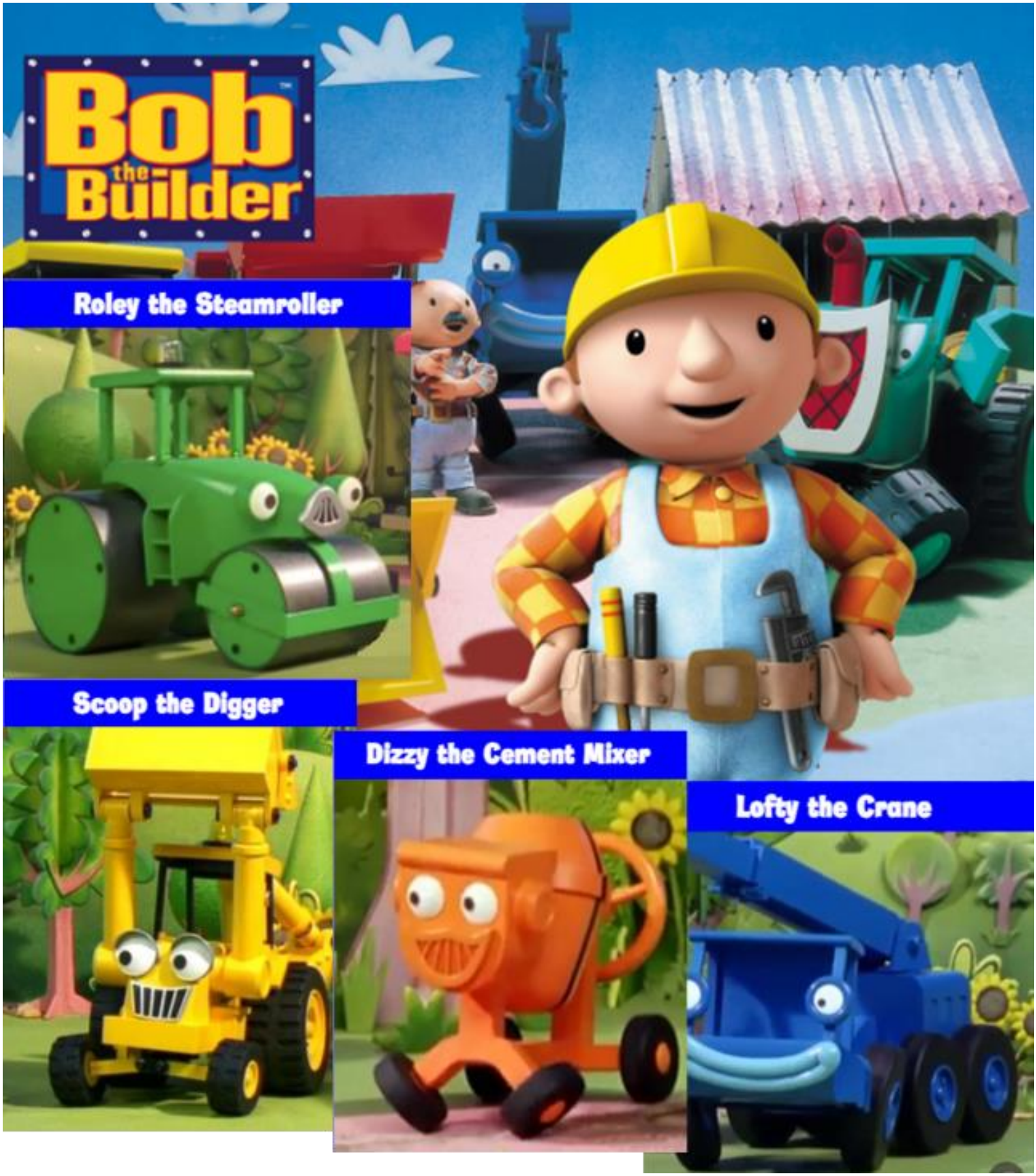
Mr. Barbetta's *"I don't care about 3D Printing. I just want the art credit."* class

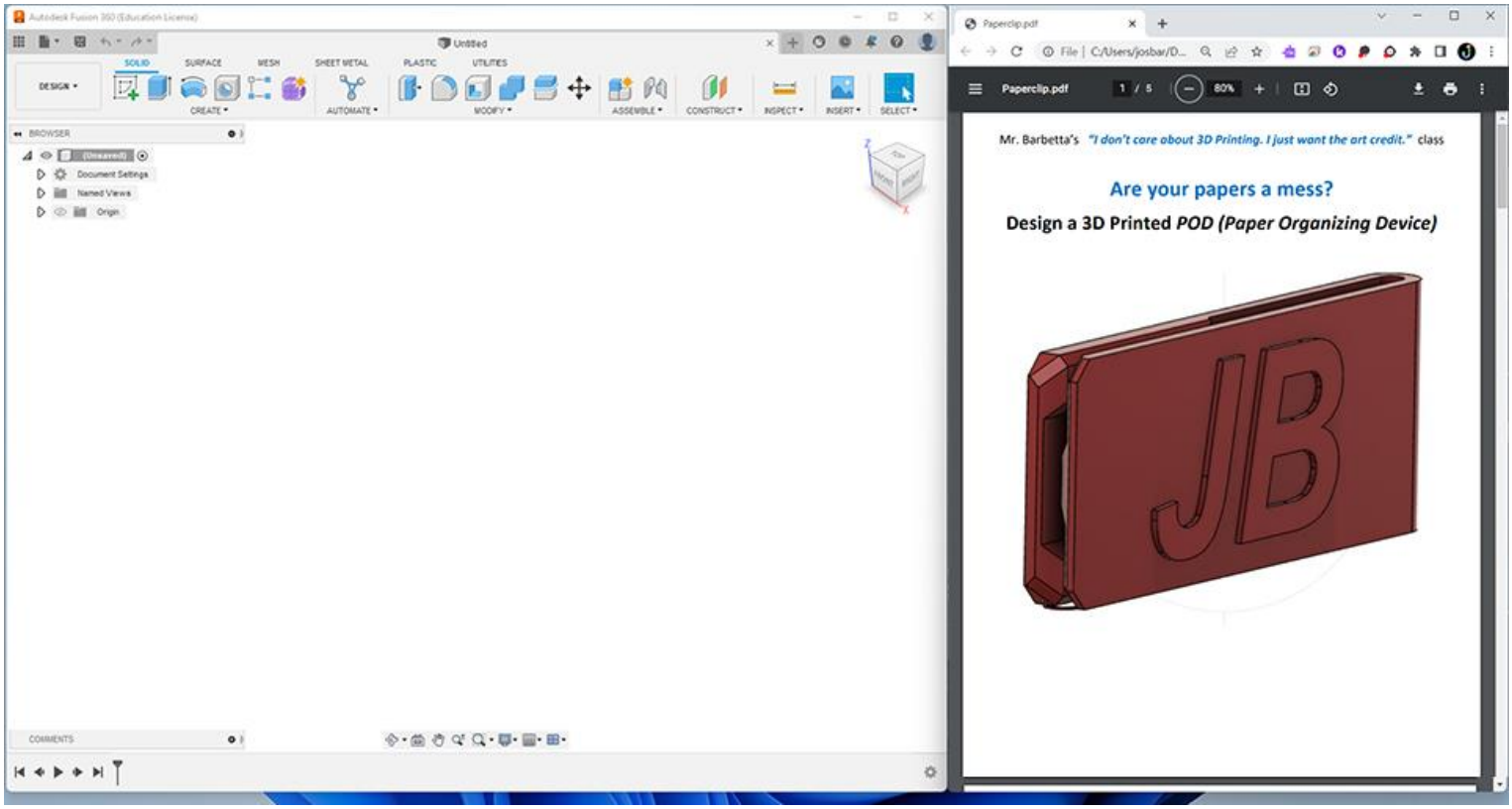
3D Printed Door Hinge Jig

Today's Lesson is Sponsored by Bob the Builder

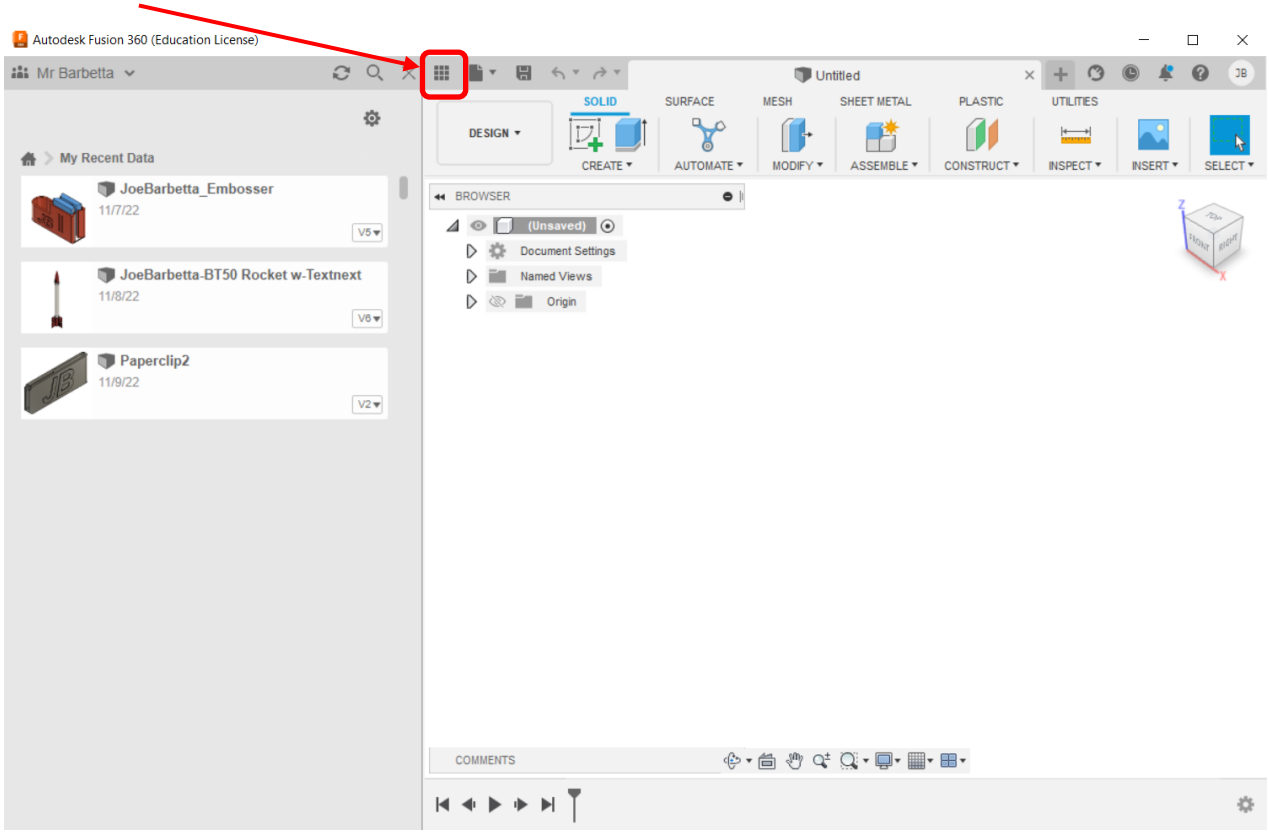


Using This Document

The best way to follow this document is to **reduce the width of the Fusion 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%.



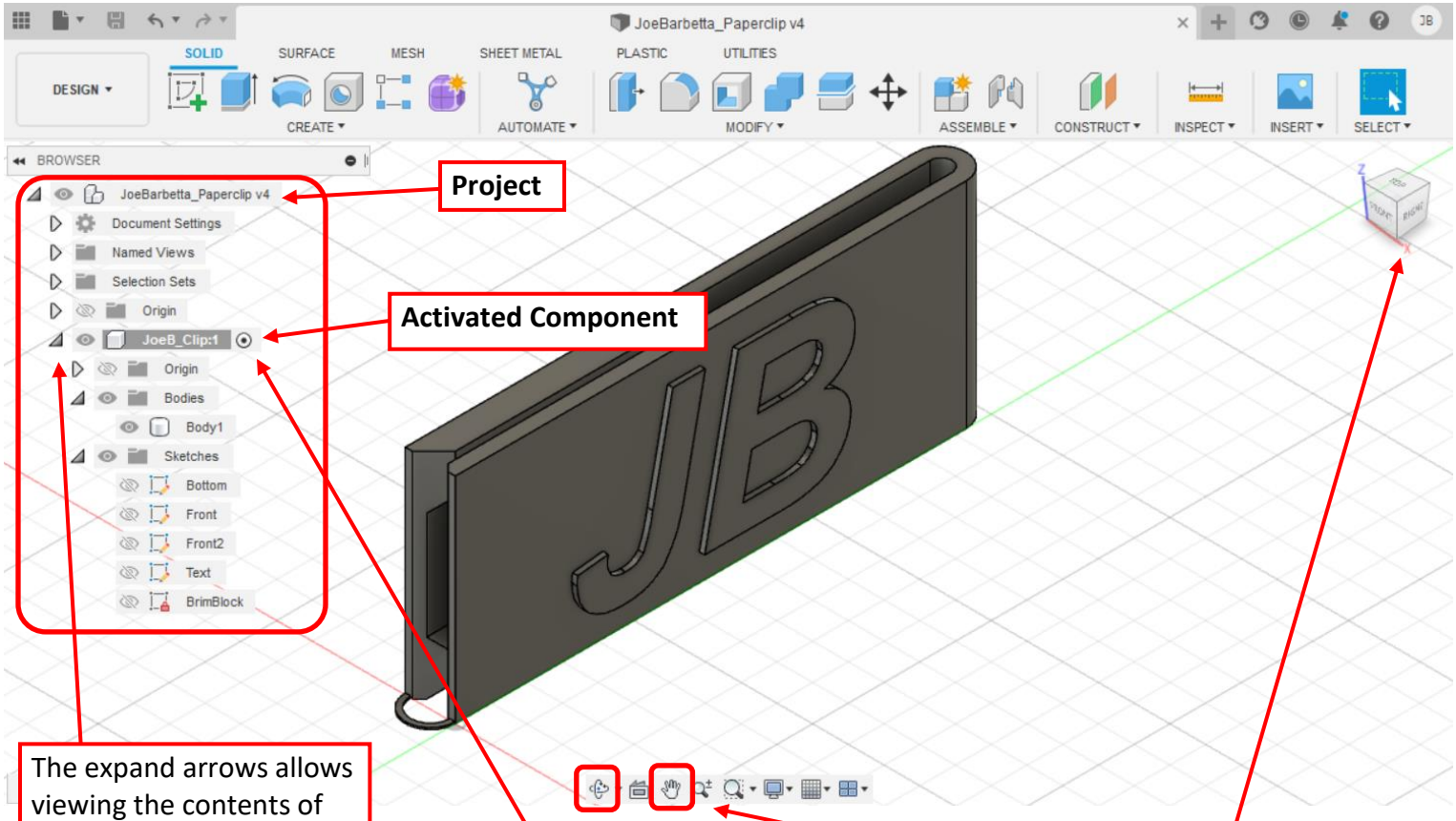
The Fusion window will not allow its width to be reduced much so for smaller computer screens a trick is to click on the **Data Panel icon** and then move the window to the left with the Data Panel off the screen.



The Fusion User Interface

Review this page and the following for information on the user interface.
The actual instructions begin on page 7 (Starting a Design in Fusion).

This is what the final paperclip will look like in Fusion See comments after this screen shot.



The expand arrows allows viewing the contents of *Components* and *Bodies* and *Sketches* folders.

Note the **circled dot** of the *Activated Component*. When one wants to work on a Component, it must be Activated by right-clicking on the Component Name and selecting **Activate** at the top of the pop-up menu.
When a project is reopened, the Project at the top of the Browser is Activated and a Component may have to be Activated again.

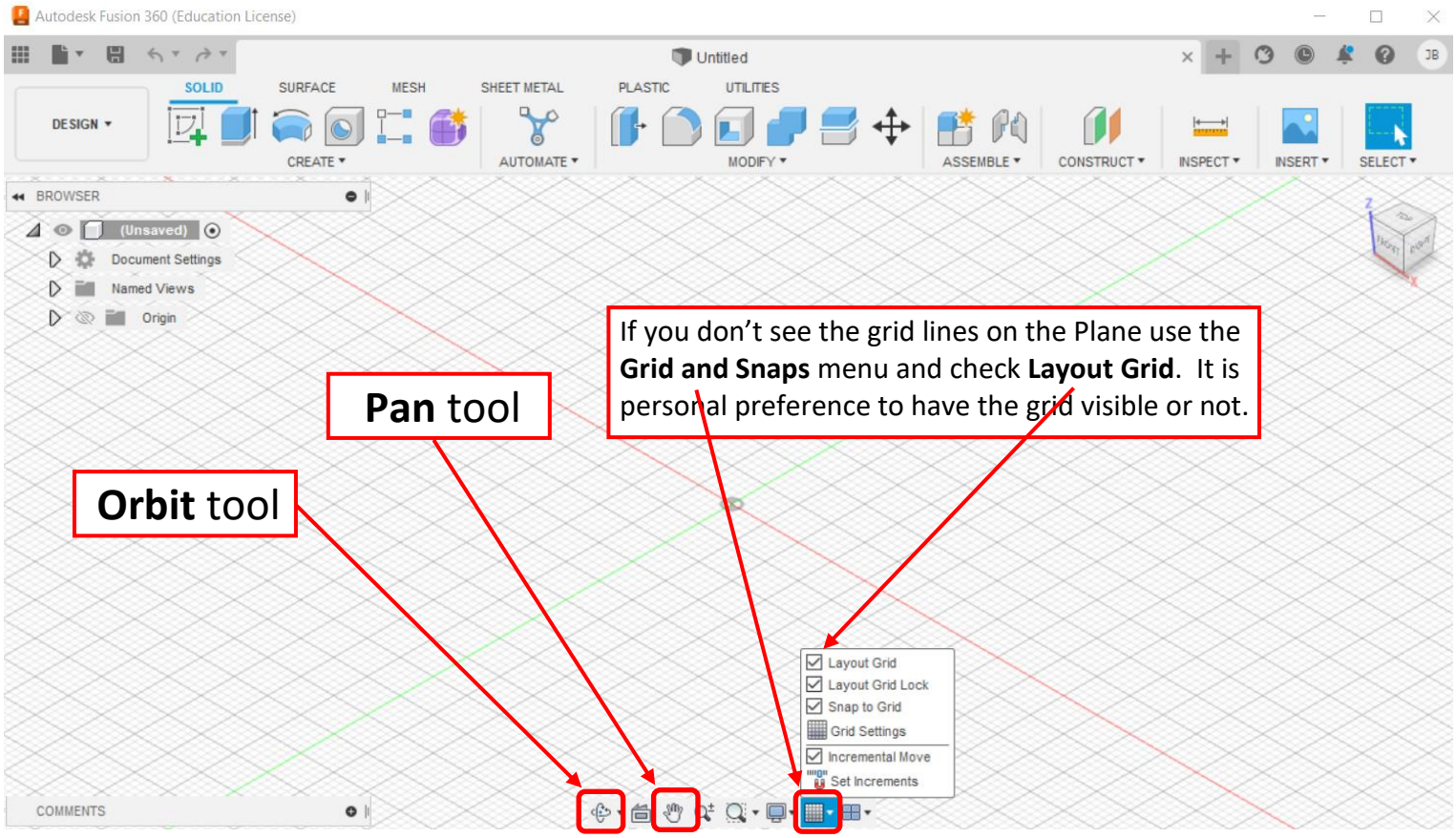
The **Orbit** and **Pan** (hand icon) tools are commonly used to adjust the view of the design. After using one the **Esc key** is needed to exit the operation.
The top right **View Cube** is another method to rotate the view of the design.

The **mouse wheel** can be used for zooming in and out. The **mouse wheel** can alsoe be pressed to pan.

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.

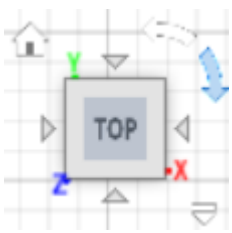


Here is a close-up of the View Cube at the top right of the window.

- 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



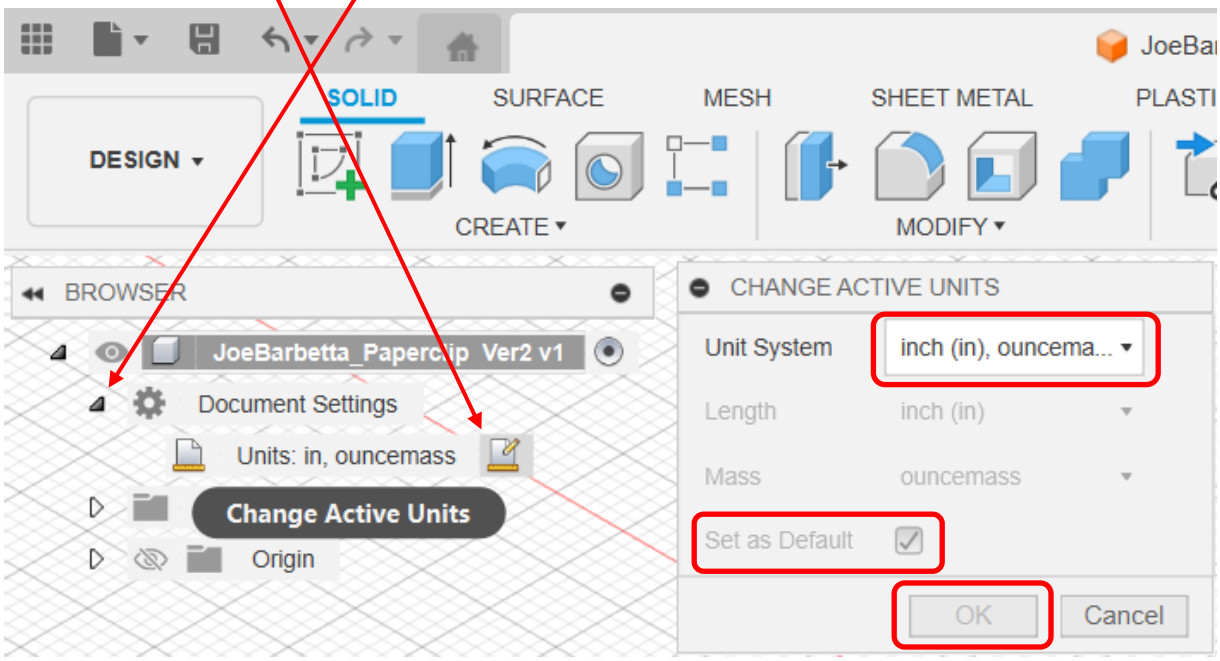
Starting a Design in Fusion

- 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 **_DoorHingeJig** e.g. **JoeBarbetta_ DoorHingeJig** (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 can be opened using "**Open…**" from the top **File** icon.

As you work you may want to occasionally save your work in case Fusion crashes.

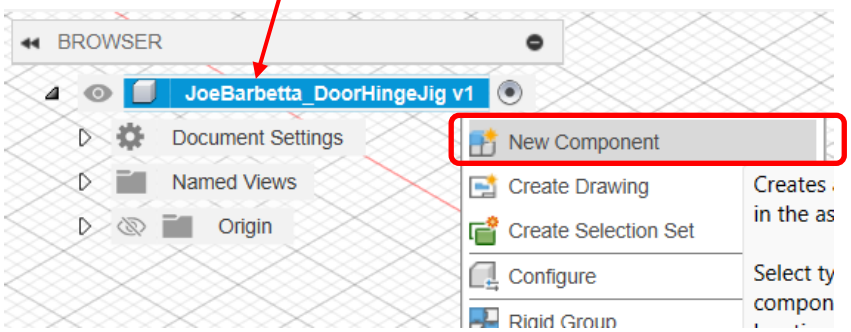
- in the left **"BROWSER"** click the **arrow next to Document Settings**
- click on the **edit icon** that appears to the left when you hover over **Units**
- ensure **Active Units** are set to **Units: in, ouncemass** and click **OK**. You can also enable **Set as Default** if it is not grayed out.

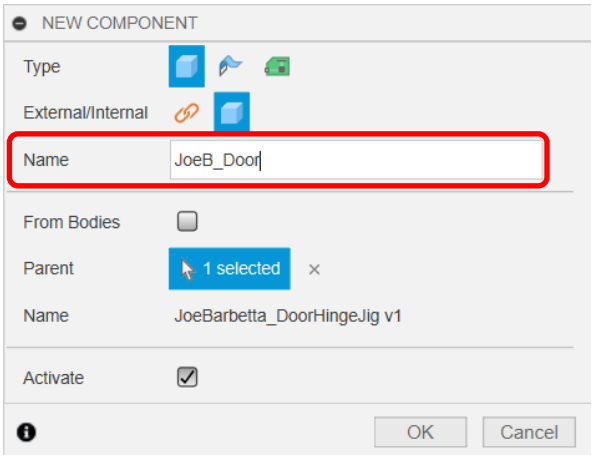


Creating a New Component

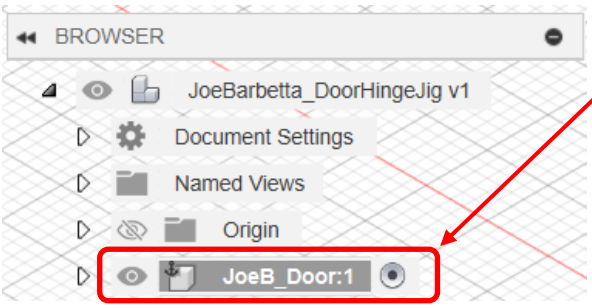
To keep a project organized it is recommended to create a new Component for each part.

- right-click on the **Project Name** at the top of the BROWSER and select **New Component**
- set **Name** as **Your first name and last name initial followed by _Door** e.g. **JoeB_Door** and click **OK**



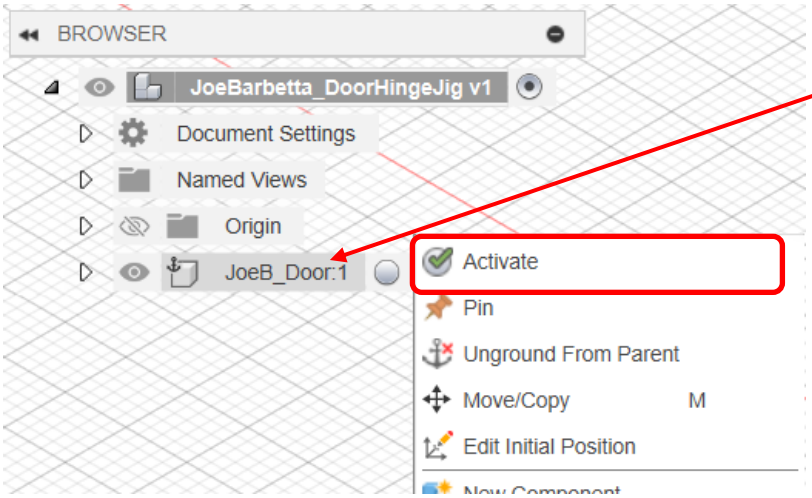


The new *Component* should show in the *Browser*.
Note that the area around the name is darkened and the circle with the dot. This indicates that it is the ***Active Component***.



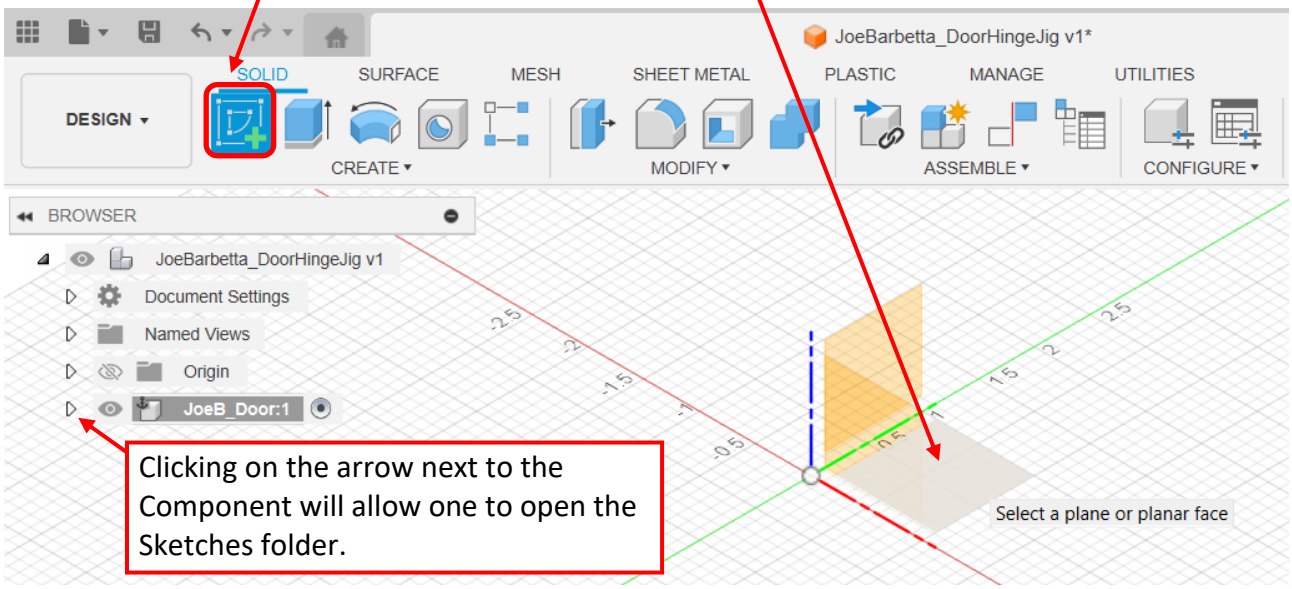
One reason to hate Fusion!

When a project is saved and then reopened, the Component that was Active is no longer Active.
If one wants to continue working on a Component, it must be reactivated by **right-clicking on the Component Name** and selecting **Activate**. One can also click on the circle to the right of the Component name.

