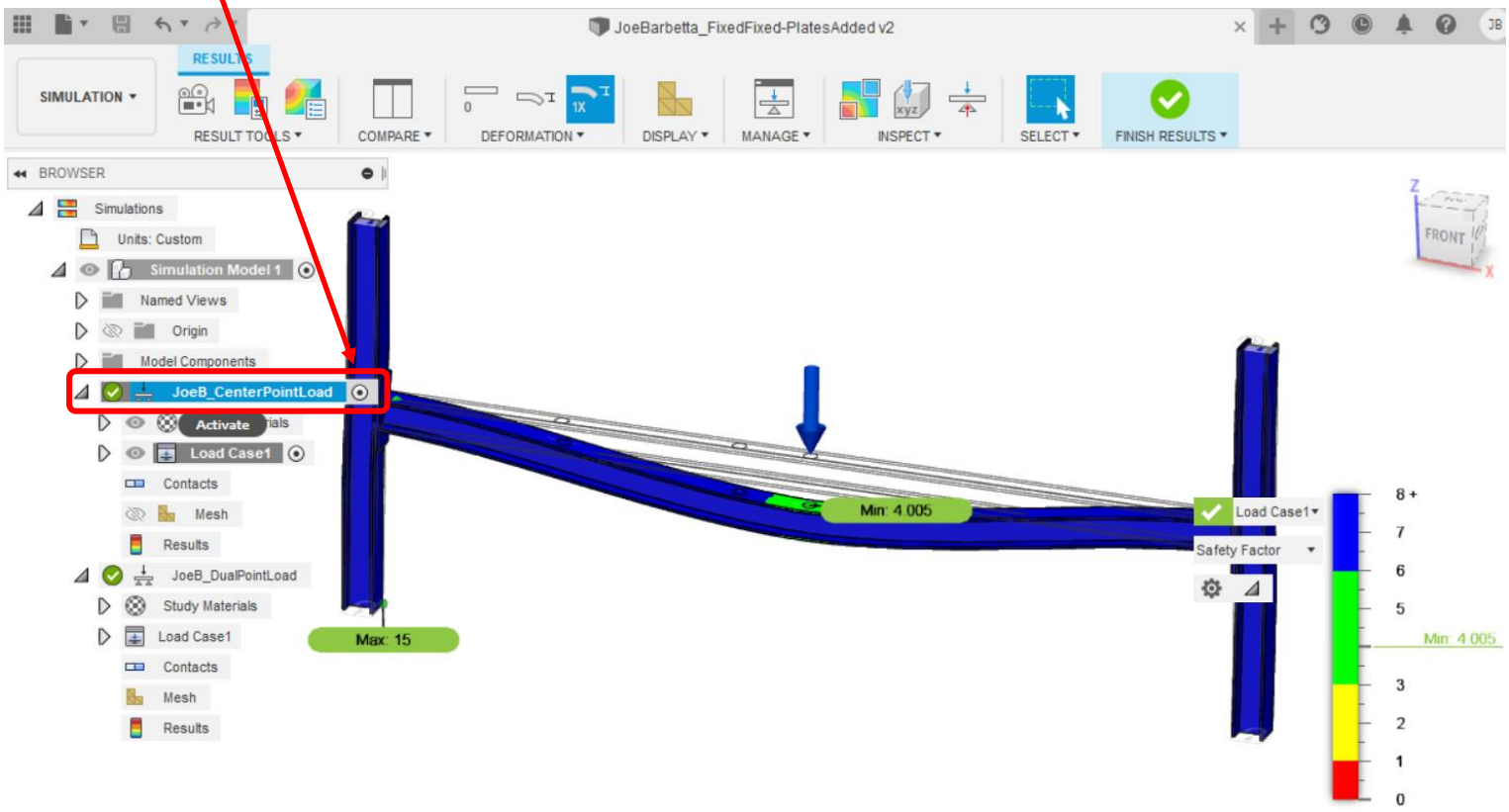


- return to the **Simulation Workspace** and note the Warnings
- right-click on **Mesh** for the **2nd study** and select **Generate Mesh**. Also use **Automatic Contacts** from the CONTACTS menu.
- right-click on **Results** for the **2nd study** and select **Solve**.





- click on the **circle for the 1st study** to make it the **Active Study**
- regenerate the **Mesh** and **Solve** the study. Note that one has to **Activate** a Study to use it and only one can be active at a time.



Deliverables:

A single pdf file including the following:

Note that you are using your previous beam and thus don't need to include the FlangeThickness derivation and calculation and Sketch screenshot.

1) Screenshots of your simulation for a **Center Point Load** showing **Safety Factor**, **Displacement(inches)**, and **Stress(psi)**
2) Screenshots of your simulation for a **Dual Point Load** showing **Safety Factor**, **Displacement(inches)**, and **Stress(psi)**
3) Screenshots of your simulation for a **Dual Point Load with Plates** showing **Safety Factor** and **Stress(psi)**
4) A simple table showing the **difference between the Safety Factor** and **Stress(psi) with and without the plates**