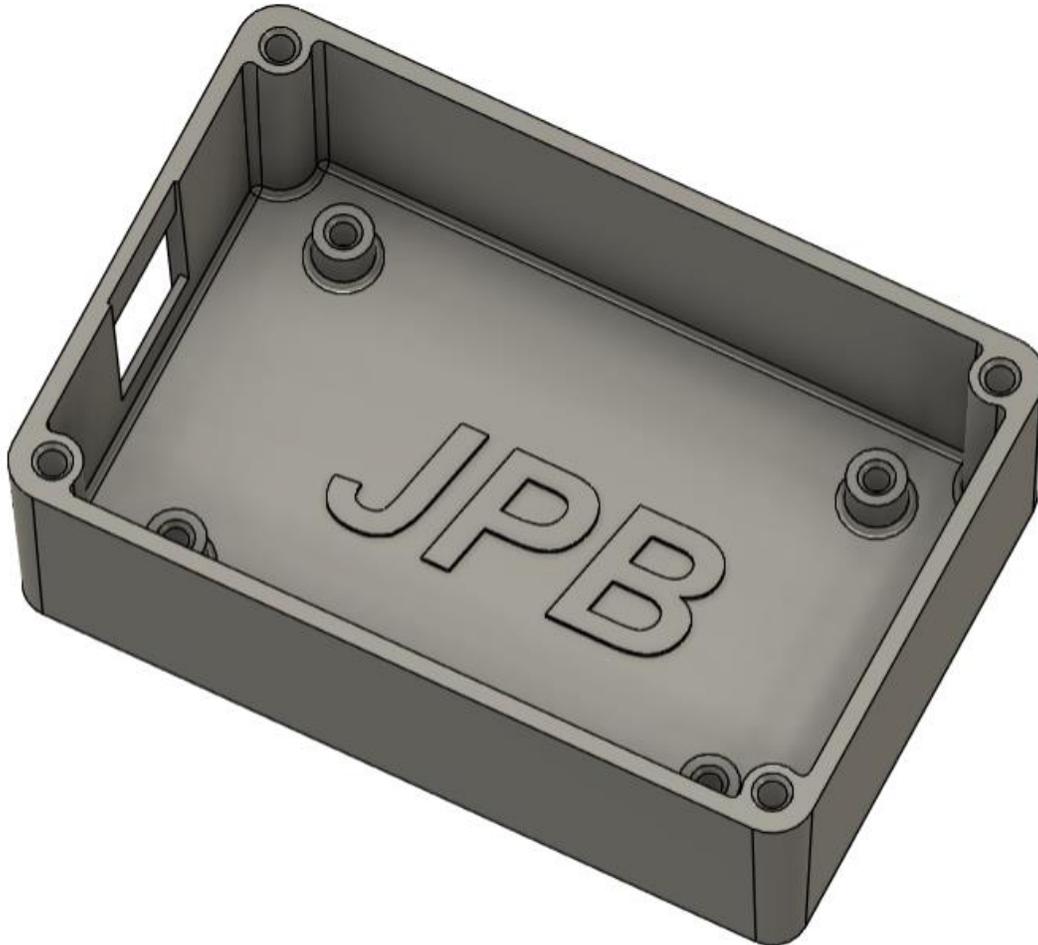


Make an Injection Molded Enclosure



Today's Lesson is Sponsored by McMaster-Carr

McMASTER-CARR

SUPPLY COMPANY

CATALOG

43

NET PRICES

McMASTER-CARR SUPPLY CO.
640 West Lake Street
CHICAGO ILLINOIS
U. S. A.

CATALOG

65

NET PRICES

ESTABLISHED 1901
McMASTER-CARR SUPPLY CO.
640 West Lake Street
CHICAGO 6 ILLINOIS
U. S. A.

Phone
ANDover 3-3161

Cable Address
"MACARCO"

McMASTER-CARR
supply company

PO. BOX 4355 • CHICAGO, ILL. 60680-4355

NET PRICES

catalog **95**

Serving industry since 1901

TELEPHONE

Sales Dept. (312) 833-0300
All Other Departments (312) 834-8000
After 5:00 P. M. (312) 834-8000

FAX (312) 834-9457

PLANT LOCATION
Berwyn, Illinois, U.S.A.

MAIL ADDRESS
PO. Box 4355
Chicago, ILL. 60680-4355 U.S.A.

McMASTER-CARR
CHICAGO, ILLINOIS

CATALOG

131

Serving industry since 1901

WEBSITE
www.mcmaster.com

EMAIL
sales@mcmaster.com
support@mcmaster.com

PHONE
(312) 833-0300
(312) 834-8000

TEXT MESSAGE
79467

STREET ADDRESS
620 N. Collins Line Road
Berwyn, IL 60404-3034

SEVENTEENTH YEAR

STATIONARY ENGINEERS DIRECTORY

1908

STEAM OF ILLINOIS ELECTRIC

CATALOG NO 28

INDUSTRIAL MERCHANDISE

NET PRICES

McMASTER-CARR SUPPLY CO.
640 West Lake Street
CHICAGO ILLINOIS

N^o 27

OUR SALES MANUAL

INDUSTRIAL MERCHANDISE CATALOG

compare our prices

McMASTER-CARR SUPPLY CO.
640 West Lake Street
CHICAGO ILLINOIS

T. J. McMASTER, PRESIDENT

W. S. CARR, SECY AND TREAS.

McMaster-Carr Supply Co.

Formerly McMASTER-DAVIS SUPPLY CO.

High Grade Steam Specialties and Engineer Supplies

Ashton Pop Safety Valves and Gages
Austin Steam and Oil Separators
Cadman Blow-off Valves and Gage Cocks
Curtis Reducing Valves
"P. B. H." Quick Closing Water Gages and Weighted Gage Cocks
"Clean Seat" Globe and Angle Valves
Canton-Hughes Simplex and Duplex Steam Pumps
Anderson Non-Return Valves

Gould Continuous Feed Water Regulator
Standard Steam and Water Boilers
The Erwood Straightway Swing Gate Back Pressure and Check Valve
Wright "Emergency" High Pressure and "Victor" Low Pressure Steam Traps
Wright "Cyclone" Exhaust Heads and Safety Water Columns
White Star Oil Filters and Continuous Oiling Systems
White Seal Sheet Packing

MECHANICAL RUBBER GOODS

Phones: Main 4400
Automatic 9791

July 25, 1903

The INTERNATIONAL STEAM ENGINEER

CONTINUED WITH THE STATIONARY ENGINEER AND MACHINIST

McMaster-Carr
Supply Co.
Chicago

General
Supplies

Catalogue No. 14

Chicago Engine Jack

With it one man can turn an engine off center quickly. Send for circular "P." Tells all.

McMaster-Carr Supply Co.
Chicago



DON'T THROW IT AWAY

Waste oil and drippings can be reclaimed again and again and each time it will give the best of service.

An Acme Oil Filter

will do the work.
The dirty oil first passes through water by gravity then is filtered through animal bone black, insuring perfect purity.
We are ready to send an Acme on 30 day's trial.

Walter L. Flower & Co.
1000 Chemical Bldg.,
ST. LOUIS, MO.

CHICAGO AGENCY
McMaster-Davis Supply Co.,
640 E. Lake Street

NEW YORK AGENCY:
James Begg & Co., 9 Dry St.

