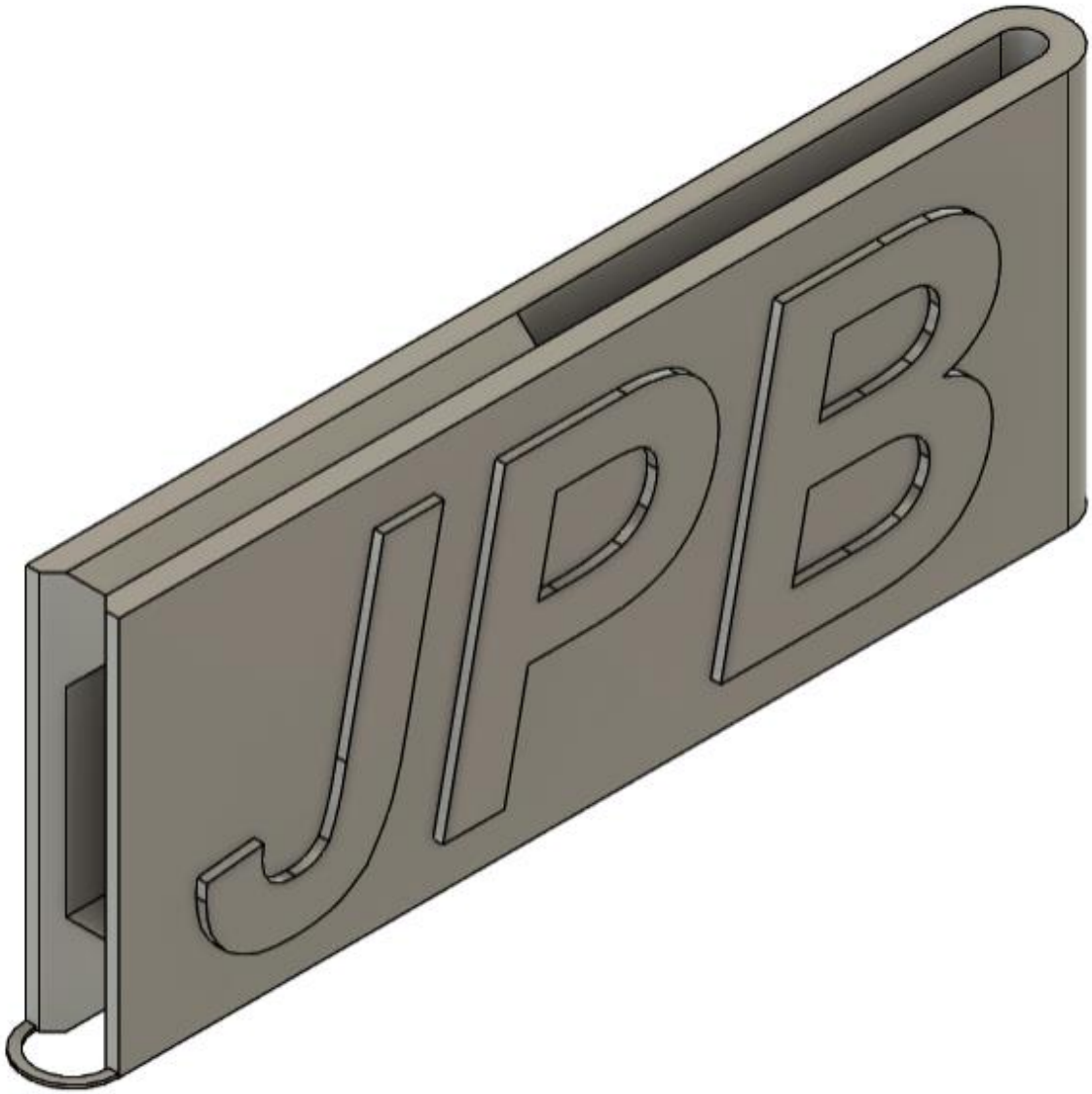
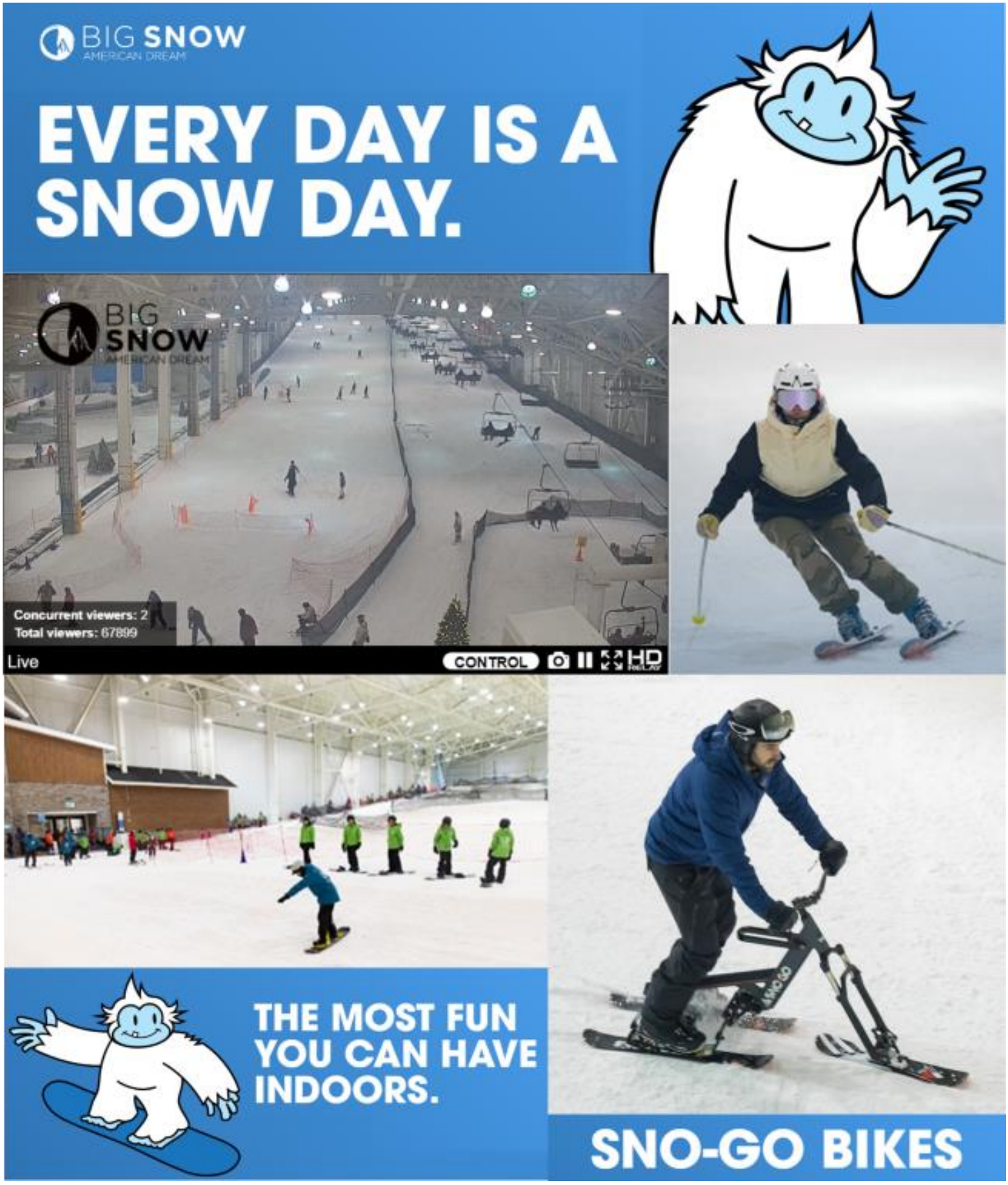
Mr. Barbetta's *"I don't care about 3D Printing. I just want the art credit."* class

Are your papers a mess?

Design a 3D Printed *POD (Paper Organizing Device)*



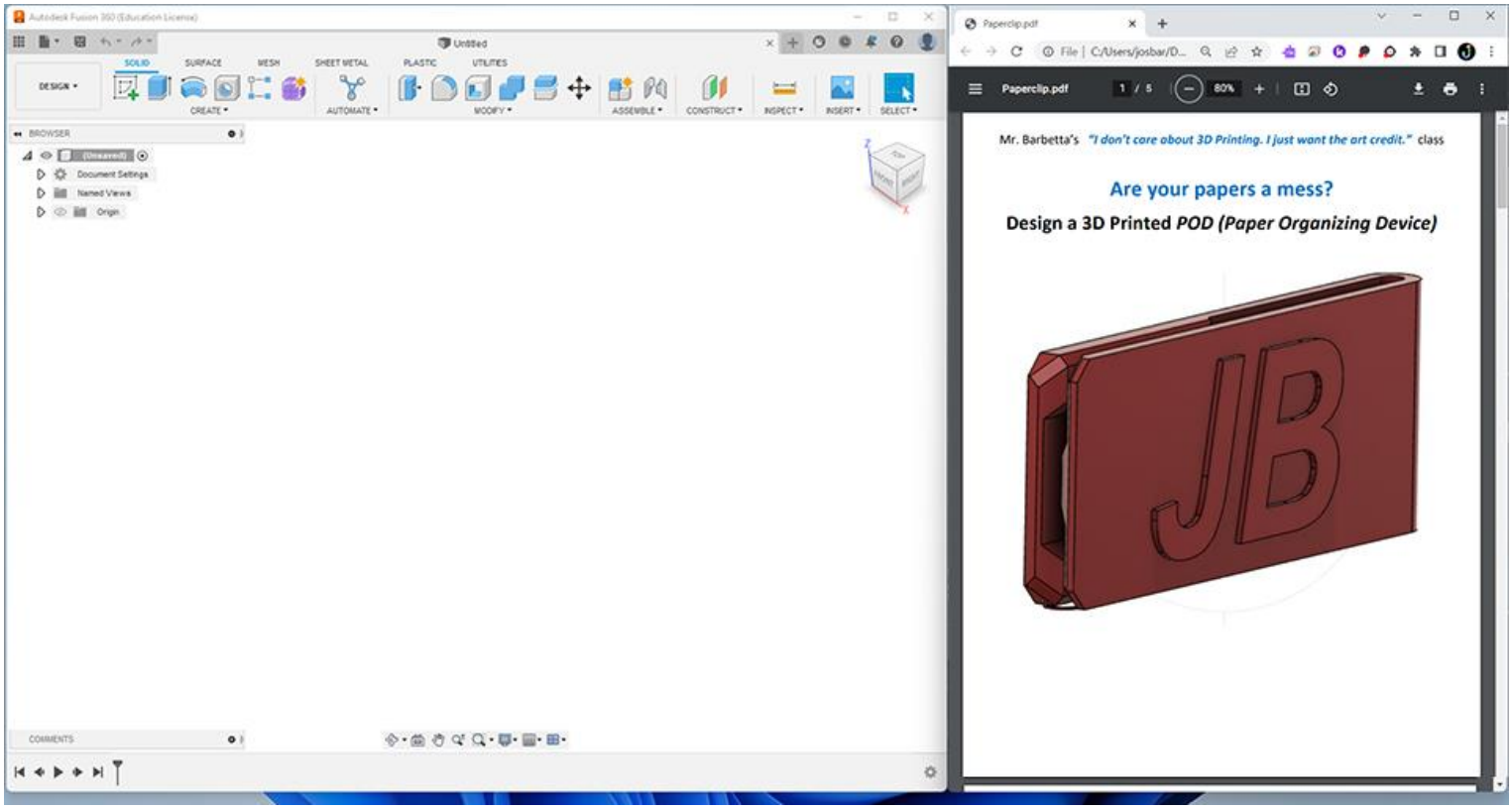
This lesson is sponsored by Big Snow American Dream



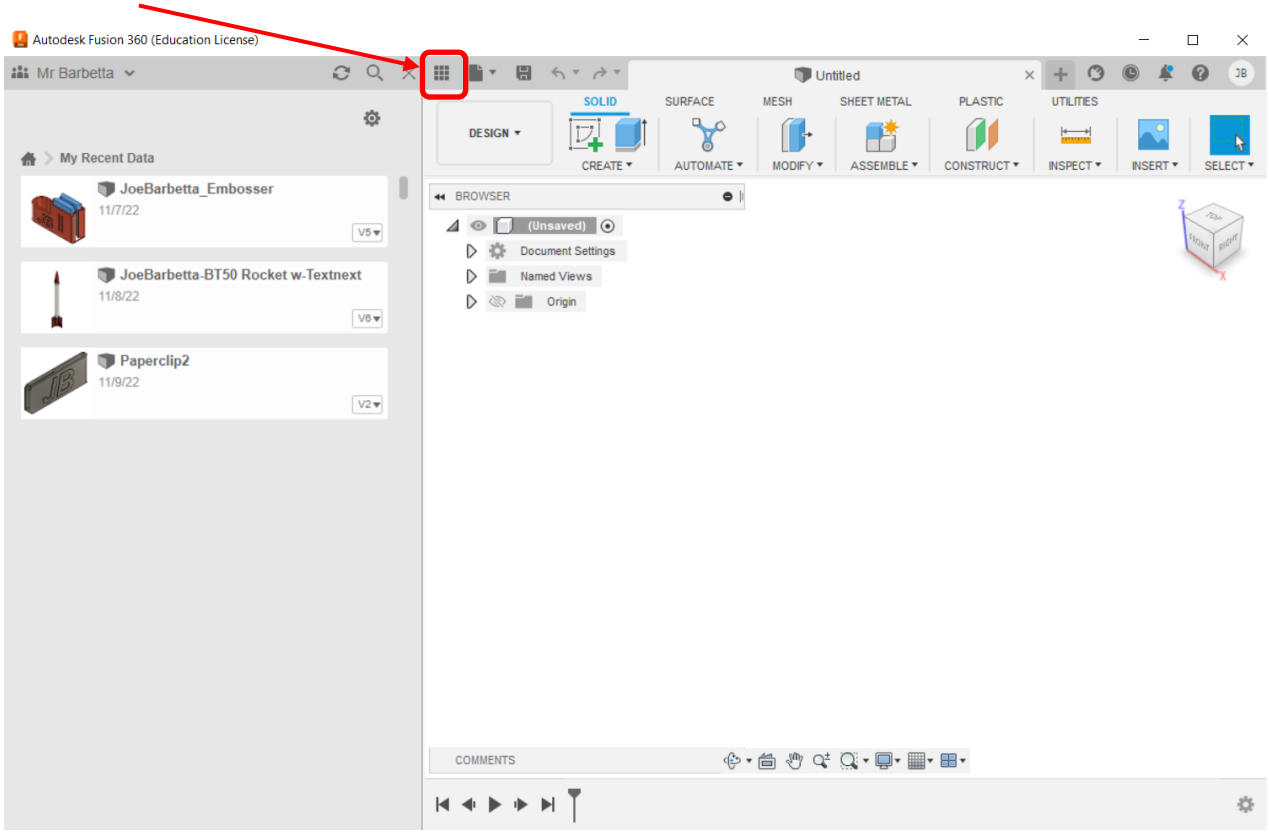
Contents

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%.



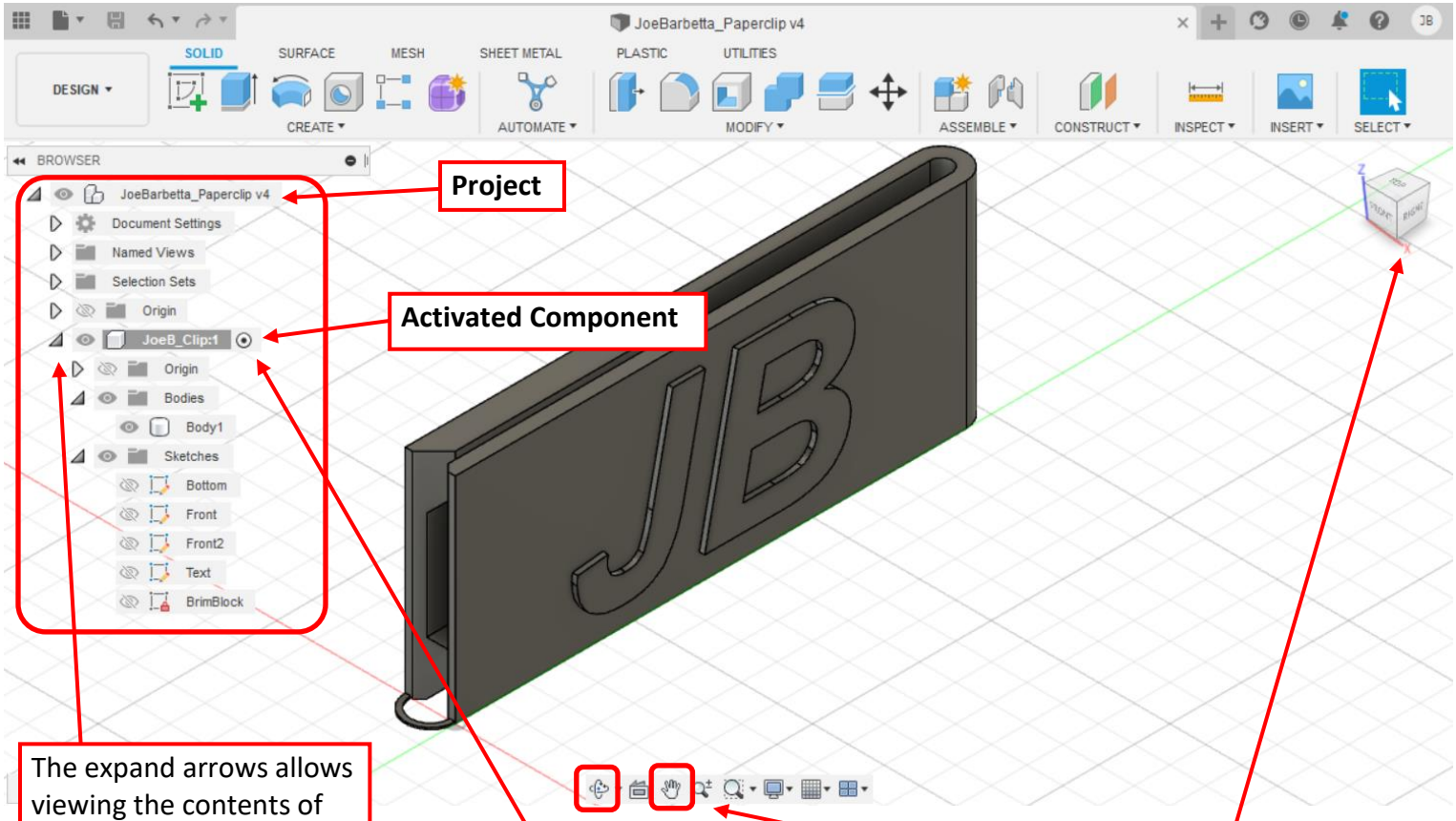
The Fusion window will not allow its width to be reduced much so for smaller computer screens a trick is to click on the **Data Panel icon** and then move the window to the left with the Data Panel off the screen.



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.

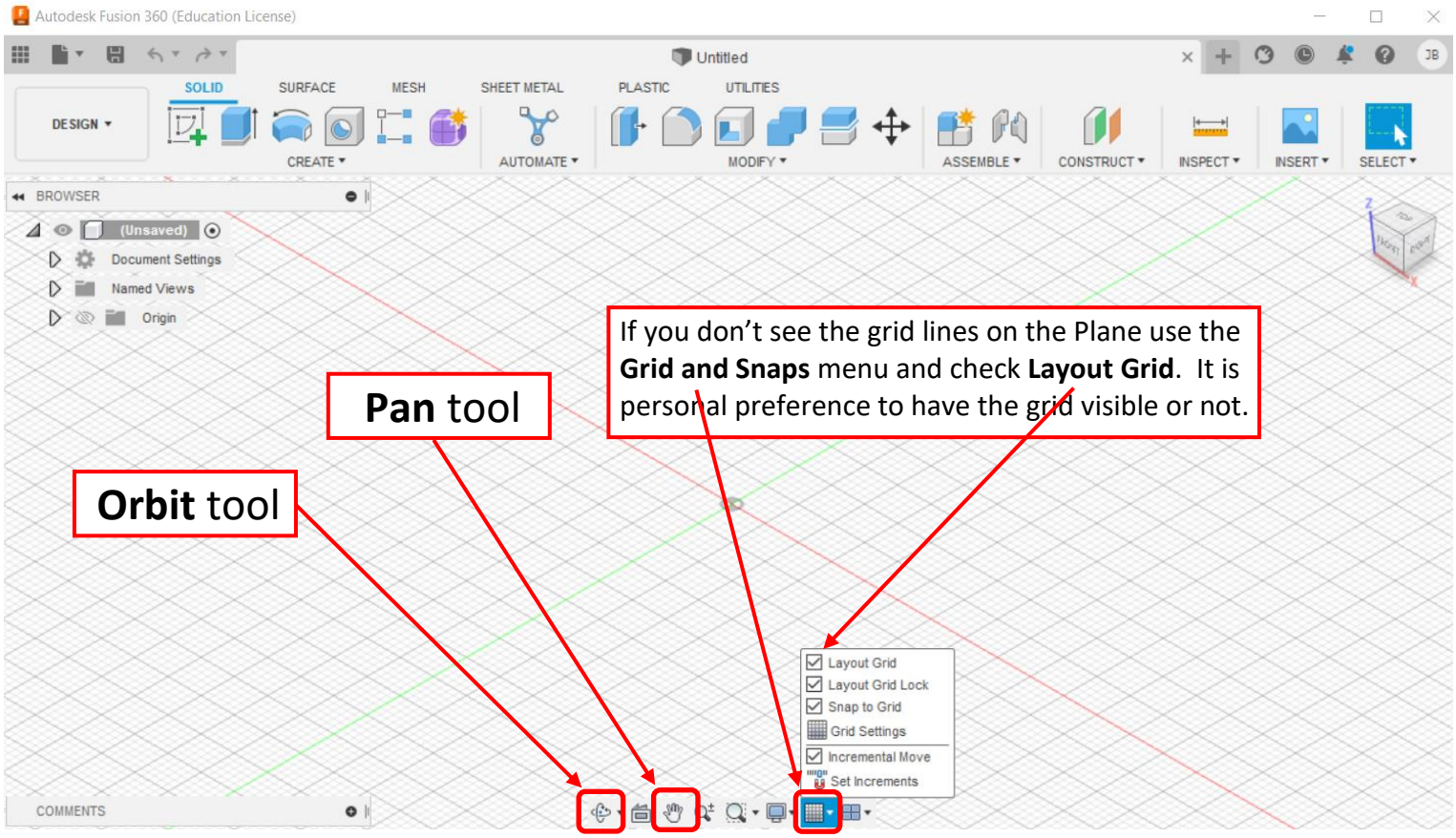
The **Orbit** and **Pan** (hand icon) tools are commonly used to adjust the view of the design. After using one the **Esc key** is needed to exit the operation.
The top right **View Cube** is another method to rotate the view of the design.

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.

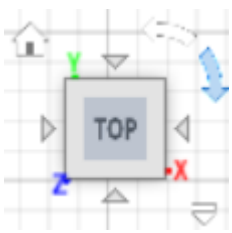


Here is a close-up of the View Cube at the top right of the window.

- 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



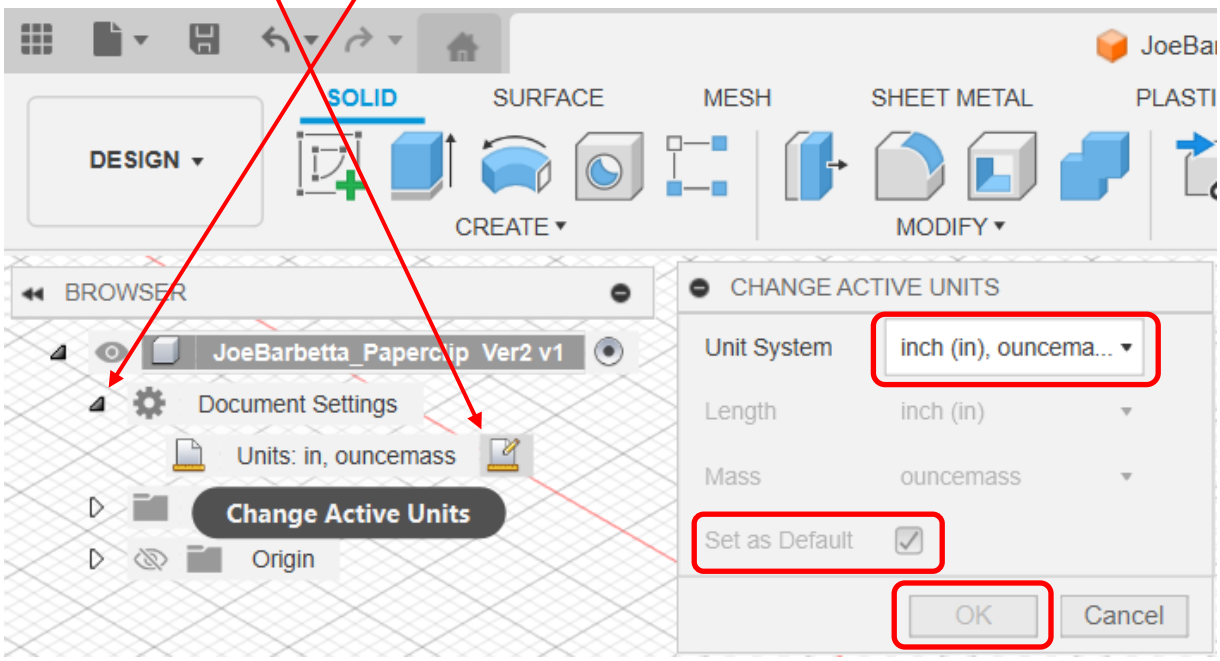
Starting a Design in Fusion

- 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 **_Paperclip** e.g. **JoeBarbetta_Paperclip** (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 360 crashes.

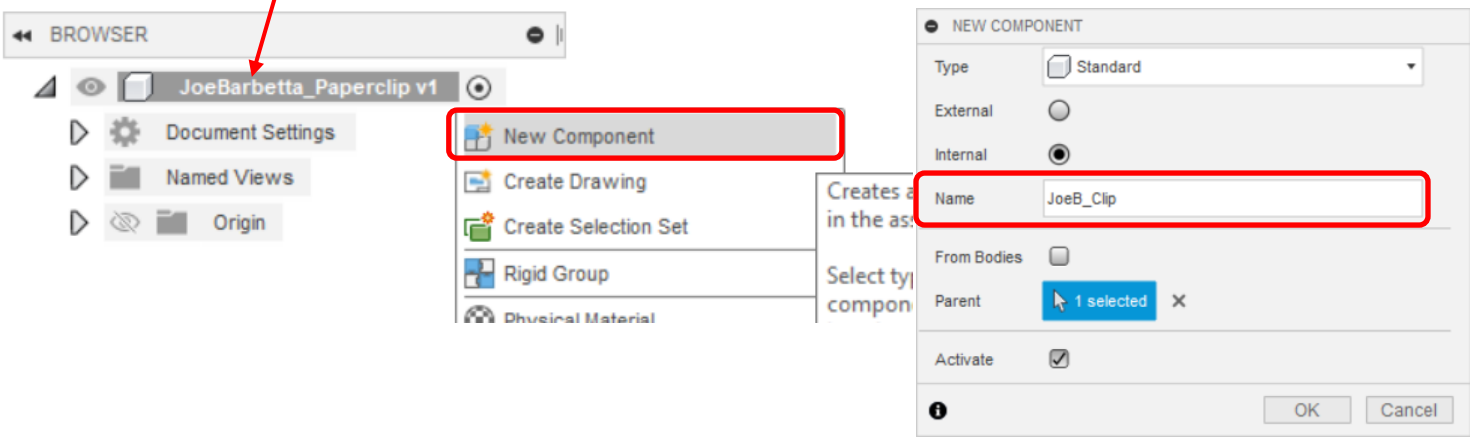
- in the left **"BROWSER"** click the **arrow next to Document Settings**
- click on the **edit icon** that appears to the right when you hover over **Units**
- ensure **Active Units** are set to **Units: in, ouncemass** and click **OK**. You can also enable **Set as Default** if it is not grayed out.



