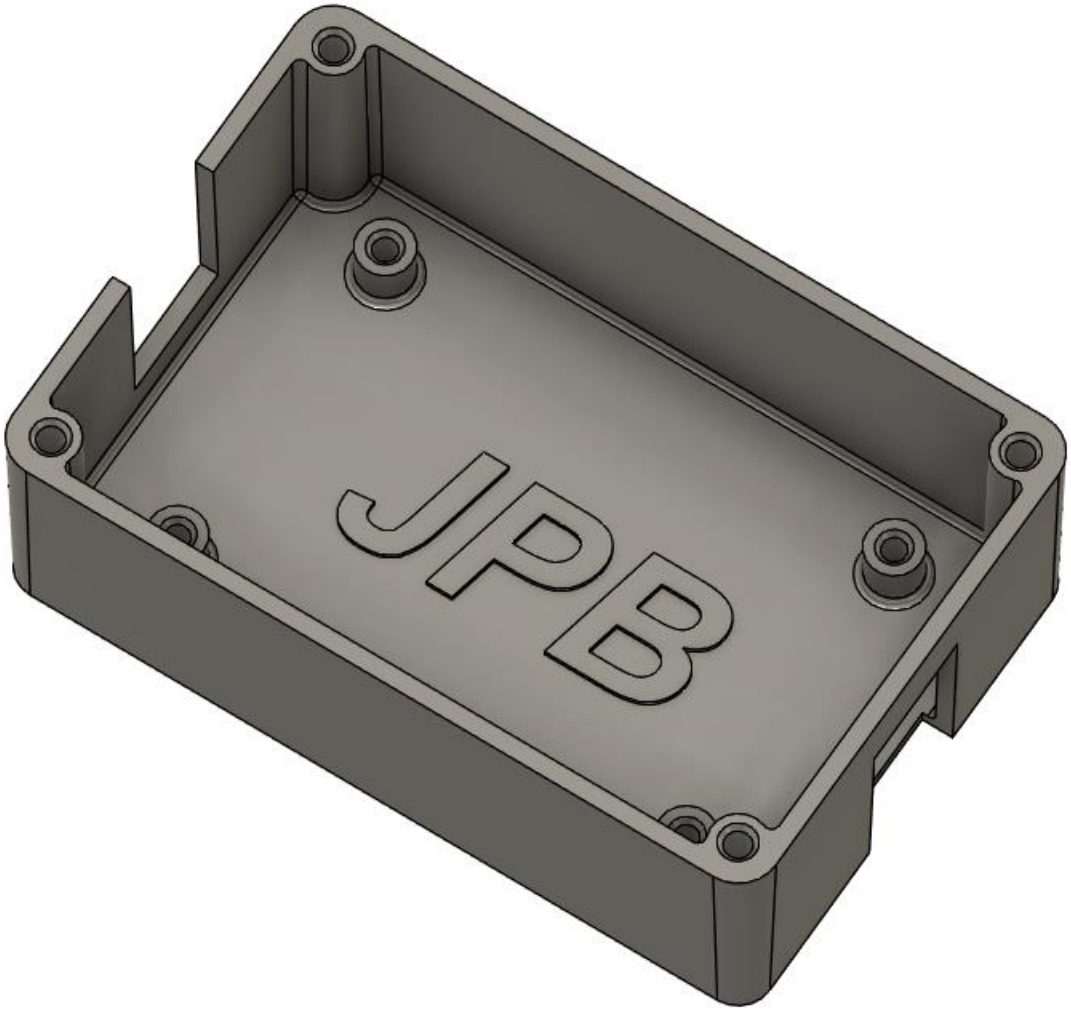
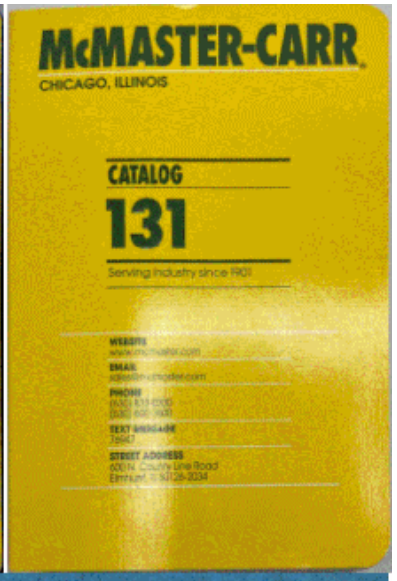
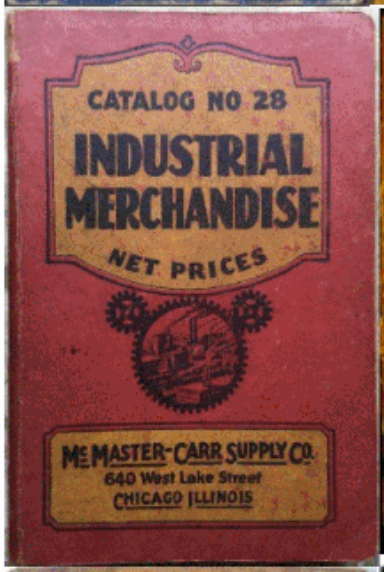
Mr. Barbetta's *"I don't care about 3D Printing. I just want the art credit."* class

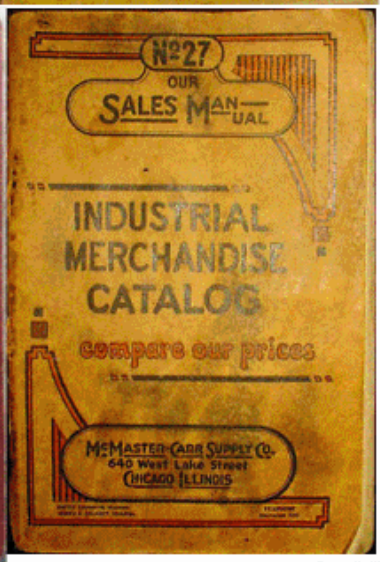
Make an Injection Molded Enclosure.



Today's Lesson is Sponsored by McMaster-Carr







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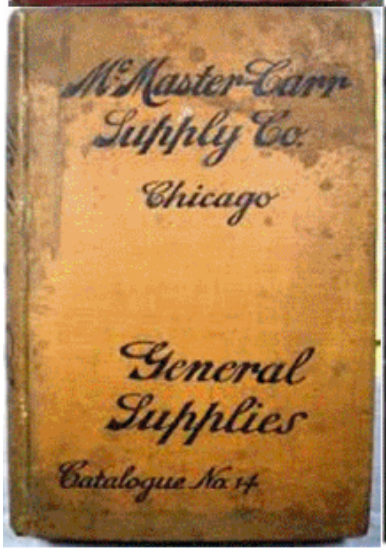

Contents

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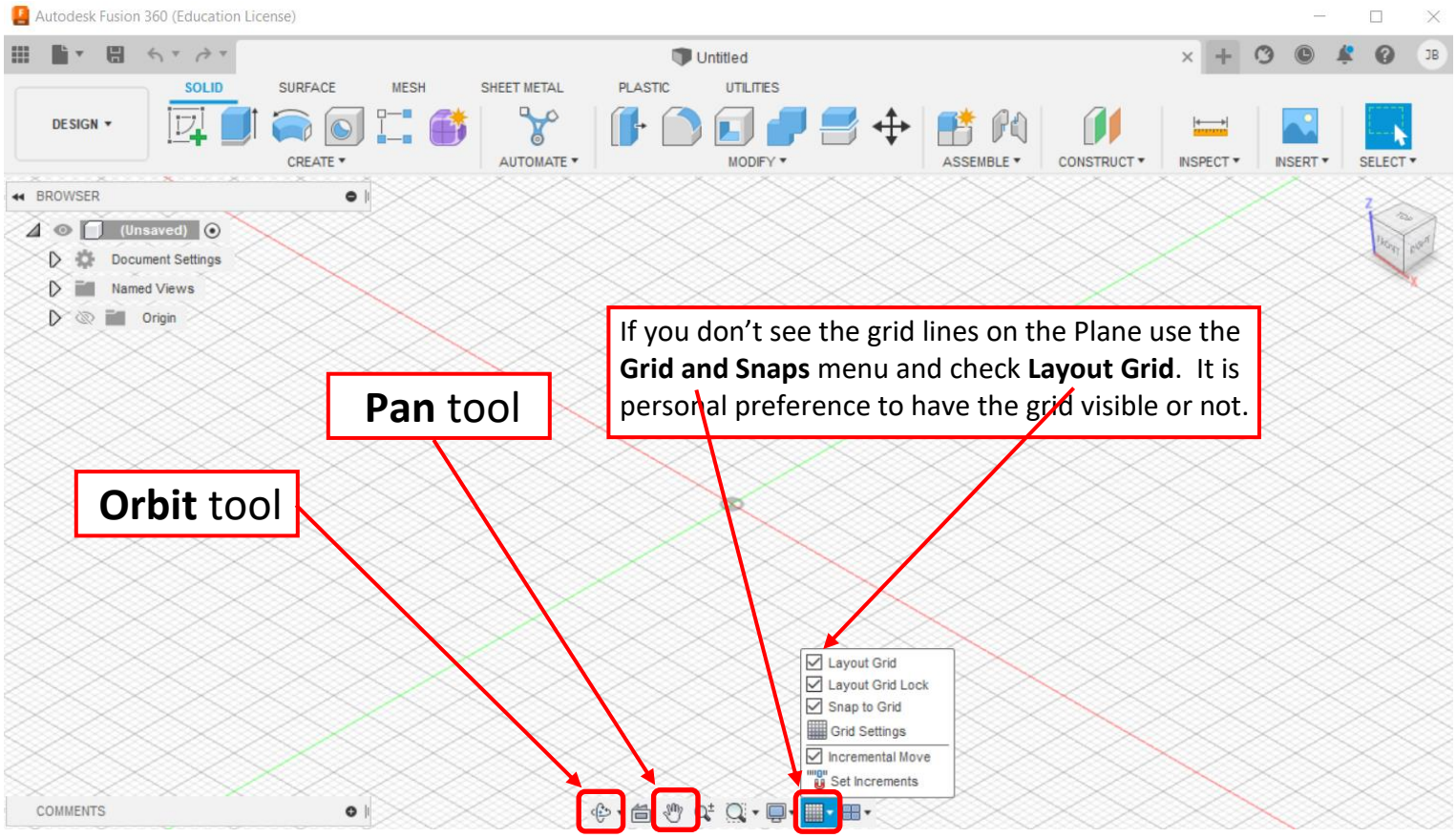
Changing the View of a Design

- 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

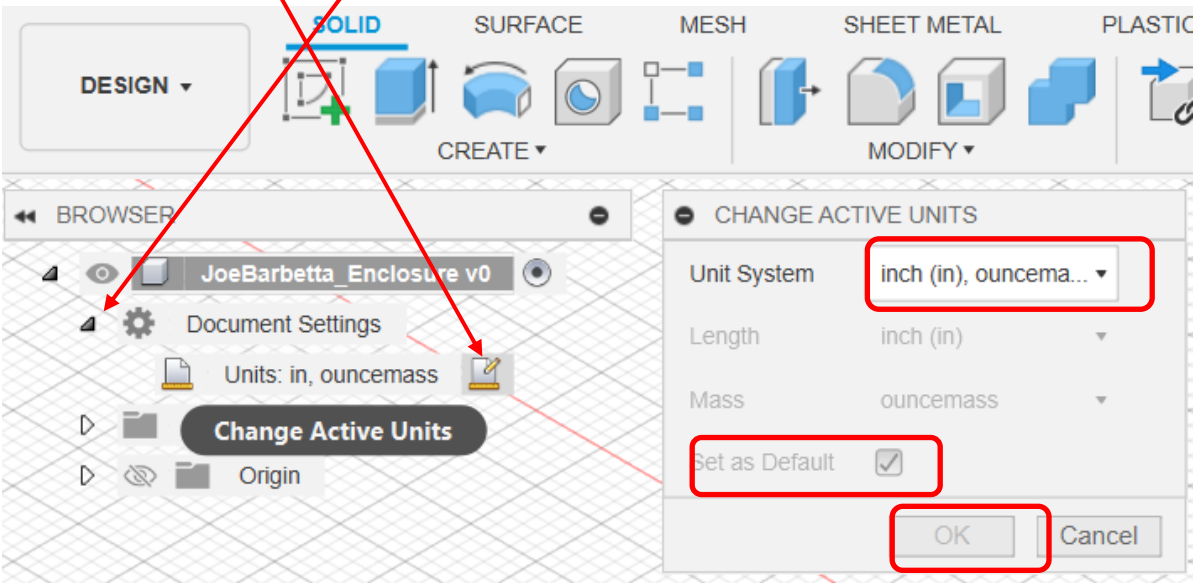
Starting a Design in Fusion (START HERE)

- 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 **_Hose** e.g. **JoeBarbetta_Hose** (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.

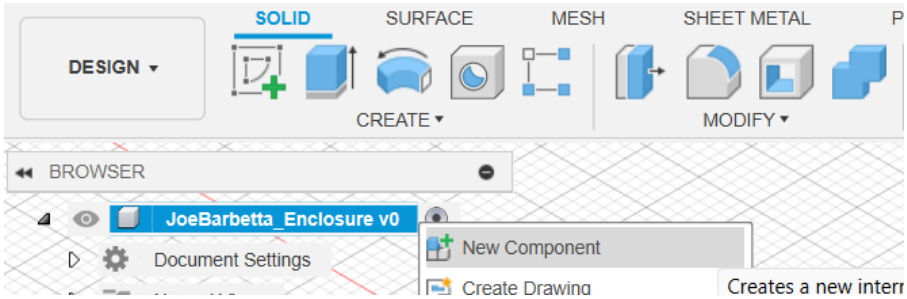


Note that the default units are in mm, which we just changed to inches.

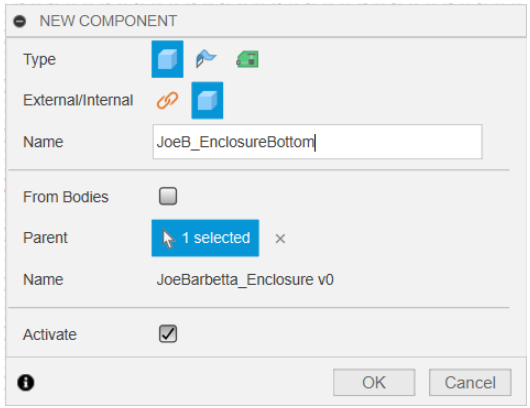
Did you know that the default units have changed over the years? The earliest version used cubits as the default unit.

Creating a New Component

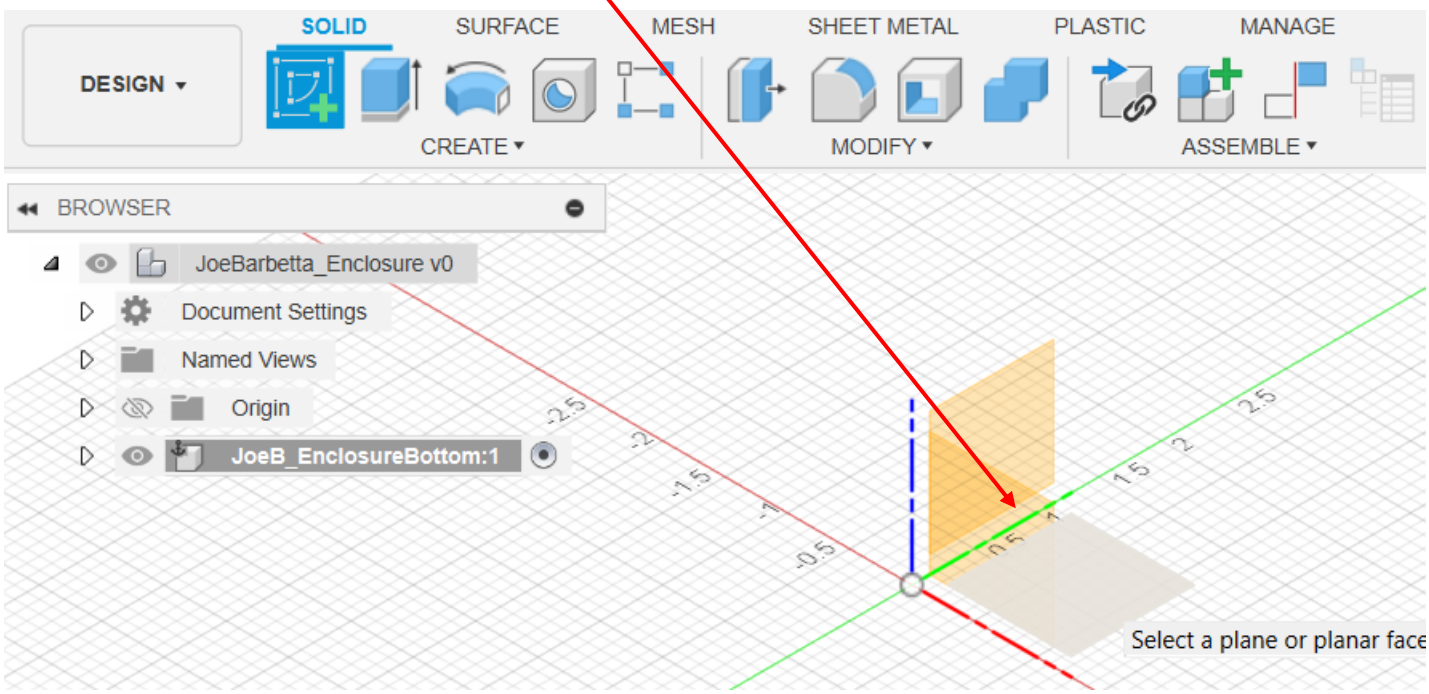
- right-click on the **Project Name** and select **New Component**



- in the **Name** box enter your **first name followed by your last name initial** and **_EnclosureBottom** (note the underscore), **e.g. JoeB_ EnclosureBottom**
- click **OK**

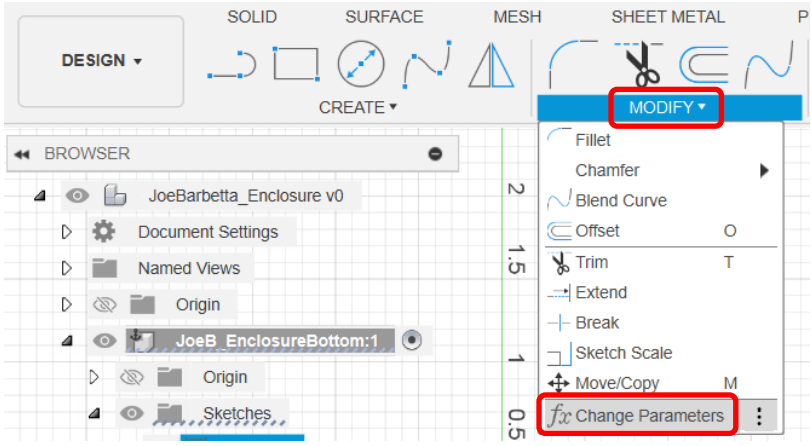


- select **Create Sketch** and click on the **bottom rhombus**

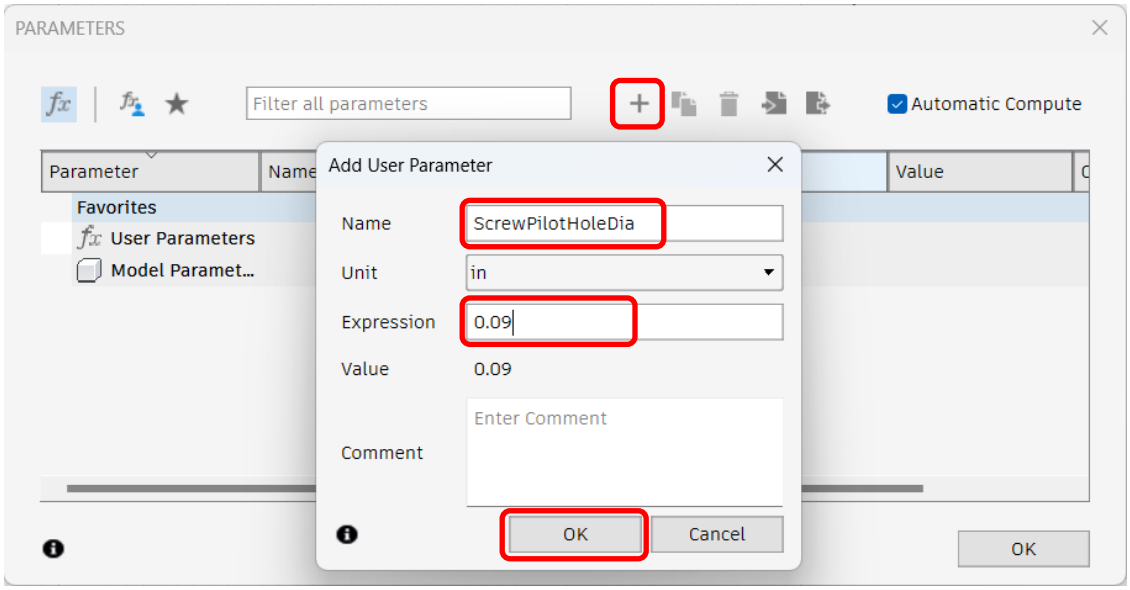


Setting User Parameters

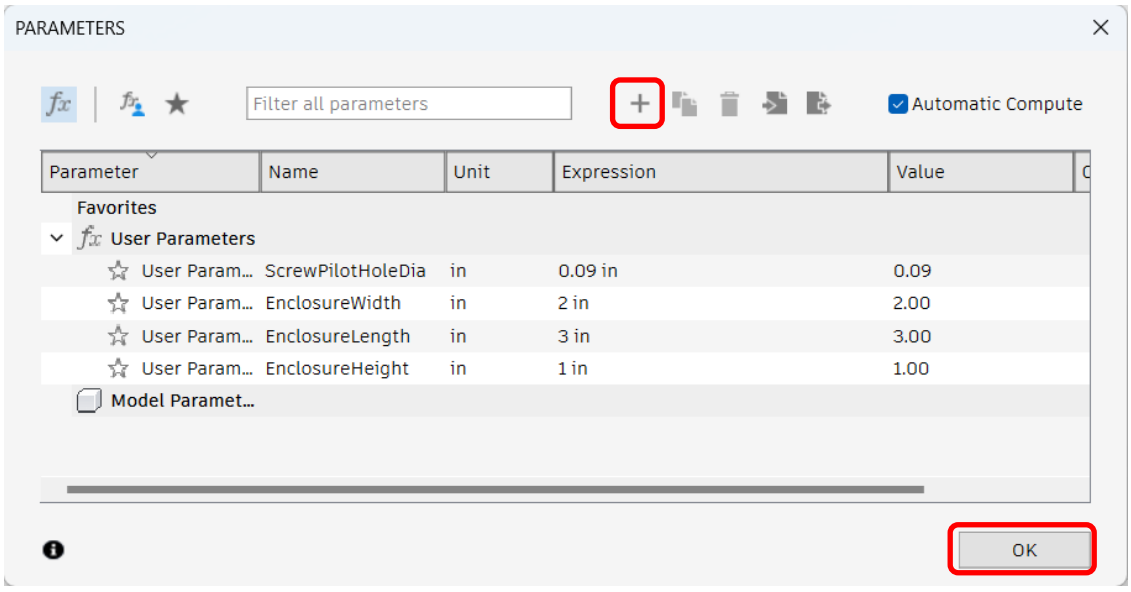
- from the **MODIFY** menu select **Change Parameters**

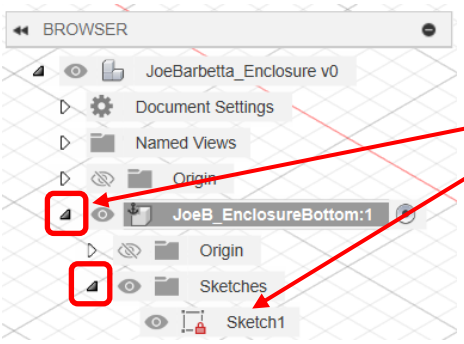


- click on the top **+** icon
- set the **Name** to **ScrewPilotHoleDia**, the **Expression** to **0.09**, and click **OK**



