Mr. Barbetta's *"I don't care about 3D Printing. I just want the art credit."* class

# Where is that socket?

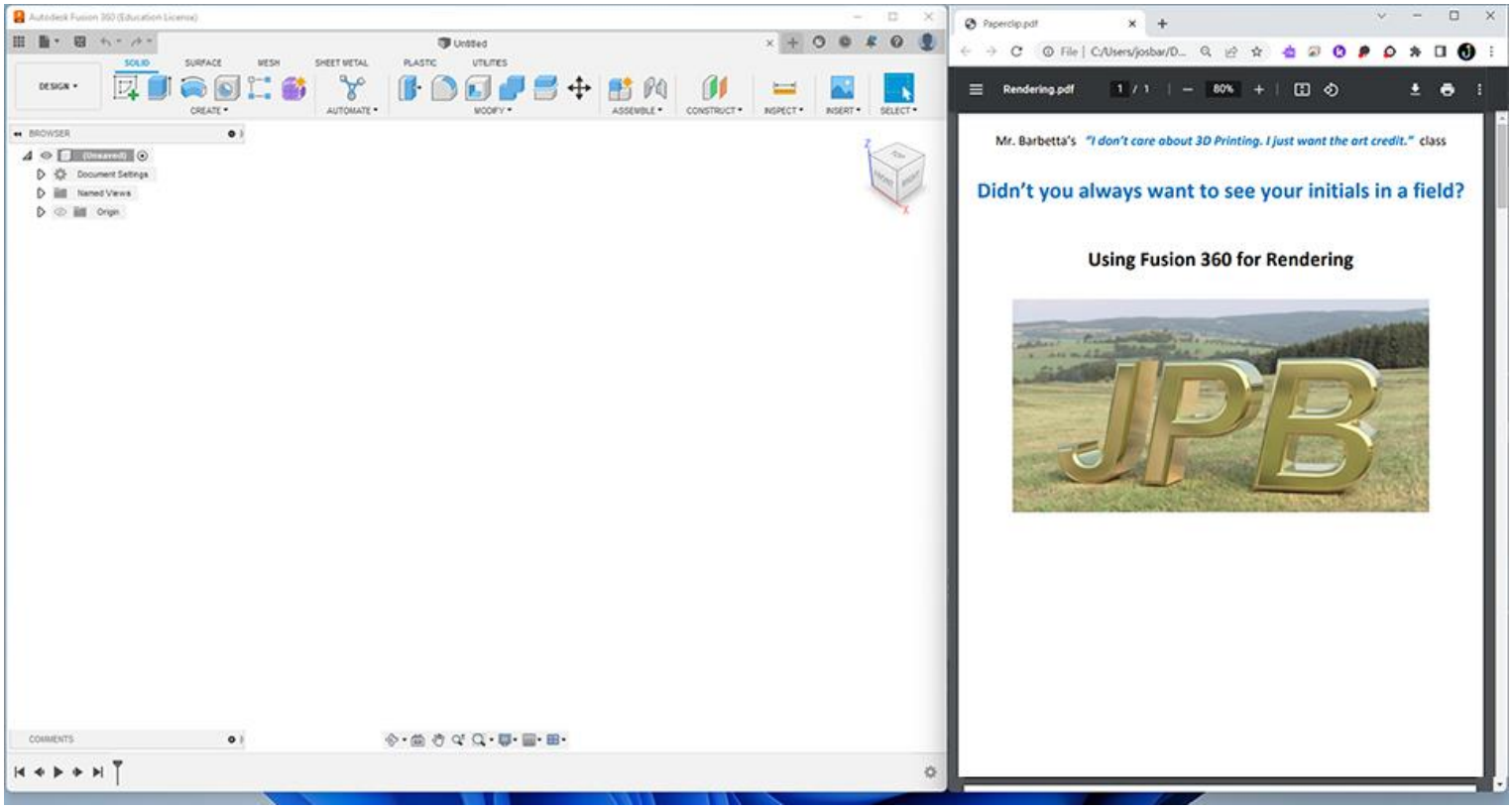

to this

**Today's lesson is sponsored by Harbor Freight.**

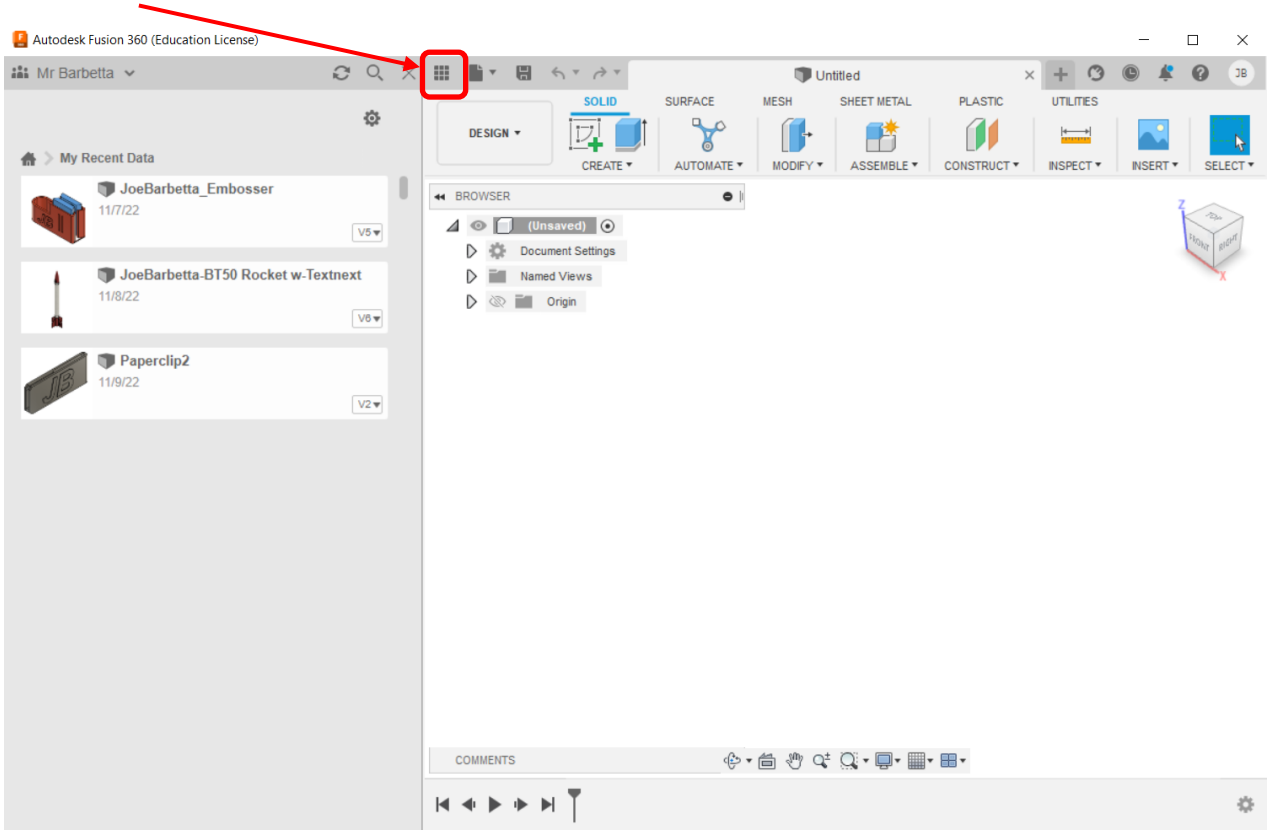
# Contents

# Using This Document

The best way to follow this document is to **reduce the width of the Fusion 360 window** and have this pdf document open in Chrome browser as shown below. This document can be **downloaded from Schoology and then dragged into Chrome** and scaled down to 80%.



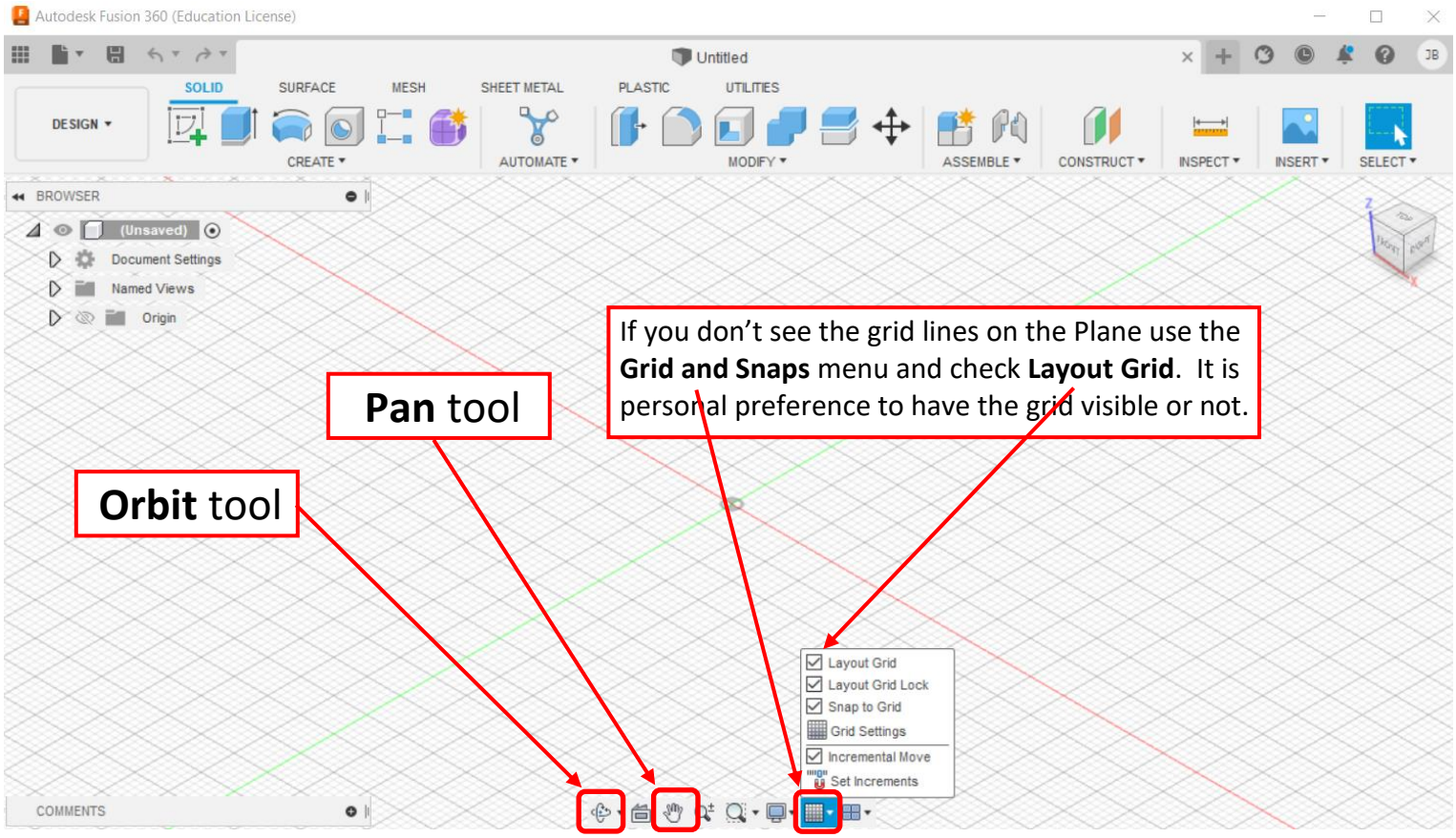

The Fusion 360 window will not allow its width to be reduced much so for smaller computer screens a trick is to click on the **Data Panel icon** and then move the window to the left with the Data Panel off the screen.



# Changing the View of a Design

- if you don't see a grid in the Fusion 360 window, as shown below, click on **Grid and Snaps** and check **Layout Grid**. Displaying the *Layout Grid* is a matter of preference. When designing for 3D printing, it can be used to represent the *build plate*.

- click on the **Orbit** tool and click somewhere on the **Grid** to practice rotating and changing the angle of the view.
- click on the **Pan** tool and then on the **Grid** to practice moving the view laterally.
- after using the *Orbit* or *Pan* tool one must press the **Esc key** to exit that mode.
- use the **Mouse Wheel** to practice Zooming in and out.

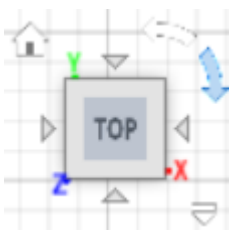



Here is a close-up of the View Cube at the top right of the window.

- click on the **View Cube** and move the cube while holding the mouse button down. This is another way to rotate the view.
- click on the Top of the View Cube and note how the view just jumped to a Top View.

The View Cube now resembles that on the right.

- click on the **Curved Arrows** at the upper right of the View Cube and practice Rotating the View.
- click on the **Arrows** at the sides of the View Cube to practice jumping to various Views.
- click on the **Home** icon to the upper left of the View Cube. This can always be used to reset the view to the Home View

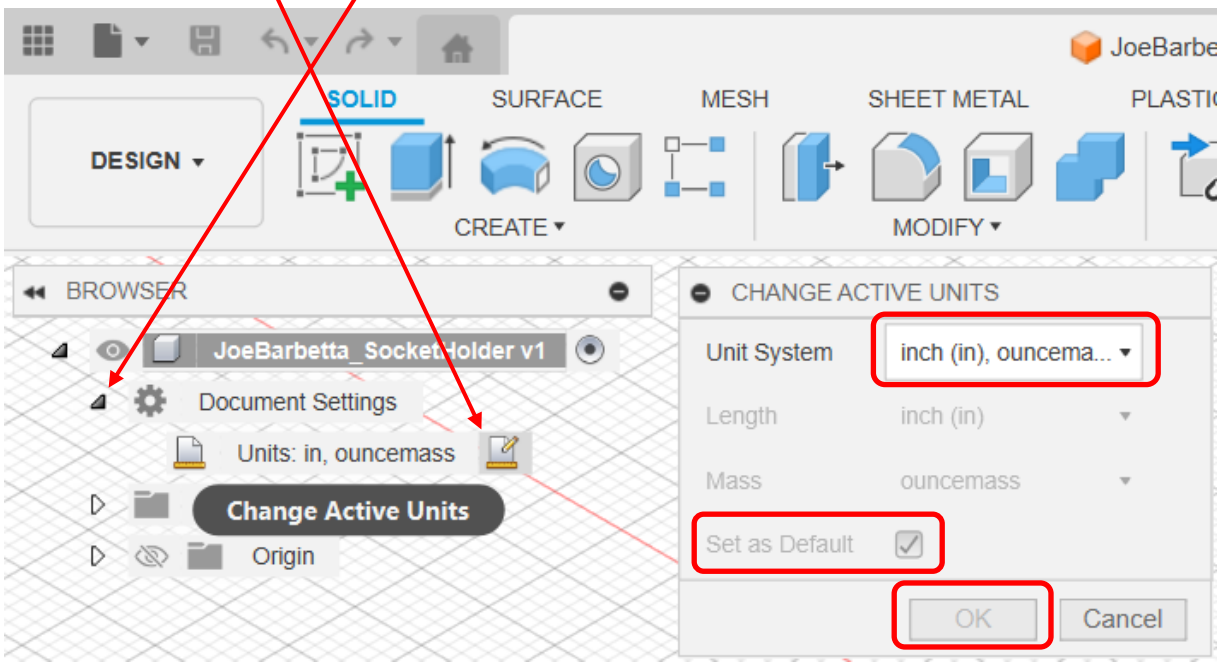
# Starting a Design in Fusion (START HERE)

- 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 **_SocketHolder** e.g. **JoeBarbetta_ SocketHolder** (note the use of the underscore)

Note that by default Fusion saves your project to "the cloud", which are the servers managed by AutoDesk. When you log into Fusion on a different computer, your projects will be available.

As you work you may want to occasionally save your work in case Fusion crashes or we lose power.

- in the left **"BROWSER"** click the **arrow next to Document Settings**
- click on the **edit icon** that appears to the right 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.



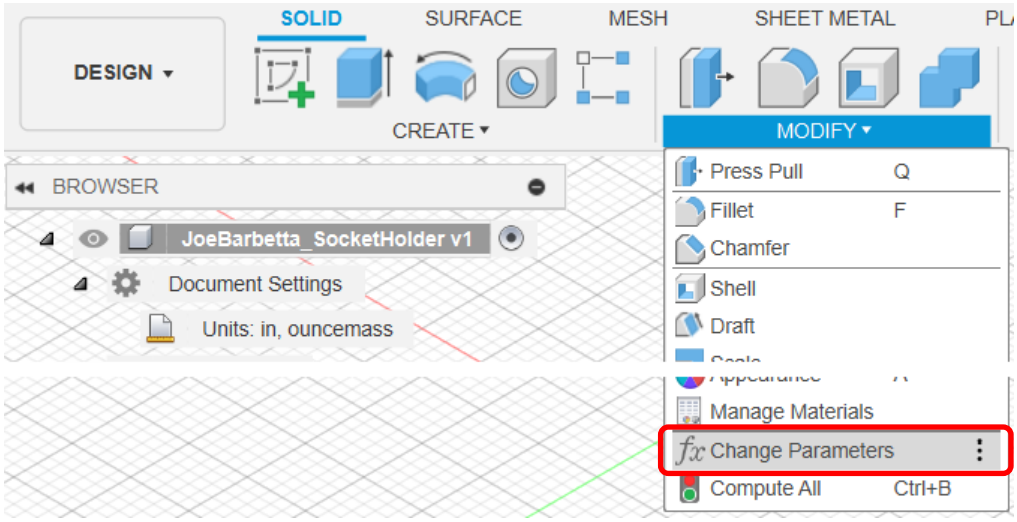

Note that the default units are in mm, which we just changed to inches.

Did you know that the default units have changed over the years? The earliest version used cubits as the default unit.

# Parametric Modelling

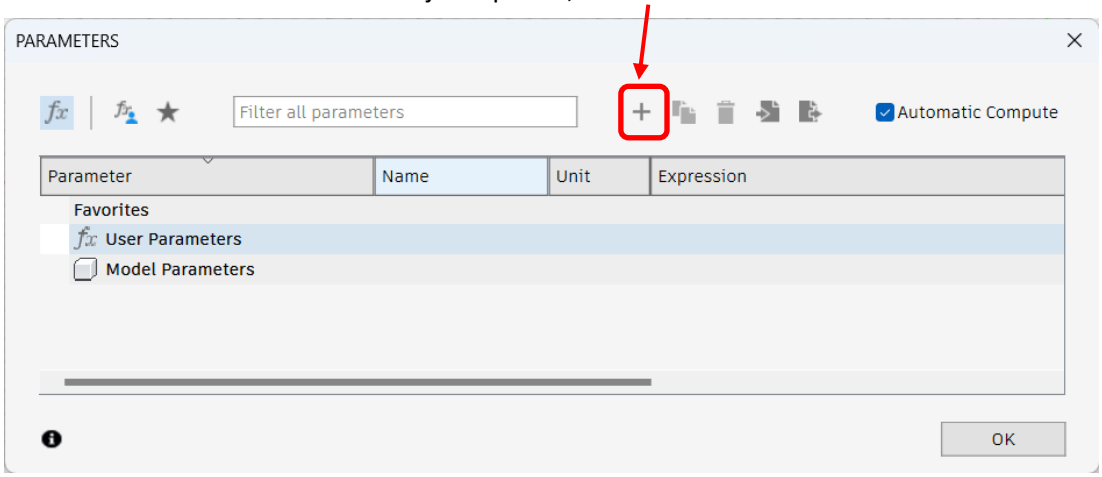
"Parametric Modelling" is a powerful feature, wherein variables can be created, which can then be used for dimensions and text. One can later change values using this window to make changes or adjustments to a design. Parameters can be added one by one, but you have a nice teacher who is willing to show you how to import a whole bunch of parameters to get started.

- from the **MODIFY** menu select **Change Parameters**
- If a window about **Parameteric Text** pops up click its **OK** button.

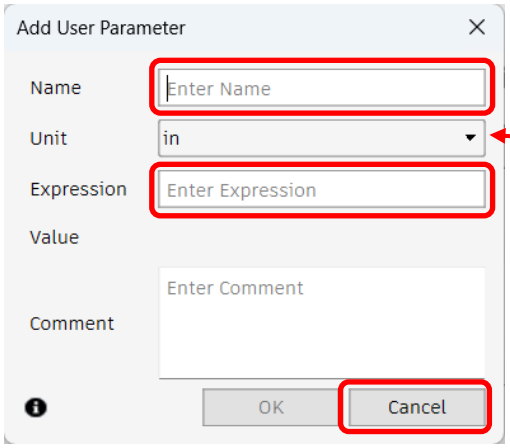



**It's all the way at the bottom.**

- in the **PARAMETERS** window that just opened, click **+**.




- In the pop-up window one can set the **Name**, **Unit**, and **Expression**. Click **Cancel** because we will use another method.




There is also a **Unit** selection. For most parameters, one will likely use **in** (inches), however, this assignment also uses **Text** and **Dimensionless** for some parameters.

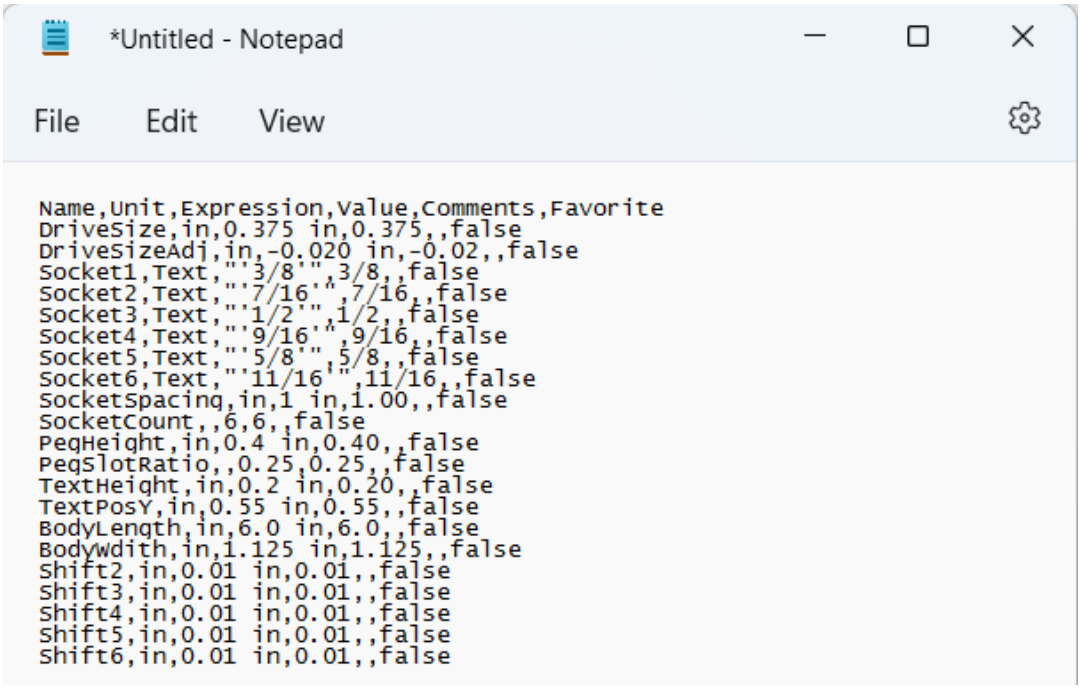
- In the lower **Windows Task Bar** click on the Search icon and enter **Notepad** in the search text box.

Note that if **Notepad** cannot be opened or if the following data cannot be copied, the values can be entered individually using the method previously shown. These steps are for using the **Import feature**. Note that Macs have a Notepad alternative.

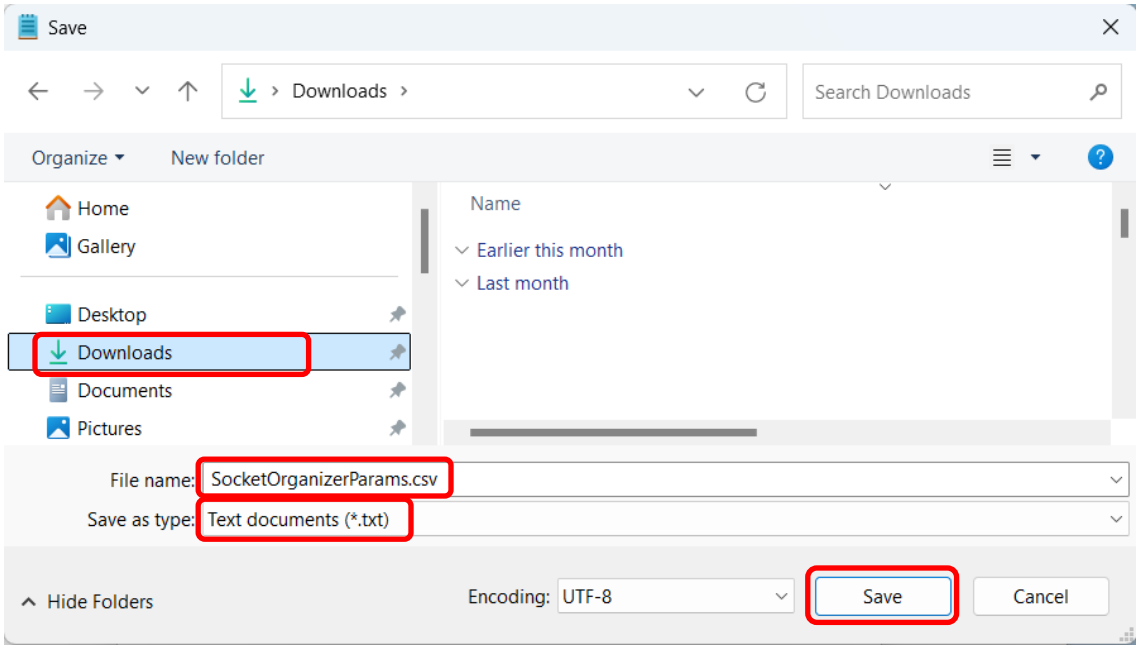
- select all of the text in the below box and right-click and select **Copy**.

```
Name,Unit,Expression,Value,Comments,Favorite
DriveSize,in,0.375 in,0.375,,false
DriveSizeAdj,in,-0.020 in,-0.02,,false
Socket1,Text,"'3/8'",3/8,,false
Socket2,Text,"'7/16'",7/16,,false
Socket3,Text,"'1/2'",1/2,,false
Socket4,Text,"'9/16'",9/16,,false
Socket5,Text,"'5/8'",5/8,,false
Socket6,Text,"'11/16'",11/16,,false
SocketSpacing,in,1 in,1.00,,false
SocketCount,,6,6,,false
PegHeight,in,0.4 in,0.40,,false
PegSlotRatio,,0.25,0.25,,false
TextHeight,in,0.2 in,0.20,,false
TextPosY,in,0.55 in,0.55,,false
BodyLength,in,6.0 in,6.0,,false
BodyWdith,in,1.125 in,1.125,,false
Shift2,in,0.01 in,0.01,,false
Shift3,in,0.01 in,0.01,,false
Shift4,in,0.01 in,0.01,,false
Shift5,in,0.01 in,0.01,,false
Shift6,in,0.01 in,0.01,,false
```

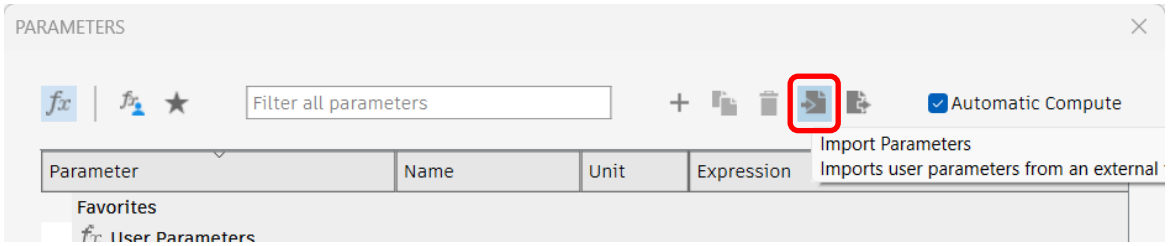
- In Notepad right-click and select **Paste**. Alternately, one can press **ctrl** and the **v** keys.



