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

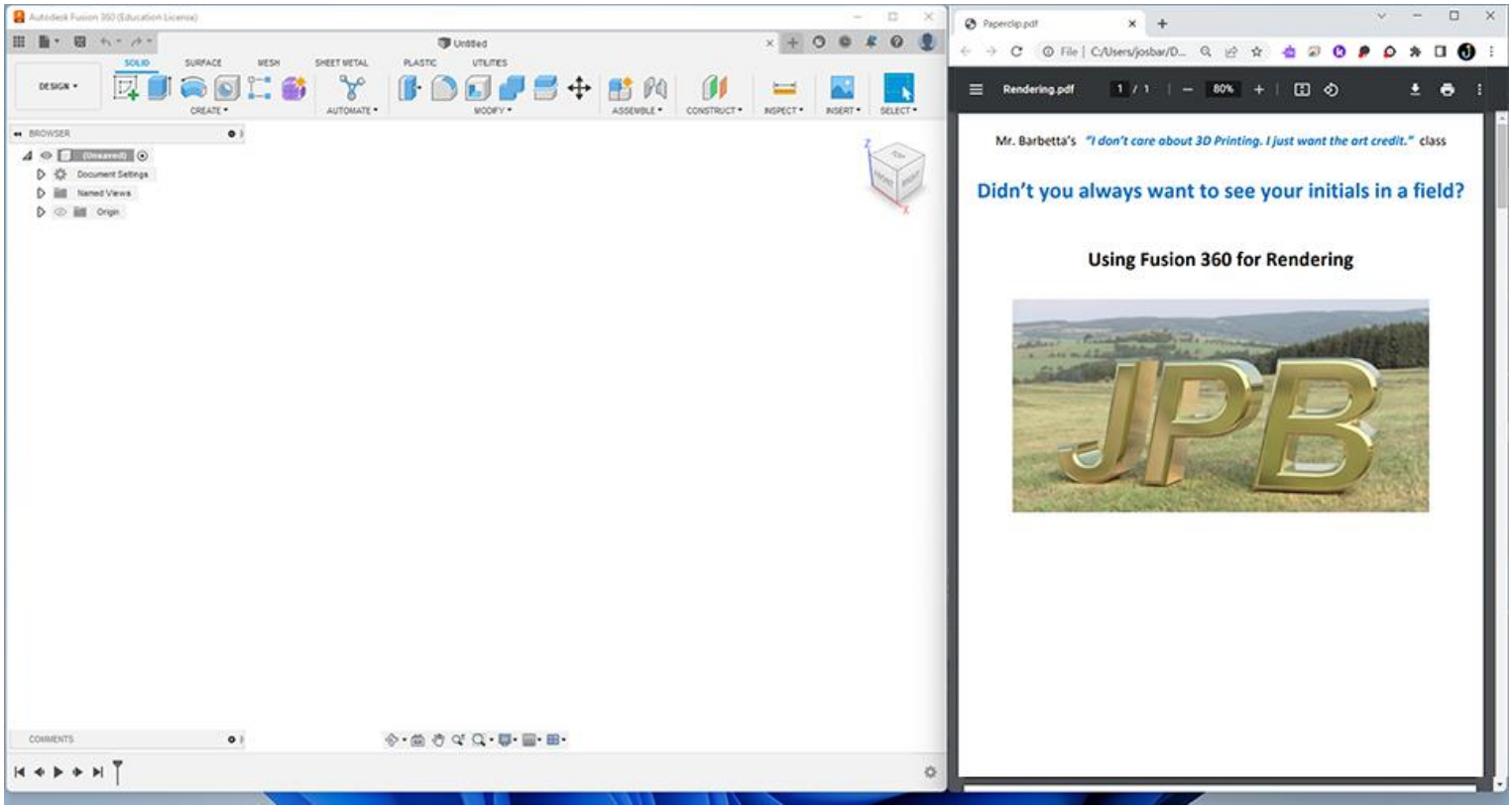
Out damn burrs! Out I say!

Today's lesson is sponsored by McMaster-Carr.

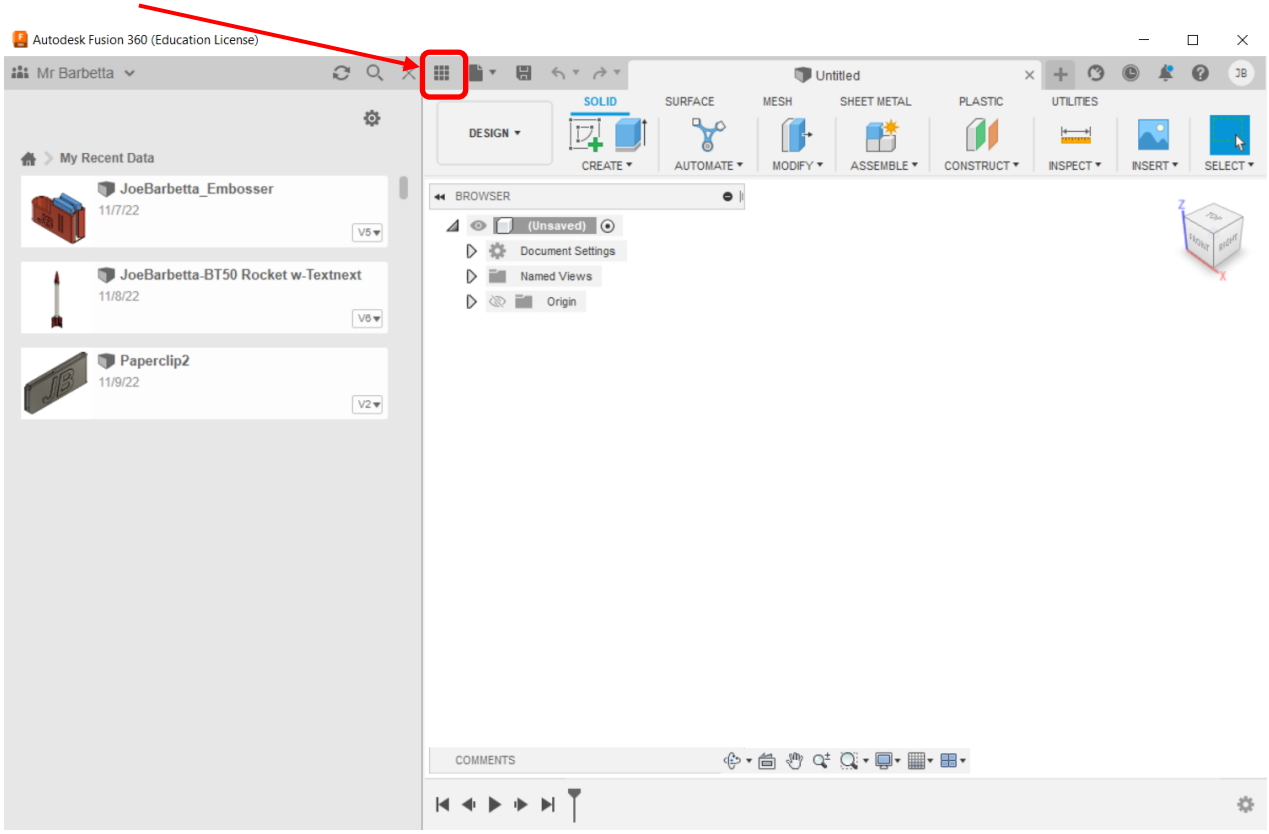
Contents

Using This Document

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



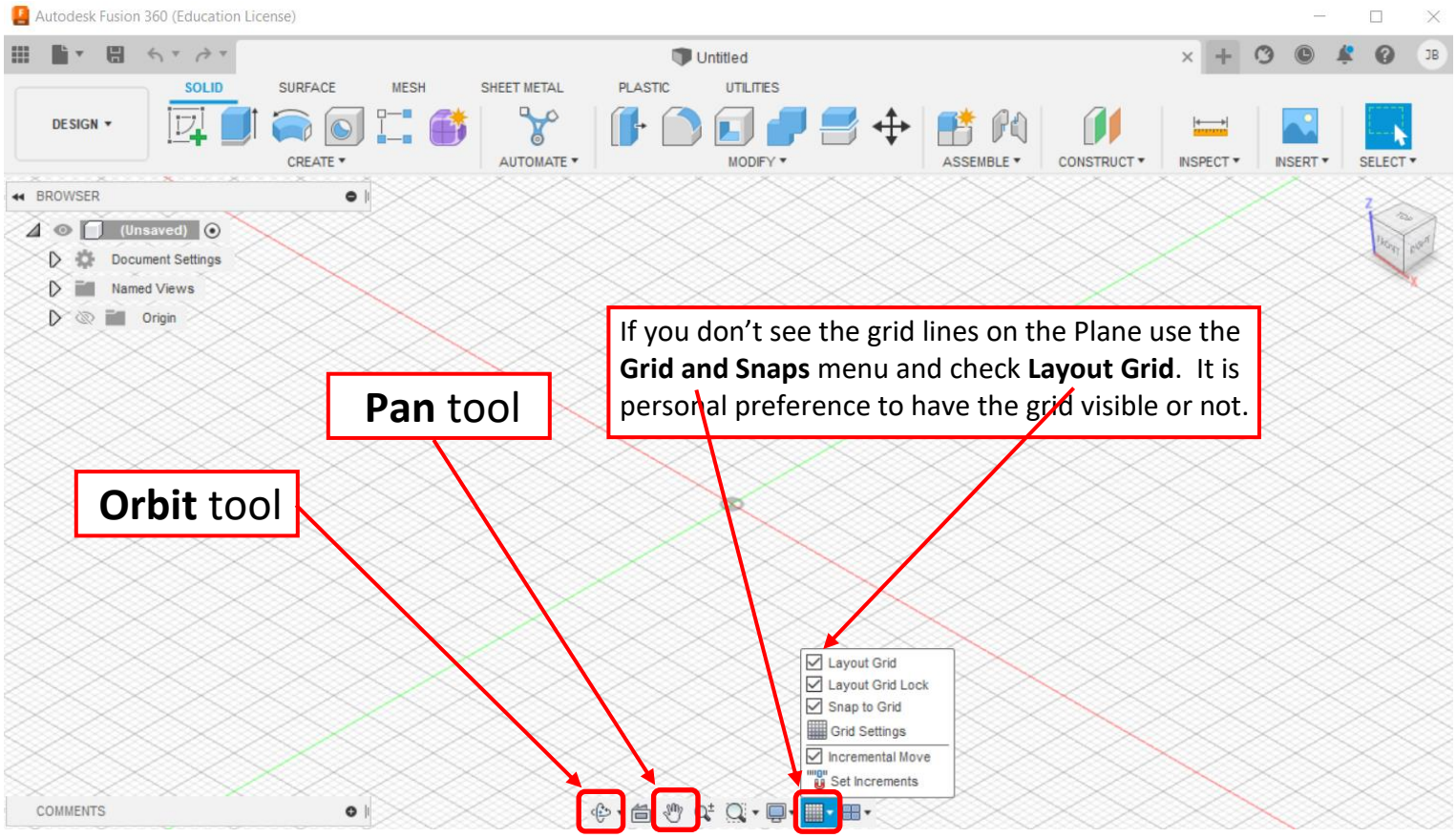
The Fusion window will not allow its width to be reduced much so for smaller computer screens a trick is to click on the **Data Panel icon** and then move the window to the left with the Data Panel off the screen.



Changing the View of a Design

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

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

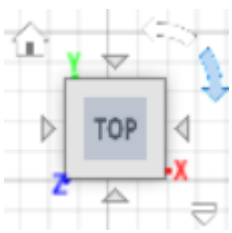


Here is a close-up of the View Cube at the top right of the window.

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

The View Cube now resembles that on the right.

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



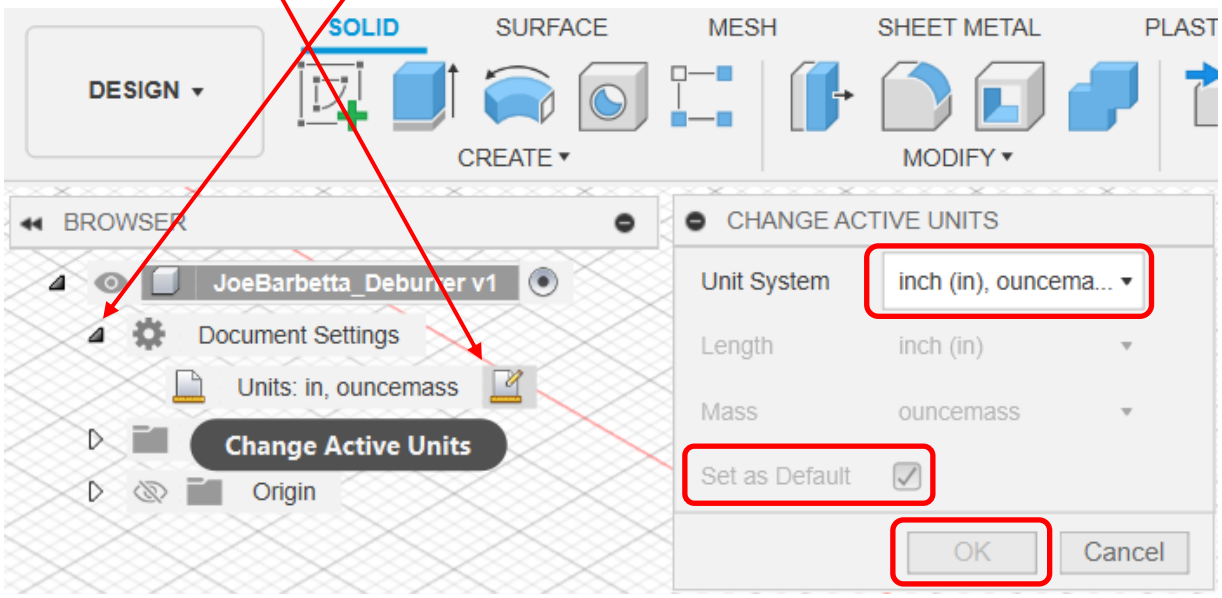
Starting a Design in Fusion (START HERE)

- open **Fusion**. If there is no icon on the Desktop, use the Windows search (magnifying glass icon) and type fusion
- from top **File** icon select **Save** and name the file.
  Use your name followed by **_Deburrer** e.g. **JoeBarbetta_Deburrer** (note the use of the underscore)

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

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

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

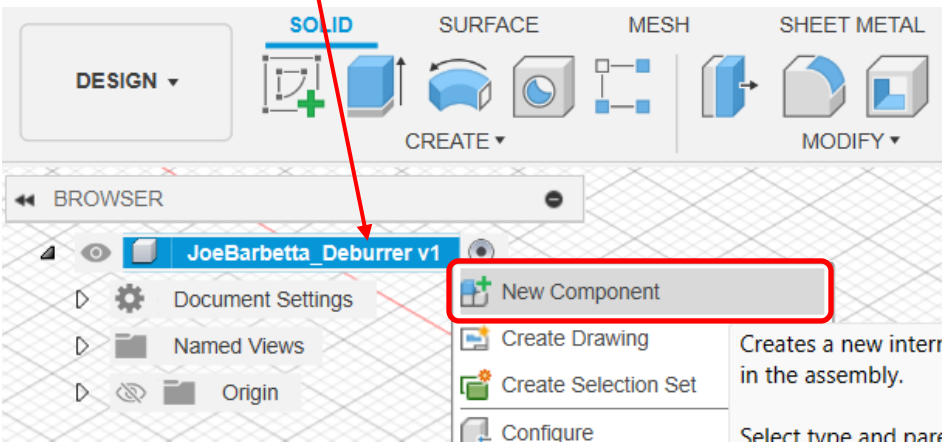


Note that the default units are in mm, which we just changed to inches.

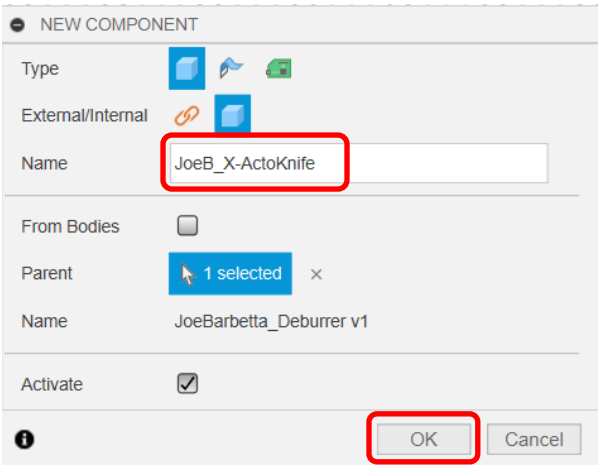
Did you know that the default units have changed over the years? The earliest version used cubits as the default unit.

Creating a Component for the X-Acto Knife

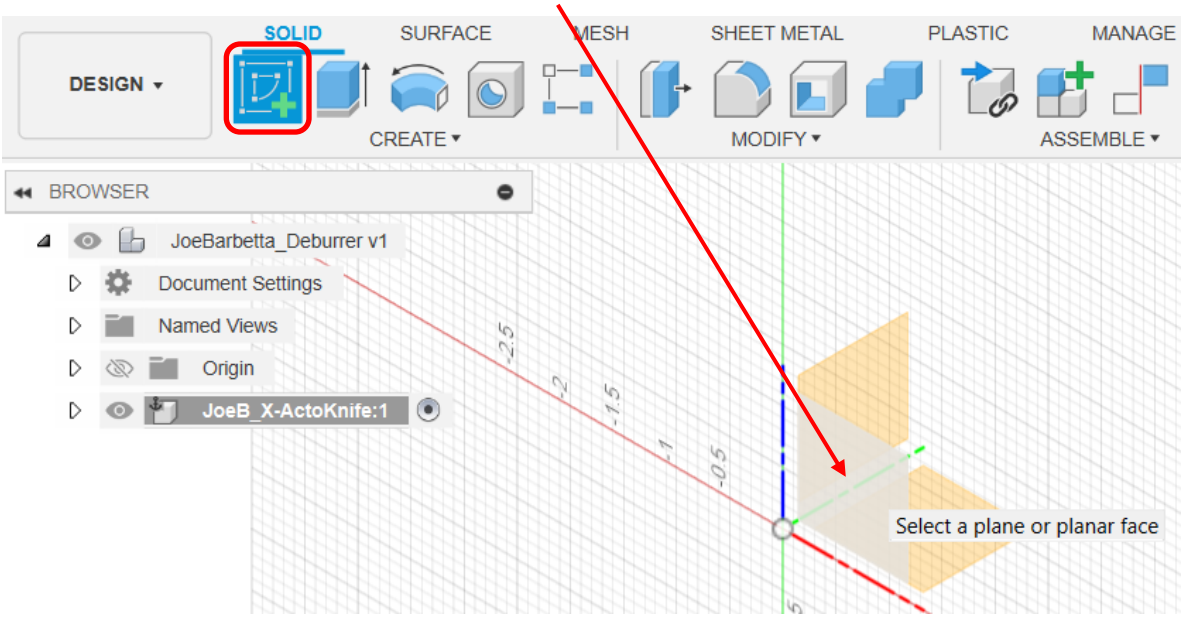
- right-click on the project name and select **New Component**



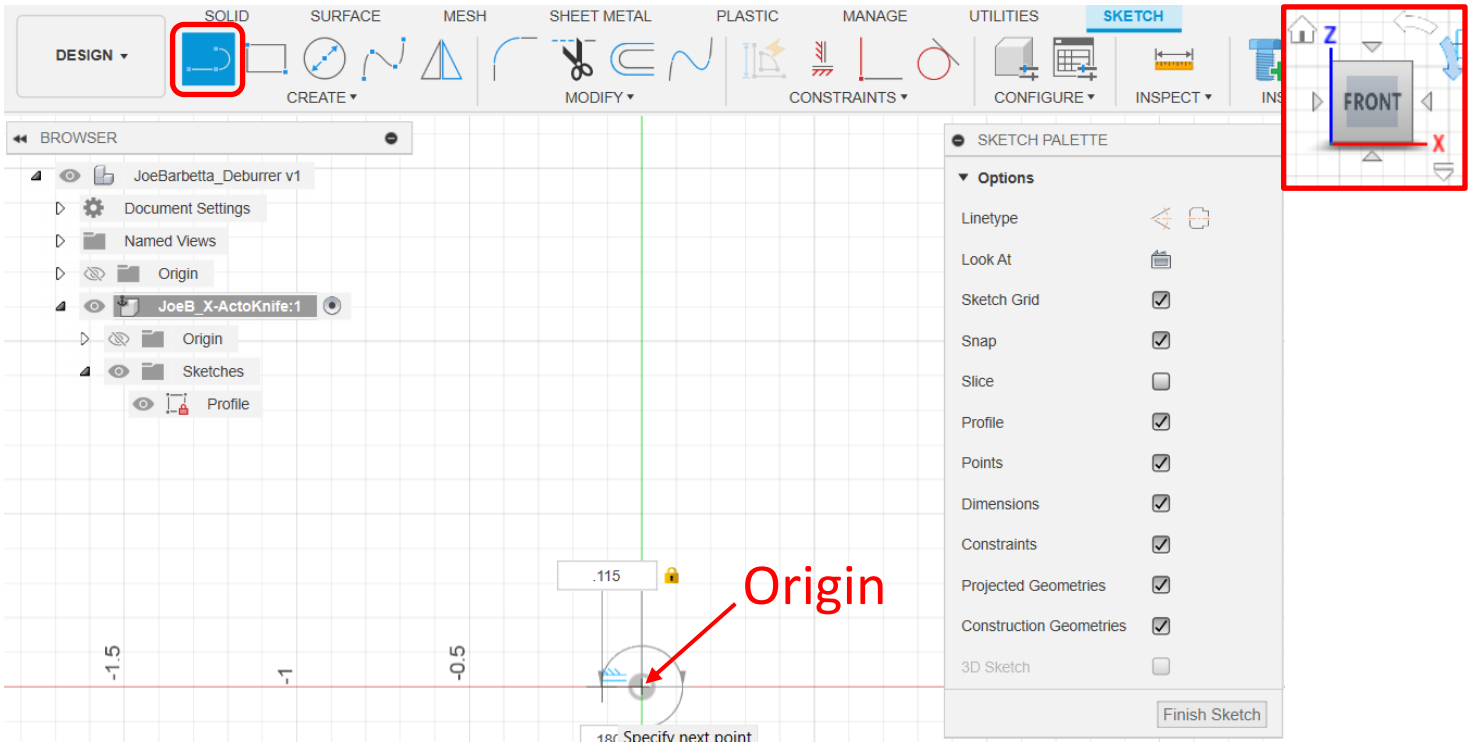
- name it using your **first name**, followed by the **initial of your last name**, followed by **_X-ActoKnife**, e.g**. JoeB_ X-ActoKnife**
- click **OK**



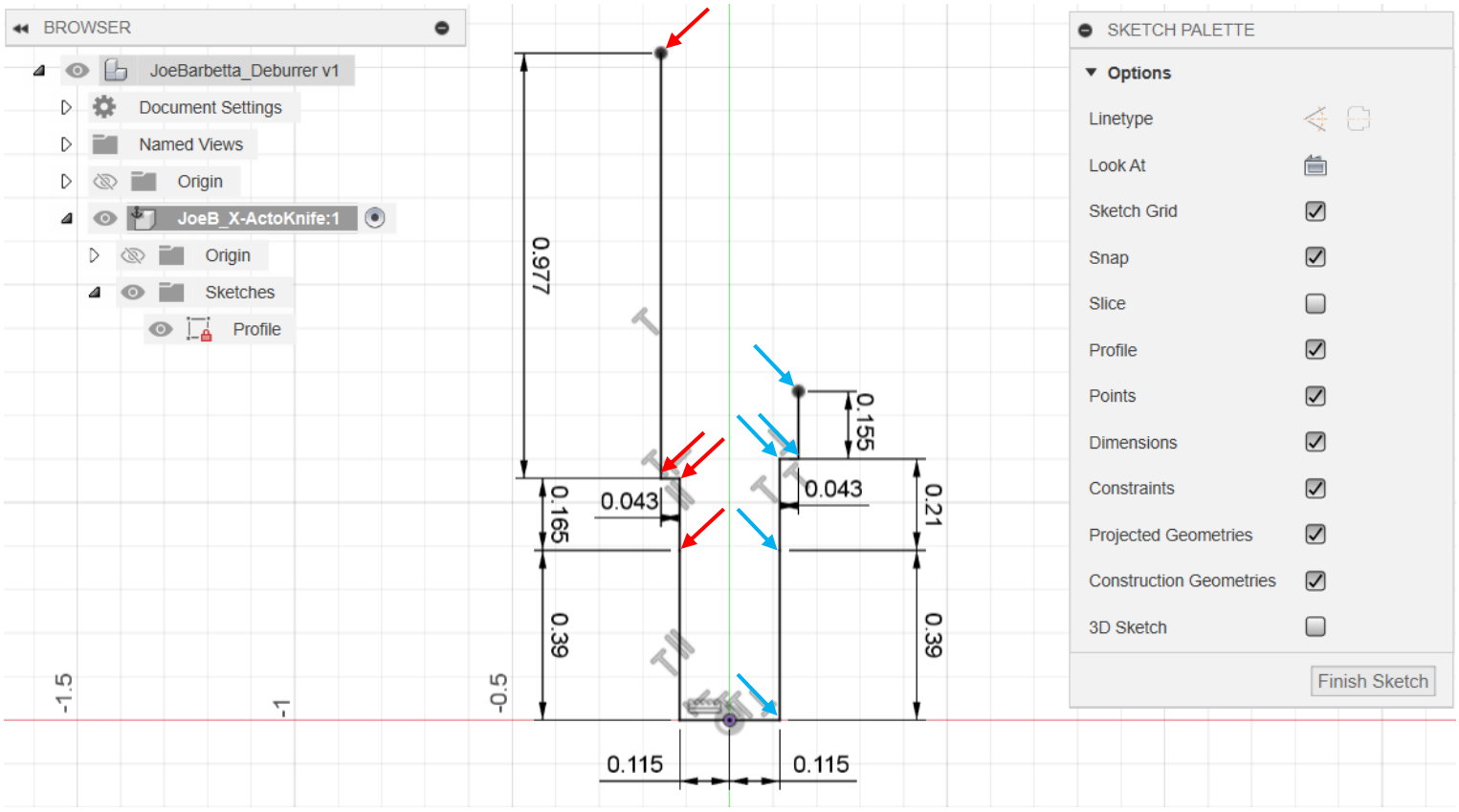
- select **Create Sketch** and click on the **front rhombus**



