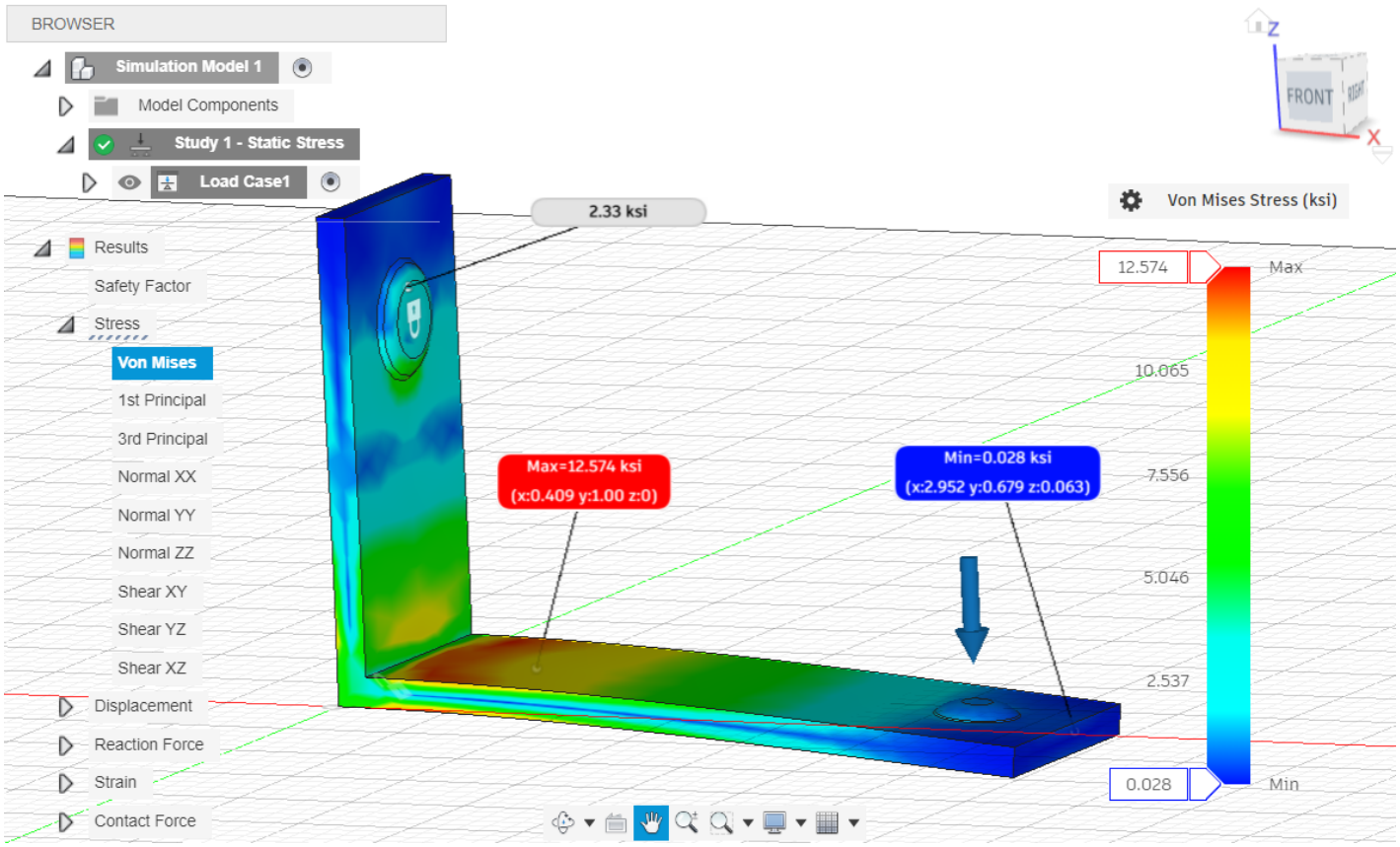
FEA (Finite Element Analysis) using Fusion

My pretty FEA results should be worth an art credit.



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

DIGGERLAND

DRIVE RIDE SOAK SLIDE

CONSTRUCTION THEME & WATER PARK

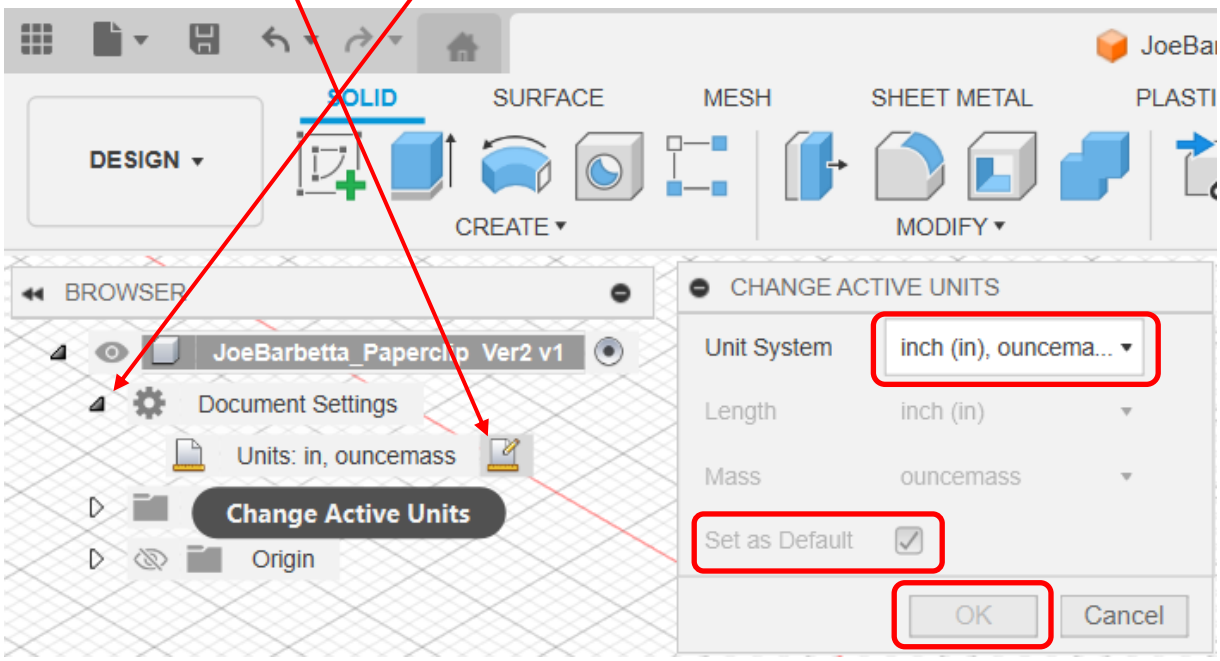
(856) 393-5962
100 Pinedge Drive
West Berlin, NJ 08091

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-Bracket-Baseline** e.g. **JoeBarbetta_FEA-Bracket-Baseline** (note the use of the underscore)

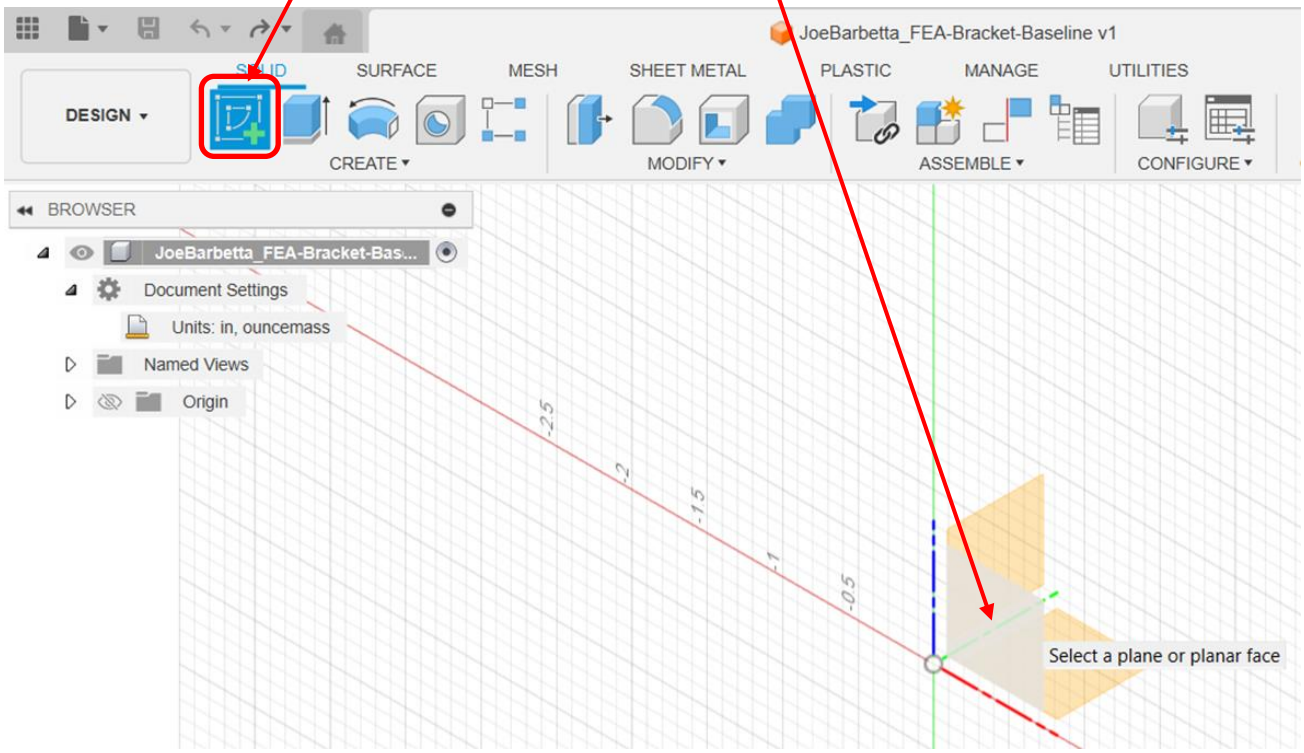
- in the left **"BROWSER"** click the **arrow next to Document Settings**
- click on the **edit icon** that appears to the left when you hover over **Units**
- ensure **Active Units** are set to **Units: in, ouncemass** and click **OK**. You can also enable **Set as Default** if it is not grayed out.



Creating the Side Sketch for the Bracket

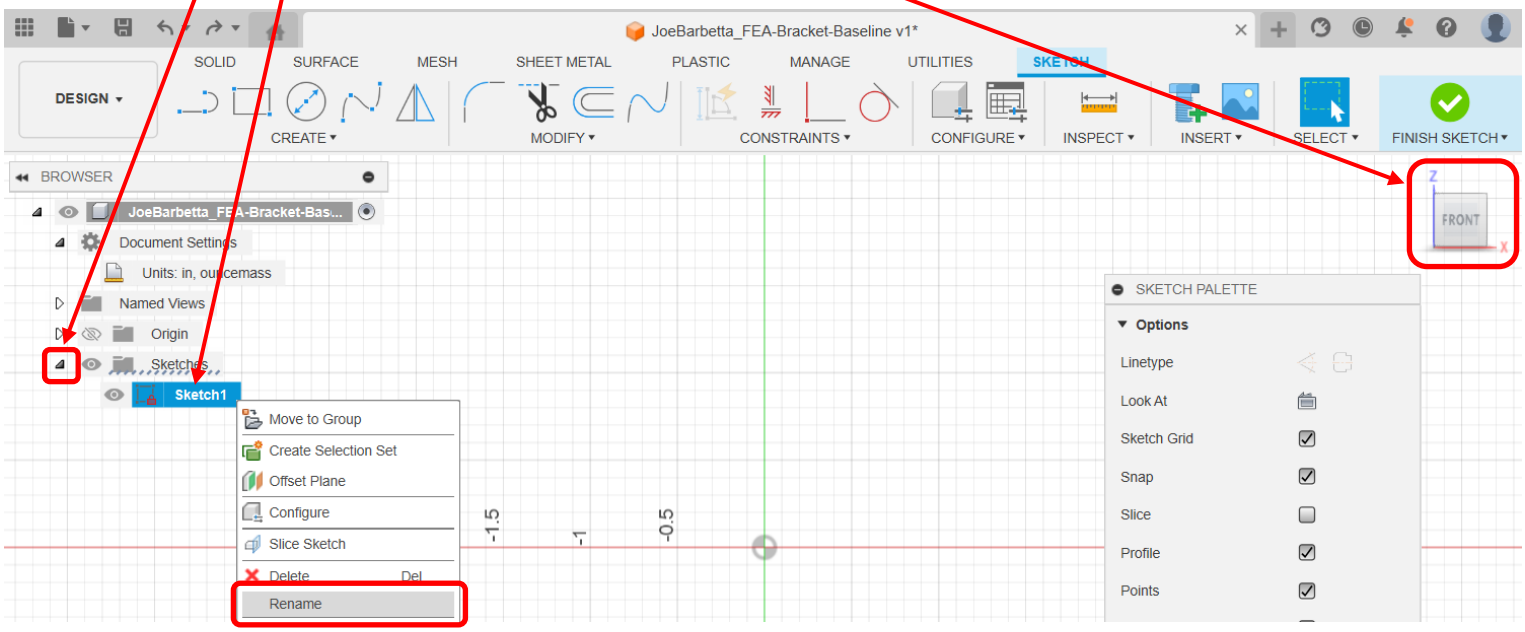
Note that a Fusion expert would suggest starting with the creation of a new Component, but you can say **“Dude, I’m creating a simple bracket.”**

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

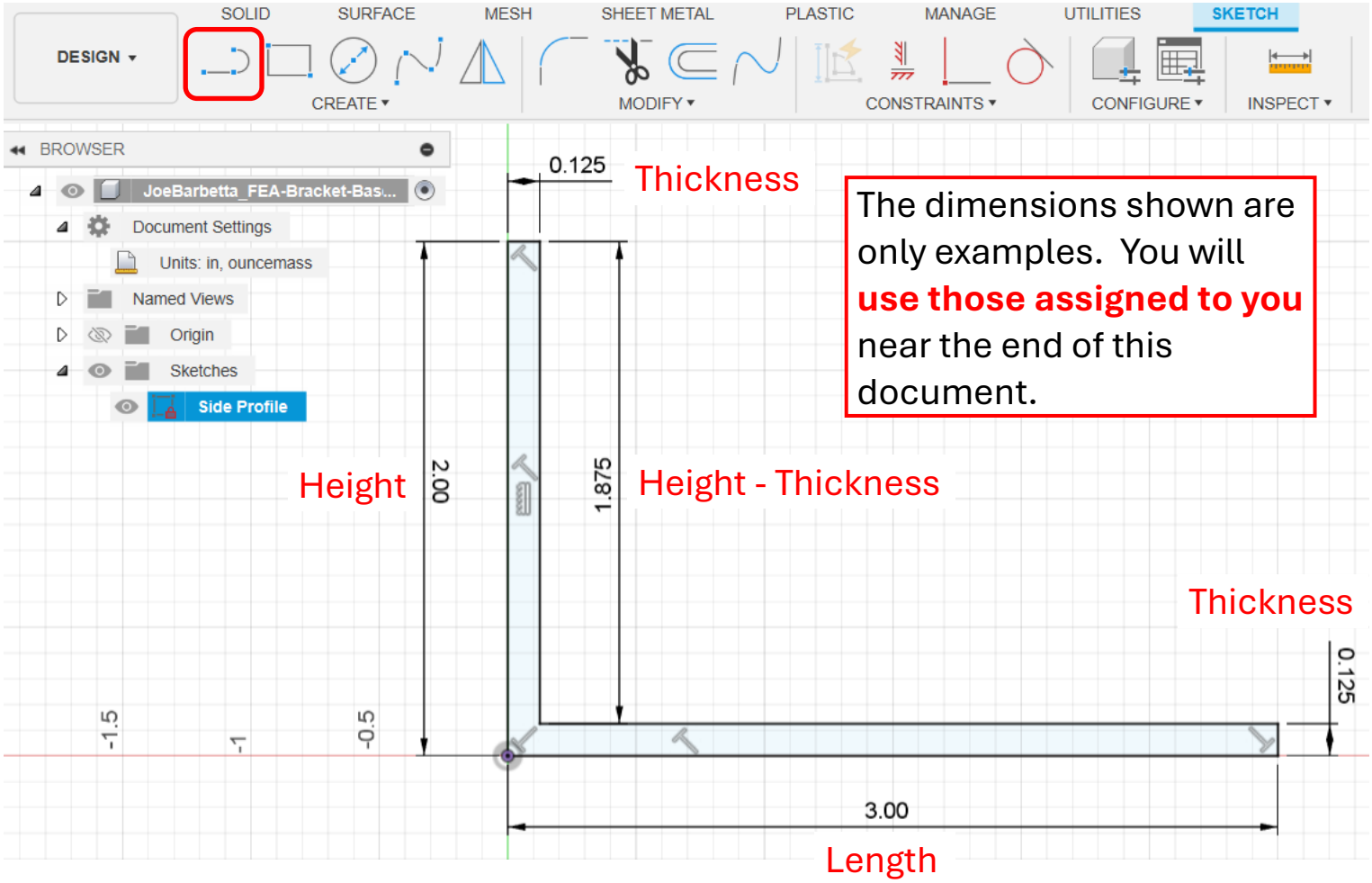


The View Cube text should show “**FRONT**”. If it does not, then redo the Create Sketch and ensure the correct rhombus is selected.