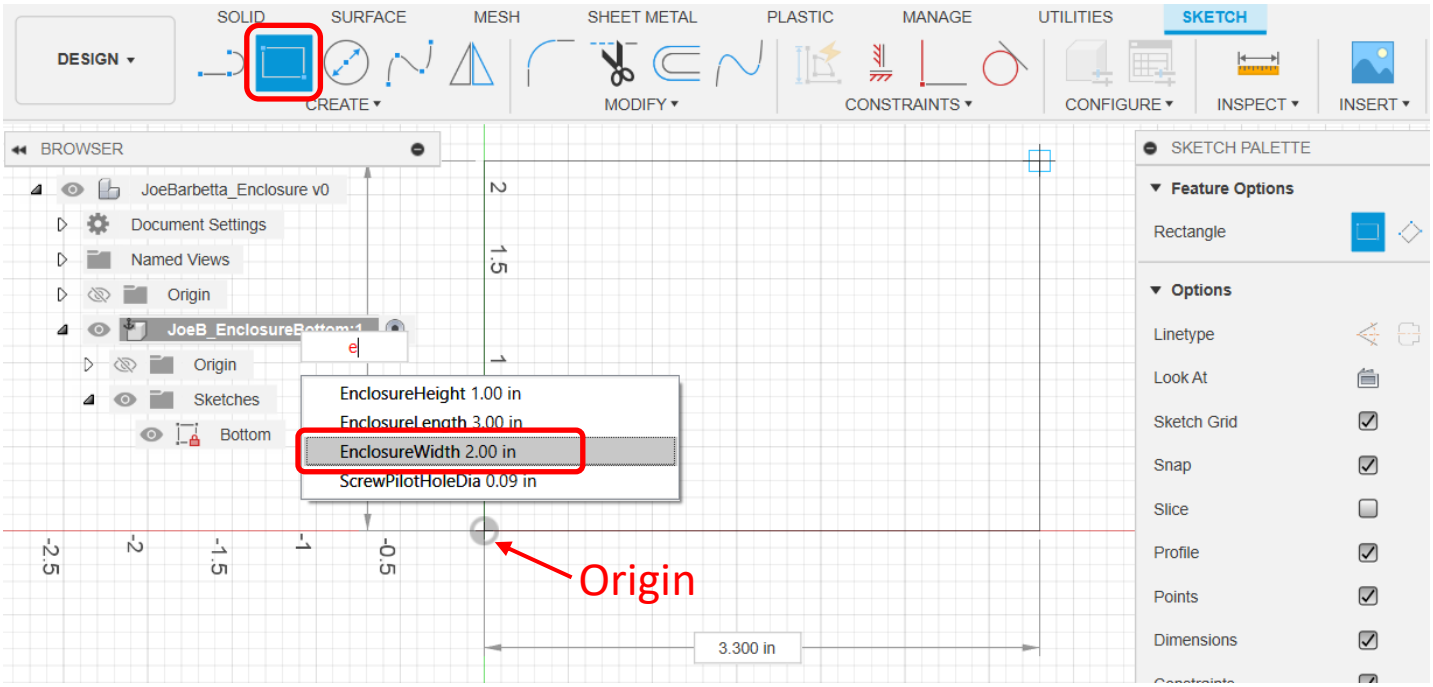
- use the **+** icon again to set **3 more** parameters: **EnclosureWidth 2.0**, **EnclosureLength 3.0**, and **EnclosureHeight 1.0**
- click **OK**





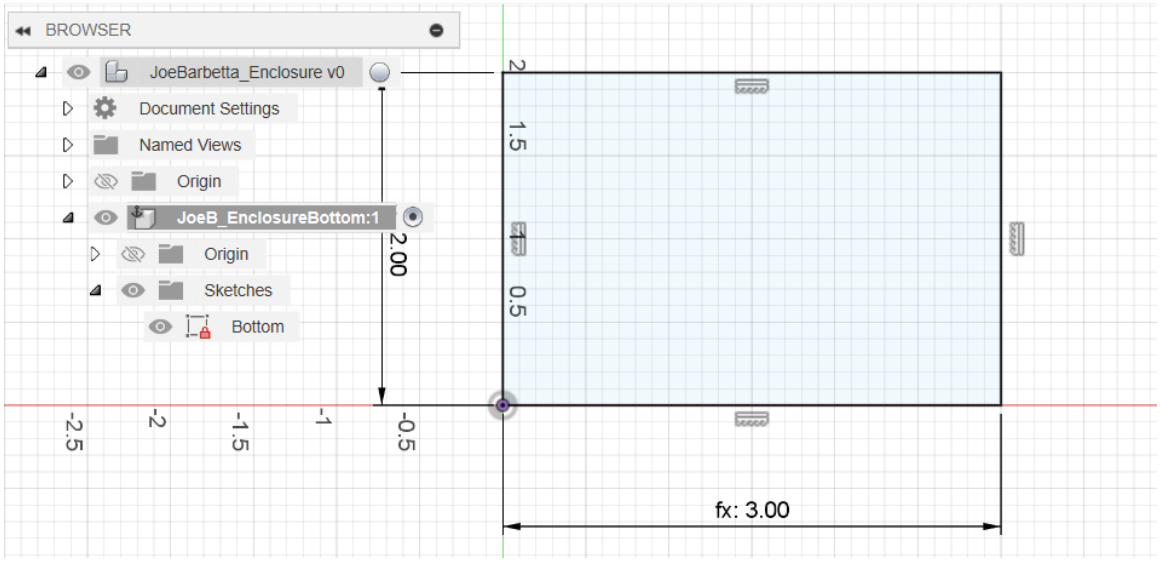
- click the **arrows** next to the **Component name** and the **Sketches** folder
- right-click on **Sketch1** and select **Rename** from the menu
- rename the Sketch to **Bottom**

- select the **Rectangle** tool
- click on the **Origin** and extend the rectangle up and to the right
- type **e**, use the **Down Arrow Key** to select **EnclosureWidth**, press the **Enter Key**

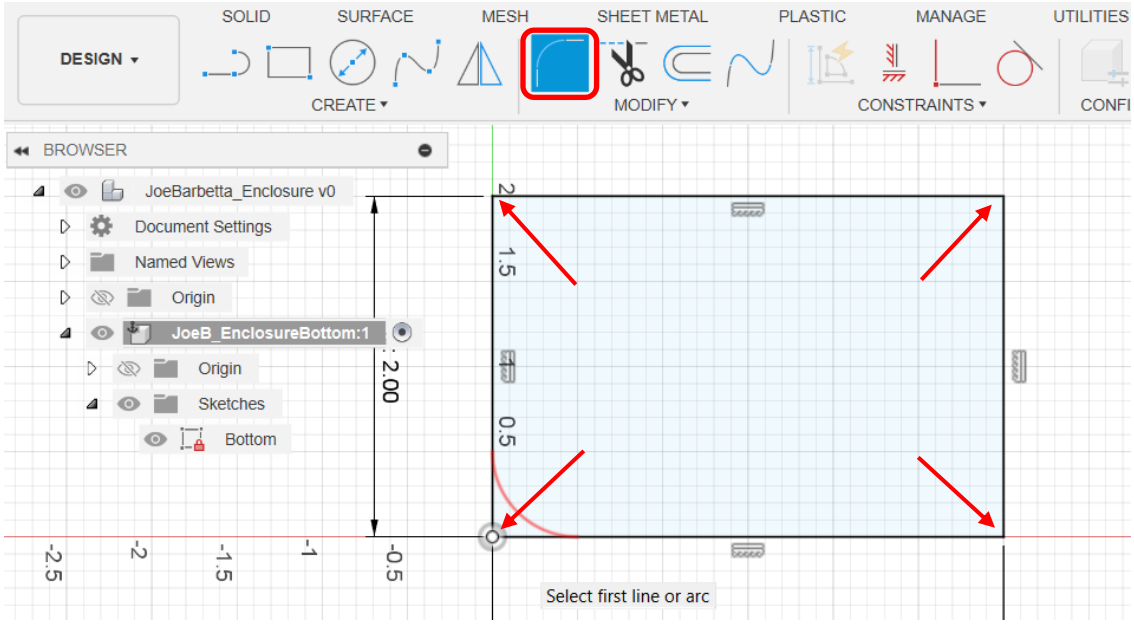


- press the **Tab key** to switch to the other dimension
- type **e**, use the **Down Arrow Key** to select **EnclosureLength**, press the **Enter Key** twice

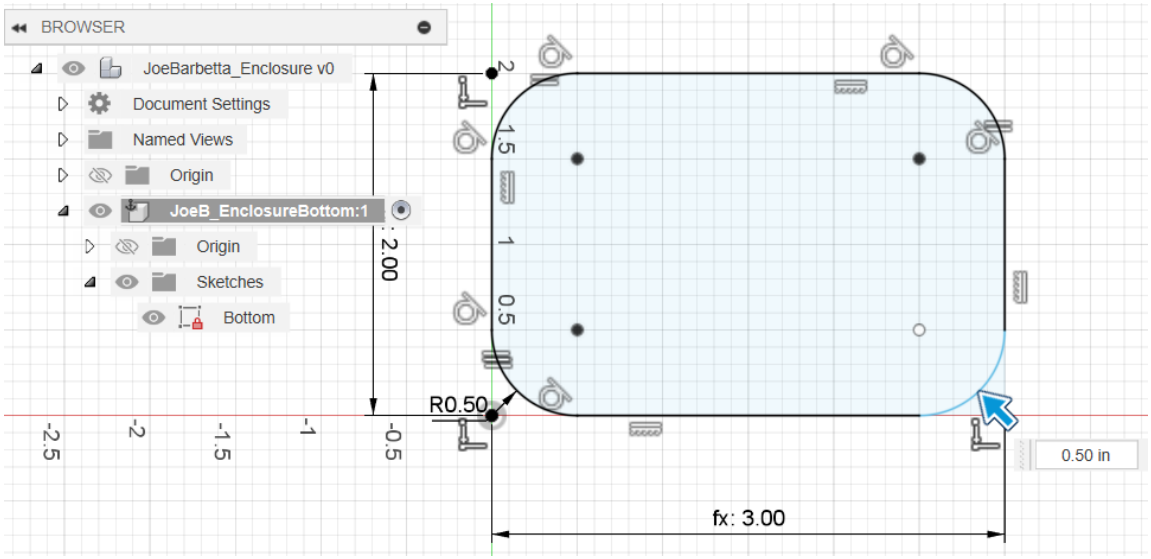
The rectangle should look like that below. It is fine if the dimension lines look different.



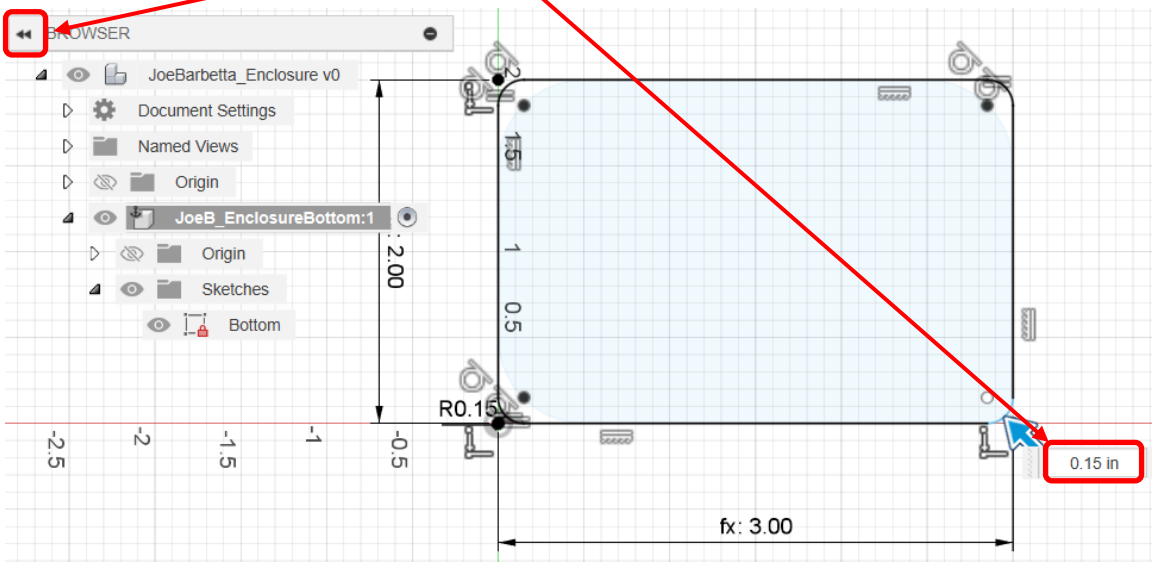
- select the **Fillet** tool. If it is not visible, find it in the MODIFY menu
- click on the **each corner** of the rectangle. Warnings that appear during the fillet operation can be ignored.



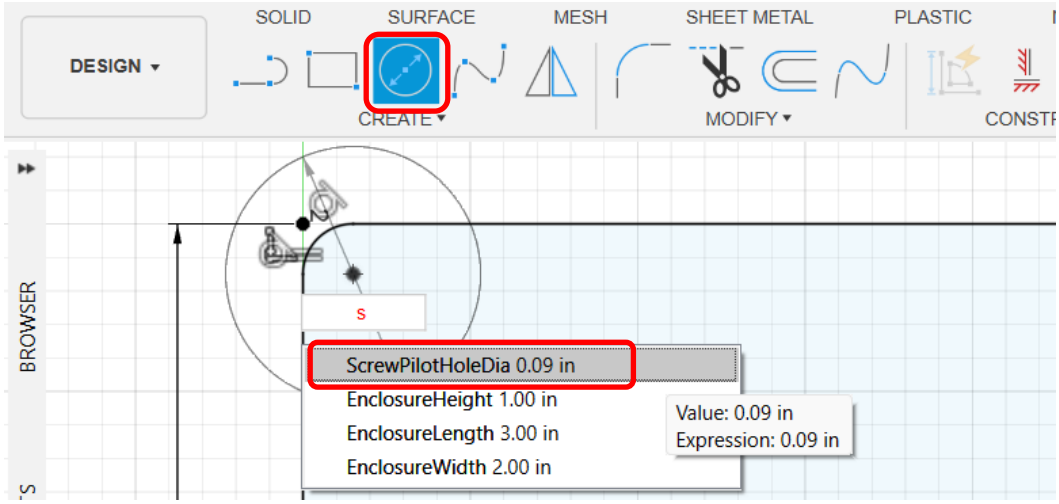
After each corner is clicked it should look like that below.



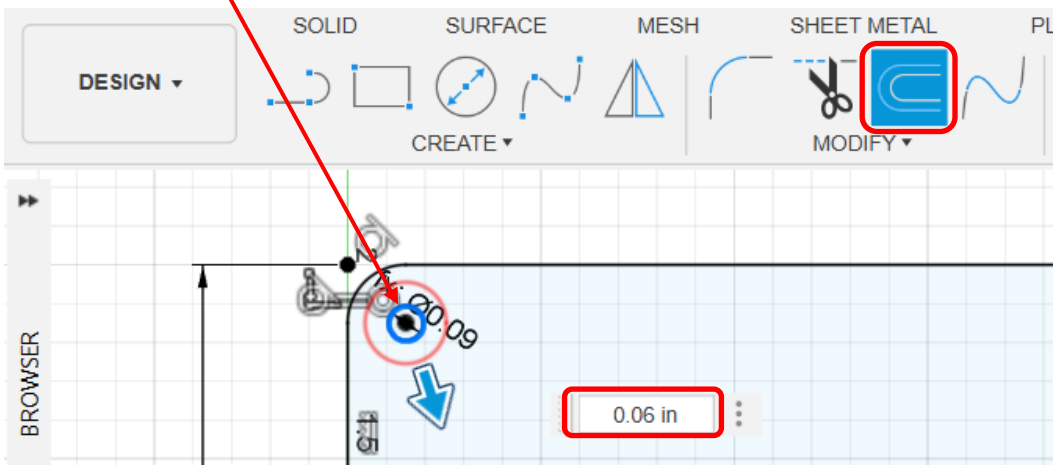
- change the value in the value box to **0.15** and press the **Enter key**
- click on the **double arrow** to hide the BROWSER



- select the **Center Diameter Circle** tool. If it is not visible, find it in the CREATE menu.
- click on the **radius center point**, extend the **circle outward**, type **s**, press the **Enter key** to select **ScrewPilotHoleDia**, and press the **Enter key** again to complete the circle

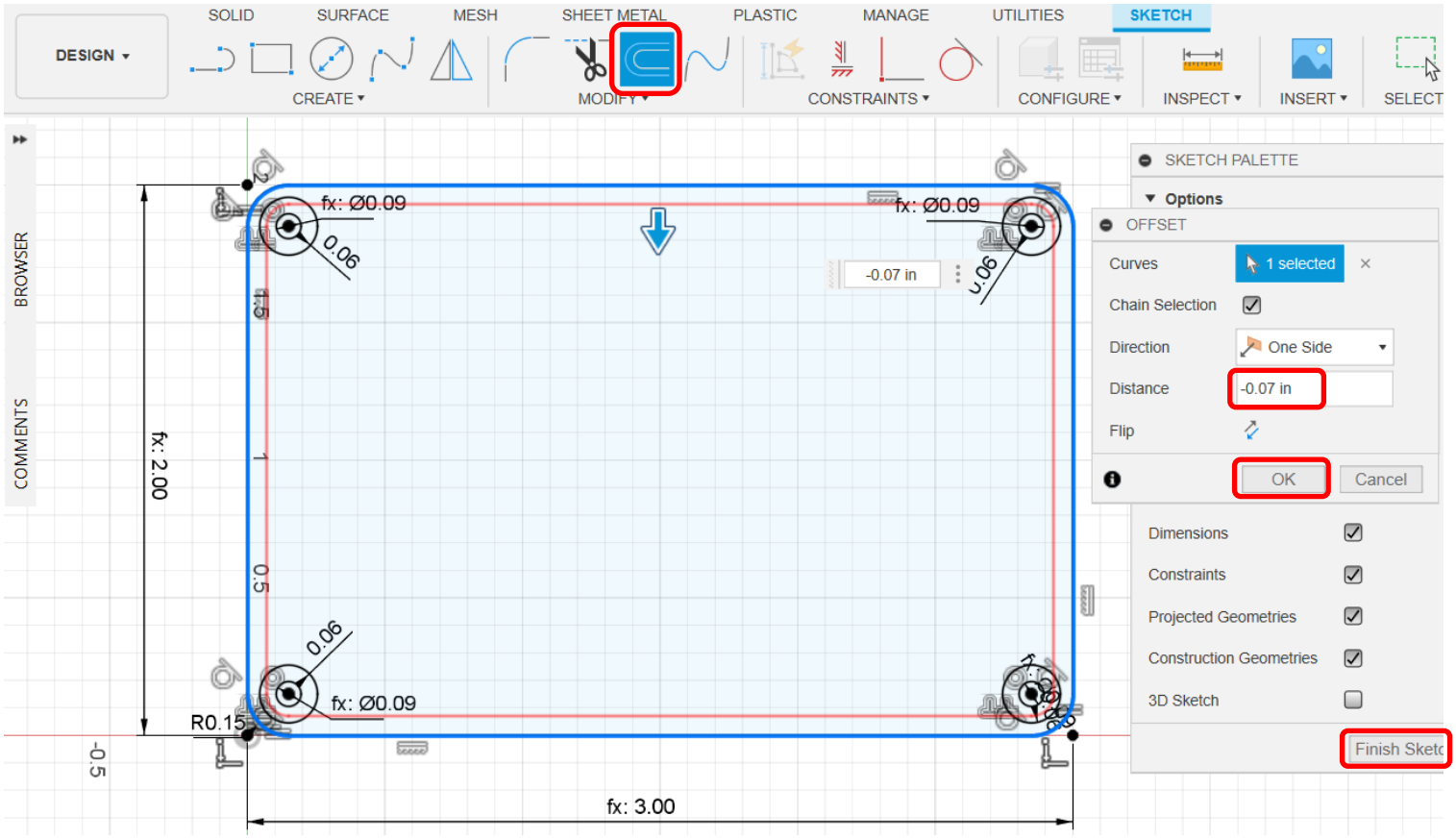


- select the **Offset** tool. If it is not visible, find it in the MODIFY menu.
- click on the **small circle** just created and enter a value of **0.06** in the value box



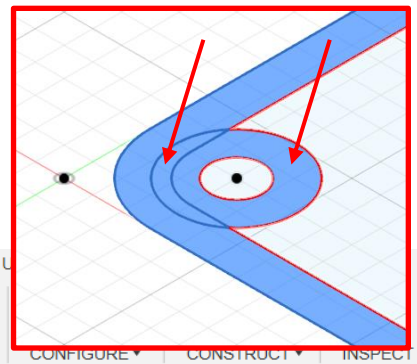
The result should look like that below.

- select the **Offset** tool again and click on the **rectangle**
- enter **-0.07** (note the minus sign) in the **Distance** box and click **OK**
- click **Finish Sketch**

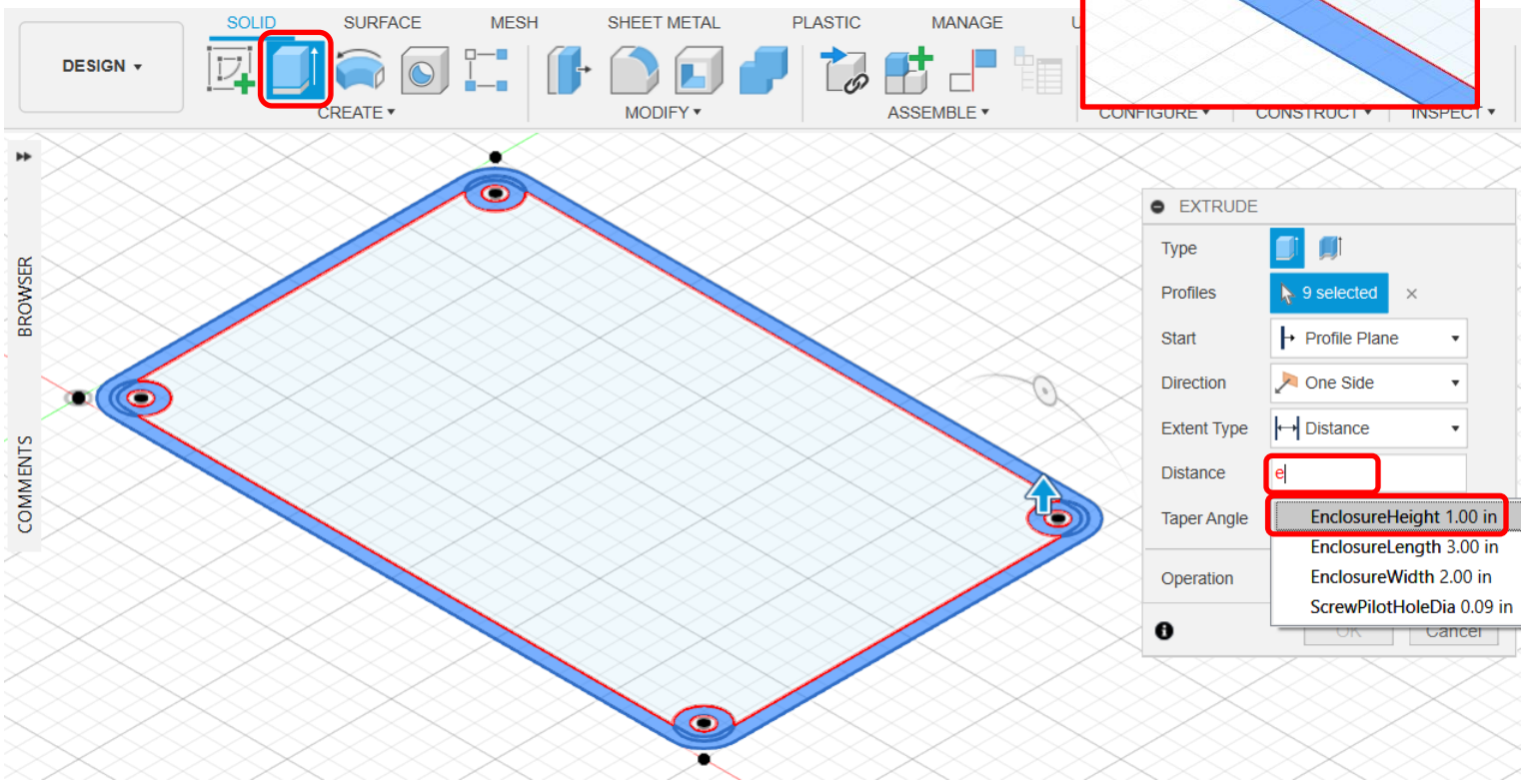


- click on the **Home** icon at the **View Cube**
- select the **Extrude** tool
- click on the outer rectanglar region and then on the 2 regions at each corner

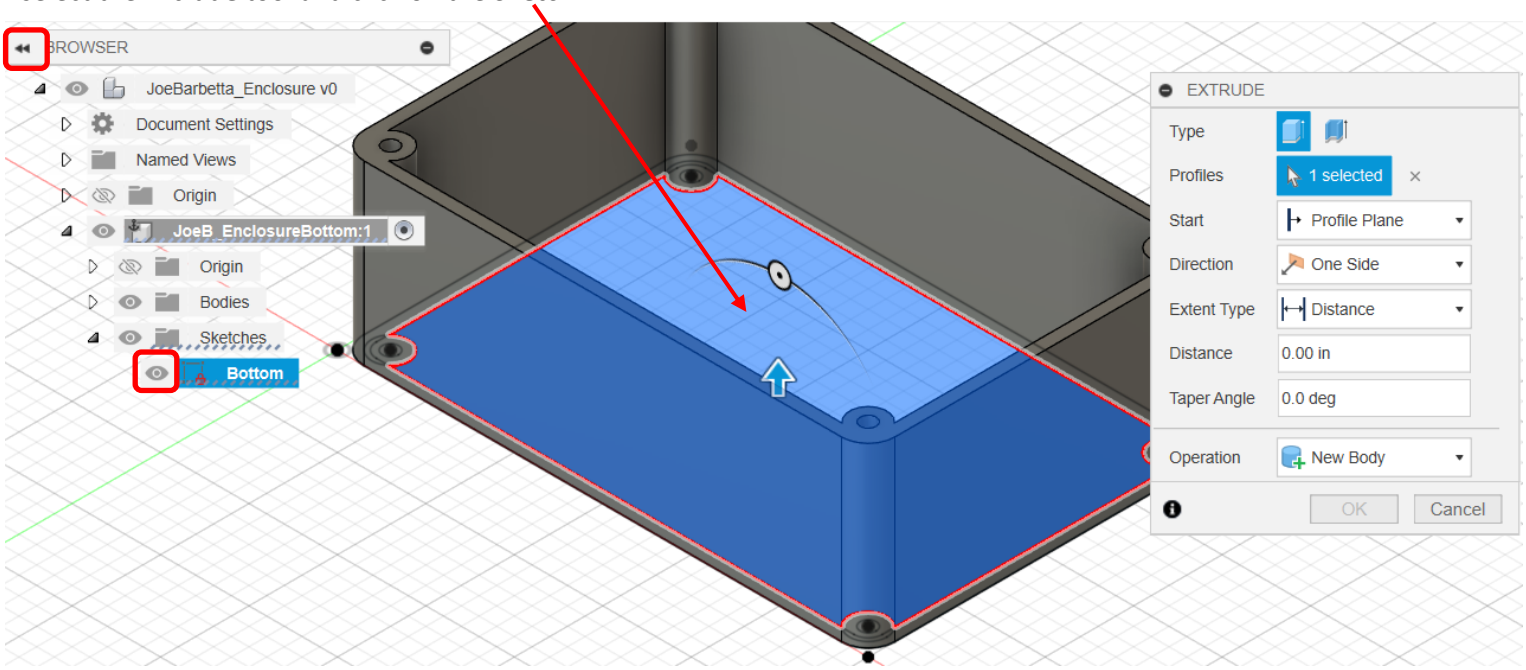
It will help to zoom into each corner when doing so. The inset image shows a closeup of a corner. Note that the inside of the small circles is not selected.