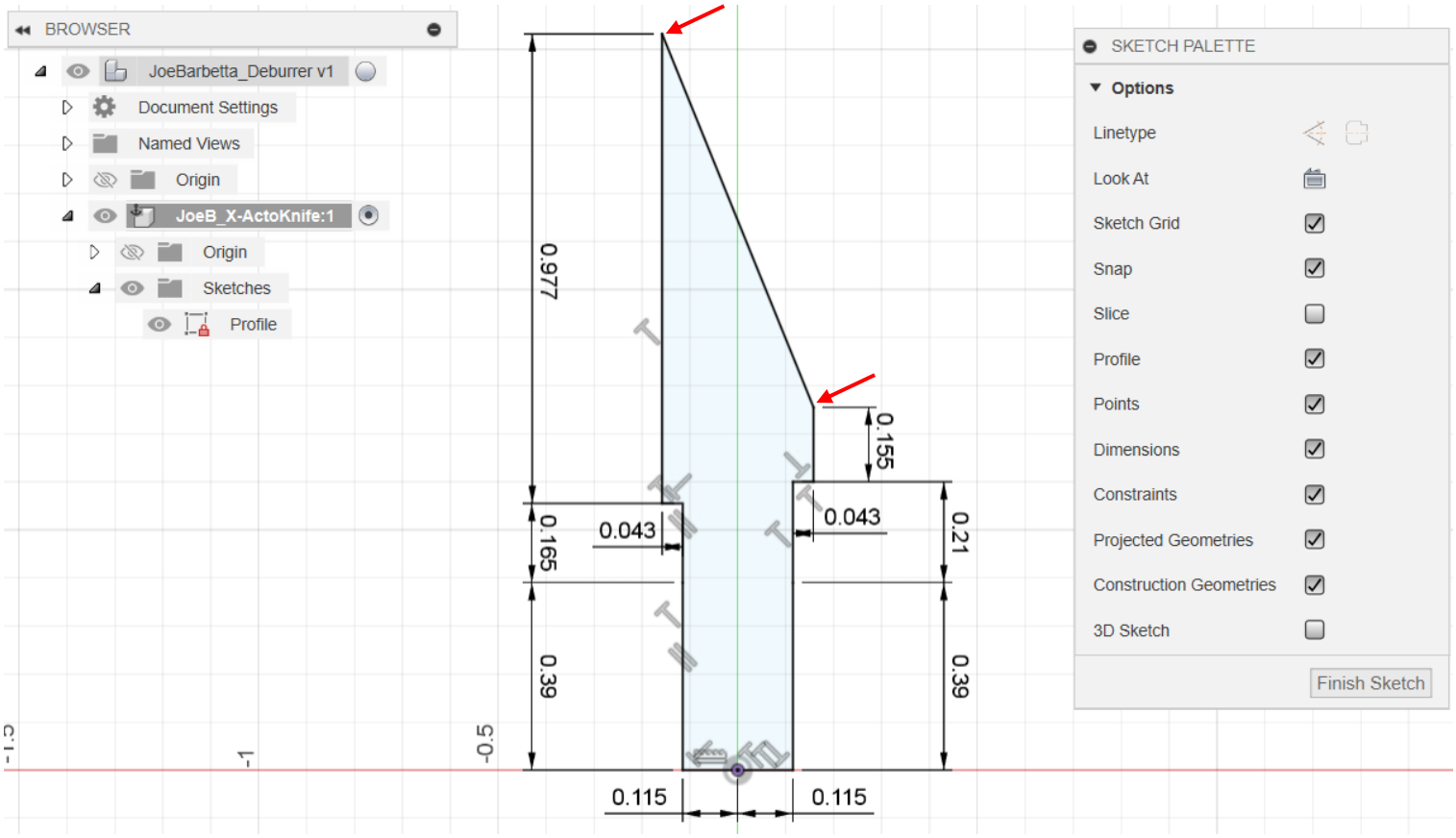
- the **View Cube** should show **FRONT**. If the view needs to be rotated click on the curved arrows at the **View Cube**
- zoom and pan to achieve a view similar to that below. Note the values on the red axis.
- select the **Line** tool. Note that if a tool icon is not visible, the tool can be found in the CREATE menu.
- click on the **origin**, extend the line over to the left, type **0.115**, and press the **Enter Key**



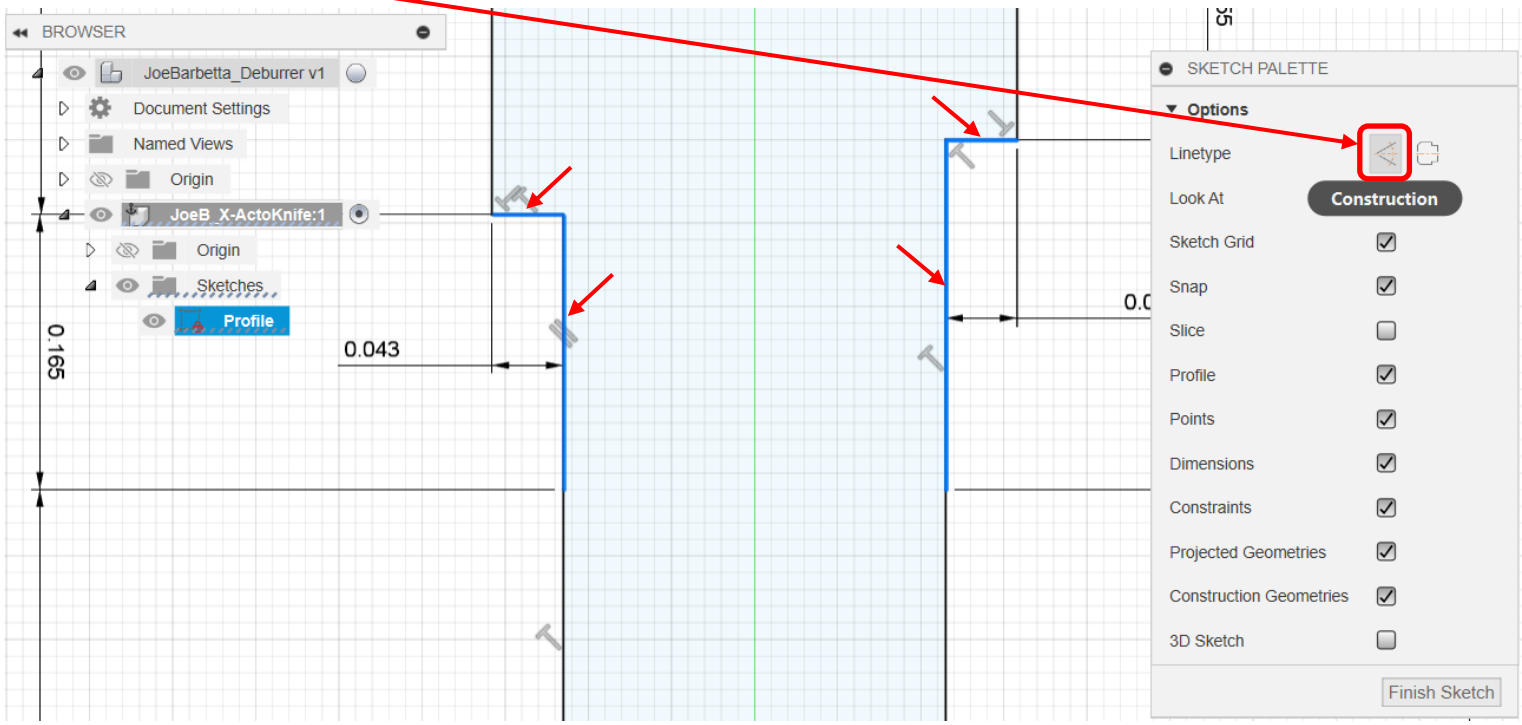
- from the **end of the line just created**, create a **0.39** line upward, then from there another line **0.165** upward, then a **0.043** line extending to the left, and then a **0.977** line upward. The red arrows show the ends of each of these lines.
- from the **Origin** draw a **0.115** line to the right, a **0.39** line upward, a **0.21** line upward, a **0.043** line to the right, and a **0.155** line upward. The blue aoorws show the ends of each of these lines.



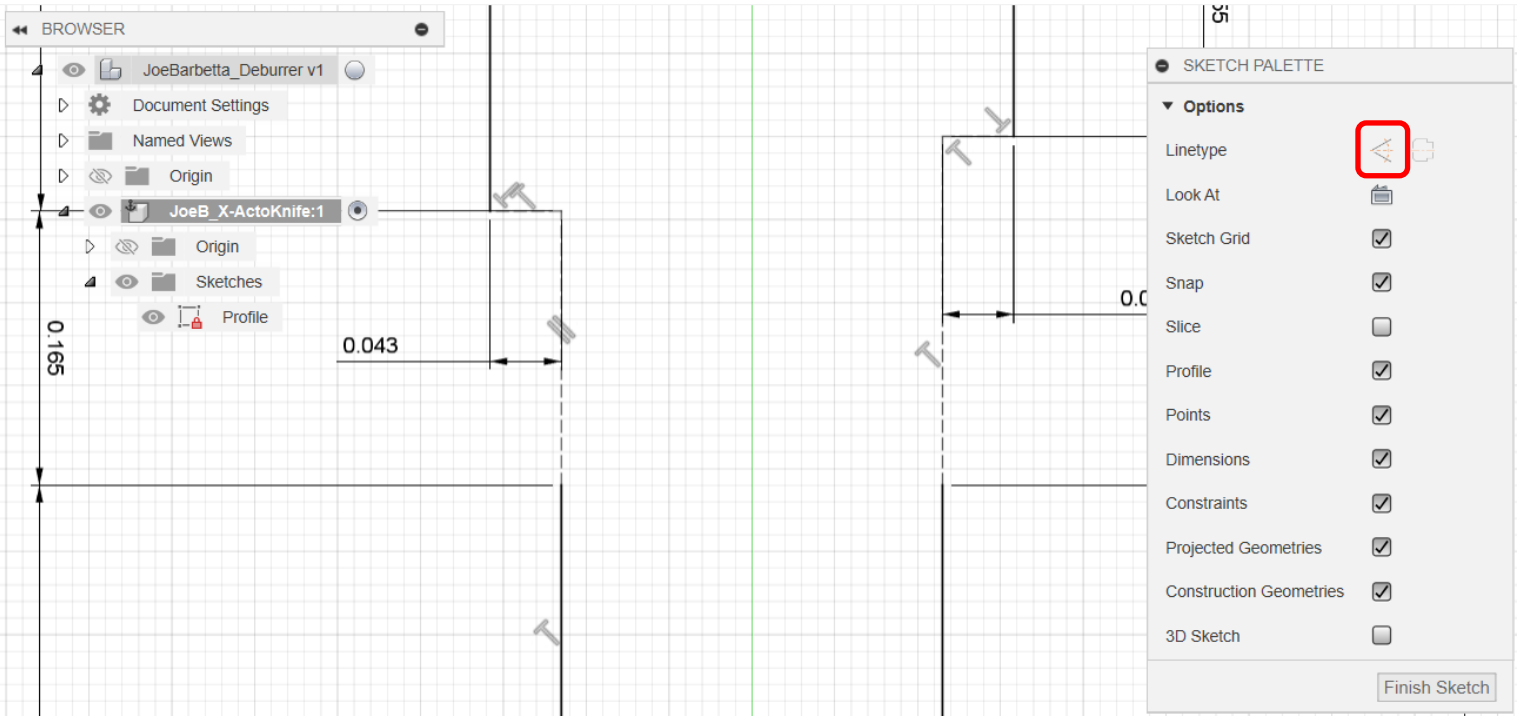
- close the shape by creating a diagonal line connecting the two points shown.



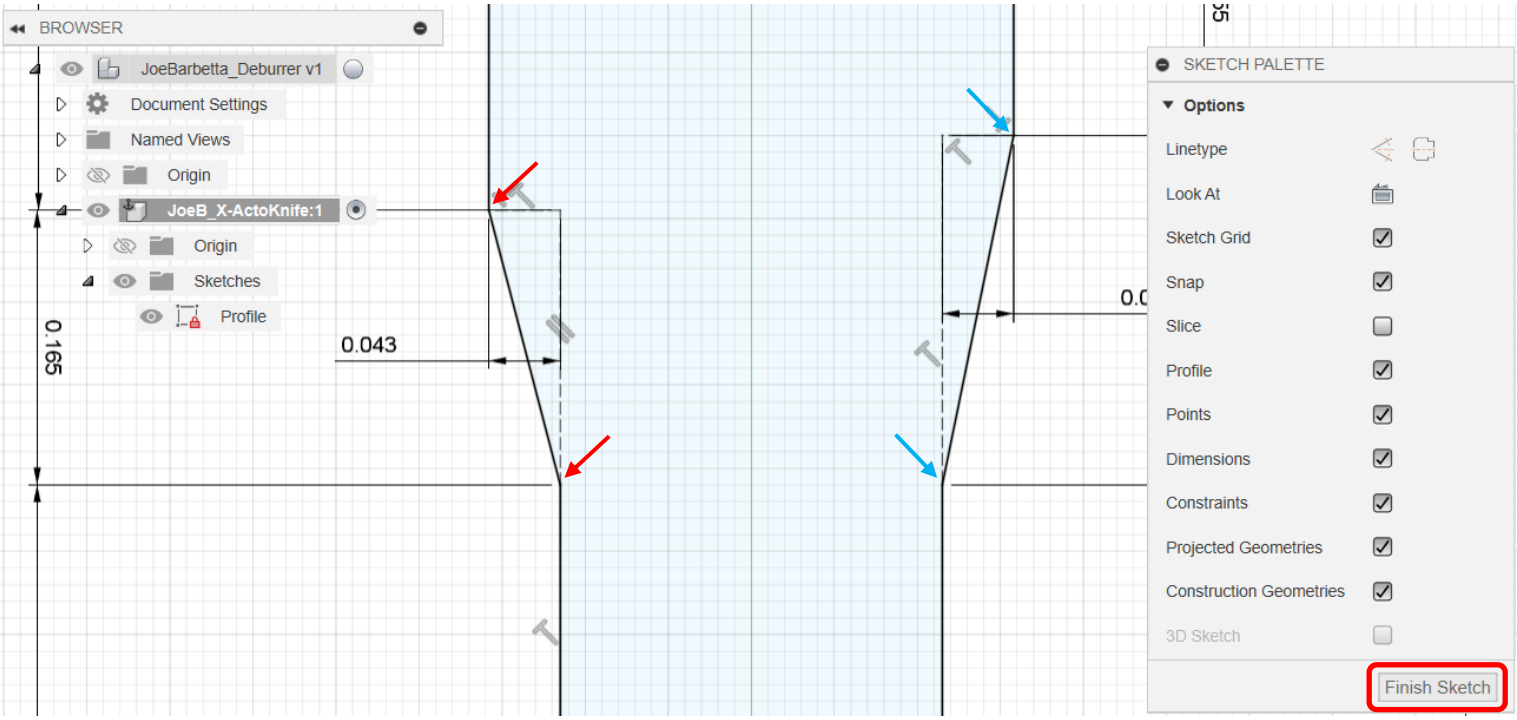
- hold the **Shift key** down and click on the **four line segments** as shown, which should highlight them blue.
- click on the **Construction** icon, which should cause them to become dashed lines.



- click **anywhere on the background** and then click on the **Construction** icon again to remove the icon's highlighting
Note that the profile is not filled anymore.

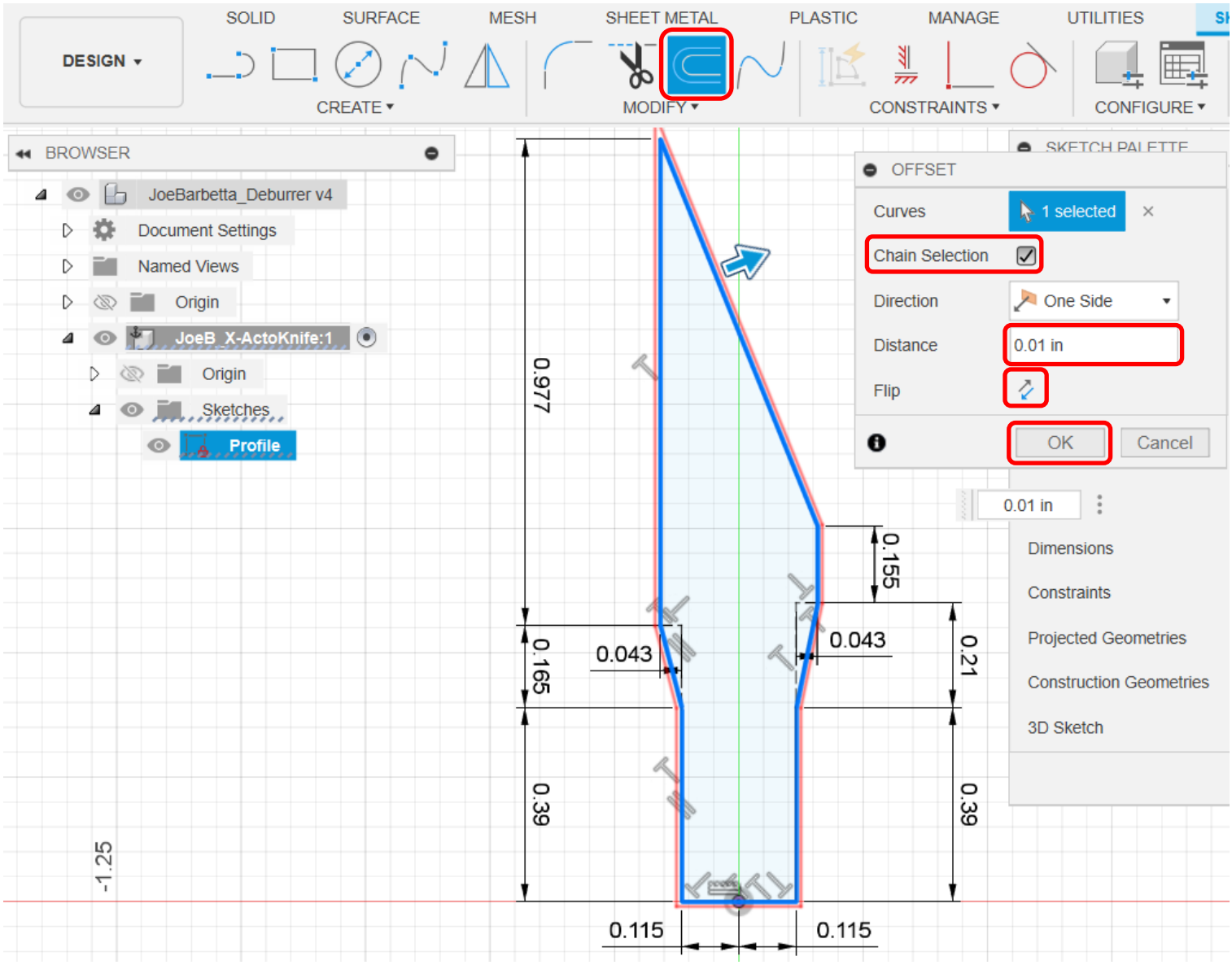


- create two diagonal lines connecting the points shown
- click on **Finish Sketch**

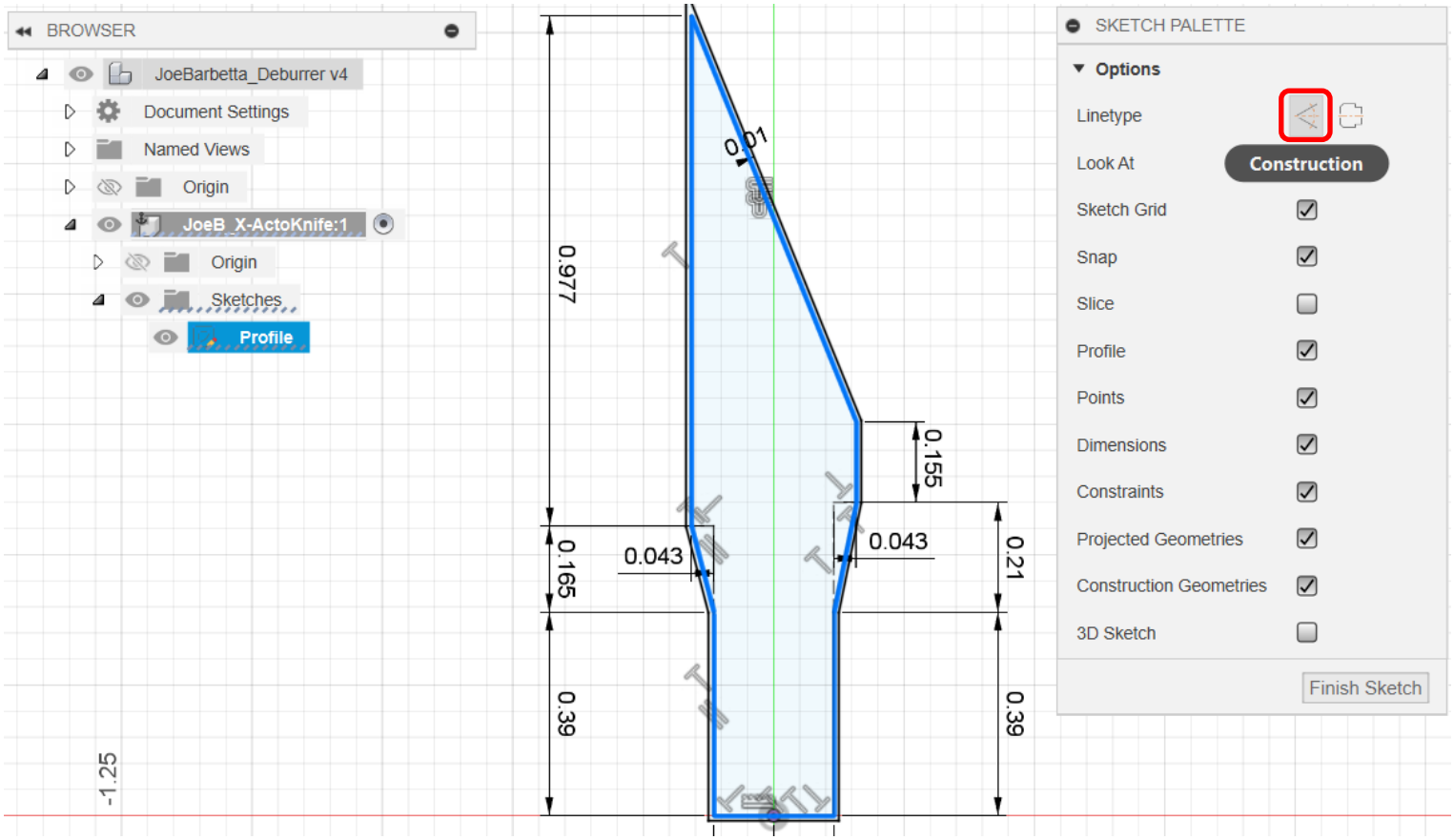


- zoom out to achieve a view similar to that below
- select the **Offset** tool. If its icon is not visible, find it in the **MODIFY** menu.
- ensure that **Chain Selection** option is checked, click on the **outline** of the knife blade, which should highlight it blue.
- enter a distance of **0.01**. The **red outline should be outside that of the blue**. If it is inside, click on the **Flip** icon.
- click **OK**

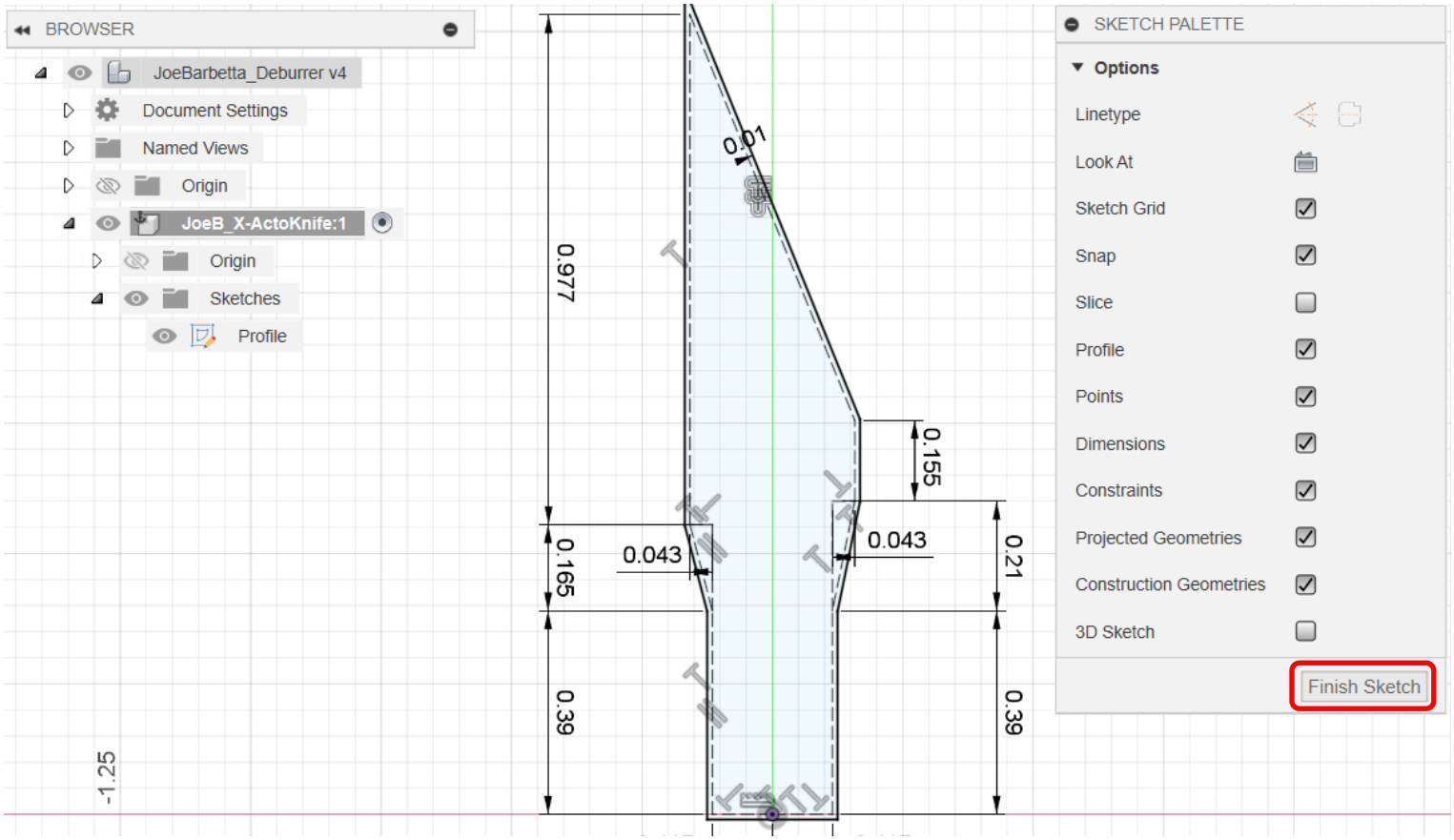
The reason this was done was to add some clearance between the blade and the shroud. We could have originally drawn the blade profile with larger dimensions, but using the offset allows adjusting the clearance if needed.



