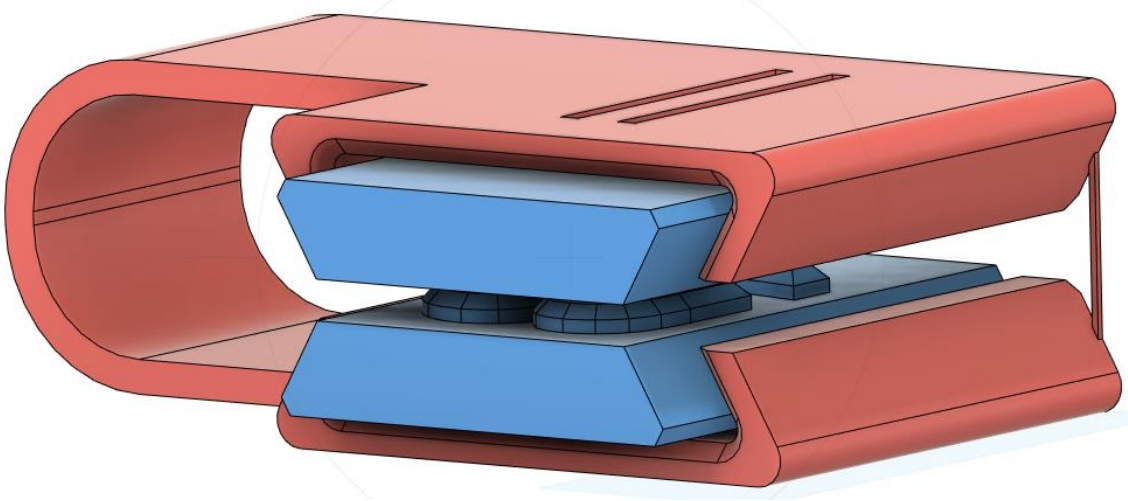
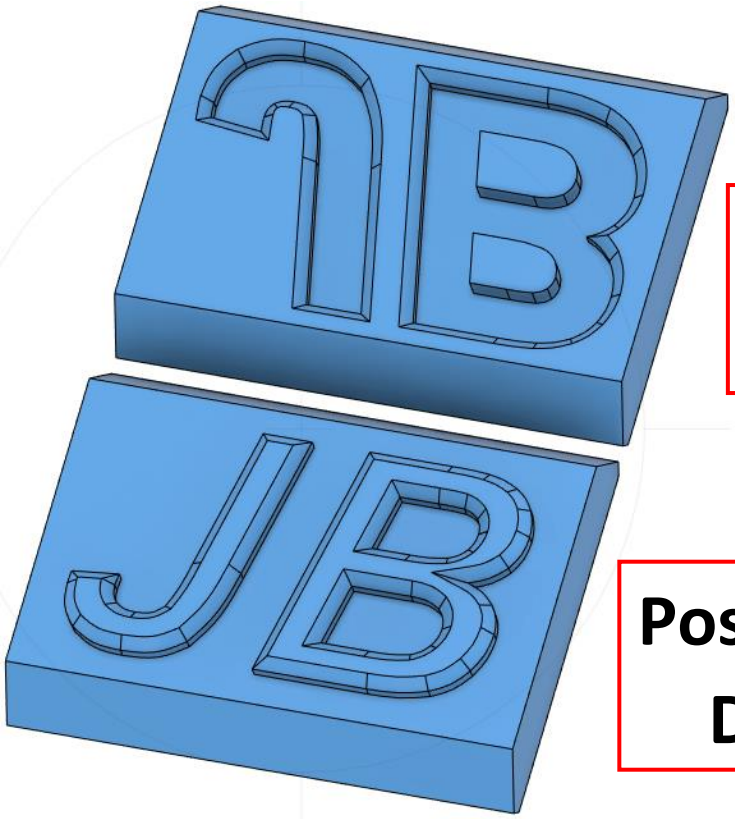
Mr. Barbetta's *"I don't care about 3D Printing. I just want the art credit."* class

Do you want to make a good impression?

Design a 3D Printed Embosser (handle and die set)





Negative Die

Positive Die

Today's Lesson is Sponsored by Field Station Dinosaurs in Leonia, NJ

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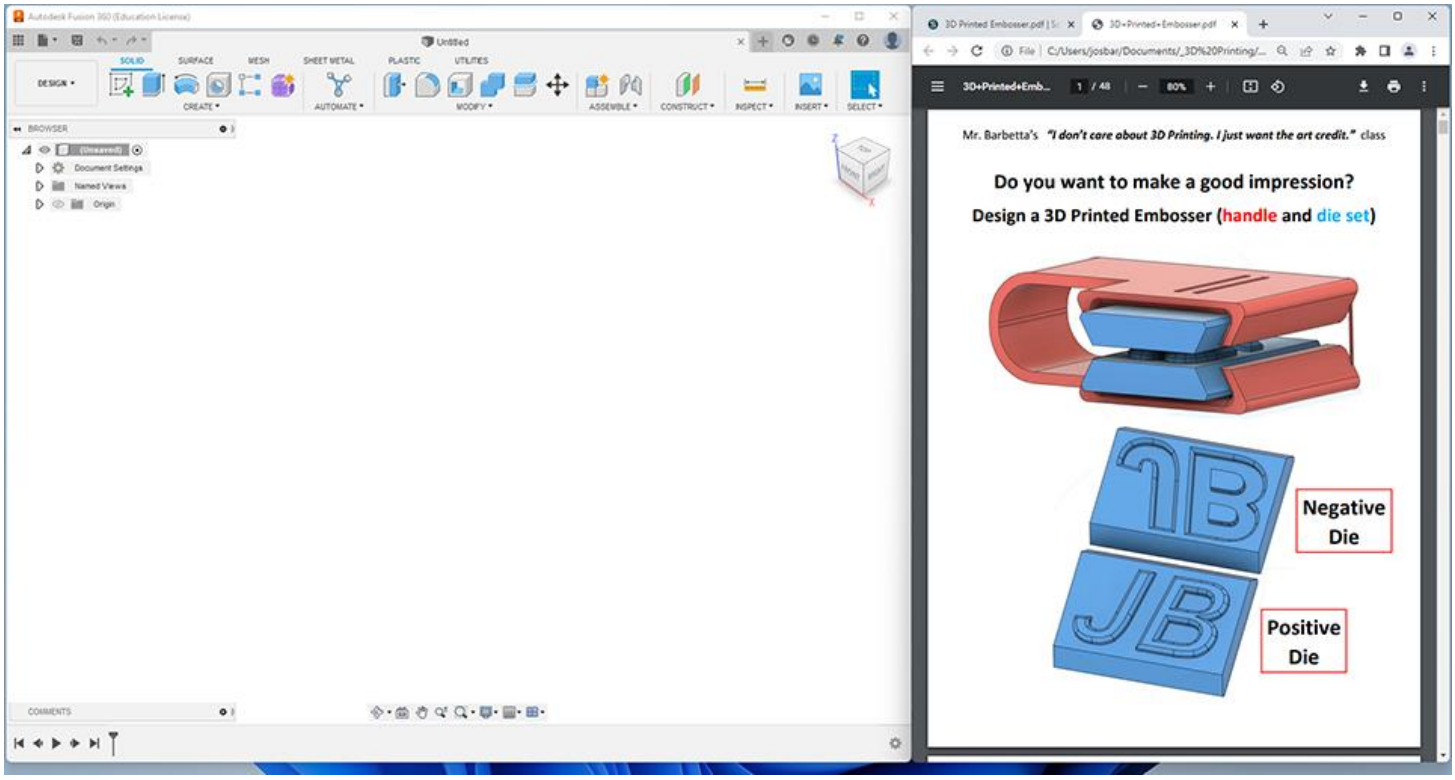
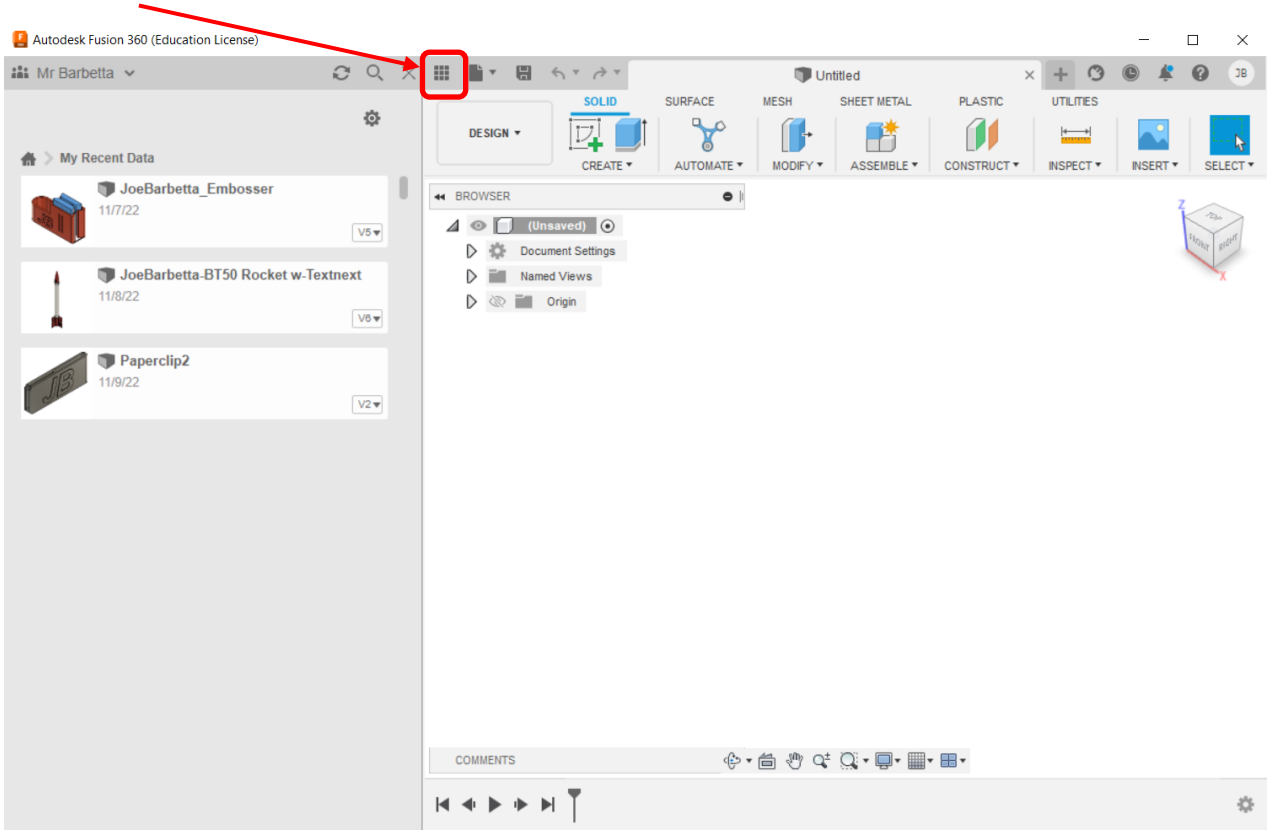
Using This Document

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%.

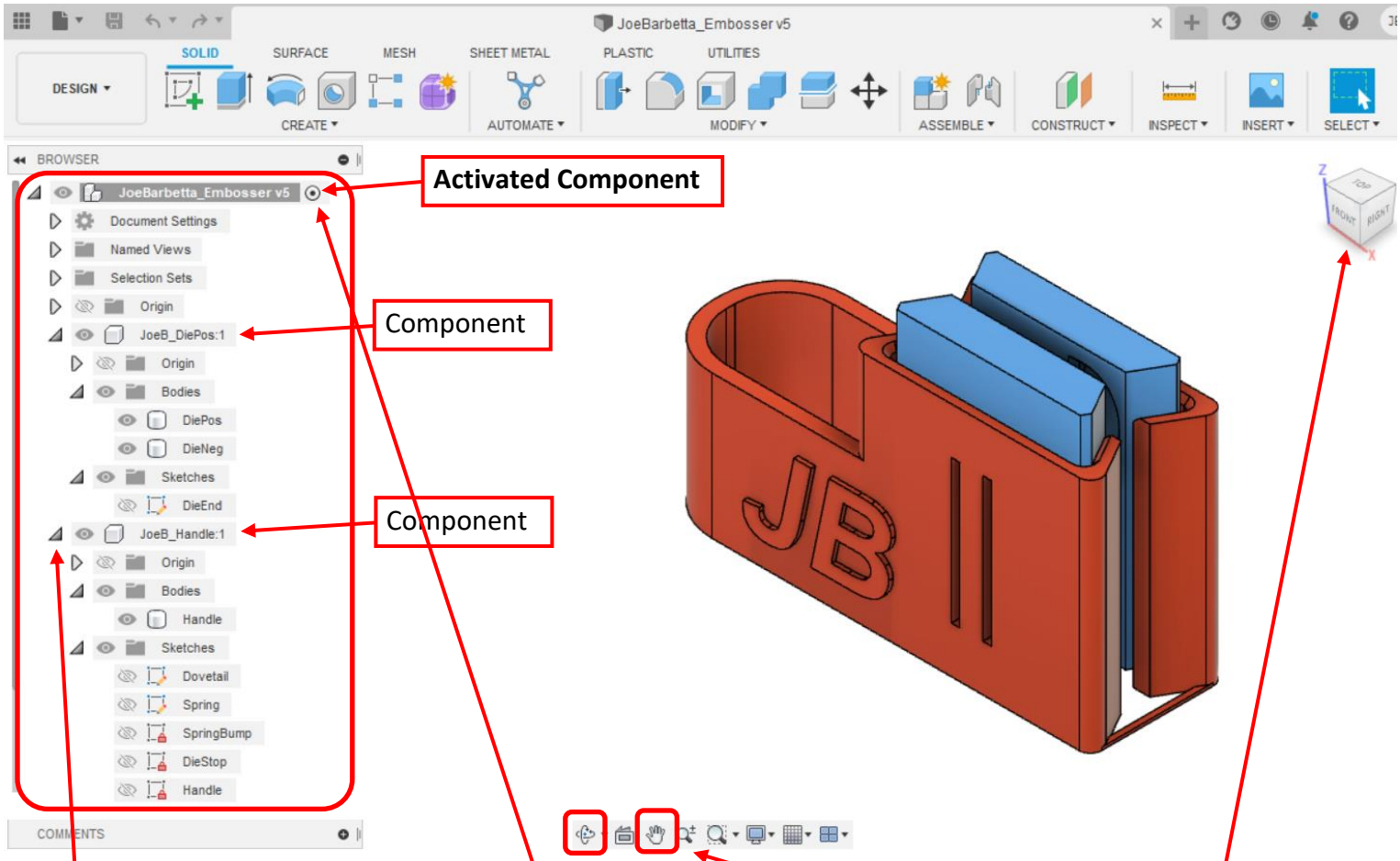


The Fusion window will not allow its width to be reduced much so for smaller computer screens a trick is to click on the **Data Panel icon** and then move the window to the left with the Data Panel off the screen.



The Fusion User Interface

This is what the final embosser 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.

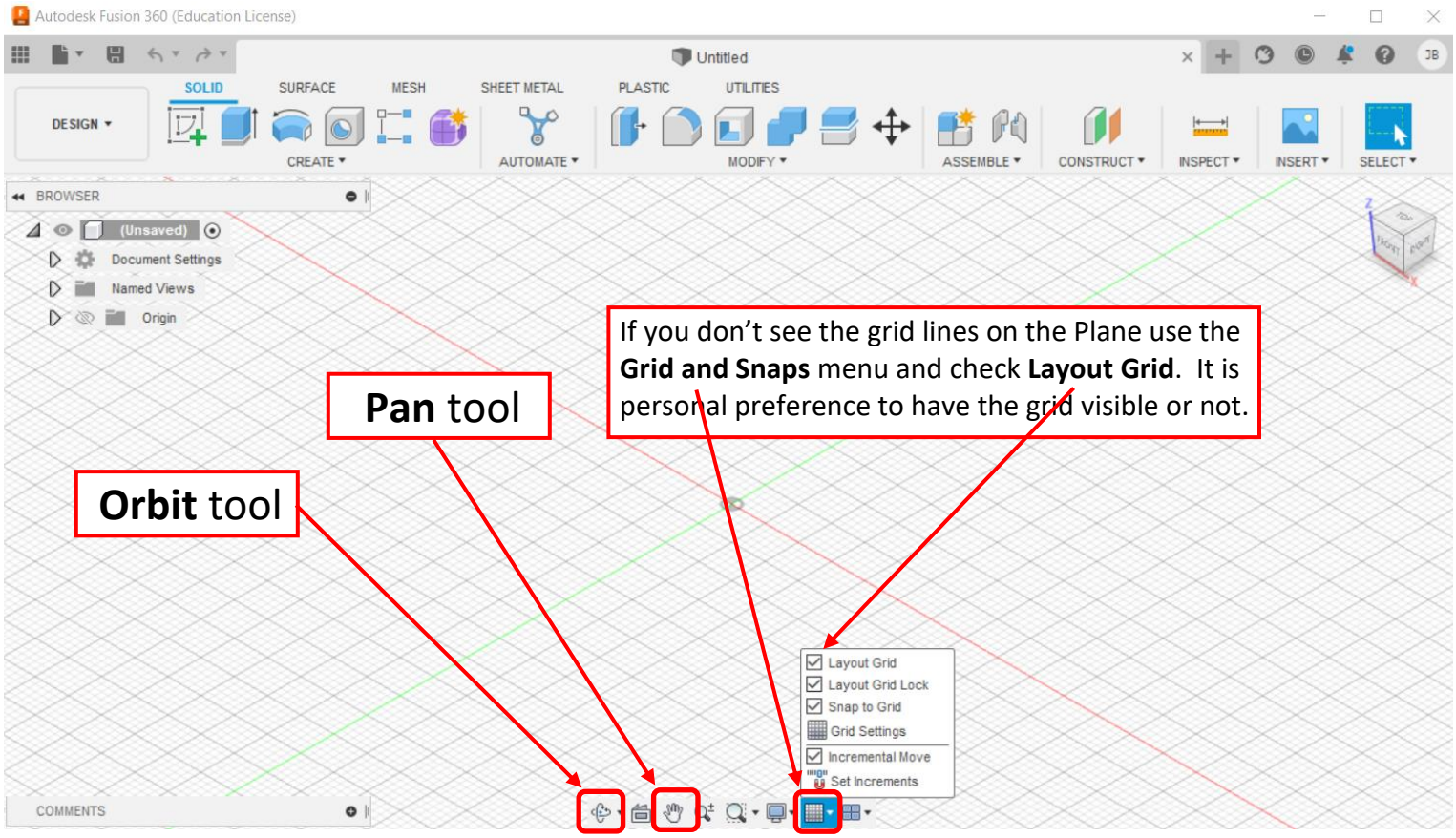
Note the **circled dot** of the *Activated Component*. When one wants to work on a Component, it must be Activated by right-clicking on the Component Name and selecting **Activate** at the top of the pop-up menu.
When a project is reopened, the Project at the top of the Browser is Activated and a Component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

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.

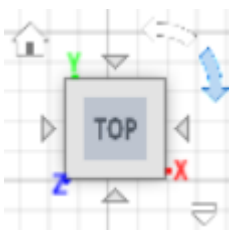


Here is a close-up of the View Cube at the top right of the window.

- 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



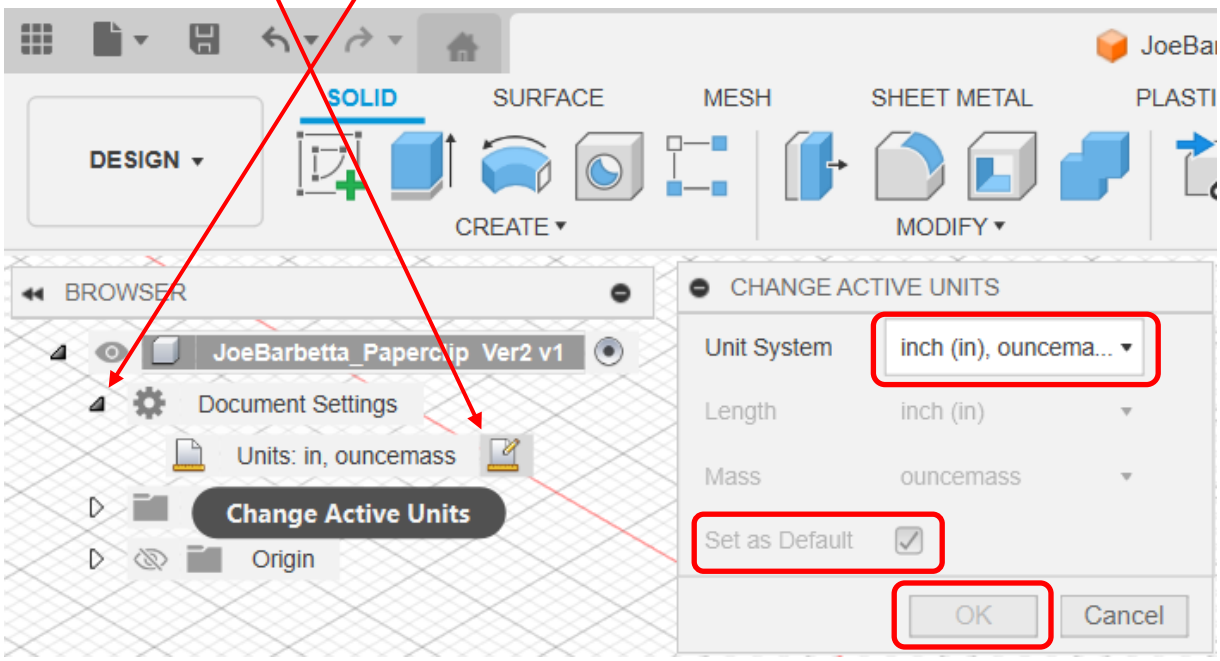
Starting a Design in Fusion

- open **Fusion 360**. 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 **_Embosser** e.g. **JoeBarbetta_Embosser** (note the use of the underscore)

Note that by default Fusion 360 saves your project to "the cloud", which are the servers managed by AutoDesk. When you log into Fusion 360 on a different computer, your projects will be available.

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

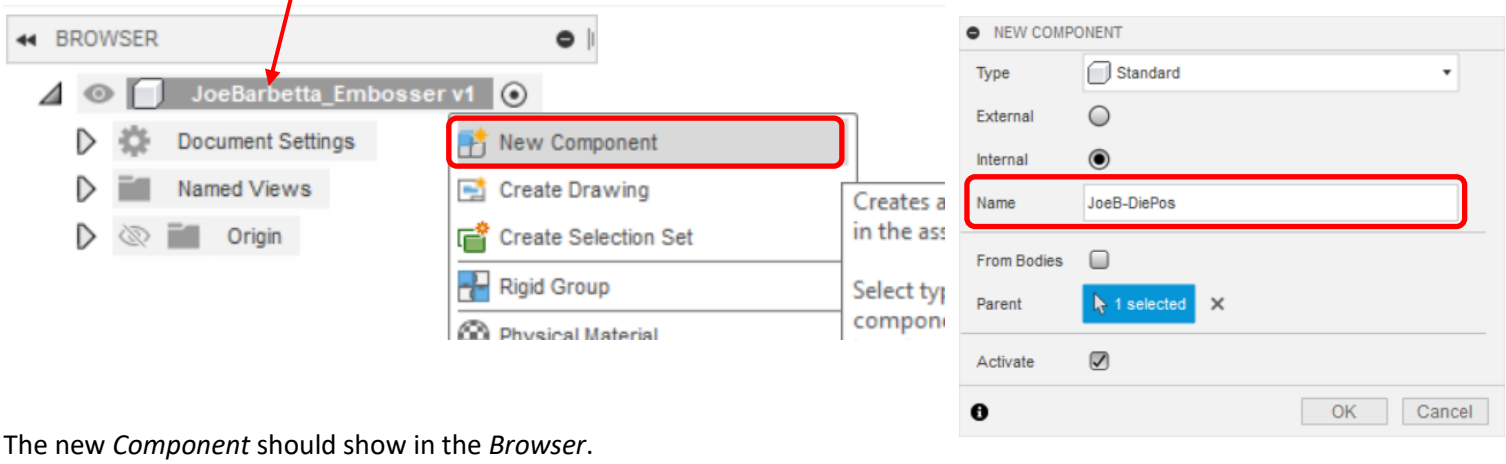
- in the left **"BROWSER"** click the **arrow next to Document Settings**
- click on the **edit icon** that appears to the left when you hover over **Units**
- ensure **Active Units** are set to **Units: in, ouncemass** and click **OK**. You can also enable **Set as Default** if it is not grayed out.



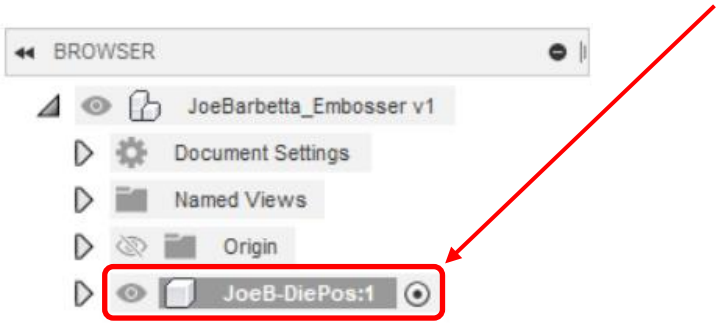
Creating a New Component

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 _DiePos** e.g. **JoeB_DiePos** and click **OK**

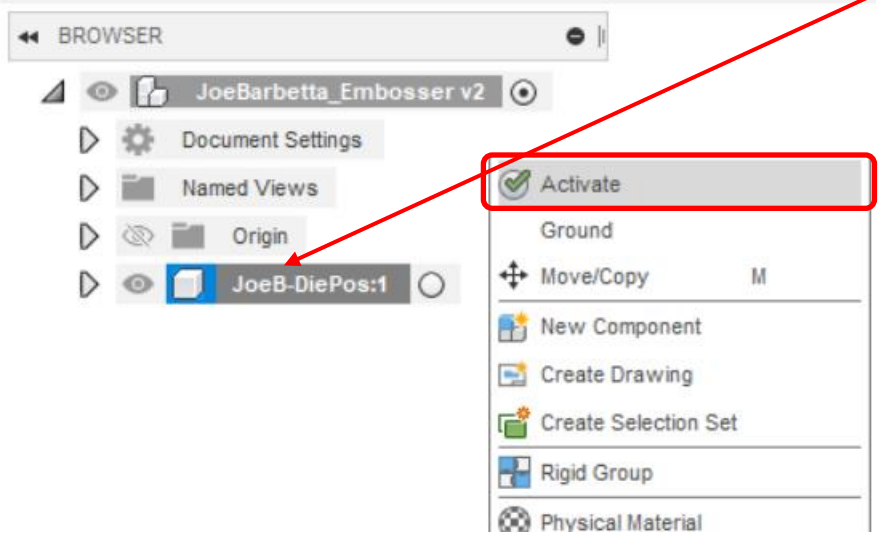


The new *Component* should show in the *Browser*.
Note that the area around the name is darkened and the circle with the dot. This indicates that it is the ***Active Component***.



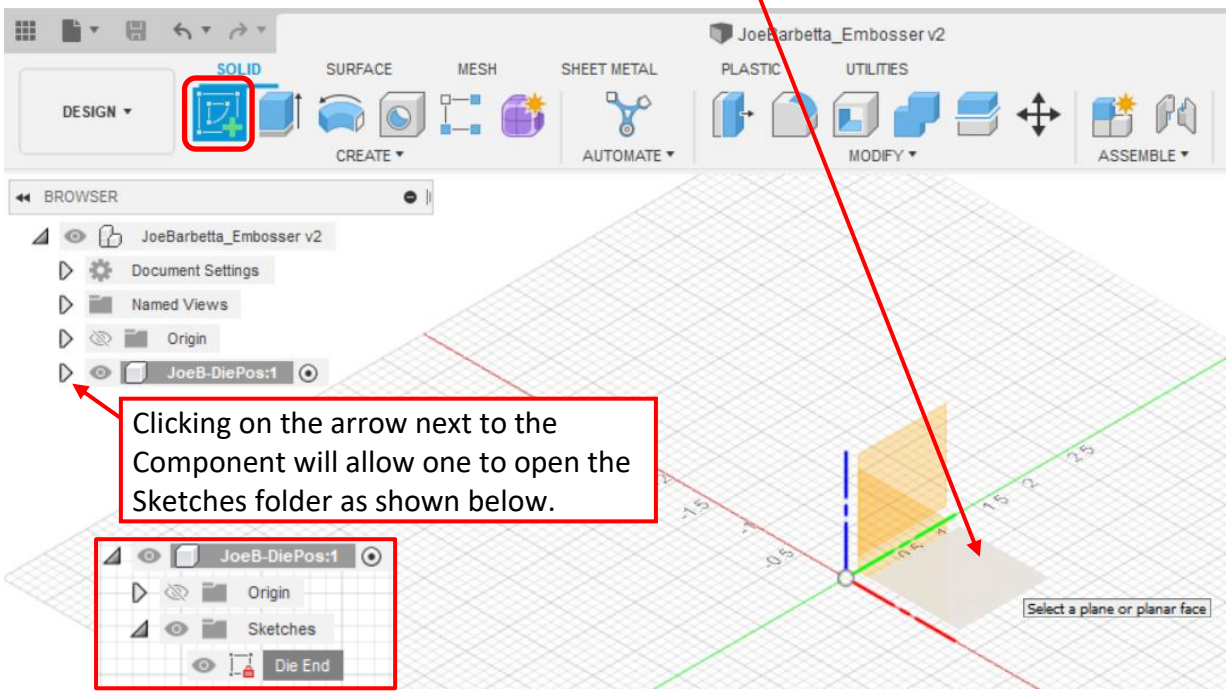
One reason to hate Fusion!

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.

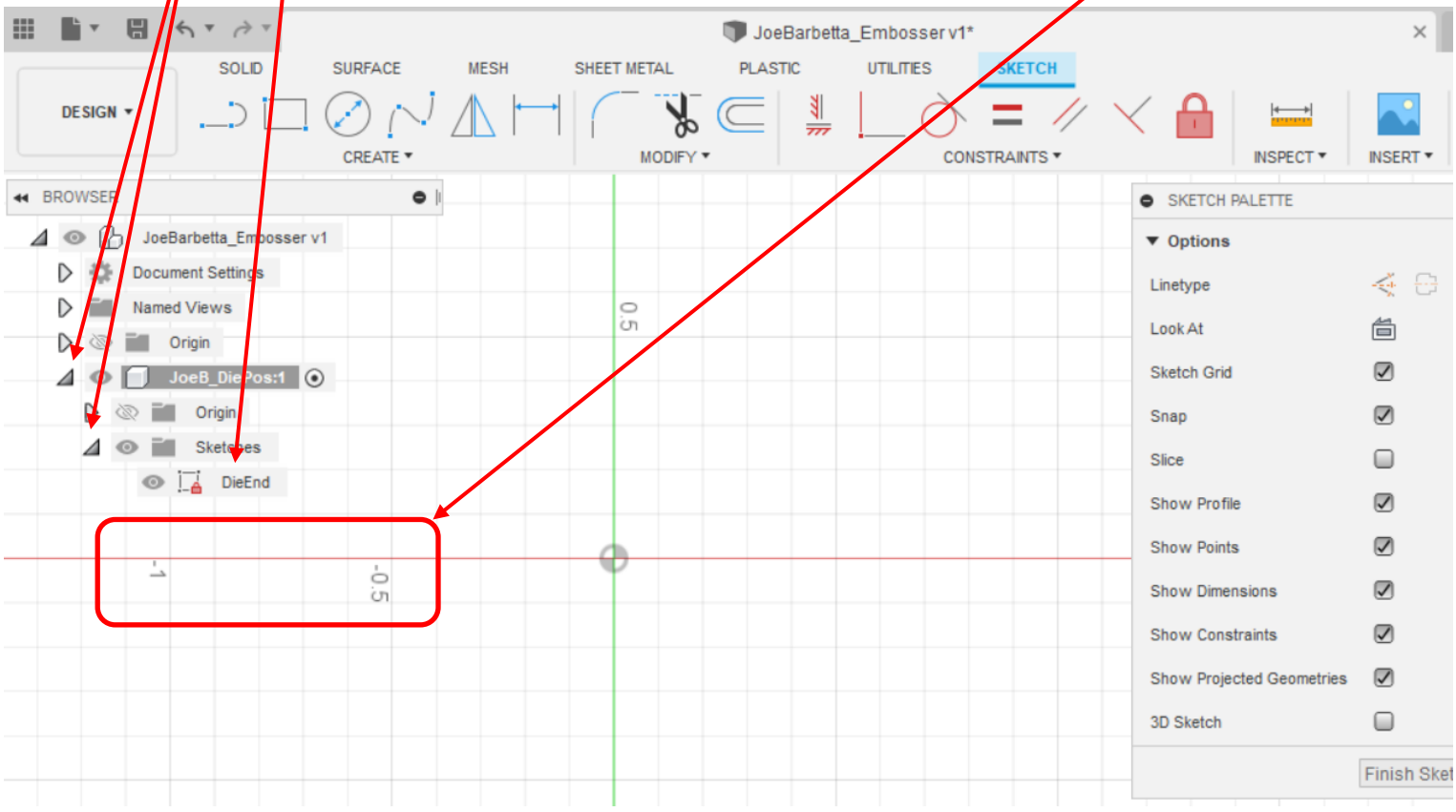


Creating the Die Sketch

- 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.



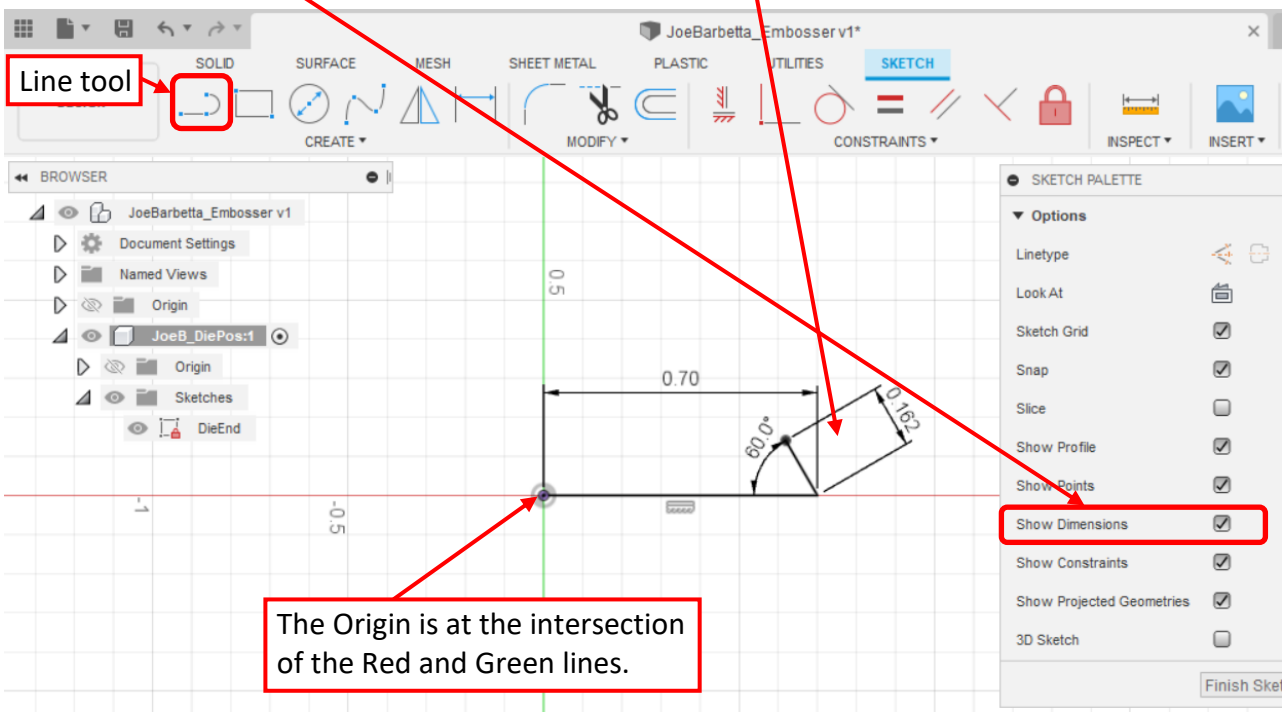
If you don't see the red and green axes as below, it's possible that the wrong Plane was selected in the previous step.
- click on the **arrows next to the new component** and then **next to the Sketches folder**
- right-click on the **Sketch name** and rename it to **Die End**.
- zoom in (and reposition if needed using the Hand icon at the bottom of the window) until **-1** and **-0.5** shows similarly to below



- click on the **Line** tool and start the line by clicking on the **Origin** and extend the line to the right and type **0.7** and click.
- continue creating an upward angled line as shown and type **0.162** for its length. Press the **Tab key** and then type **60** for the angle and click again. **If the angle is appearing on the opposite side, use its supplement** (180 – 60 = **120**).
- press the **Esc key** to stop creating more line segments. Do Not click Finish Sketch.

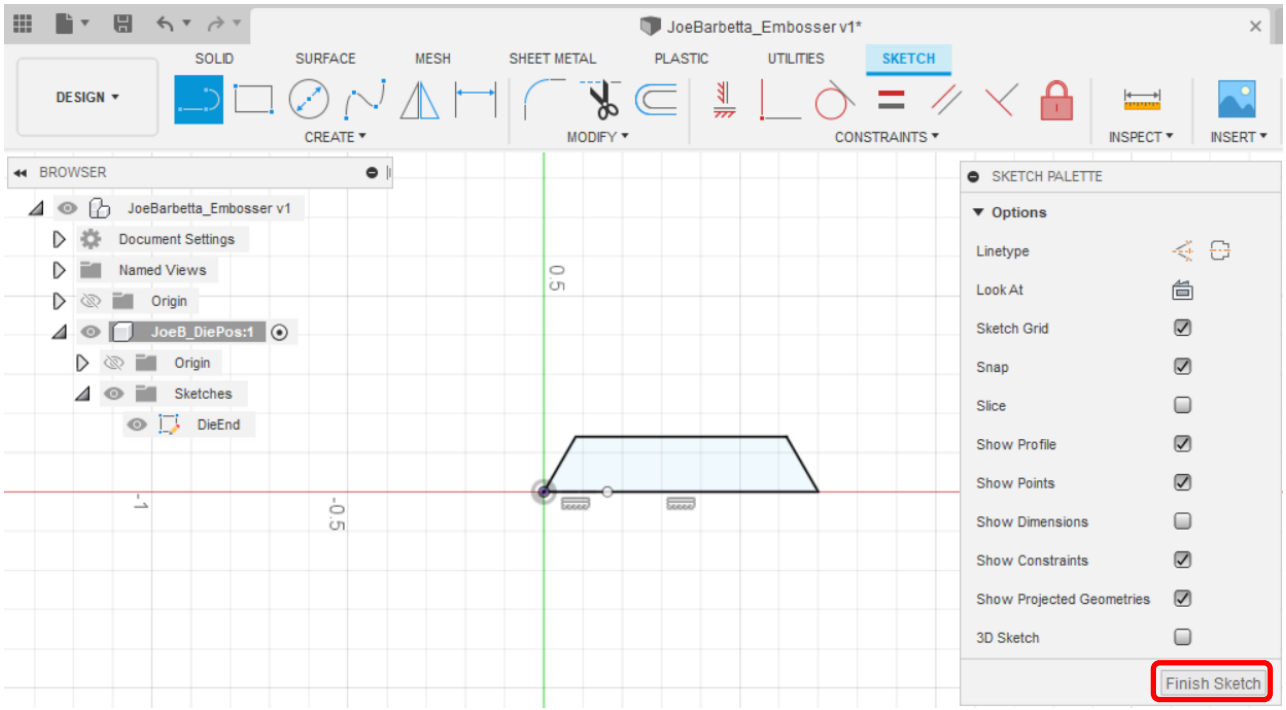
Note that the thinner dimensions lines with their values may look different. It is the two thicker lines that are important.

- uncheck **Show Dimensions** and the annoying dimension lines will disappear.



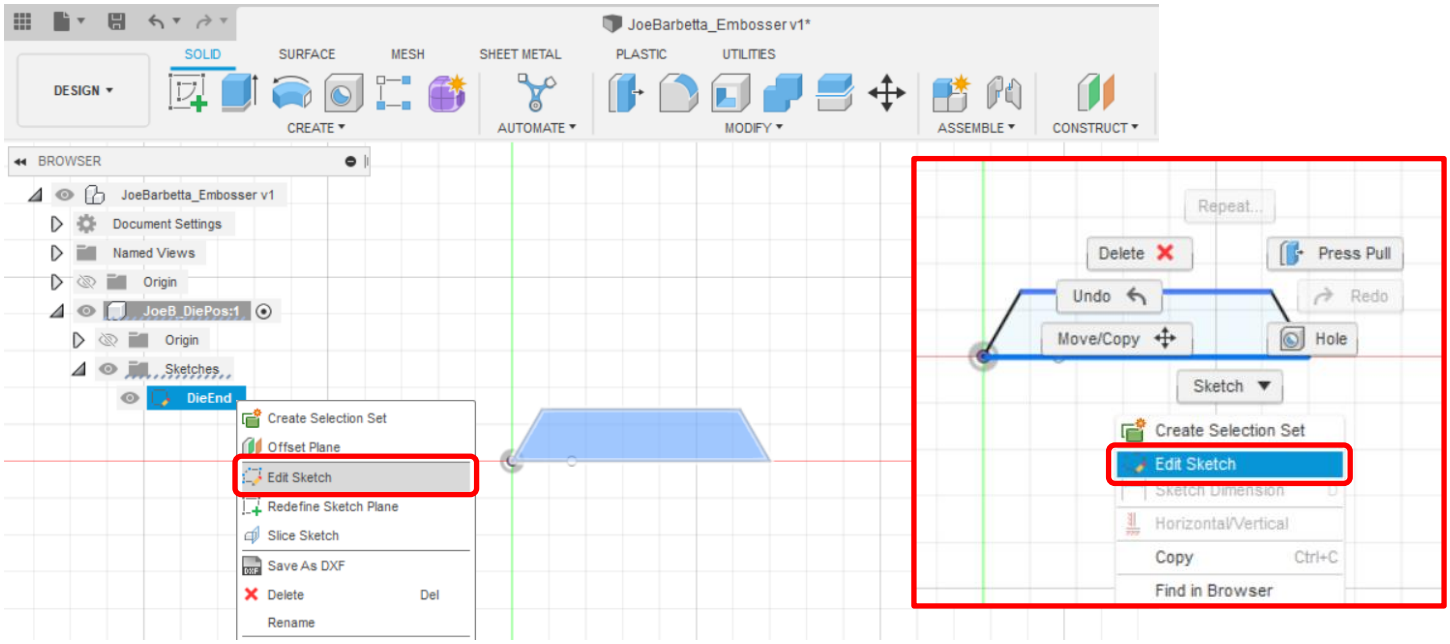
If arcs are resulting, you may be holding the mouse button when you start to extend the line. Use a **short click** and then start extending the line.

- click on the **Line** tool again and create a line from the Origin point extending upward and to the right. Type **0.162** for the length, press the **Tab key**, enter **60** for the angle, and click.
- continue extending the line to the right until it **snaps to the top of the other line** and click.
- click **Finish Sketch** now that the Sketch is complete and the profile is filled with light blue.

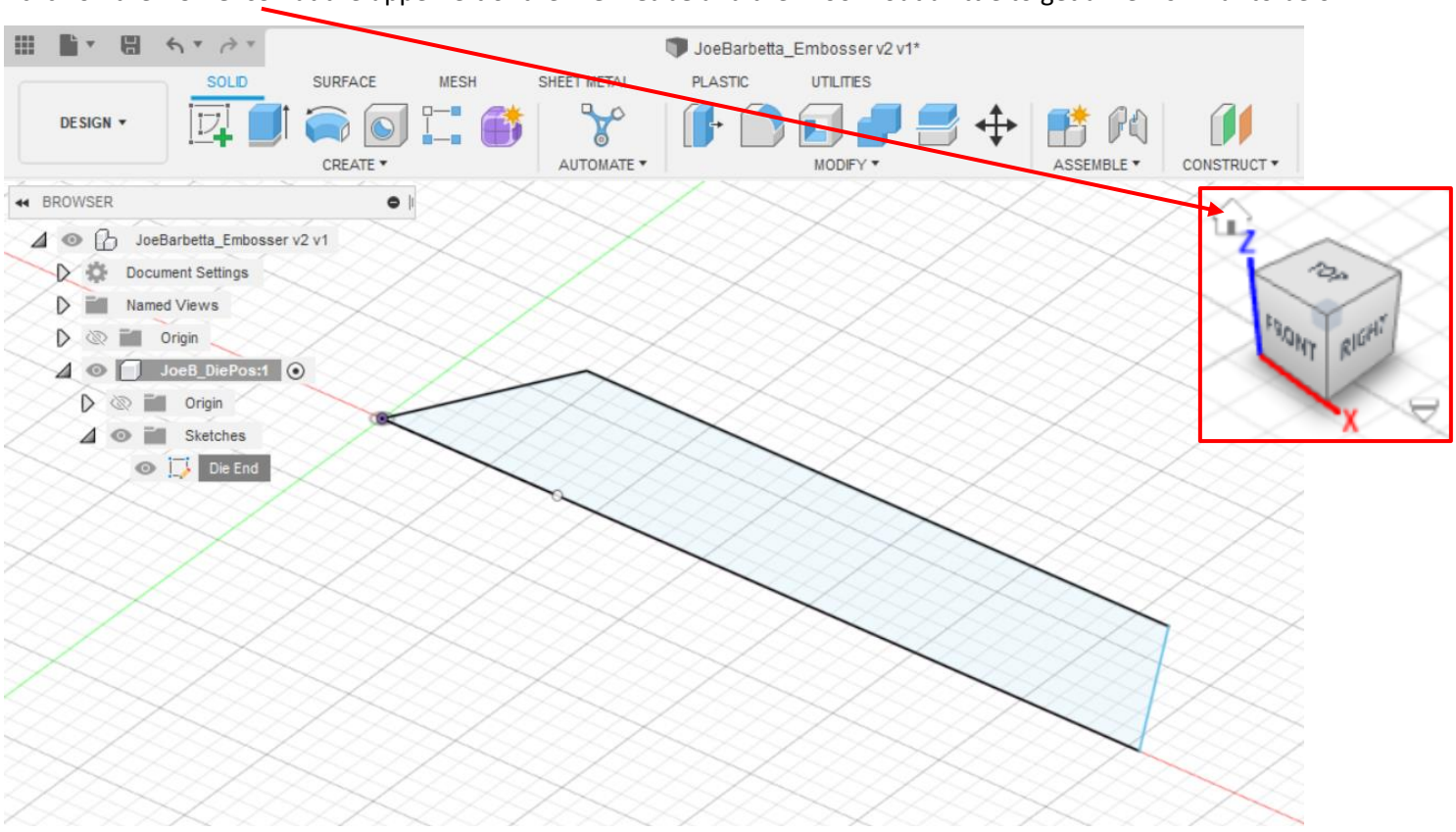


Editing a Sketch

Our Sketch is complete. However, if we clicked Finish Sketch too early, need to make changes to a sketch, or reopened our project, we would use the Edit Sketch feature. One common mistake is starting a new Sketch when an existing Sketch should be completed. One can right-click on the **Sketch Name** and select **Edit Sketch**. Alternatively, as the insert shows, one can right-click on a **Sketch Line** and select **Edit Sketch**.