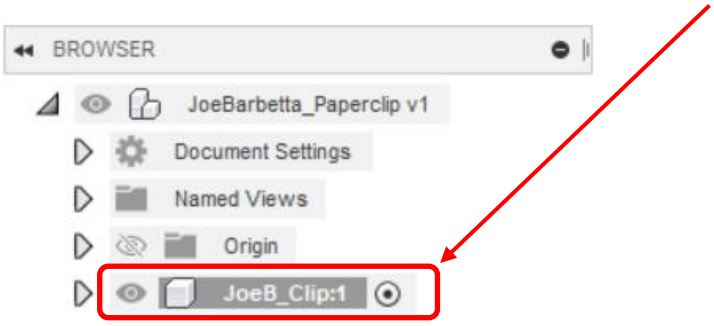
Creating a New Component

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 _Clip** e.g. **JoeB_Clip** and click **OK**

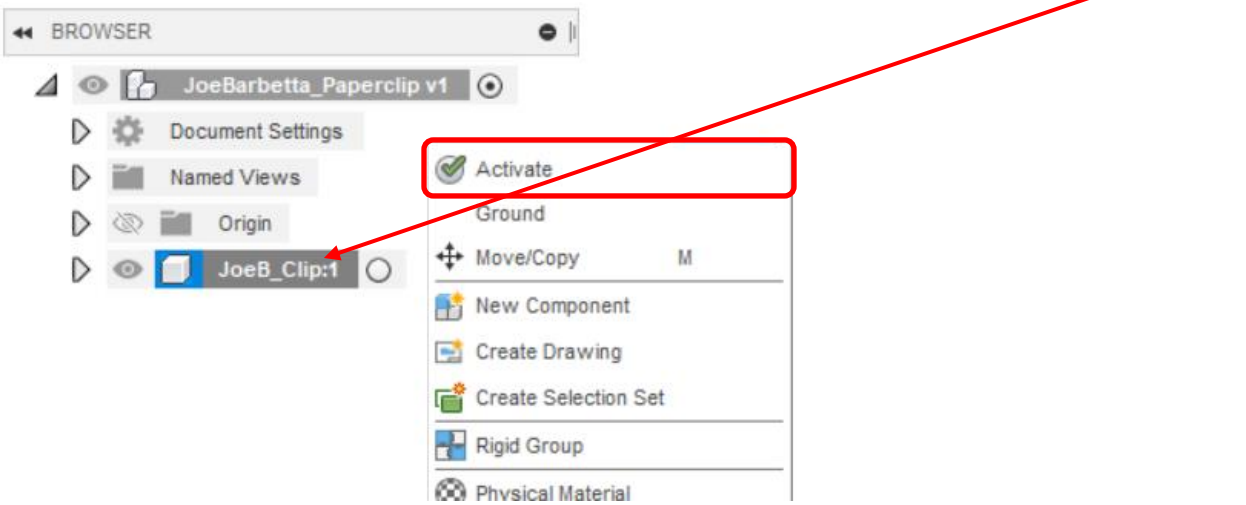


The new *Component* should show in the *Browser*.
Note that the area around the name is darkened and the circle with the dot. This indicates that it is the ***Active Component***.



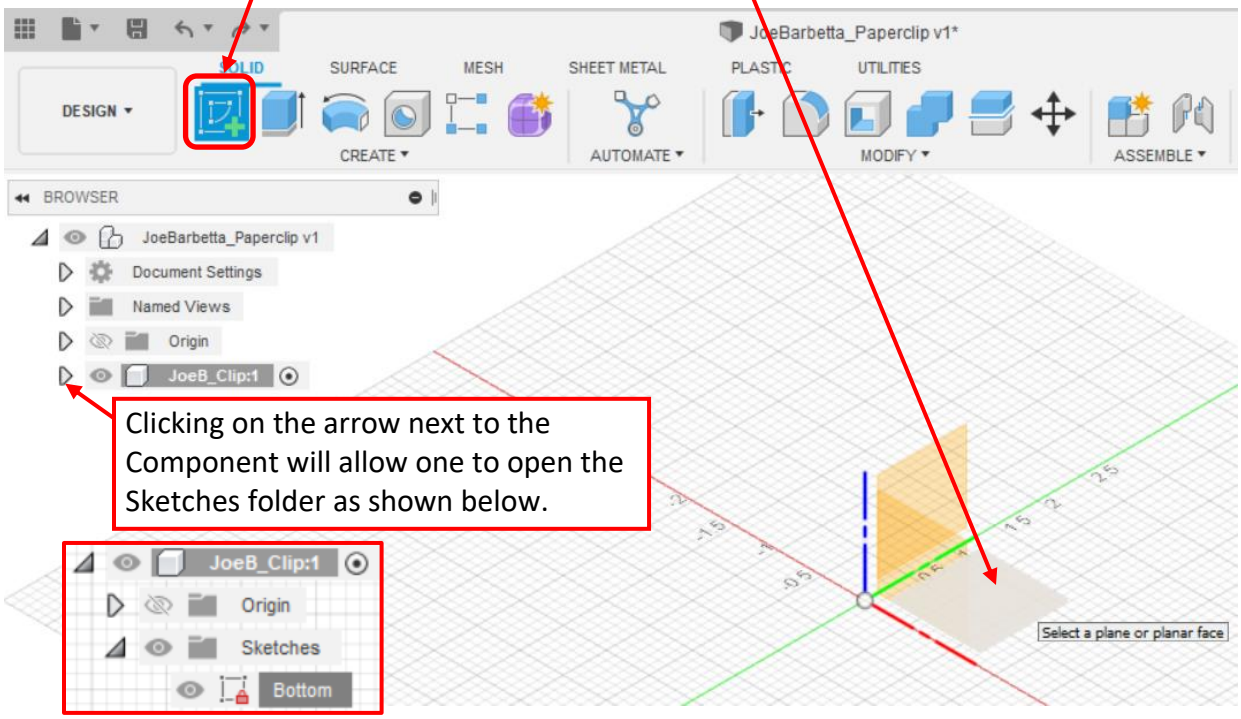
One reason to hate Fusion!

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.

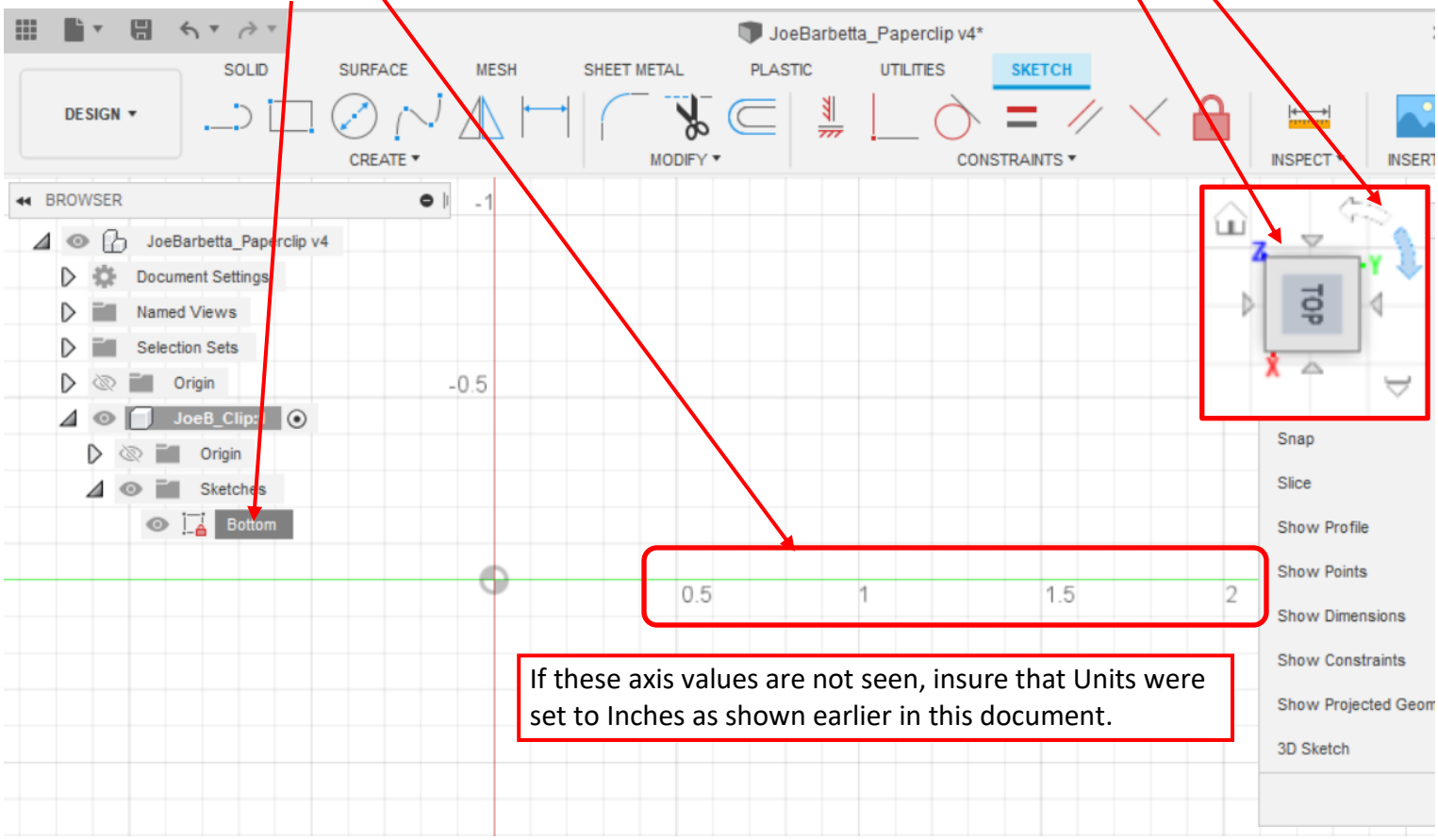


Creating the Bottom Sketch

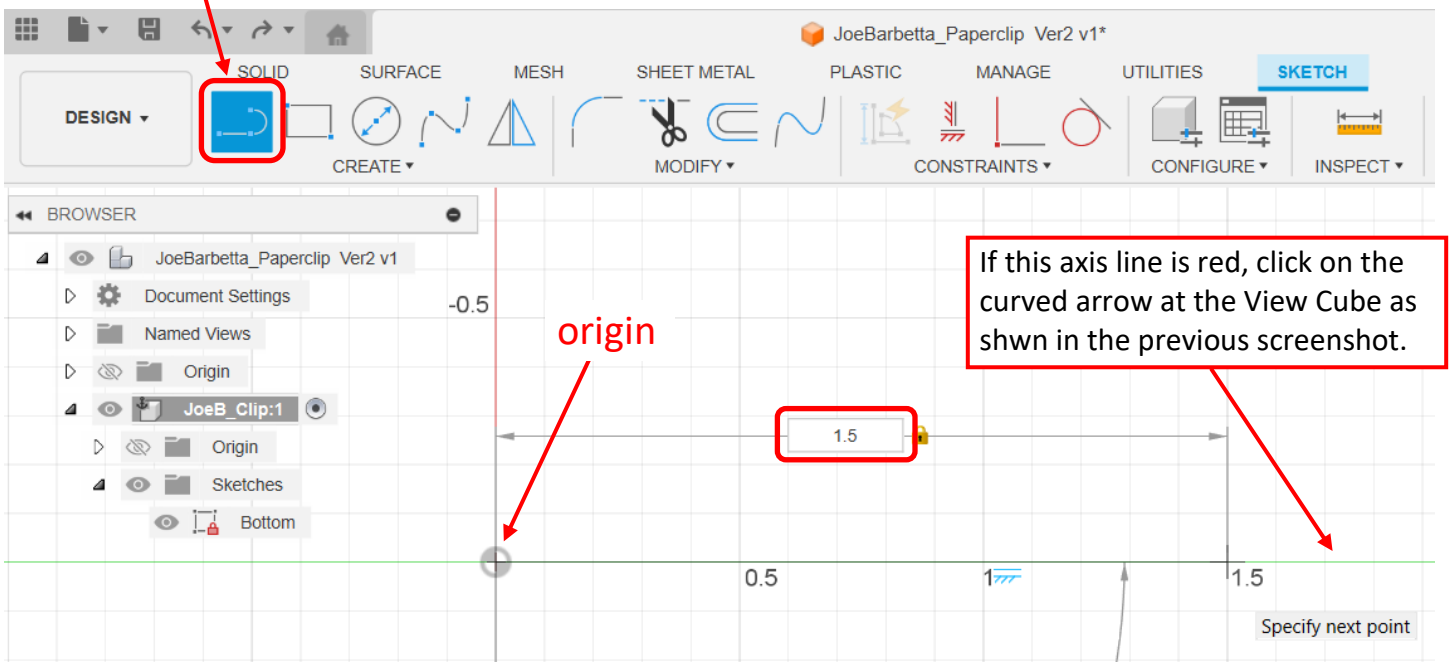
- select the top **Create Sketch** tool and click on the **bottom rhombus** to select the X-Y Plane.
If a tool can't be found, one can always look in the **CREATE** and **MODIFY** menus for it.



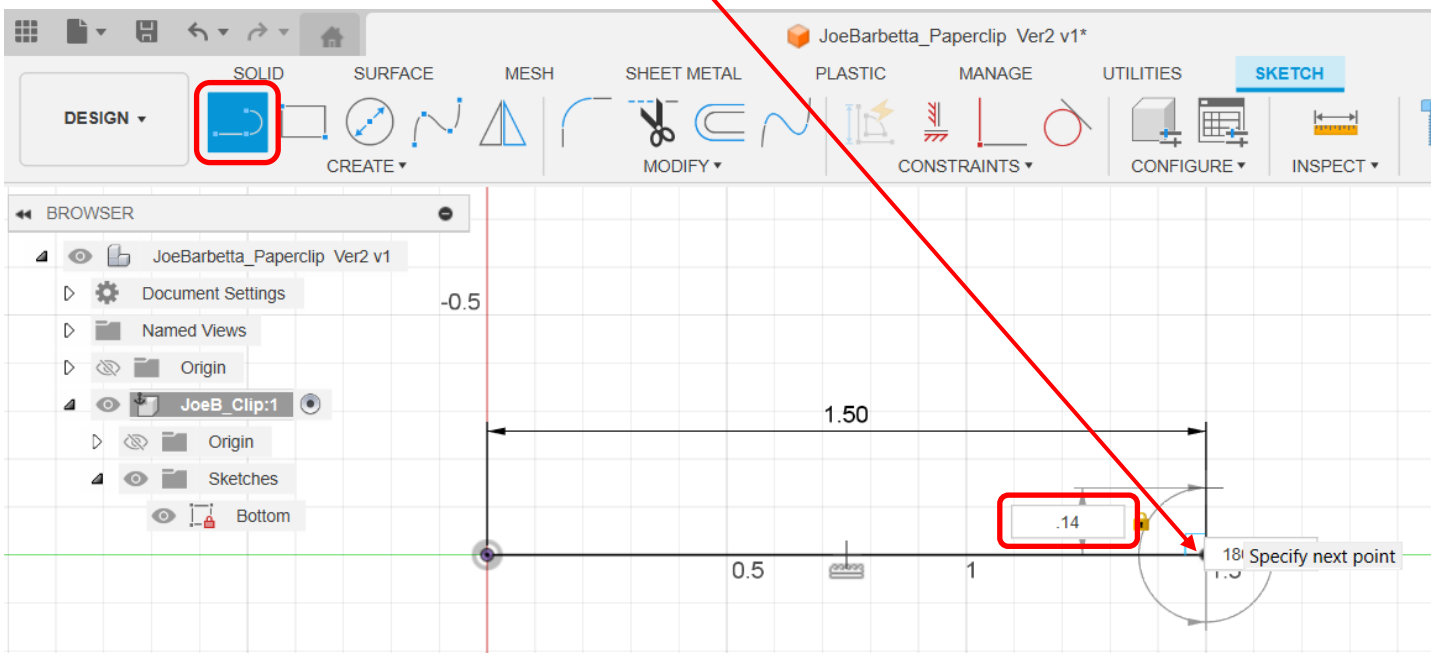
- **zoom in and pan** (hand icon) to achieve a view similar to below. You may have to use the **curved arrow** to rotate the view. If the curved arrows don't show, click on the center of the View Cube.
- note the **values on the green axis** as an indication of the desired view and the position of the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



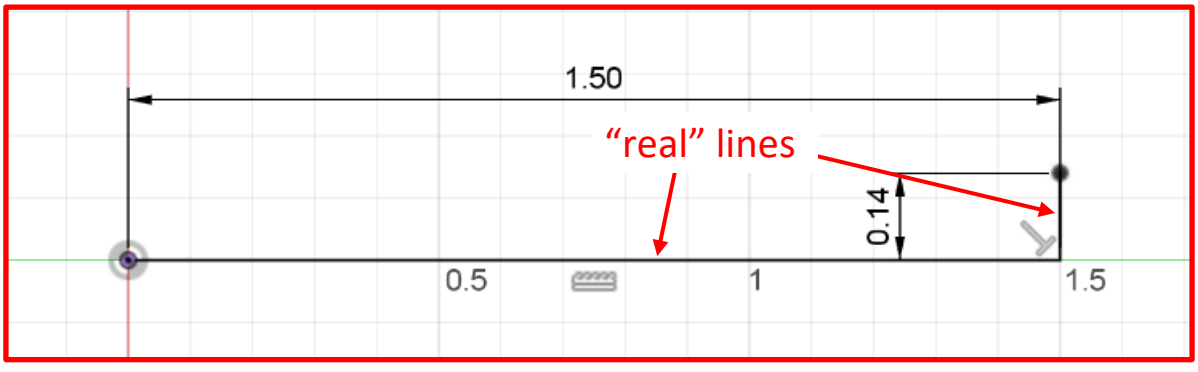
- click on the **Line** tool and then click on the **Origin** (where the green and red lines cross) to start the line.
- extend the line to the right (keeping it horizontal), type **1.5** and then the **Enter key**.



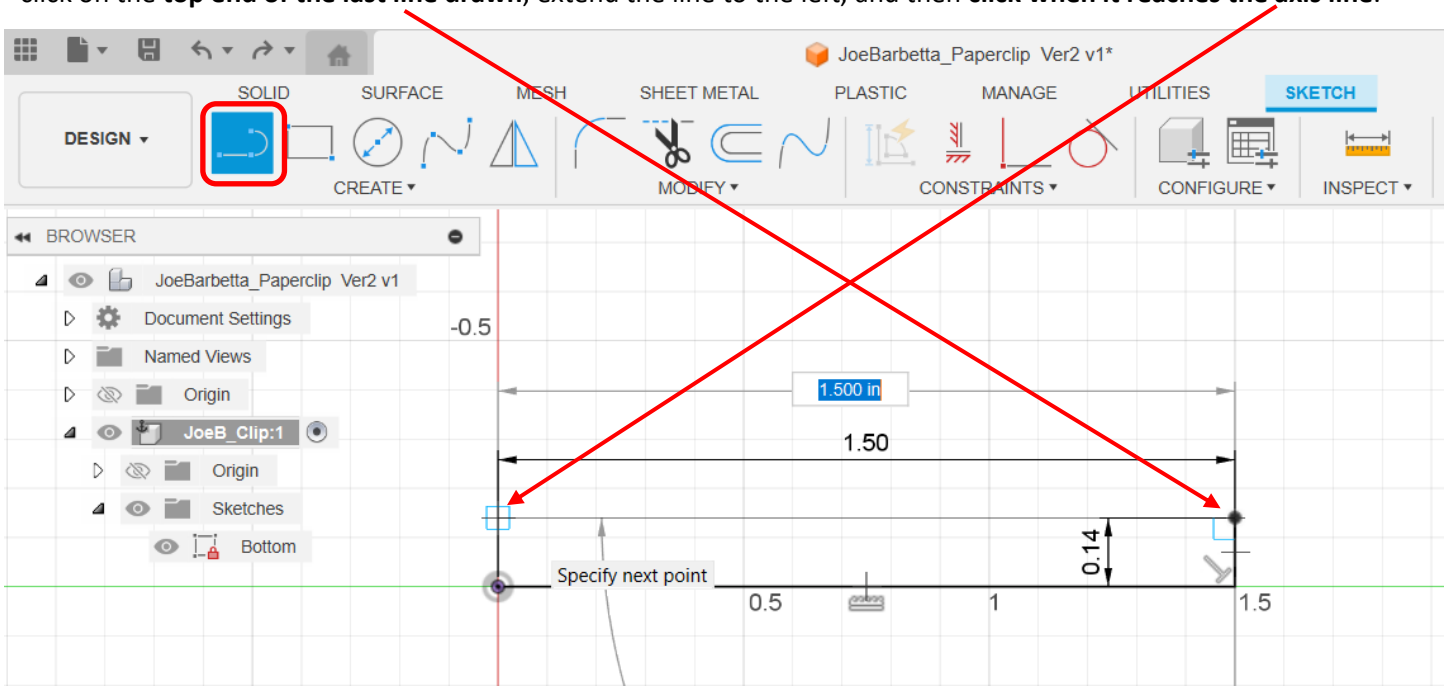
- click on the **Line** tool again and then **click on the end of the line** just drawn
- extend this line up (keeping it vertical), type **0.14** and then the **Enter key**.



At this point the **thicker lines** are those that were just created. The **thinner lines** are just dimension lines.

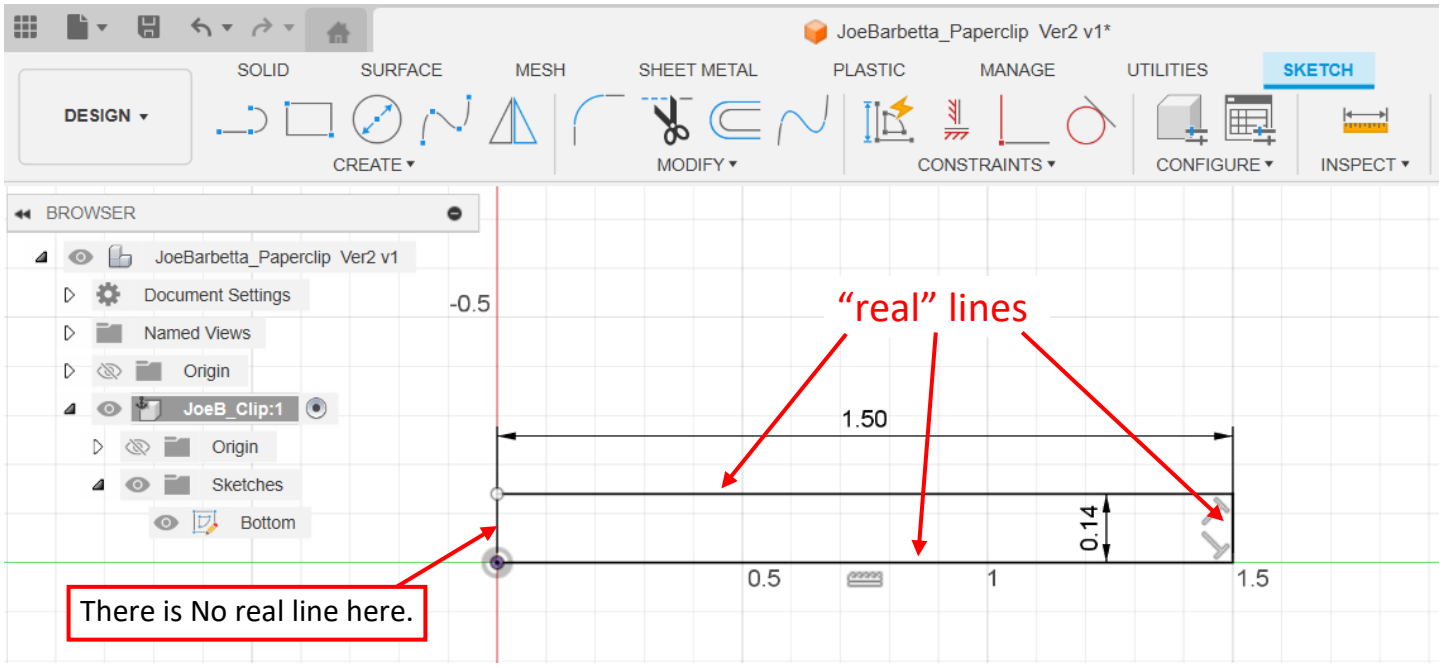


- click on the **Line** tool again. The Line tool is your bestie in Fusion.
- click on the **top end of the last line drawn**, extend the line to the left, and then **click when it reaches the axis line**.



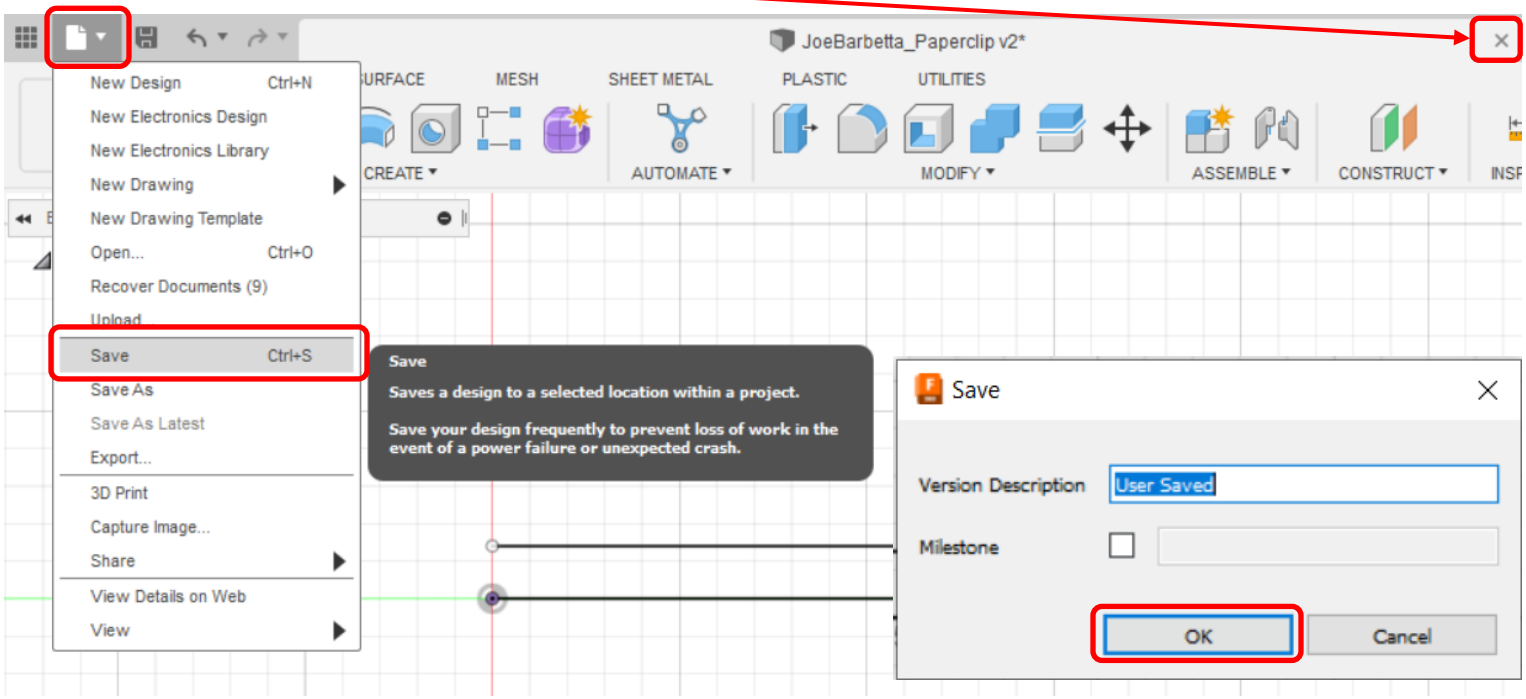
The result should appear as below. Note that **Dimension Lines** may appear in different locations.