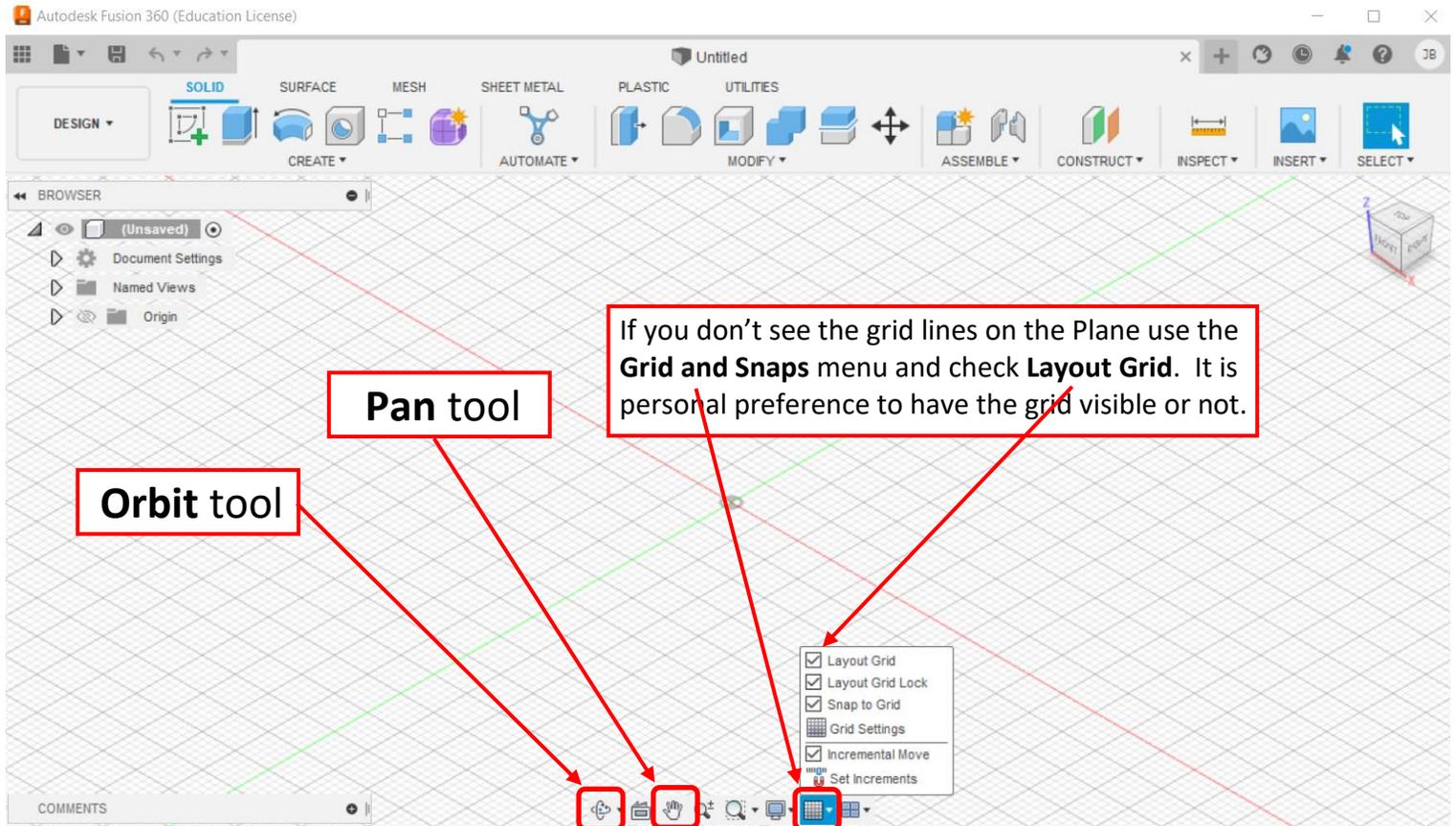


Contents

Changing the View of a Design.....	4
Starting a Design in Fusion (START HERE)	5
Creating a New Component	6
Setting User Parameters	7
Adding Drafts.....	15
Adding Draft to Holes.....	19
Adding Side Holes.....	22
Creating the Side Hole – Increasing Draft	25
Creating the Side Hole – Measuring Draft	27
Creating the Side Hole – Creating a Construction Midplane	28
Creating the Side Hole – Creating a Section Analysis.....	29
Creating the Side Hole – Creating a Cross-section Sketch	30
Adding Bosses	40
Adding Text	46

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.

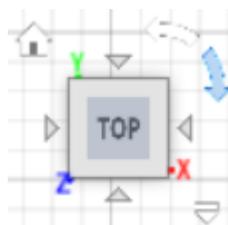
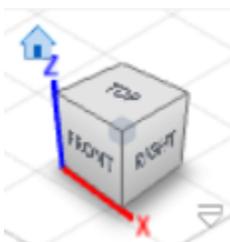


Here is a close-up of the View Cube at the top right of the window.

- 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



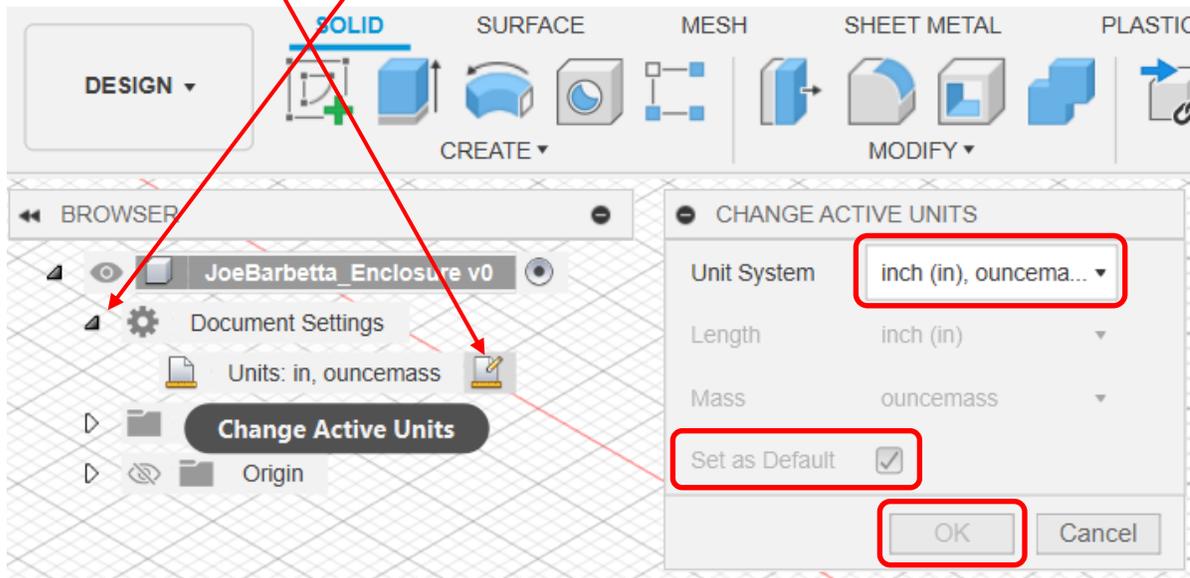
Starting a Design in Fusion (START HERE)

- 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 **_Enclosure** e.g. **JoeBarbetta_Enclosure** (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.

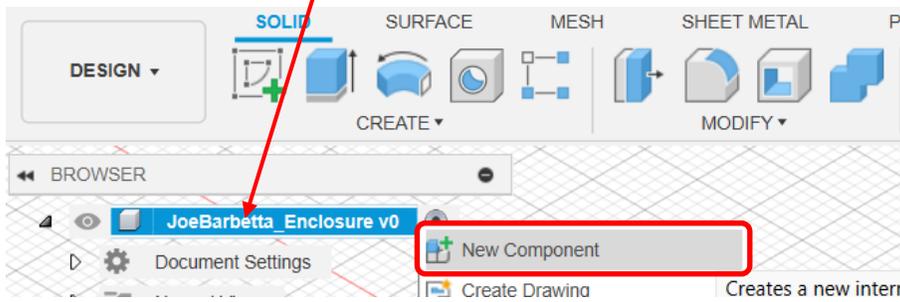


Note that the default units are in mm, which we just changed to inches.

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

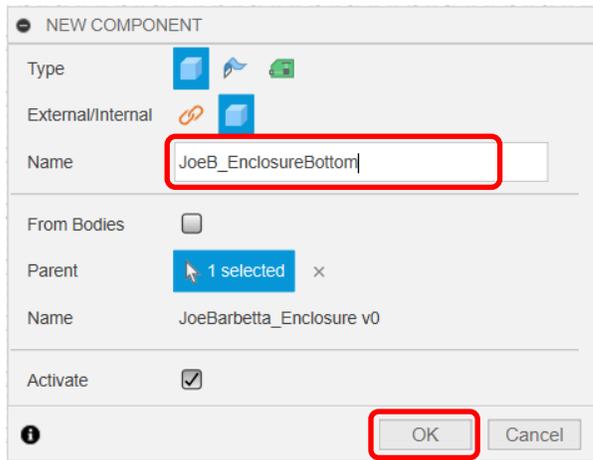
Creating a New Component

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

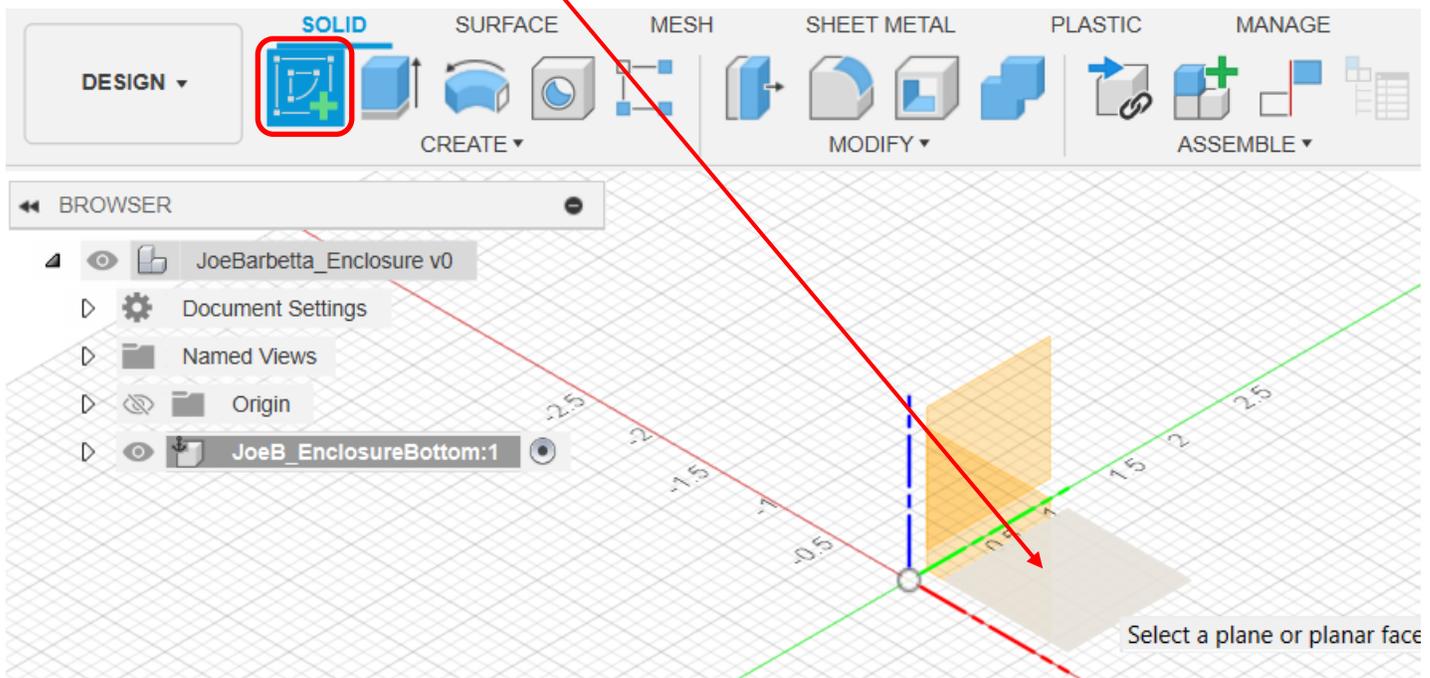


- in the **Name** box enter your **first name followed by your last name initial and _EnclosureBottom** (note the underscore), e.g. **JoeB_EnclosureBottom**