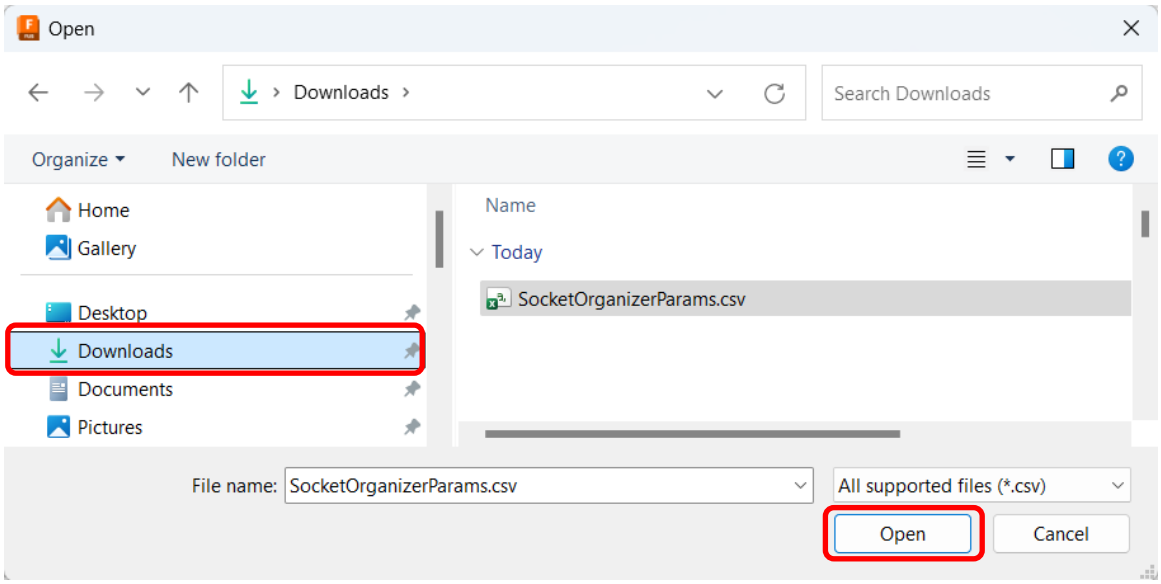
- from the **File** menu select **Save**
- select the **Downloads** folder
- change **Save as type** to **All files (*.*)**
- change the **File Name** to **SocketOrganizerParams.csv**.  csv stands for comma separated values
- click **Save**



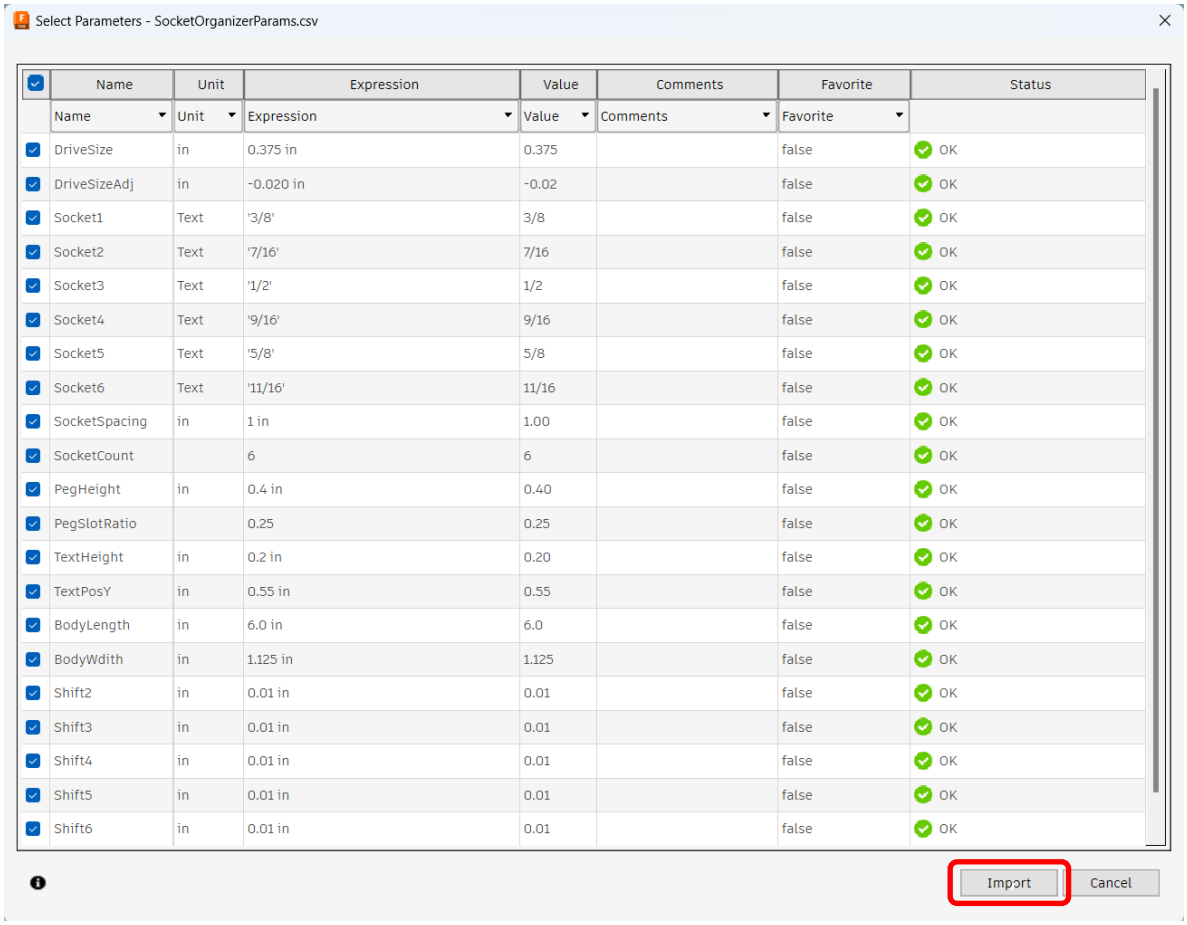

- in the PARAMETERS window click on the **Import Parameters** icon
- at the bottom of the next IMPORT window, click **Select from my computer…**



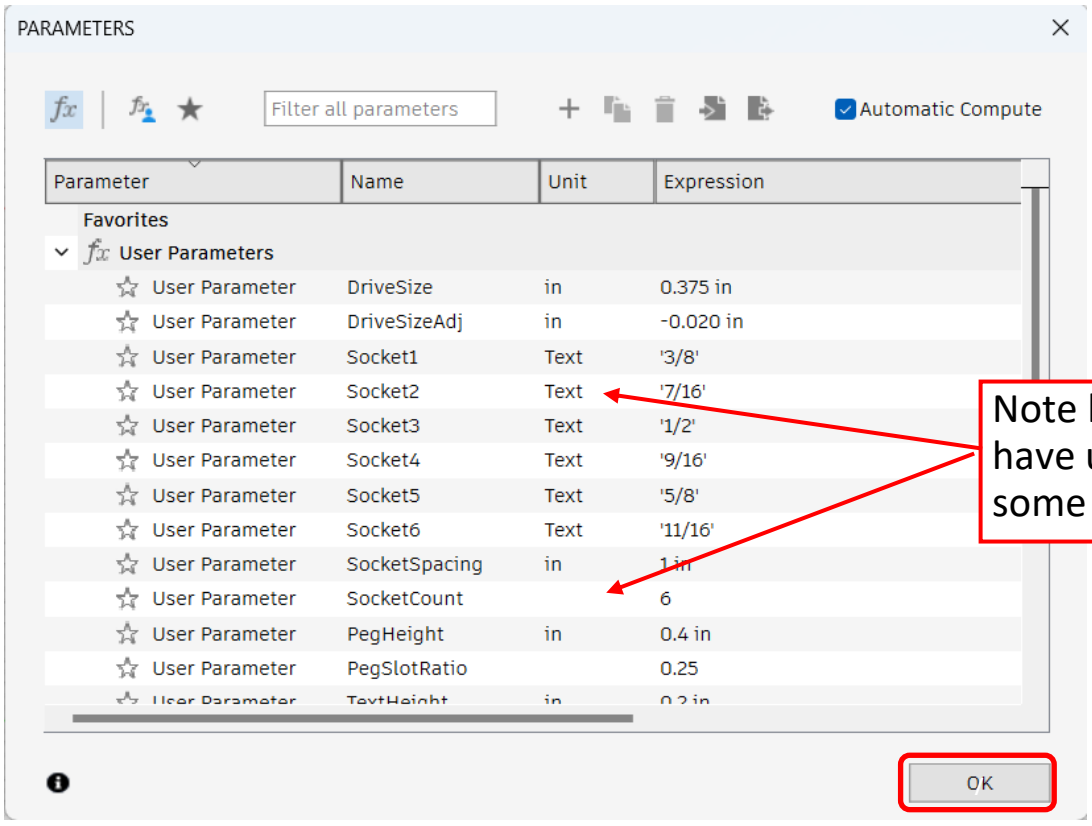

- select the **Downloads** folder and the file from Notepad.  Click **Open**.



A screen will show, which lists all the parameters from the opened file. If Fusion detects any errors, they will be shown.
- click **Import**




All the parameters should now show on the window. (not all are shown below to allow the window to fit) Click **OK**.

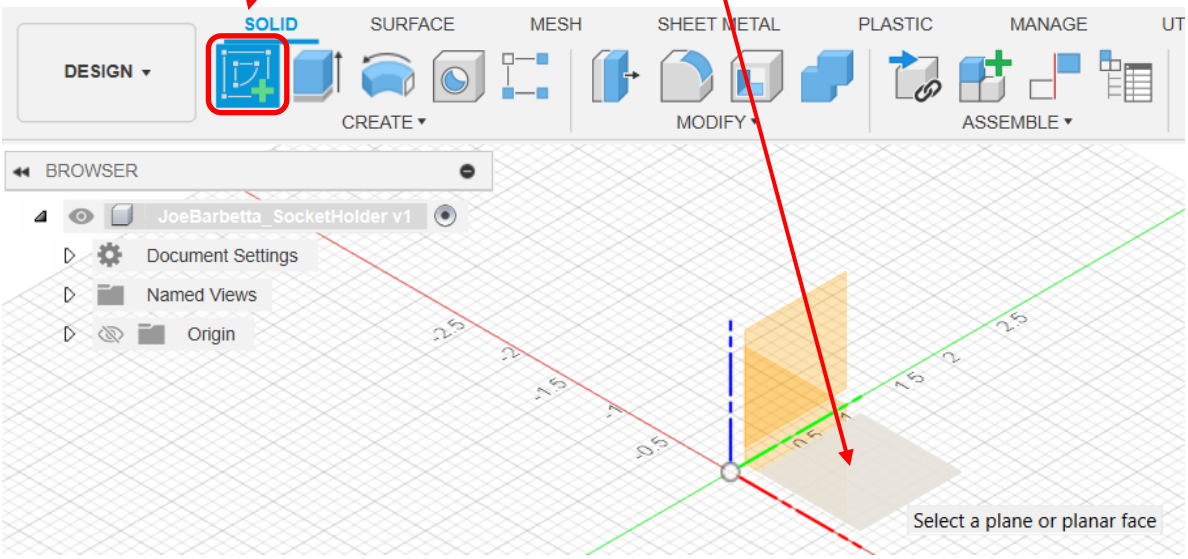



Note how some parameters have units set as **Text** and some **don't show a unit**.

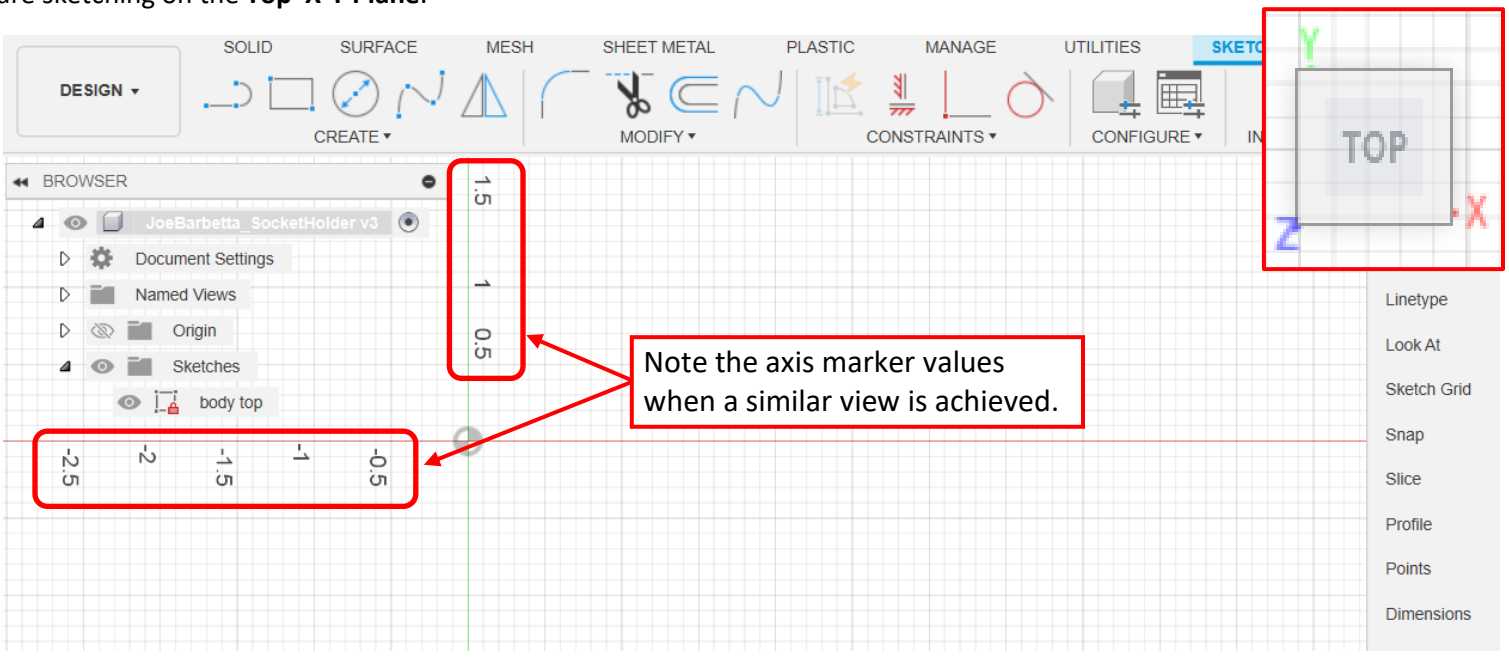
# Creating the First Sketch

Note that a Fusion expert may tell you to create a Component first. Just say **"Dude. I'm just making a hole gauge."**

- select the top **Create Sketch** tool and click on the **bottom rhombus** to select the X-Y Plane.
If a tool can't be found, one can always look in the **CREATE** and **MODIFY** menus for it.

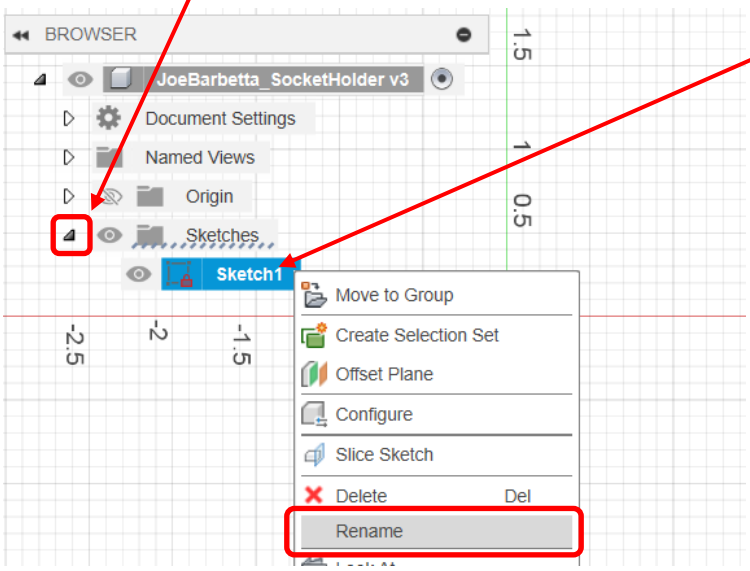



- zoom in as shown below. The scale labels can give an idea of how far one is zoomed in. The **View Cube** should indicate you are sketching on the **Top X-Y Plane**.

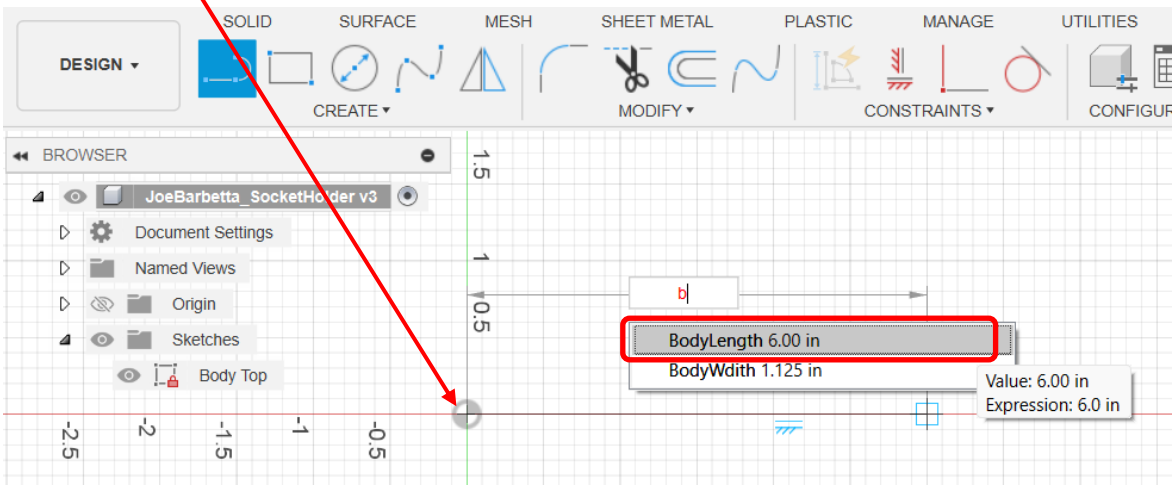


Whenever a new Sketch is created, it should be named.

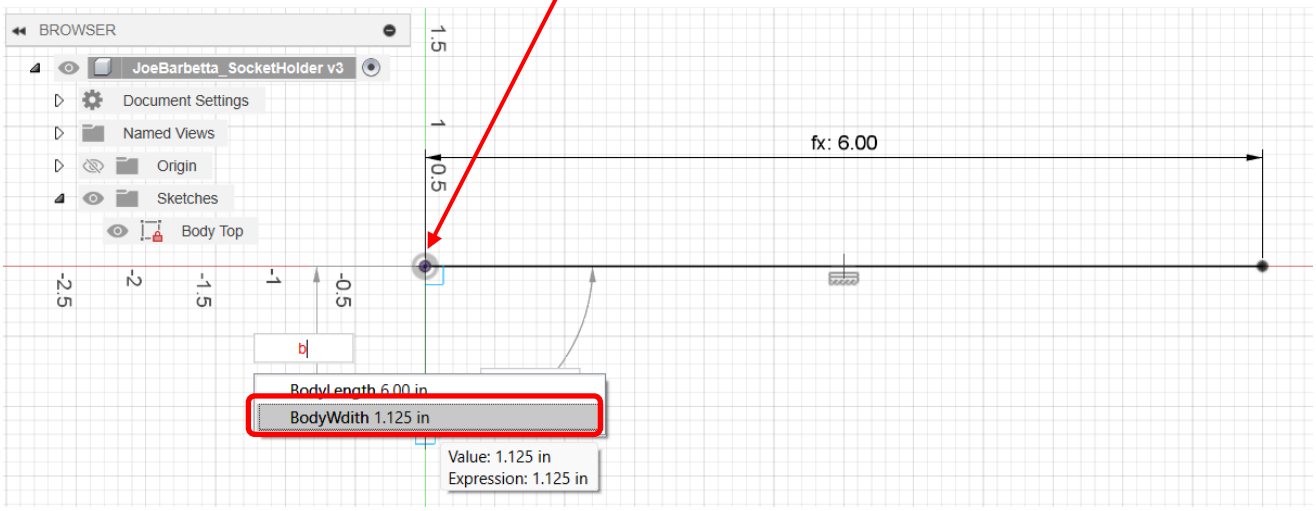
- click on the **arrow** to open the Sketches folder, **right-click** on the default name **Sketch1** and select **Rename**. Change the name to **Body**.



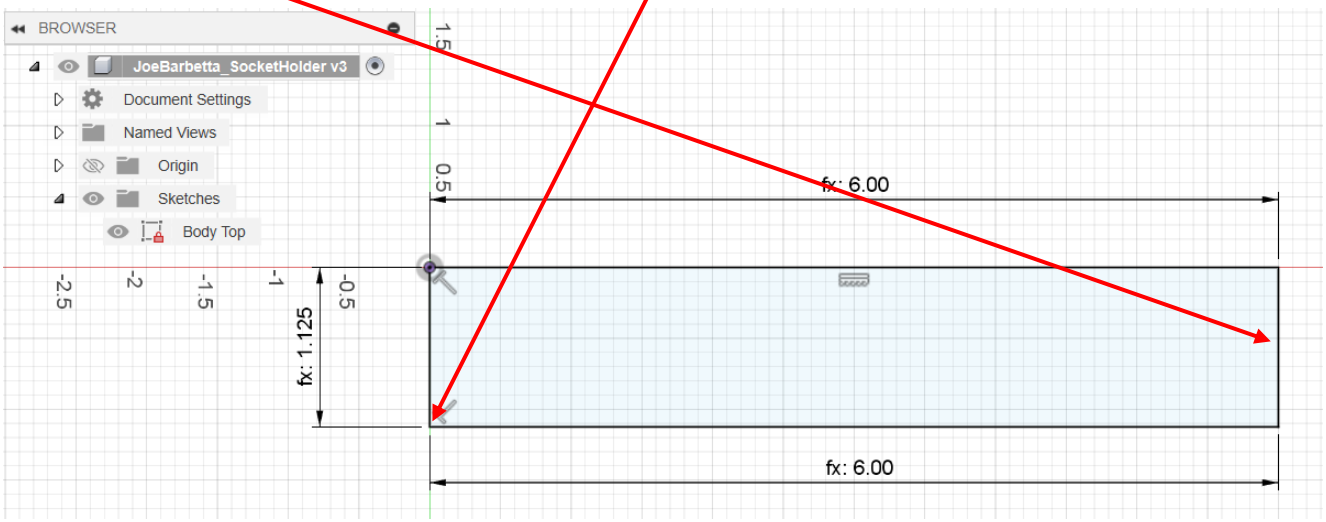

- select the **Line** tool
- click on the **Origin**, extend the line **to the right**, type **b**, select **BodyLength**, and press the **Enter key**.



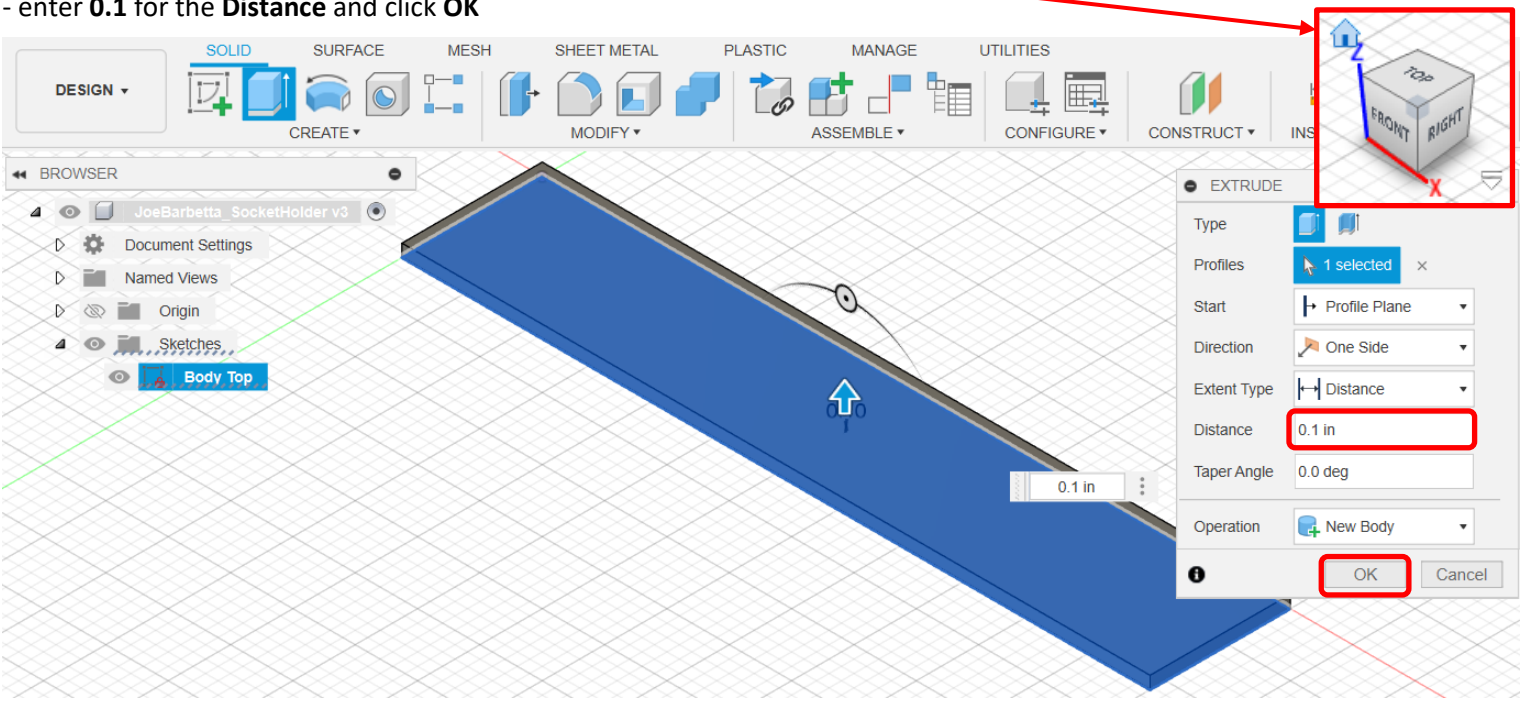

- select the **Line** tool again and click on the **Origin** again, extend the line down and use **BodyWidth**.



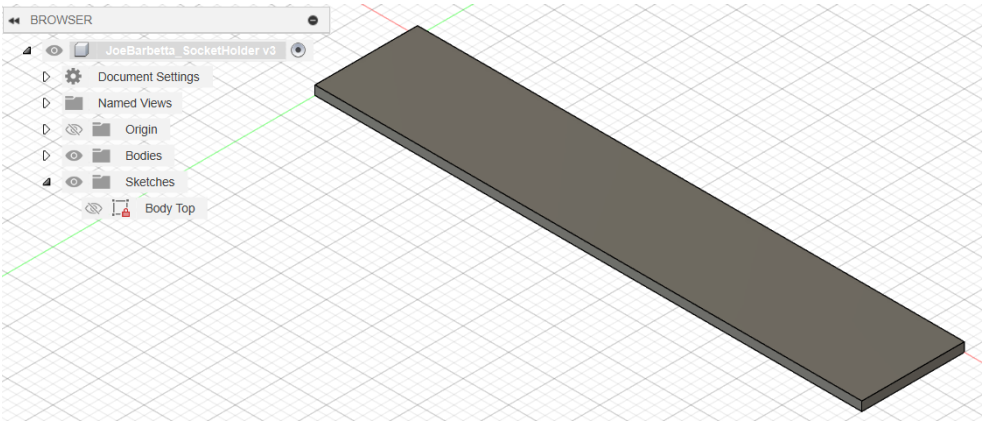
- select the **Line** tool again and start a line from the **last point** and extend the line to the right and use **BodyLength** again.
- then draw **another line** up to close the profile.



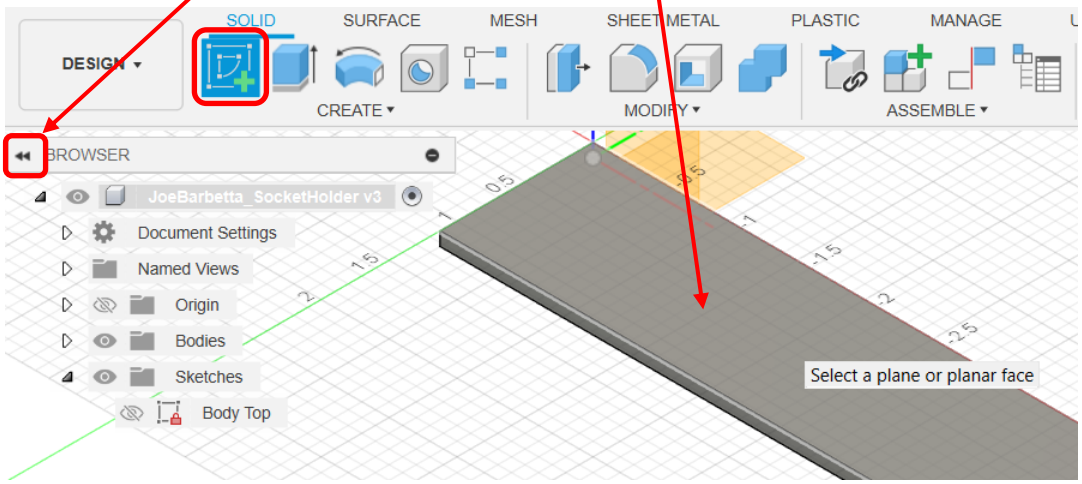

- click on the **Home** icon at the **View Cube**
- select the **Extrude** tool. If the Extrude tool is not visible, find it in the CREATE menu.
- enter **0.1** for the **Distance** and click **OK**



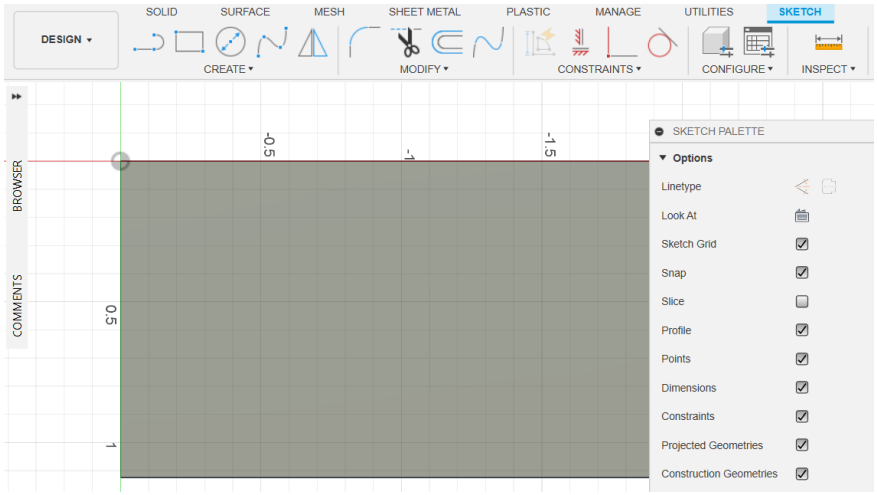

The body should look like that below.



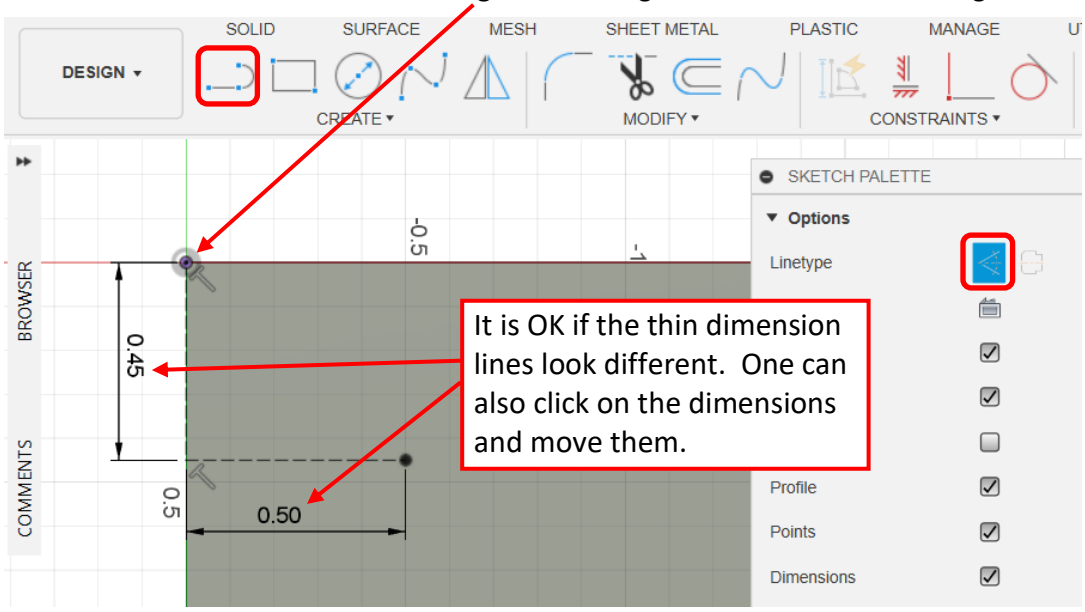
- select the **Create Sketch** tool and click on the **top surface** of the body
- rename the new Sketch as **Top Features**
- click on the **arrows** next to BROWSER to close it to make more room. It can be reopened when needed.



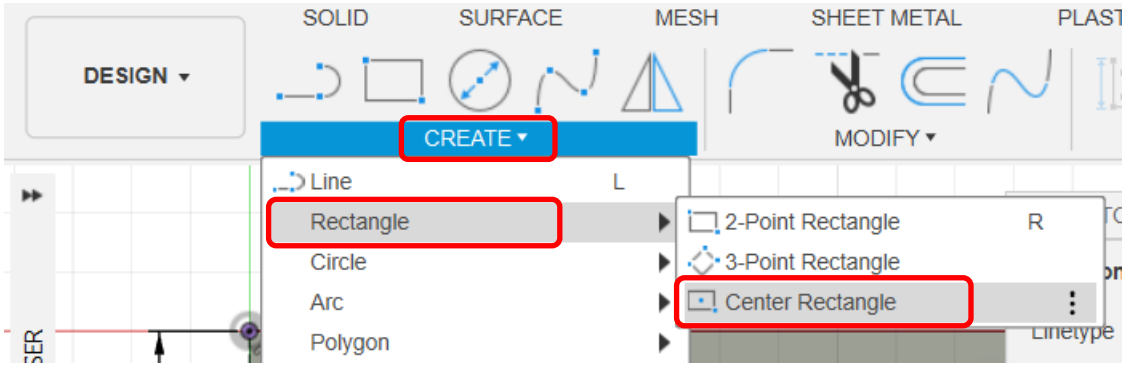

- Zoom in and pan (holding the mouse wheel down) to achieve a view similar to that below.