At this point only 3 real lines have been created.



Saving and Reopening a Project

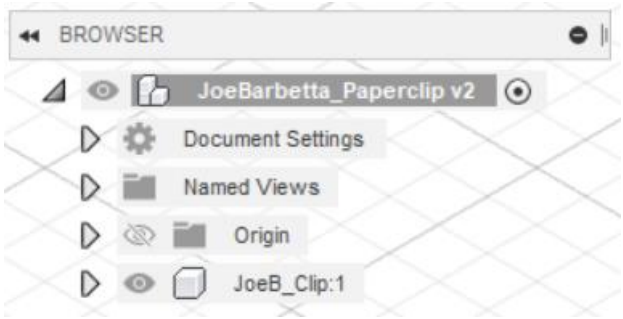
- click on the **File** icon and select **Save**. When the Save window shows click **OK**. You can keep the default Version Description.
- close the Project by clicking the **X** at the right of the **Project tab**.

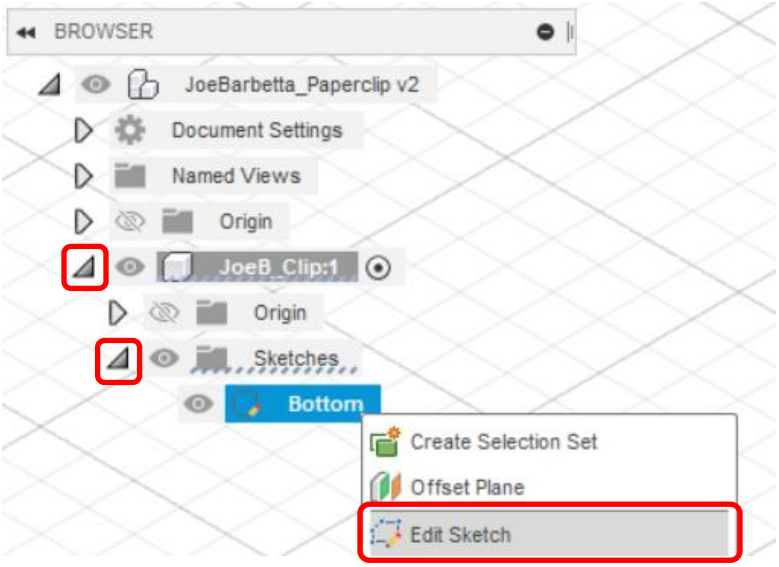


Select **"Open…"** from the top **File** icon and select the fle you just saved.

Note that when the Project is reopened, the BROWSER will be collapsed and the *Component* last worked on will **Not be Active**.

- right-click on the **Component** last worked on and select **Activate** or click on the **Circle** next to it.
- click on the **Expand Arrow** next to the **Component** and then that next to the **Sketches** folder
- right-click on the **Sketch** last worked on and select **Edit Sketch**

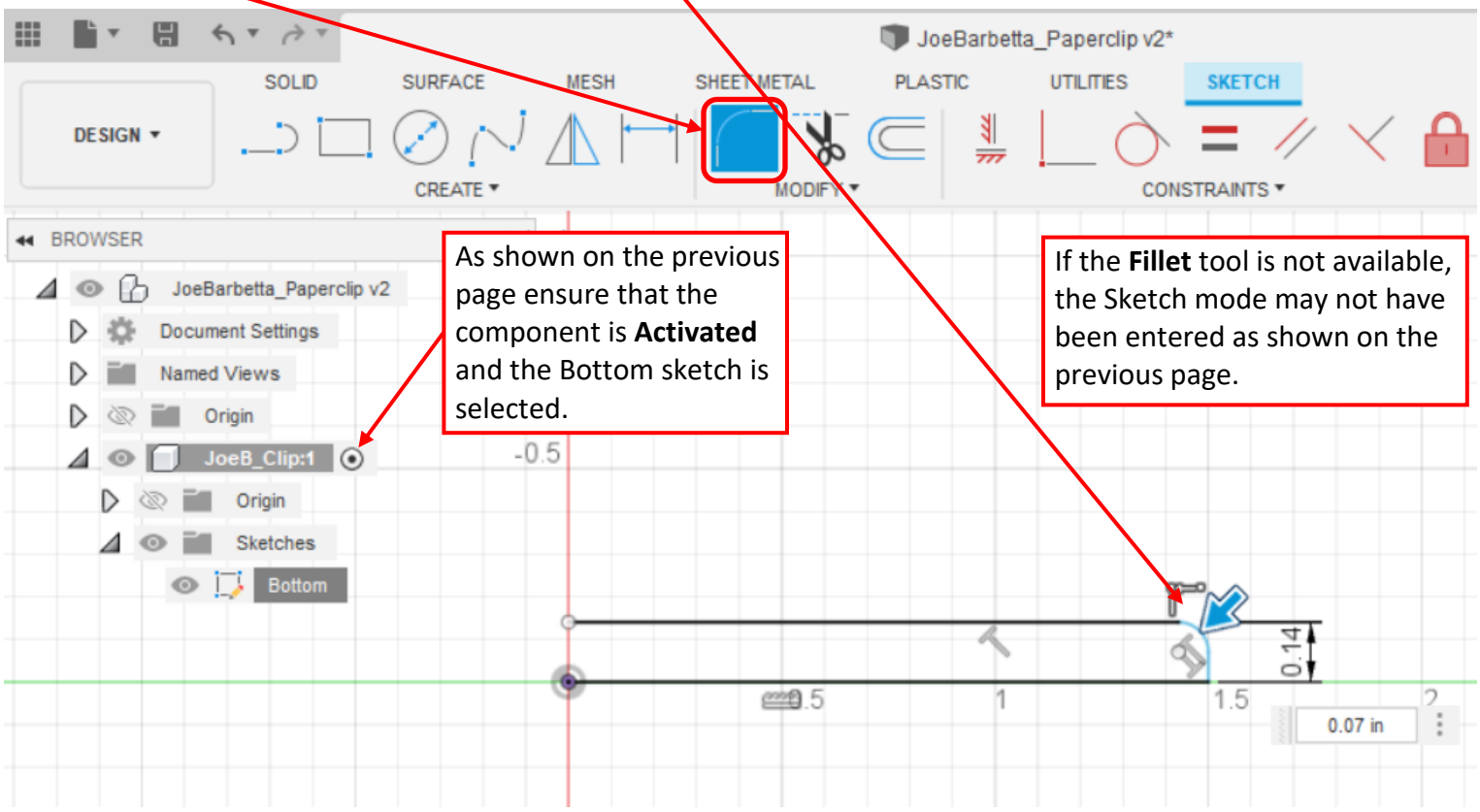




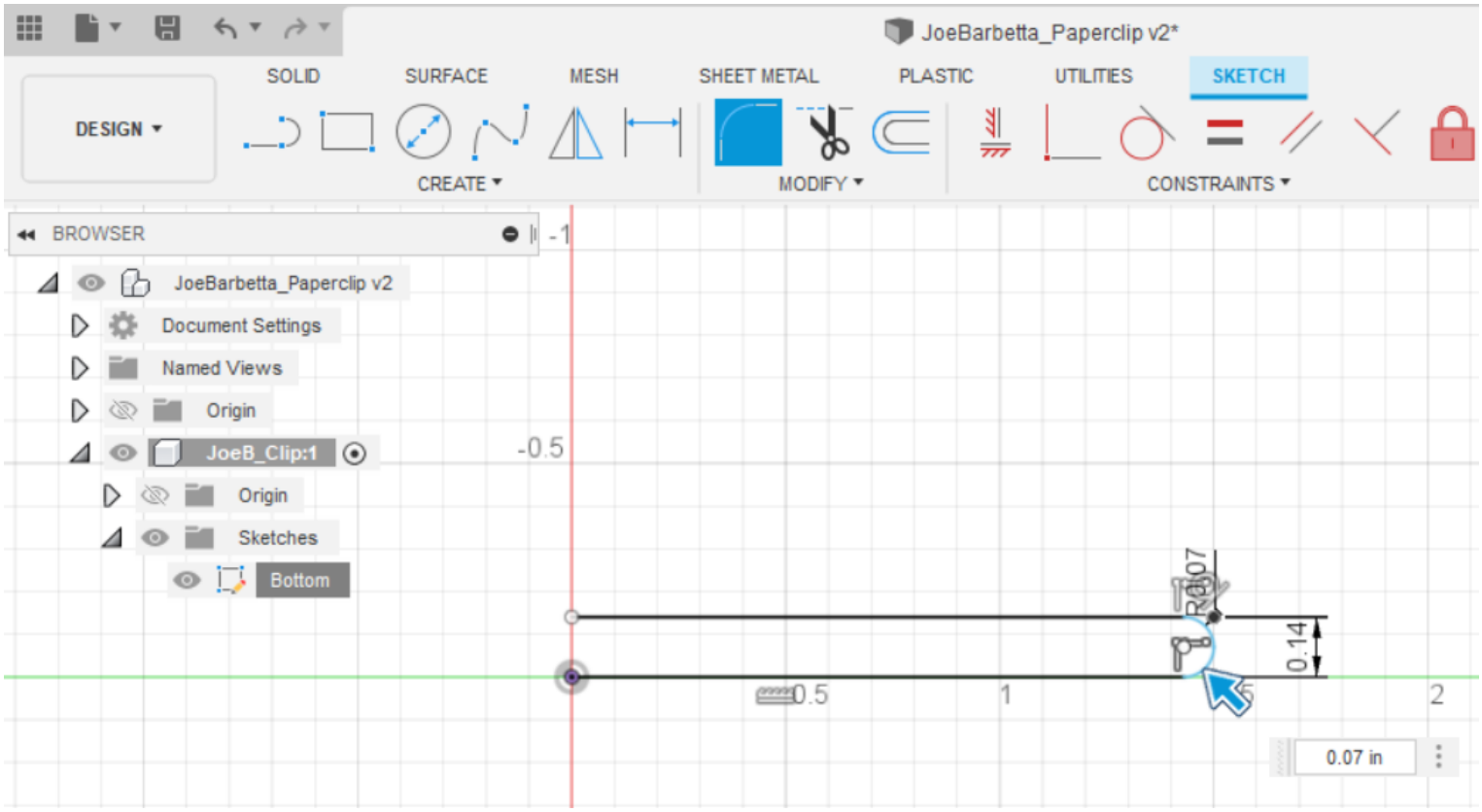
It is important to use **Edit Sketch** so that you continue to work on the same Sketch that was not yet completed.

Using the Sketch Fillet Tool

- click on the **Fillet** tool and then click on the **top right corner** of the intersecting lines and enter **0.07**

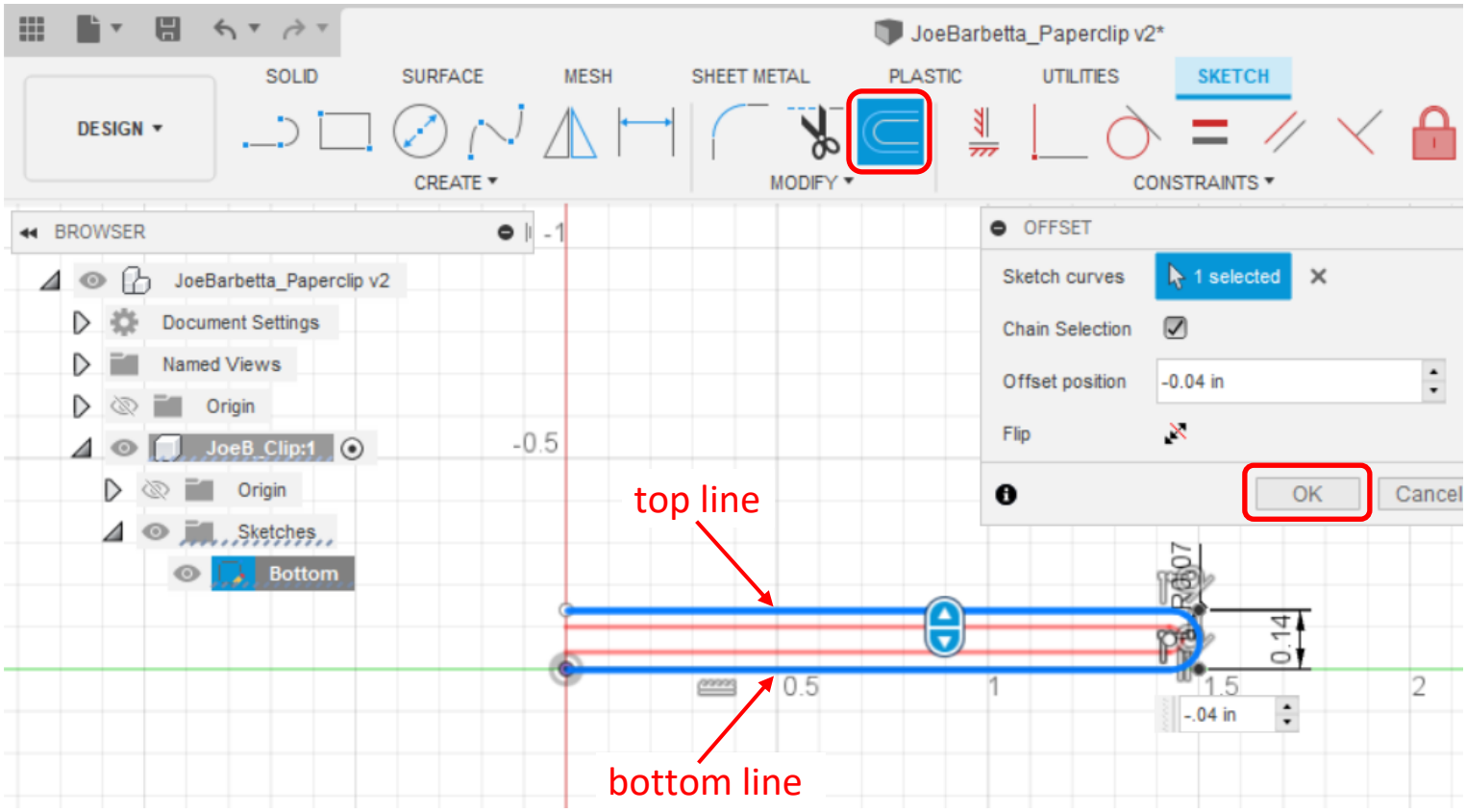


- do the same for the bottom right corner

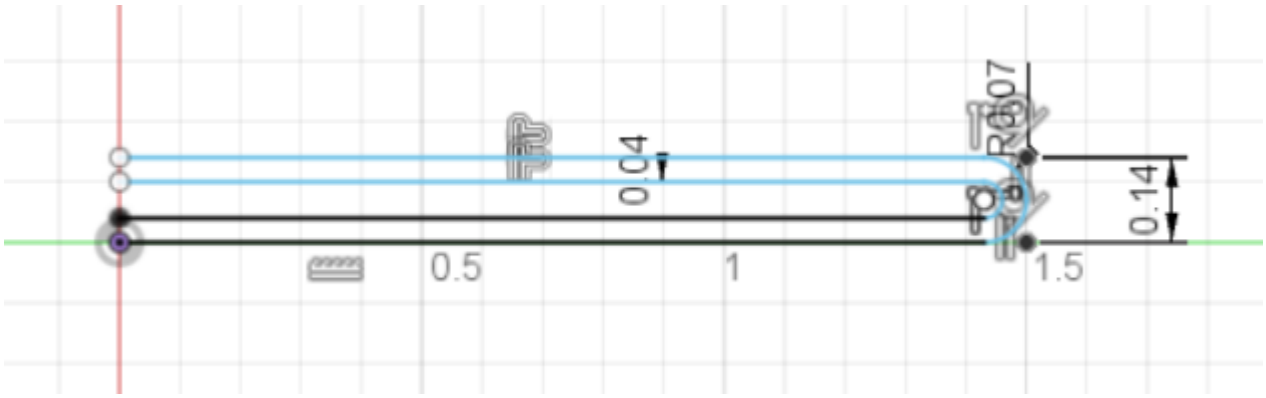


Using the Offset Tool

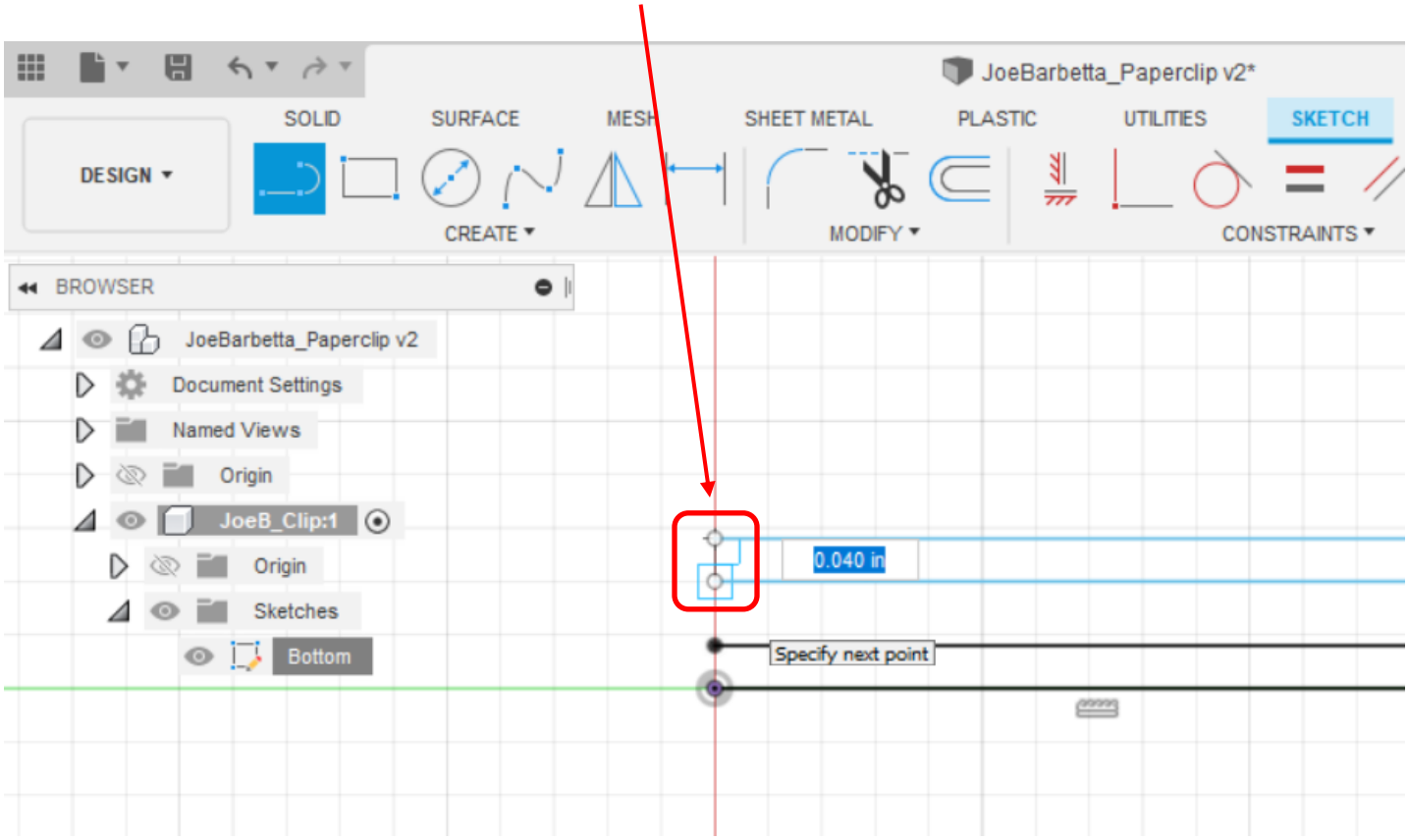
- click on the **Offset** tool and then the **top and bottom lines**, which should make them turn dark blue.
-Enter **-0.04** (note the minus sign). The red lines will first be outside of the blue lines, but upon entering the negative value the **red lines should be inside the blue lines** as shown below.
- click **OK**



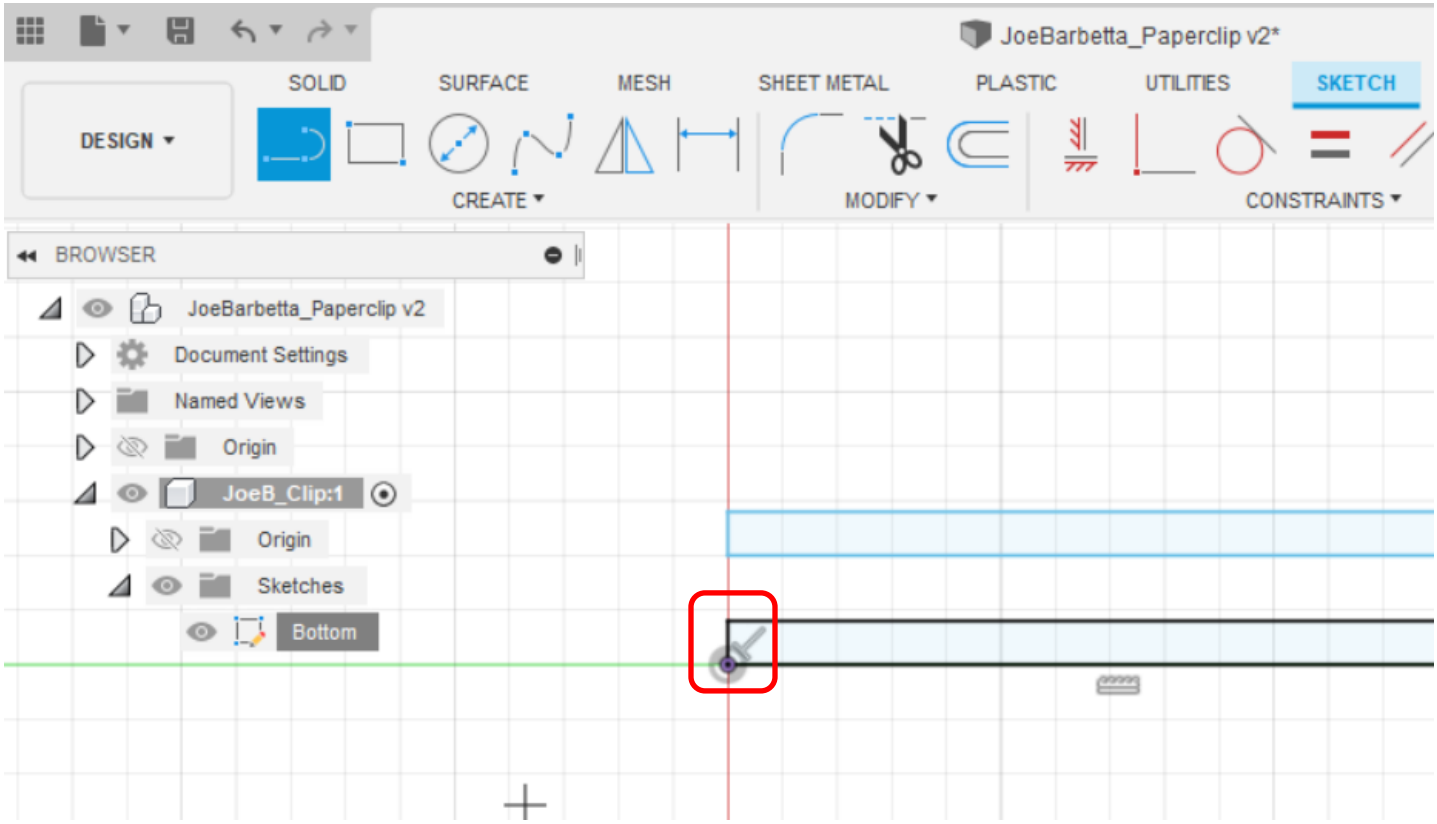
Ignore any annoying *Constraint* symbols and the fact that lines **may be blue or black** at this point.



- click on the **Line** Tool and create a line **between the two end points**

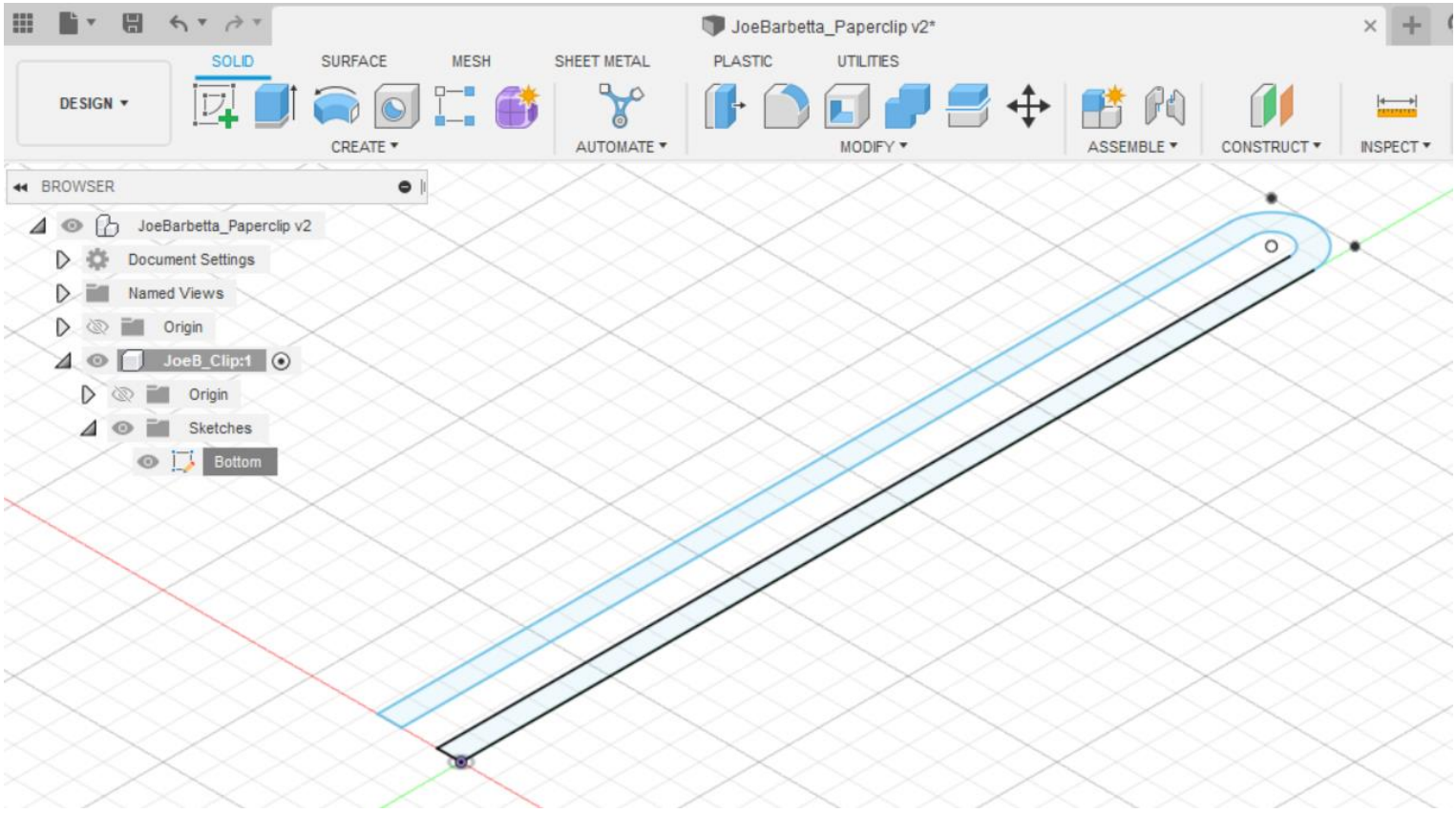


- connect the **lower end points** as well and click **Finish Sketch**. The **interior should turn light blue**, which indicates that we now have a ***Profile***. A *Profile* is a closed *Sketch Object* that can be *Extruded*.