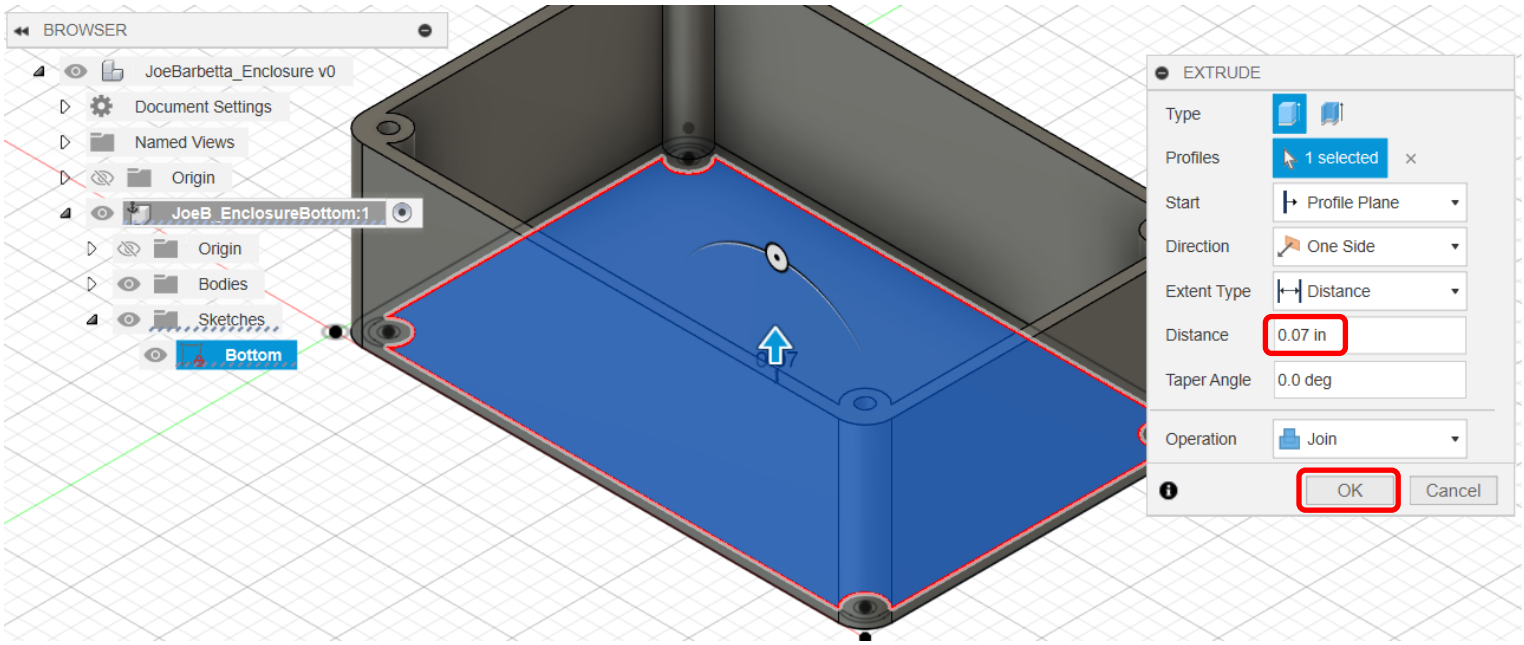
- in the **Distance** box type **e** and select **EnclosureHeight** and click **OK**



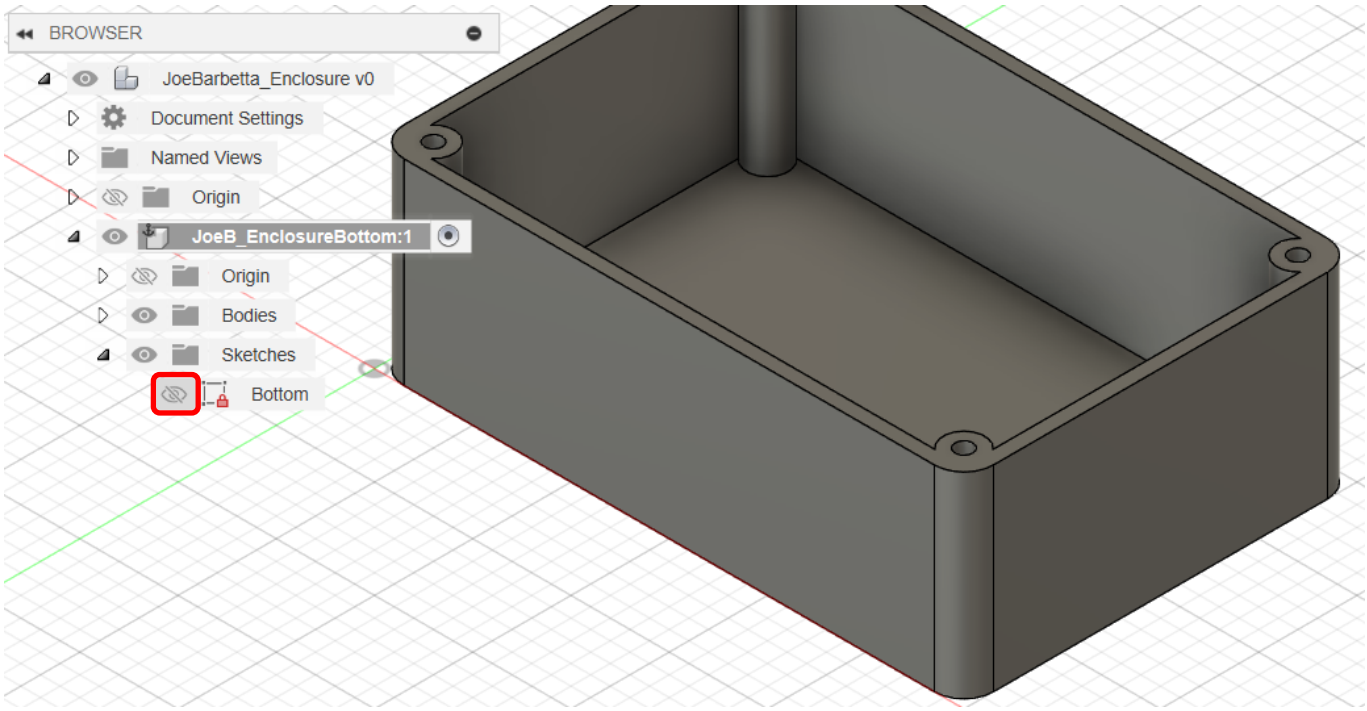
- click on the **double arrow** to reopen the BROWSER
- click on the **eye** icon for the **Bottom** sketch to make it visible again
- select the **Extrude** tool and click on the **Sketch**



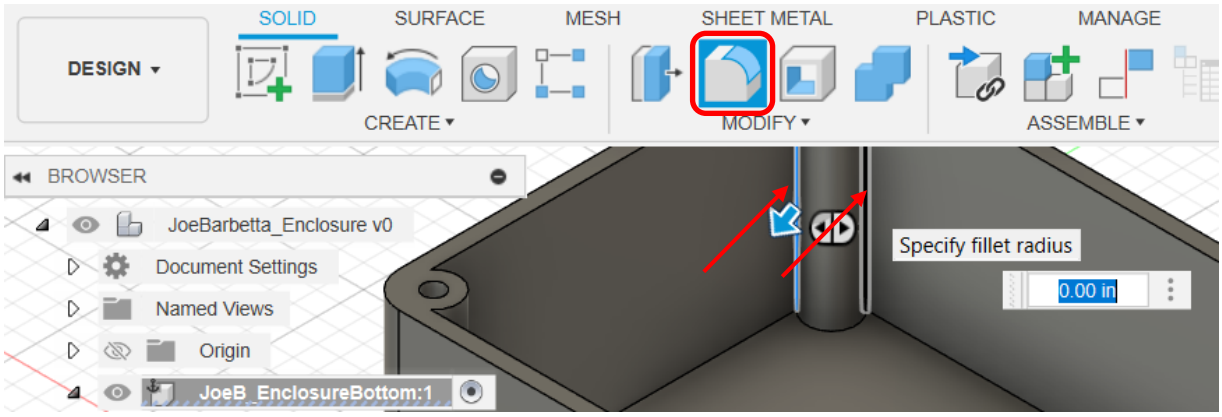
- for **Distance** enter **0.07** and click **OK**



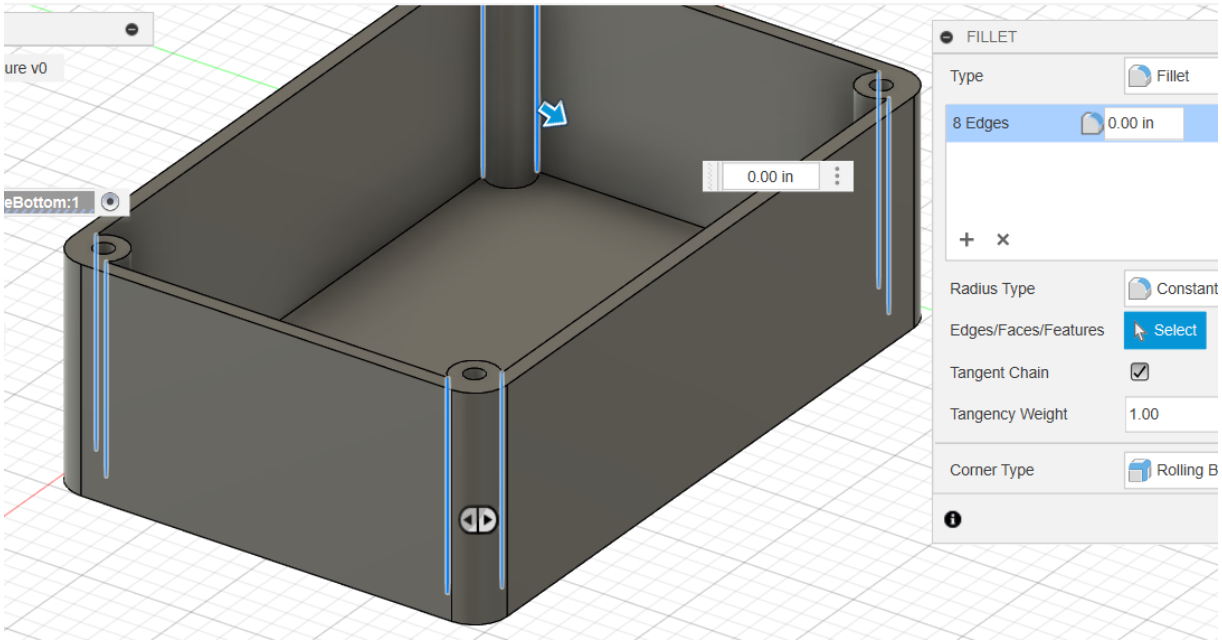
- click on the **eye** icon for Button to hide it



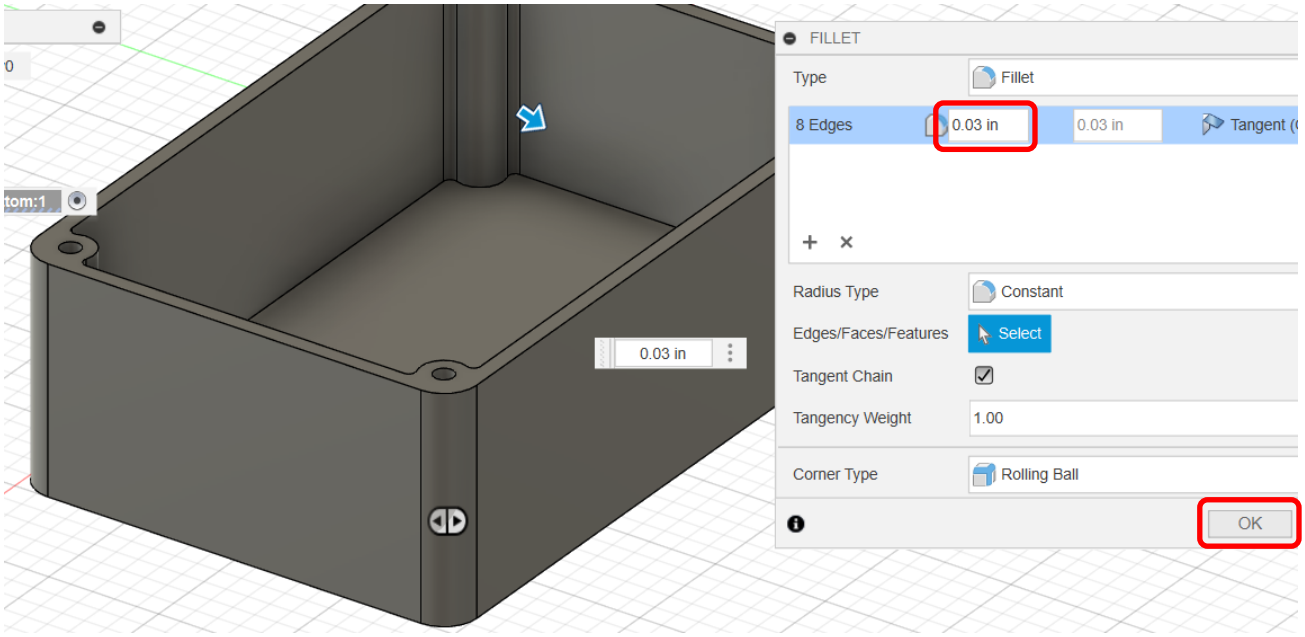
- select the **Fillet** tool. If it is not visible, find it in the MODIFY menu
- click on **edges** at the **intersection on the cylindircal feature and the inside walls**



- adjust the **View Cube** to access **all 4** interior corners. There should be **8 edges** highlighted in blue as shown below.

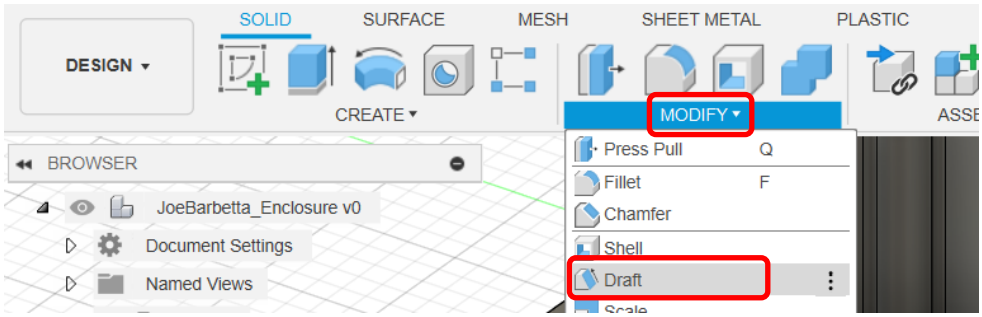


- enter **0.03** and click **OK**

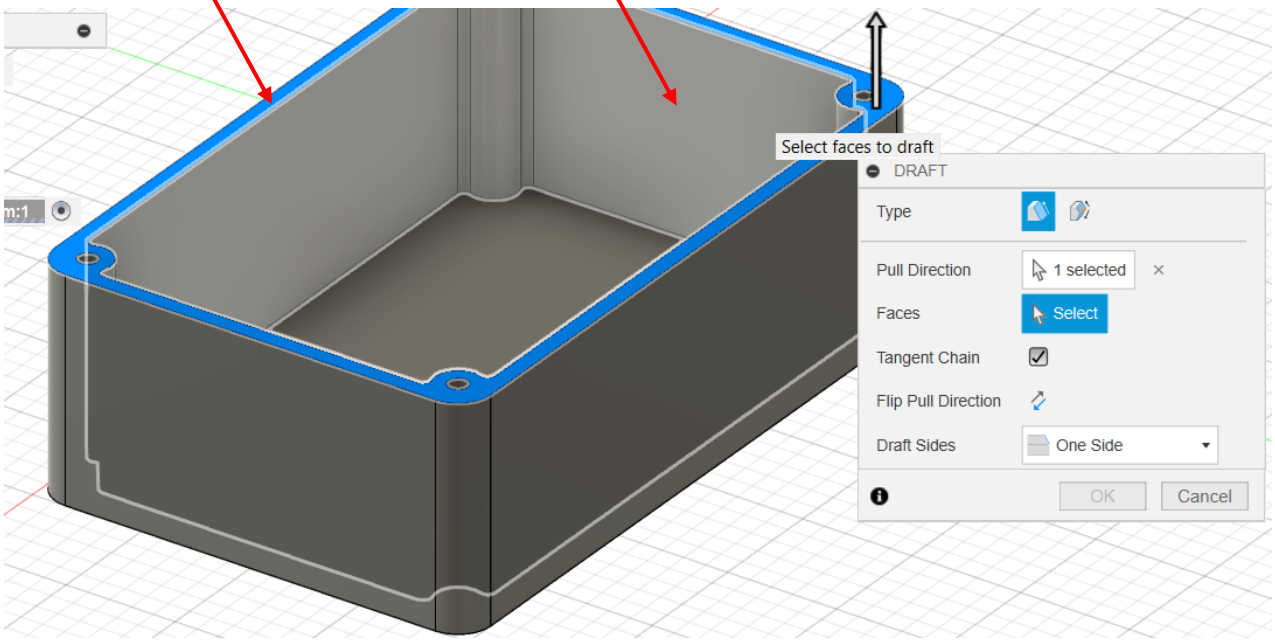


Adding Drafts

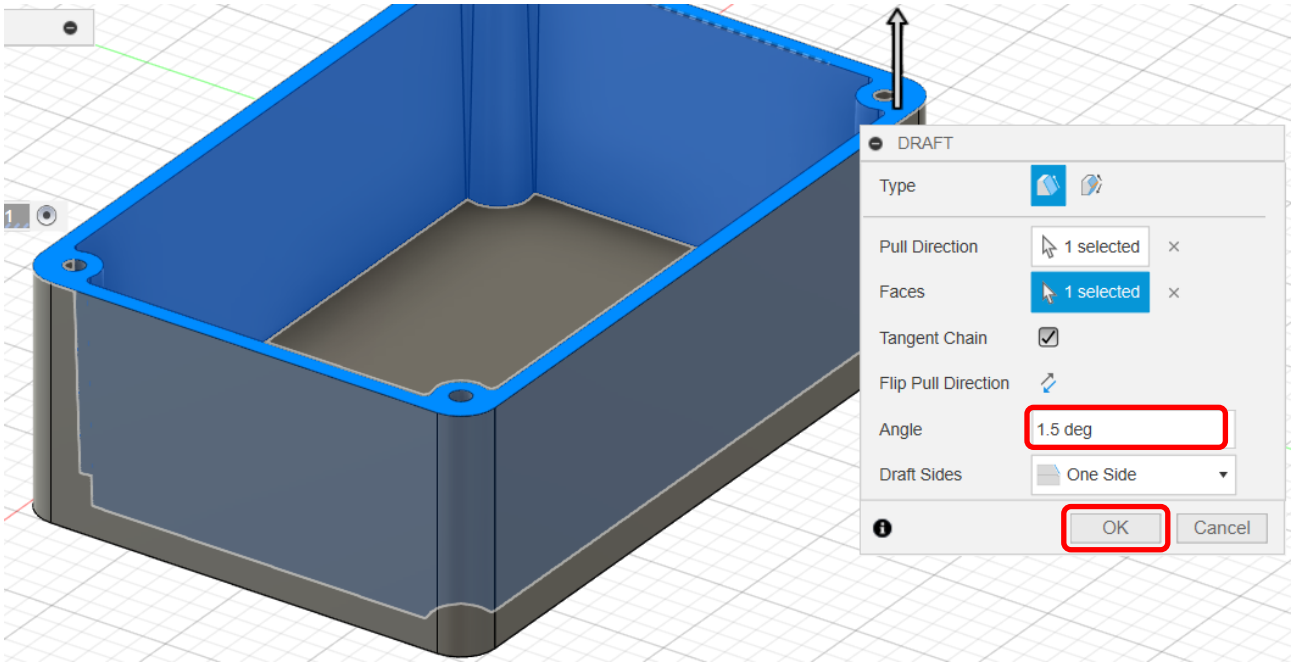
- from the **MODIFY** menu select **Draft**