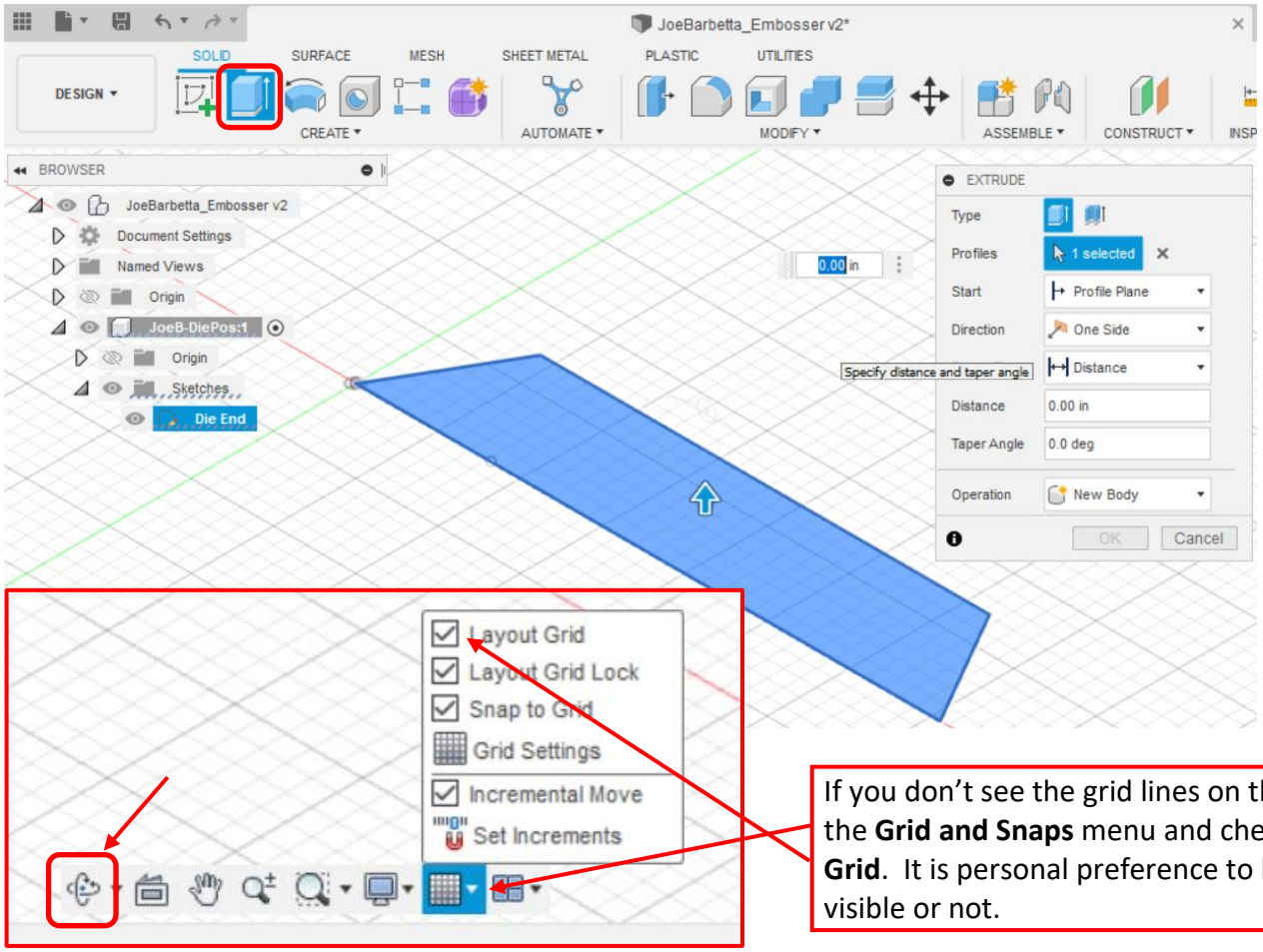


- click on the **Home icon** at the upper left of the **View Cube** and then zoom out a little to get a view similar to below.



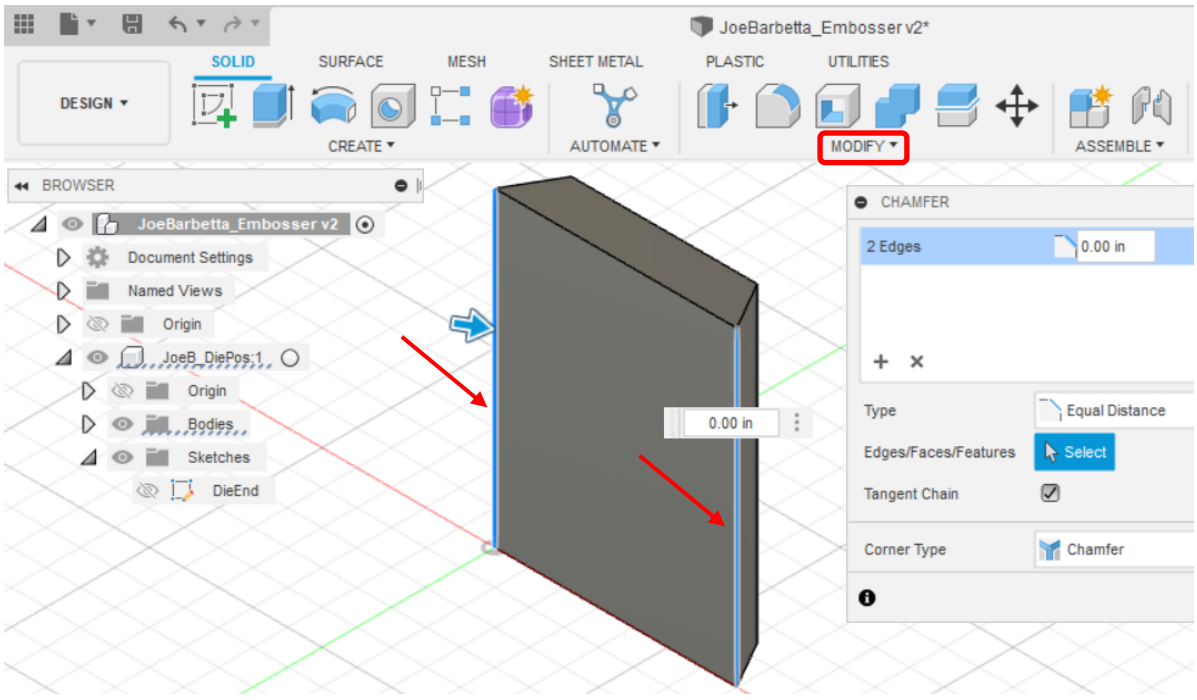
Extruding the Sketch

- select the **Extrude** tool and enter **0.9** and click **OK**. If the Profile (interior of the trapezoid) didn't turn blue automatically, then click on it to do so.



If you don't see the grid lines on the Plane use the **Grid and Snaps** menu and check **Layout Grid**. It is personal preference to have the grid visible or not.

- zoom out a little to achieve a view similar to below
- select the **Chamfer** tool in the top **MODIFY** pull-down menu
- click on the **two edges** (the edges will turn blue) as shown and enter **0.03** and click **OK**

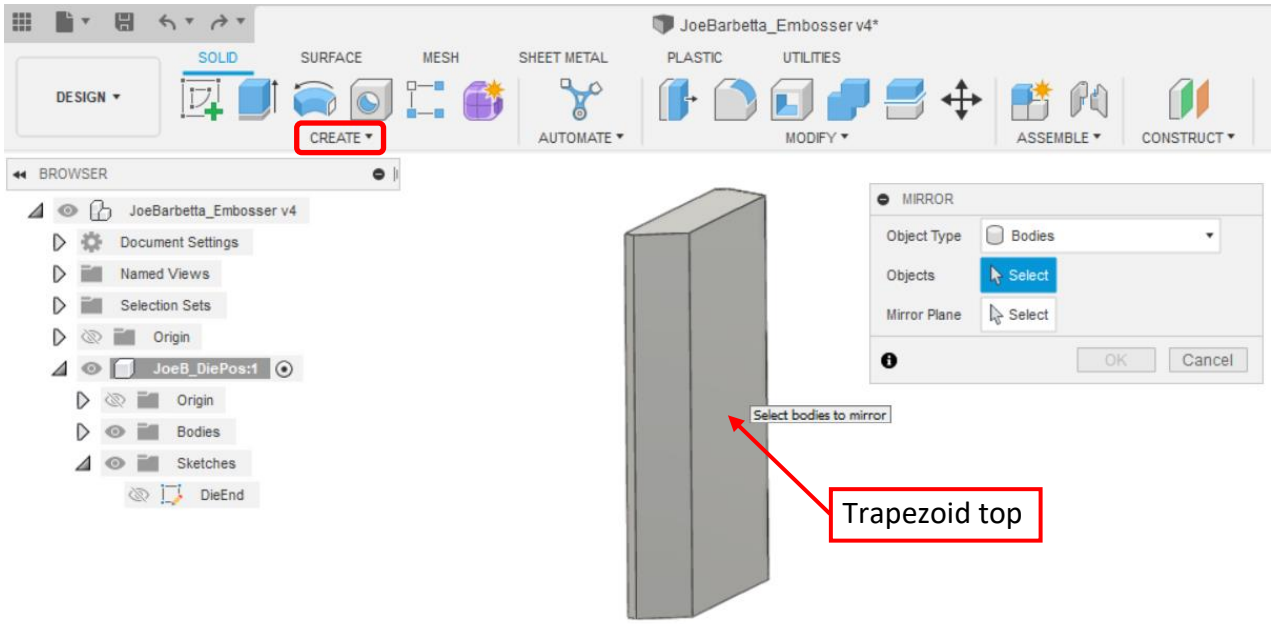


Mirroring an Object

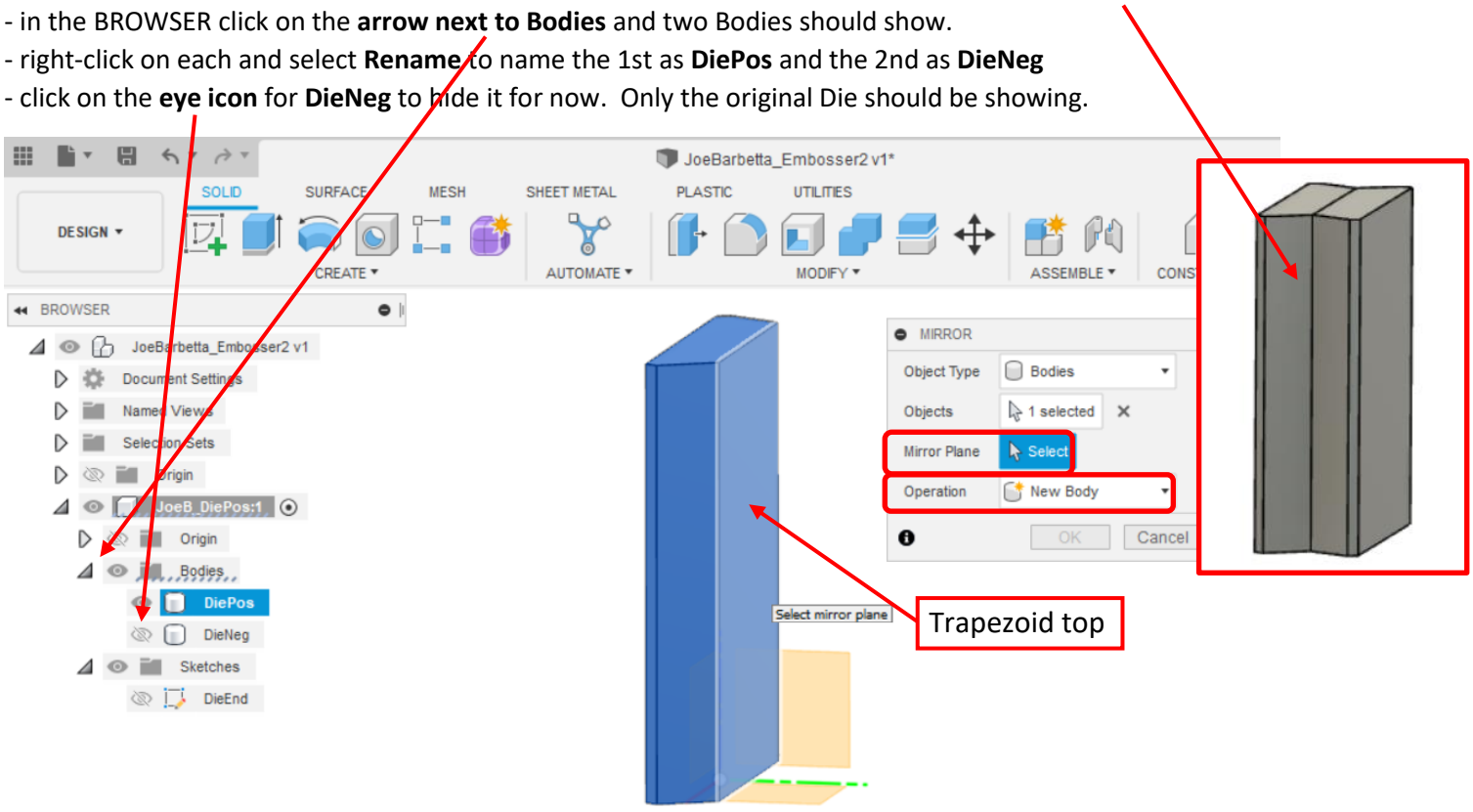
Designs will often have features that are mirror images of each other. It is a great convenience that most CAD programs offer the ability to create a duplicate body that mirrors the original body.

We just created the *Positive Die*. Now we can create a mirrored part for the *Negative Die*.

- adjust the view so that the trapezoid top is visible as shown below.
- in the **CREATE** pull-down menu select **Mirror** (it's far down in the list) and **click on the Die body**.

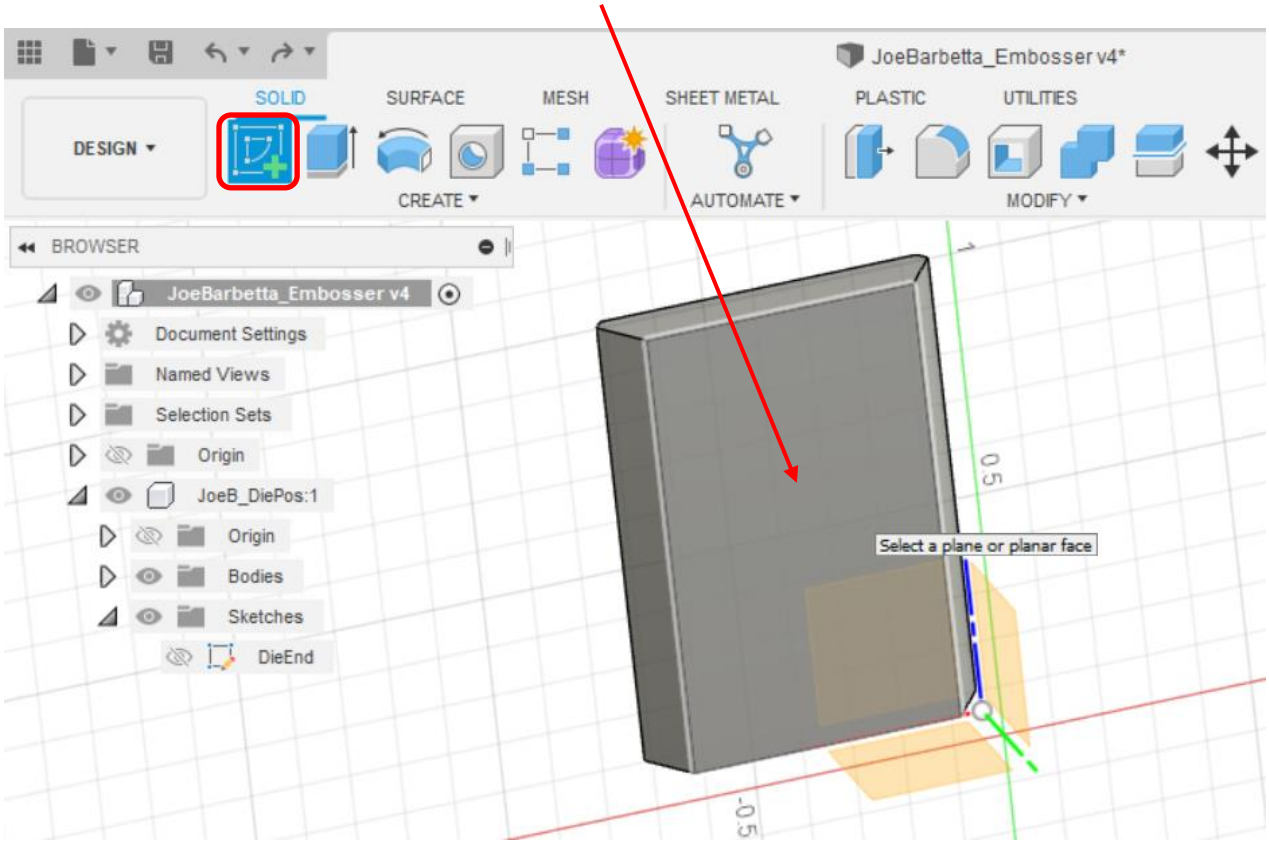


- click on **Select** next to **Mirror Plane**.
- click on the **trapezoid top** again. Note that the **"top"** is the large surface.
- set the **Operation** is set as **New Body** and click **OK**. Verify that a line appears between the two halves.
- in the BROWSER click on the **arrow next to Bodies** and two Bodies should show.
- right-click on each and select **Rename** to name the 1st as **DiePos** and the 2nd as **DieNeg**
- click on the **eye icon** for **DieNeg** to hide it for now. Only the original Die should be showing.

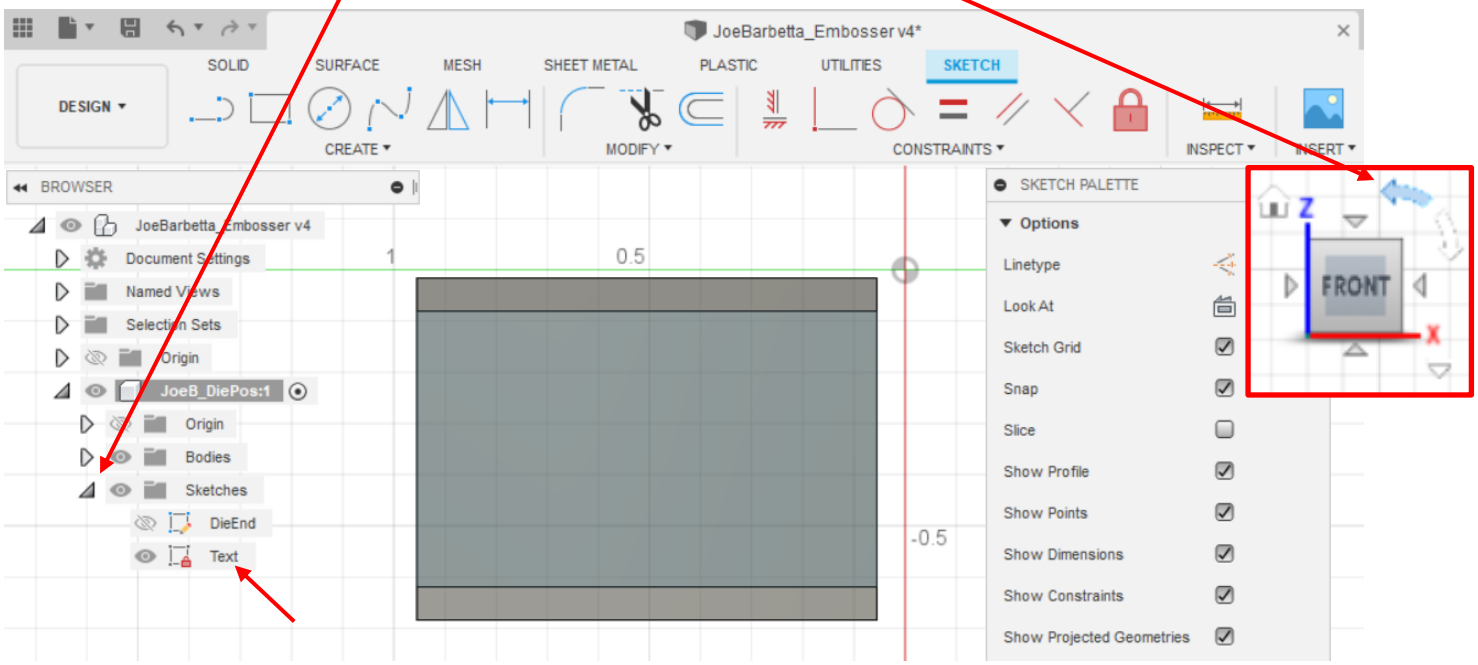


Creating Text for the Die

- adjust the View so that the *Face* where you want text faces you. In this case the trapezoid top.
- select the **Create Sketch** tool and **click on that face**.



- you may need to rotate the view using the **curved arrows at the upper right of the View Cube**. If the curved arrows do not appear, click on the FRONT face of the View Cube.
- right-click on the new **Sketch name** and select **Rename** to rename the new Sketch to **Text**. If you don't see the Sketch names you may have to click on the **arrow next to the Sketches folder**.

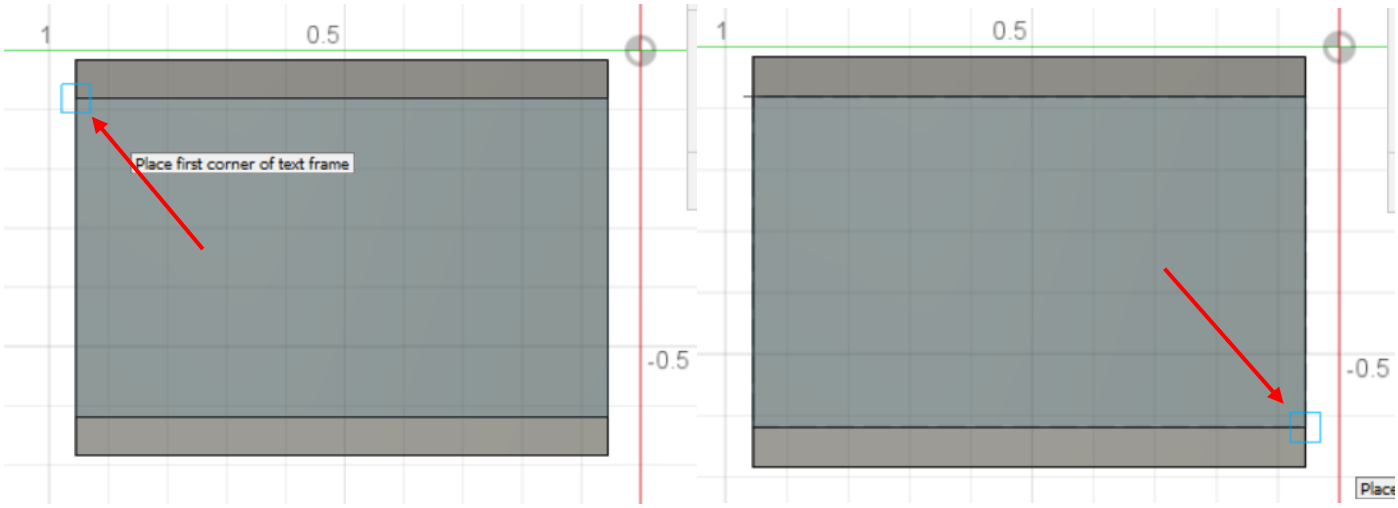


- from the **CREATE** pull-down menu select **Text**

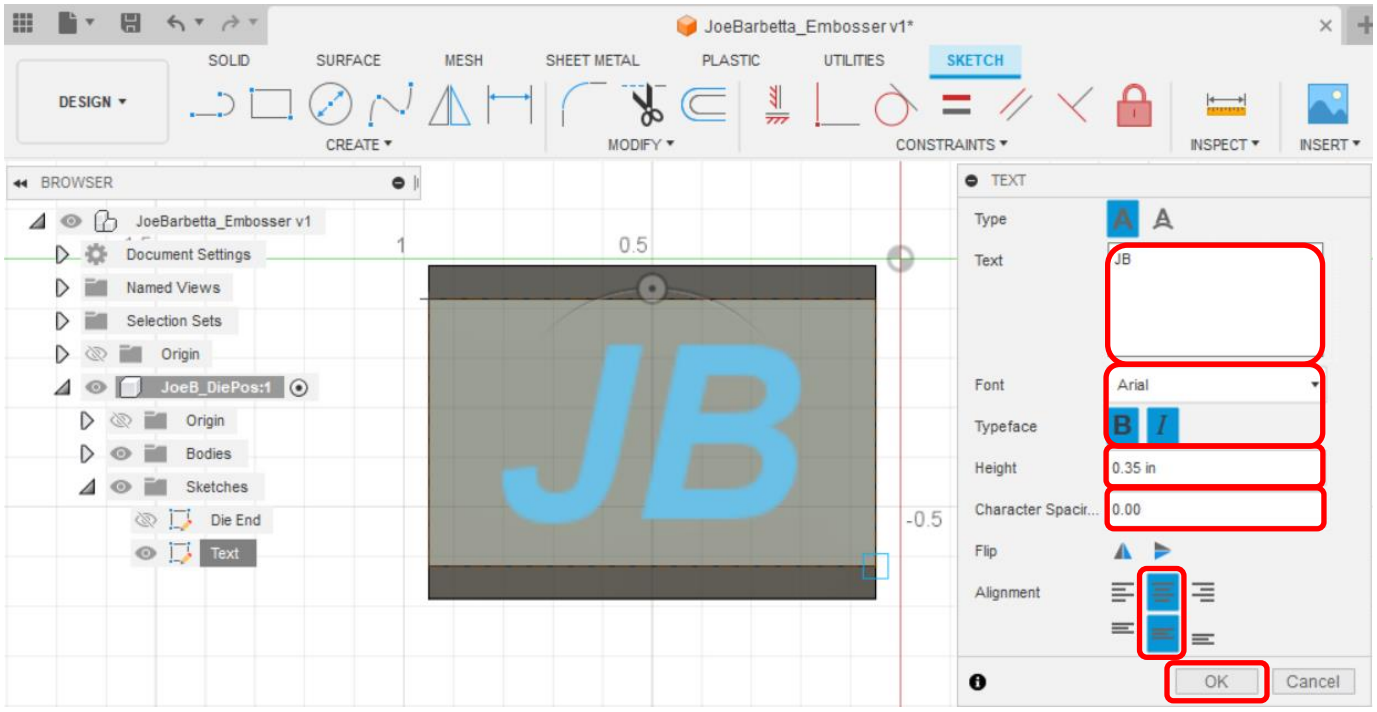
At this point one needs to define the extents of the text region by clicking on the desired upper-left location and then clicking on the desired lower-right location.

Here are two close-up views of the object. **Click on the top-left corner** of the object's surface and then **click on the bottom-right corner** as shown by the arrows below.

There is a bug in Fusion 360, wherein it may not allow you to click on the top-left corner if it is too close to the BROWSER folders. You may have to use the bottom Pan tool (hand icon) to move the object to the right.

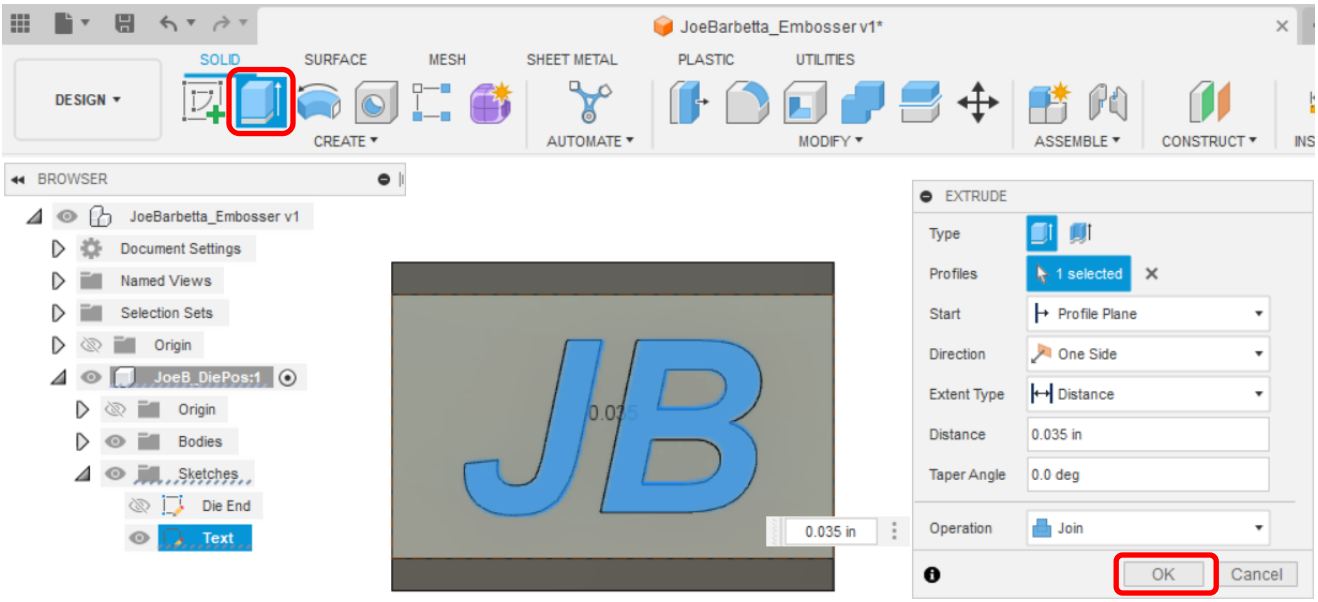


- in the **TEXT** window enter your initials for **Text** and click on the **Center** and **Middle** icons for **Alignment**.
- set the value for **Height** to **0.35**
- as per personal preference **Bold** and/or **Italics** can be selected for the **Typeface**. It is recommended to keep the font as Arial. Other fonts may not produce adequate results.
- click **OK** when done and then **Finish Sketch**

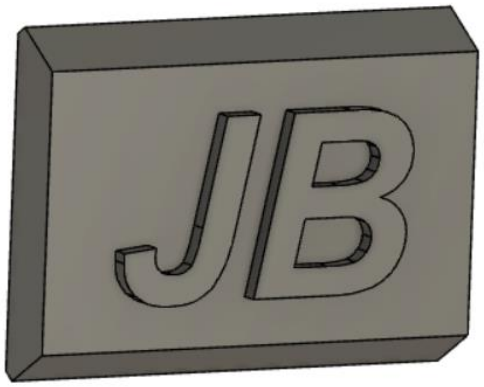


- right-click on the text and select **Create Selection Set**. Nothing will seem to happen, but this is needed for the next Extrude operation.

- select the **Extrude** tool, enter **0.035**, and click **OK**



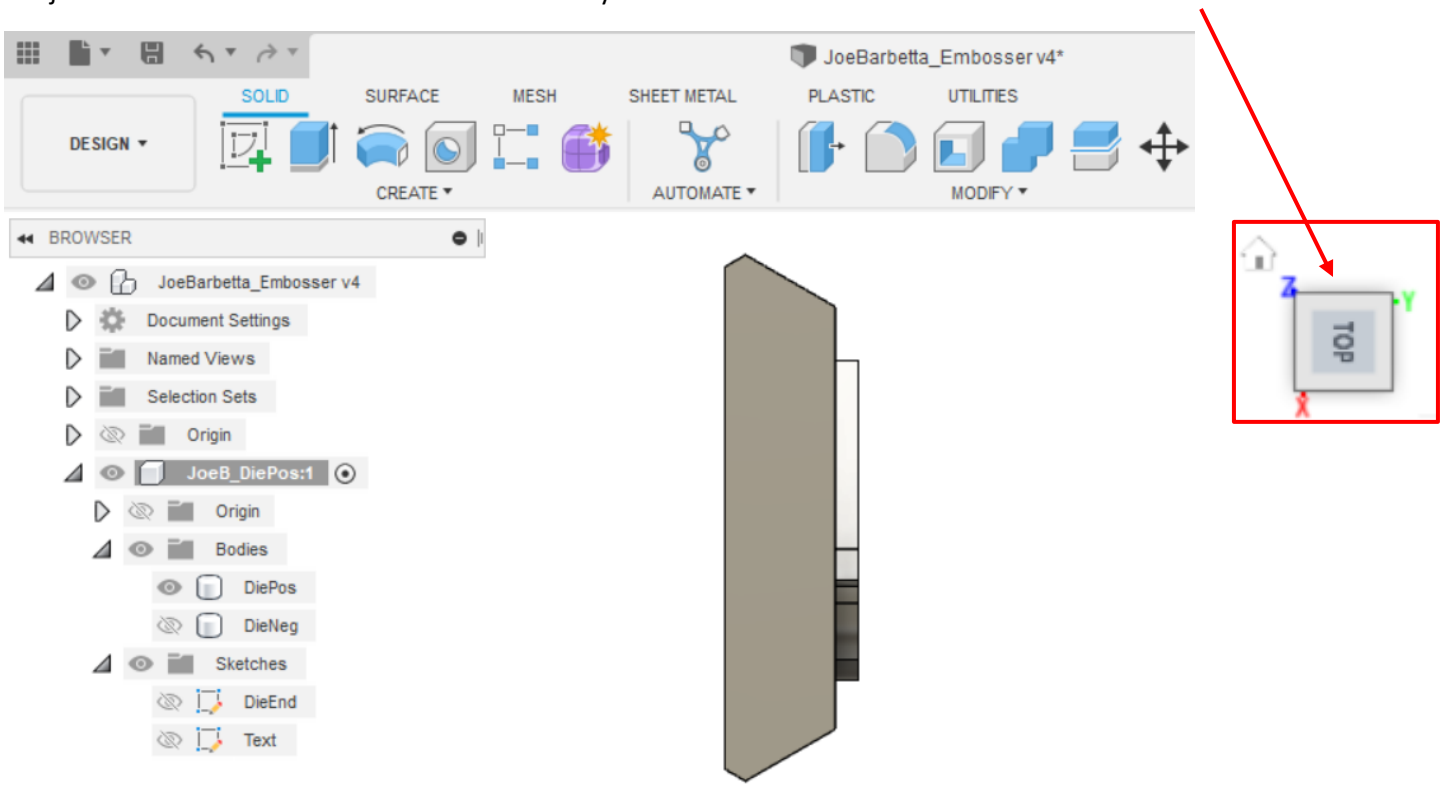
When the view is rotated, one can see the extruded text. Note that one can always right-click on the Text sketch in the Sketches folder and then right-click on the text and select Edit Text to change the text and its attributes.



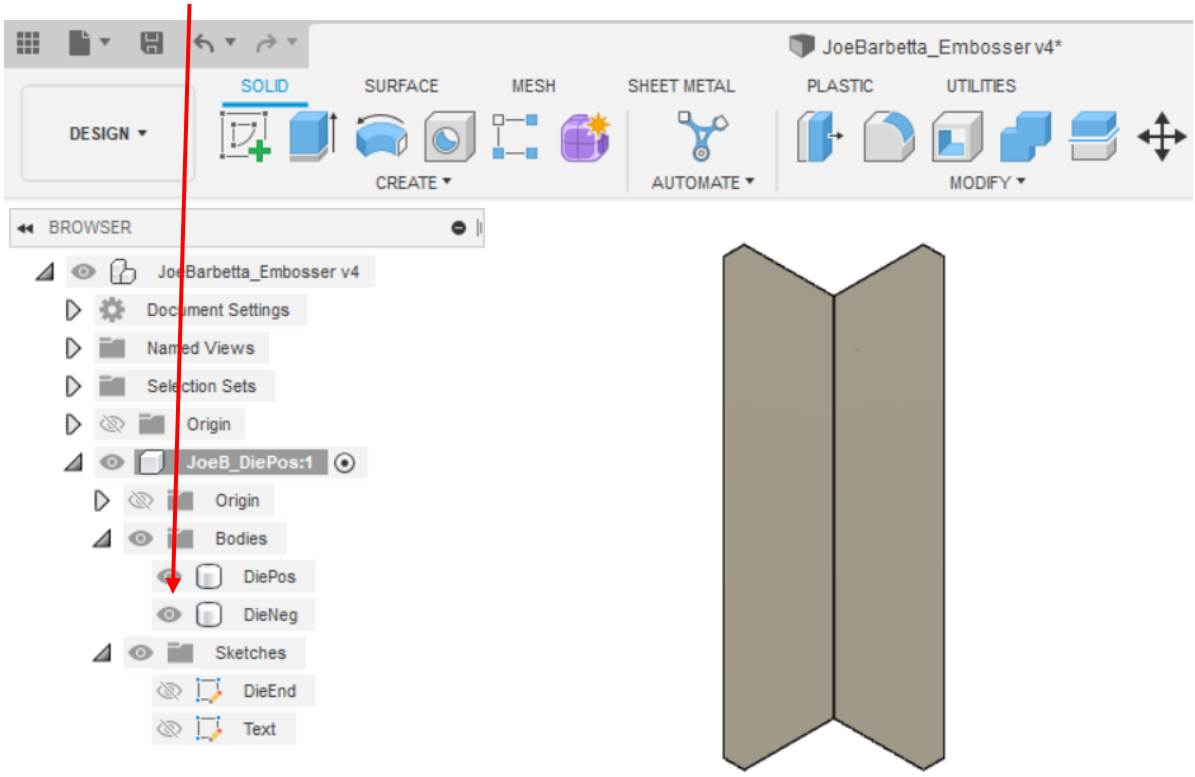
Cutting Material from an Object

A powerful feature of many CAD programs allows the use of one object to cut material from another object.

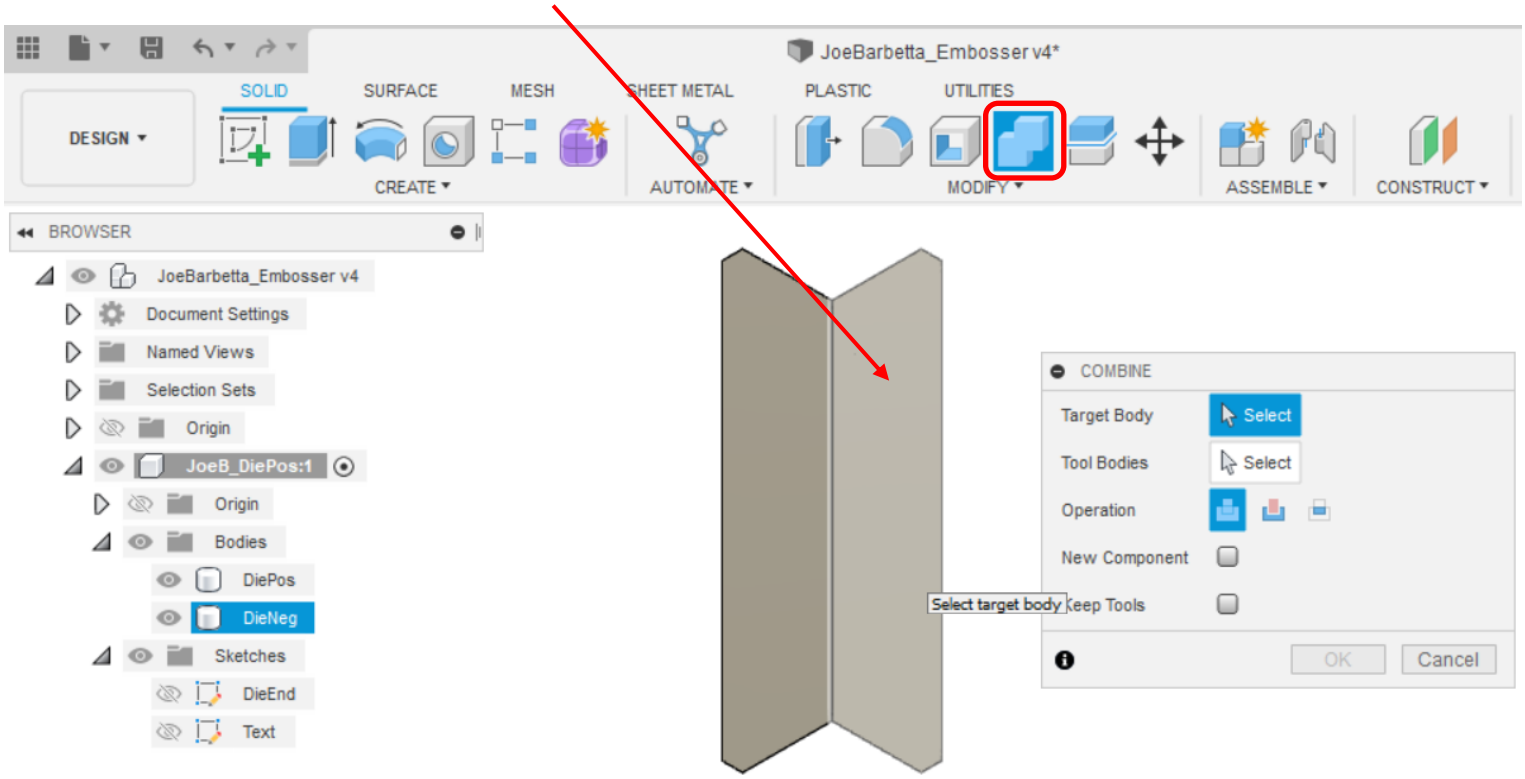
- adjust the view to see the end of the DiePos Body as shown below. The View Cube should show TOP.



- click on the **eye icon** for the **DieNeg** *Body* to make it visible. Both Dies should now be visible.

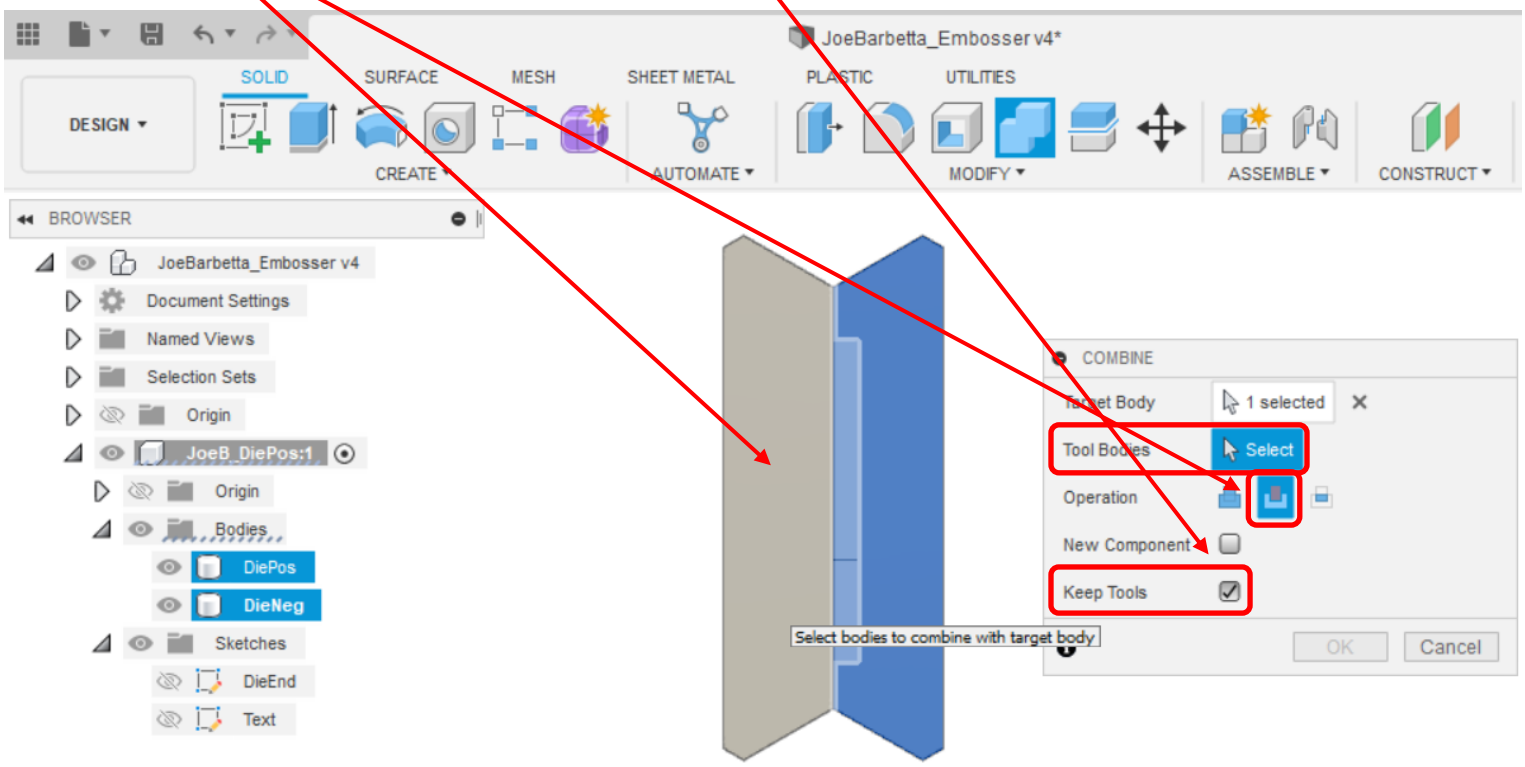


- select the **Combine** tool and click on the **DieNeg** body. If the tool not visible select Combine from the MODIFY menu.

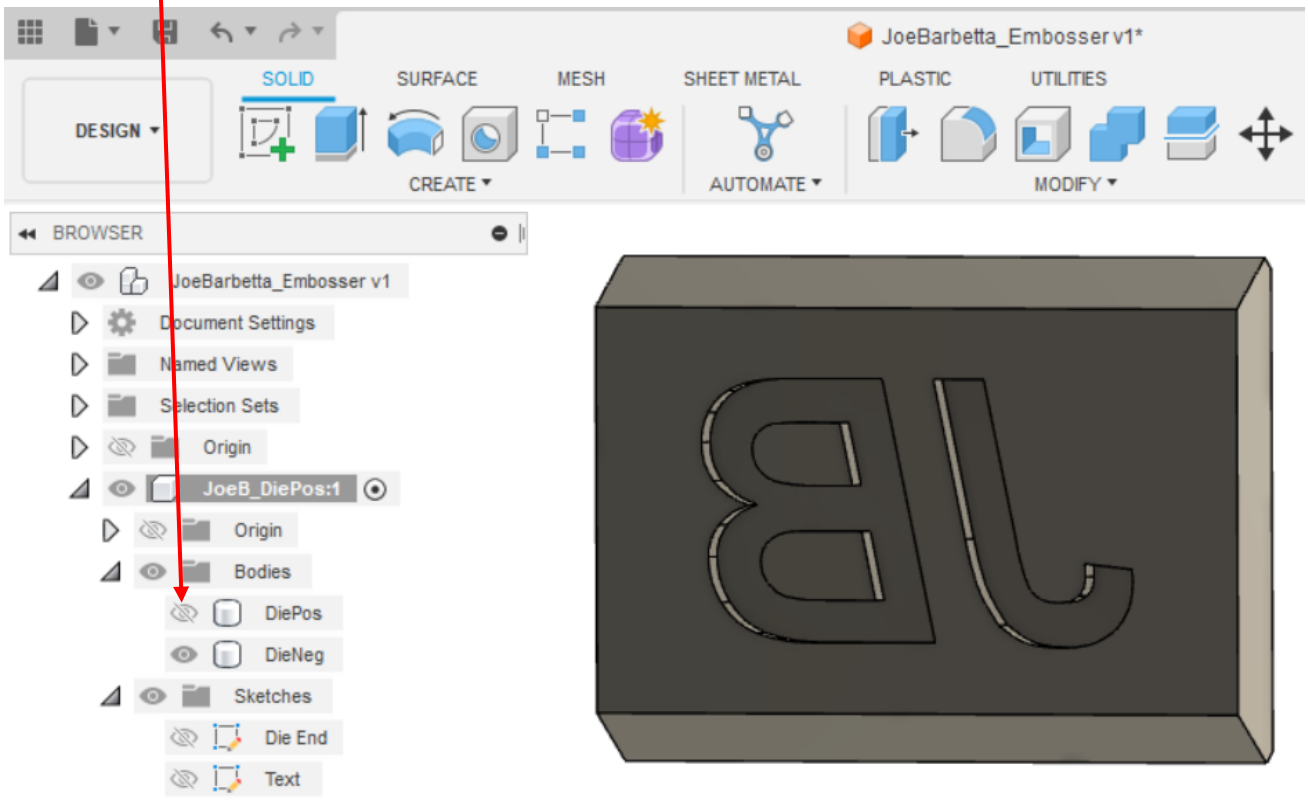


Here we are using the *Combine* tool to allow the *Tool Body*, which has the extruded text, to Cut away from the *Target Body* because we want to cut the letters out. The Combine tool can also be used to combine two or more objects into on when the Join Operation is used.

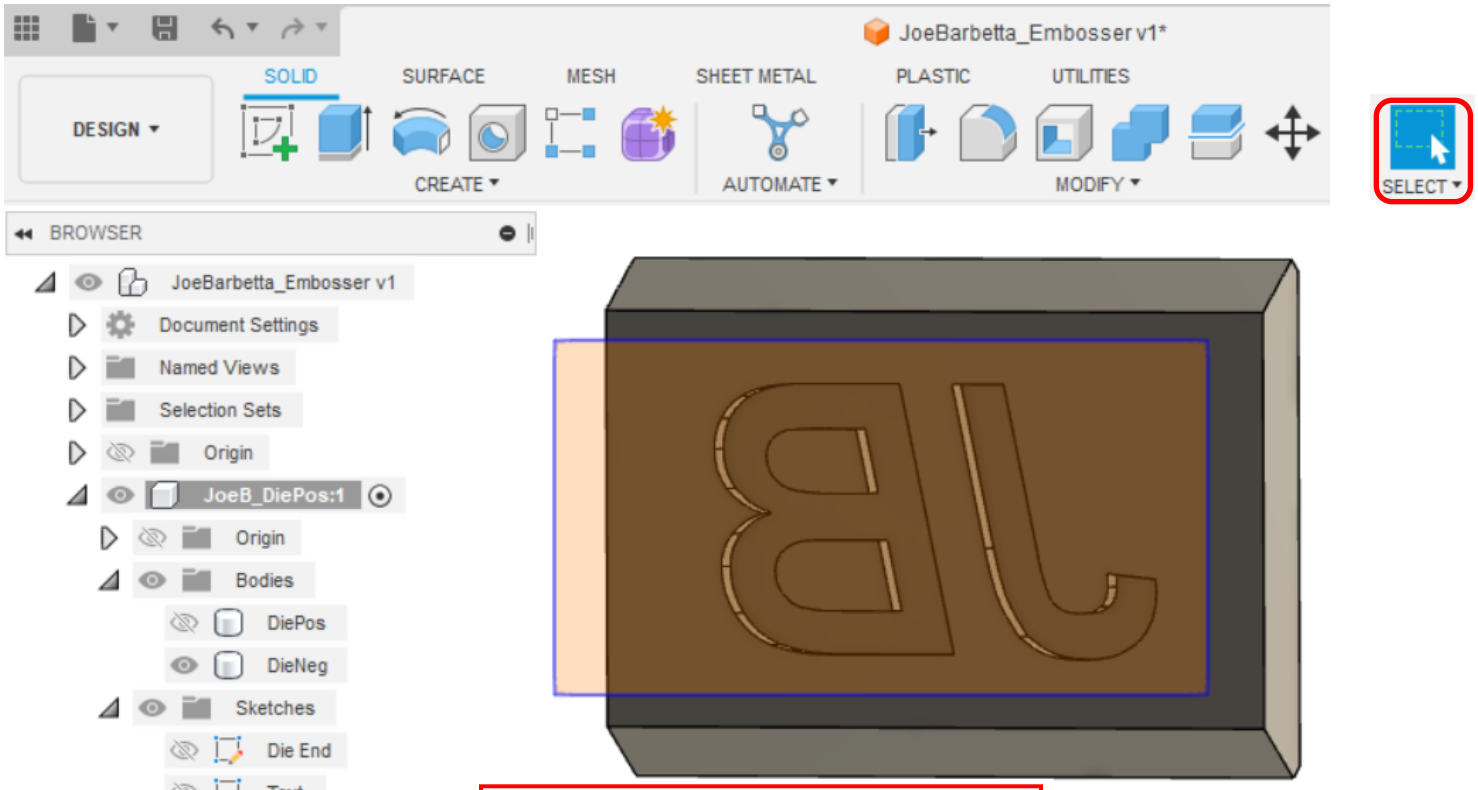
- ensure that **Select** next to **Tool Bodies** is enabled (highlighted in blue)
- click on the **DiePos** *Body*
- select the **middle Cut icon** next to **Operation** and enable **Keep Tools** and click **OK**



- click the **eye icon** for the **DiePos** *Body* and adjust the View to see that the letters were cut from the DieNeg *Body*



- click on the **Select** tool at the extreme right of the top tool bar
- **drag the selection rectangle** from the **top-left position to the bottom-right position** to encompass the entirety of the letters, which should turn the letters blue when selecting is complete. If the entire surface is blue then too much was selected.