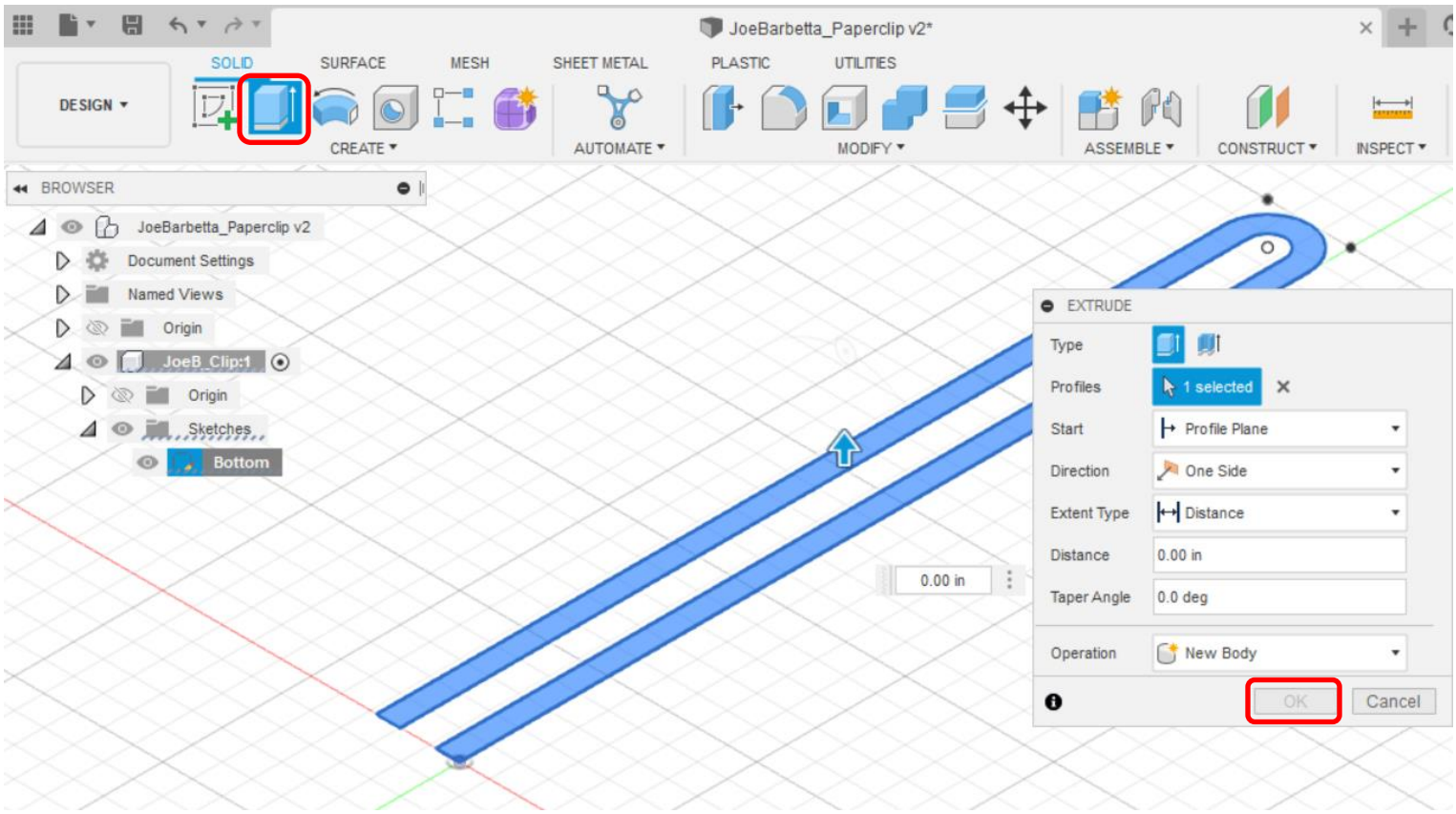


Extruding a Sketch Profile

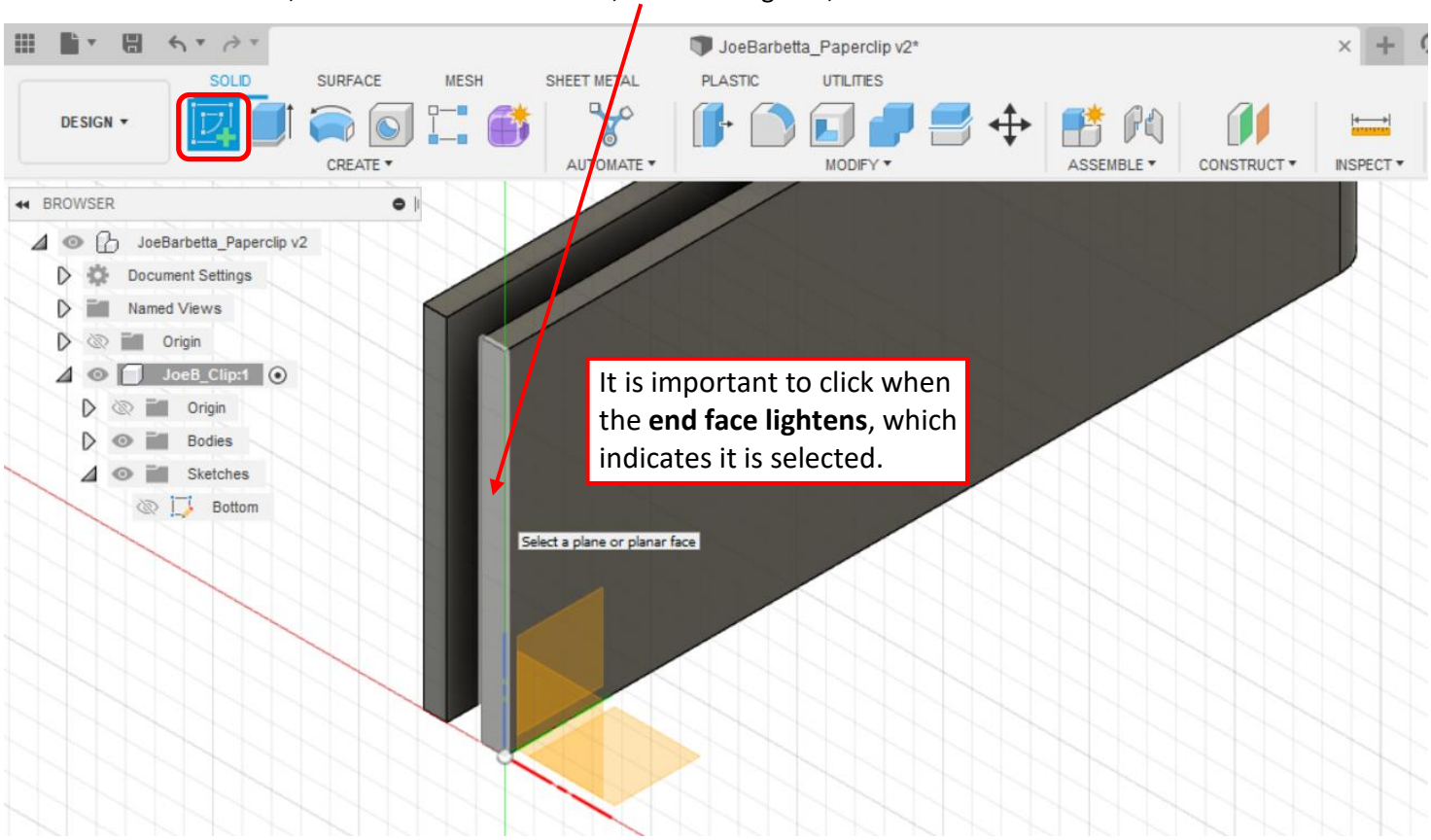
- click the **Home Icon** at the upper left of the **View Cube** and the Profile should look as below.



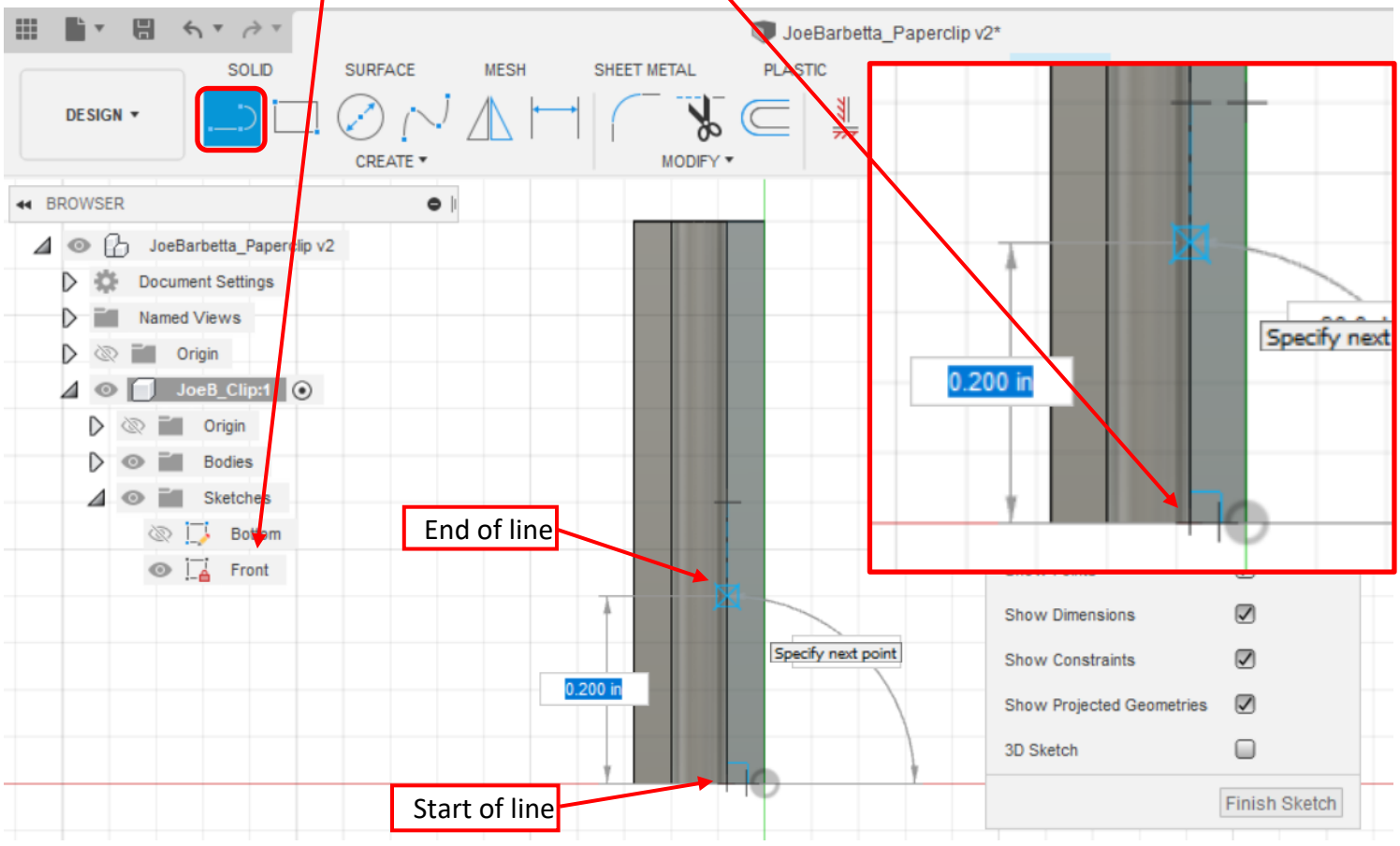
- click on the **Extrude** Tool and enter **0.6** and click **OK**. The result should be a dark gray Body.



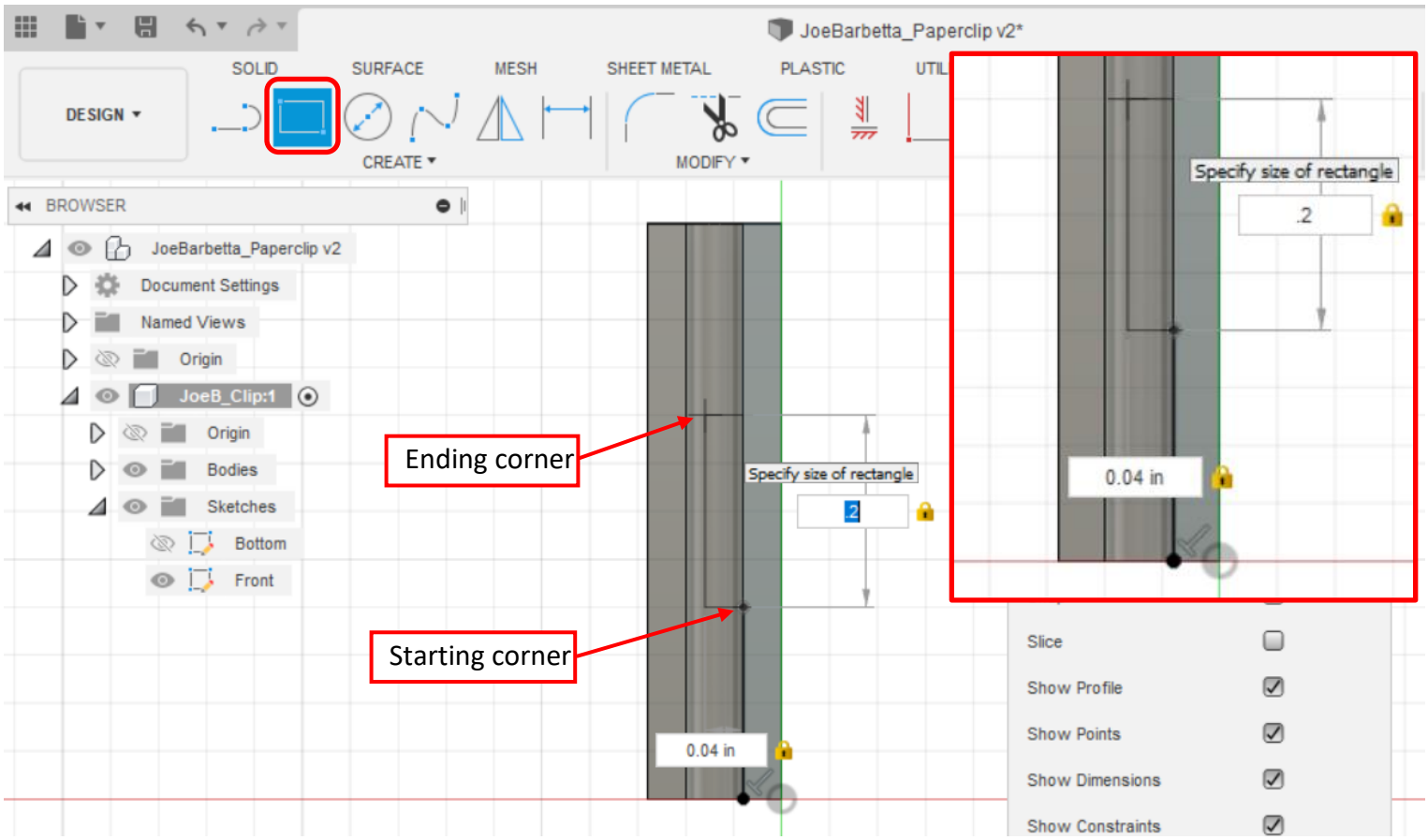
- zoom to get a view similar to below.
- click on the **Sketch** tool, hover over the **face** shown, which will lighten, and click.



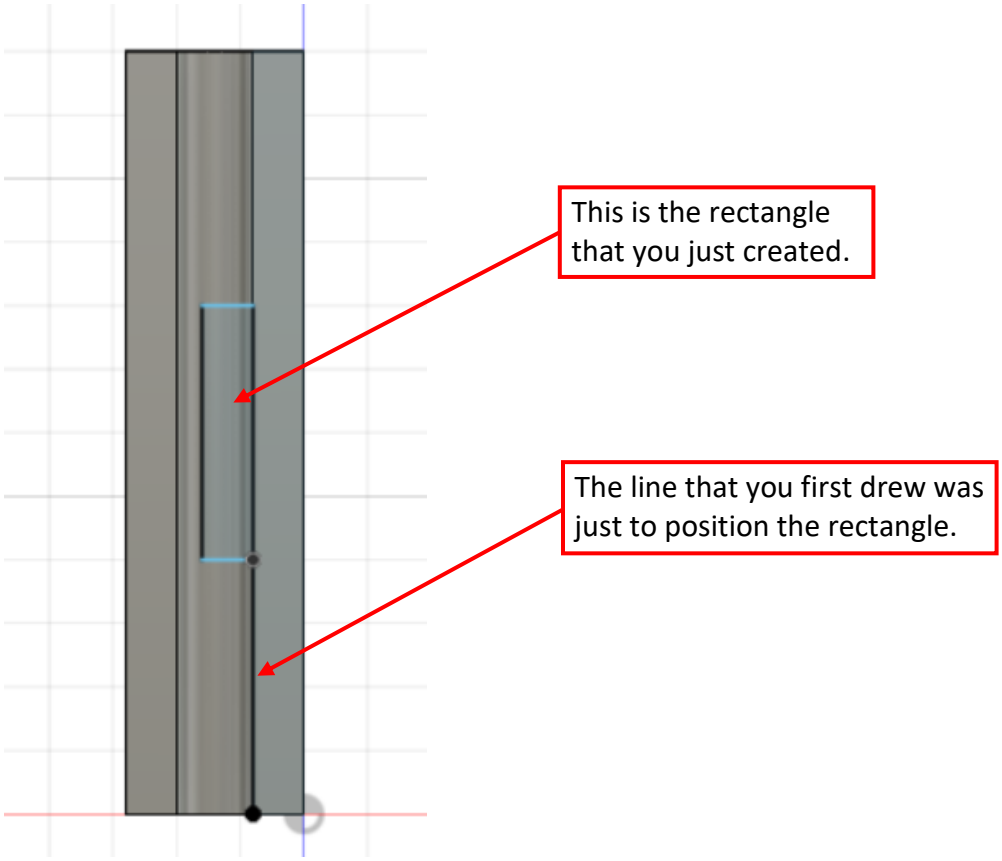
- right-click on the **new Sketch name** and select **Rename** and name the Sketch to **Front**
- click on the **Line** tool and create a line from the **bottom left corner** of the right body section and up by **0.2**



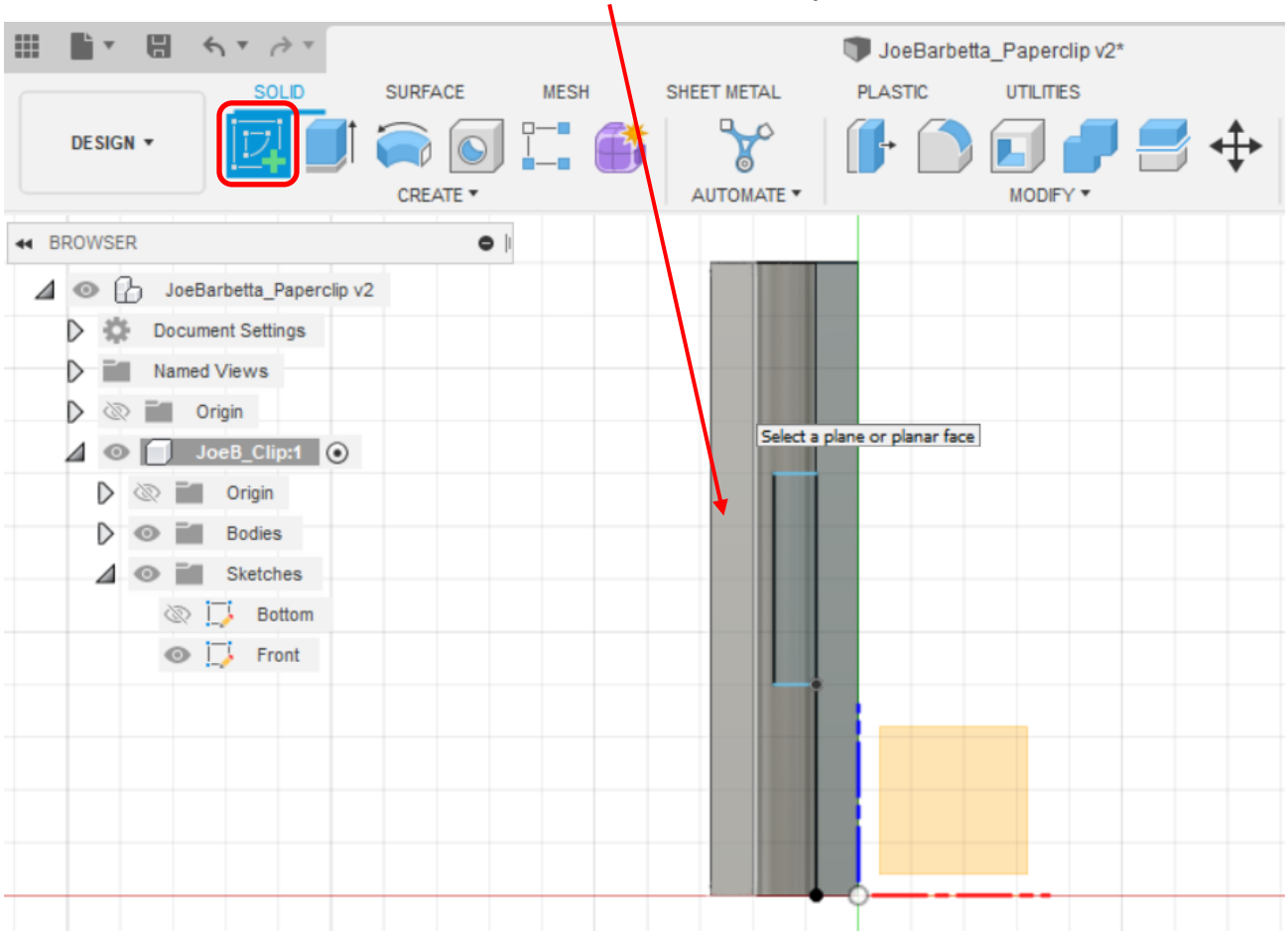
- click on the **Rectangle** tool and click on the **top point** of the line just created. Enter **0.04** for the width and **0.2** for the height by using the **Tab key** to switch between the two dimensions.



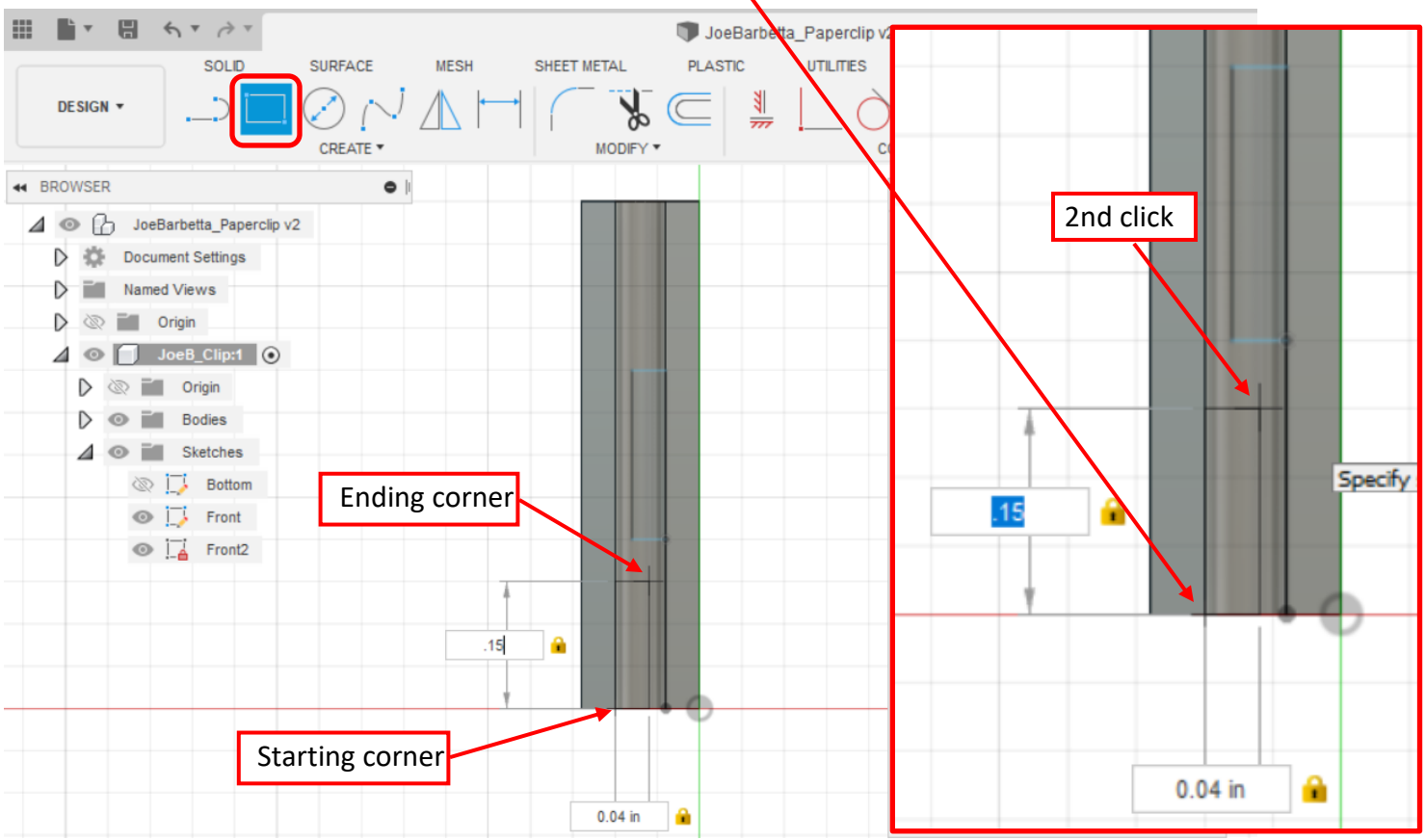
- click **Finish Sketch** and the front should look lake that below.