- click **OK**

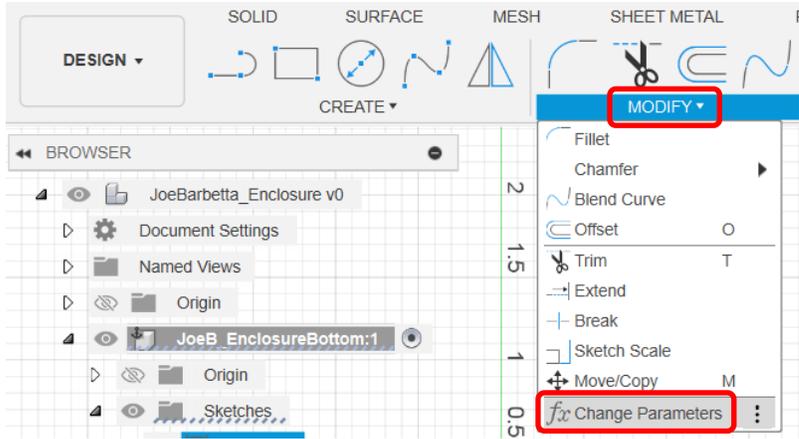


- select **Create Sketch** and click on the **bottom rhombus**



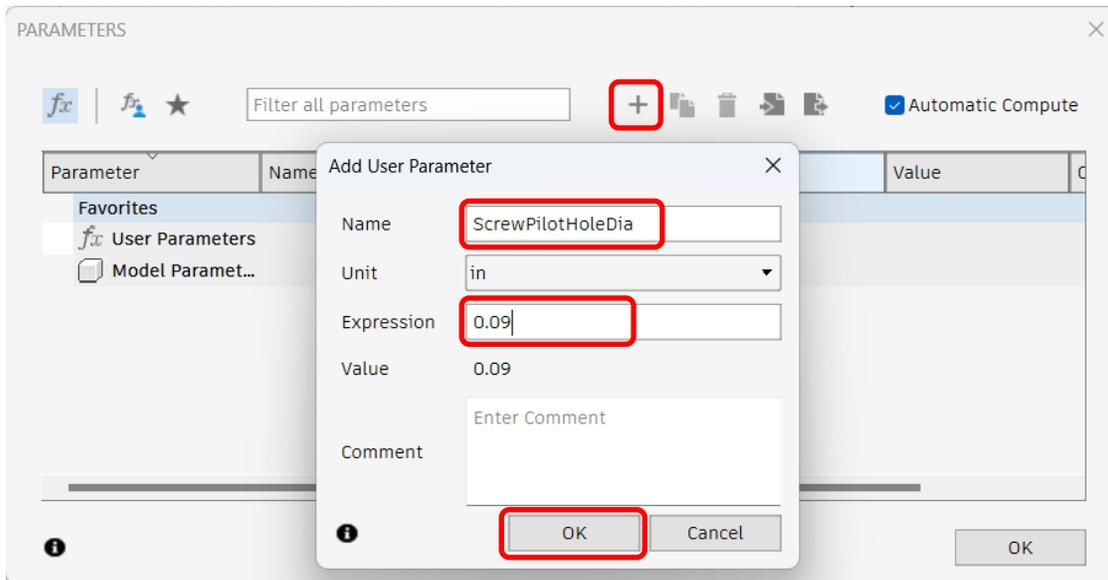
Setting User Parameters

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



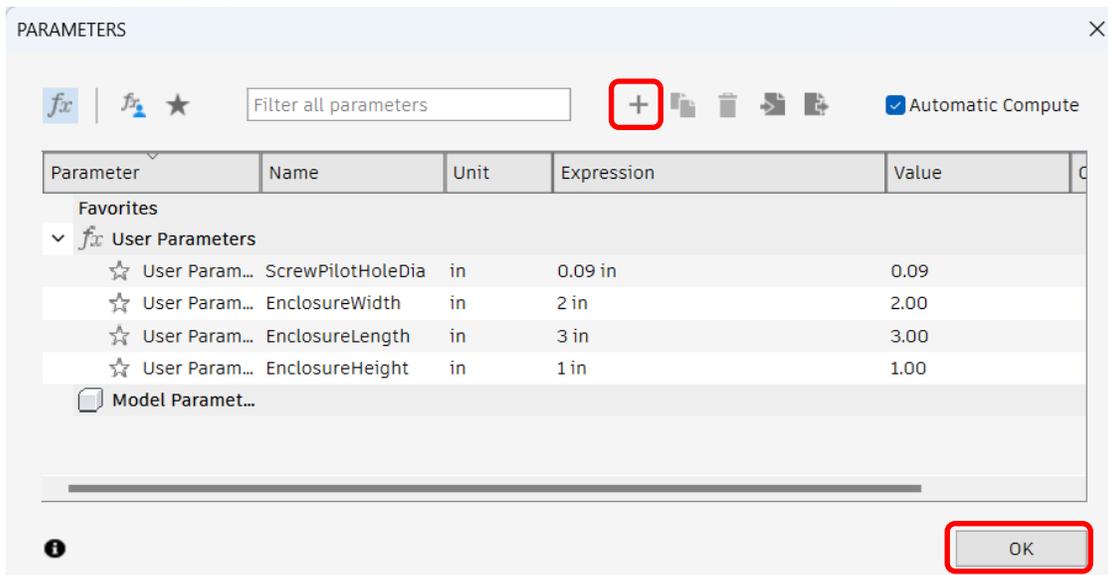
- click on the top + icon

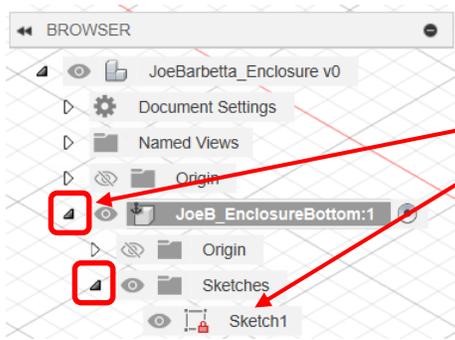
- set the **Name** to **ScrewPilotHoleDia**, the **Expression** to **0.09**, and click **OK**