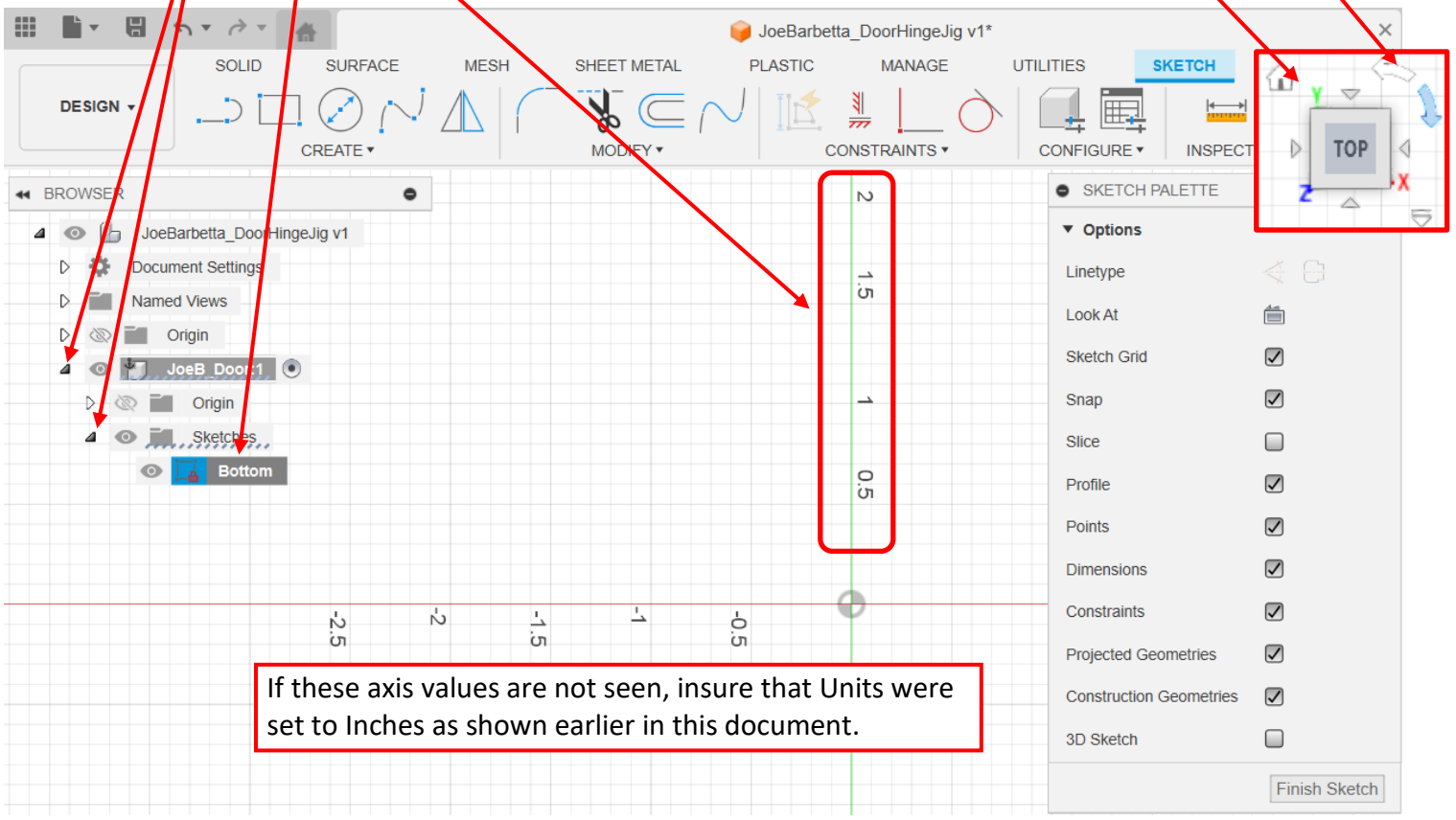


Creating the Bottom Sketch for the Door

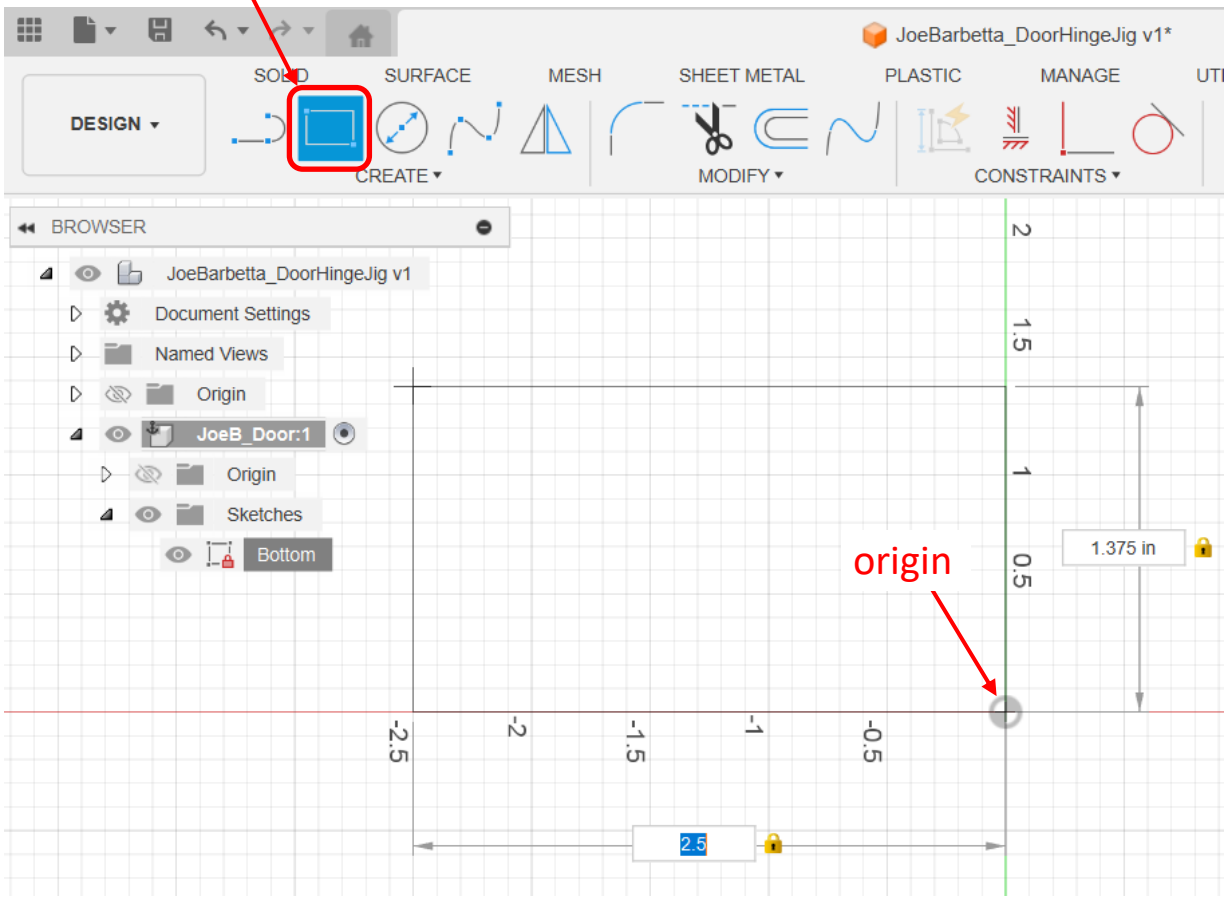
- select the top **Create Sketch** tool and click on the **bottom rhombus** to select the X-Y Plane.
If a tool can't be found, one can always look in the **CREATE** and **MODIFY** menus for it.



- **zoom in and pan** (hand icon) to achieve a view similar to below. You may have to use the **curved arrow** to rotate the view. If the curved arrows don't show, click on the center of the View Cube.
- note the **values on the green axis** as an indication of the desired view and the position of the **View Cube**
- click on the **arrows** next to the **Component** name and **Sketches** folder.
- right-click on the **Sketch Name** and select **Rename** and name it **Bottom**

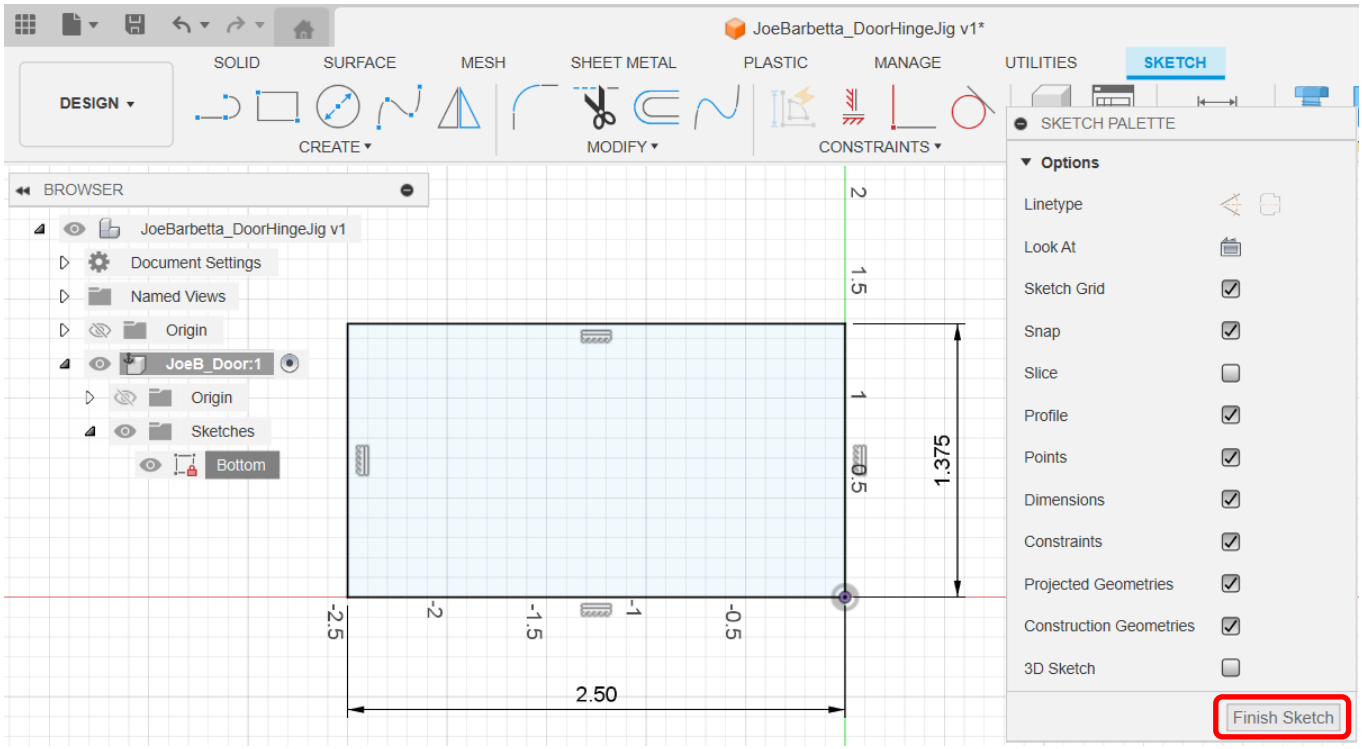


- select the **Rectangle** tool
- click on the origin (where the axes cross) and start dragging the other corner up and to the left
- type **1.375**, press the **Tab key**, type **2.5**, and then press the **Enter key**

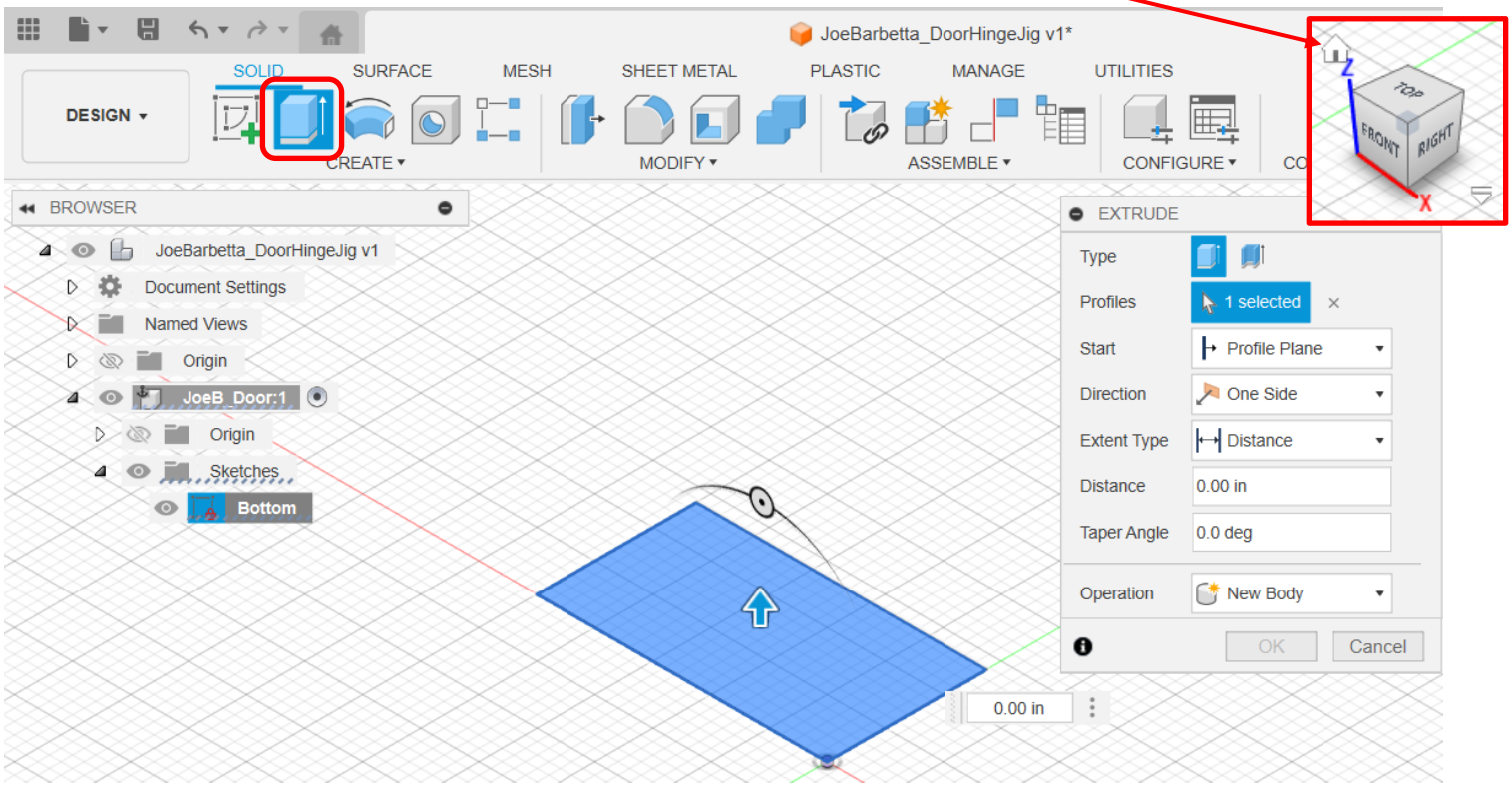


The rectangle should look like that below. Note that the dimensions lines show the height and width of the rectangle.

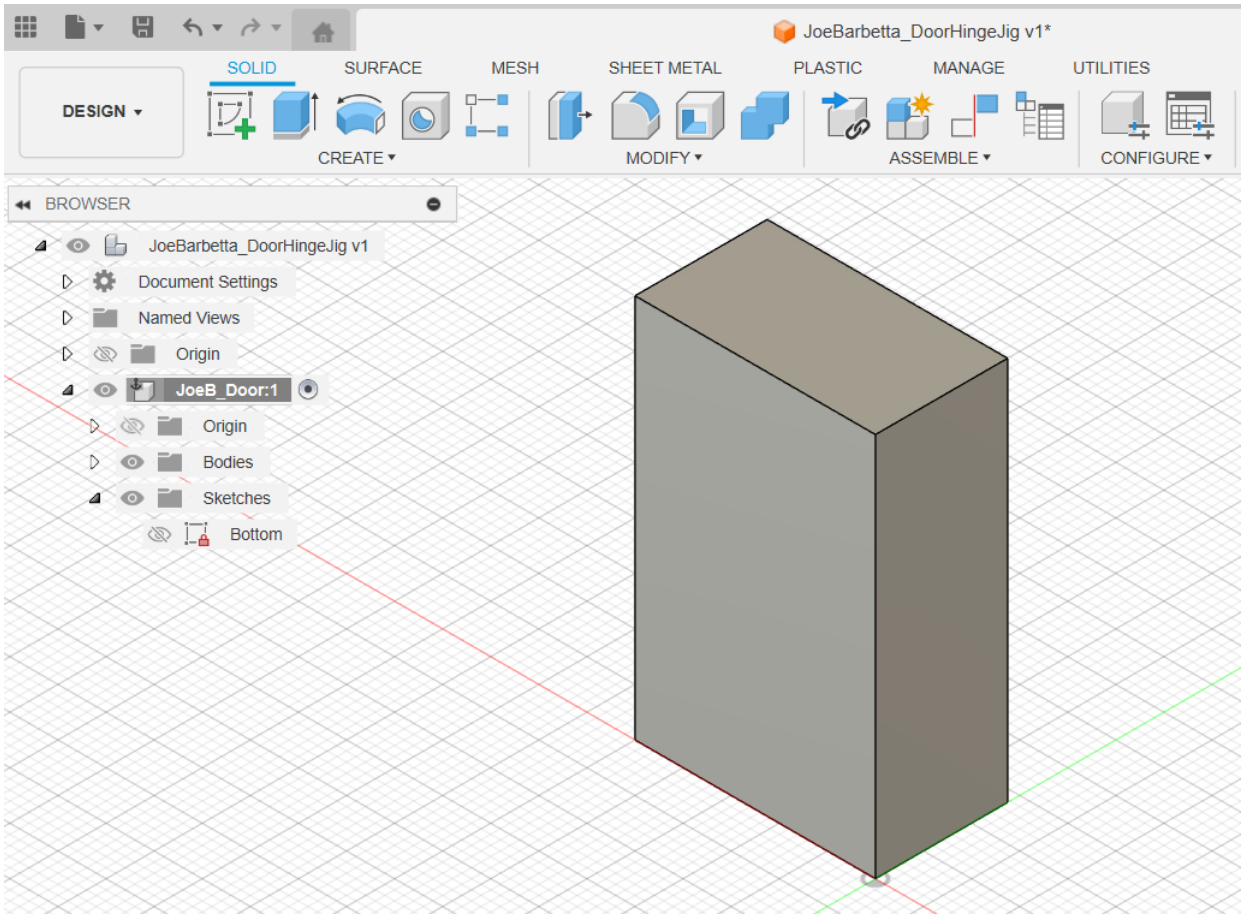
- click **Finish Sketch**



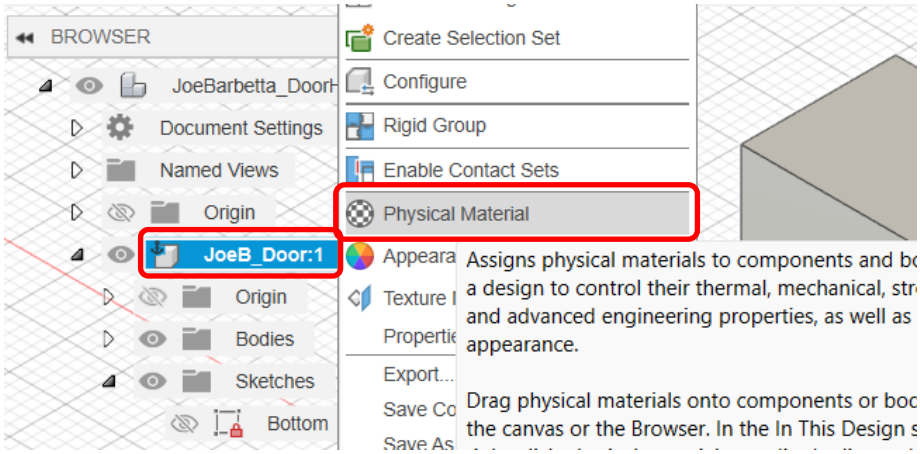
- click on the **Home icon** at the top-left of the **View Cube**
- use the **mouse wheel to zoom** to get a view similar to below
- select the **Extrude** tool, type **4** and then press the **Enter key**



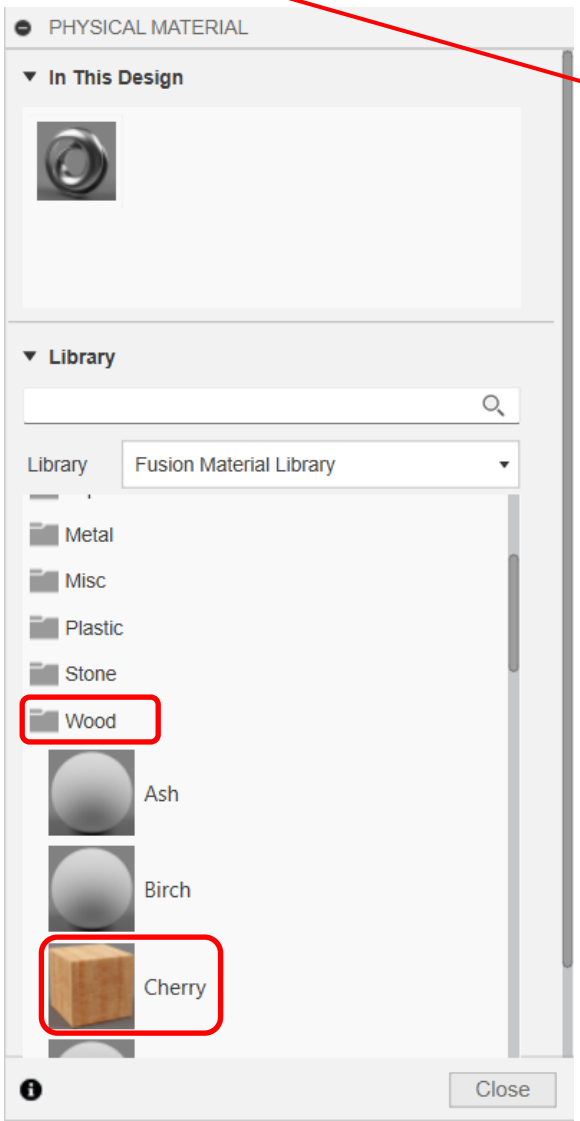
There should now be a solid body as shown below.

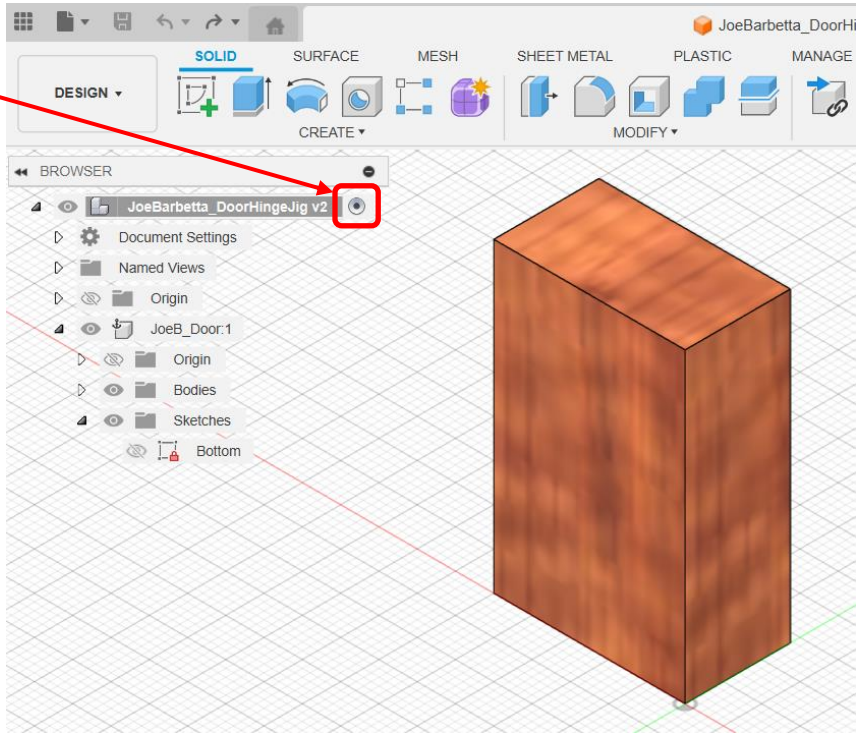


-right click on the **Component** name and select **Physical Material** from the menu.



- scroll down to the **Wood** folder, **click on the folder**, and then **drag the Cherry block onto the body** that was created
- click on the **Close** button
- the body should look as it does on the right
- click on the **circle** next to the project name to activate the overall project

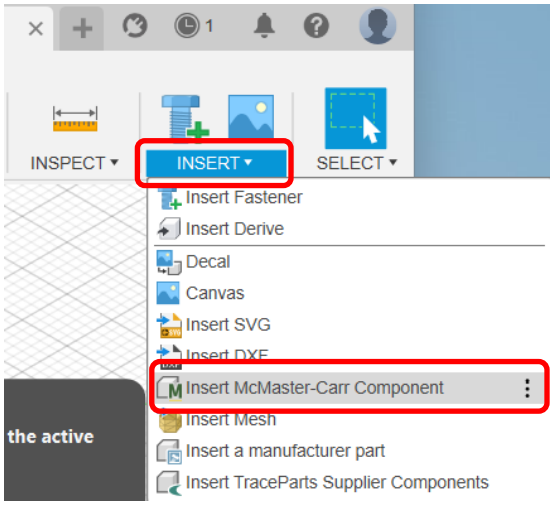




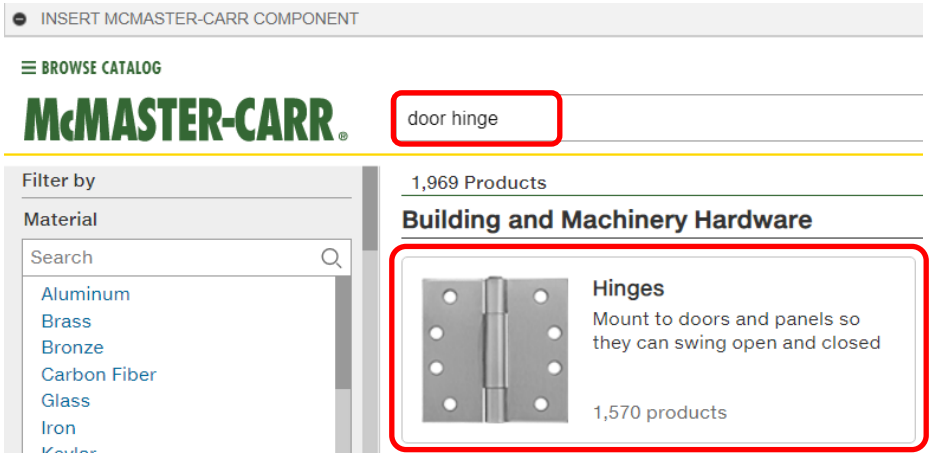
We love McMaster-Carr

Even if the hinge may be purchased elsewhere, McMaster-Carr is a great resource to get CAD models to insert into a design to speed up the design process. Here we search for door hinges and click through categories to find a good match.

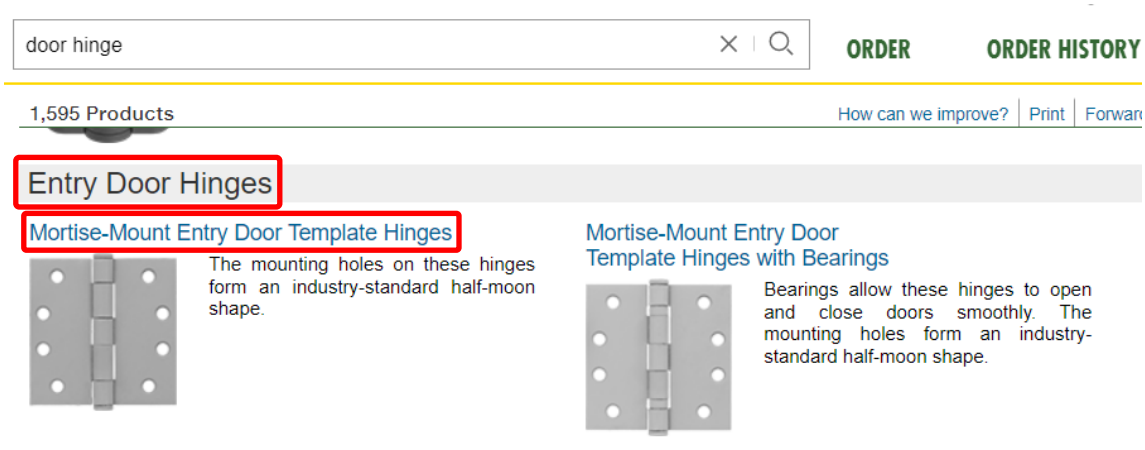
- from the top right of the Fusion screen, select **Insert McMaster-Carr Component** under the **INSERT** menu



- drag the bottom right of the McMaster-Carr window to enlarge it to make it easier to search
- search for **door hinge** and click on the first category **Hinges**



- scroll down to **Entry Door Hinges** and yell **I can't believe how many hinges they sell !**
- click on **Mortise-Mount Entry Door Template Hinges**



- scroll down to the first item under **Chrome-Plated Low-Carbon Steel – Dull** and click on the **blue McMaster-Carr number**

32 Products

How can we improve? | Print | Forward

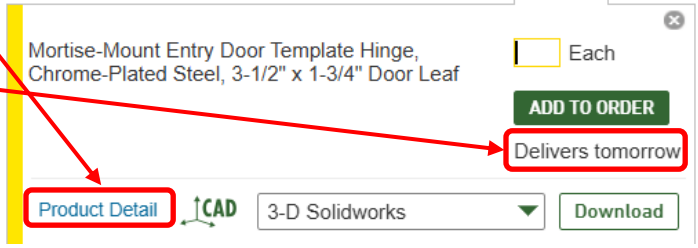
Ht.	Wd.	Ht.	Wd.	O'all Wd.	Leaf Thick.	Range of Motion	Daily Openings	Capacity, lbs.	Pin Type	Pin Dia.	Pin Material	Mounting Fasteners Included		Each
For Hollow Metal and Wood Doors														
Primed Low-Carbon Steel														
3"	1 1/2"	3"	1 1/2"	3"	0.097"	180°	45	40	Removable	15/64"	Steel	Yes	1494A42	$16.17
3 1/2"	1 3/4"	3 1/2"	1 3/4"	3 1/2"	0.119"	180°	205	45	Removable	17/64"	Steel	Yes	1494A11	7.97
4"	2"	4"	2"	4"	0.129"	180°	205	60	Removable	17/64"	Steel	Yes	1494A12	9.60
4 1/2"	2"	4 1/2"	2"	4"	0.134"	180°	205	75	Removable	5/16"	Steel	Yes	1494A43	9.64
4 1/2"	2 1/4"	4 1/2"	2 1/4"	4 1/2"	0.134"	180°	205	75	Removable	5/16"	Steel	Yes	1494A13	10.04
5"	2 1/4"	5"	2 1/4"	4 1/2"	0.145"	180°	205	100	Removable	21/64"	Steel	Yes	1494A15	29.47
5"	2 1/2"	5"	2 1/2"	5"	0.145"	180°	205	100	Removable	21/64"	Steel	Yes	1494A14	28.43
Chrome-Plated Low-Carbon Steel—Dull														
3 1/2"	1 3/4"	3 1/2"	1 3/4"	3 1/2"	0.119"	180°	205	45	Removable	17/64"	Steel	Yes	1494A52	9.47
4"	2"	4"	2"	4"	0.129"	180°	205	60	Removable	17/64"	Steel	Yes	1494A53	11.45
4 1/2"	2"	4 1/2"	2"	4"	0.134"	180°	205	75	Removable	5/16"	Steel	Yes	1494A54	8.43

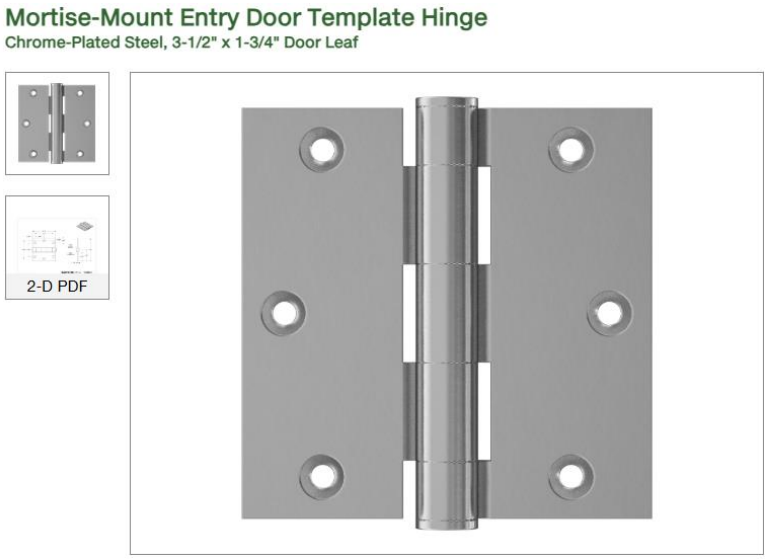
- click on **Product Detail** to get the drawings shown below.