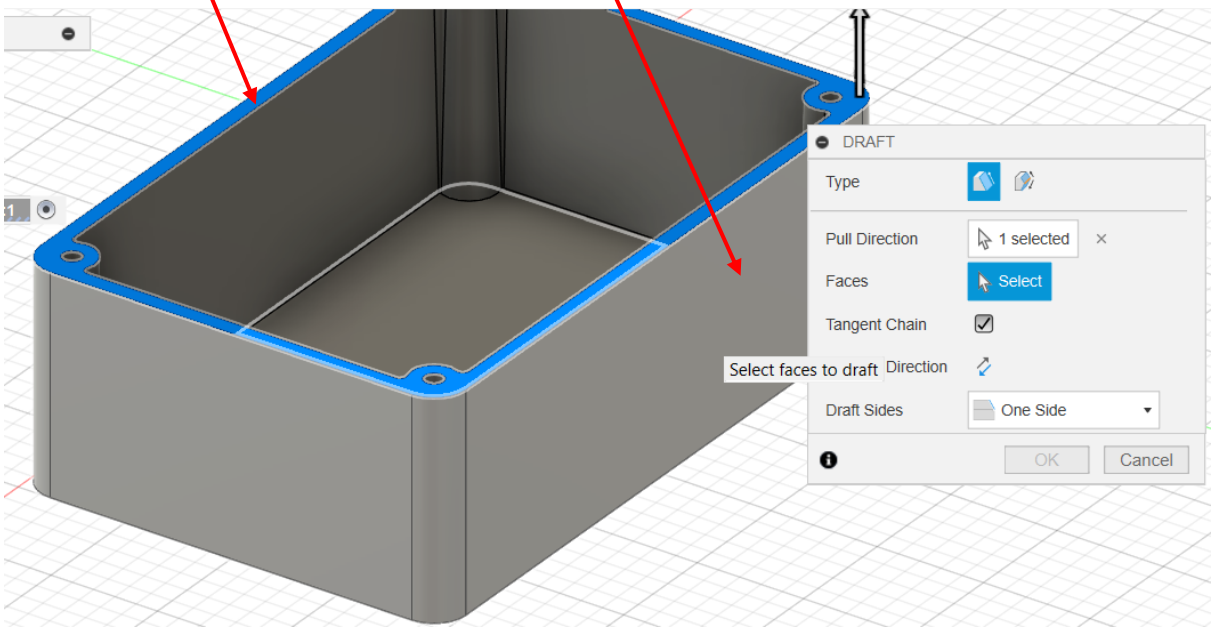
- click on the **top surface** and then click on an **interior wall**



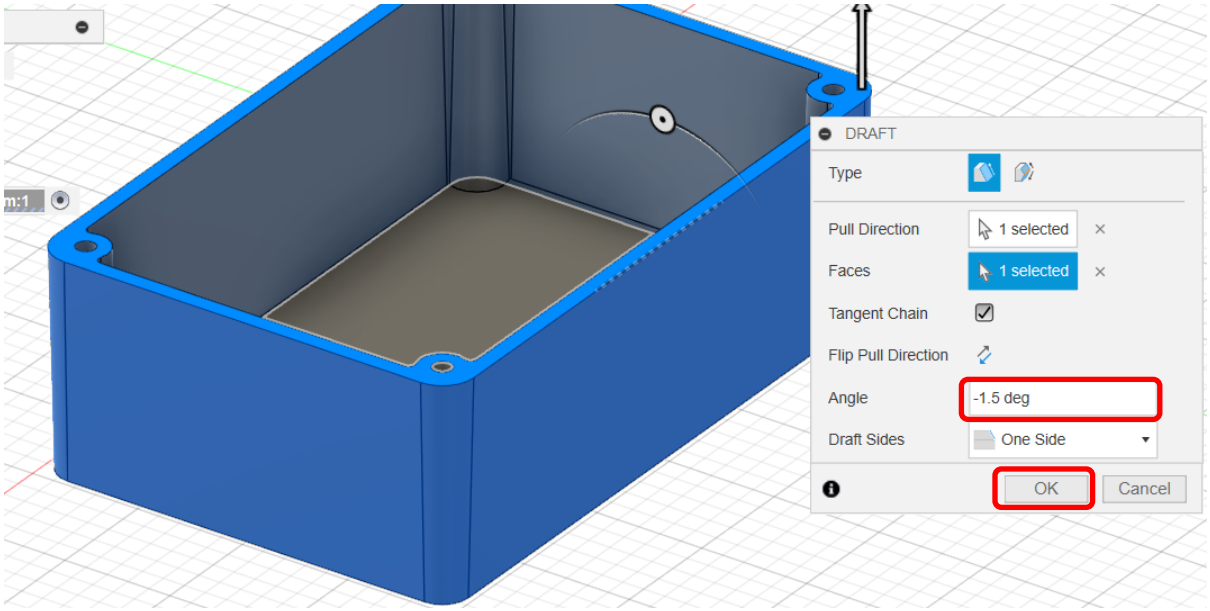
- enter **1.5** for **Angle** and click **OK**

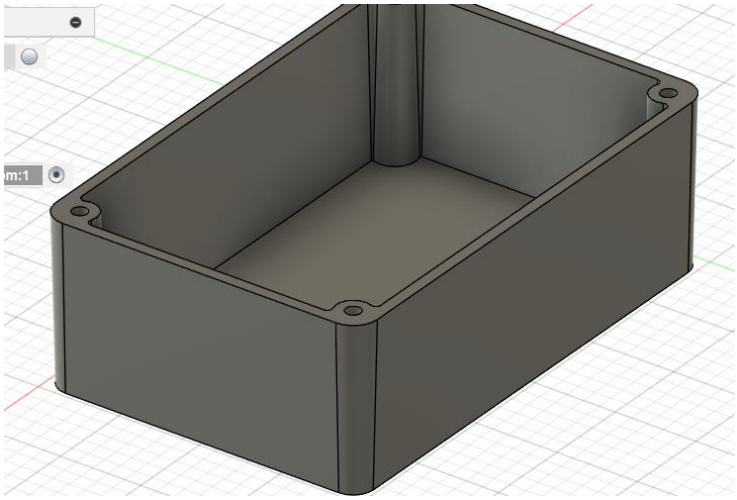


- select the **Draft** tool again
- click on the **top surface** and then click on an **exterior wall**



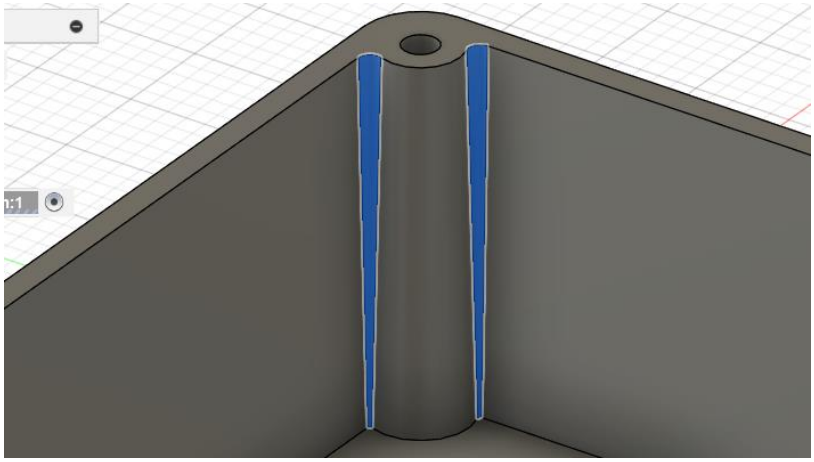
- enter **-1.5** (note the minus sign) for **Angle** and click **OK**



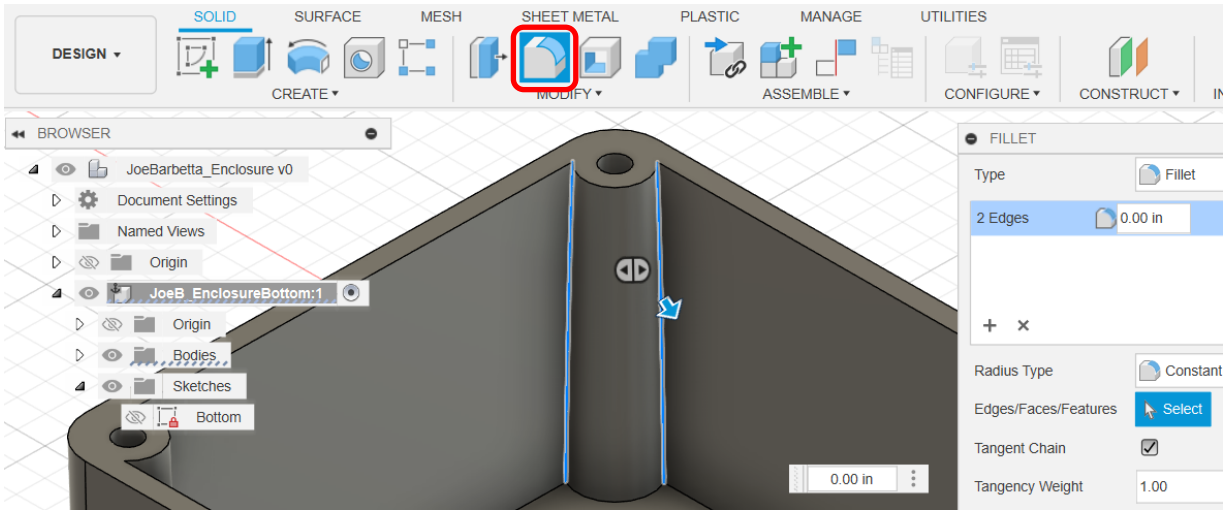


This is the result of appling draft.

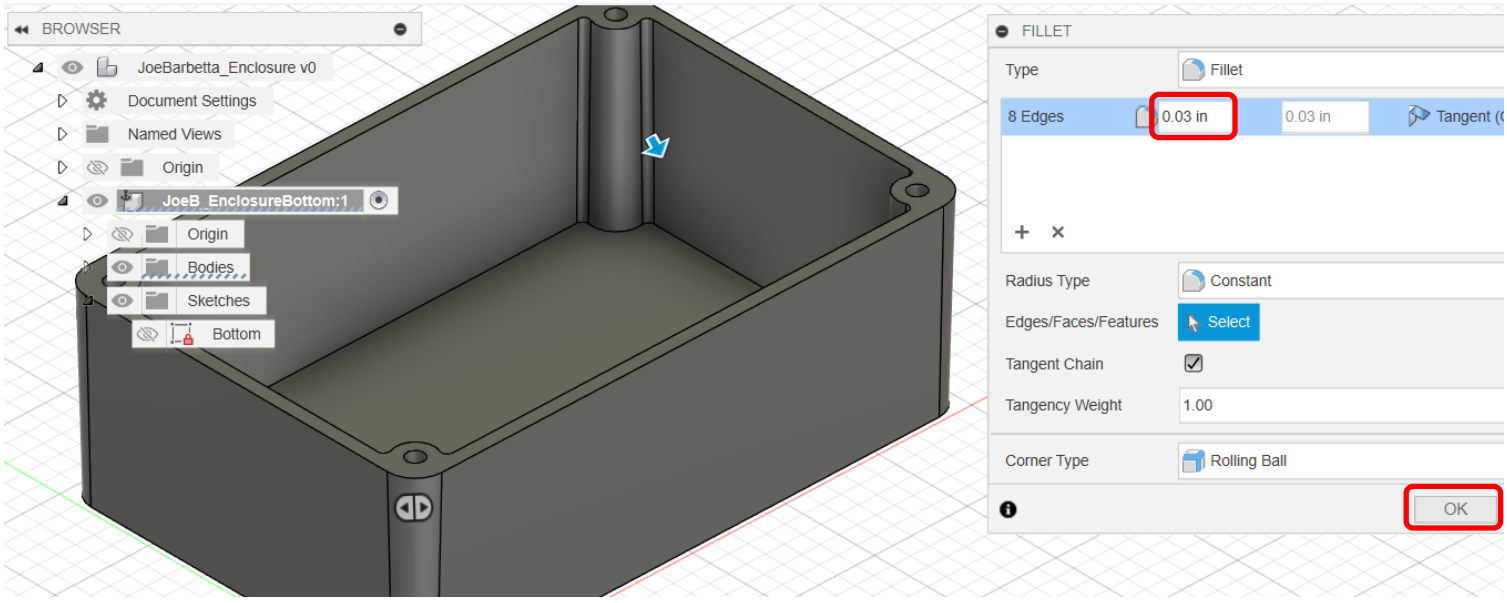
- zoom into an interior corner
- **hold the Shift key down** and select the **2 fillets** and press the **Delete key**
- perform the same operation at the **3 other interior corners**



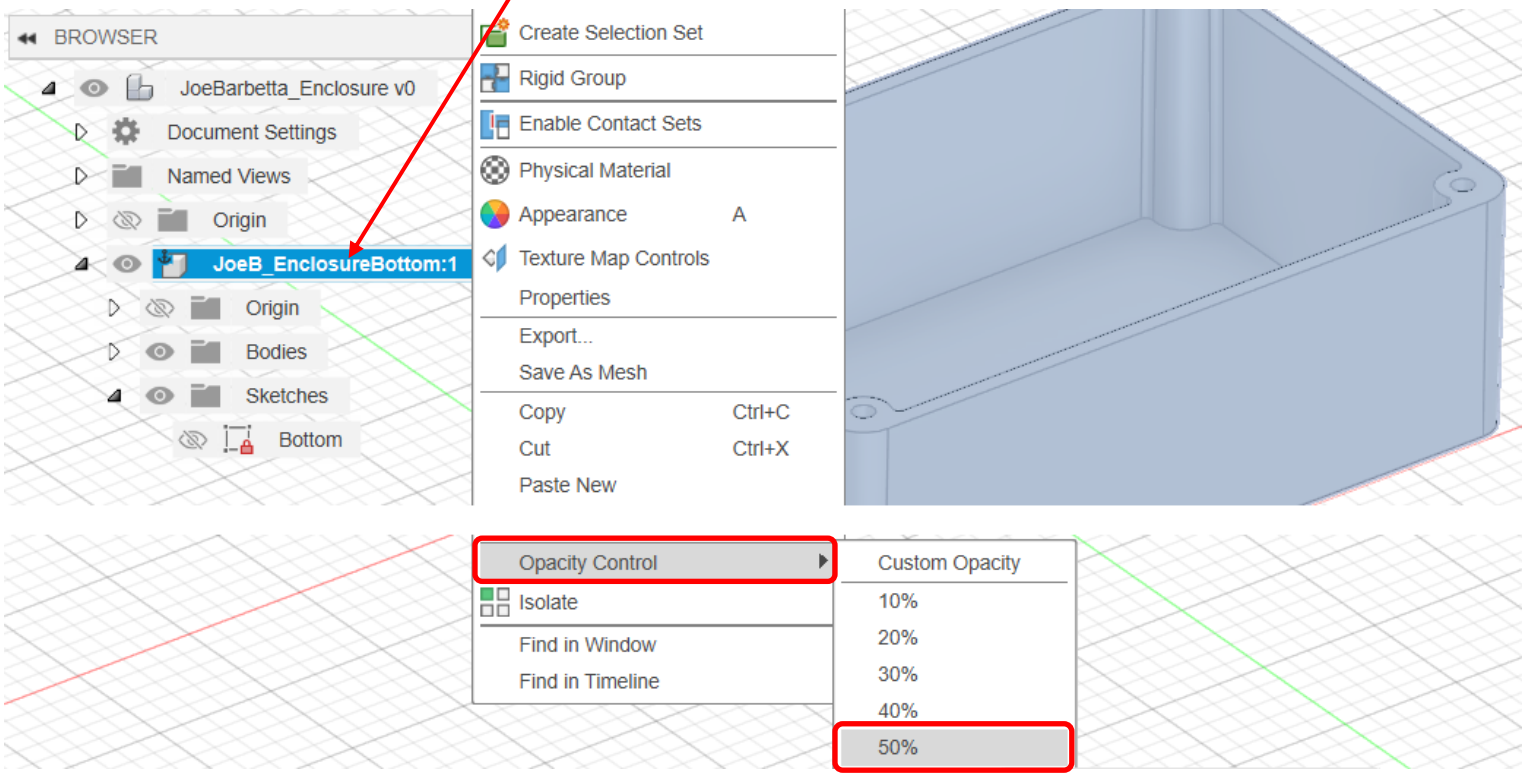
- select the **Fillet** tool
- as done previously, zoom into each interior corner to select **2 edges at each interior corner**



- enter **0.03** and click **OK**

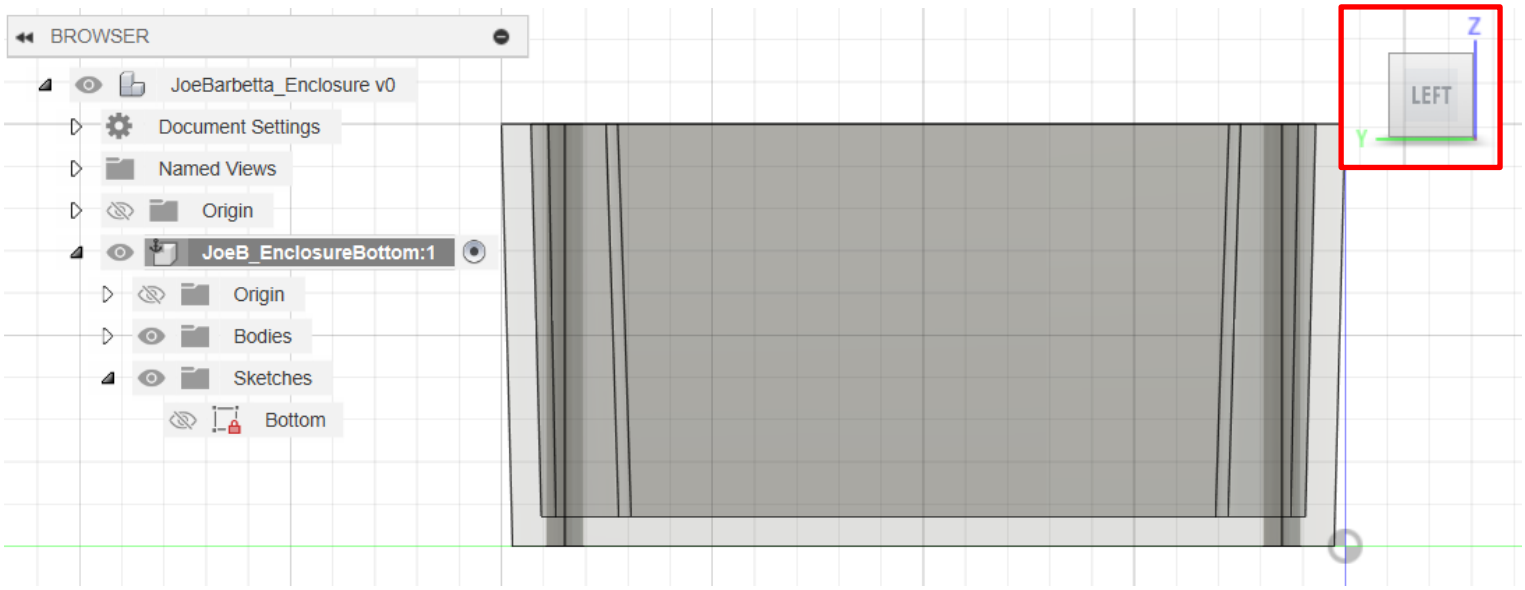


- right-click on the **EnclosureBottom Component Name** and select **Opacity Control** and **50%**



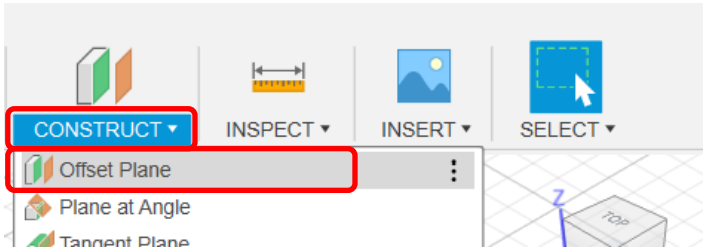
- click on the **LEFT** labeled face of the **View Cube**.

The angled walls show the draft.

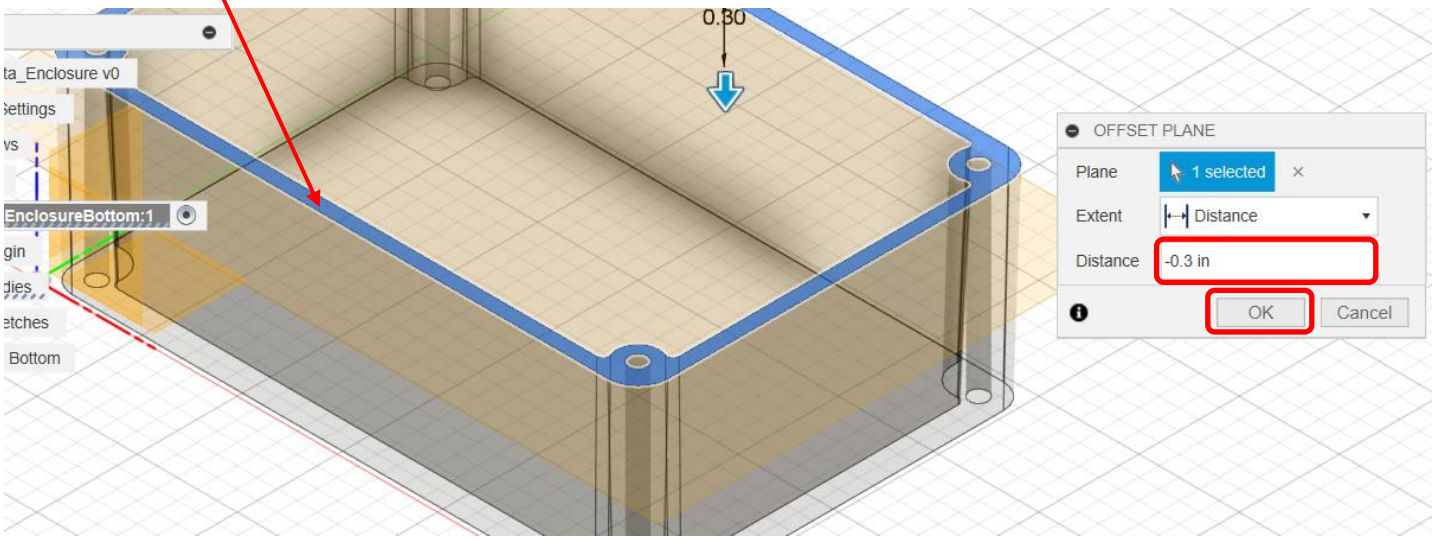


Adding Draft to Holes

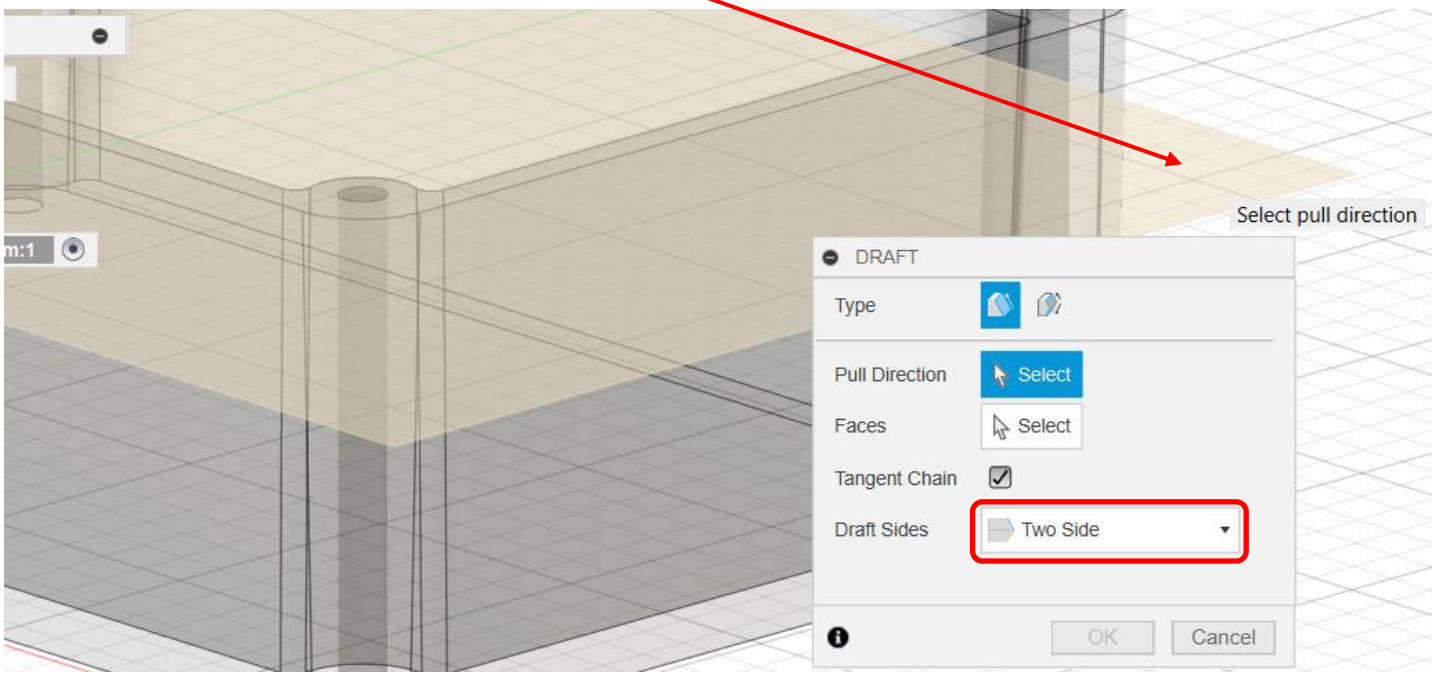
- from the **CONSTRUCT** menu select **Offset Plane**