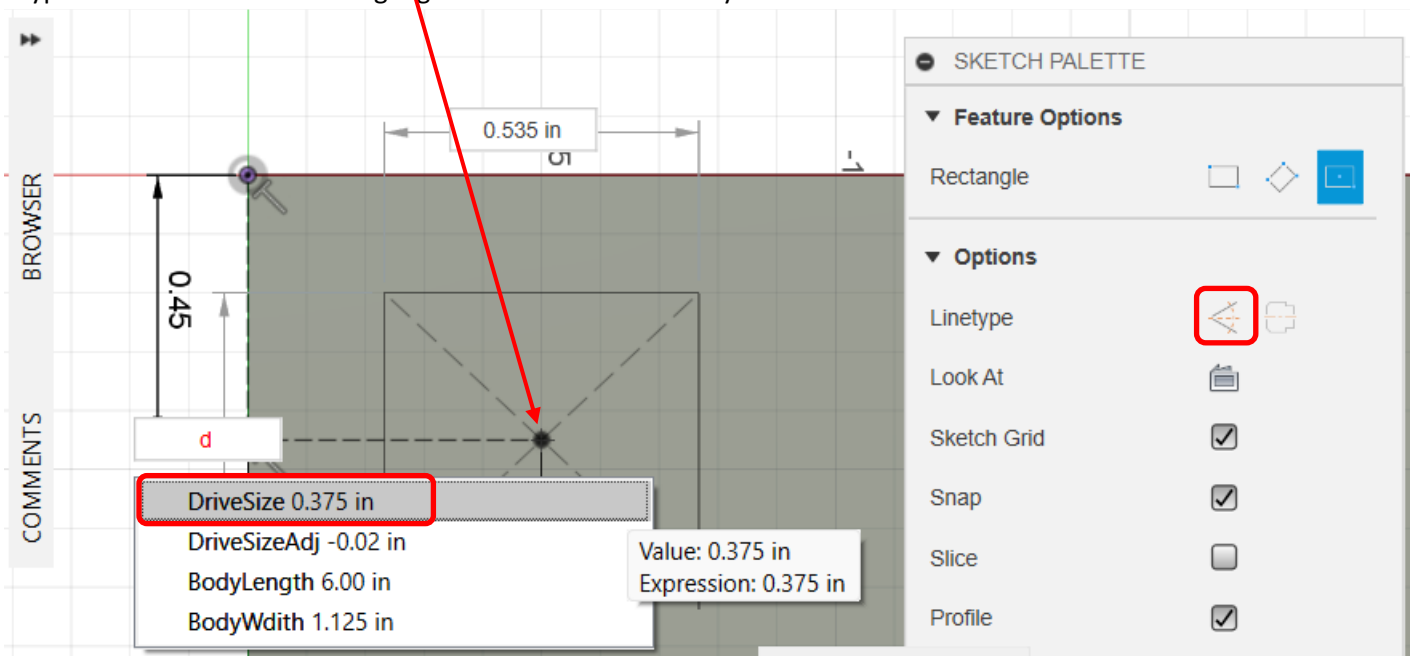

- click on the **Construction** icon for Linetype to highlight it. This will cause drawn line to be dashed.
- create a line downward from the **Origin** with a length of **0.45** and one to the right with length of **0.5**



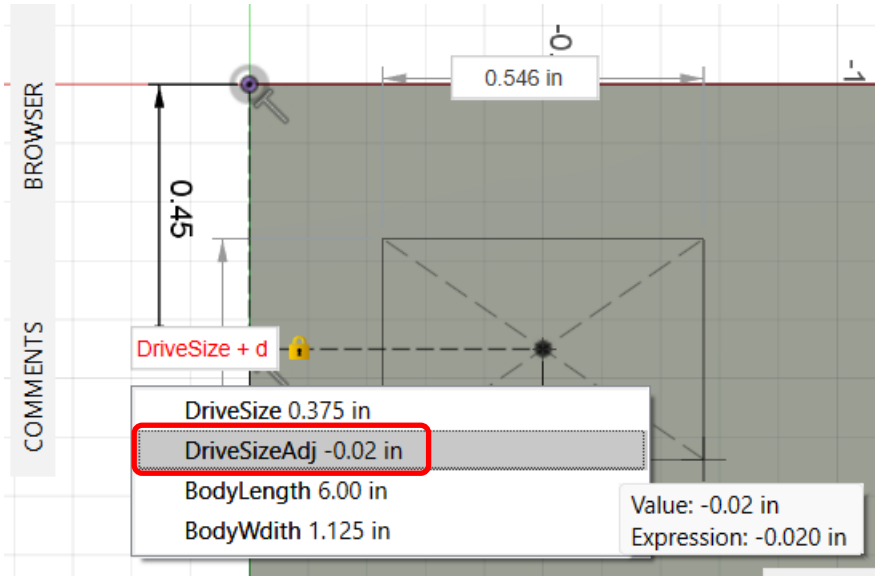
- from the **CREATE** menu select Rectangle and **Center Rectangle**




- click on the **Construction** icon to **turn off the highlighting** to draw normal lines
- click on the **endpoint of the 0.500 line** and expand the rectangle out. The next few steps are tricky. Try not to say bad words.
- type **d**. **DriveSize** should be highlighted. Press the Enter key.

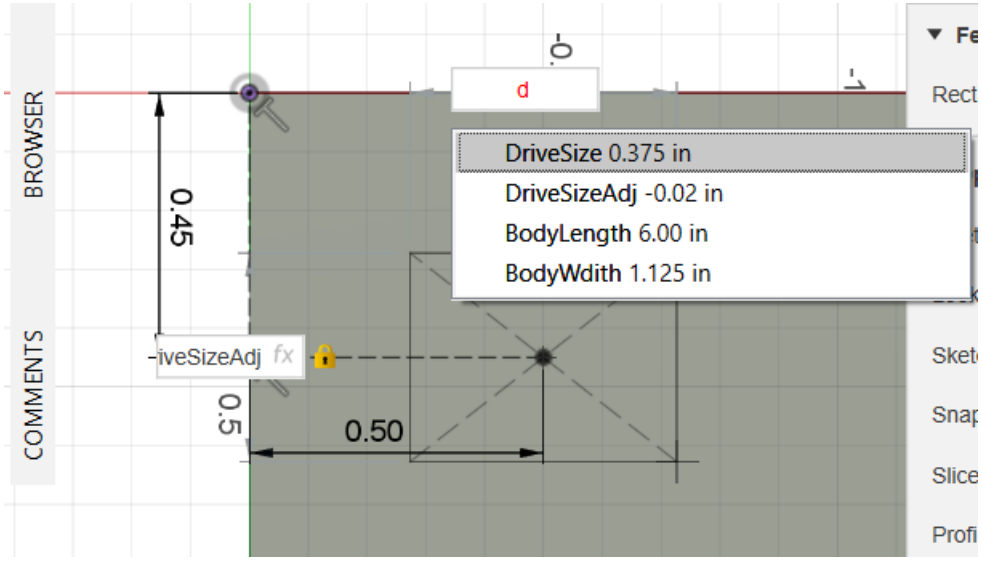



- then type **+** (plus sign) and **d** and use the down arrow to select **DriveSizeAdj** and press the **Enter Key**.

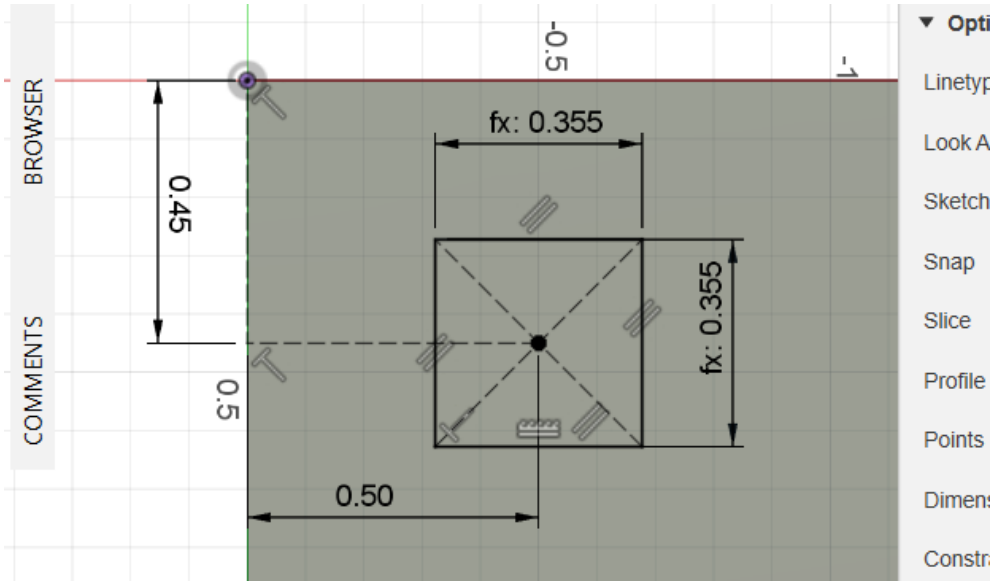



Note that a dimension can be a single parameter or an equation using one or more parameters.
Here we used **DriveSize + DriveSizeAdj**. This is done because a common DriveSize can be set, i.e. 1/4, 3/8, or 1/2 and an adjustment value can be applied after preforming a test print.

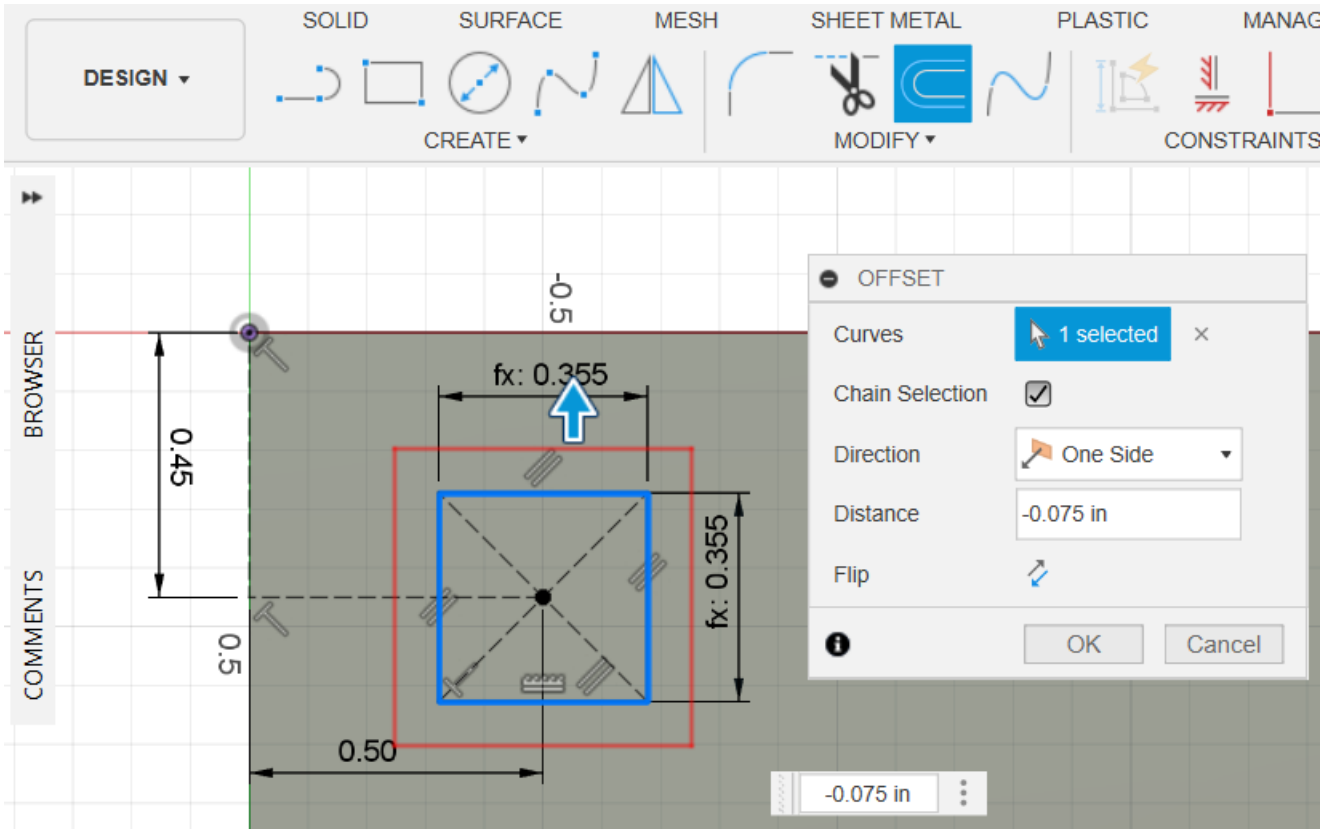
- press the **Tab key**, which should then select the other dimension of the rectangle.
- Type **d**, press the **Enter key**, type **+**, arrow down to **DriveSizeAdj**, and press the **Enter key**
- press the **Enter key** again to complete the rectangle.



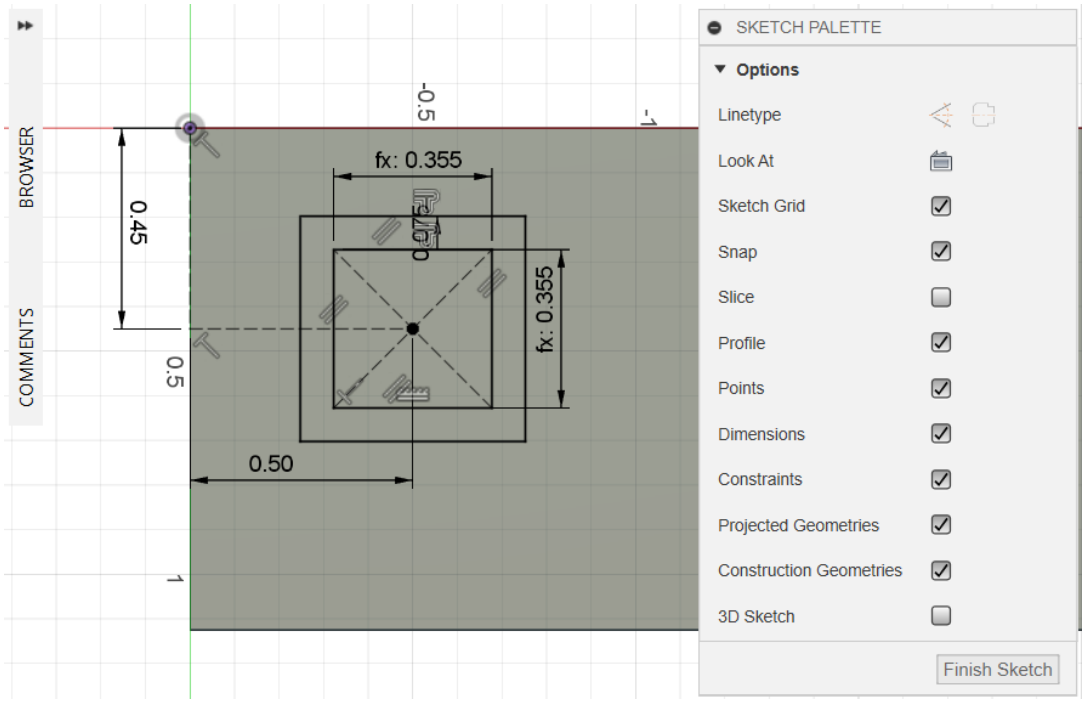

- move the dimension lines to achieve a view similar to that below



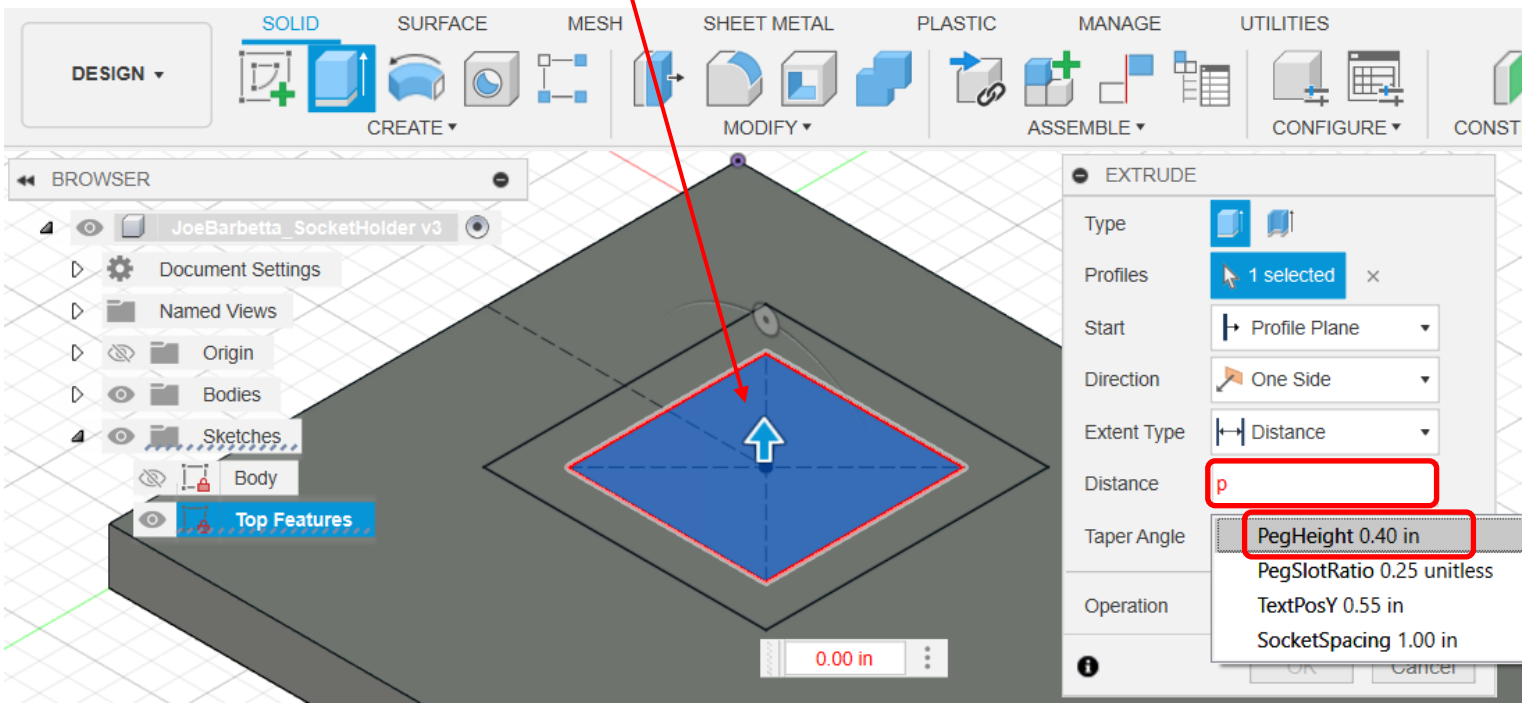
- select the **Offset** tool.  If it is not visible, select it from the **MODIFY** menu.
- click on an edge of the rectangle
- enter **-0.075** (note the minus sign).  If the red rectangle appears inside the blue rectangle, click the **Flip** icon.
- click **OK**.



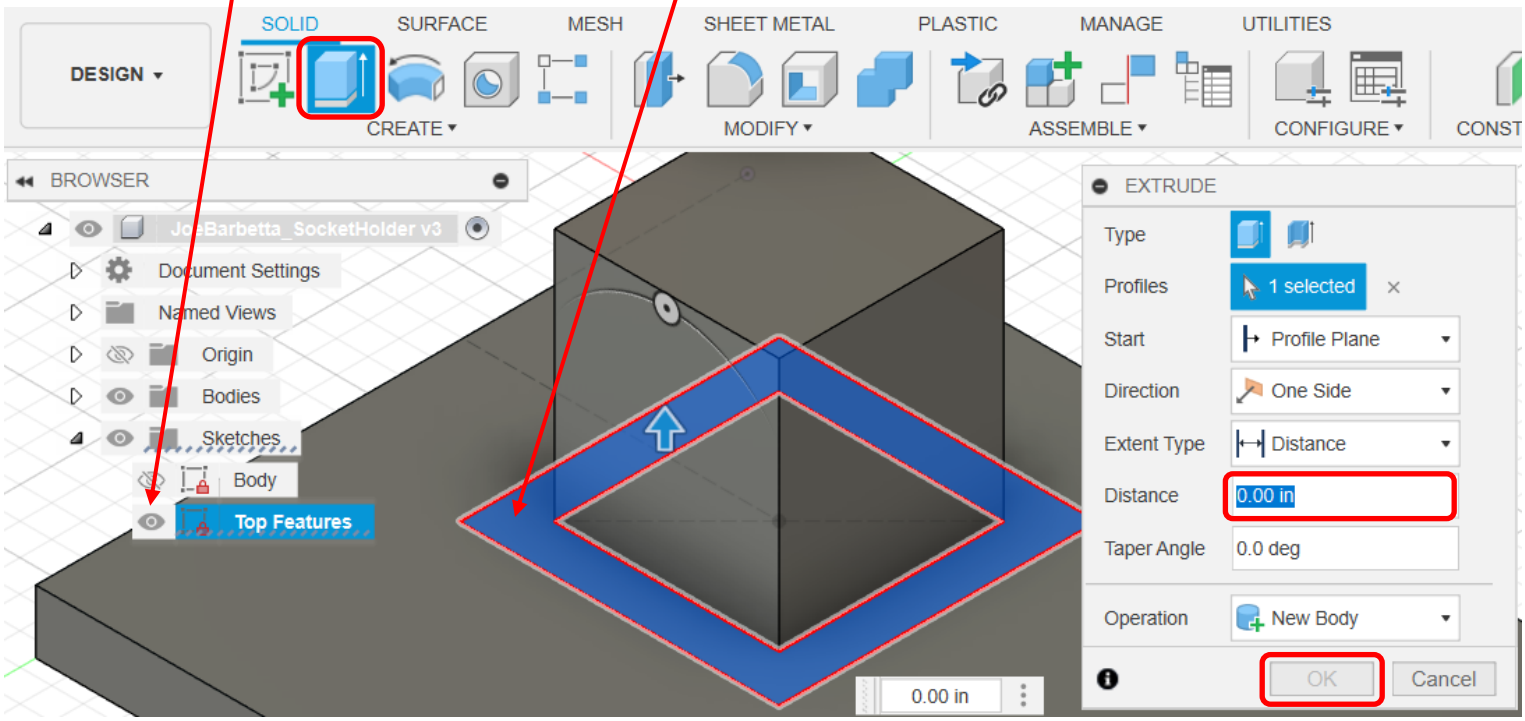

- click Finish Sketch



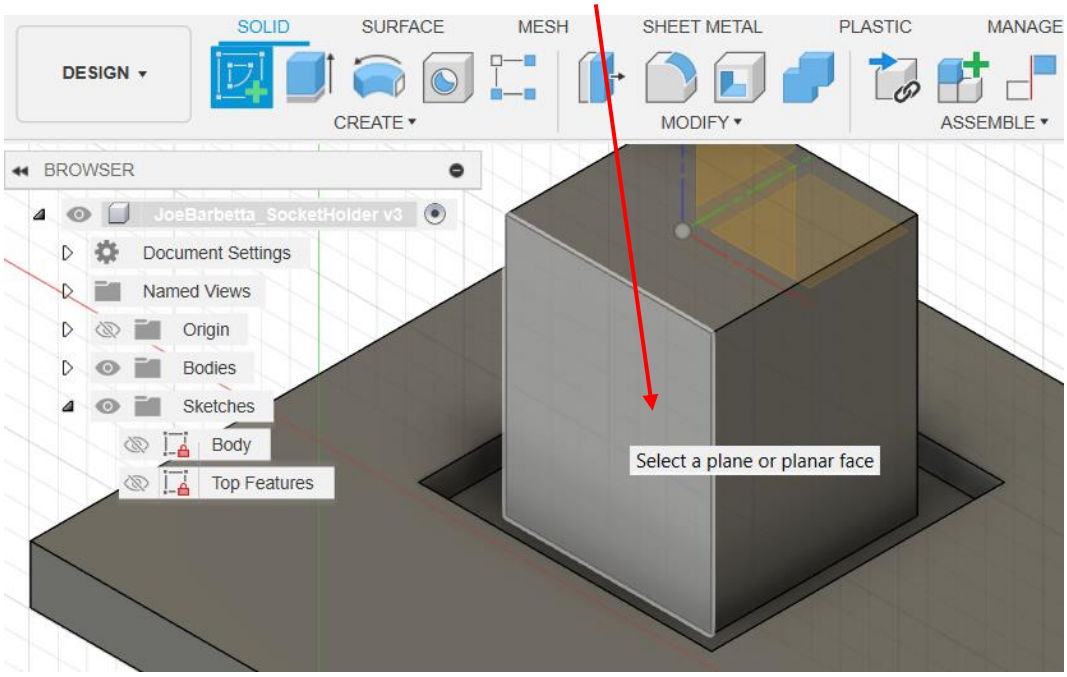
- click on the **Home** icon at the **View Cube** and Zoom in to achieve a view similar to that below
- click on the **BROWSER arrows** to reopen it
- select the **Extrude** tool and click on the **inner rectangle**
- in the **Distance** box type **p** and select **PegHeight**
- click **OK** in the EXTRUDE window



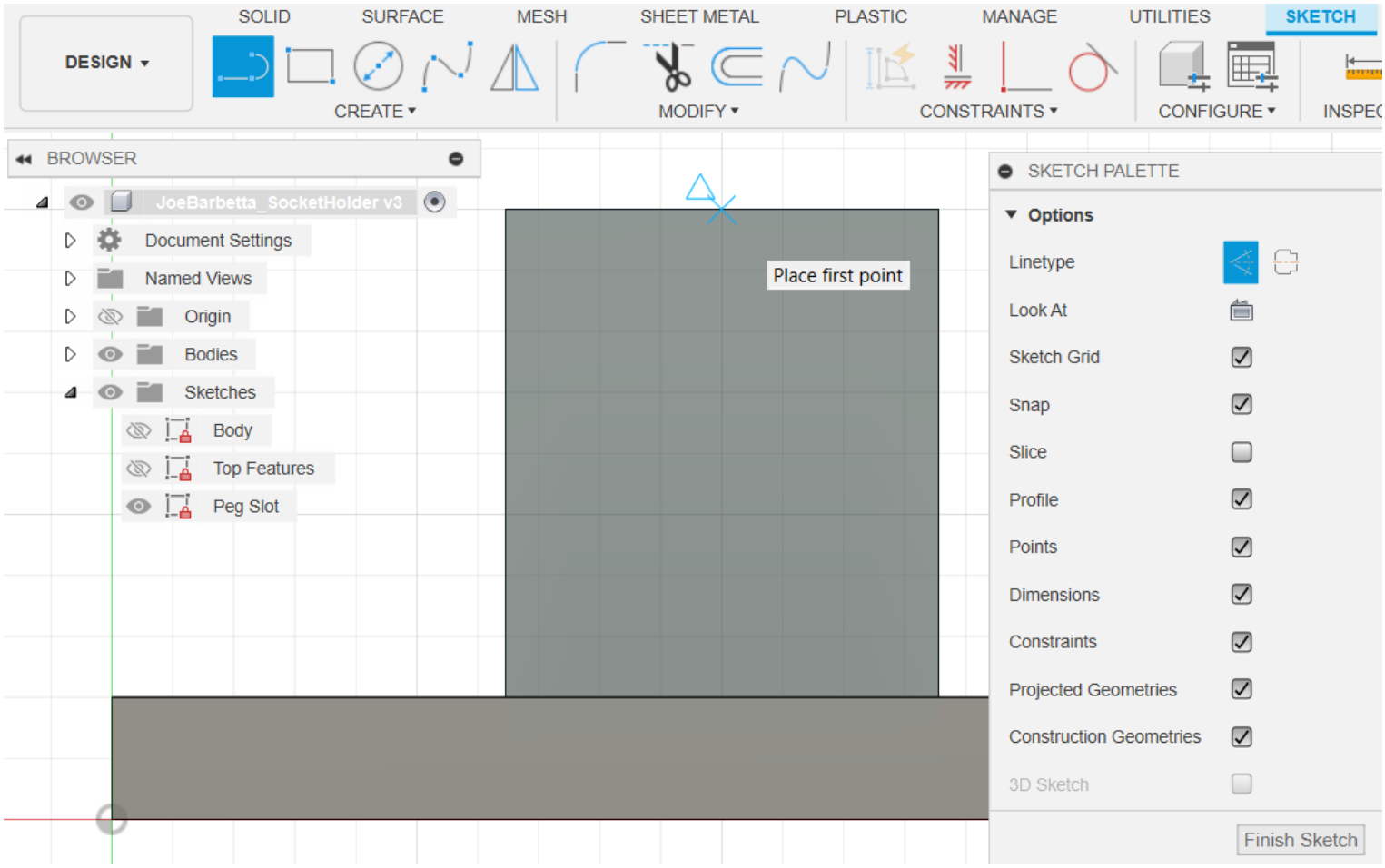

- click on the **eye icon** next to **Top Features** to make the Sketch visible again
- select the **Extrude** tool and click on the **area around the protrusion** to highlight it blue
- enter **-0.05** (note the minus sign) in the **Distance** box and click **OK**



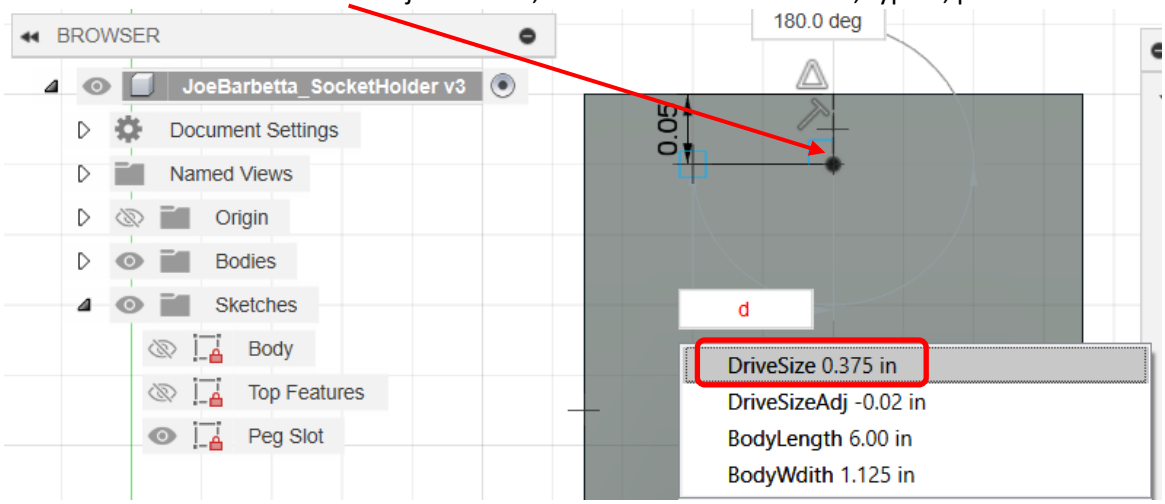
- click on the **eye** icon again to hide the sketch
- select the **Create Sketch** tool and click on the **surface** of the protrusion



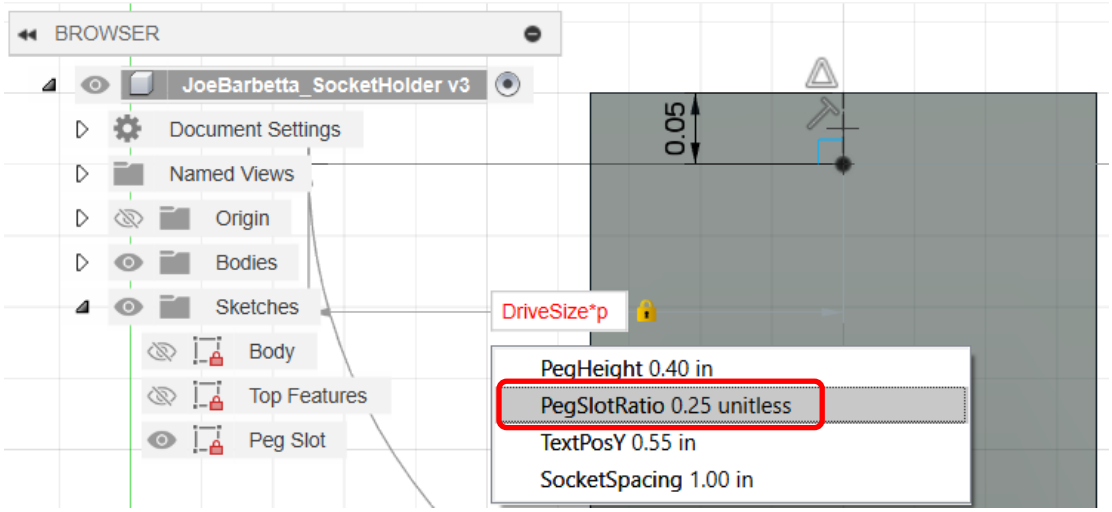

- zoom in similar to that below
- rename the Sketch to Peg Slot
- select the Line tool and click the Construction icon to highlight it blue
- move the mouse over the top edge. When the mouse is in the center of the edge a blue triangle should appear. Click when this happens. Drag the line down and enter 0.05.