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

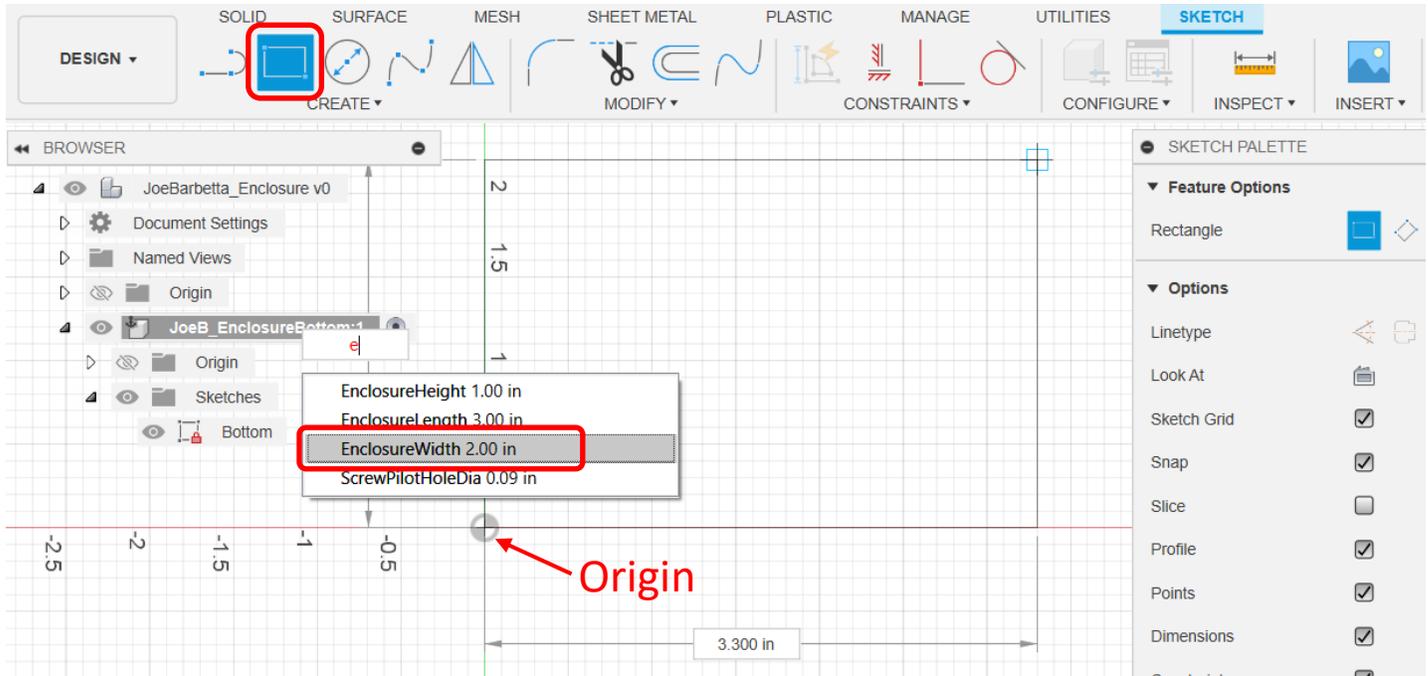
- click **OK**





- 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**

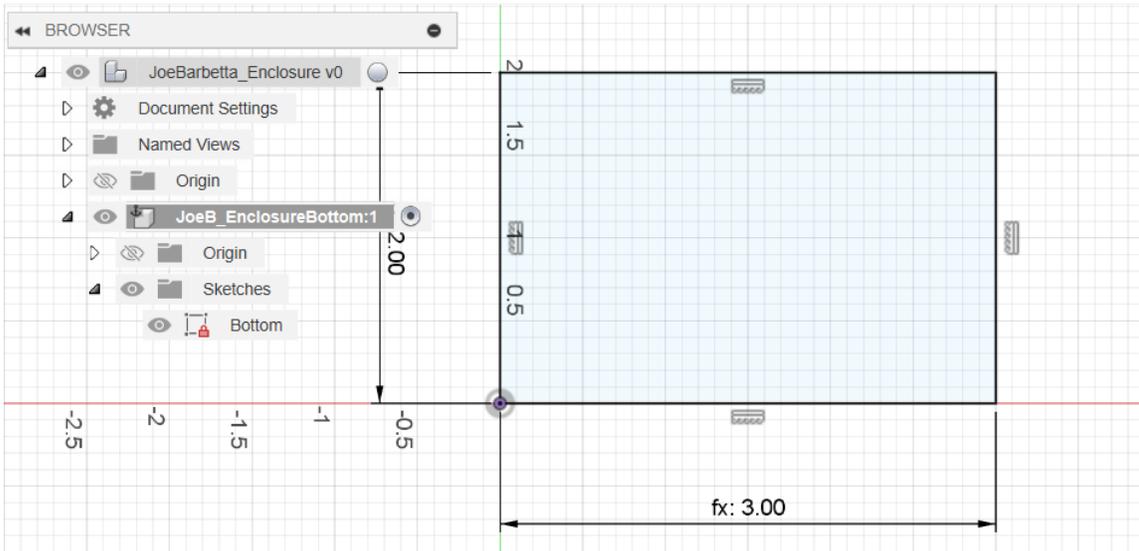
- select the **Rectangle** tool
- click on the **Origin** and extend the rectangle up and to the right
- type **e**, use the **Down Arrow Key** to select **EnclosureWidth**, press the **Enter Key**



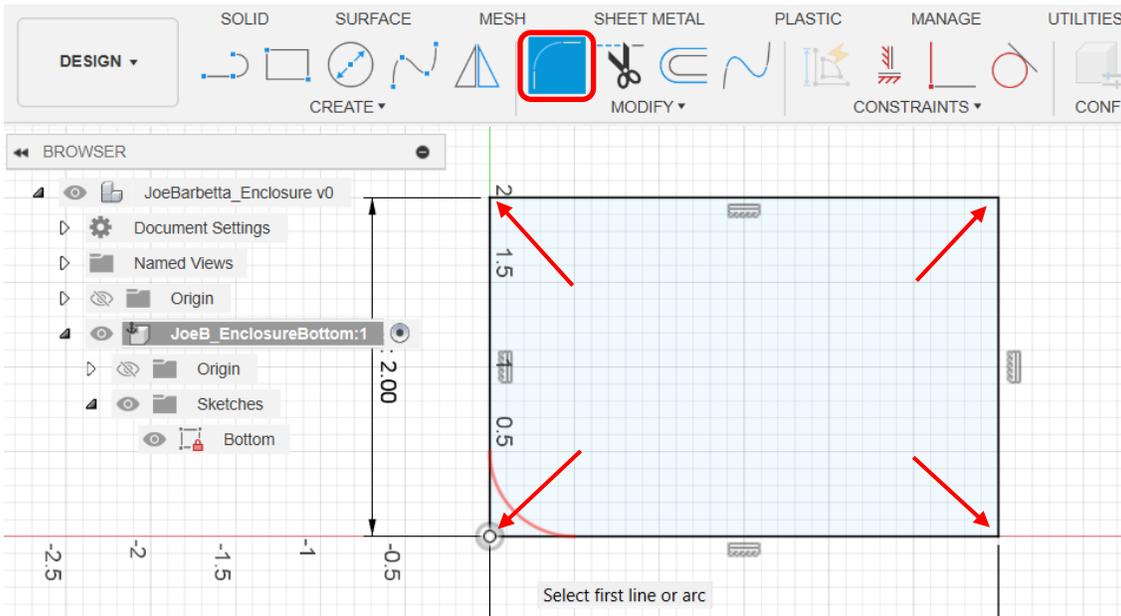
- press the **Tab** key to switch to the other dimension
- type **e**, use the **Down Arrow Key** to select **EnclosureLength**, press the **Enter Key** twice



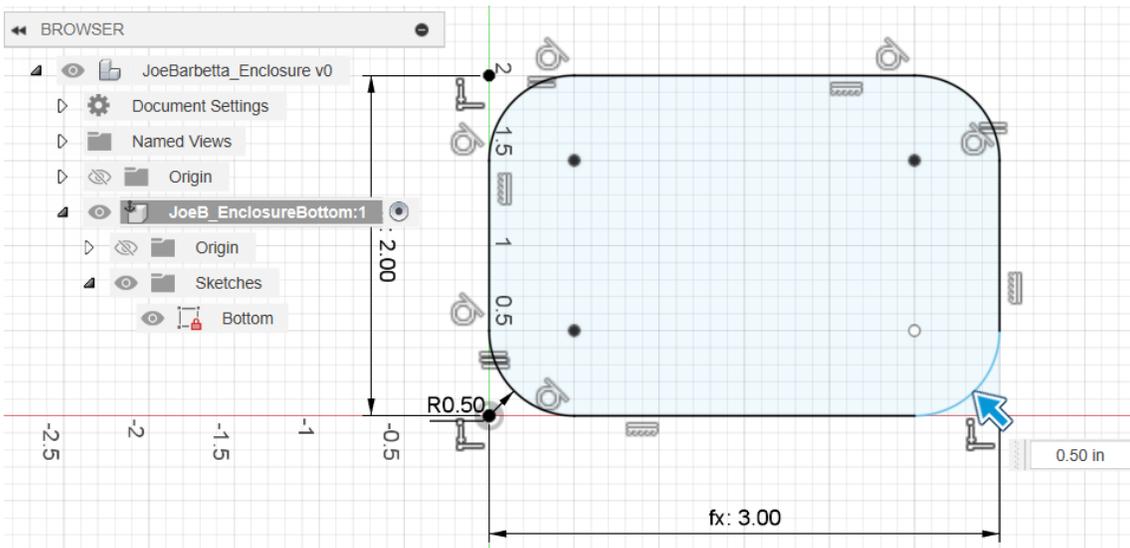
The rectangle should look like that below. It is fine if the dimension lines look different.



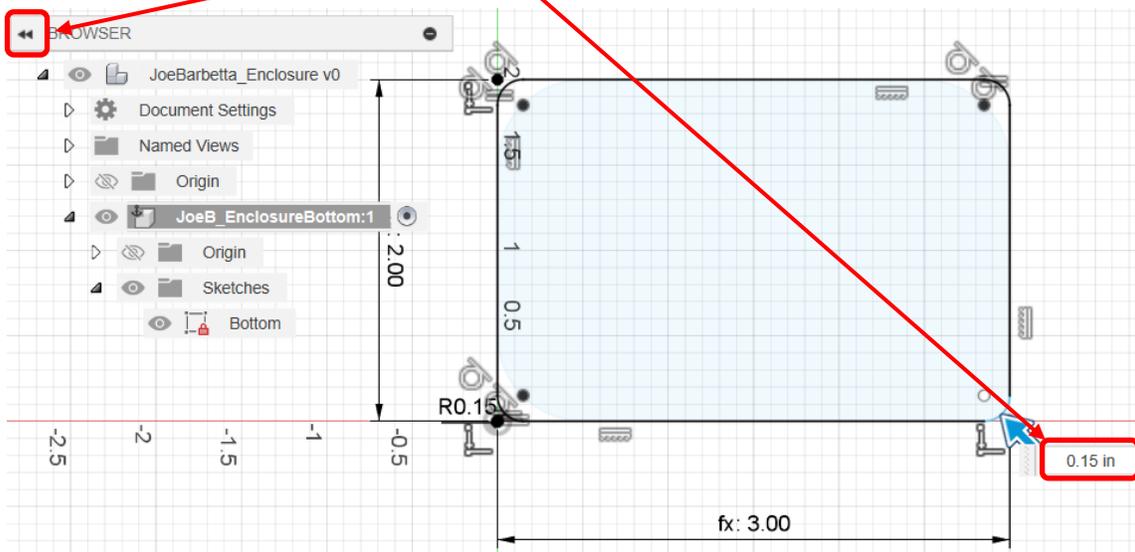
- select the **Fillet** tool. If it is not visible, find it in the MODIFY menu
- click on the **each corner** of the rectangle. Warnings that appear during the fillet operation can be ignored.