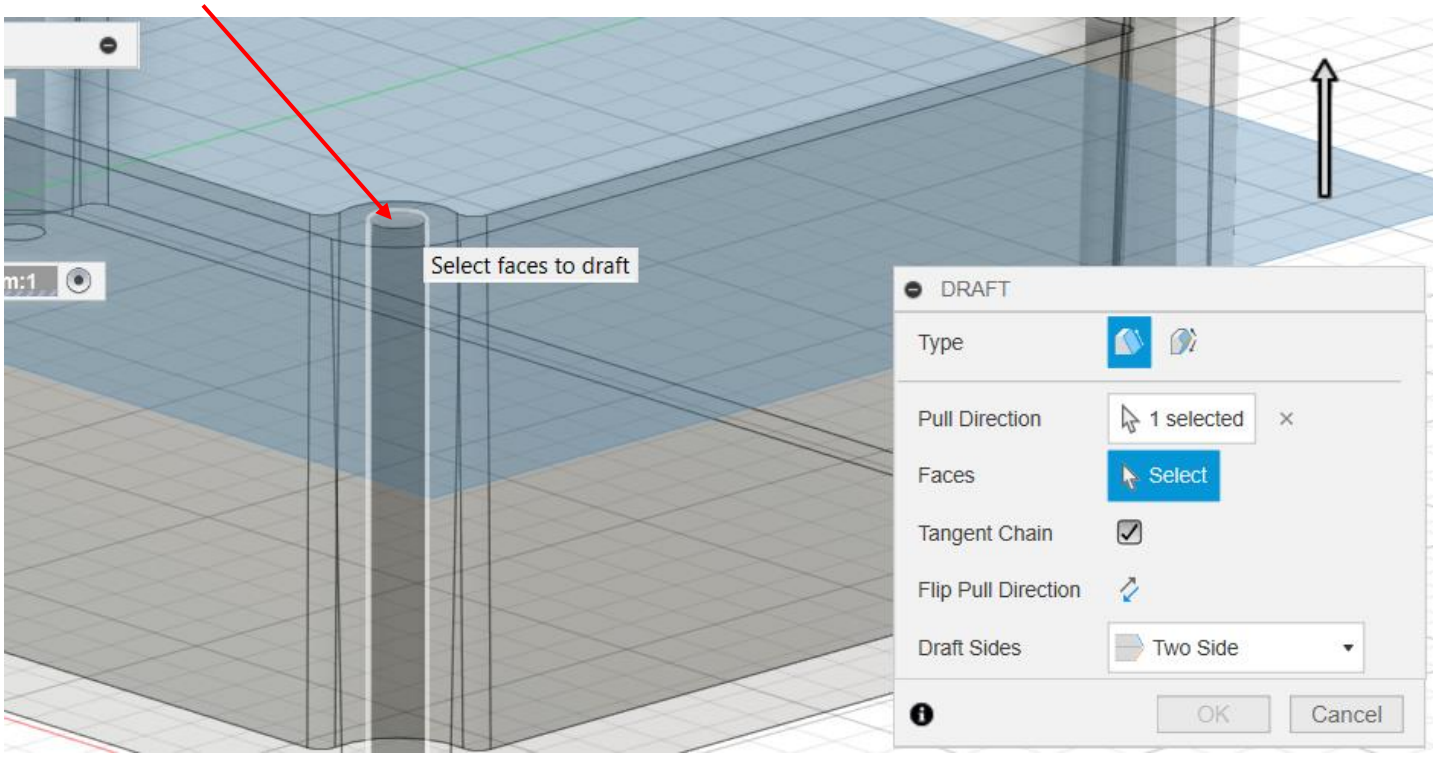
- click on the **top surface** and enter a value of **-0.3** (note the minus sign) for **Distance** and click **OK**



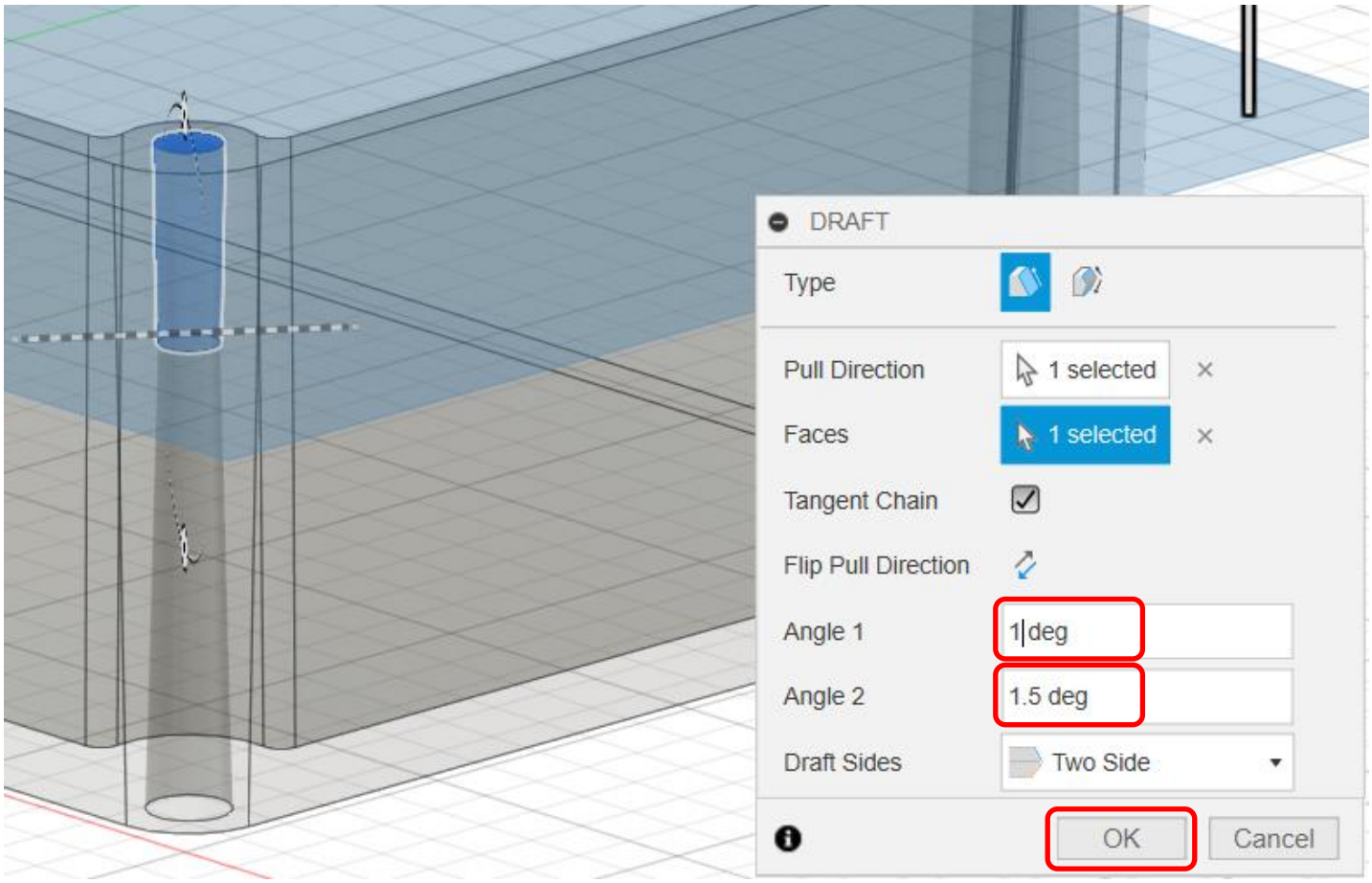
- click on the **Construction Plane** just created
- change the **Draft Sides** to **Two Sides**



- click on the **inside** of the hole



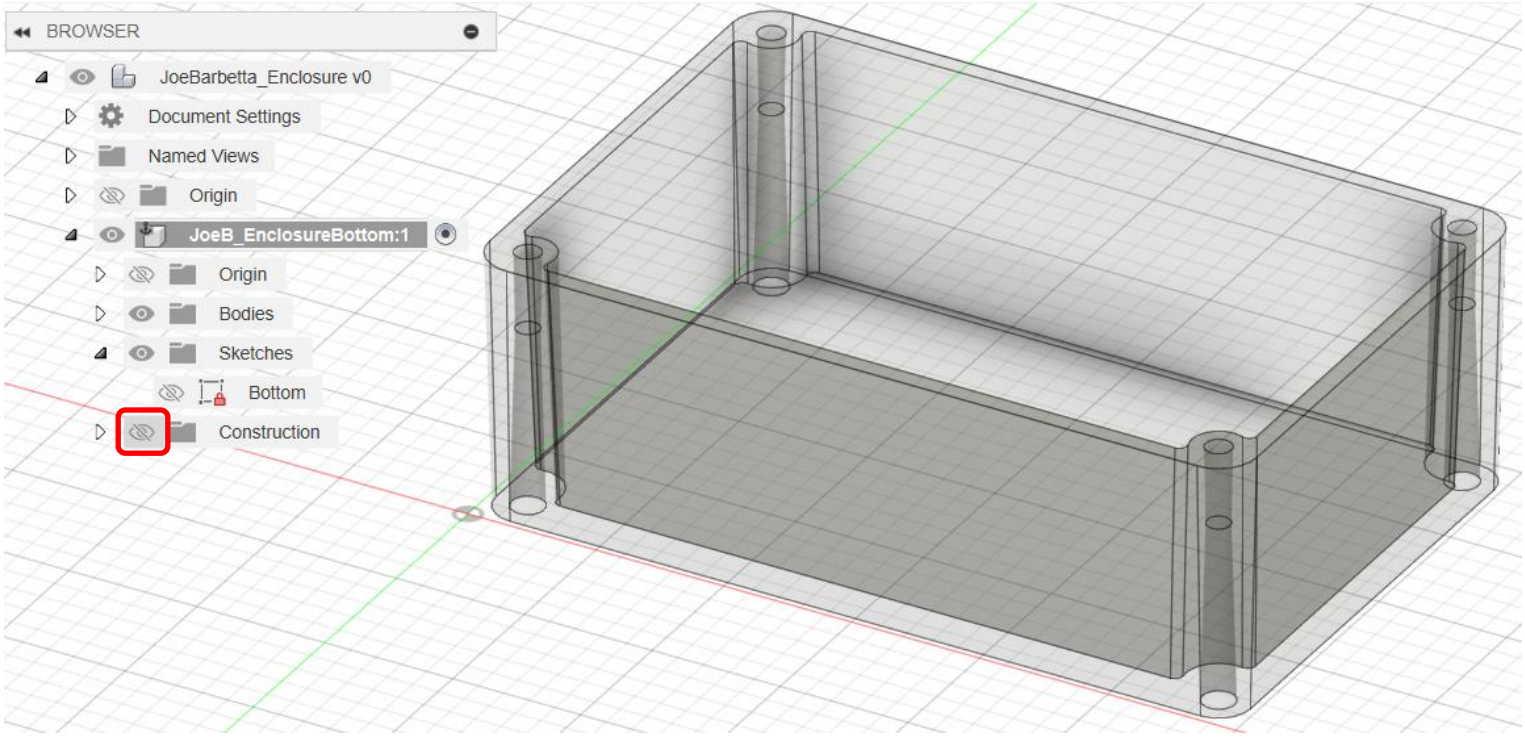
- enter a value of **1** for **Angle 1** and a value of **1.5** for **Angle** 2, and click **OK**



- perform the same operation on the **other 3 holes**. The Draft tool should remember the 2 angles each time, but verify this when drafting the other holes. You may have to reselect **Two Side** for **Draft Sides** each time.

- yell **"I am a Draft King!"**

- click the **eye** icon next to the **Construction** folder to hide the Construction plane
- right-click on the **Component name** and set **Opacity** back to **100**



Minimizing ***sink marks***

One may question the reason for the bottom section of the hole. If the hole just extended down from the top by the 0.3", there should not be a need for a hole extending all the way to the bottom.

Without the bottom hole section, there would be a large mass of plastic in the corners of the enclosure. When the plastic cools, it contracts and this would cause material to get pulled in around that section. ***Sink marks*** would appear on the exterior walls at the corners.

These sink marks impact the aesthetics of the part, but if extreme can result in undercuts in both exterior and interior walls, making part ejection difficult.

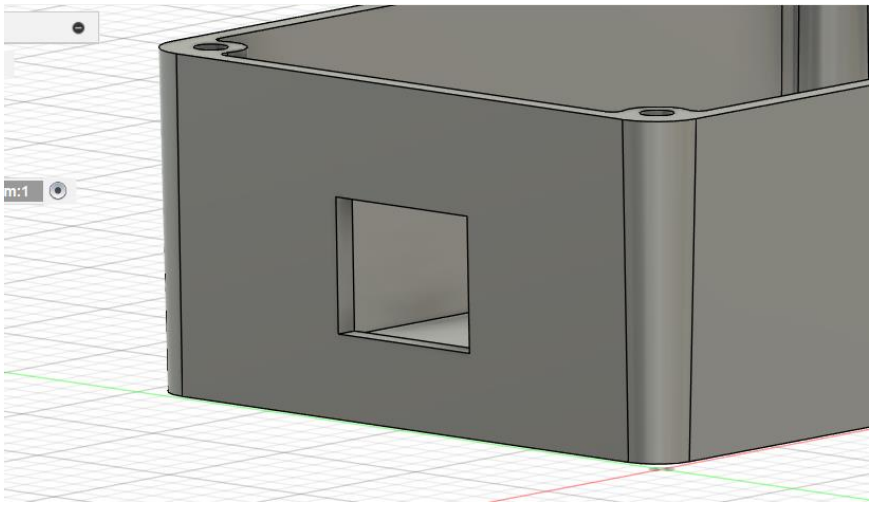
mitigation technique used in the assigment could be optimized further, however, for simplicity it is being limited to the bottom tapered hole.

Adding Side Holes

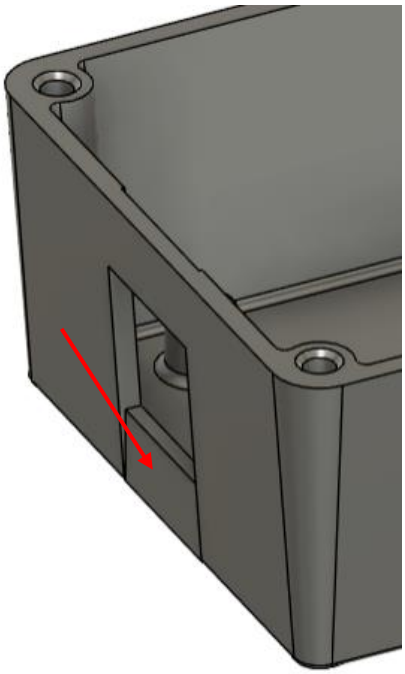
A common requirement for an enclosure are holes for connectors, for example a 0.500 x 0.500 in hole as shown below.

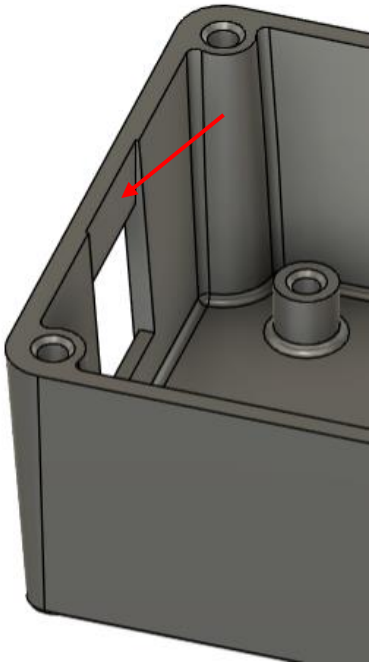
A hole, such as this seems simple, but the hole in a side wall is considered an ***undercut*** and it would require a ***side action*** in the mold. This is a movable part in the mold, which is positioned in the hole location when plastic is injected into the mold, but must then slide out of the hole before the part is ejected. It this part, a core, isn't moved it will prevent the part from being ejected. It will be stuck in the mold.

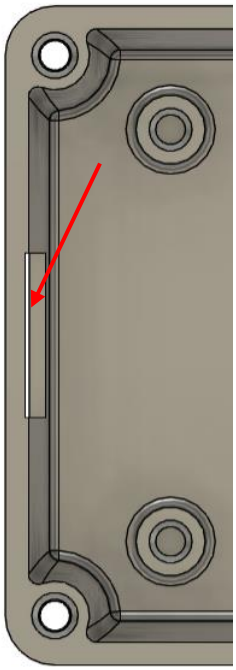
Adding side actions to a mold increase the complexity and cost of the mold. It is common practice to modify a design to allow the use of a ***straight-pull mold***. This is a mold with **only 2 halves** and no movable side actions.



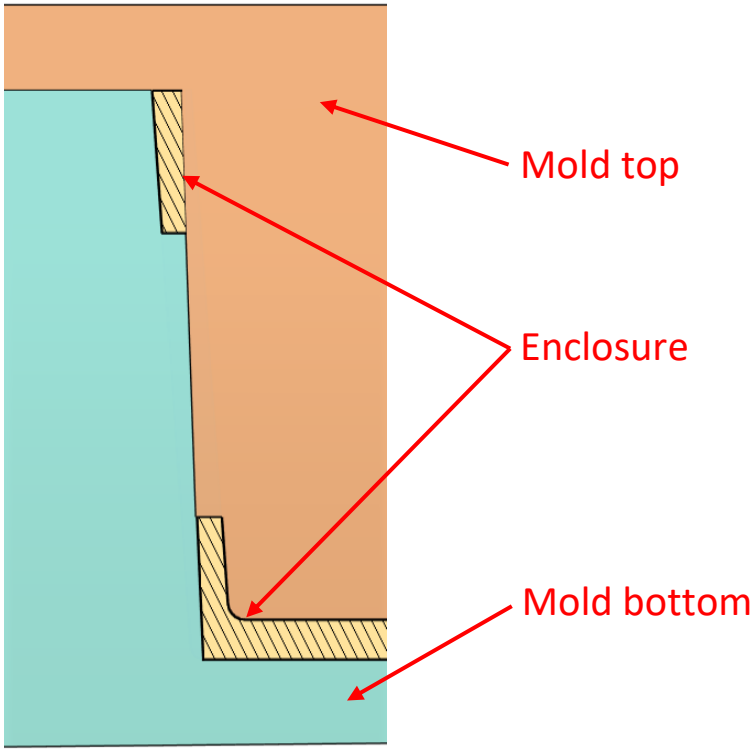
One method to avoid the need for side actions is to apply a **larger draft** on walls where there will be a side hole. The below pictures show a **4 degree draft**. One then removes material below the hole on the outside wall and also remove material above the hole on the interior wall. The 3rd picture is a top view that shows that material above the hole does not cover material below the hole. A thin space is also visible to help show that this is true.

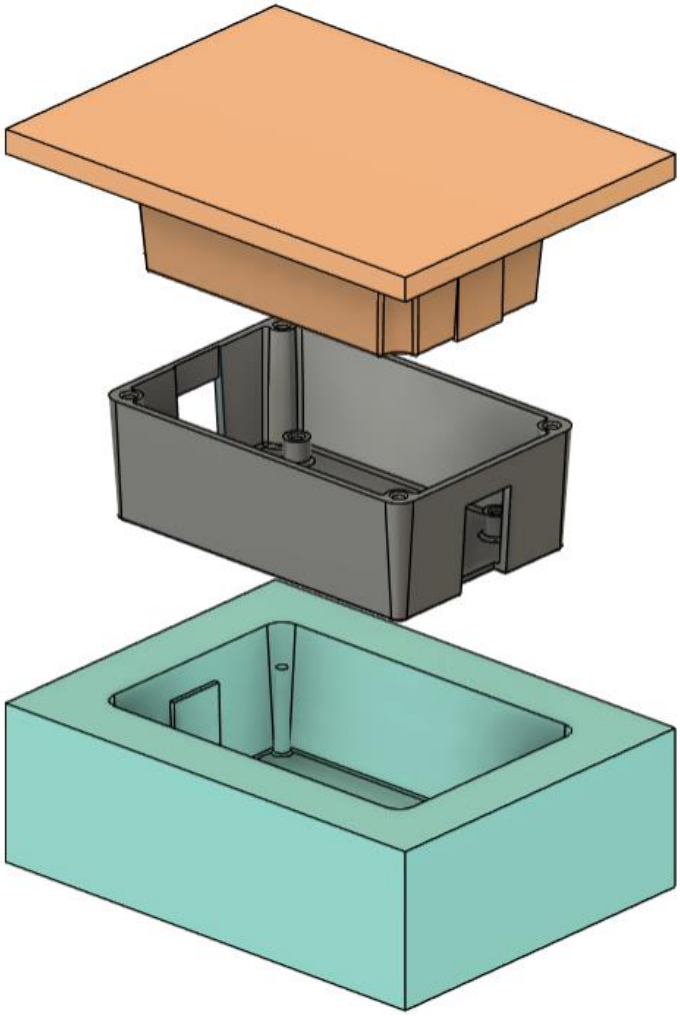




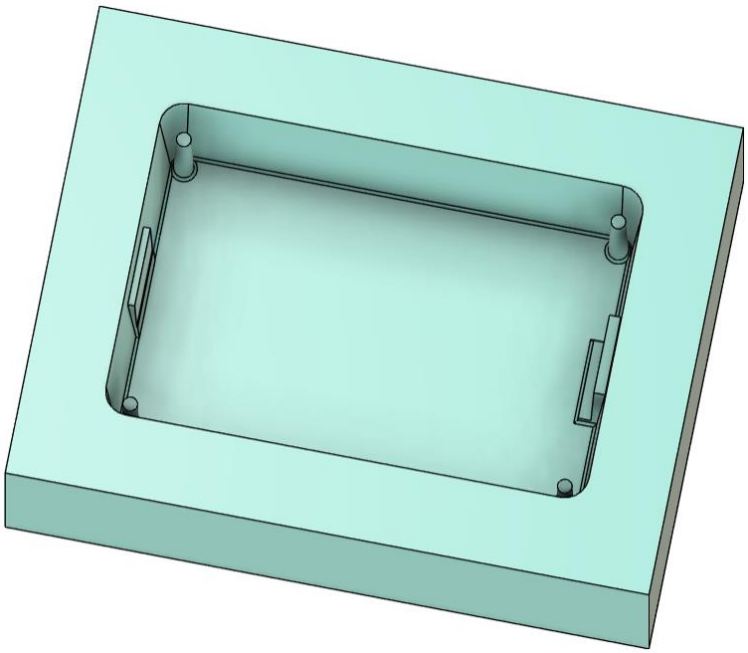


This is a cross-section of the hole and mold.