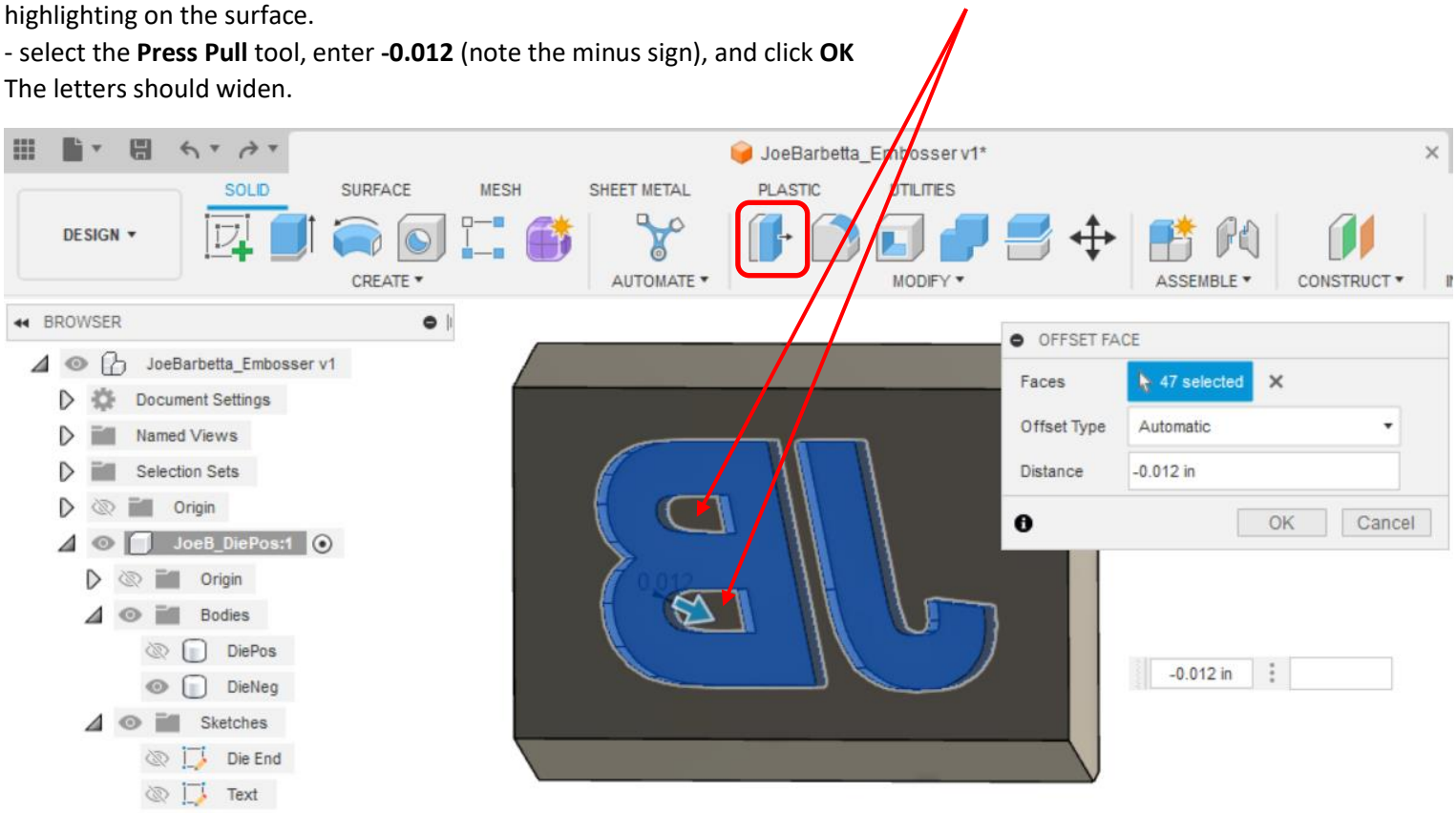
The selection rectangle is encompassing the text and **not** the edges of the face.

Using the Press Pull Tool

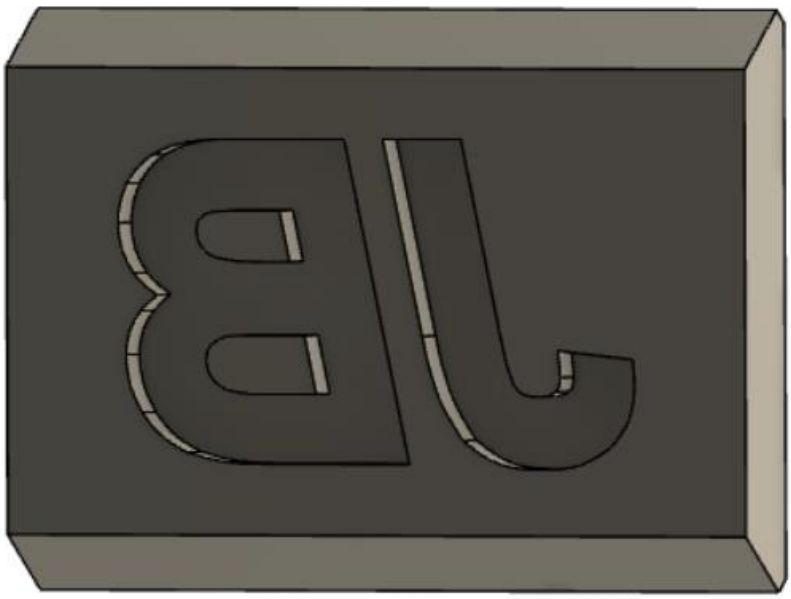
The Press Pull tool allows one to Press or Pull on one or more Faces to adjust dimensions of an Object. One can directly click on one or more faces or use the Select tool to select many faces as done here. We are using it here to create some clearance between the side faces of the letters of the two dies.

- hold the **Shift key** and click on the **"islands" of any letters**. For example, with the B below, doing so will remove the blue highlighting on the surface.
- select the **Press Pull** tool, enter **-0.012** (note the minus sign), and click **OK**

The letters should widen.

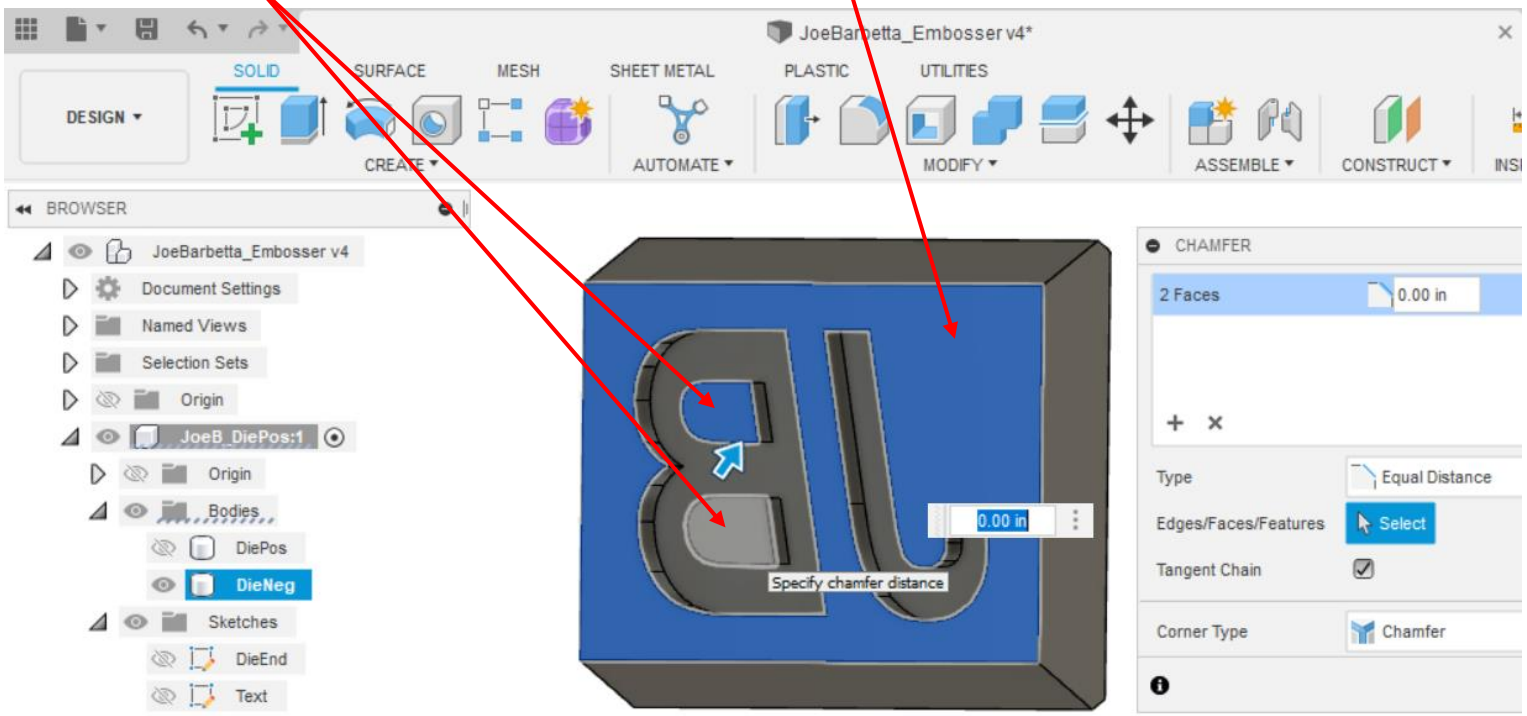


The result should look as that below.



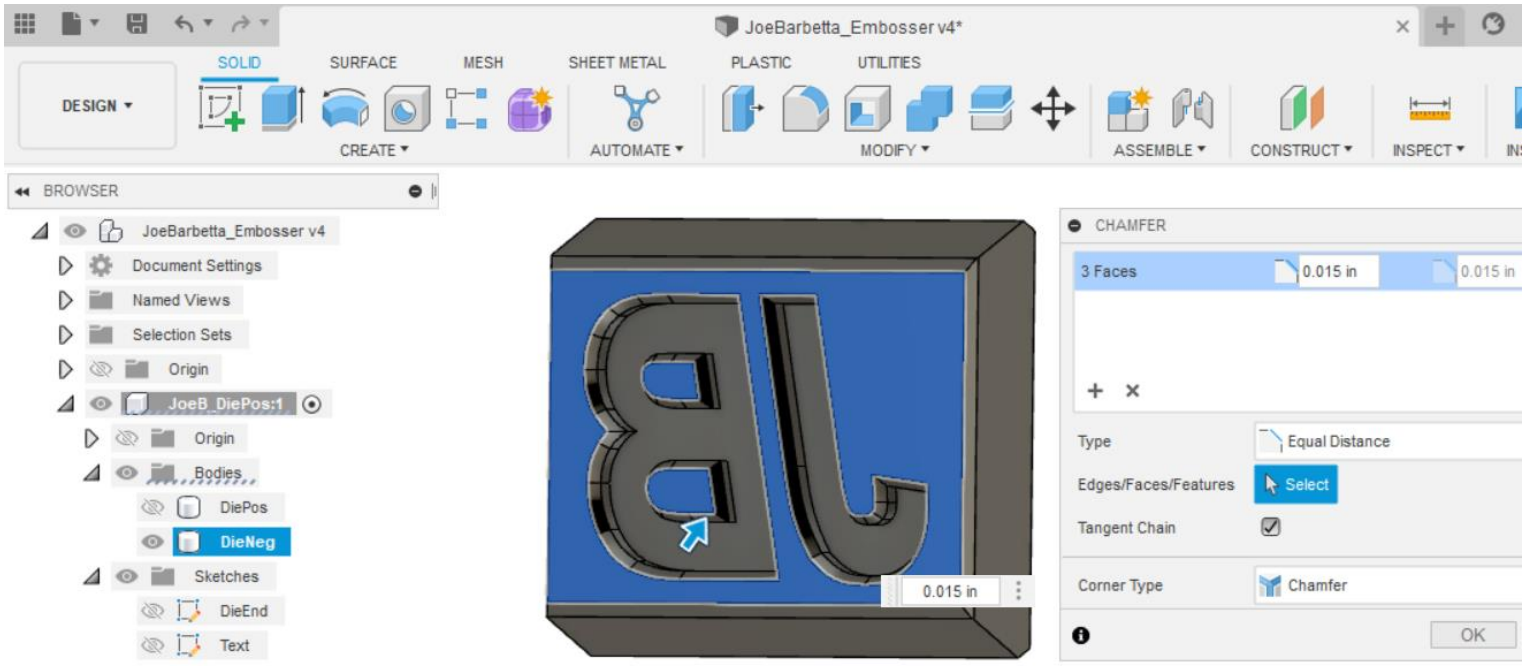
To help ensure the embosser doesn't rip the paper, we will add chamfers to the edges of the letters. Normally, one clicks on edges to chamfer, but because there are many along the curves of the letters, we will use the feature wherein one can select a Face.

- from the **MODIFY** pull-down menu select **Chamfer**. Click on the **surface of the die outside of the letters**.
- click on **interior Faces** for letters that have interior sections



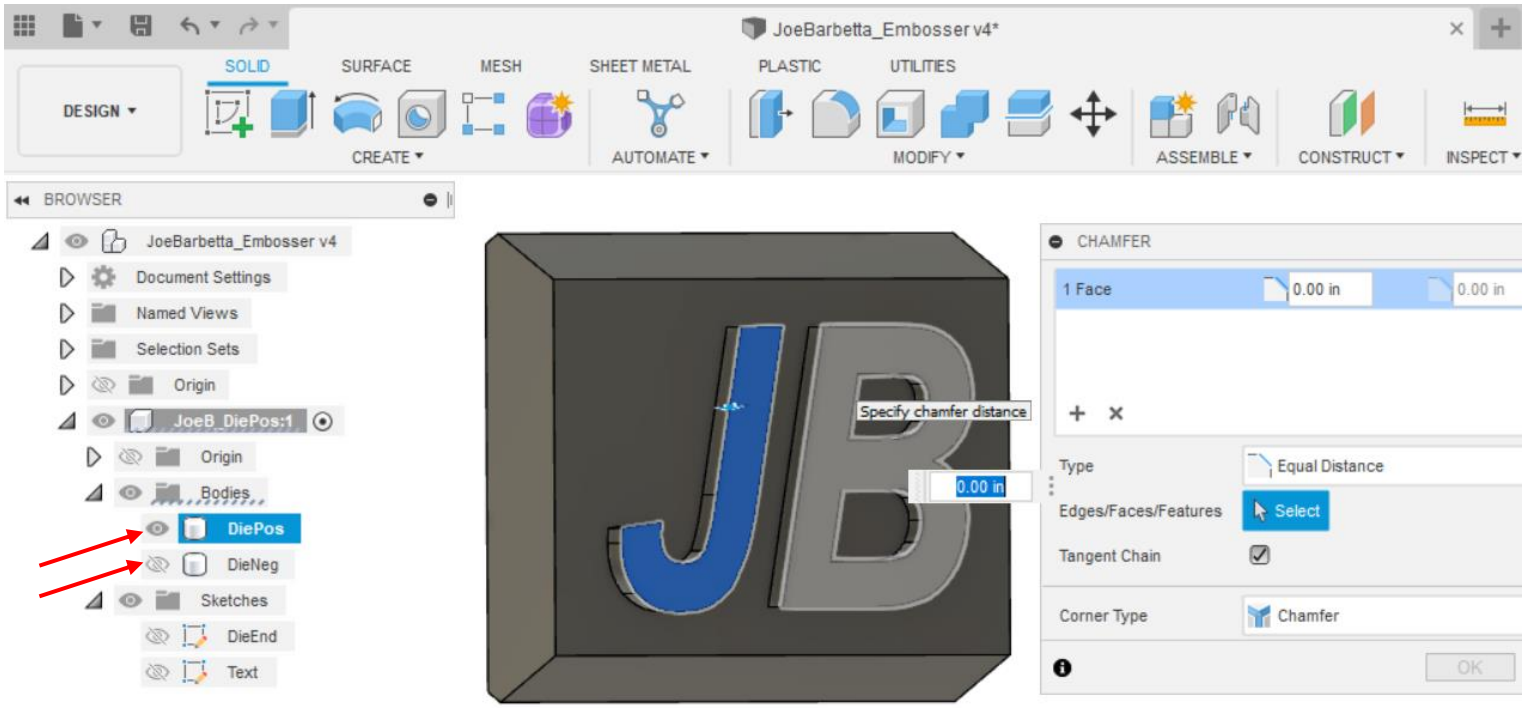
- enter **0.015** and click **OK**

If there is an error try a smaller value such as 0.010. Sometimes there may still be an error if certain letters are used or the letters are too close together. The chamfer isn't that important so this step can be ignored if not working.

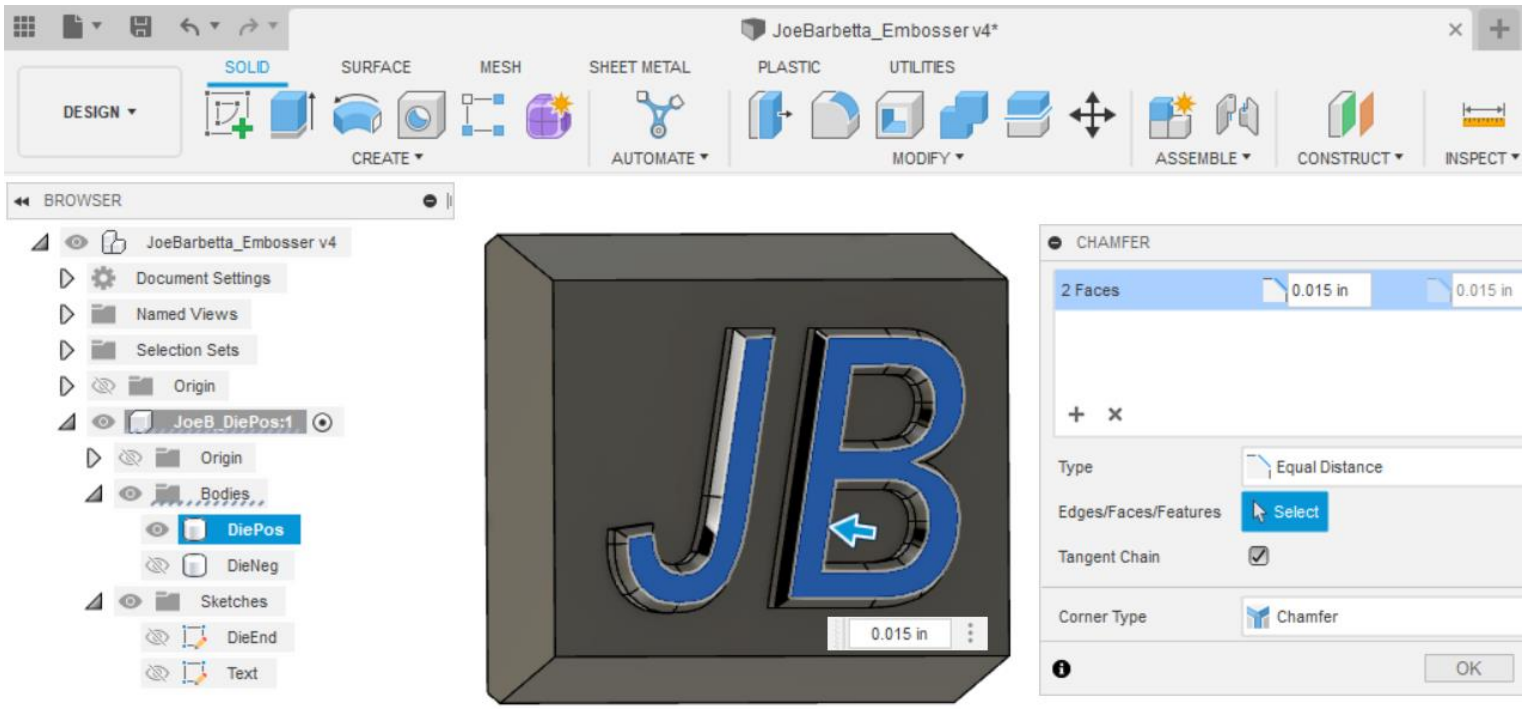


Now we want to do similarly for the Positive Die.

- click on the **eye icons** to now make **DiePos visible** and **DieNeg hidden** and adjust the view to see the letters.
- from the **MODIFY** pull-down menu select **Chamfer**. Click on the **surface of each letter**.

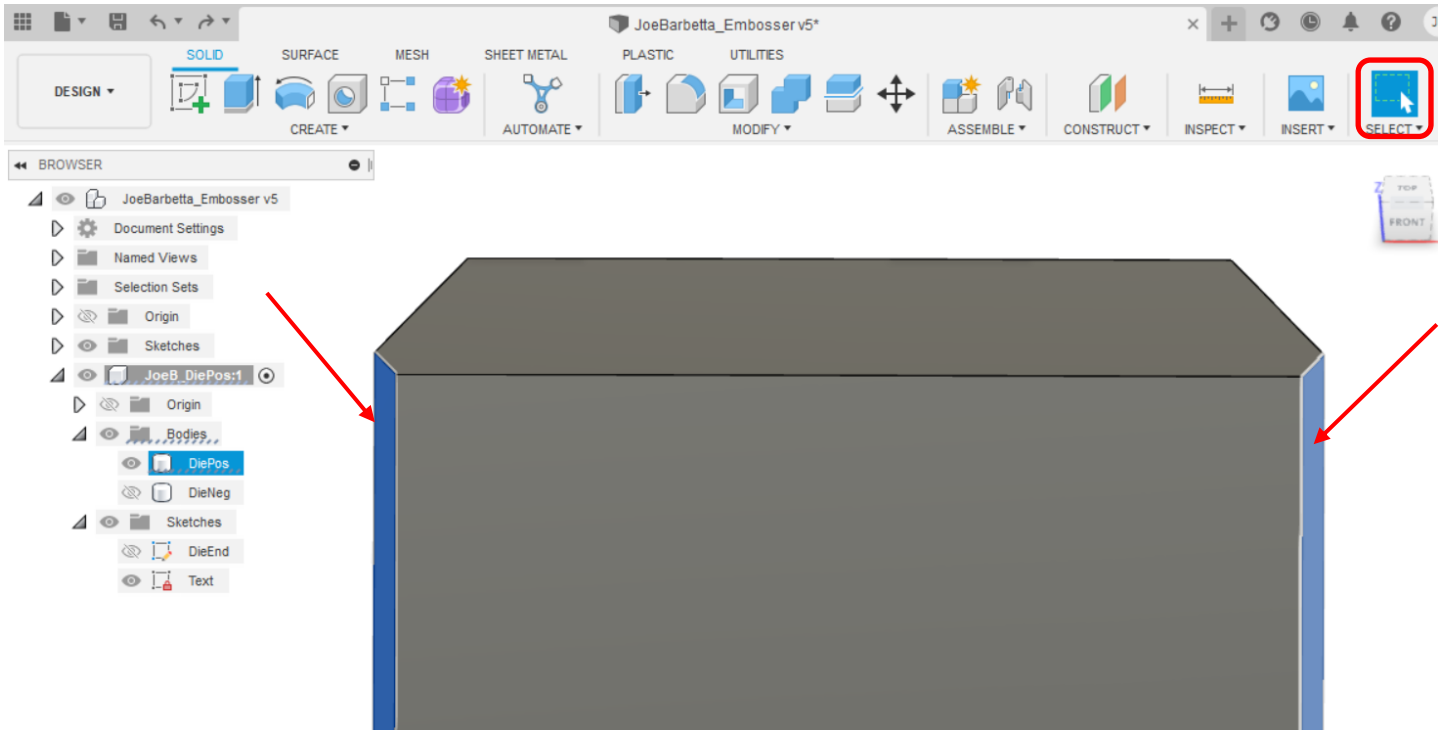


- enter **0.015** and click **OK**. As before, if an error is occurring this step can be ignored.

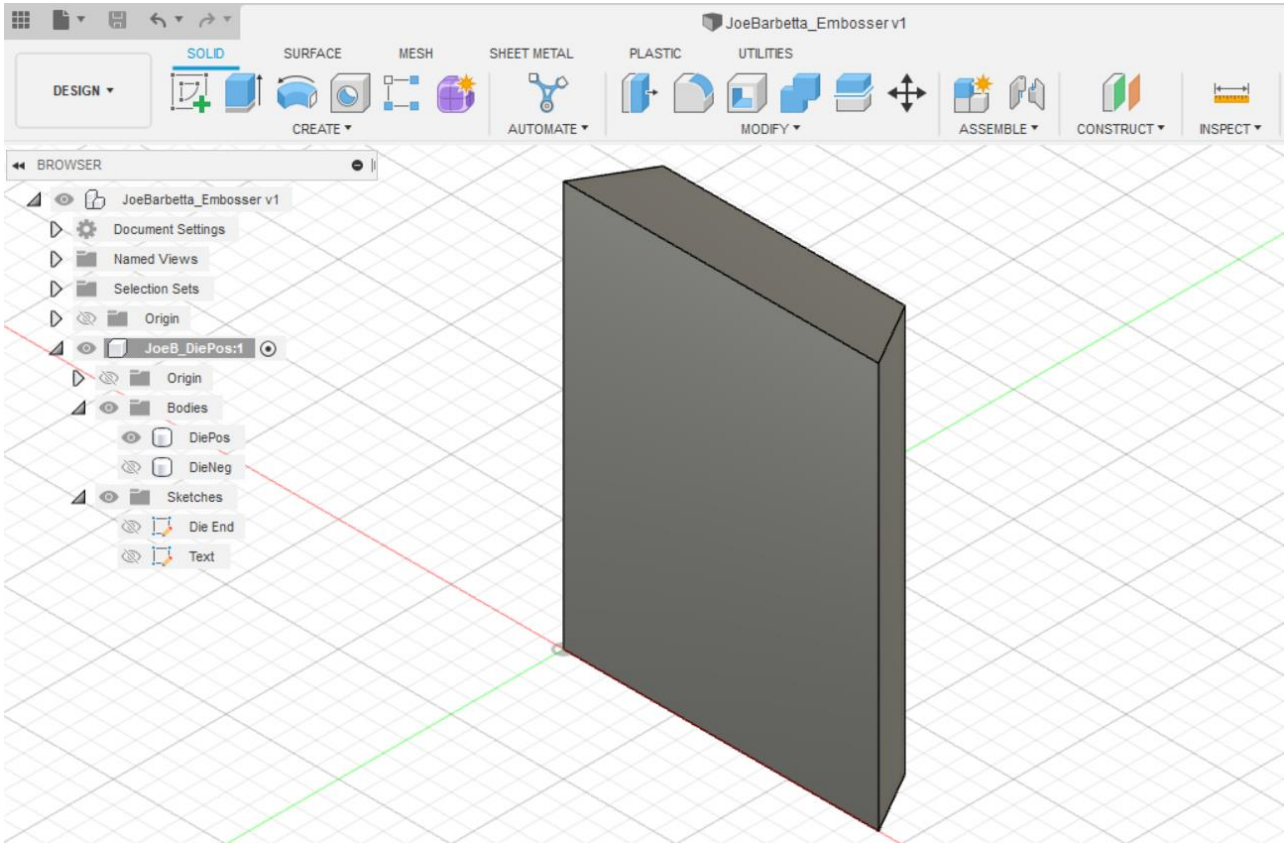


- adjust the view to access the two edge Chamfers
- click on the **Select** tool, hold the **Shift key**, and click on each **chamfer** as indicated by the arrows.
- press the **Delete key** and they should disappear.

Why did we add these chamfers in the first place? Your teacher had a brain fart and should have waited to the last steps to add them.



- select the **Home view** and the Die should look like that below



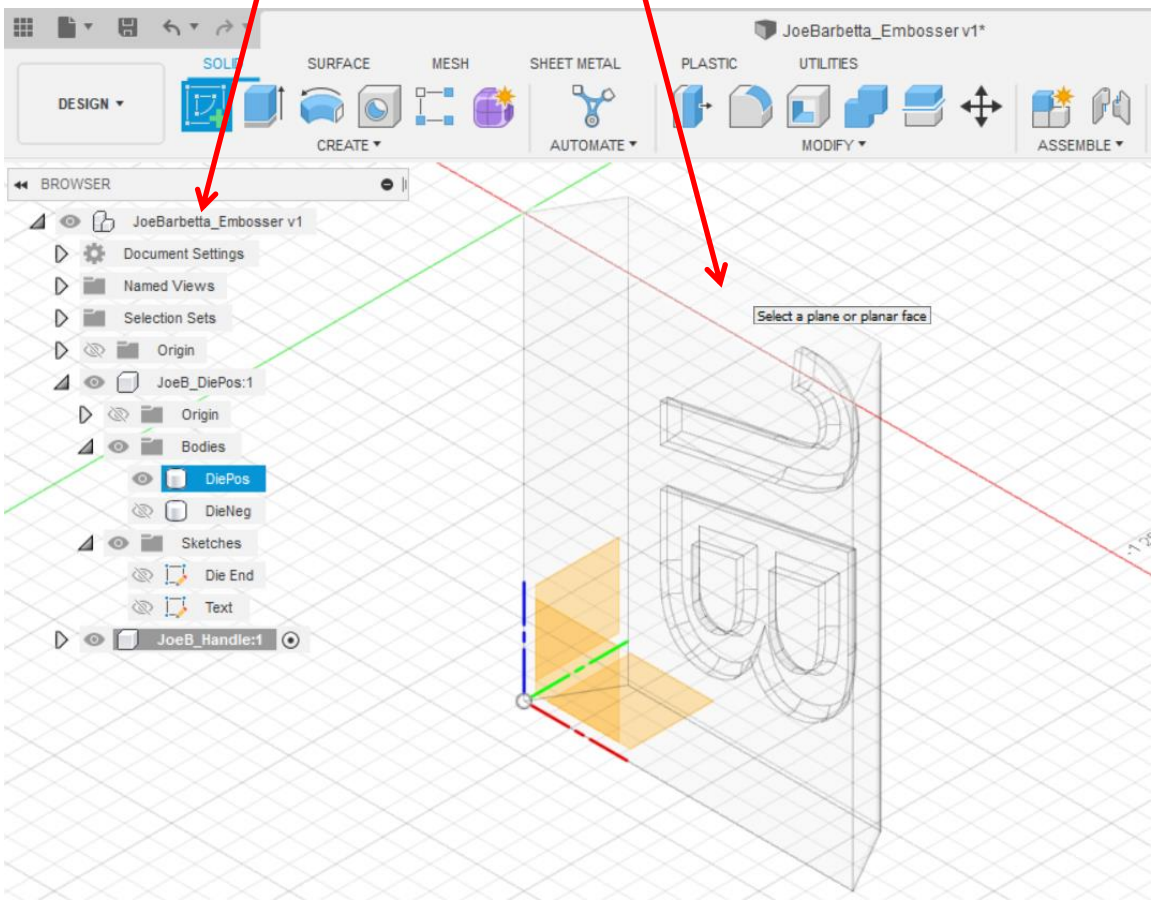
Creating the Handle with Dovetail Channels

- right-click on the **Project Name** in the **BROWSER** and select **New Component**
- set **Name** as **Your first name and last name initial followed by _Handle** e.g. **JoeB_Handle** and click **OK**

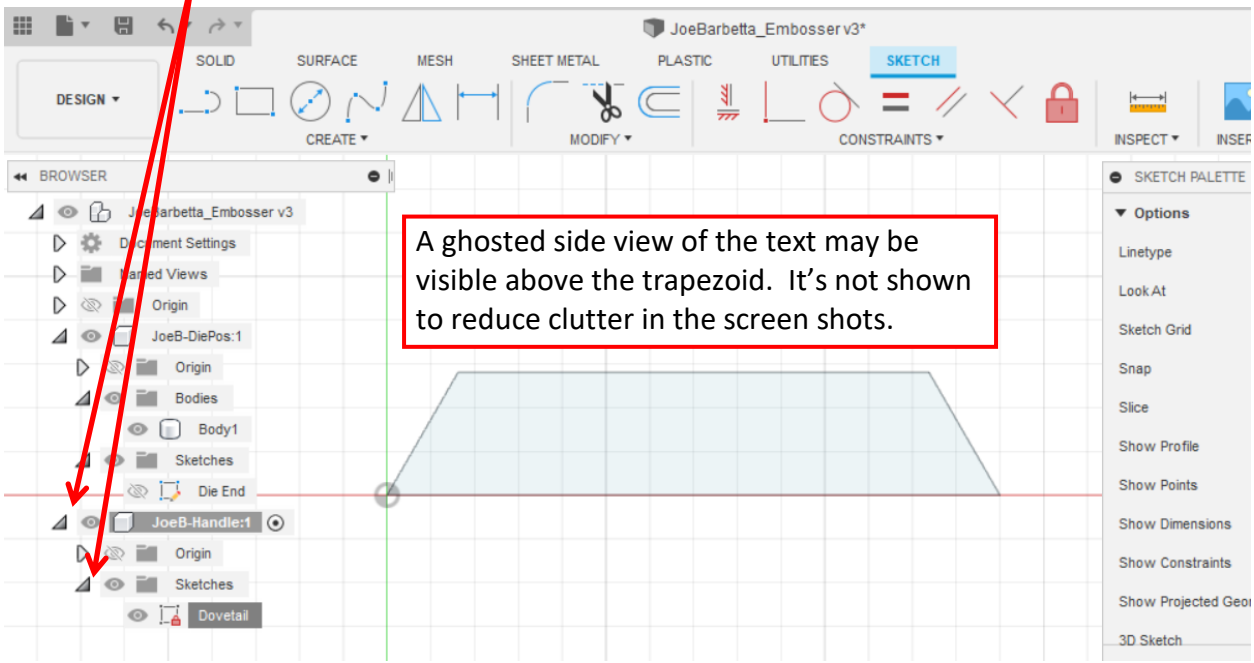
Note how the DiePos Component becomes ghosted, which indicates that it is no longer the Active Component.

- select the **Create Sketch** tool and click on the **top face of the ghosted DiePos body**.

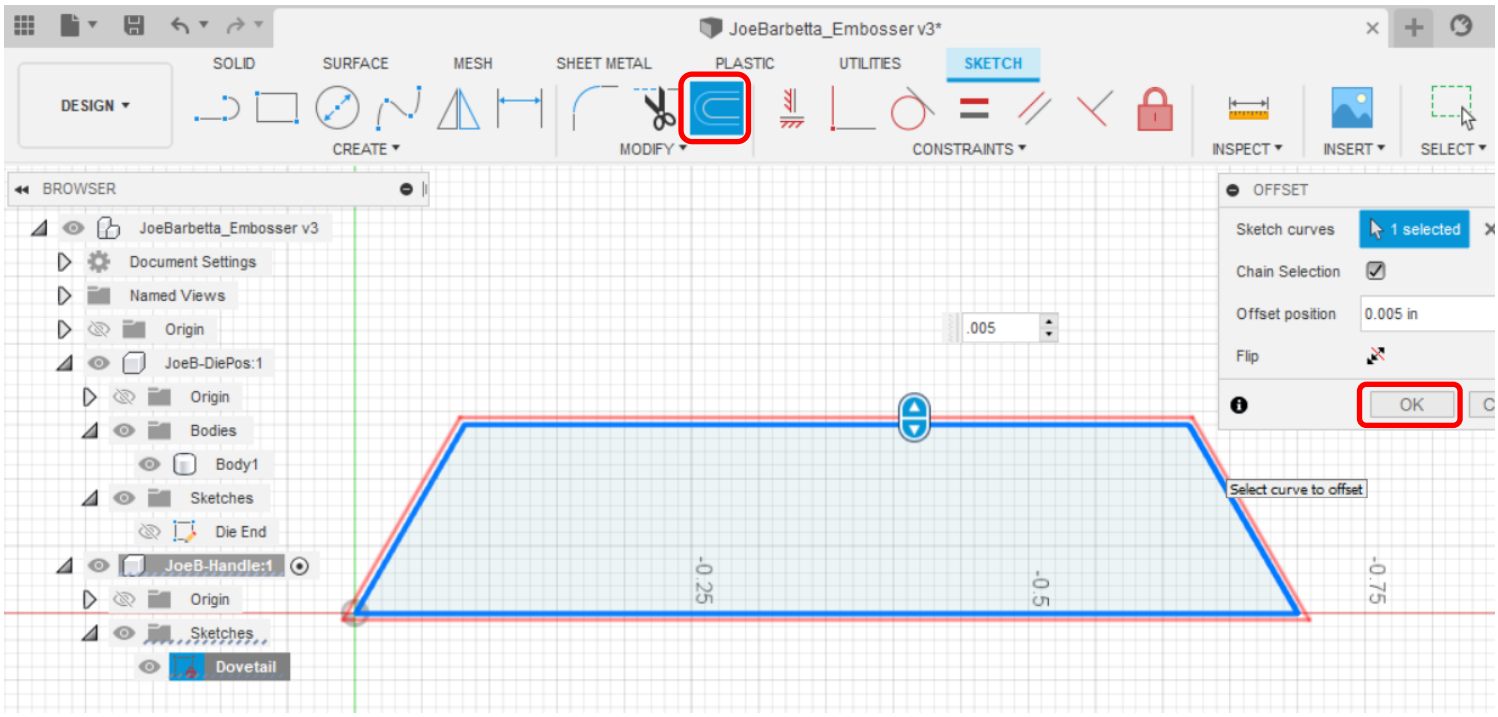
Remember Not to click Finish Sketch until directed to do so.



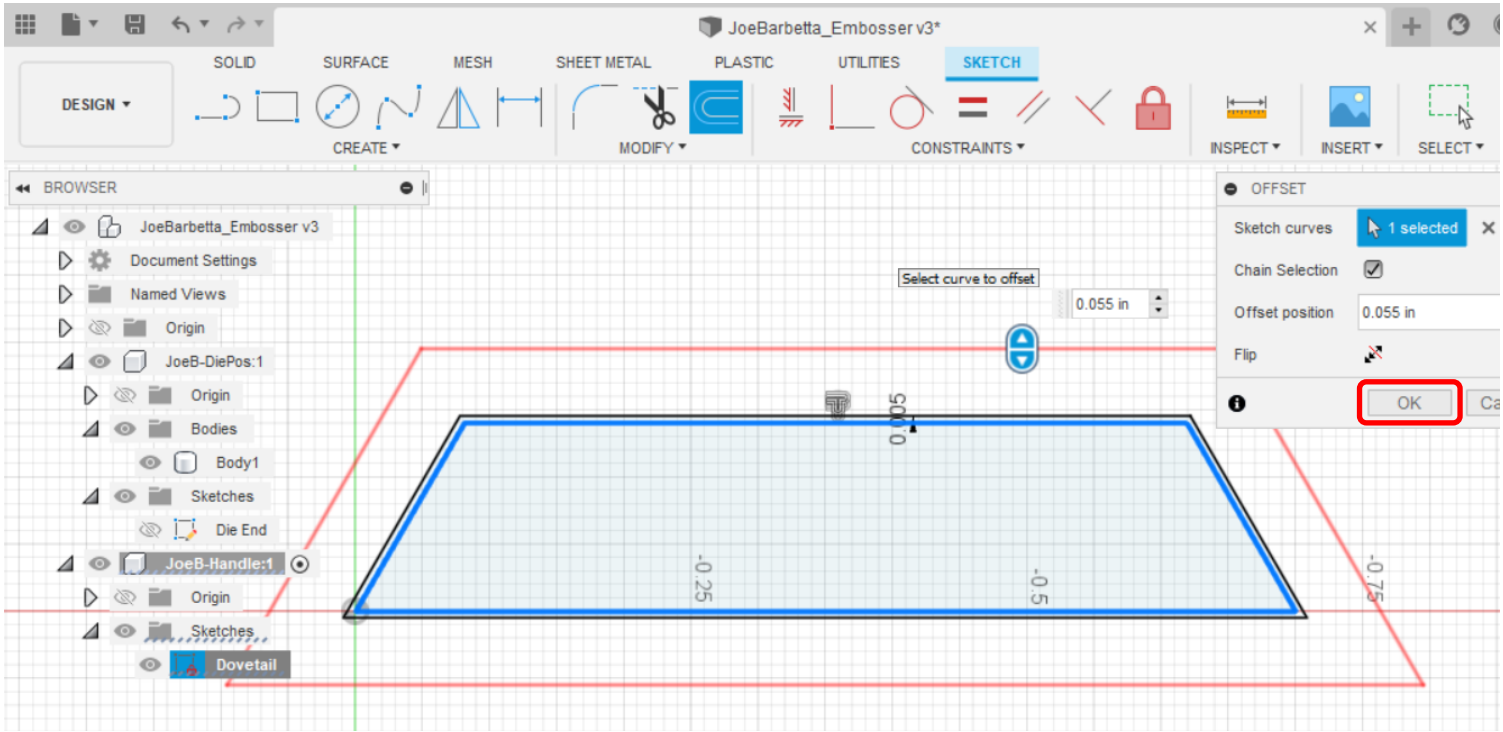
- if needed rotate the view using the **curved arrows at the top right on the View Cube** and zoom in.
- click on the arrows in the **BROWSER** to open the **Handle** Component and its **Sketches** and rename the Sketch to **Dovetail** by right-clicking on the Sketch name and selecting **Rename**. The Dies will slide into the *Dovetail slot*.



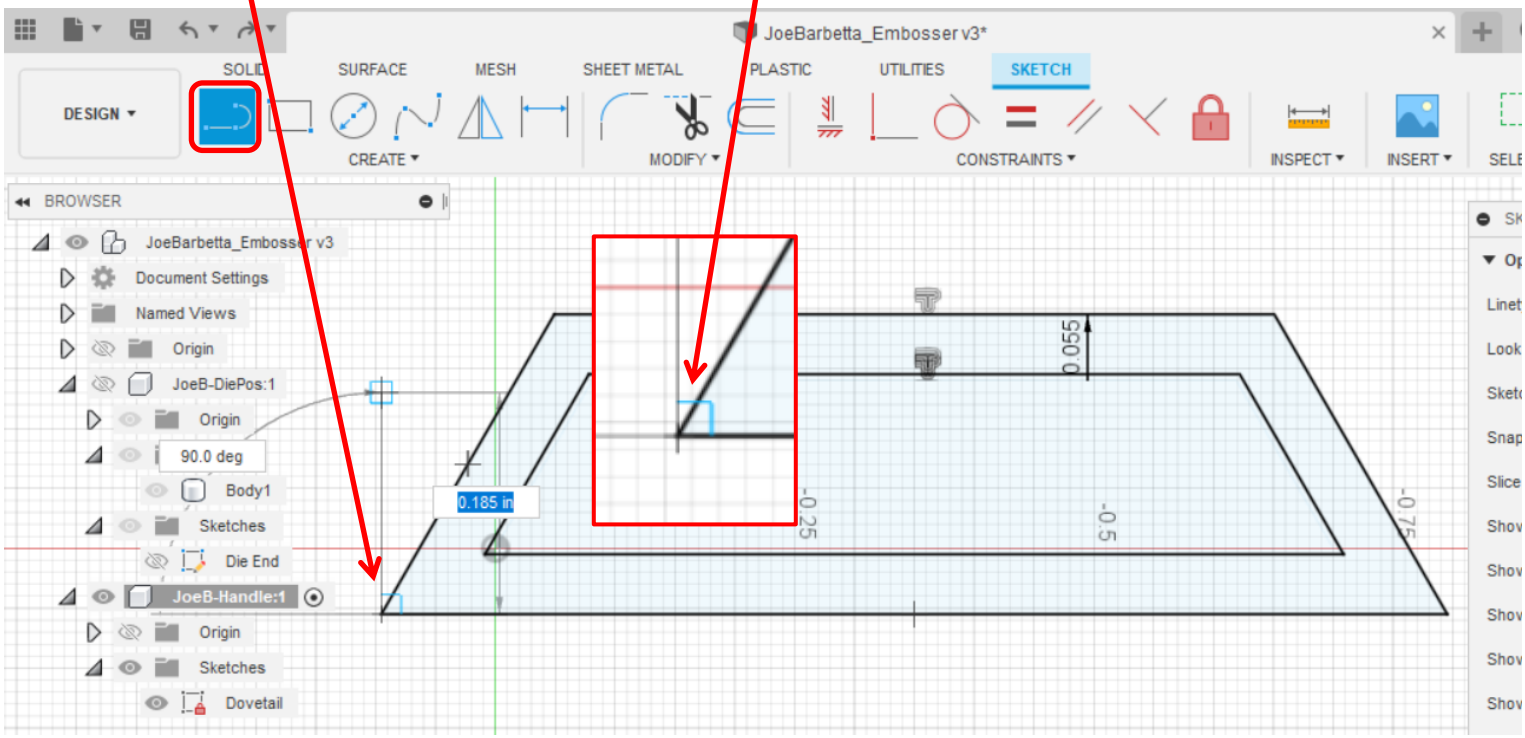
- select the **Offset** tool and click on the border of the ghosted DiePos body and enter **0.007**” and click **OK**



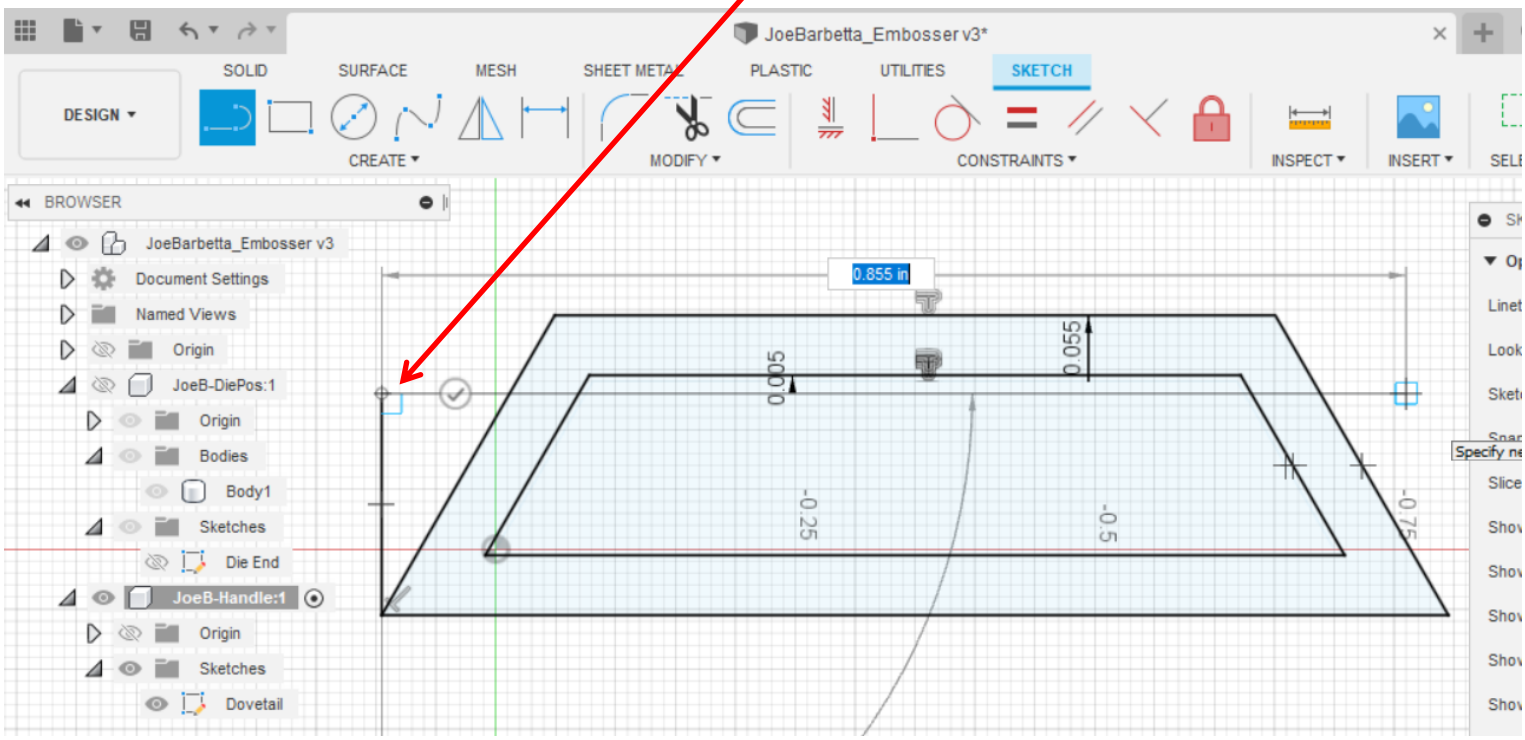
- select the **Offset** tool again and click on the border of the ghosted DiePos body again and enter **0.055**” and click **OK**
If there is an error you must click on the original DiePos body border and not the result of the last Offset.