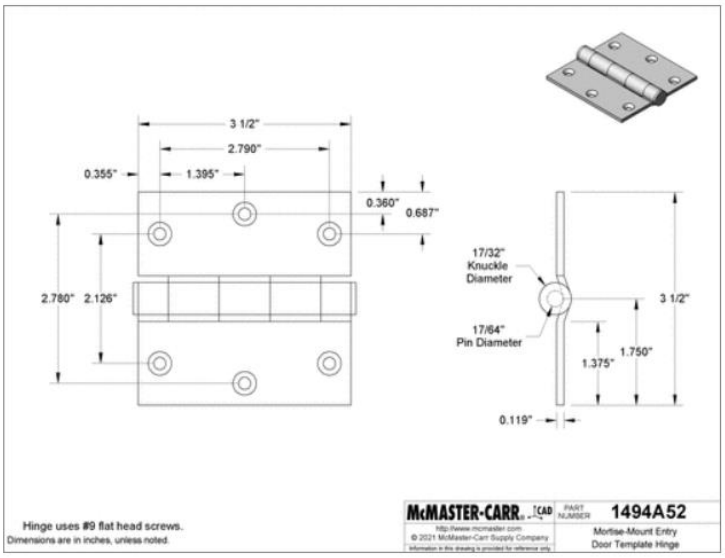
Chrome-Plated Low-Carbon Steel—Dull

3 1/2" 1 3/4" 3 1/2" 1 3/4" 3 1/2" 0.119" 180° 205 45 Removable 17/64" Steel Yes **1494A52** 9.47

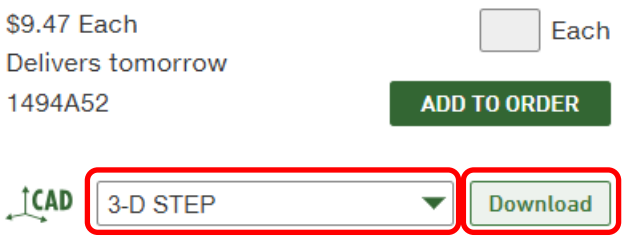
Wow! We can get this tomorrow.
It is difficult to find something that McMaster-Carr doesn't have in stock.



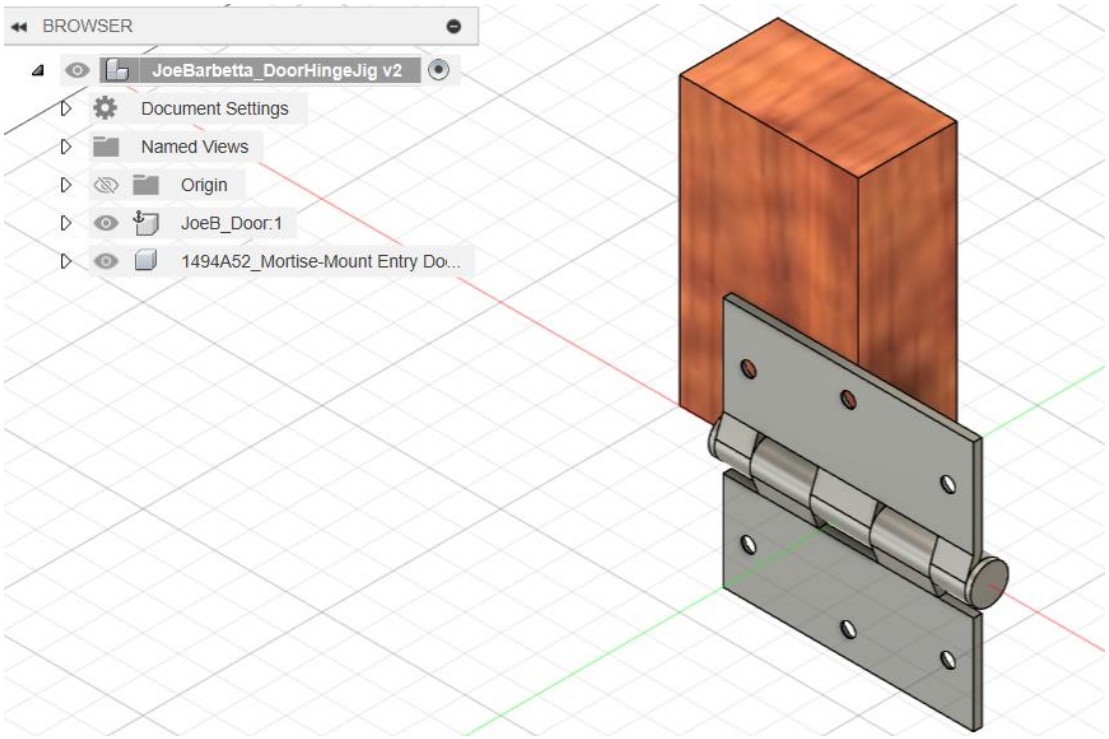




- in the upper right of this screen find the below section
- select 3-D STEP and click Download

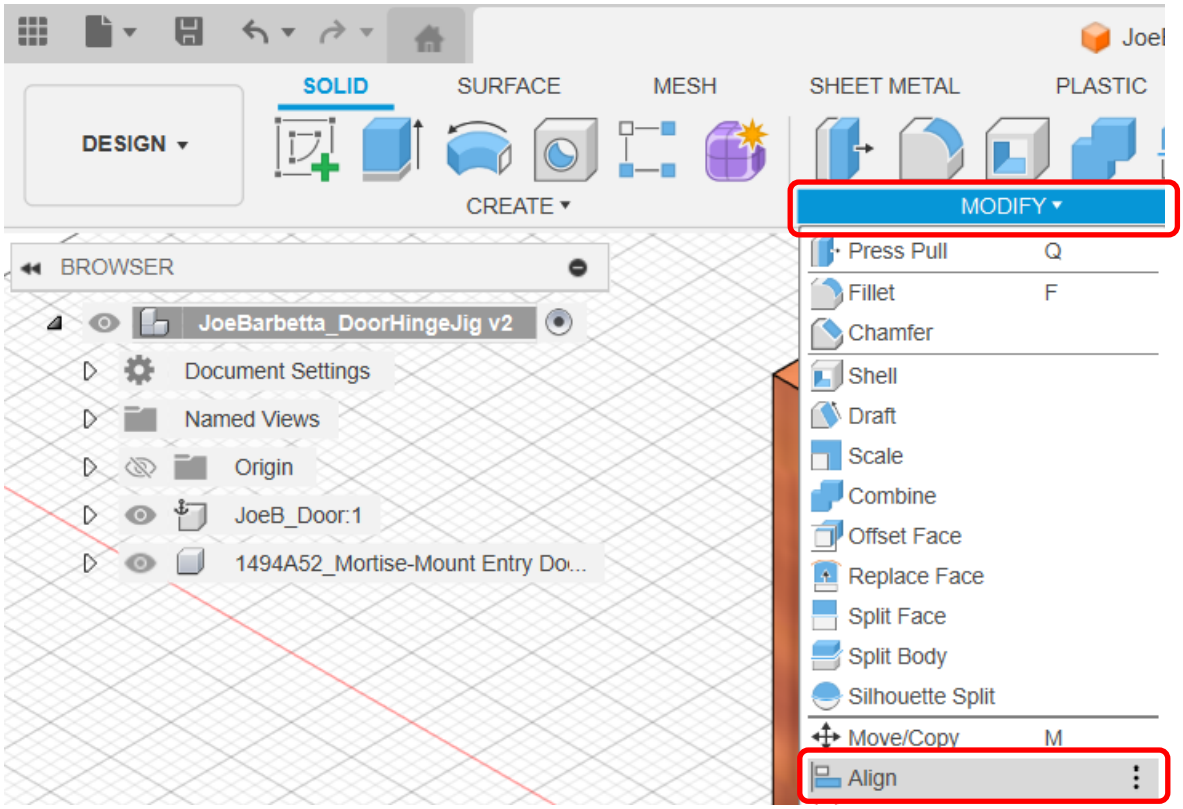


- marvel at the beautiful hinge that is now in your project

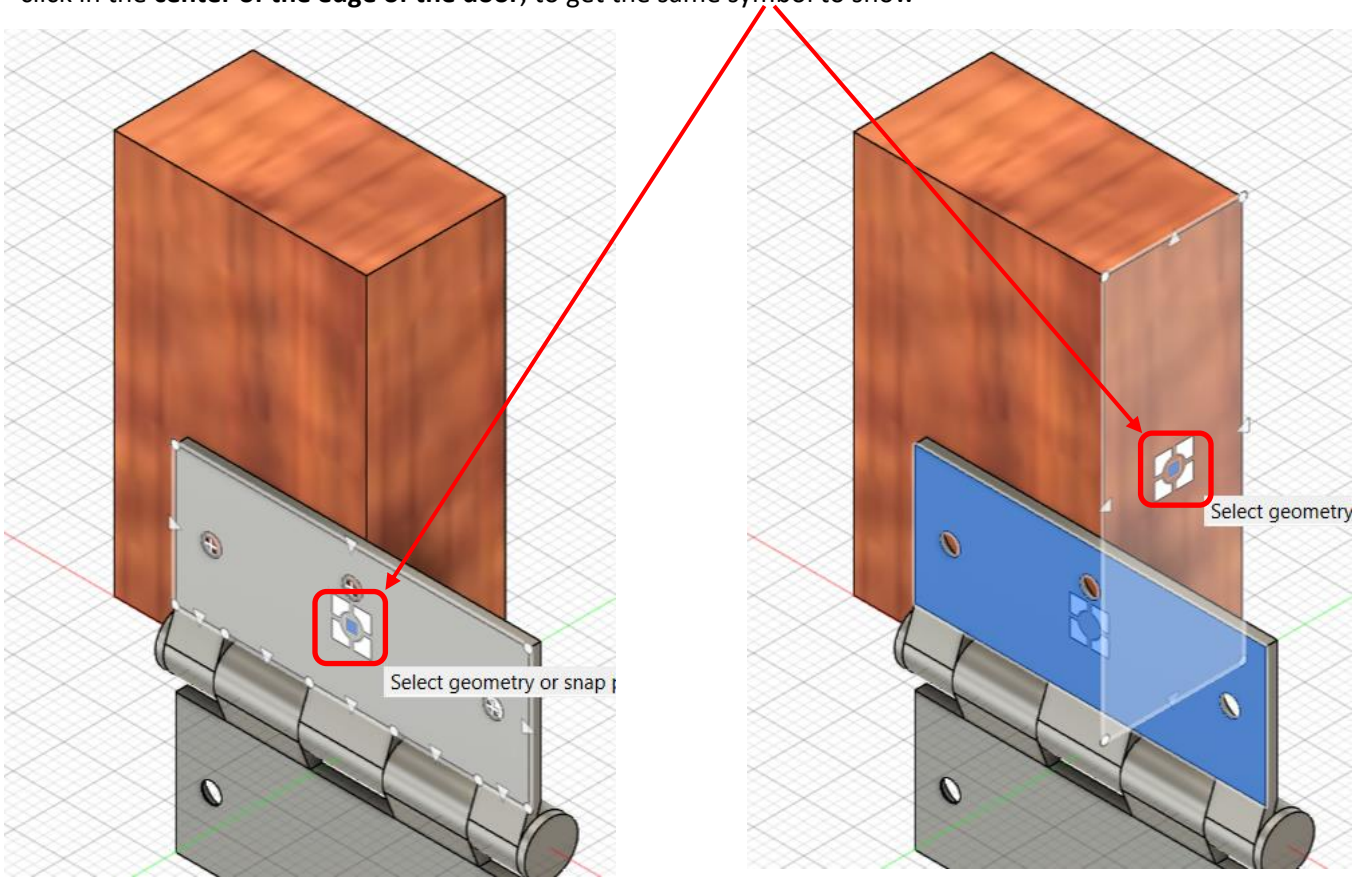


Aligning Components

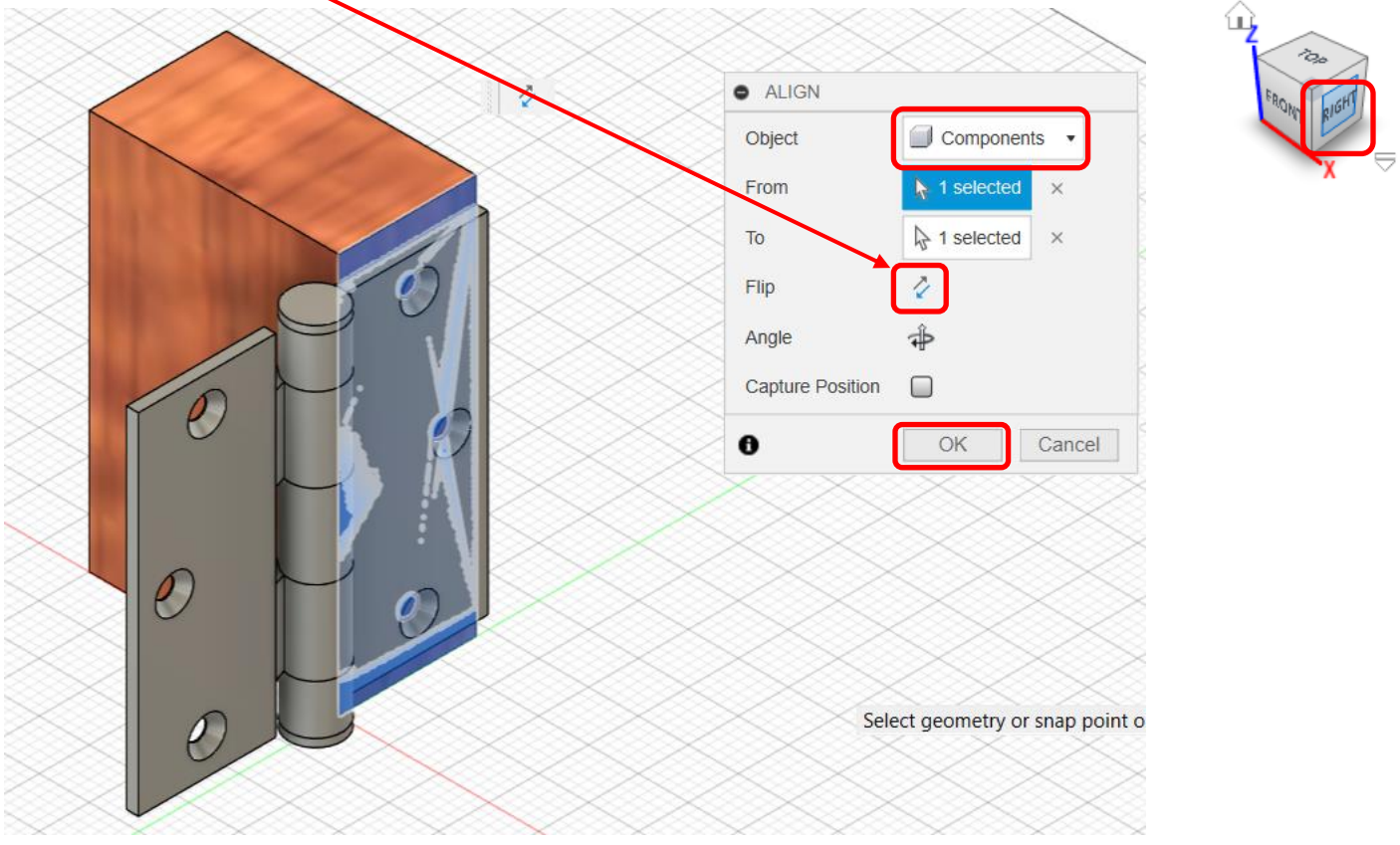
-from the **MODIFY** menu select **Align**



- click in the **center of the hinge leaf**, which should show the indicated symbol
- click in the **center of the edge of the door**, to get the same symbol to show



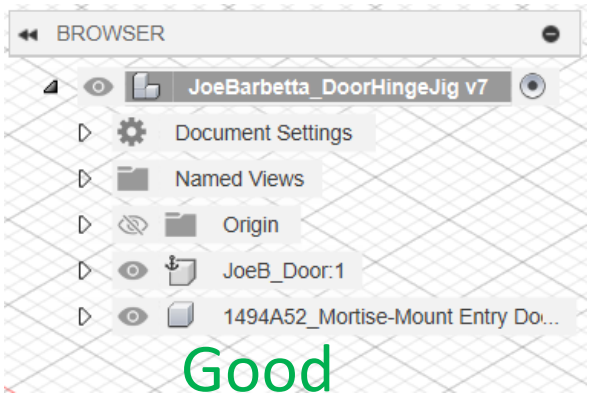
- ensure that **Components** is selected for the Object
- click on the **Flip** icon and then **OK**
- click on the **RIGHT** face of the **View Cube.** **If the align fails say bad words and continue to the next page.**

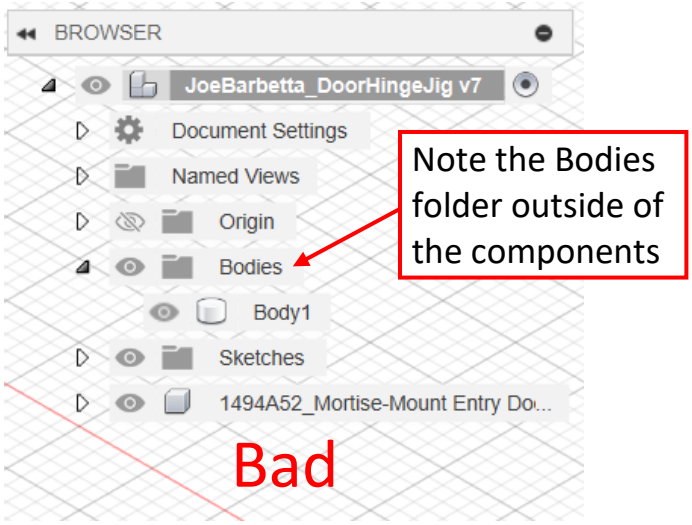


If the Align worked and you didn't have to say bad words, skip this page.

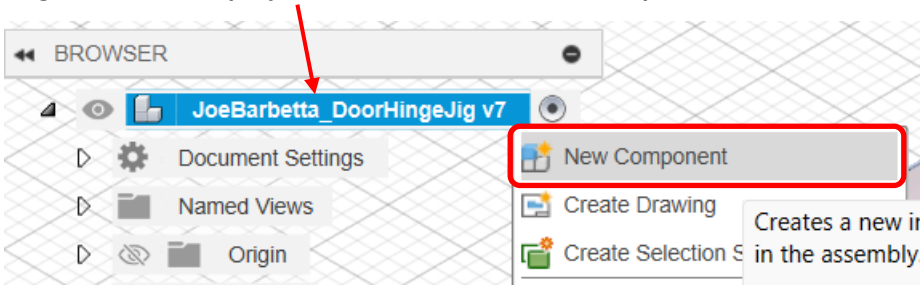
If the Align did not work and you saud bad words read below.

One possibility is that the Door body is Not in a Door Component and a Bodies folder exists outside of a component.

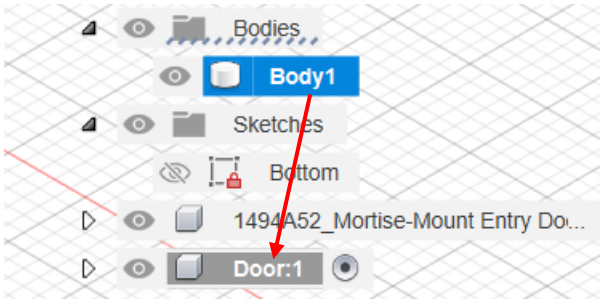




- right-click on the **project name** and select **New Component** and set the Component name to **Door**



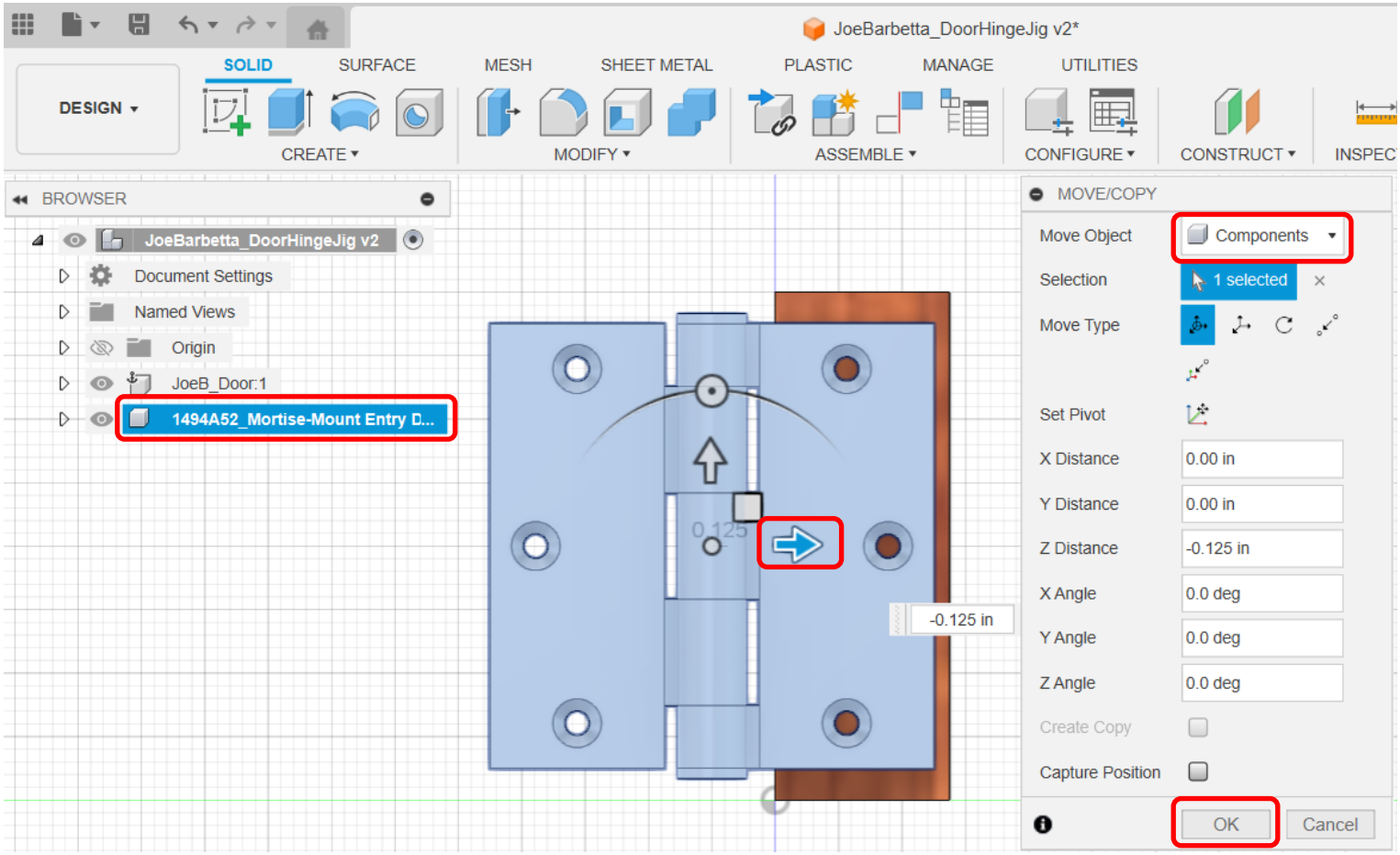
- hold the left mouse button down on **Body1** and **drag it onto the name of the new component**. Ignore any warning message that may be shown.



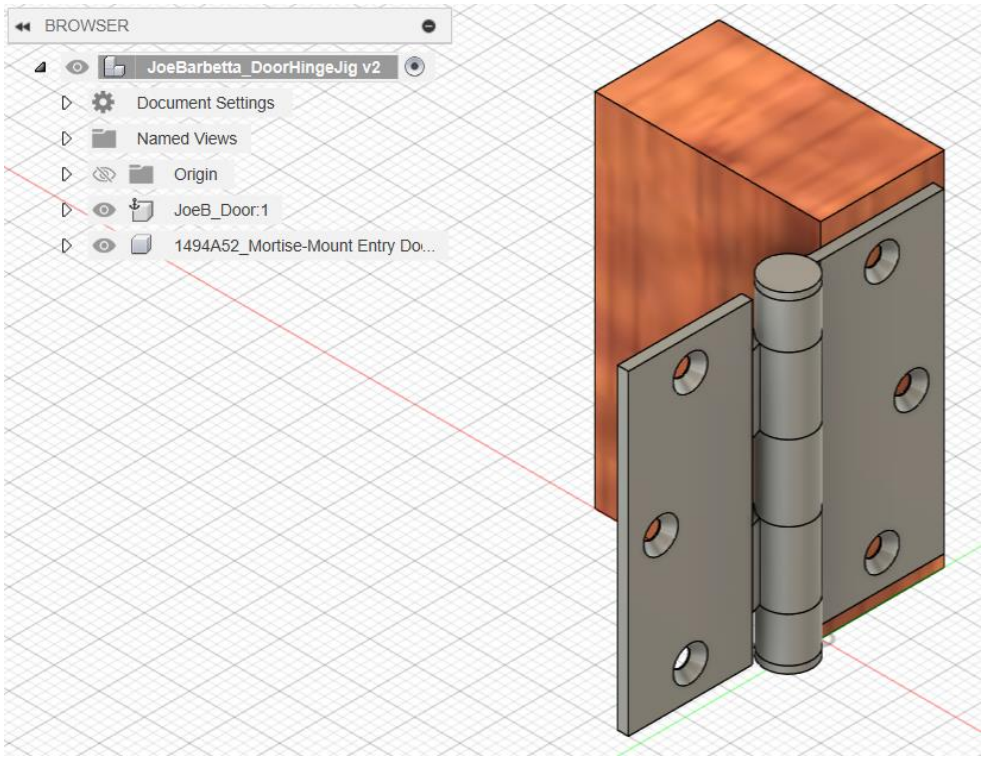
- if there is an anchor symbol on the hinge component, right-click on it and select **Unground From Parent**.

Now you can stop saying bad words and return to the previous alignment steps.