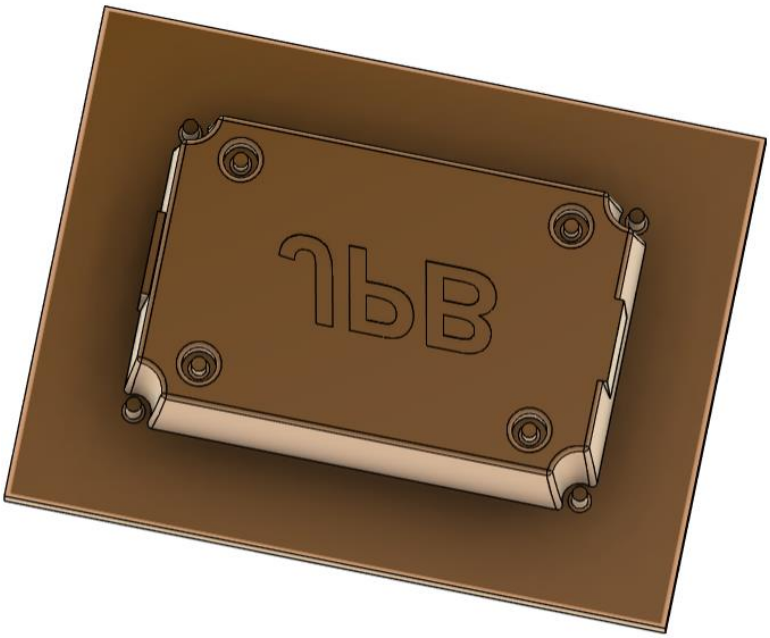


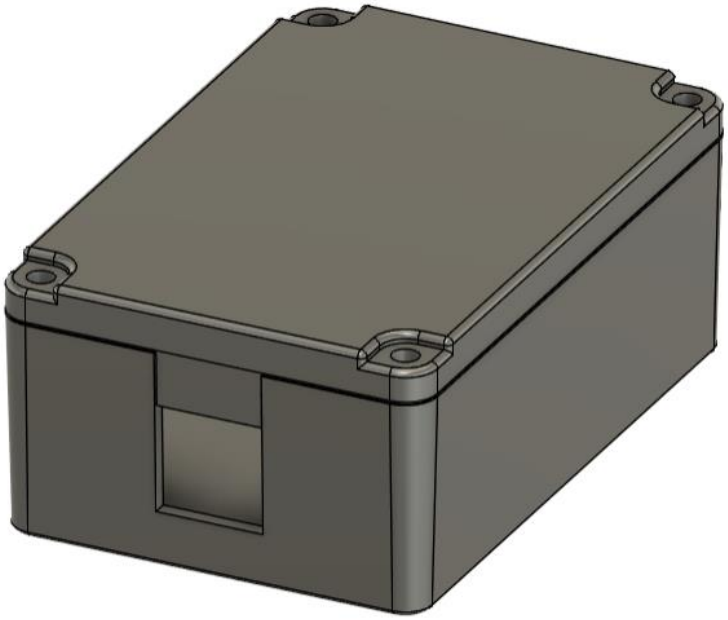


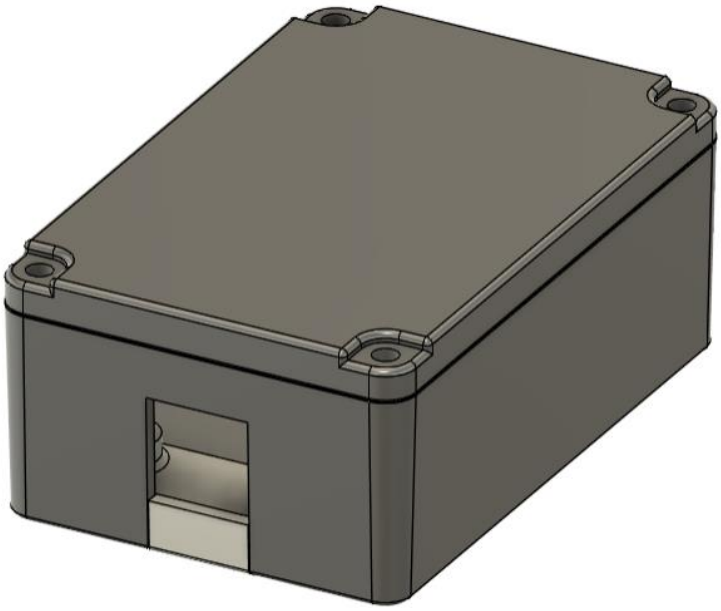
Mold bottom



Mold top



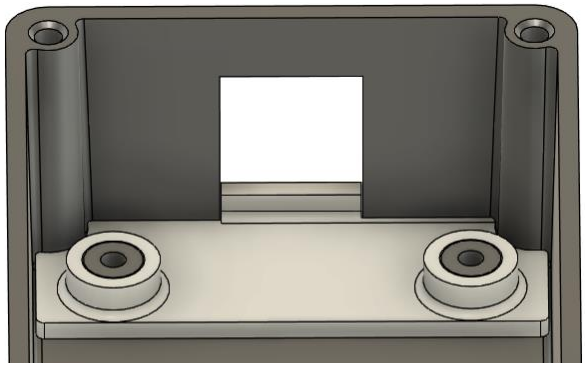


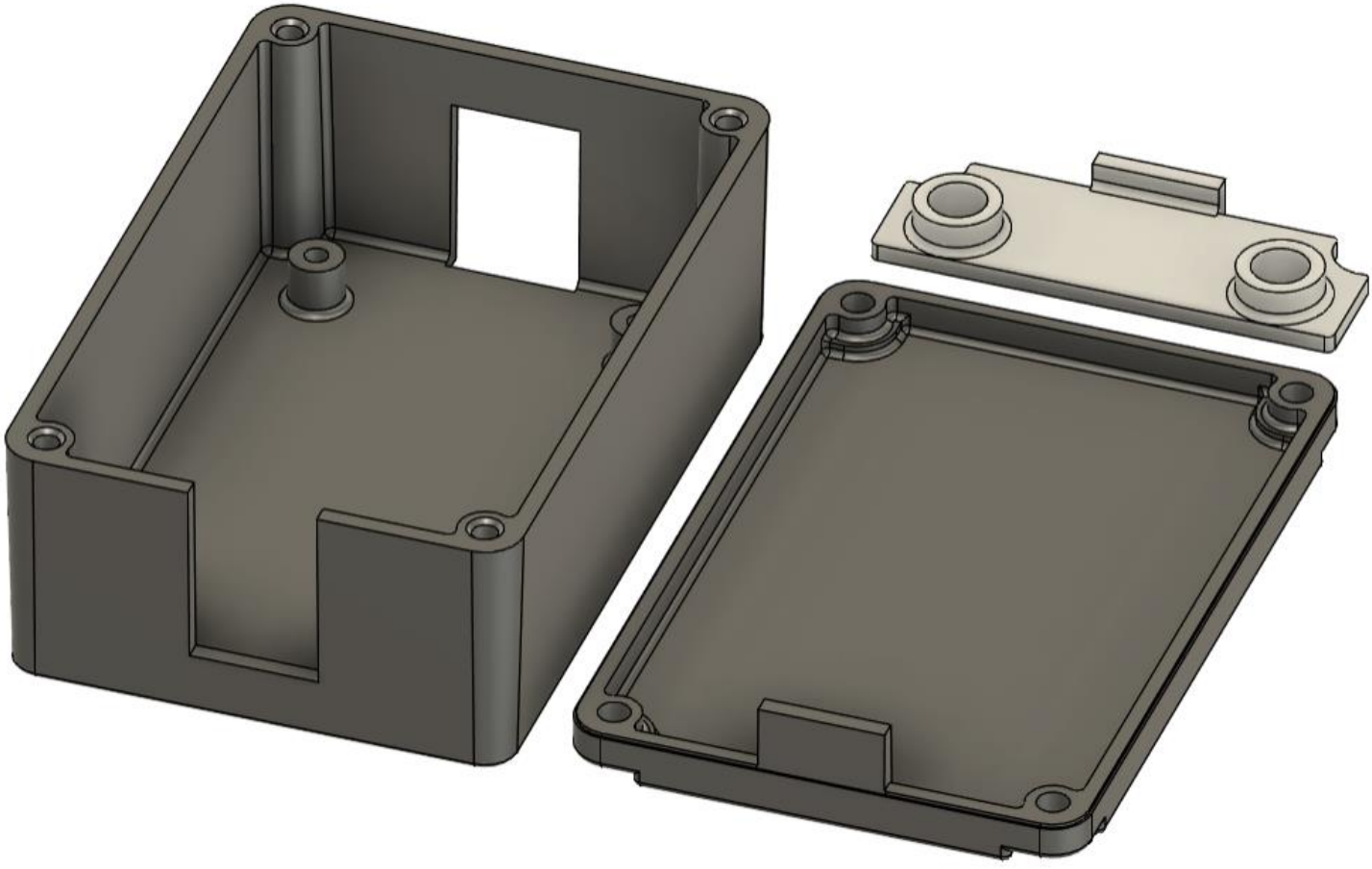


The above pictures shows the enclosure using 2 of the methods introduced.

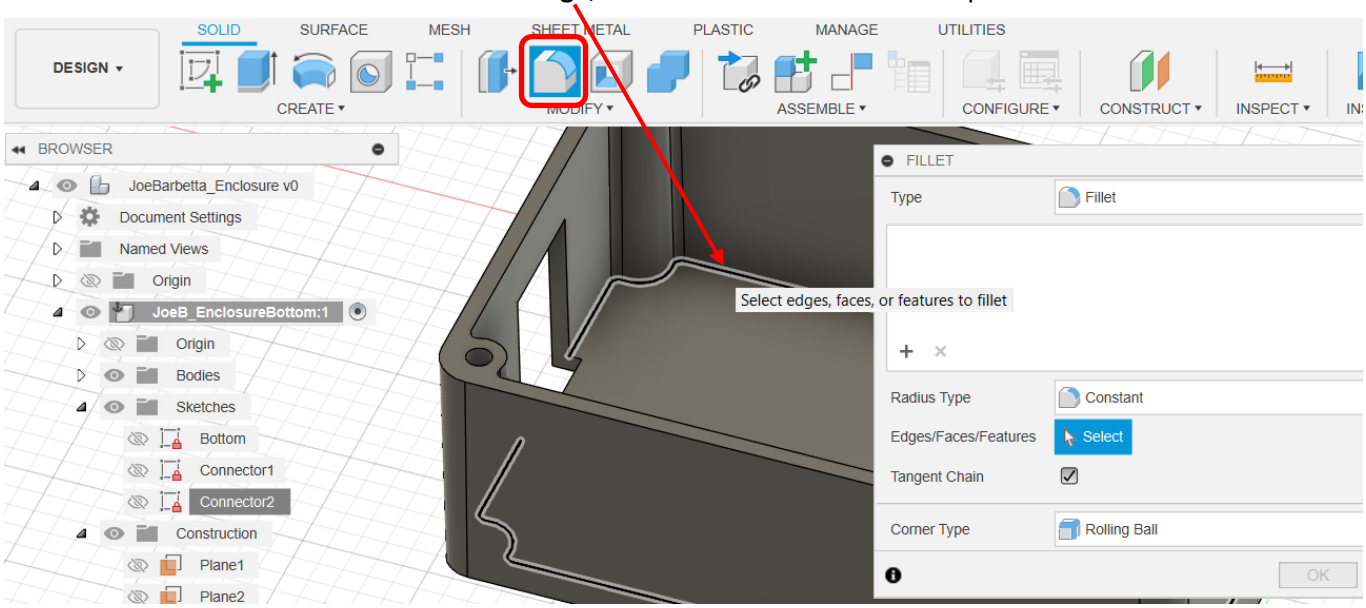
The left picture above, uses a protrusion from the top cover to fill the top of the connector hole.

The right picture above using an extra insert part, shown on the right. It is common to have a ***family mold***, which will produce both the bottom shell and top cover. This insert would just be another cavity in the mold.

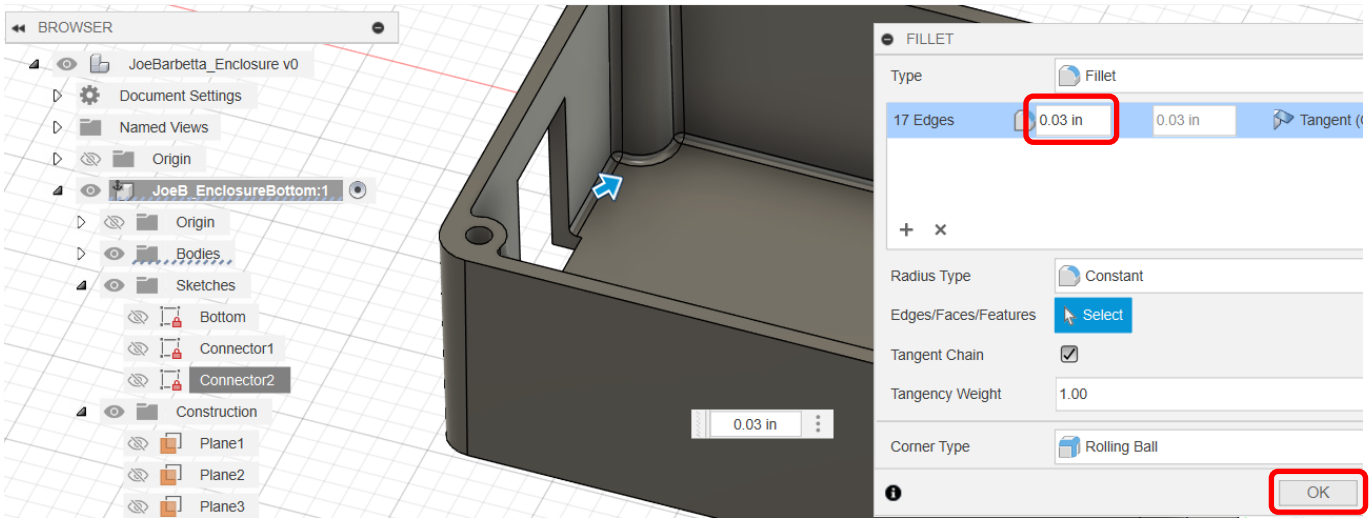




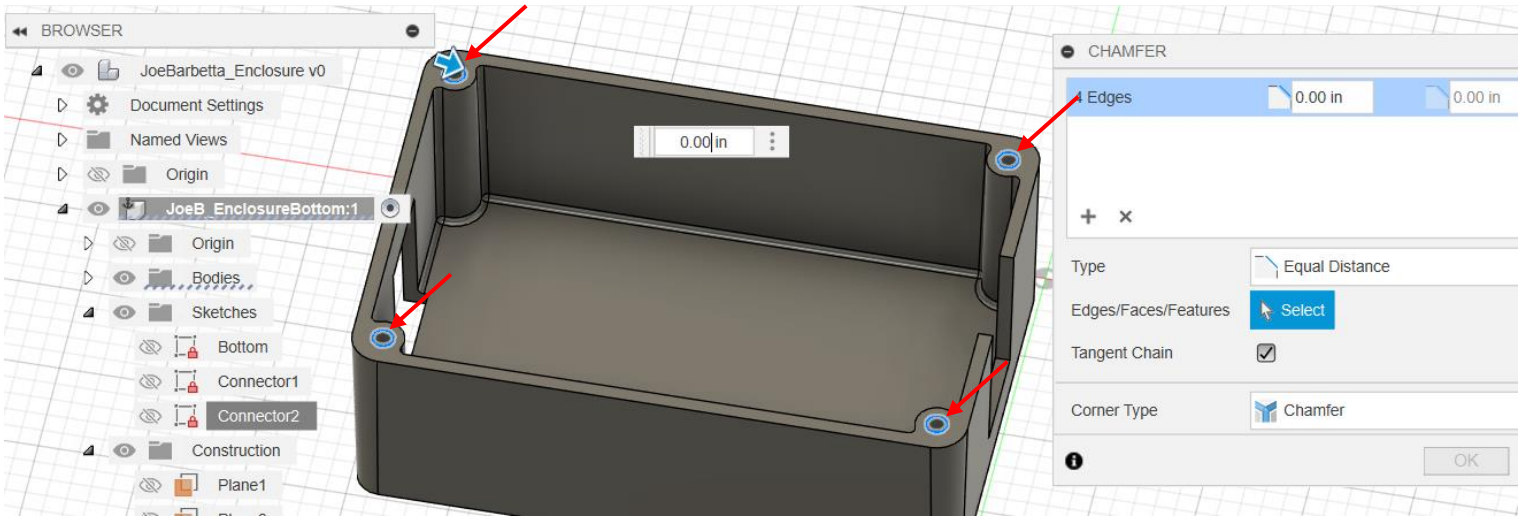
- select the **Fillet** tool and click on a **bottom edge**, which should select the entire perimeter



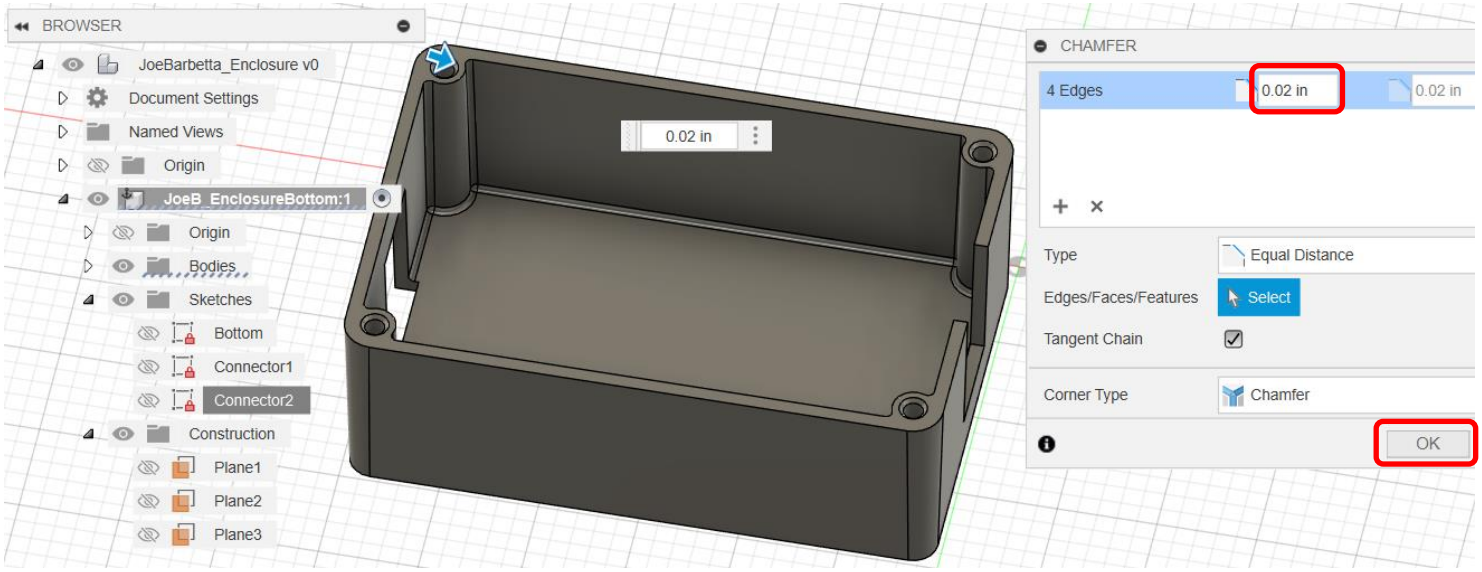
- enter a value of **0.03** and click **OK**



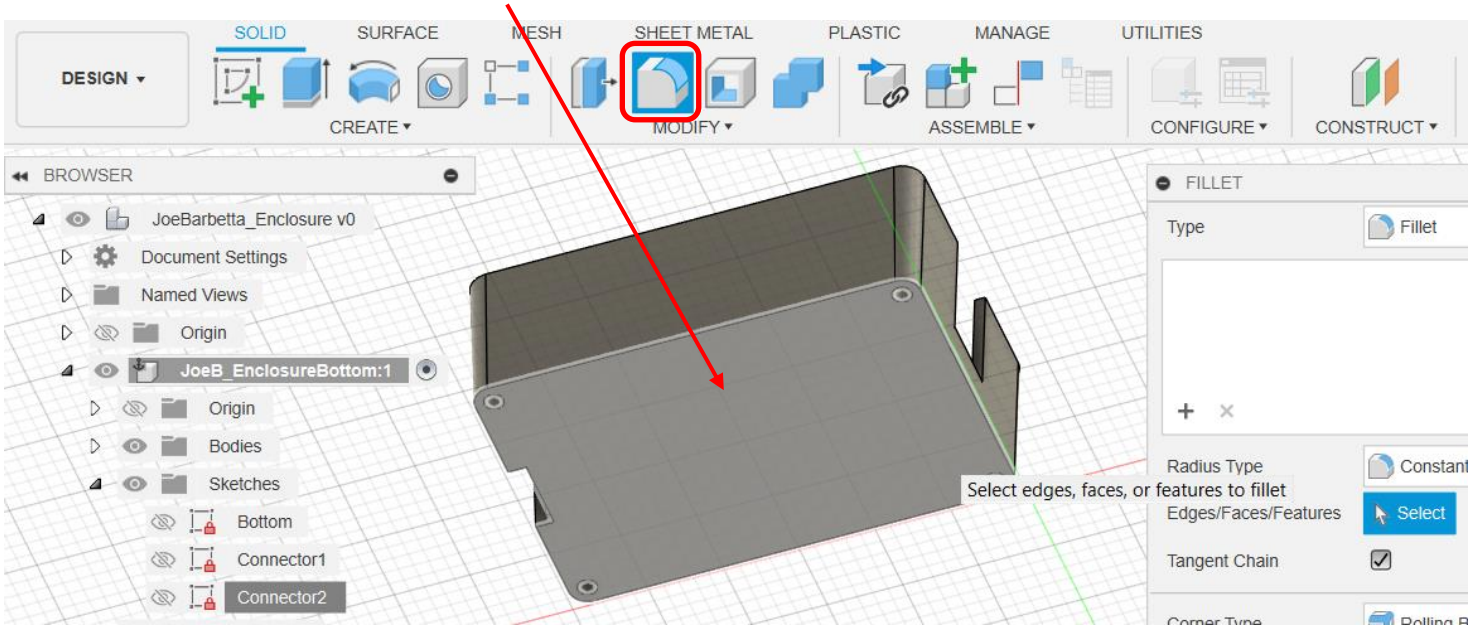
- from the **MODIFY** menu, select the **Chamfer** tool
- click on the **edge of each of the 4 holes**



- enter a value of **0.02** and click **OK**

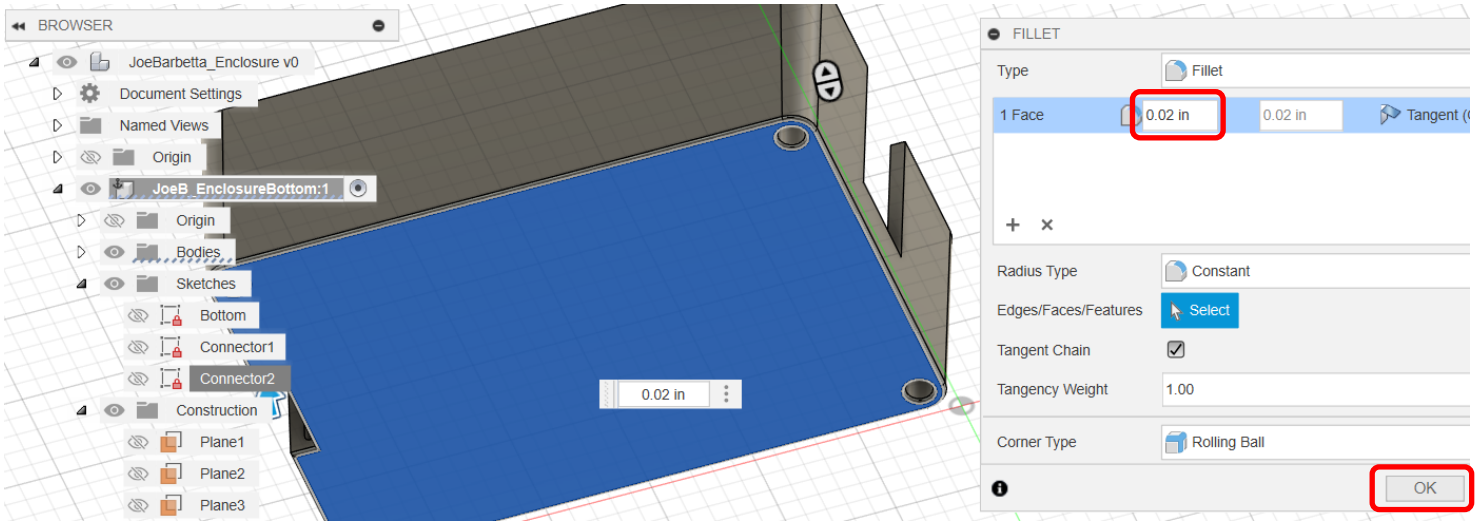


- select the **Fillet** tool and click on the **bottom of the enclosure**



- enter a value of **0.02** and click **OK**

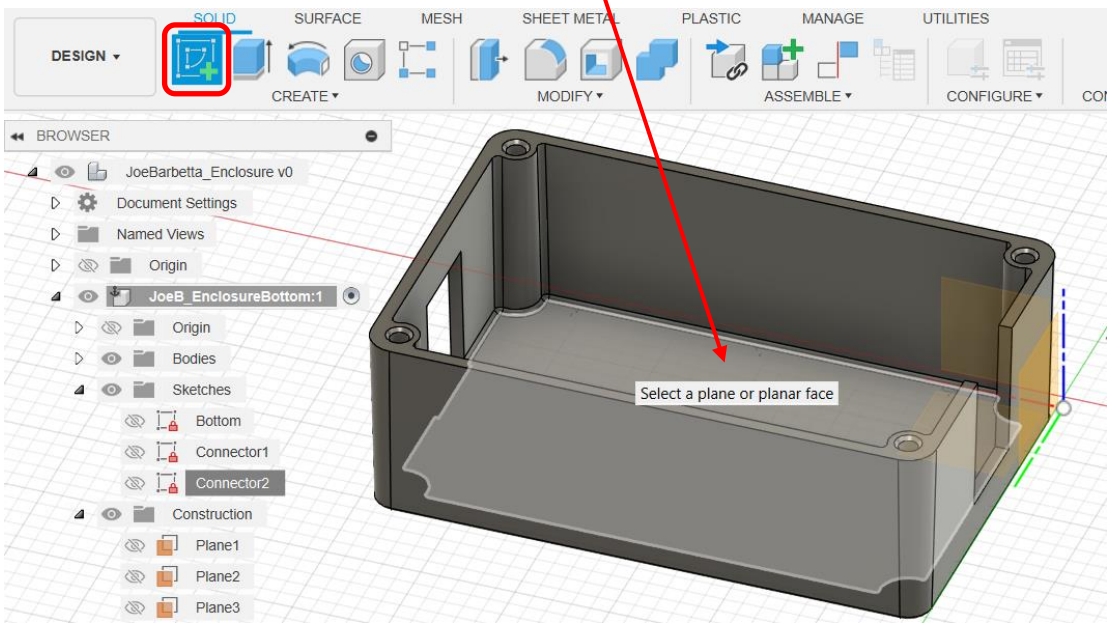
Note that **when a surface is selected**, **all of the features** on the surface are filleted.