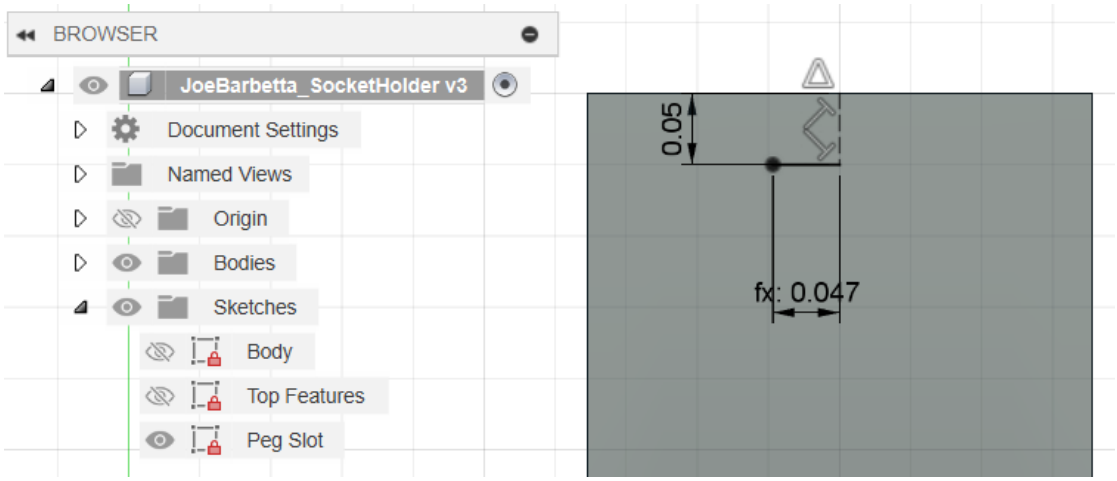
- select the **Line** tool again
- click on the **Construction** icon again to remove the highlighting
- click one the **bottom of the line** just drawn, extend the line **to the left**, type **d**, press Enter to select **DriveSize**



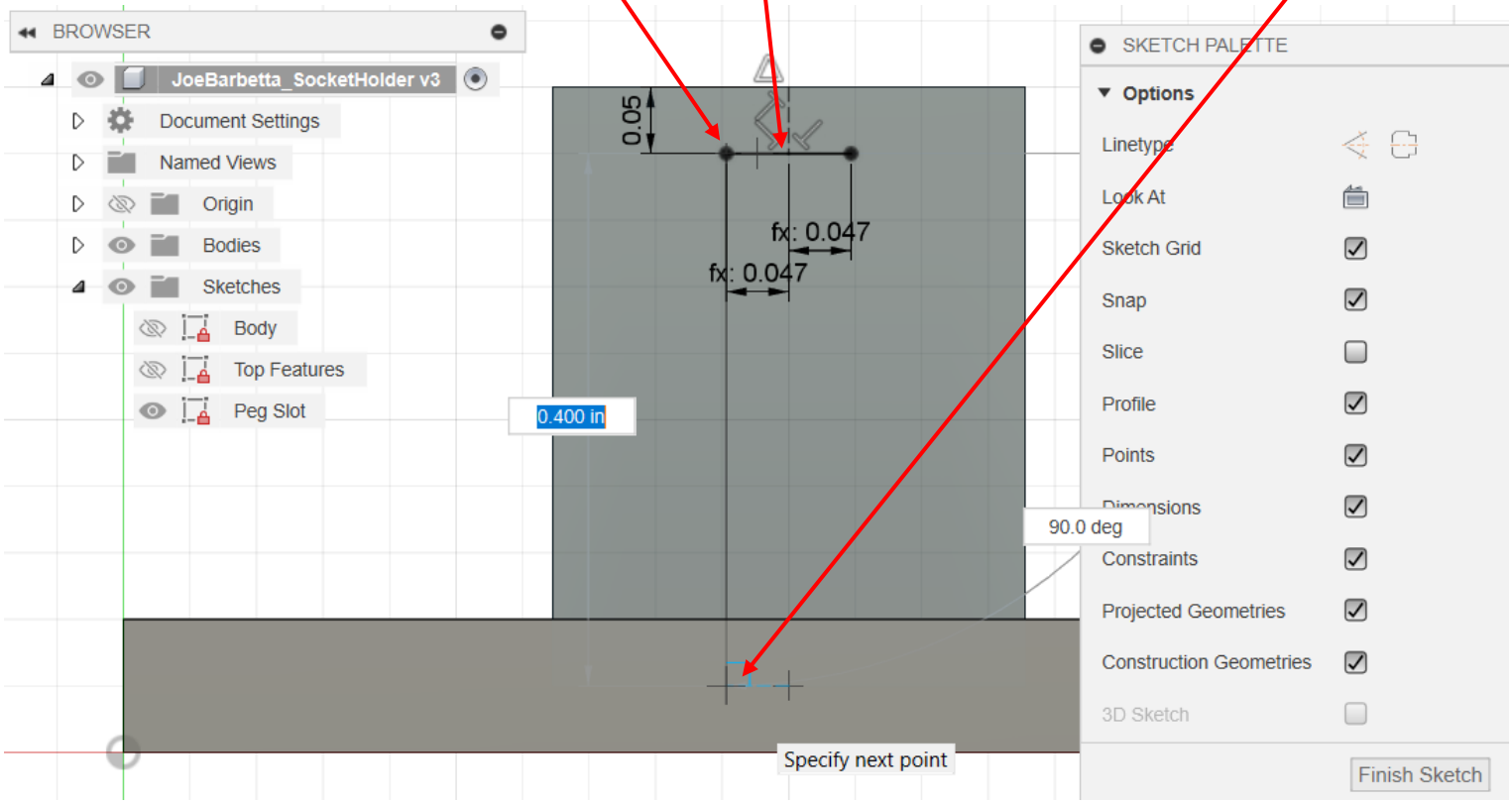

- type an **asteric**, type **p**, use the down arrow to select **PegSlotRatio**
- type **/ 2** and press Enter



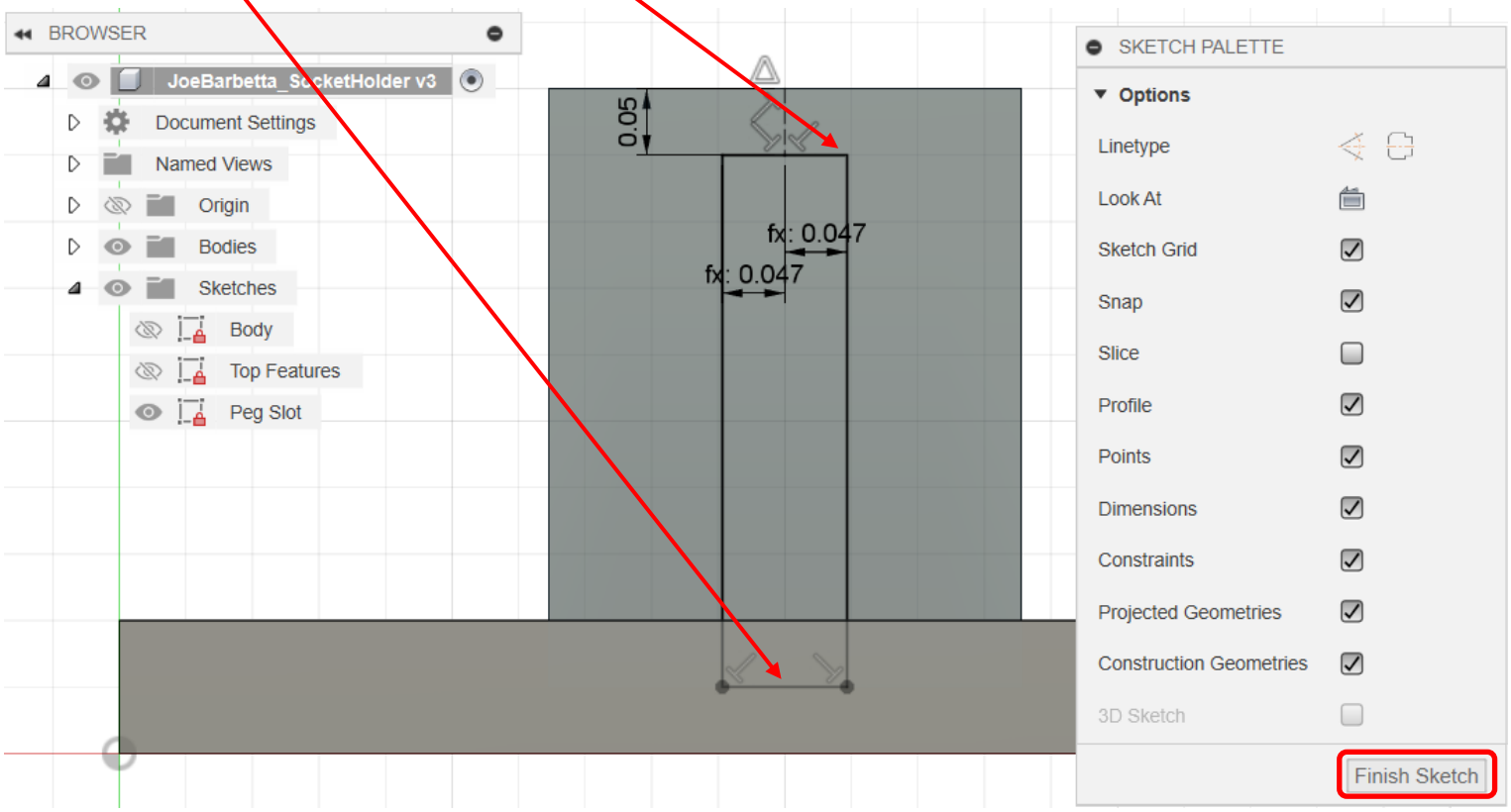

The result should look like that below. We ended up setting its dimension with the equation **DriveSize * PegSlotRatio / 2**.



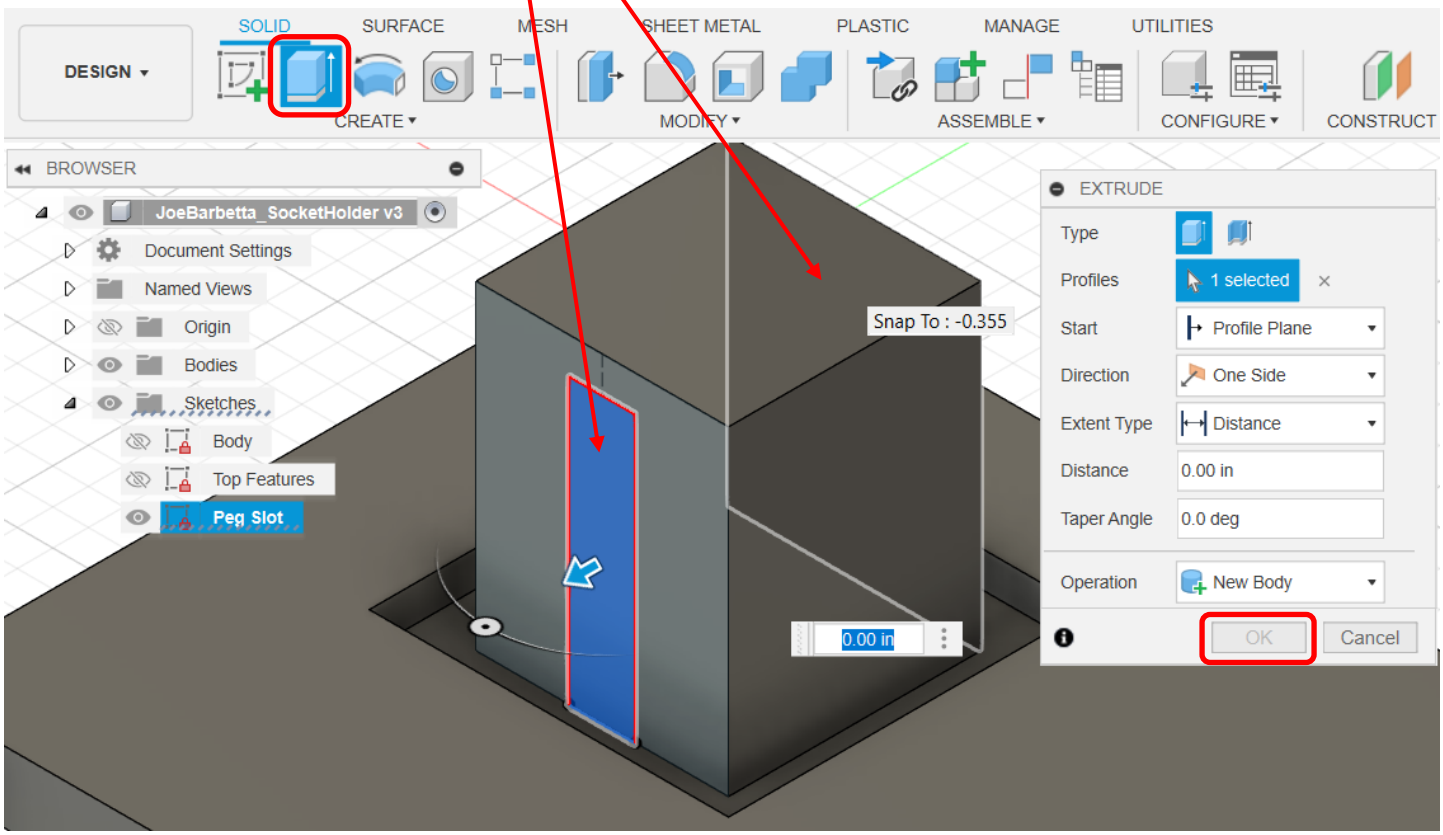
- use the **Line** tool and, as before start a line at the **bottom of the 0.05 line**, extend to the right, and implement the equation **DriveSize * PegSlotRatio / 2** again.
- create another line downward from the **point indicated** and as it nears the bottom it should snap to the **position shown**, which is the hidden surface that was created when we extruded a region down 0.05.



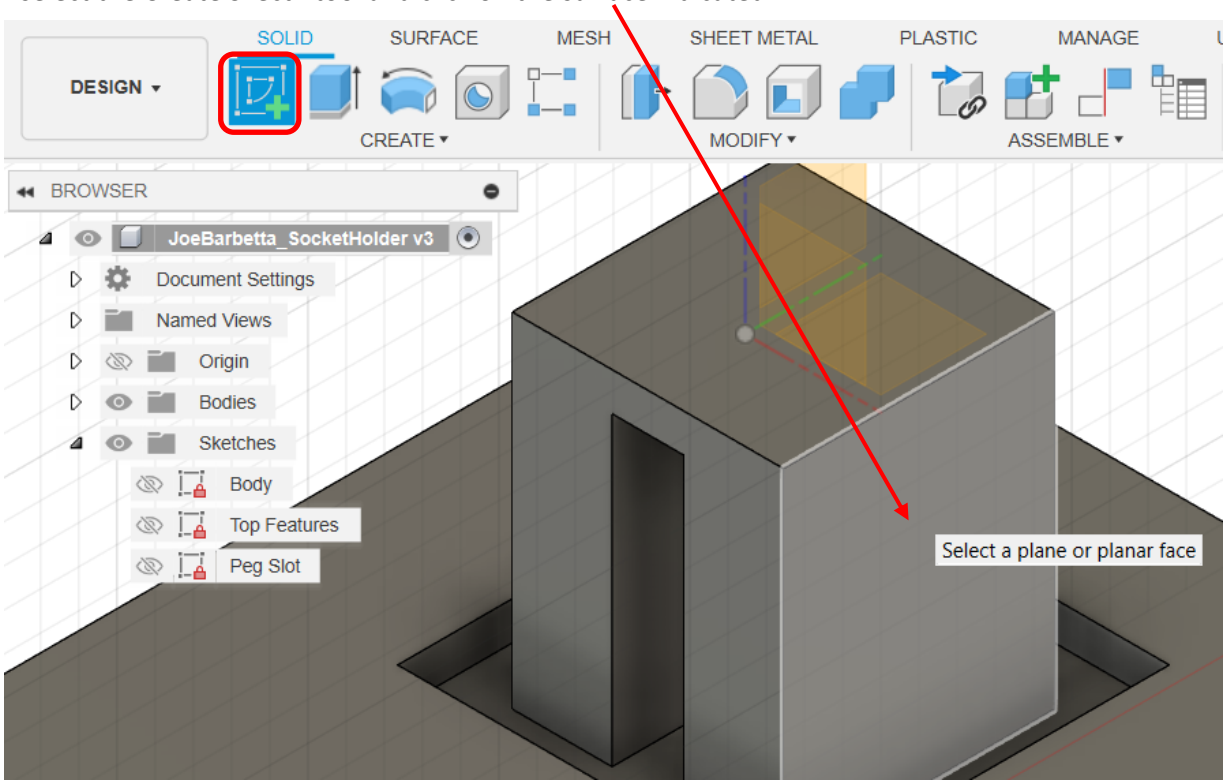

- create **2nd line** downward from the **point indicated** as done with the previous line.
- create a **3rd horizontal line** to connect to two endpoints and click **Finish Sketch**.



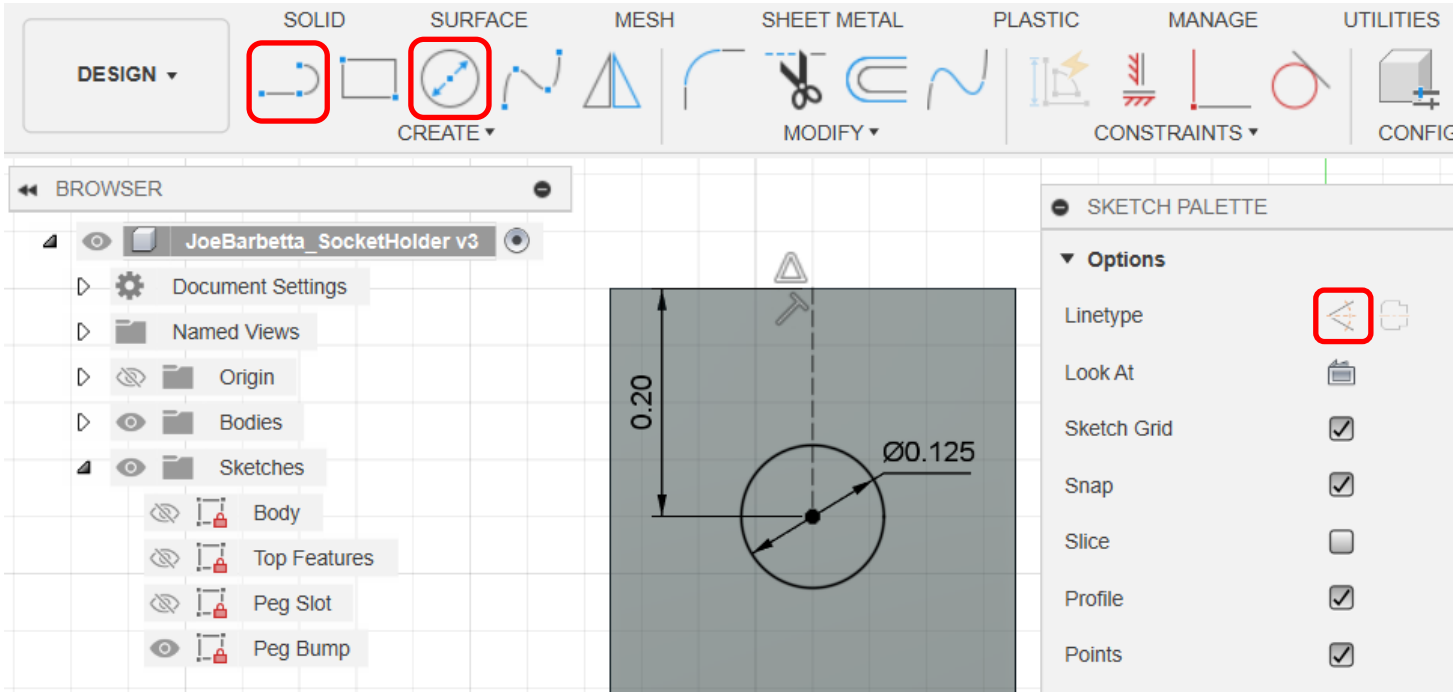
- return to the **Home** view and zoom in to achieve the below view
- select the **Extrude** tool, click on the **rectangular profile** just created
- press the left mouse button on the area shown until a list of faces shows and select the one corresponding to the rear surface and **click**. This is being done to select through the body. This should result in a rectangular hole being cut through the peg.
- click **OK**



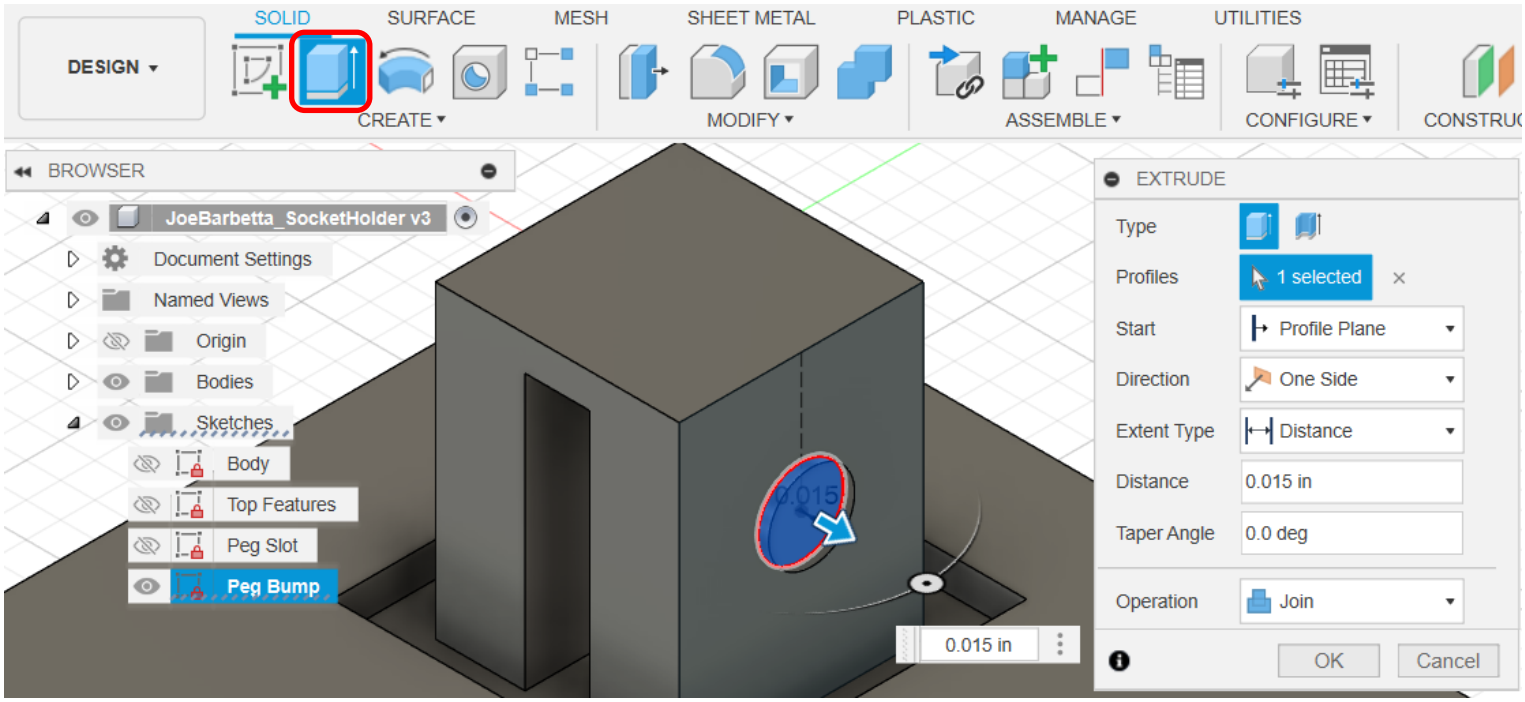

- select the Create Skecth tool and click on the surface indicated



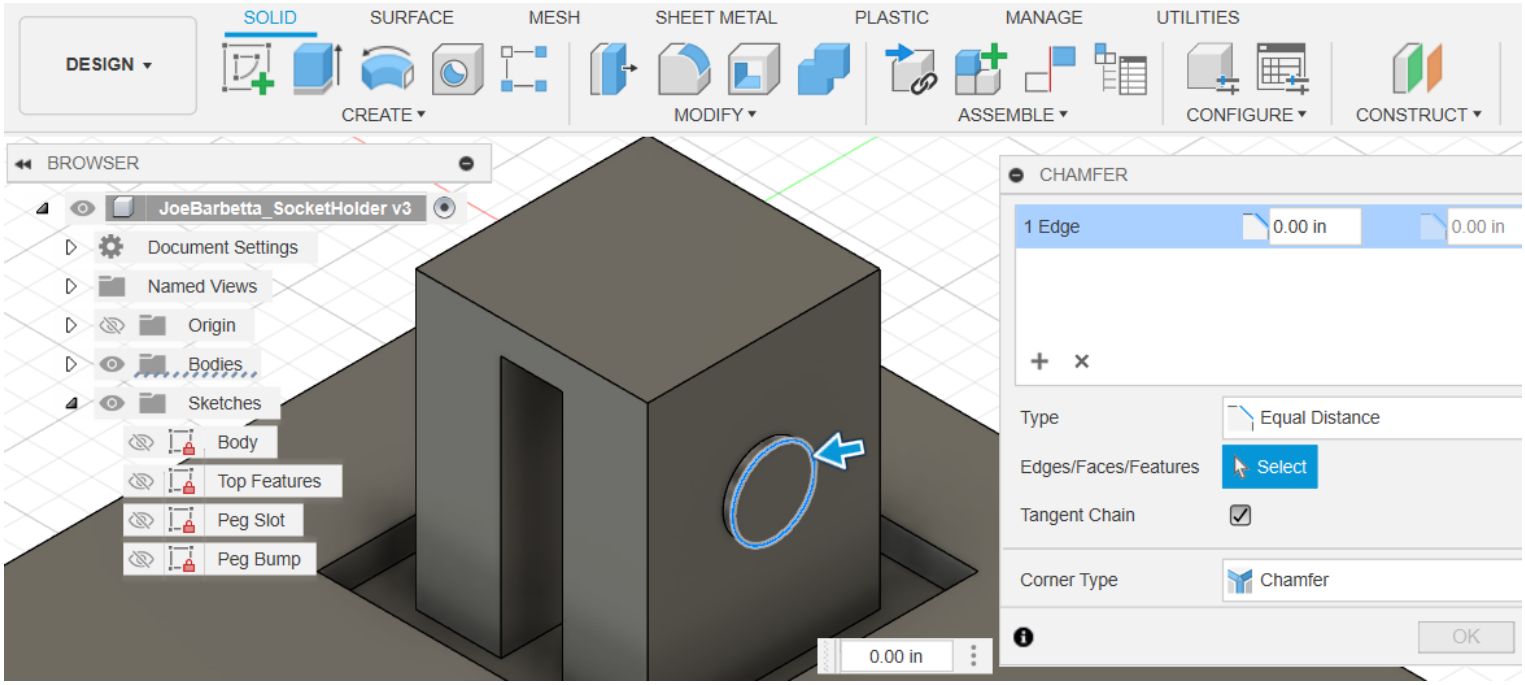
- rename the Sketch **Peg Bump**
- create a **Construction** line from the **center of the top edge** downward by **0.20**
- from its bottom end create a circle 0f **0.125** and click **Finish Sketch**



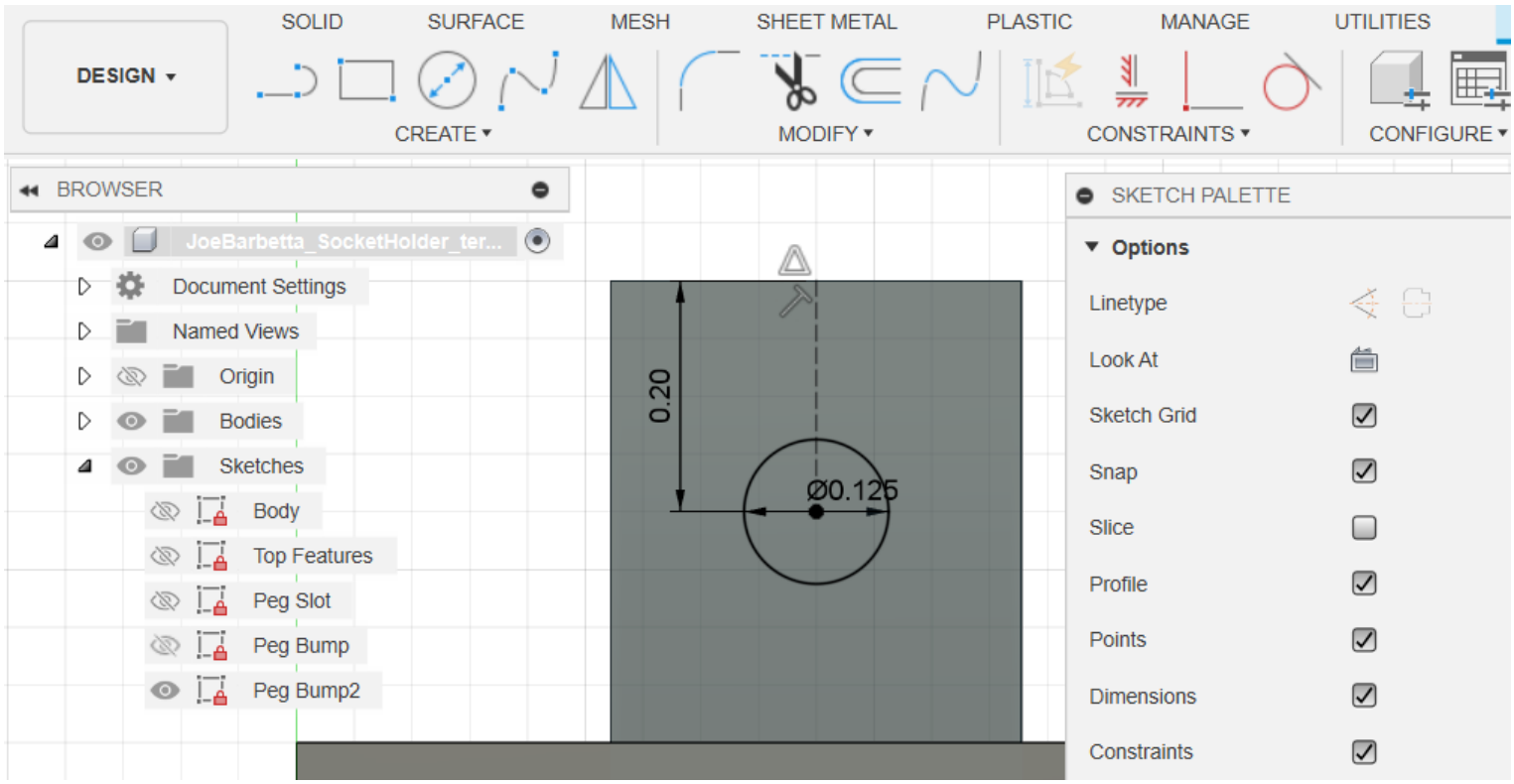

- extude the circular region out by **0.015**



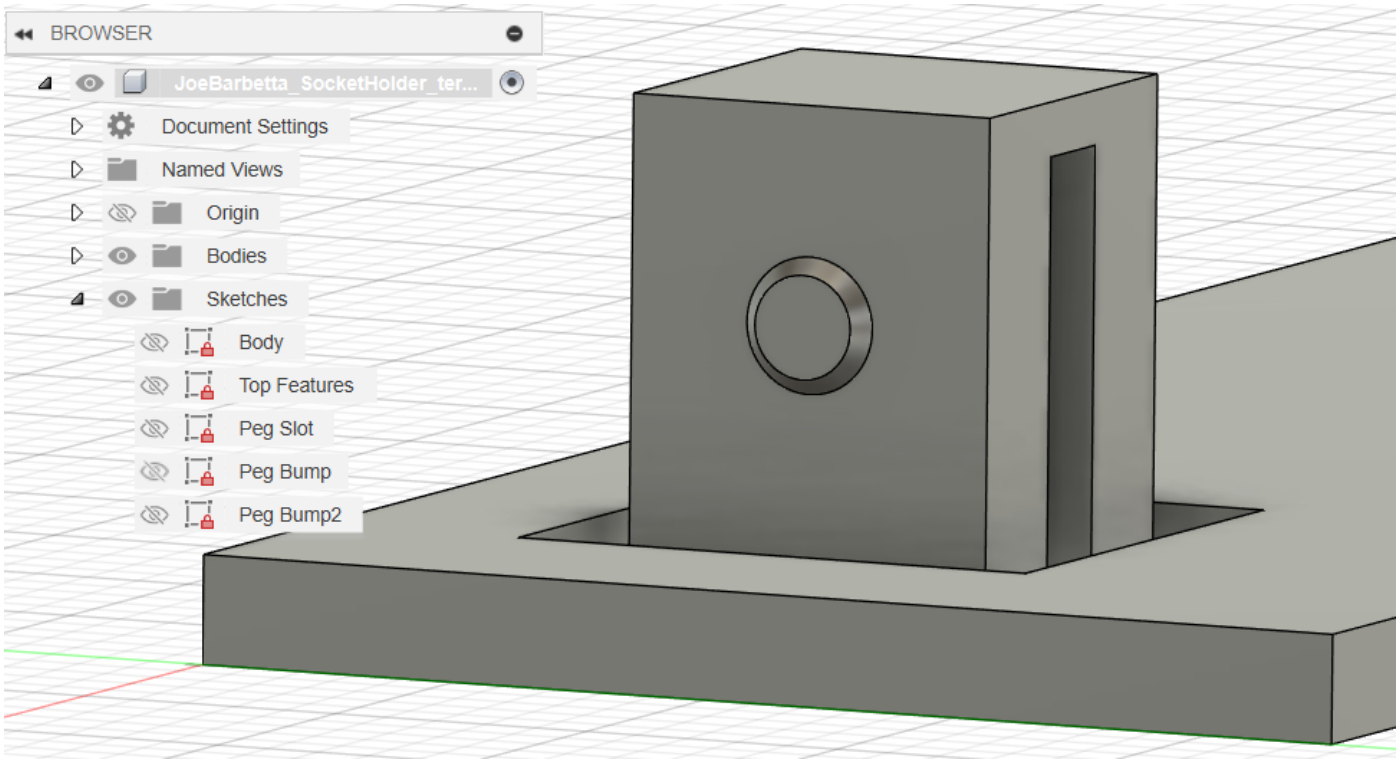
- from the **MODIFY** menu select the **Chamfer** tool and create a chamfer of **0.015**