- from the **MODIFY** menu select **Move/Copy**
- change the **Move Object** to **Components**
- click on the **Hinge Component in the Browser** and then the Blue horizontal arrow
- enter **-0.125** (note the minus sign), which should shift the hinge slightly to the left and click **OK**

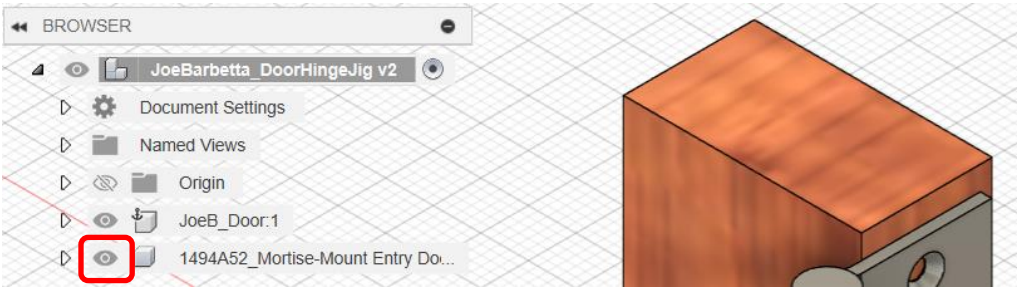


- click on the **Home icon** at the **View Cube** to get a view similar to that below

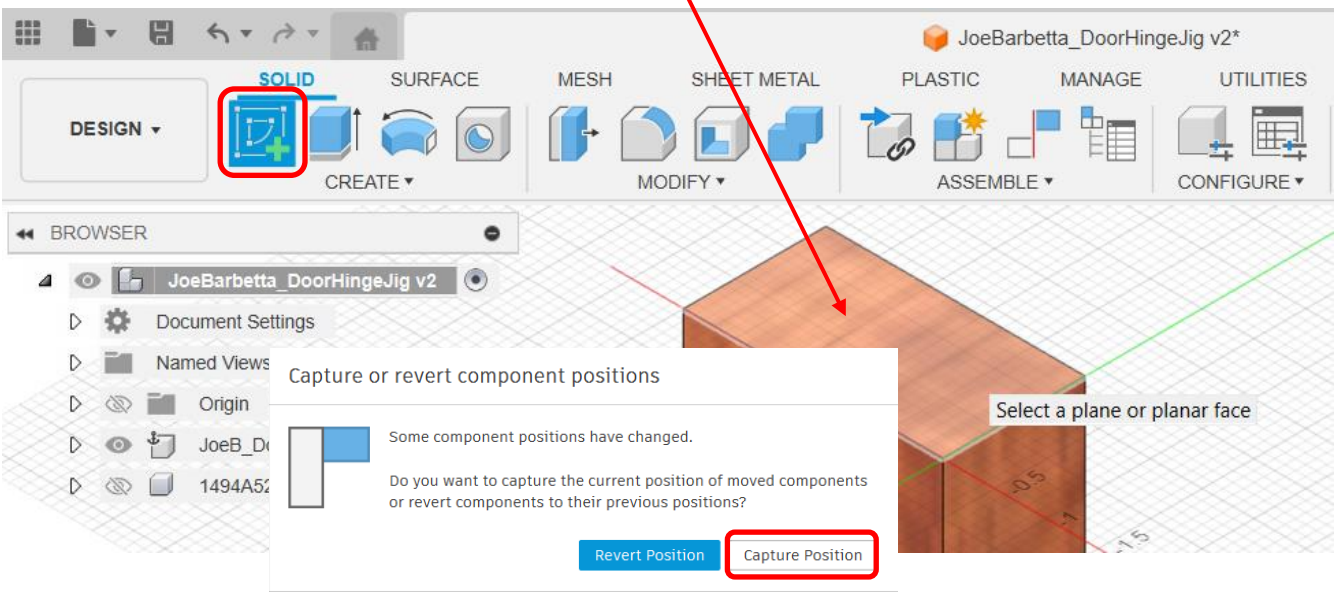


All this work and still no jig, lets get started

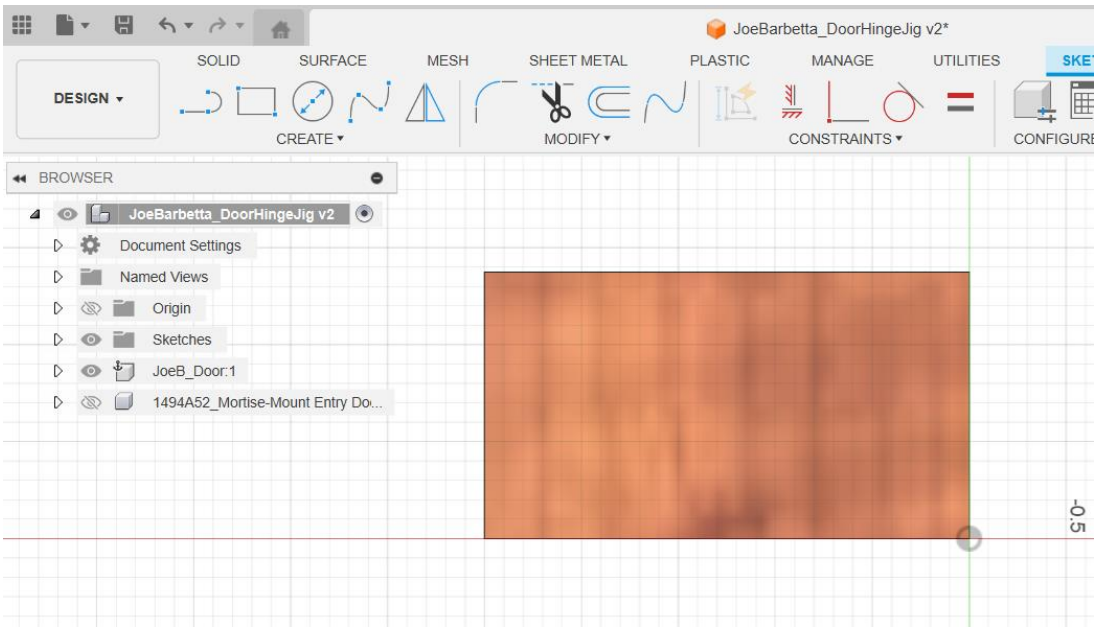
- click on the **eye icon** next to the Hinge component to hide the hinge



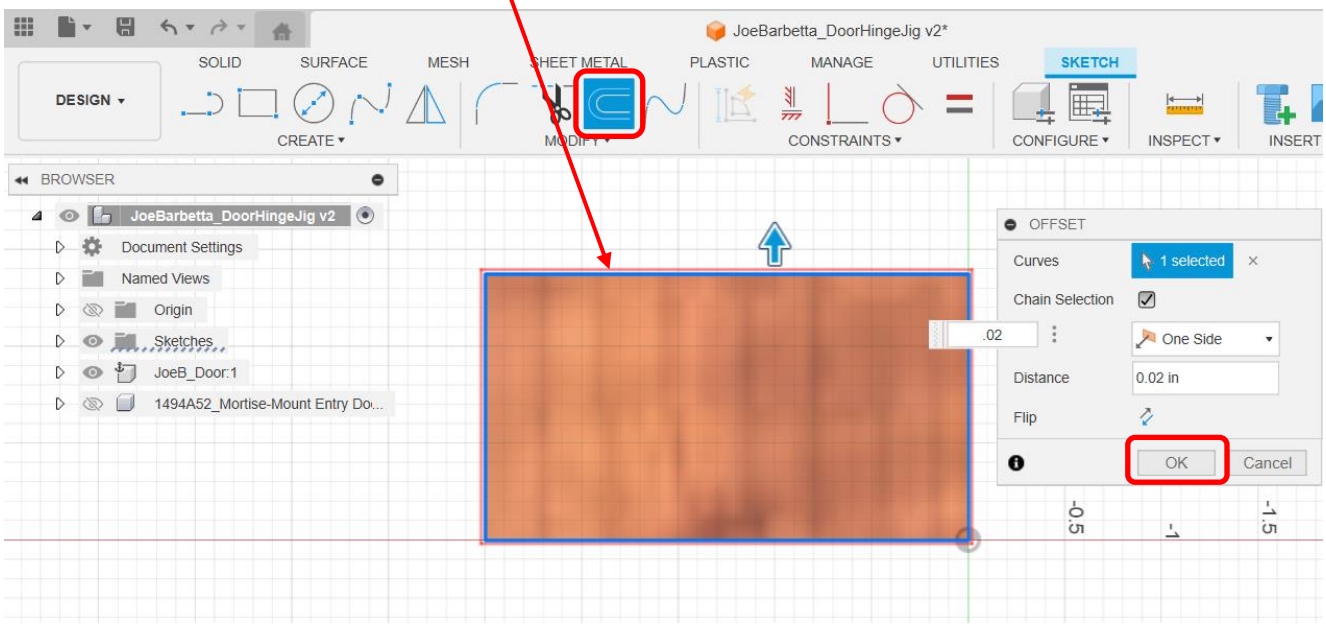
- select the **Sketch** tool and click on the **top** of the Door
- if a **Capture …** window pops up, click **Capture Position**



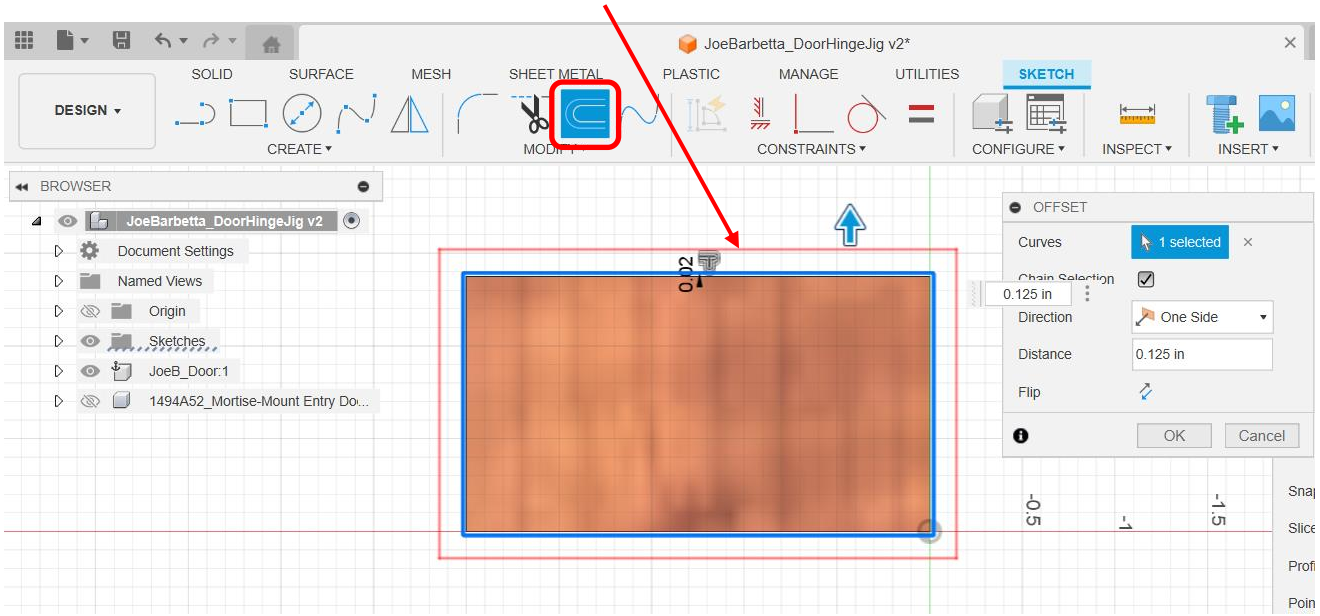
- **Zoom in** to achieve a view similar to that below



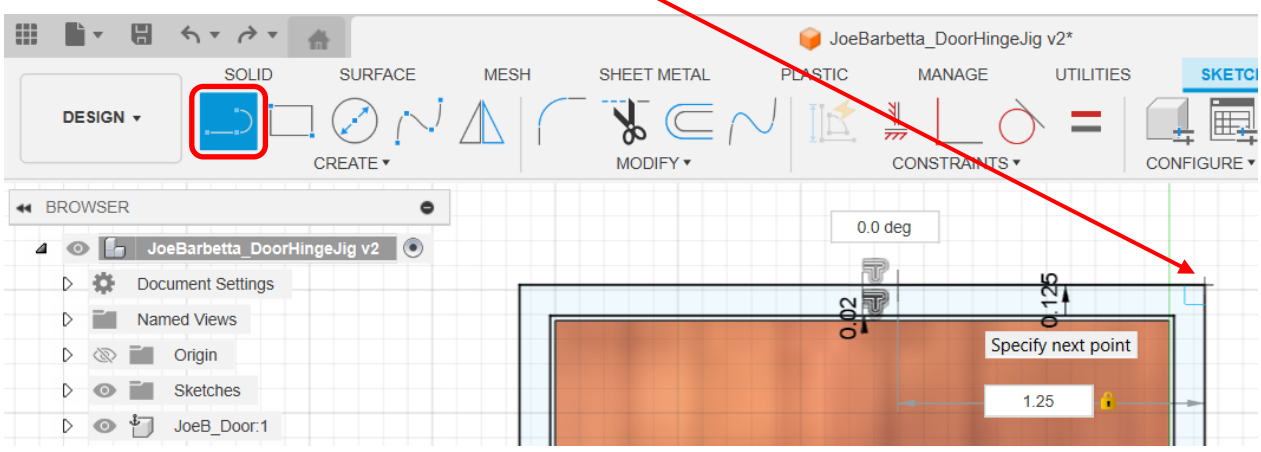
- select the **Offset** tool and click on an **edge** of the door top
- type **0.02** and press the **Enter key**



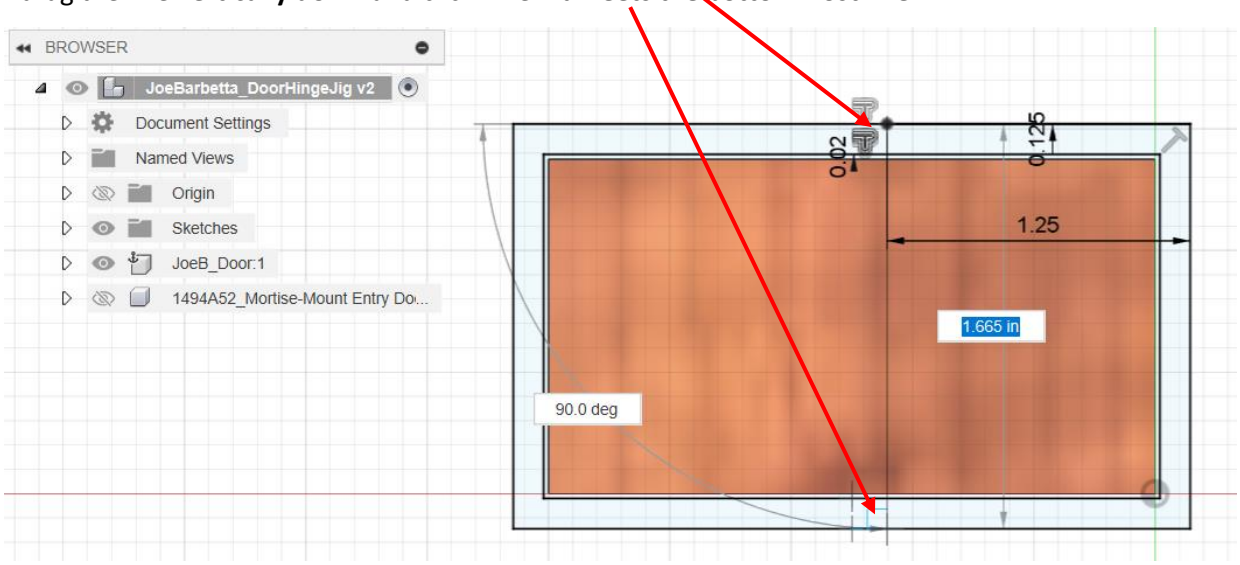
- use the **Offset** tool again, click on one of the **outer lines**, type **0.125**, and then press the **Enter key**



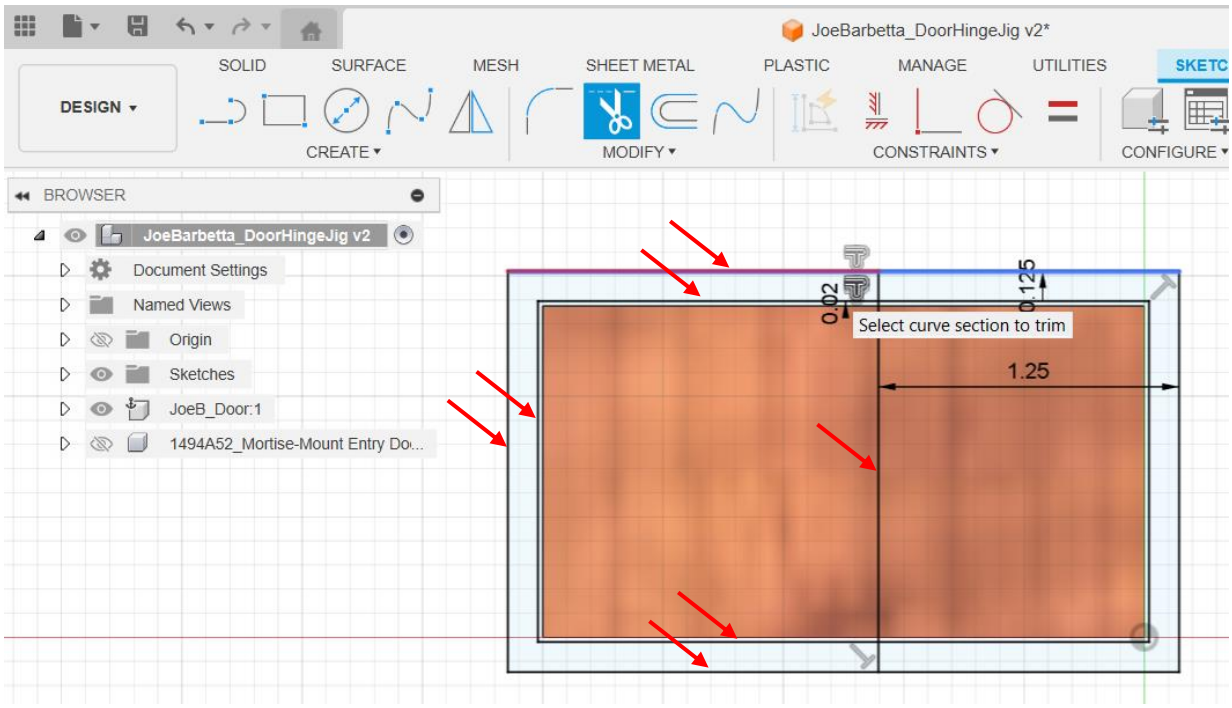
- select the **Line** tool and click on the **top right corner** of the outer rectangle
- **drag the line to the left** keeping it horinzontal, type **1.125**, and press the **Enter key**



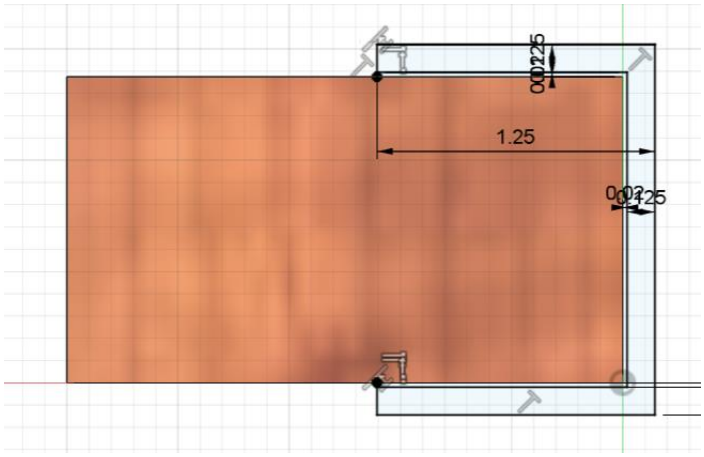
- select the **Line** tool again and **start this line at the end of the last line drawn**
- drag the line **vertically down** and click when it **meets the bottommost line**



- select the **Trim** tool and click on the **7 lines pointed to by arrows**



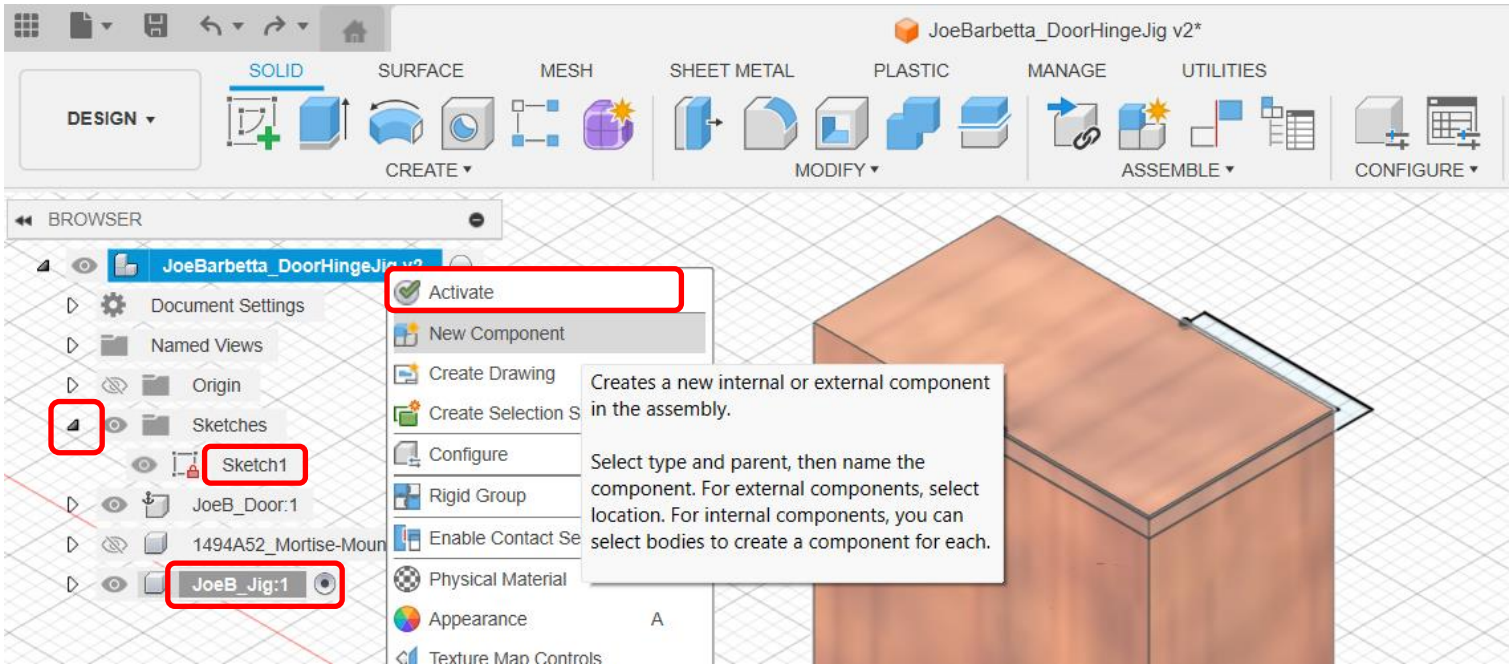
- the resulting lines should look similar to that below. It is OK if the thin lines with values look different.



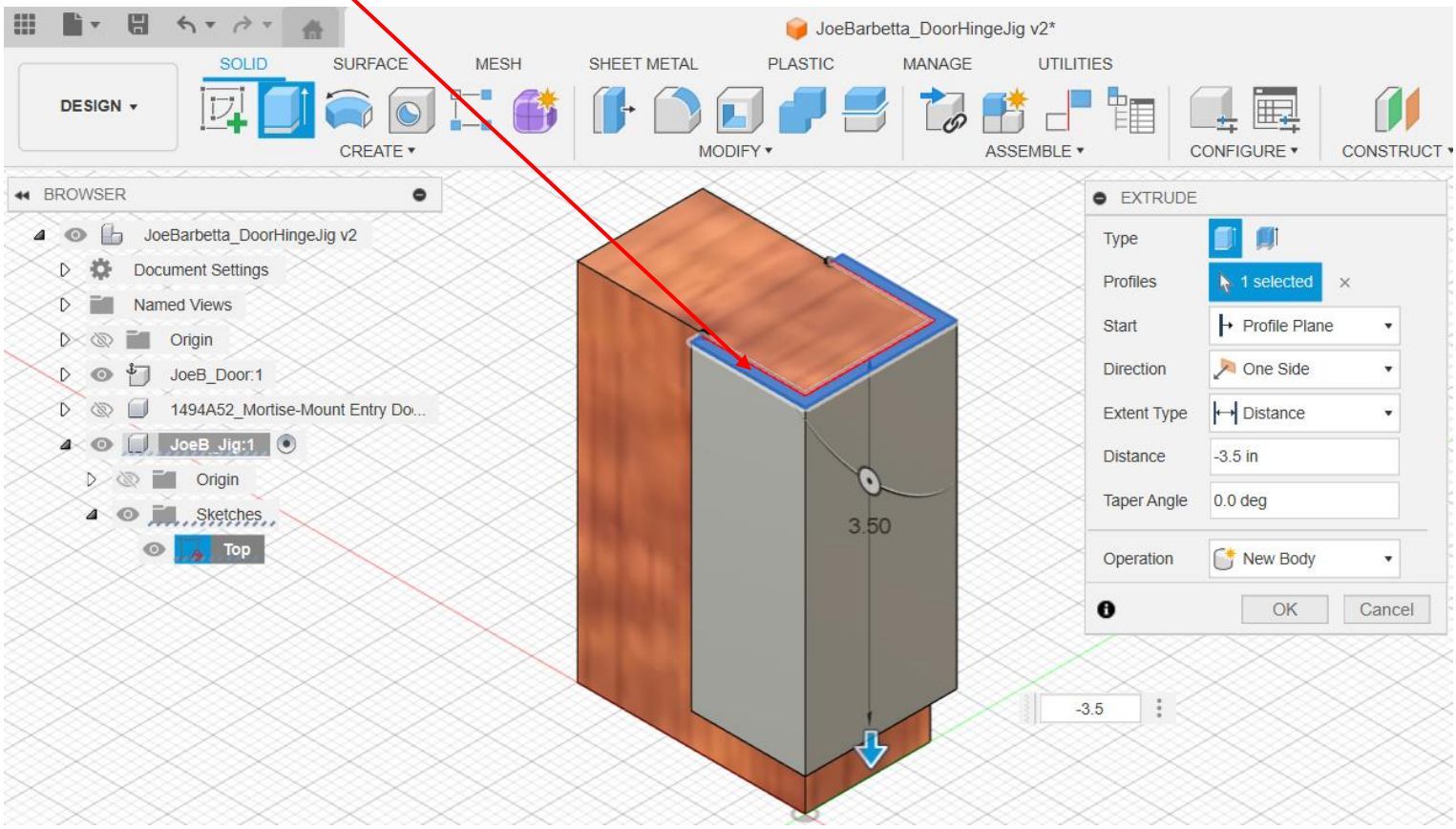
A Fusion expert may criticize the method used here because there are two tiny line segments left and construction lines were not used. You can say, **“Dude, I’m just trying to get my art credit.”**

Oh No! We forgot to make a Component for the Jig! Let's fix this.

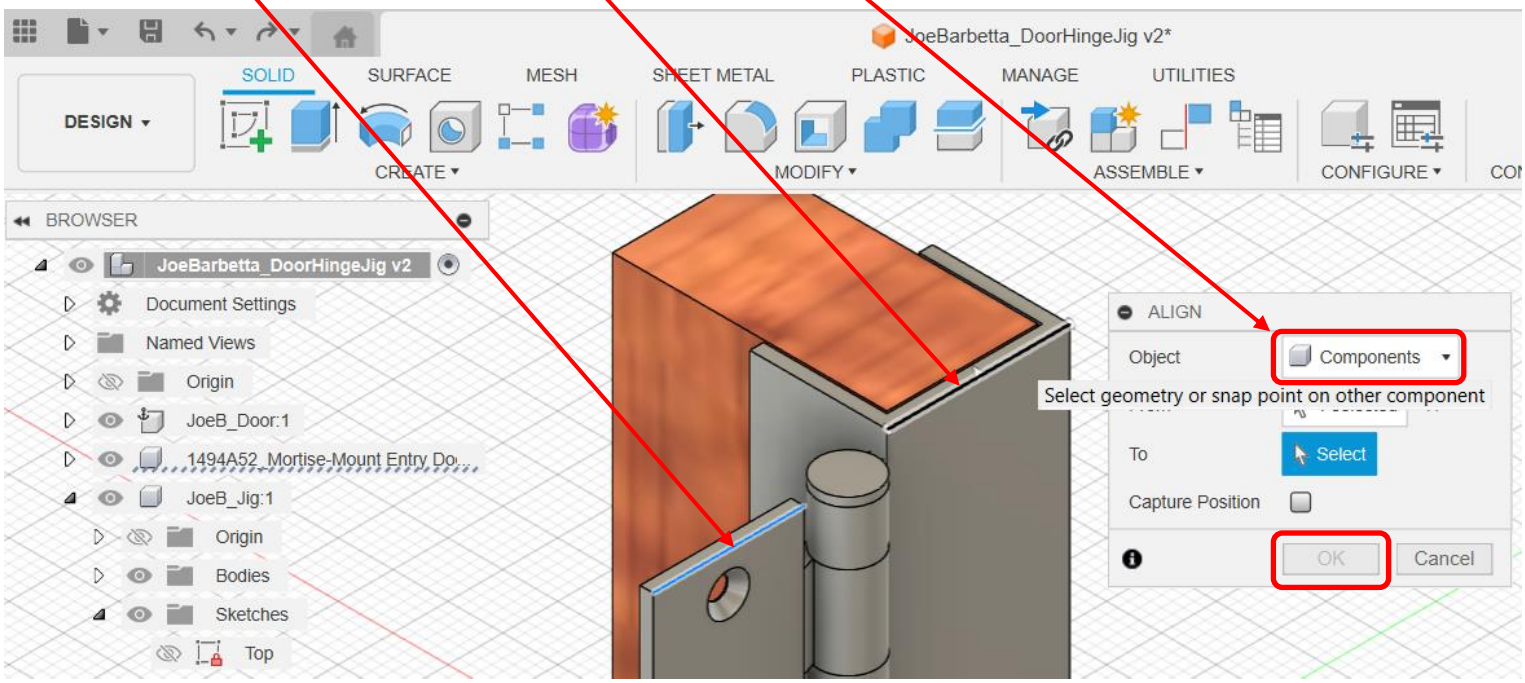
- right click on the **Project name** and select **New Component**
- name the component with your **First name**, **last name initial**, **underscore**, and **Jig**, e.g. **JoeB_Jig**
- click on the **arrow** to open the Sketches folder and **drag Sketch1 onto the new component name**



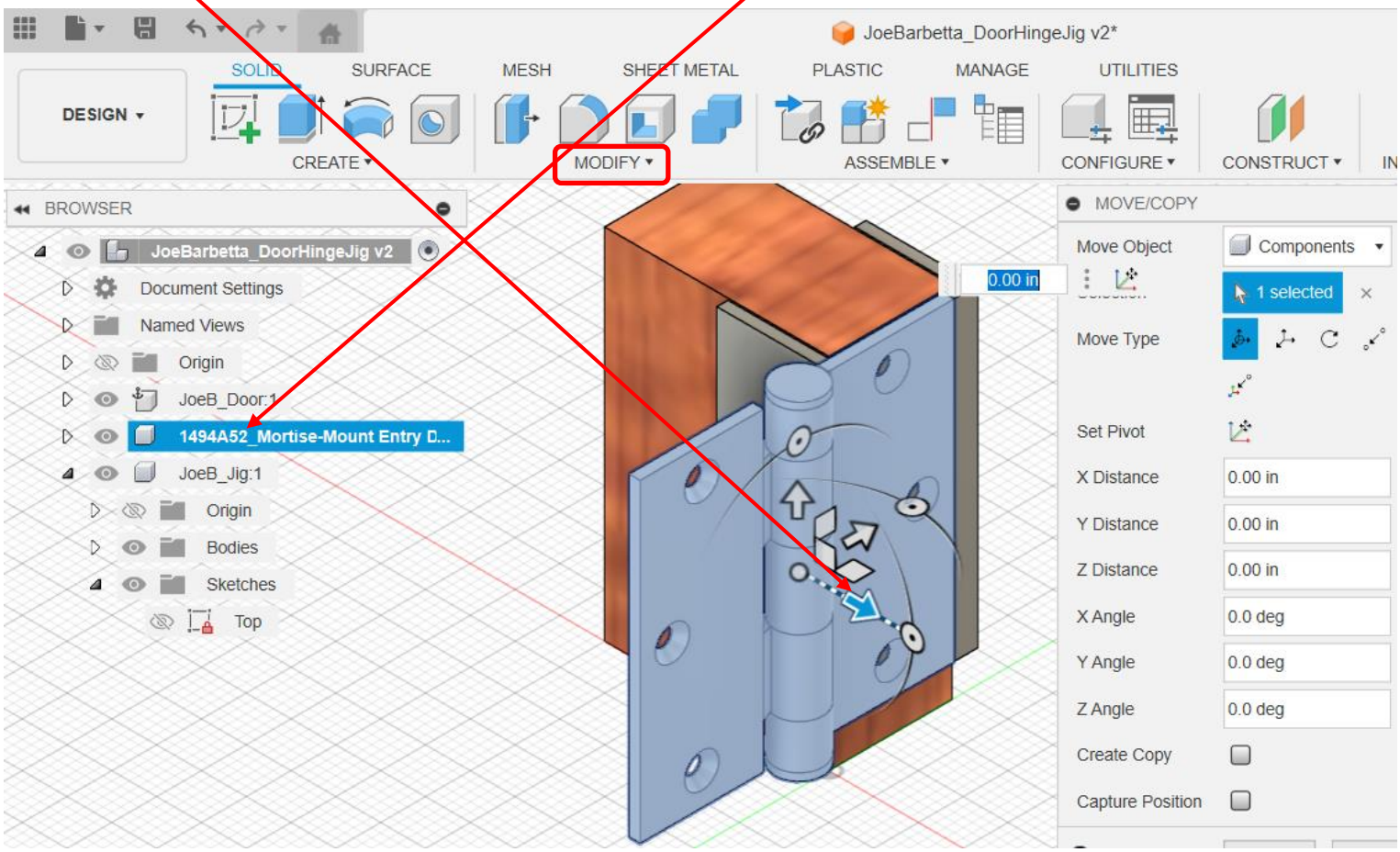
- select the **Extrude** tool
- click on the **sketch profile**
- type **-3.5** (note minus sign) and press the **Enter Key**