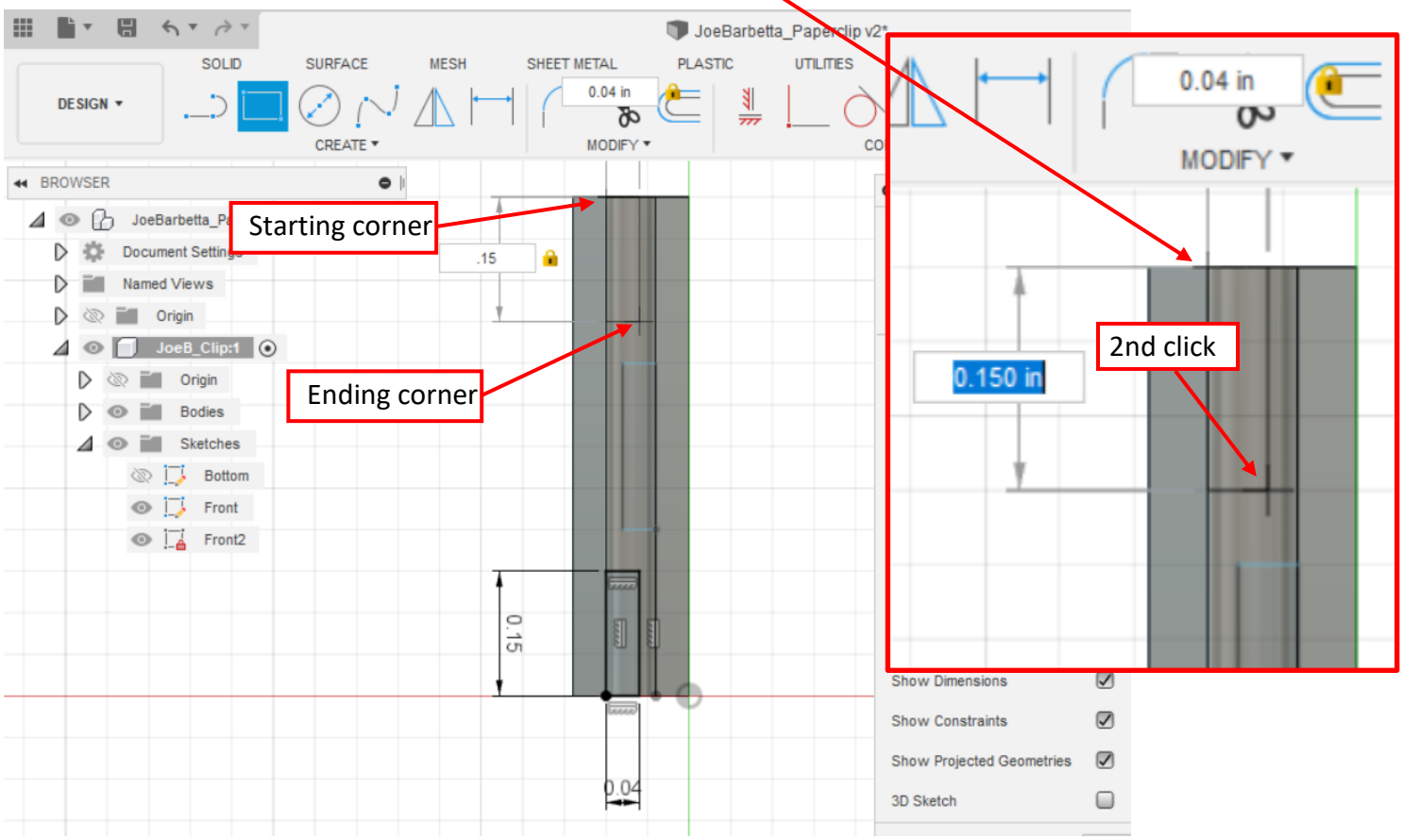
- click on the **Create Sketch** tool and click on the **face of the left body section**



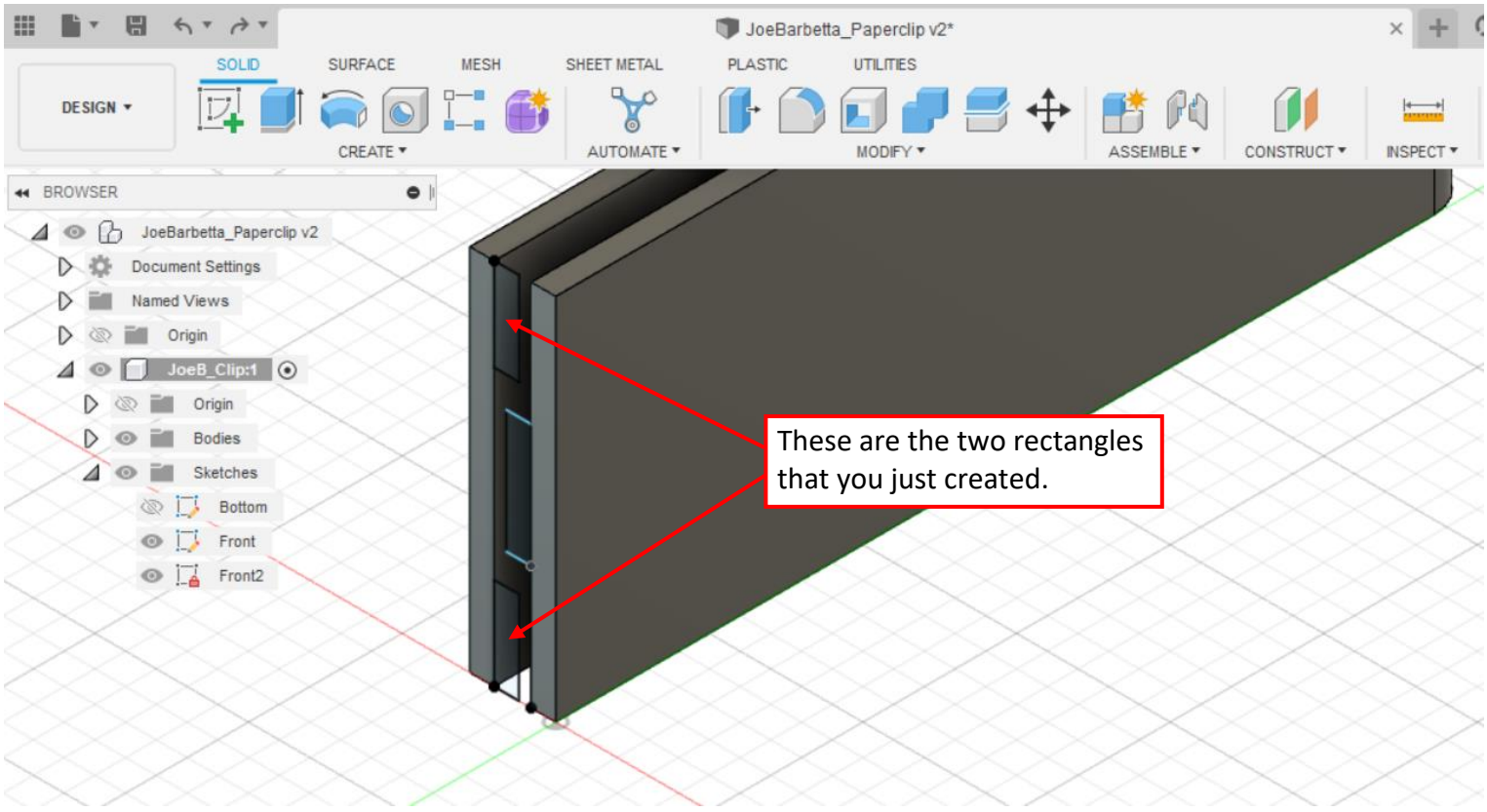
- click on the **Rectangle** tool and click on the **bottom right corner** of the left body section. Enter **0.04 for width** and **0.15 for height** using the **Tab key** to switch between the two dimensions if needed.



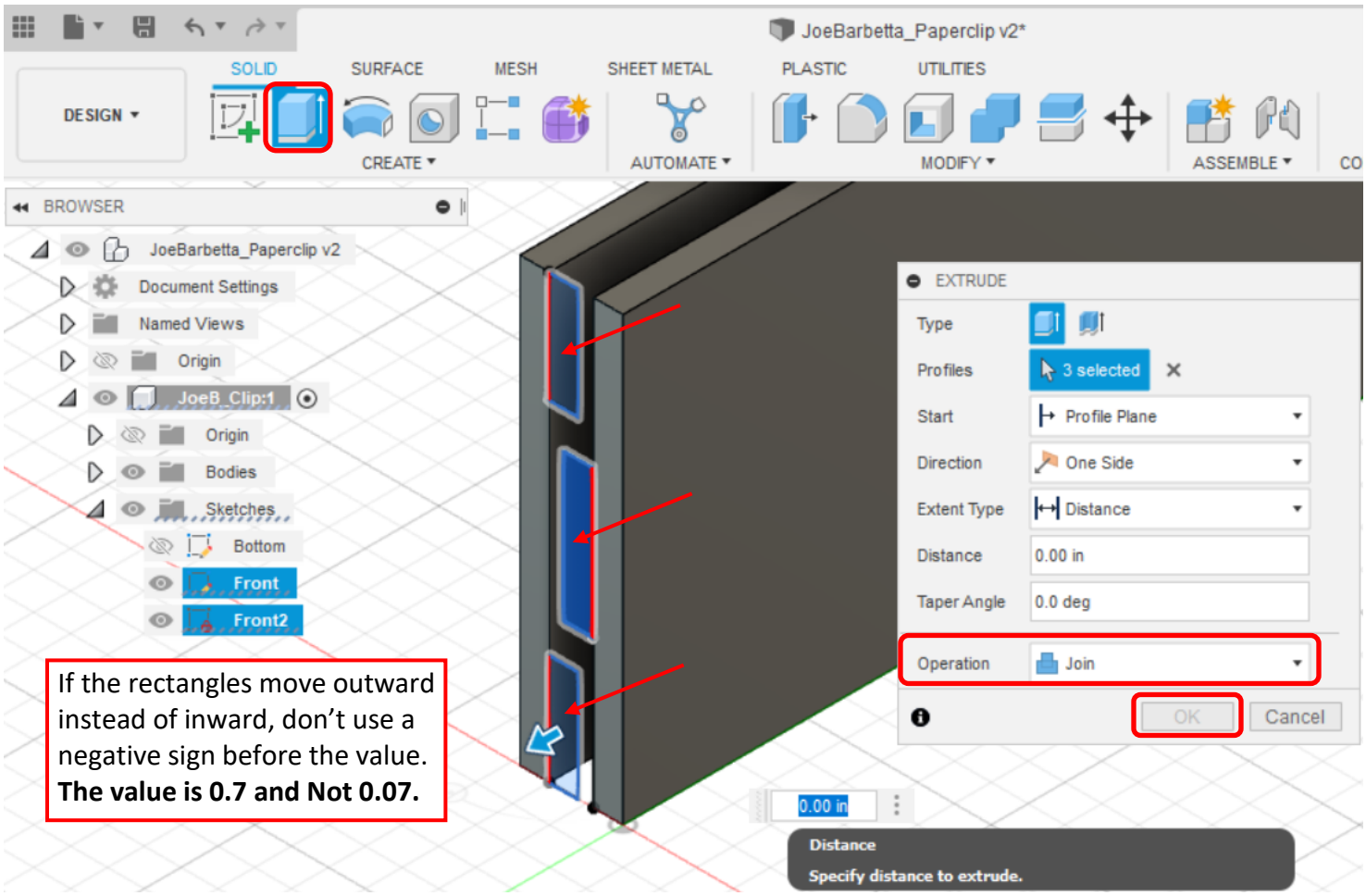
- click on the **Rectangle** tool again and click on the **top right corner** of the left body section. Enter **0.04 for width** and **0.15 for height** using the **Tab key** to switch between the two dimensions if needed.



- click **Finish Sketch**

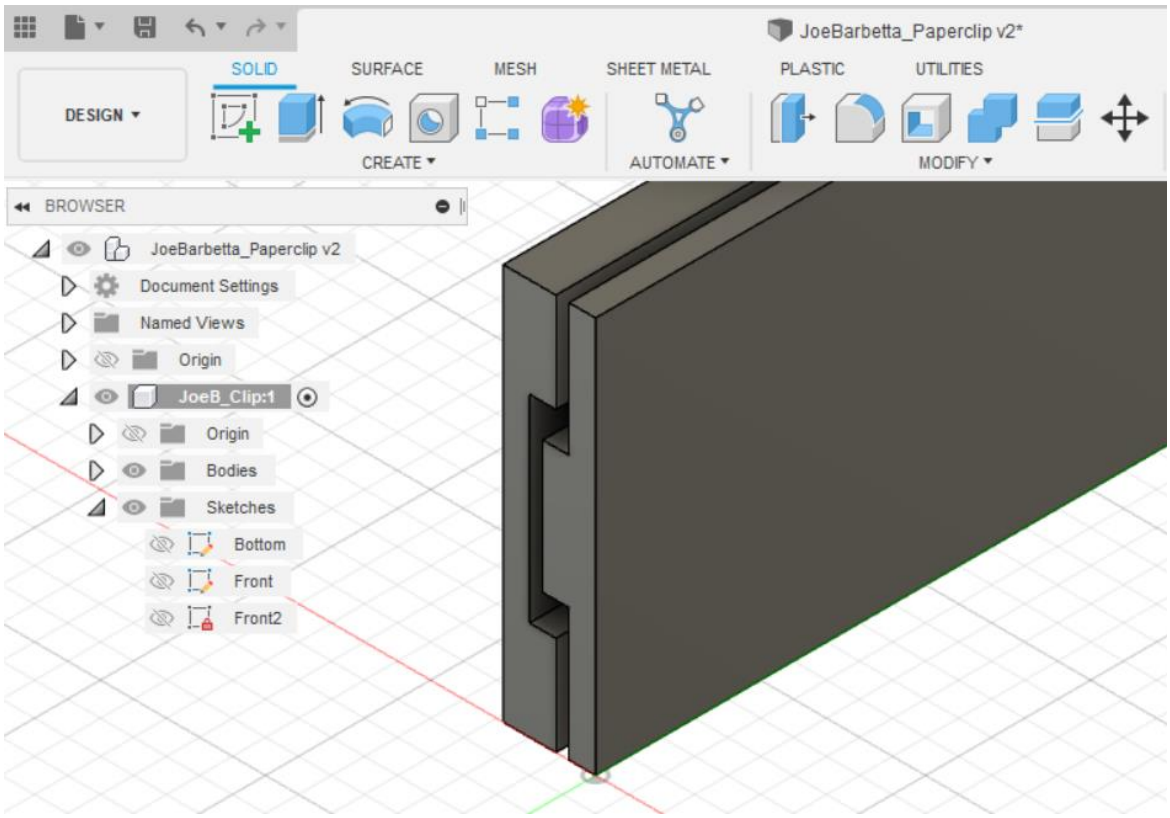


- click on the **Extrude** tool and click on the **three rectangles** just created
- ensure the **Operation** is set to **Join**. Enter **-0.7** (note the minus sign) and click **OK**.



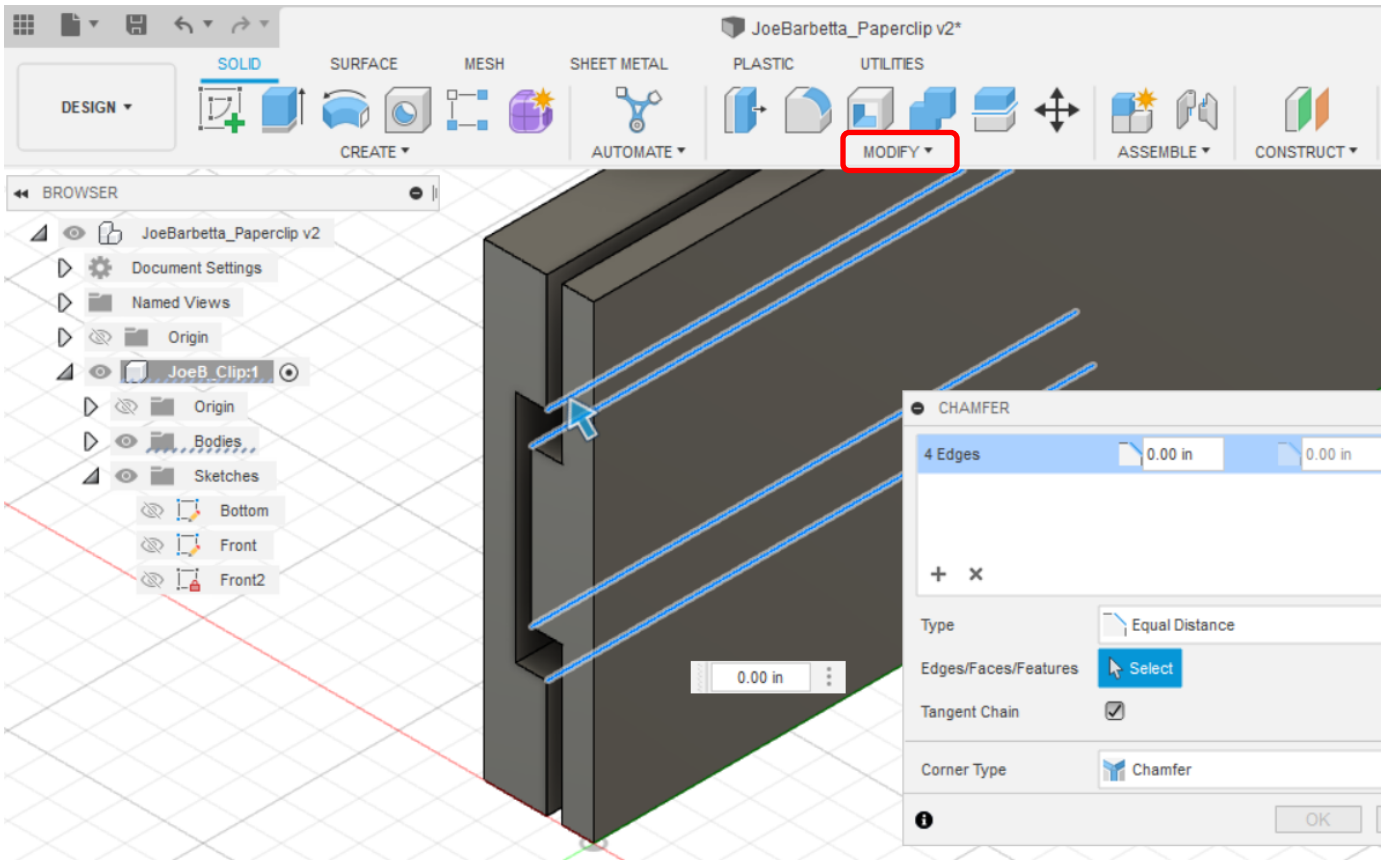
If the rectangles move outward instead of inward, don't use a negative sign before the value. **The value is 0.7 and Not 0.07.**

The result should look like that below.

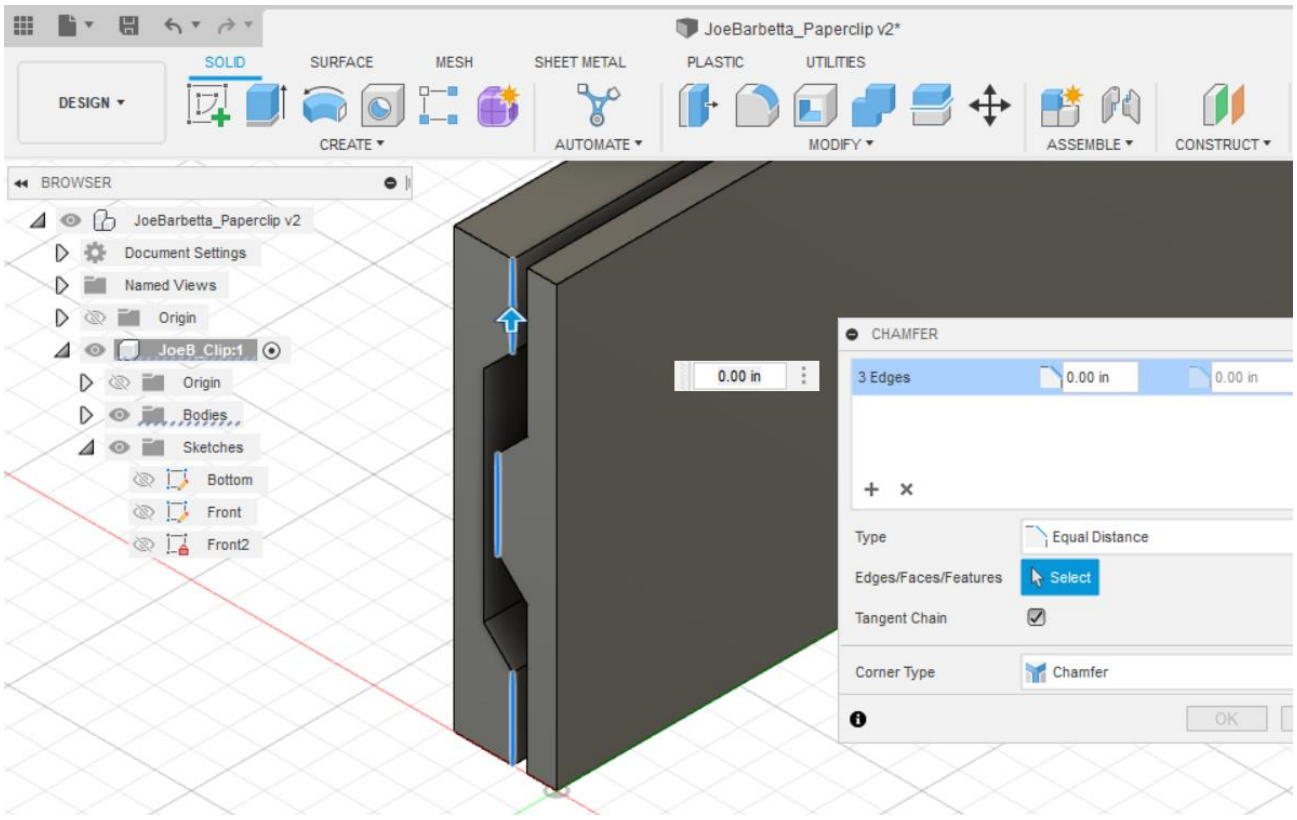


Using the Chamfer Tool

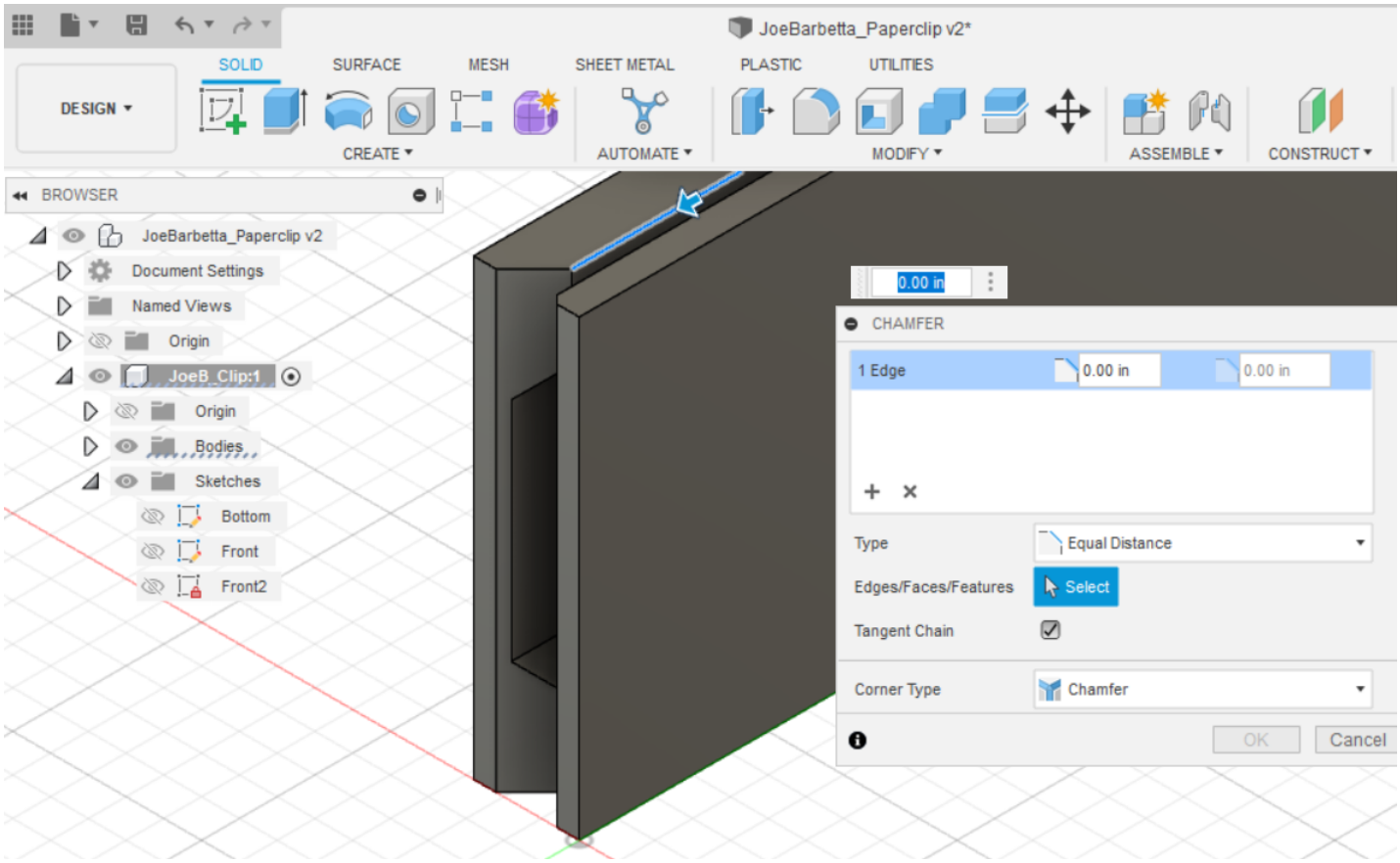
- select the **Chamfer** tool from the **MODIFY** pull-down menu and carefully click on the **edges** as shown, which will turn them blue. One can click on them "through the body". Enter **0.039** and click **OK**.



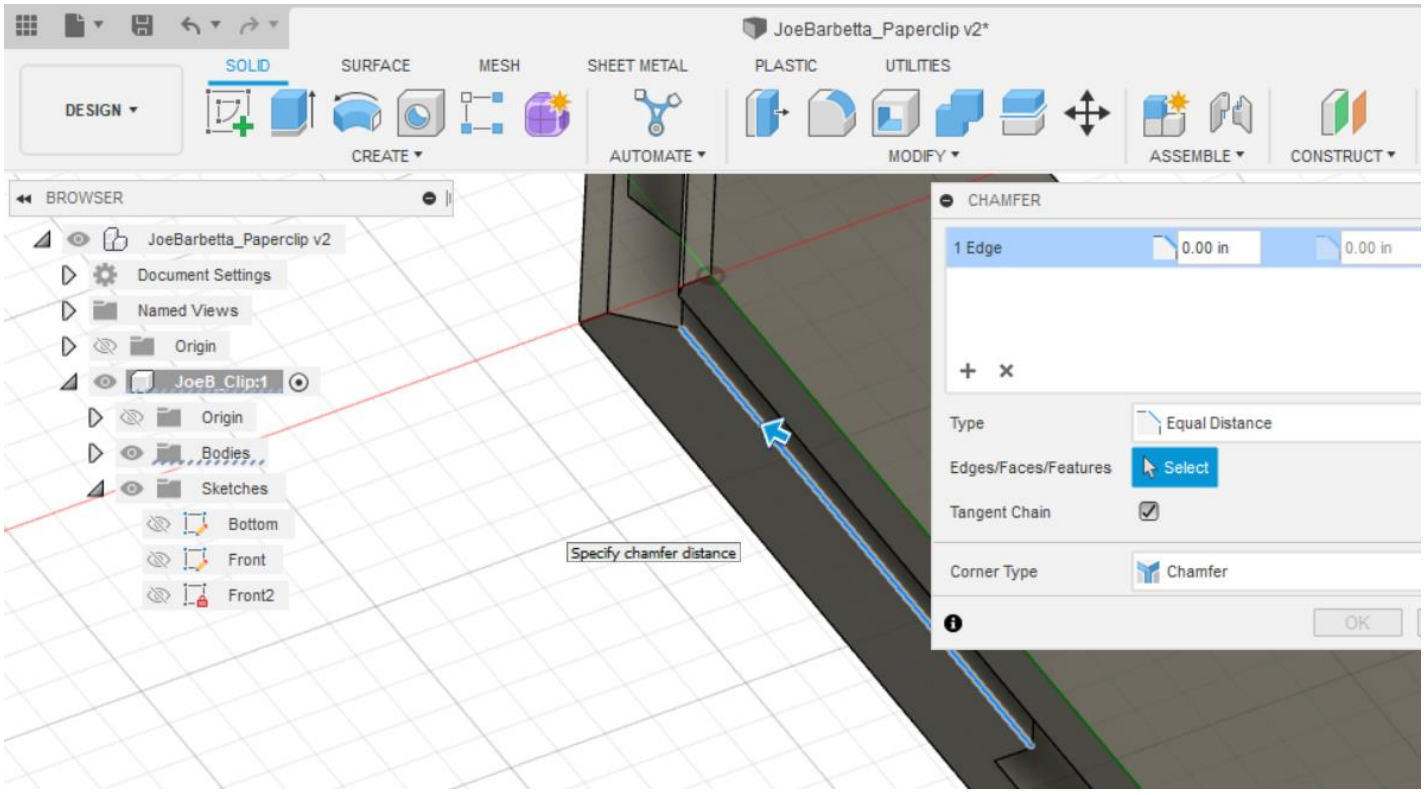
- select the **Chamfer** tool again and click on the **3 edges** as shown, enter **0.06**, and click **OK**.