- double-click on the **original profile**, which is now the inner profile, to highlight it blue
- click on the **Construction** icon, which should cause these lines to become dashed

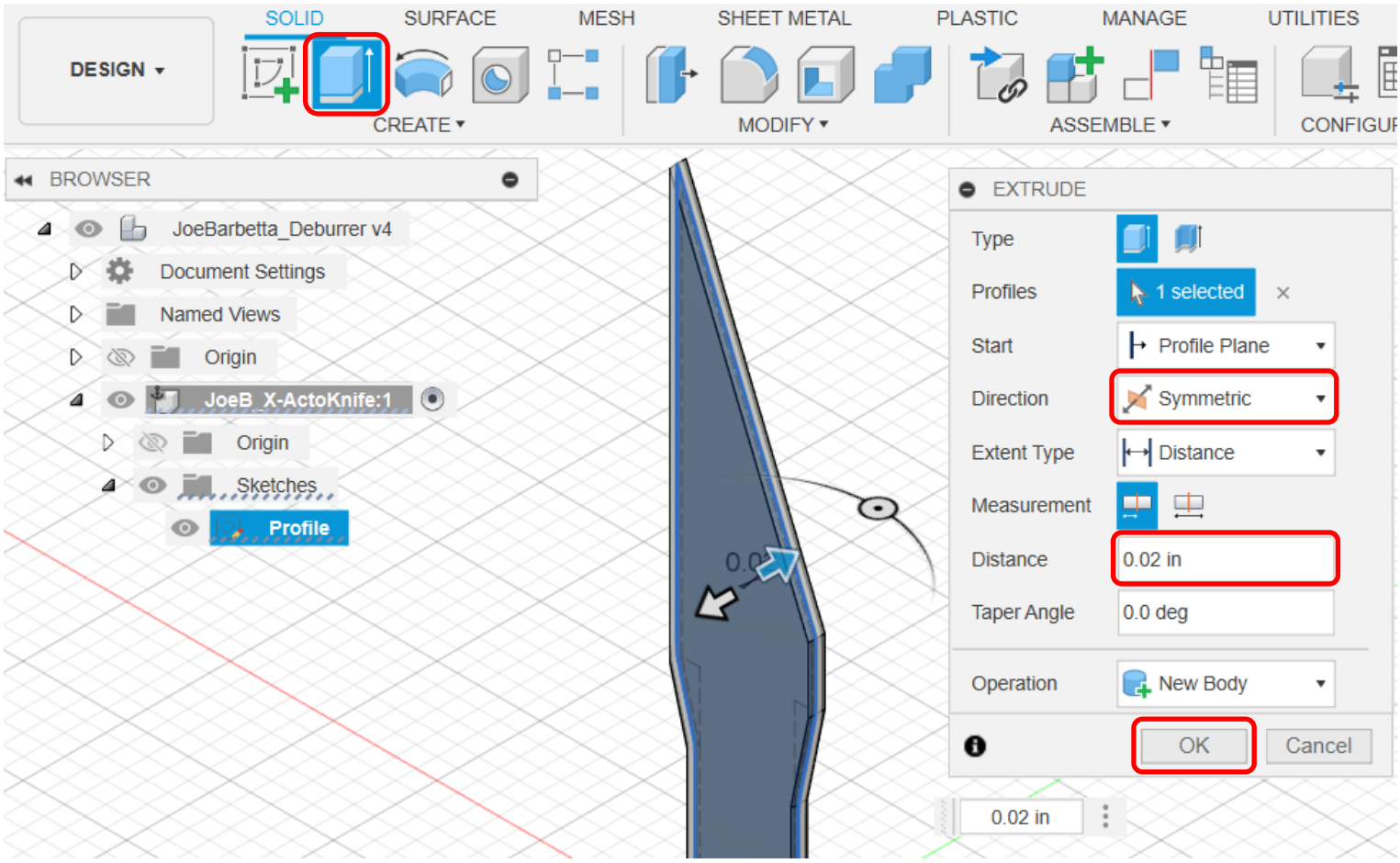


- click **Finish Sketch**

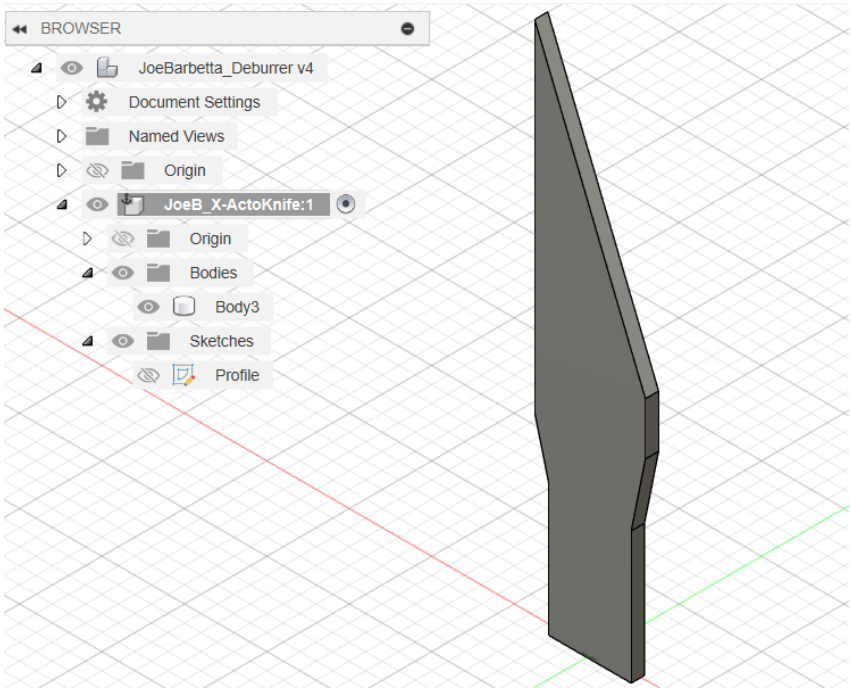


- click on the **Home** icon near the **View Cube** and zoom to achieve a view similar to that below
- select the **Extrude** icon, change **Direction** to **Symmetric**, and **distance** to **0.02**.
- click **OK**

Because the Direction is set to Symmetric, the final thickness will be 0.04. Note that an actual X-Acto blade is 0.02 thick. The extra 0.01 on each side is for clearance.

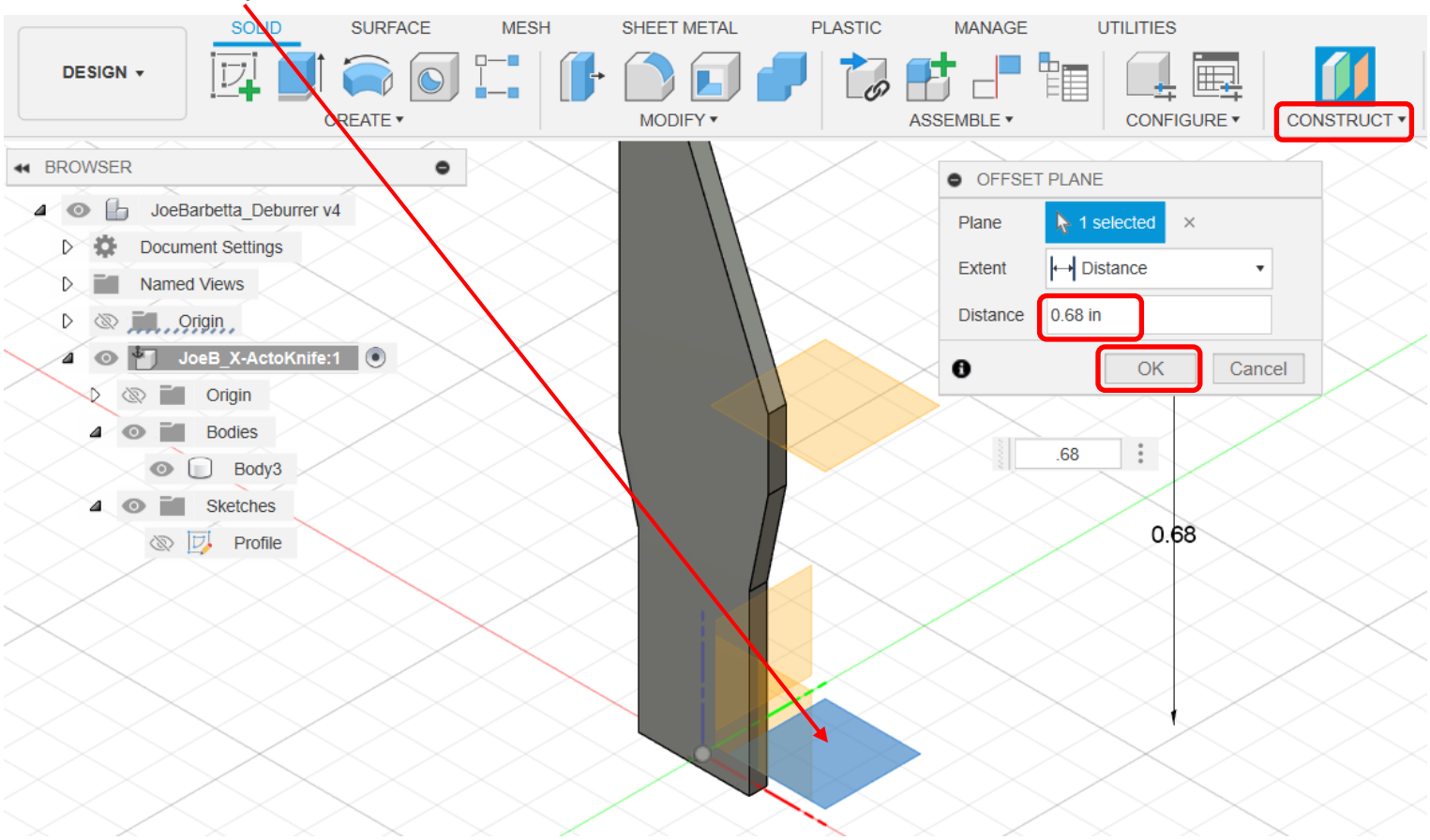


This is the finished knife blade. There is no need to add chamfers for the sharp edge because this is just being used to cut away from a body that will become the shroud.

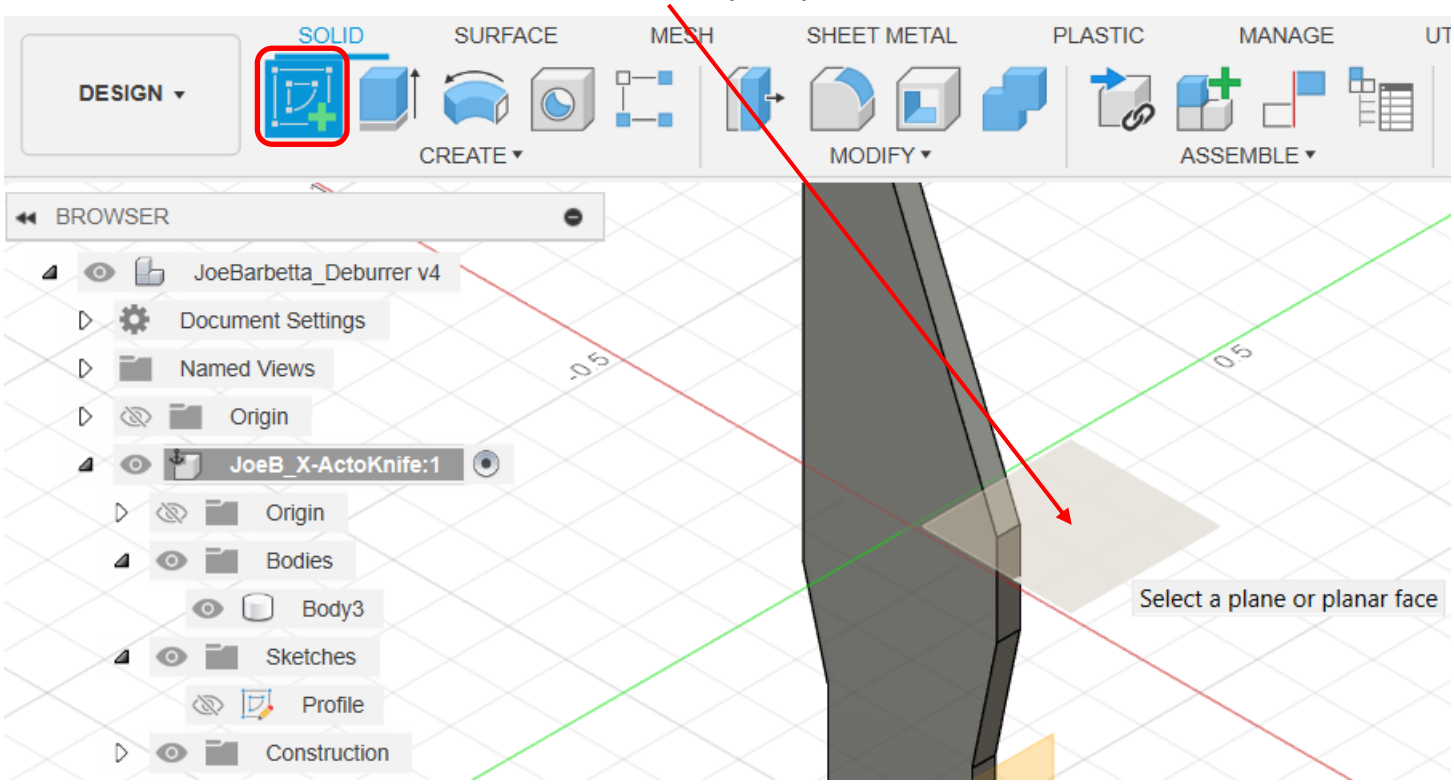


Creating the Handle

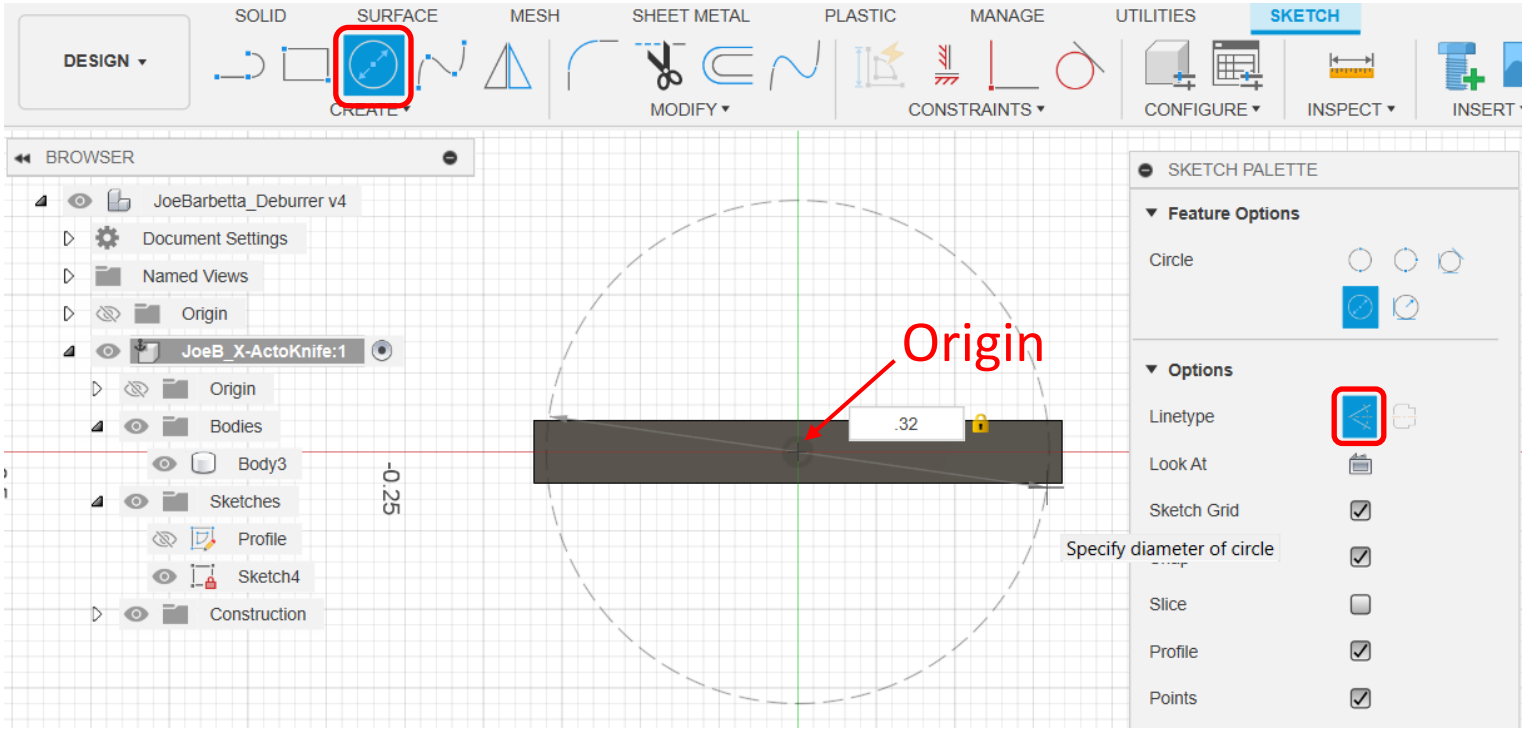
- from the **CONSTRUCT** menu select **Offset Plane**. It should be the first item in the list.
- click on the **bottom plane** and enter **0.68** for **Distance** and click **OK**



- select the **Create Sketch** tool and click on the **Construction plane** just created

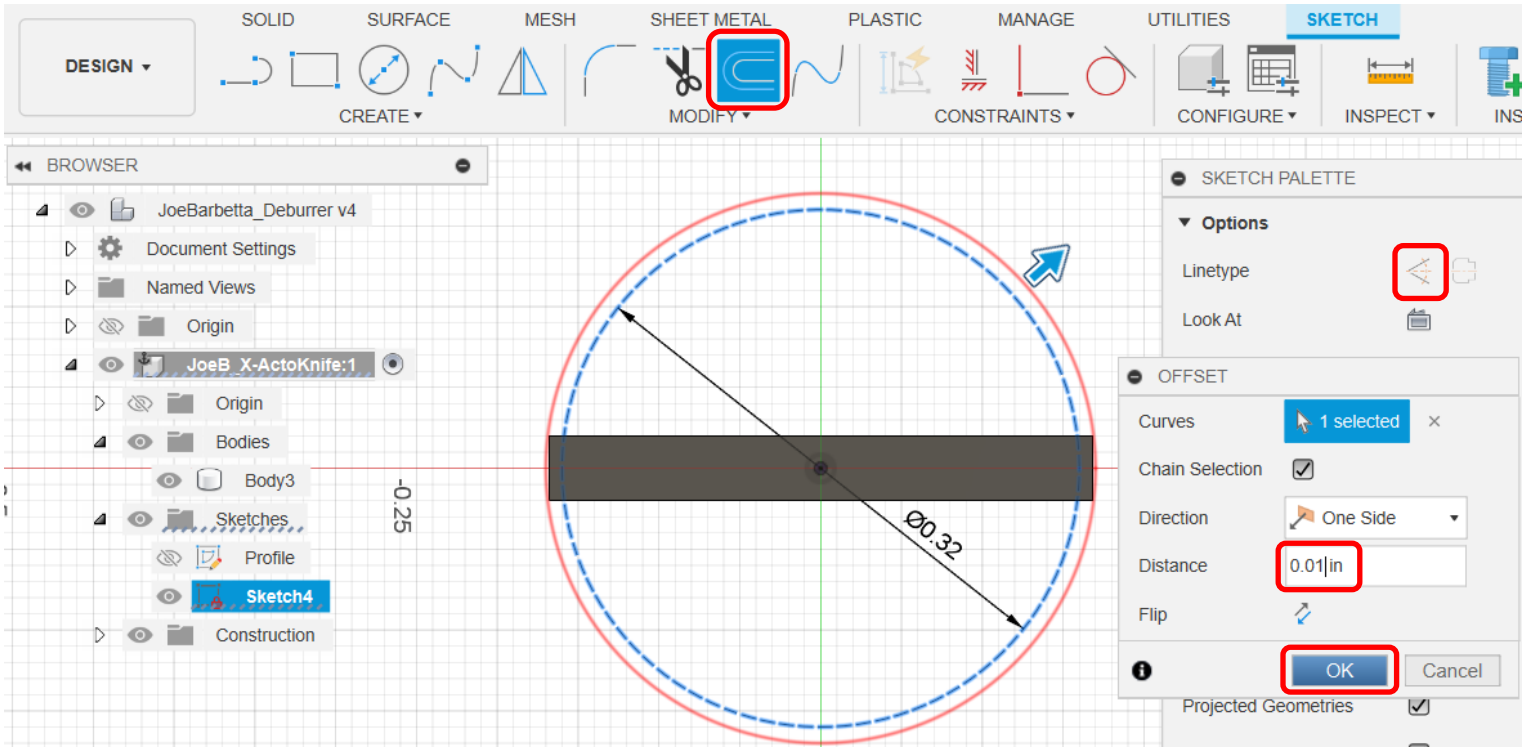


- zoom in to achieve a view similar to that below
- select the **Center Diameter Circle** tool. If the icon is not visible find it in the **CREATE** menu.
- click on the **Origin**, which should be in the center of the rectangular blade section
- click on the **Construction** icon to highlight it, extend the circle out, type **0.32**, and press the **Enter key**

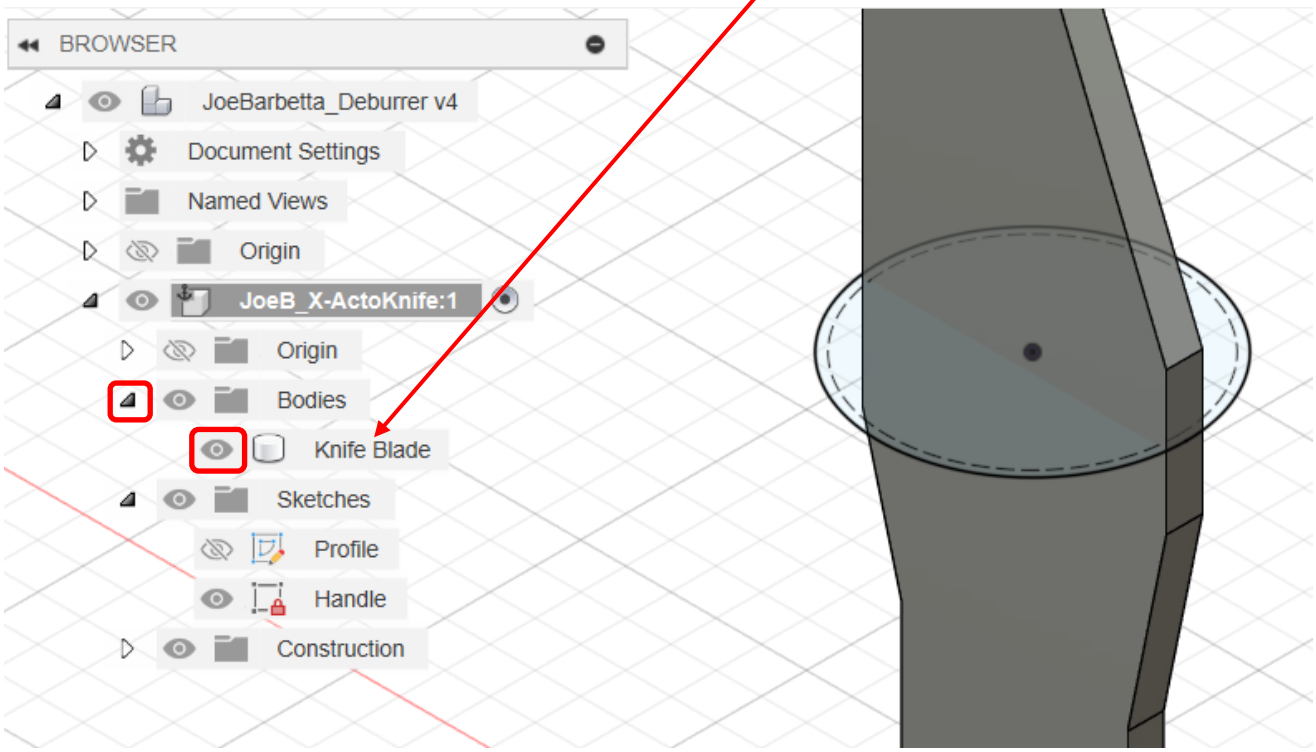


- click on the **Construction** line icon to turn off the highlighting
- select the **Offset** tool and click on the **dashed circle**, enter **0.01** for **Distance**.
- The red circle should be outside of the blue dashed circle. If it is inside, click the Flip icon.
- click **OK** and then **Finish Sketch**

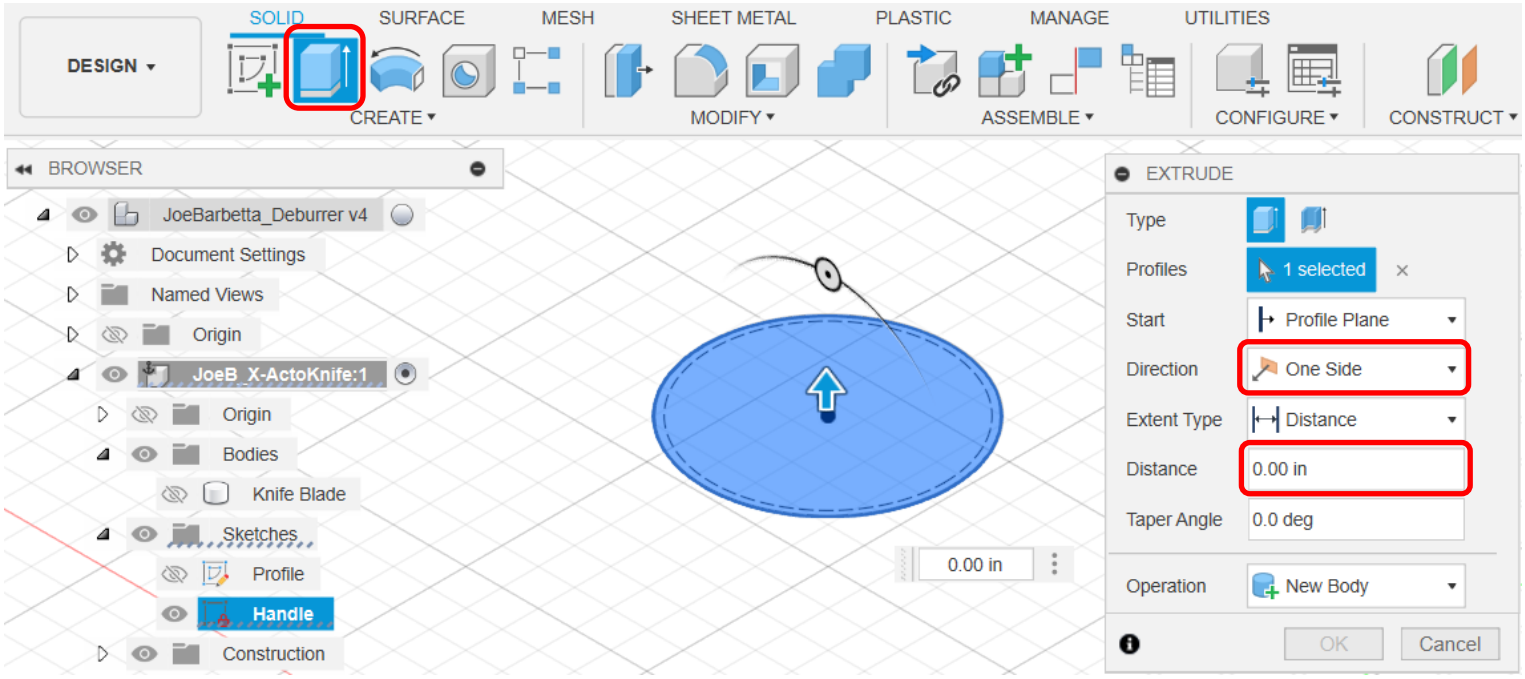
This 0.01 is also a clearance value.