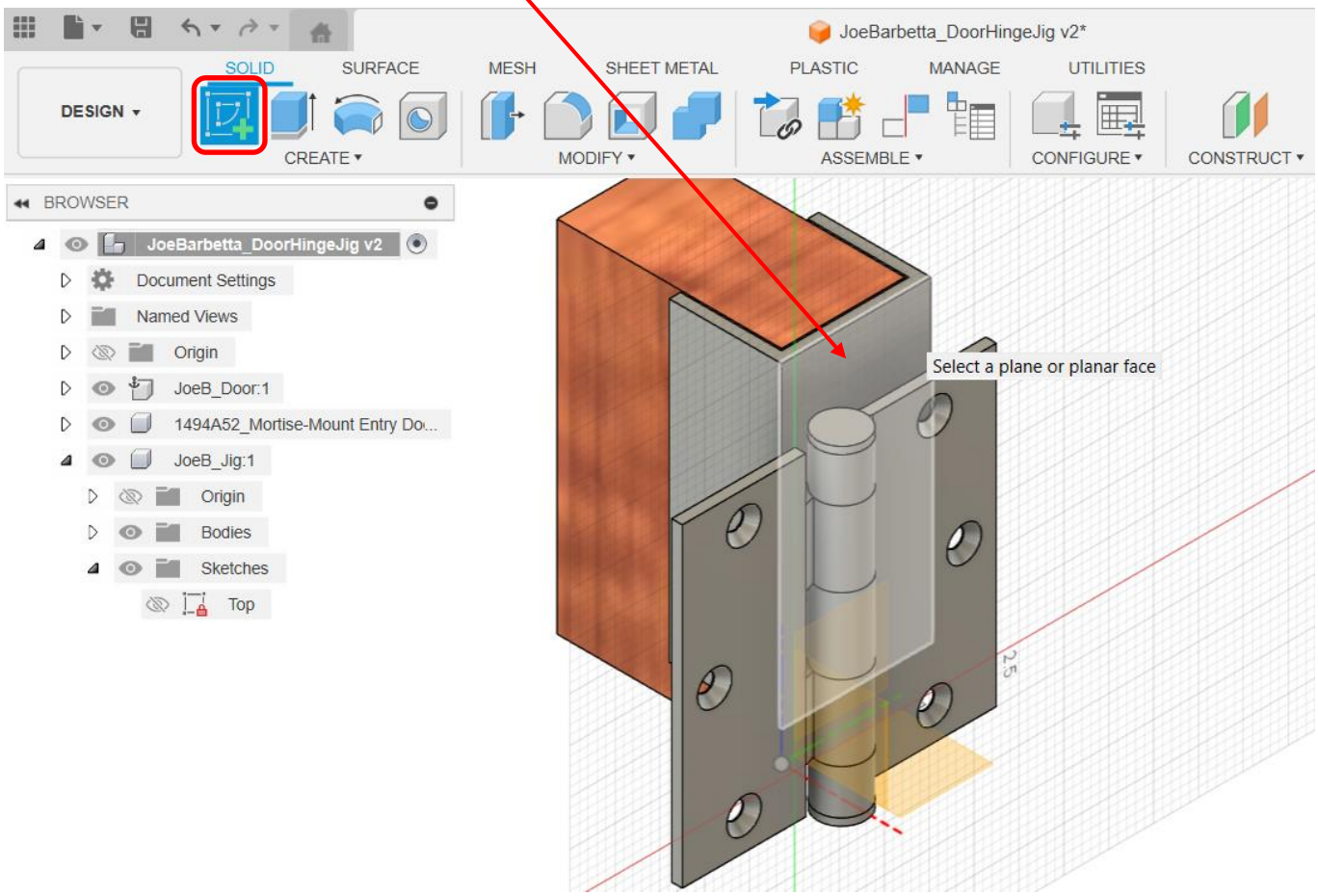
- click on the **circle of the Project Name** to Activate the entire project
- click on the **eye icon** for the hinge
- under the **MODIFY** menu select **Align** and ensure that **Components** is selected
- click on the **top edge** of the hinge, then the **top edge** of the jig, and click **OK**



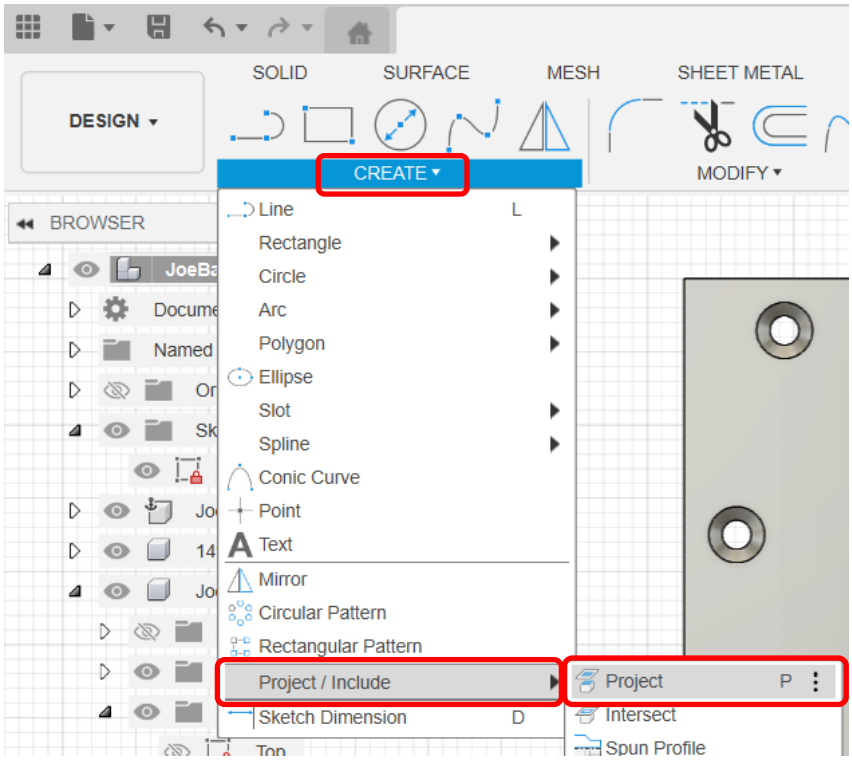
- under the **MODIFY** menu select **Move/Copy** and click on the hinge **Component name**
- click the **blue arrow** facing down and to the right and enter **1.00**. This is just to move the hinge away from the jig.



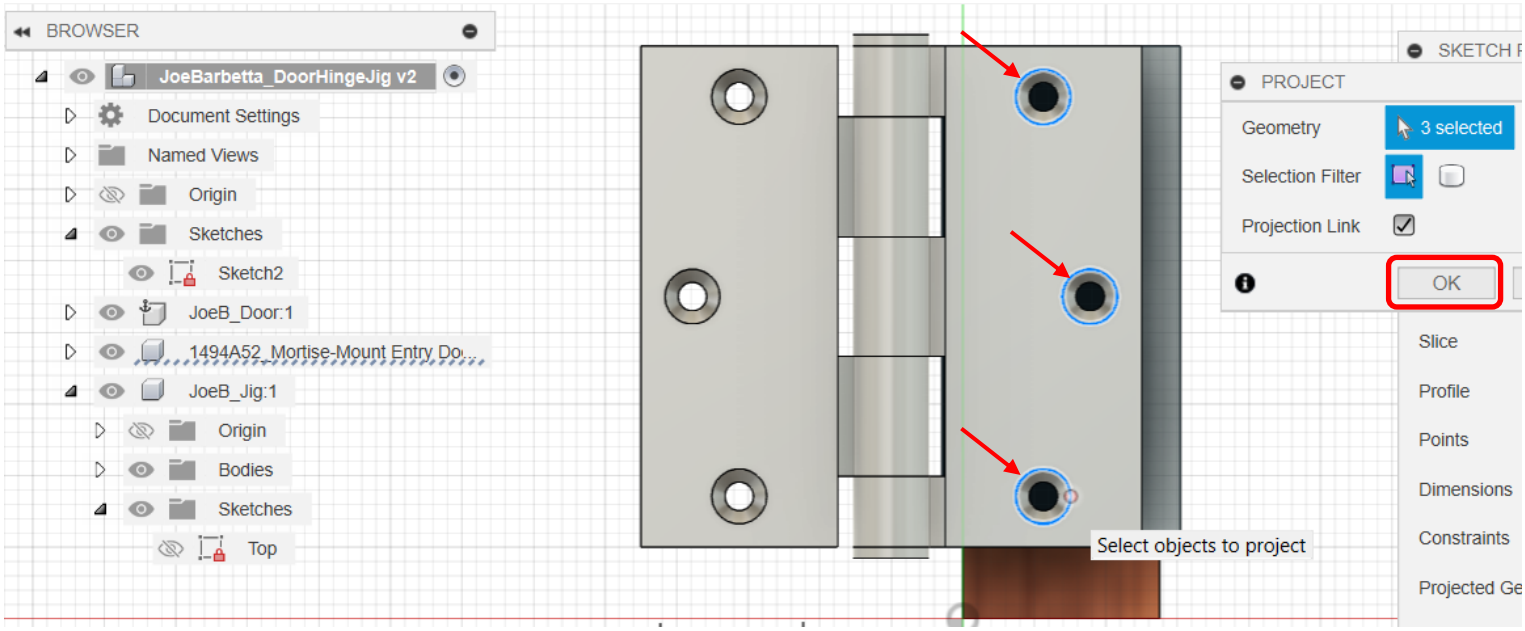
- select the **Sketch** tool and click on the **face of the jig**
- click on **Capture Position** if the Capture or Revert Component Positions window shows



- under the **CREATE** menu select **Project / Include** and **Project**



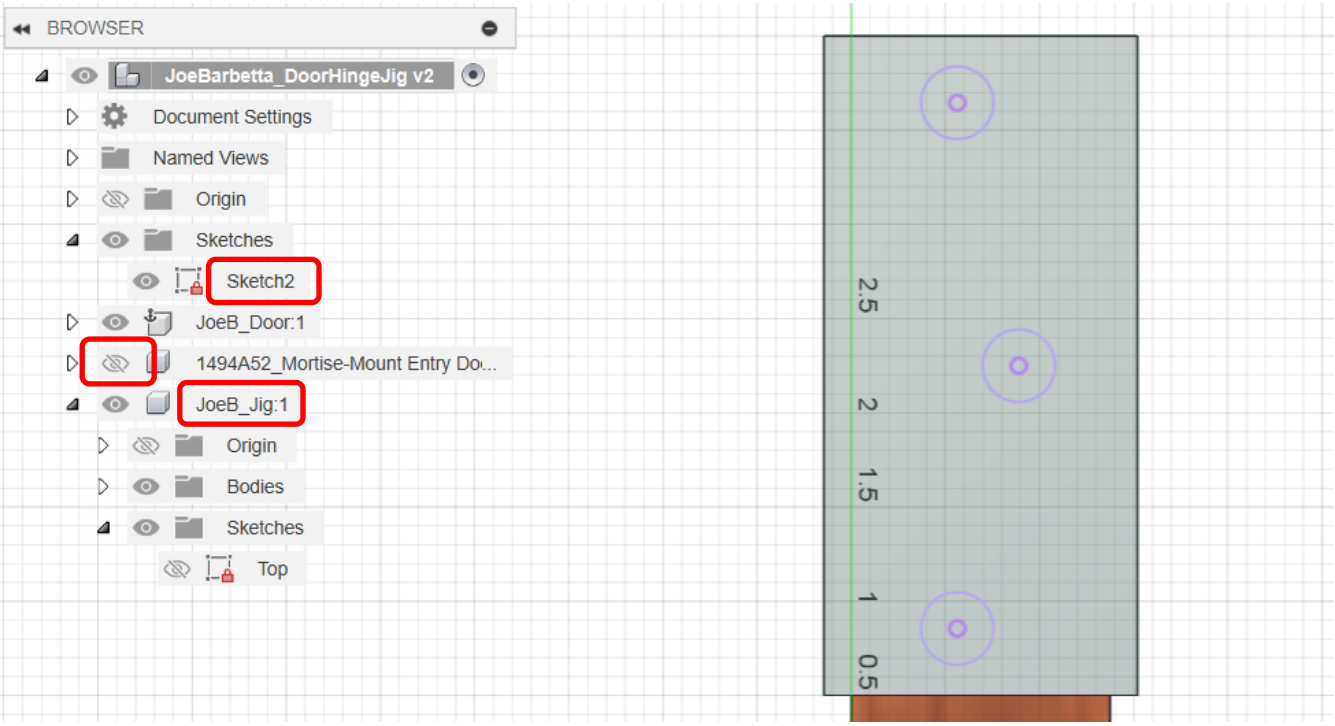
- click on the **edges of the 3 holes** and then **OK**



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

Oh No! Forgot to activate the Jig Component before creating the last sketch!

- drag the **last Sketch** created onto the **Jig Component name** and click on the **Circle** next to it to **Activate**
- right click on the name of that Sketch and Rename to **Holes**



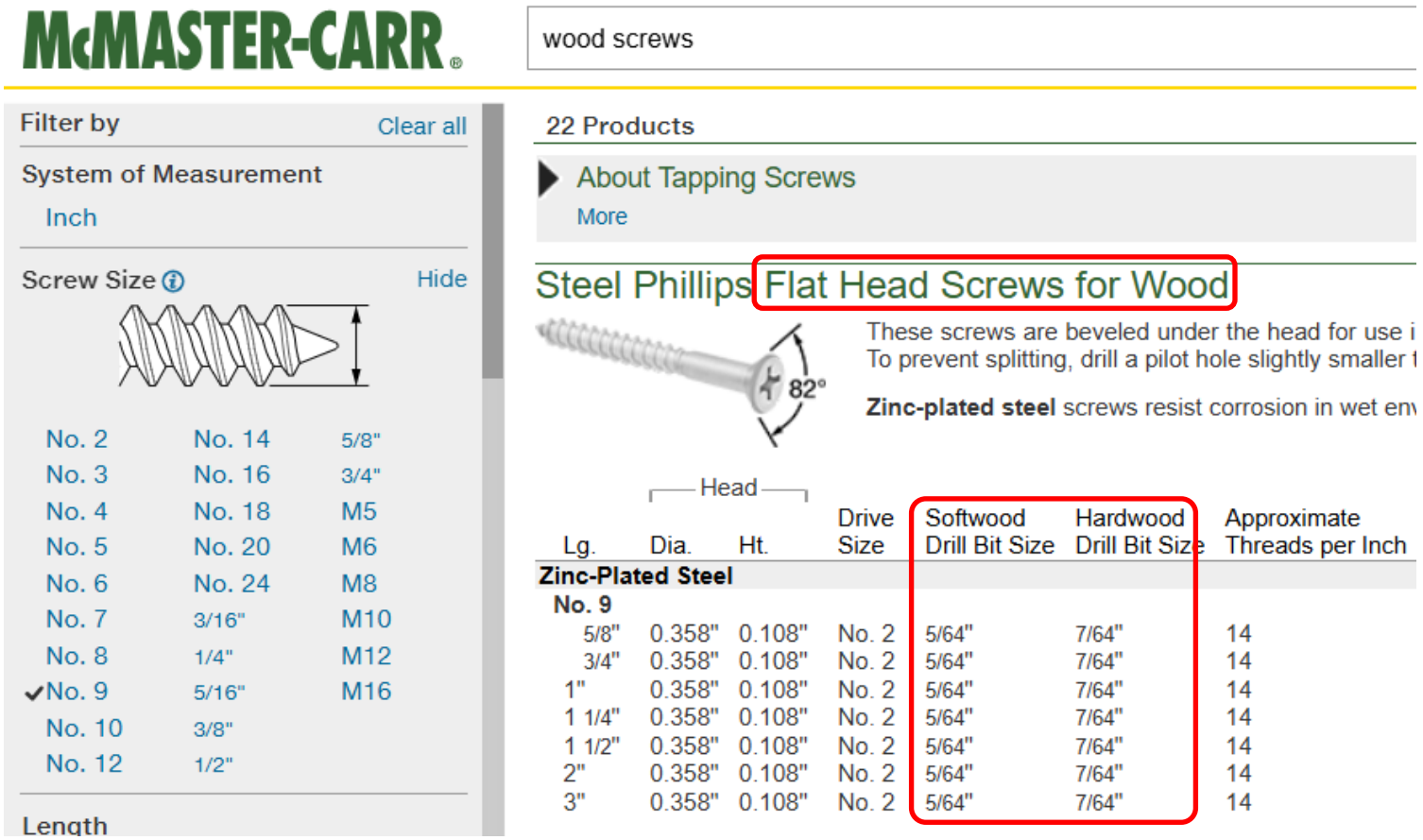
The information page on for the hinge specified the use of No. 9 screws for mounting.

This would be a wood screw, unless the door is made of Unobtanium.

Here No. 9 Flat Head Wood screws were found in McMaster-Carr. To prevent wood from splitting, a pilot hole should be drilled into the door for each screw location. This also reduces the torque needed to prevent stripping of the screw head.

It is important to use a specified pilot hole size. If this hole is too small, the wood may split. If the hole is too large, the screw may pull out. Note that there are drill sizes for **softwood** vs **hardwood**. A cheaper door is likely **softwood**, ie. Pine, Fir, or Spruce. A more expensive door may be a **hardware**, ie. Maple, Walnut, Oak. **Softwoods** are harvested from **conifers** (trees with needles and pine cones). **Hardwords** are harvested from **deciduous** trees (with leaves). Conifers grow more quickly, which contributes to softwood being cheaper.

We will assume softwood for now, which shows a specified pilot hole of **5/64”**. Note that this pertains to a **No 9.** screw. Other size screws would have different pilot hole sizes specified. **Note that the size, No. 9, only reflects the diameter of the screw and Not its length.**

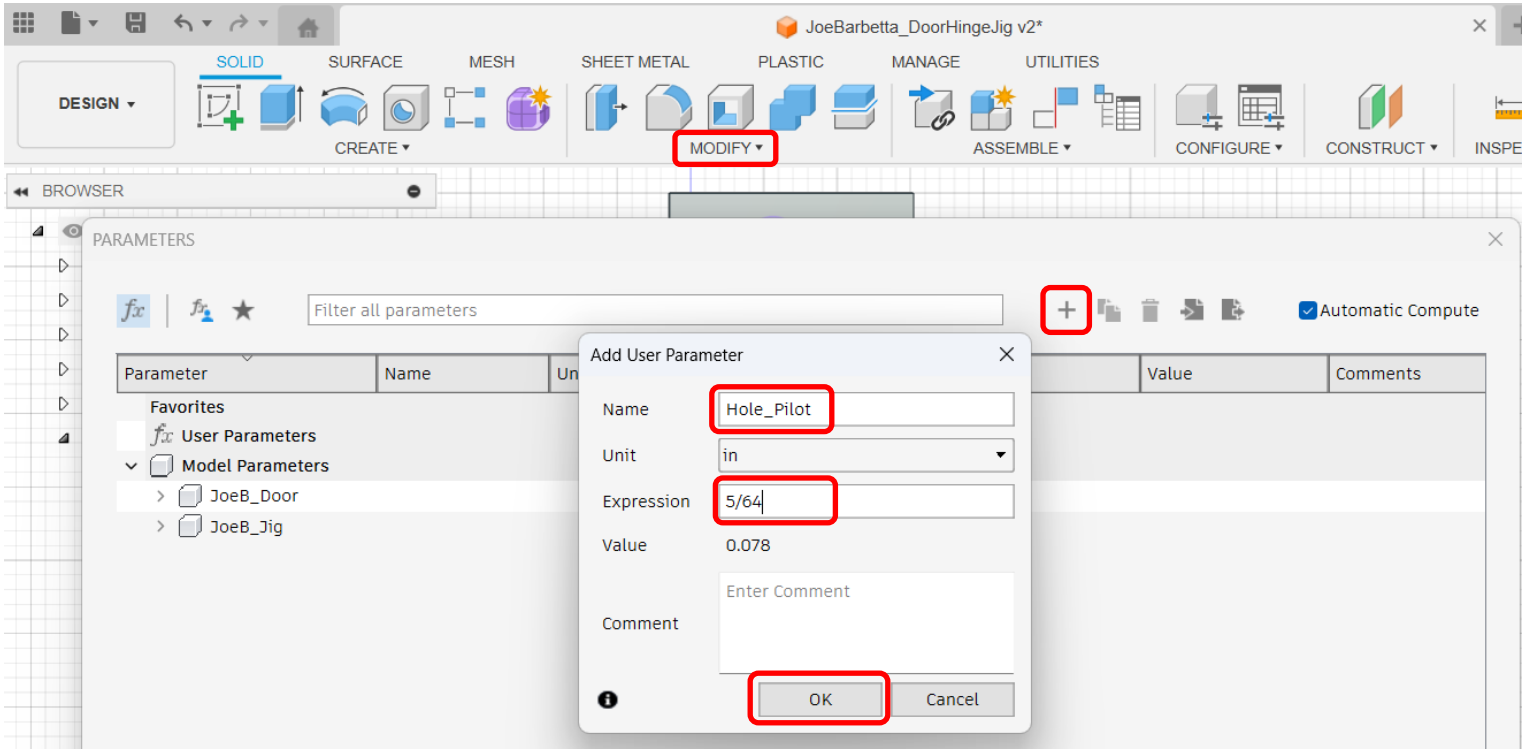


User Parameters

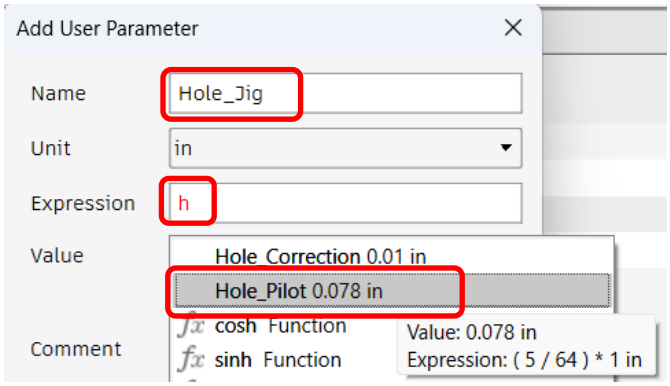
If there are multiple features with the same dimension, it can be helpful to define User Parameters. A parameter essentially works as a variable that is chosen for a dimension. This parameter can then be used as a dimension for multiple features, such as holes. For example, if a parameter is set for the diameter of a group of holes, and one wishes to change this diameter, the one parameter value can be changed and all the holes will be modified. Otherwise, one would have to manually redefine the dimension of each hole separately. This not only saves time, but also ensures that all the features using that parameter will be changed. When doing so manually, a feature could be missed.

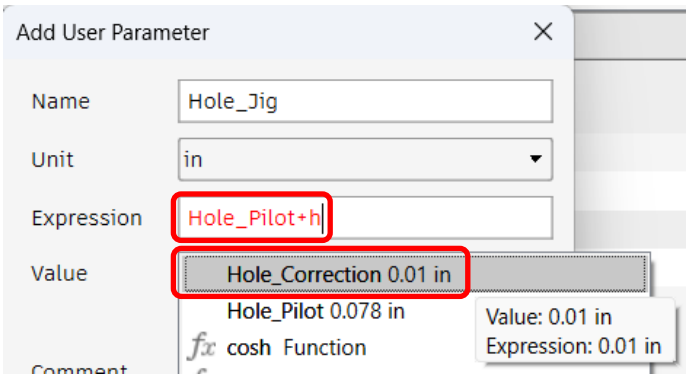
- from the **MODIFY** menu select **Change Parameters**, which appears near the bottom of the list.
- if a message window pops up, click its **OK** or **Close** button
- click on the **+** icon and enter **Hole_Pilot** for the **Name** and **5/64** for the **Expression** and then click **OK**.
- click on the **+** icon again for **Hole_Correction** and **0.01**

Consider using names that can be easy to access. For example, one can start each dimension associated with a hole starting with "**Hole**". When later applying a parameter, typing **h** will show all the **Hole** parameters. It should also be noted that a fraction or formula can be used for the Expression.

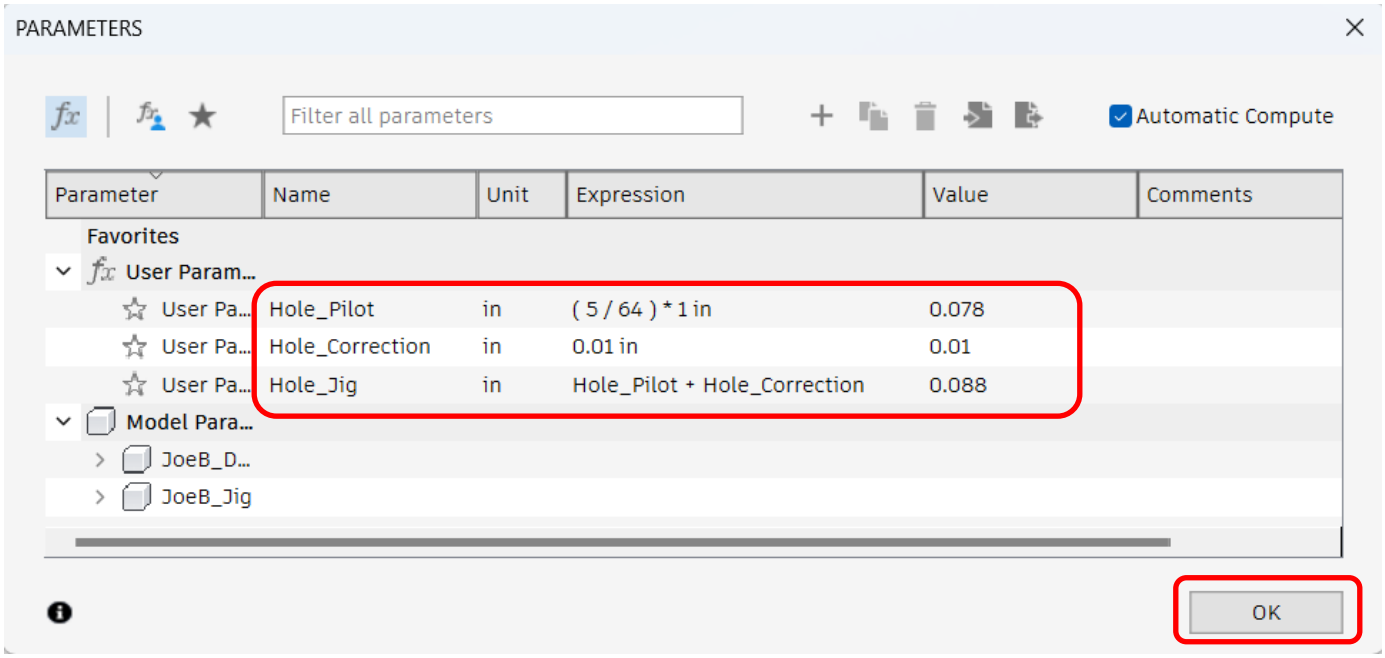


- use the **+** icon again to create a 3rd parameter named **Hole_Jig**
- in the expression box type **h** and select **Hole_Pilot**
- then type a **+** sign and then type **h** and select **Hole_Correction**, as shown on the right picture and click **OK**