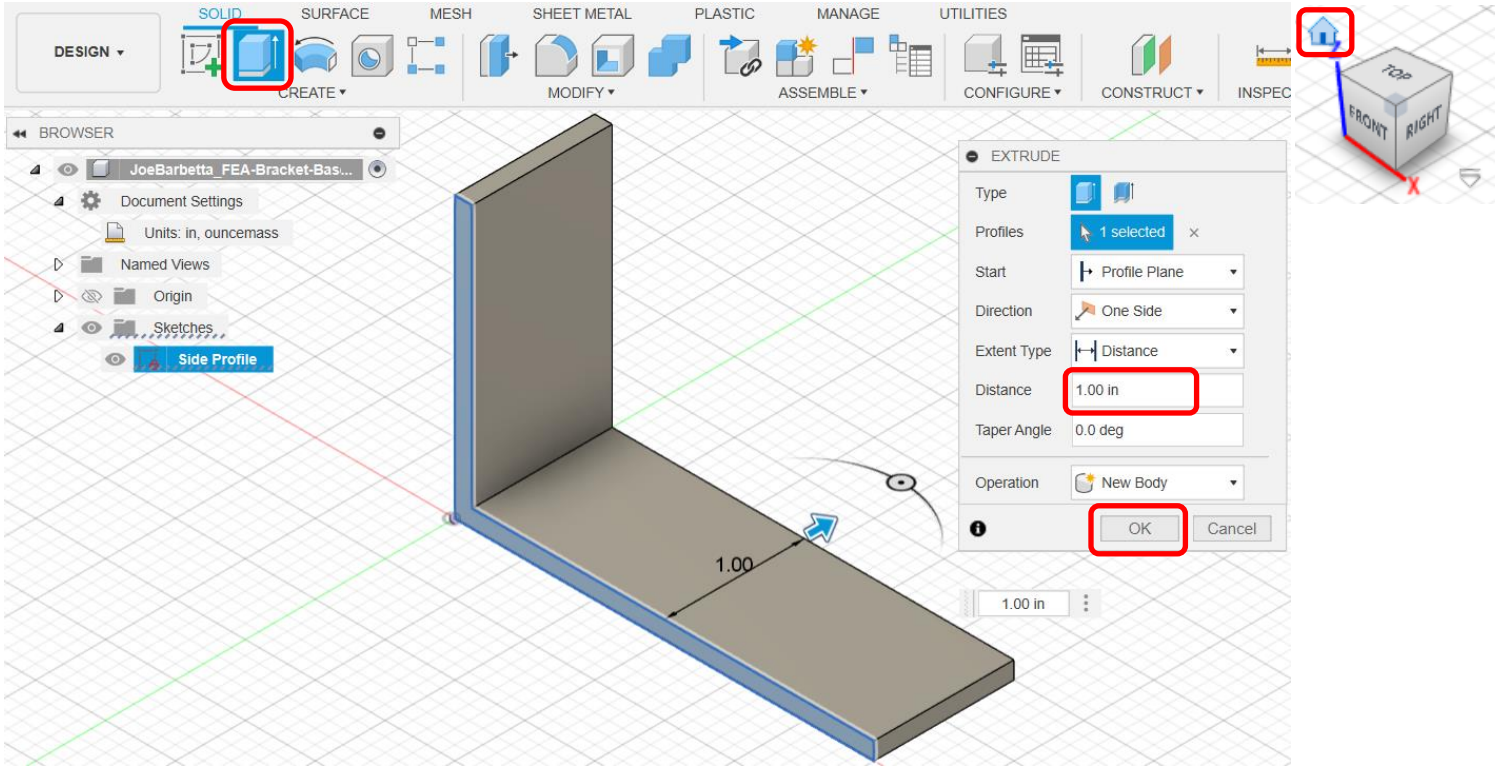
- click on the **arrow** next to the **Sketches** folder to view any sketches
- right-click on the **Sketch** and select **Rename** and enter the text **Side Profile**.



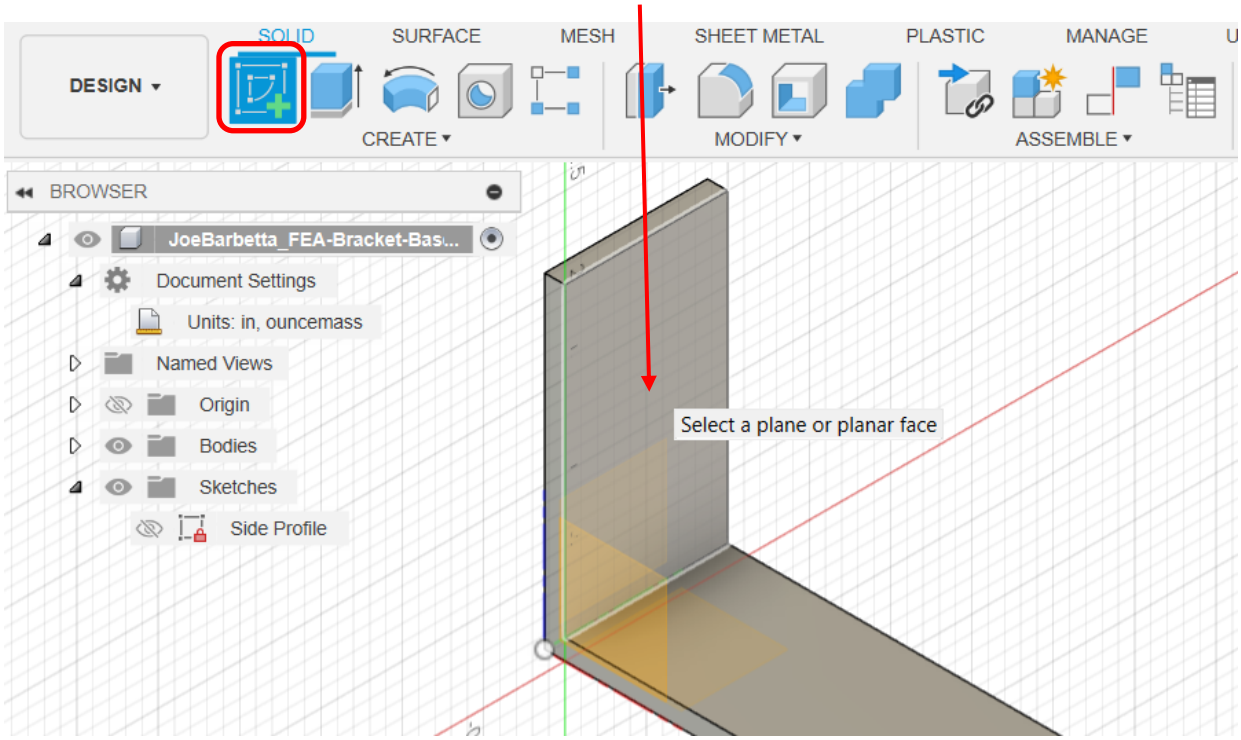
- use the **Line** tool to sketch the profile of the bracket using the **dimensions assigned to you**
- click **Finish Sketch** when done



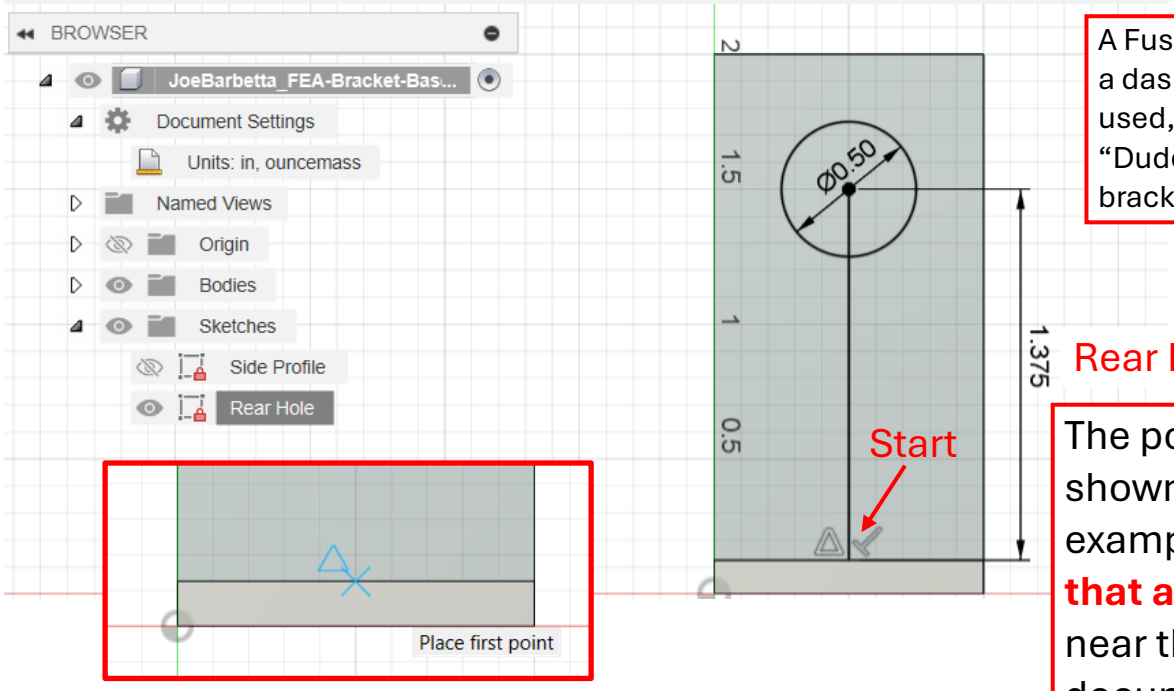
- click on the **home icon** at the **View Cube** to achieve a view similar to that below
- use the **Extrude** tool to extrude the profile **1.00 in** and click **OK**



- select the **Create Sketch** tool and click on the **front face**



- select the **Line** tool and **start a line near the bottom in the center of the bottom edge**. As the inset picture shows, a **blue triangle** will show when the mouse is at the center of the edge.
- continue the line upwards and enter **your assigned value for the rear hole position**
- at the top of the line just drawn select the **Center Diameter Circle** and enter **0.5 for the diameter**. If a tool is not visible at the top find it in the **CREATE** menu.

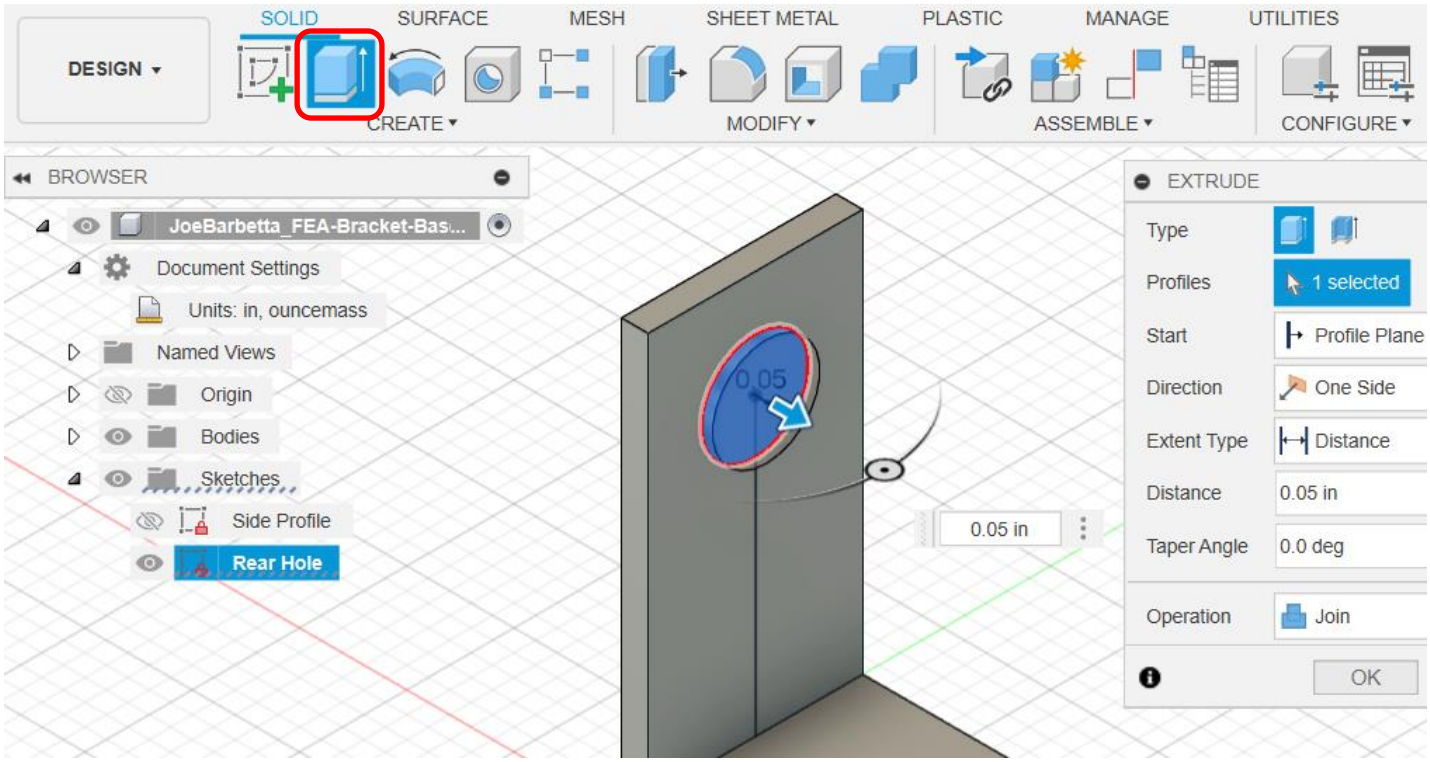


A Fusion expert would want a dashed Construction line used, but you can say. "Dude, this is just a simple bracket.

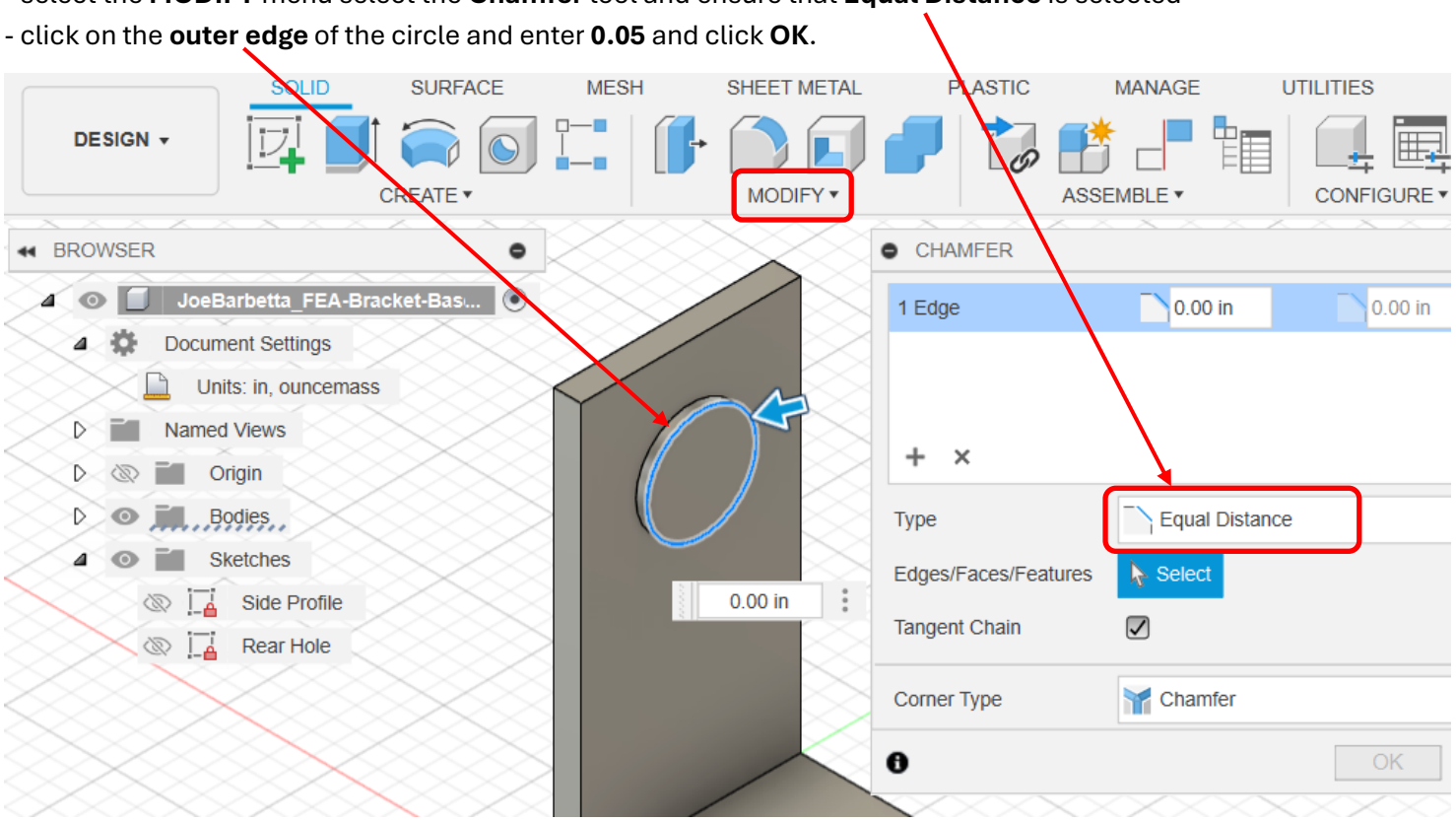
Rear Hole Position

The position dimension shown is only an example. You will **use that assigned to you** near the end of this document.

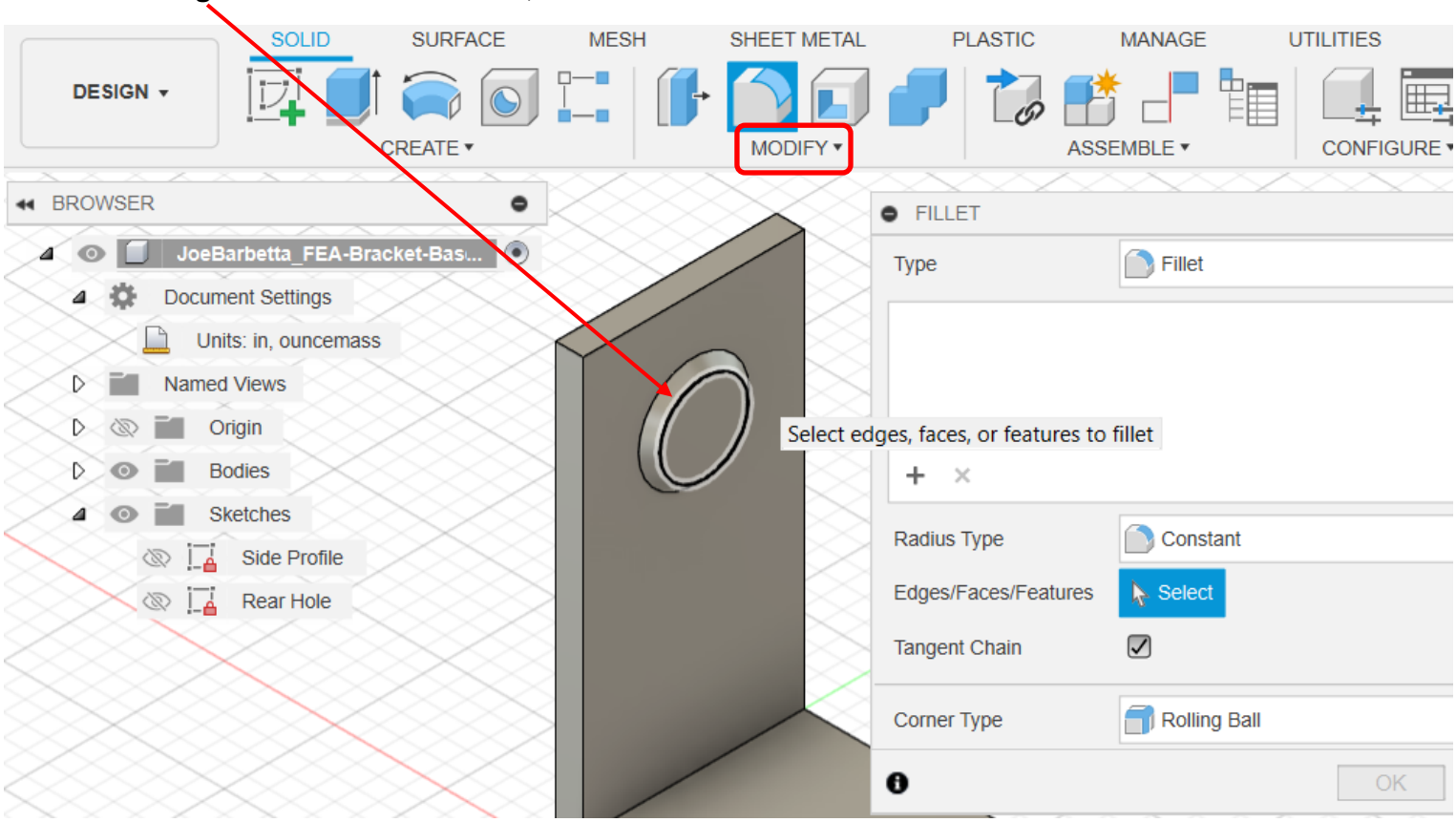
- select the **Extrude** tool and extrude the circle out by **0.05 in**.