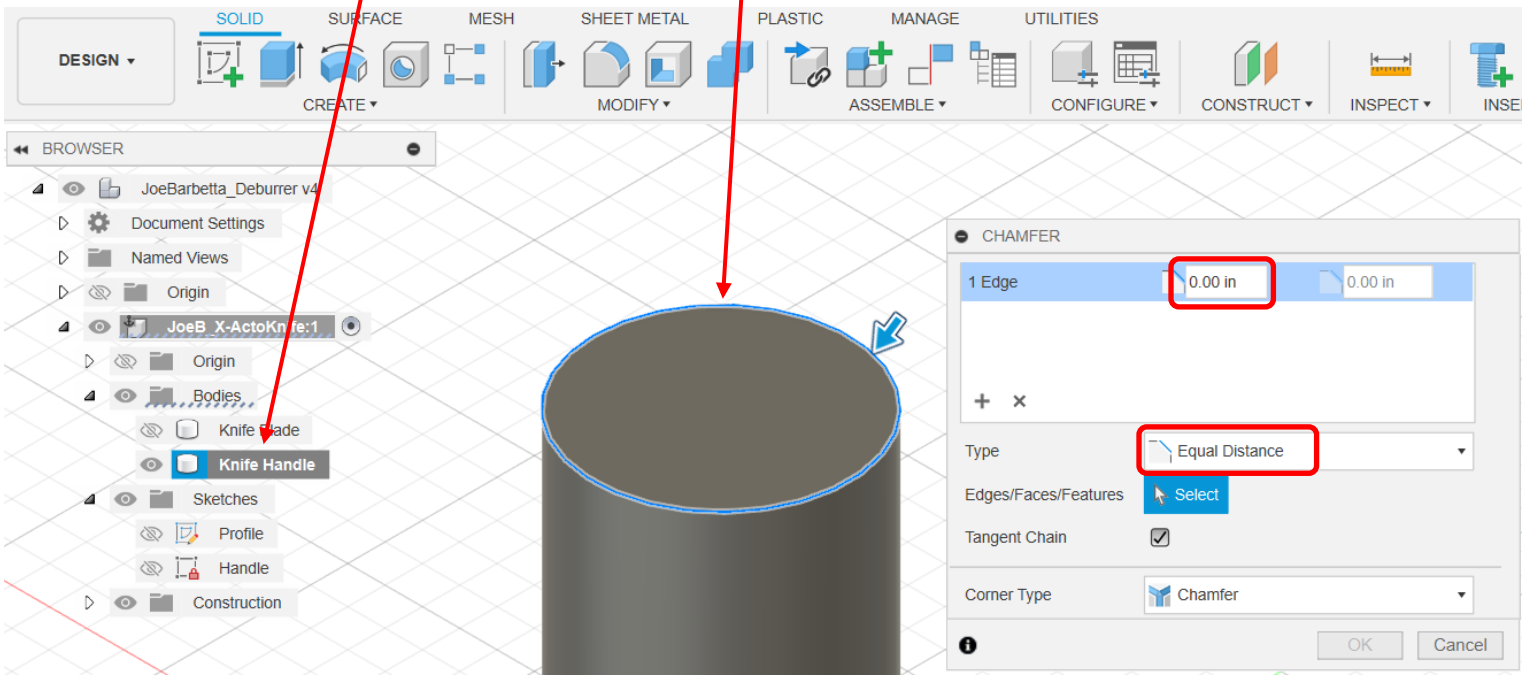
- zoom in to achieve a view similar the that below
- click on the arrow to open the **Bodies** folder and right-click on **Body1** and select **Rename** to change the name to **Knife Blade**
- click on the **eye** icon next to the body to hide it



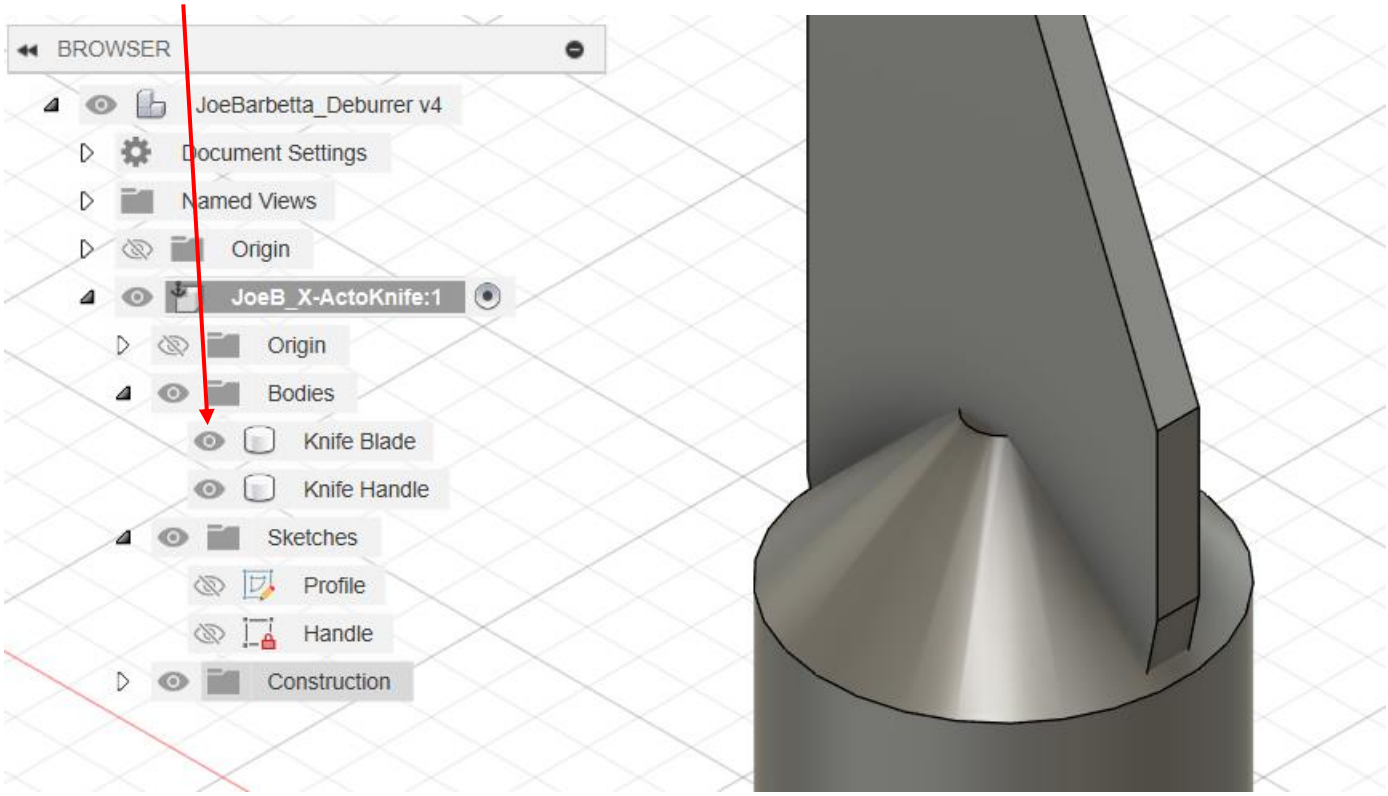
- select the **Extrude** tool
- ensure the **Direction** is **One Side**
- enter **-1.0** (note the minus sign), which should cause the circle to be extruded downward
- click **OK**



- rename the new Body to **Knife Handle**
- from the **MODIFY** menu select the **Chamfer** tool
- ensure the **Type** is set to **Equal Distance** and click on the **top edge** of the cyclinder
- enter **0.14** and click **OK**

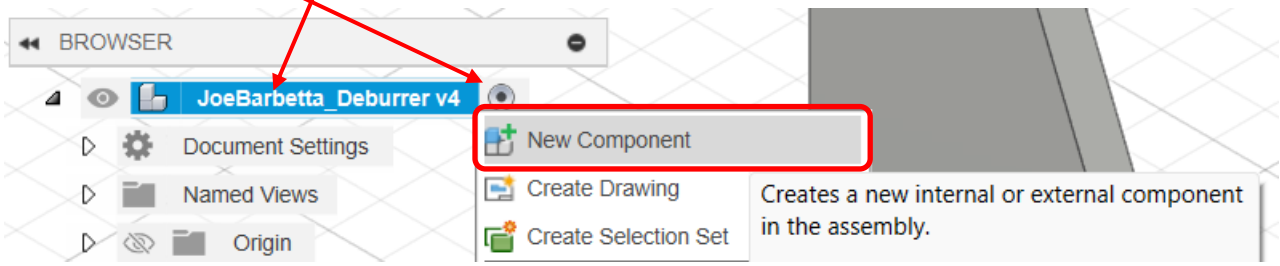


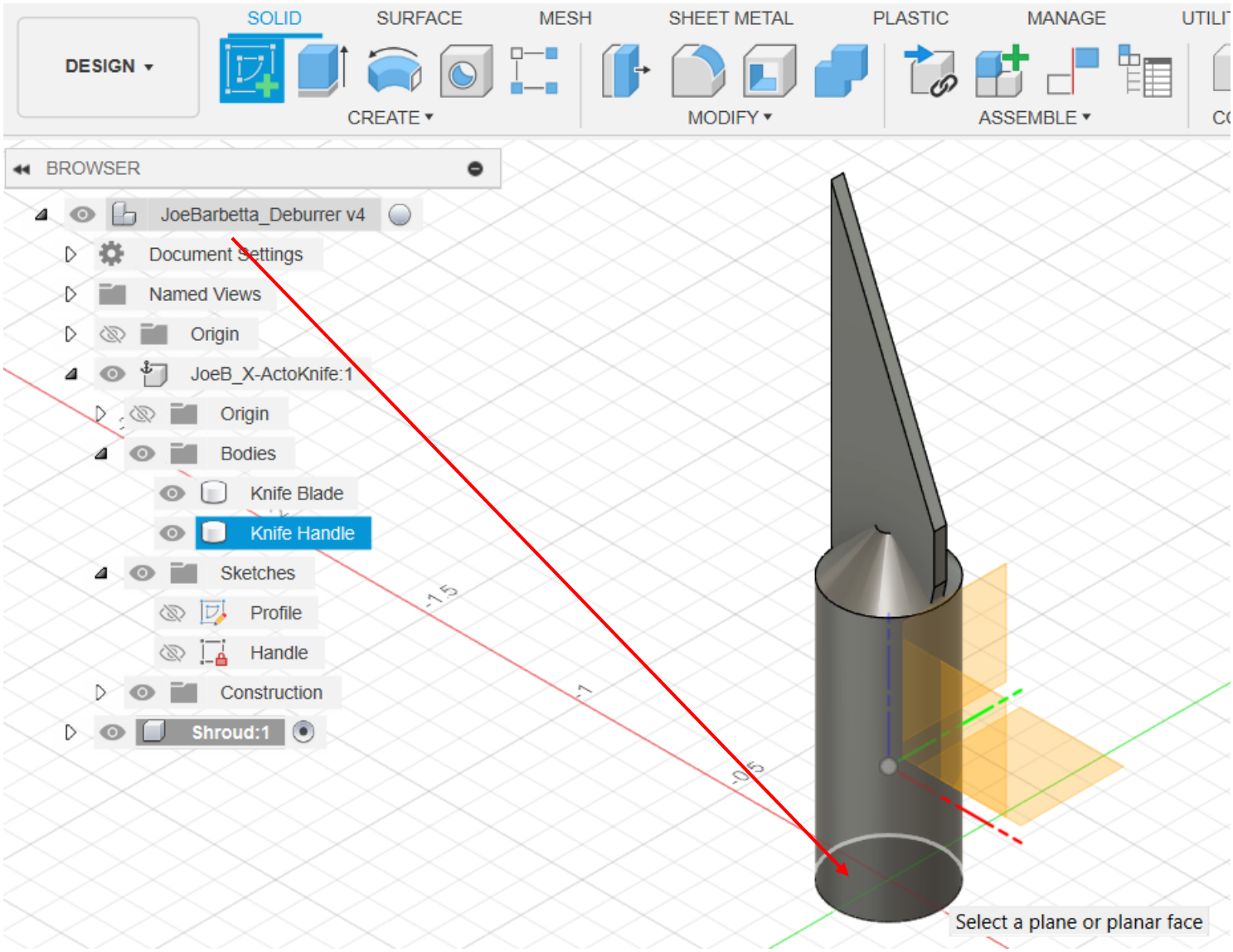
- click on the **eye** icon for **Knife Blade** to make it visible



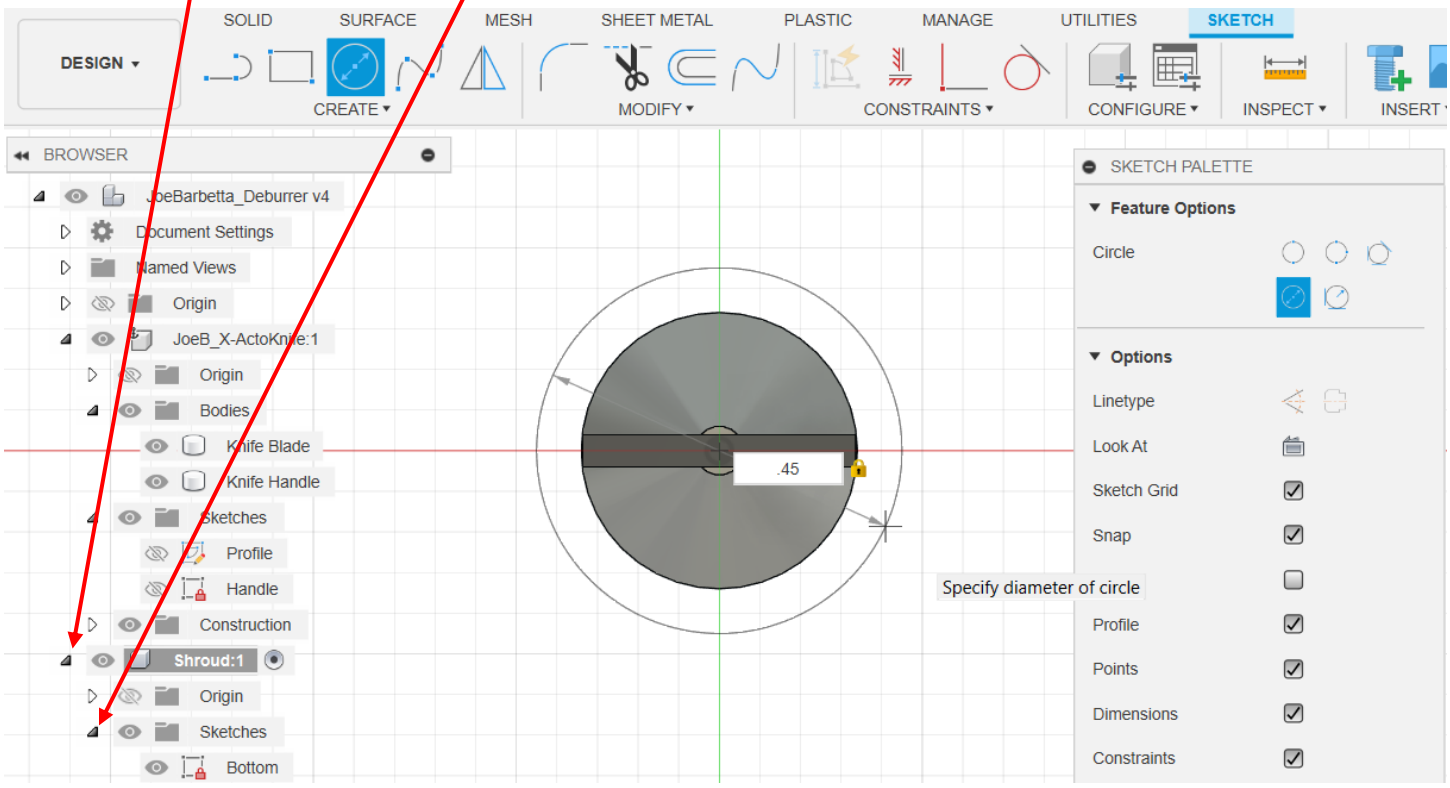
Creating the Shroud Body

- click on the **circle** for the project to activate it
- right-click on the **project name** and select **New Component**

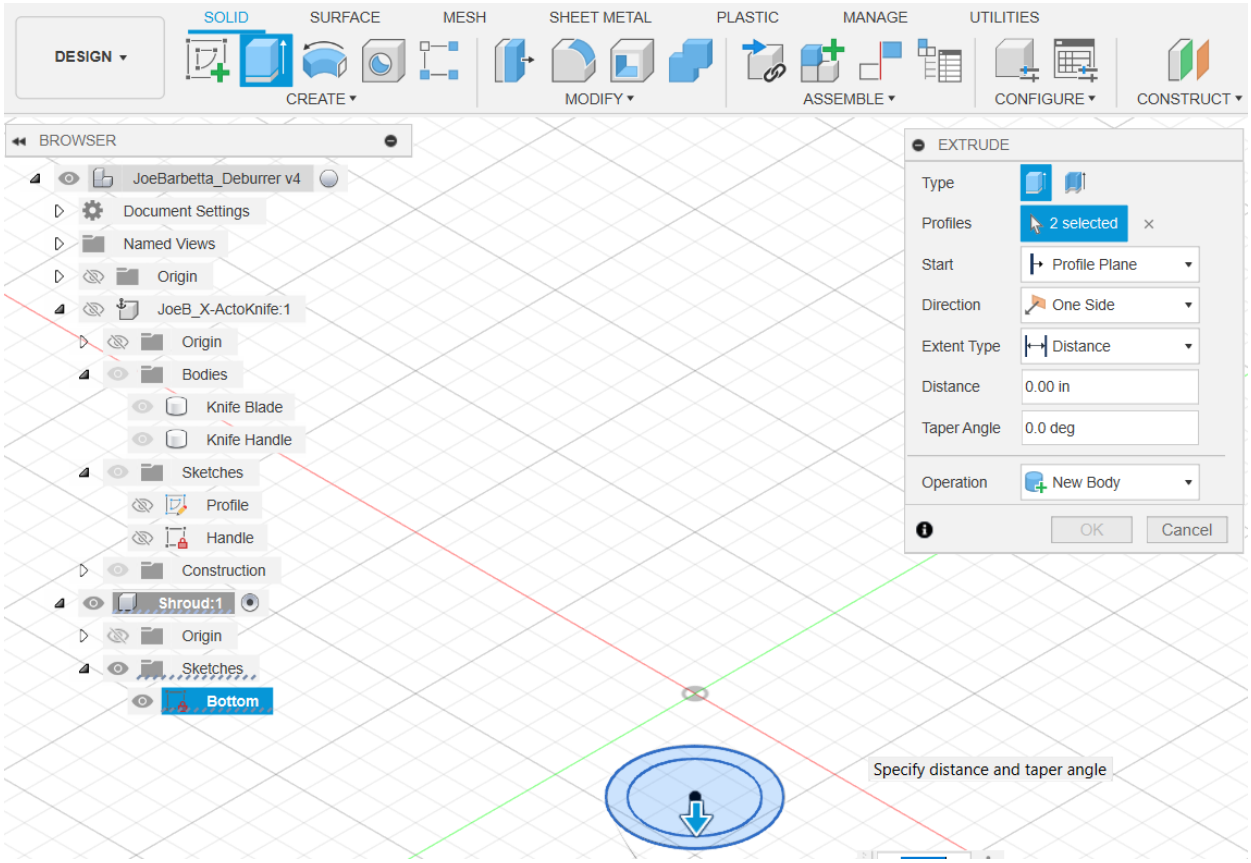




- open the **Shroud folder** and its **Sketches folder** and rename the Sketch to **Bottom**
- select the **Center Diameter Circle** icon, click on the **Origin**, extend the circle out, and enter **0.45**, and click **Finish Sketch**

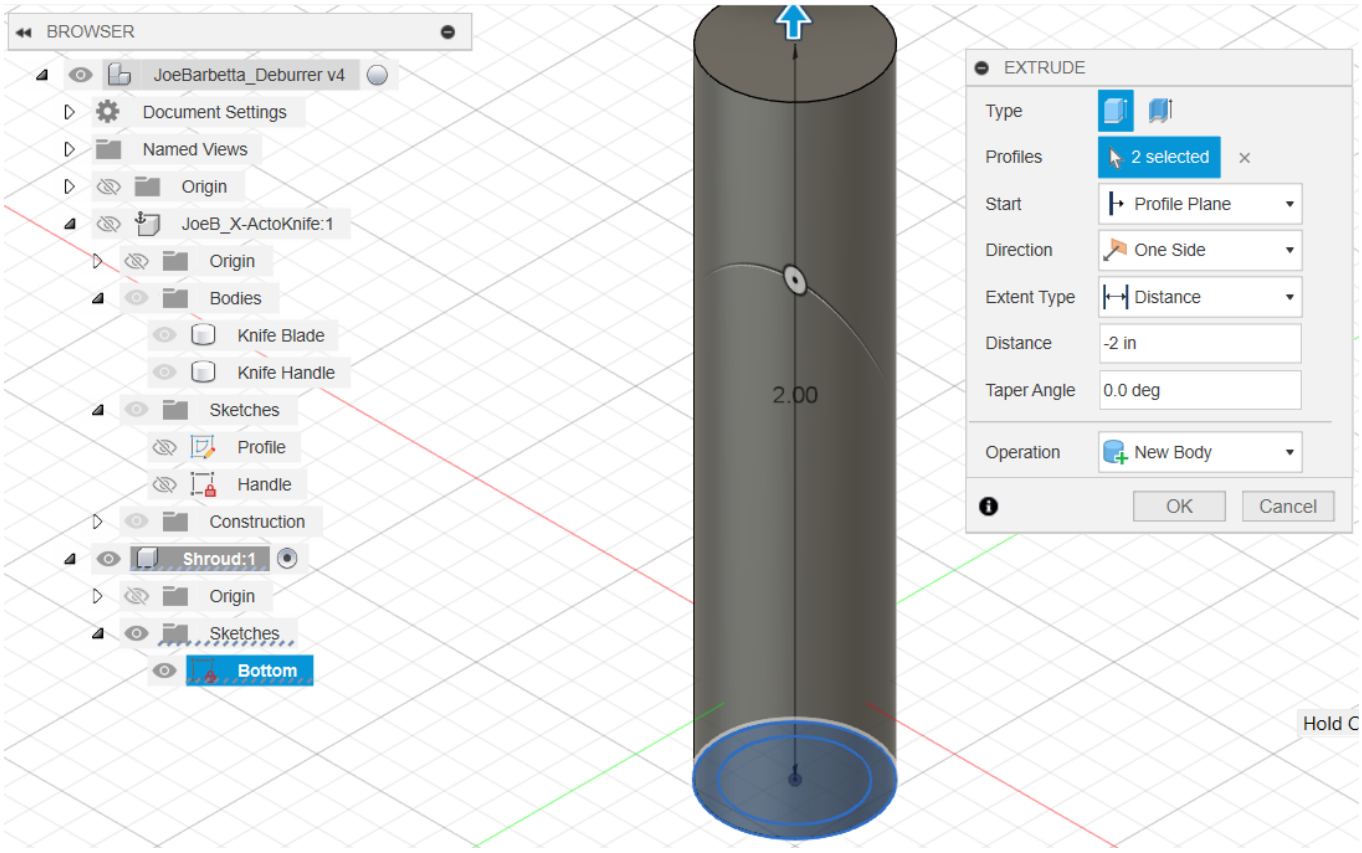


- return to the Home view and click on the eye icon for the X-ActoKife Component to hide it
- select the Extrude icon and click on both the inner circle and outer circle to highlight both

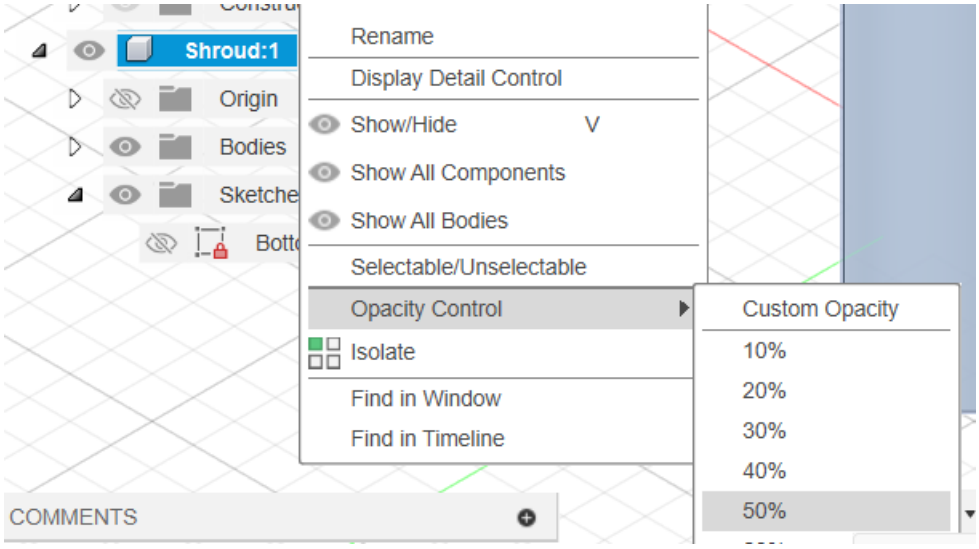


- enter -2 (note the minus sign) to extrude the cylinder upwards. If the cylinder extrudes down, then use a positive value.