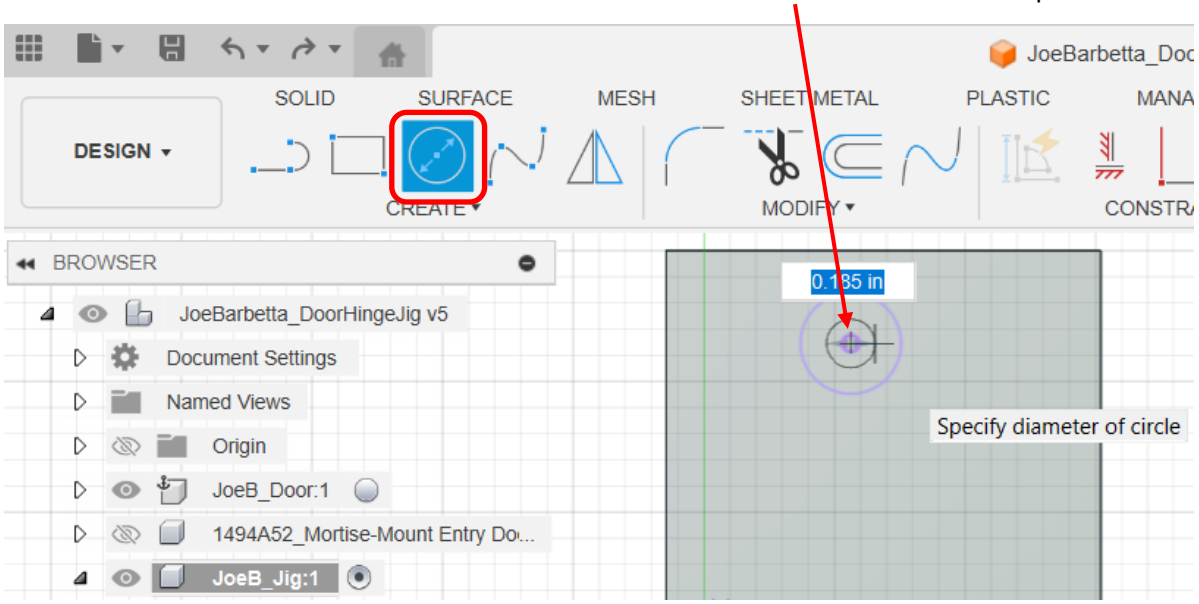




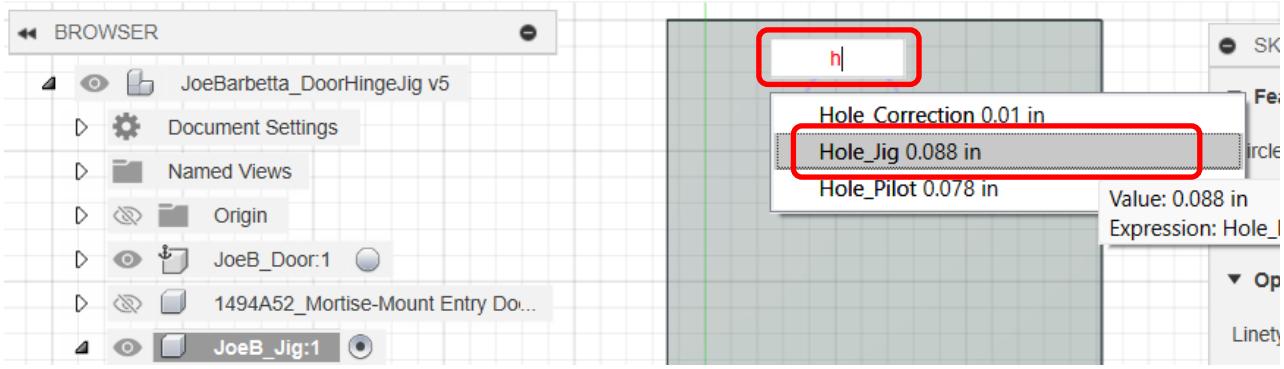
The 3 parameters enter should appear as below.
- click **OK**



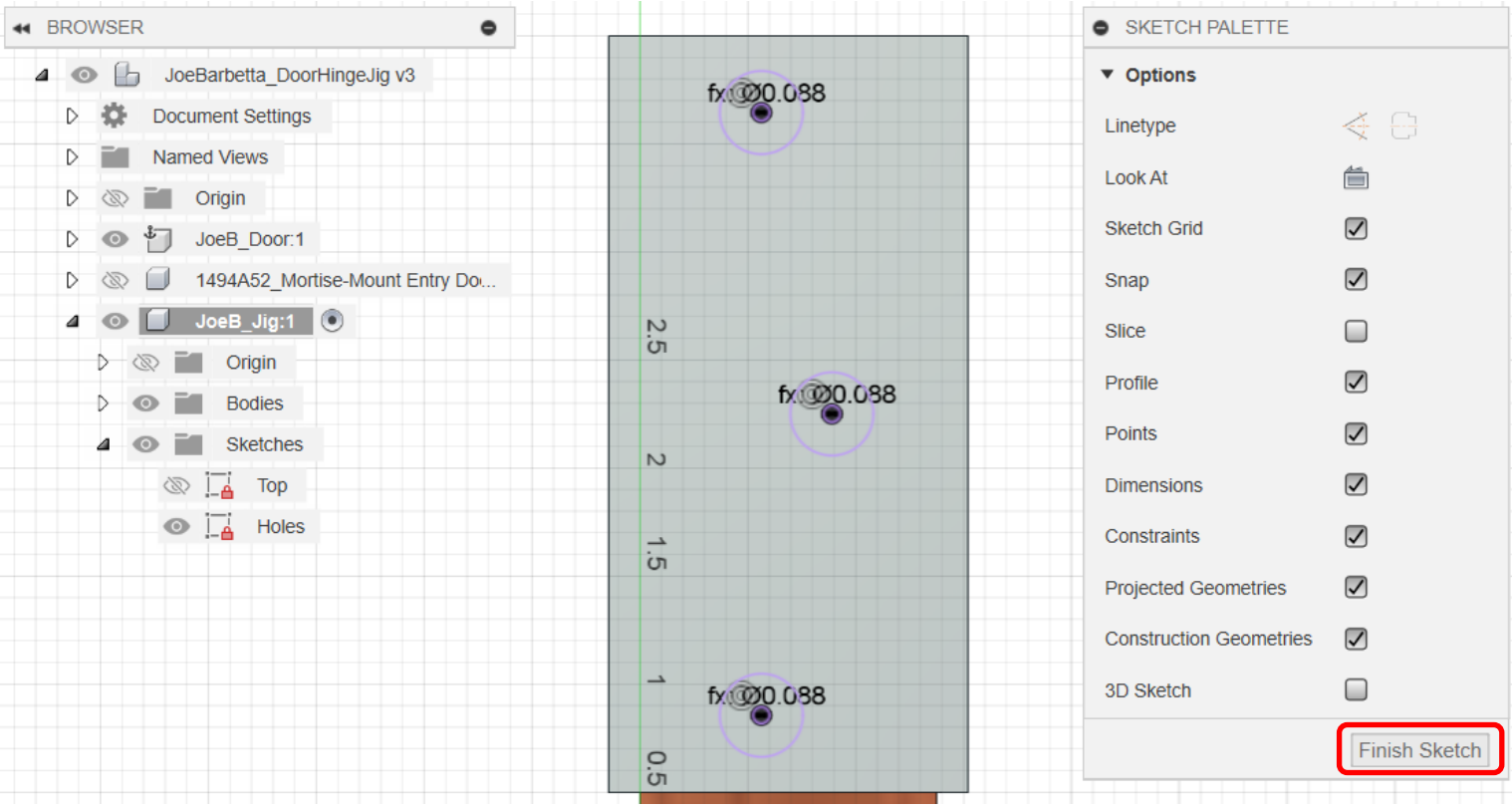
- if the Sketch mode has been exited, one can right click on the **Holes** sketch and select **Edit Sketch**
- select the **Center Diameter Circle** tool and click on the **center of the violet circle** and pull the circle edge out



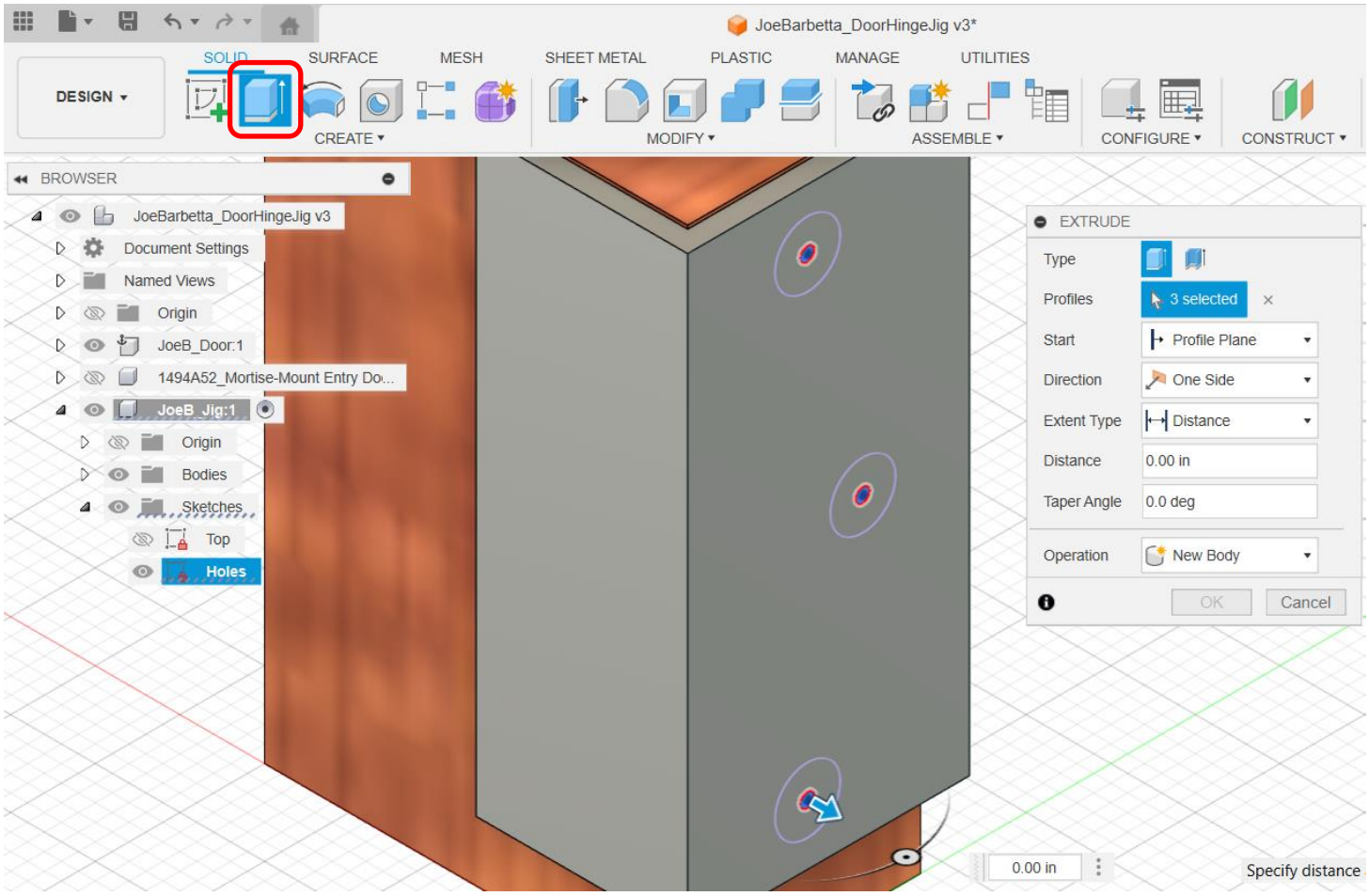
- type **h** and then select **Hole_Jig 0.088** in from the list



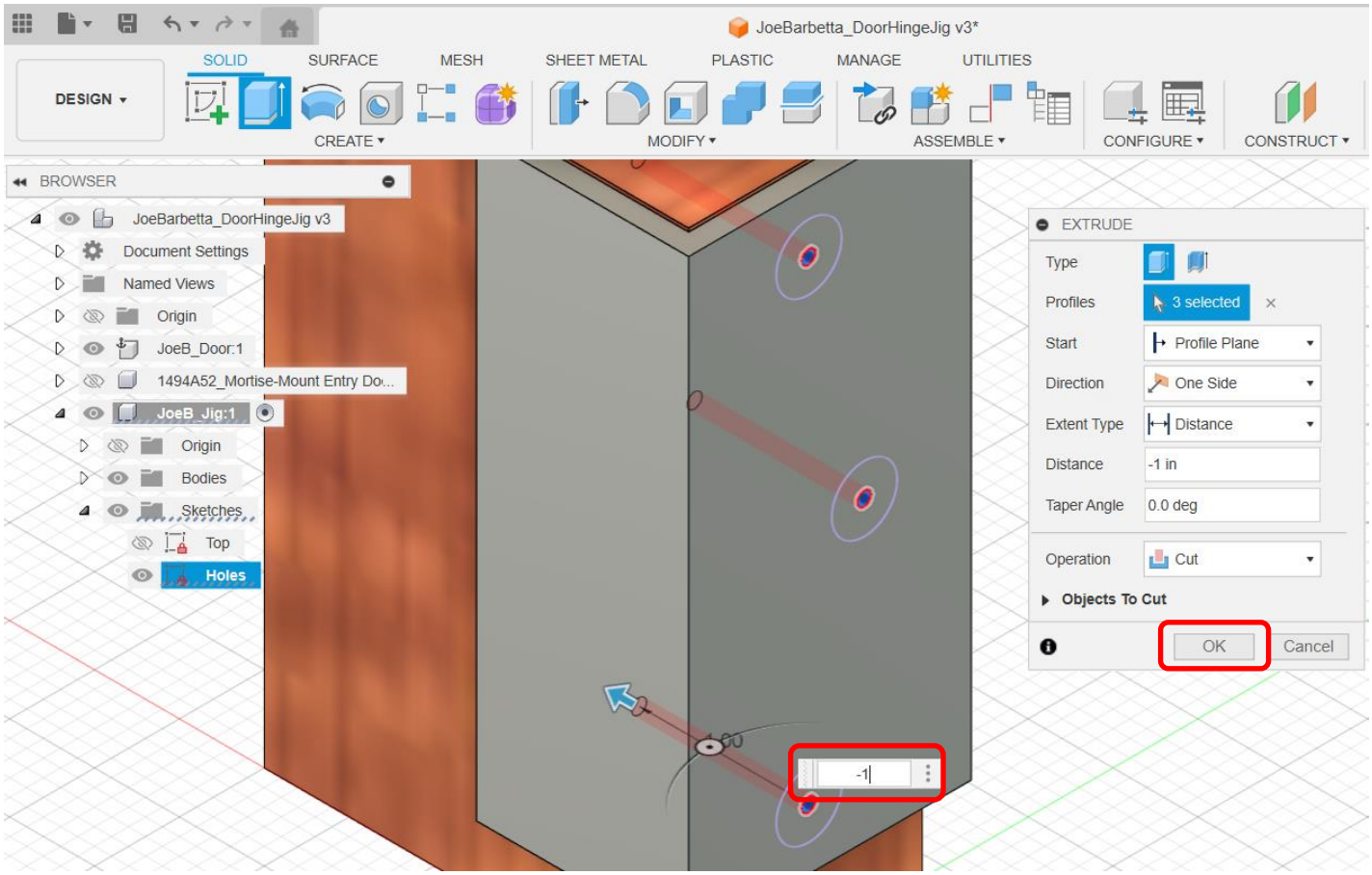
- perform the same steps for the **other two holes** and then click **Finish Sketch**



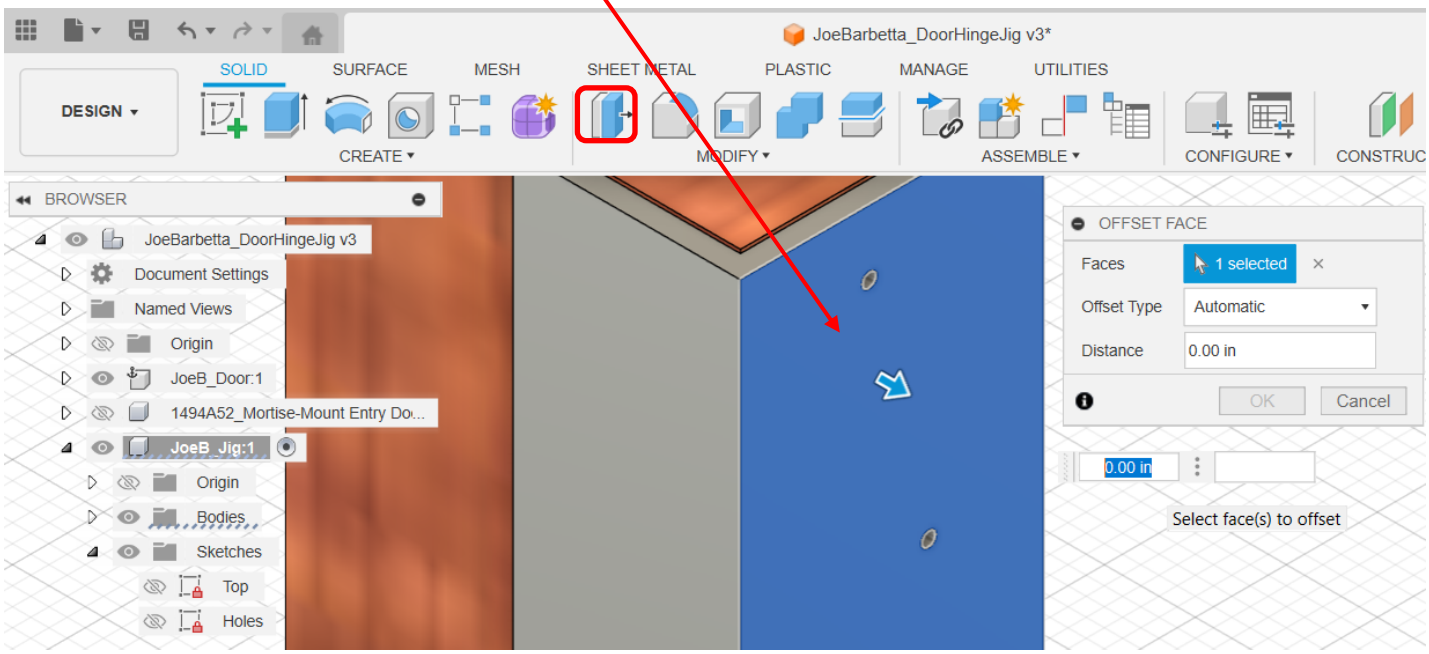
- click on the **Home icon** at the **View Cube** and adjust the view similar to that below
- select the **Extrude** tool and click on the **three small circles**



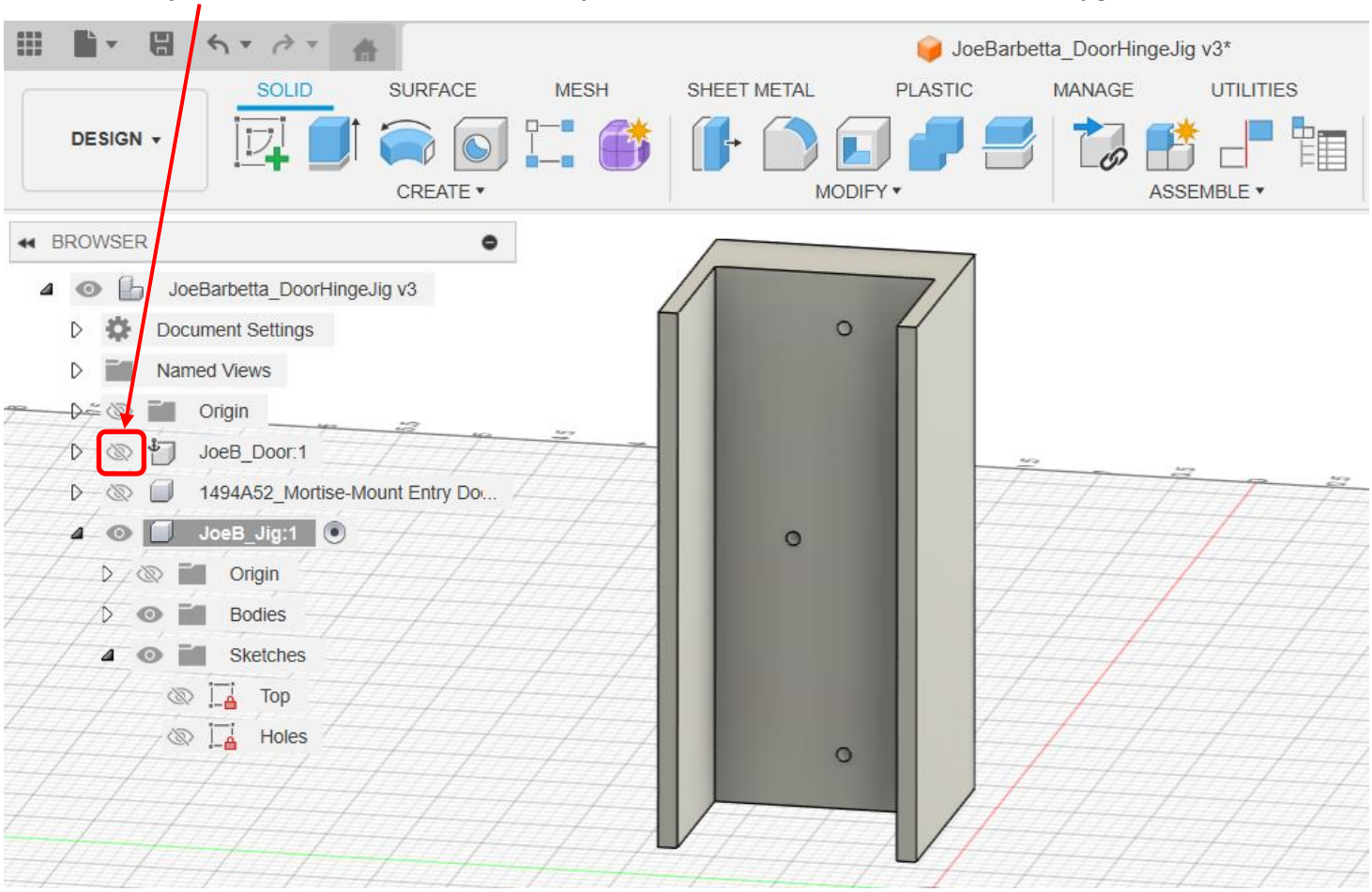
- type **-1** (note the minus sign) and click **OK**



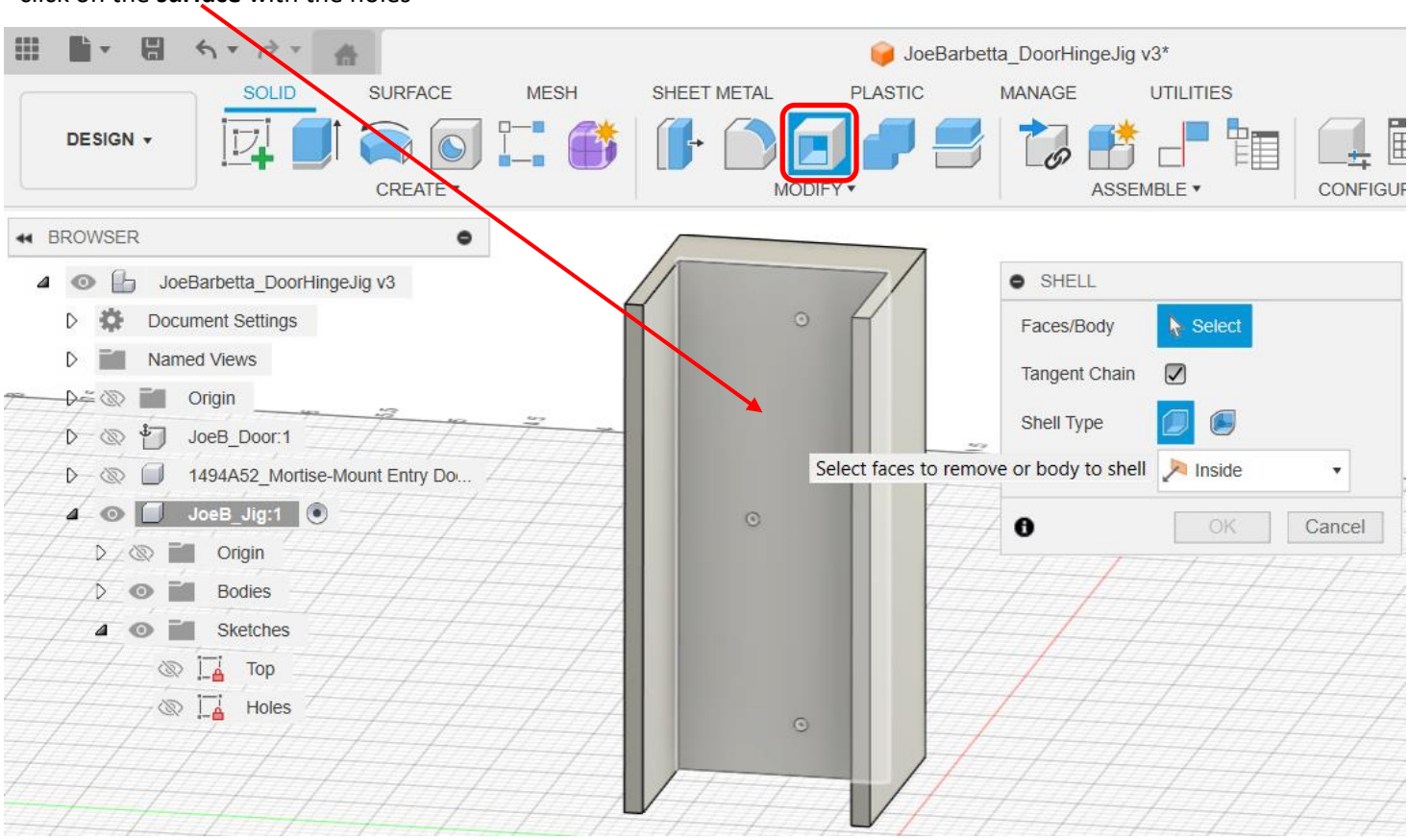
- select the **Press Pull** tool and click on the **surface with the holes** and type **0.5**, and click **OK**.
This was done to create deeper holes in the jig to help keep the drill bit perpendicular to the door surface.



- click on the **eye icon** for the **Door** to hide it and **adjust the view** to see the other surface of the jig

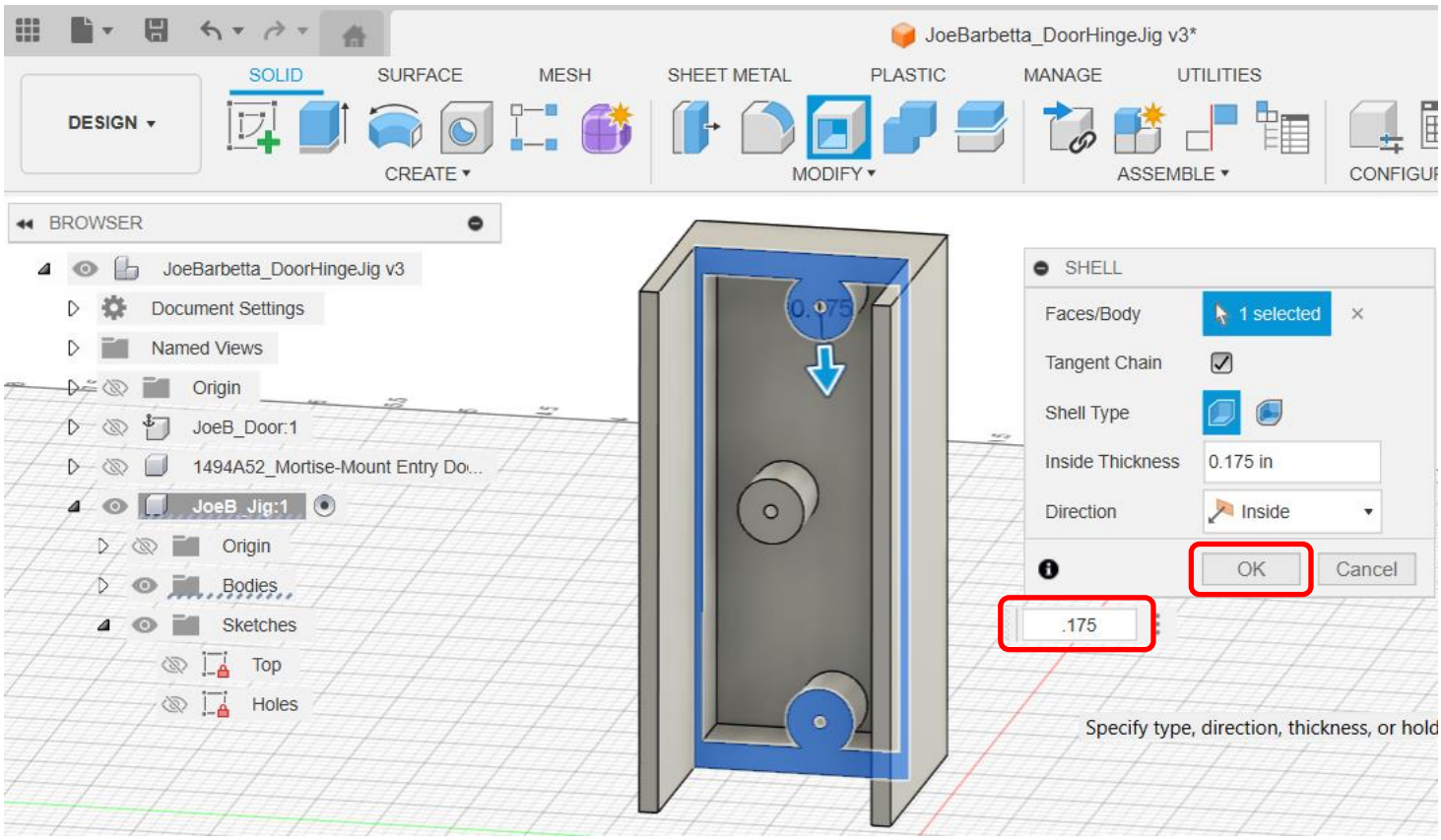


- select the **Shell** tool. If the tool isn't visible one can find it in the **MODIFY** menu.
- click on the **surface** with the holes