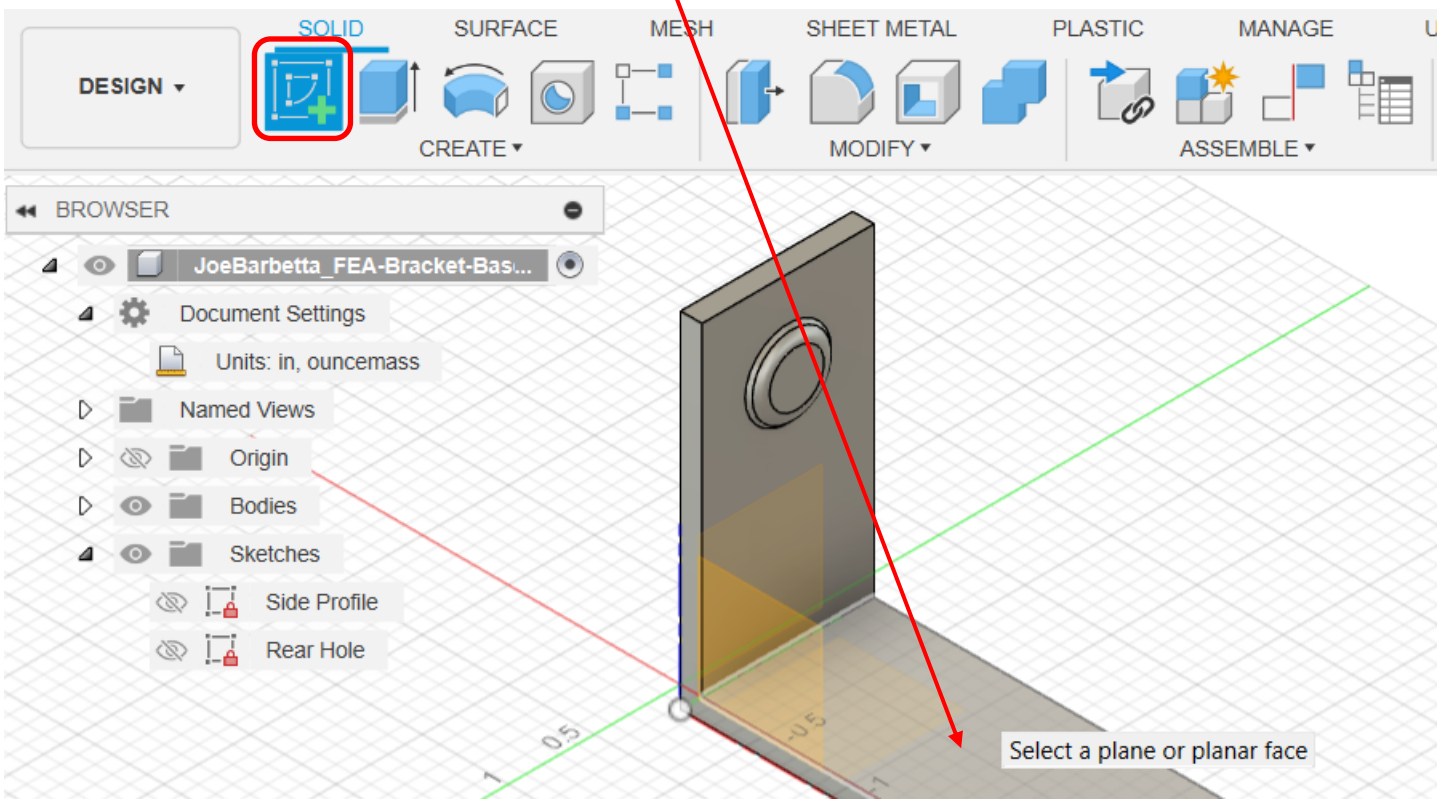
- select the **MODIFY** menu select the **Chamfer** tool and ensure that **Equal Distance** is selected
- click on the **outer edge** of the circle and enter **0.05** and click **OK**.



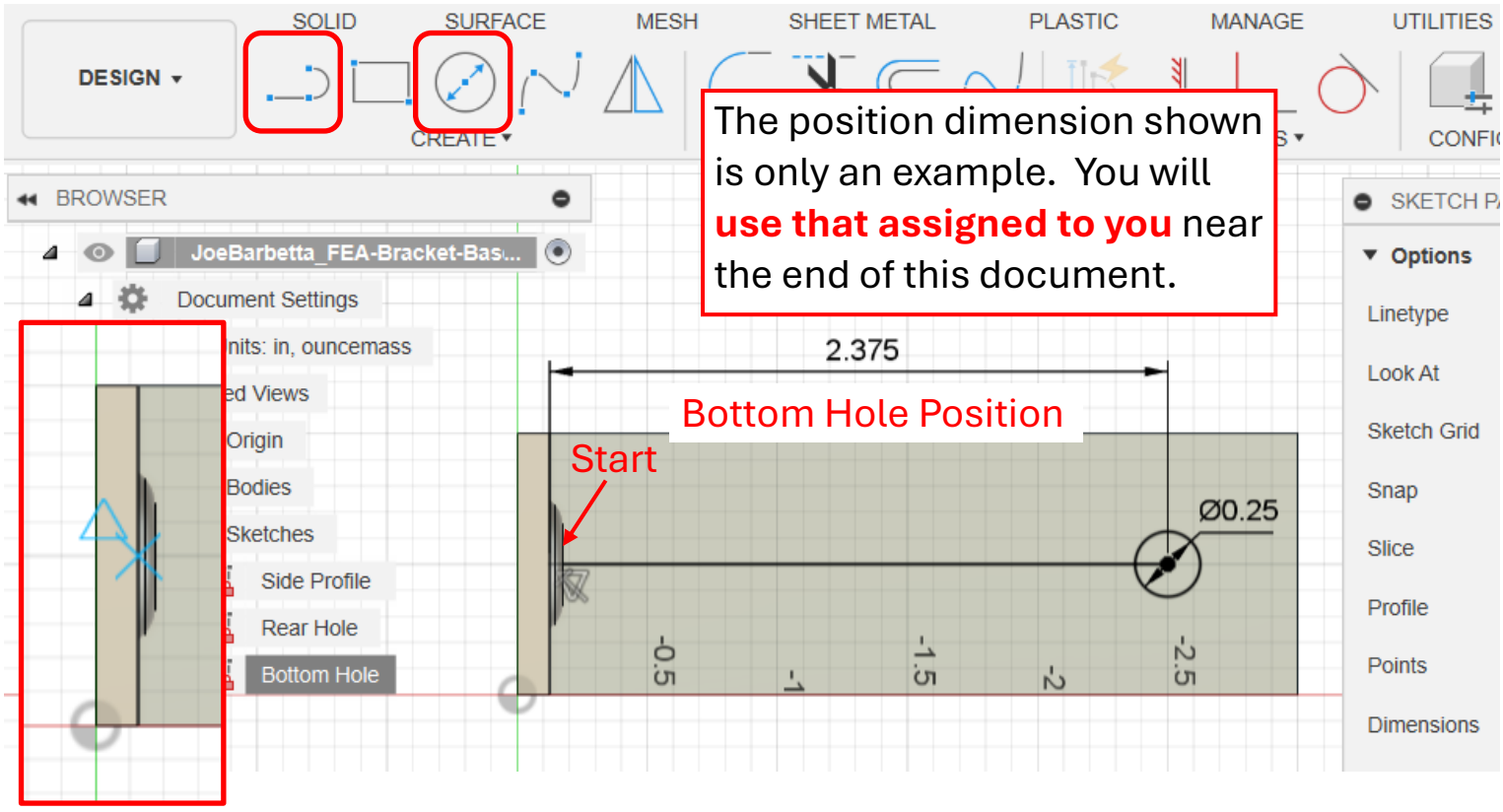
- from the **MODIFY** menu select the **Fillet** tool. PRONOUNCE FILLET USING THE **T** AND NOT LIKE THE PIECE OF FISH.
- click on the **edge** of the chamfer as shown, enter **0.1** and click **OK**.



- select the **Create Sketch** tool and click on the **bottom face**. Rename the Sketch to **Bottom Hole**

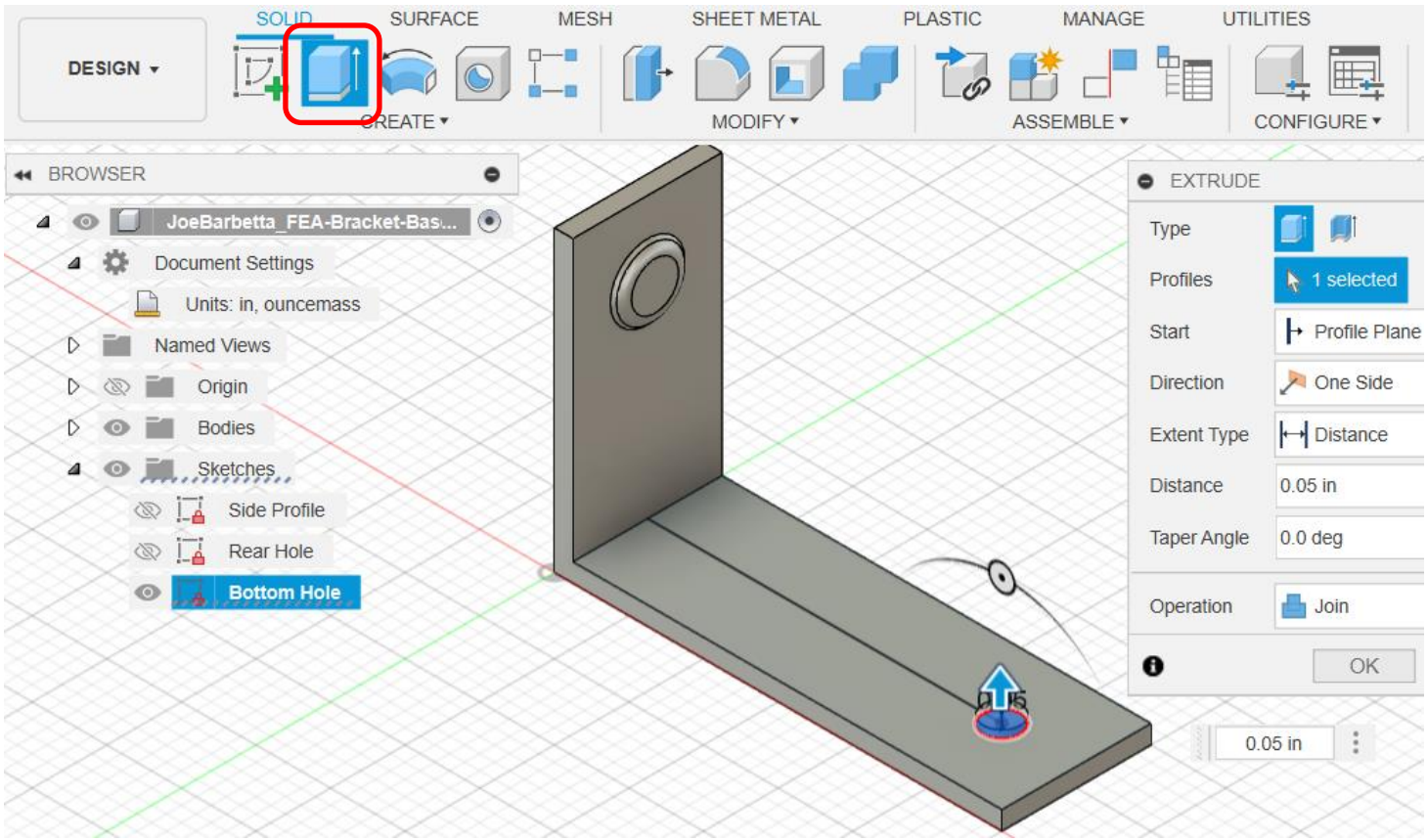


- select the **Line** tool and start the line from the **center of the edge** as shown in the inset picture
- extend the line to the right and enter your assigned **Rear Hole Position**.
- select the **Center Diameter Circle** tool and enter **0.25 for the diameter** and click **Finish Sketch**.

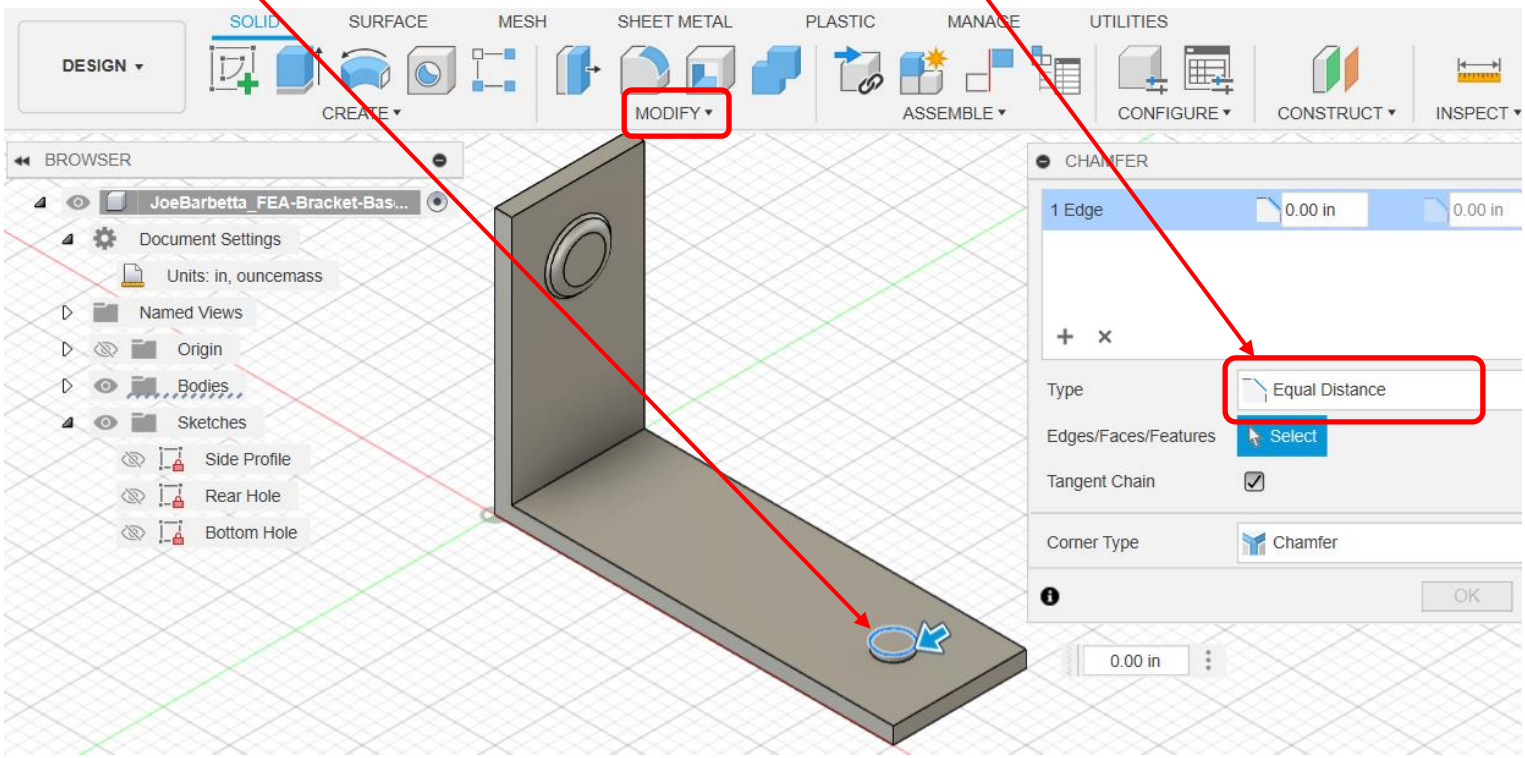


The position dimension shown is only an example. You will **use that assigned to you** near the end of this document.

- select the **Extrude** tool and extrude the circle out by **0.05 in**.



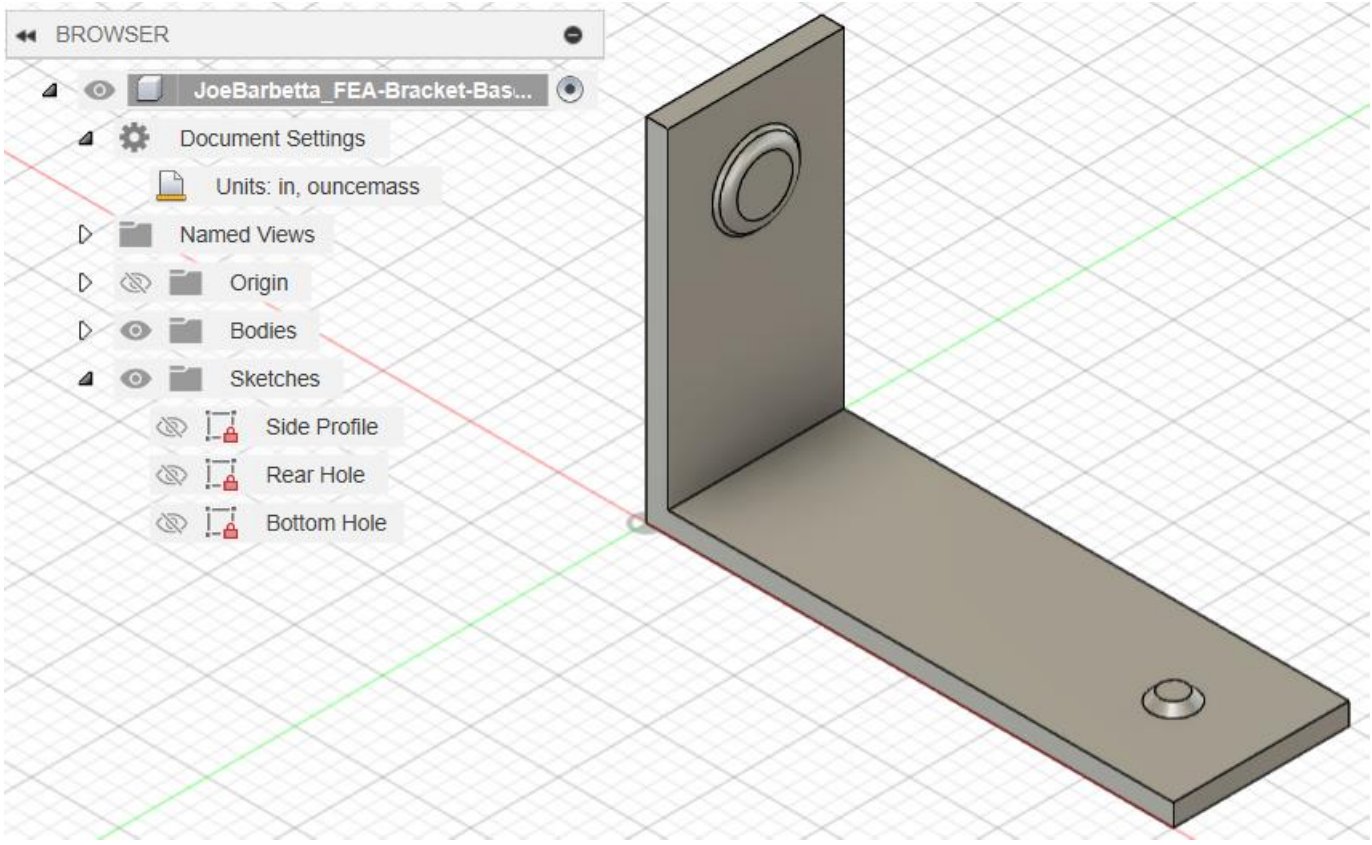
- select the **MODIFY** menu select the **Chamfer** tool and ensure that **Equal Distance** is selected
- click on the **outer edge** of the circle and enter **0.05** and click **OK**.