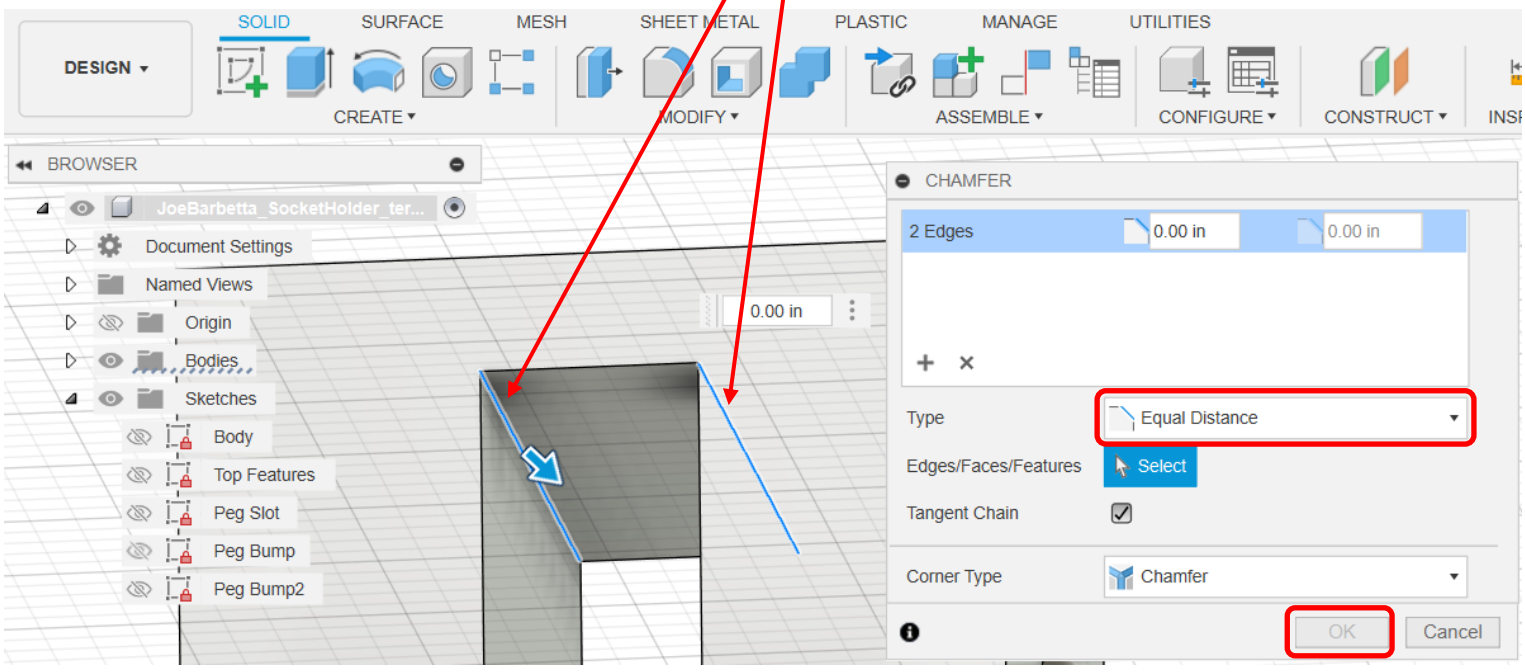

- rotate the view to create the same **sketch** on the **opposite side of the peg** and rename it **Peg Bump2**



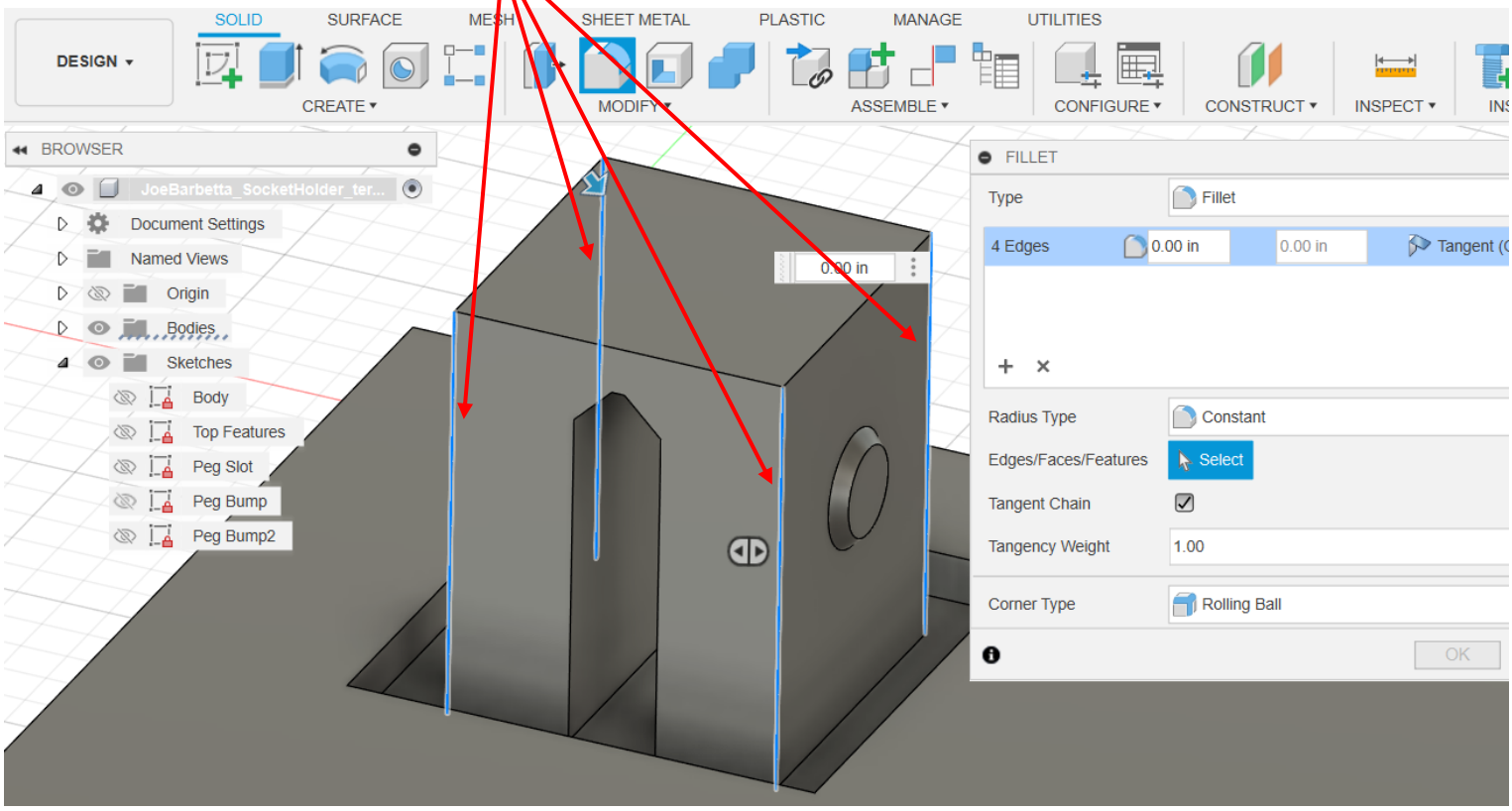
- extrude the circle by **0.015** and add a chamfer of **0.015**.
Both sides of the peg should now have the same bump.



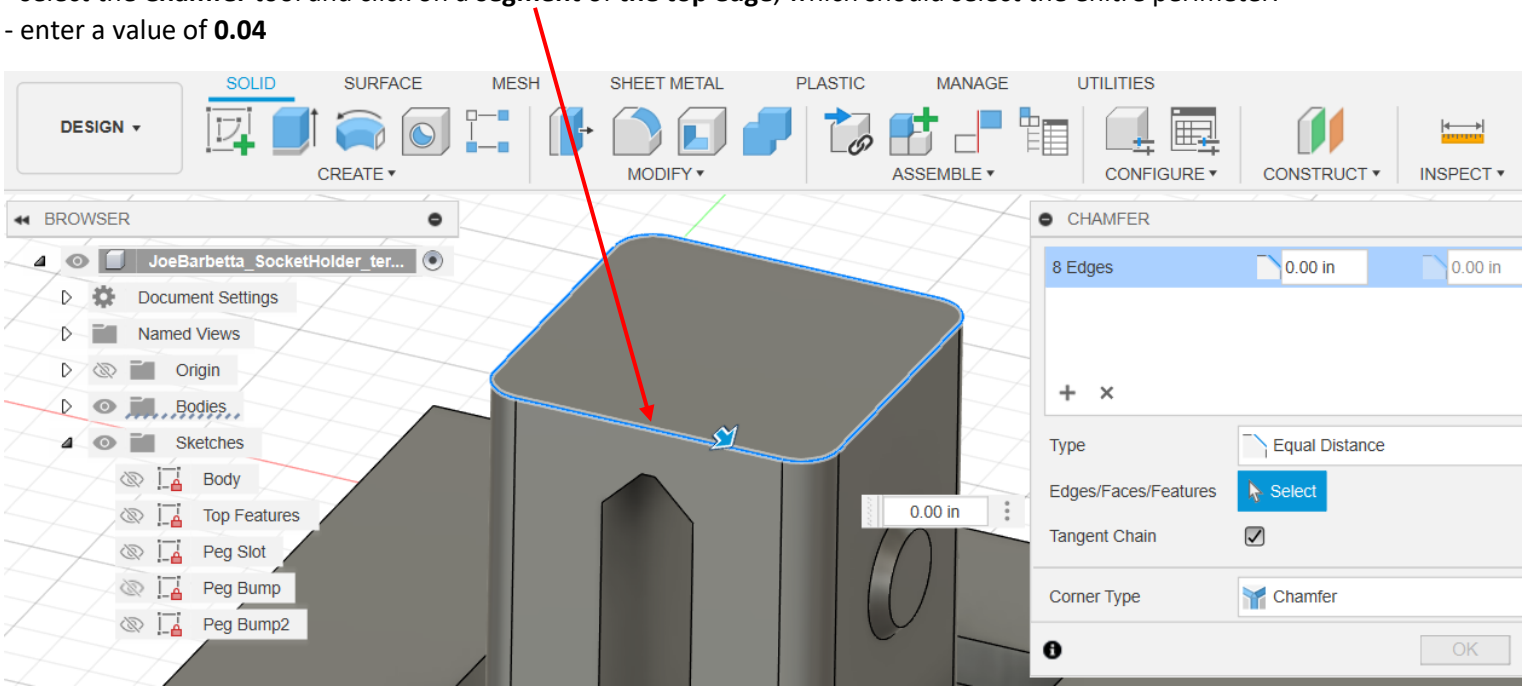

- rotate the view to access the **underside of the slot top**
- from the **MODIFY** menu select the **Chamfer** tool and select the **2 edges** as shown. Note that for the hidden edge, one may need to hold down the mouse button to select it. However, sometimes the selection mode will allow a single quick click.
- ensure that the Type is set to **Equal Distance** and enter a value of **0.04** and click **OK**.
These chamfers will help with the printing of the top "bridge".



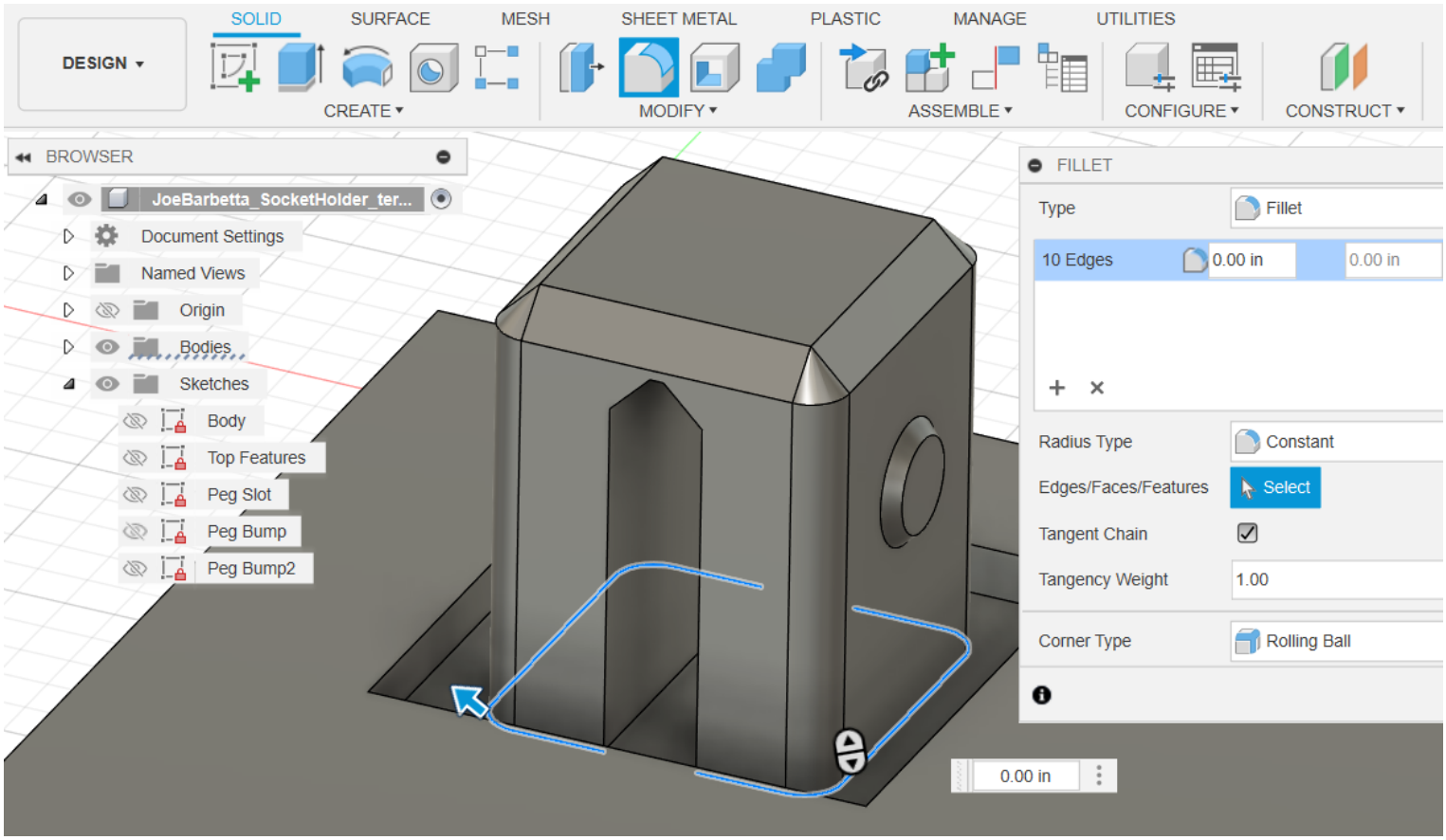
- select the Fillet tool and select the **4 vertical edges**
- enter **0.04**



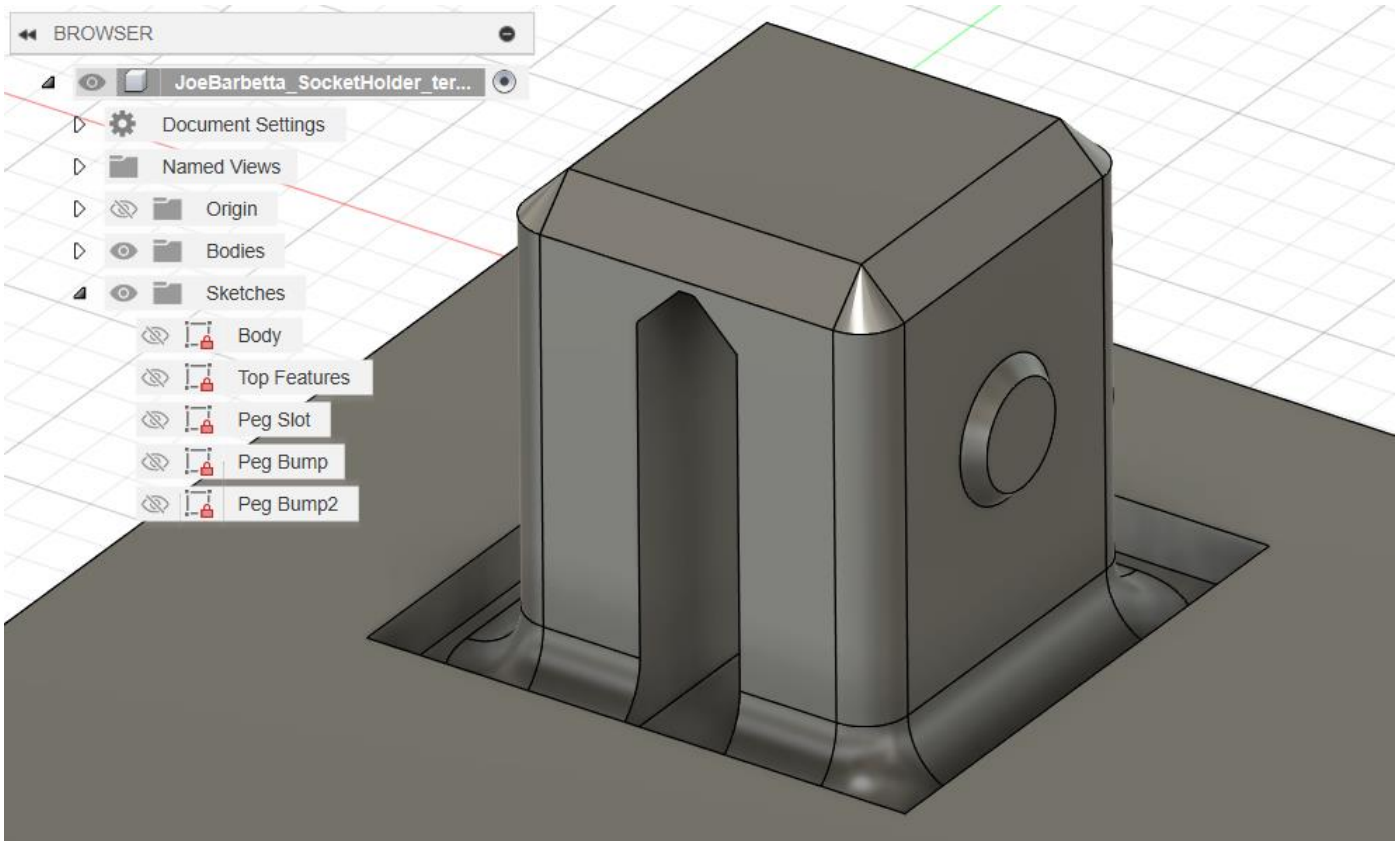

- select the **Chamfer** tool and click on a **segment** of **the top edge**, which should select the enitre perimeter.
- enter a value of **0.04**



- select the **Fillet** tool and select the **bottom edges of the peg** and enter a value of **0.06**
Fillets can be added to any sharp internal corner to reduce stress concentrations to prevent breakage.

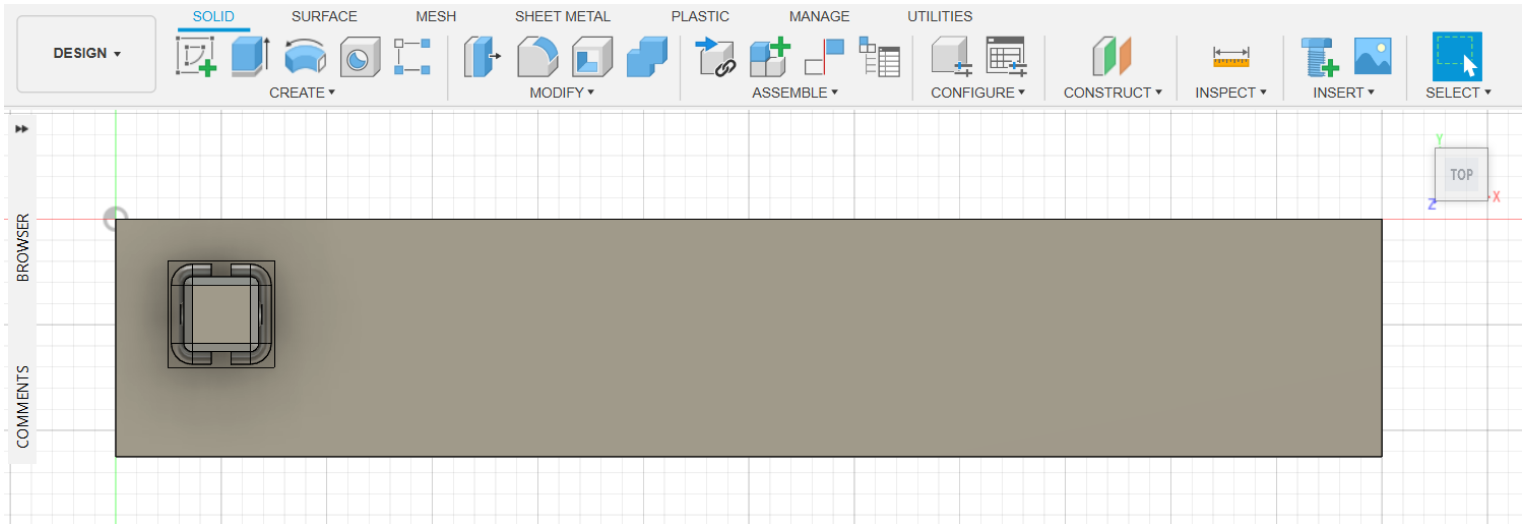



The peg should look like that below.



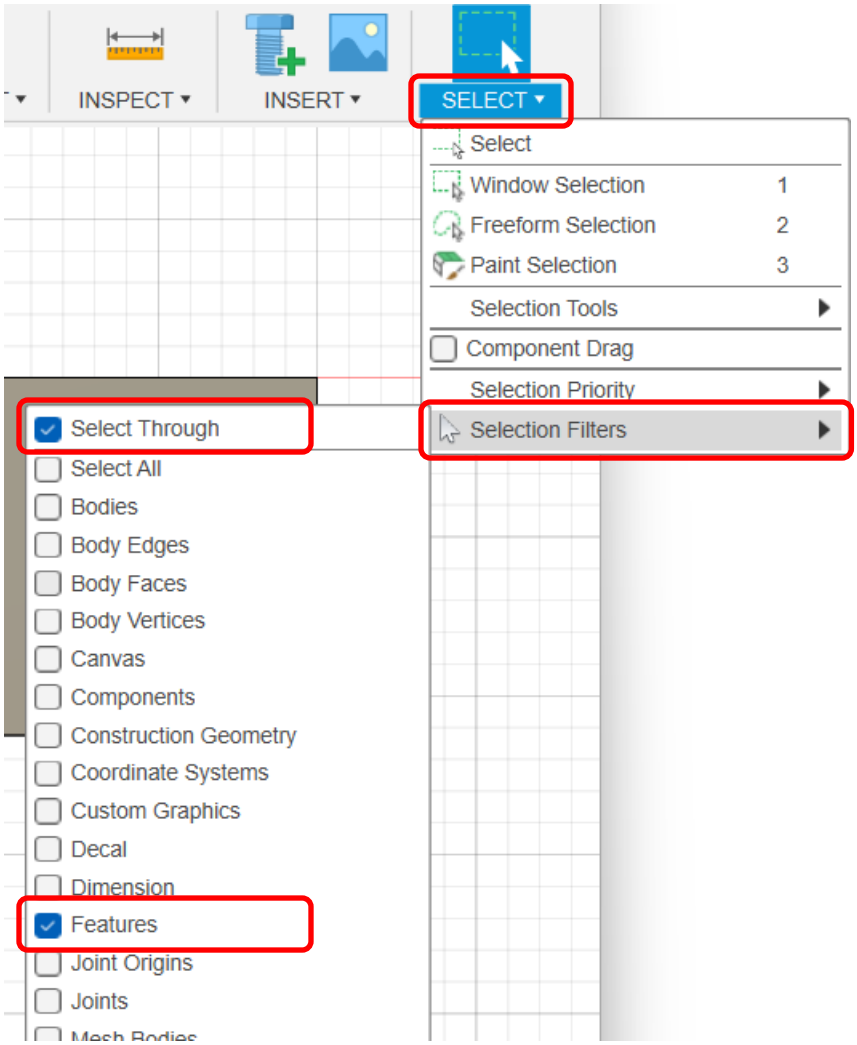
# Patterning Features

- adjust the view as shown below

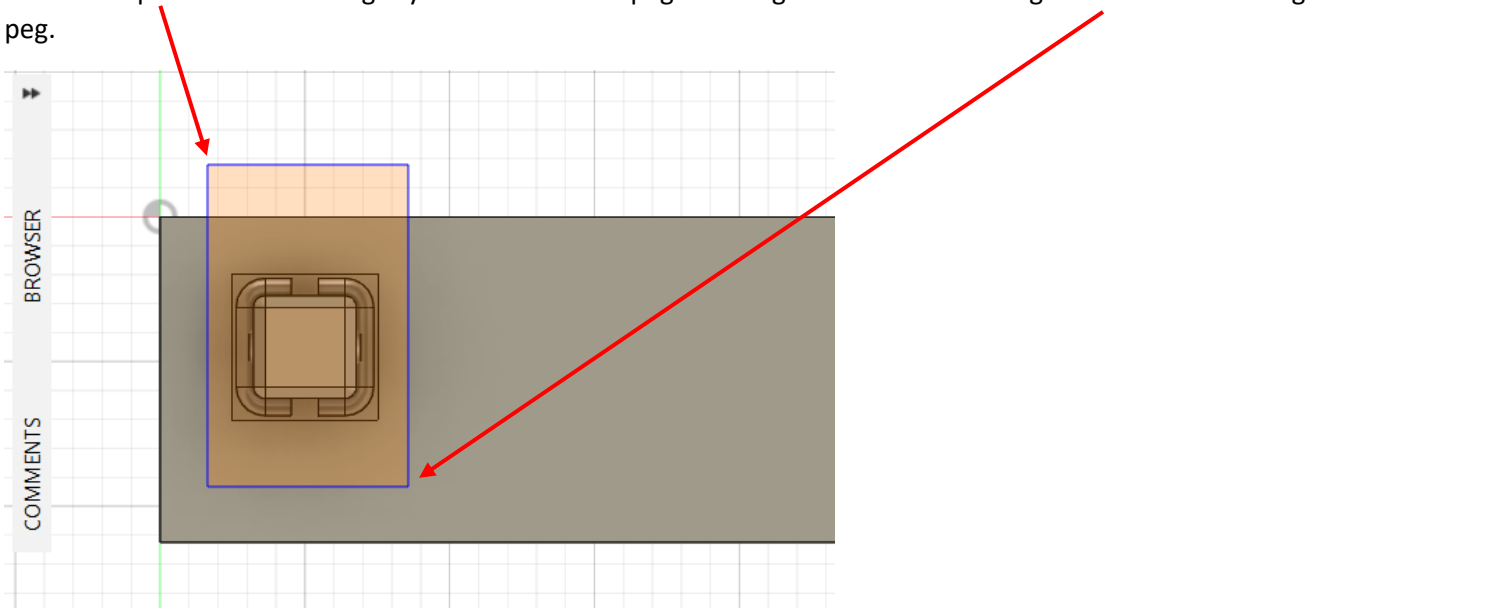



- from the top right **SELECT** menu, click on **Selection Filters**, which should show a long list with all the items checked
- click on **Select All** to uncheck it and then click on **Features** to select it.
- ensure that the top **Select Through** option is checked

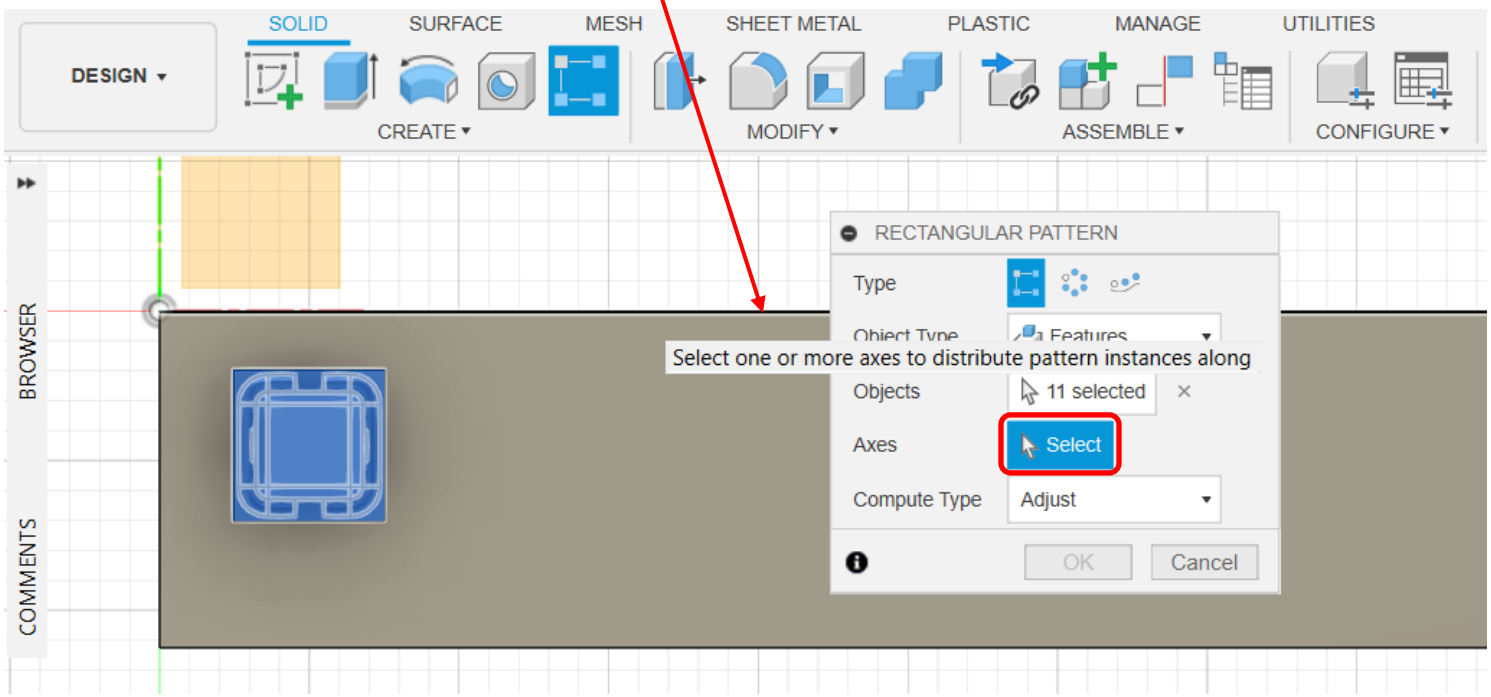
The result should look like that below. Note that the entire list is not shown.