- starting at the left bottom corner of the outer trapezoid, create a line upward, type **0.185**, and **click**
As the line is being extended up, ensure the **blue right-angle symbol** is visible and shown in the close-up view.

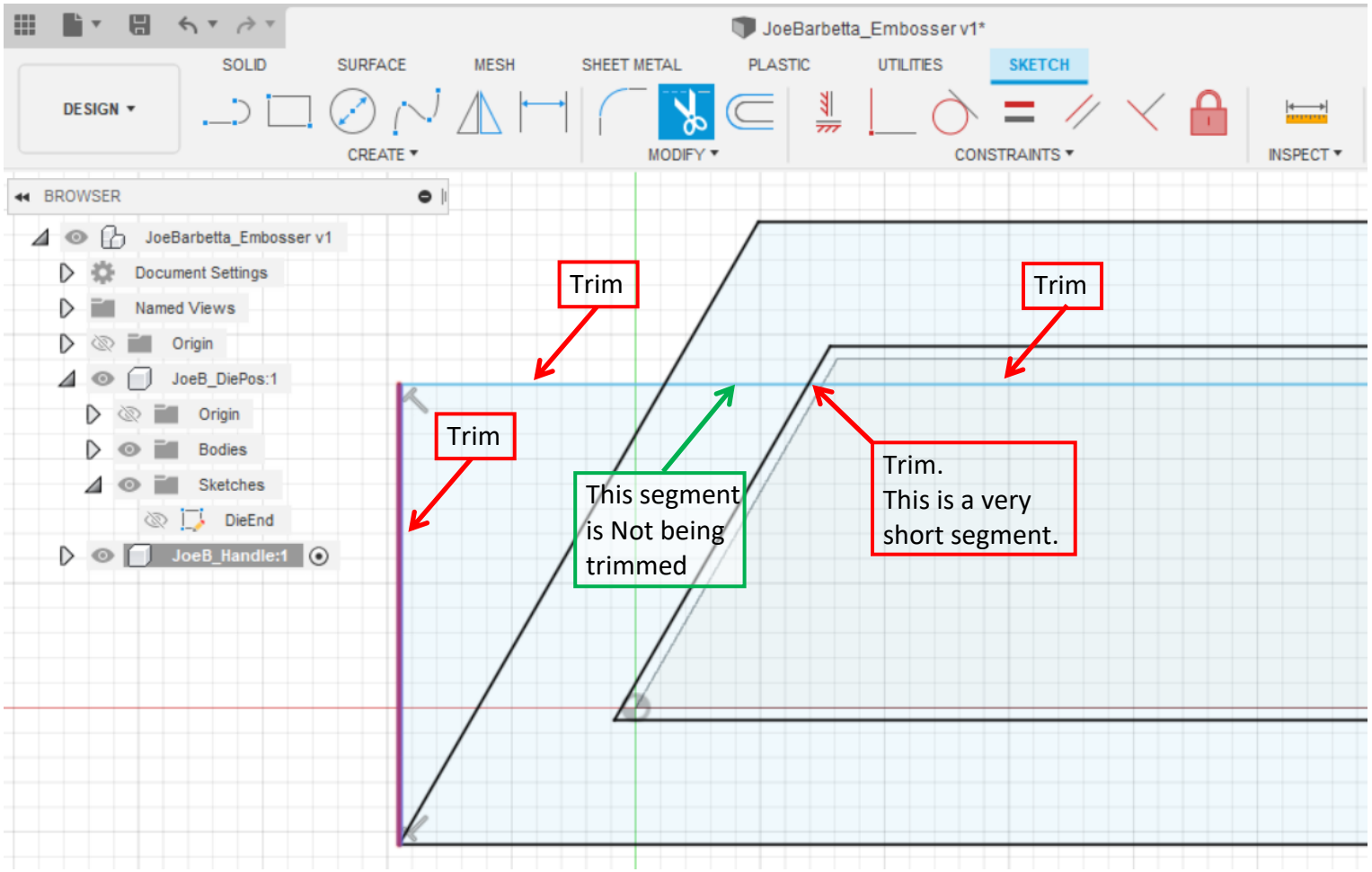


- continue drawing a line from that last position over to the right, type **0.855**, **click**, and press the **Esc key**
It's not critical how long this line is, as long as it passes through all lines of the trapezoid.
As this line is being extended to the right, ensure the **blue right-angle symbol** is visible.



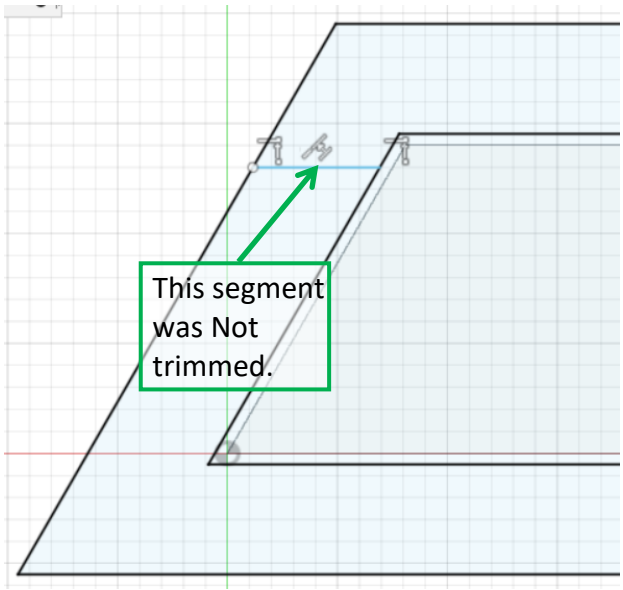
Using the Trim Tool

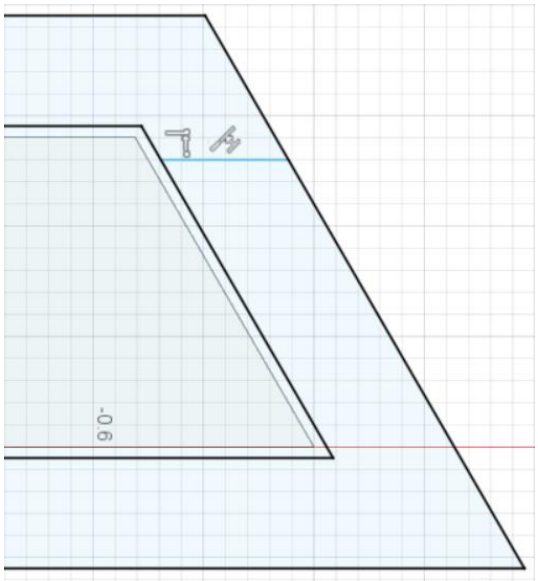
- select the **Trim** tool and **click on the blue line segments** indicated by the arrows. Note that the 3rd arrow points to a very short segment. View the photos under this one to see the results. .You can ignore any warnings.



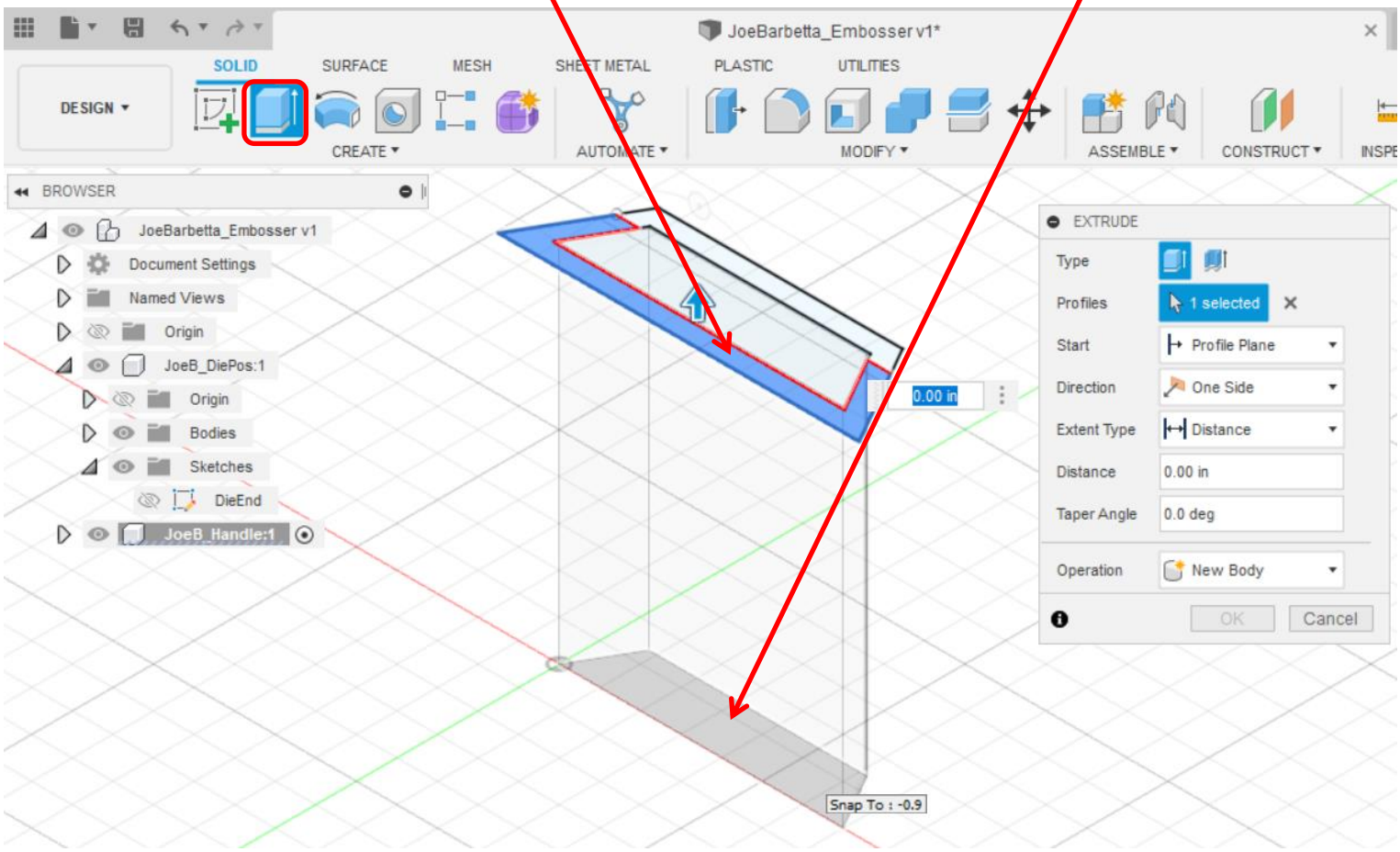
- perform the same trimming at the right side of the trapezoids

Here you can see the results when zoomed in to each end of the trapezoid. Note that each blue line extends between two black lines. Click **Finish Sketch**.

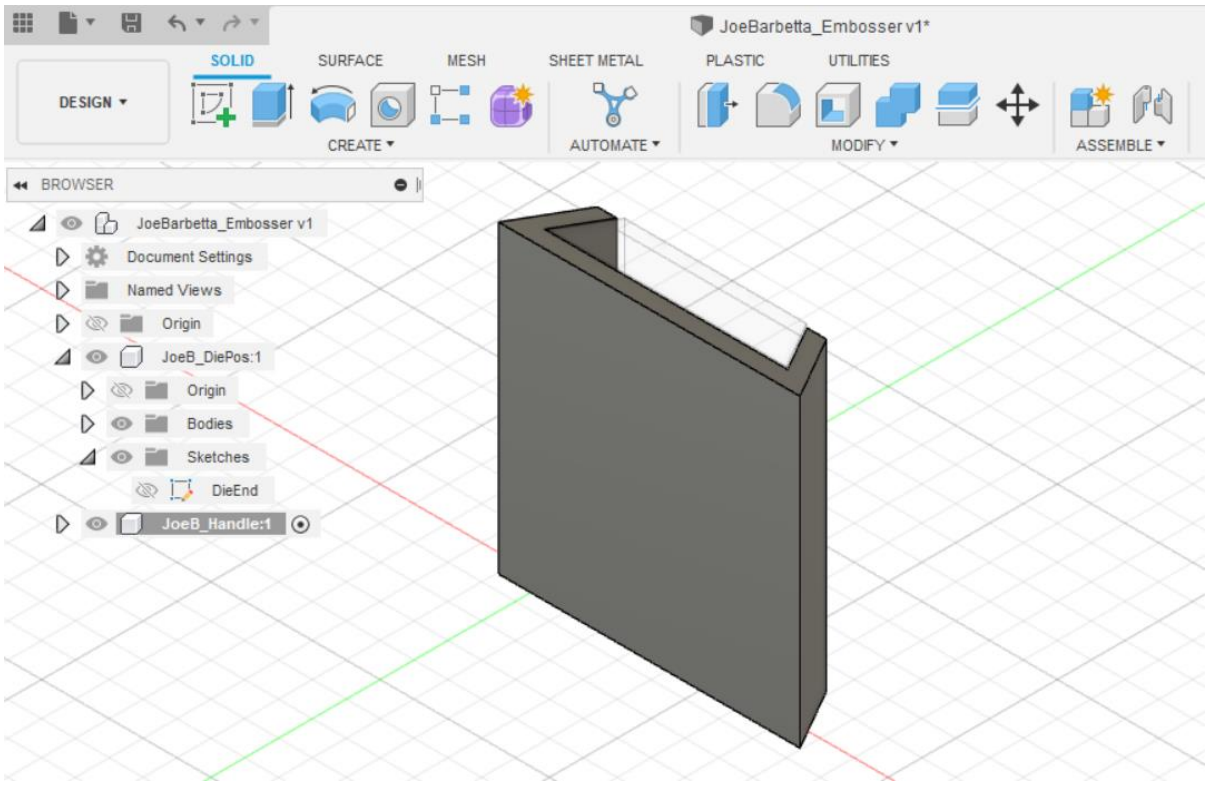




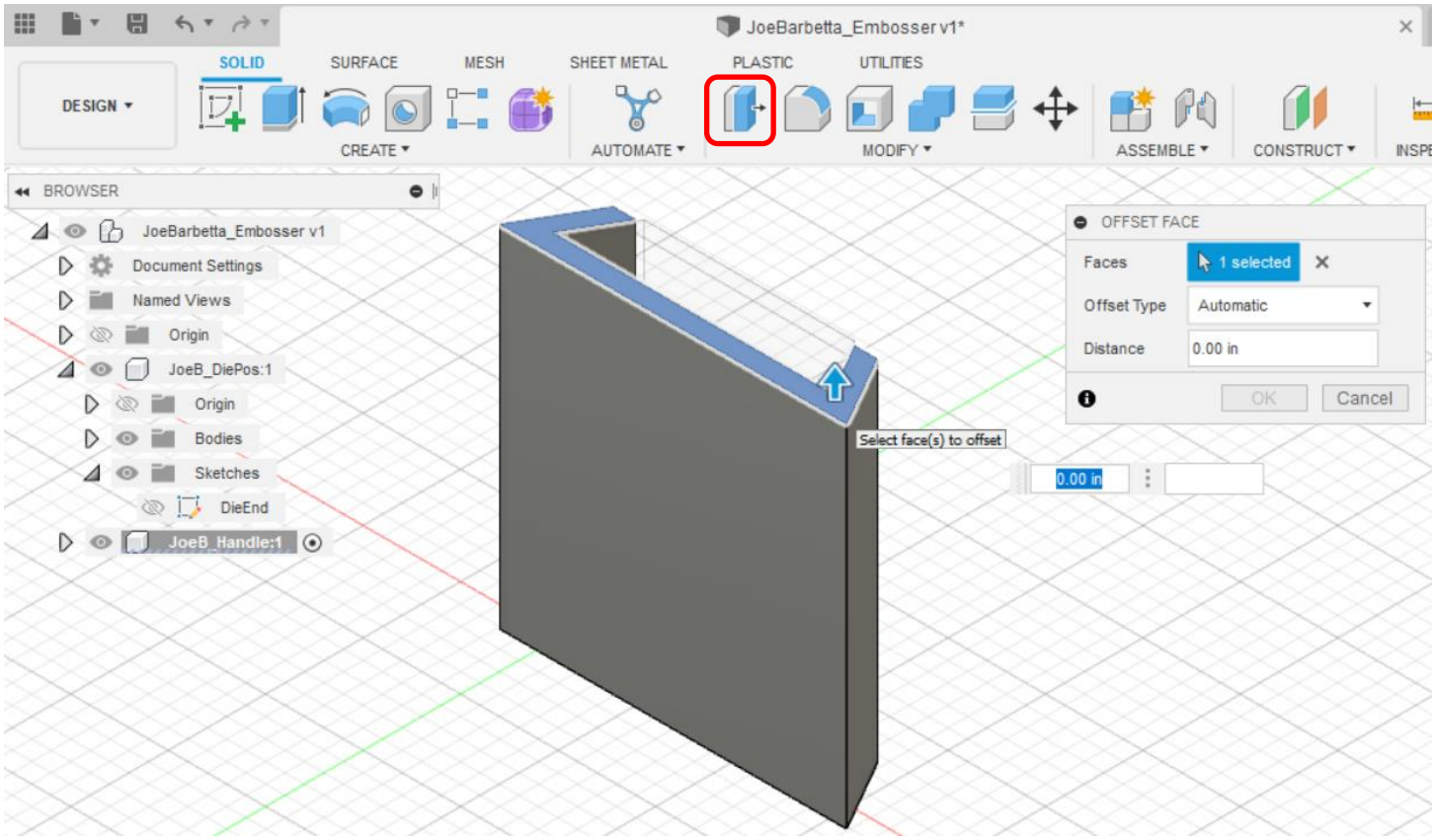
- select the **Home** view
- select the **Extrude** tool and click on the top region (it will turn blue) and then click on the bottom region on the lower Plane and click **OK**. Ensure that only the larger region of the trapezoid is extruded down and not the smaller region.



The Dovetail channel should like as below.

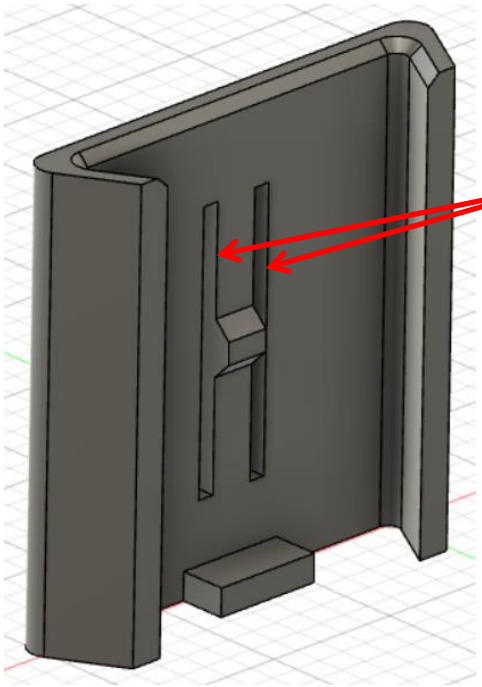


- select the **Press-Pull** tool and click on the top region (it will turn blue), enter **-0.075** (note the minus sign), and click **OK**
The end should move down slightly.



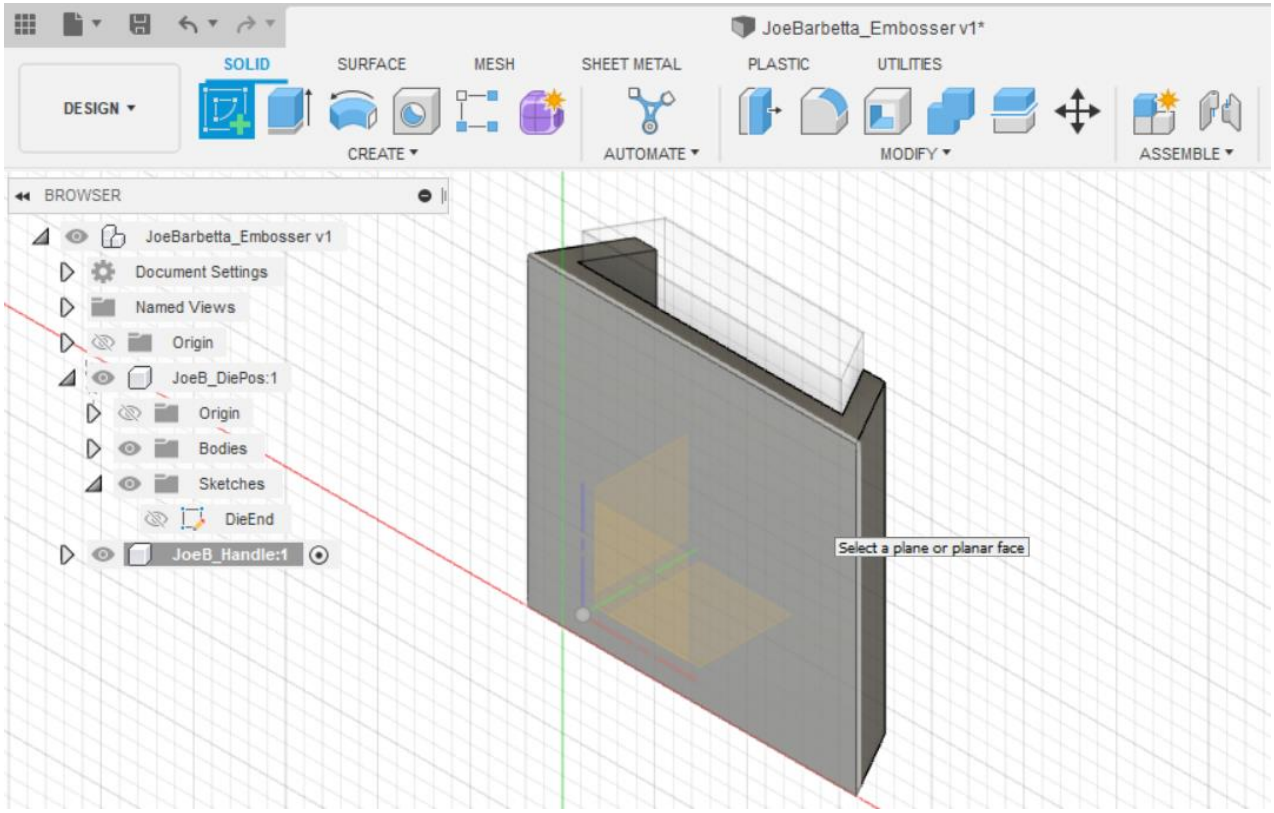
Creating a Spring Mechanism

Unless you're ready to spend several thousand dollars on an injection mold, we must deal with lower accuracies of 3D Printing. We can't always count on the Die being held securely in the Dovetail channel. The following steps will cover the creation of two thin slots with a bump in the middle. When a Die is slid into the channel, the bump will be displaced outward slightly and maintain pressure on the Die.



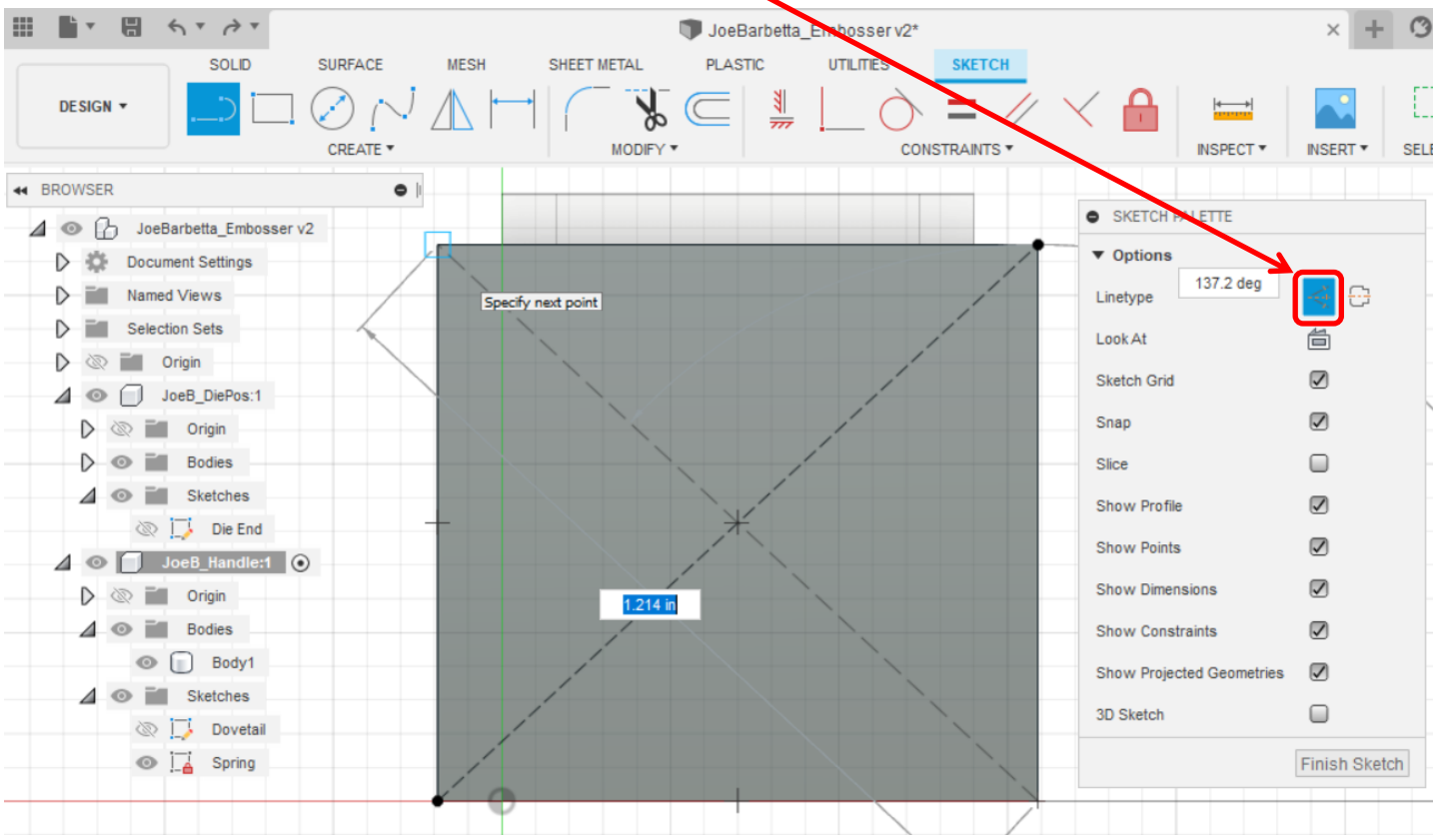
The two slots allow some deflection of the plastic between the slots. When a Die is inserted it will displace the bump outward and it will impart a force on the Die to hold it in place.

- select the **Sketch** tool and select the rear of the dovetail channel.



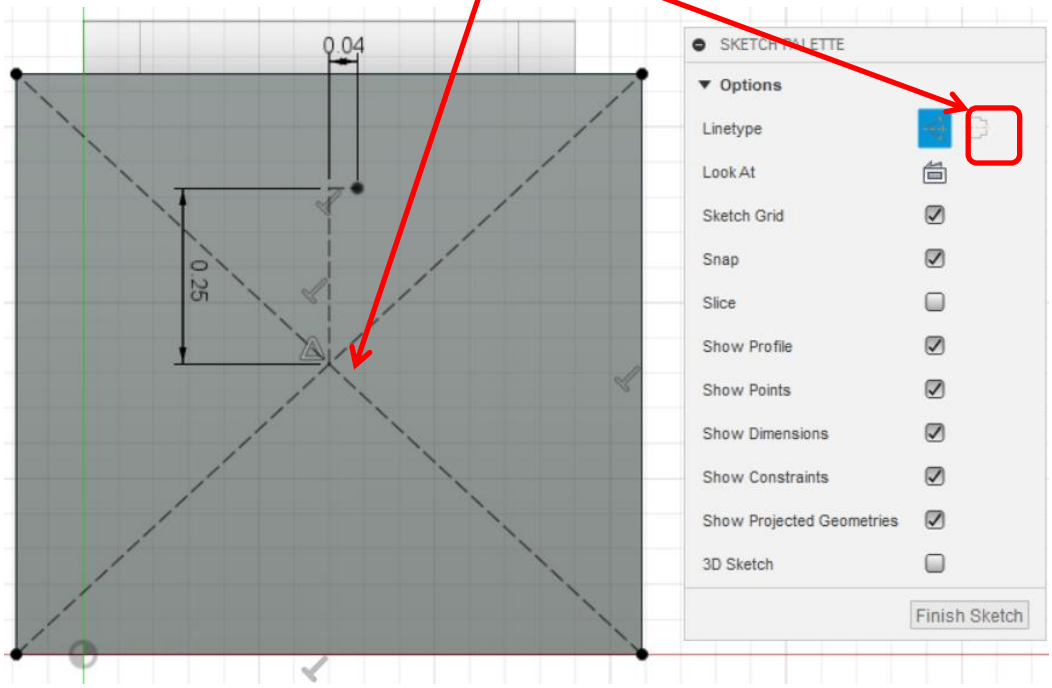
X Marks the Spot! A common method for finding the center of a surface when either hand drawing lines or using CAD is to create lines connecting opposite corners. The intersection marks the center of the surface.

- select the **Line** tool and then click on the **Construction icon** to the right of **Linetype** to turn it blue.
- click on diagonally opposite corners to create two lines as shown below. No dimensions need to be entered.

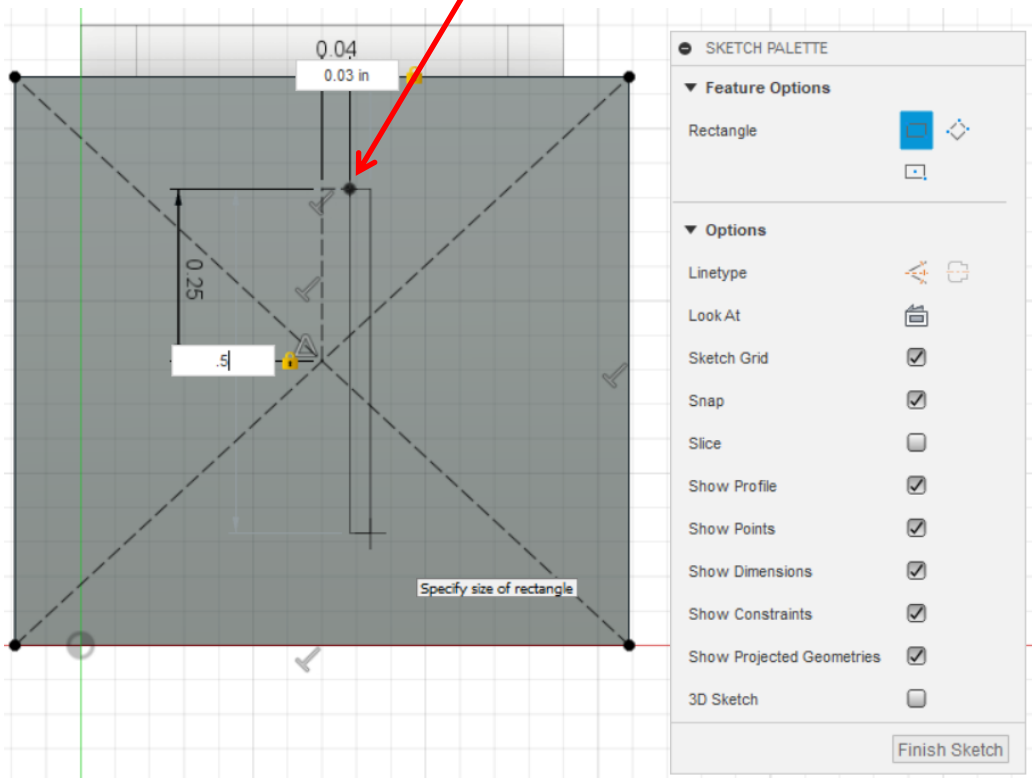


- create a Line starting at the **intersection of the lines** and upward by **0.25** and then to the right by **0.04** press the **Esc key**. Don't worry if the thin black dimension lines look different. It's the dashed lines that count. Dashed lines indicate that they are *Construction Lines*, which are used for positioning other lines.

- after creating these lines, click on the **Construction icon** again to remove the blue highlight.

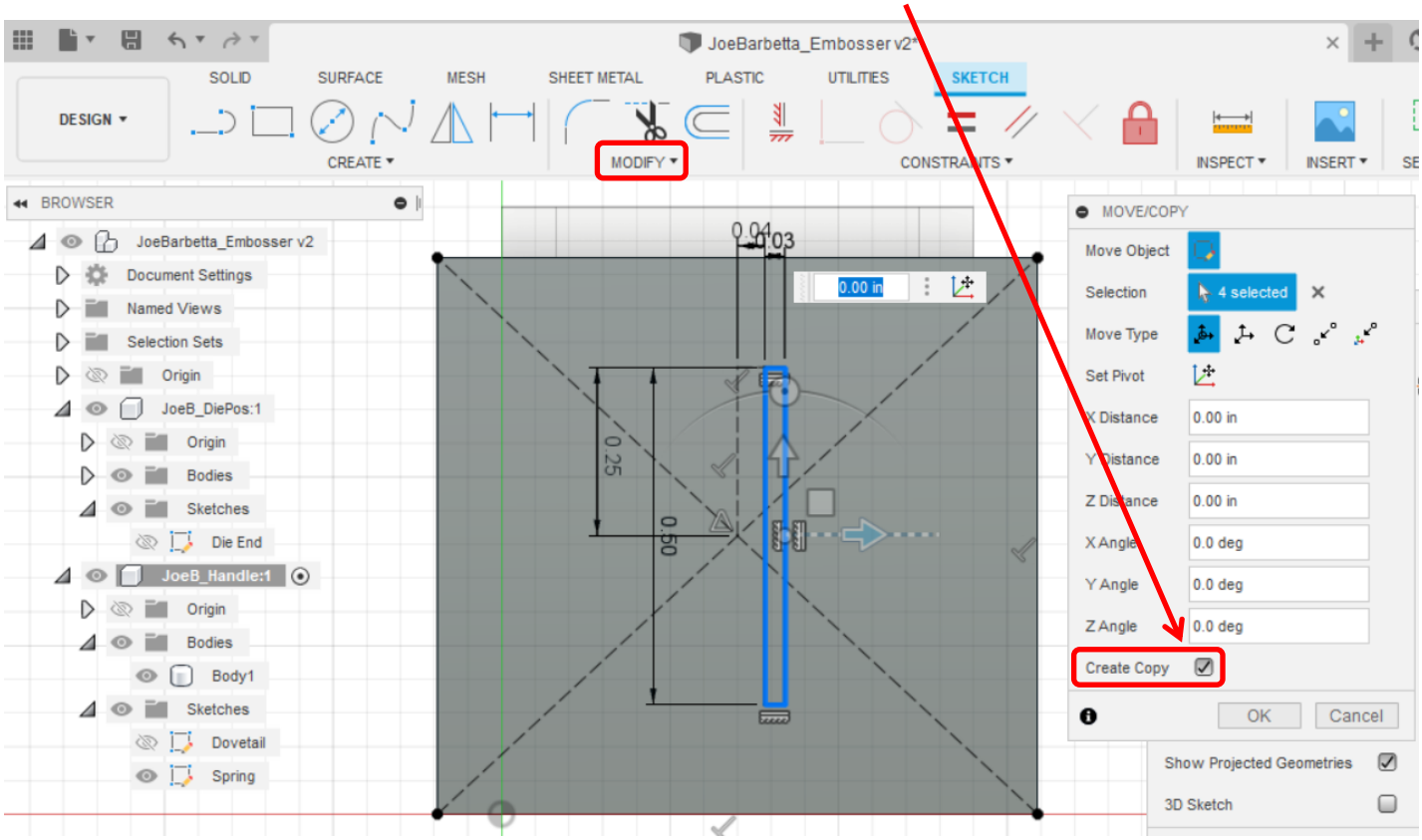


- use the **Rectangle** tool and click on the **end of the dashed line**, as shown by the red arrow below. Enter **0.03** for the width and **0.5** for the height. One can use the **Tab** key to switch between the two dimensions if needed.

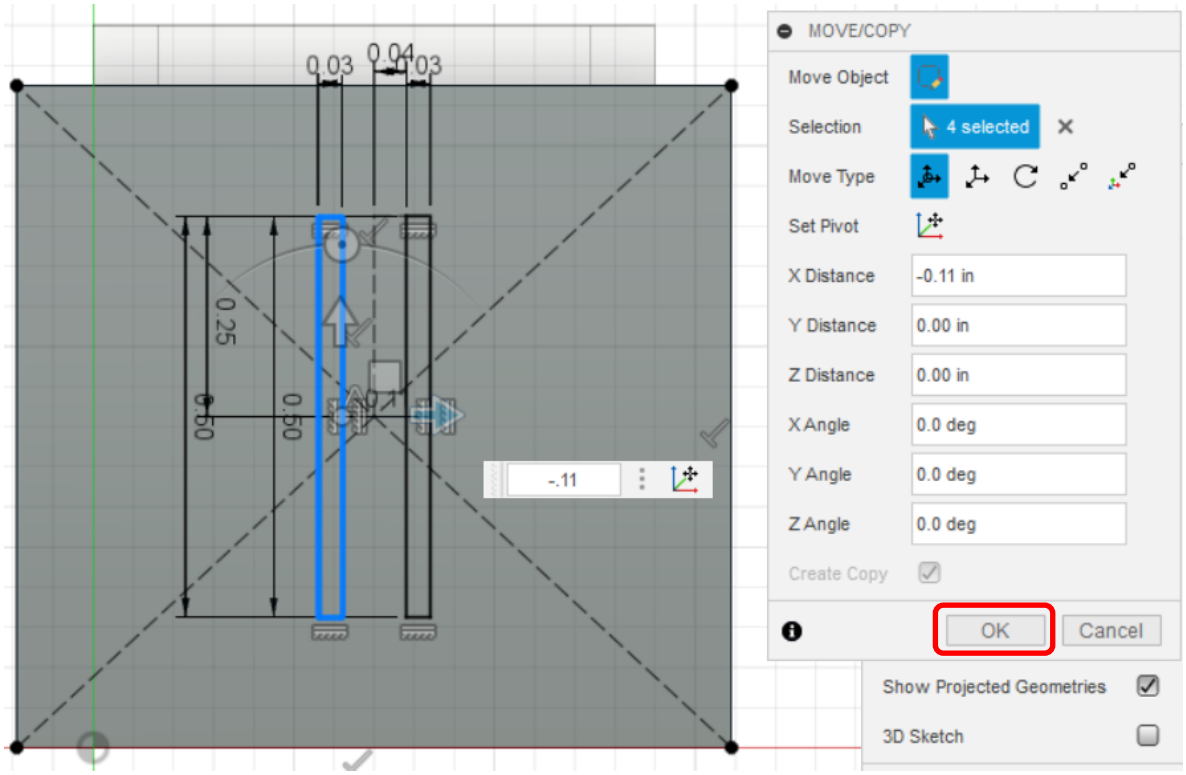


Using the Move/Copy tool

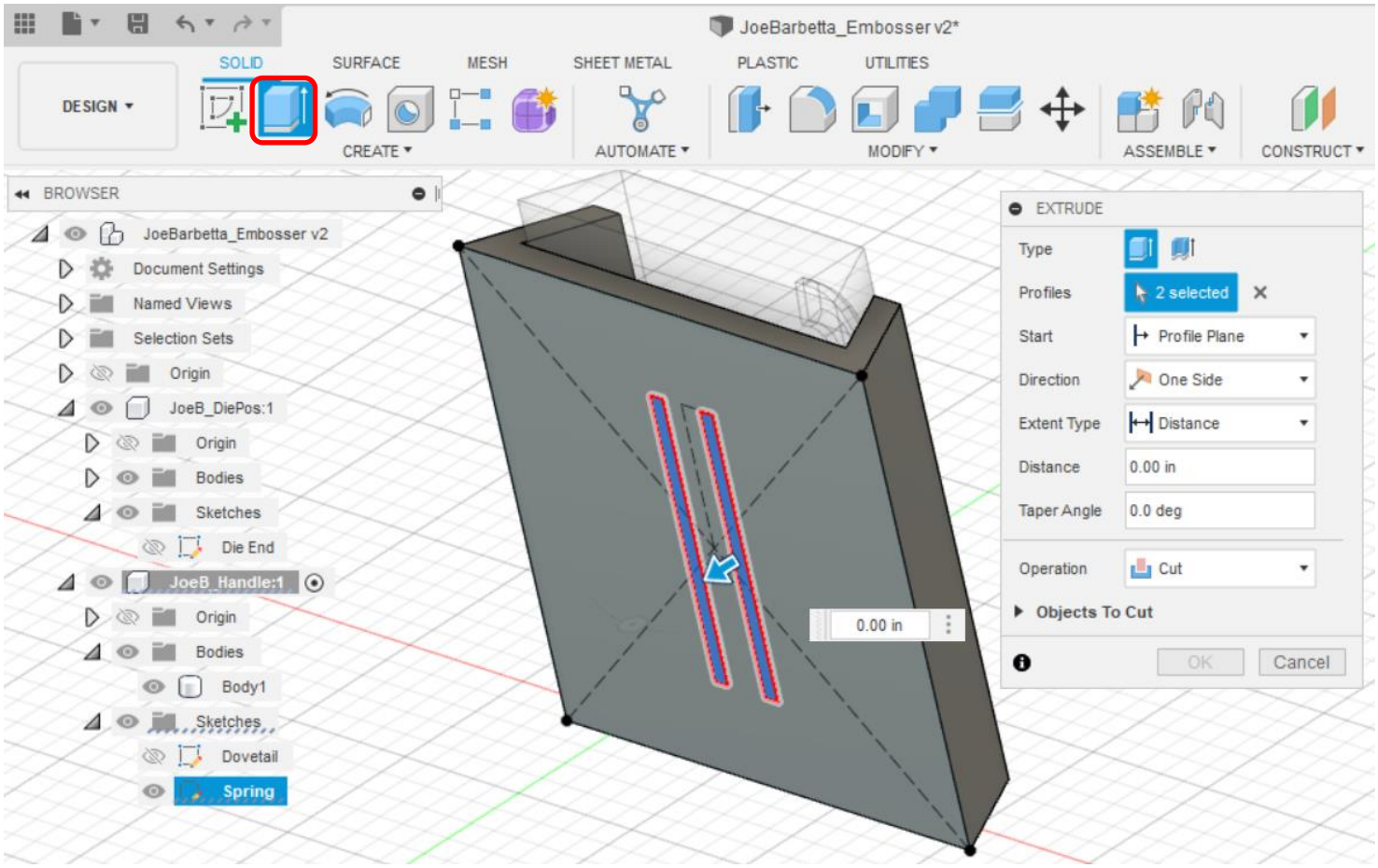
- select **Move/Copy** in the top pull-down **MODIFY** menu
- **double-click on an edge** of the rectangle you just created, which should turn each edge dark blue.
- near the bottom of the MOVE/COPY window click on the box next to **Create Copy** to enable it.



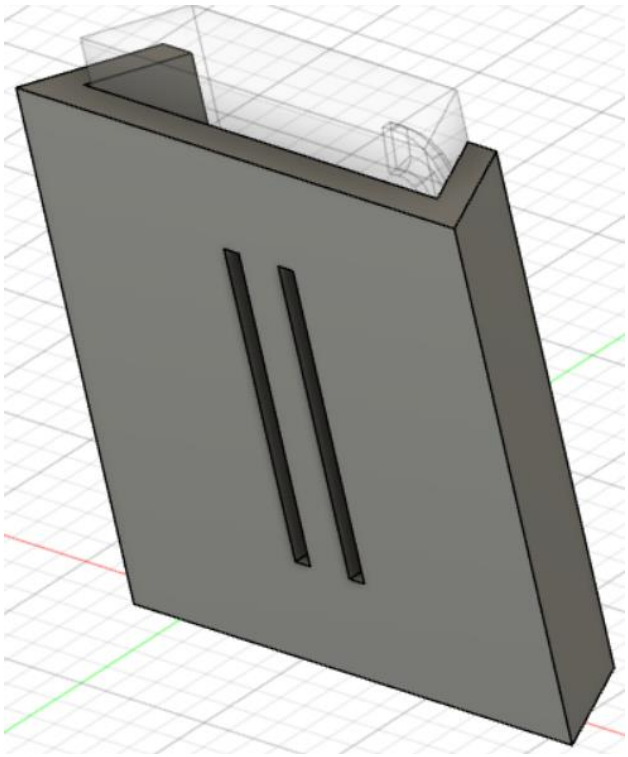
- **drag the horizontal arrow to the left** and enter **- 0.11** (note the minus sign) in the pop-up box
- click **OK** and then **Finish Sketch**



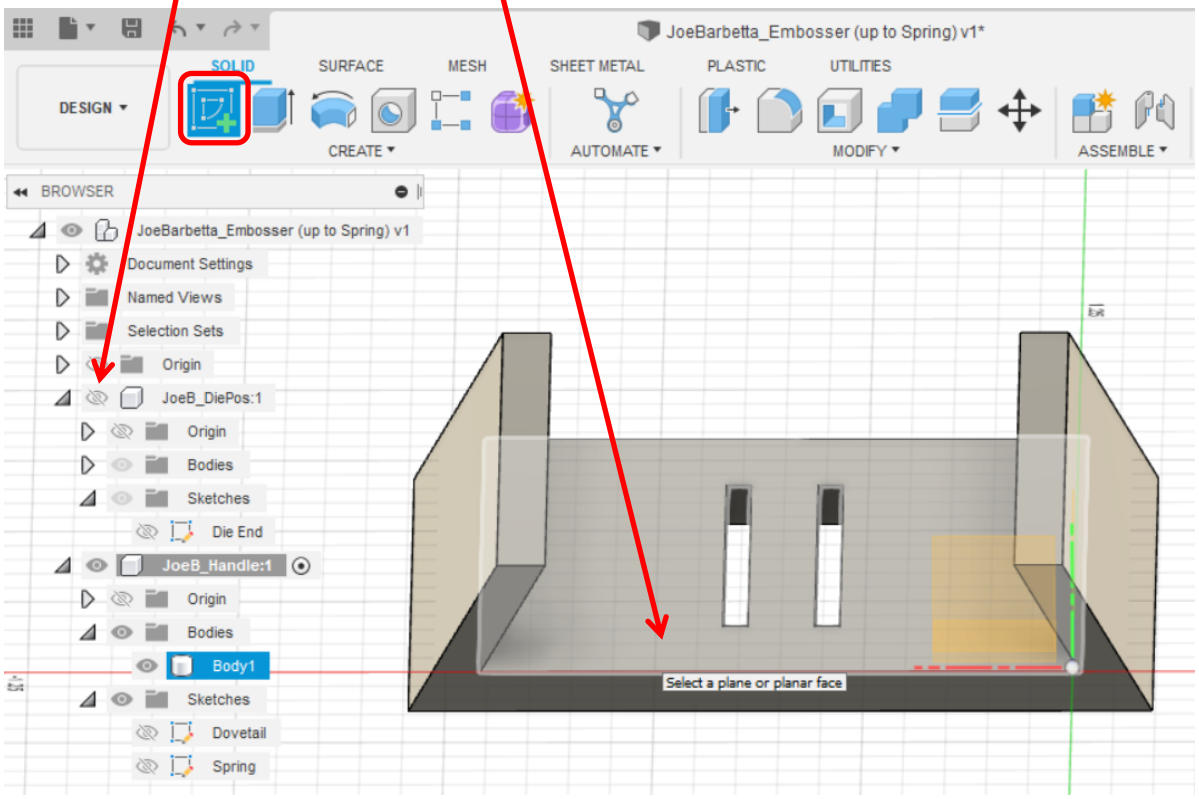
- change the view similar to below
- select the **Extrude** tool and click on each thin rectangular sections to turn them blue
- enter **-0.05** (note the minus sign) and click **OK**.



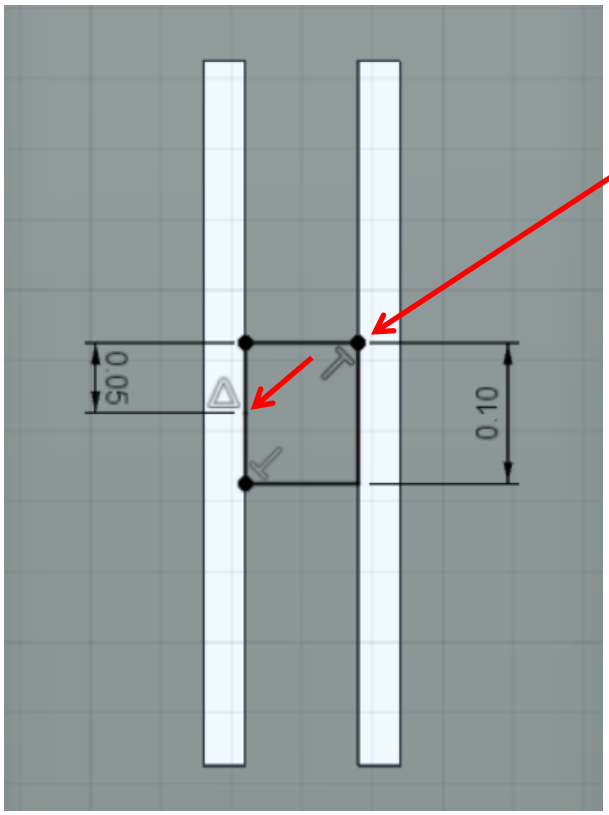
As shown below, slots should appear.