Here's your bracket.

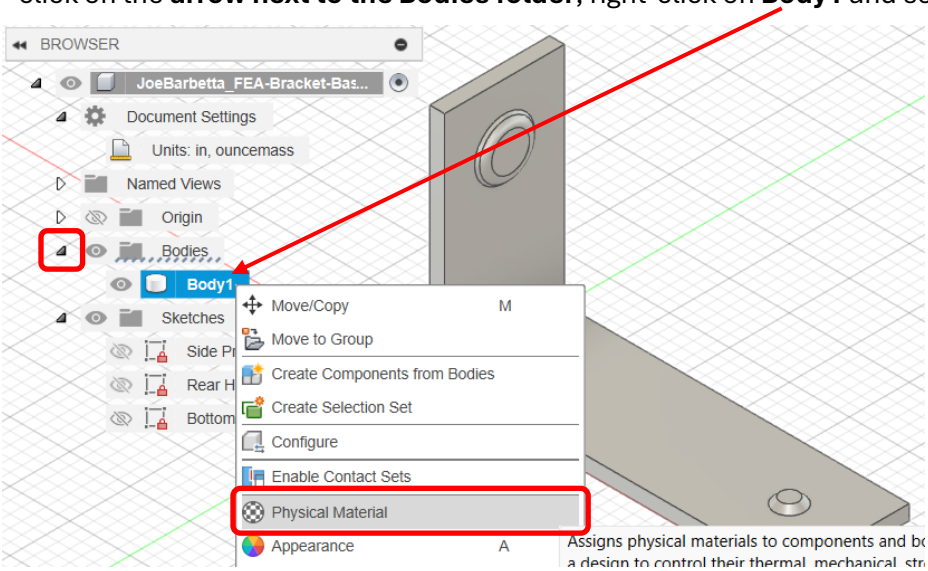
What the Sigma! Shouldn't a bracket have holes!

This is modified for the best application of FEA. Where the holes would be there are protrusions. The shallow protrusions simulate the pressure of the underside of the screw heads. The fillets and fillet help prevent FEA artifacts.

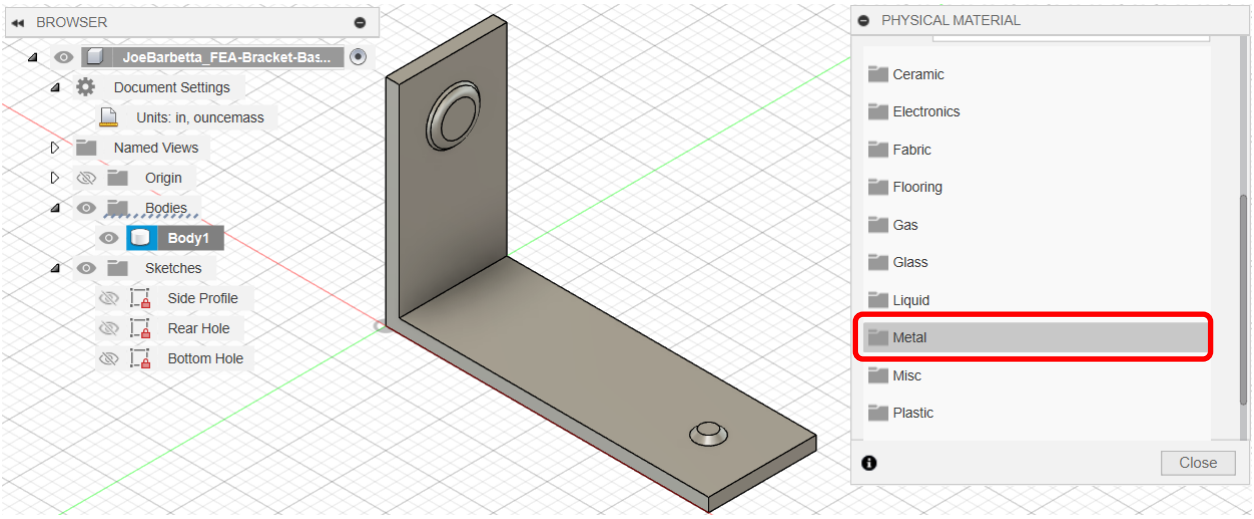


Assigning a Physical Material

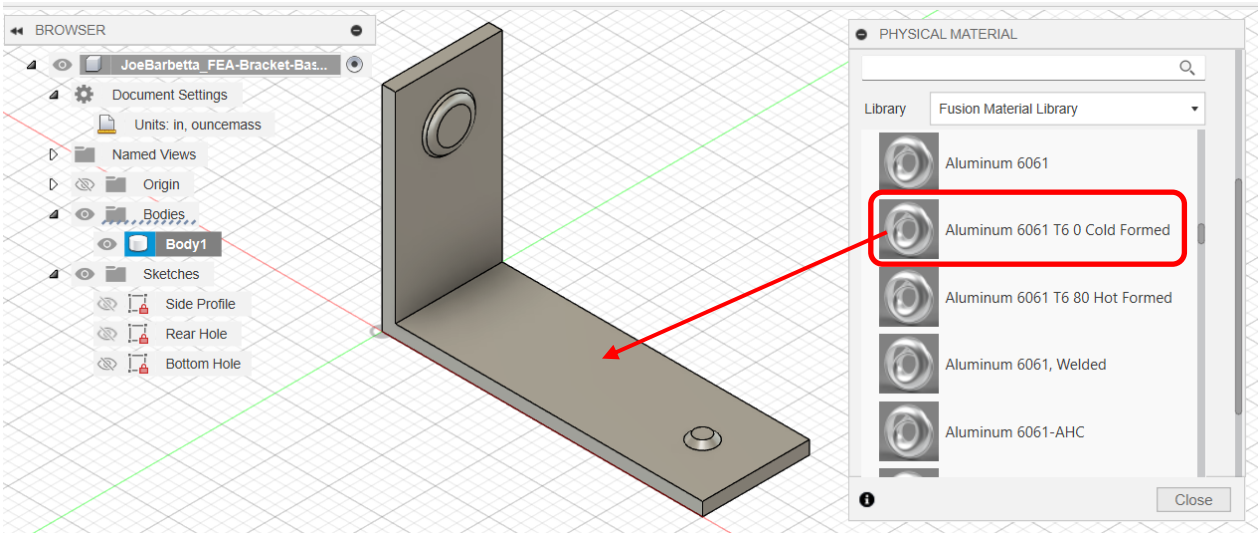
- click on the **arrow next to the Bodies folder**, right-click on **Body1** and select **Physical Material**



- scroll down to the Metal folder

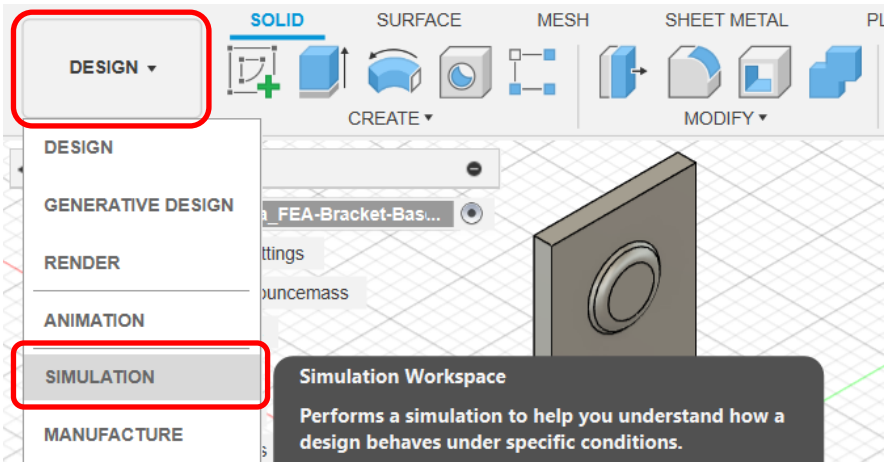


- scroll down to the **Aluminum 6061 T6** and **drag the icon onto the body**, which should lighten the body color



Performing a simulation

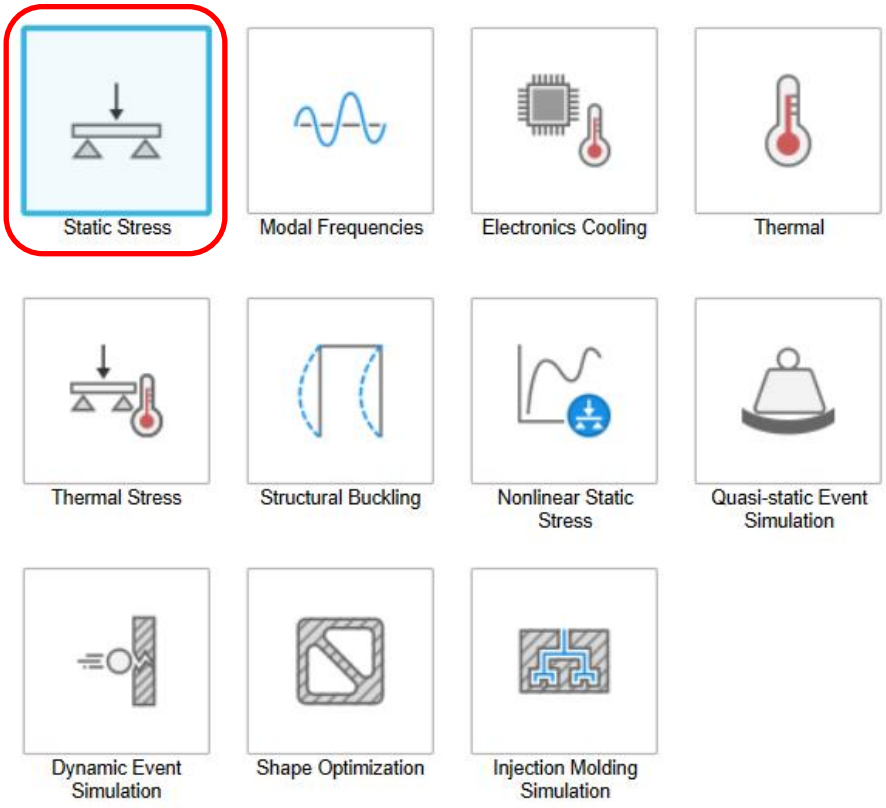
At the upper left switch from the **DESIGN** Workspace to the **SIMULATION** Workspace. Note that one can switch back to the DESIGN Workspace at any time to make design modifications.

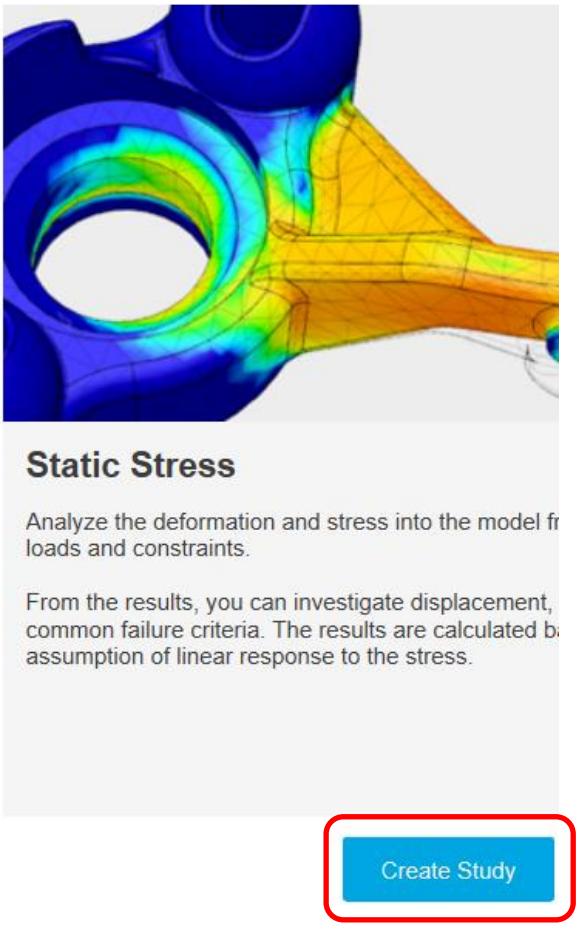


When entering the SIMULATION Workspace for the first time, one is presented with many options.

Look at all the options. **Fusion is the gas!**

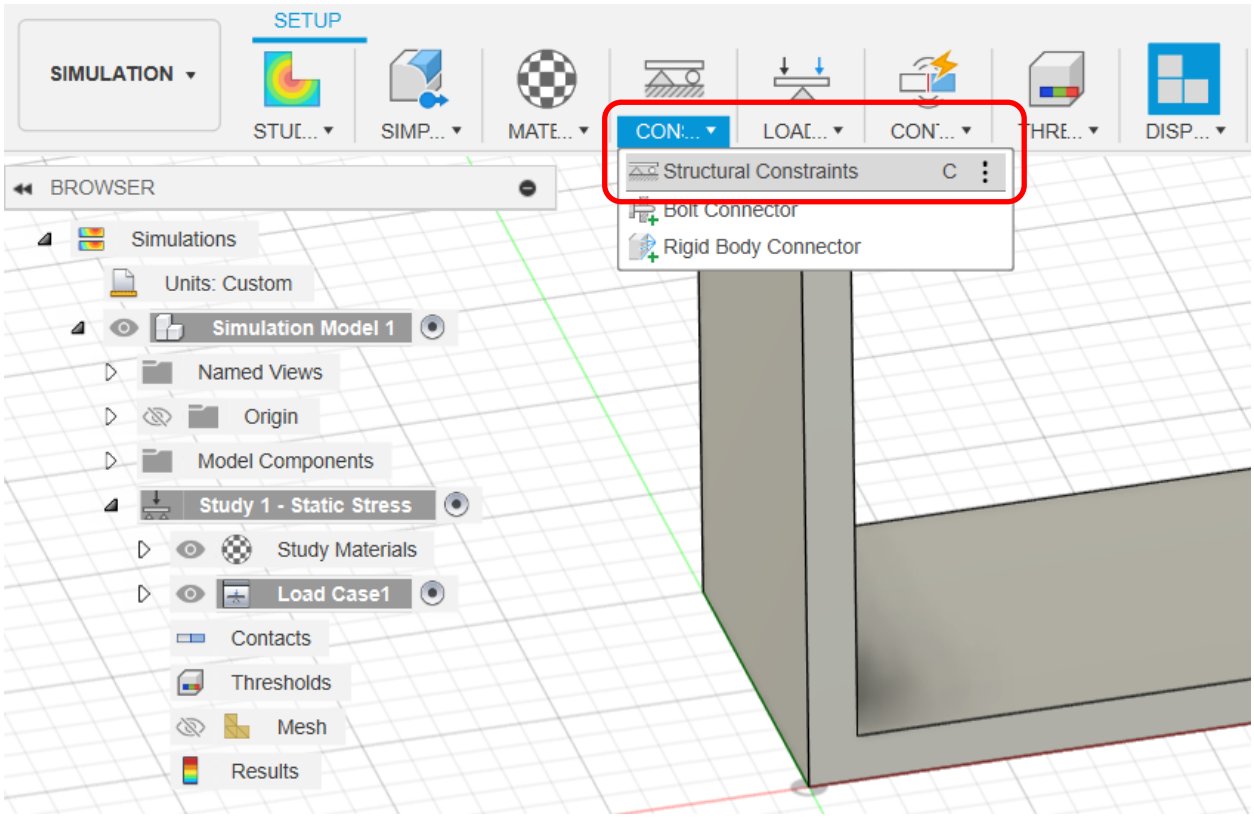
Select **Static Stress** and then click **Create Study**.