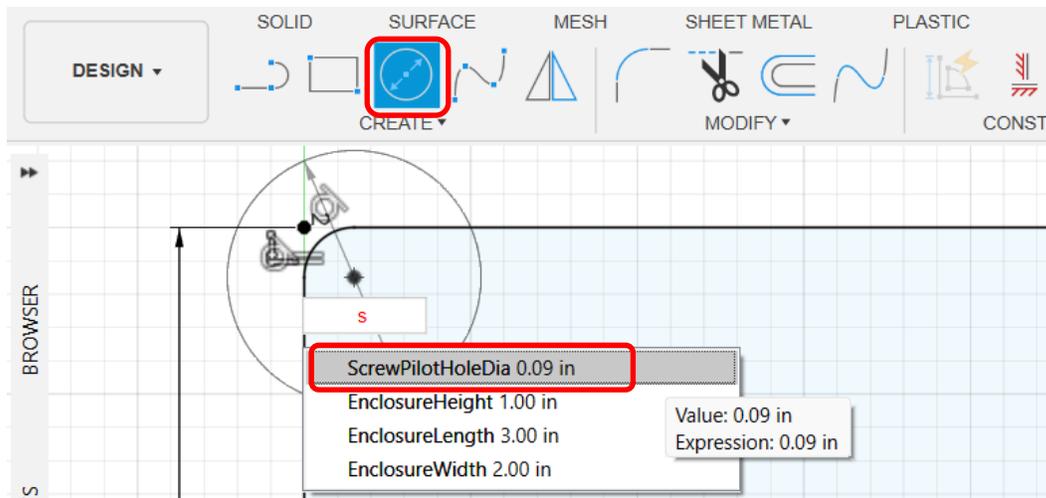
After each corner is clicked it should look like that below.



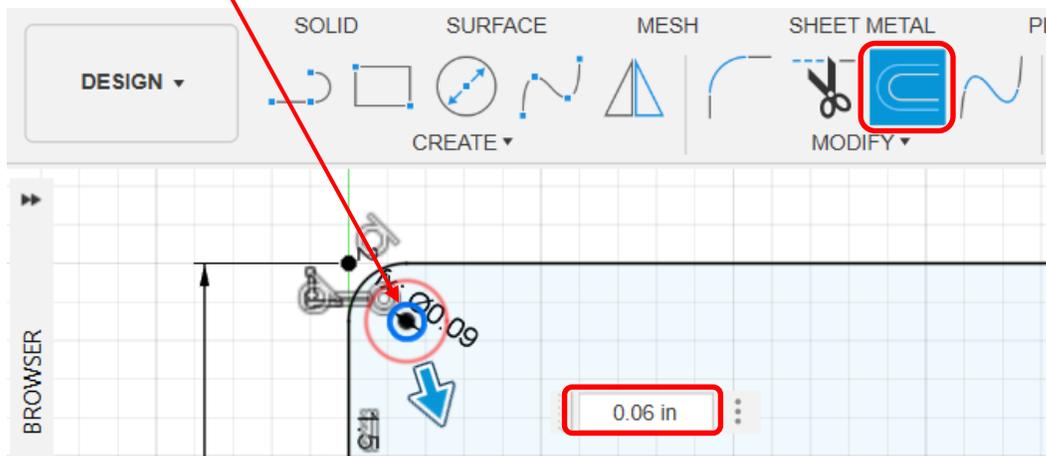
- change the value in the value box to **0.15** and press the **Enter** key
- click on the **double arrow** to hide the BROWSER



- 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



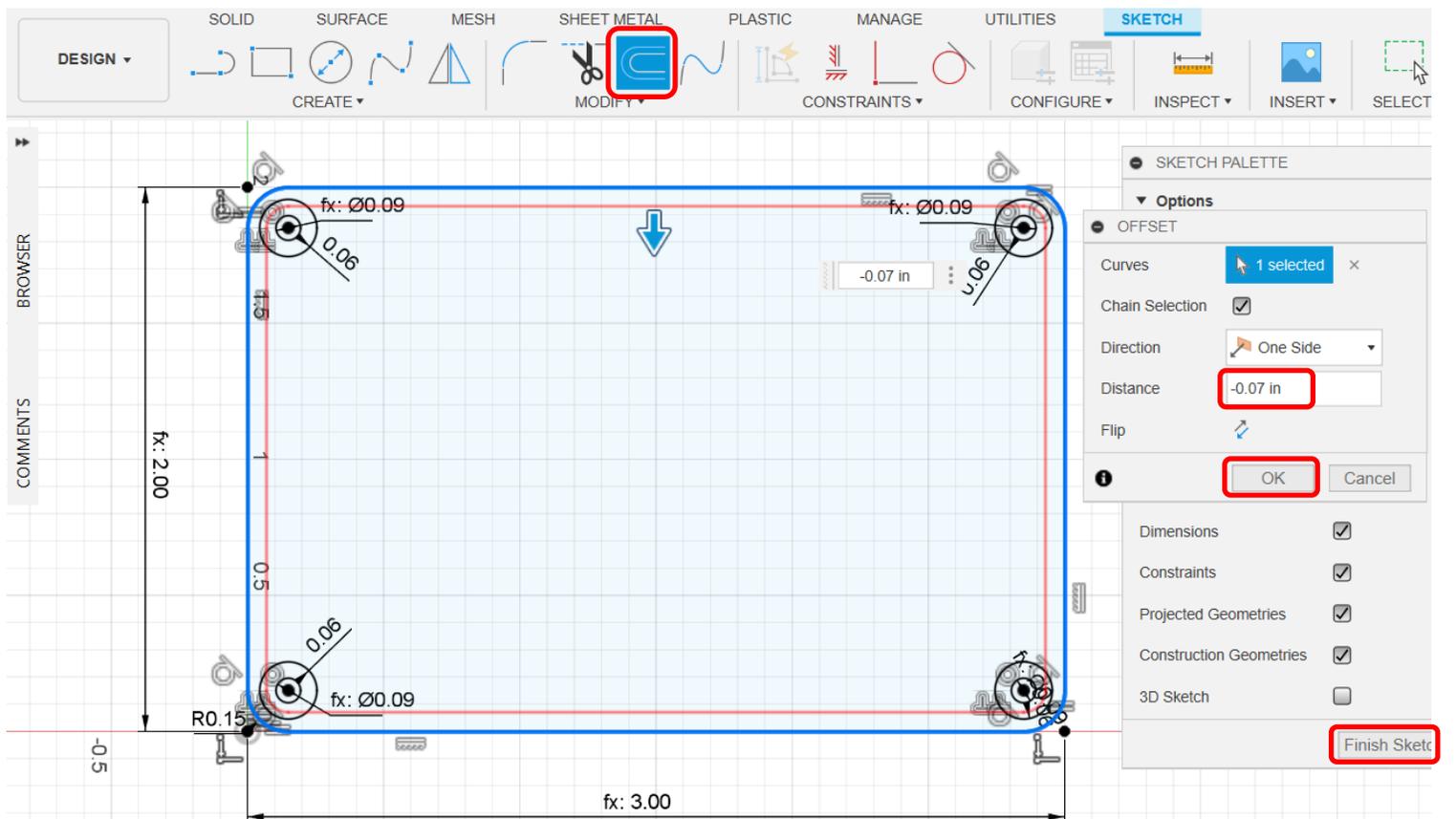
- select the **Offset** tool. If it is not visible, find it in the MODIFY menu.
- click on the **small circle** just created and enter a value of **0.06** in the value box



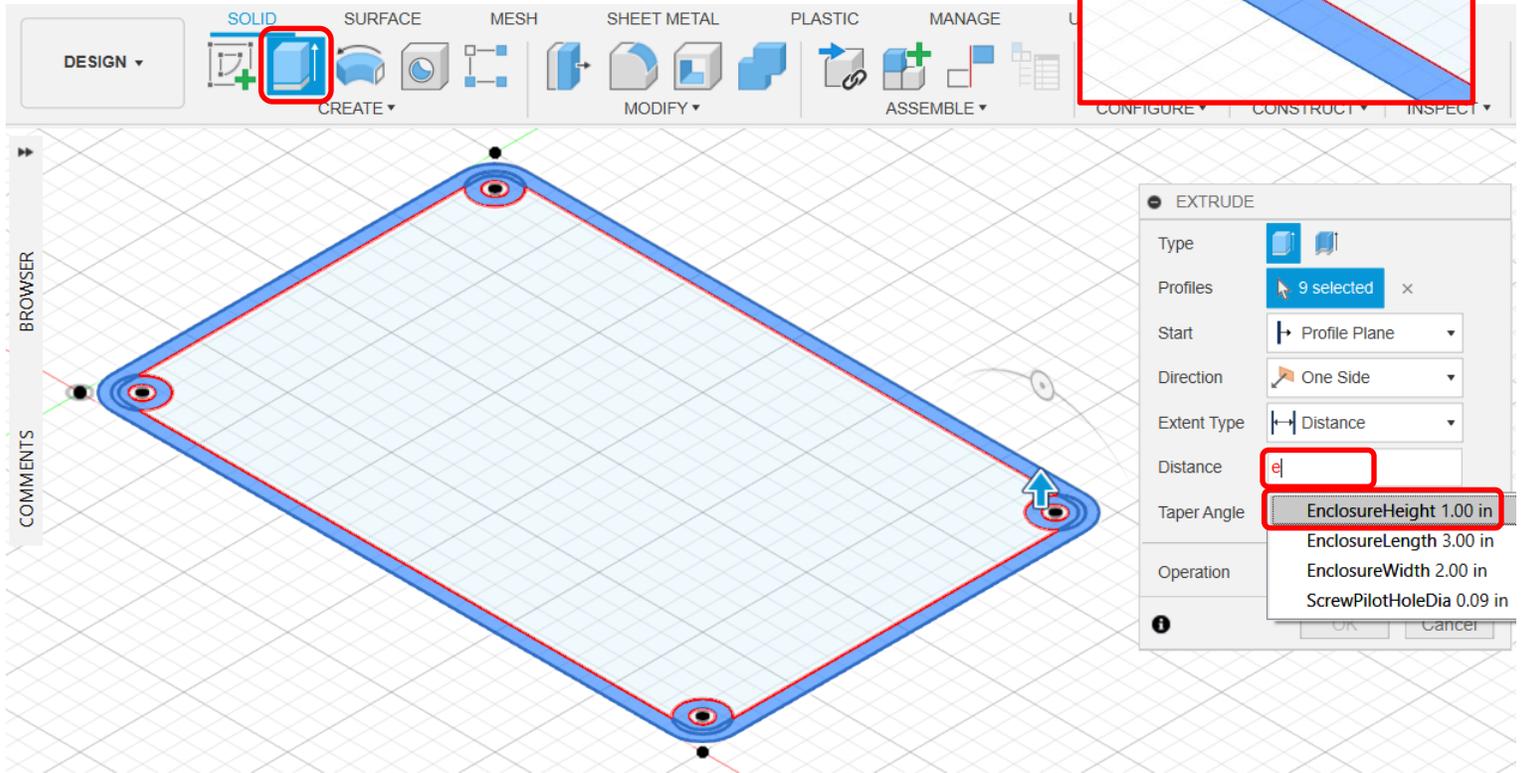
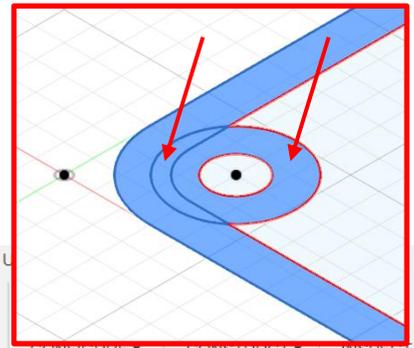
The result should look like that below.