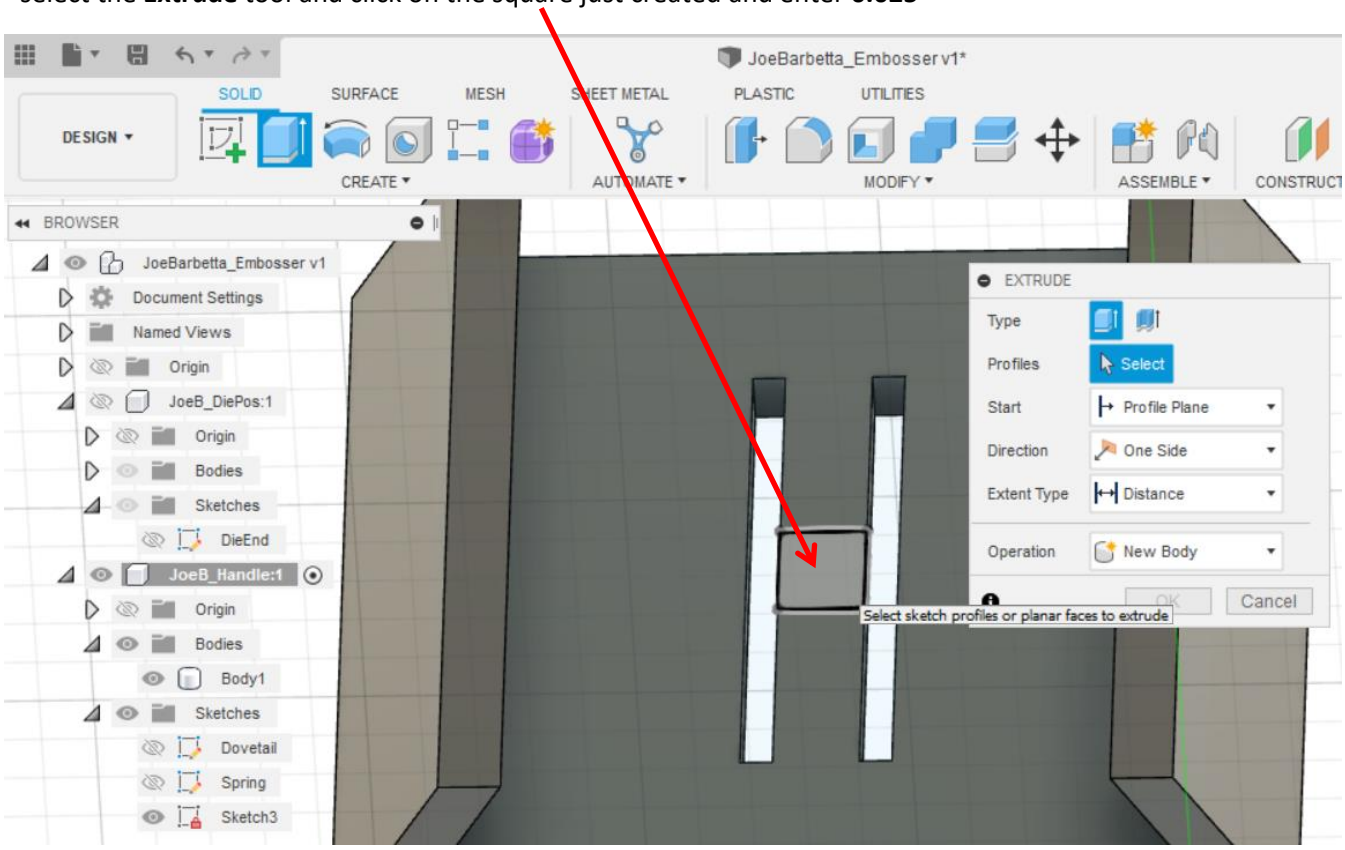
- change the view similar to that below
- click on the **eye icon** next to the **DiePos** *Component* to hide it
- select **Create Sketch** and click on the **Face** as indicated by the arrow



- select the **Line** tool and hover over the **center of the left slot edge**, as shown by the red arrow to snap to it
- create a line upward by **0.05**, then over to the right where the line will **snap to the edge of the right rectangle**, then down by **0.1**, **back to the edge of left rectangle** and **up to the starting point**
- click **Finish Sketch**

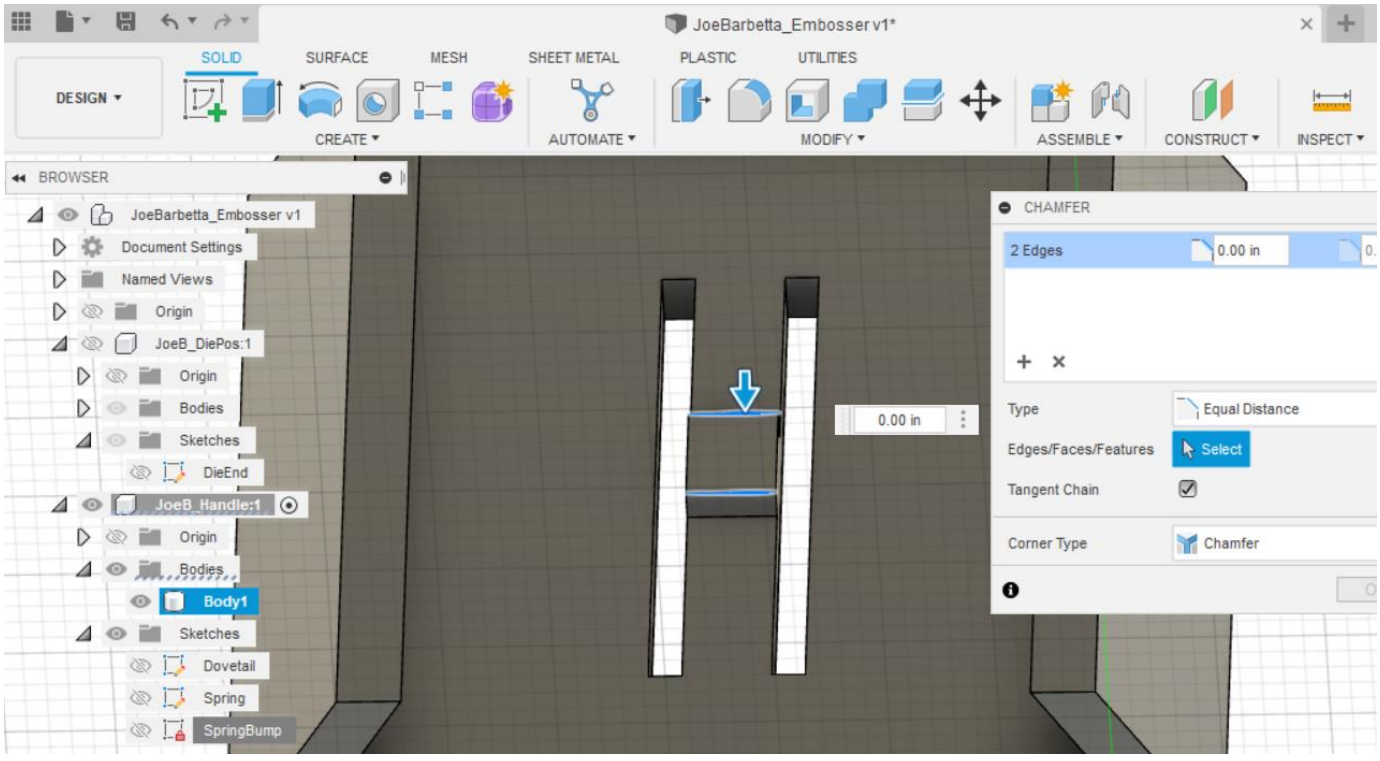


- adjust the view angle similar to that below
- select the **Extrude** tool and click on the square just created and enter **0.025**



- in the top **MODIFY** pull-down menu select **Chamfer**
- **click on the edges** as shown below to highlight them
- enter **0.025** and click **OK**

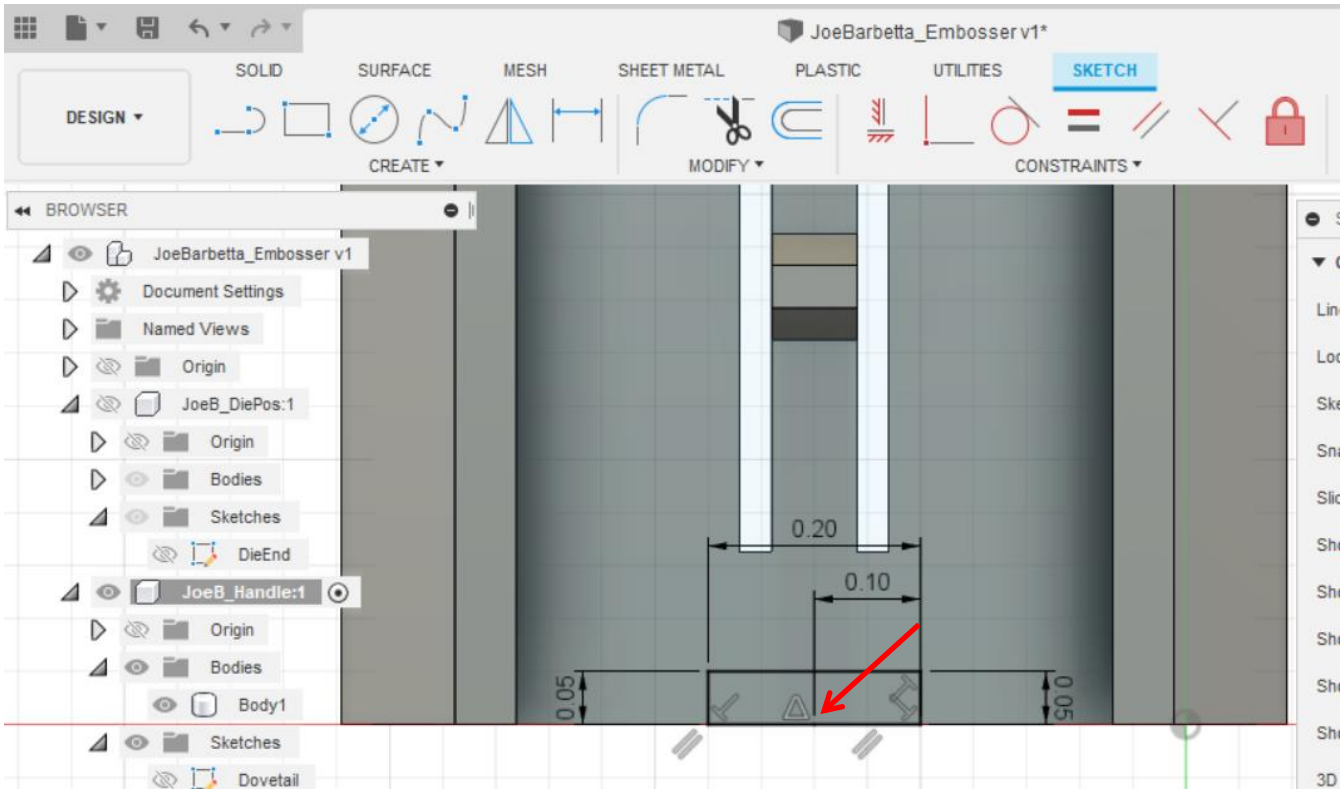
There should now be angles on the bump.



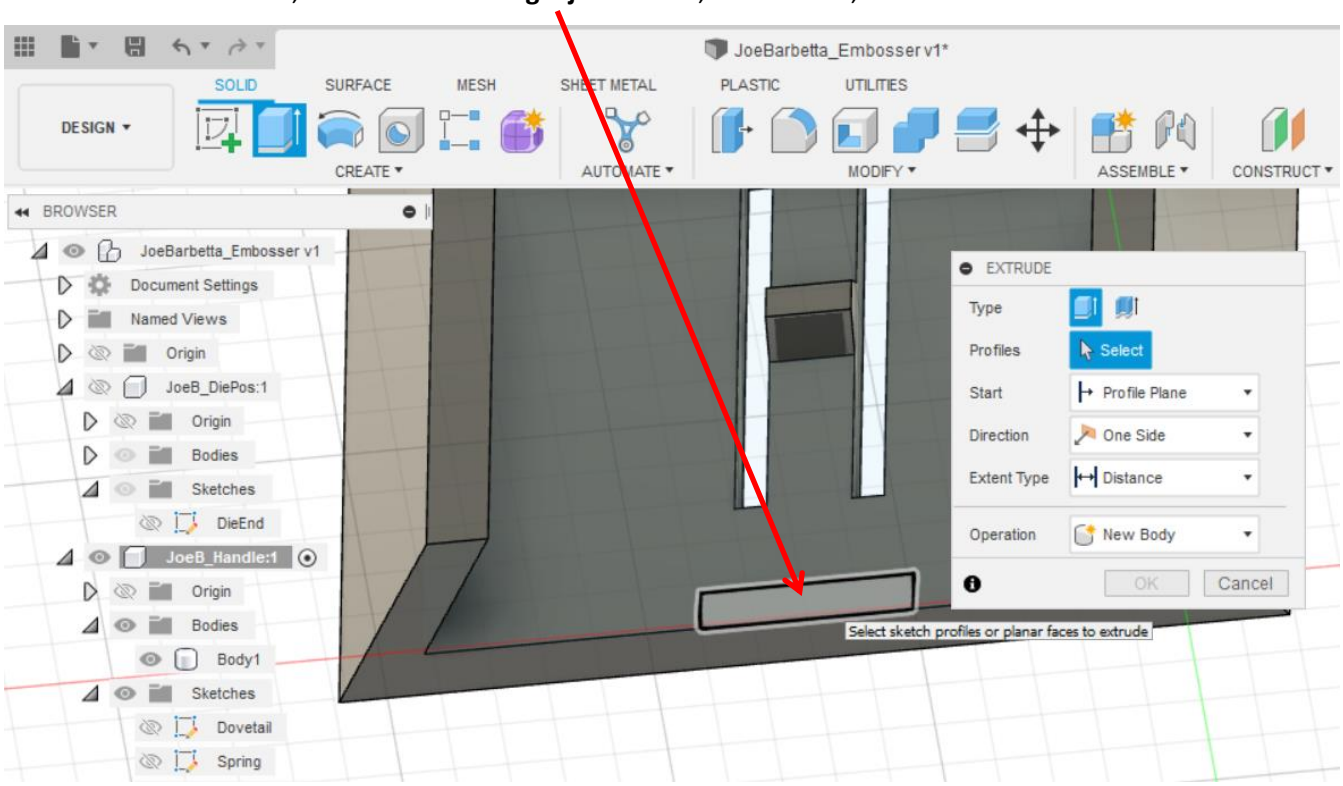
When the Die is inserted into the dovetail channel a “stop” will help maintain its position.

- start a **Sketch** on the same surface again and use the **Line** tool to create the rectangle with dimensions shown below. Use the **snap to center** feature to start the 1st line segment at the red arrow location and over towards the right by **0.10**, followed by a line upward by **0.05**, then over to the left by **0.2**, then down by **0.05** and then to the start point to close the rectangle.

- click **Finish Sketch**



- select the **Extrude** tool, click on the **rectangle just drawn**, enter **0.085**, and click **OK**

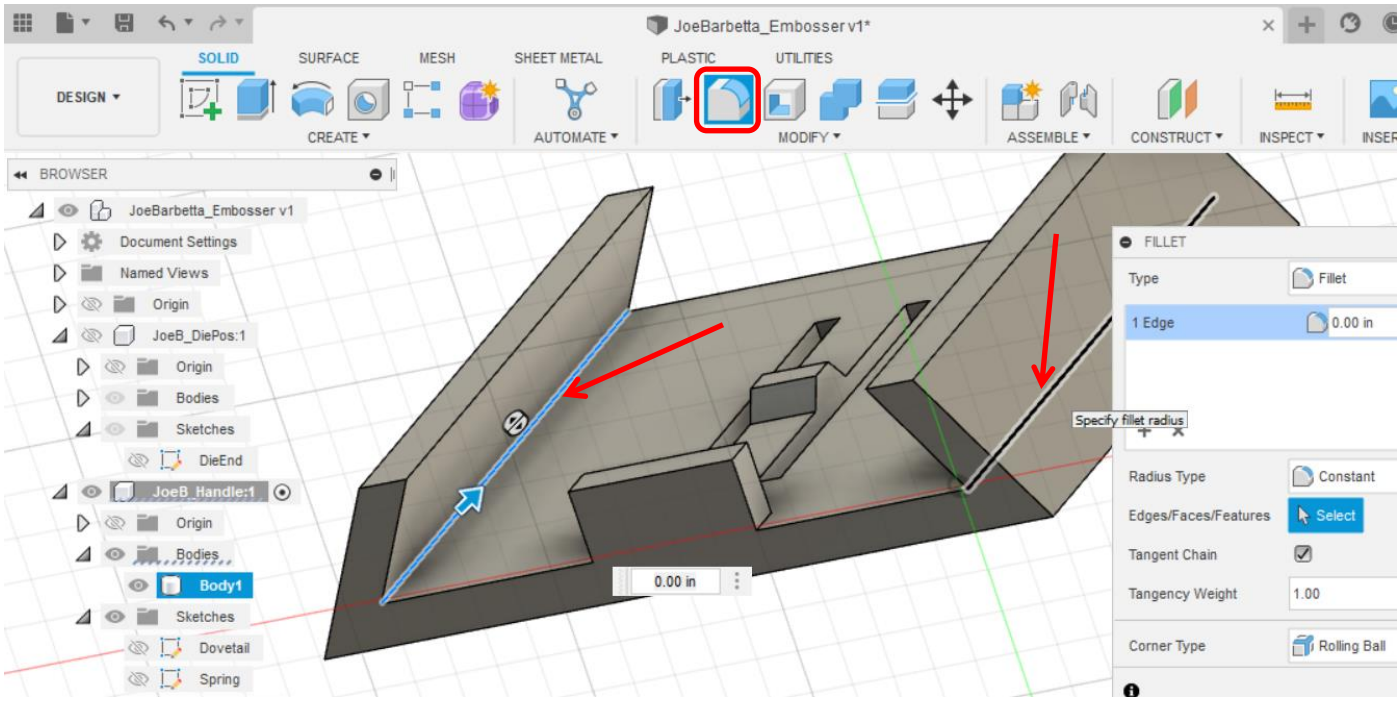


Eliminating Sharp Internal Angles

Sharp internal angles in any material result in weak points where fractures can start due to stress concentrations.

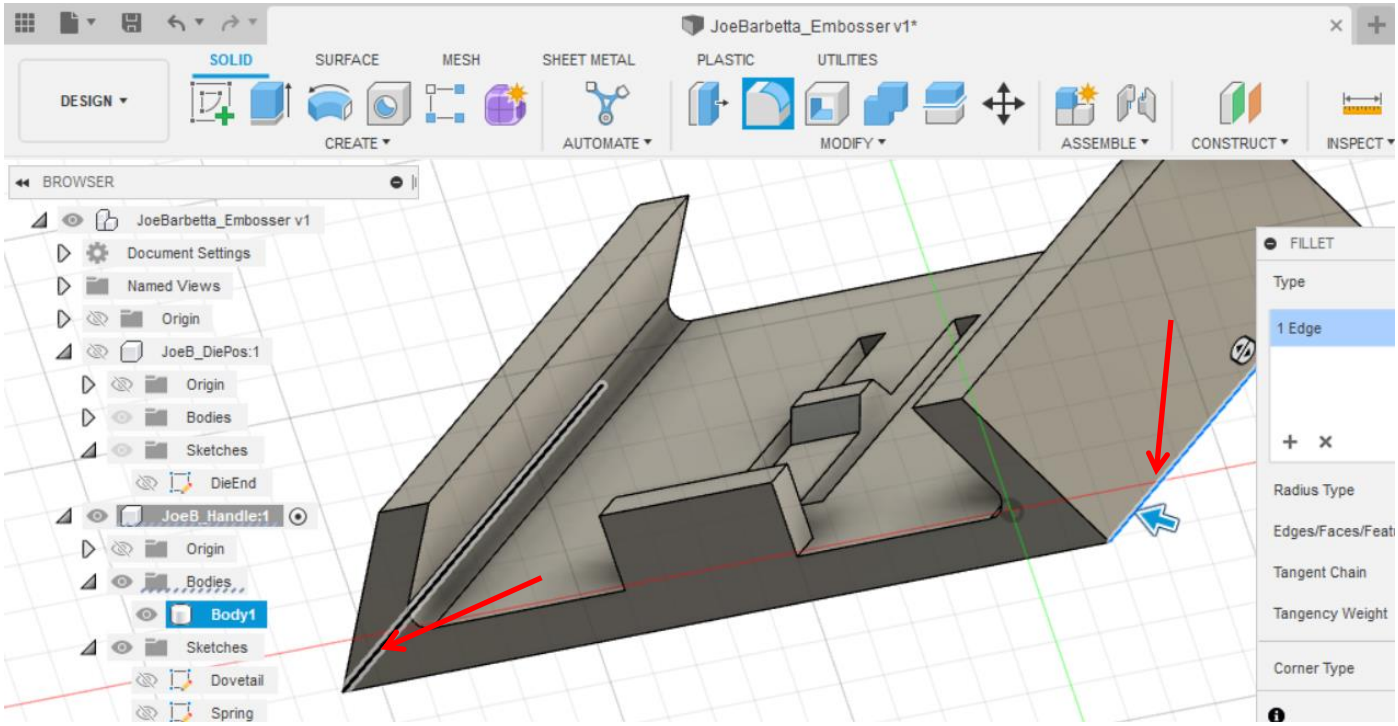
- select the **Fillet** tool (pronounced with the **t** and not like fillet of fish) and click on the two edges as shown. Note that for the edge on the right, the software allows clicking on an edge "through" the object.

- enter **0.02** for the fillet radius and click **OK**. We don't want to use a larger fillet radius to prevent interference with the Die.



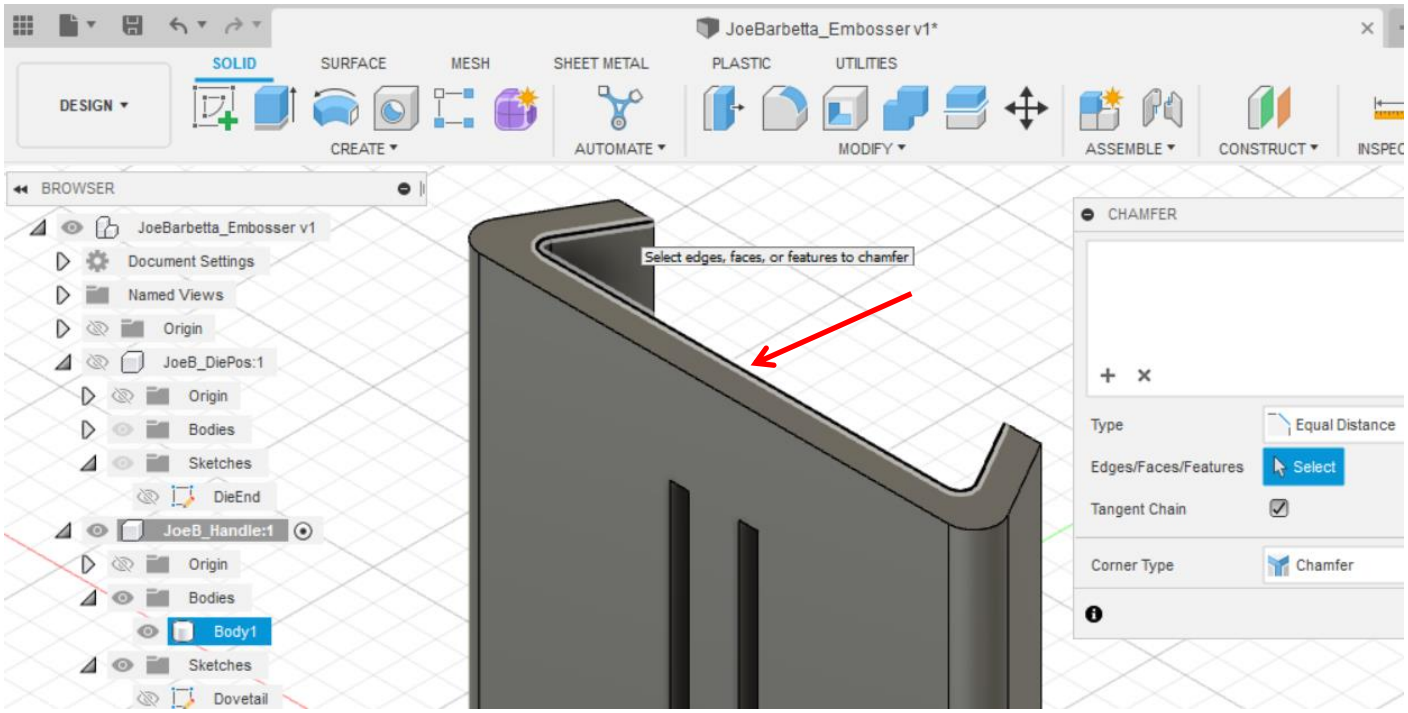
- select the **Fillet** tool again to create a **0.04** radius fillet on the external sharp angles

This is more for aesthetics.

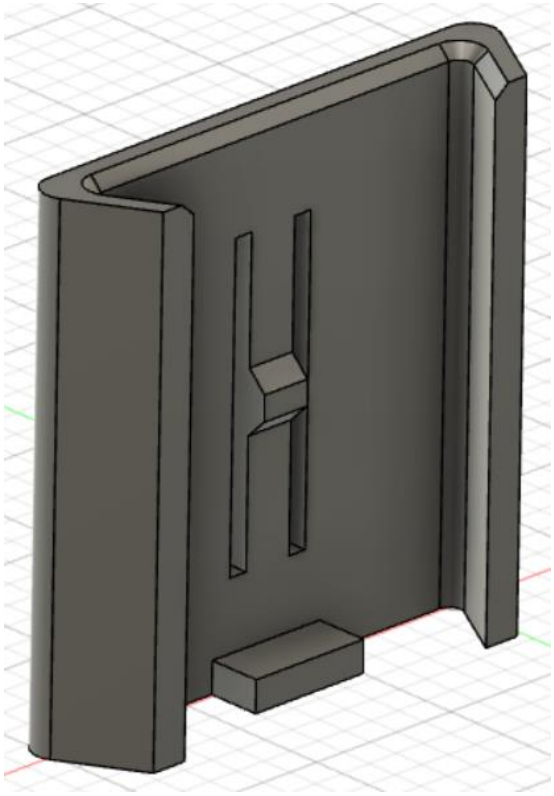


- change the view to show similarly as below
- select the **Chamfer** tool in the **MODIFY** pull-down menu
- **click on the internal edge** as shown below. Note how the curved and angled line segments become highlighted as well.
- enter **0.02** and click **OK**

This chamfer will make it easier to insert the Die into the dovetail channel.

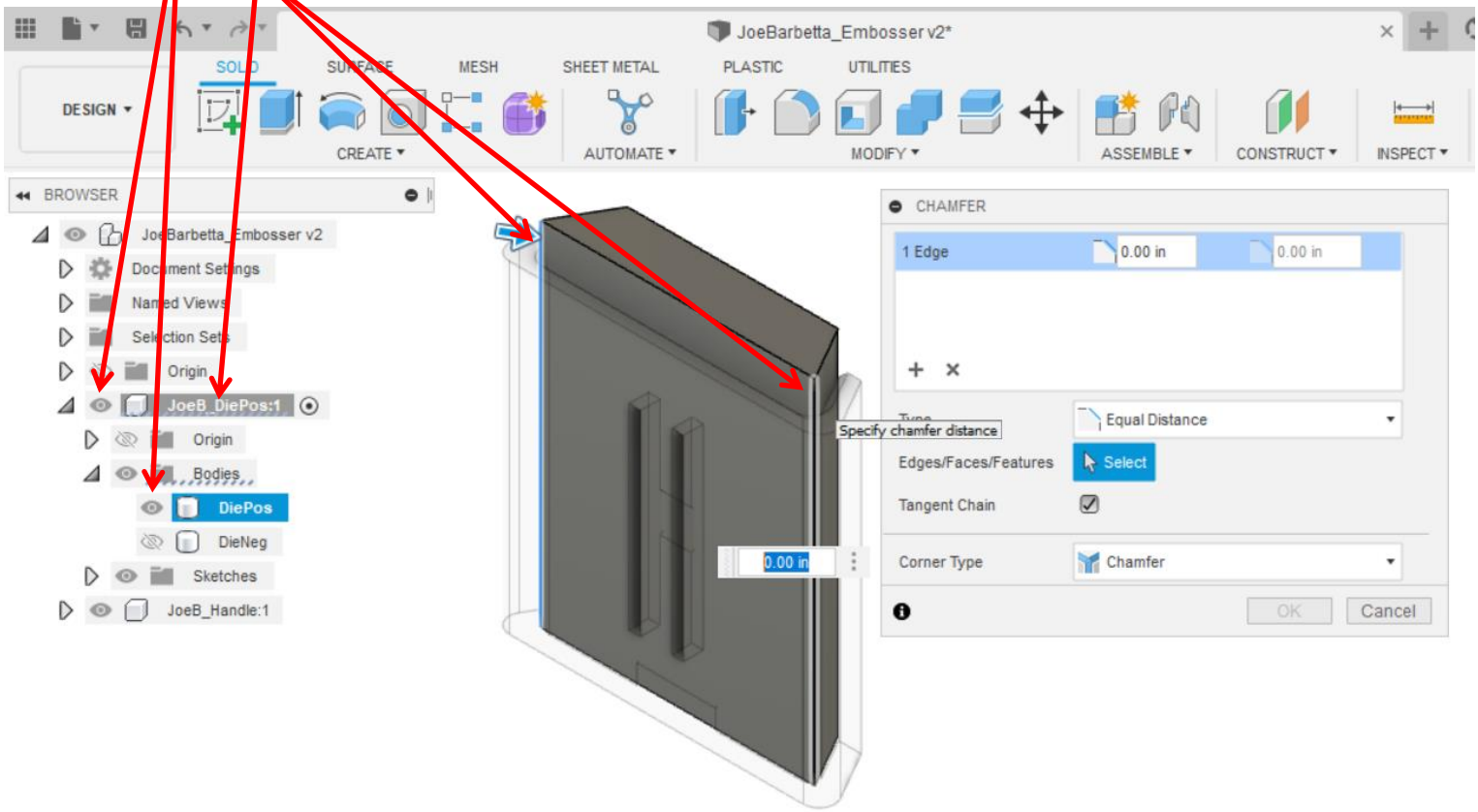


The dovetail channel should now look as that below.

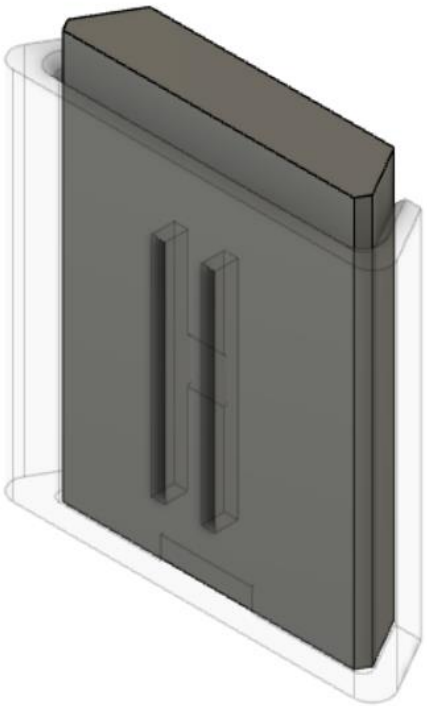


Now we can put these chamfers back on the Die.

- click on the **Home Icon** at the **View Cube**
- right-click on the **DiePos** Component and select **Activate**
- click on the **eye icons** next to the **DiePos Component and Body** to make them visible
- select the **Chamfer** tool in the top **MODIFY** pull-down menu
- click on the **two edges** as shown and enter **0.04** and click **OK**

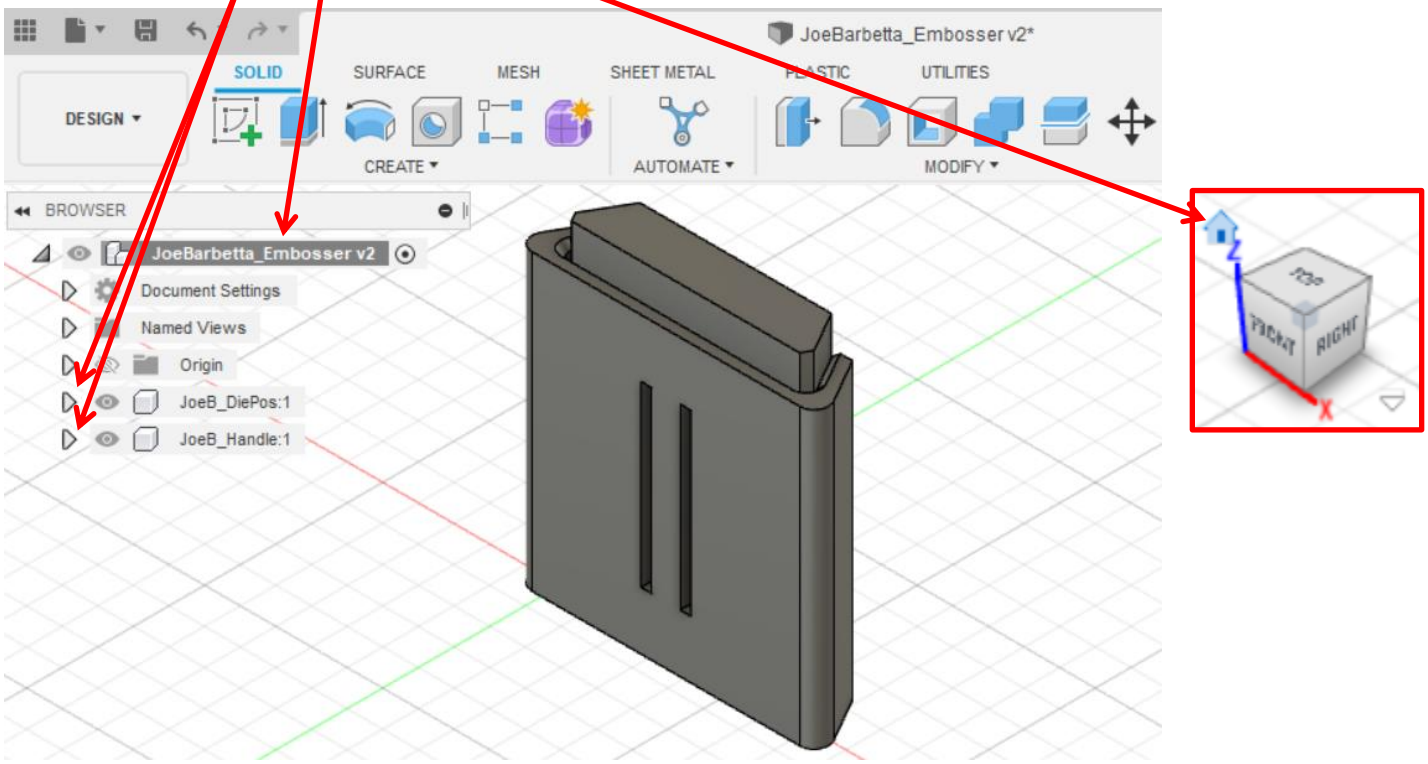


The Die should look like this.

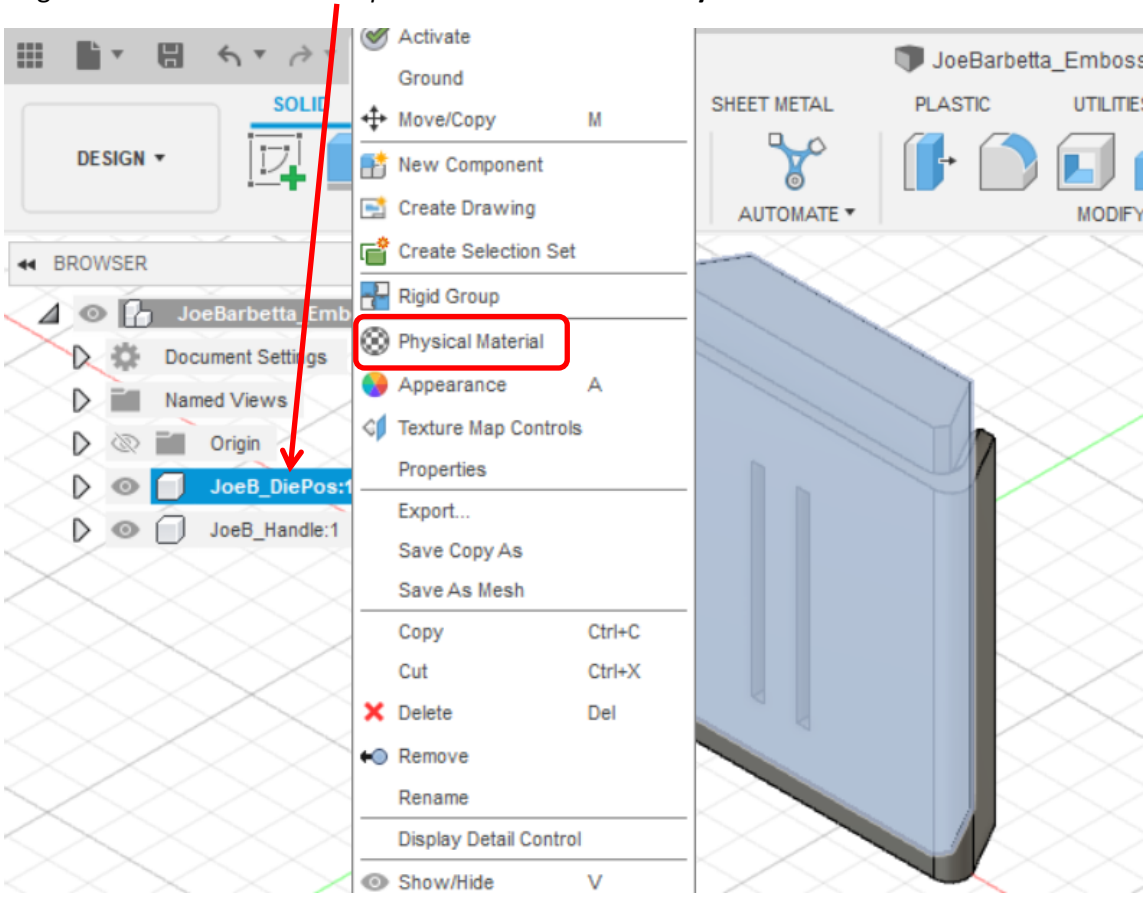


Setting Materials and Colors

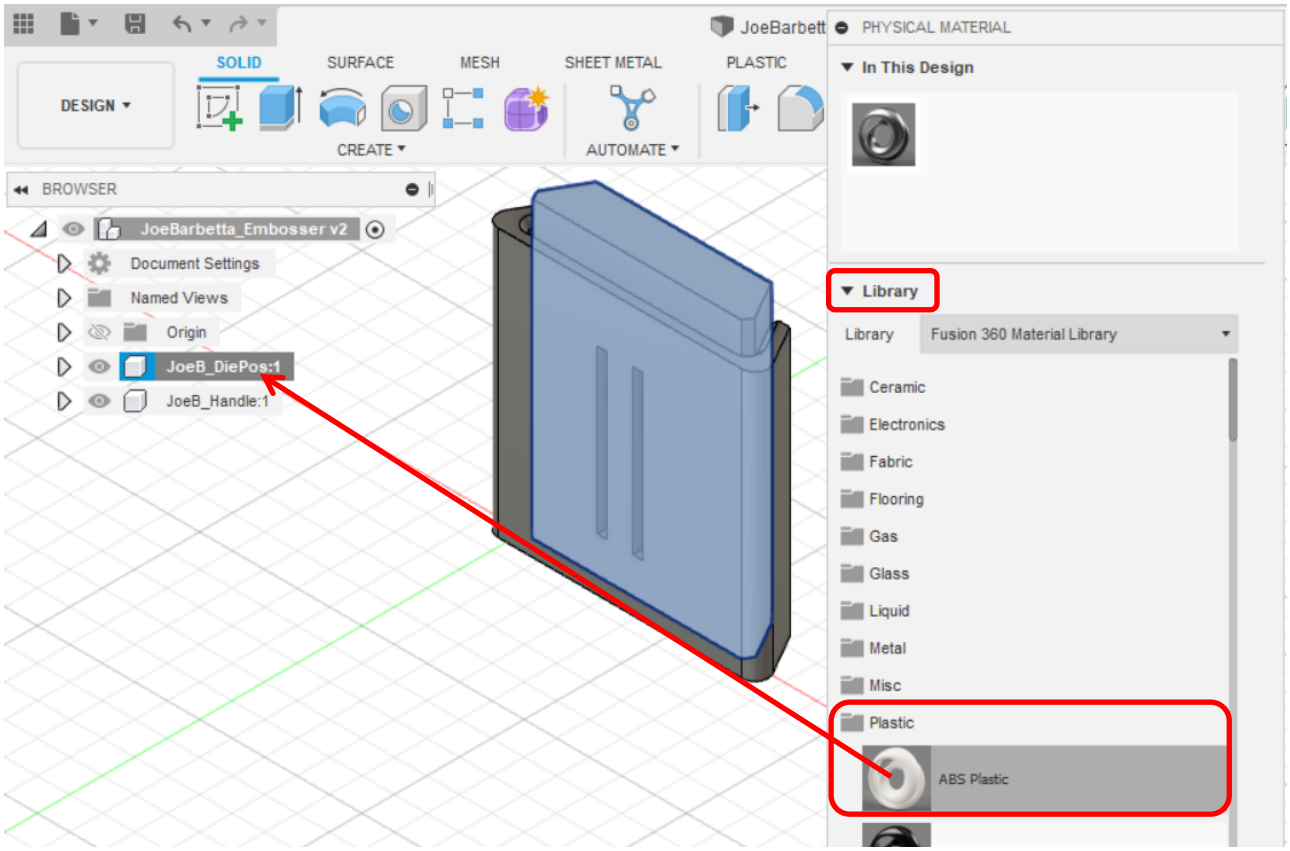
- right-click on the *Project name* in the BROWSER and select **Activate** to view all *Components* in the *Design*
- click on the **Home icon** at the **View Cube**
- click on the **Expand Arrows** to close the details of each Component



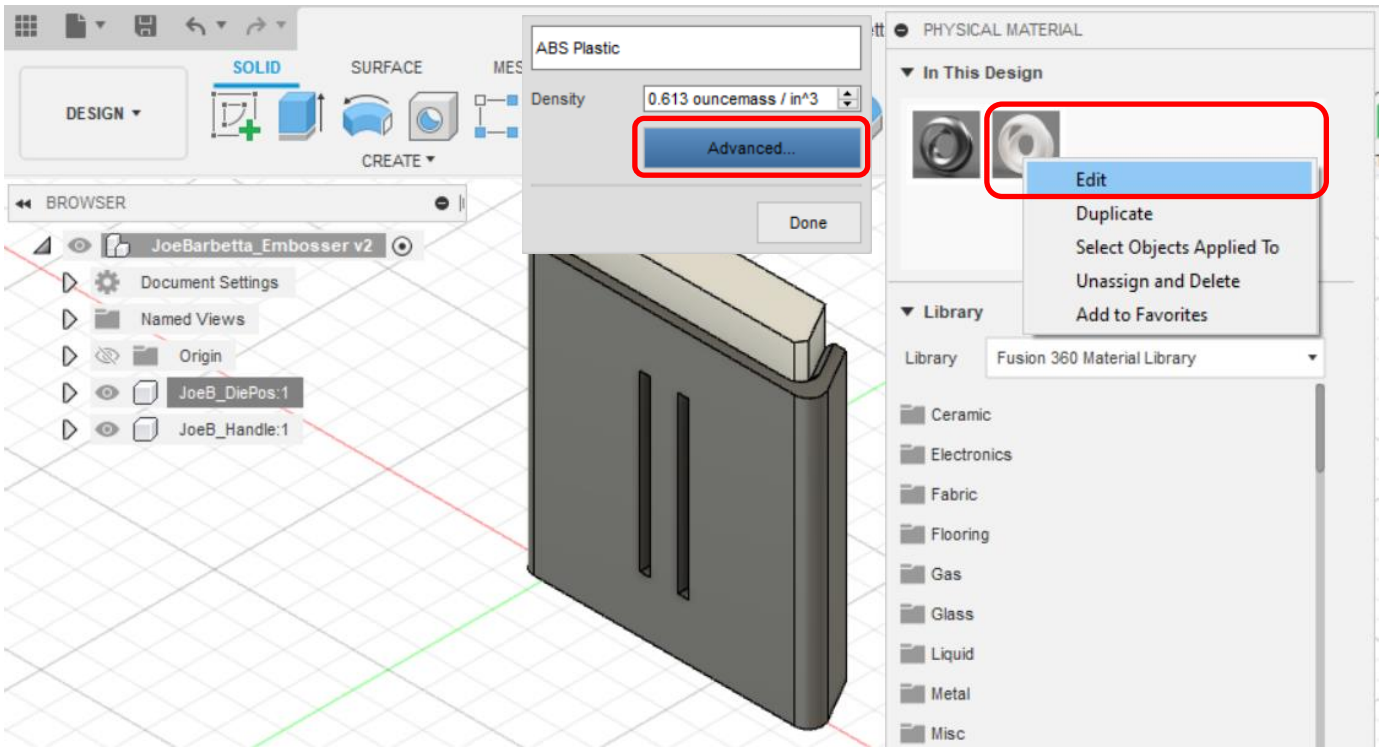
- right-click on the **DiePos** *Component* name and select **Physical Material**



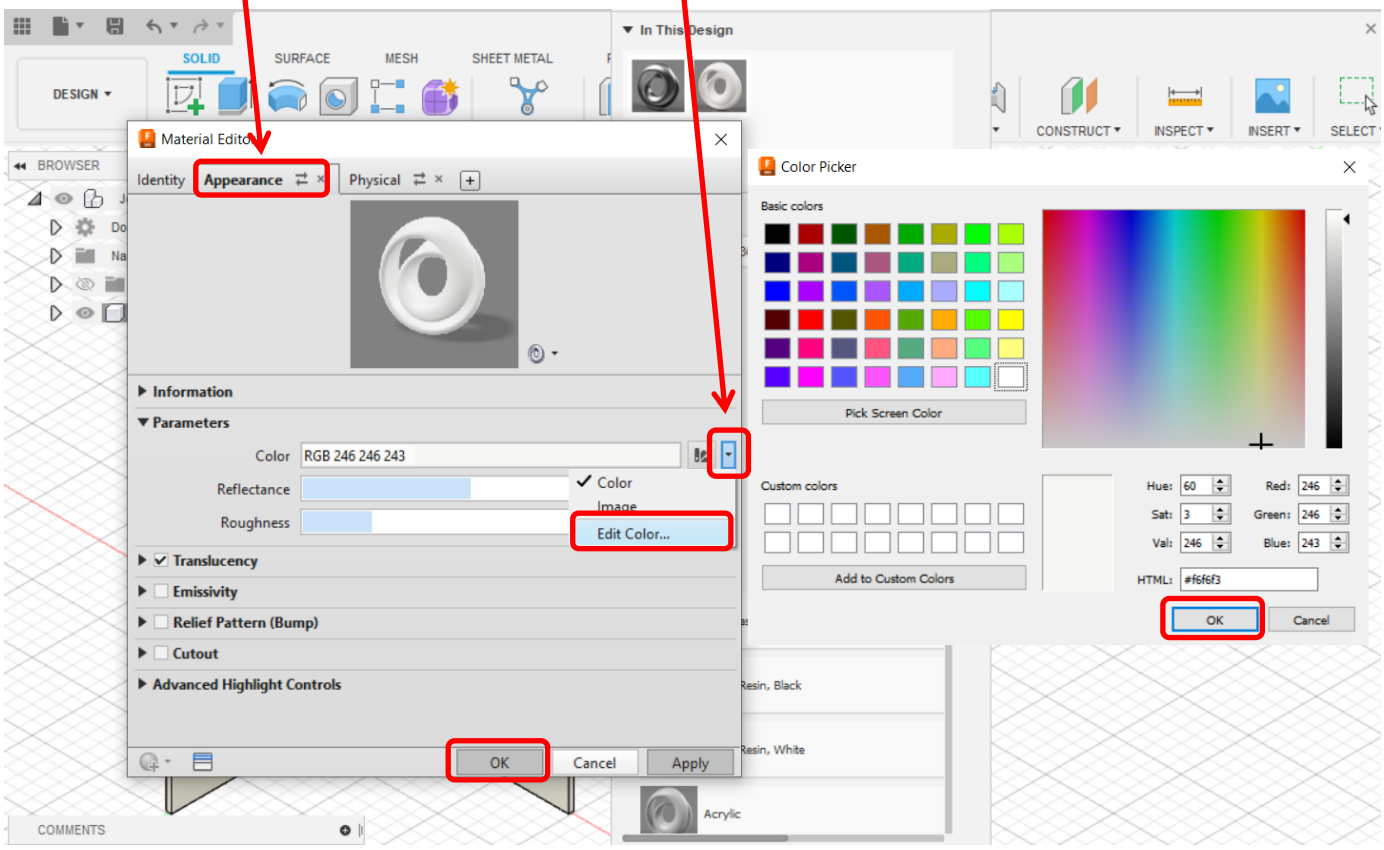
- in the **Library** section of the PHYSICAL MATERIAL window open the **Plastic** folder. You may have to scroll to find it.
- click on the **ABS Plastic** icon and **drag it** onto the **Die Component Name**. The component should turn white.