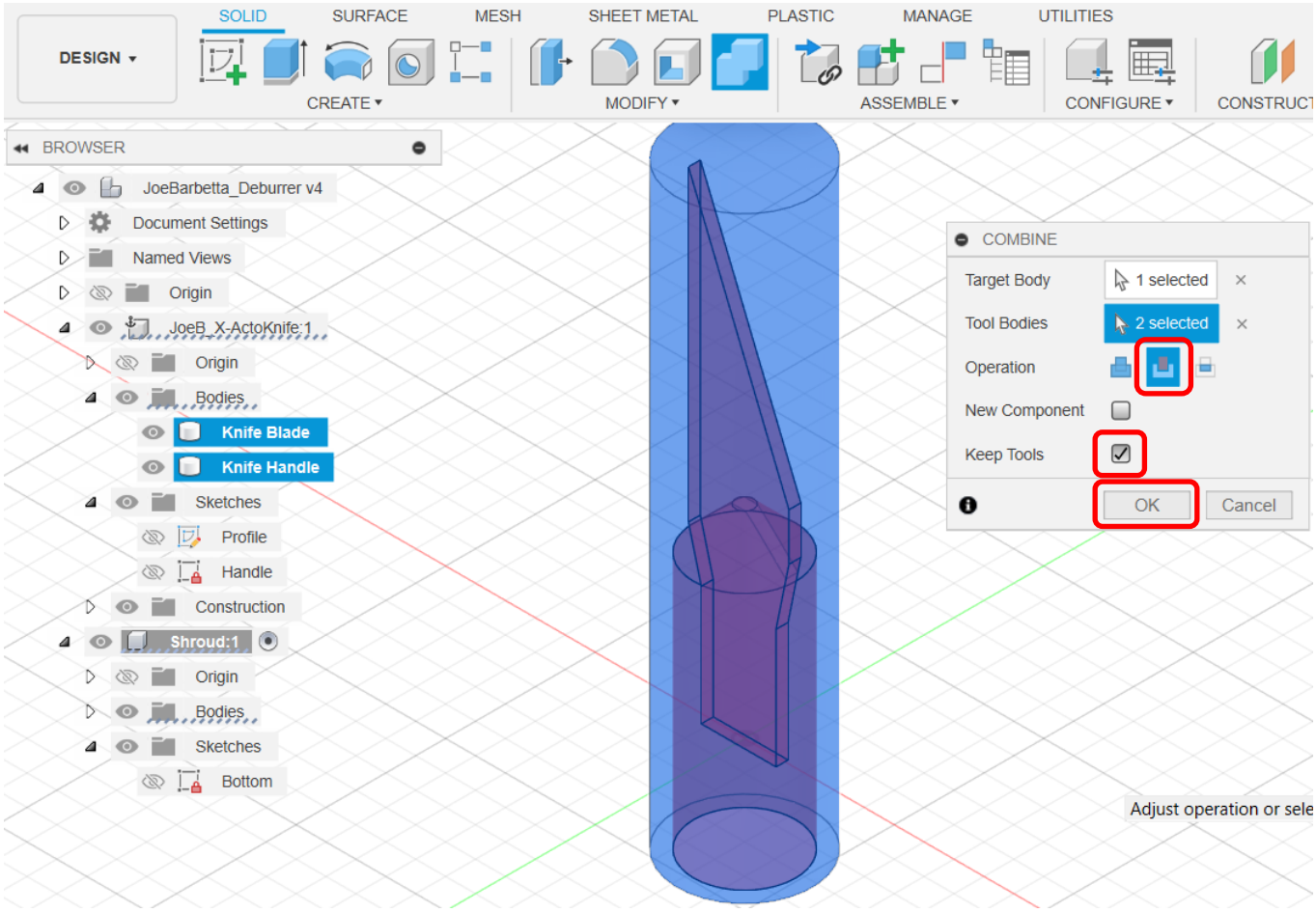
- click OK



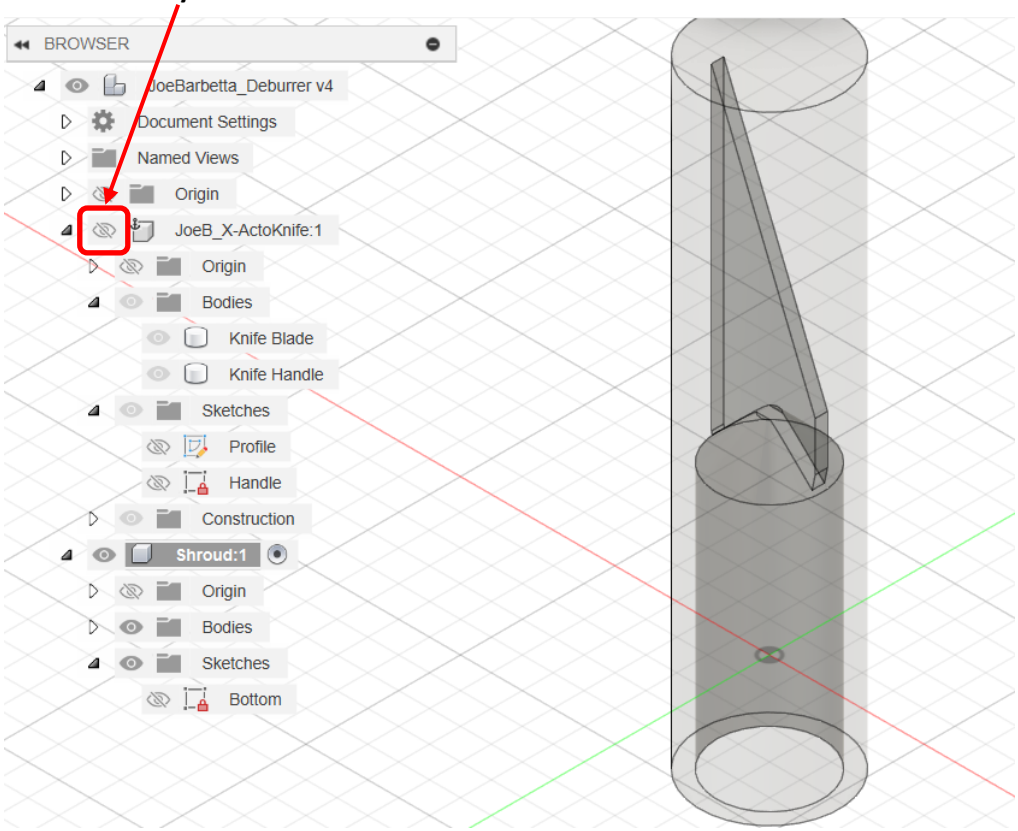
- right-click on the **Shroud** Component, select **Opacity Control** and **50%**



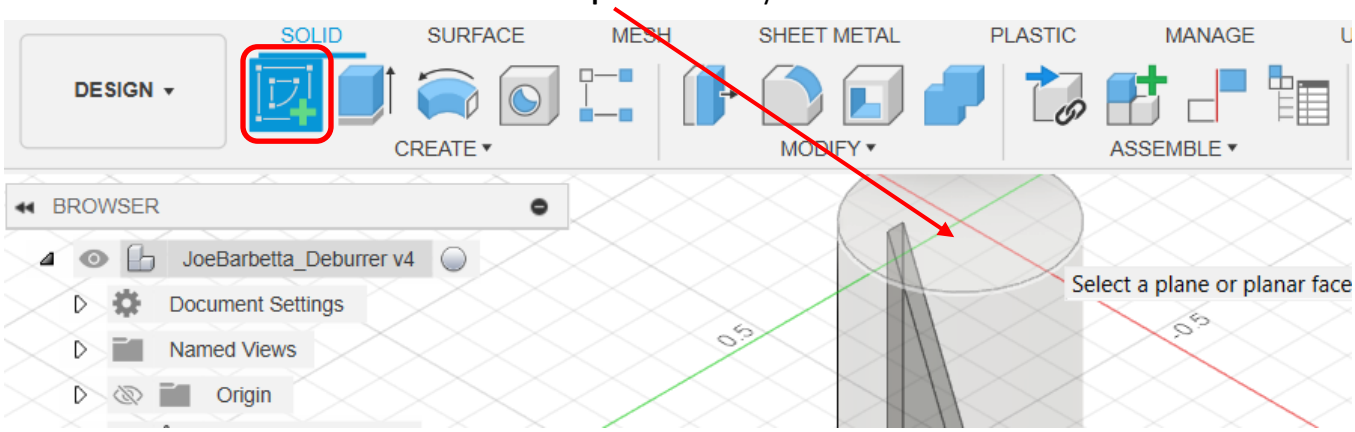
- select the **Combine** tool. If the icon is not visible, find it in the **MODIFY** menu.
- ensure that **Cut** is selected for the **Operation** and **Keep Tools** is checked
- click on the **Shroud cylinder** and then on the **Knife Blade** and **Knife Handle** and click **OK**



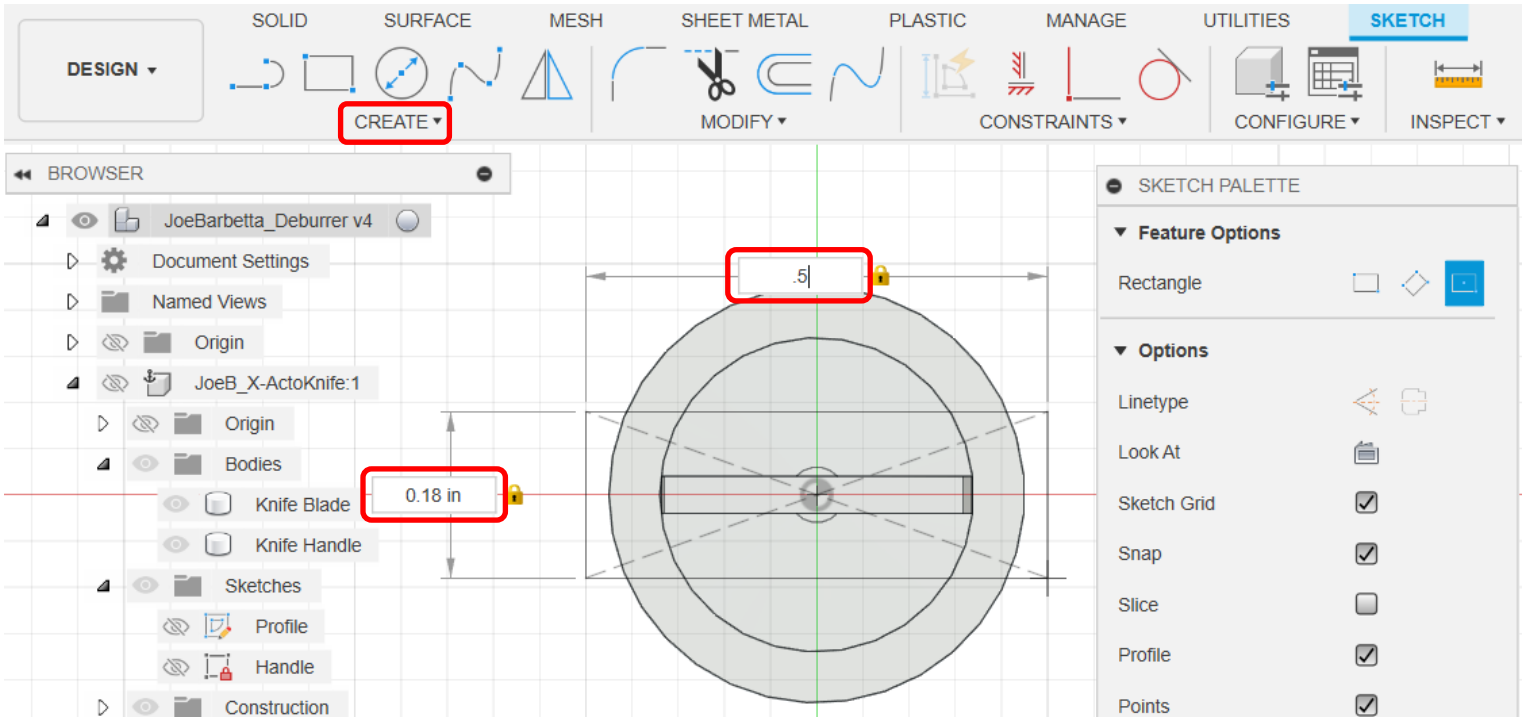
- click on the **eye** icon for the **X-ActiKnife** to hide it



- select the **Create Sketch** tool and click on the **top face** of the cylinder

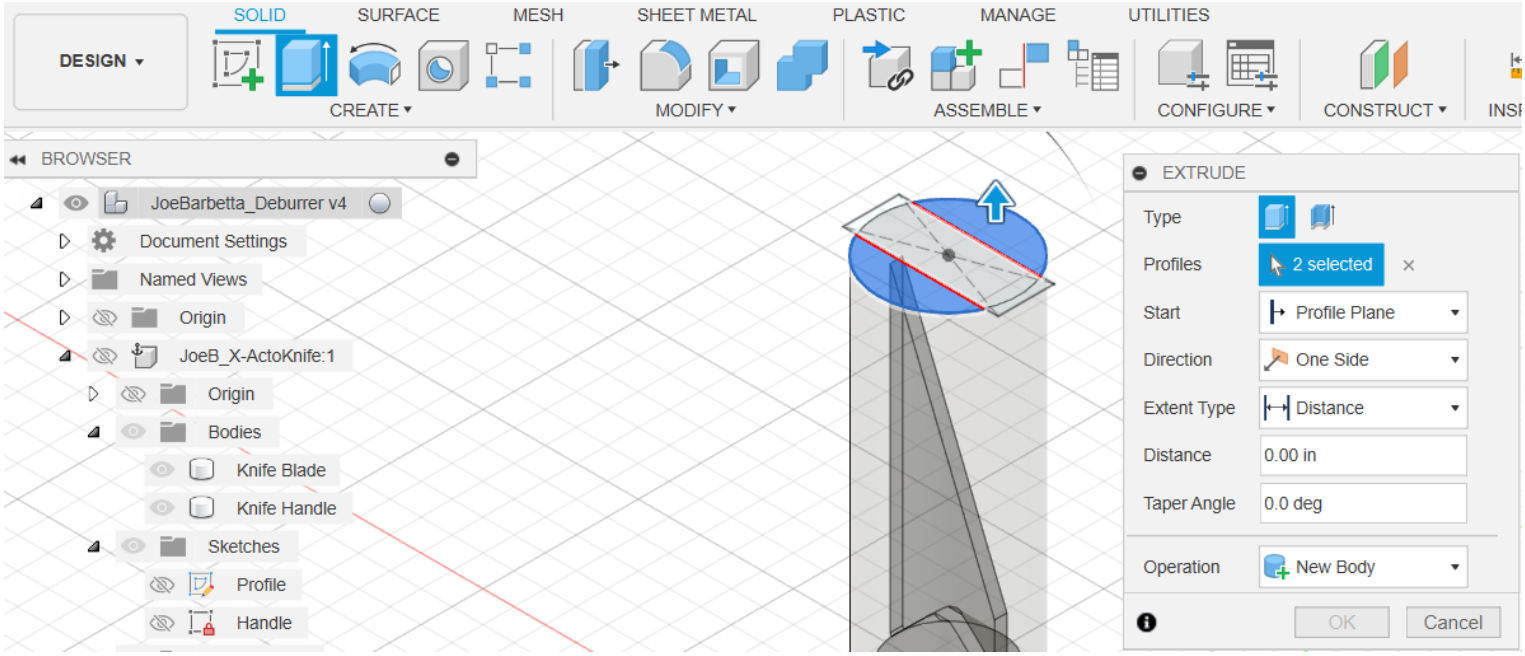


- from the **CREATE** menu select **Rectangle** and **Center Rectangle**
- click on the origen and set the **Height** to **0.18** and the **Width** to **0.5**. The Tab key can be used to switch between them.
- click **Finish Sketch**

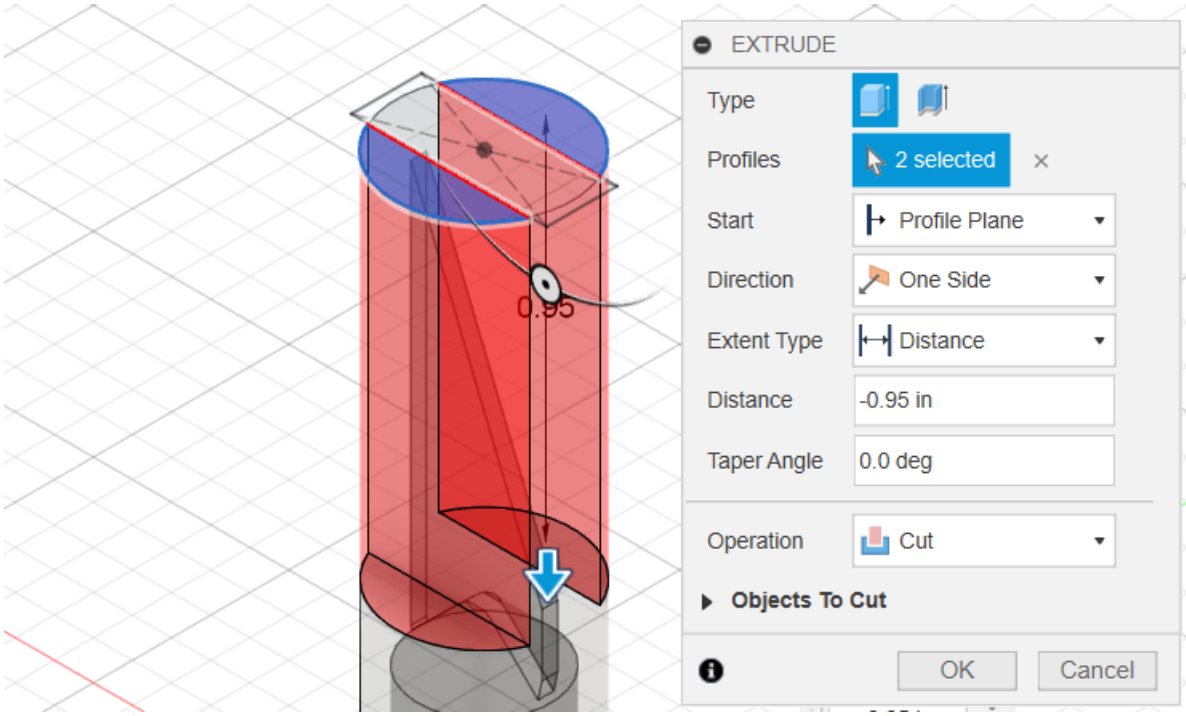


- return to the **Home** view and zoom into the top of the cylinder

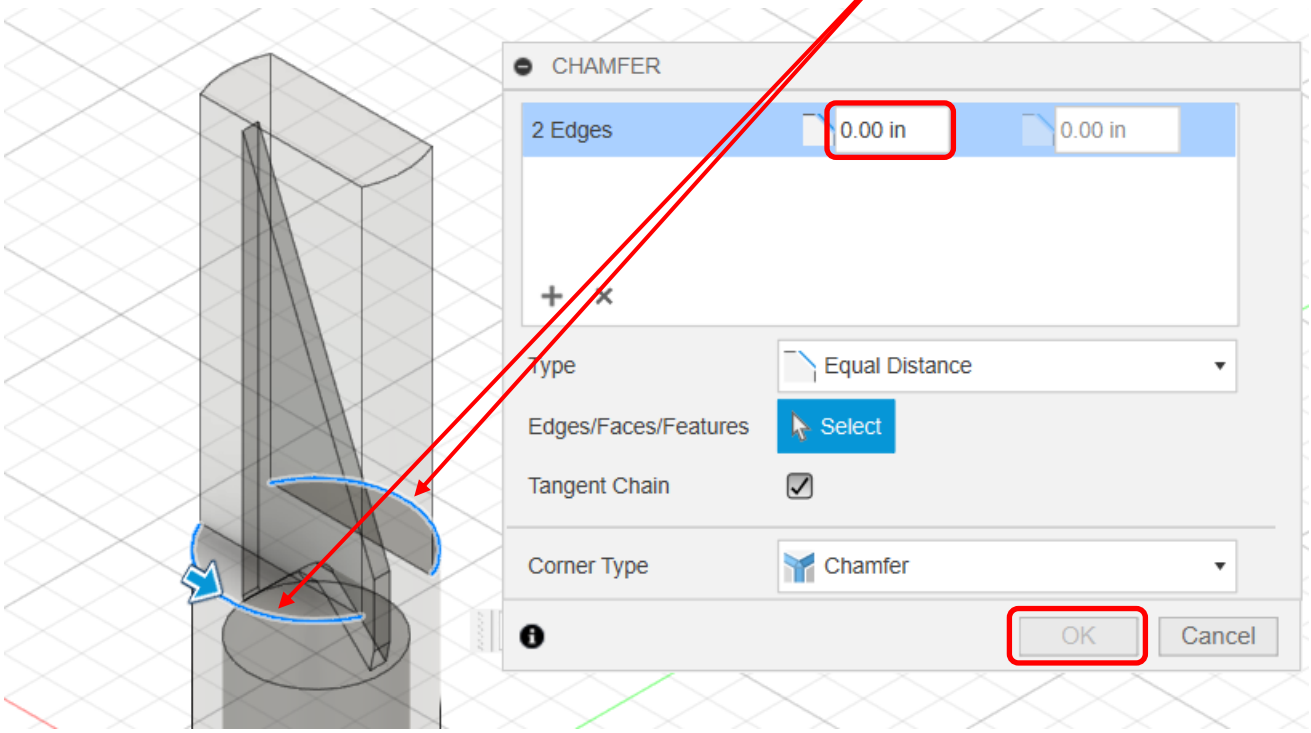
- select the **Extrude** tool and click on the **two regions as shown** to highlight them blue



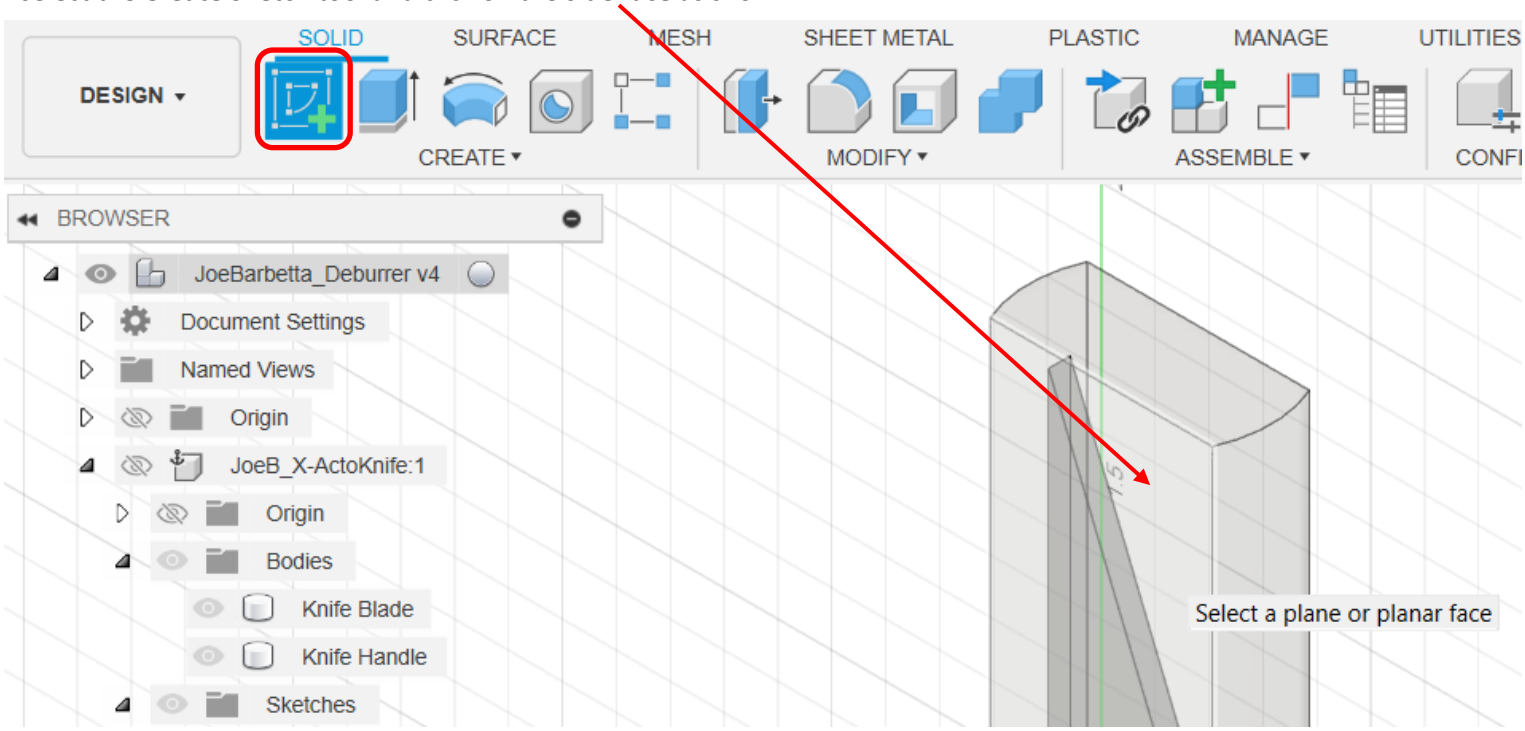
- enter **-0.95** (note the minus sign) for **Distance**. Yell out **"That looks slay!"**
- click **OK**



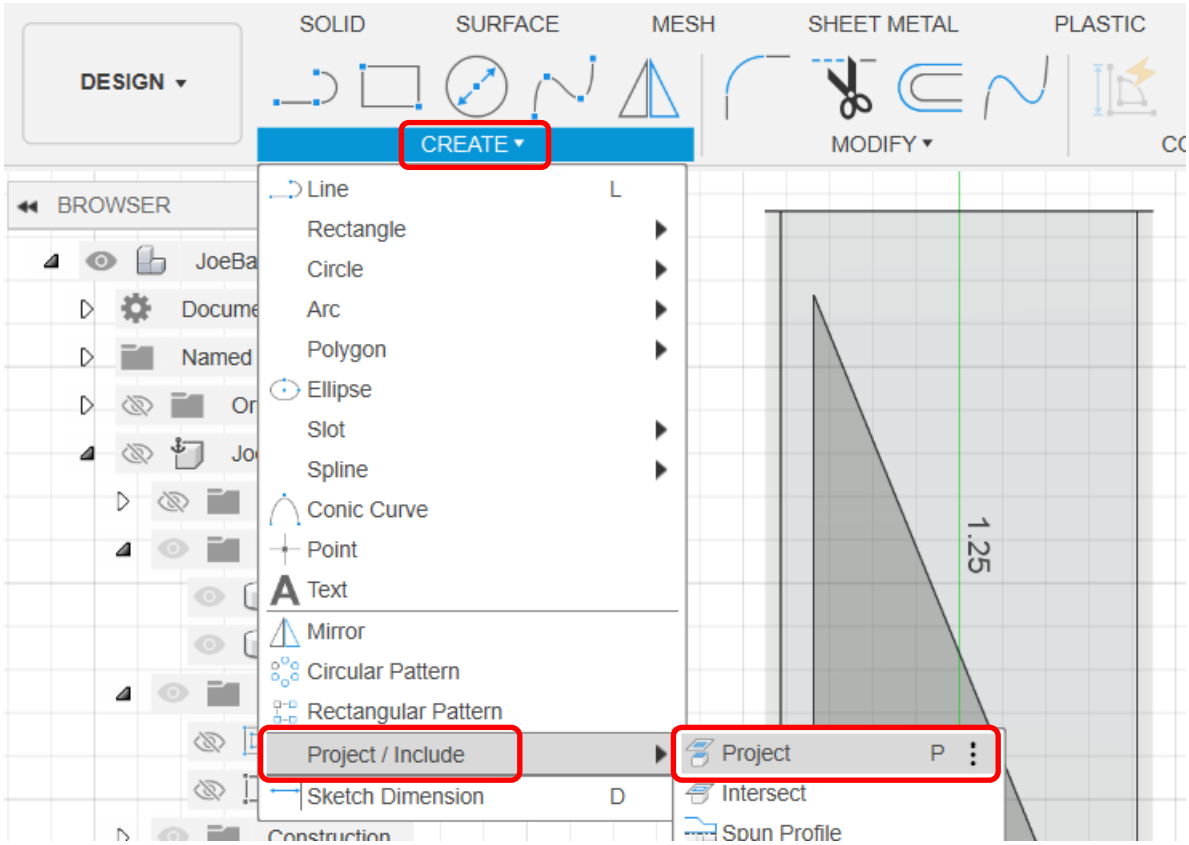
- from the **MODIFY** menu select the **Chamfer** tool and click on the two **circular edges** as shown
- ensure **Equal Distance** is set and enter **0.125** and click **OK**