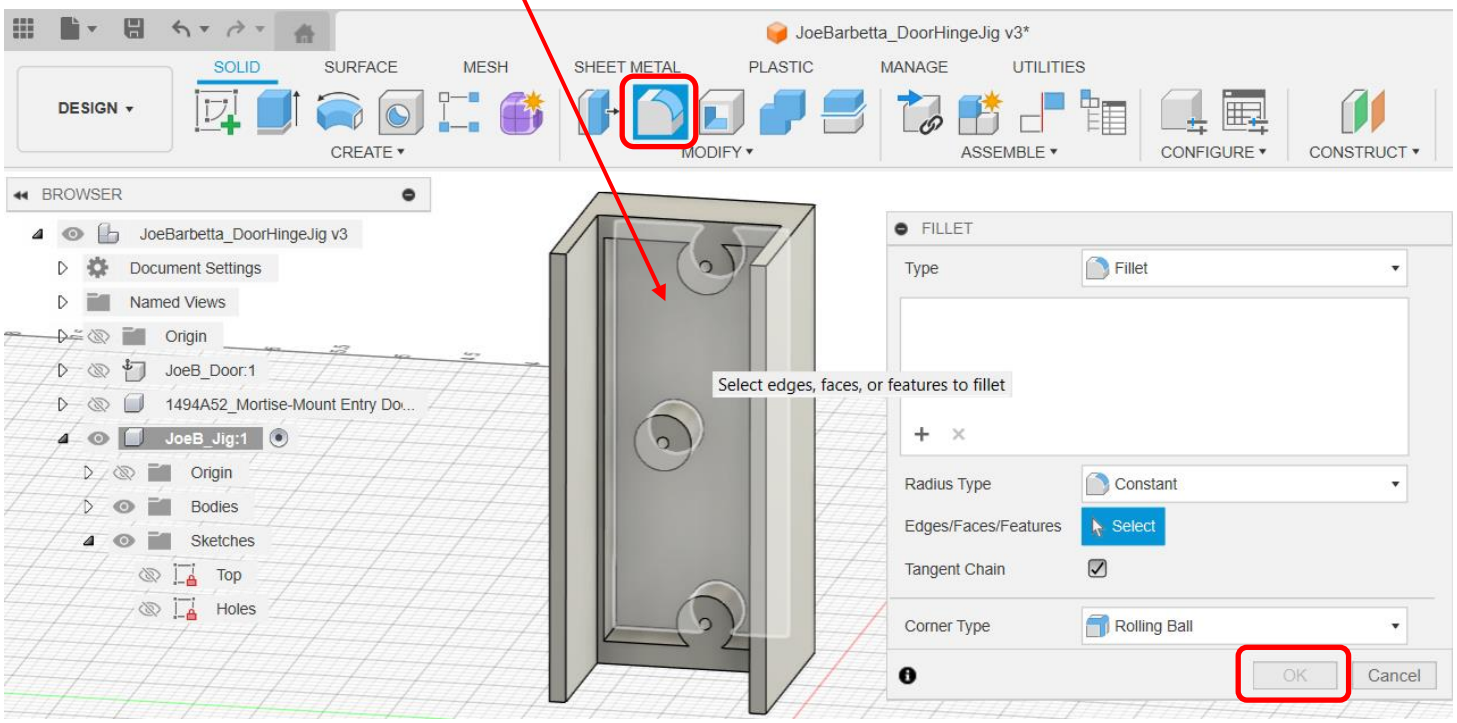


- type **0.175** and click **OK**

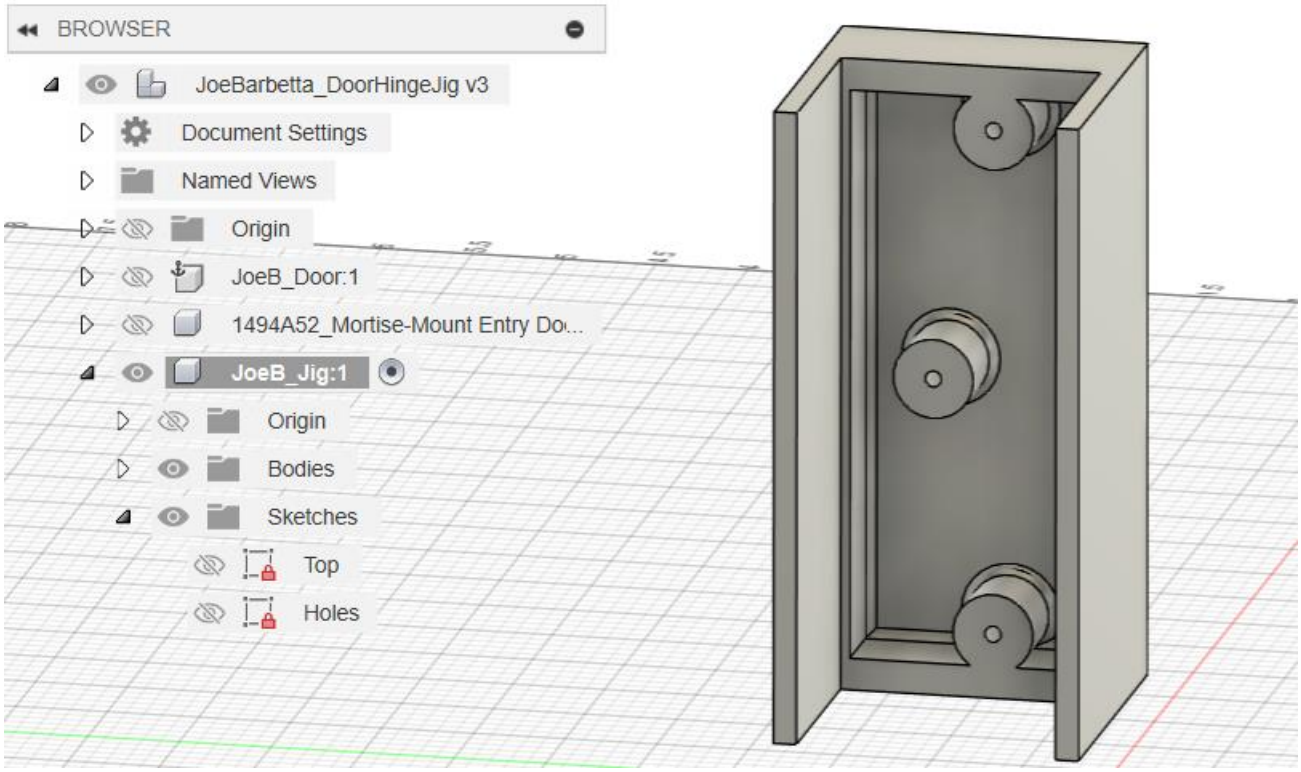
This is done to reduce the amount of plastic use. It also makes the design look cool.



- select the **Fillet** tool and click on the **surface** as indicated
- type **0.05** and click **OK**

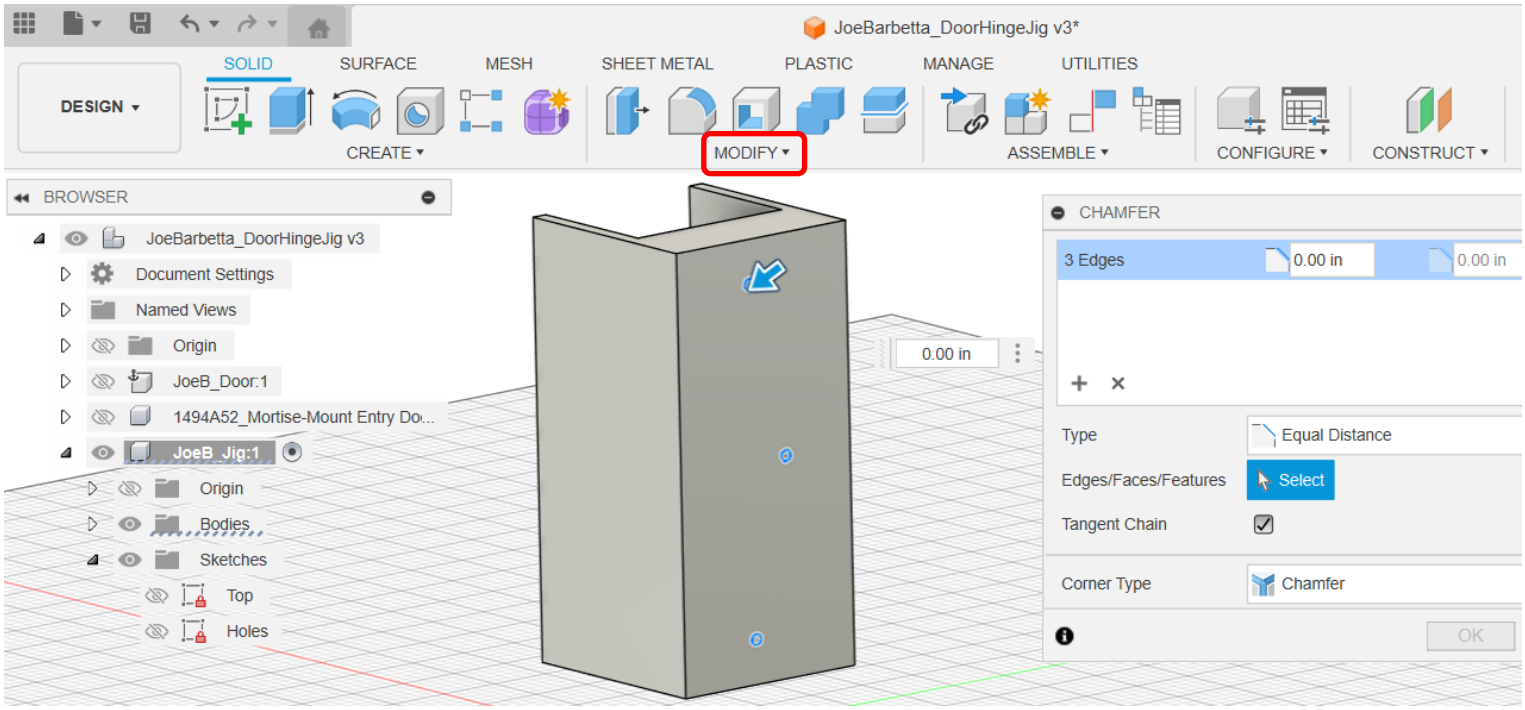


This is the result. The purpose of the fillets adds some strength.

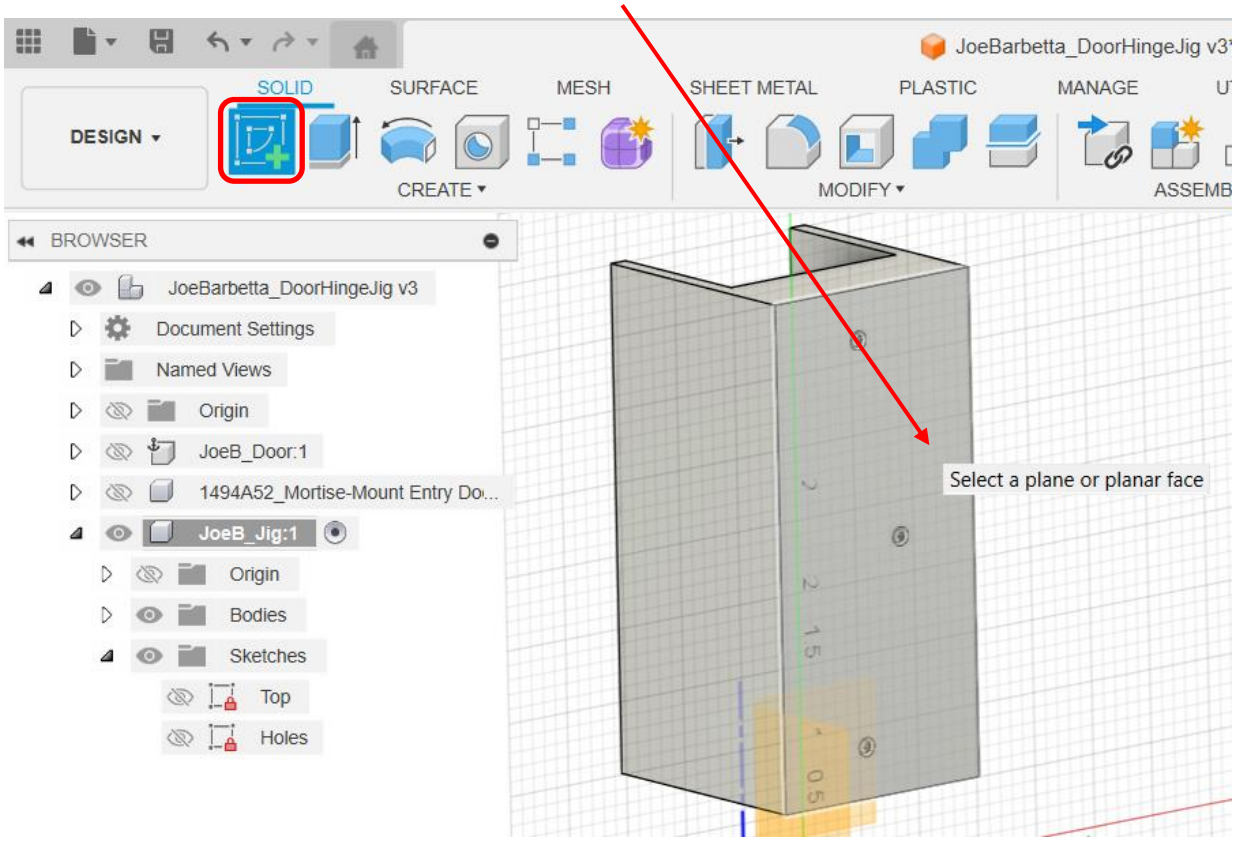


- use the **View Cube** to achieve a view similar to that below.
- under the **MODIFY** menu select **Chamfer**
- type **0.02** and then click **OK**

This chamfer will make it easier to insert a drill bit into the hole.

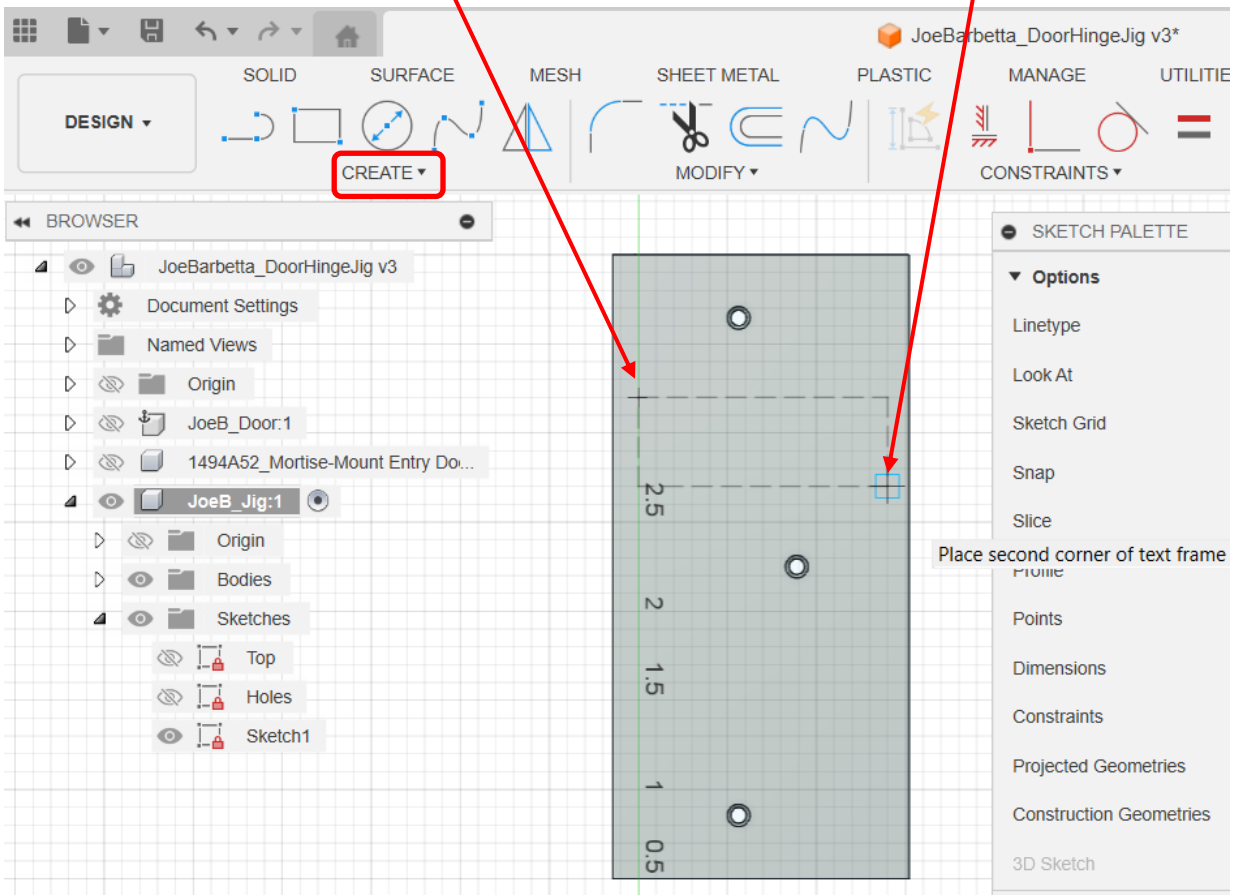


- select the **Create Sketch** tool and click on the **surface with the holes**

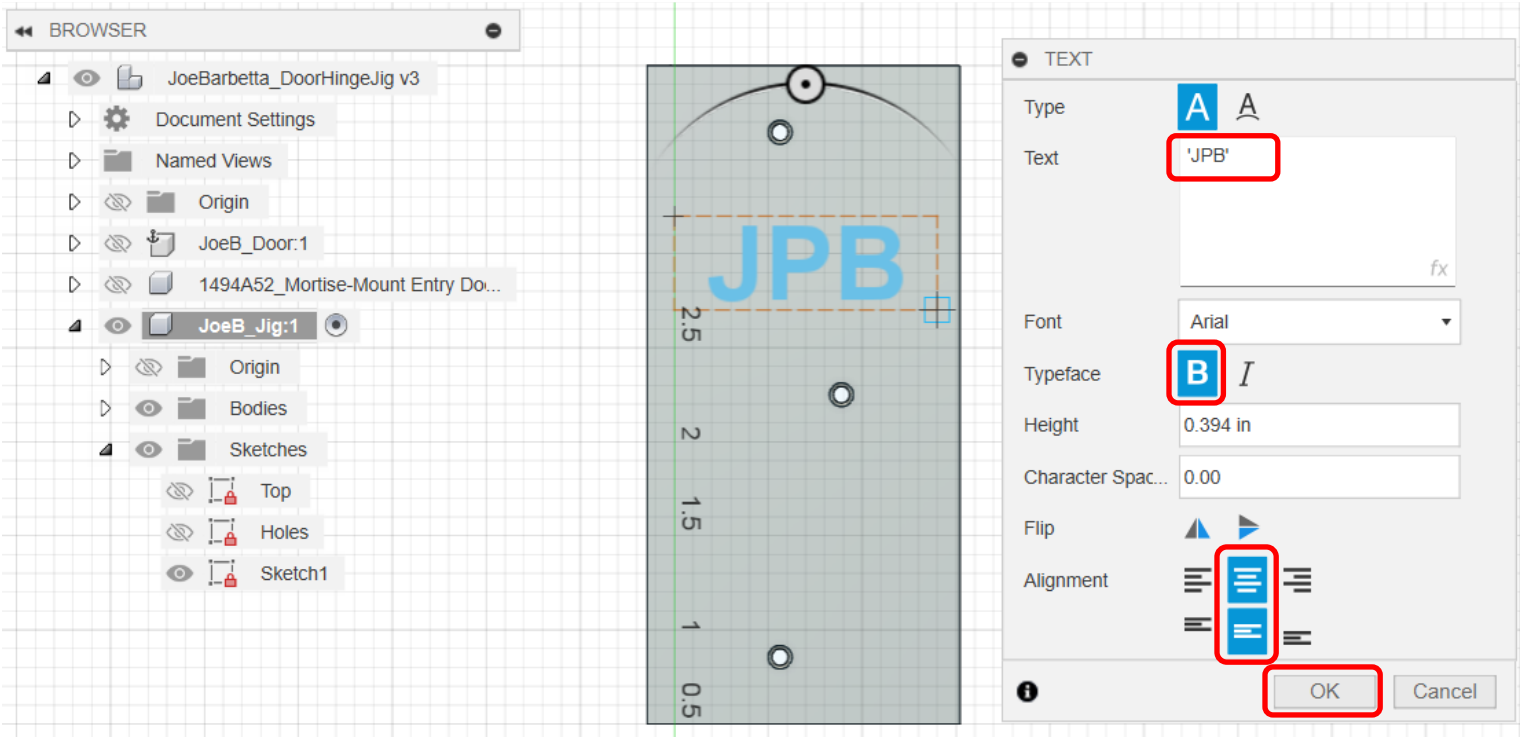


- under the **CREATE** menu select the **Text** tool
- click on a location near that indicated and extend the other corner to the right and down and click again

Note that the position of this rectangle is Not critical. It is just to position the text.

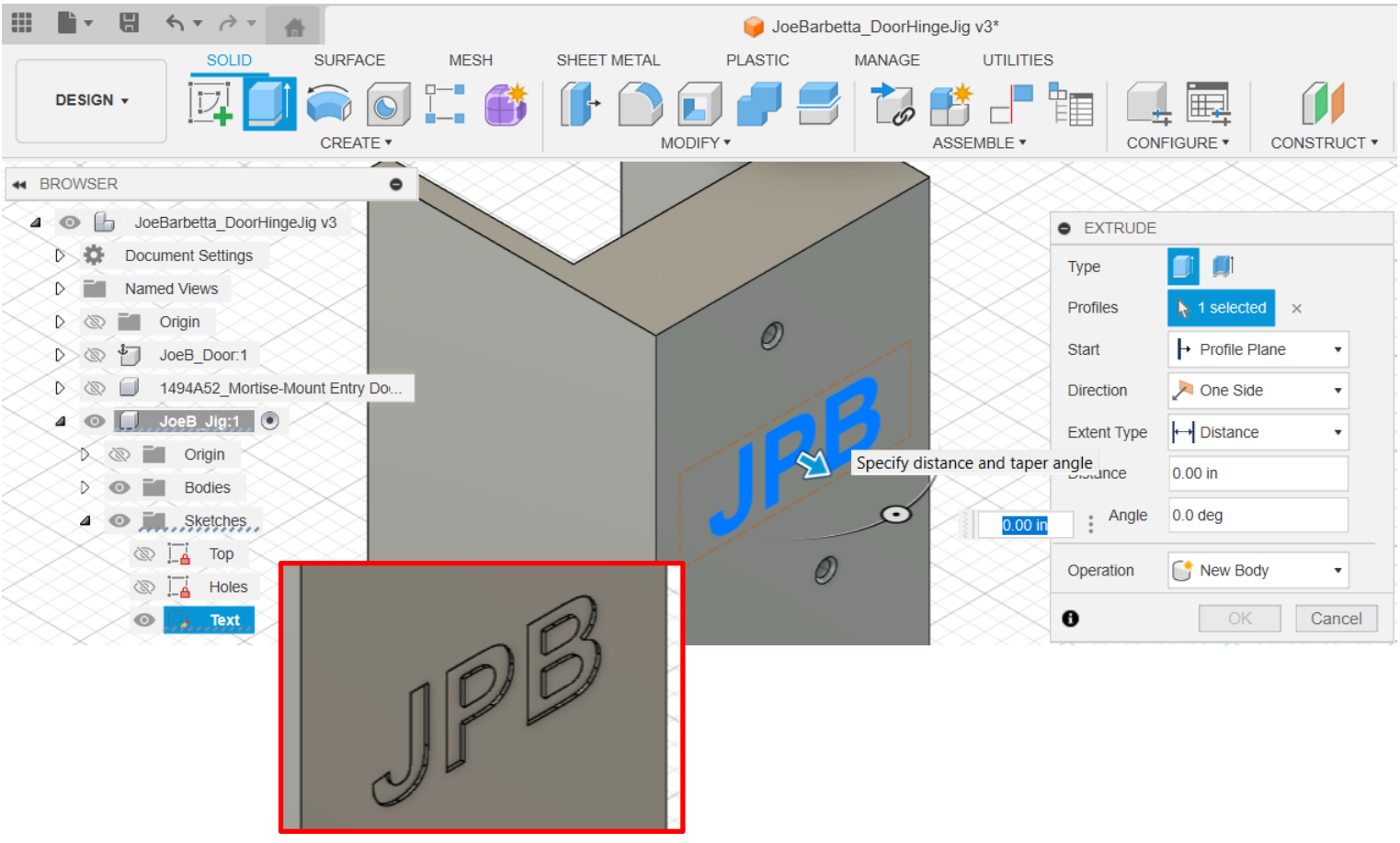


- in the Text box enter your initials
- click on **B** to make the text bold and the **middle Alignment icons** and click **OK**

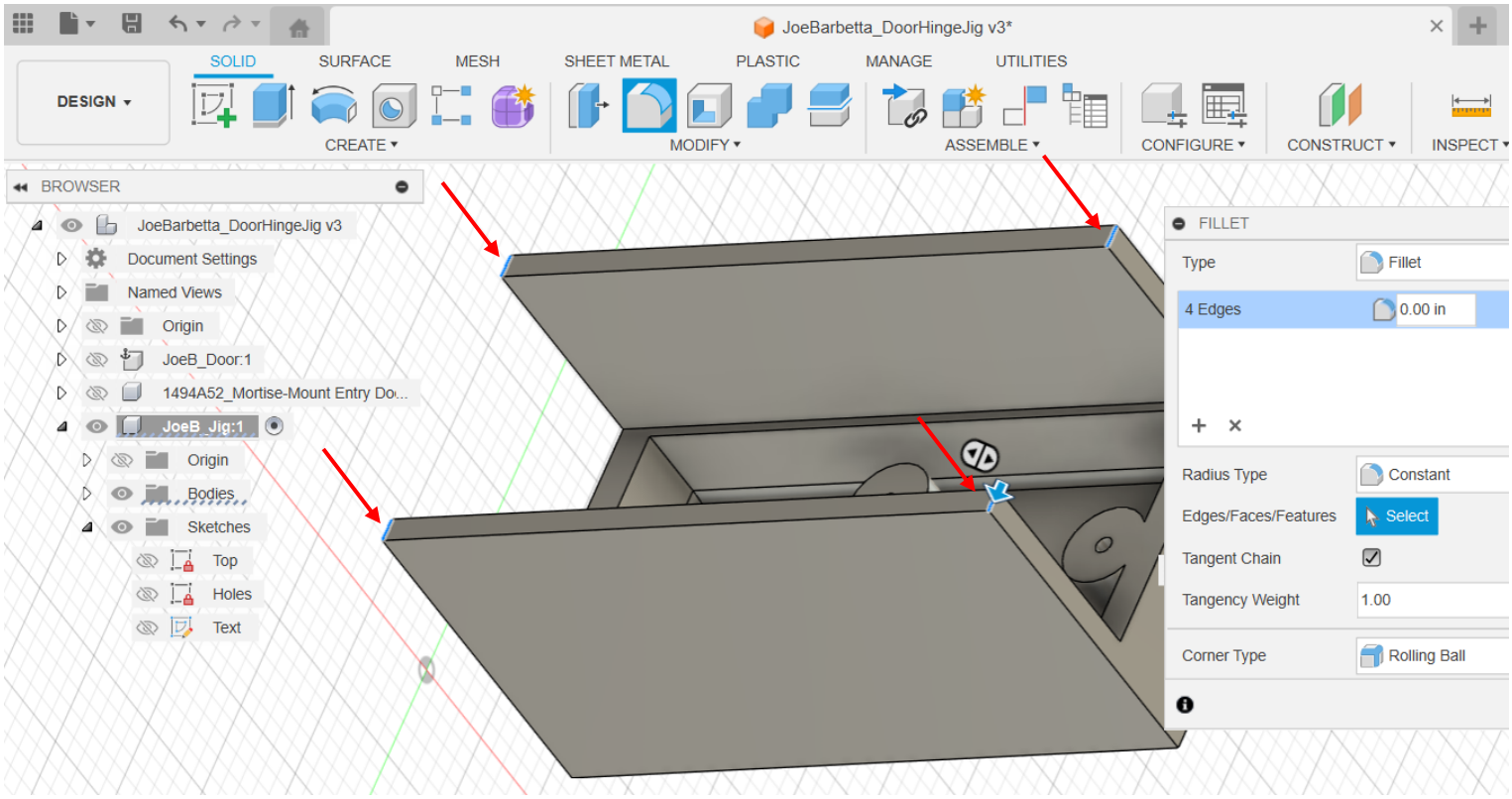


- select the **Extrude** tool and click on one of the letters
- type **-0.02** (note the minus sign) and click **OK**

The text should be embossed inward and shown in the inset picture. If it is protruding out, the minus sign may not have been used. This is very important because this will be the bottom surface on the 3D Printer build plate.

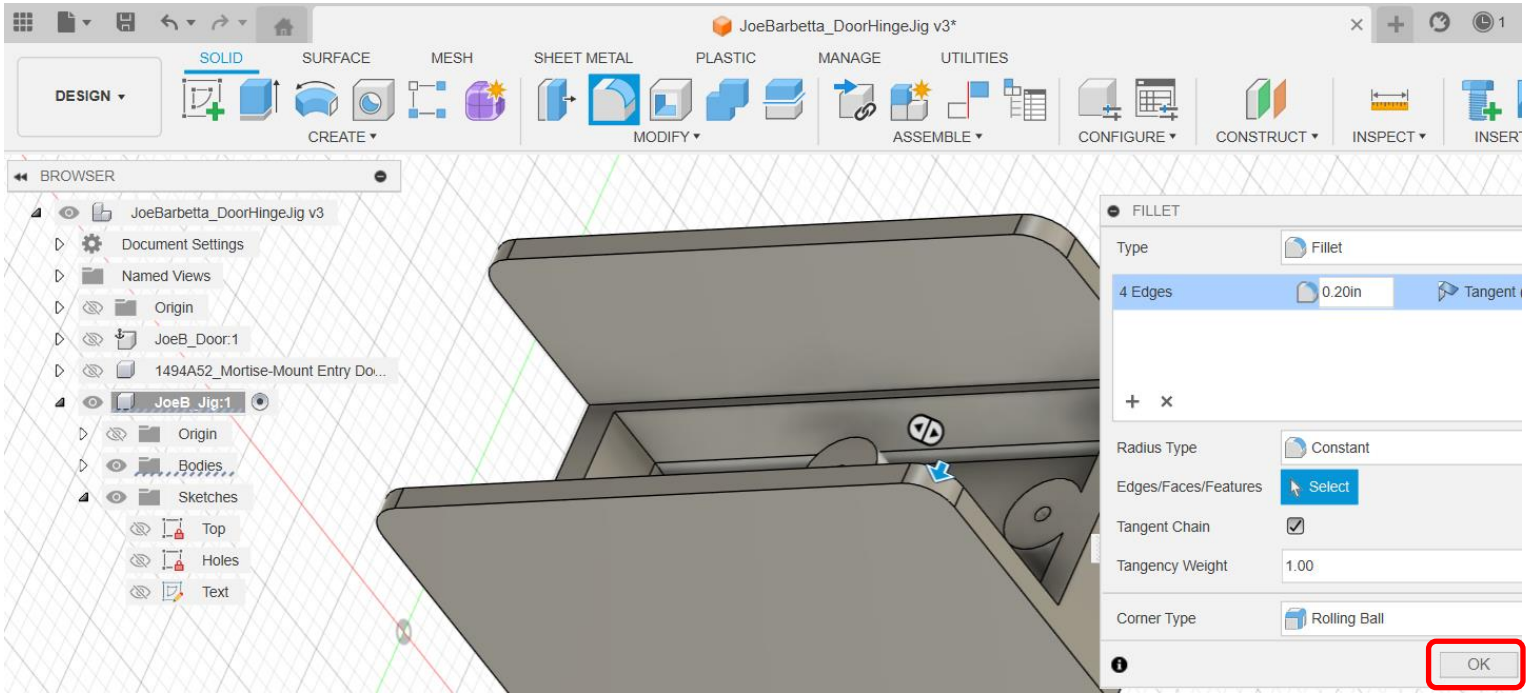


- adjust the view similar to that below
- select the **Fillet** tool and click on the **4 small edges**



- type **0.2** and click **OK**

These filleted edges will help prevent scratching of the door surface. It also look neat.