- select the **Chamfer** tool again and click on the **top edge** as shown, enter **0.04**, and click **OK**.



- rotate the View to access the bottom edge as shown below.
- select the **Chamfer** tool again and click on the **bottom edge** as shown, enter **0.04**, and click **OK**.



3D Printing 45 Degree Rule

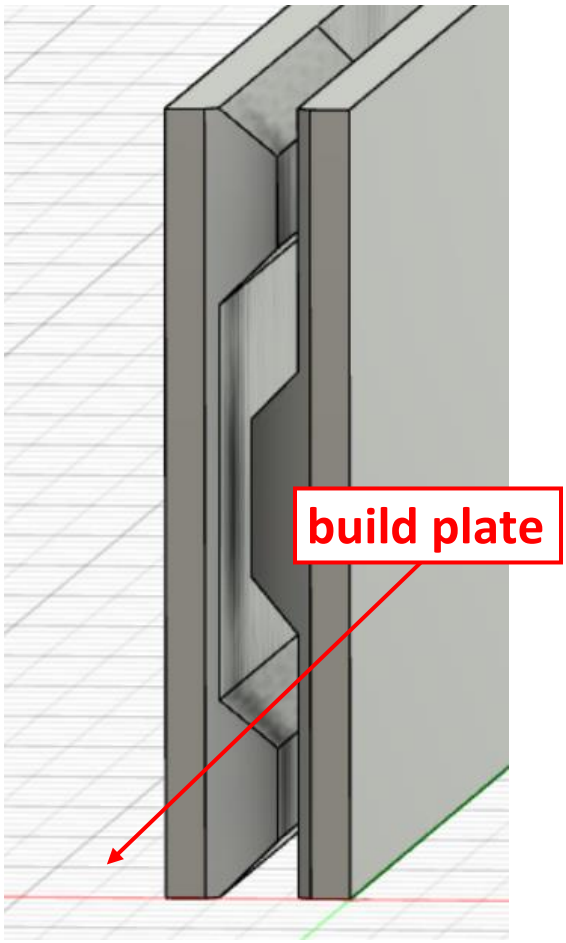
This page is just for information and the steps continue on the following page.

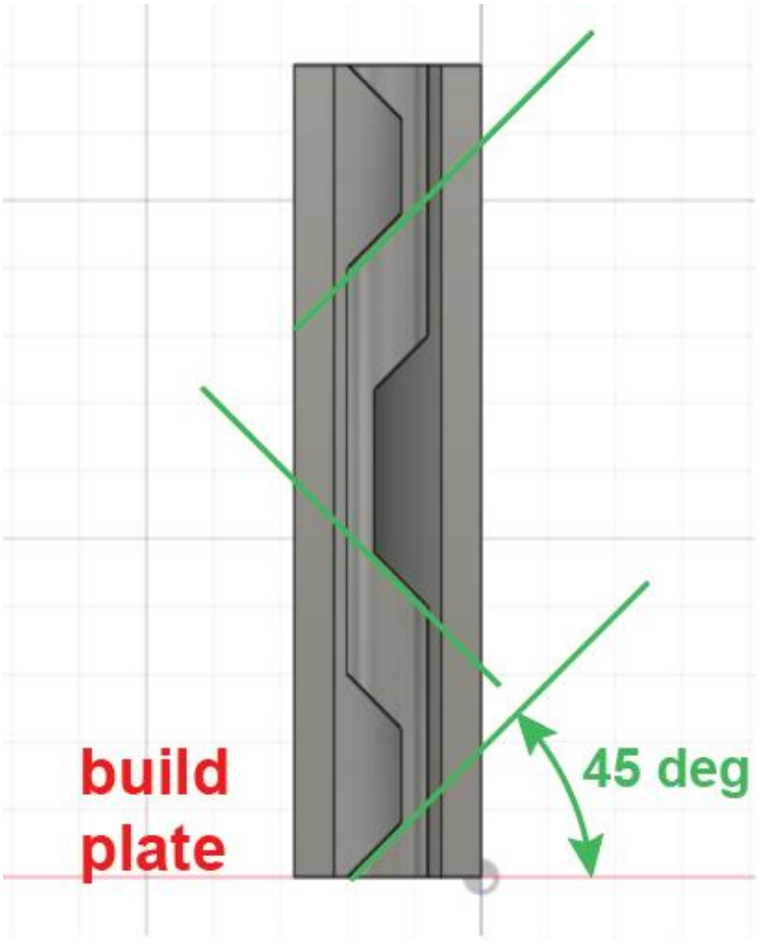
A slightly angled Front View should look as below.

Our design is optimized for 3D Printing by eliminating the need for supports. *Supports* are extra plastic elements that are printed to allow overhangs to be printed. These supports are then broken away, which would be difficult for this design.

The right side picture illustrates the design's adherence to the **45 Degree Rule of 3D printing**, wherein the angle of any overhang surface with regard to the build plate is equal to or greater than 45 degrees.

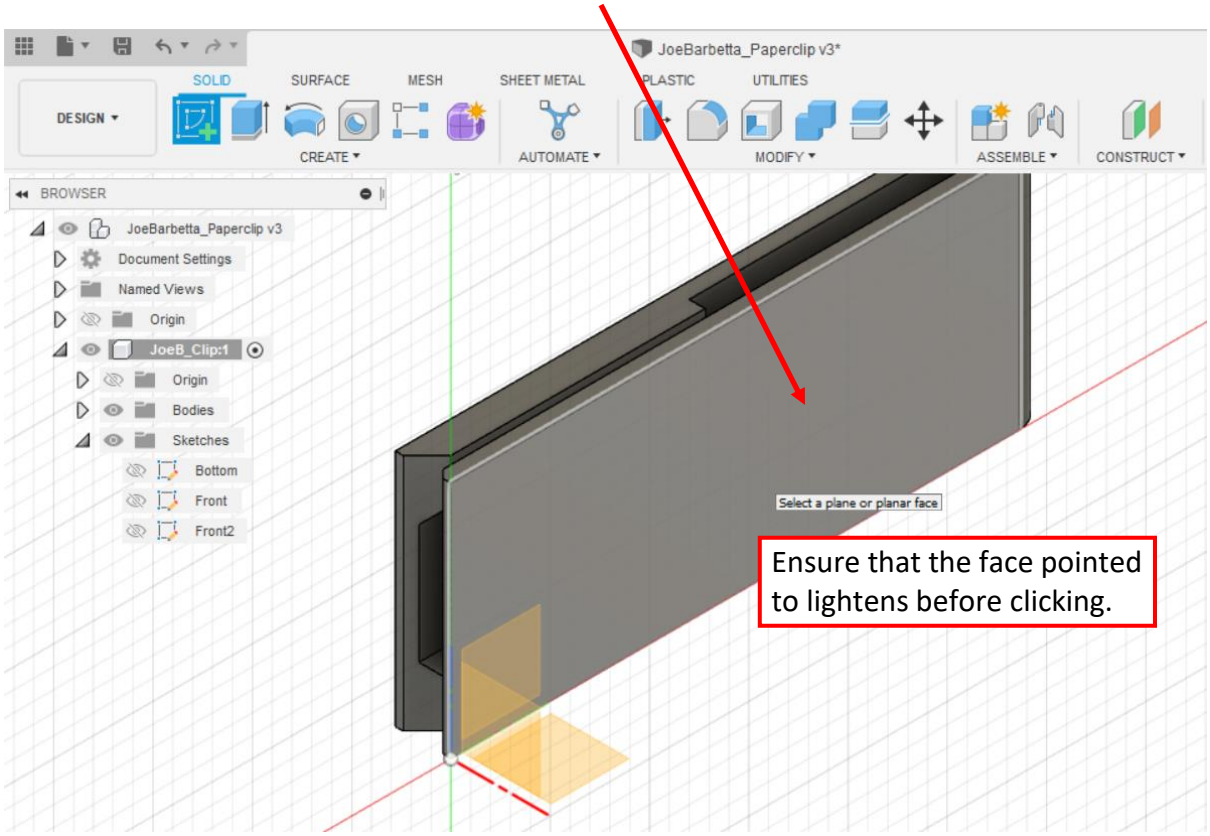
This is a **DFM (Design For Manufacturing)** consideration, which incorporates design features to optimize manufacturing.



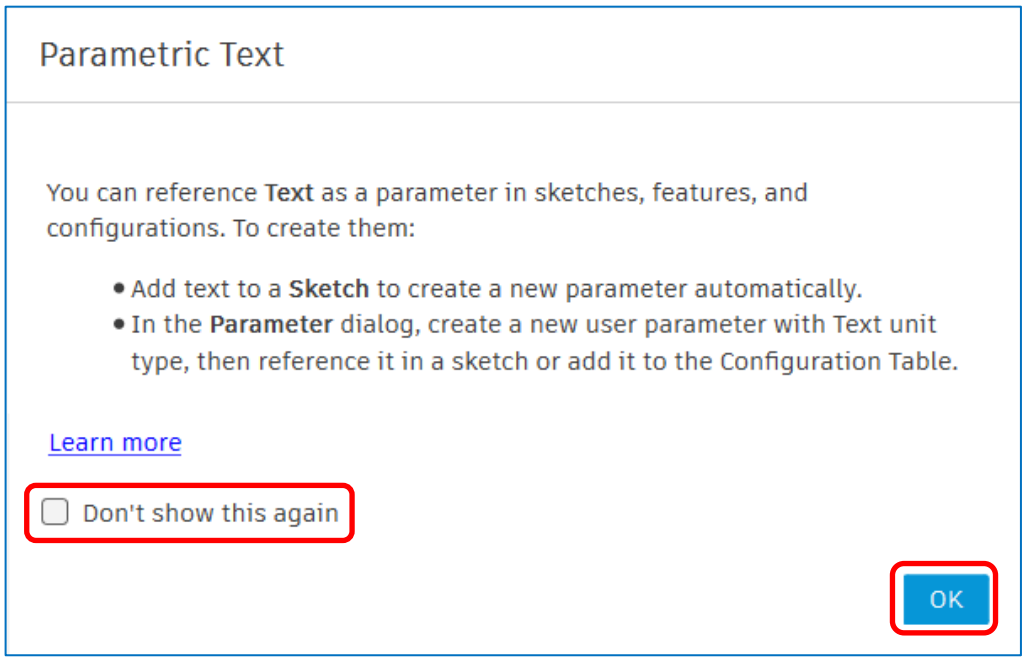


Creating Text

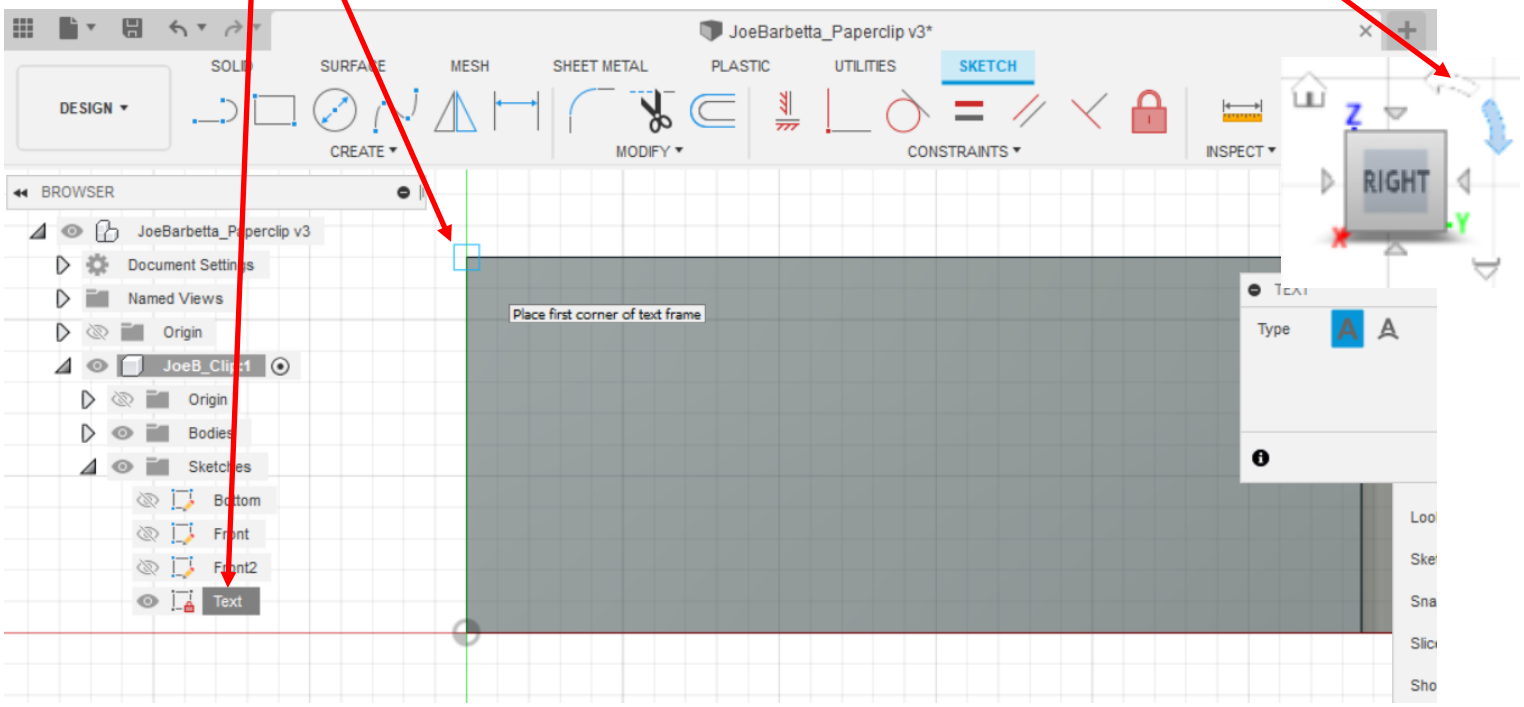
- adjust the **View** so that the *Face* where you want text can be click on.
- select the **Create Sketch** tool and **click on that face**.



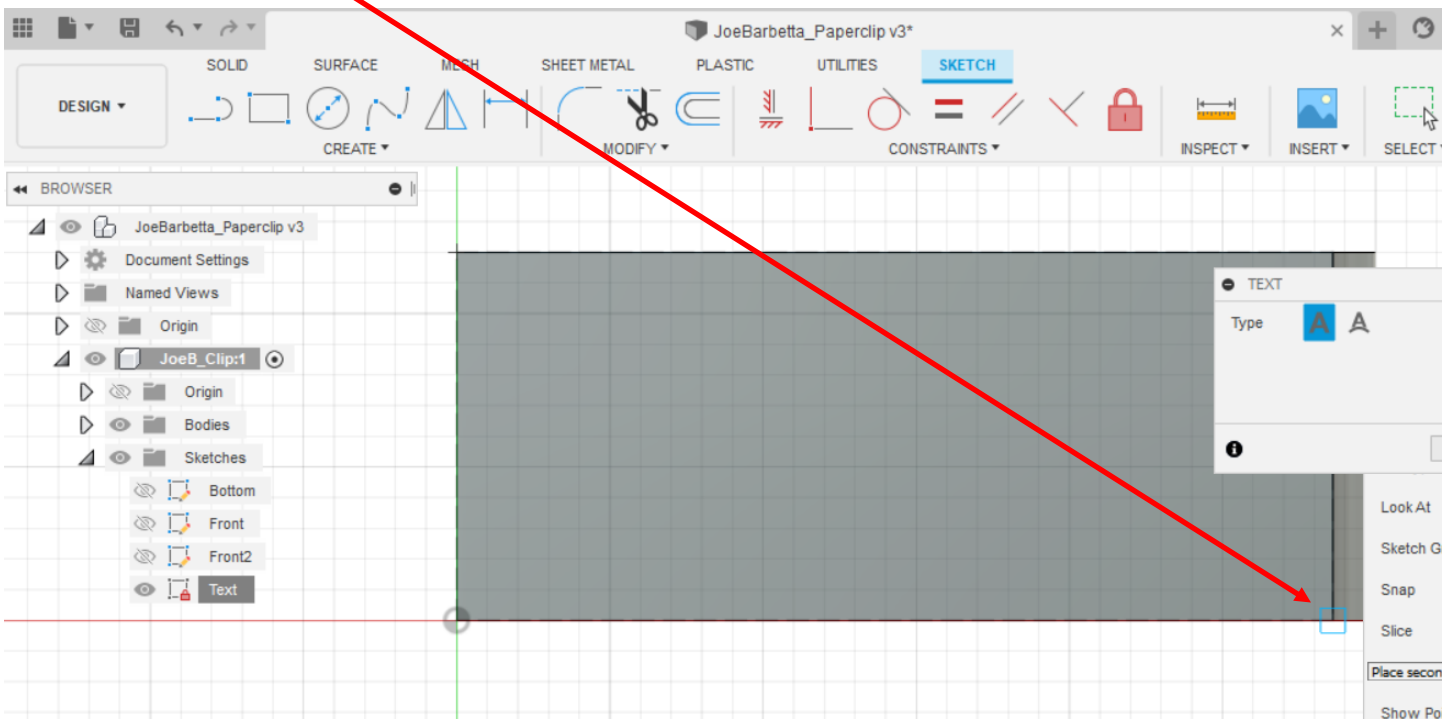
- select the **Text** tool in the **CREATE** pull-down menu
- if this window like this pops up **check the Don't show this again** check box and then click **OK**. It essentially announces a new feature where a Parameter can be used to specify the text. It's a feature that we will Not be using.



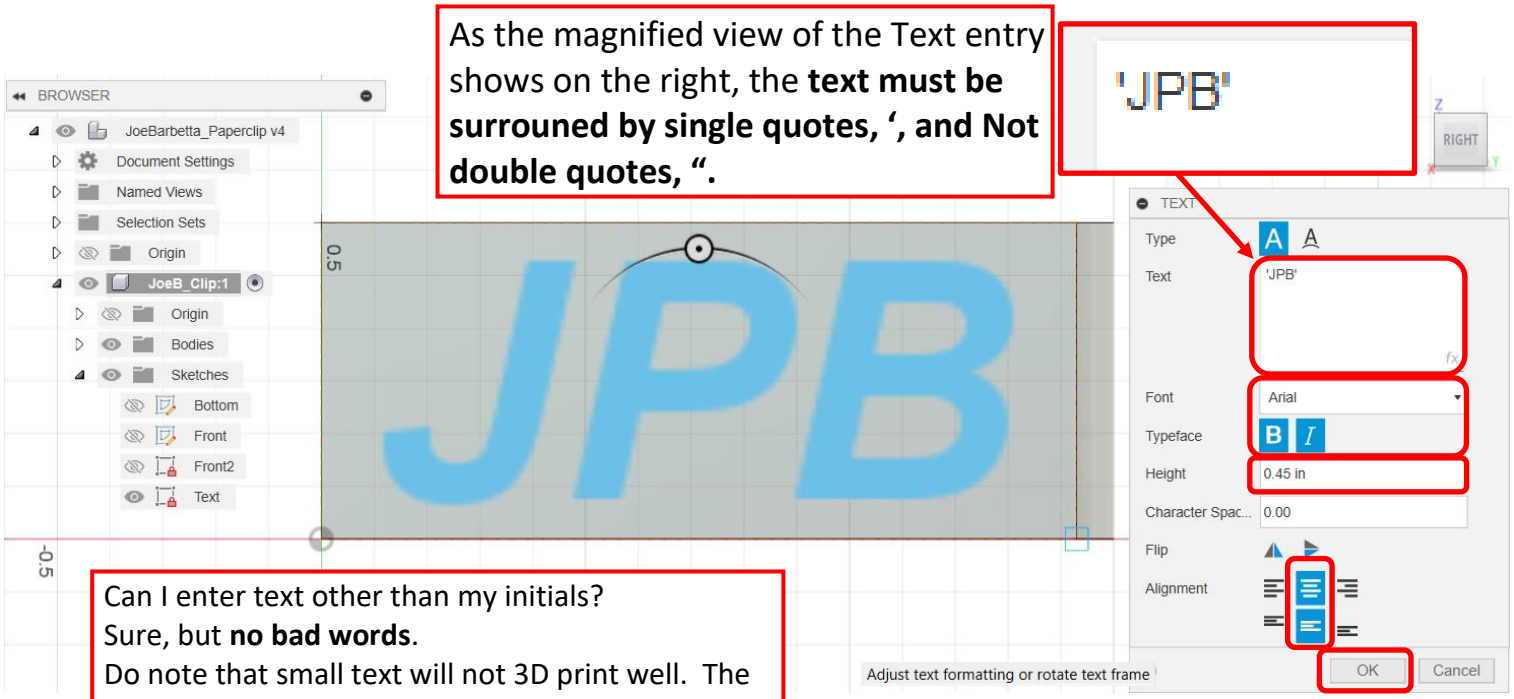
- if the desired orientation isn't achieved, click on a **Curved Arrow** at the upper right of the **View Cube** to rotate the View.
- right-click on the **name of the new Sketch** and select **Rename** and change the name to **Text**
- click on the **upper left corner** of the Face to start creating the text region rectangle. If this point cannot be selected, you may have to move the View to the right and away from the BROWSER region. This is due to a bug in Fusion 360.



- click on the **lower right corner** of the Face to complete the text region rectangle

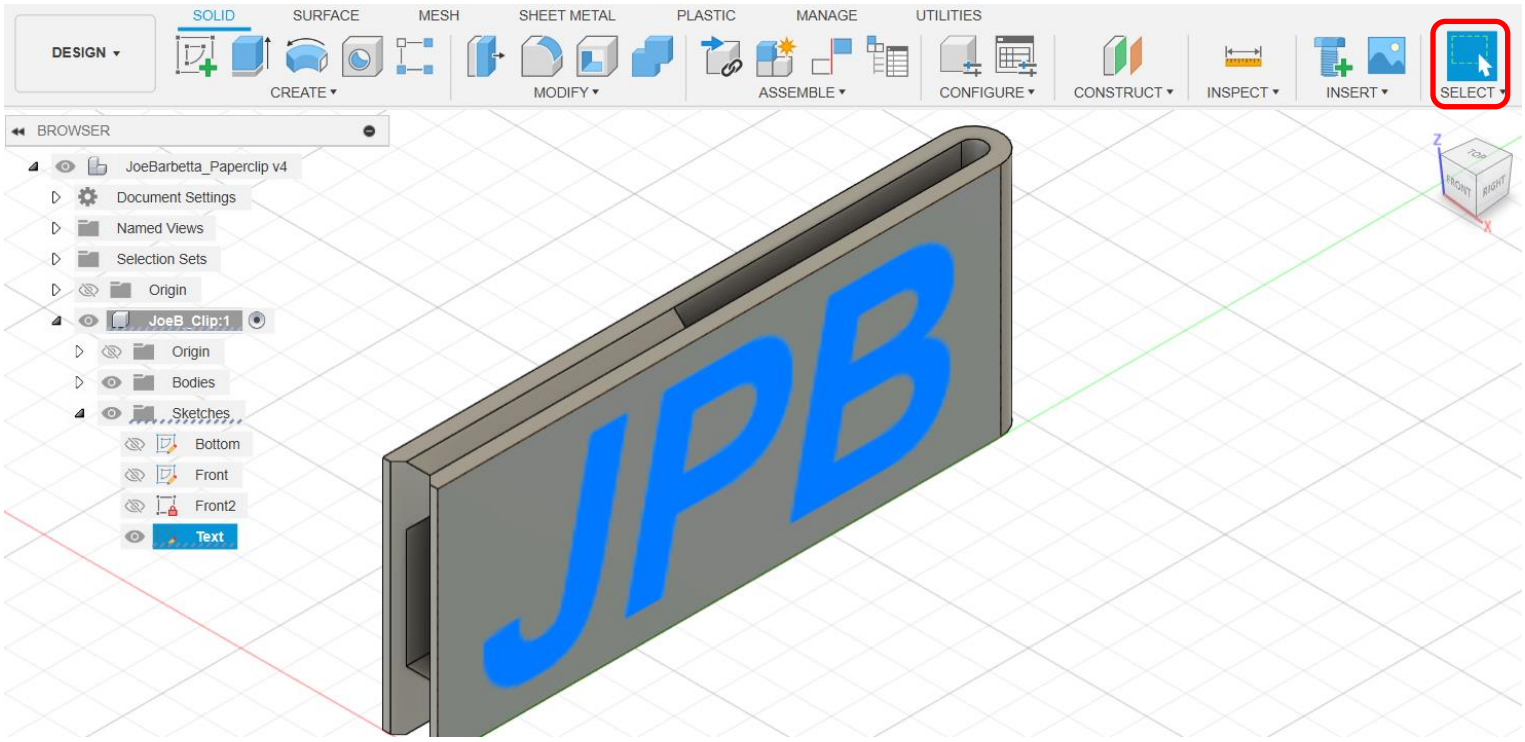


- in the **TEXT** window enter your three initials for **Text**. See the note in the top red box for the use of single quotes.
- click on the **Center** and **Middle** icons for **Alignment**.
- try different values in the **Height** box to set the desired height. Here **0.45** is used.
- as per personal 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** when done and then **Finish Sketch**

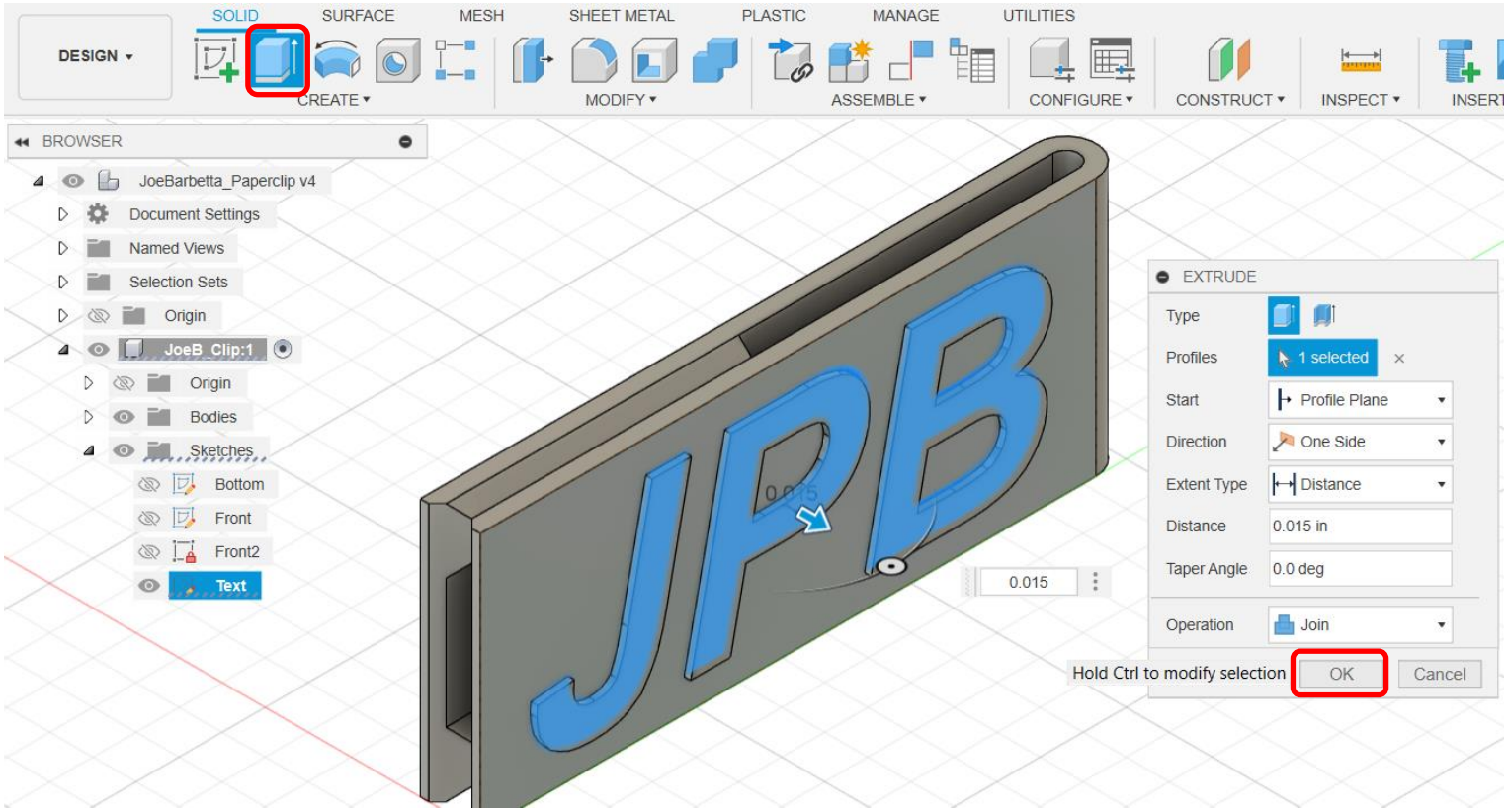


Can I enter text other than my initials?
Sure, but **no bad words**.
Do note that small text will not 3D print well. The larger the text the better. One could get away with two lines of text if each line is at least 0.20 in high.