- select the **Create Sketch** tool and click on the **side face** as shown

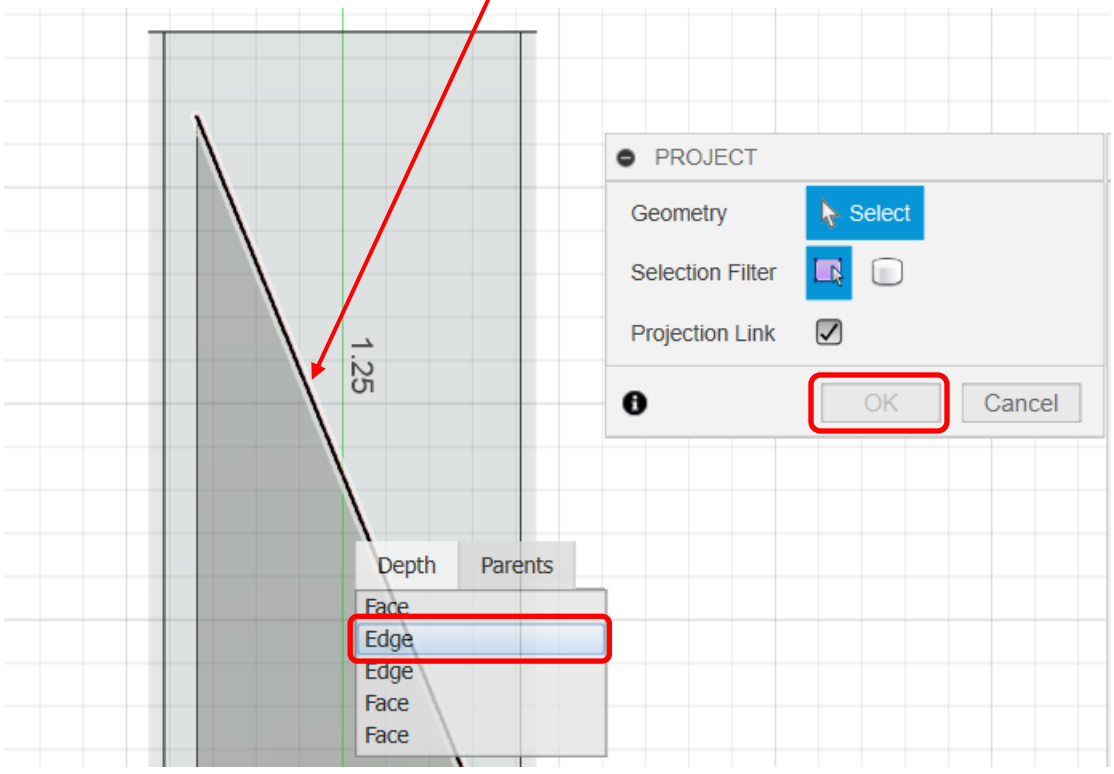


- from the **CREATE** menu select **Project / Include** and **Project**

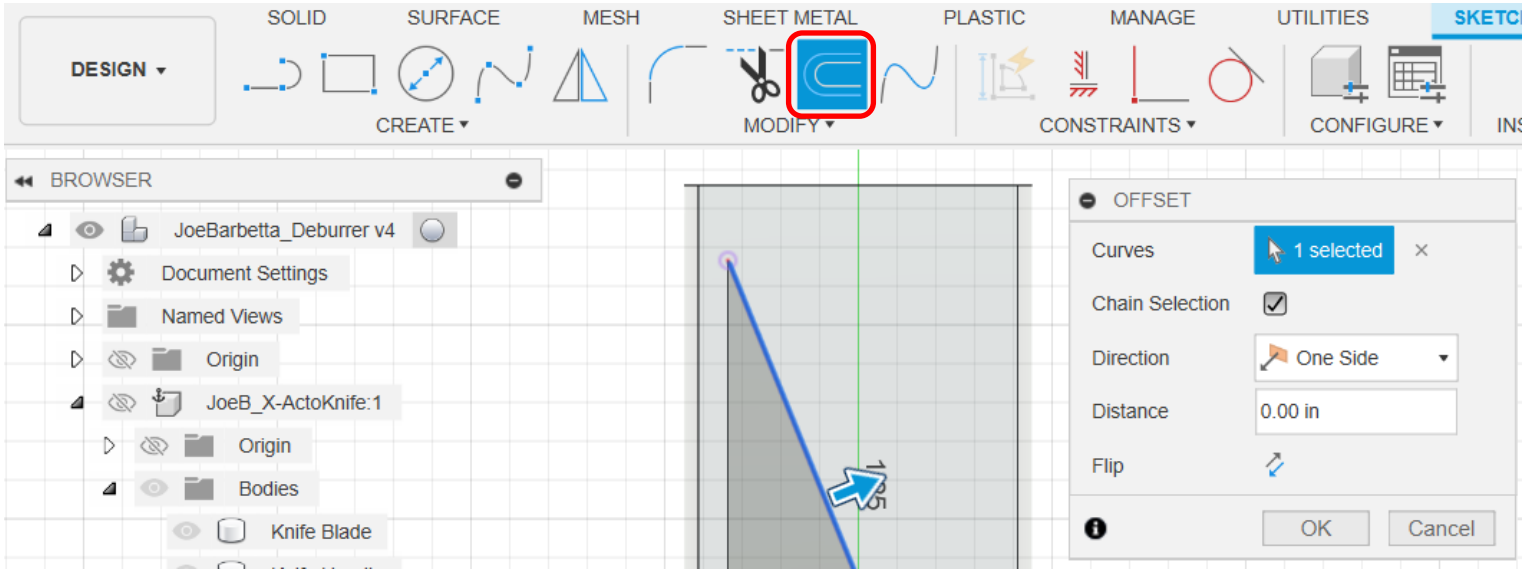


- hold the **Mouse Button down** over the **blade edge** until the **pop-up menu** shoes and select the **Edge** option that causes the blade edge to become highlighted. It may be hard to see, but there will be a violet line and a small violet circle at edge end of the line segment. This operation produced this new line on the sketch.

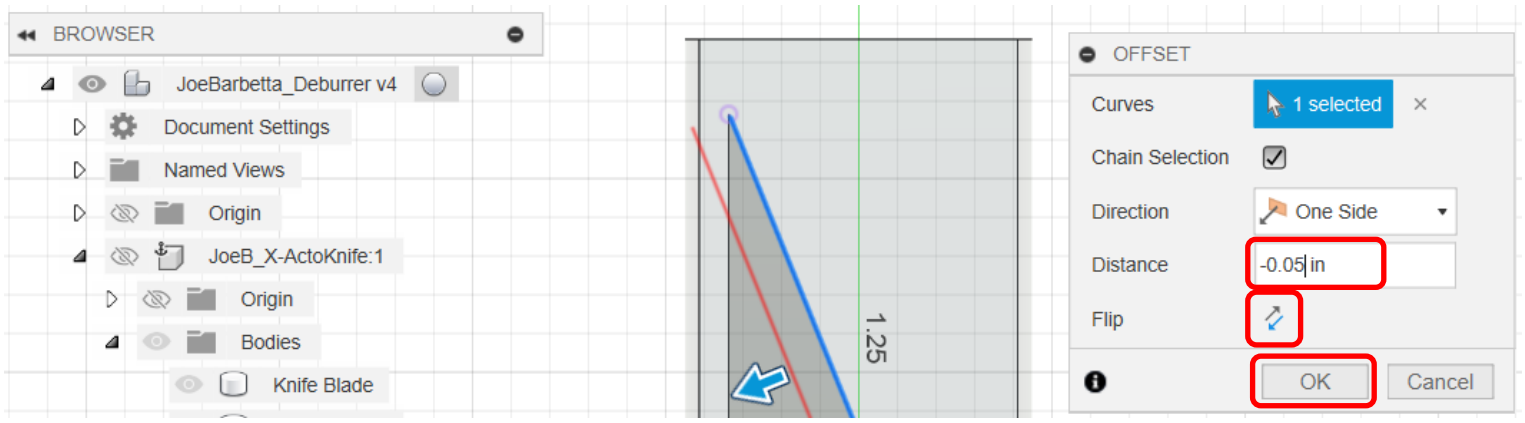
- click **OK**

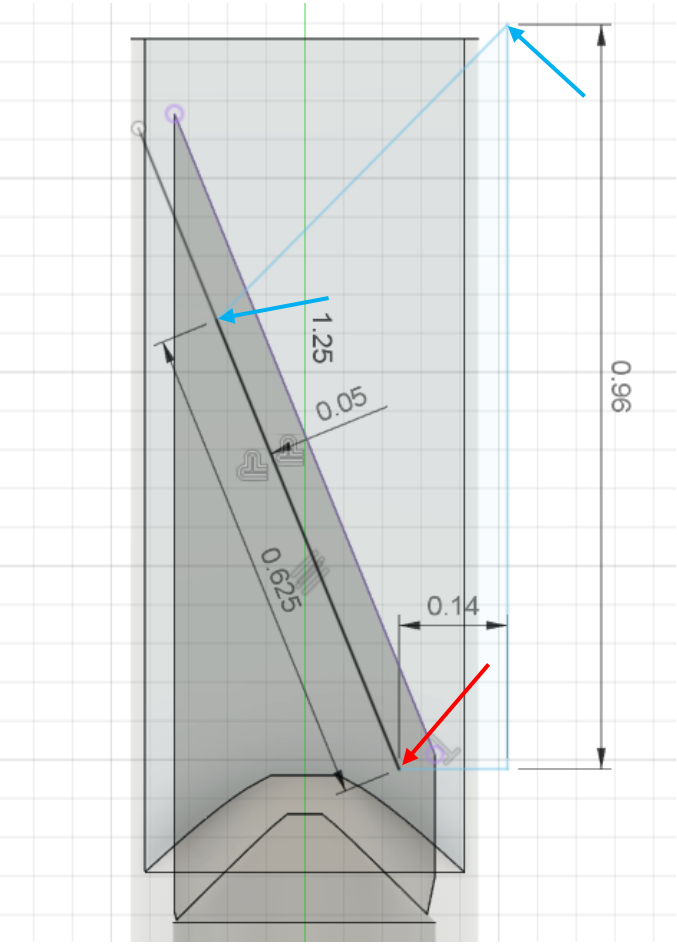


- select the **Offset** tool and click on the **knife blade edge**, which will cause it to turn blue



- enter **-0.05** (note the minus sign) for **Distance**. The red line should be to the left of the blue line. If it is not, click on the **Flip** icon.
- click **OK**





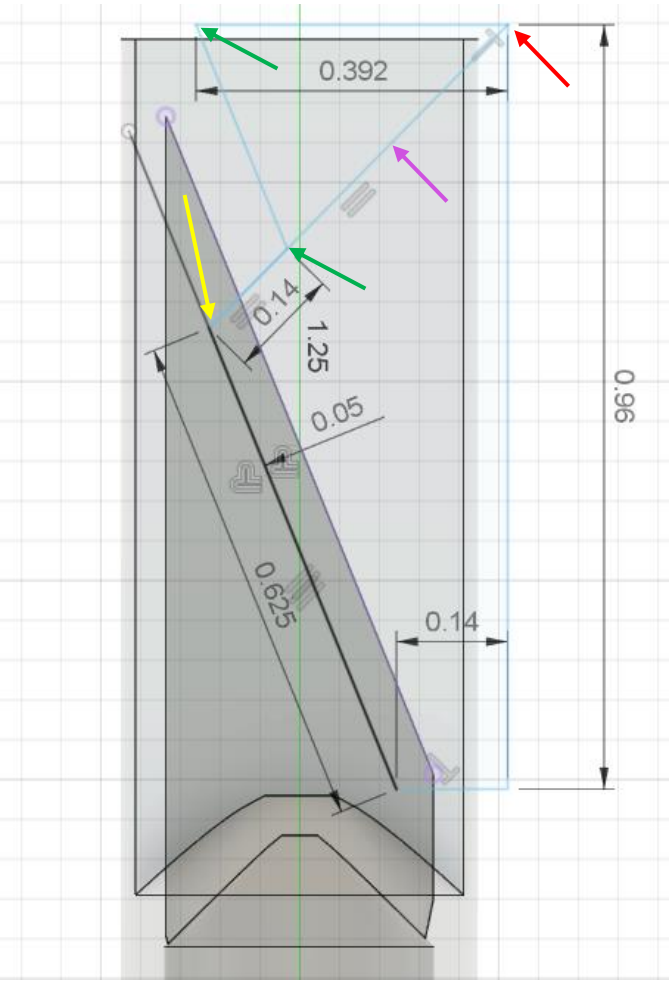
- select the Line tool and click on the point indicated by the **red arrow**

- extend the line to the right and enter **0.14**

- from the end of that line create a **0.96** line upward

- from the original point indicated by the **red arrow**, entend a line upward and to the left that covers the diagonal line and enter **0.625**

- create a final line that connects the two points indicated by the **blue arrows**. No length value will be entered for this last line.

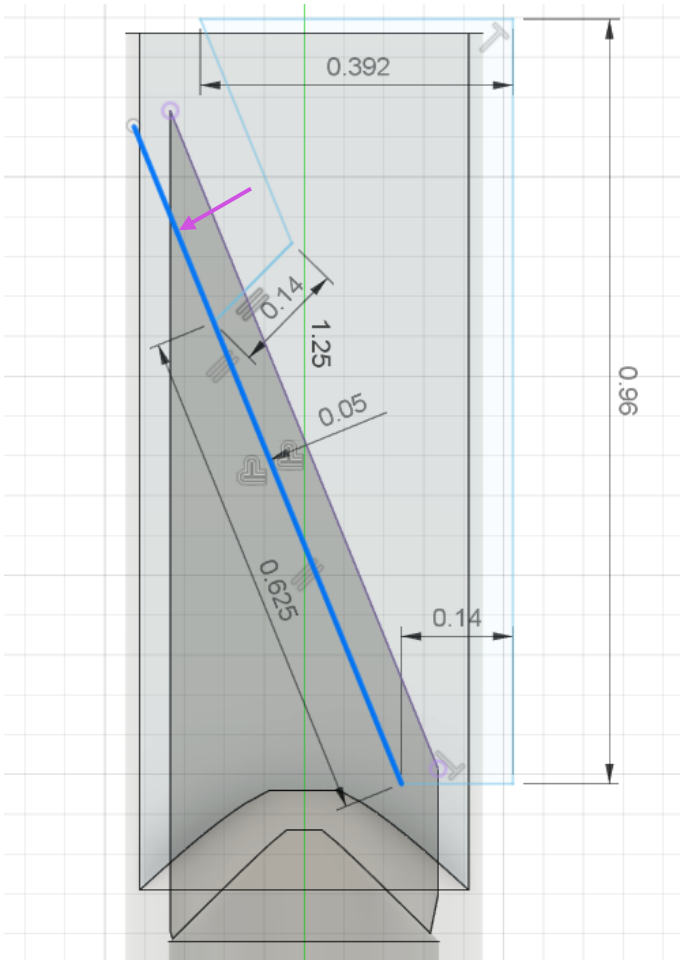


- start a Line at the corner point indicated by the **red arrow**, extend it to the left, anf enter **0.392**

- start a line from the point indicated by the **yellow arrow**, entend a line upward and to the right that covers the diagonal line and enter **0.14**

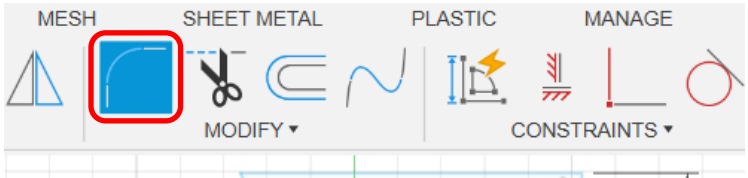
- create a final line that connects the two points indicated by the **green arrows**. No length value will be entered for this last line.

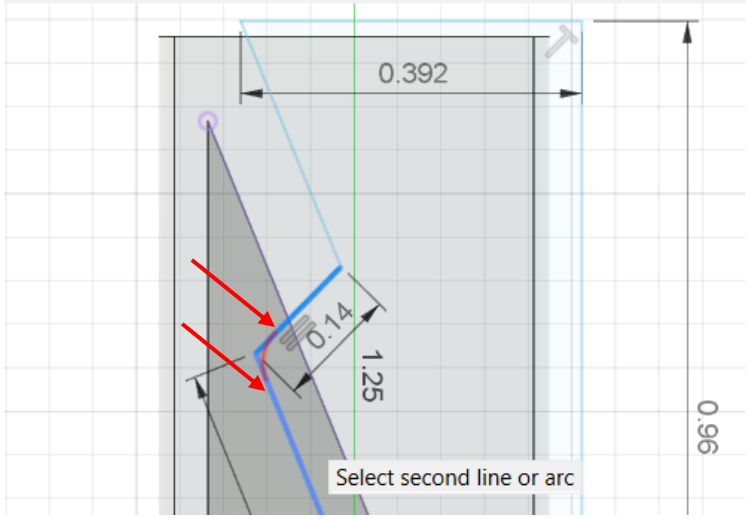
- click on the line indicated by the **violet arrow** and press the **Delete key**. Not that the line tool is Not being used here. The line is just being selected and then deleted.

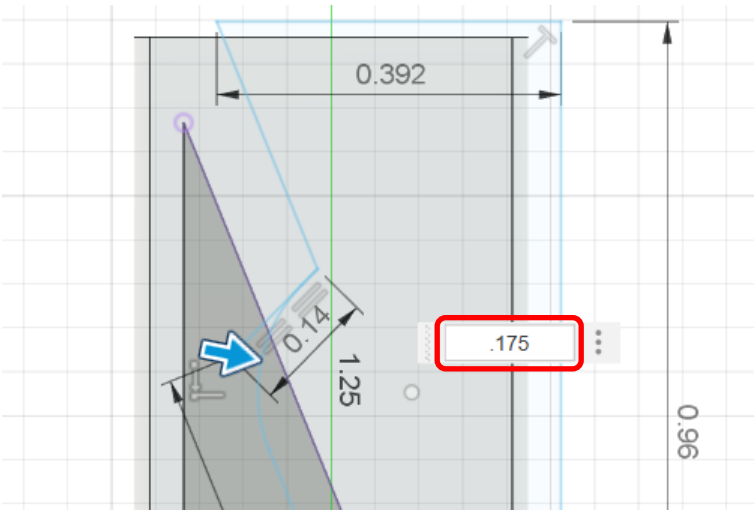


- click on the line indicated by the **violet arrow** and press the **Delete key**. The lower **0.625** section of this line should remain.

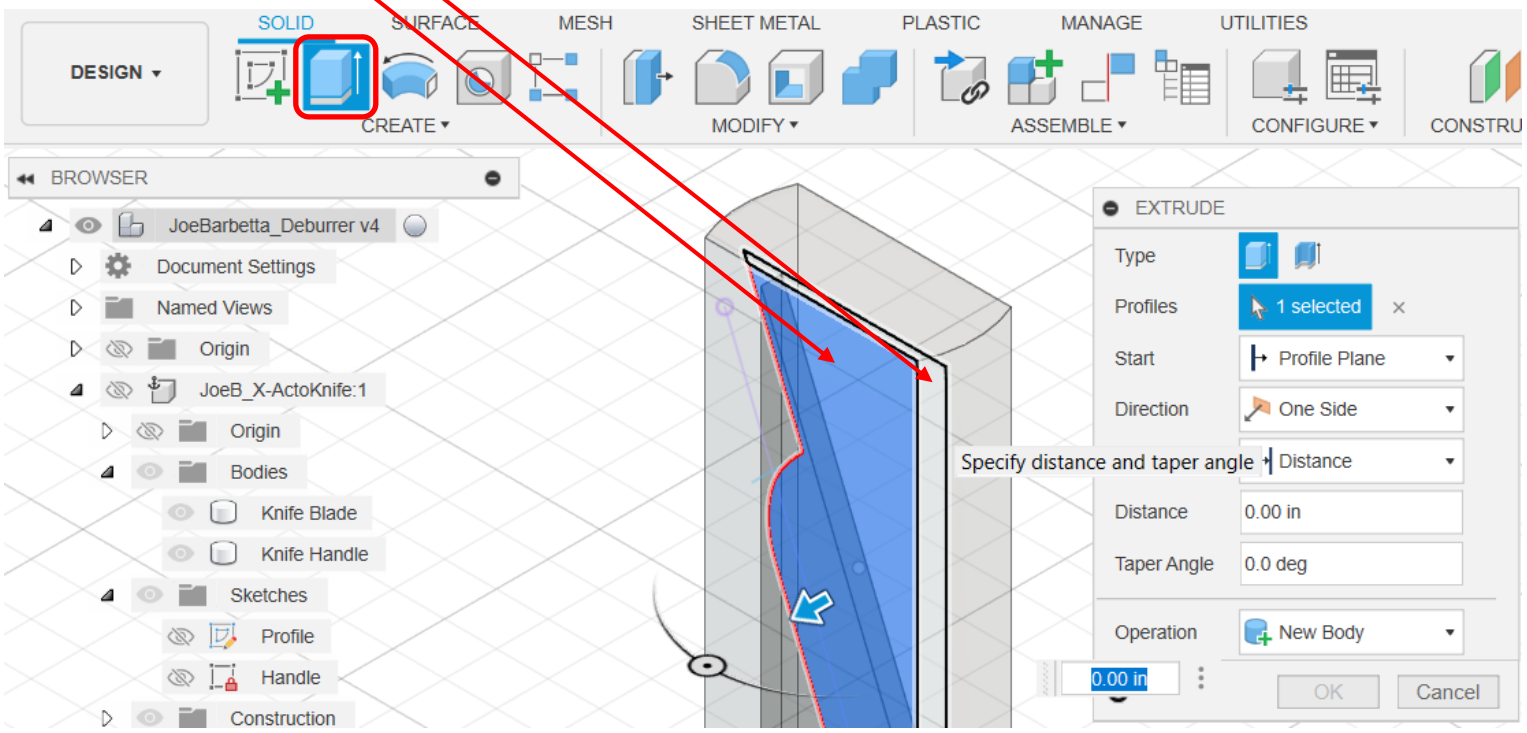
- select the **Fillet** tool. If it is not visible find it in the **MODIFY** menu.
- click on the two line segments indicated by the **red arrows** and enter **0.175** as shown on the right picture.



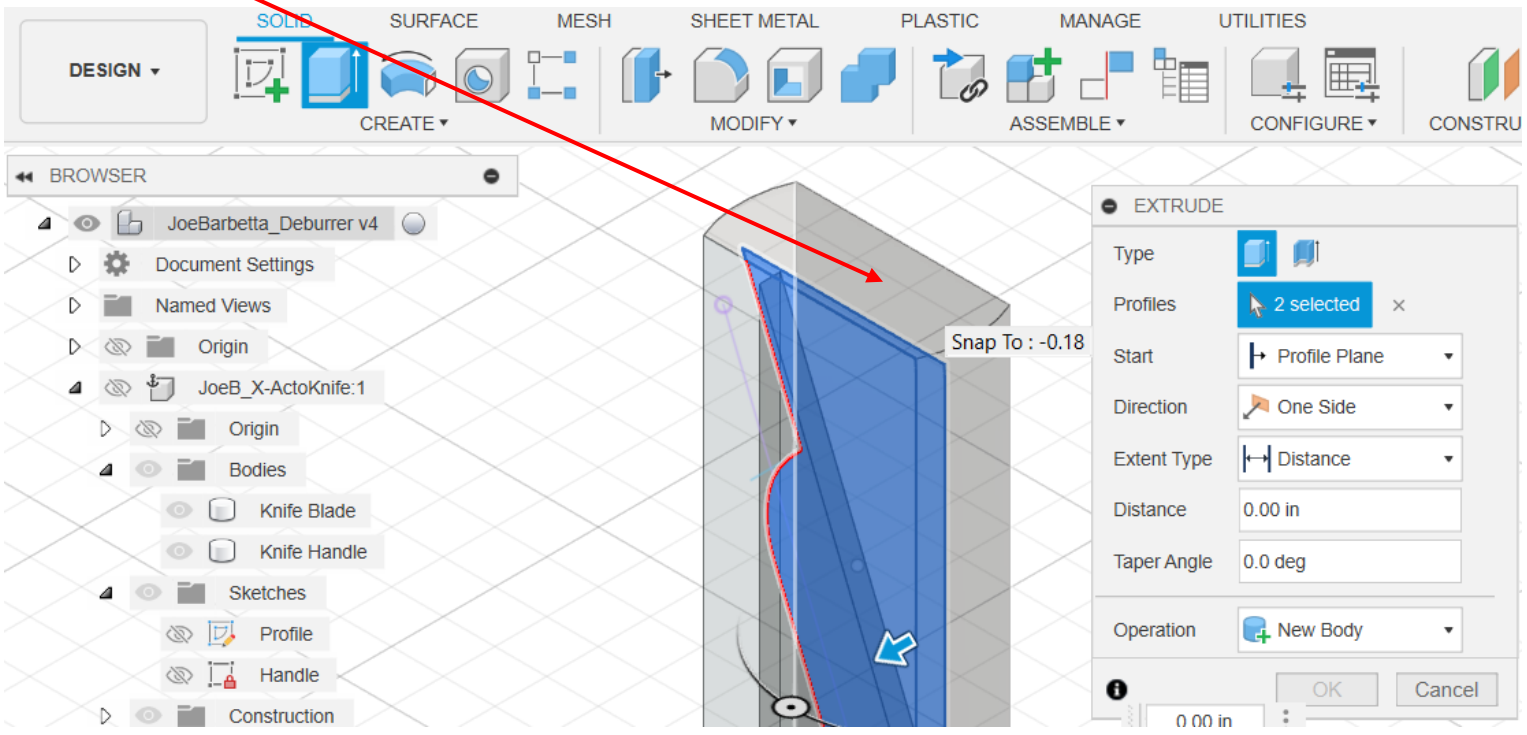




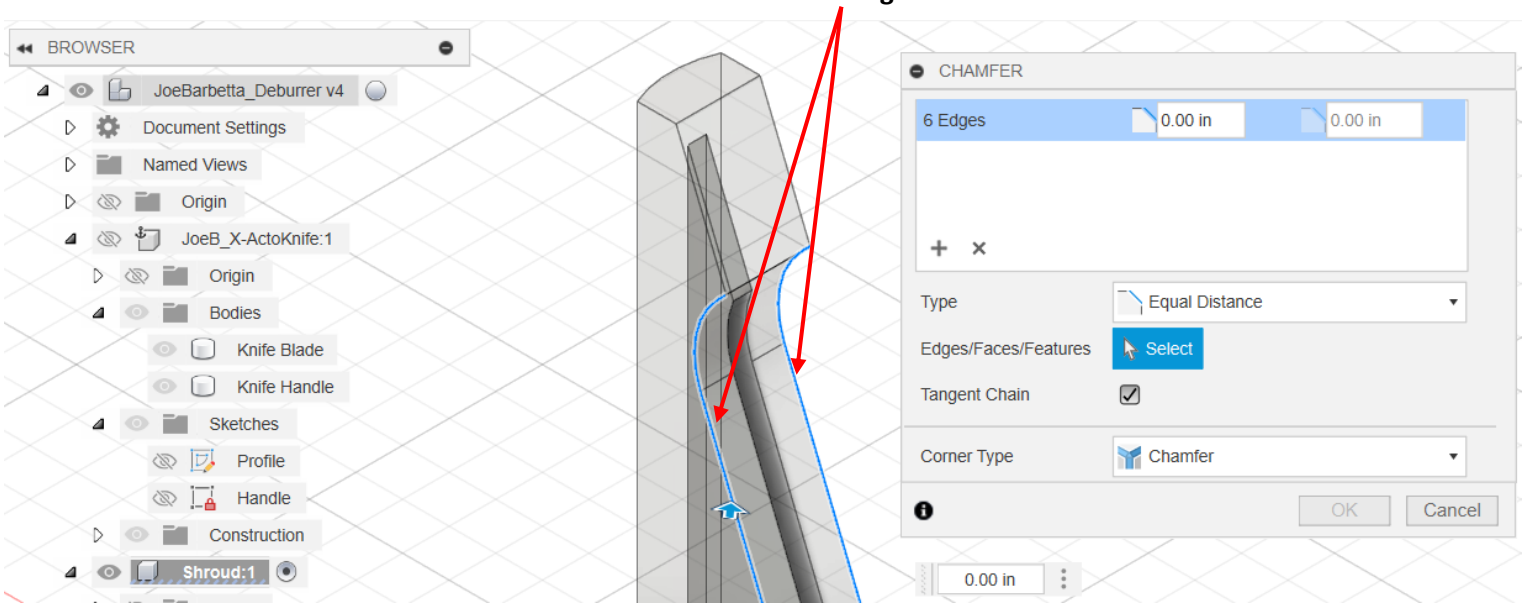
- click **Finish Sketch**
- return to the **Home** view and zoom in to achieve a view similar to that below
- click on the region indicated, which should highlight it blue
- click on the remaining smaller region to highlight it blue as well