Creating a Test Print

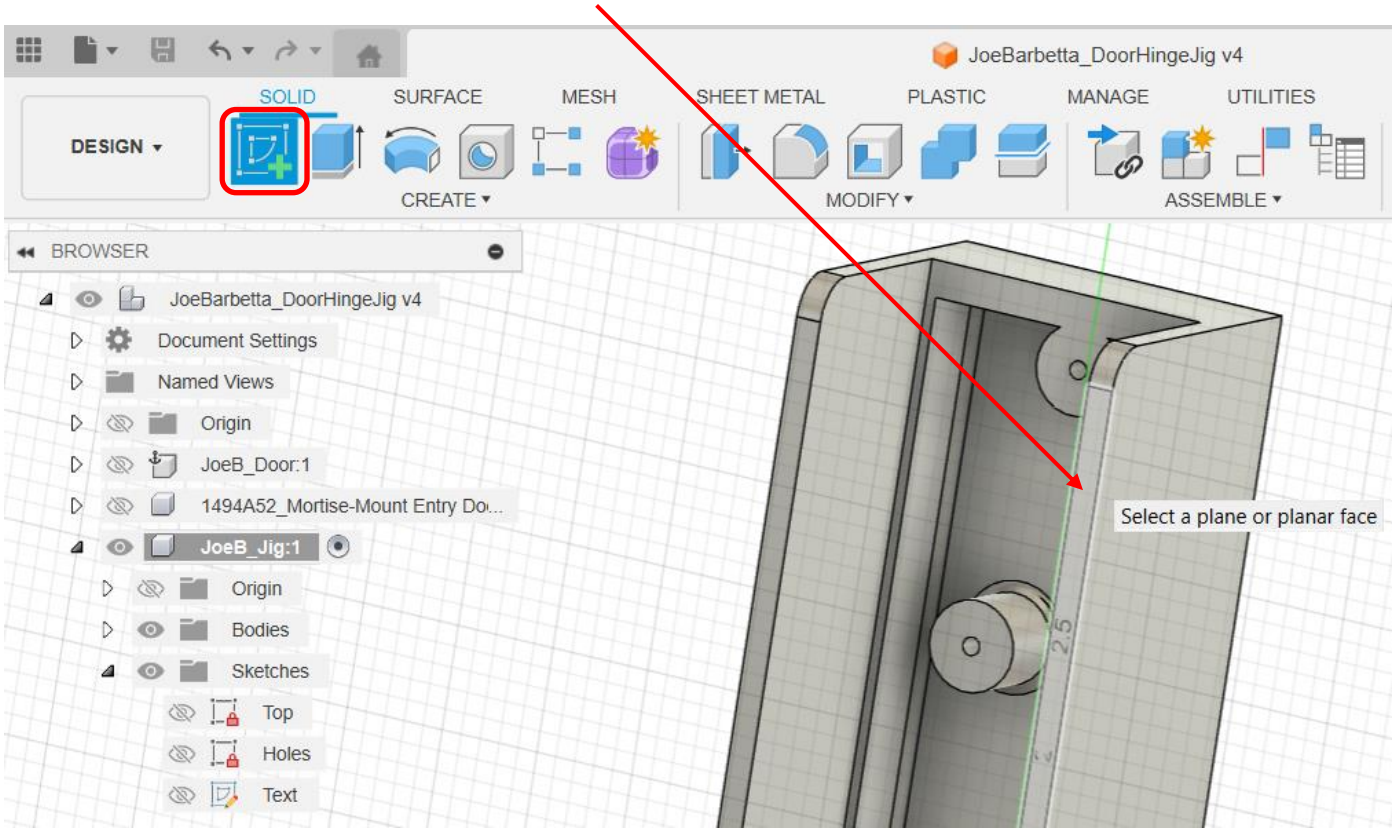
There are two dimensions that are important. The hole diameter and the distance between the extensions that hug the door. If the hole is too small the drill bit will not fit or create a lot of friction, which will start softening the plastic.

Because plastic is extruded through a small nozzle, often 0.4 mm, it can expand slightly. This expansion have vary due to may factors, such as the printer used, settings, filament, etc. Thus the resultant hole diamter can vary up to 0.02”. Note that the **Hole_Correction** parameter, which is initially set to **0.01**, compensates for the hole typically being smaller than that desired.

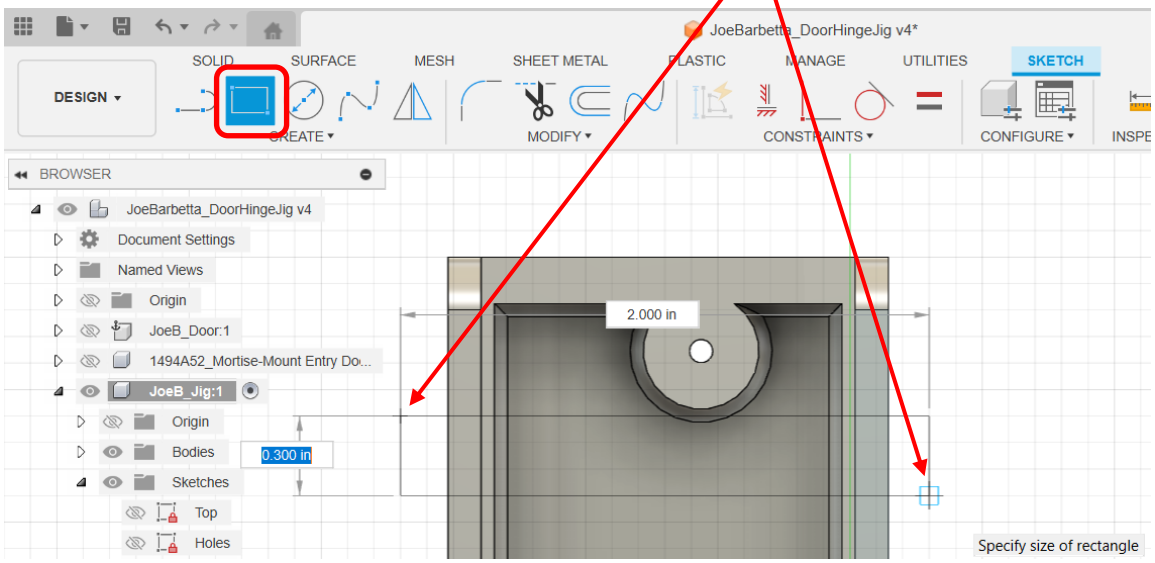
It should be noted that the position of the center of the holes is generally very accurate and that is why 3D Printing is a great solution for creating jigs.

Instead of printing any part in its entirety, it is prudent to cut down the object to print only features, for which dimensions need to be verfified. This will reduce the amount of wasted plastic and time if adjustments must be made. If features, such as holes, need adjustments they can be applied to the design before a final full print is done.

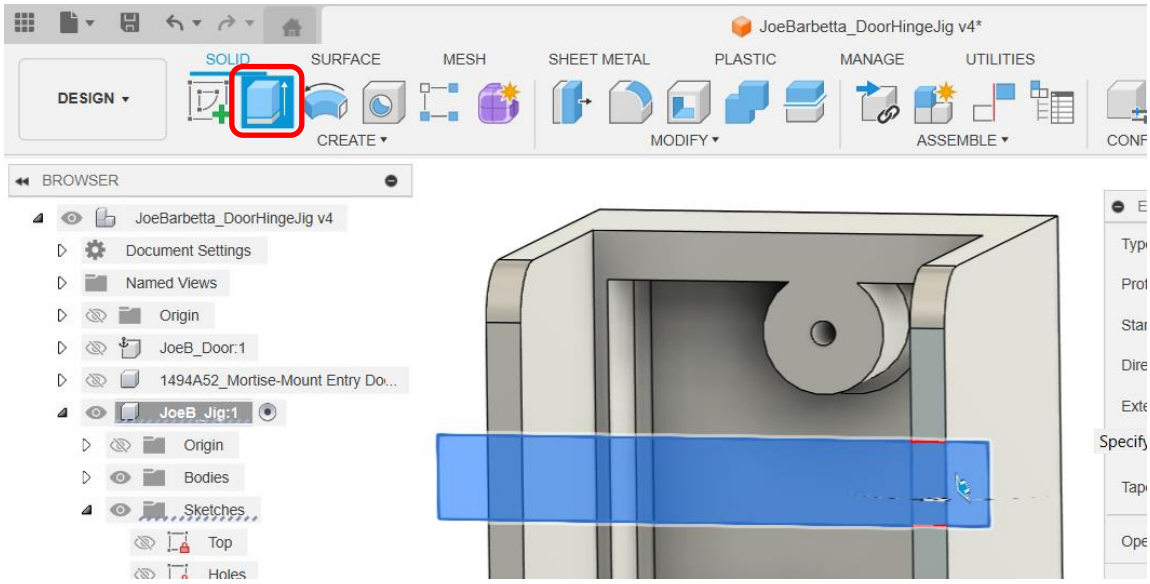
- select the **Create Sketch** tool and **click on the surface** as indiated



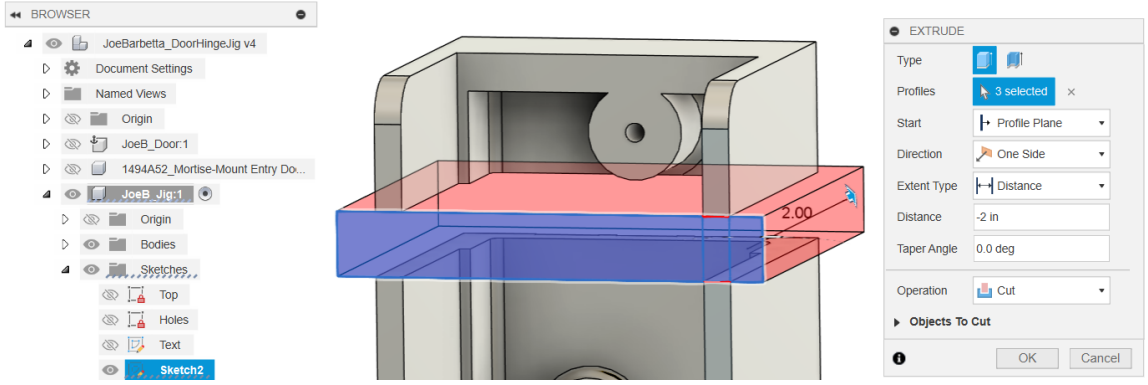
- select the **Rectangle** tool and draw a rectangle **between the two points** as indicated
The postion of the rectangle is not critical. It should be under the top hole.



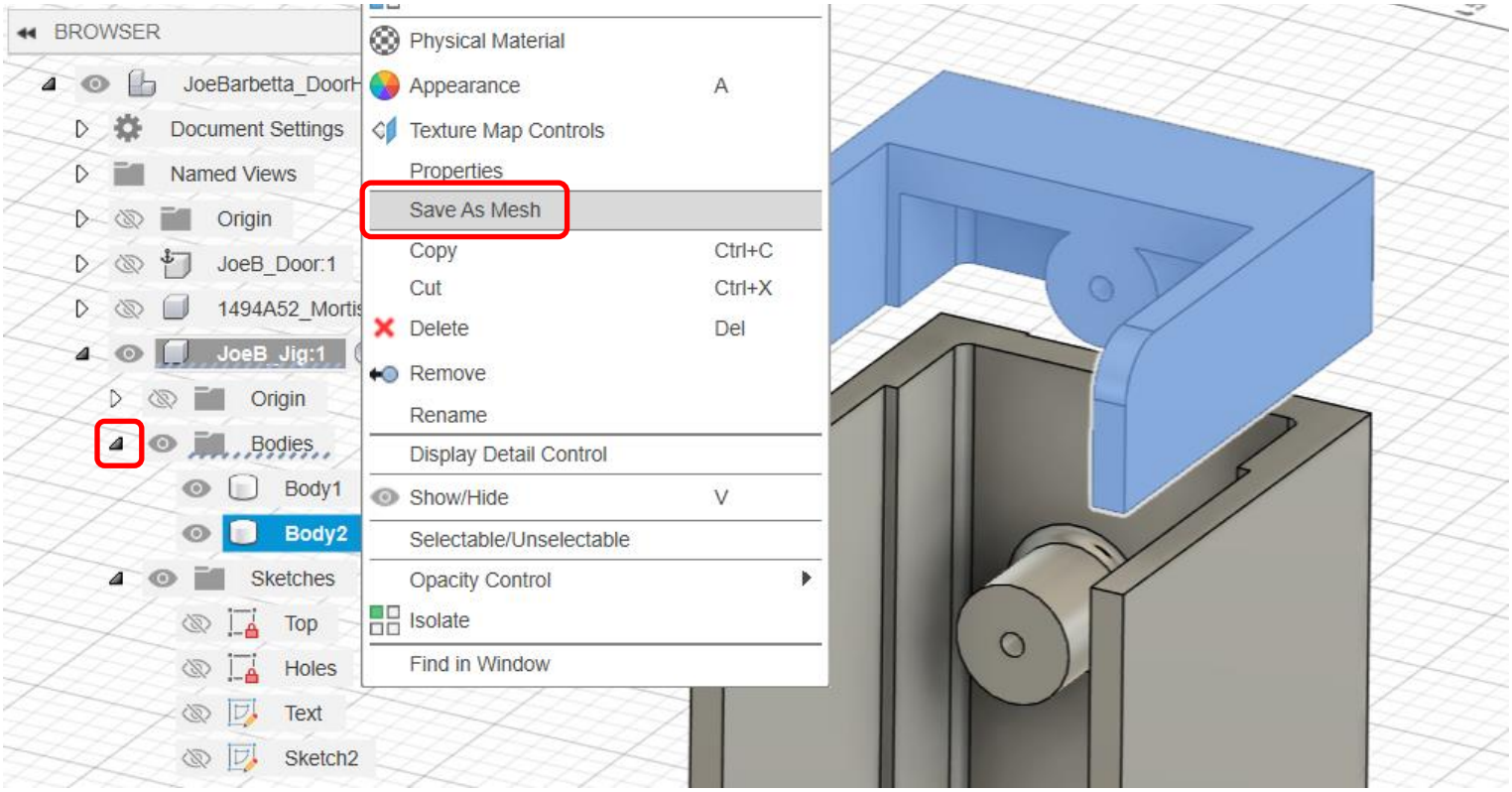
- select the **Extrude** tool and click on multiple regions of the rectangle so that it is **fully dark blue** as shown



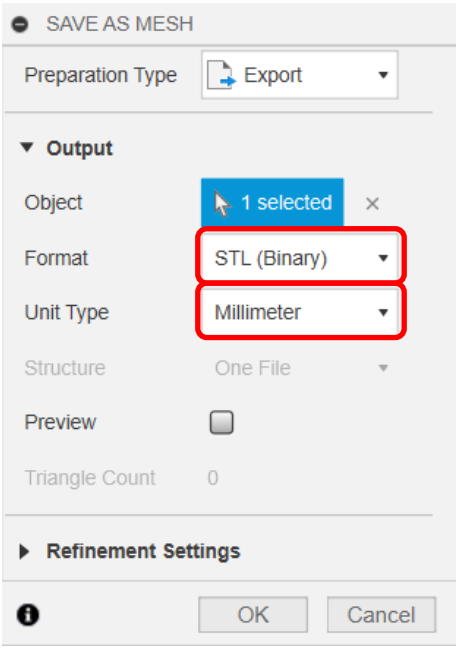
- type **-2** (note the minus sign) and click **OK**
The 2 inches isn't critical. This operation is used to cut all the way through to separate the body into two bodies.
Note that there are other methods to split a body, such as the Split Body tool. Extruding a sketch, such as that done here provides more flexibility in isolated sections for a test print.

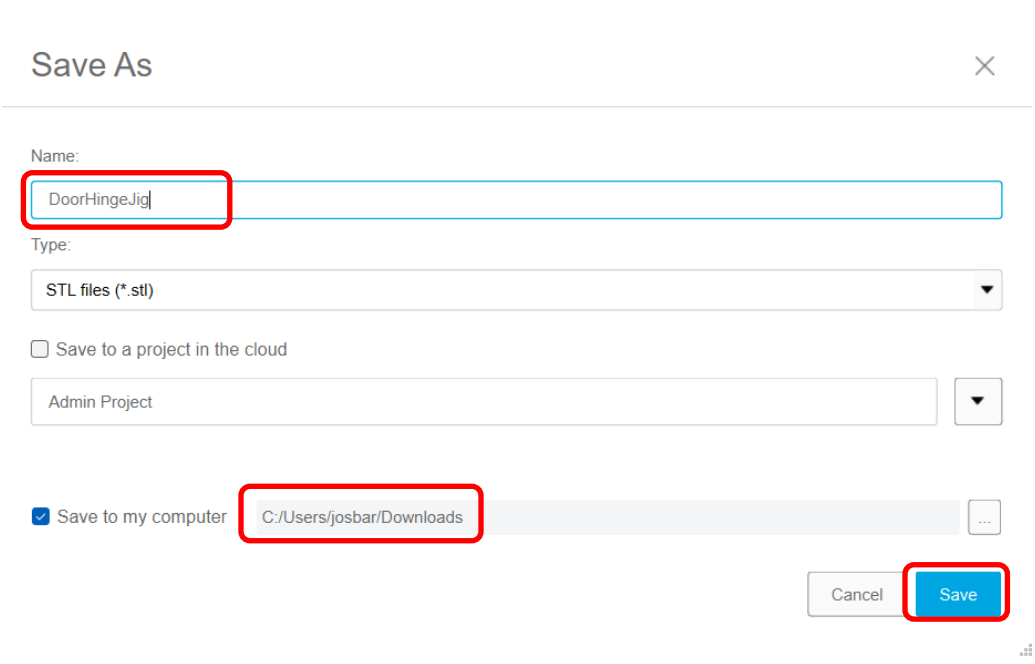


- click on the **arrow** next to the **Bodies** folder
- right click on the body for the smaller section, which will trun it blue, and select **Save As Mesh**

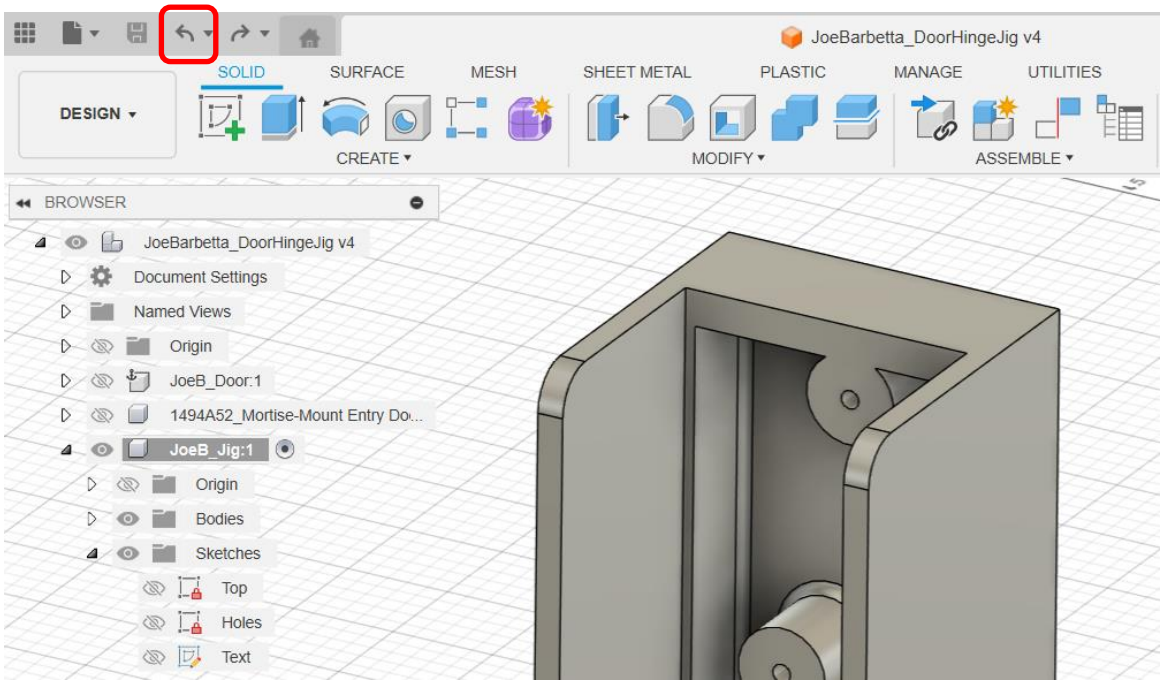


- ensure that the **Format** is **STL (Binary)** and the **Unit Type** is **Millimeter** and click **OK**
- enter a **name** for the test object
- make note of the save location, it is normally Downloads, and click **Save**





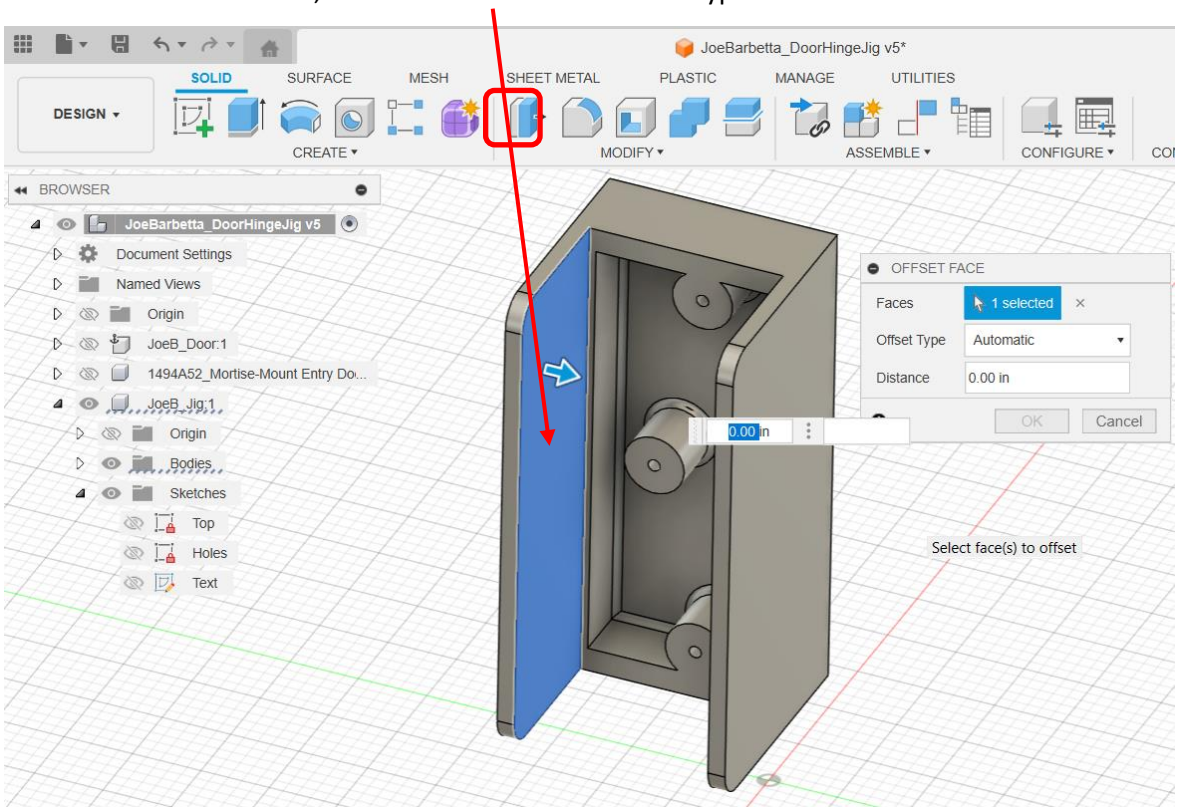
- click the **Undo** icon multiple times until the last extrude and sketch is undone to restore the orginal part



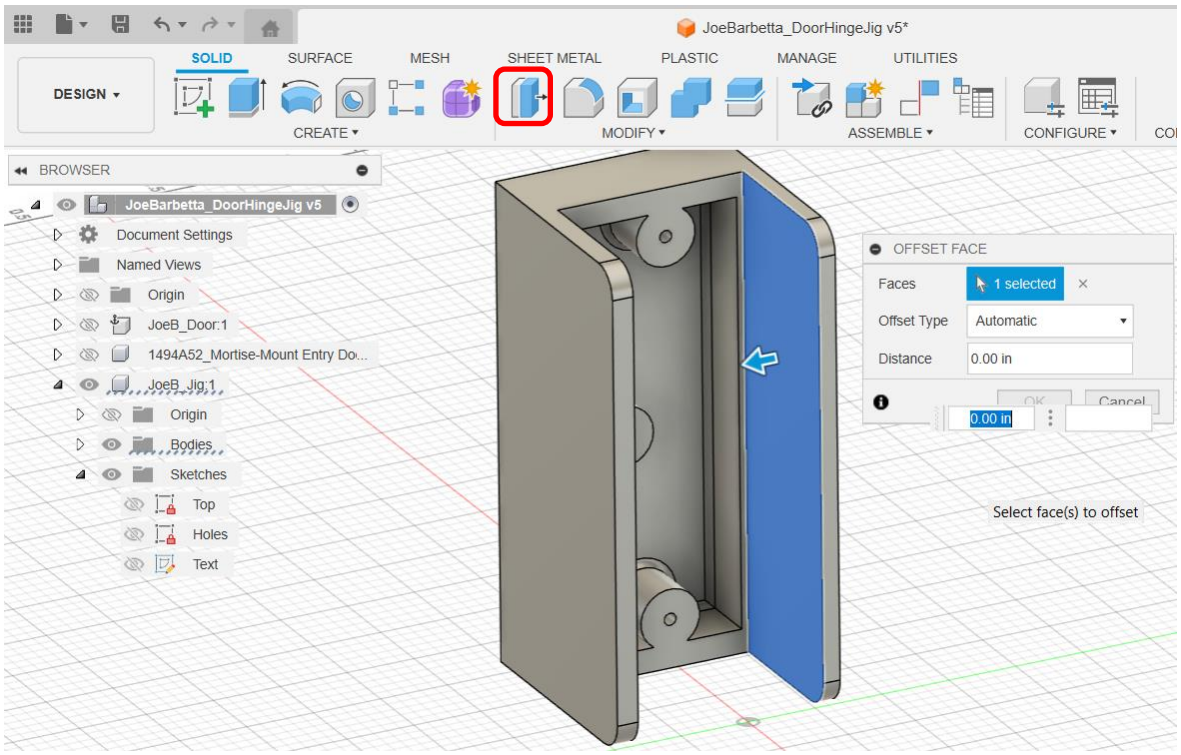
Making Adjustments

Imagine you place the test print on the edge of the door and it seems too loose and you determine that it is about 0.06” too wide. Also imagine that the hole is a little too tight on the drill bit. Enlarging it by 0.010” should fix this. We didn’t set parameters that would affect the fit on the door, but we can force surfaces, called Faces in Fusion, to achieve our adjustments. If we move each surface in by 0.03” we will achieve a 0.06” reduction. We do want to move each surface in the same amount to not shift the hole layout.

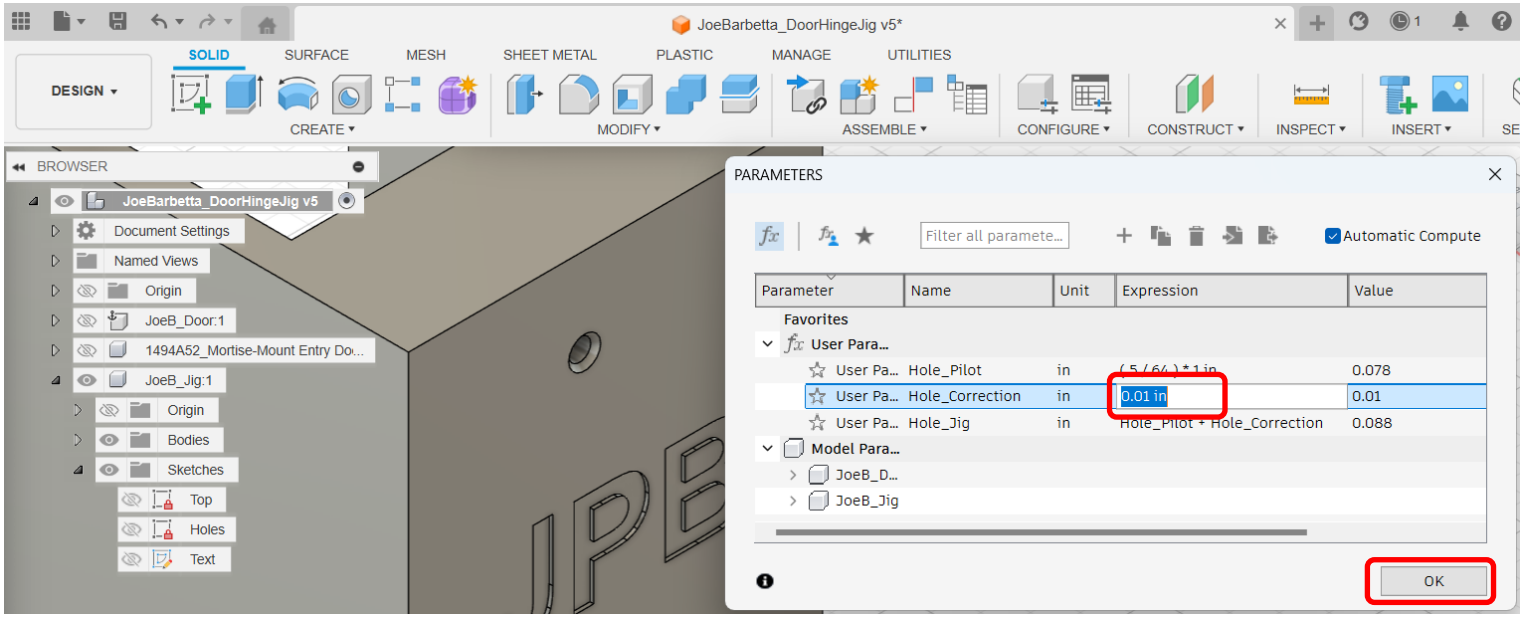
- select the **Press Pull** tool, click on the face as shown and type **0.03** and click **OK**.



- adjust the view to access the opposite surface
- use the **Press Pull** tool again to pull this surface out by **0.03** as well



- click on the Home icon near the View Cube and then zoom into one of the holes
- under the MODIFY menu select Change Parameters near the bottom of the list
- click on the **0.01** for **Hole_Correction**, change it to **0.02**, look at the hole, and press the **Enter key**
- Yell out. **That was amazing!**
- click OK



- click on the **Home icon** at the **View Cube**
- click on the **eye icon** for the **Door** to show the wood
- ensure that Sketches folder is open with the Sketches shown
- adjust the view similar to that below and submit a screenshot of your masterpiece