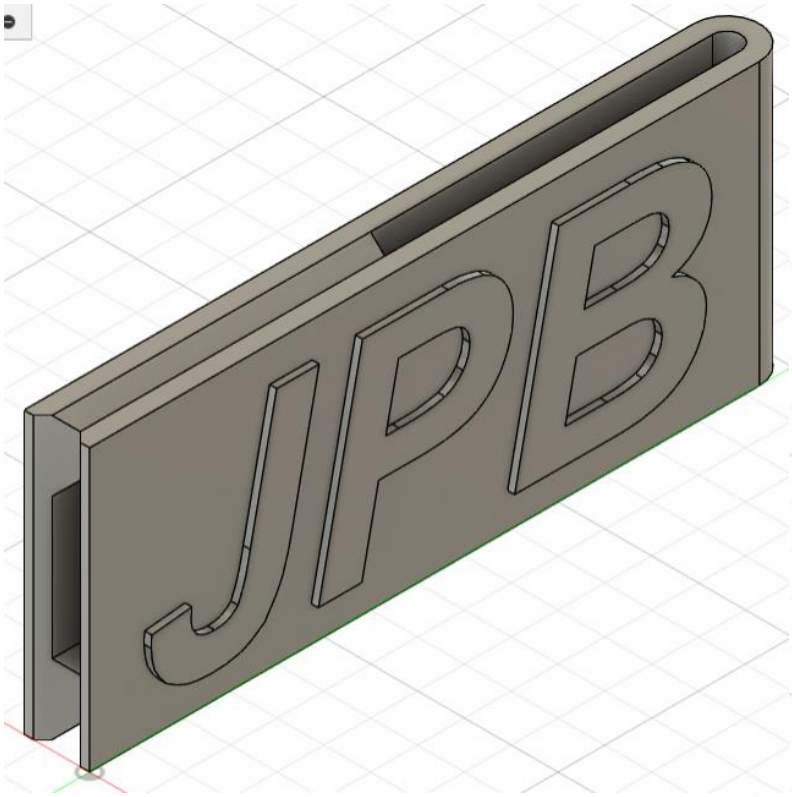
- click on the **Home** icon next to the **View Cube**
- click on the **SELECT** tool and then **click on any letter of the text**, which should make it turn bright blue.



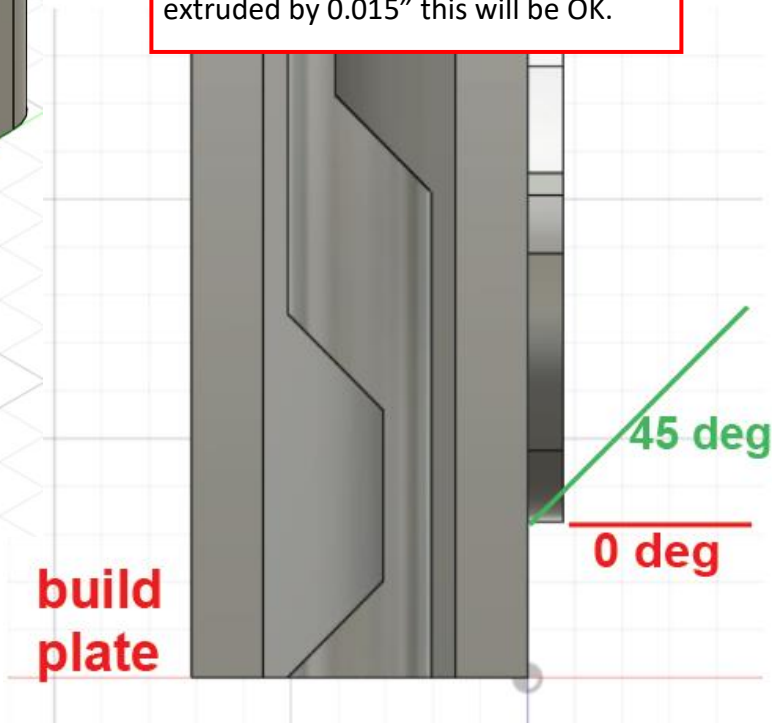
- select the **Extrude** tool, type **0.015**, and click **OK**



Here is the result.



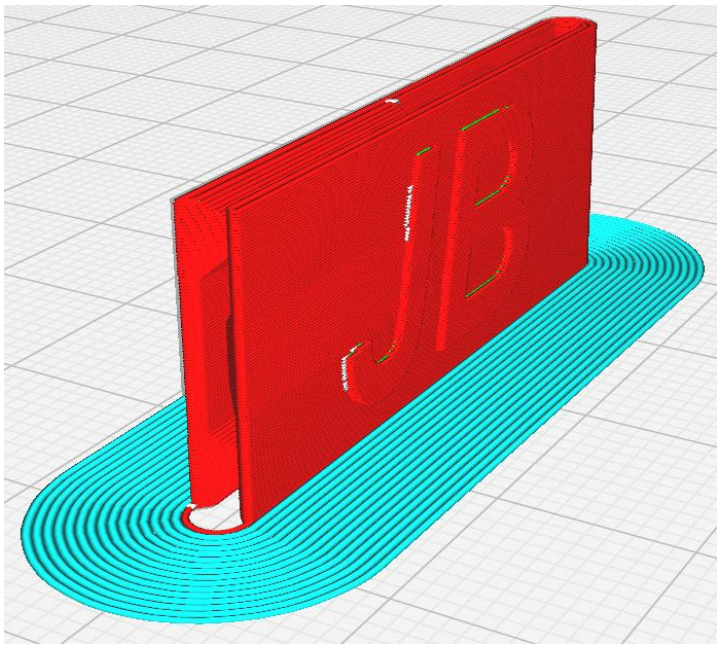
Note that the bottoms of the extruded text do violate the 45 Degree Rule. As shown below the bottom faces are parallel and thus 0 degrees relative to the build plate.
However, because the text is only extruded by 0.015” this will be OK.



Fighting Cura Brim Generation

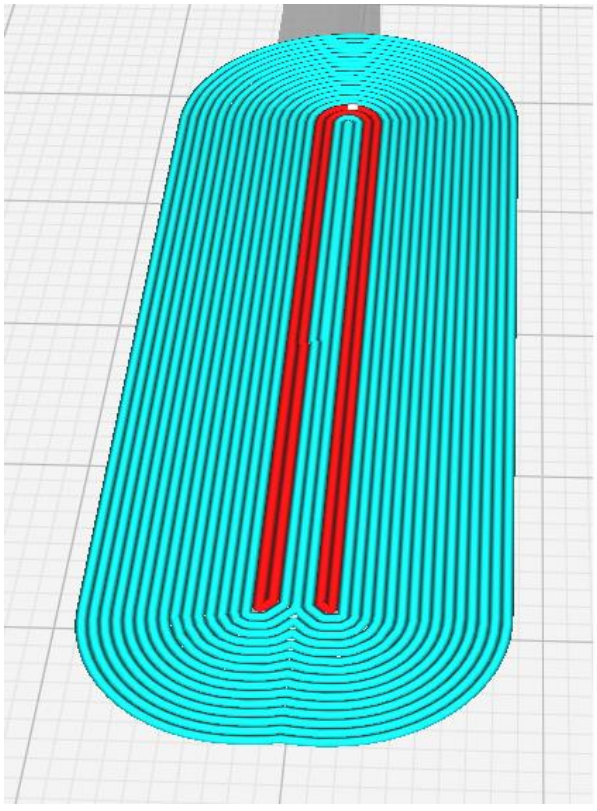
This page is just for information and the steps continue on the following page.

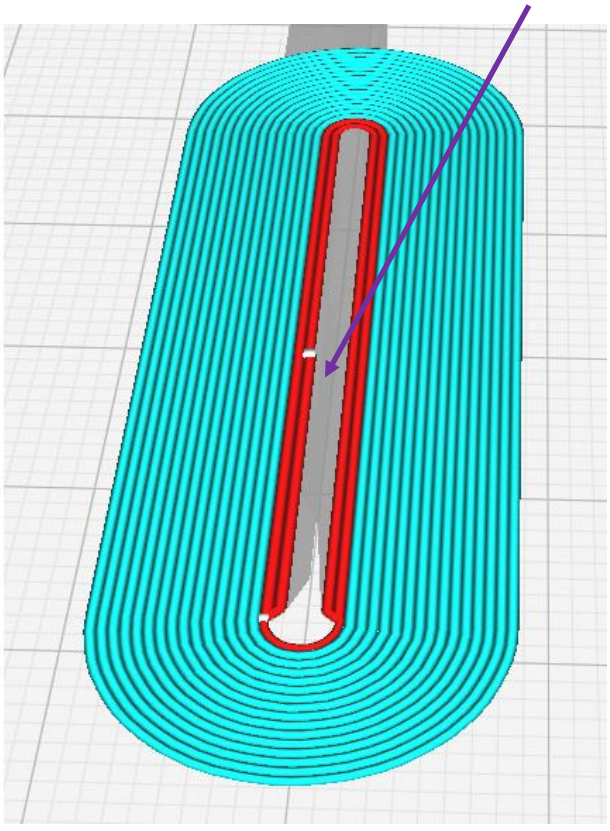
By default the Cura Slicer software will generate an extension of the first layer around the bottom of parts. This aids "**bed adhesion**" by preventing sections of the first layer from separating from the build plate. After printing is finished, the brim is removed by hand. Cura has a **Brim Only on 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.



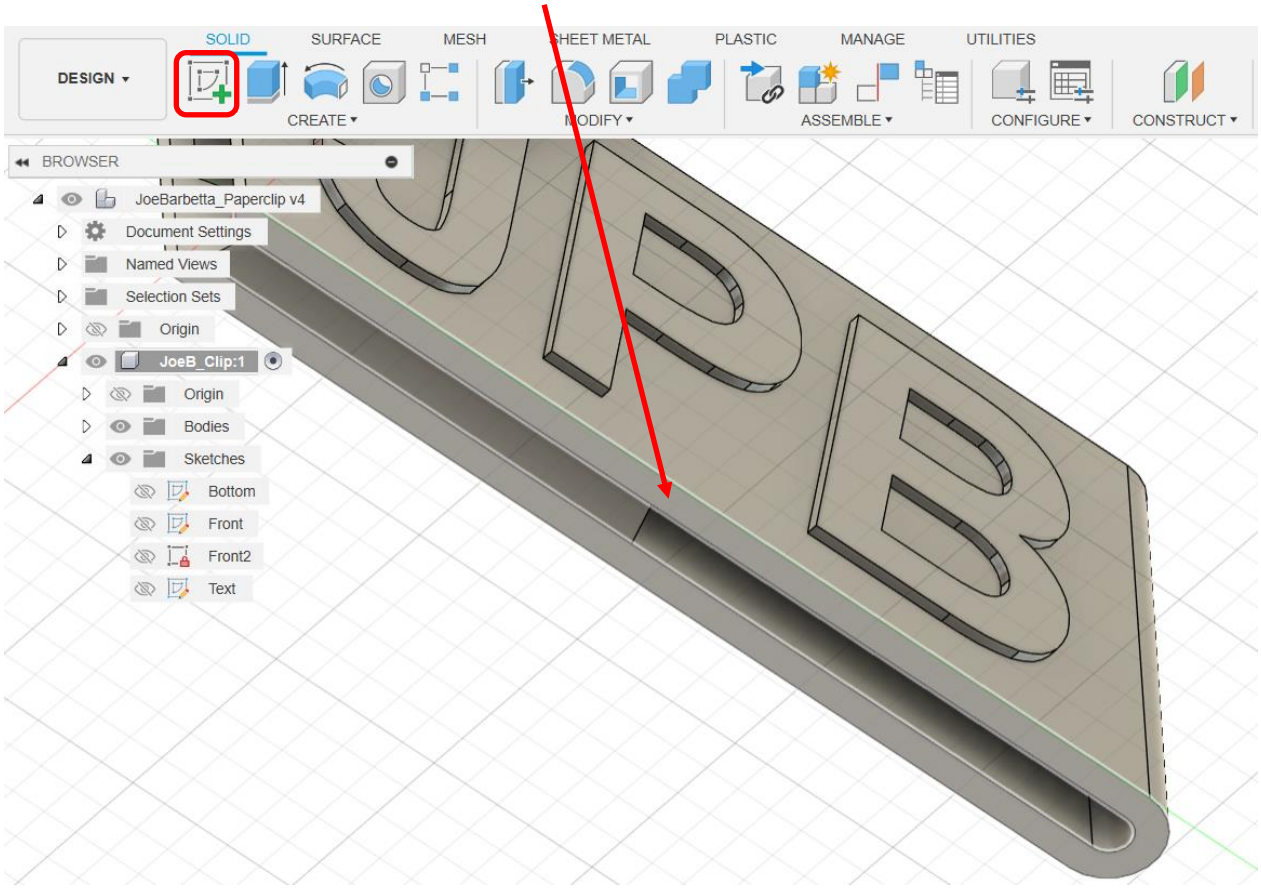
This is the view of the paperclip when it is opened in the Cura slicer program and sliced. To ensure that the first layer properly adheres to the build plate a Brim is generated as shown in blue.

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**.

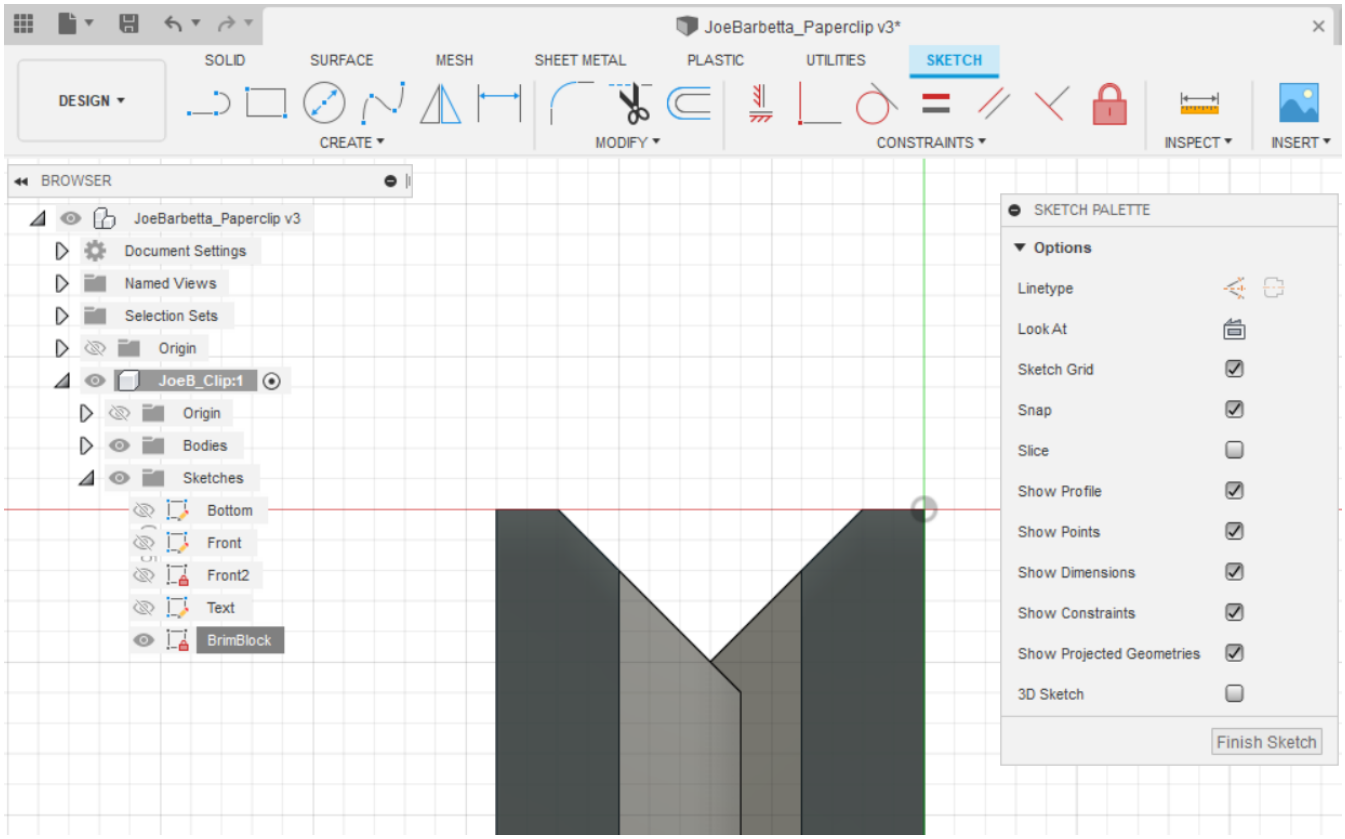




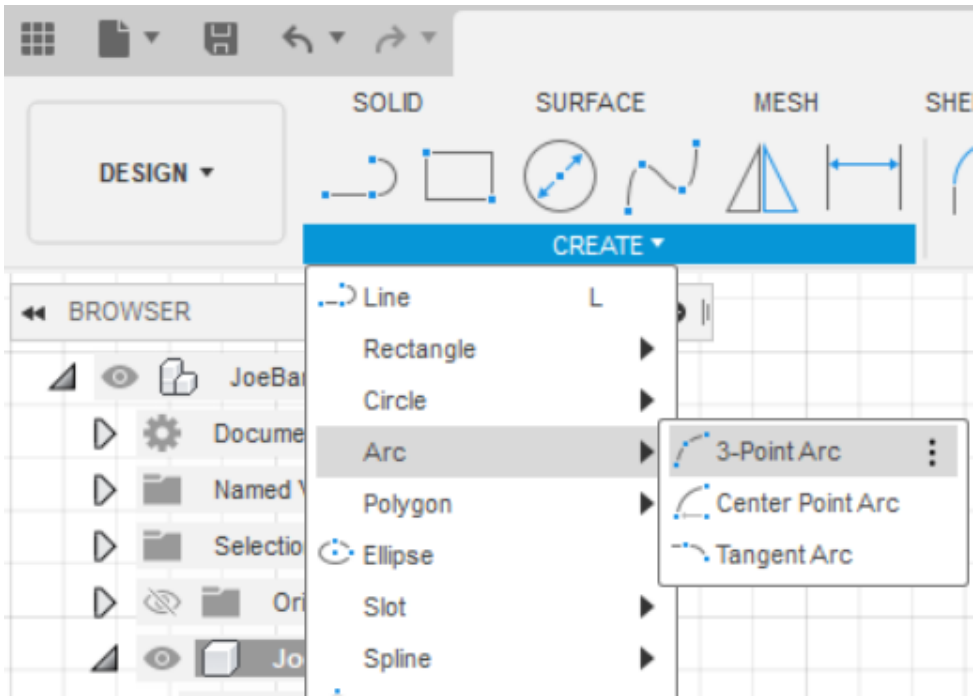
- adjust the View to access the bottom of the paperclip
- click on the **Sketch** tool and click on the **bottom face**. When the mouse is first over the bottom it's color should lighten.



- zoom into the top of Sketch with some room above it as shown below
- rename the Sketch to **BrimBlock**

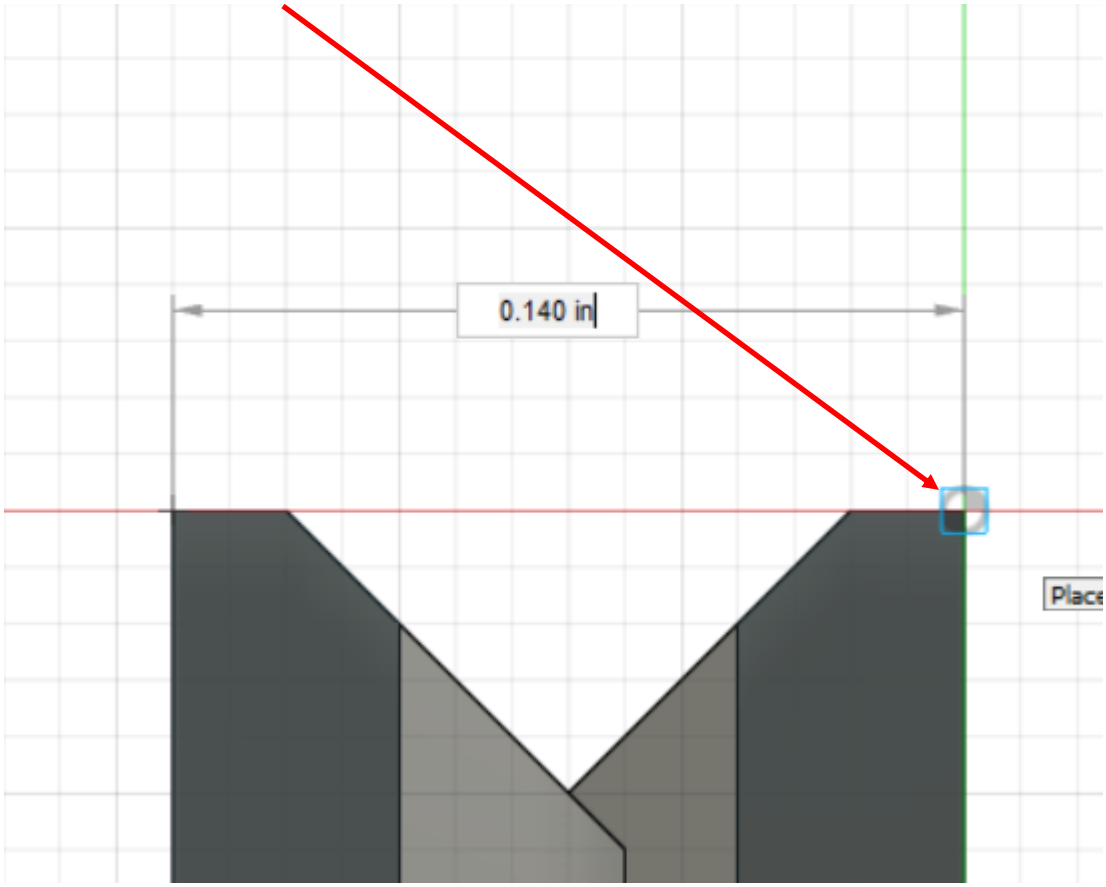


- from the **CREATE** pull-down menu select **Arc** and **3-Point Arc**

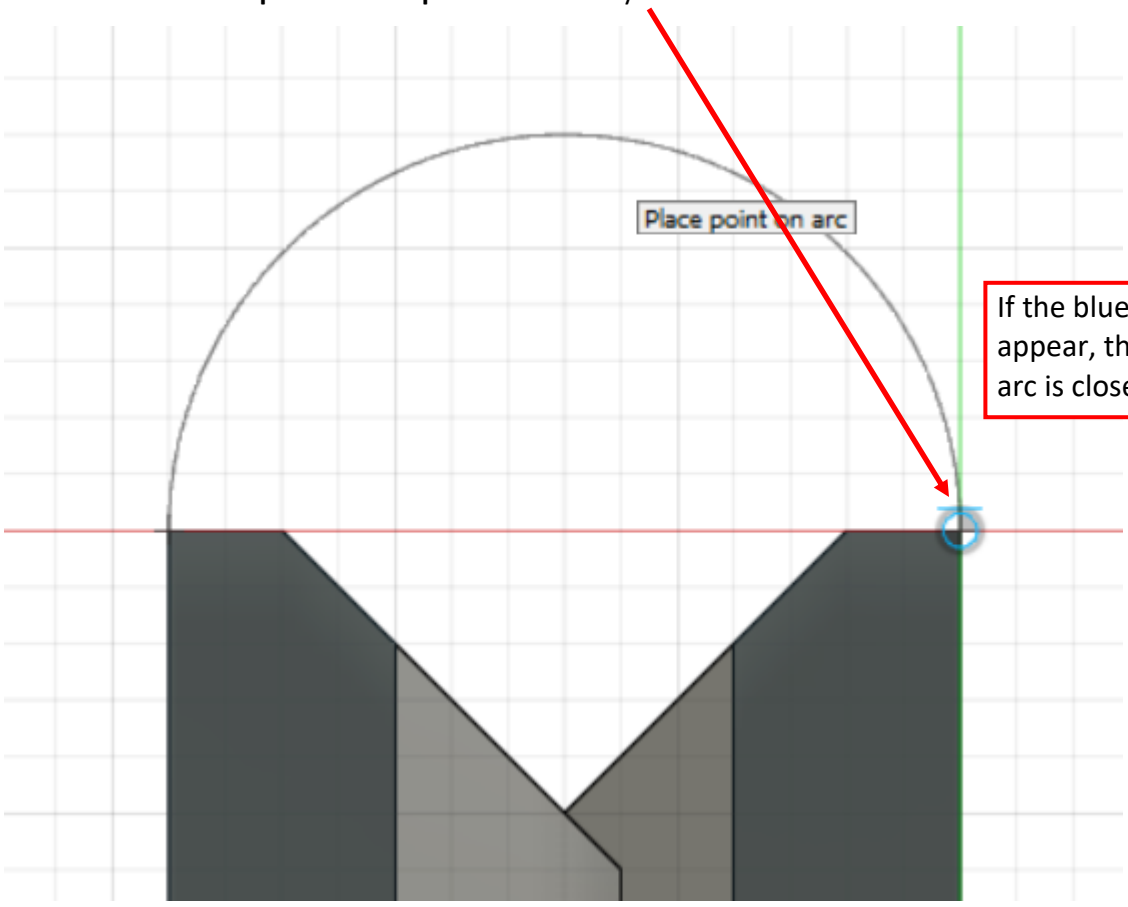


- click on the **upper left point** to specify one end of the **Arc**

- click on the **upper right point** to define the other end of the **Arc**

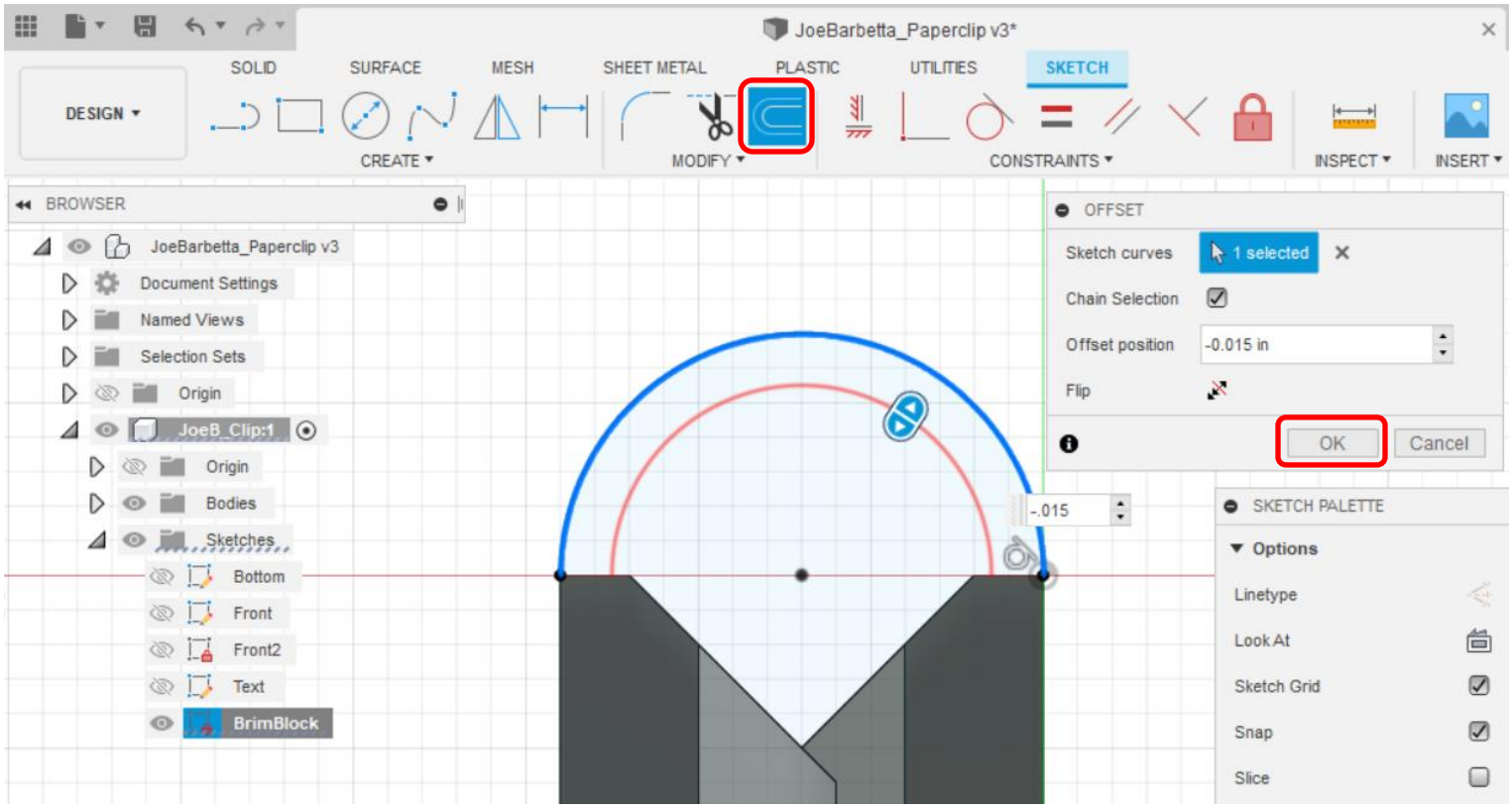


- move the mouse to **pull the Arc up** until the blue symbol indicates that the Arc is a semicircle and click