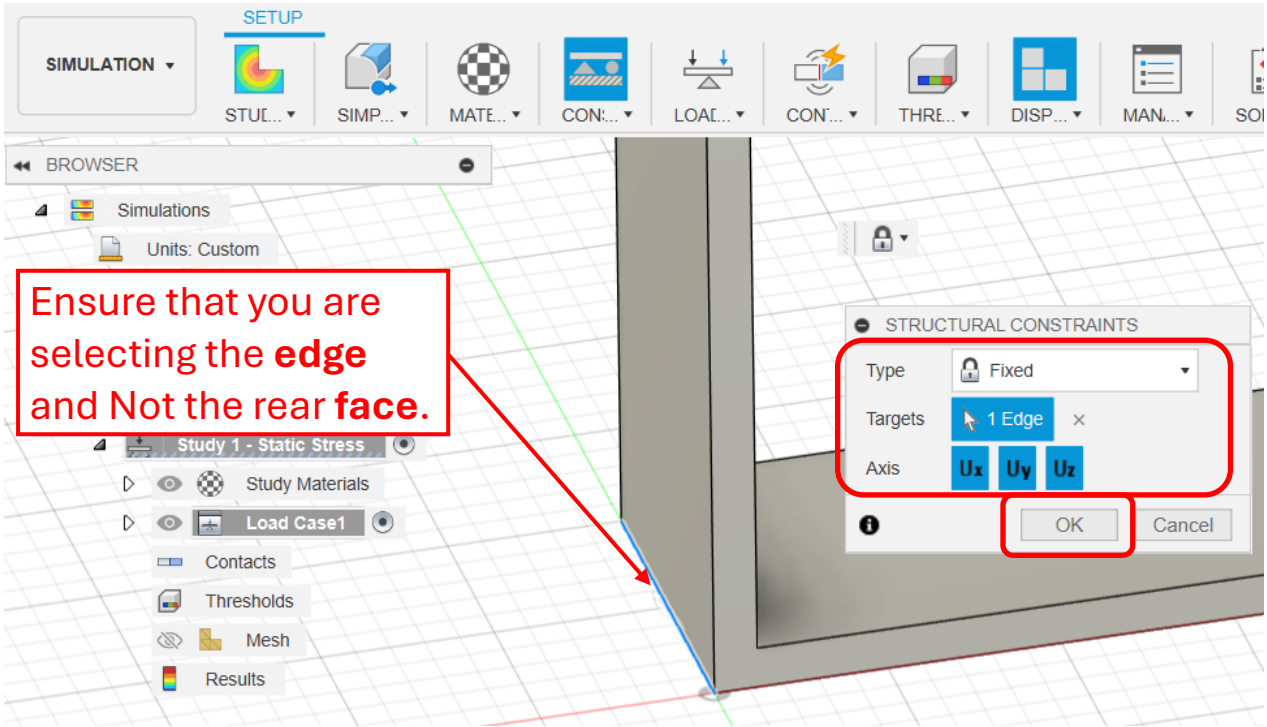


Adding Constraints

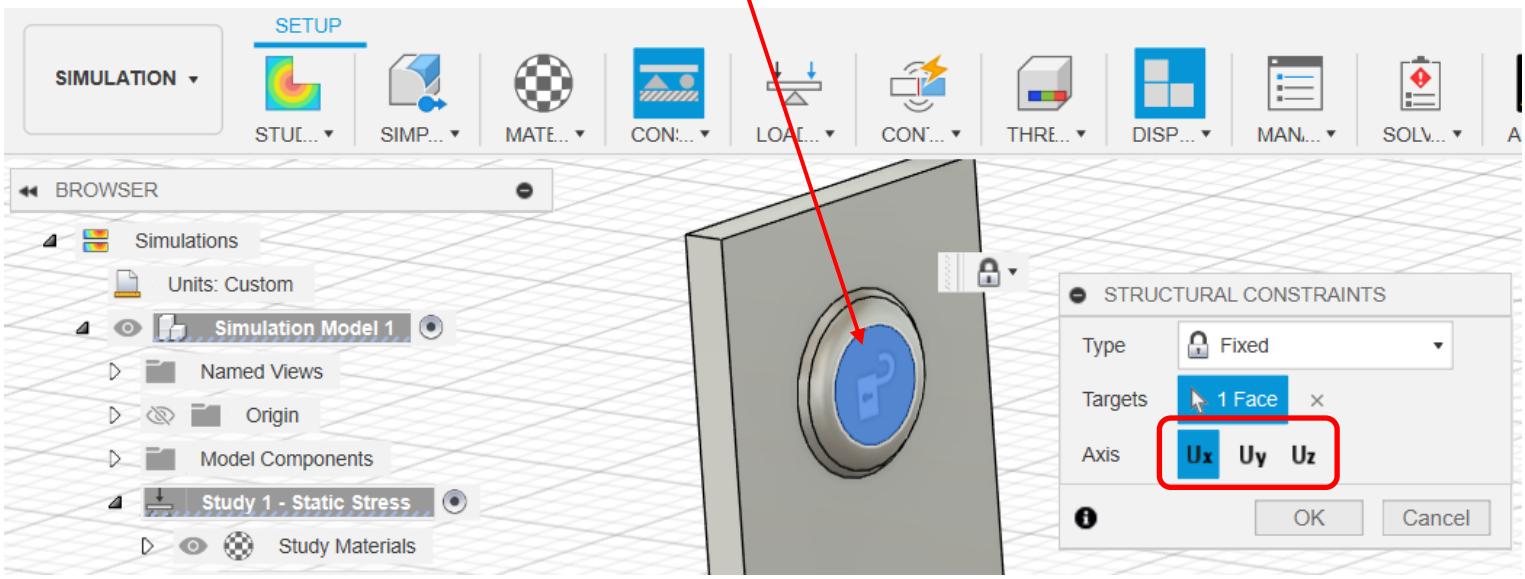
- rotate the view to access the bottom rear edge of the bracket



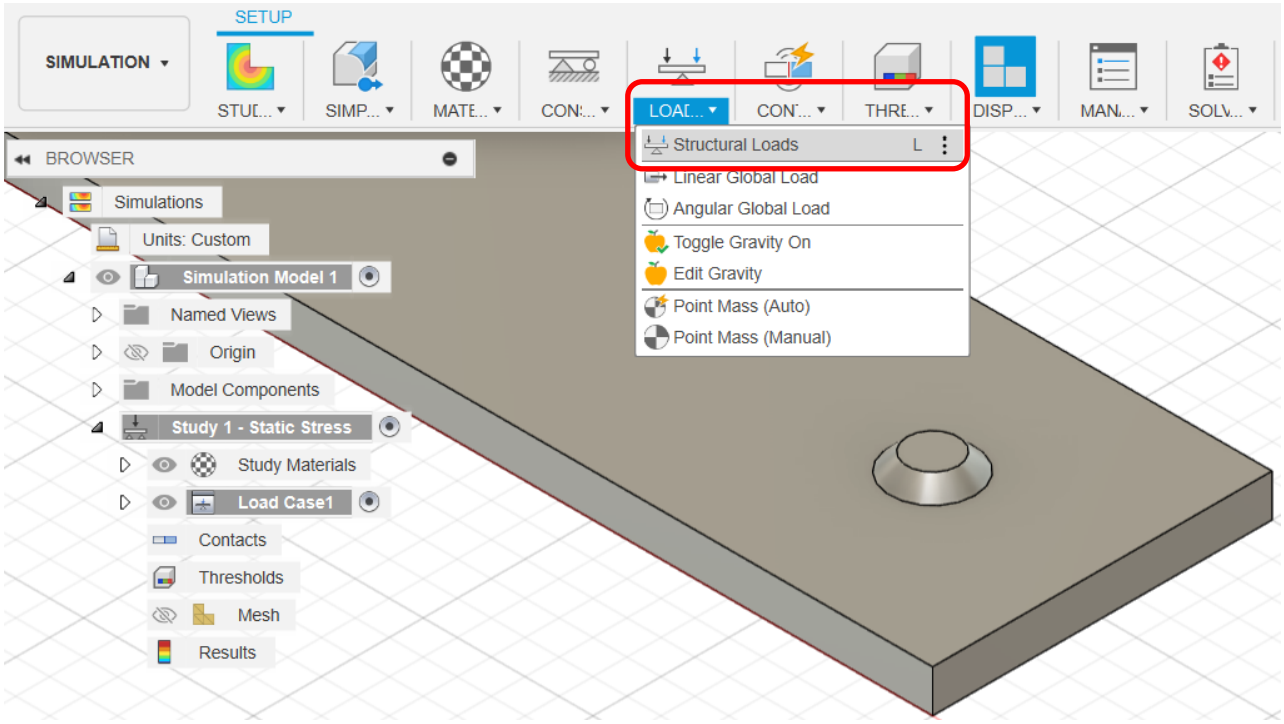
- click on the **edge** as shown. Keep the defaults in the STRUCTURAL CONSTRAINTS window. Click **OK**.



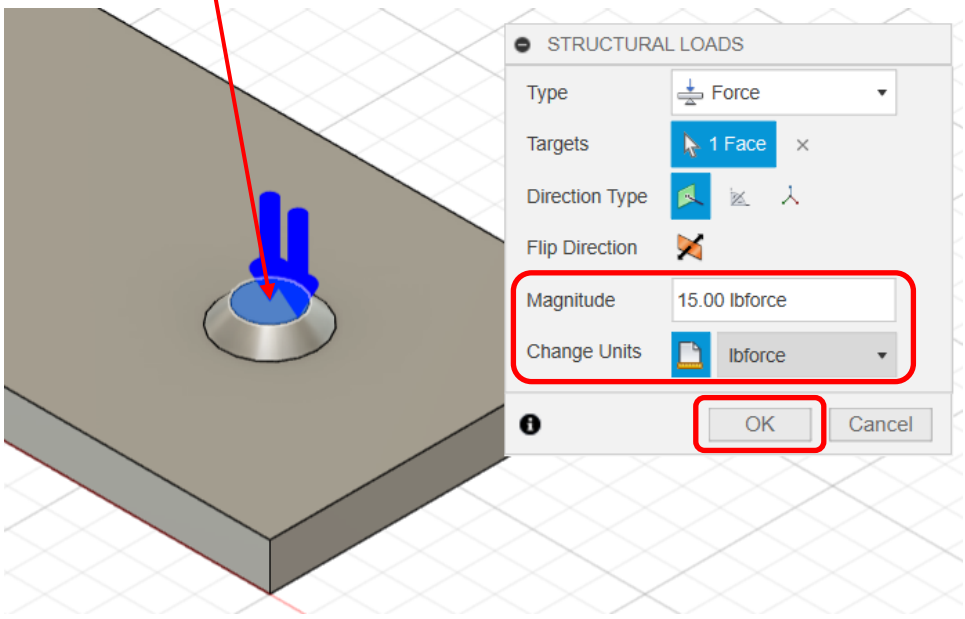
- rotate the view to access the top screw loaction
- select **Structural Constraints** again and **click on the face** of the protrusion.
- click on **Uy** and **Uz** to turn them off. This allows using the x-axis only for the contraints. Click **OK**.



- Zoom in and adjust the view to access the bottom screw location
- from the **LOADS** menu select **Structural Loads**



- **click on the face** of the protrusion.
- enter **your assigned force**, click on **Change Units** and change the units to **lbforce**, then click **OK**.



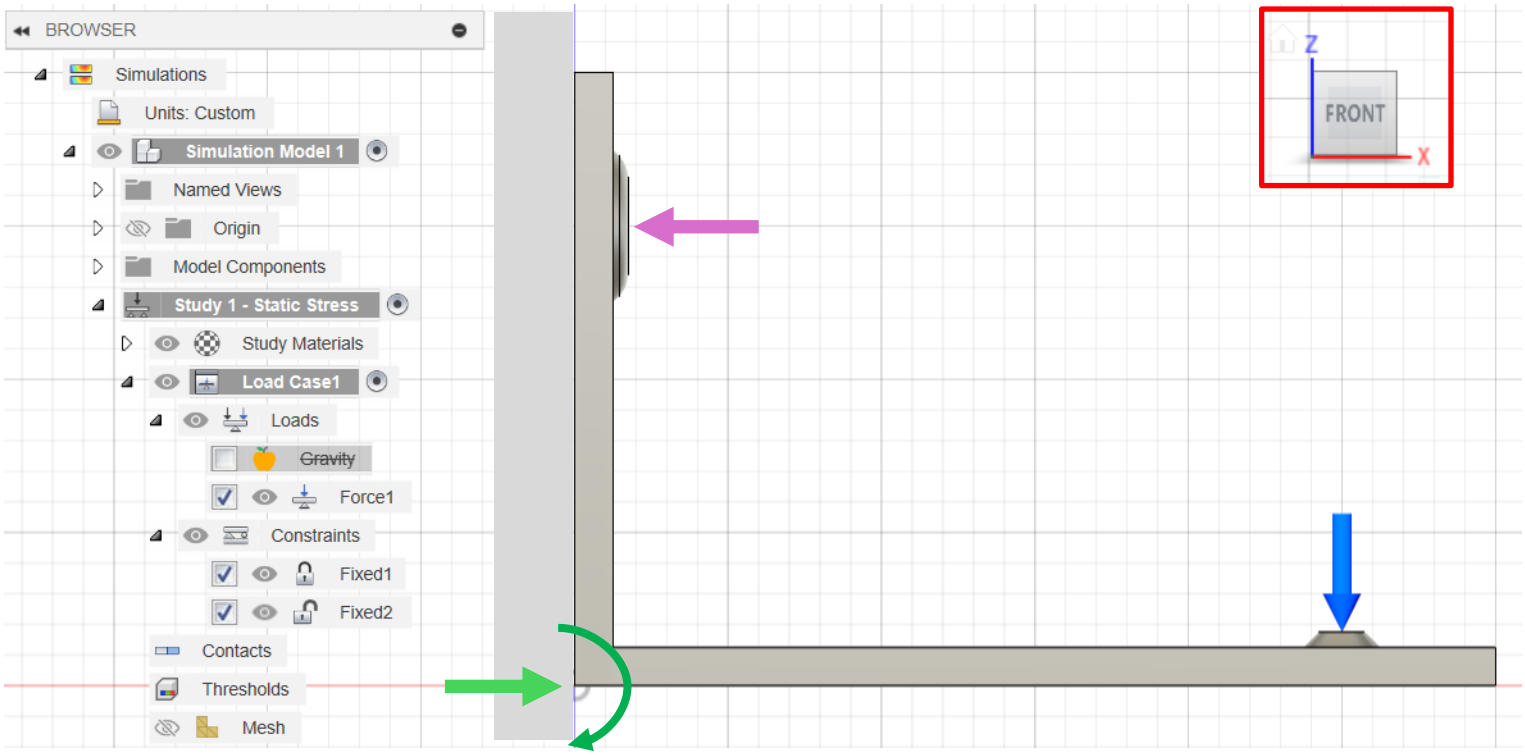
The load value shown is only an example. You will **use that assigned to you** near the end of this document.

- click on the **FRONT** surface of the **View Cube** to achieve the side view of the bracket

The gray block was added here (not in Fusion) just to illustrate a wall that the bracket is connected to.

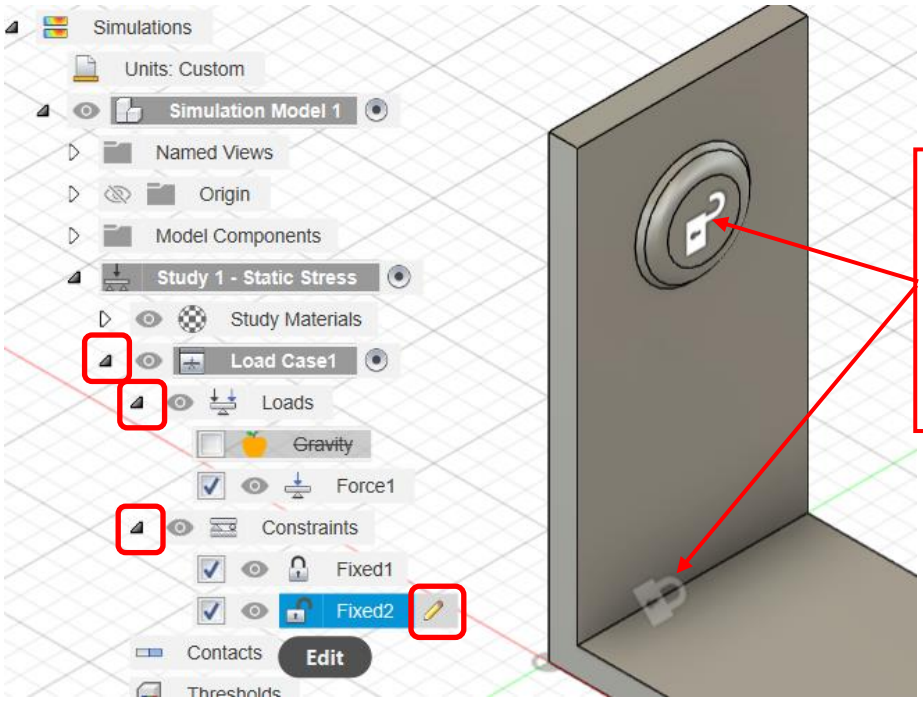
The Constraint that was added at the bottom rear edge (represented by the green arrow) was placed at the edge to allow the bracket to rotate around this point. The amount of rotation will be tiny, but this provides a pivot point. A moment will exist here as a force is applied near the end of the bracket. The wall also "pushes" against this edge as represented by the green arrow.

The violet arrow represents the force of the screw holding the top of the bracket against the wall. This force is "trying" to pull the screw out of the wall. Note that there is often one or more screws securing the bracket against the wall. However, the topmost screw will experience the greatest "pull-out" force and this problem was simplified by only including this screw. Note that any of these screws are also subjected to a downward shear force, but a common failure mode of such as appication is the failure of the top screw, which is under tension.



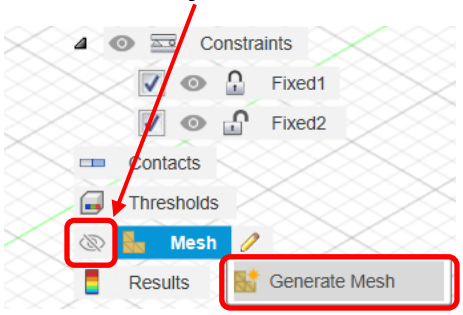
- return to the **Home view** and click on the arrow next to **Load Case** and then **Loads** and **Constraints**

When hovering over a Load, eg Force1, or a Constraint, eg Fixed1, a pencil icon shows. This can be clicked on to make any changes.

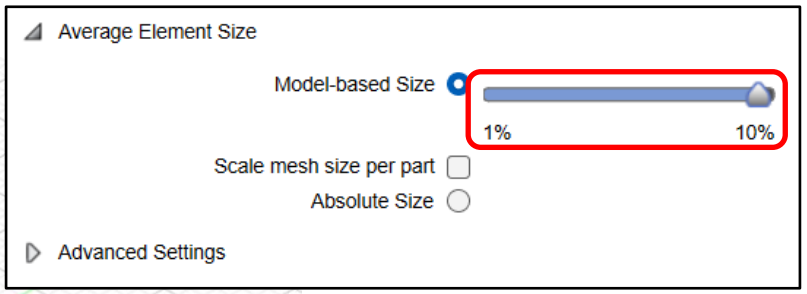


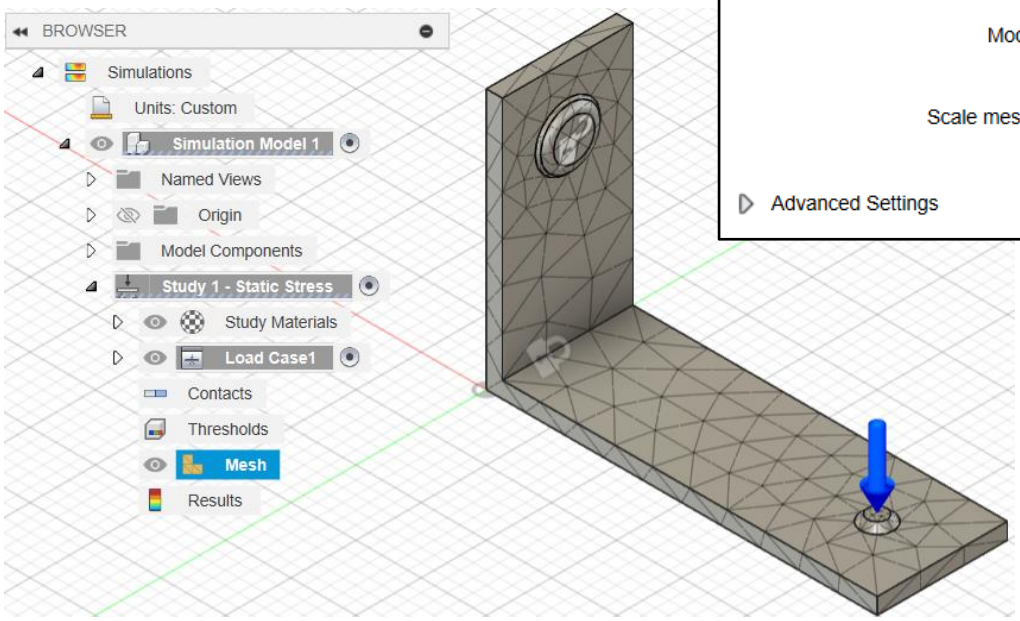
Note the icons for the Constraints. The top one is shown Unlocked because an axis was disabled. The bottom one is shown Locked because all 3 axis were kept as enabled.

- near the bottom of the left BROWSER items, right-click on **Mesh** and select **Generate Mesh**
- click on the **eye** icon next to **Mesh**. The view should look like the picture lower down.