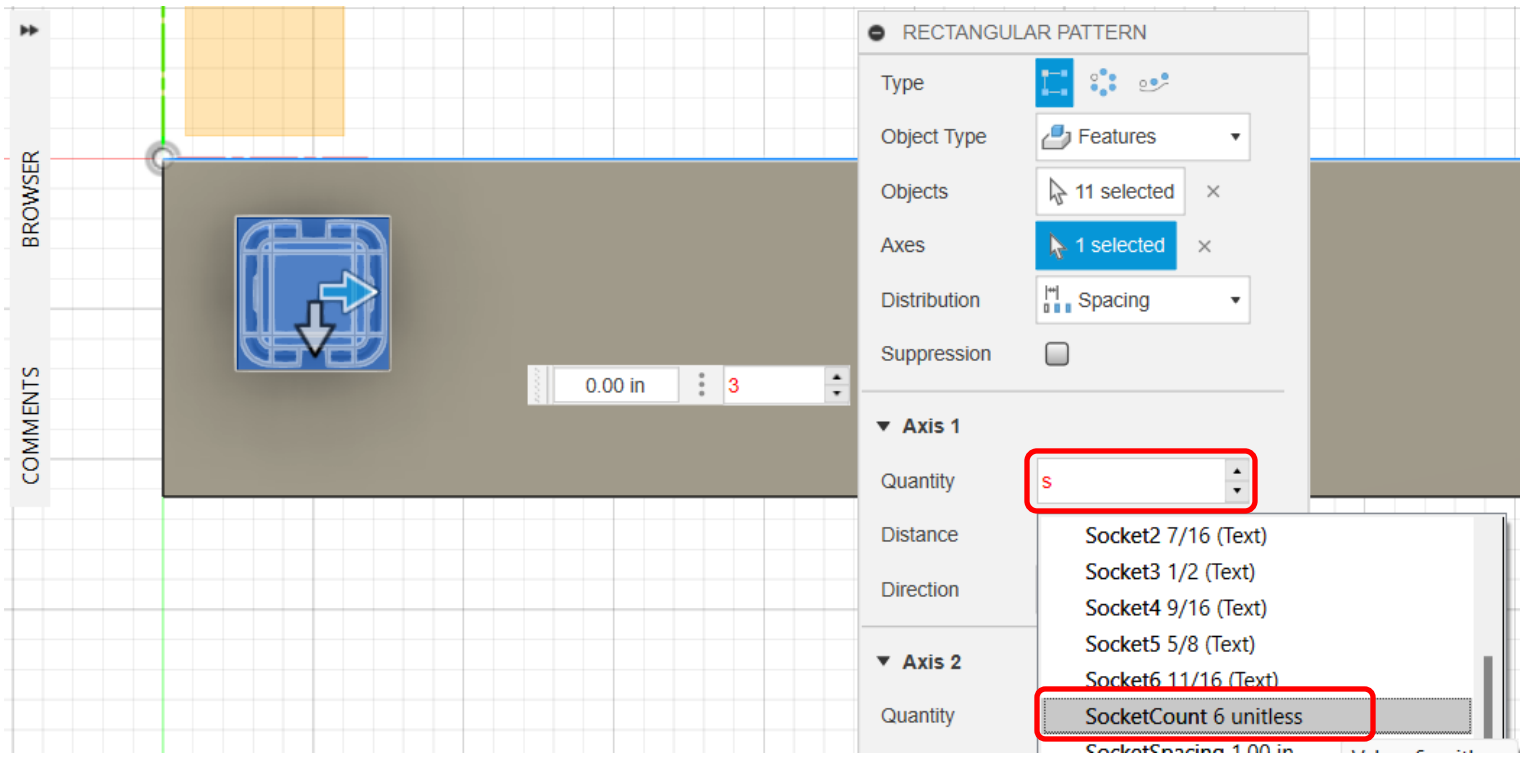
- click on a point above and slightly to the left of the peg and drag the selection rectangle down and to the right to select the peg.



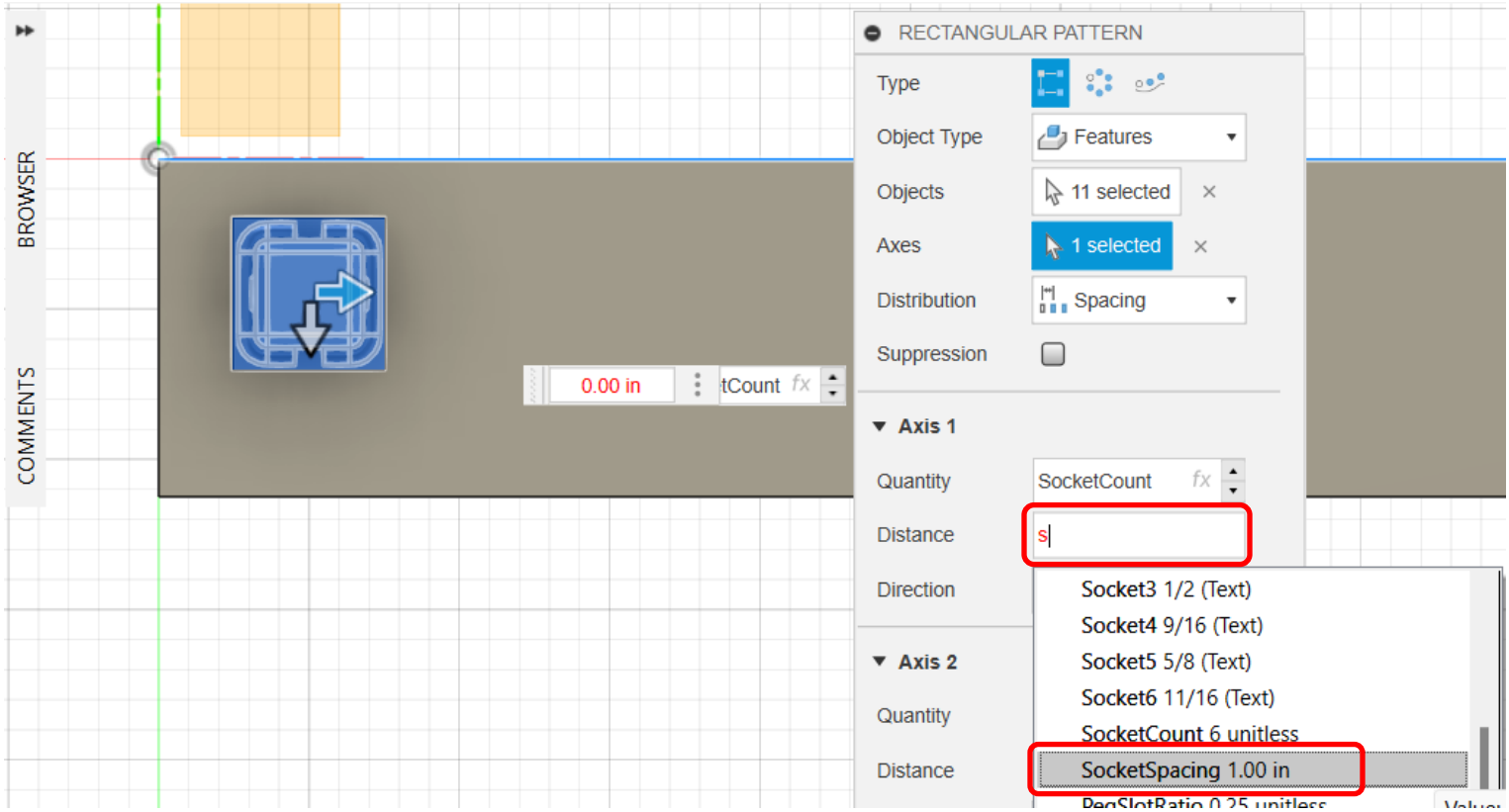

- from the **CREATE** menu select **Pattern** and **Rectangular Pattern**
- click on **Select** next to **Axes** and then click on the **top edge** of the body



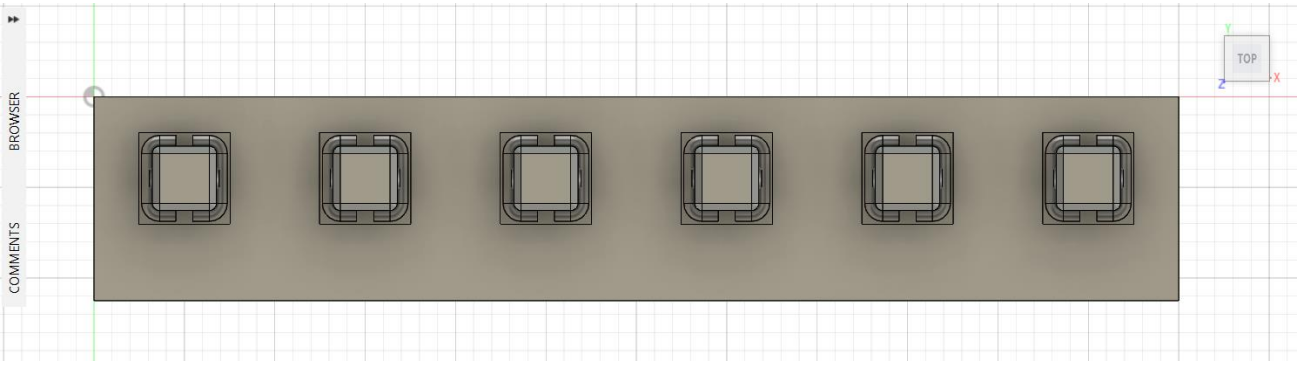
- in the 1st Quantity box type **s** and select **SocketCount**



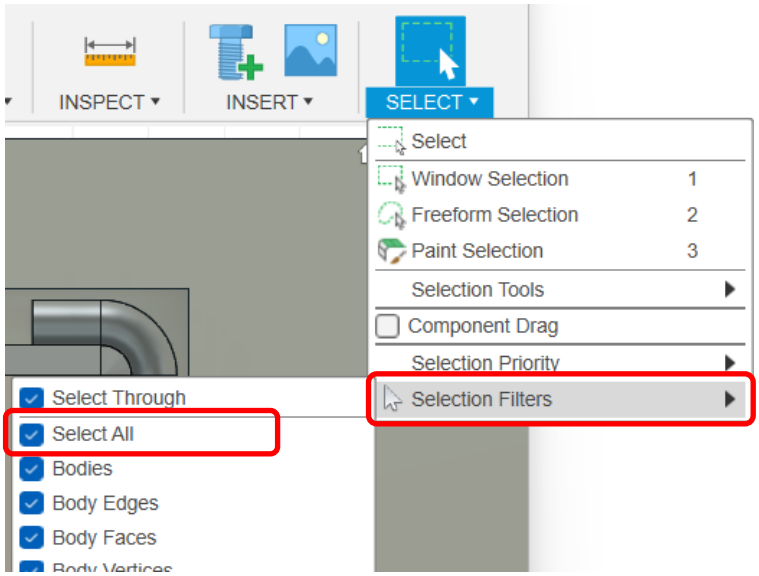

- in the Distance box under it type **s** and select **SocketSpacing**
- click **OK**



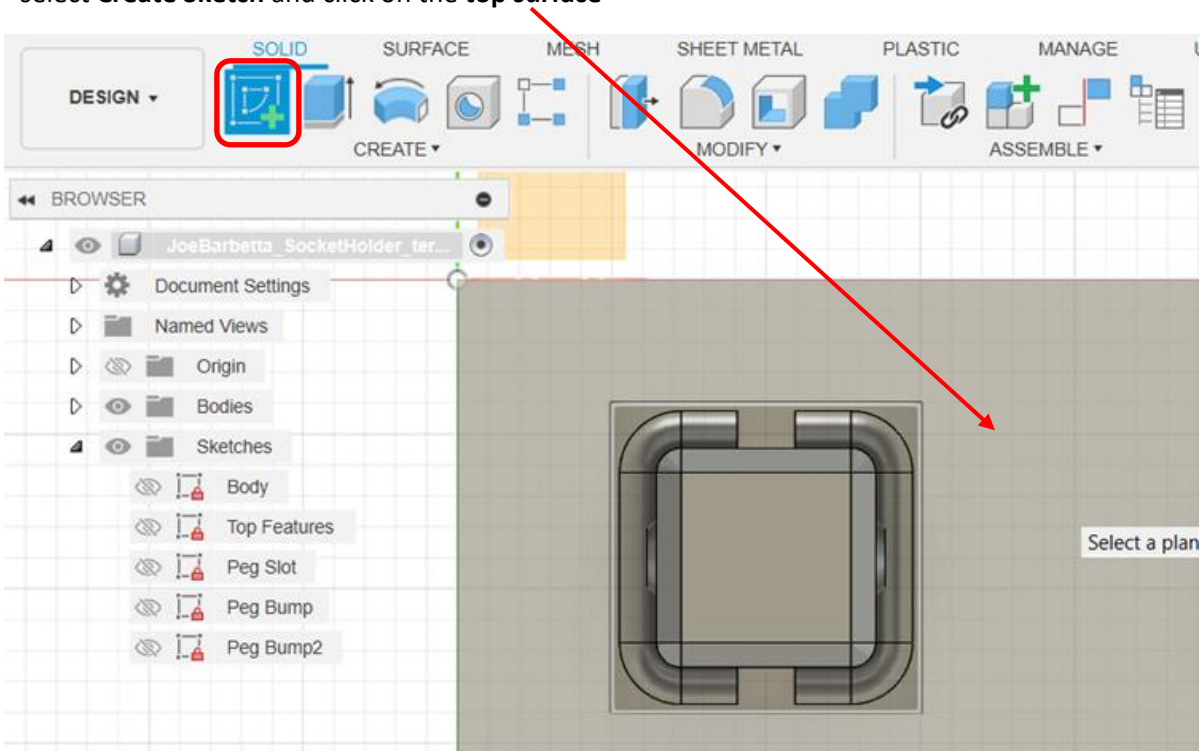
There should now be 6 pegs.



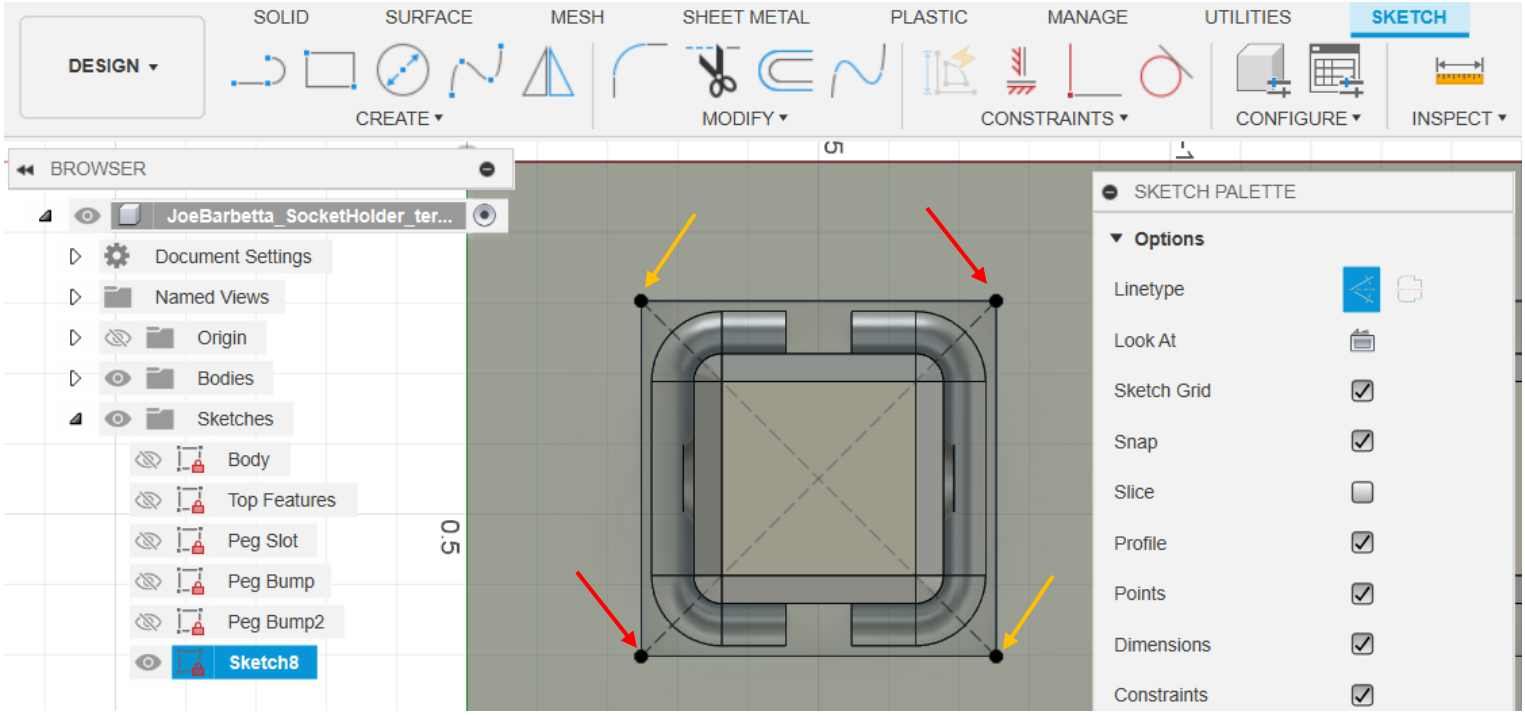

- open the **Selection Filters** again and click **on Select All** to bring the filters back to the defaults



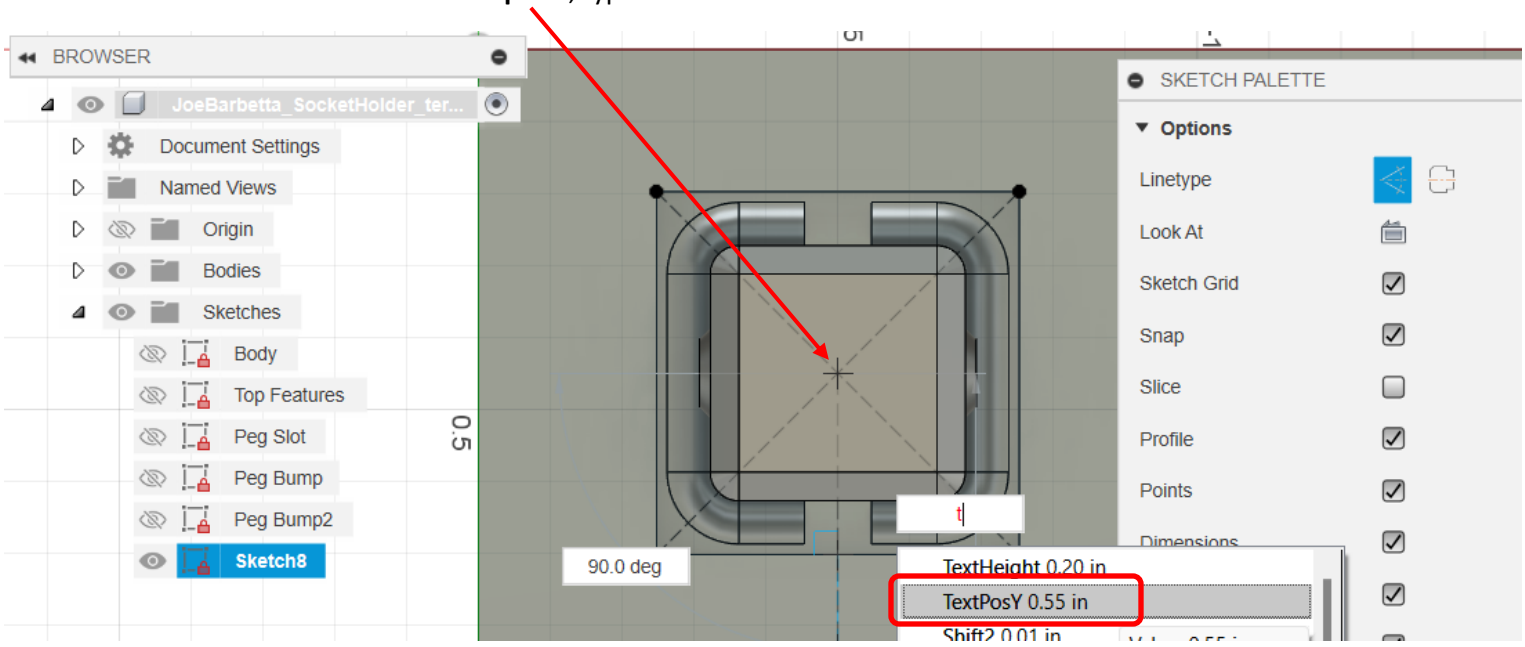

- zoom into the 1st peg as shown below
- select **Create Sketch** and click on the **top surface**



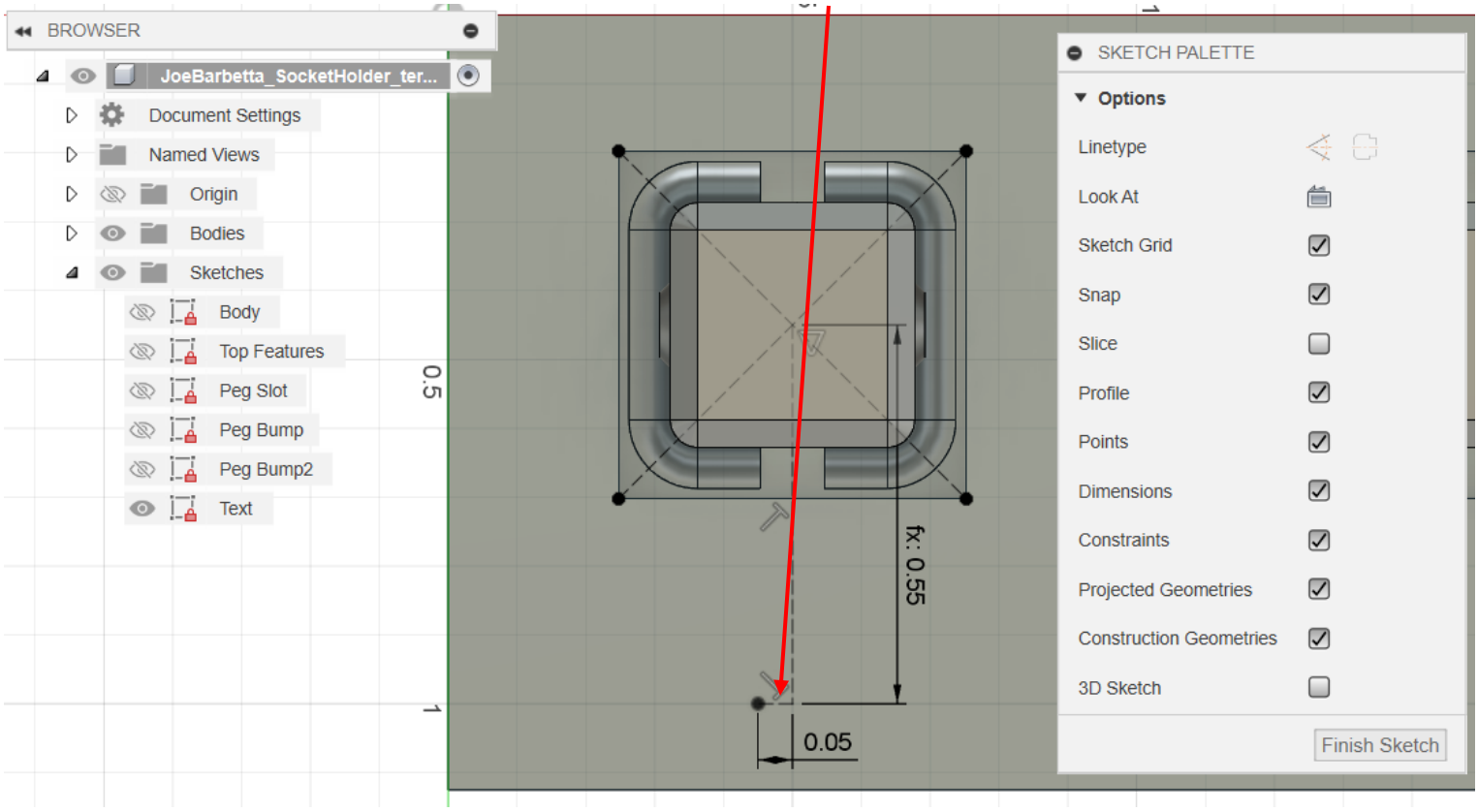
- select the **Line** tool and click the **Construction** line icon to highlight it
- create **two lines** connecting the **opposite corners** of the large square.  This is an old trick to identify the center of a square or rectangle in CAD, on paper, or on an actual piece of material.



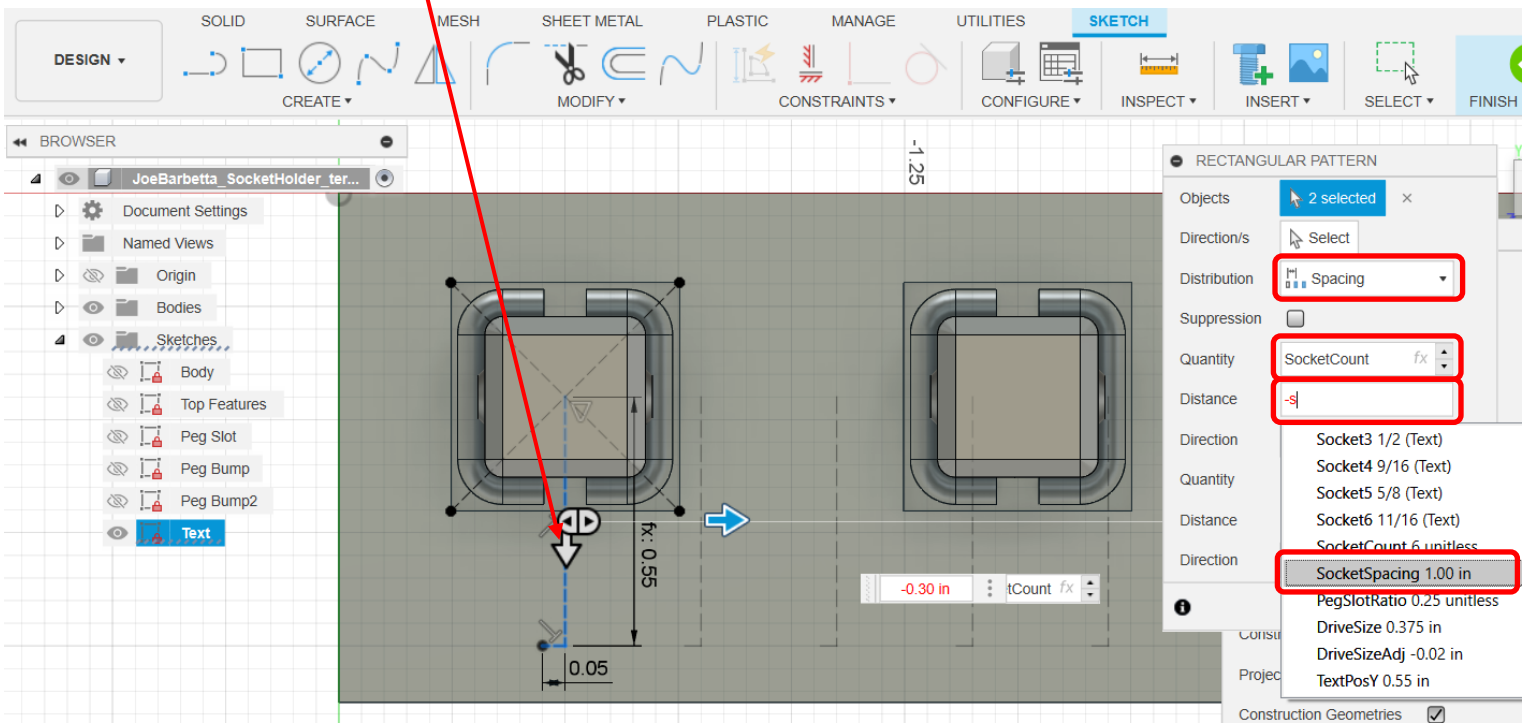

- create a line downward from the **center point**, type **t** and select **TextPosY**



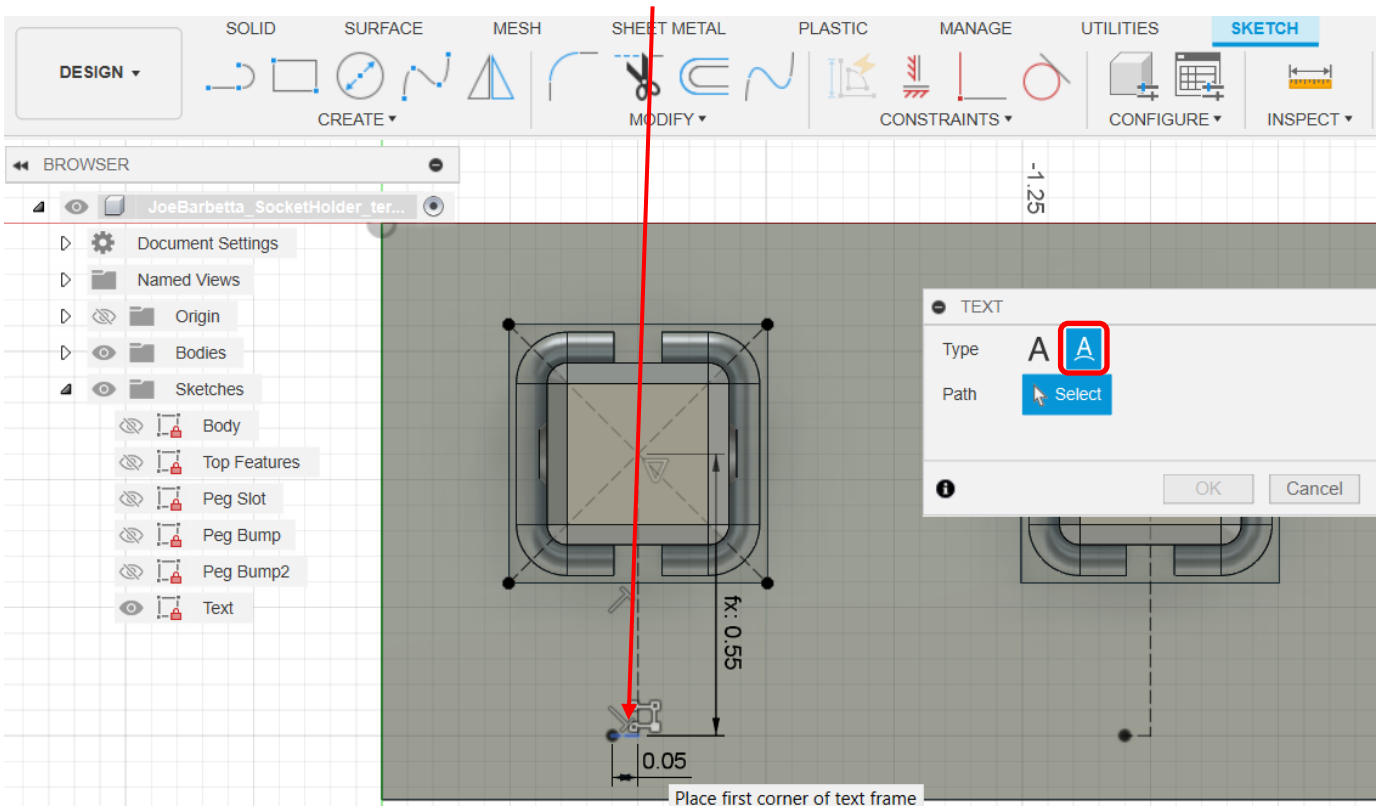
- from the bottom of that line create a short line to the left of length **0.05**