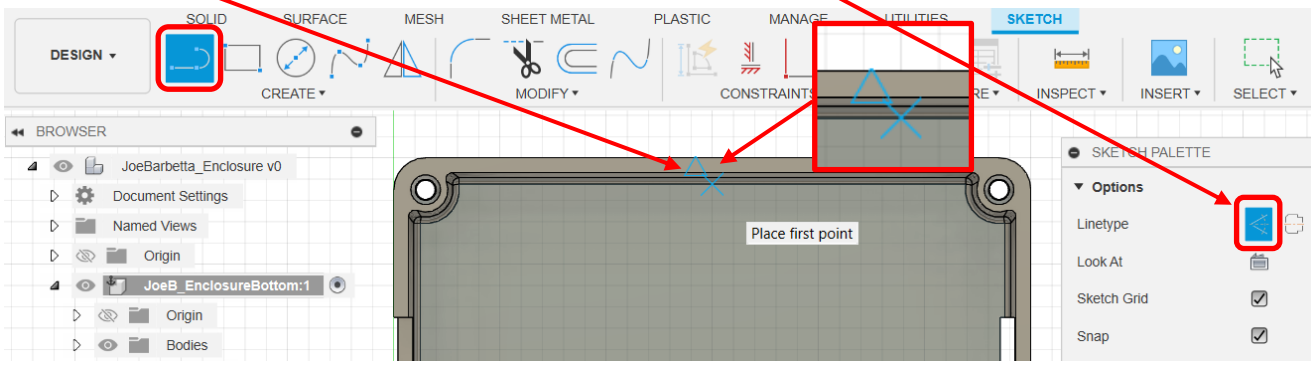
Adding Bosses

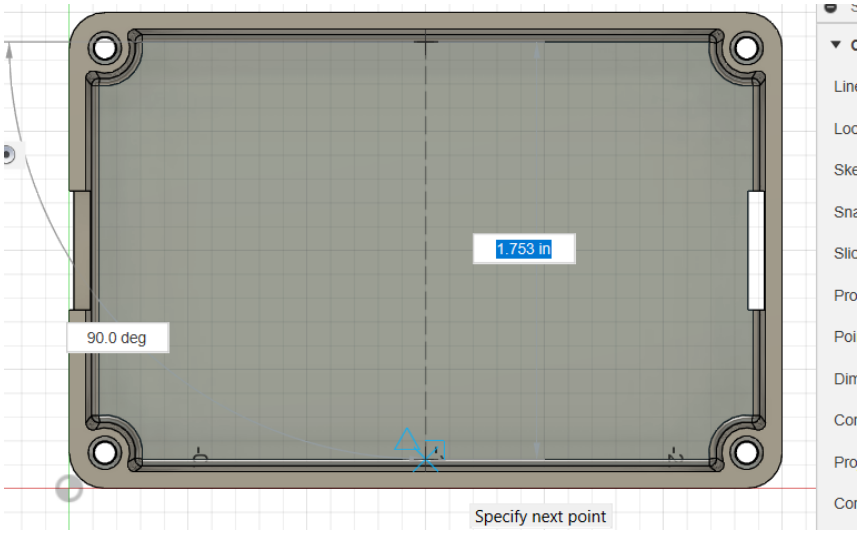
Bosses are protrusions on the interior bottom of an enclosure to support internal comonents, such as a circuit board.

- select **Create Sketch** and click on the **interior bottom surface**



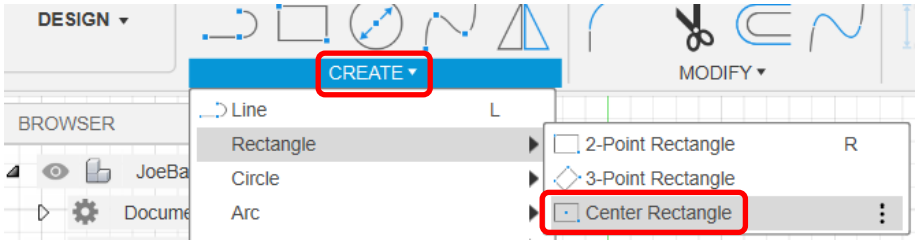
- click on the **Construction line icon** to highlight it blue
- select the **Line** tool and move the mouse over the center of the edge shown
- when a **blue triangle** appears, which indicates the center of the edge, **click on that point**



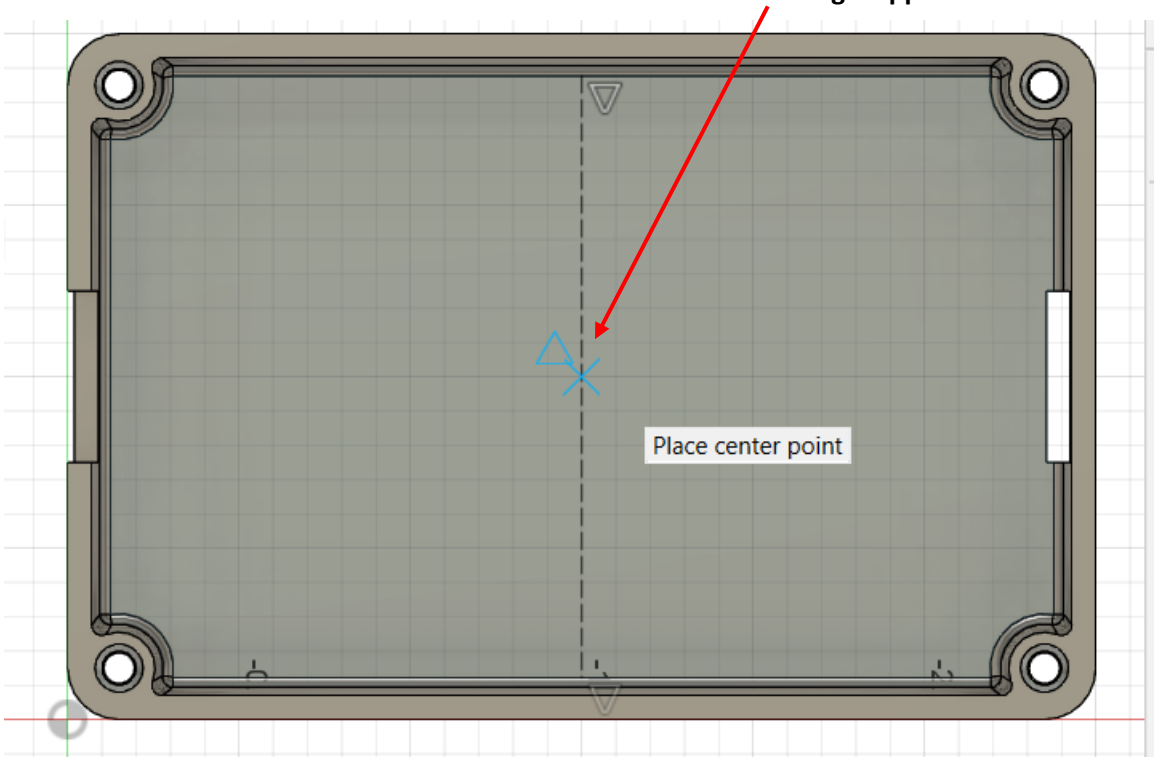


- **extend the line downward** to the opposite edge
- when a **blue x and triangle** appear, click on that point

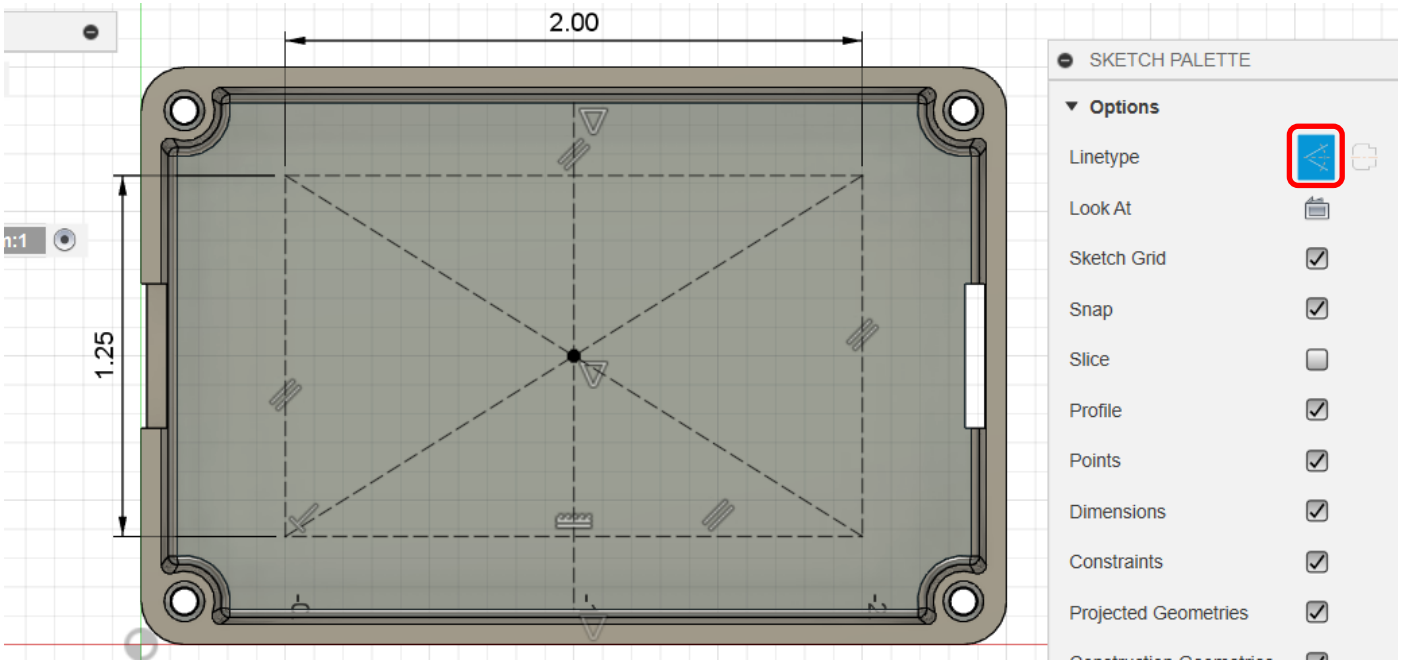
- from the **CREATE** menu, select **Center Rectangle**



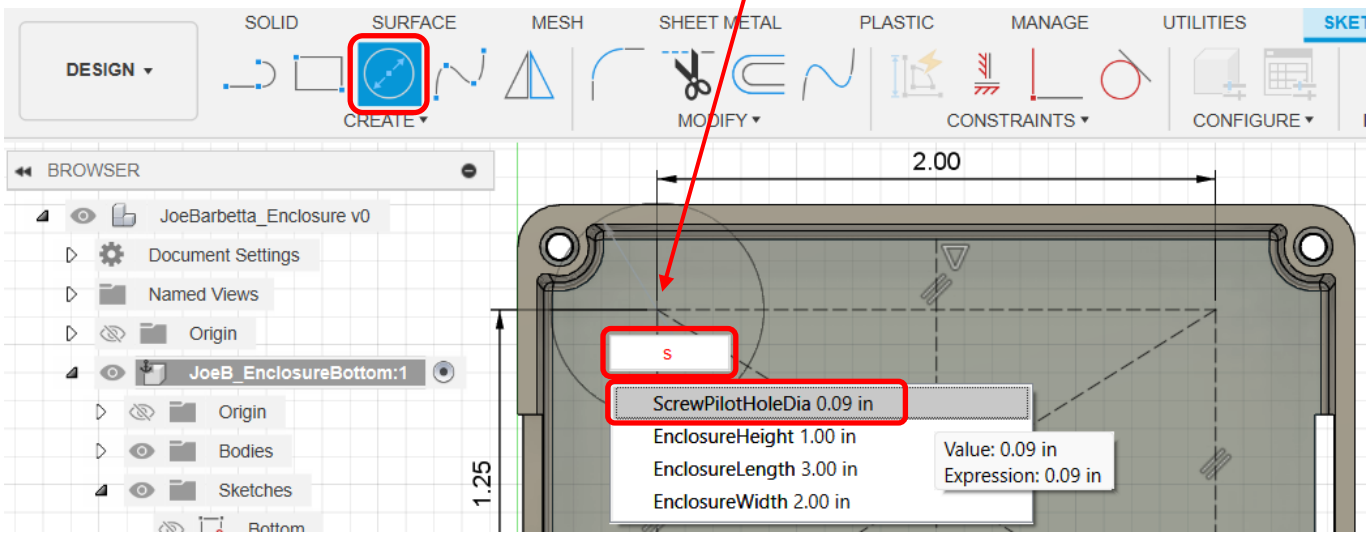
- move the mouse over the **dashed line** and **click when the blue triangle appears**



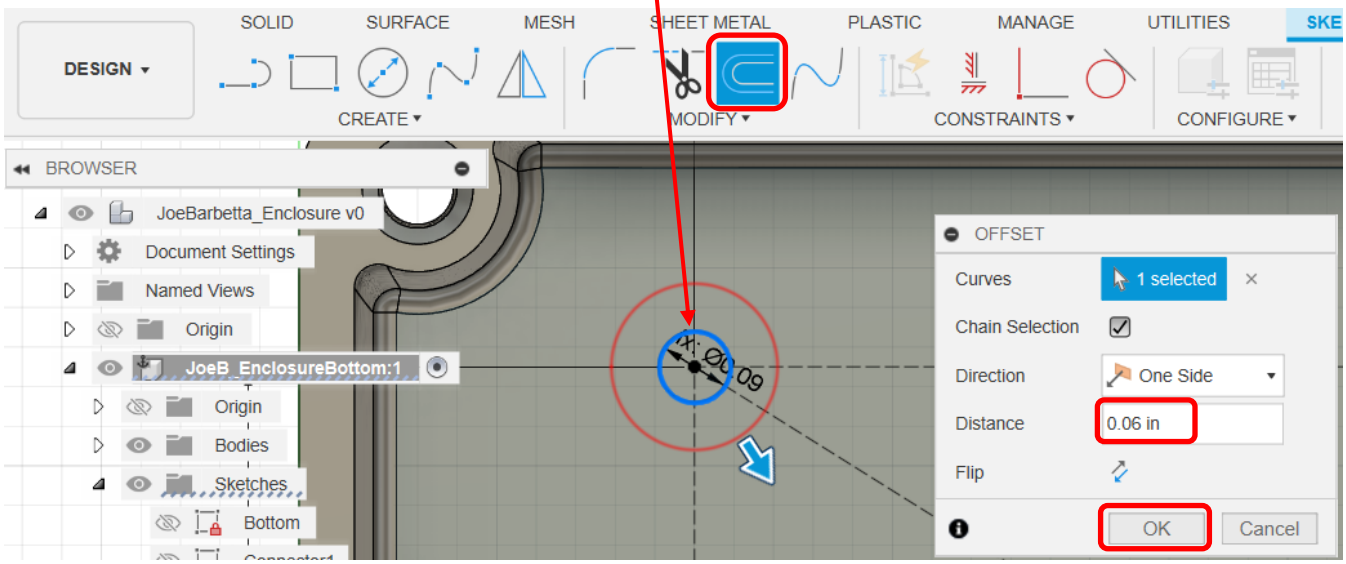
- extend the **rectangle outward** and enter values of **1.25** and **2.00** as shown
- click on the **Construction** line icon again to **remove the blue highlighting**

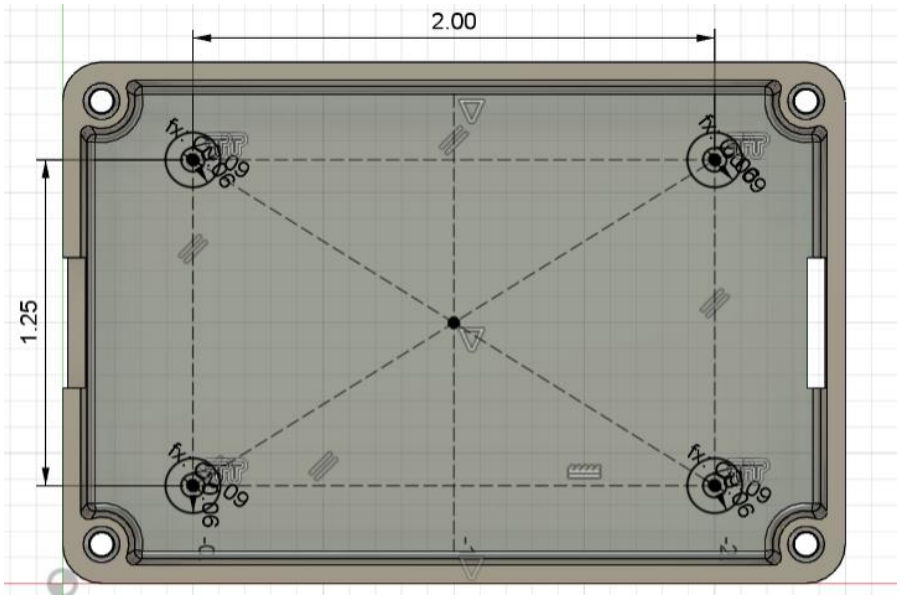


- select the **Center Diameter Circle** tool and click on the **top left corner** of the rectangle just created
- **extend the circle outward**, type **s**, select **ScrewPilotHoleDia**, and press the **Enter key**



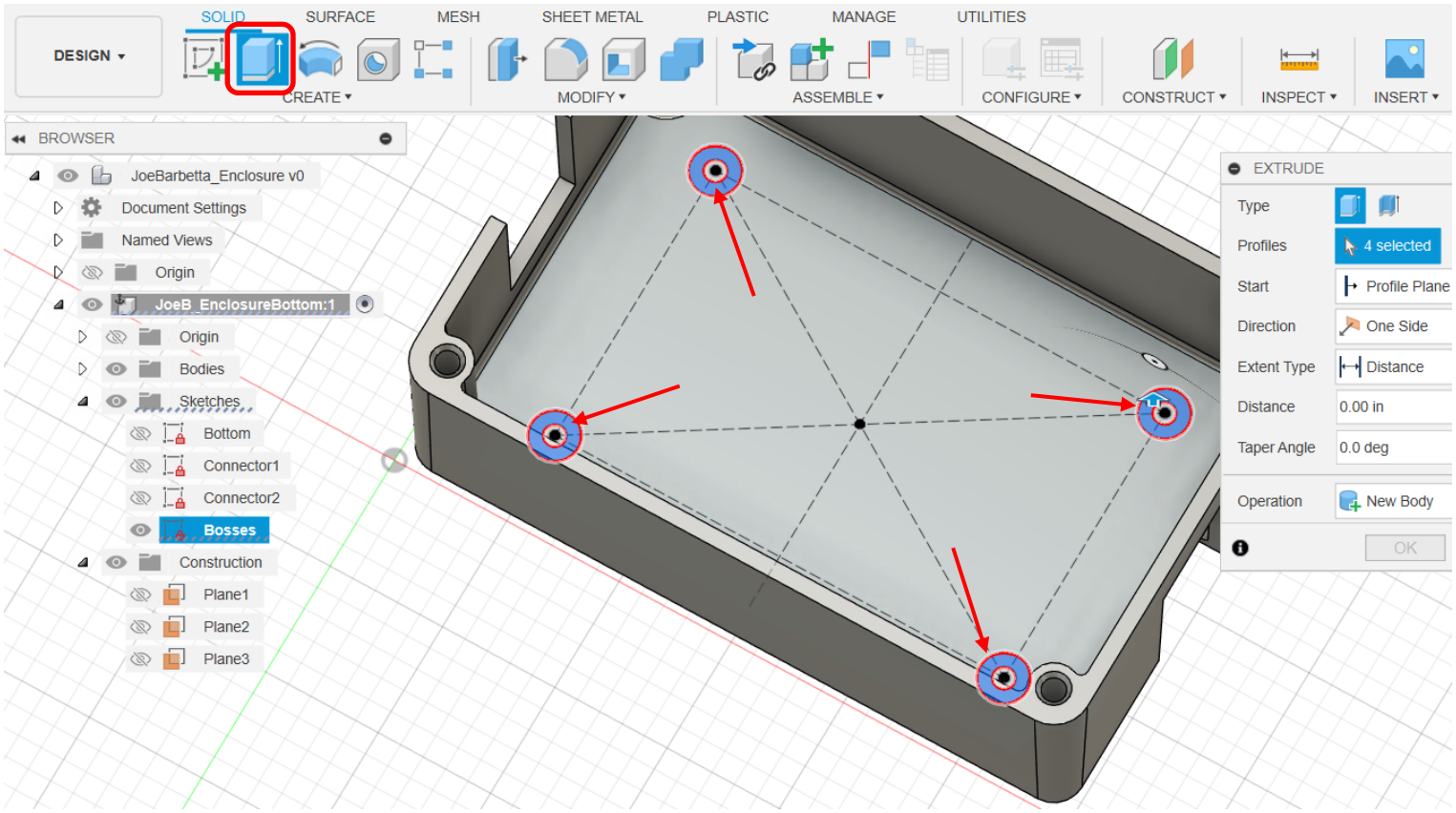
- select the **Offset** tool and click on the **circle just created**
- **extend the circle outward**, enter a **Distance** of **0.06**, and click **OK**



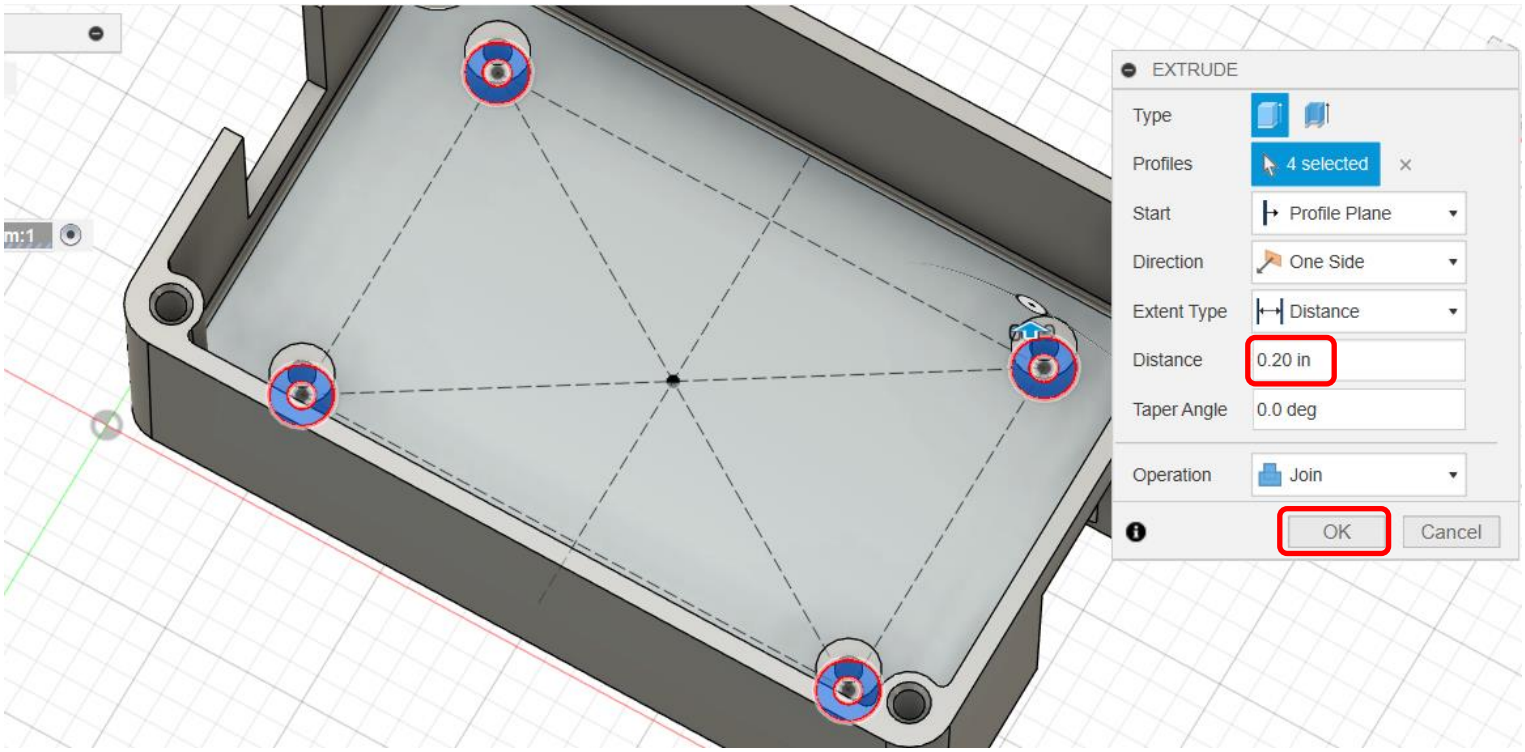


- perform the **same circle and offset operations** at the **other 3 corners**
- rename the Sketch to **Bosses**
- click **Finish Sketch**

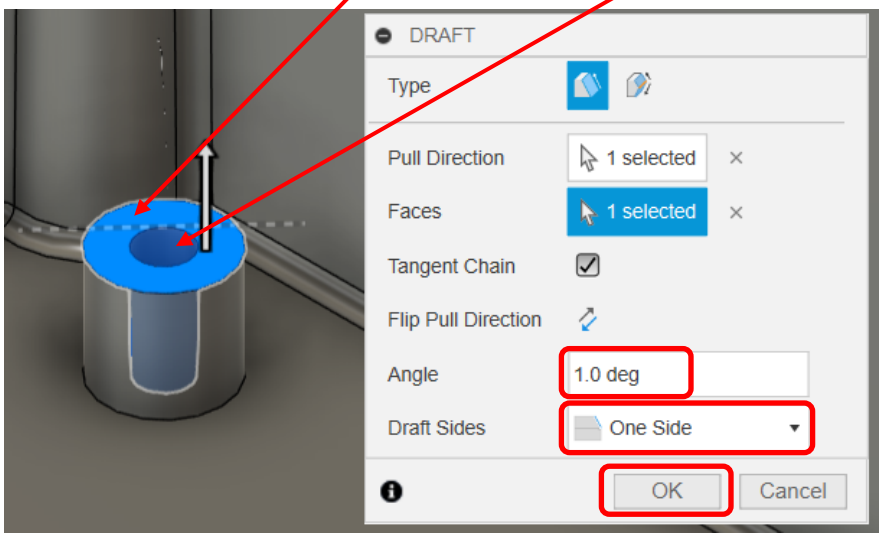
- select the **Extrude tool** and click on the **region between the two circles at each corner**. Ensure that the **center circles do not become selected**.