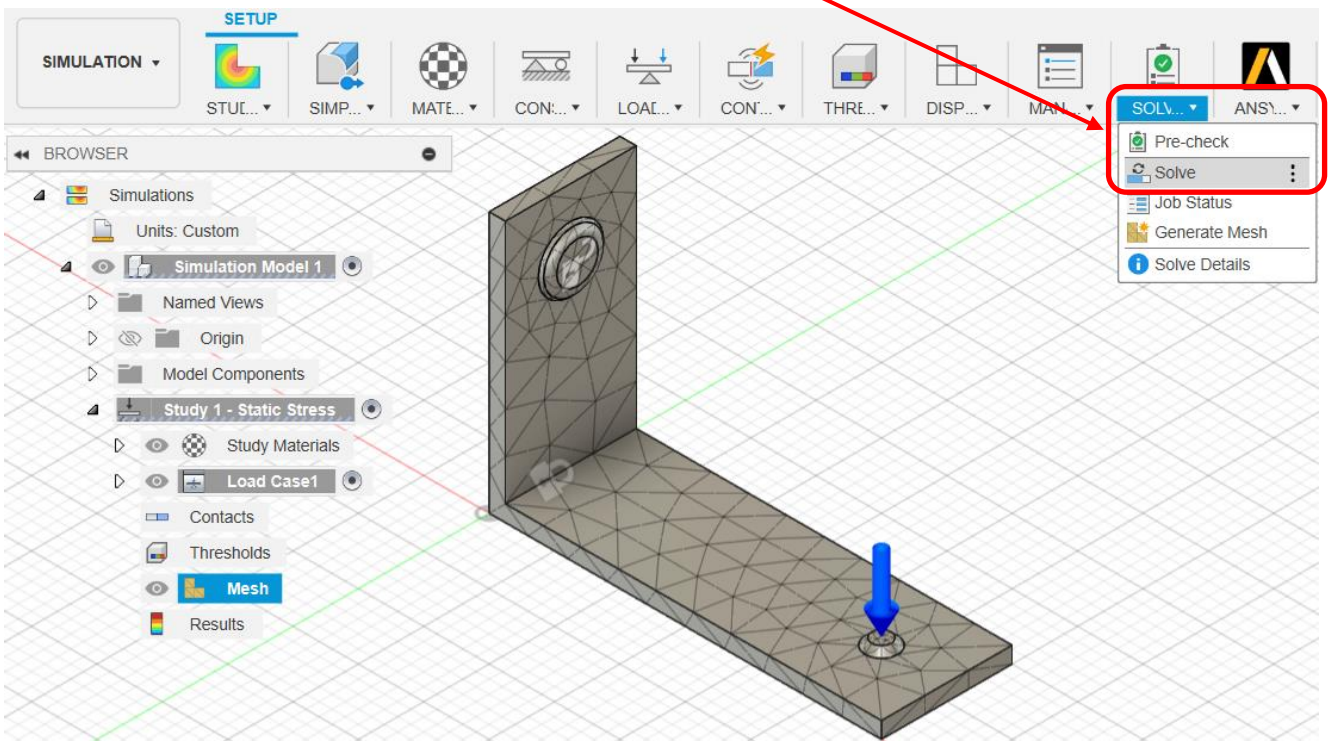


One can click on the pencil icon next to Mesh to access the below settings. We will keep the 10% default value to lesson the solve time. Moving the slider to the left will produce more mesh elements, which can lead to more accurate simulations.

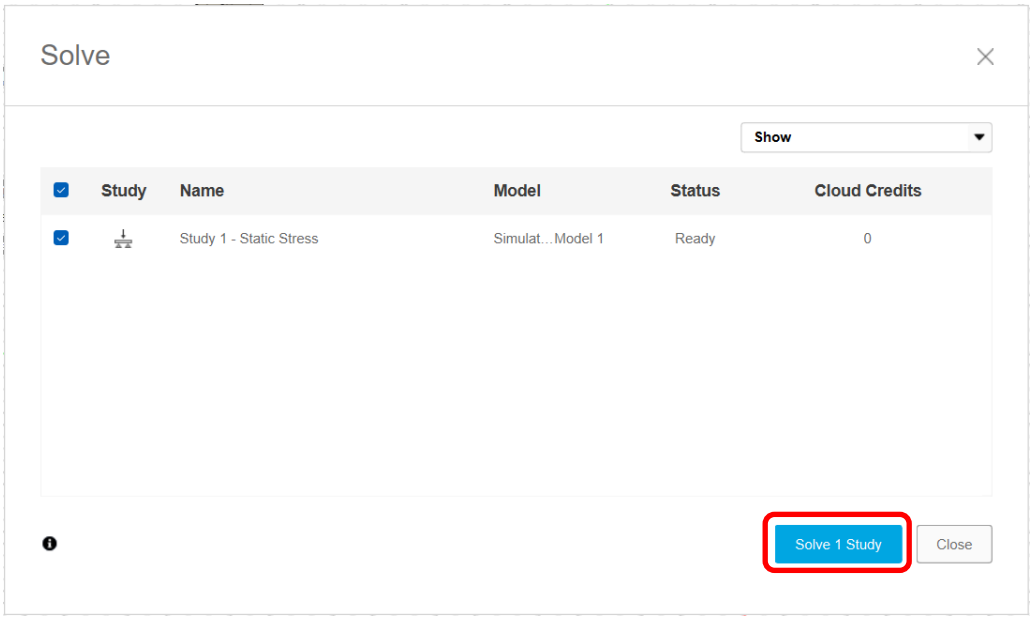




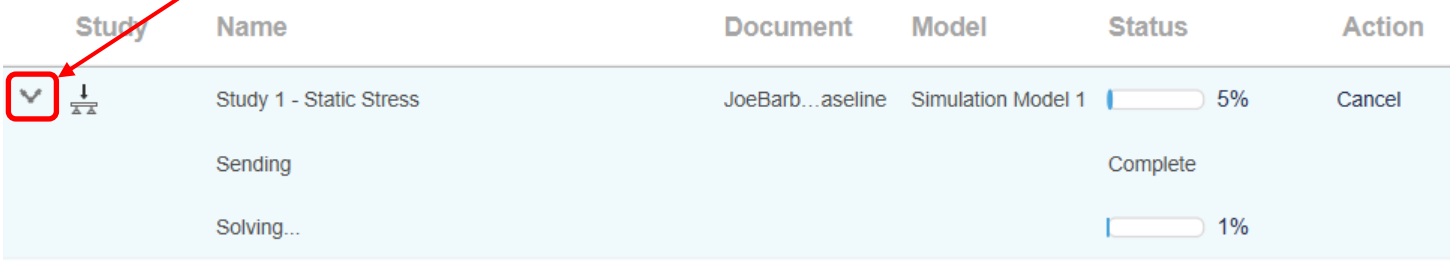
- from the top SOLVE menu select **Pre-check** and then **Solve**



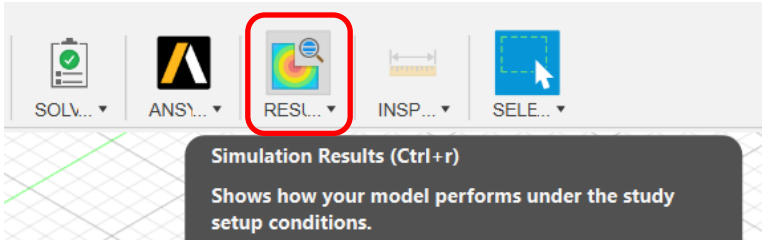
- click on **Solve 1 Study**



- click on the **progress** icon. The Sending task will complete fairly quickly. The Solving task can take 1 to a few minutes.

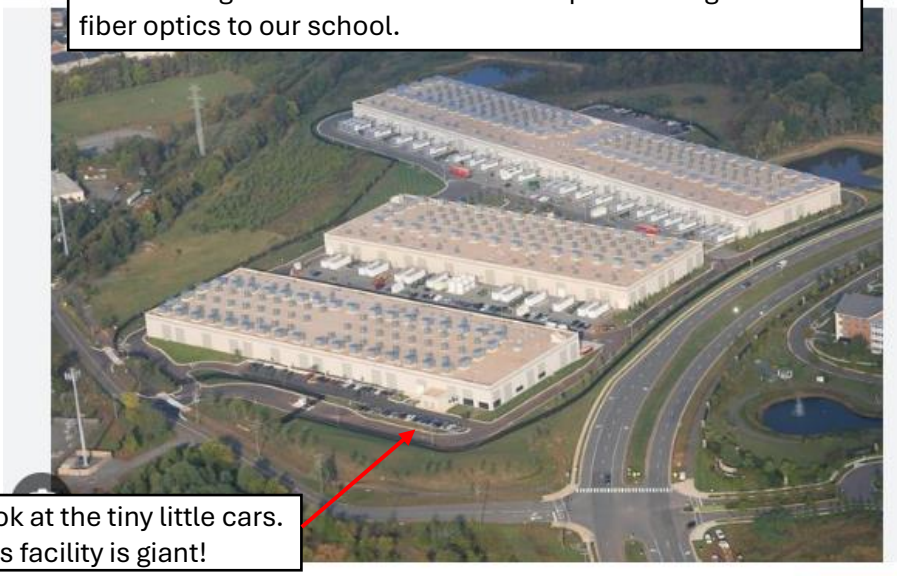


- when the simulation is solved close the progress screen and click the **Simulation Results** icon



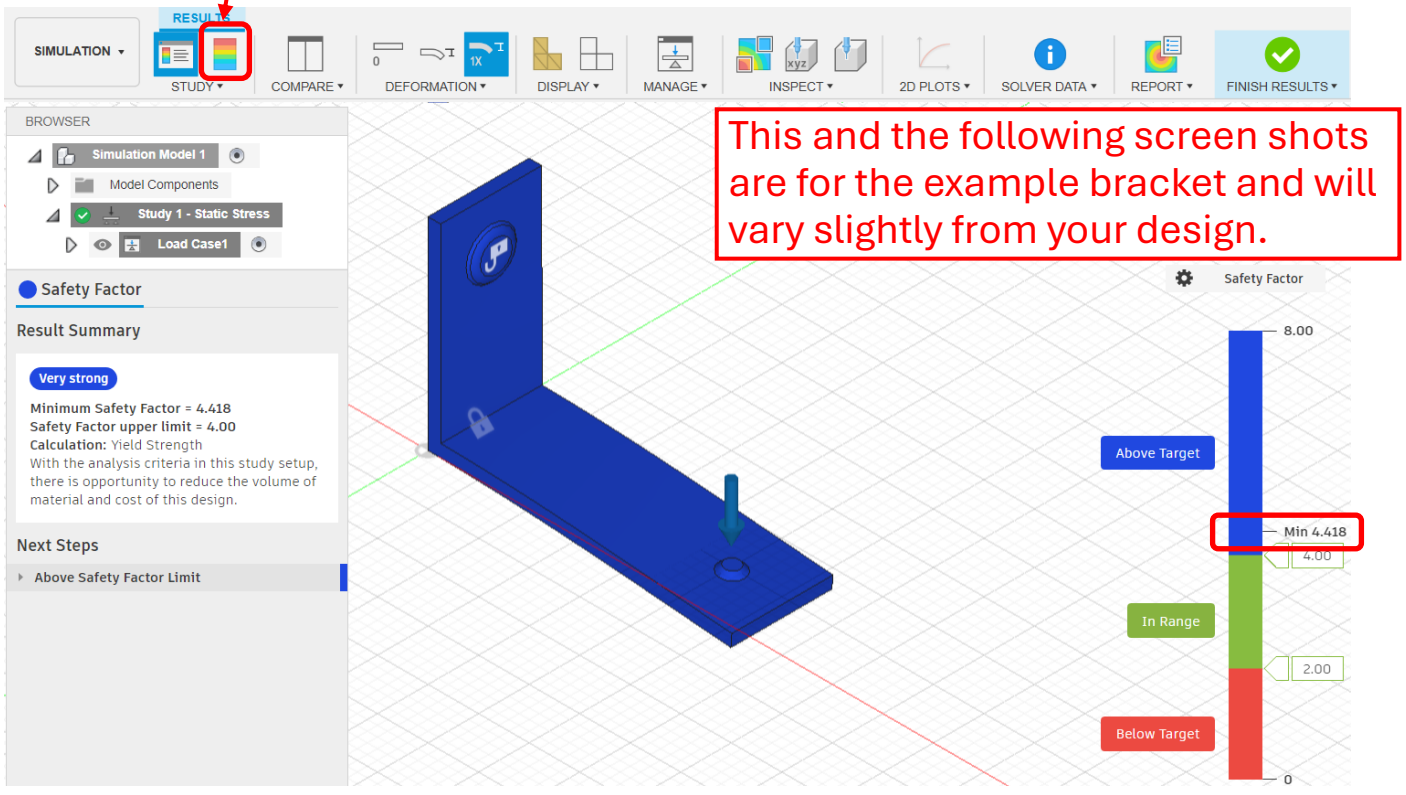
- watch the neat “data from the cloud” animation

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.

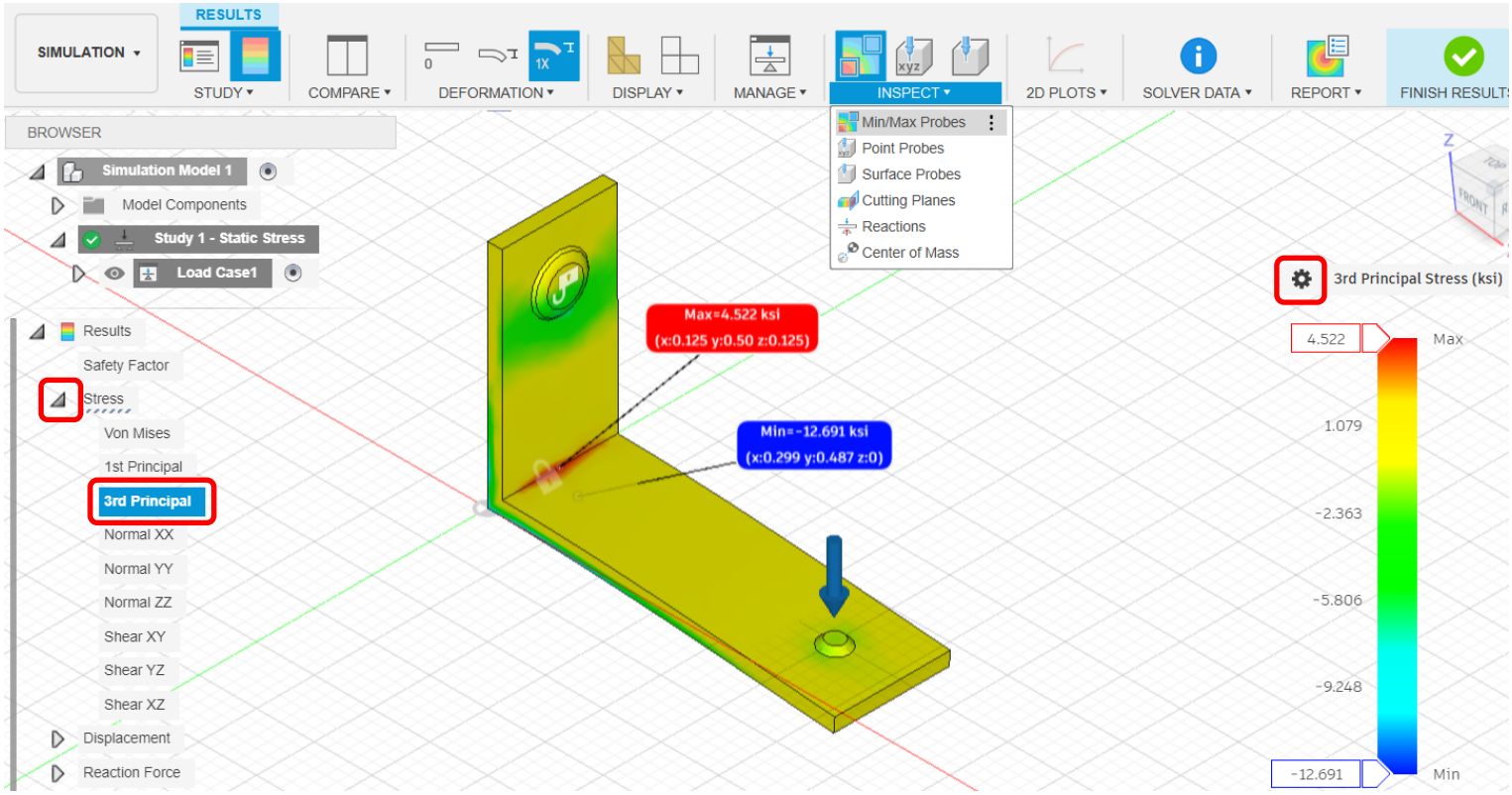


- click on the **Results** icon. The default screen is that of the **Guided Results**

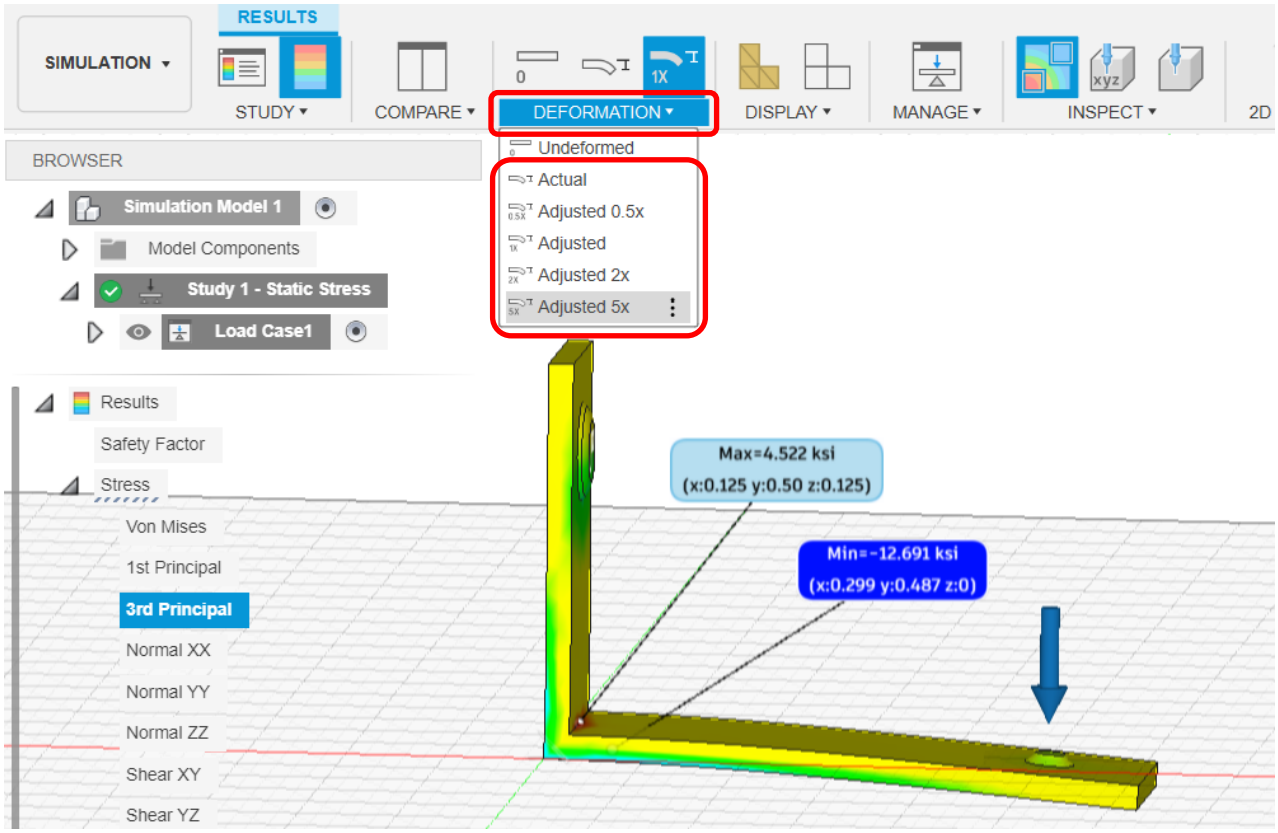
Note that the **Safety Factor** is stated as **4.418** for this example



- click on the **arrow** next to **Stress** and then select **3rd Principal**
- click on the **gear** icon above the scale graphic and change the units to **ksi**. **1 ksi = 1000 psi (lbs/in^2)**. This the below example shows the maximum stress of **4.522 ksi (4,522 psi)** occurs at the inside of the bend. It is common for stress concentrations to occur at sharp inner corners.