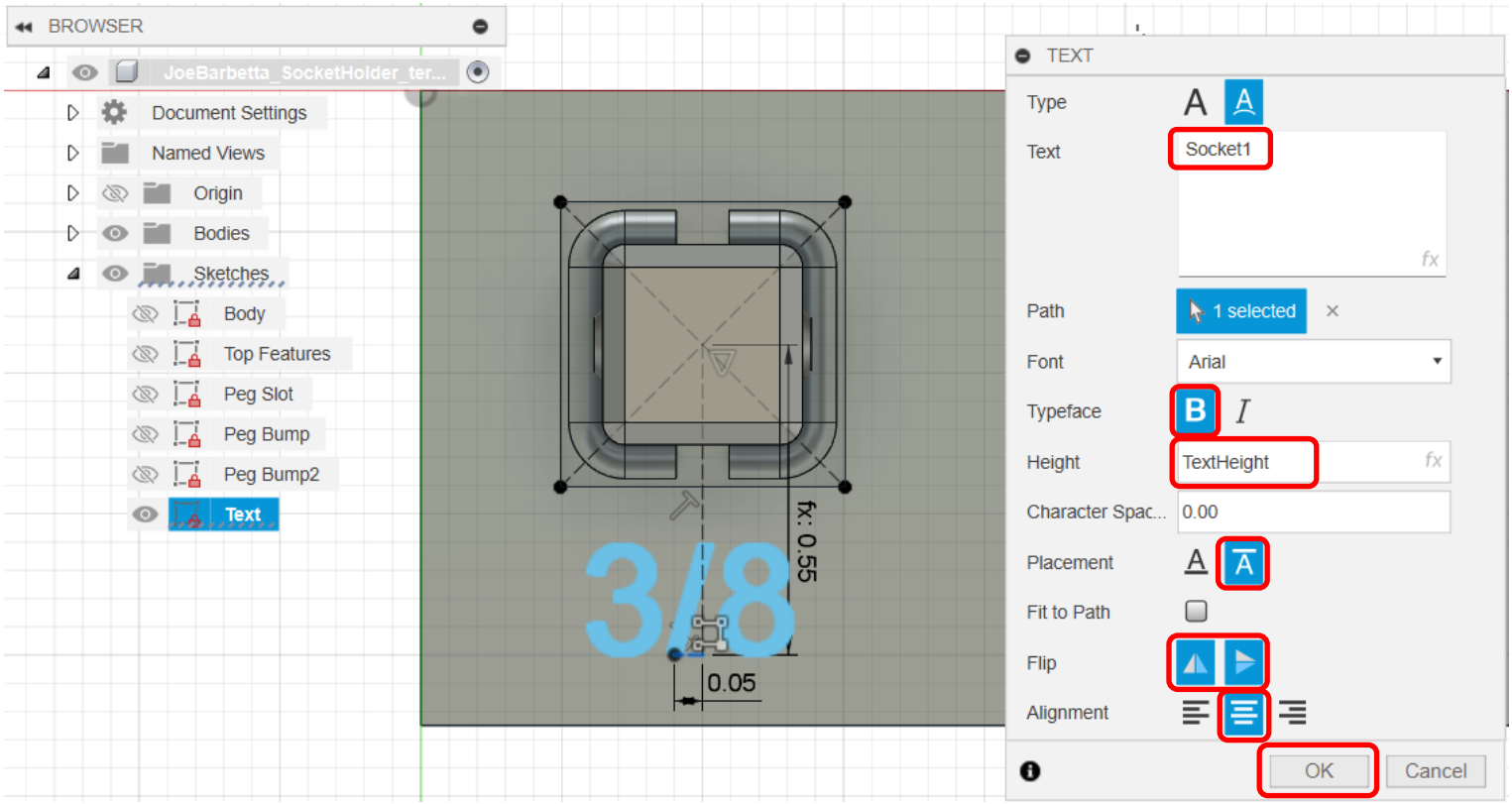

- zoom out a little
- from the **CREATE** menu select **Rectangular Pattern** and ensure **Distribution** is set to **Spacing**
- **double-click on the dashed vertical line**, which should highlight it and the small 0.05 in line blue
- pull the **blue arrow** a little to the right
- in the the **Quantity** box type **s** after the minus sign and select **SocketCount**.  Make sure to keep the minus sign.
- in the **Distance** box type **s** and select **SocketSpacing**
- click **OK**



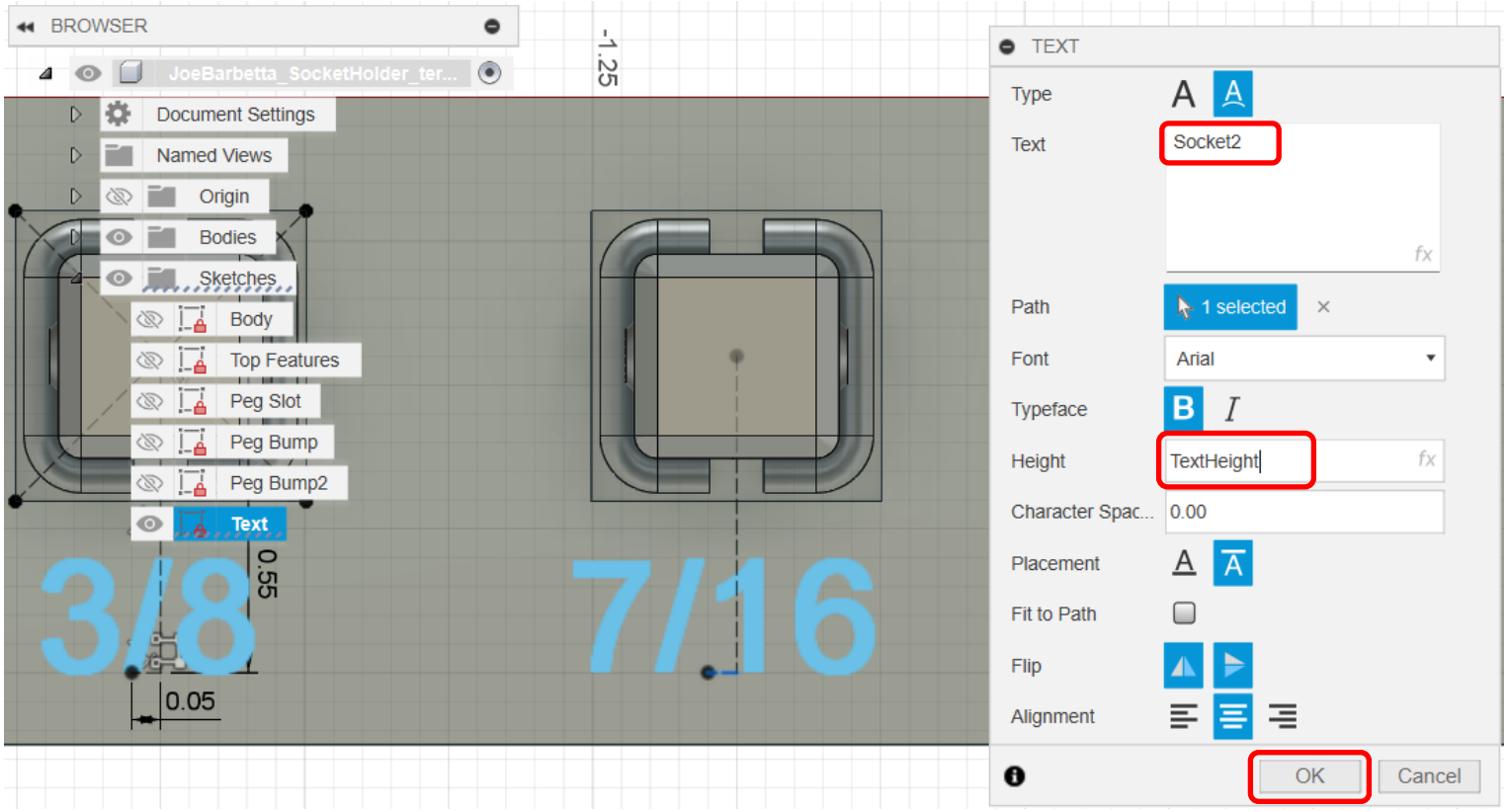
- from the **CREATE** menu select **Text**
- if a **Paremetric Text** window shows click **OK** to close it
- select the **Text on Path** icon and click on the **small 0.05 in line**



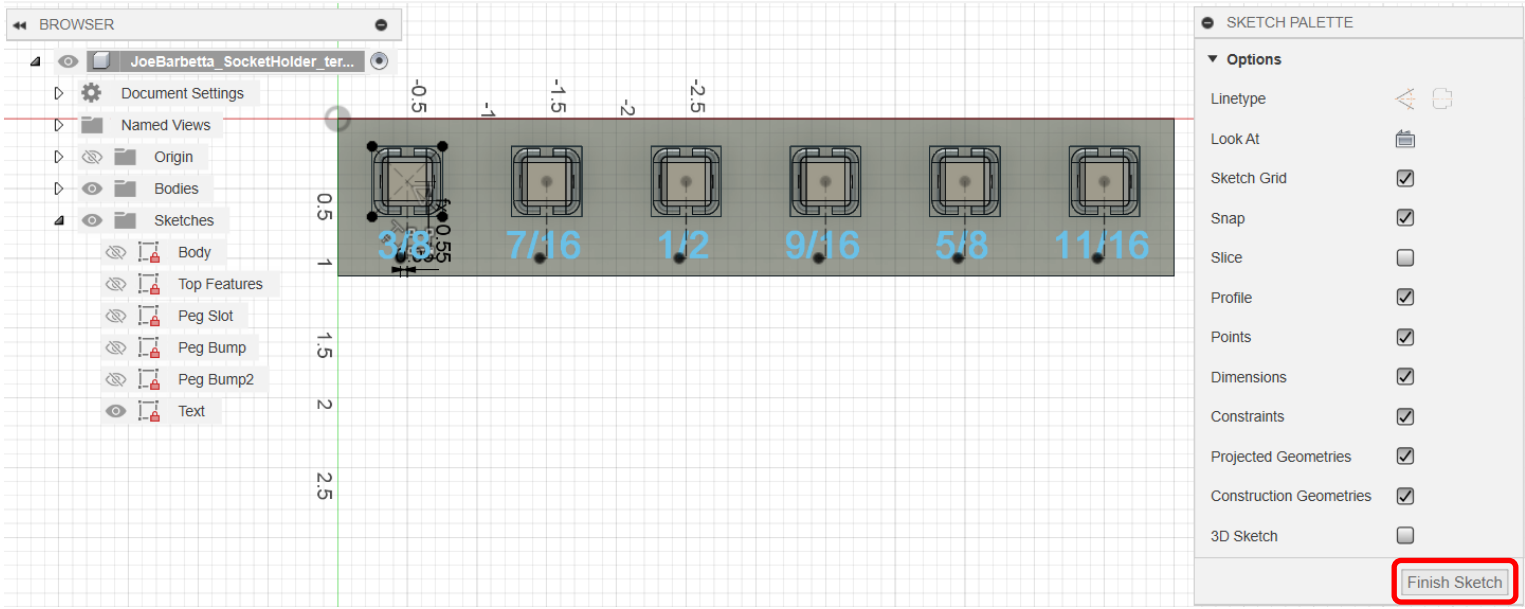

- in the **Text** box **select the two single quotes**, type **s**, and select **Socket1 3/8 (Text)** and click on the **B** icon for bold text
- in the **Height** box type **t** and select **TextHeight**
- set the **Placement**, **Flip**, and **Alignment** options as shown and click **OK**



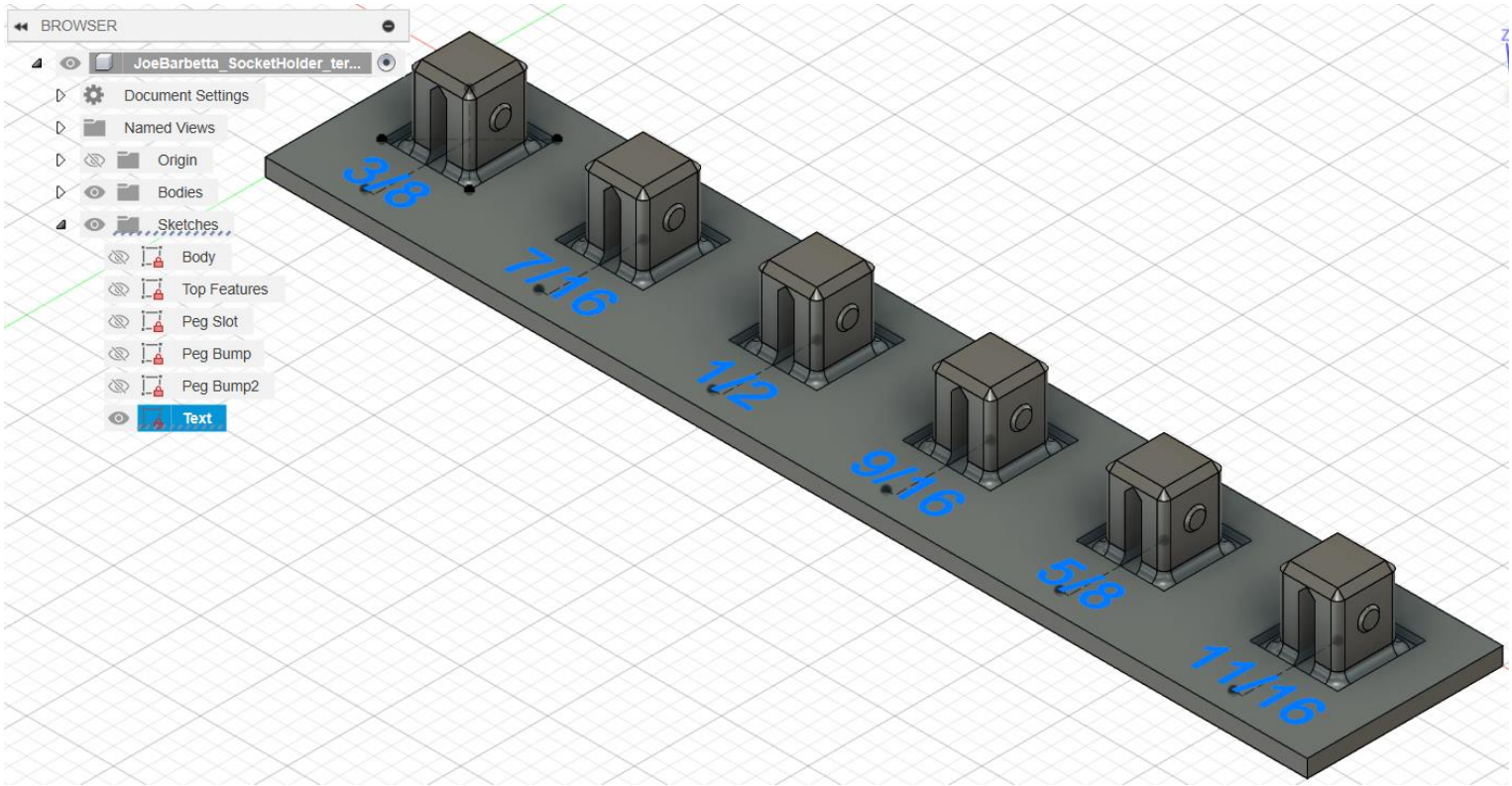
- pan over to the **2nd peg** and use the **Text** tool and peform the same operations.  Note that only the **Text** box and **Height** should need setting.  The other options should reflect the previous settings.
- click **OK**



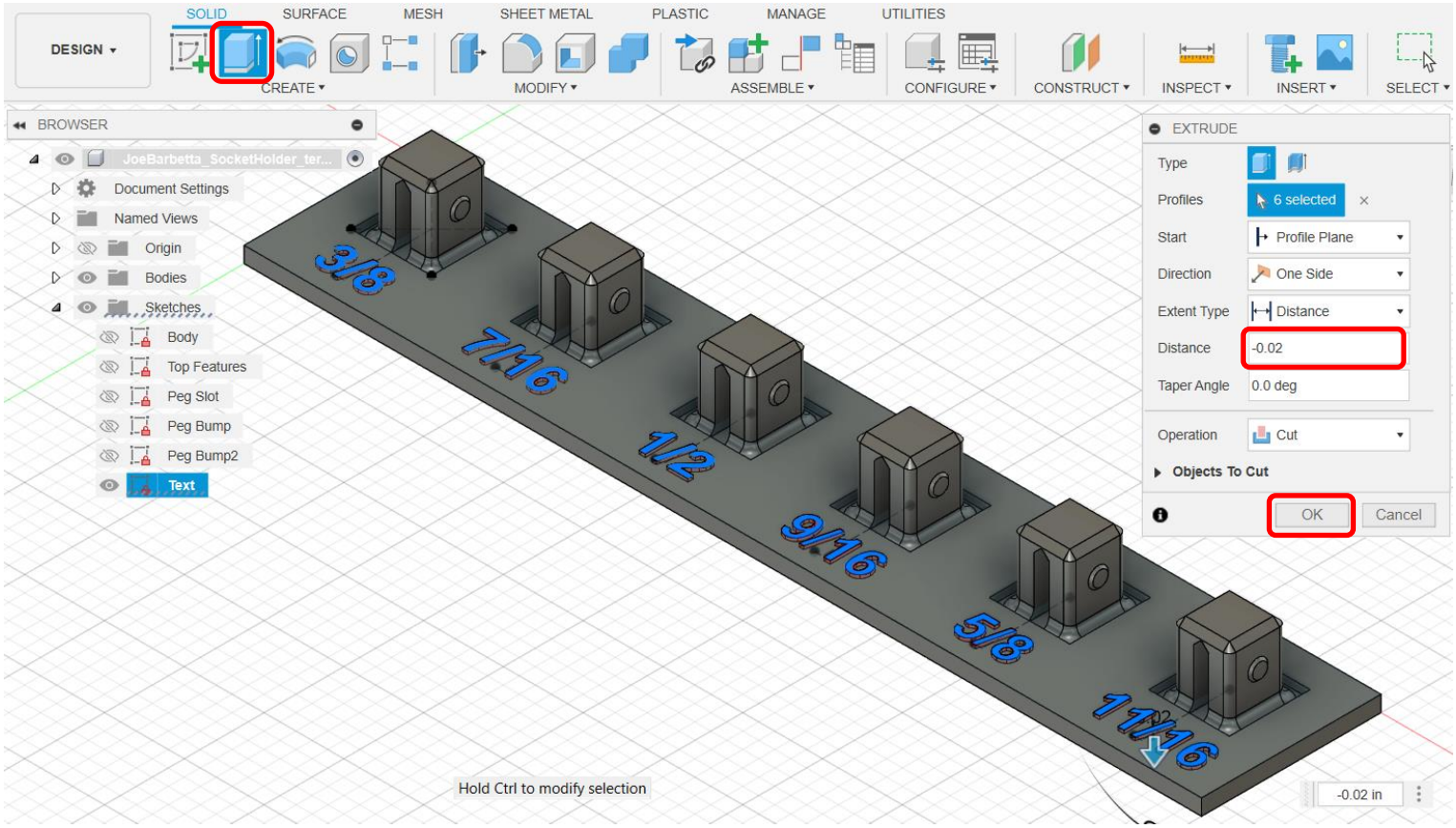

- when all 6 are done click **Finish Sketch**



- hold the **Shift key** and **click on each text value** to cause it to turn dark blue

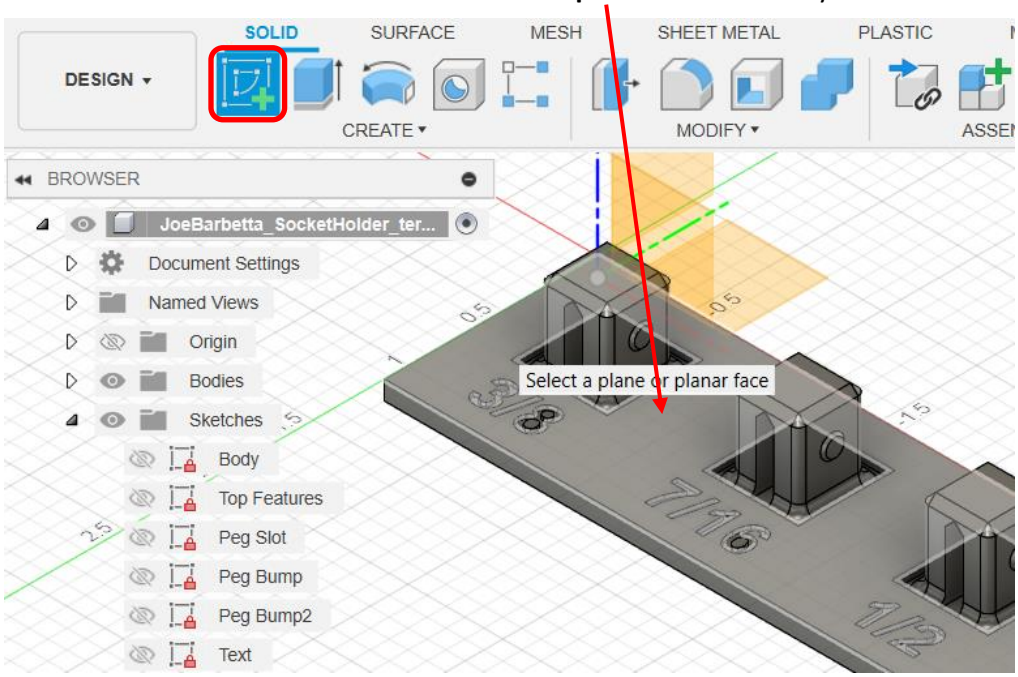



- select the **Extrude** tool
- for **Distance** enter **-0.02** (note the minus sign) and click **OK**

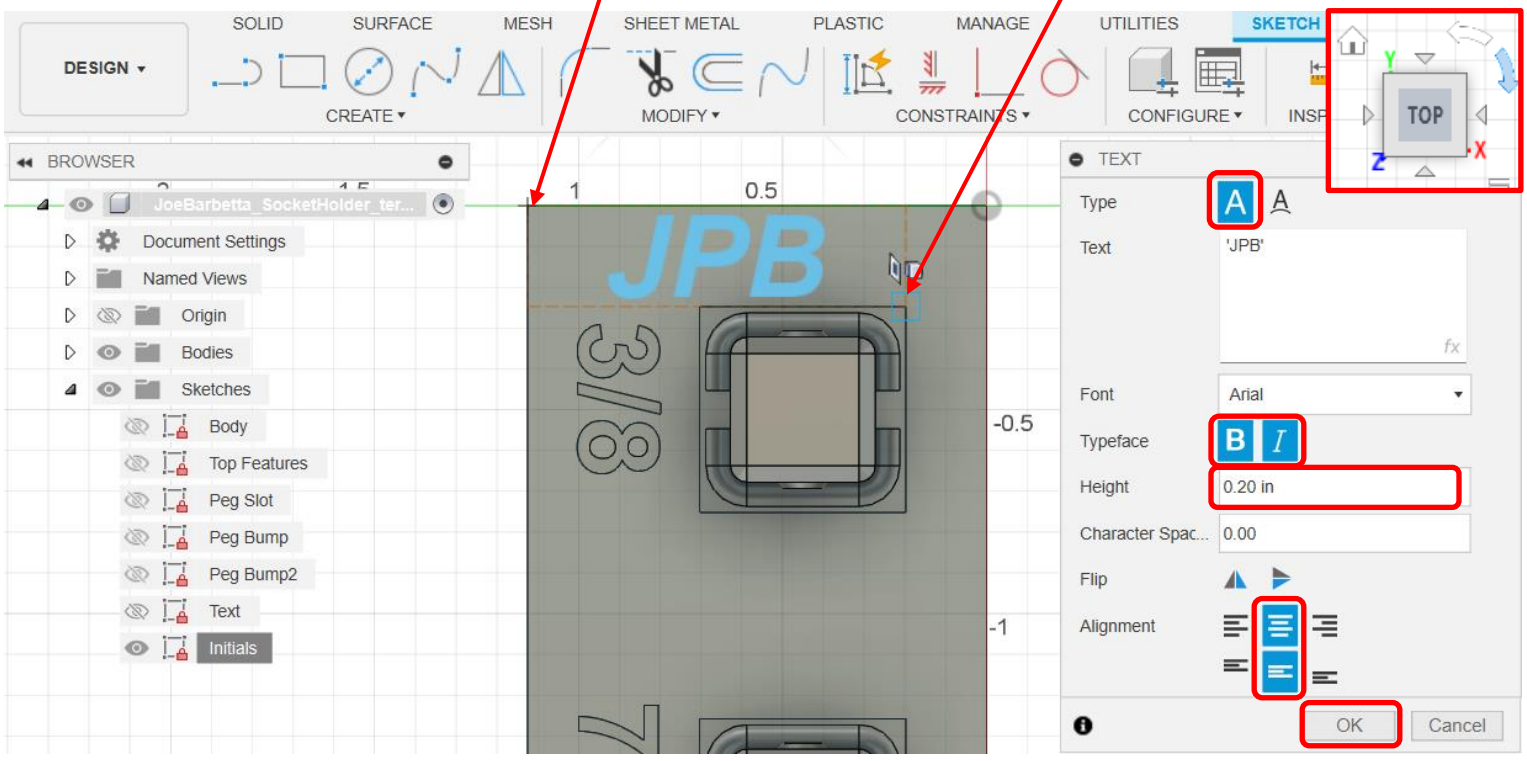


# Adding you intials

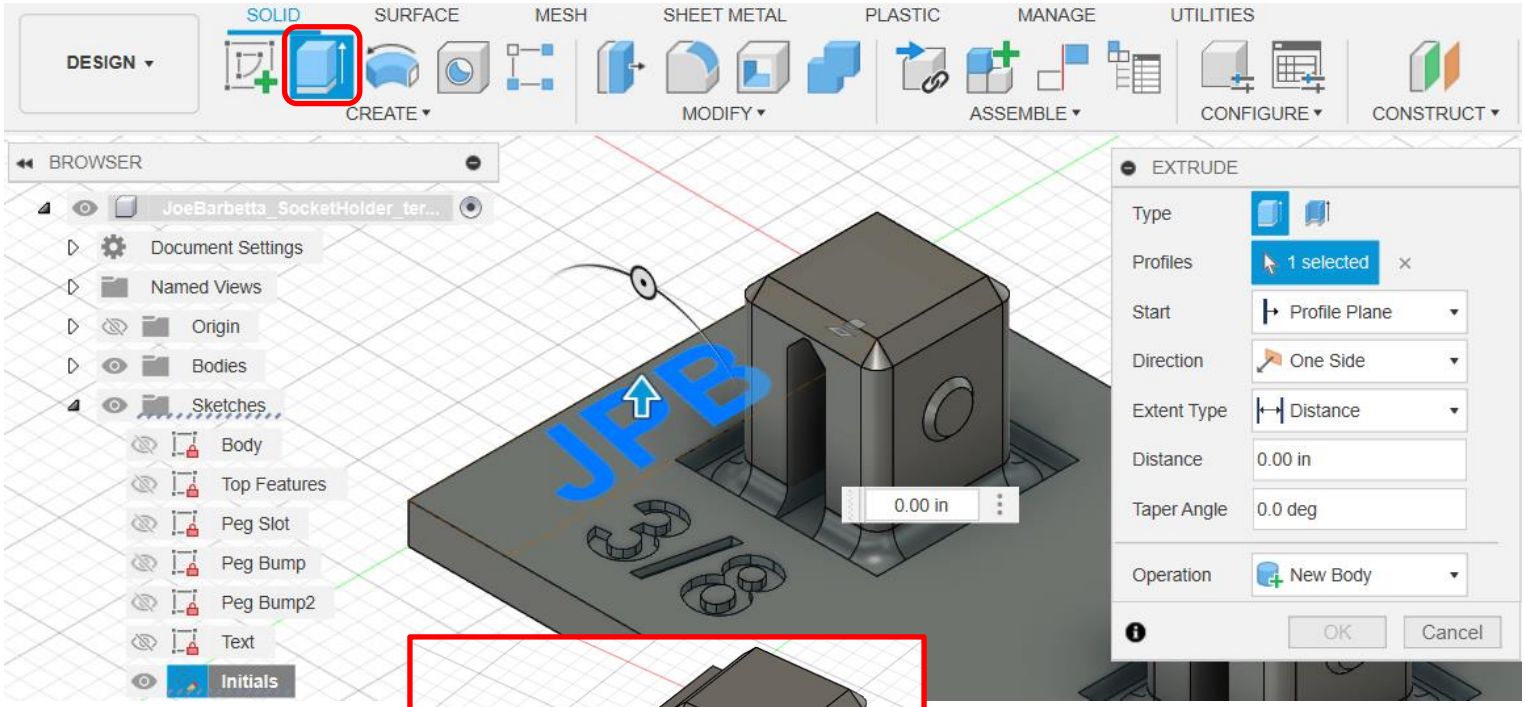
- select the **Create Skect** tool and click on the **top surface** of the body

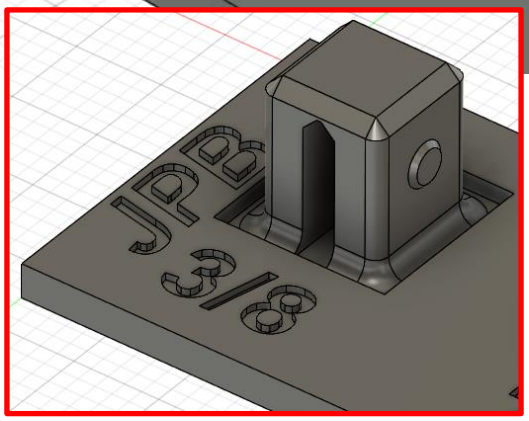



- if needed click on a **Curved arrow** at the **View Cube** and zoom to achieve the below view
- rename the Sketch to **Initials**
- from the **CREATE** menu select **Text** and click on the ***normal* Text** icon
- create the frame rectangle starting the the **top left corner of the body** and ending at the **top-right corner on the peg**.
- enter your **3 initials** with a **preceeding and ending single quote**.
- set the **TEXT options as shown** and click **OK** and **Finish Sketch**.



- select the **Extrude** tool, click on the **initials**, and enter **-0.02** (note the minus sign) for **Distance** and click **OK**

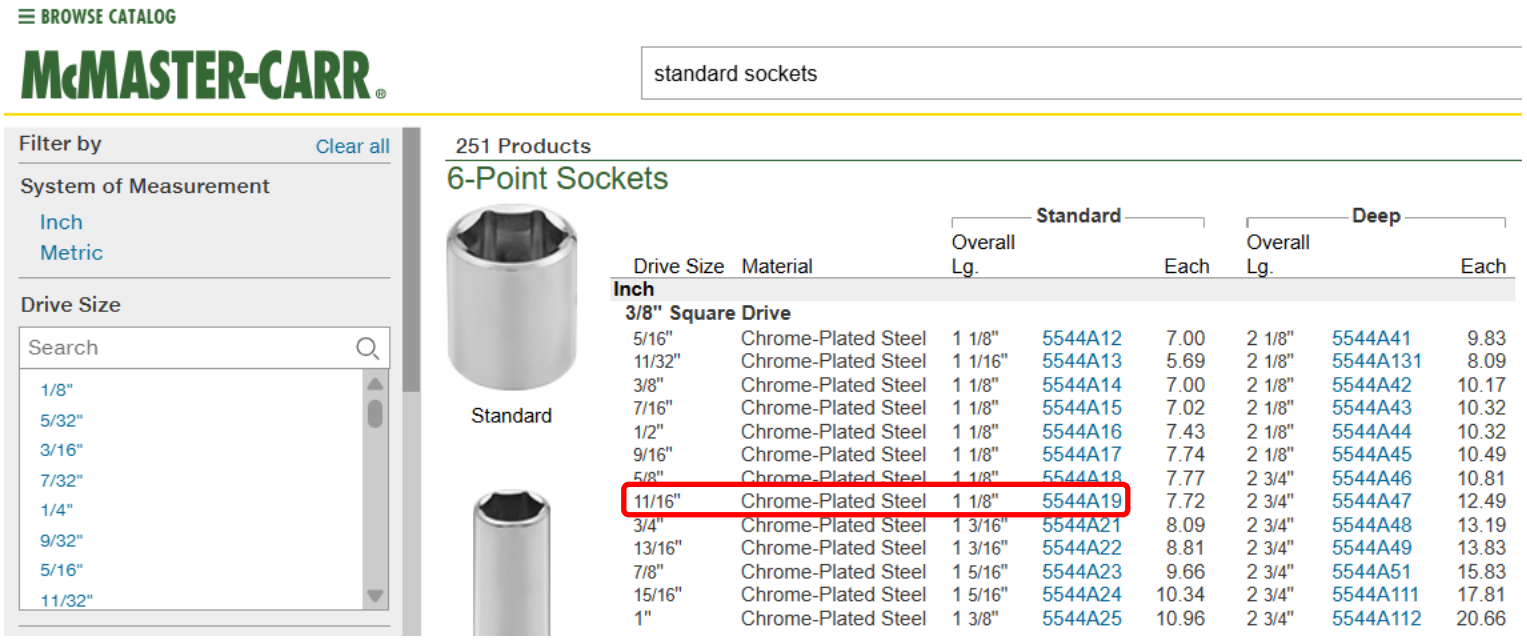


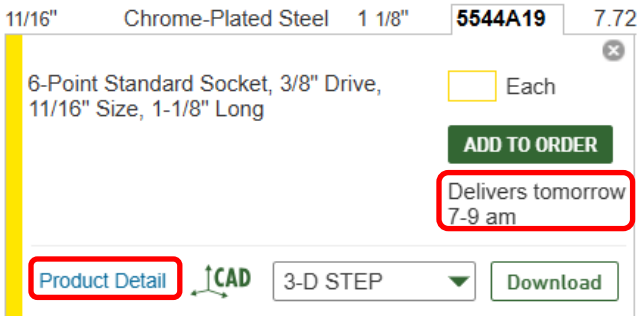

The result should look like this.