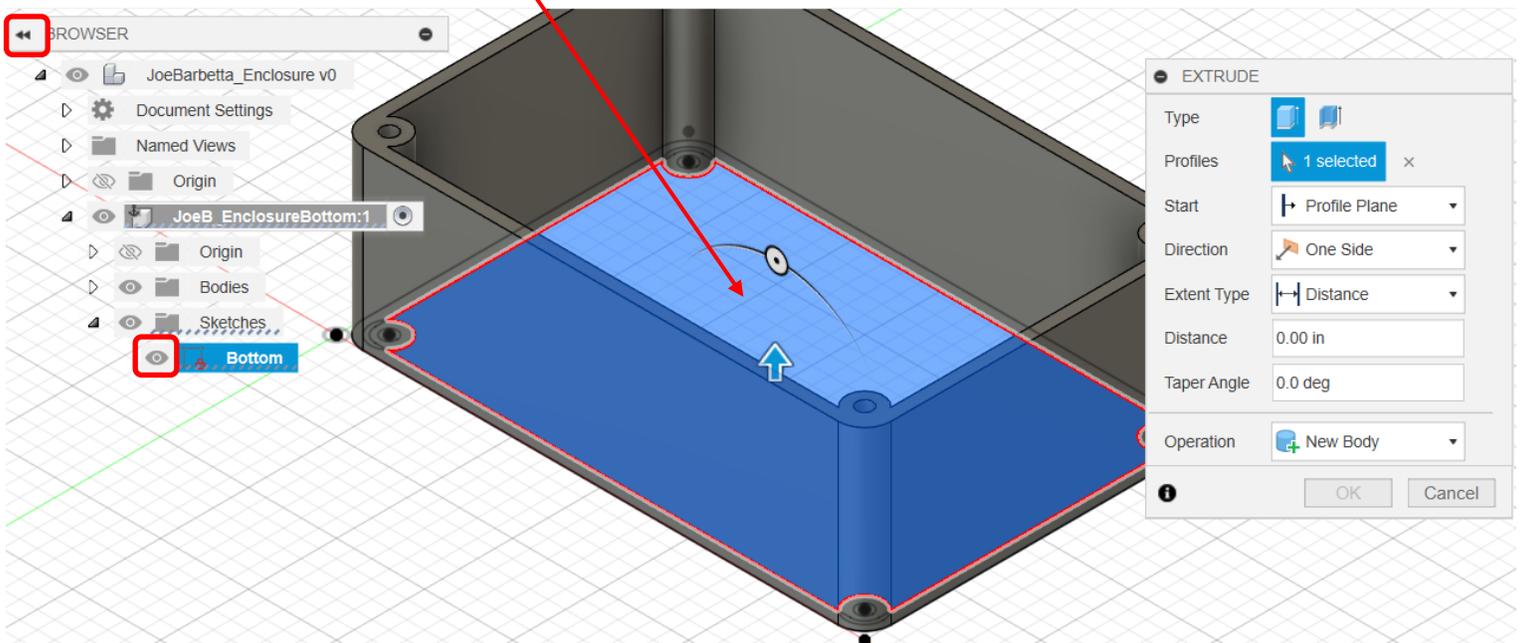
- select the **Offset** tool again and click on the **rectangle**
- enter **-0.07** (note the minus sign) in the **Distance** box and click **OK**
- click **Finish Sketch**



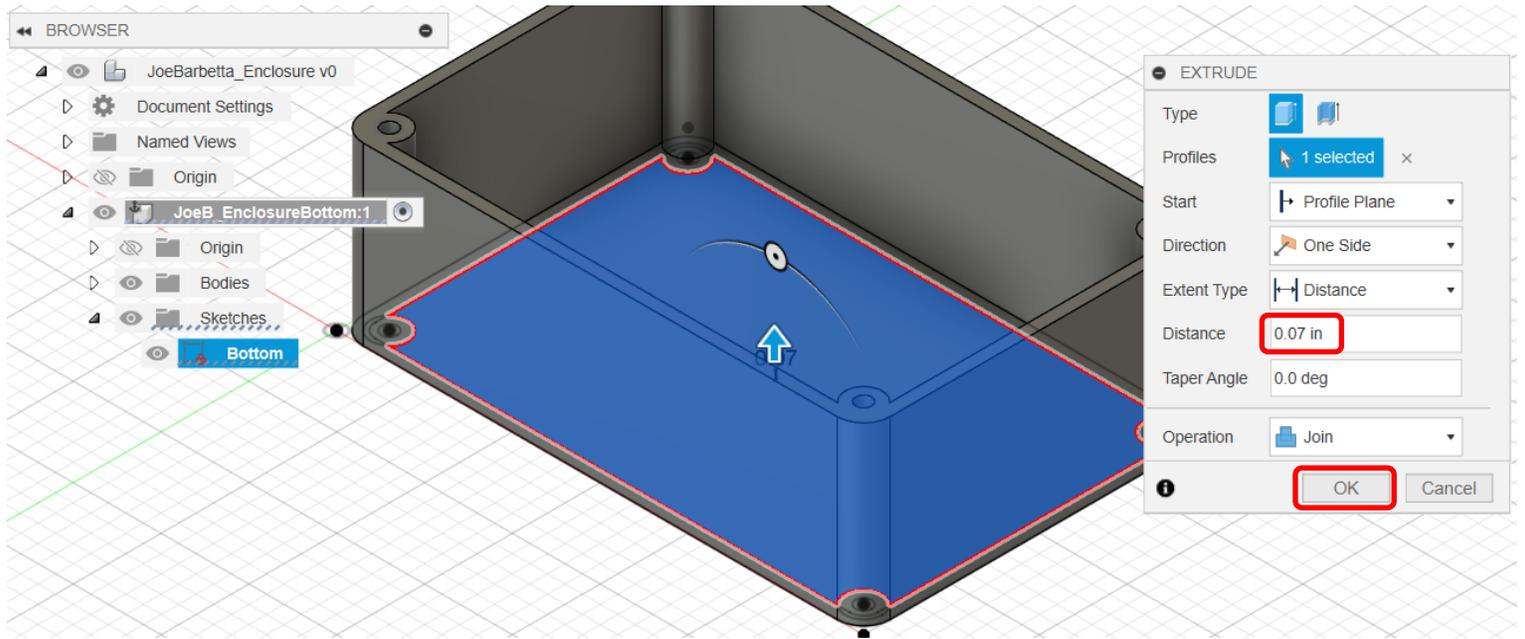
- click on the **Home** icon at the **View Cube**
 - select the **Extrude** tool
 - click on the outer rectangular region and then on the 2 regions at each corner
- It will help to zoom into each corner when doing so. The inset image shows a closeup of a corner. Note that the inside of the small circles is not selected.
- in the **Distance** box type **e** and select **EnclosureHeight** and click **OK**