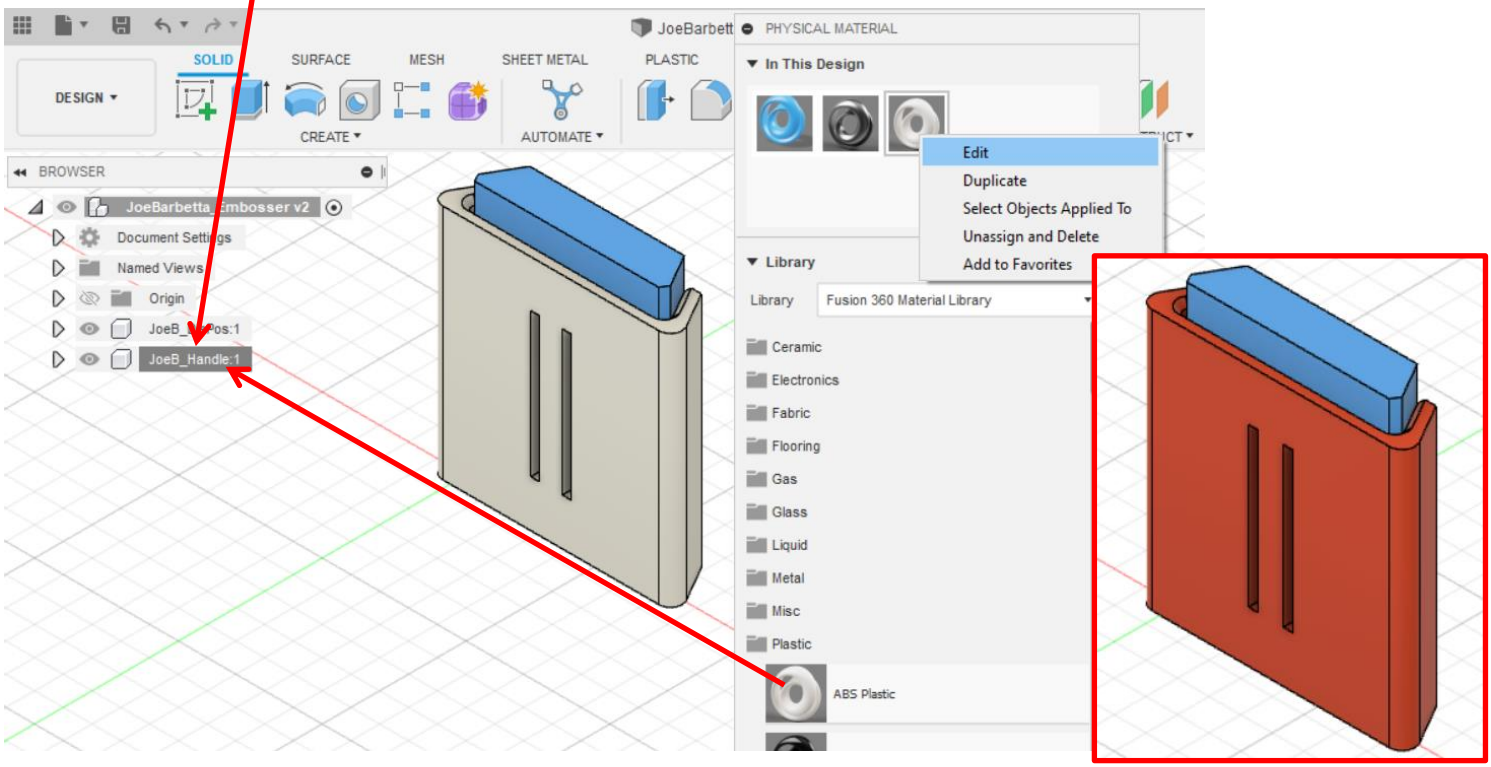
- right-click on the new white icon that appeared in the top "*In This Design*" section and select **Edit**
- click **Advanced..** button in the pop-up window



- select top **Appearance** tab
- in the **Parameters** section click on the **small down arrow** all the way to the right of *Color* and select **Edit Color...**
- select from the **Basic colors** on the left or **color selector** on the right and click **OK**. Select your favorite color.
- click **OK** on the *Material Editor* window and click **Close** on the *PHYSICAL MATERIAL* window



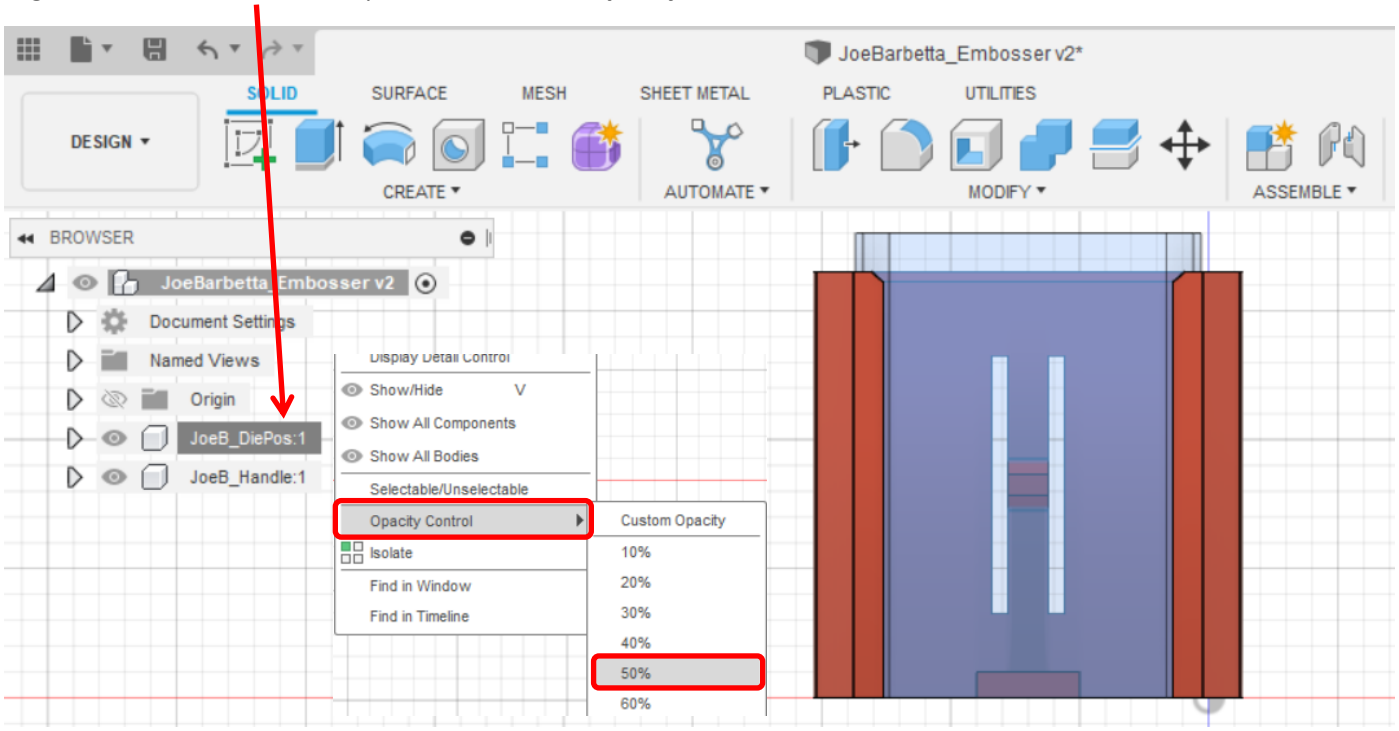
- right-click on the **Handle** *Component* and select **Physical Material**
- from the Library drag **ABS Plastic** onto the Handle Component Name
- as done previously, right-click on the new icon and select **Edit** to follow the steps to select a different color.



Using Opacity Control

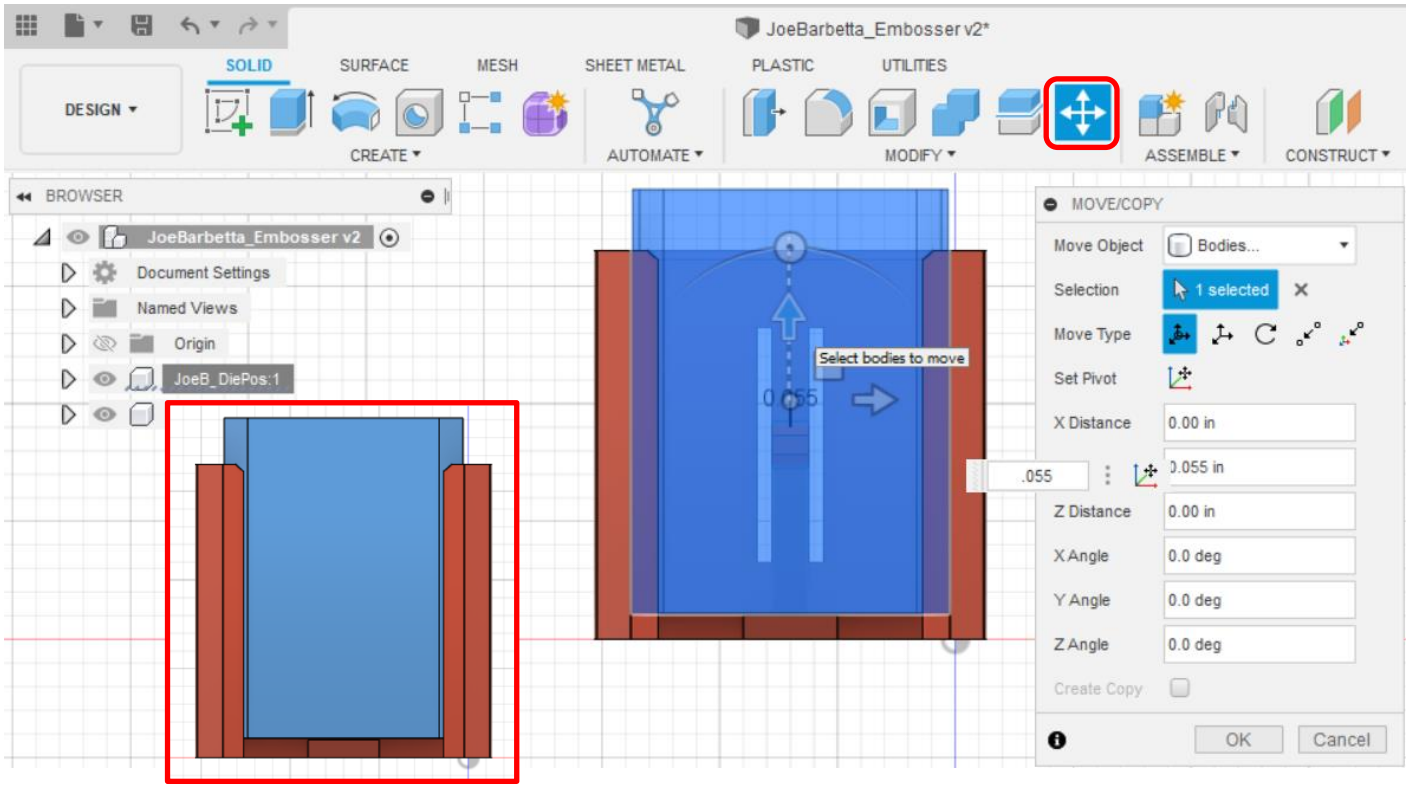
This feature of many CAD programs allows one to set component translucency to see through components.

- adjust the View to see into the Dovetail Channel
- right-click on the **DiePos** Component and select **Opacity Control** and **50%**



Moving a Component

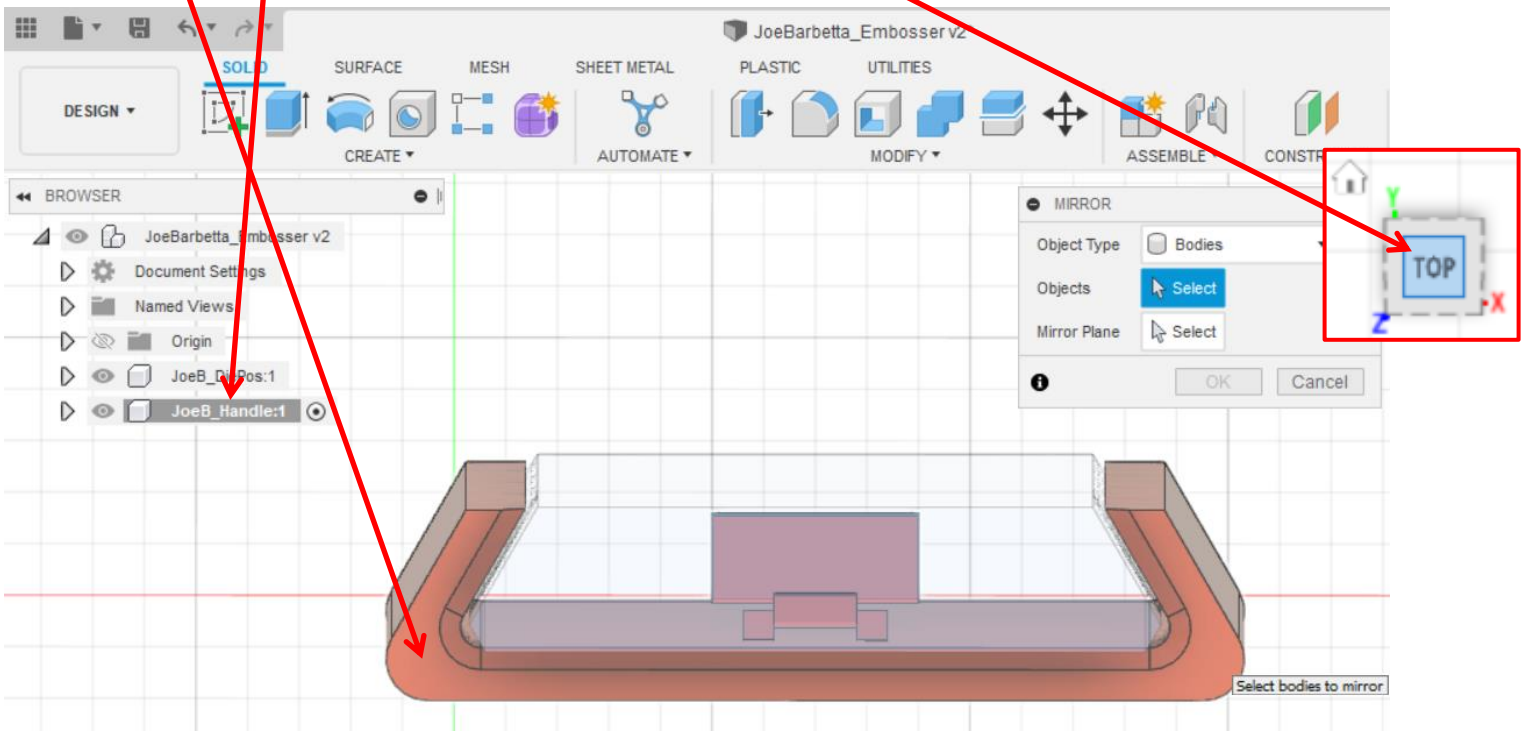
- select the **Move/Copy** tool and click on the DiePos Component **away from any text**
- click on the **upward arrow** and enter **0.055** and click **OK**.
- right-click on the DiePos Component and select **Opacity Control** to set the opacity back to **100%**



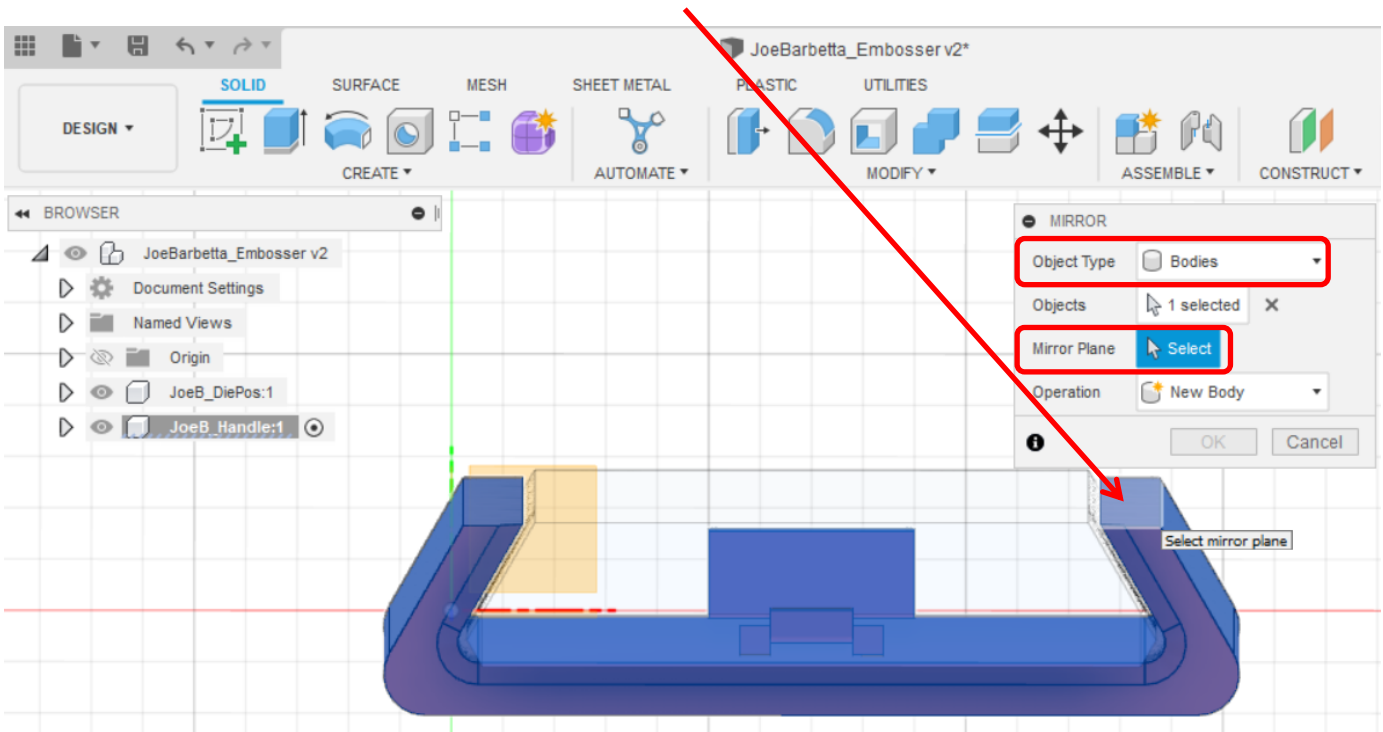
Mirroring the Dovetail Channel

Designs will often have features that are mirror images of each other. It is a great convenience that most CAD programs offer to create a duplicate body that mirrors the original body.

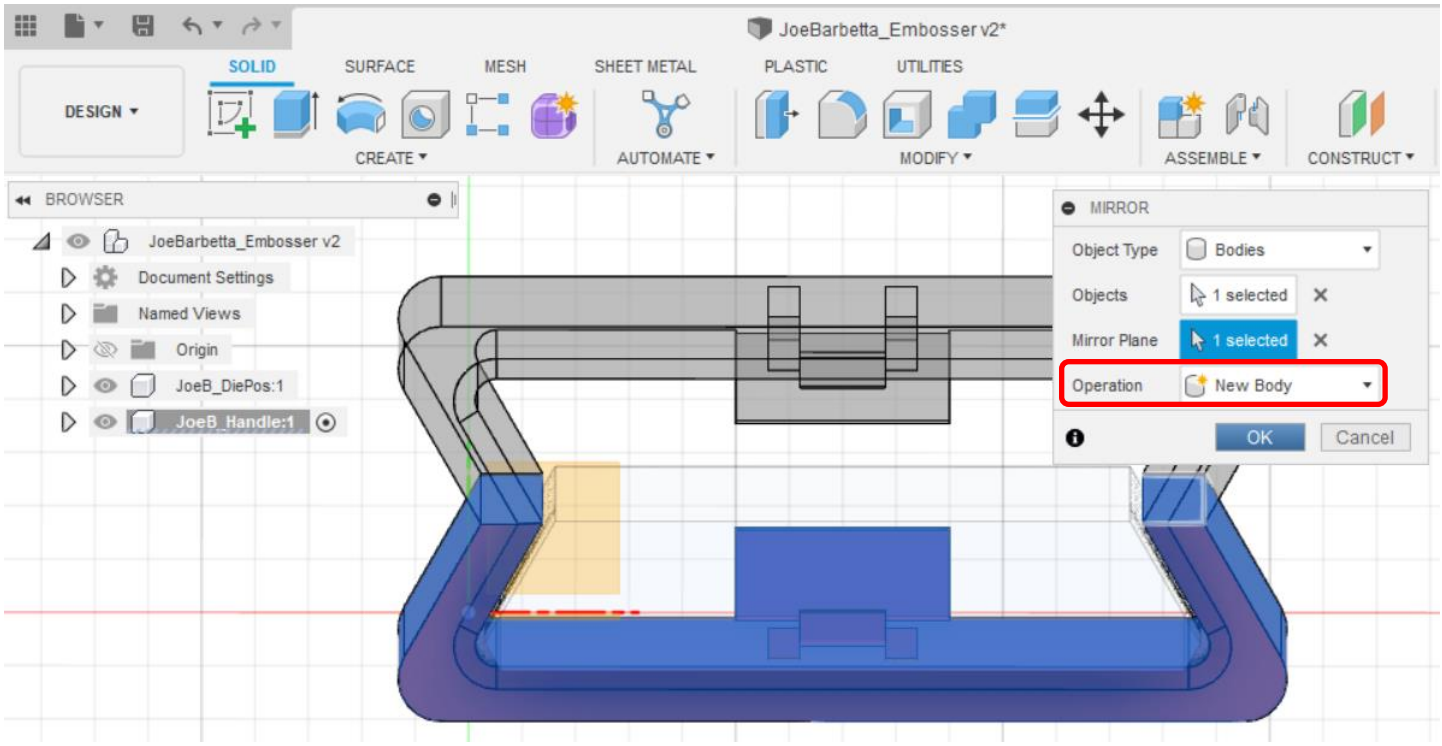
- right-click on the **Handle Component Name** and select **Activate** to ensure it is the *Active Component*
- **adjust the View** as shown below. This is essentially a **Top View** with a slight angle.
- from the top **CREATE** pull-down menu select **Mirror** near the end of the list. If there is a pop-up select Capture Position.
- click on the **top surface** of the dovetail channel



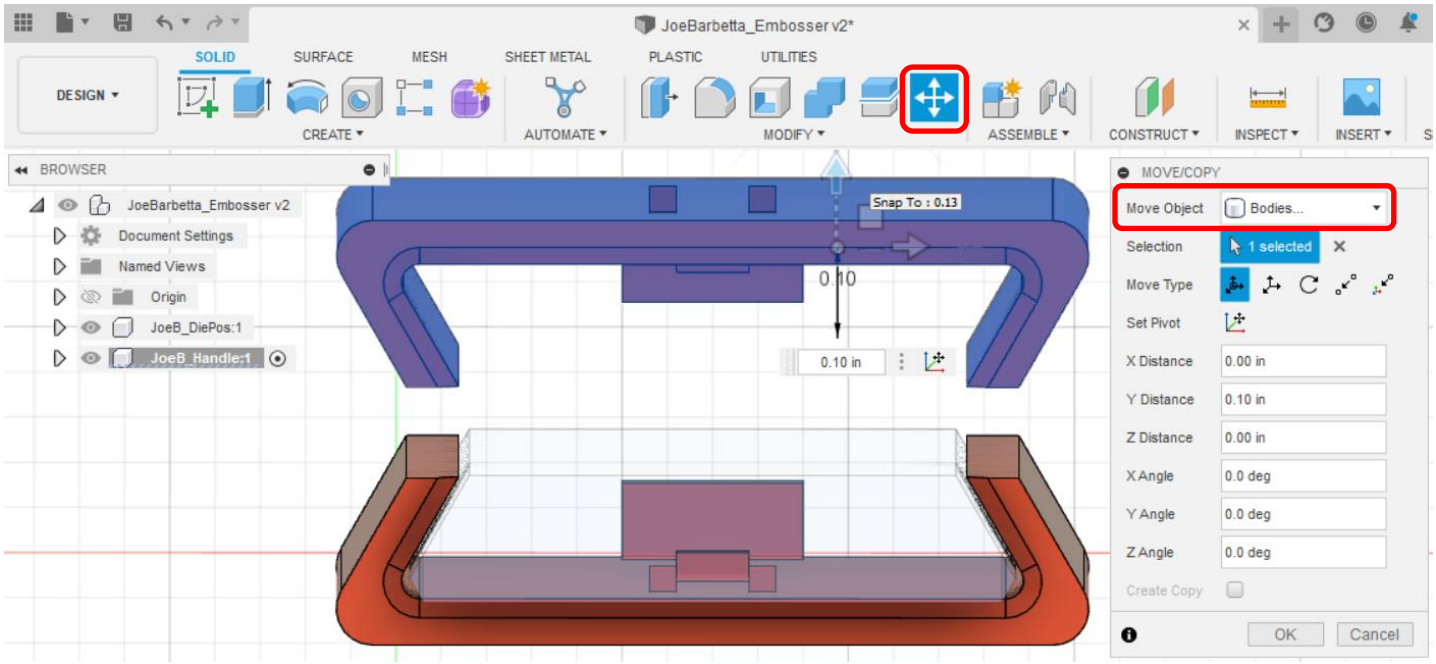
- ensure **Object Type** is set to **Bodies**
- click on **Select** next to **Mirror Plane** and click on the **surface of the dovetail extension**



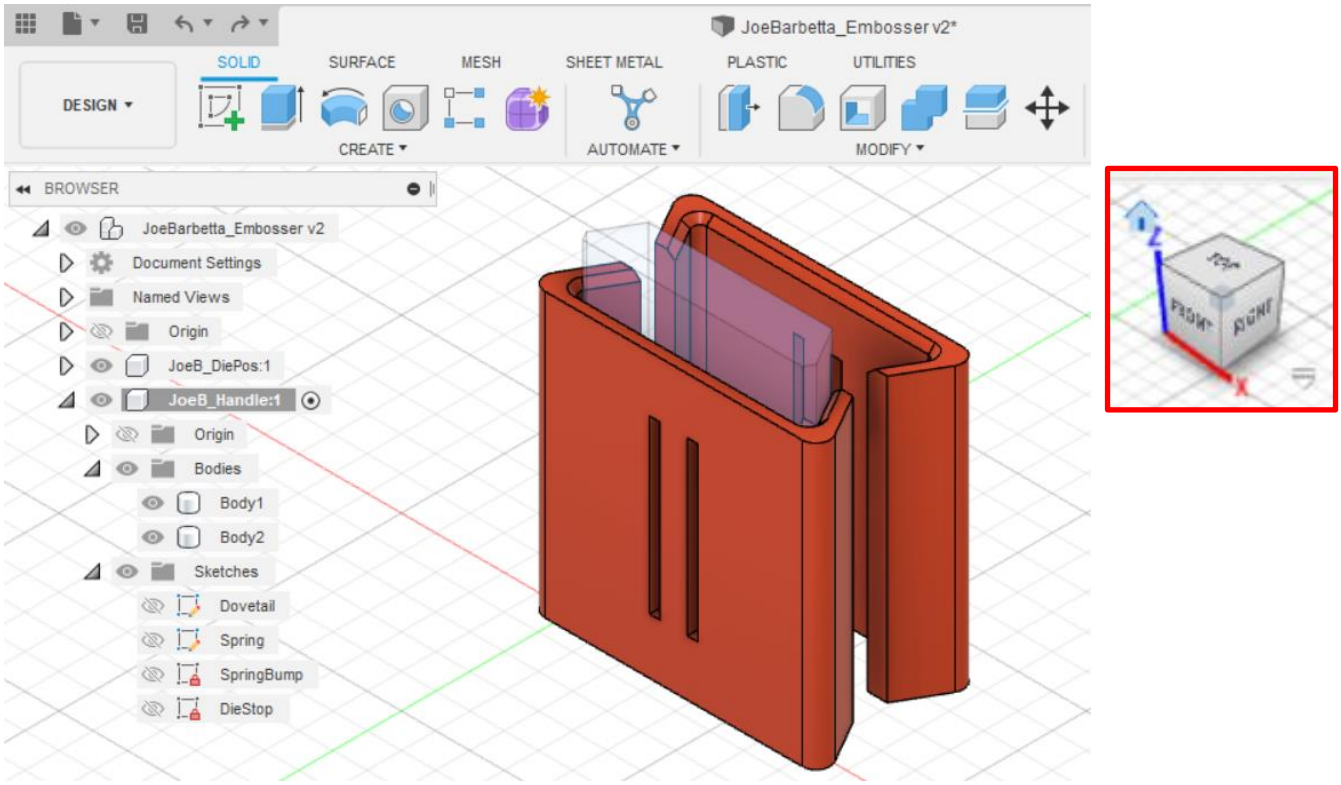
- before clicking **OK** ensure that **Operation** is set to **New Body**. It may have automatically switched to Join when the Mirror Plane was selected. Click **OK**



- select the **Move/Copy** tool, click on the **newly created dovetail channel**
- ensure that **Move Object** is set to **Bodies**
- click on the **upward move arrow**, enter **0.1**, and click **OK**



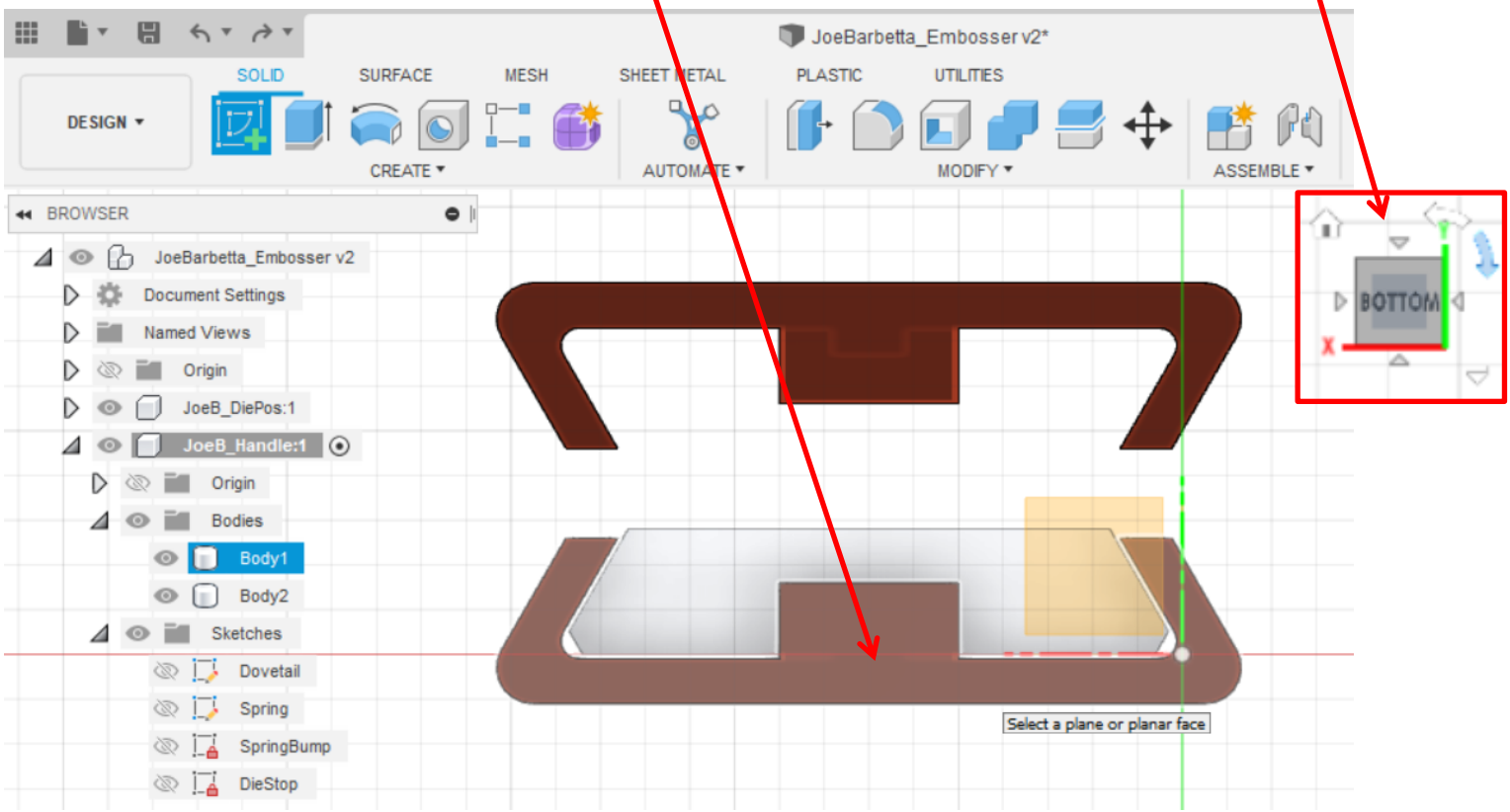
- click on the **Home** icon at the **View Cube** and your design should look similar to below



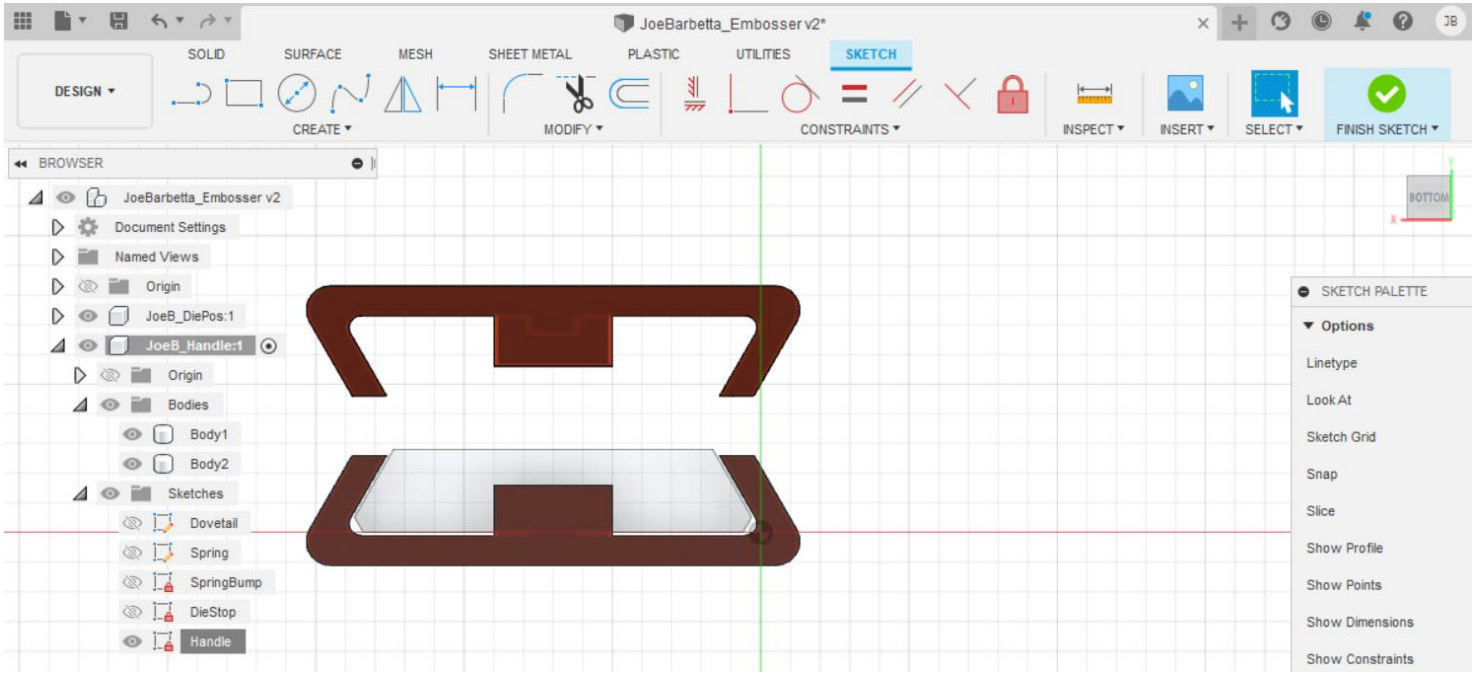
Now we need to create the section of the handle that connects the two dovetail channels.

- change the view so that we are looking at the **bottom of the channels**. The **View Cube should indicate BOTTOM** as shown. The curved arrows at the upper right of the View Cube can be used to rotate the view if needed.

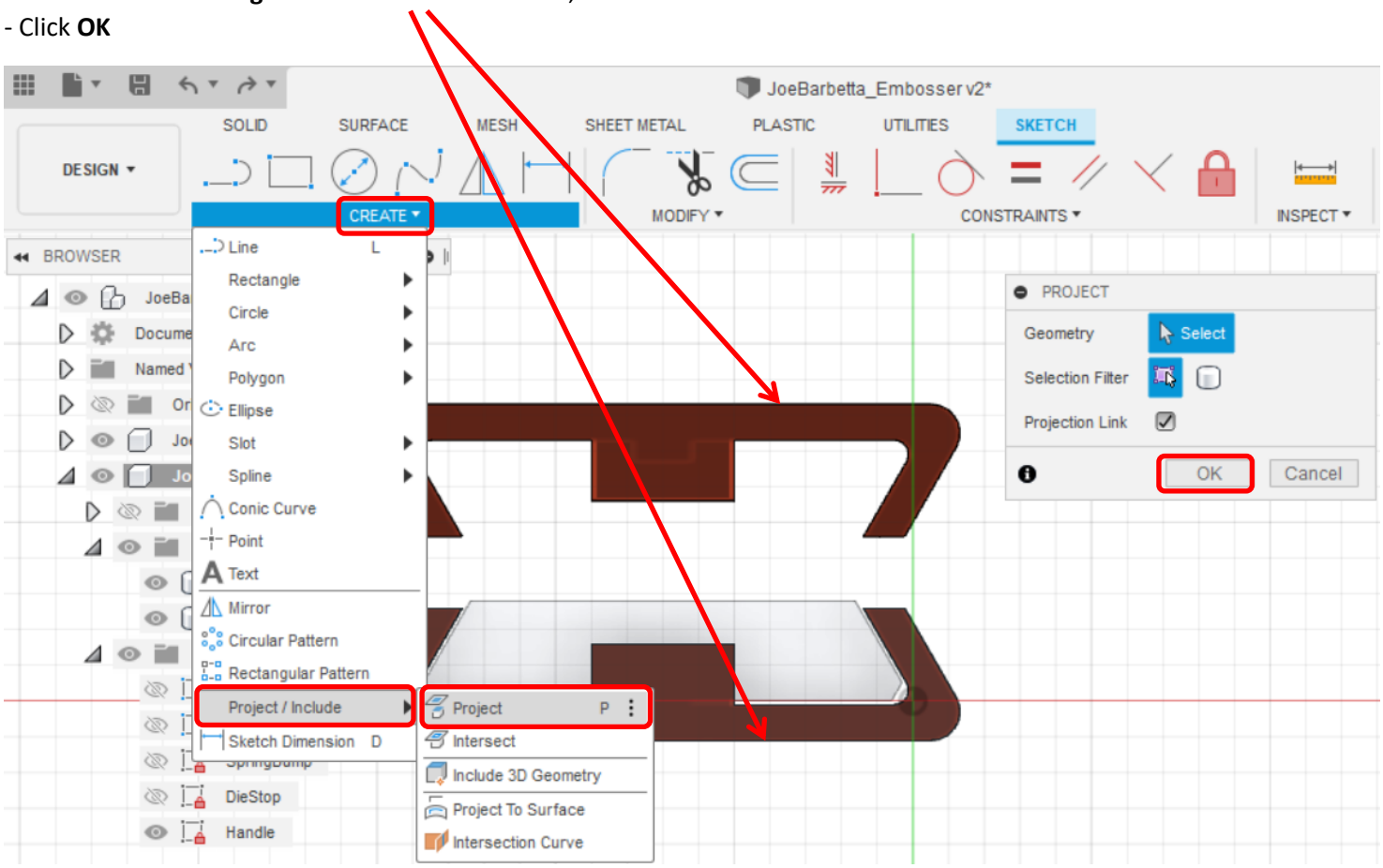
- select **Start Sketch** and click on the lower dovetail channel.



- position the dovetail channels to left room to the right for sketching.



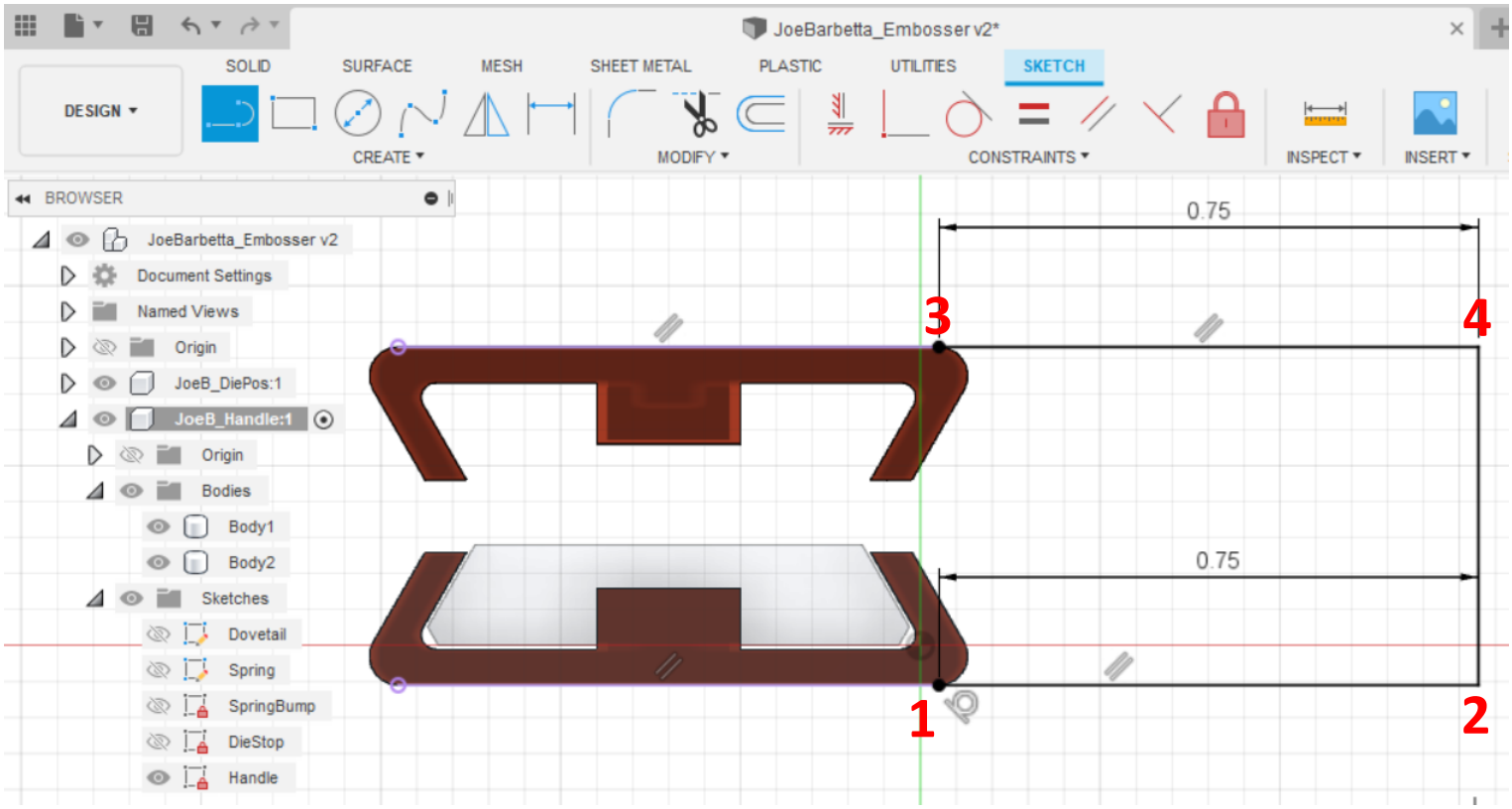
- near the bottom of the **CREATE** pull-down menu select **Project/Include** and **Project**
- click on the **outer edges of the dovetail channels**, which should cause them to turn blue.
- Click **OK**



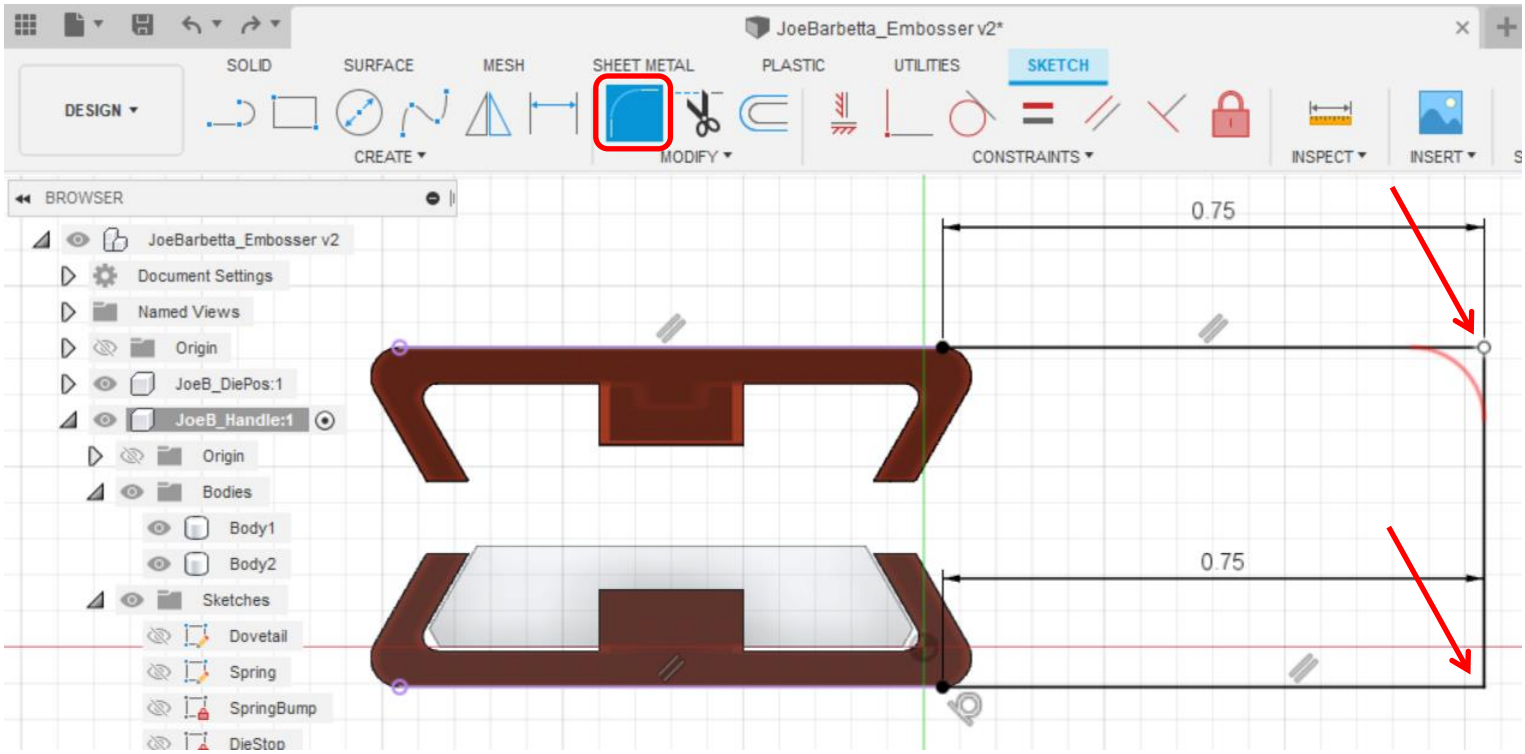
- select the **Line** tool

- create a line from point **1** (should snap at the violet circle) over to the right near **2**, enter **0.75**, click, and press the **Esc key**.