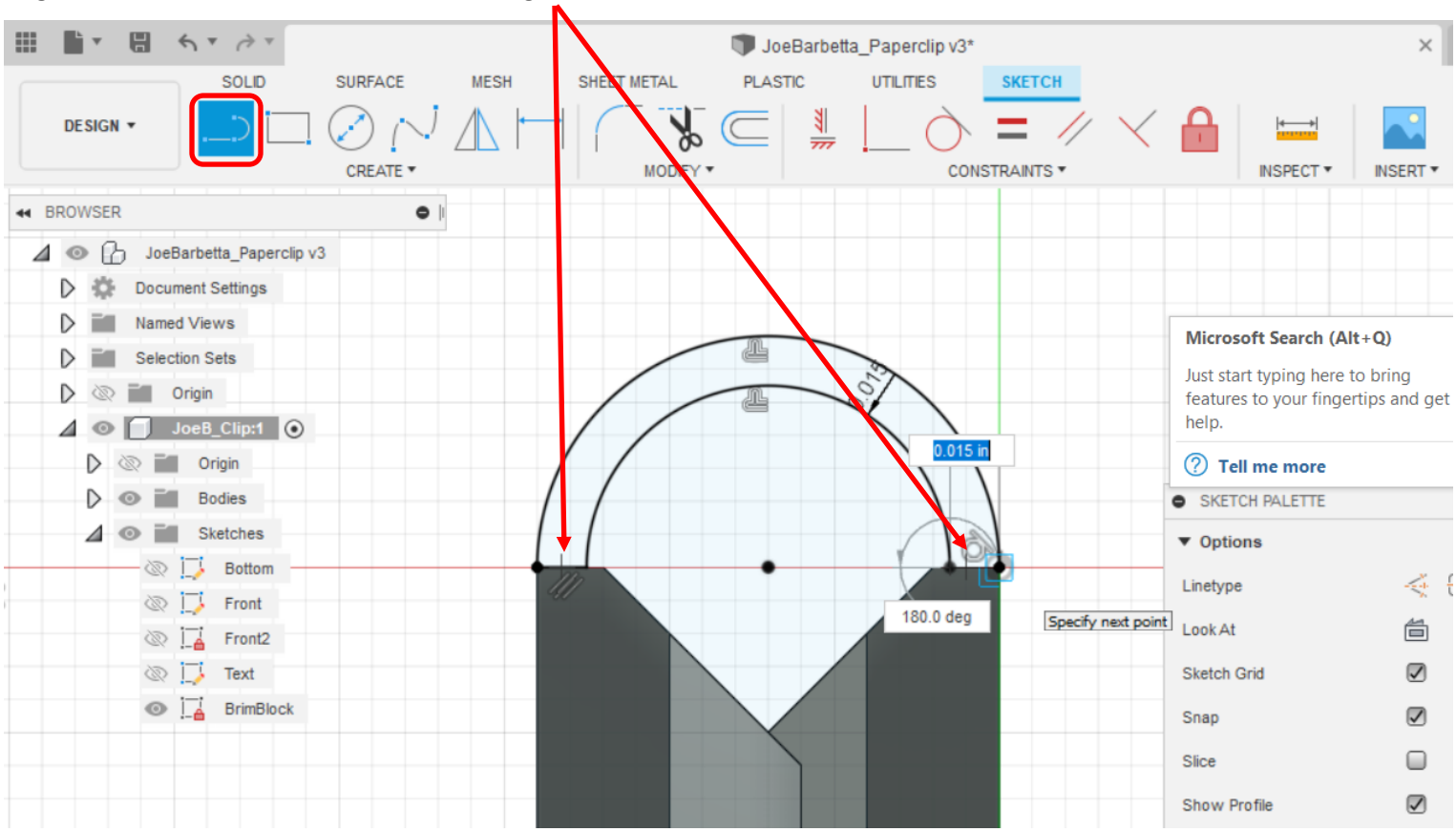


If the blue symbol doesn't appear, then just click when the arc is close to the shown shape.

- click on the Offset tool and enter **-0.015** (note the minus sign). This will move the red arc to the inside of the blue arc.
- click on **OK**

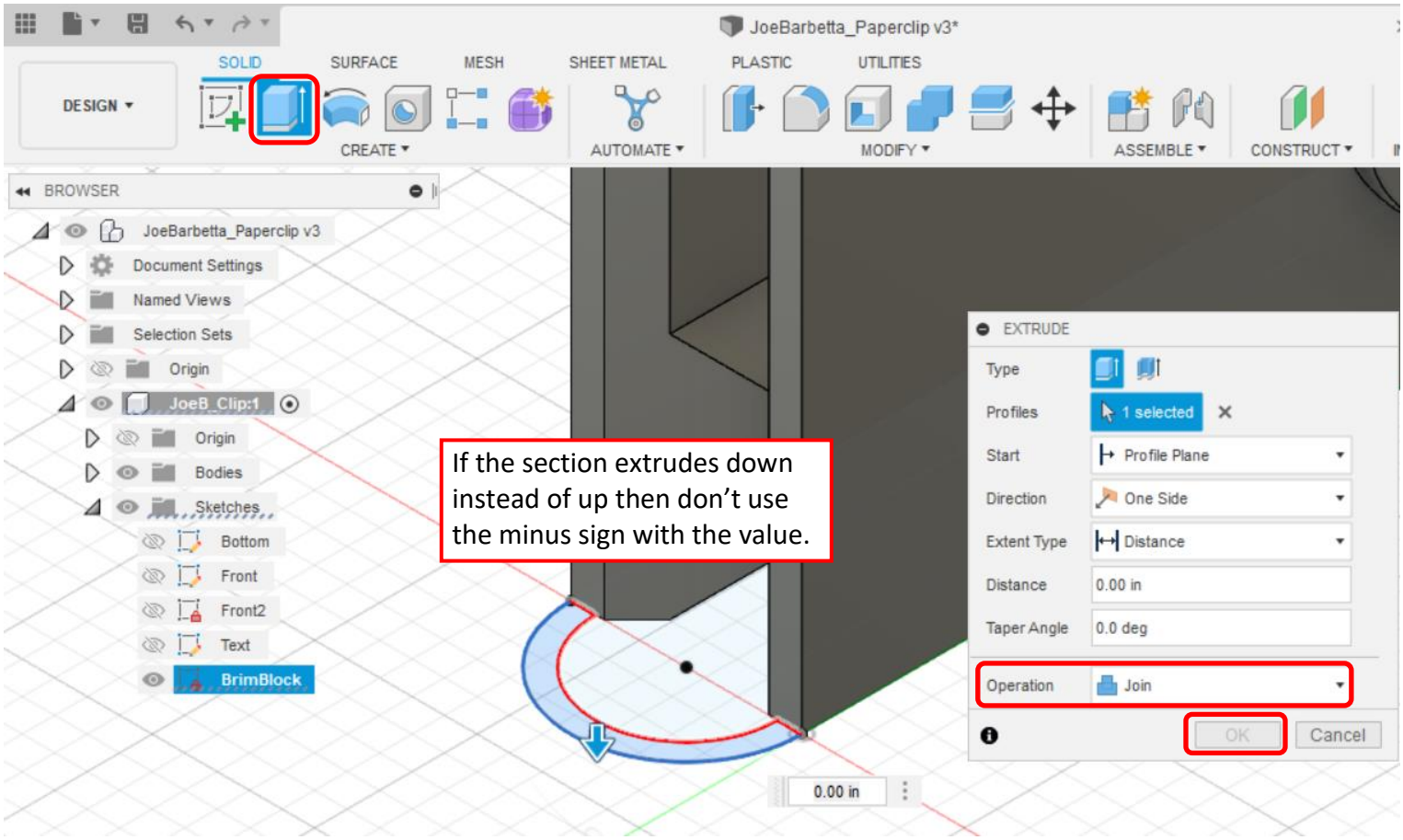


- right-click on the Line tool and create line segments between the outer and inner arcs and click **Finish Sketch**



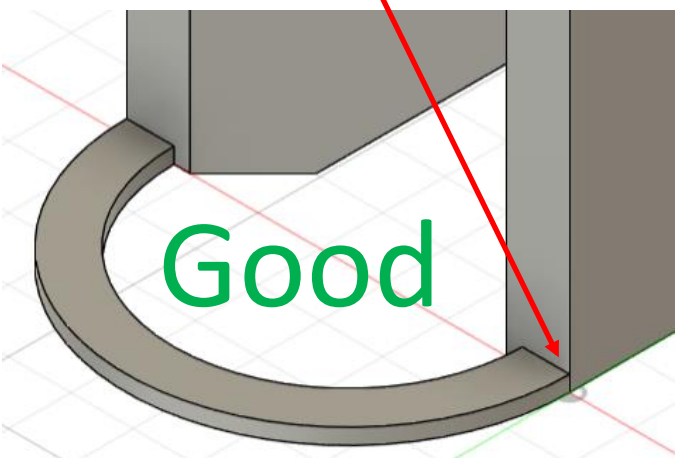
- click on the **Home** icon and zoom into the arc profile just created

- click on the **Extrude** tool and enter **-0.005** (**note the minus sign**)
- ensure the **Operation** is set to **Join** and click **OK**. The profile should extrude up a small amount.

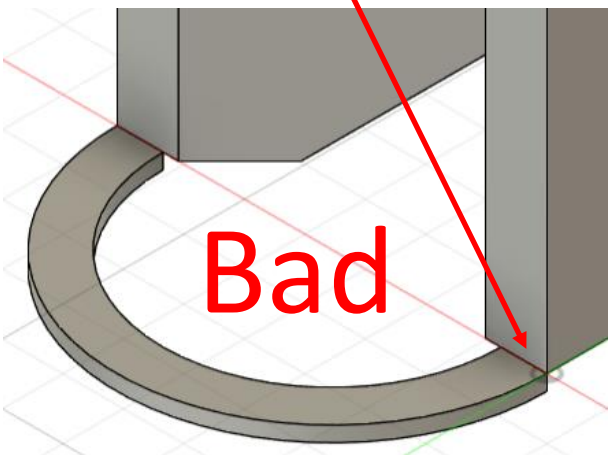


- ensure the Brim Block is correct

The **bottom of the Brim Block should be level with the bottom** of the clip body.



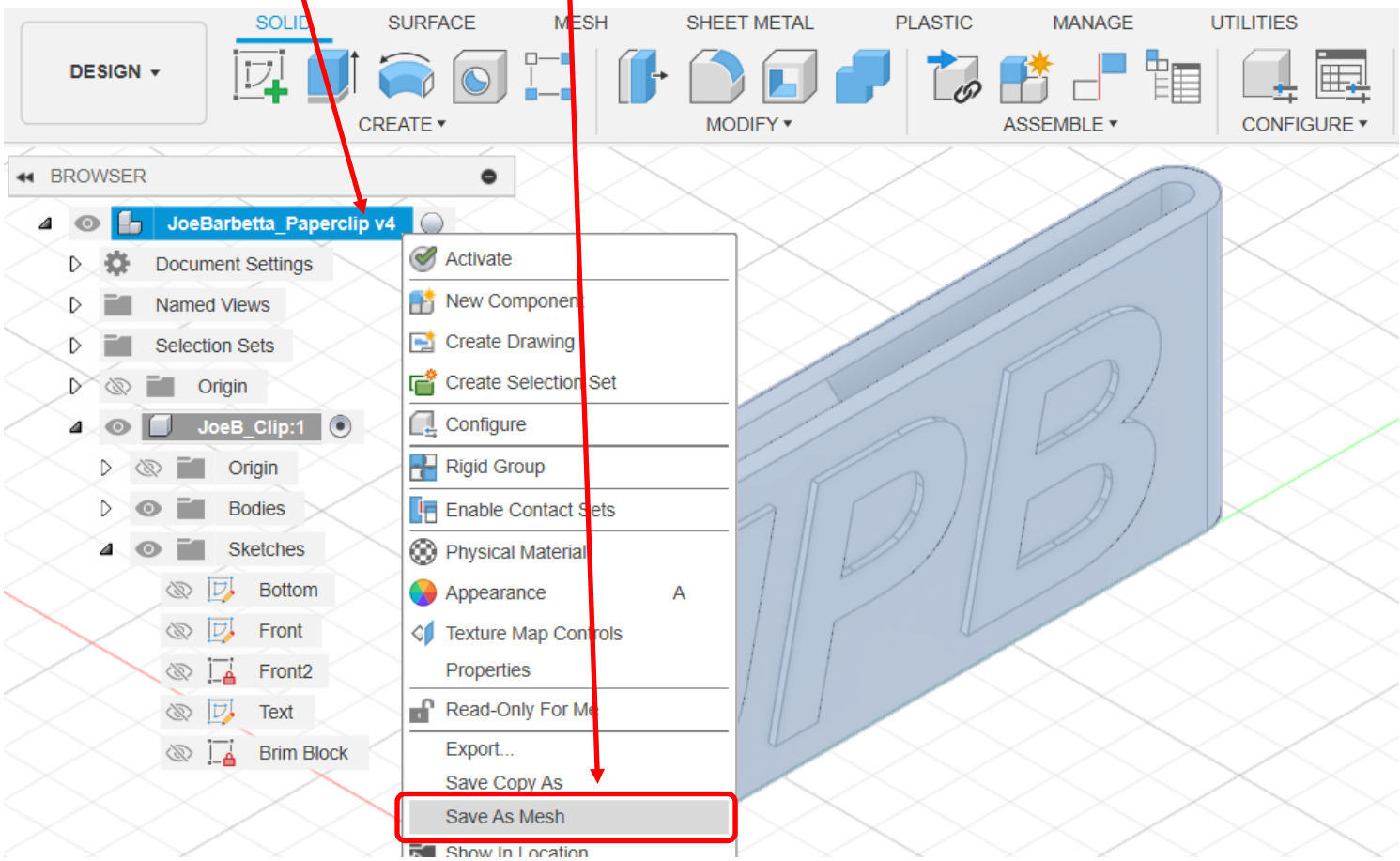
Here the Brim Block is lower than the clip body. Redo the extrusion step to it looks like the left picture.



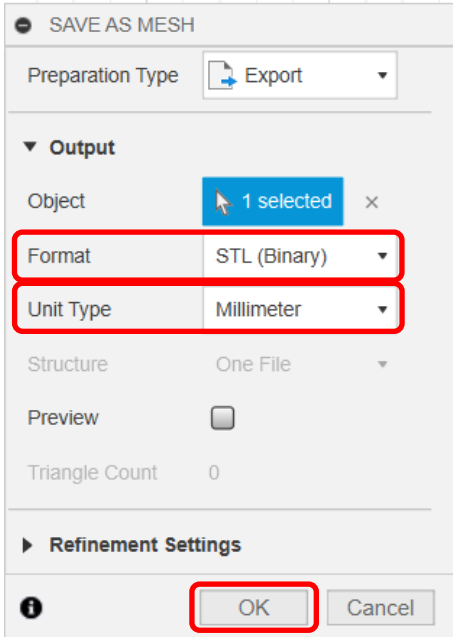
Exporting STL Files

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.

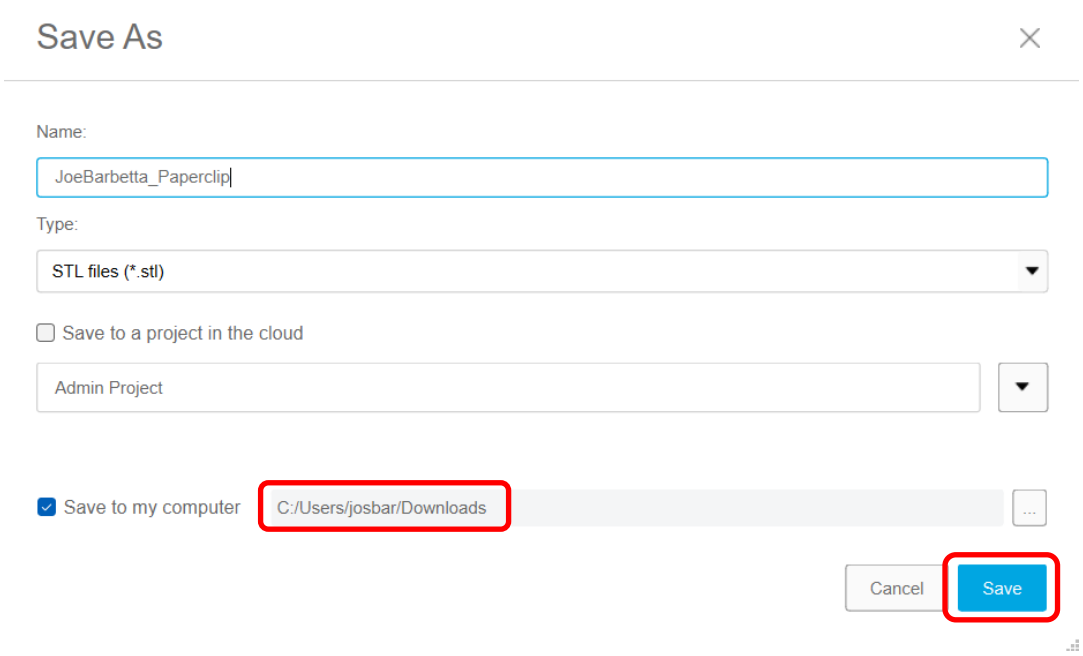
- right-click on the **Project Name** and select **Save As Mesh**



This window will show each time Save As Mesh is used. Ensure that **Format is set to STL (Binary)** and **Unit Type is set to Millimeter** and then click **OK**. You will then be prompted to save the file. The default location is the **Downloads** folder.



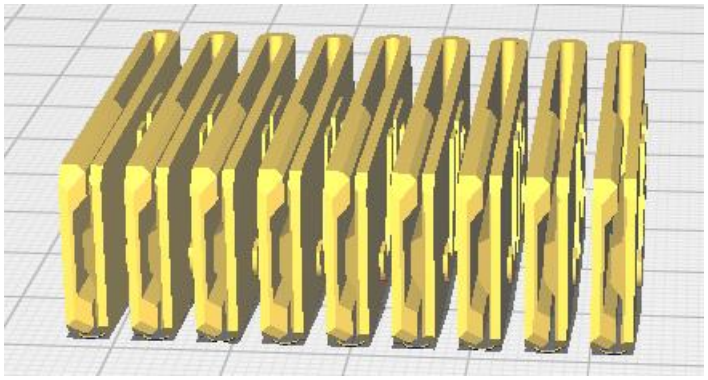
- click **Save**. By default Fusion 360 will save the STL file to the **Downloads** folder.



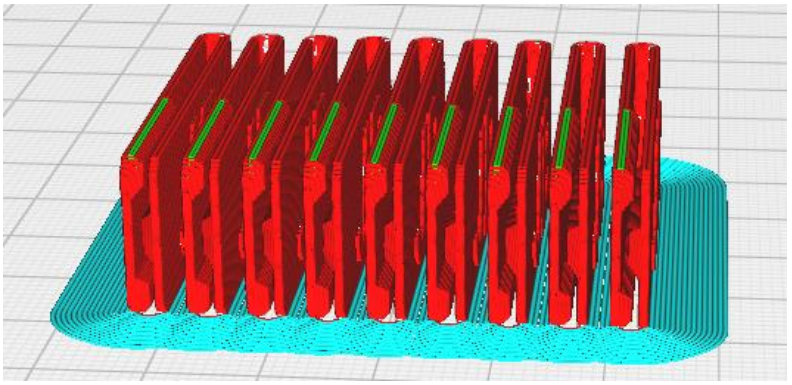
You will not be doing anything to create the below drawings. They just show the process of printing the paper clips.

Once you submit your STL file, it and those of your colleagues, are opened in the Cura slicer program. The program "slices" them using 0.15mm layers and adds a brim. Cura outputs a single .ufp file that is loaded into an Ultimaker 3D printer.

A group of paperclips in the Cura slicer program



The group after slicing and brim generation



A group of paperclips printed on an Ultimaker 3D printer

The color of 3D printed objects depends on the color of the filament that is loaded into the 3D printer.

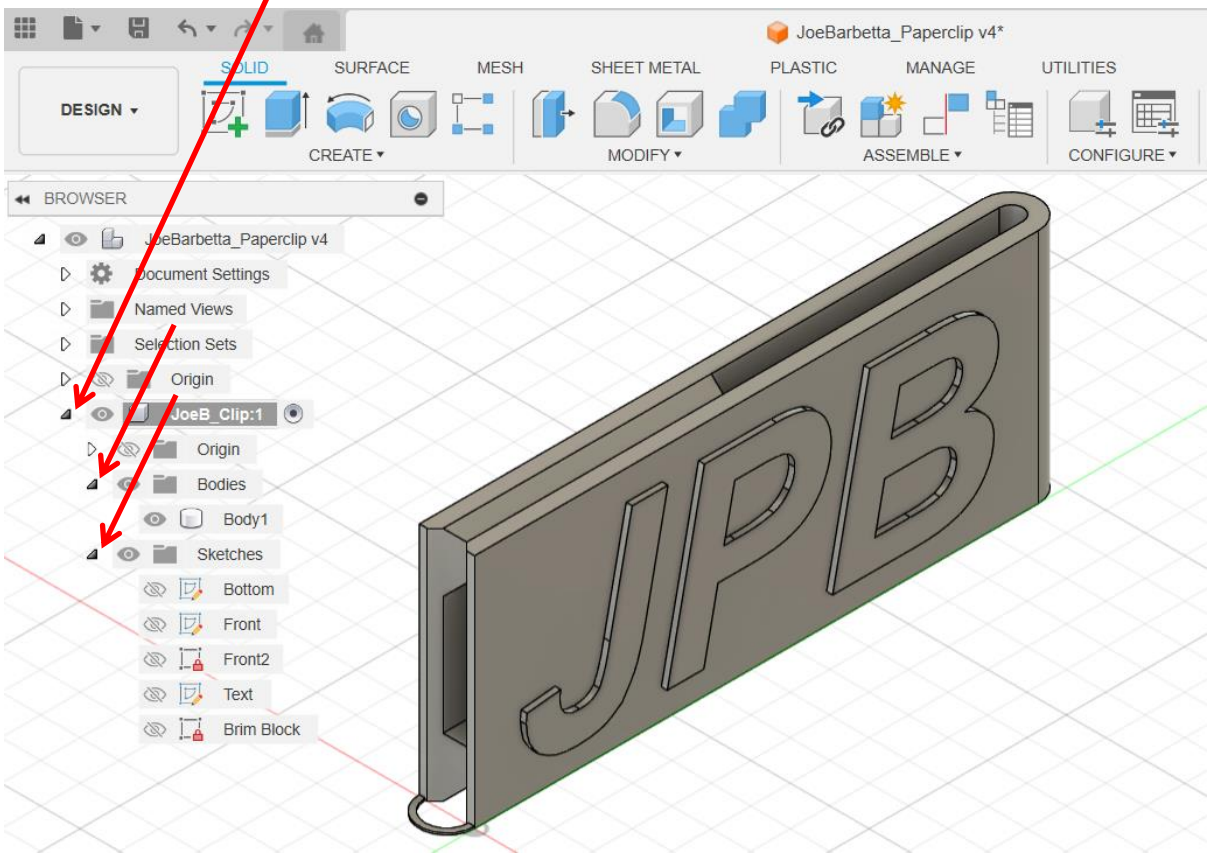
Creating a PNG file for submission

- reset the view to the Home view and position the paperclip near the BROWSER section as shown below.

- if needed click on the **Expand Arrows** to ensure that the **Component, Bodies, and Sketches folders are open**.

- press the **Shift**, **Windows**, and **S keys** together and then stretch the selection rectangle over the screen to include the BROWSER section and paperclip as shown below.

- click on the window that pops up at the bottom right of the screen and then click on the top right File icon of the Clipping window and save the file. It should save as a .png, which will be uploaded to Schoology. It should be viewable in Schoology.



Please submit both the above **PNG screenshot** and **.stl file** on Schoology.

The .stl file will Not show its image in Schoology as the .png does.