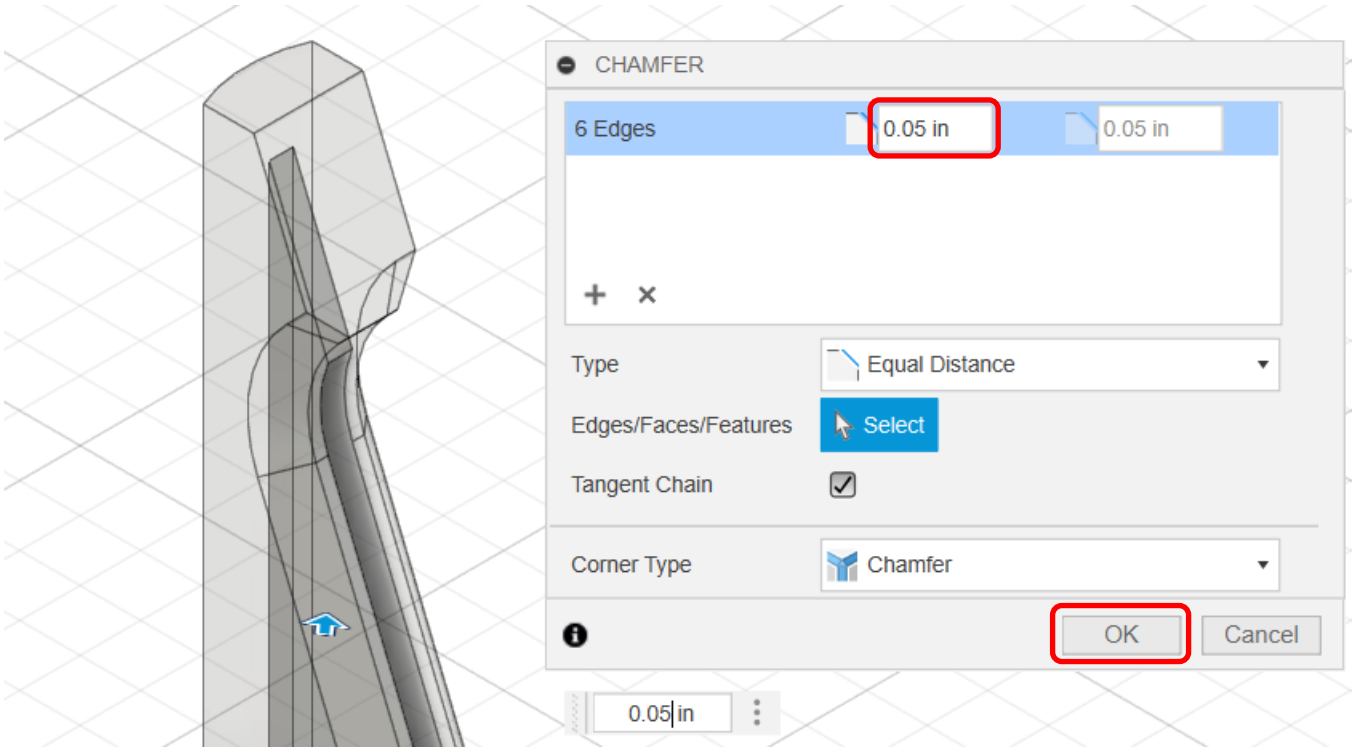
- click on the **top face**, which should select the rear face and cause a **Snap To: -0.18** box to show and click **OK**



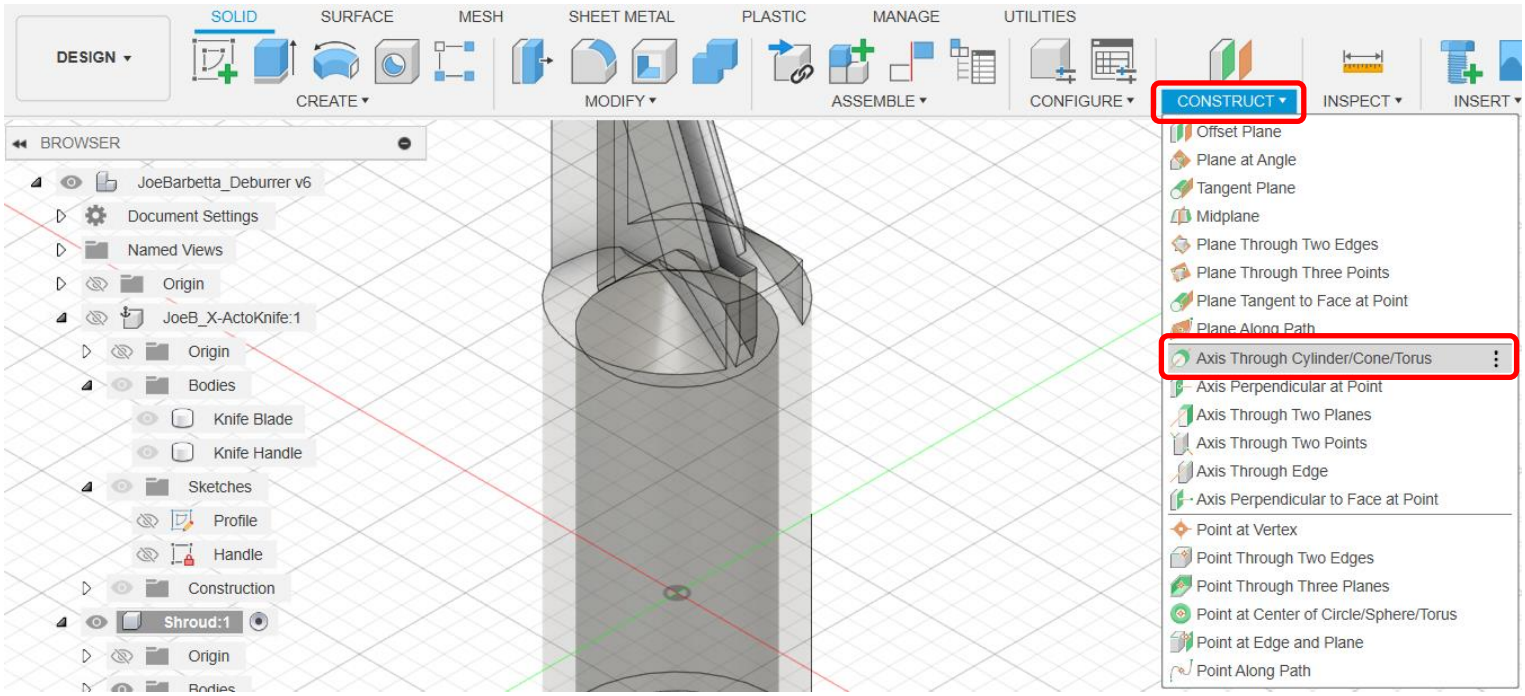
- select the **Chamfer** tool from the **MODIFY** menu and click on the **two edges** as shown



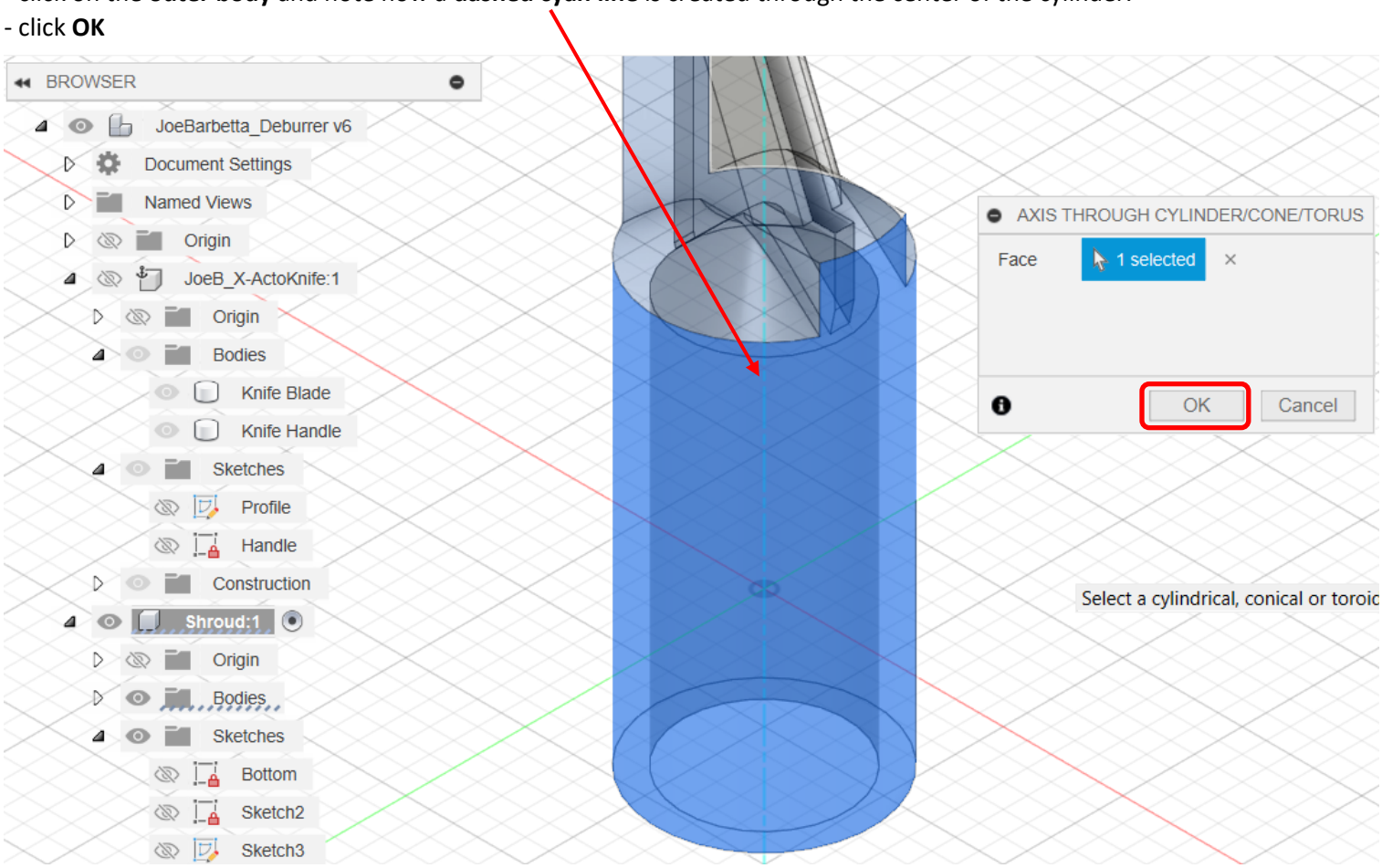
- enter **0.05** and click **OK**



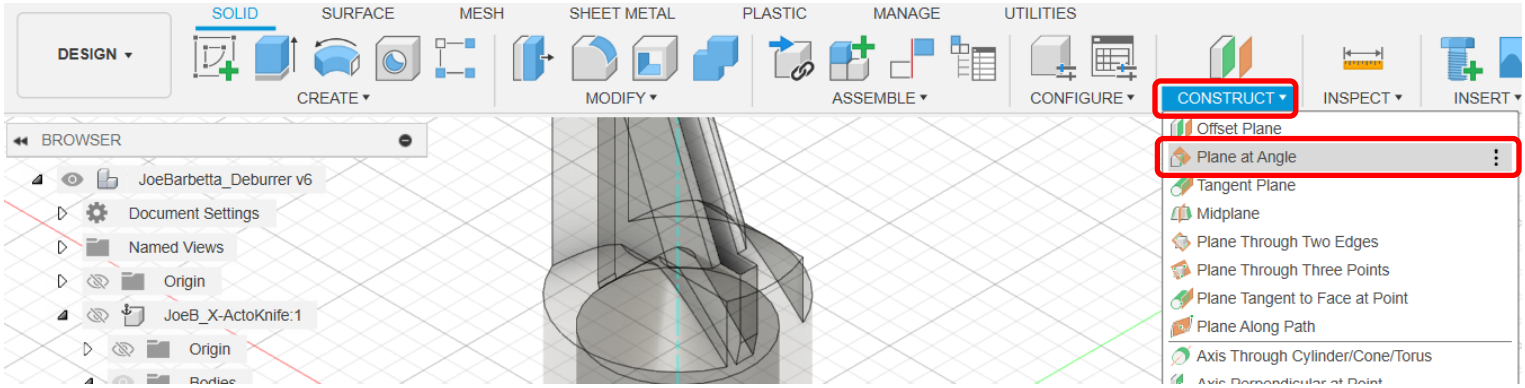
- pan the view to access the bottom portion of the shroud
- from the **CONSTRUCT** menu select **Axis Through Cylinder/Cone/Torus**



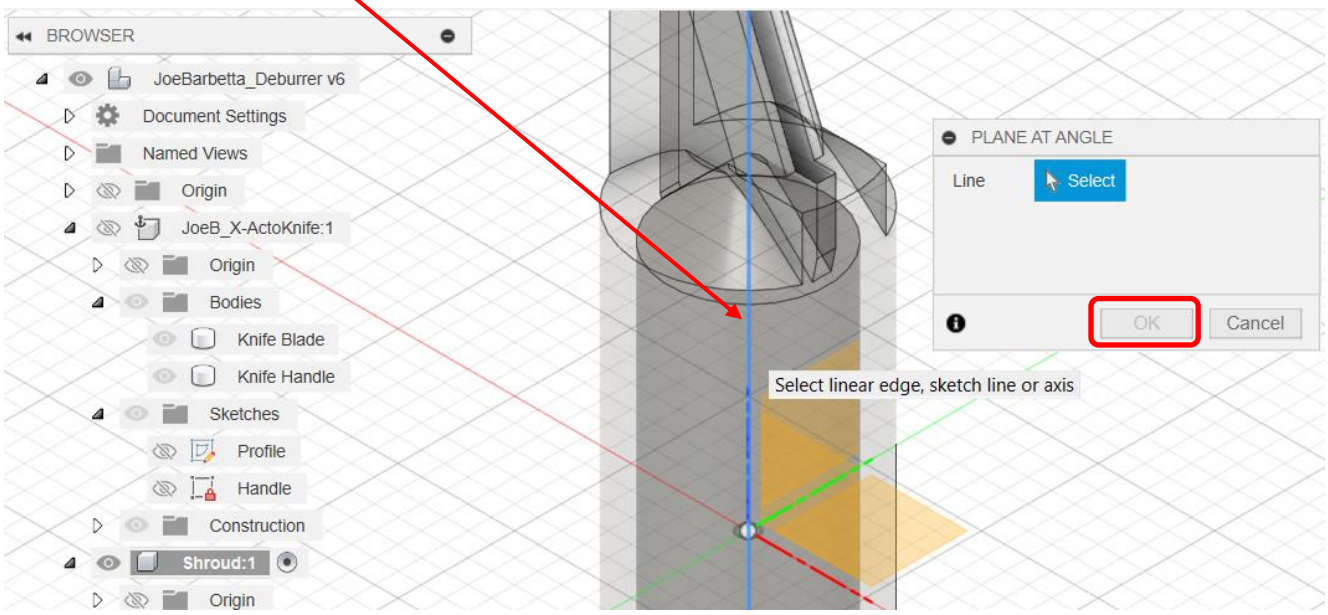
- click on the **outer body** and note how a **dashed cyan line** is created through the center of the cylinder.
- click **OK**



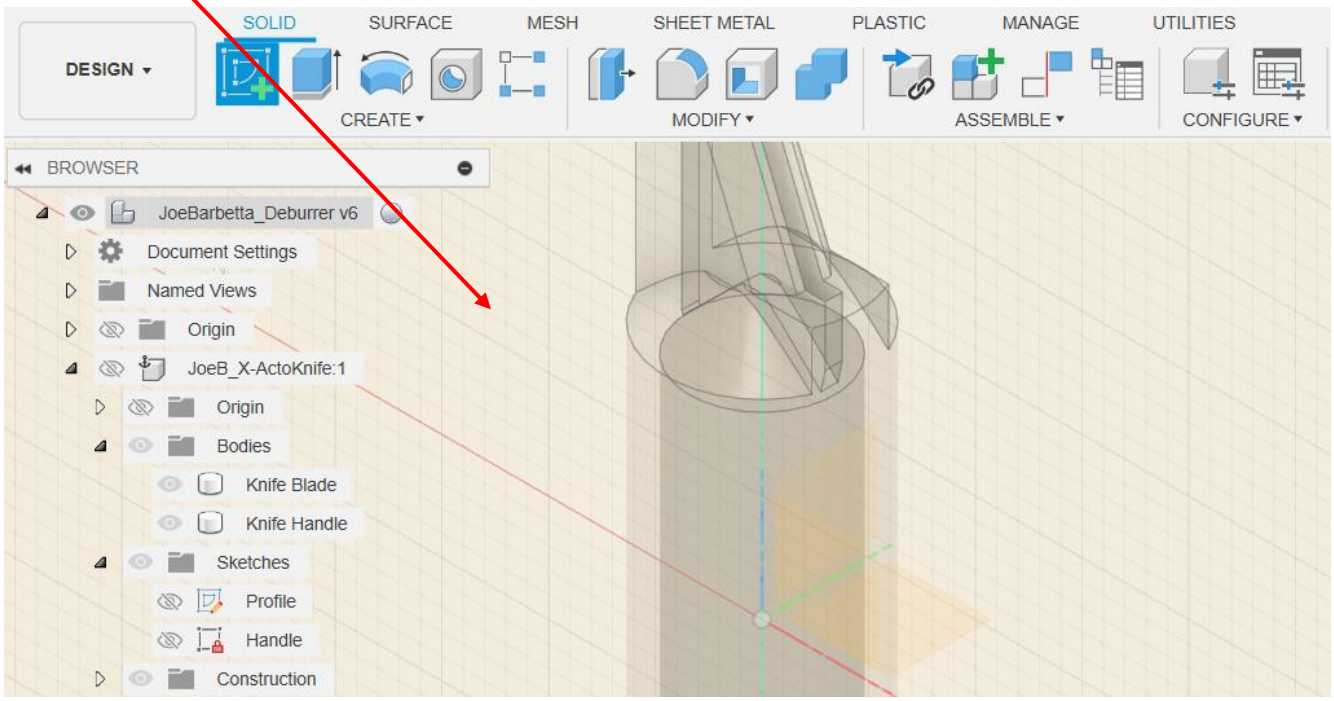
- from the **CONSTRUCT** menu select **Plane at Angle**



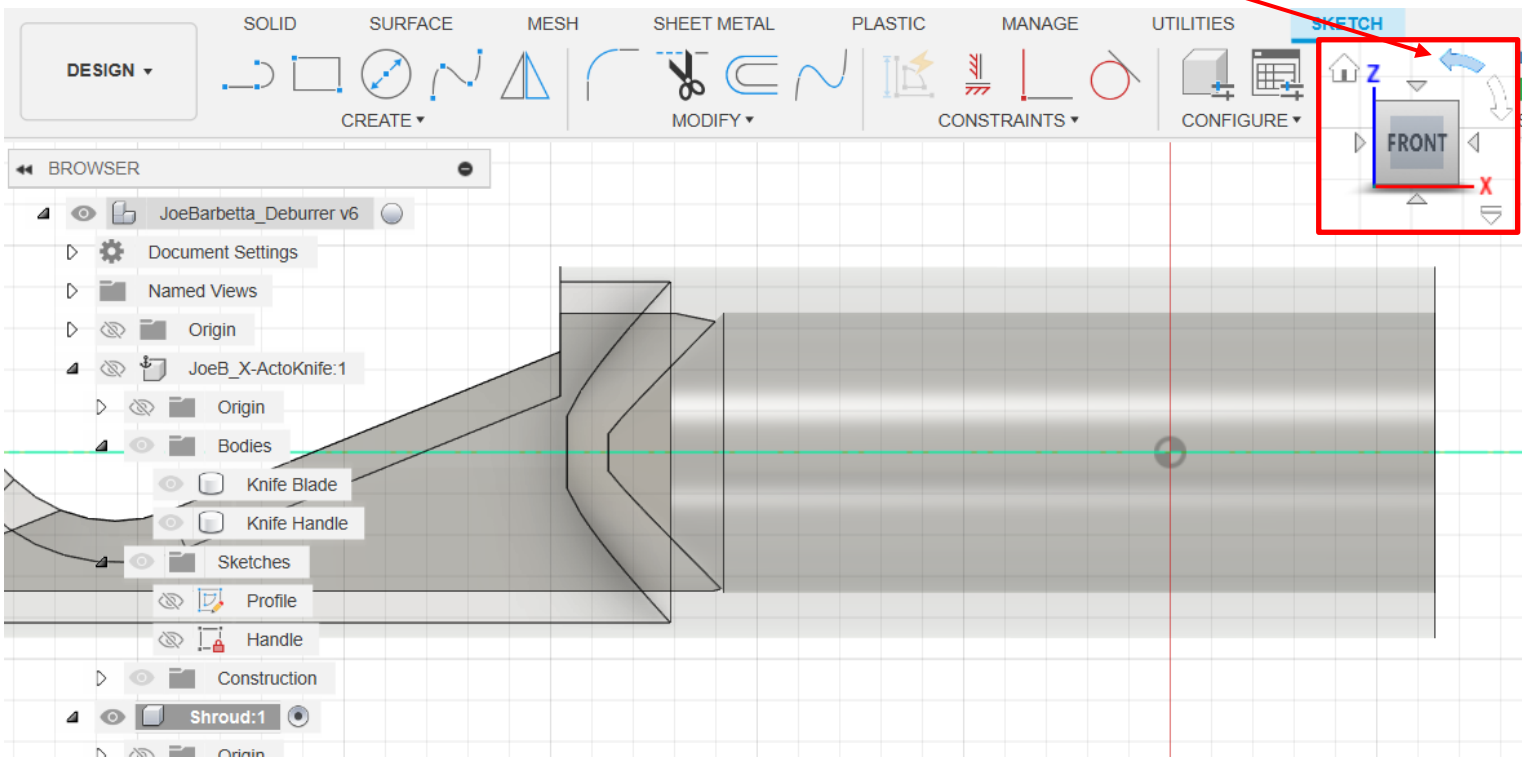
- click anywhere on the **cyan axis**, which should highlight it blue, and click **OK**



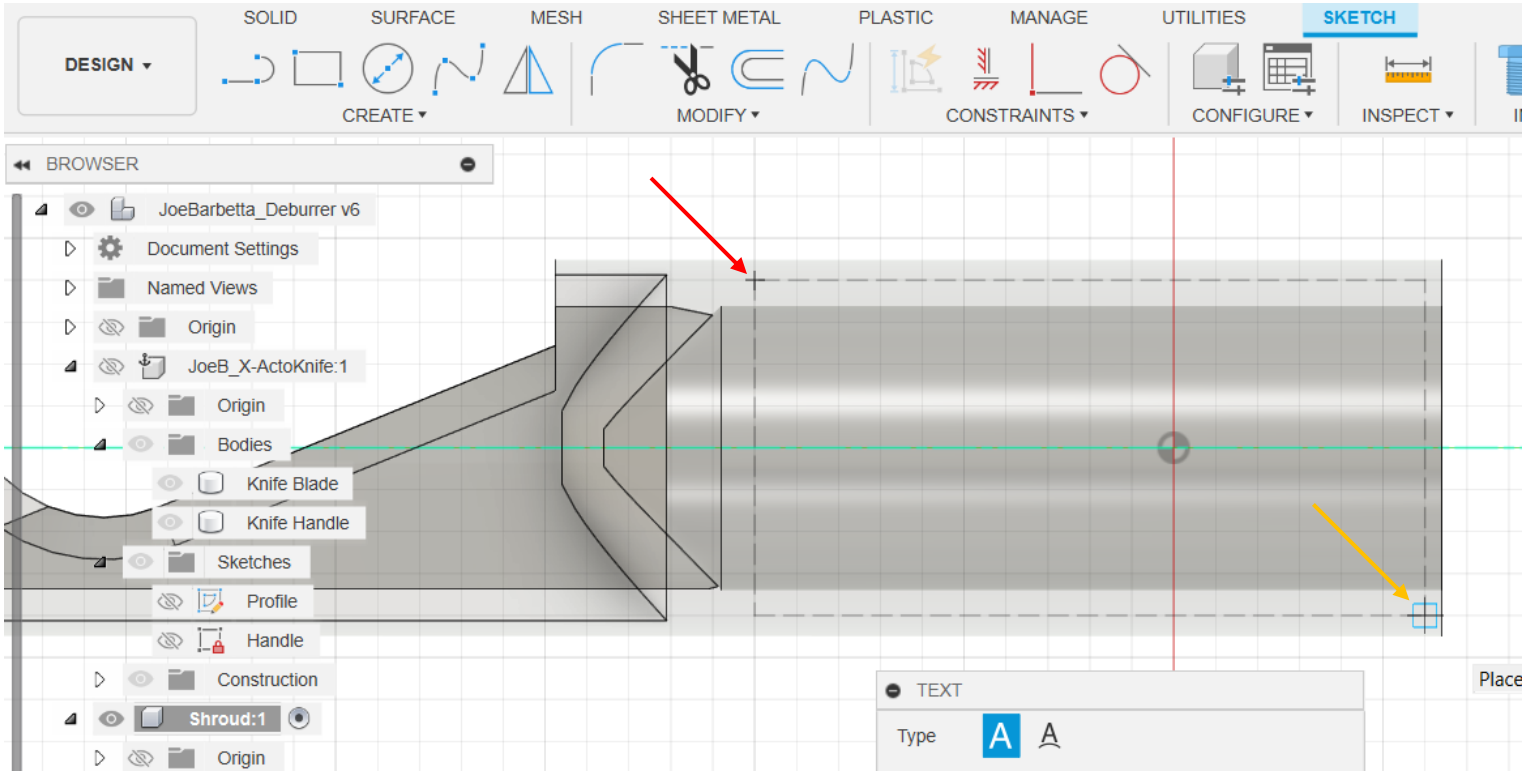
- select **Create Sketch** and click anywhere on the plane