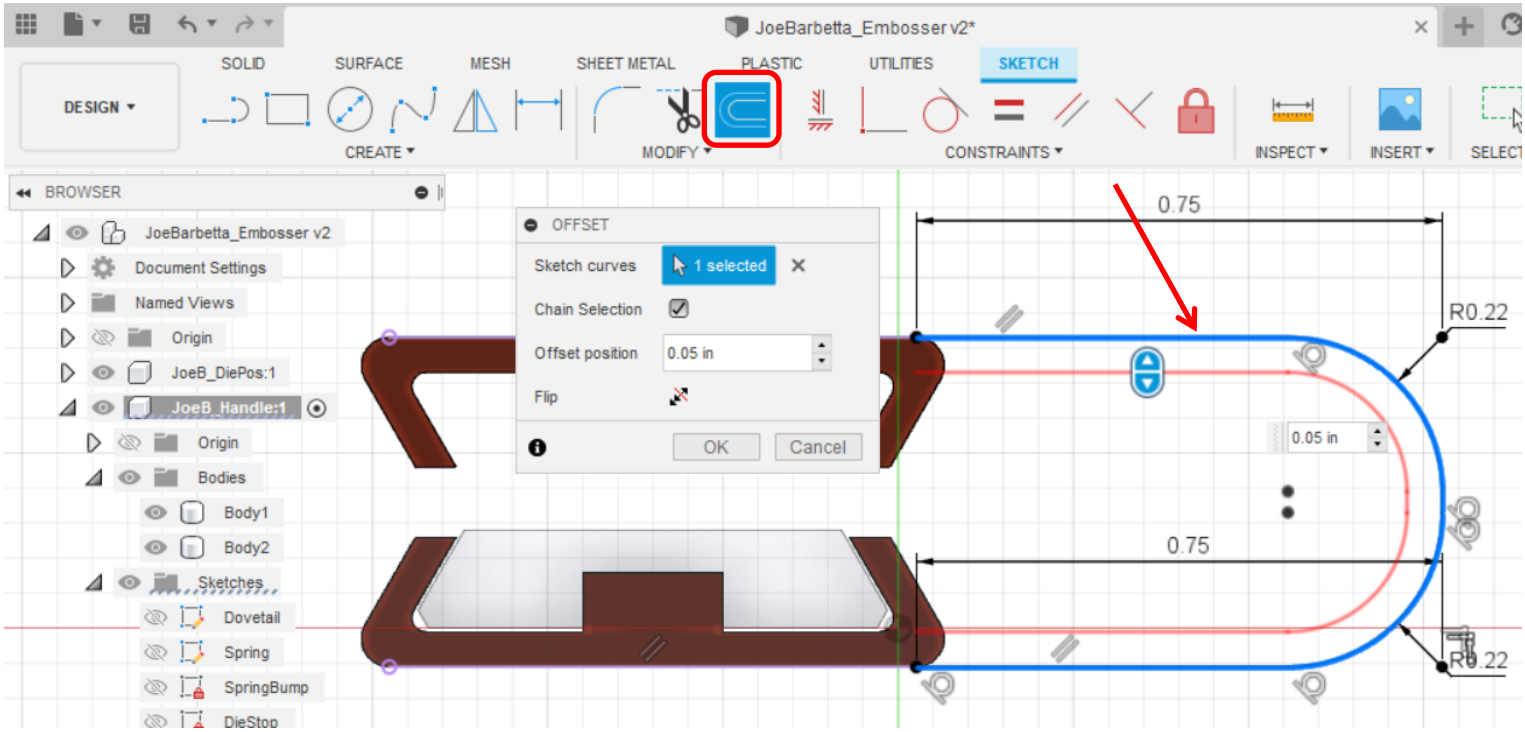
- create a line from point **3** (should snap at the violet circle) over to the right near **4** and enter **0.75**, click, and continue down to end at point **2** Don't worry if the thin black dimension lines look different.



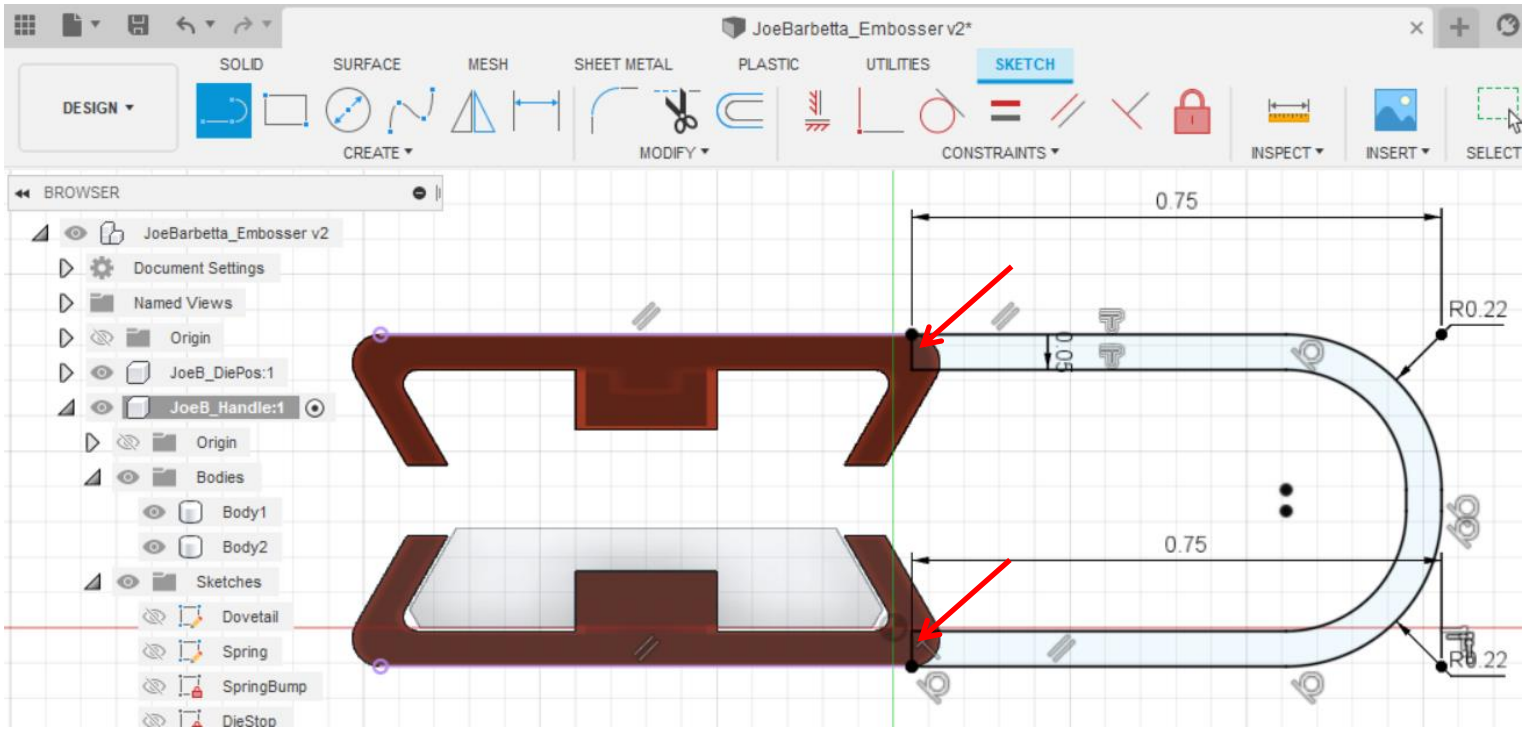
- select the **Fillet** tool and click on the corner point as shown and enter **0.22** and do the same for the lower point. You can ignore any errors.



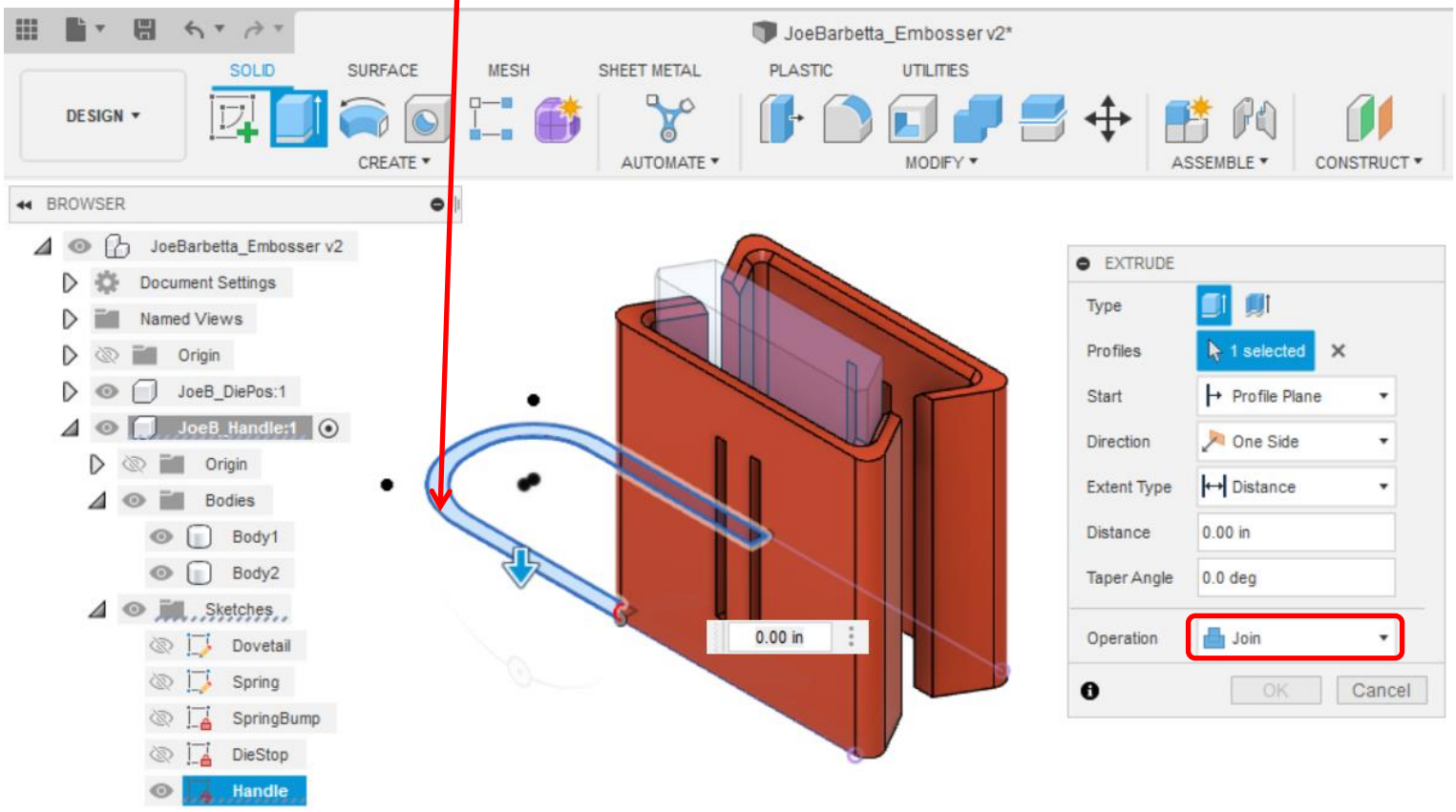
- select the **Offset** tool, **click on the handle outline**, enter **0.05**, and click **OK**



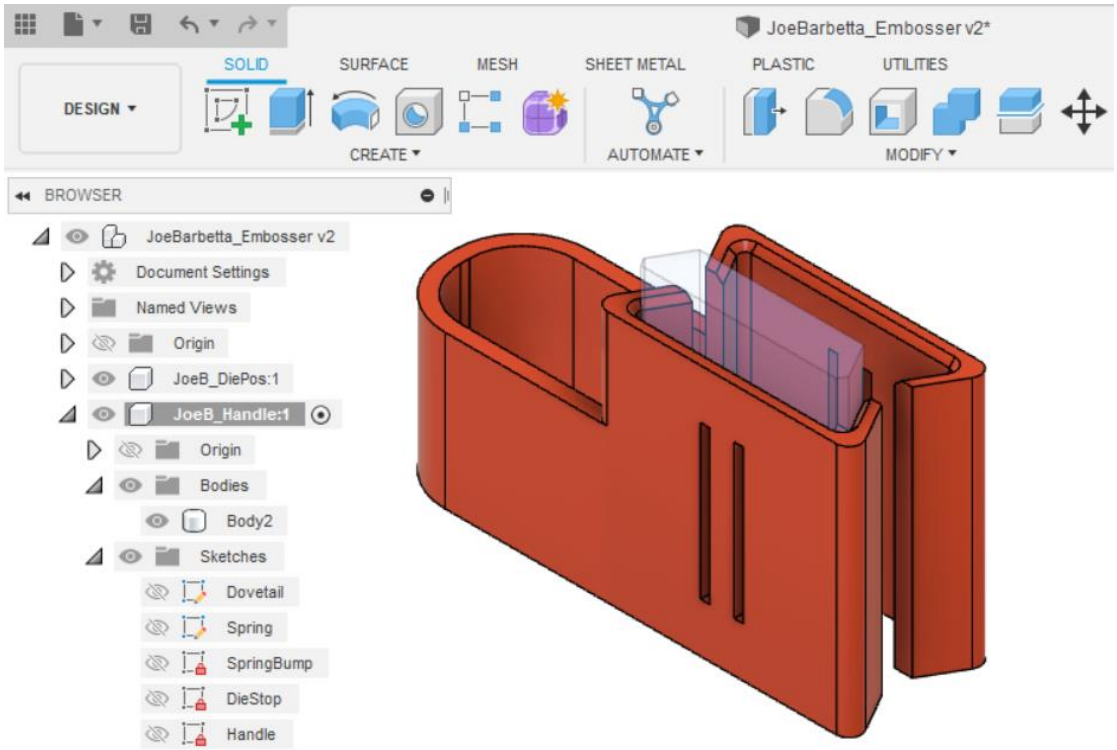
- select the **Line** tool and create **two small line segments** to close the handle outline and cause its interior to turn light blue.
- click **Finish Sketch**



- click on the **Home** icon at the **View Cube** to reset the view
- select **Extrude** and **click on the Handle profile**. Type **-0.5** (note the minus sign), ensure **Operation is set to Join**, and click **OK**. Because we created the Sketch when looking at the bottom of the dovetail channels, we need the negative value.



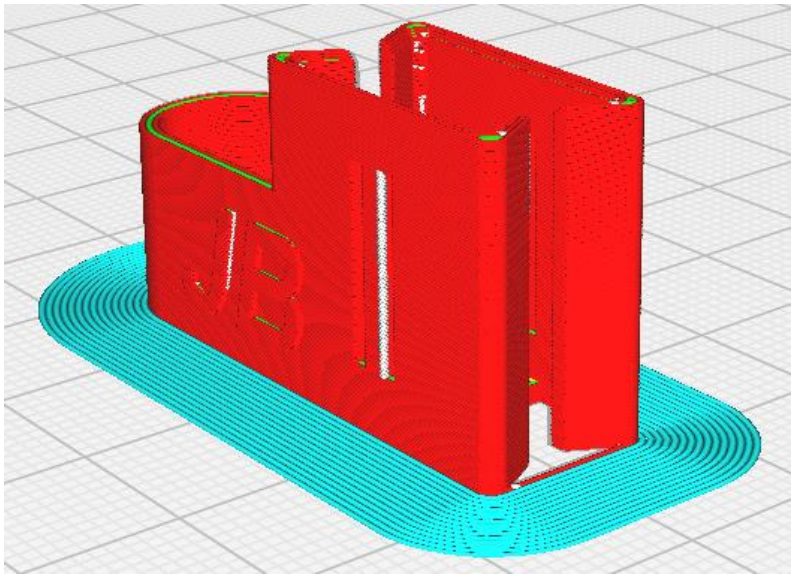
The Handle should look like that below.



Fighting Cura Brim Generation

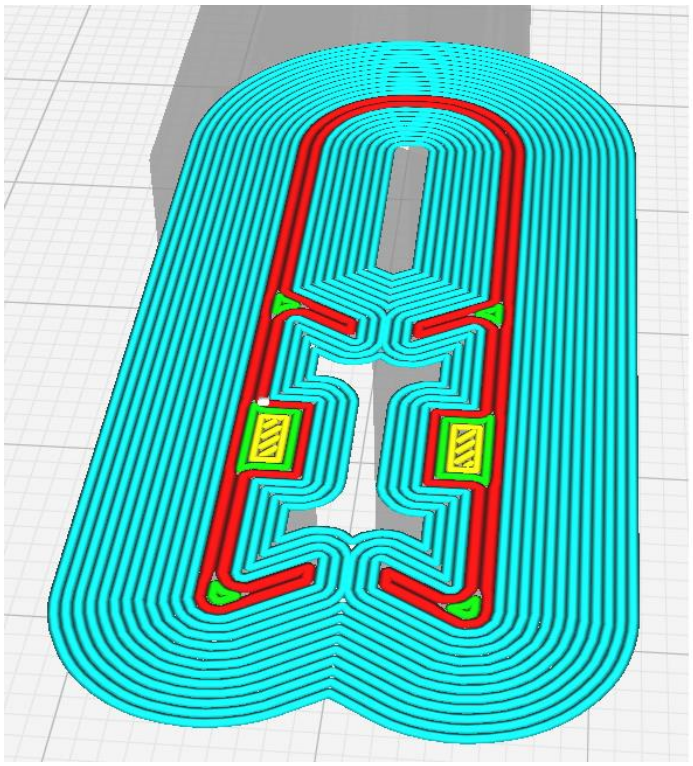
This page is just for information and the steps continue on the following page.

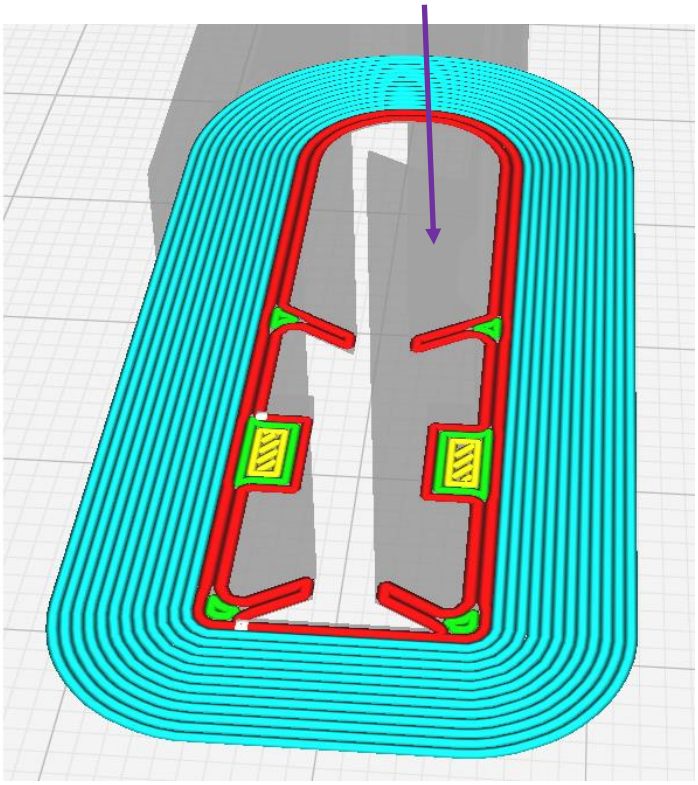
By default the Cura Slicer software will generate an extension of the first layer around the bottom of parts. This aids "**bed adhesion**" by preventing sections of the first layer from separating from the build plate. After printing is finished, the brim is removed by hand. Cura has a **Brim Only on Outside** setting, but because the handle geometry is not closed the inner region will have a brim. We can prevent this by adding a small "blocking" member.



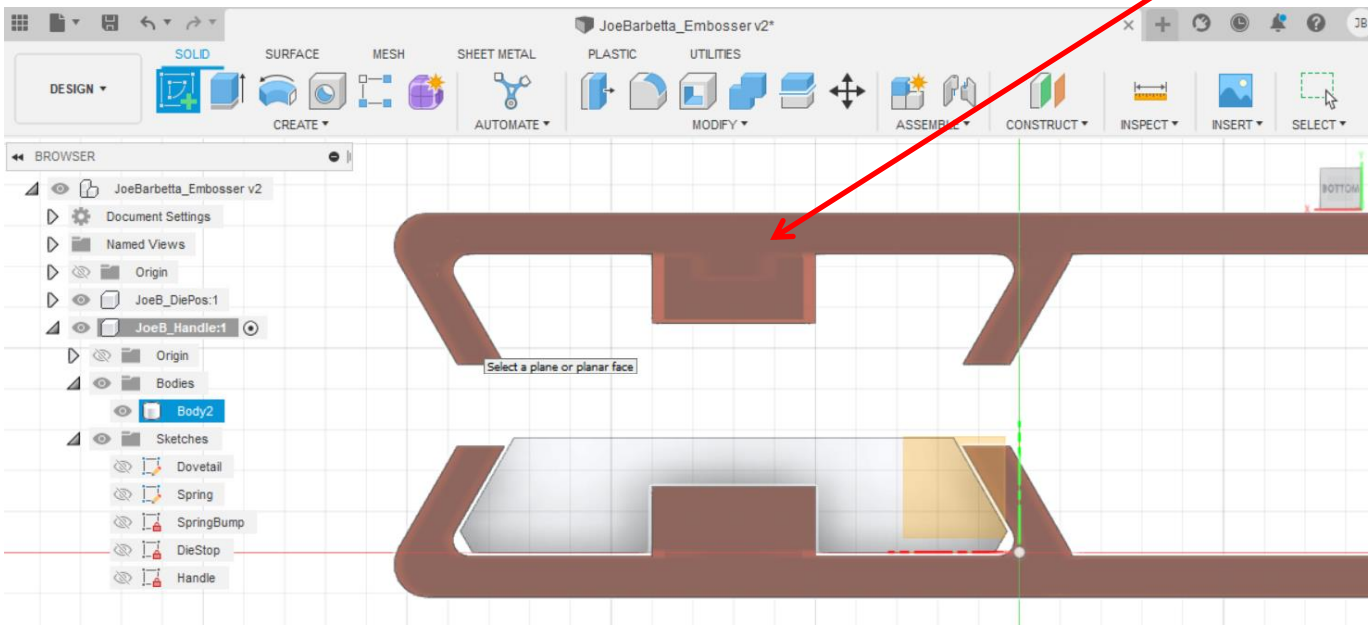
This is the view of the Handle when it is opened in the Cura slicer program and sliced. To ensure that the first layer properly adheres to the build plate a **Brim** is generated as shown in blue.

The two below screenshots show the first layer as shown in Cura. Even with the **Brim Only on Outside** setting enabled, the interior area of the paperclip has a Brim as shown in the left screenshot. This interior brim will be difficult to remove. The right side screenshot shows the result with the Brim Block, wherein there is **no brim in the interior area**.

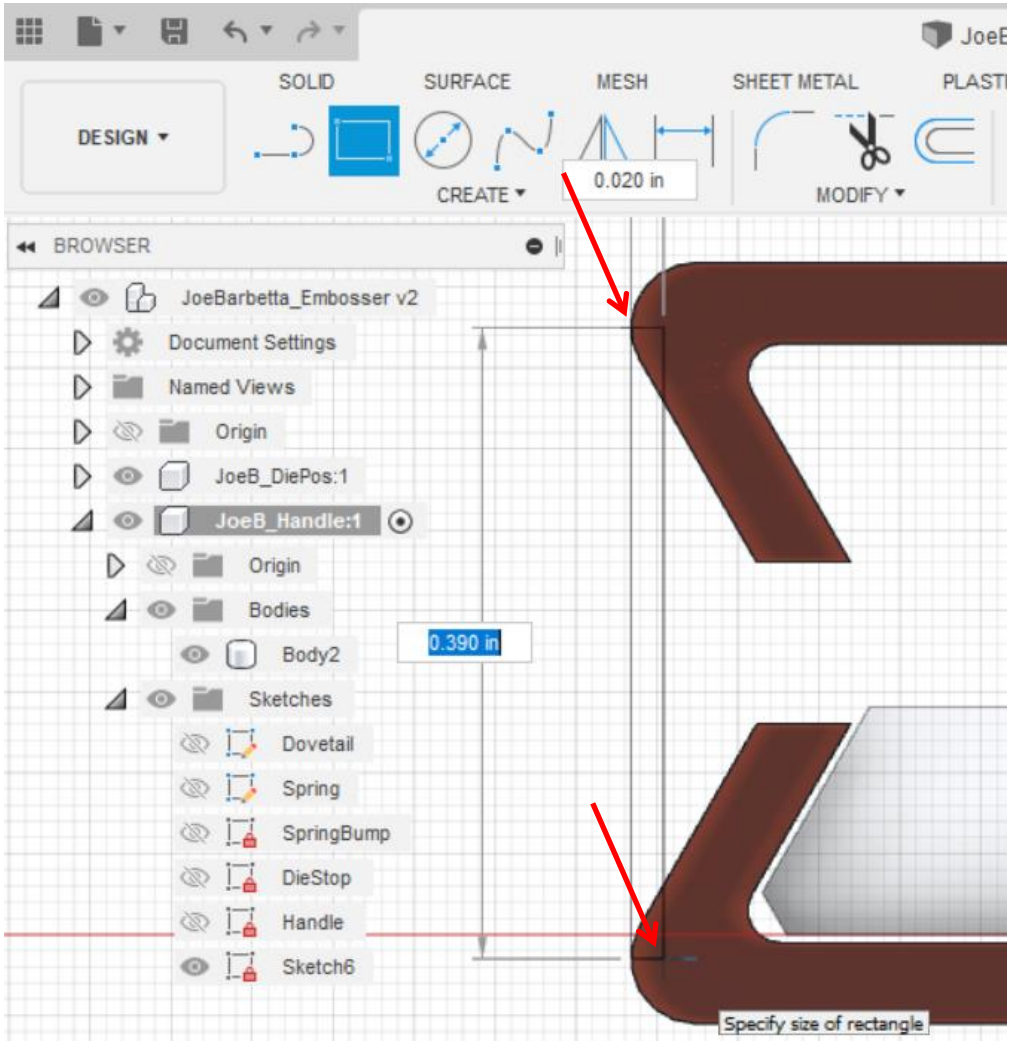




- as done earlier, change the view to a Bottom View, select **Create Sketch**, and click on the **bottom surface**.

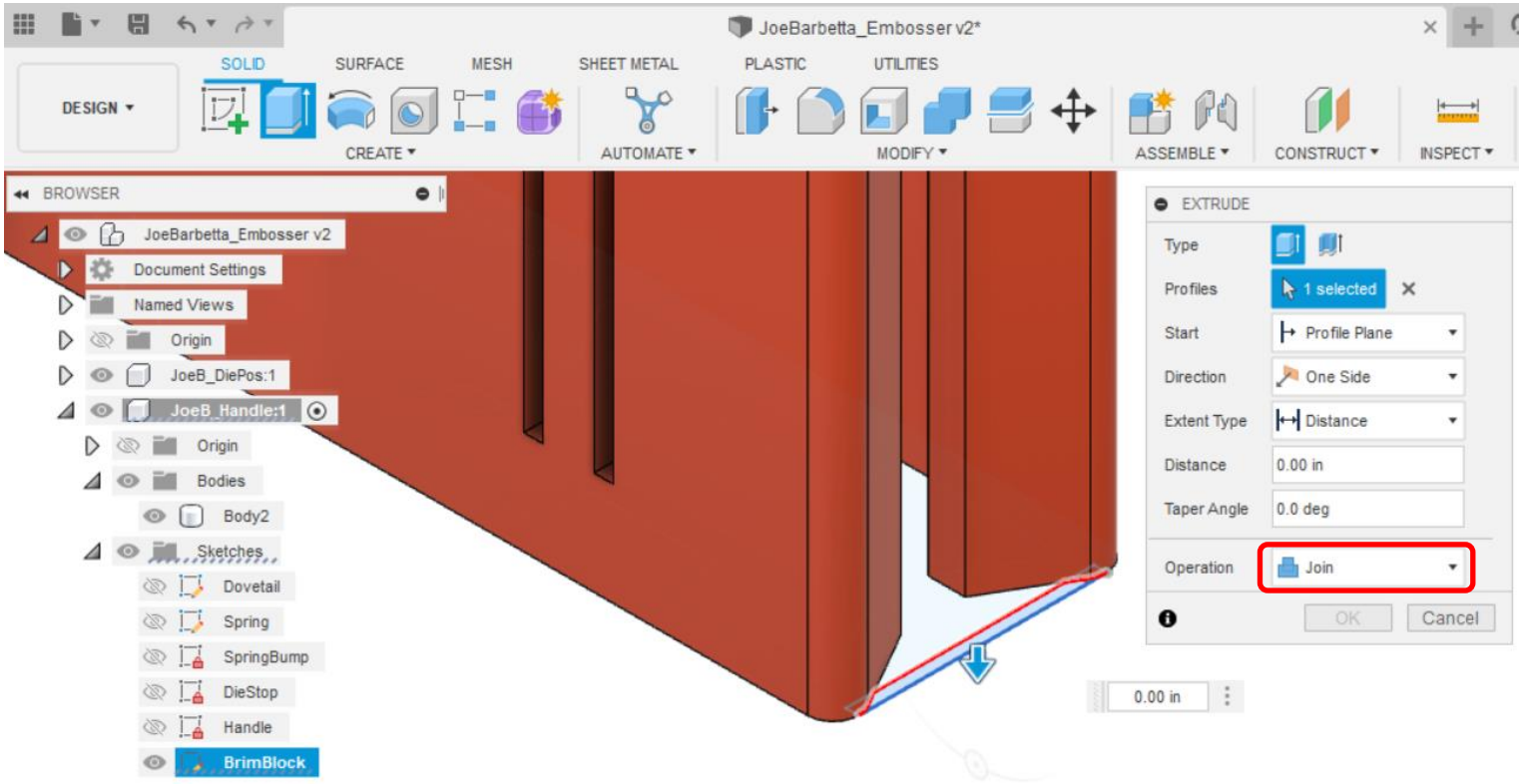


- select the **Rectangle** tool and start the rectangle at the top point and use a width of **0.02** and a height of **0.39**
- click **Finish Sketch**. The placement and dimensions of this rectangle are not critical. We just want to close up the end.

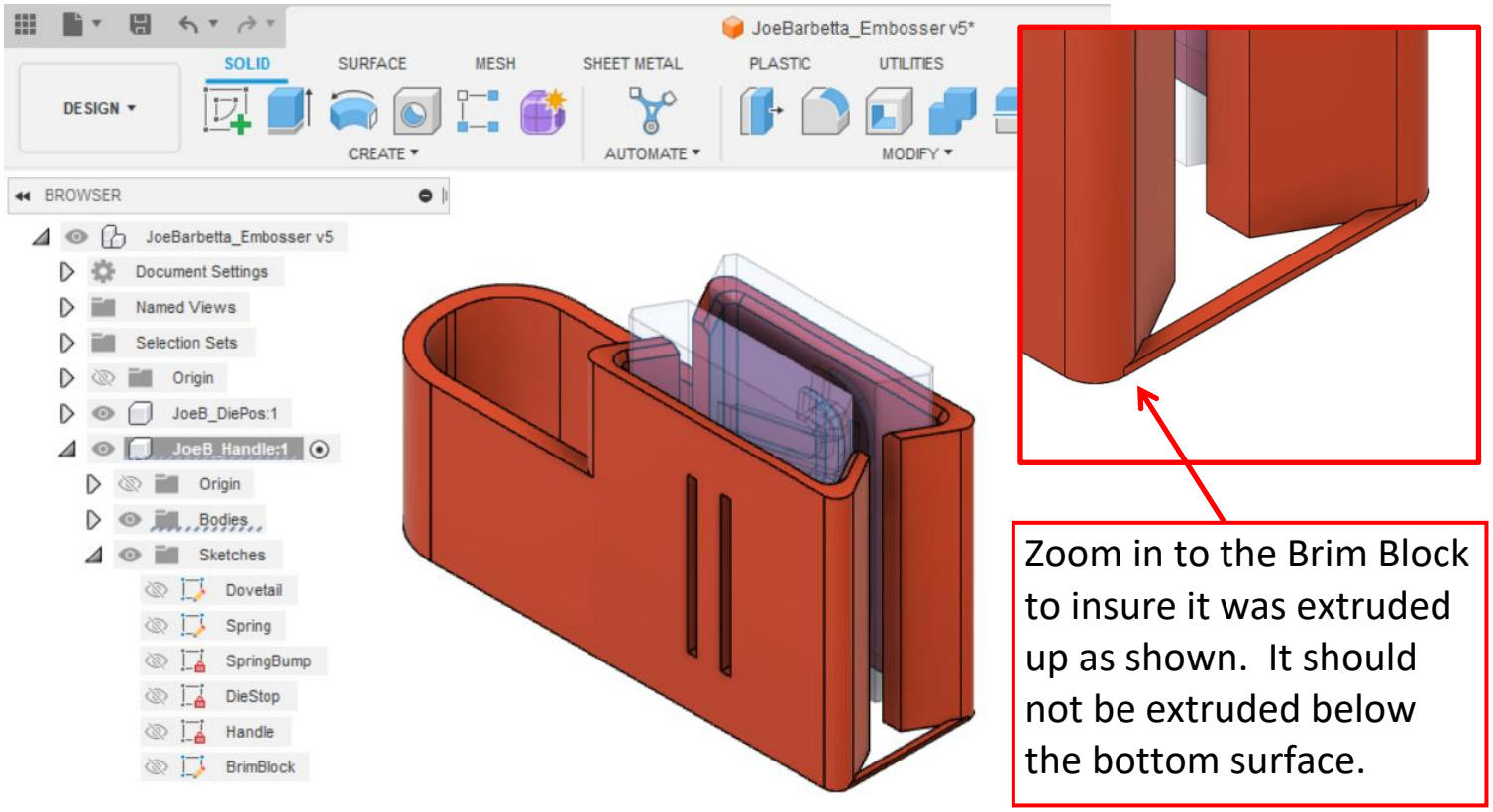


- reset to the **Home** View again and zoom into the are where the rectangle was just created
- select the **Extrude** tool, enter **-0.01** (note the minus sign), ensure the **Operation** is set as **Join** and click **OK**.

Note that this section we created is so thin that it will just break away with the brim. However, it will be effective in blocking brim generation inside the handle.



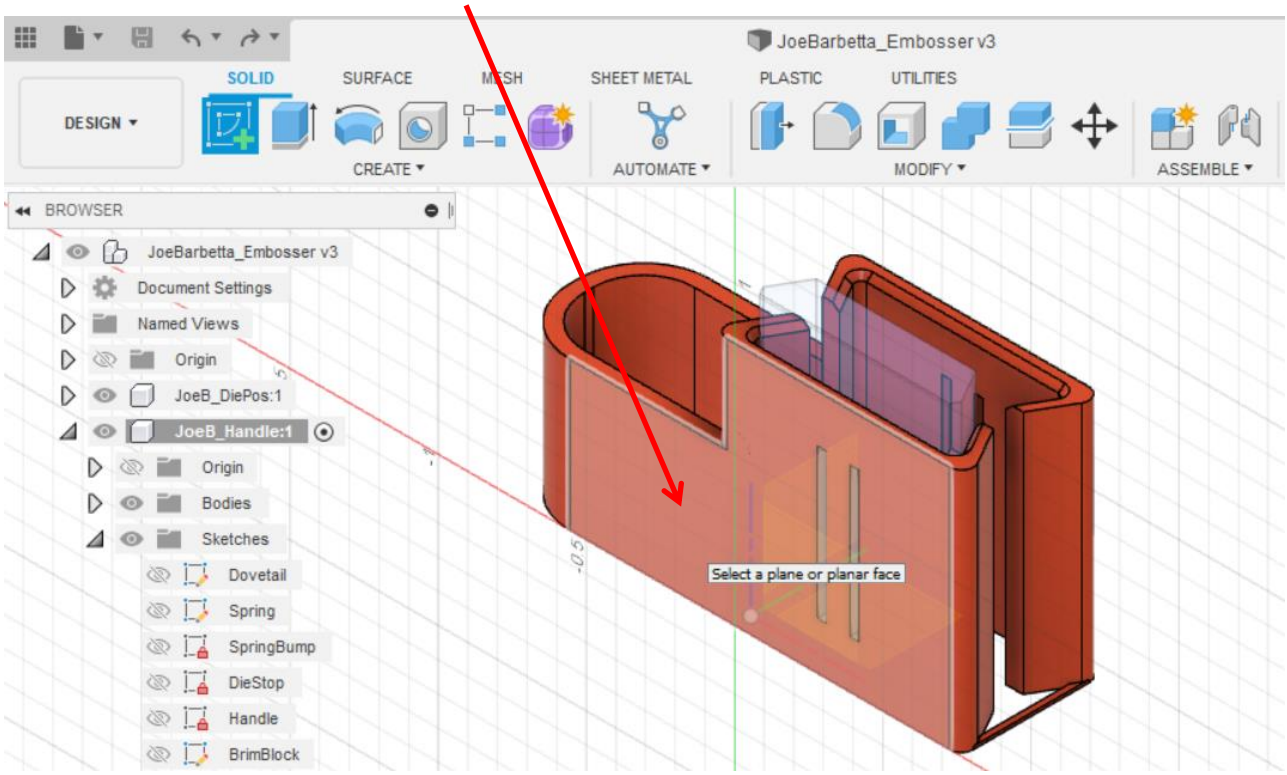
- click on **Home icon** at the **View Cube** and the design should look like that below.



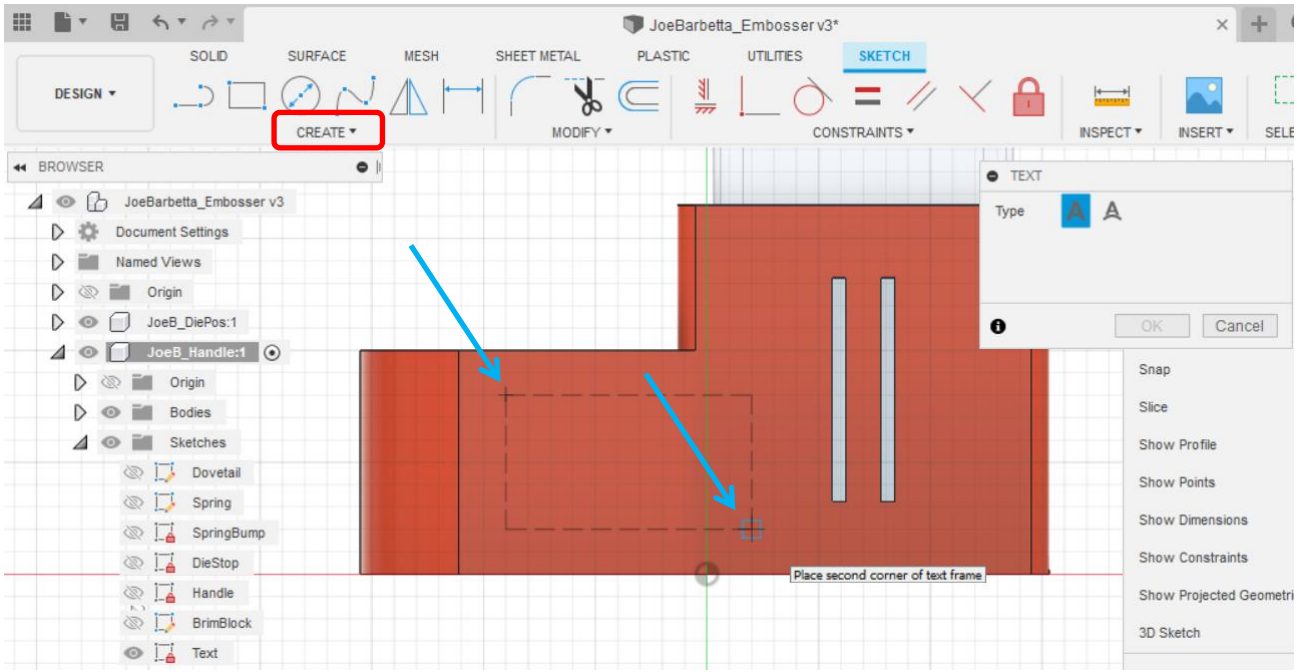
Adding Text to the Handle

To identify the Handle as yours, you want to add your initials to the Handle.

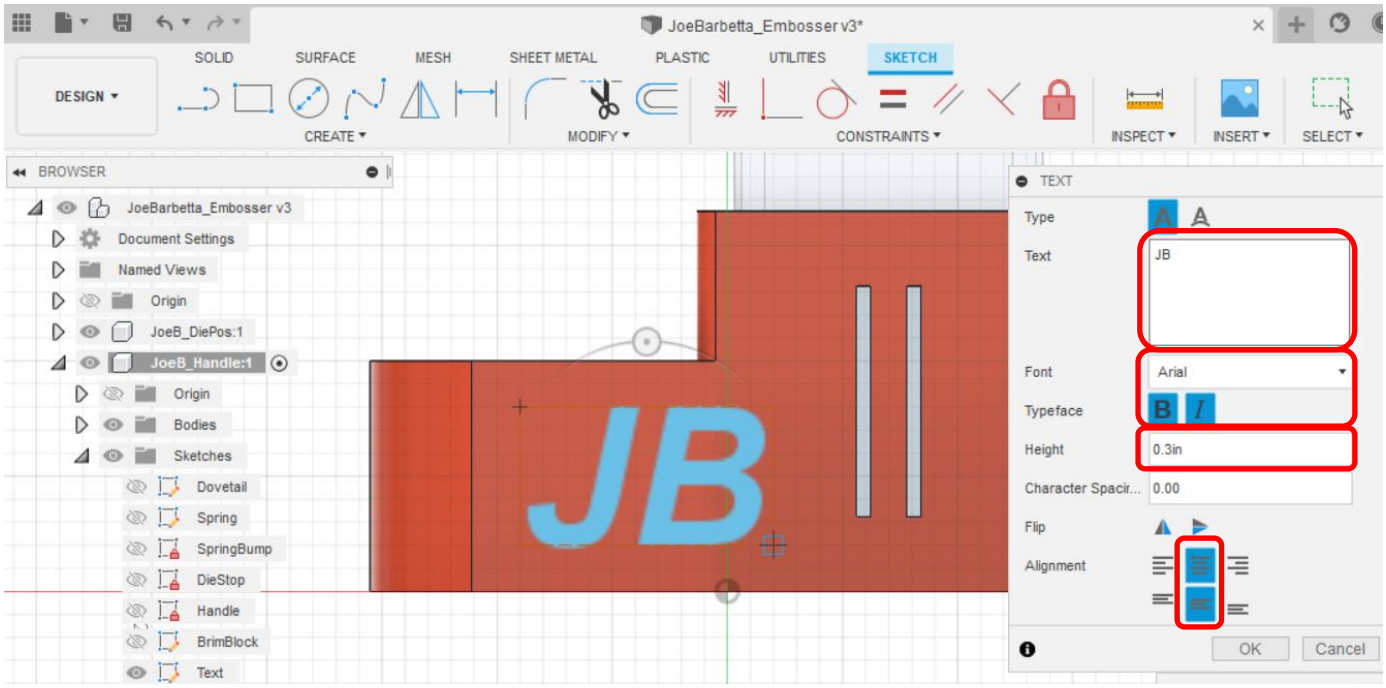
- reset to the **Home** View
- select **Start Sketch** and **click on the side face** of the Handle



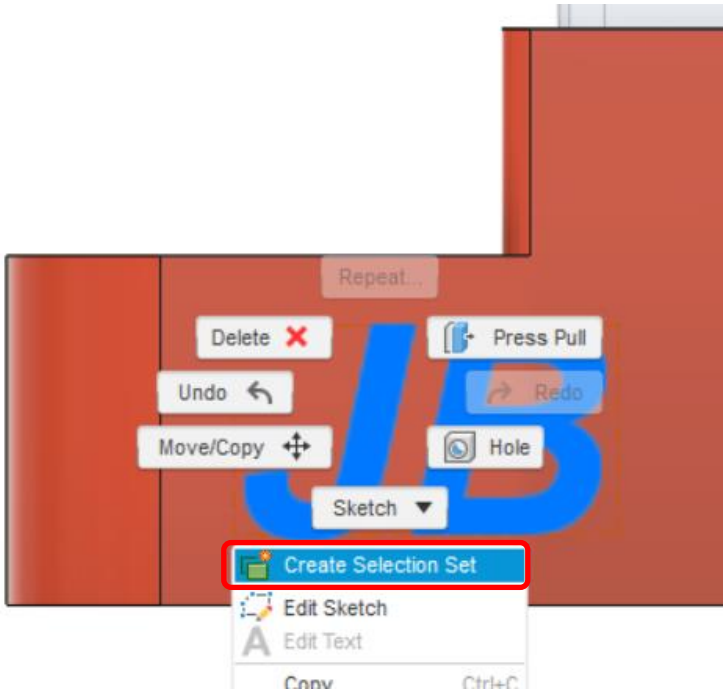
- from the **CREATE** pull-down menu select **Text**
- define the text region by **clicking on a position for the top-left corner and then for the bottom-right corner**, as shown by the blue arrows. If you can't establish the top-left shift the view to the right away from the Browser area.



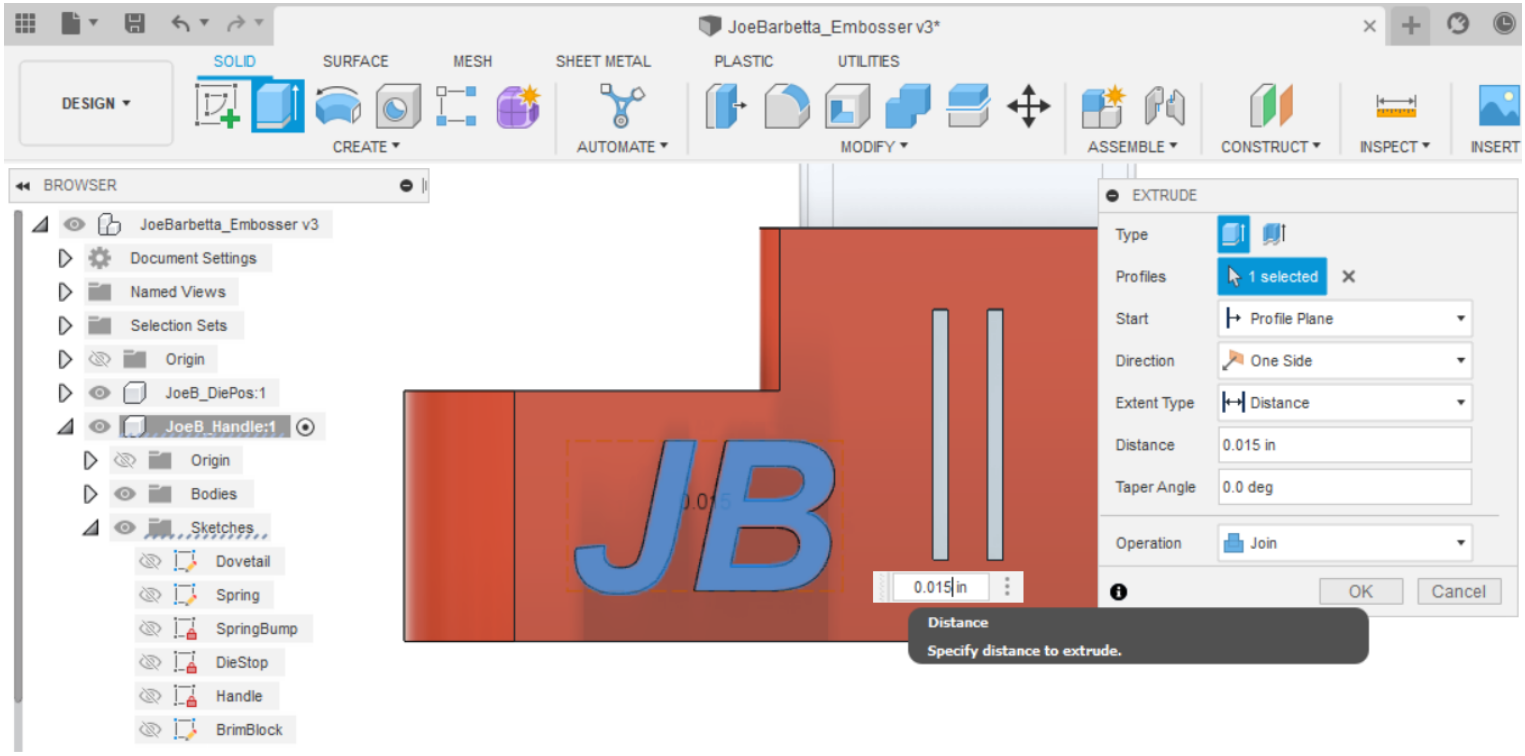
- in the **TEXT** window enter your initials for Text and click on the **Center** and **Middle** icons for **Alignment**.
- try different values in the **Height** box to set the desired height. Here **0.3** is used.
- as per personal preference **Bold** and/or **Italics** can be selected for the **Typeface** and the **Font** can be changed from the default of **Arial**. Note that not all fonts can be extruded and if the later Extrude step fails, a new font must be selected.
- click **OK** and **Finish Sketch** when done.