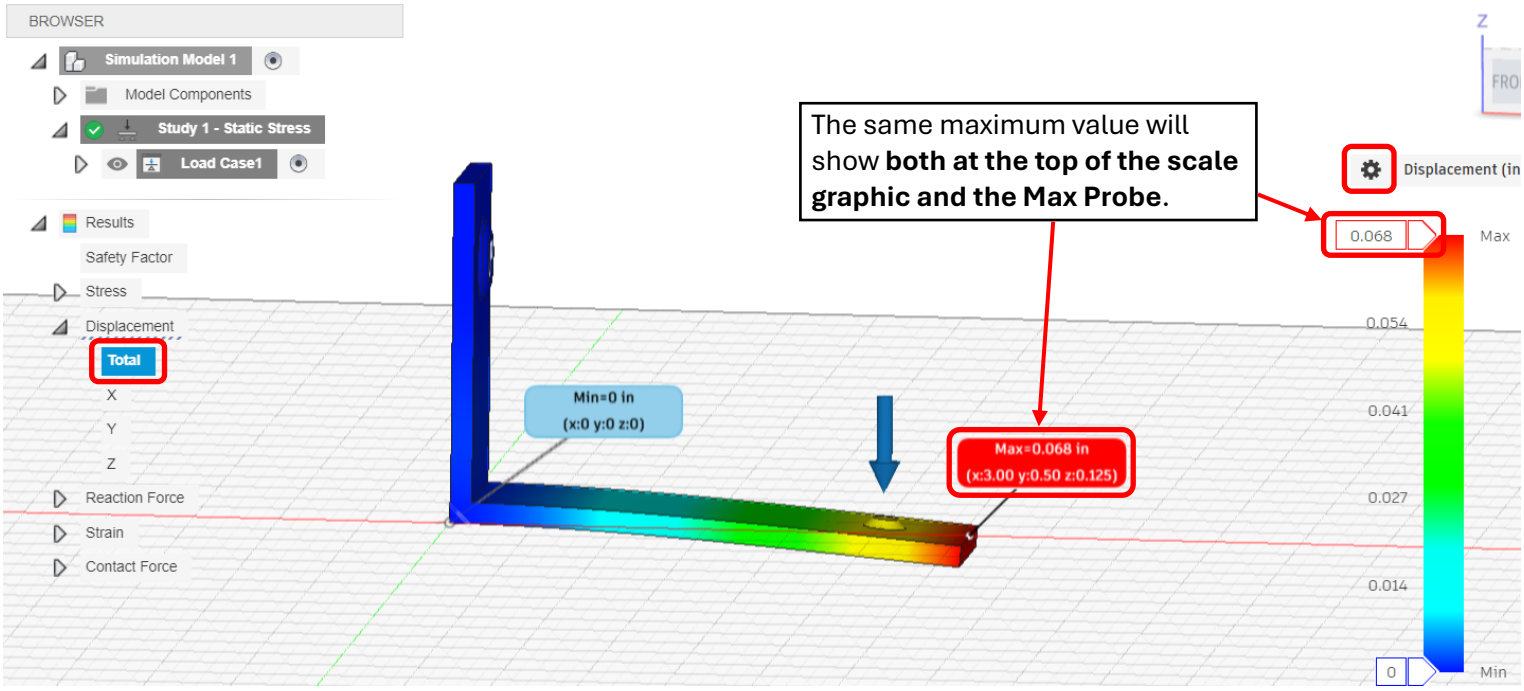


- adjust the view to see more of a side view
- under **DEFORMATION** select **Adjusted 5x** and then **Actual** and note the difference. By default, Fusion displays an Adjusted view to exagerate deformations, which is helpful to see where deformations occurs.

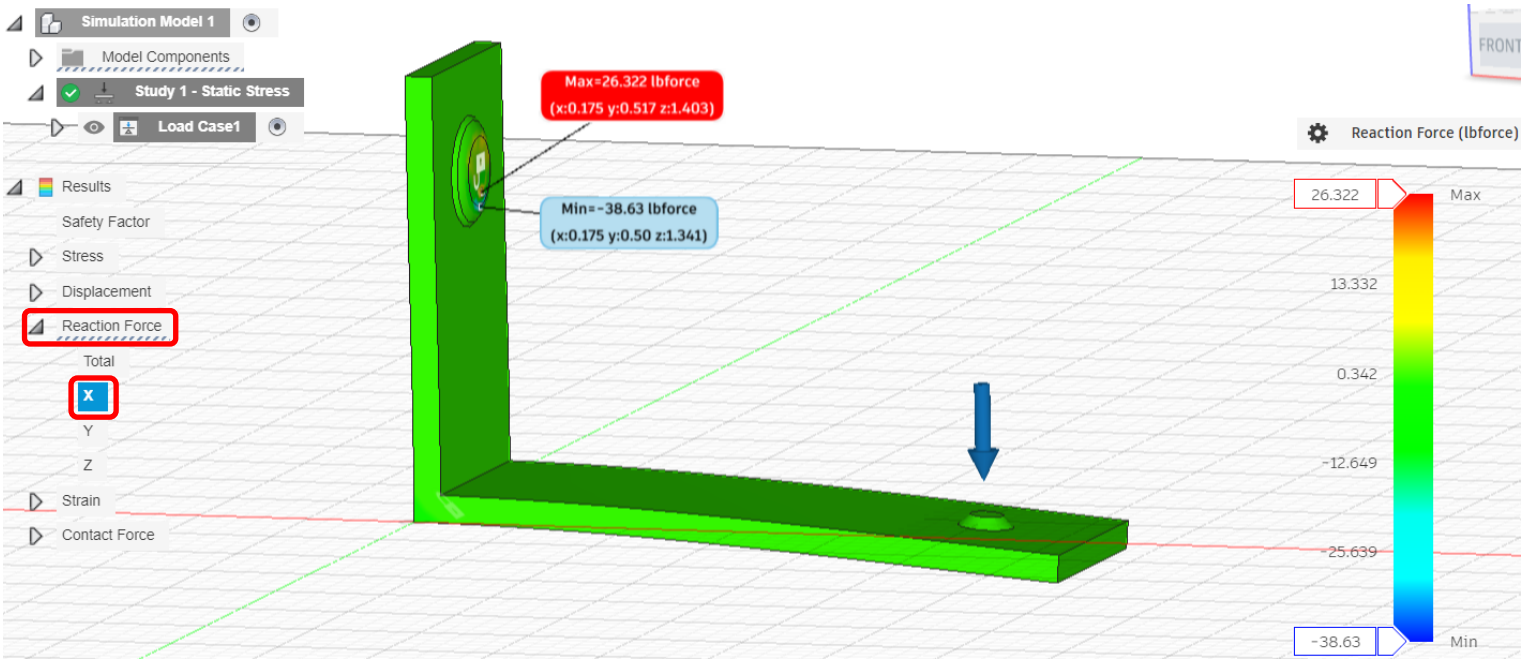


- click on the **arrow next to Displacement** and select **Total**.
- click on the **gear** icon to change the units to **inches** if needed.

Note the **Max deflection** at the end of the bracket of 0.068 in for this example.

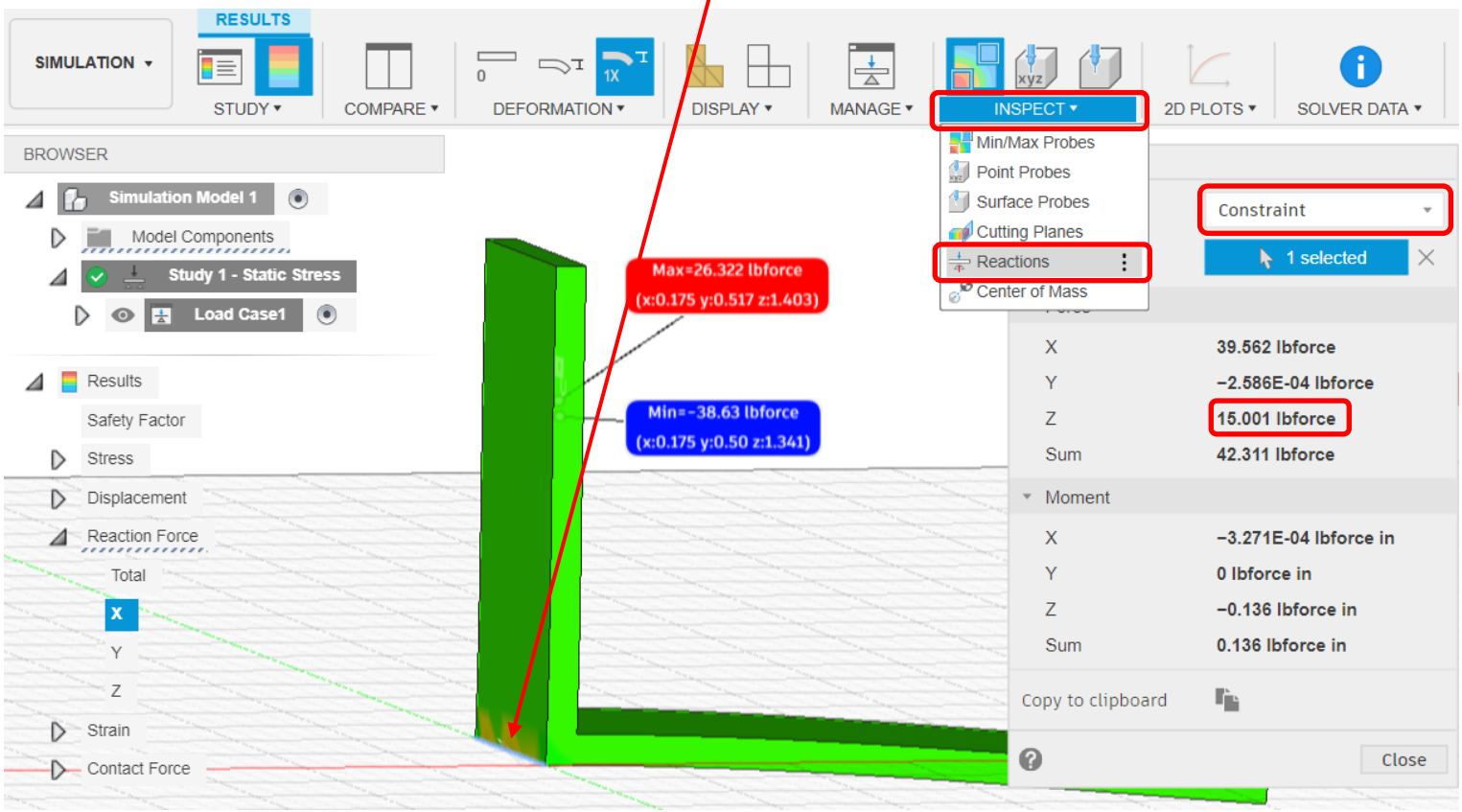


- open the **Reaction Force** options and select **X**



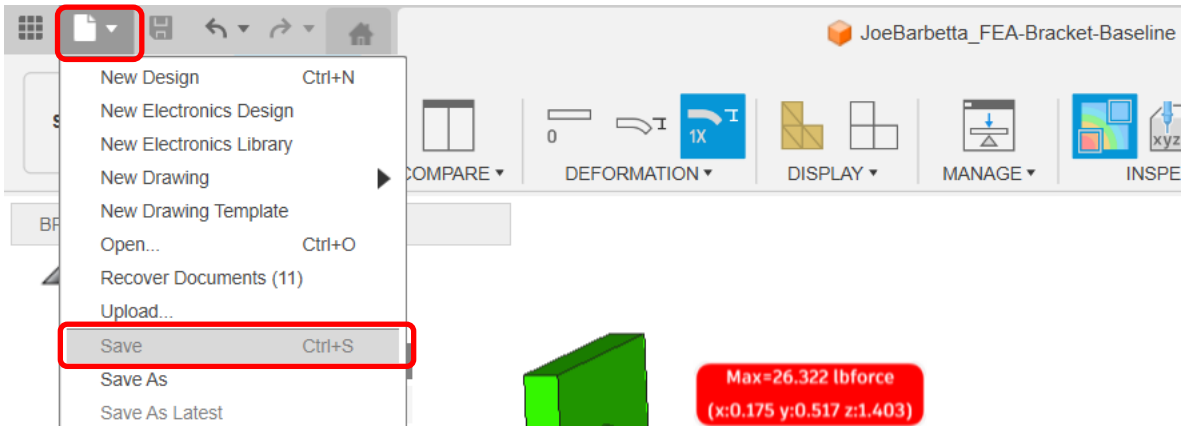
- rotate the view to access the rear bottom edge of the bracket
- from the **INSPECT** menu select **Reactions**
- ensure that **Constraint** is selected and click on the **rear bottom edge** of the bracket

Note that the **Z** Force is equal to the load.



- **Save** your project. The name should end with **Baseline**, which was set at the start of this document.

Later on, when modifications are made, the name can be changed. This will allow multiple projects to be saved, allowing one to open a previous design.

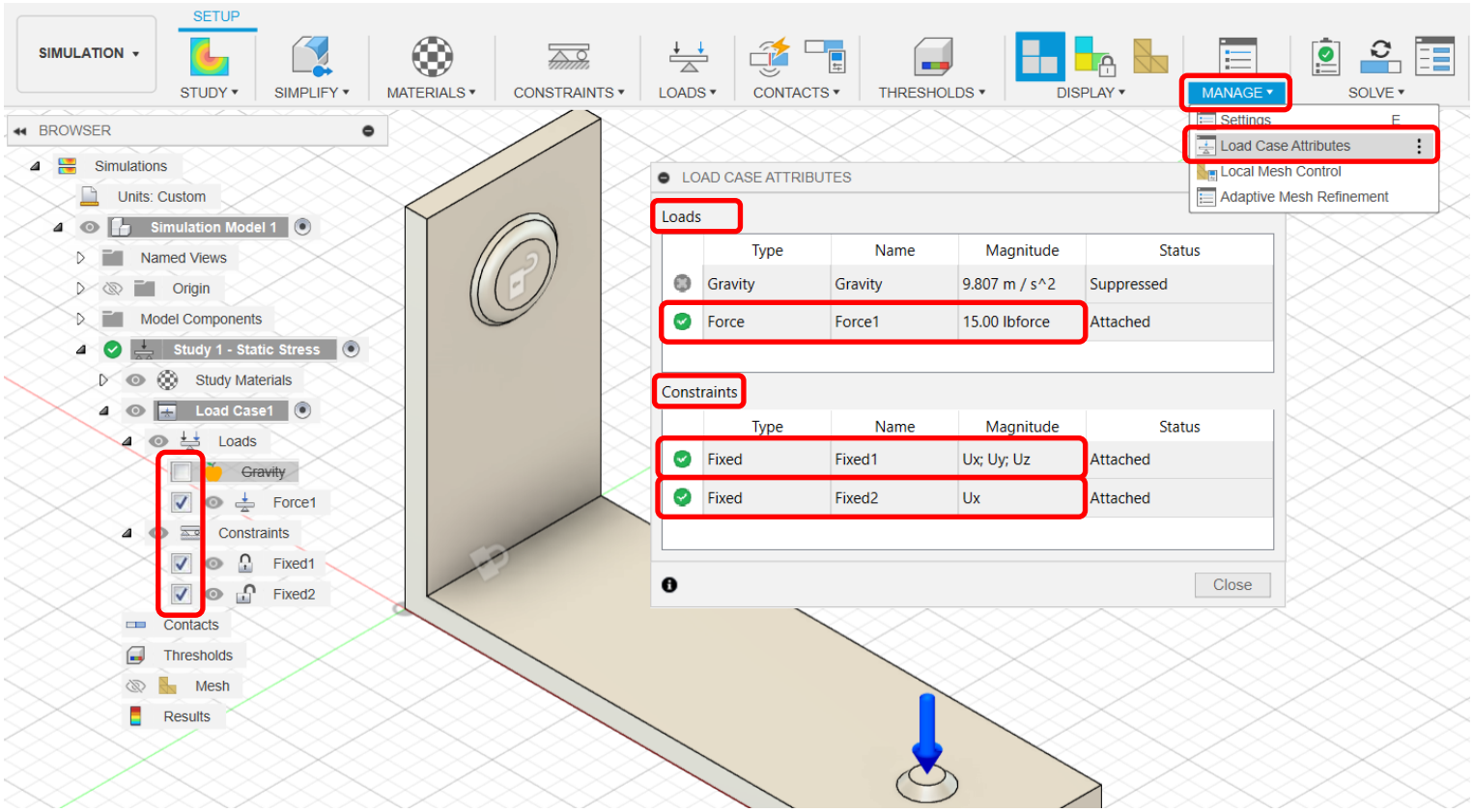


Listing Loads and Constraints

- from the **MANAGE** menu select **Load Case Attributes**
- using the **bottom-right border** of the LOAD CASE ATTRIBUTES window, enlarge the window to view all the items

Note that it shows the **Magnitude** of the Load **(Force**) applied and the **Axes** of the **Constraints**. It shows that for one constraint we added, the Y and Z axes were disabled.

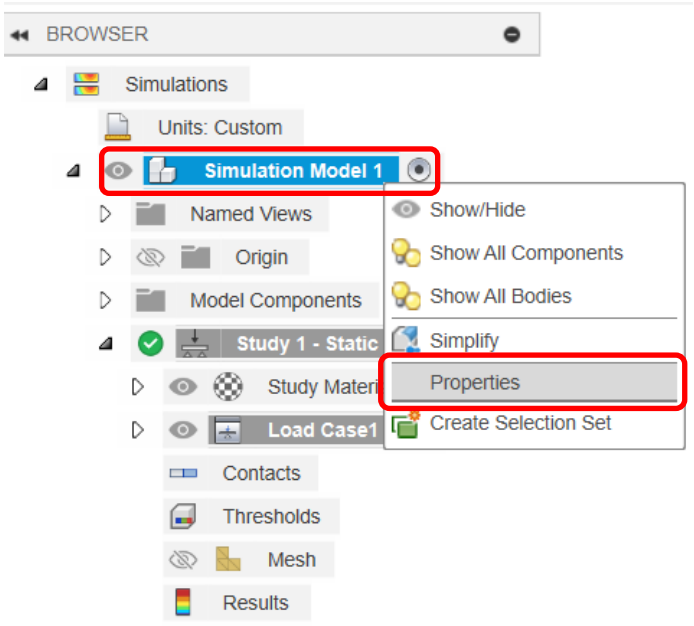
Note that the force of gravity can also be enabled. For many situations, such as this bracket, gravity on the bracket itself barely plays a role. For a large structure, the use of gravity would be more relavent. There are check boxes in the BROWSER under Loads and Constraints to toggle their enabled or disabled status.