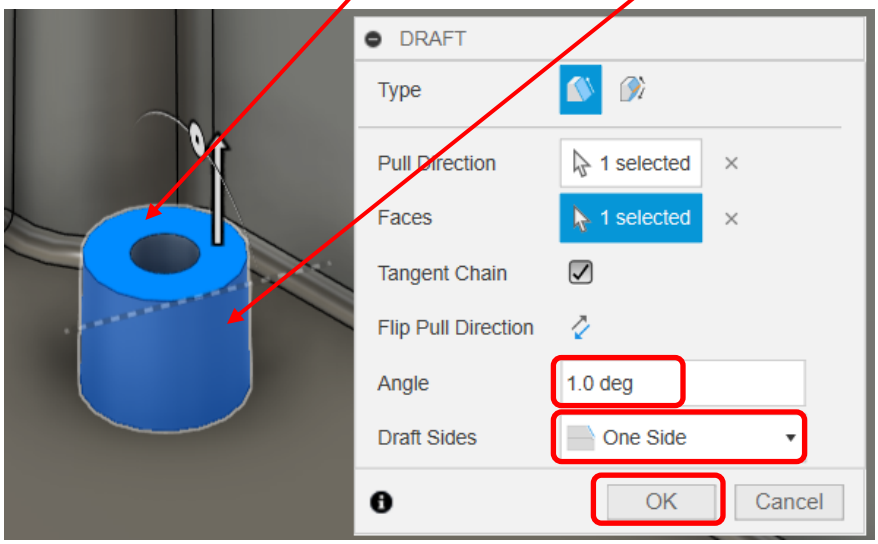
- enter **0.20** for Distance and click **OK**



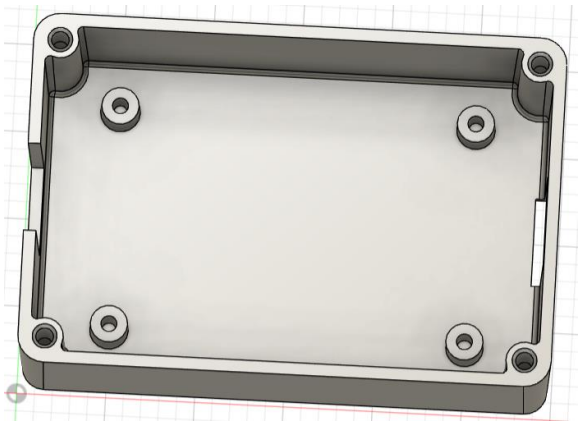
- zoom into one of the bosses
- from the **MODIFY** menu, select **Draft**
- click on the **top surface of the boss** and then on the **interior of the hole**
- ensure that **Draft Sides** is set to **One Side**, enter a value of **1.0** for **Angle**, and click **OK**



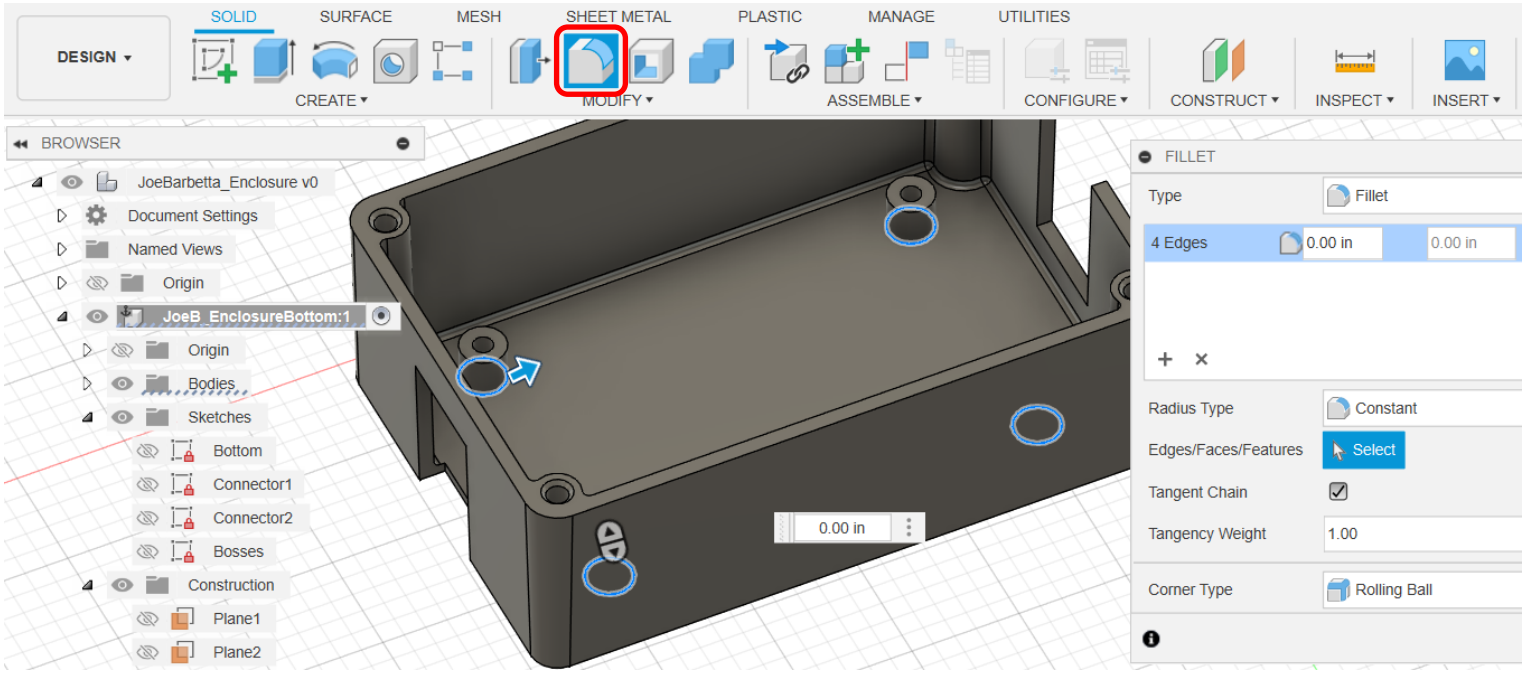
- select the **Draft** tool again
- click on the **top surface of the boss** and then on the **Exterior of the hole**
- ensure that **Draft Sides** is set to **One Side**, enter a value of **1.0** for **Angle**, and click **OK**



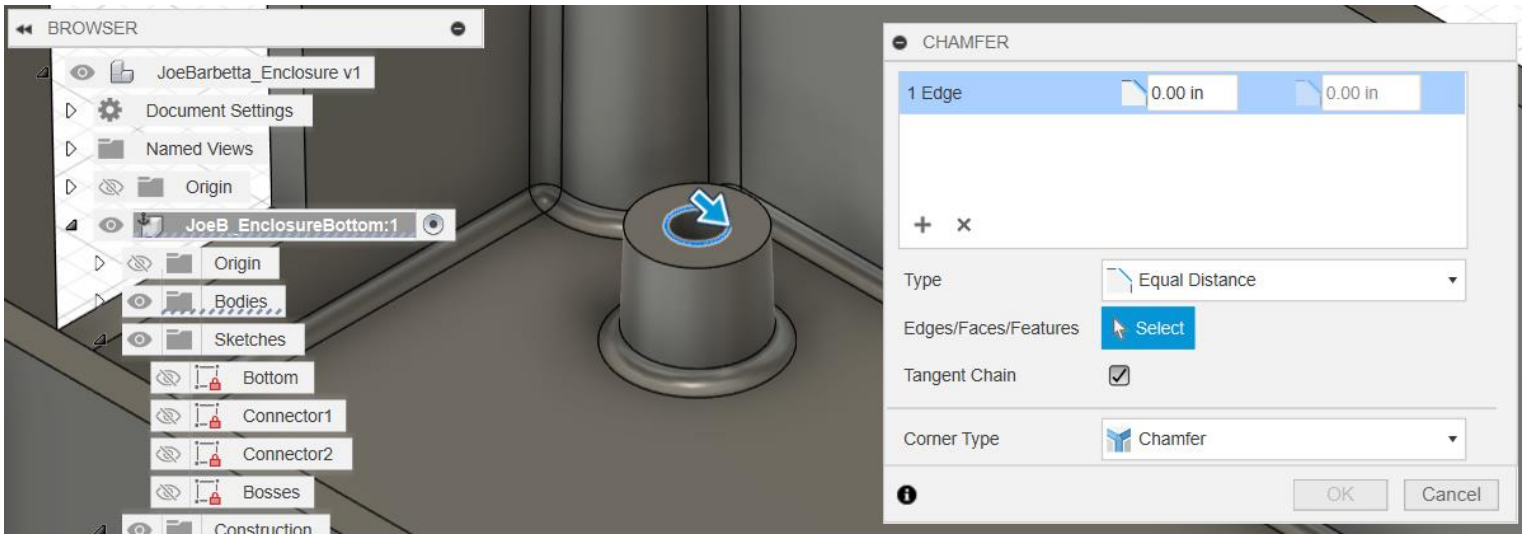
- perform the above **2 Draft operations** on the **other 3 bosses**



- select the **Fillet** tool
- click on the **bottom edge of each boss**. The view will need to be rotated to access each boss.
- enter a value of **0.03** and click **OK**

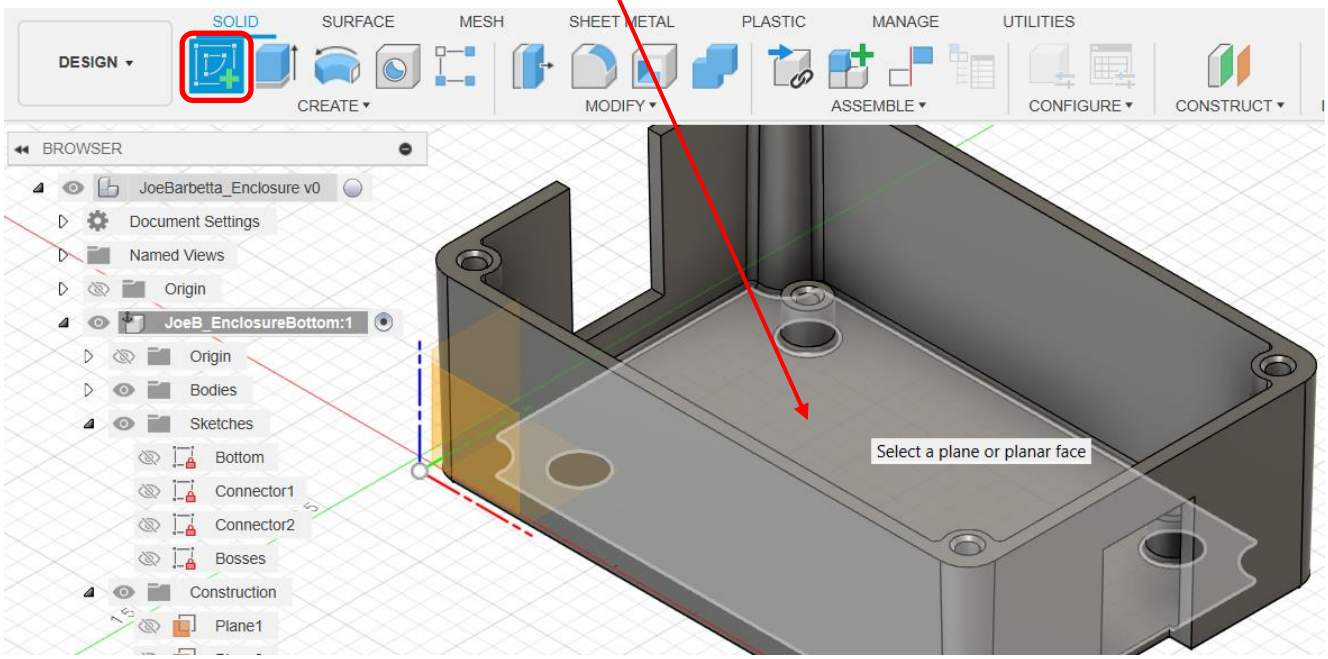


- from the **MODIFY** menu, select the **Chamfer** tool
- click on the **hole edge of each boss**, enter a value of **0.02**, and click **OK**

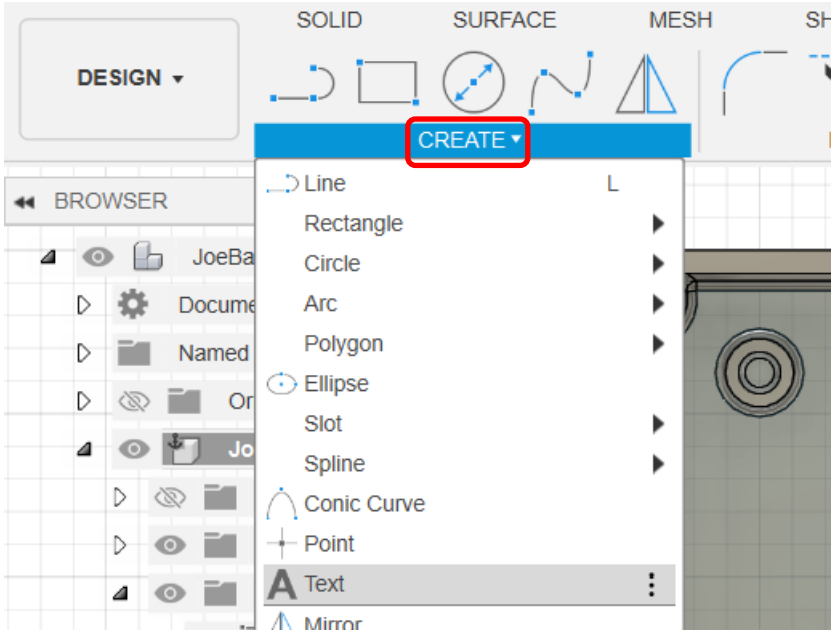


Adding Text

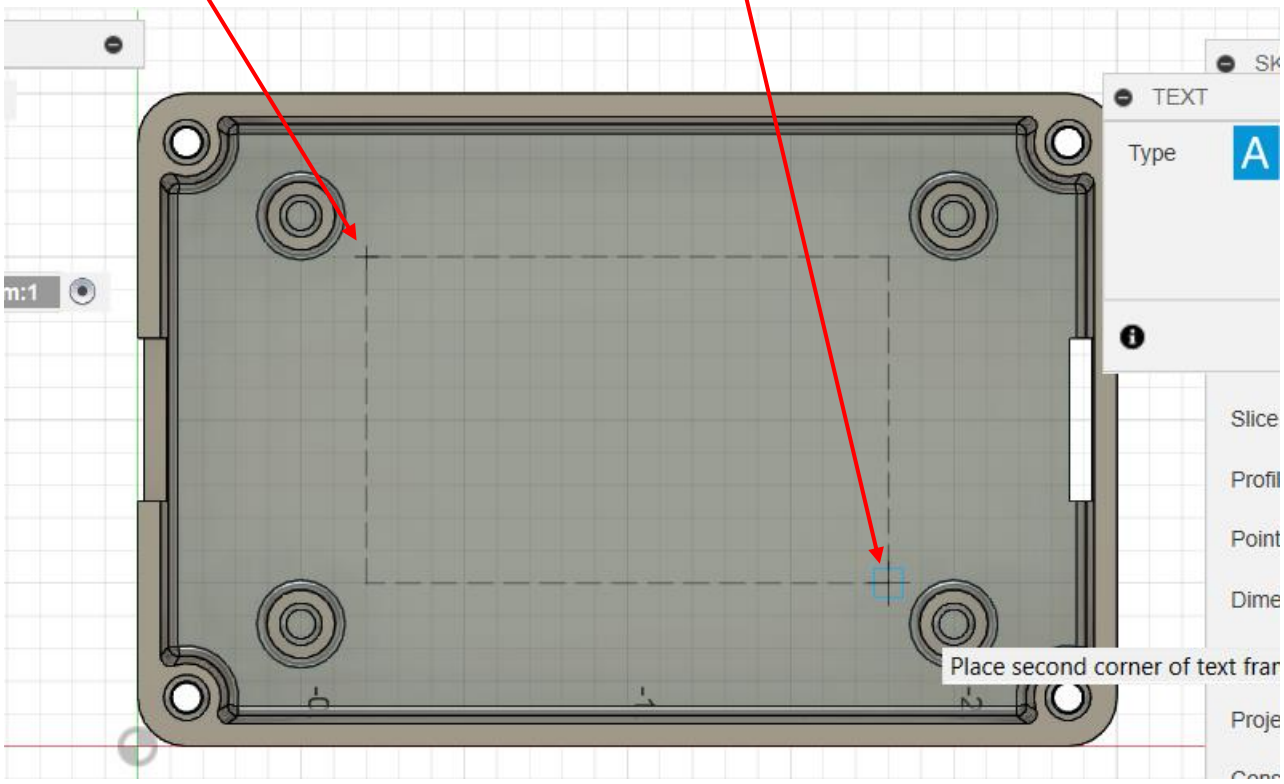
- select **Create Sketch** and click on the **interior bottom surface**
- rename the **Sketch** to **Text**



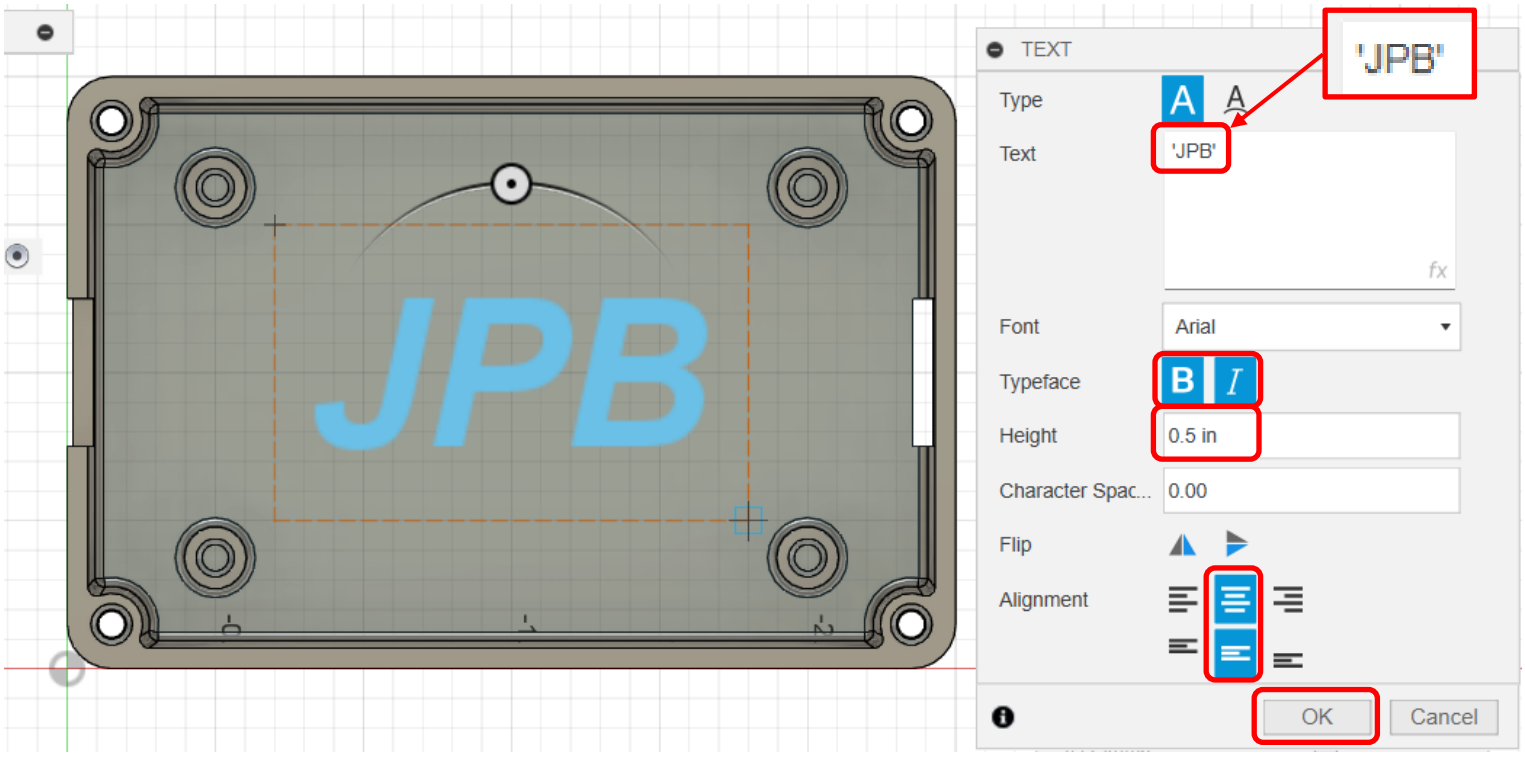
- from the **CREATE** menu, select **Text**
- if a message window appears regarding **Parametric Text**, click is **OK** button



- if a message window appears regarding **Parametric Text**, click is **OK** button
- click on a **point near the top left boss**. These point locations are not critical.
- extend the rectangle down and to the right and click on a **point near the bottom right boss**



- in the Text box enter your **3 initials preceeded by and followed by a single quote**
- click on the **Bold** and **Italic** isons to highlight them
- enter **0.5** for **Height**
- select the **2 center Alignment options**
- click **OK** and **Finish Sketch**



- select the **Extrude** tool
- click on the **text** just created
- enter **0.02** for **Distance** and click **OK**