# We love McMaster-Carr

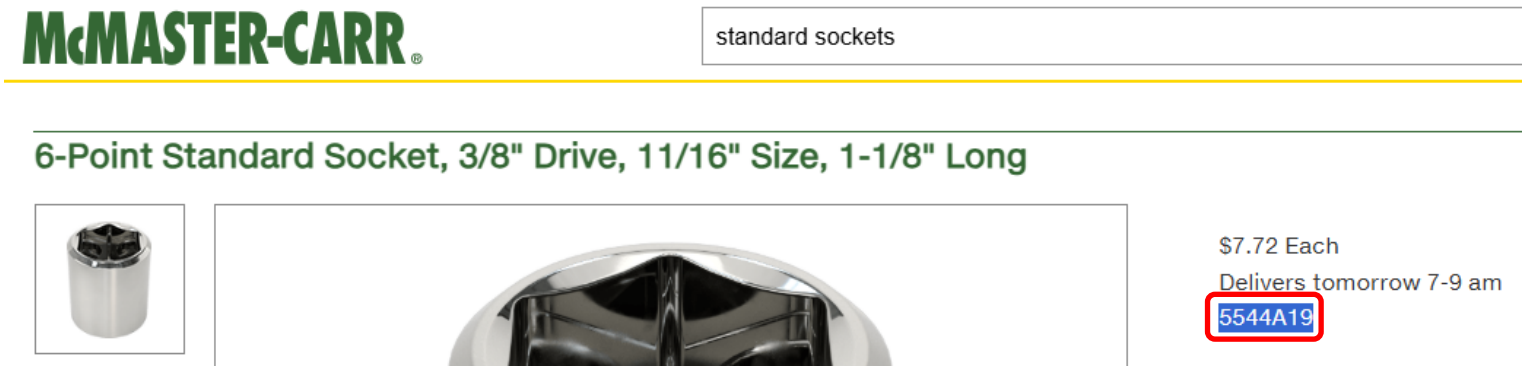
- go to **mcmaster.com** and search for **standard socket** and select **6-Point Sockets**.
- scroll down to the **3/8" Square Drive** section and select the largest socket you Socket Holder will accommodate, which for this example is 11/16"



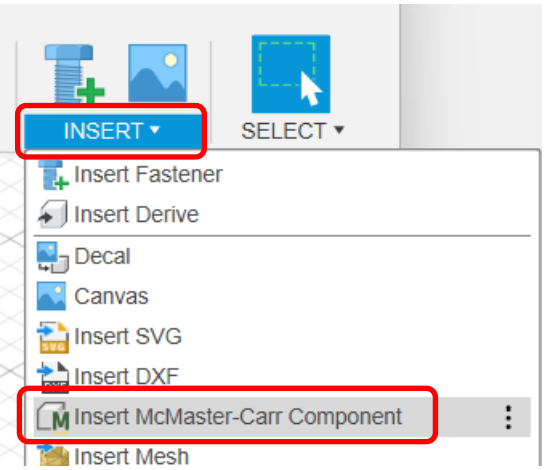

- click on its blue number and note how it can be delivered the same day if ordered in the morning or the next morning if ordering later in the day. **Yell "I love McMaster-Carr!"**
- click on **Product Detail**



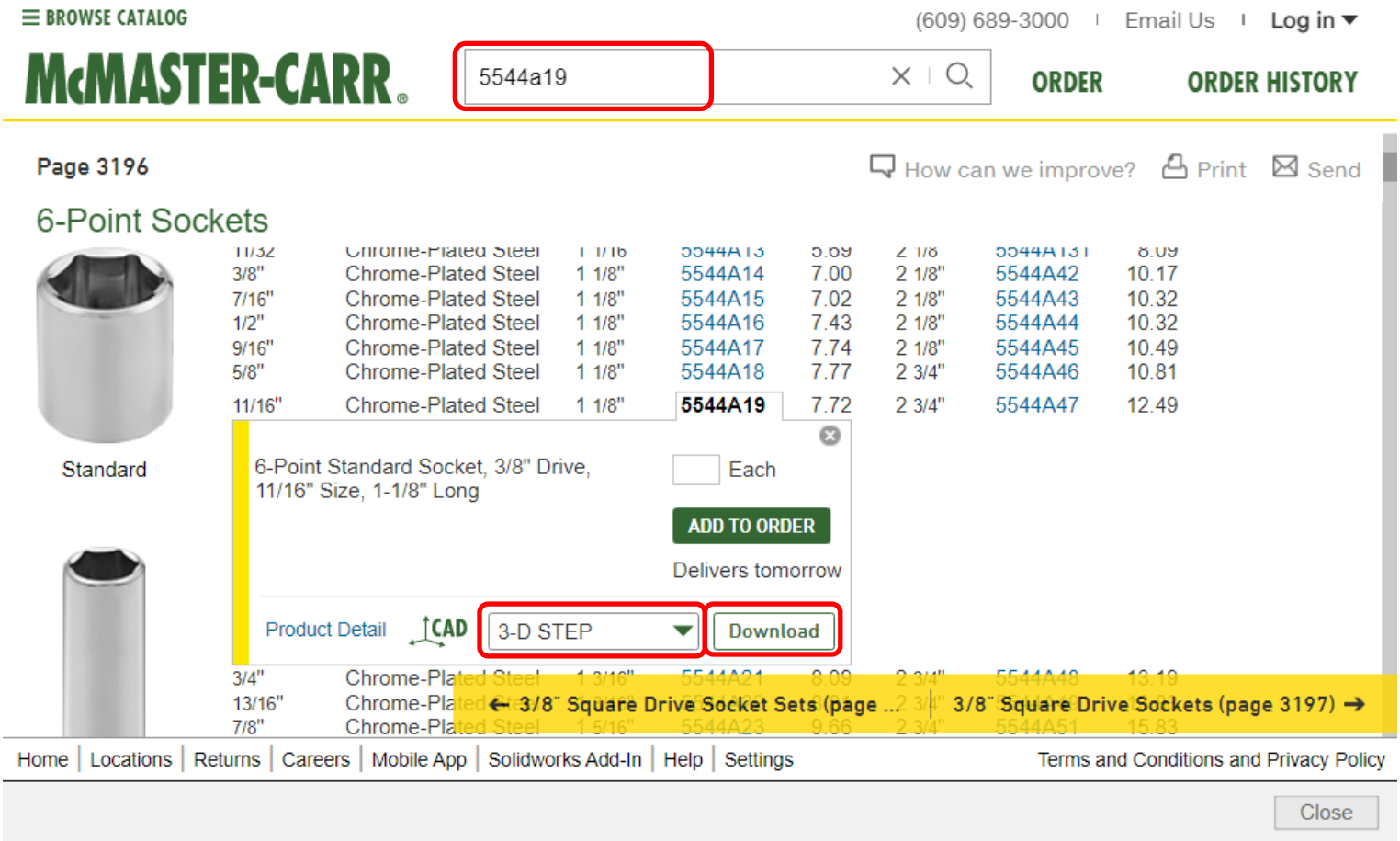

- select the **stock number**, which will highlight it and press **ctrl + c** to copy the number



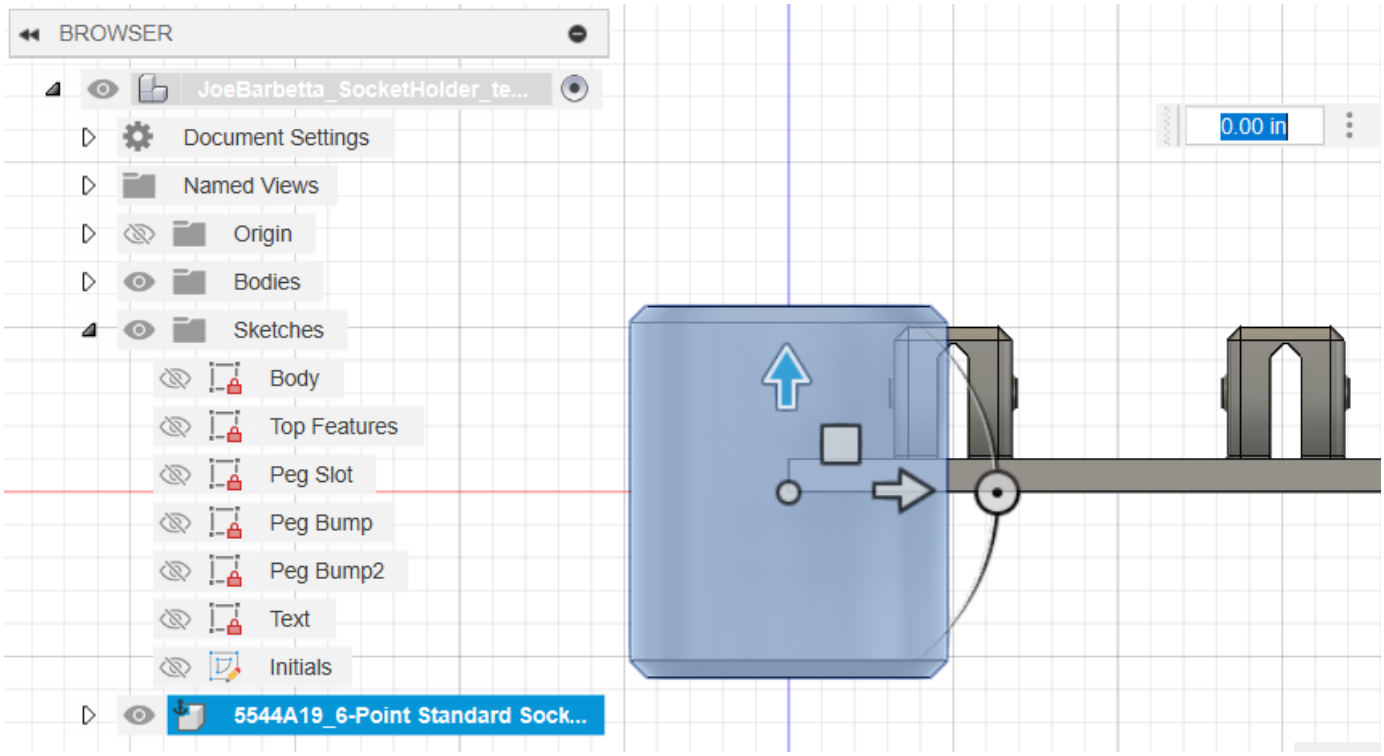
- from the top-right **INSERT** menu select **Insert McMaster-Carr Component**



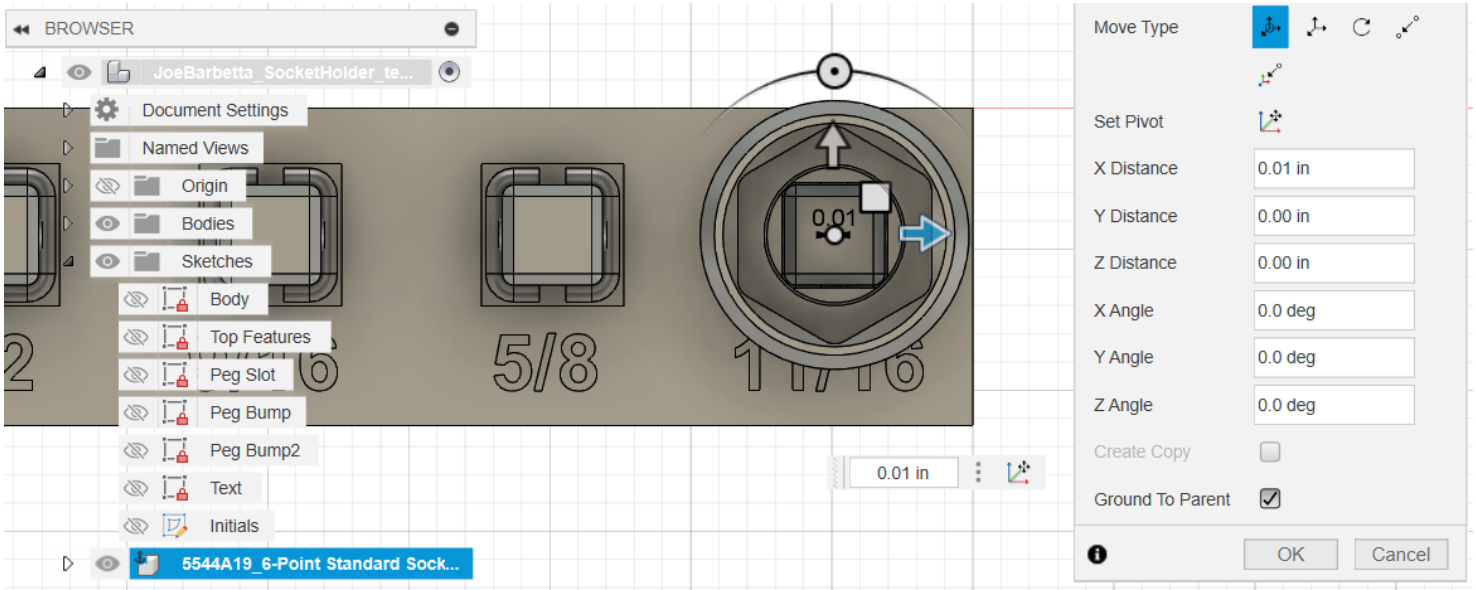

- **paste the number in the search box and press Enter**, which should open up an order window for the part
- select **3-D STEP** and click **Download**



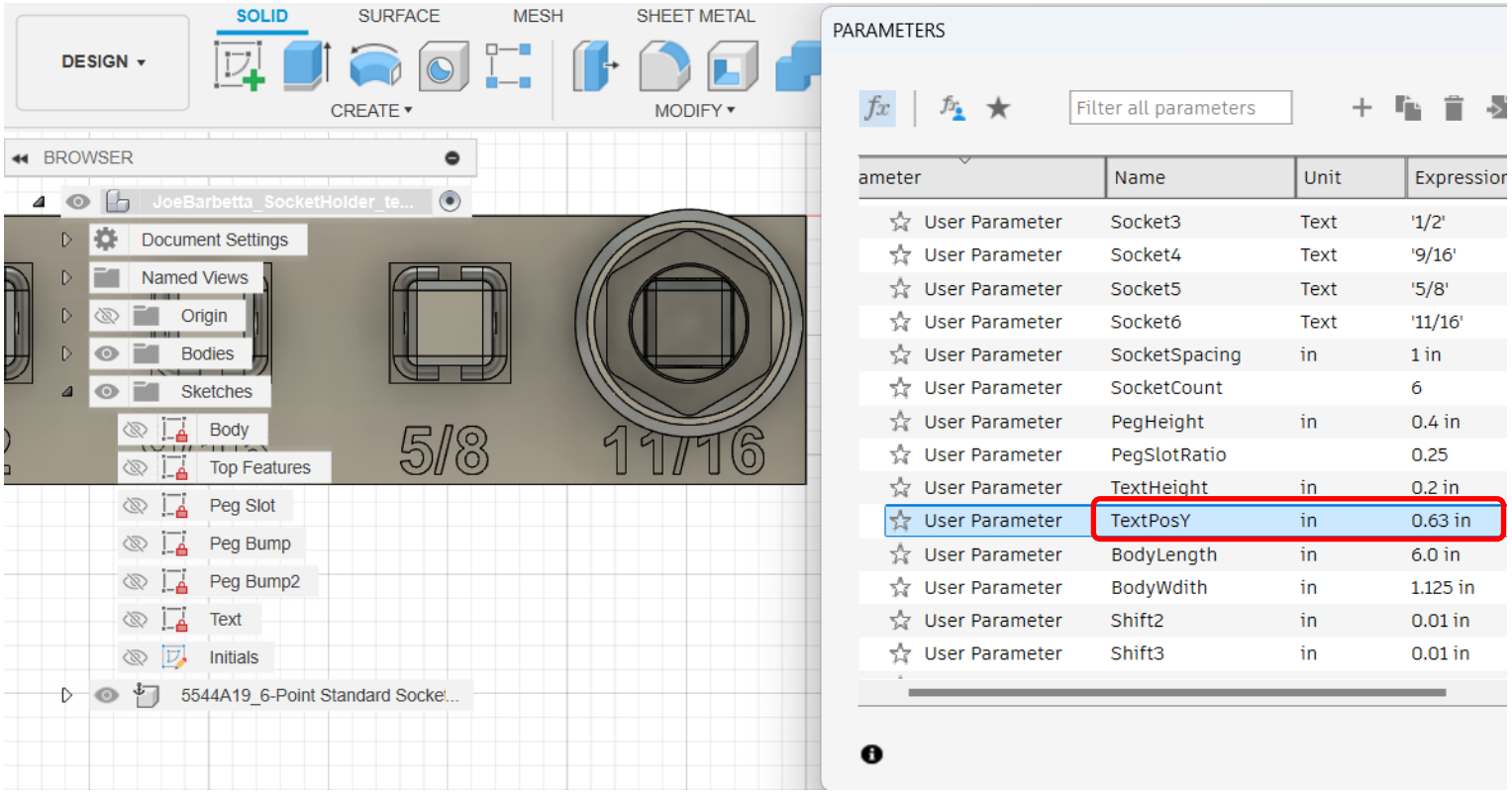
- change the view to a side view and move the socket just above the base of the socket holder




- change the view to a top view and move the socket centered over the corresponding peg. There is no need to use an alignment feature. Just eye it up. This is only to determine the text position.



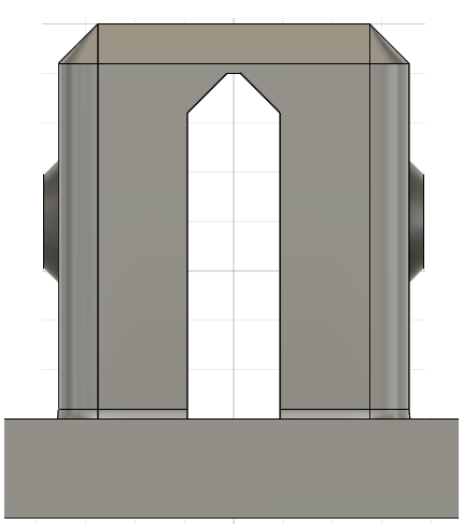
- adjust the **TextPosY** parameter to position the text by entering new values. Here it was found that changing the **TextPosY** value from its original **0.55** value to **0.63** value moved the text to a position where it could be read. Note that there should be some space between the text and the bottom edge. If more space is desired for the text, the **BodyWidth** can be increased as well.

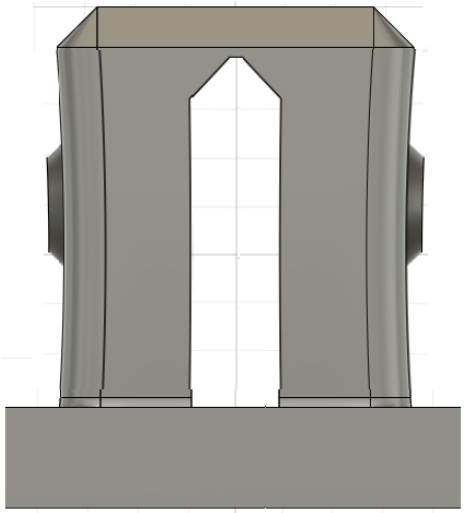


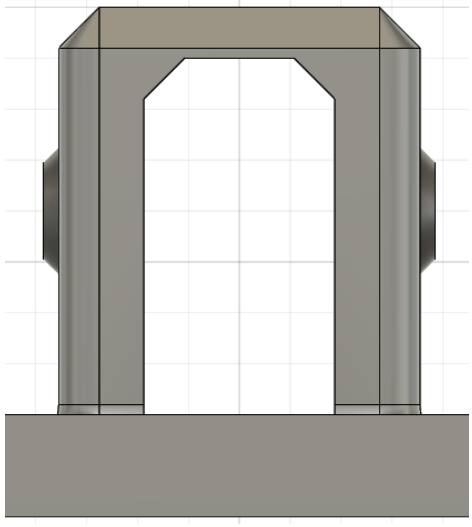
# Adjusting for fit

As illustrated on the right, when a socket is placed over a peg the side force on the bumps should cause a lateral deformation of the two “walls” of the peg. The flex of the walls will be influenced by their thickness and it is the **PegSlotRatio** parameter that controls the thickness. It defines the ratio of the distance between the two walls to that of the width of the peg. We set it to 0.25 initially, but testing found that thinner walls allowed a flex that allowed easier atachment and release of sockets.

- from the MODIFY menu select Change Paremeters and change the **PegSlotRatio** from **0.25** to **0.50**. This should result in pegs as shown on the right. If it is increased too much the walls may be too thin and they may break. Note that this change affects all of the pegs, which demonstrates the usefulness of parameters.



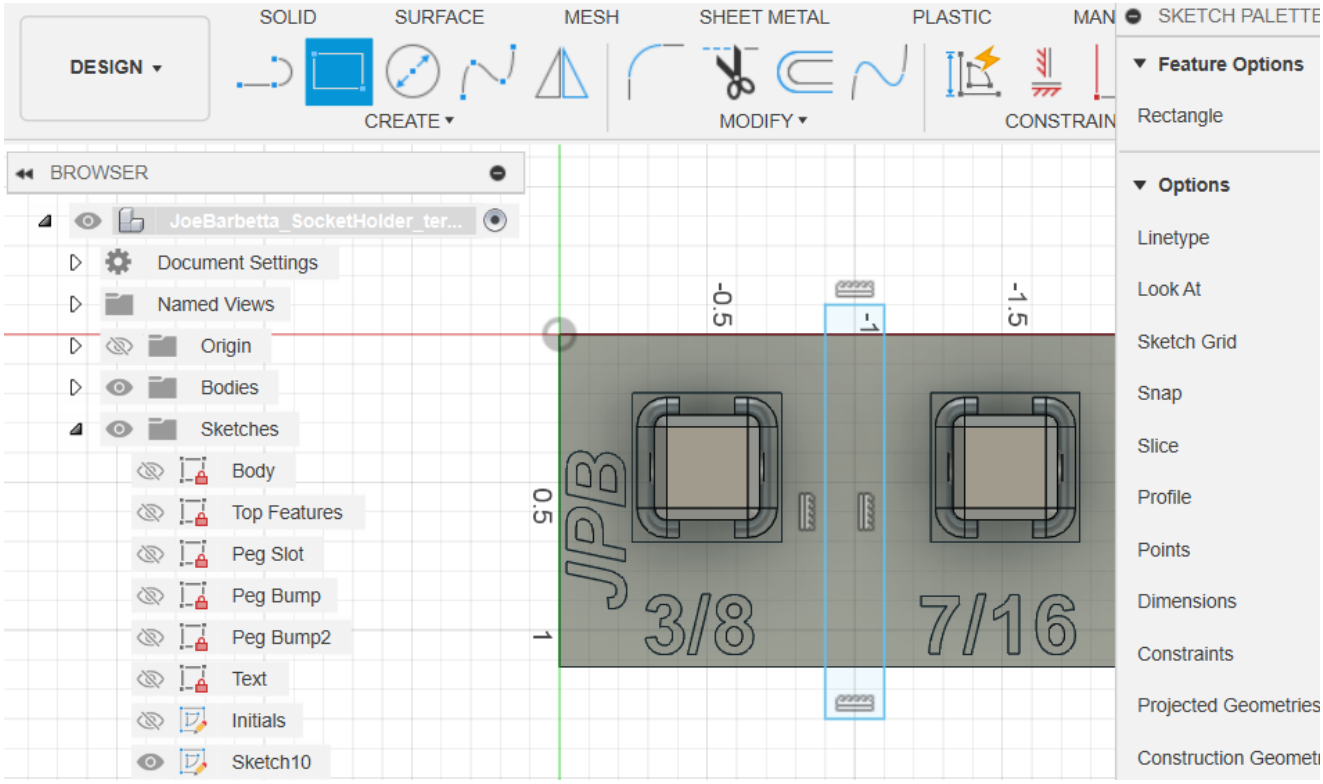




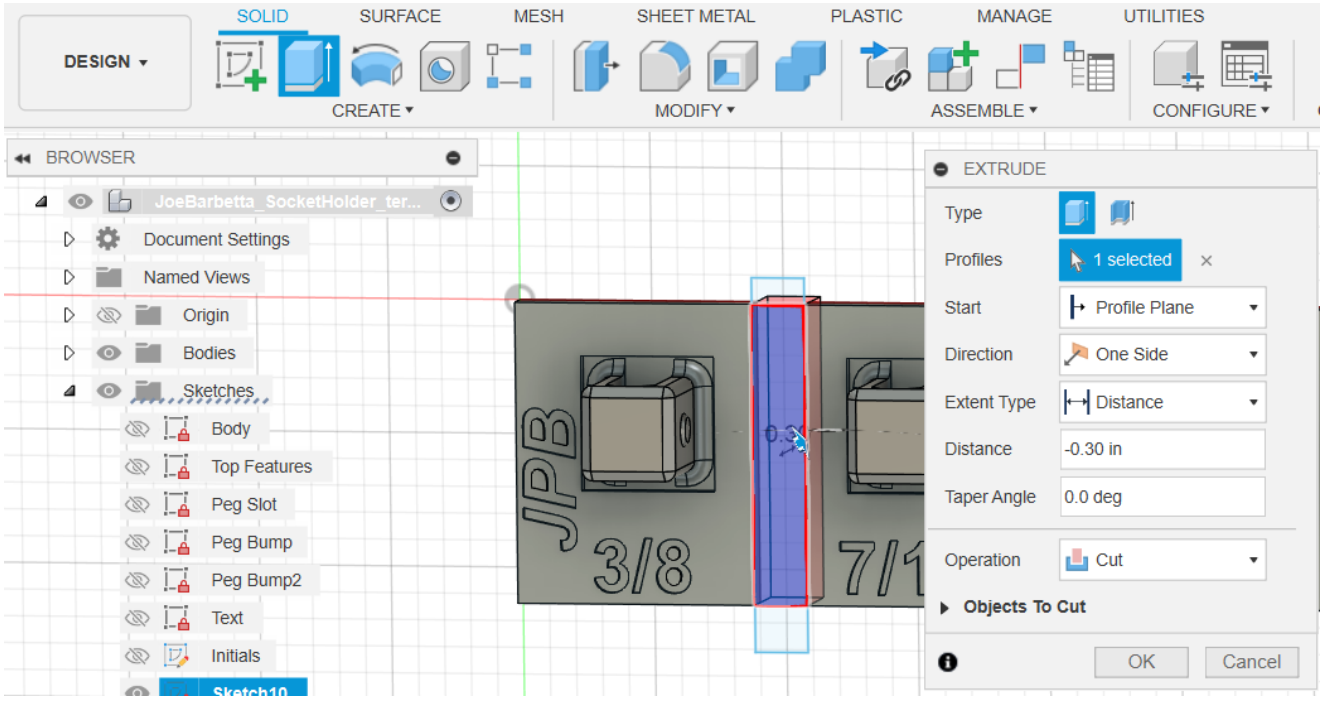
# Performing a Test Print

It would be a waste of time and plastic to print the entire socket holder and then finding that the sockets are too loose or too tight.  Whenever there is a critical feature, such as a hole or element that needs to have an adequate fit with a component, the design should be cut down to print only the features that need testing.

**- save the present design**.  This will allow one to revert back to the design before it was cut up for testing.
- create a skecth on the top surface.  There is no need to rename it because it is temporary.  Click **Finish Sketch**.

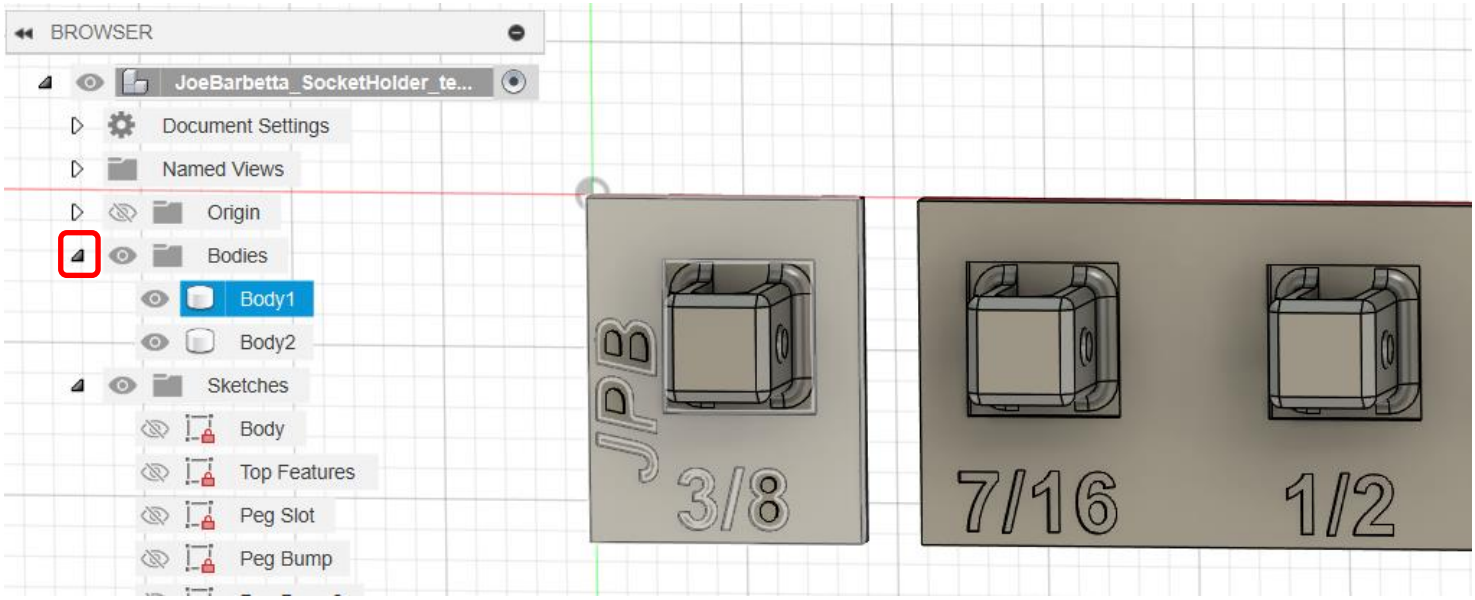



- Extrude the rectangle towards the rear to cut through the body and click OK



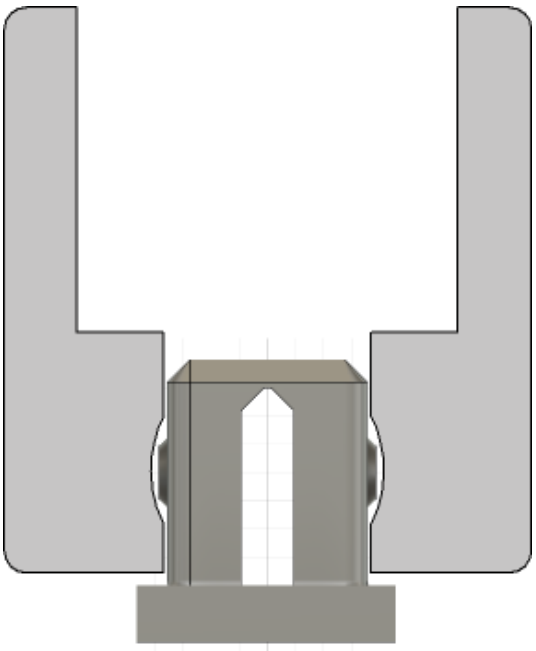
- open up the Bodies folder and hover on the isolated body to see which body in the Browser gets highlighted. In this case it is Body1. **Right-click on Body1** (or whichever becomes highlighted) and select **Save As Mesh**. Give it the design name followed by test.

- close the design **without saving** and **reopen the design previously saved**. Alternatively, one can use the undo button to reverse the cut that was made. Sometimes, there are many changes made for a test print and then it will be easier to revert back to the design saved before the changes to prepare a test print.




The next step would be to open the .stl in a Slicer program such as Ultimaker Cura and print the item. Different sockets can be placed on the peg to determine adjustments that should be made.

The parameter tht can adjust the fit is **DriveSizeAdj**, which was originally set as **-0.020**. It is negative to account for the bumps that extend out by **0.015**. Because there are two bumps the peg width maximum at the bumps is increased by **0.030**.



To the left is a cross-section of a 11/16” socket with a 3/8” drive. The *drive* is the width of the square hole that attaches to a socket. As mentioned previously 1/4” and 1/2" drives are also common.
Most, but not all, sockets have internal indentations. The ratchets drive square will have small spring loaded balls that fit in these indentations to help secure the socket to the ratchet.
In the case of our holder, the small protrusions will fit in these indentations. However, if the fit is too tight it will be difficult to remove the socket. The test print will verify the fit and then a decision can be made on how to change the DriveSizeAdj parameter. It could make sense to perform a 2nd or even 3rd test print to ensure a good fit is achieved before performing the final print.