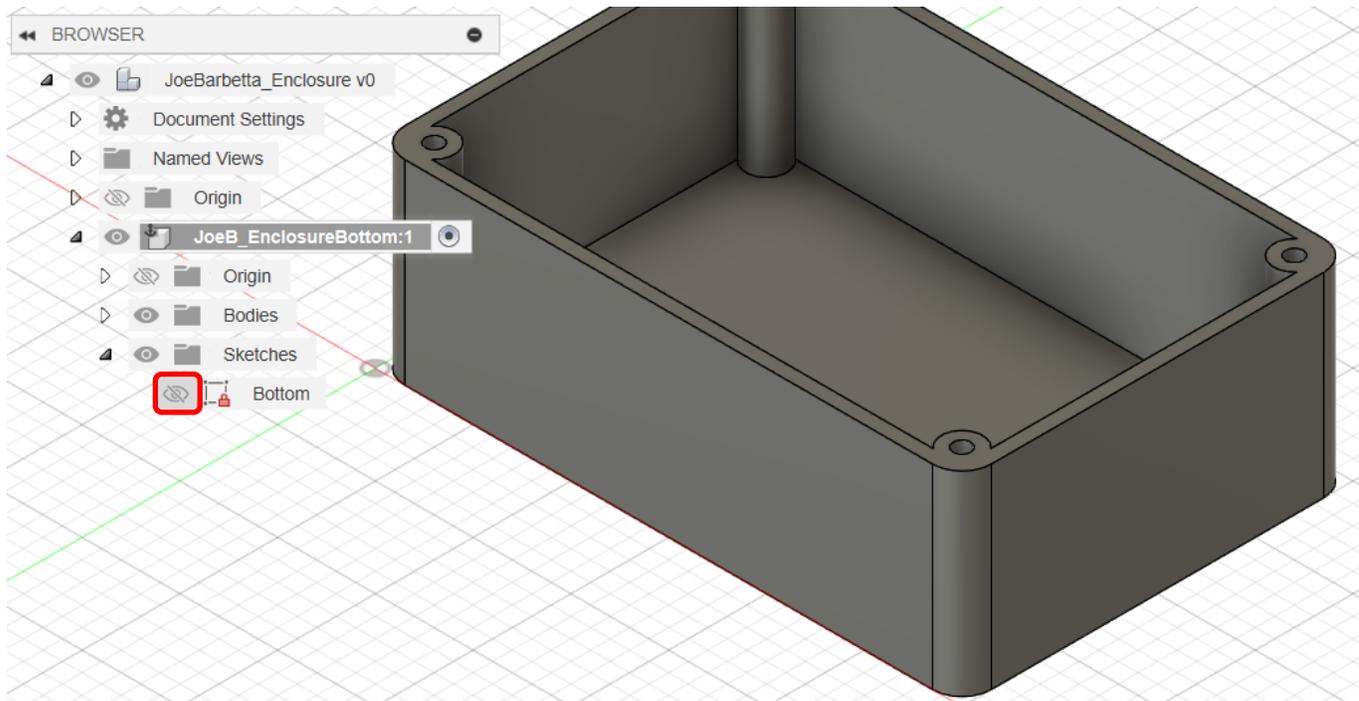
- click on the **double arrow** to reopen the BROWSER
- click on the **eye** icon for the **Bottom** sketch to make it visible again
- select the **Extrude** tool and click on the **Sketch**



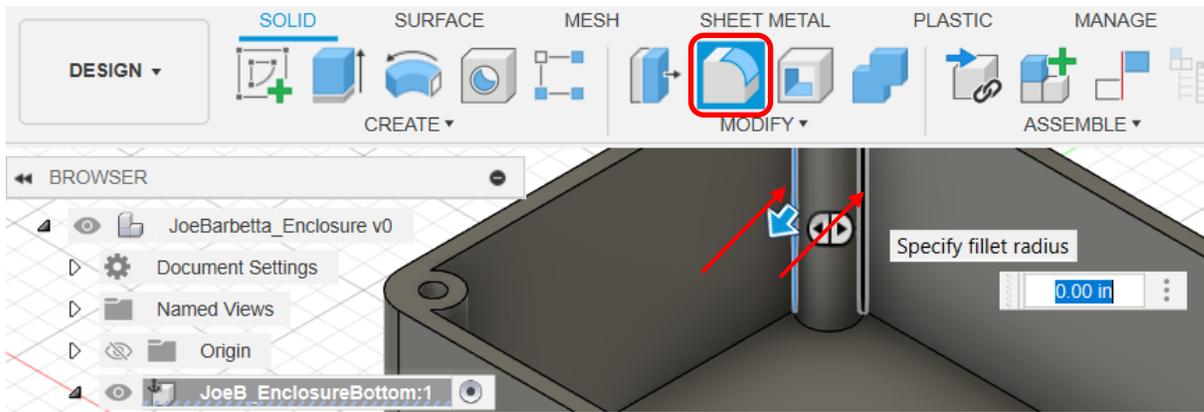
- for **Distance** enter **0.07** and click **OK**



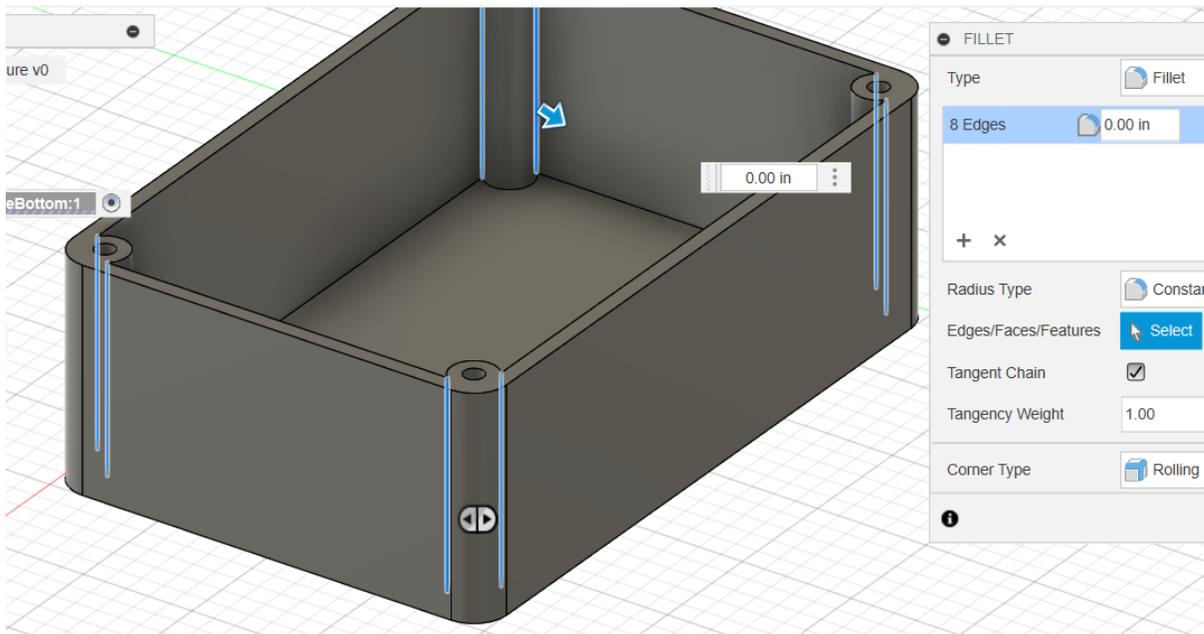
- click on the **eye** icon for **Bottom** to hide it



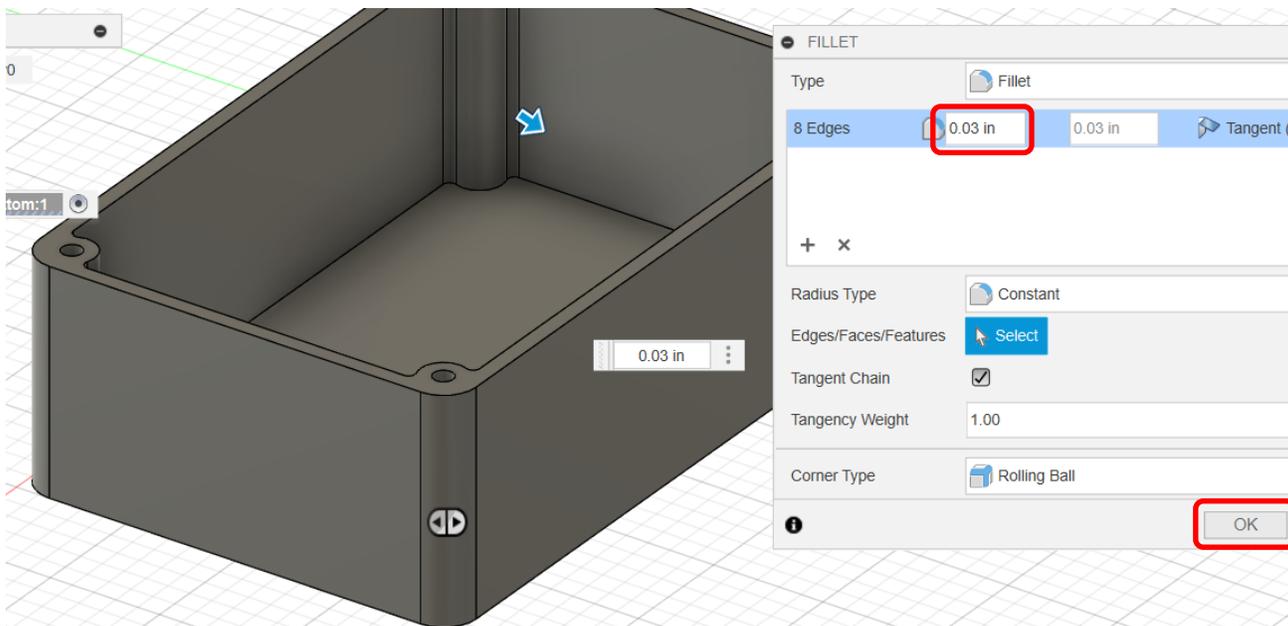
- select the **Fillet** tool. If it is not visible, find it in the MODIFY menu
- click on **edges** at the **intersection on the cylindrical feature and the inside walls**