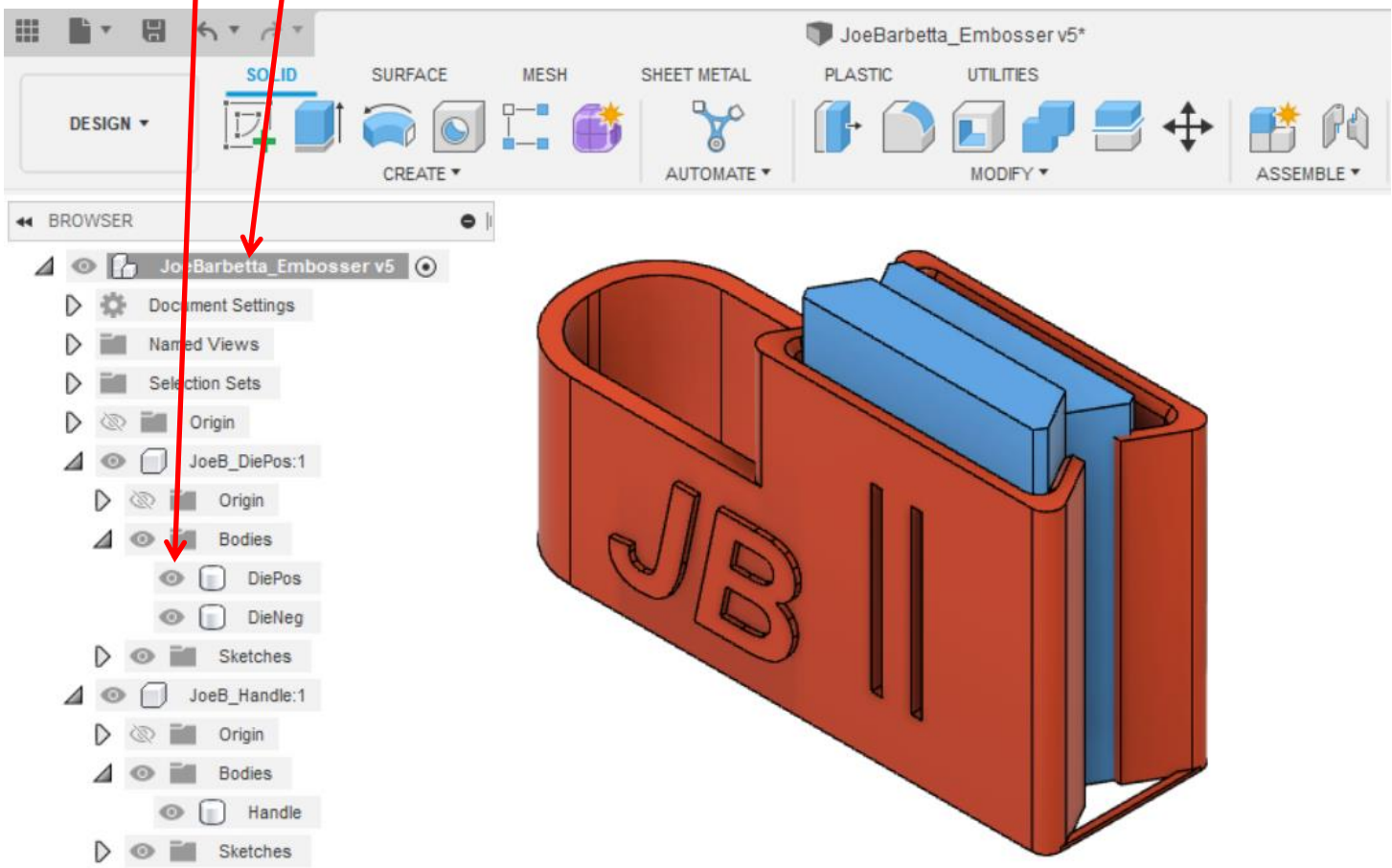
- right-click on the **text** and select **Create Selection Set**. Nothing will seem to happen, but this is needed for the next Extrude step.



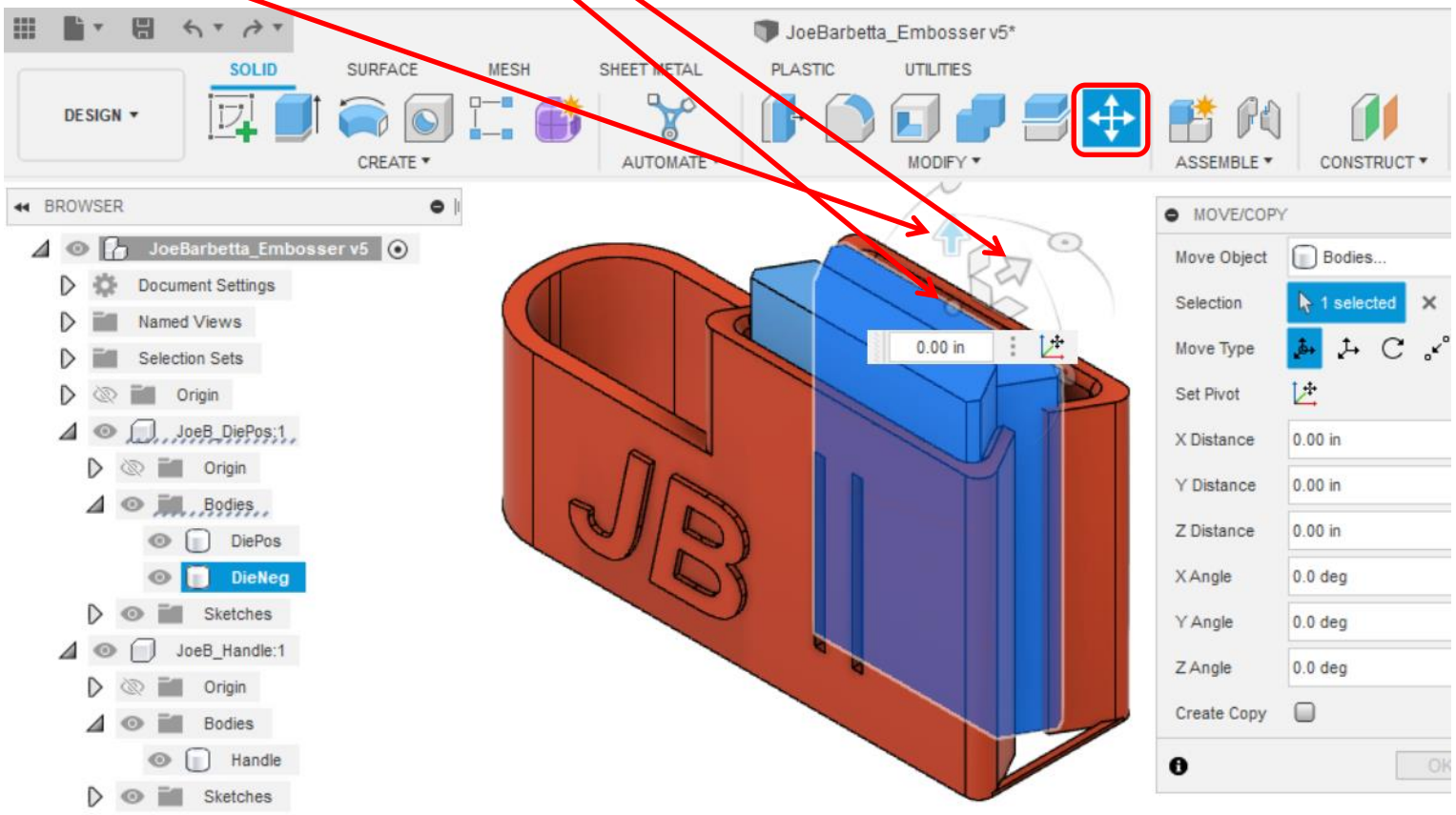
- select the **Extrude** tool and enter **0.015** and click **OK**.



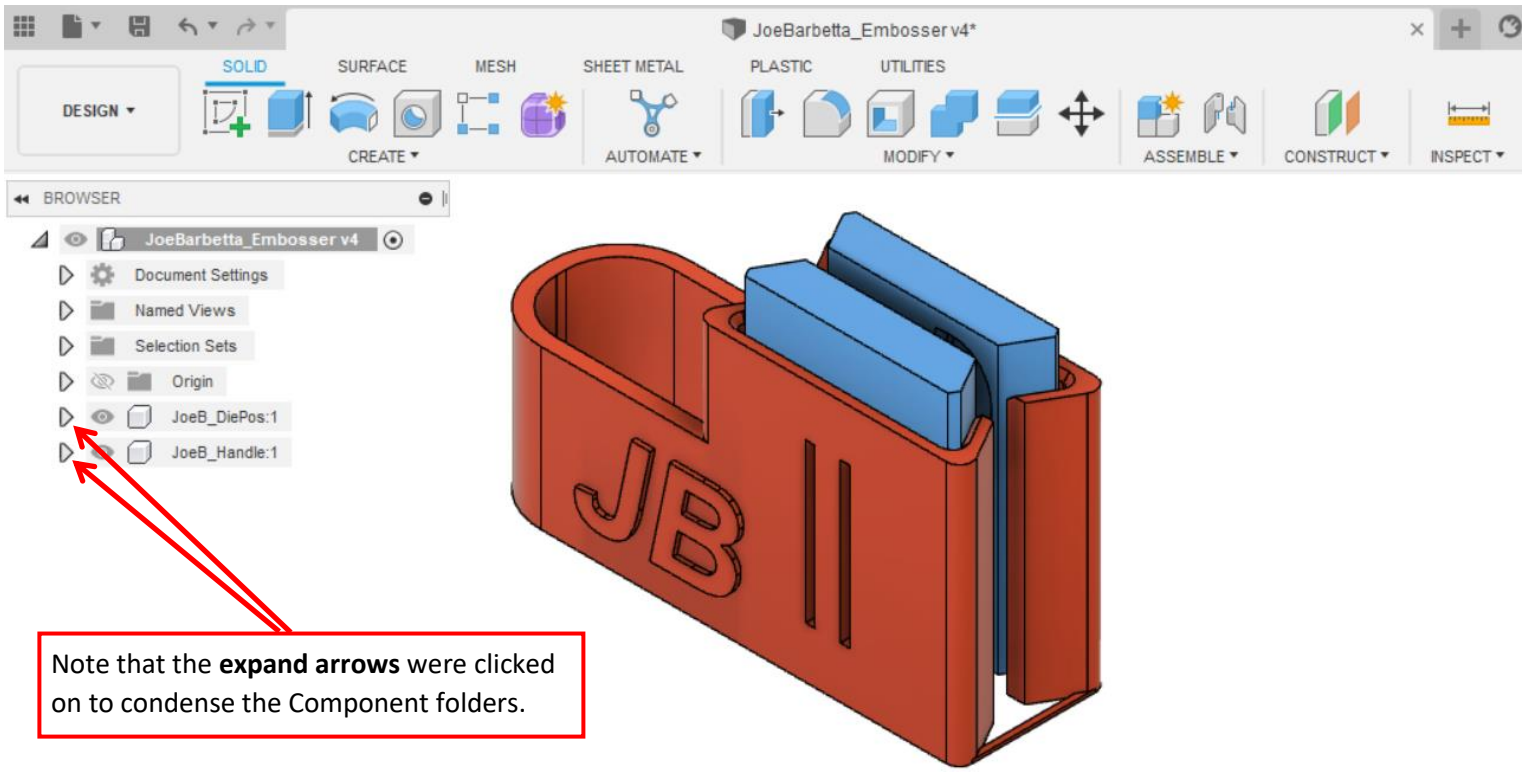
- right-click on the **Project Name** and select **Activate** to allow viewing all components
- click on **Home icon** at the **View Cube** and the design should look like that below.
- click on the **Eye icons** next to any hidden Bodies to make them visible



- click the **Move/Copy tool** and the **top edge of the rear Die**
- click on the **Up Arrow**, type **0.055**, click the **Rear Arrow**, type **0.07**, click **OK**



The result show look as below.

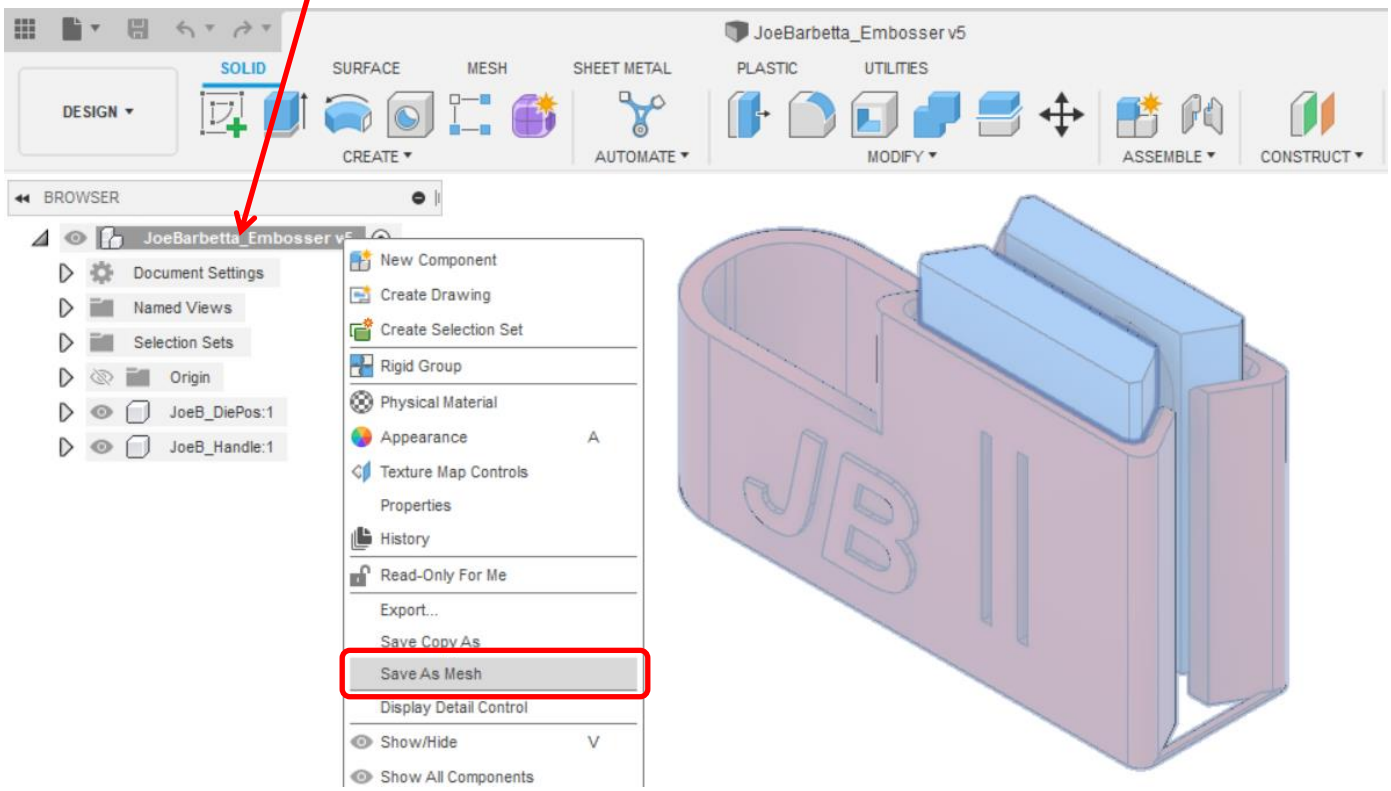


Note that the **expand arrows** were clicked on to condense the Component folders.

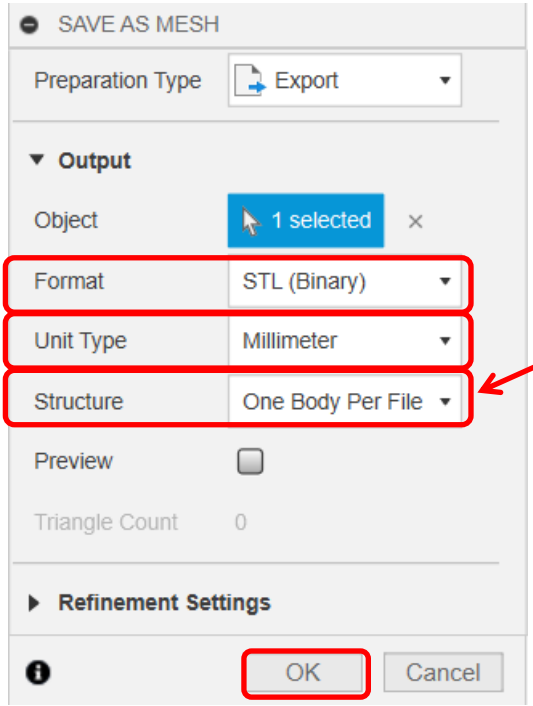
Exporting STL Files

There are various methods for creating STL files. One is using Export from the File menu, however, this can be slow because it sends the job to the cloud. This alternative method is faster.

- right-click on the **Project Name** in the BROWSER and select **Save As Mesh**



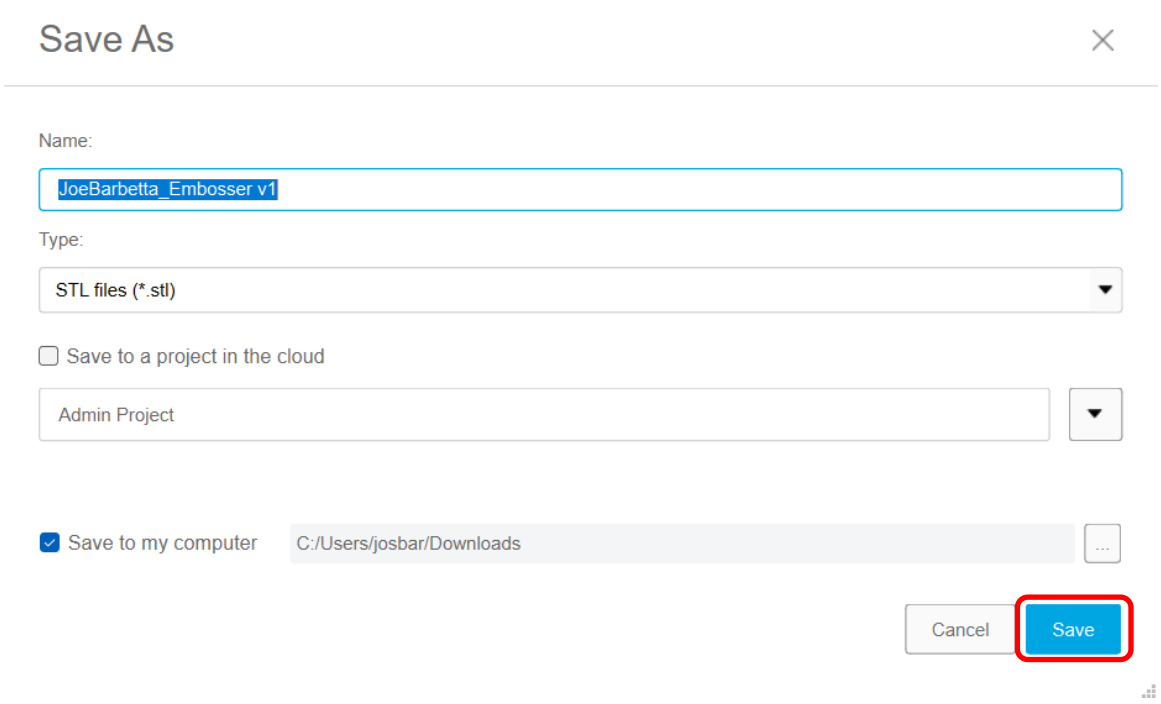
This window will show each time Save As Mesh is used. Ensure that Format is set to **STL (Binary)**, Unit Type is set to **Millimeter**, and Structure is set to **One Body Per File**. Then click **OK**. You will then be prompted to save the file as shown on the following page.



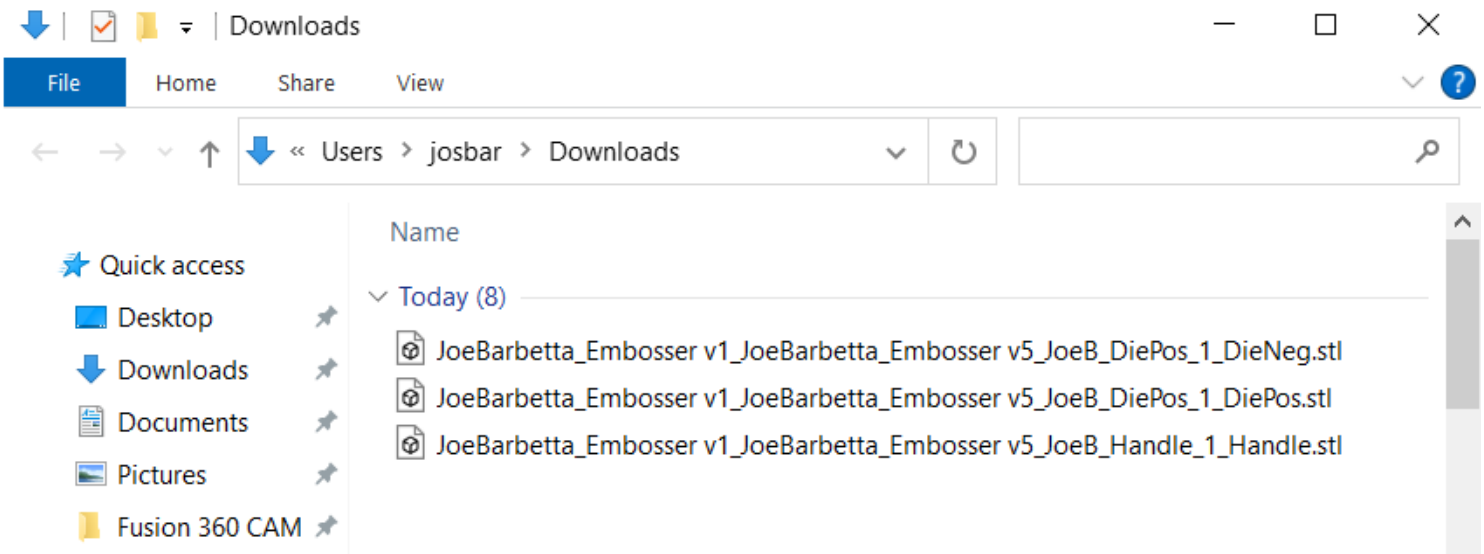
Ensure **One Body Per File** is selected. Otherwise, only one STL file will be created and when imported into the slicer software, it will be impossible to rotate the dies as needed.

When the **Save STL** window appears the File name should match the Project Name.

- click **Save**. Note that the default location is the **Downloads** folder.



- open the **Downloads** folder on the computer. Three files should have been generated with the endings **DieNeg**, **DiePos**, and **Handle**. If only a single file was generated, redo the Exporting STL Files step and ensure that **Structure is set to One Body Per File** in the **Save As Mesh** window. These **3 files** are to be submitted.



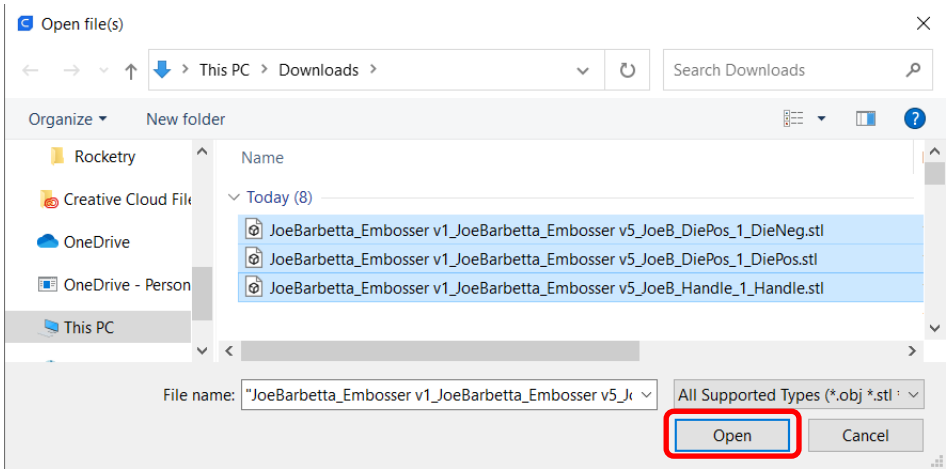
Using the Cura Slicer Program

If this is the first time you are using Cura on this computer, use the **Cura Slicer Program** document on Schoology to get started.

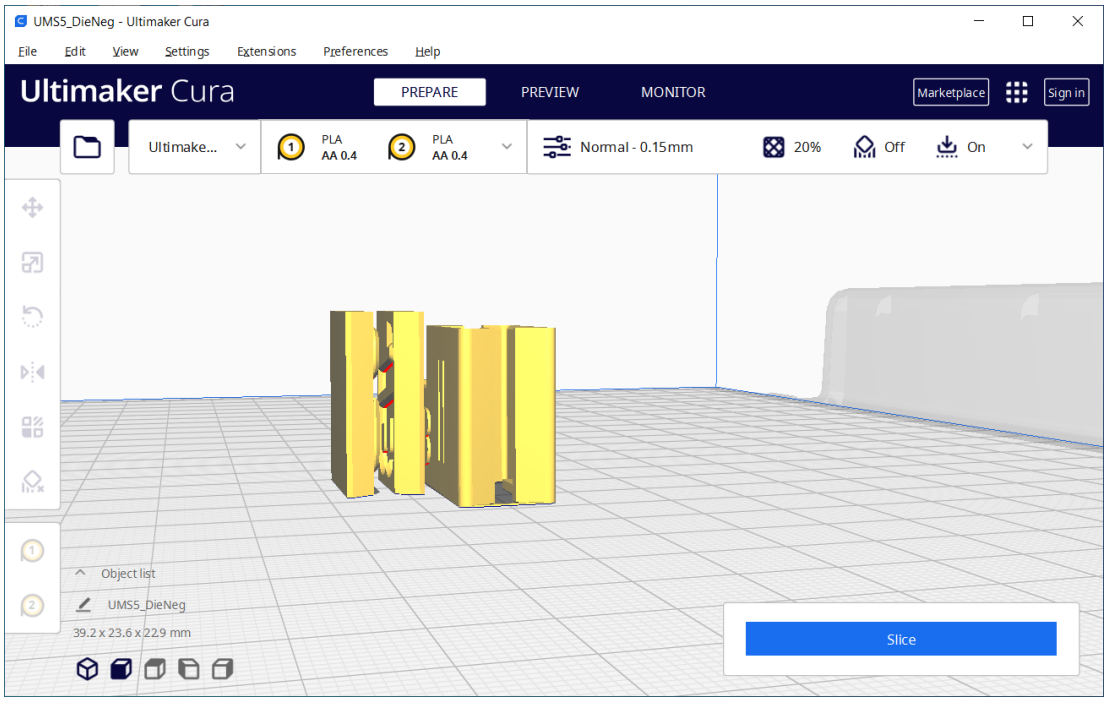
A *slicer* is software that "slices" one or more objects defined by STL files. Each slice comprises the data that the 3D printer uses for that layer. A new file is then generated that is loaded onto the 3D printer.

If the **Cura** icon is not on the Desktop, click on the Windows **Search** tool, which can a magnifying glass icon.
Type Cura in the search box and you should find it.
If Cura prompts you to sign in or create an account, **chose not to**. If it is the first time Cura is used on that computer, it may prompt you to select a printer. Select "**Non Networked Printer**" and then **Ultimaker 5S**, which may be the top default printer. You may also have to click **Next** a few times.

- from the top **File** menu select **Open File(s)**. The top navigation bar of the Open file(s) can be used to navigate to the **.stl file location**, for which the default for Fusion 360 is **Downloads**. Hold the **Shift key** and click on the three **.stl files** and click **Open**.



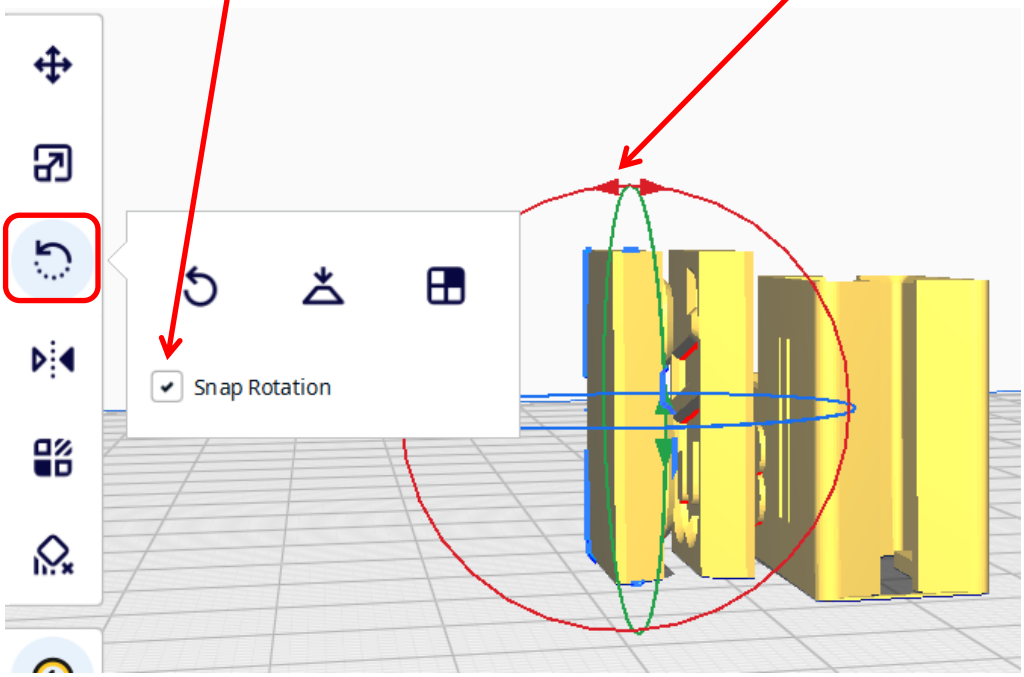
Here are the three objects in Cura after zooming and adjusting the view angle.



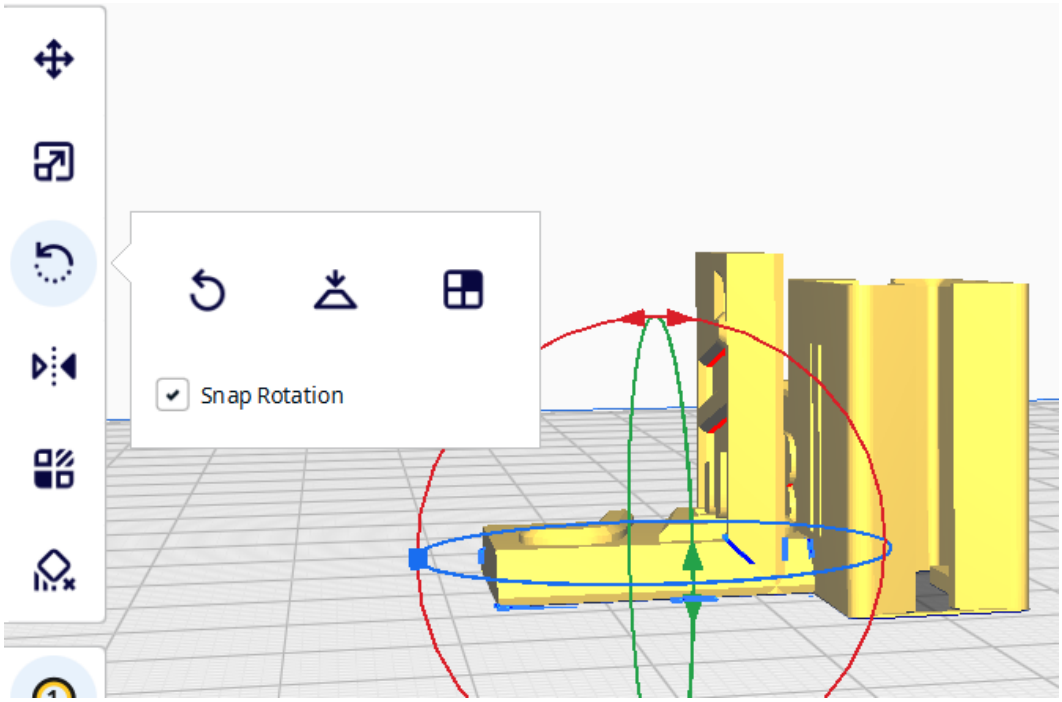
This program is annoying because it uses different methods to change the object view compared to Fusion.
- Turning the mouse wheel zooms in and out. Holding the mouse wheel down allows panning.
- Holding the right mouse button changes the view angle.
- The **View Cube** options at the screen bottom left allows jumping to different views.
The objects will be placed on the plane representing the 3D printer's built plate with the same orientation as in Fusion.

It is very common for objects to **need rotating** to optimize printing. The two dies must be rotated so that the text surface is facing up.

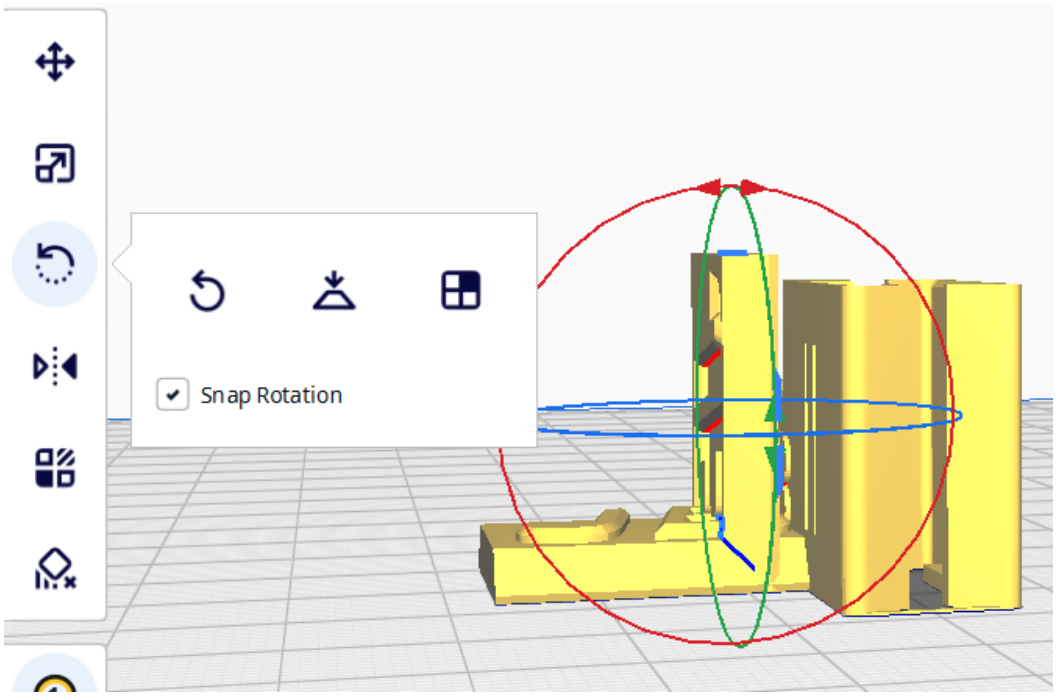
- click on one Die to select it and then select the Rotate tool, which will cause rotation circles to appear around the object.
- ensure that **Snap Rotation** is checked and then click on the proper **arrow** to rotate the object **90 degrees** on the desired axis and in the desired direction. One can always Undo under Edit to reverse any changes.



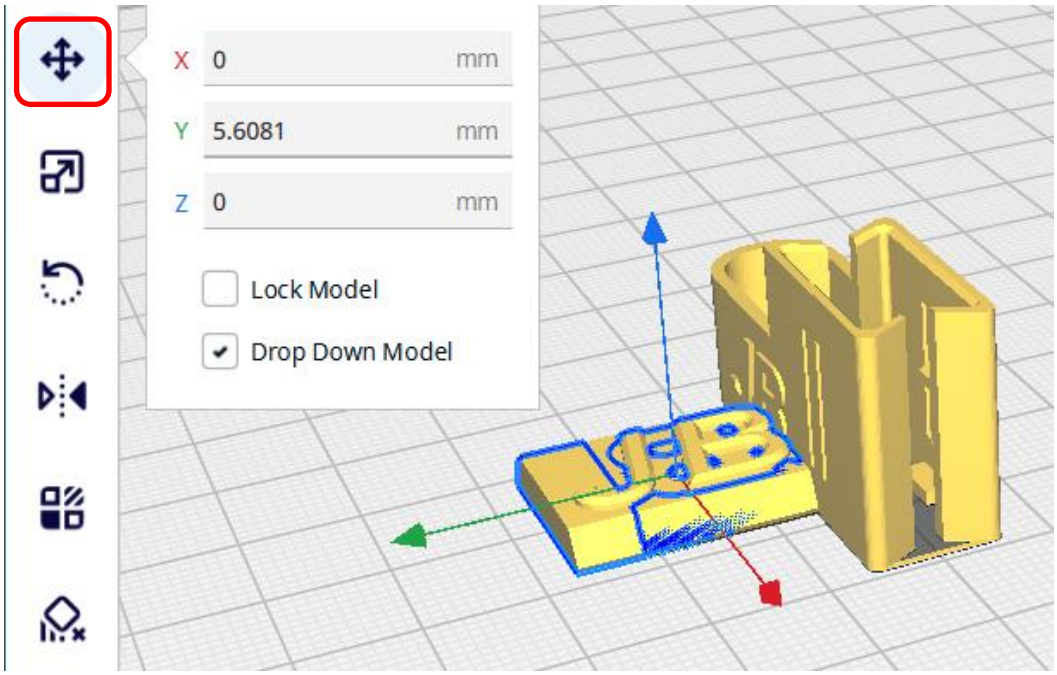
This is the result. Note that the object is intersecting the other objects, but that will be fixed.

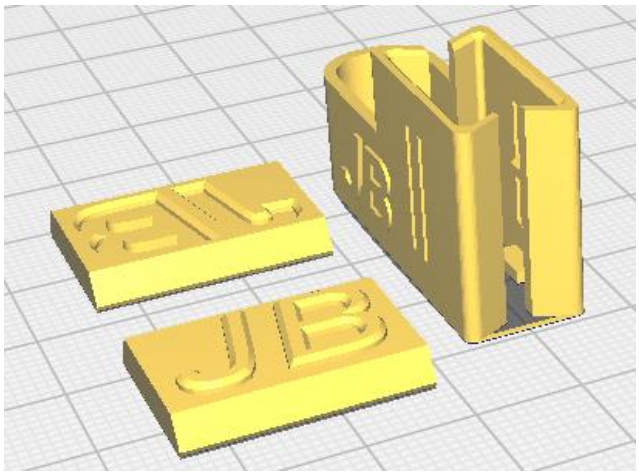


- click on the 2nd Die and rotate it in the opposite direction by **90 degrees**



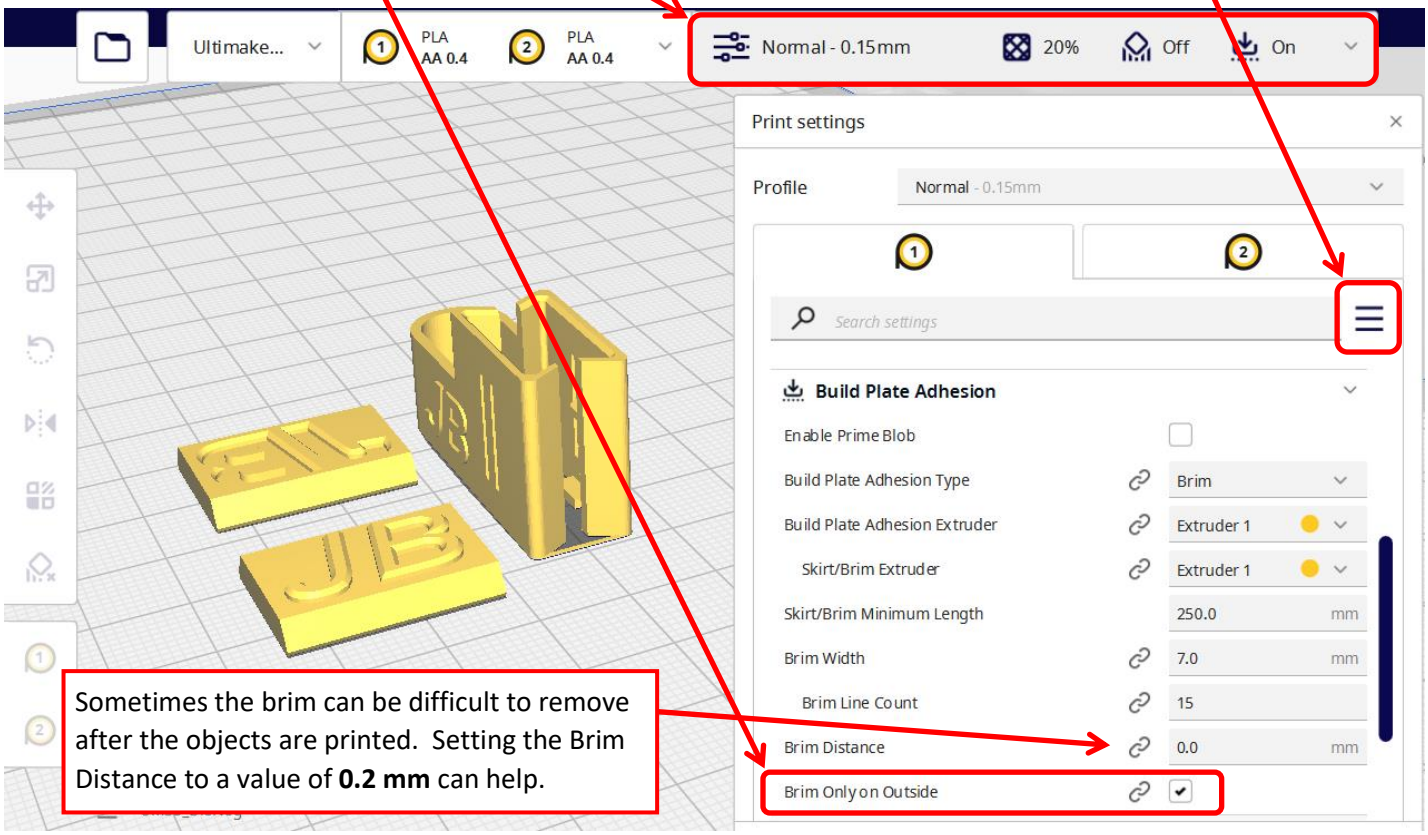
- select the **Move** tool and click on a Die and then use the **red and green arrows** to move it away from the other objects



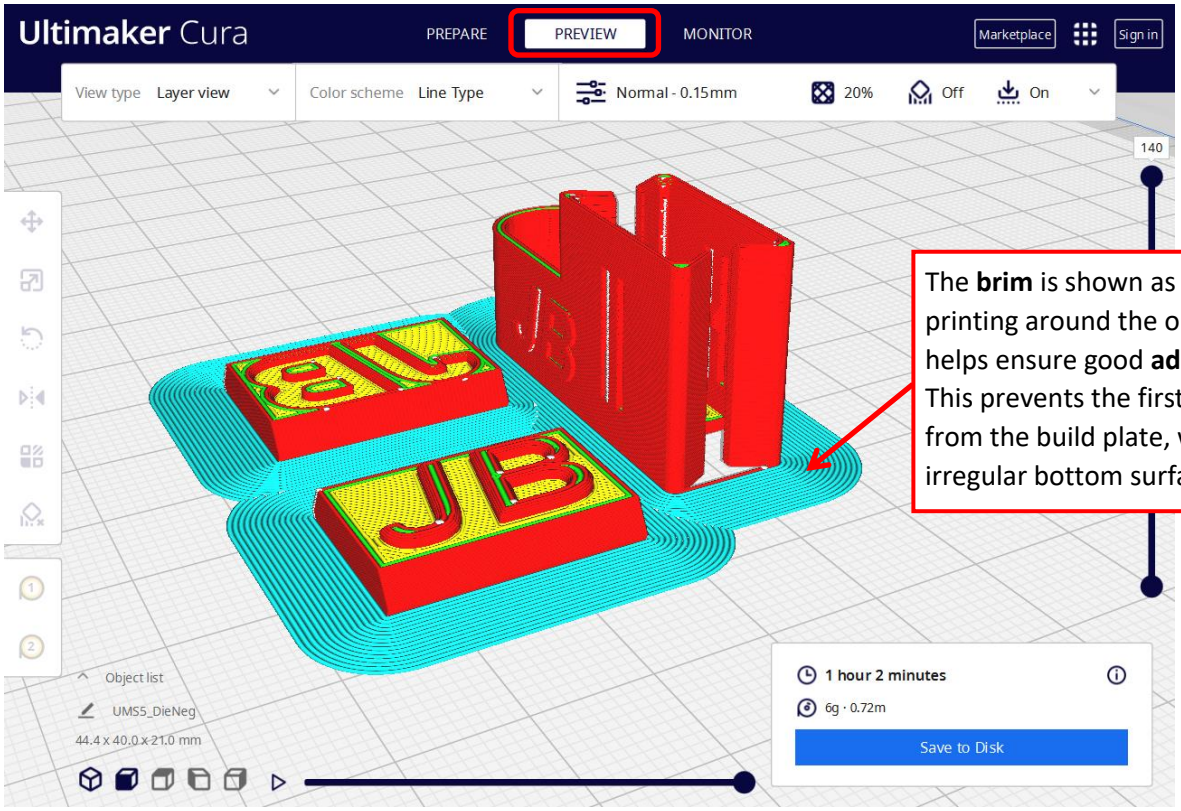


Here is an example of the two Dies and Handle spaced appropriately from each other. Note that the Handle did not need to be rotated.

- click anywhere on the **print settings bar** to open the Print settings window and then the **view menu** and select **All**
- scroll down to the **Build Plate Adhesion** category and click on it to open its settings
- ensure that **Brim Only on Outside** is checked



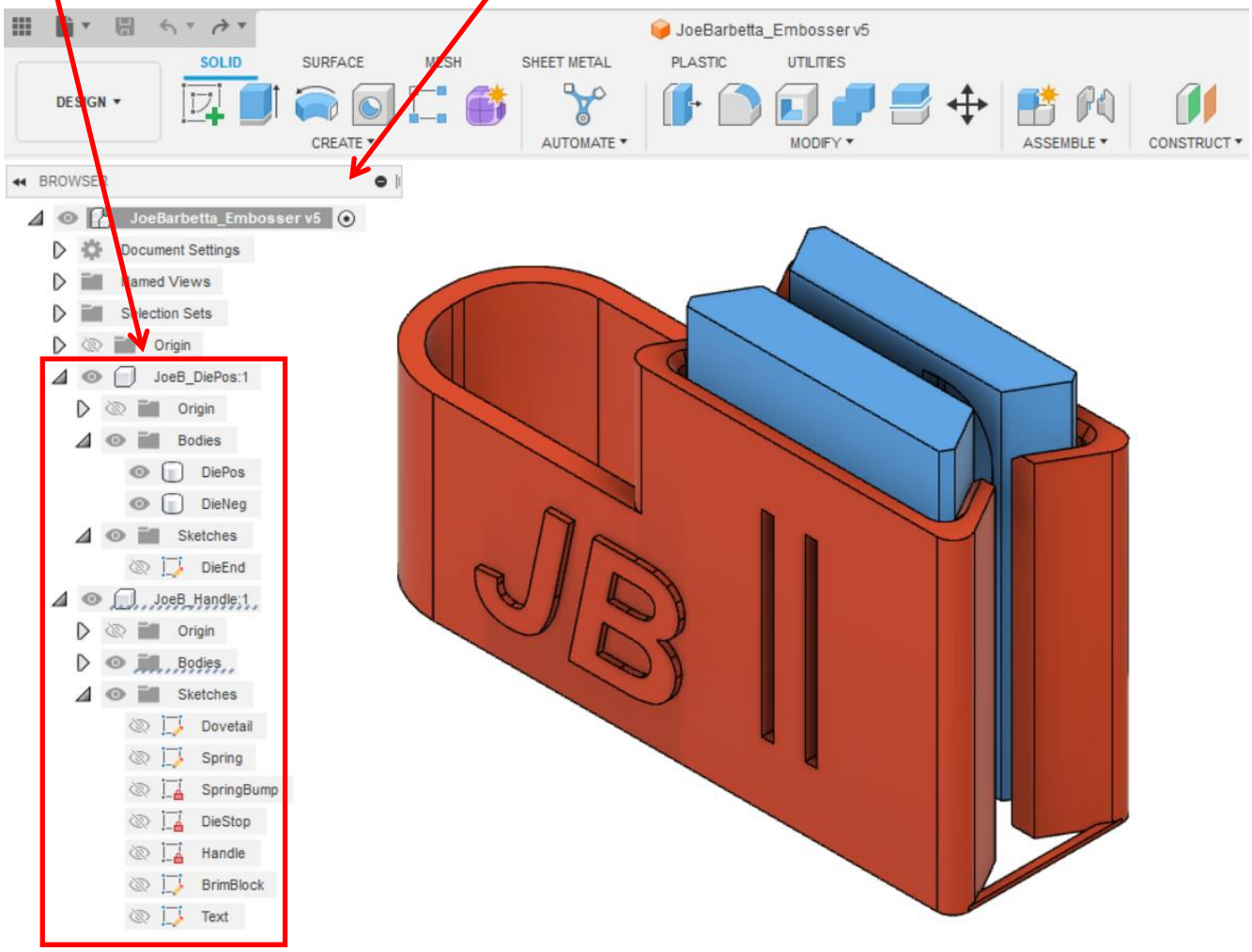
- close the Print settings window and click on the large **Slice** button on the bottom right
- click on **PREVIEW** and Cura will show all the layers.
- if one were to print the object, a USB drive can be inserted in the PC and then Save To External can be clicked.



What to Submit

Upload the **3 .stl files**, a **screen shot of Cura** as shown on the previous page, and a **screen shot of Fusion 360** as explained below.

Prepare Fusion 360 to take a screen shot by **Activating the Project Name** and expanding the Components to show **Bodies and Sketches**. Upload a .png screenshot as shown below.



What happens next? See the next page.

The .stl files from multiple students are loaded into Cura. Below is what Cura looks like after 17 embossers are loaded. In the bottom right it is shown that this batch will take almost 18 hours to print. The printer will run overnight and then the handles and dies will be separated from the brim material and distributed to students.