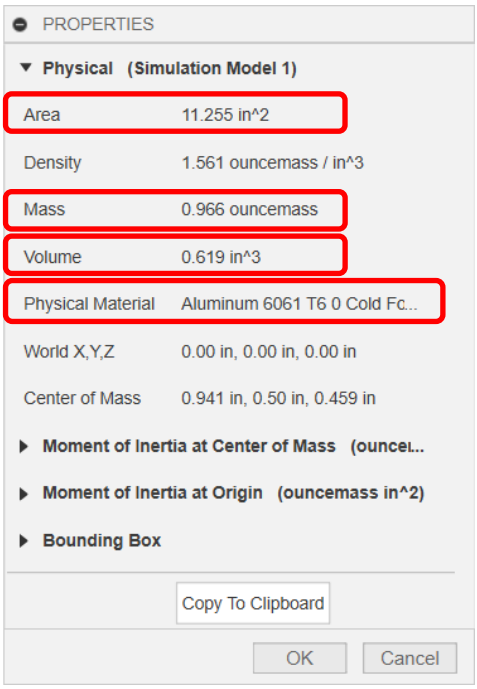


Determining the mass of design body

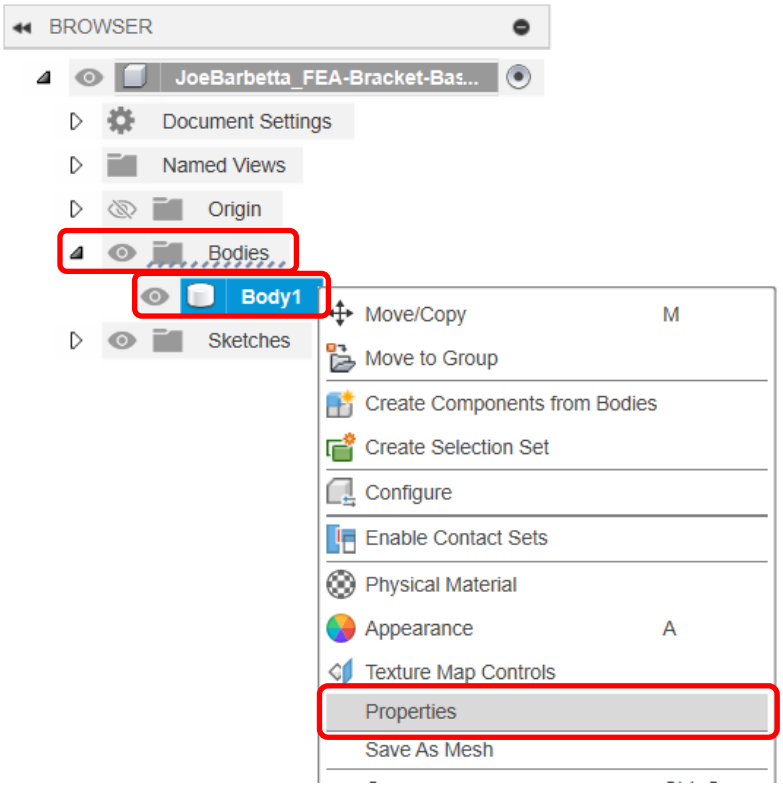
The mass of your design can be determined from the **PROPERTIES** window in either the **SIMULATION** or **DESIGN** Workspaces. The **Area** (surface area), **Volume**, and **Physical Material** can also be determined.

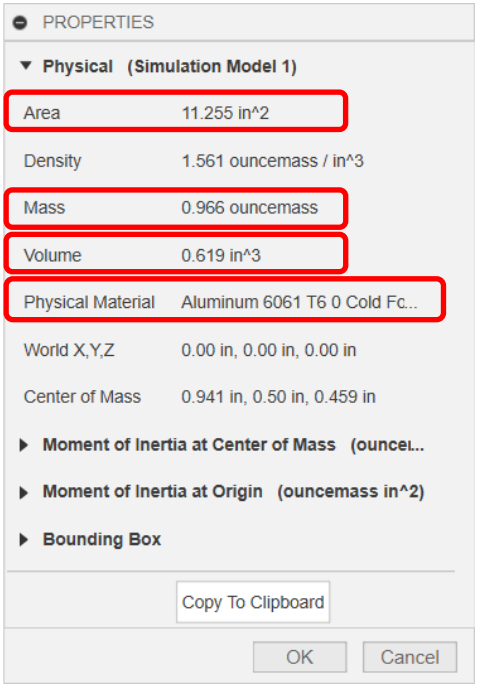
- when in the **SIMULATION** Workspace, right-click on **Simulation Model** and select **Properties**





- when in the **DESIGN** Workspace, right-click on the **Body** in the **Bodies** folder and select **Properties**



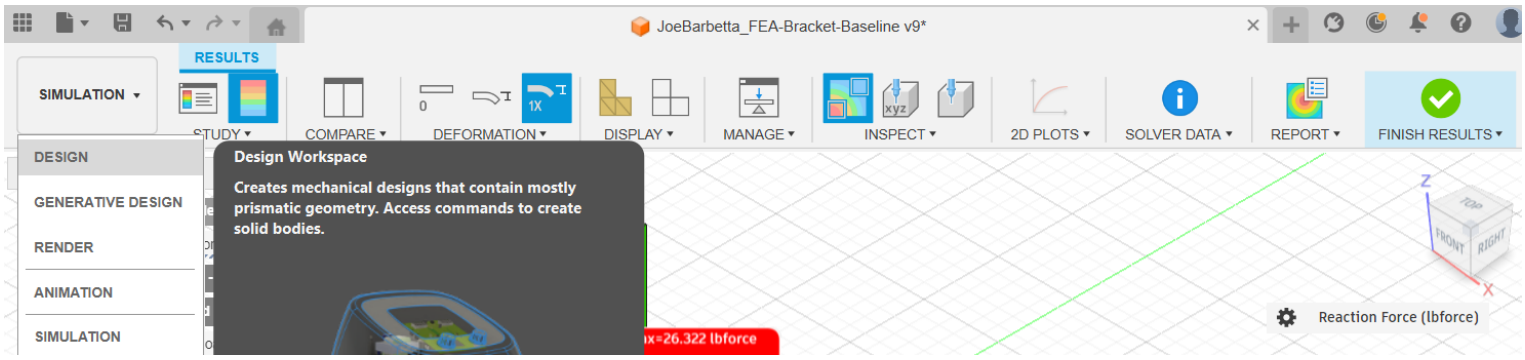


Modifying the design

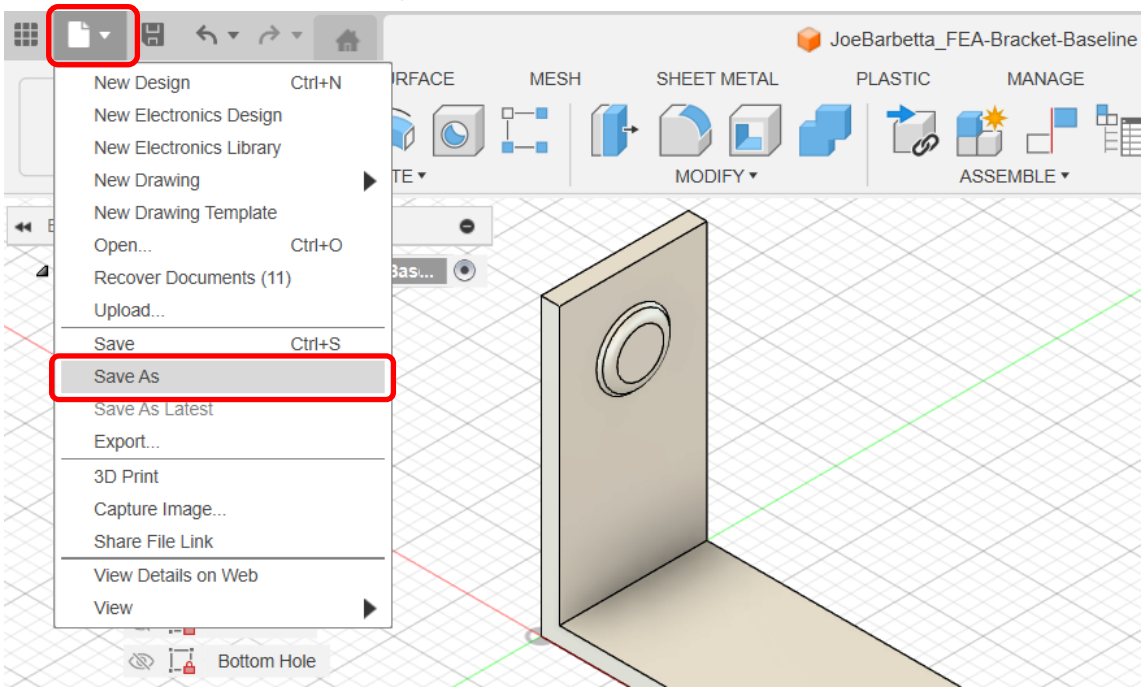
- **Save** your project. The name should end with **Baseline**, which was set at the start of this document.

Later on, when modifications are made, the name can be changed. This will allow multiple projects to be saved, allowing

- click on **FINISH RESULTS** and then switch back to the **DESIGN** Workspace to make modifications.



- start by using **Save As** and changing "**Baseline**" to something like "**Mod 1**" to ensure the modification is not written over the Baseline project. Do the same with each modification. This will allow one to switch back to a the original or any modification if one wished to repeat the simulation.



- make design modifications to strenthen your bracket. **The bracket mass cannot be increased by more than 25%.** The document section "Determining the mass of design body" instructs one on determining the mass.

These modifications **should Not affect the dimensions of the original bracket or the screw positions**. Note that the "bumps" added to the bracket surfaces simulate the screw positions.

These modifications should reduce the deflection (**Total Displacement**) and reduce the maximum stress (**3rd Pricipal**)

- return to the **SIMULATION** Workspace, generate a new mesh and run a new simulation

Assigned dimensions and load

Assigned dimensions and load

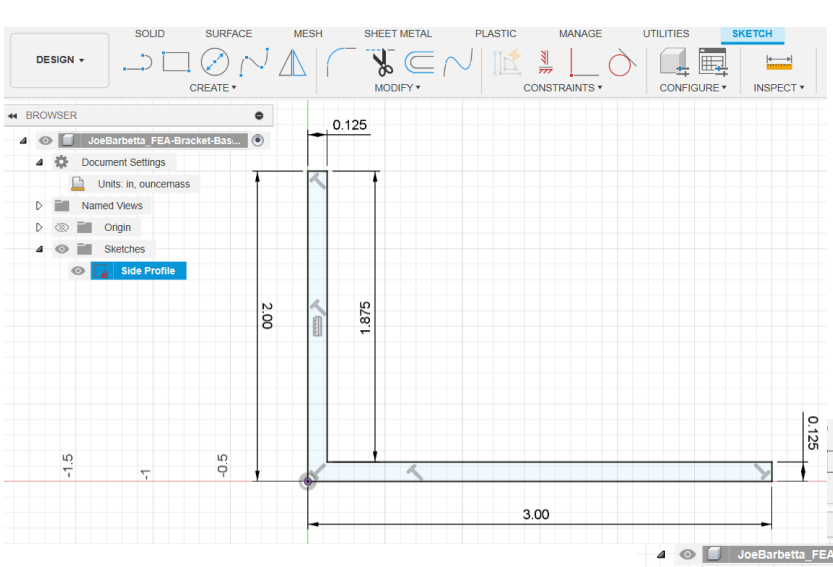
using the First letter of your First Name

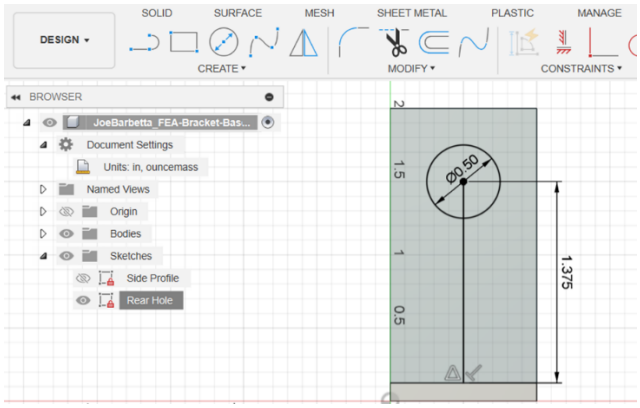
	Height	Length	Load(lbs)
Example	2.00	3.00	15
A,B,C,D	1.75	3.00	16
E,F,G,H	2.00	2.75	17
I,J,K,L	1.75	2.75	17
M,N,O,P	1.75	3.00	14
Q,R,S,T	2.25	3.00	14
U,V,W,X	2.25	2.75	18
Y,Z	2.25	2.75	18

All dimensions in inches
Thickness = 0.125
Width (amount extruded) = 1.000
Rear Hole Pos = Height - 0.625
Bottom Hole Pos = Length - 0.625

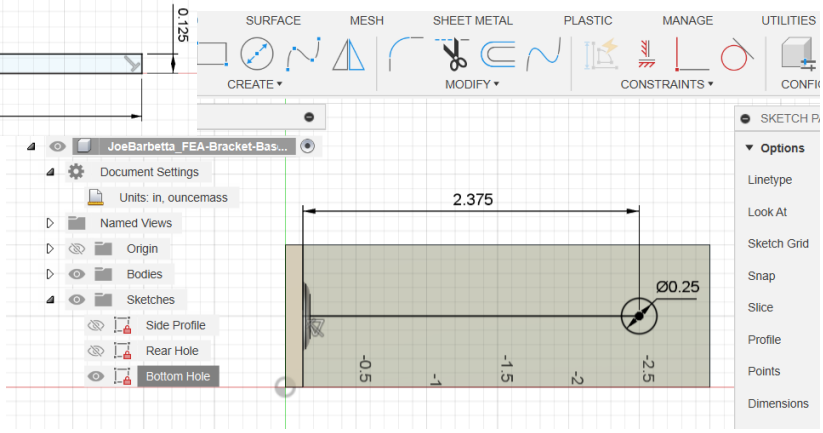
Submission requirements

- screenshots of the Side Profile, Rear Hole, and Bottom Hole, for your **baseline** as examples show below

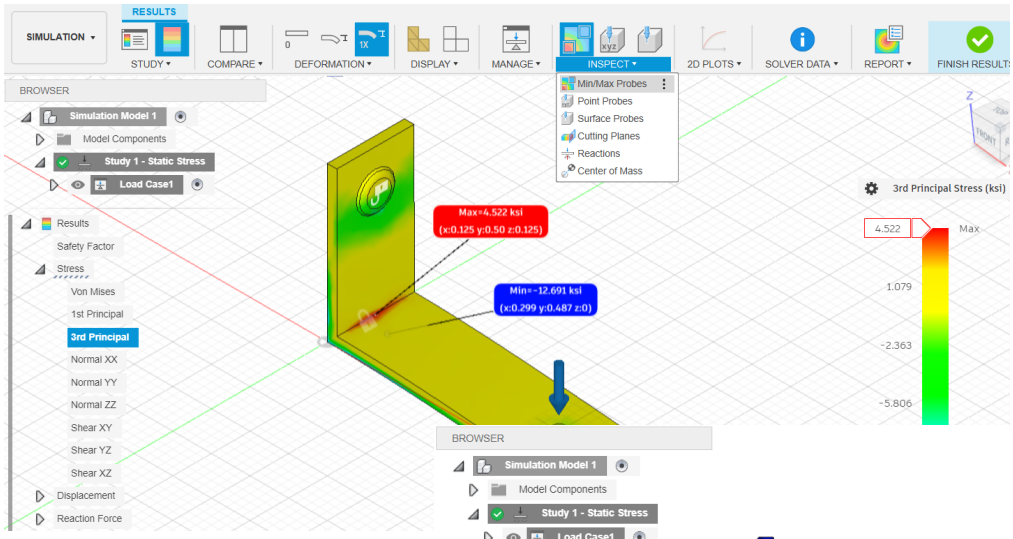




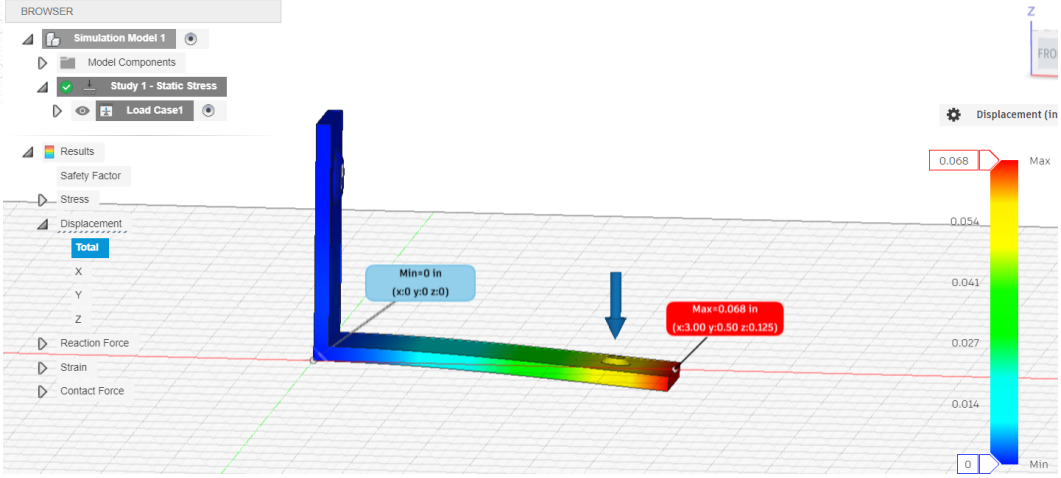
The sketches shown here are only examples. Some dimensions will be different for your assignment!



- **Stress** and **Displacement** screen shots of the **baseline** (original) design as shown below



This and the following screen shots are for the example bracket and will vary slightly from your design.



Continued on next page.

- **Stress** and **Displacement** screen shots of the the **first and second modified designs**.
- **comparison table**

Each value in the table **must include units**, except for Safety Factor, which is dimensionless.
Each Optimization value **shall be followed by a percentage change in parenthesis**. This is calculated as

Percentage change = (new - baseline) / baseline * 100

Use a **"+"** sign if the value is positive. An increase in strength will result in an increase in Safety Factor and a decrease in both Stess and Displacement. Also ensure that the statement "Percentages are those compared to Baseline." is included after the table.

	Baseline	**1st Optimization**	**2nd Optimization**
Safety Factor (Min)	4.418	5.234 (+18.5%)	6.654 (+50.6%)
Stress (Max)	4.522 ksi	3.257 ksi (-28.0%)	2.163 ksi (-52.2%)
Displacement (Max)	0.068 in	0.042 in (-38.2%)	0.032 in (-52.9%)

Percentages are those compared to Baseline.