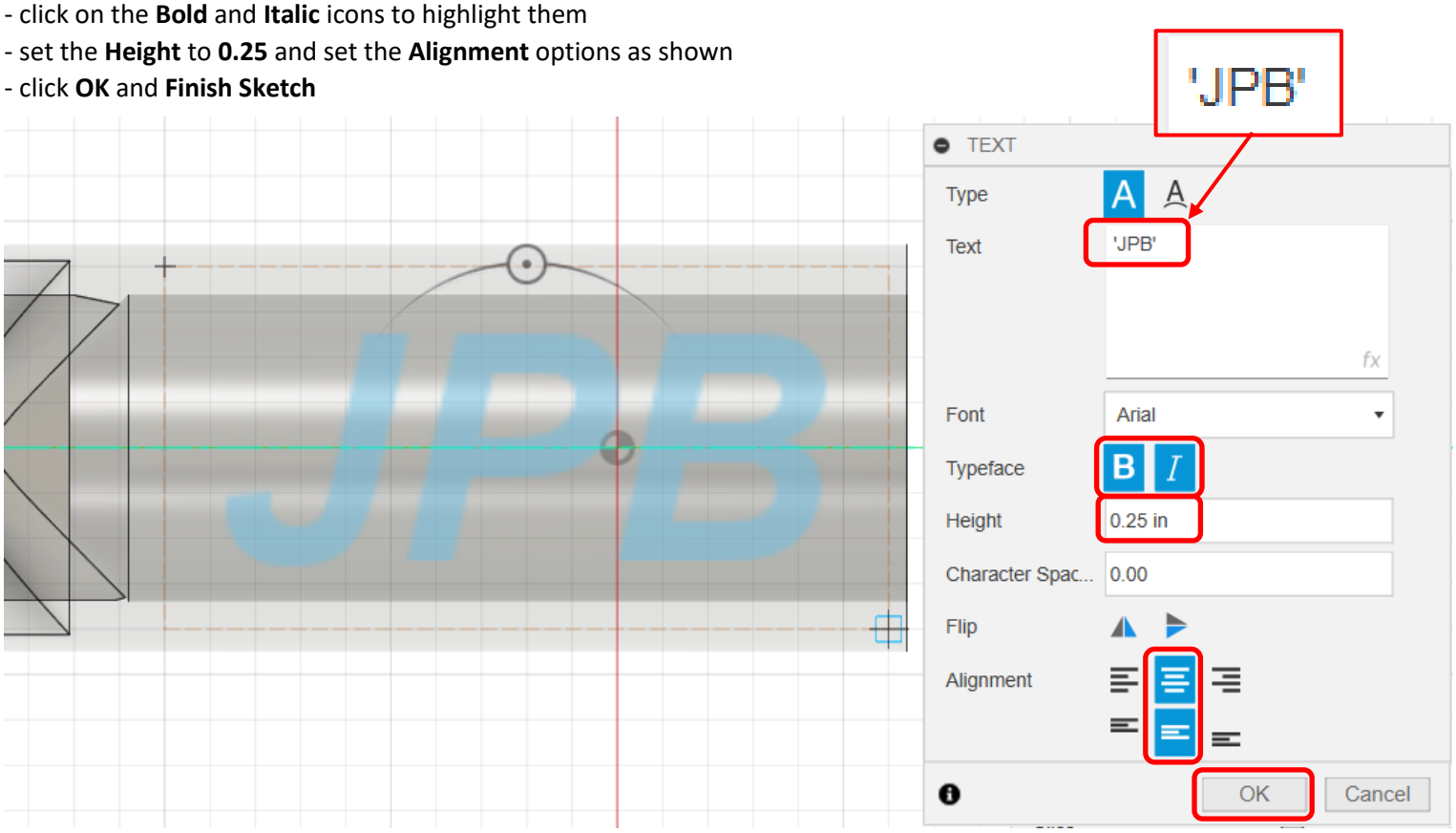
- if needed click on the **rotation arrows** that appear when moving the mouse over the **View Cube**
- zoom and pan to achieve a view similar to that below



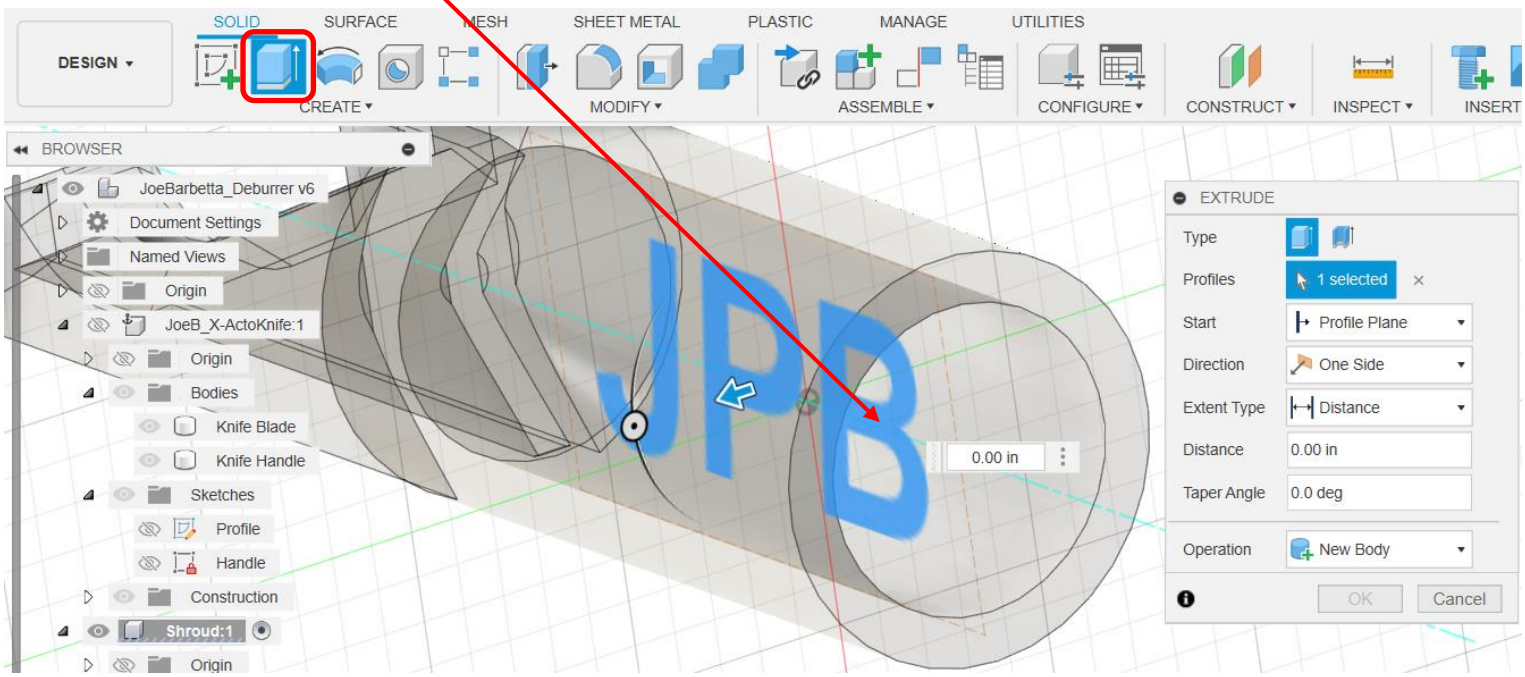
- from the **CREATE** menu select **Text**
- if a message window about Parametric Text appears, click its **OK** button
- click at a point near the **red arrow** to start the frame rectangle and click again near the **yellow arrow**. The positions are not critical. One just wishes to define an area on the cylinder.



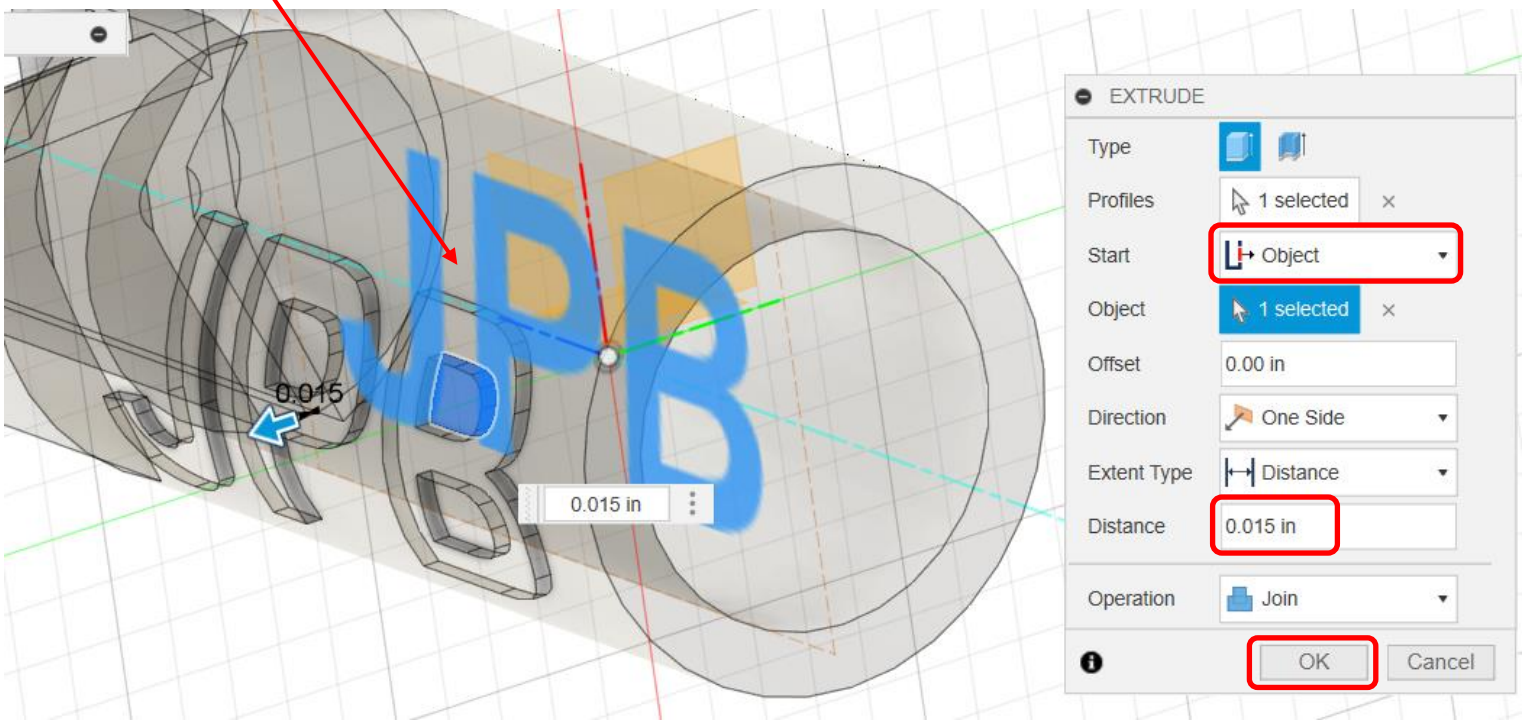
- in the **Text** box enter your **3 initials**. The initials must be **preceeded and followed by a single quote** as shown below.
- click on the **Bold** and **Italic** icons to highlight them
- set the **Height** to **0.25** and set the **Alignment** options as shown
- click **OK** and **Finish Sketch**



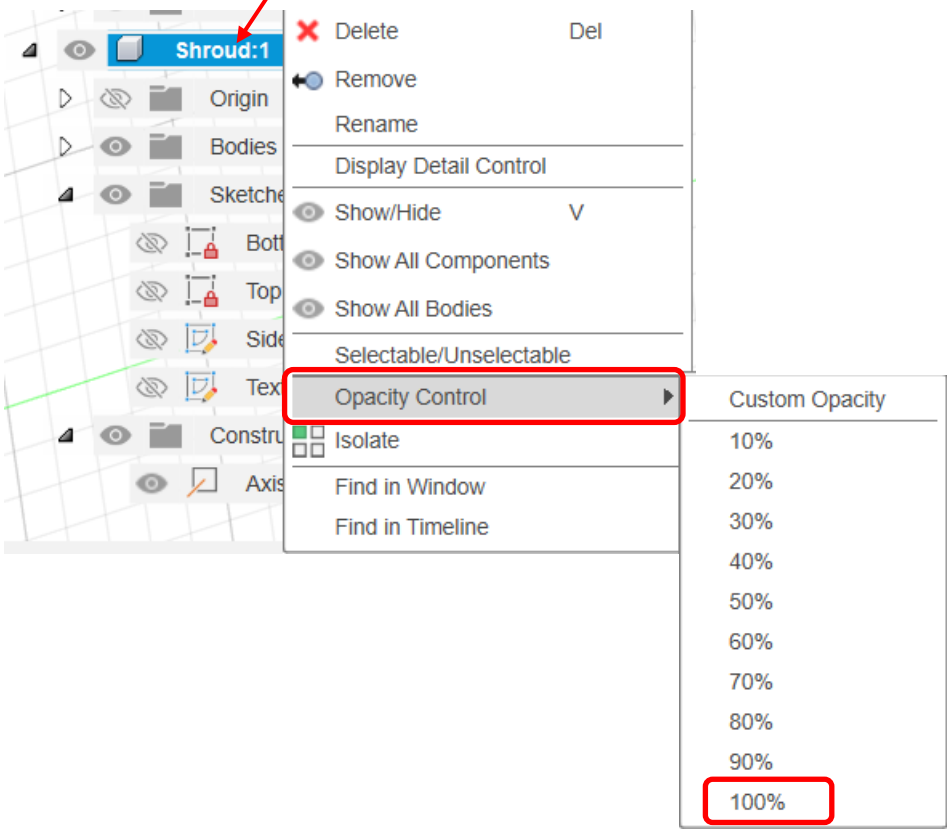
- rotate the view to achieve that similar to that below
- click on the **text** through the end hole to select it
- select the **Extrude** tool



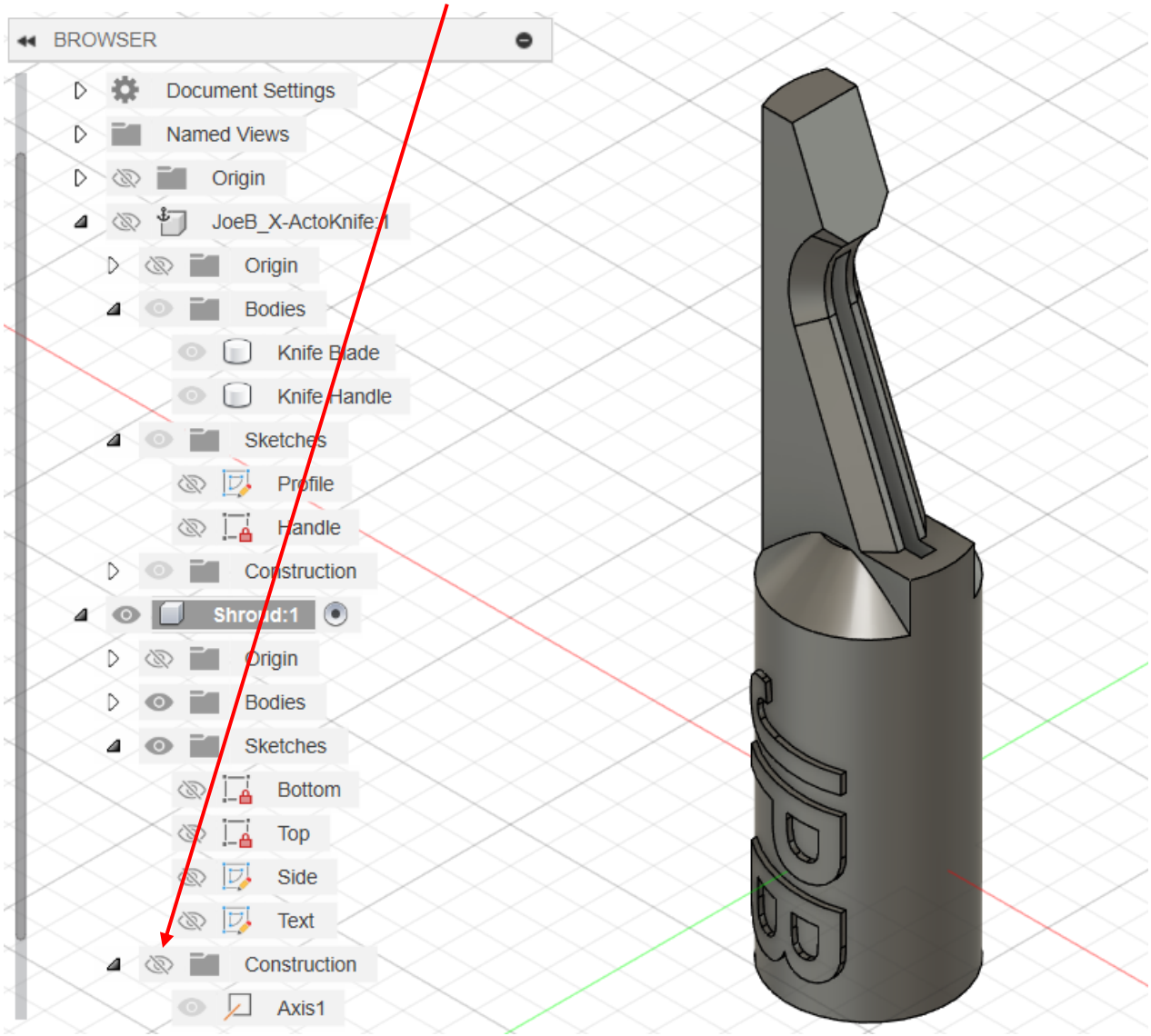
- select the **Object** option for **Start** and set the **Distance** to **0.015**
- click on the **surface of the cylinder**, which should cause the letters to protrude slightly from the surface
- click **OK**



- right-click on the **Shroud** Component name and select **Opacity Control** and select **100%** at the bottom of the list

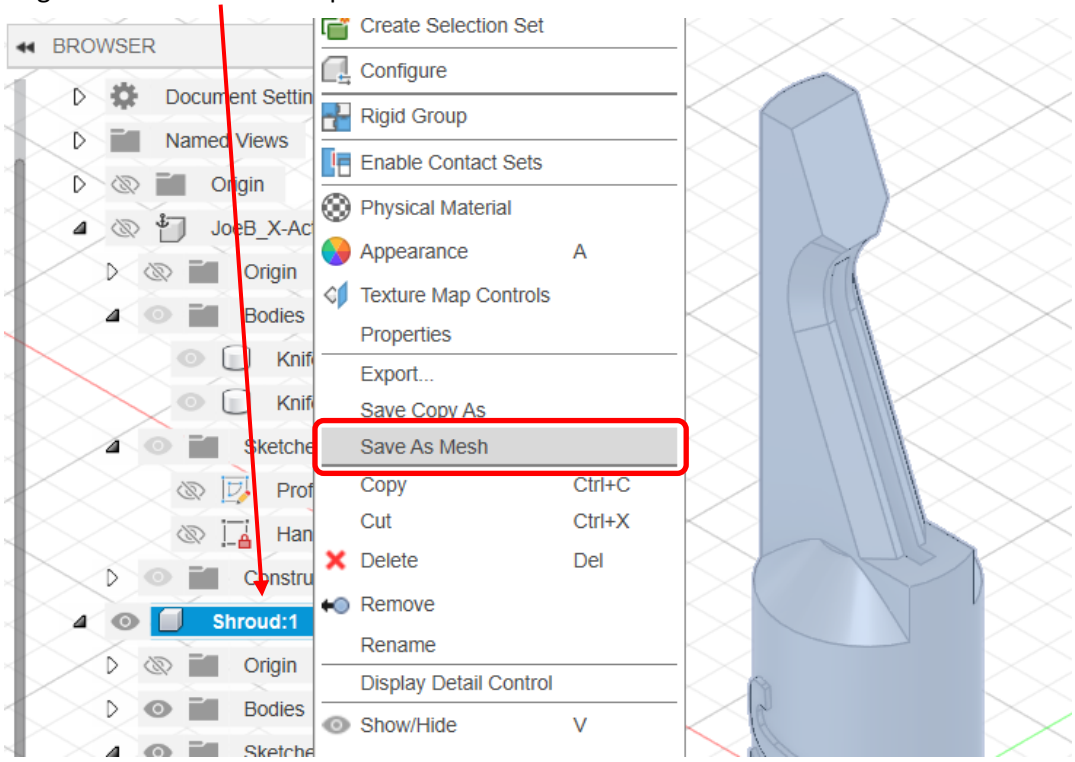


- return to the **Home View** and click on **eye** icon next to **Construction** to hide the axis

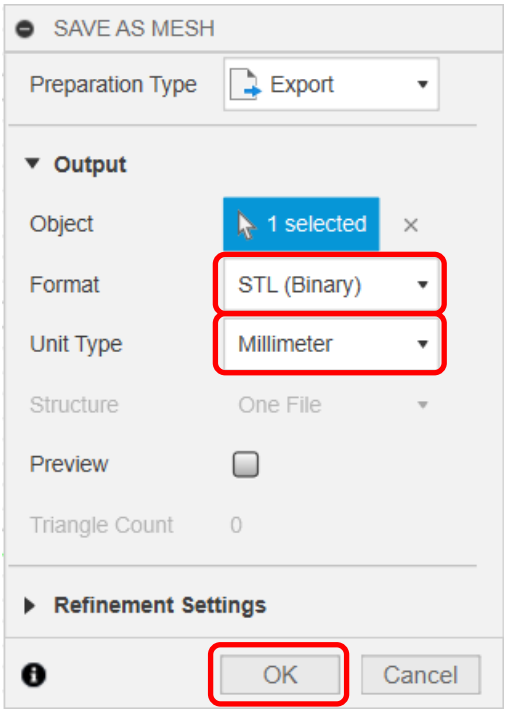


Creating an STL file

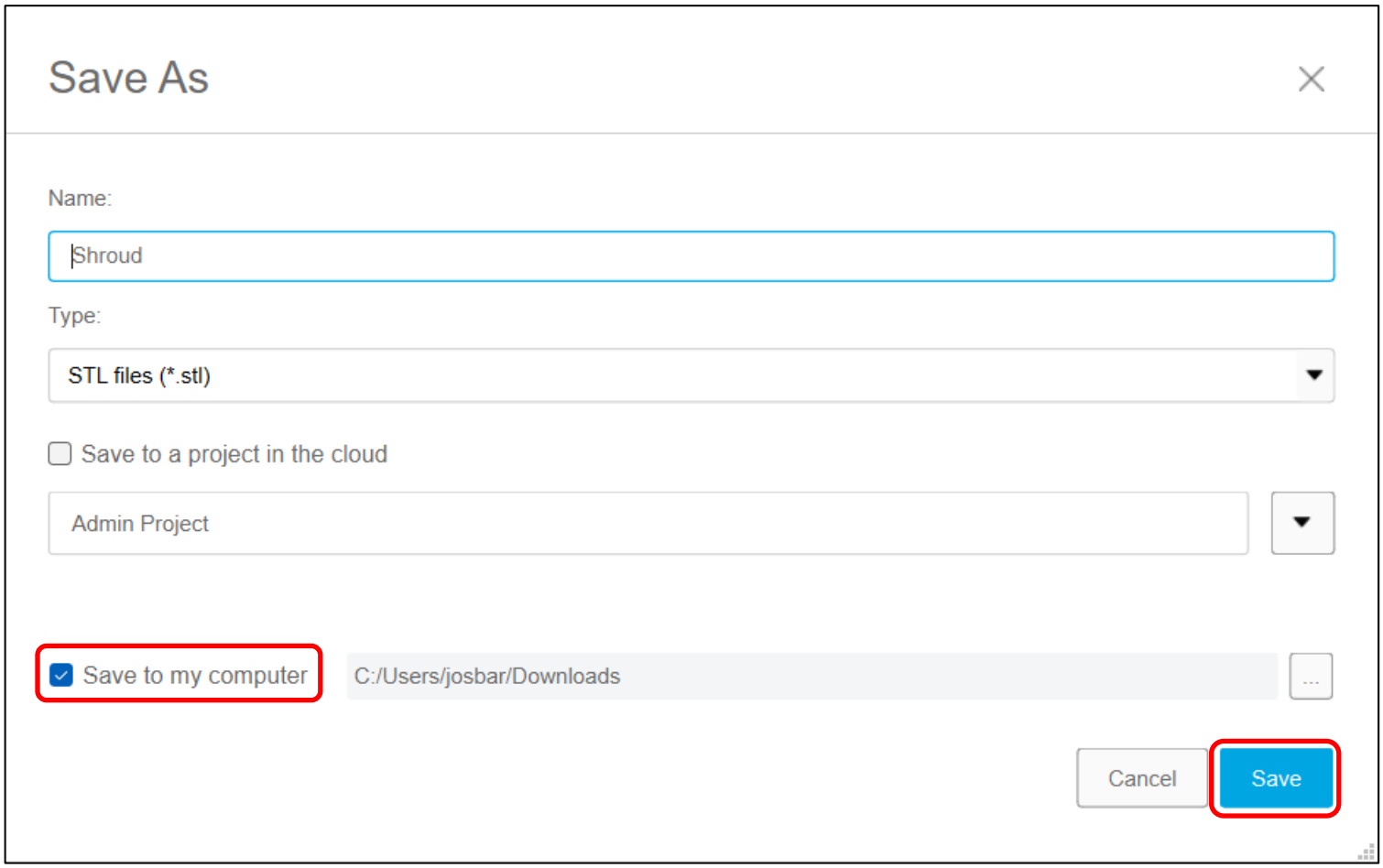
- right-click on **Shroud** Component name and select **Save As Mesh**



- set Format to **STL (Binary)**, Unit Type to **Millimeter**, and click **OK**



The name should be Shroud.
- ensure that **Save to my computer** is **checked** and note the save location. The **Downloads** folder should be the default.
- click **Save**



- the shroud should be printed upright as shown. **No supports are needed.** The right picture shows the shroud on an X-Acto knife.