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

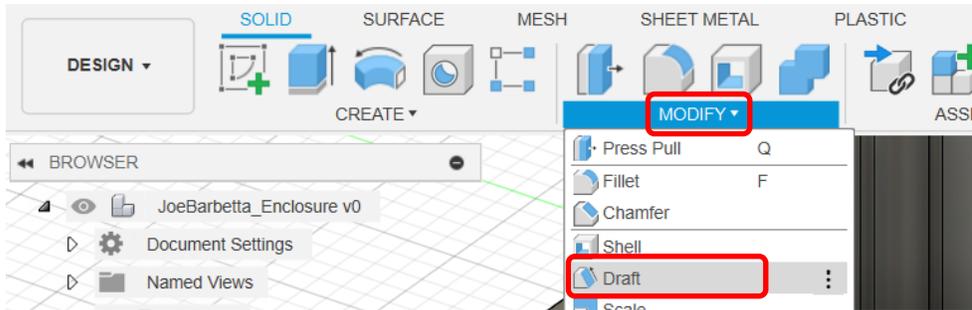


- enter **0.03** and click **OK**

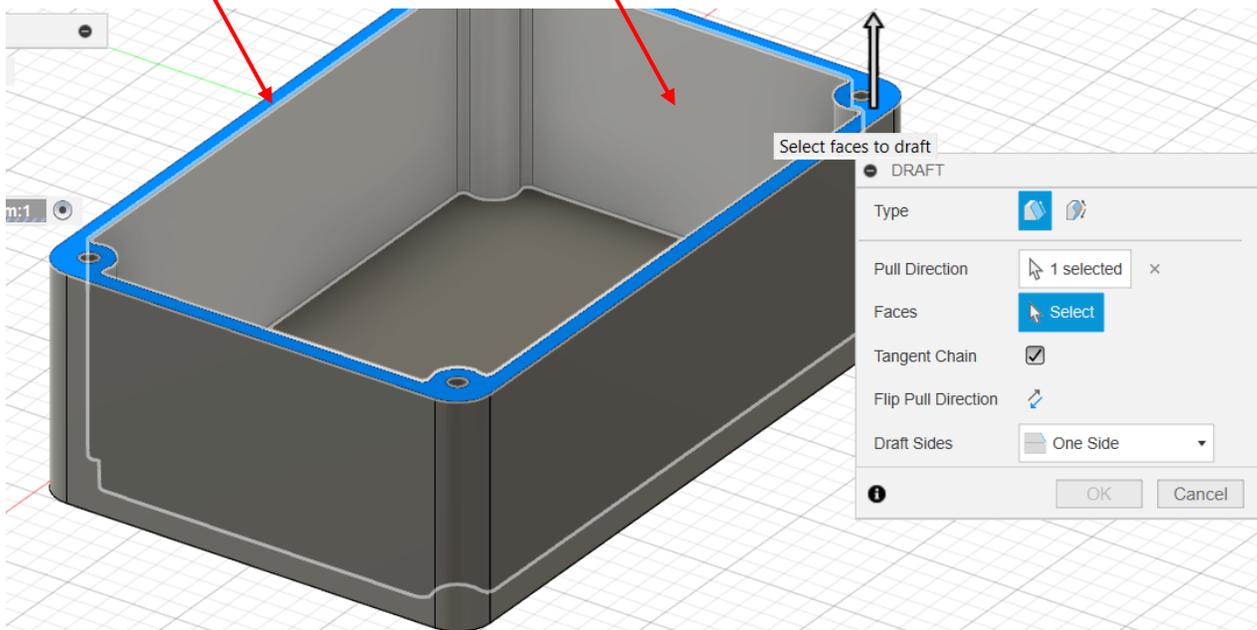


Adding Drafts

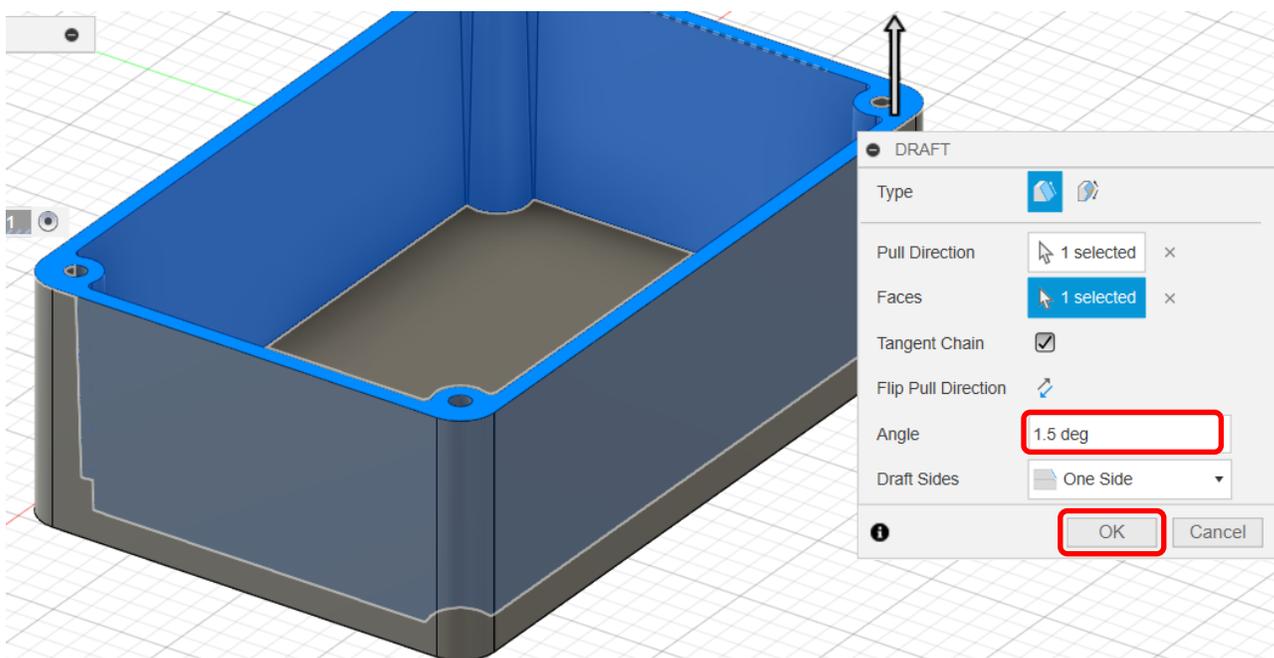
- from the **MODIFY** menu select **Draft**



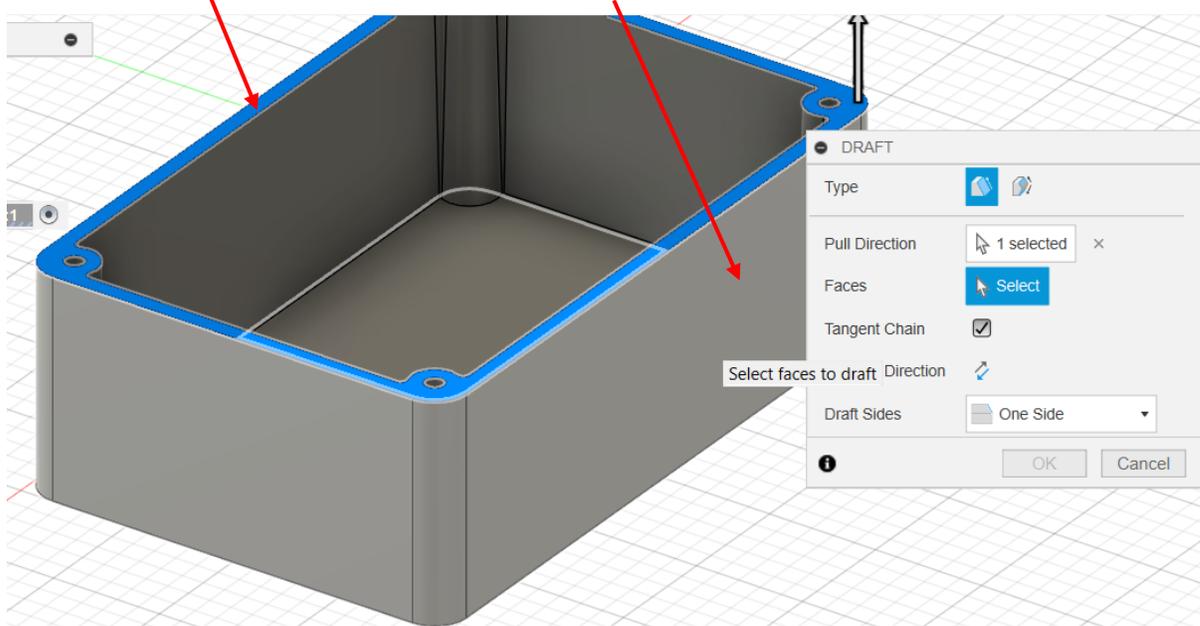
- click on the **top surface** and then click on an **interior wall**



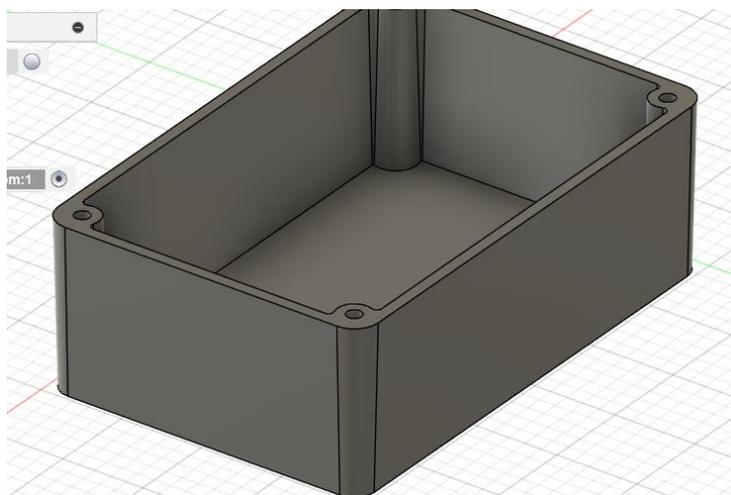
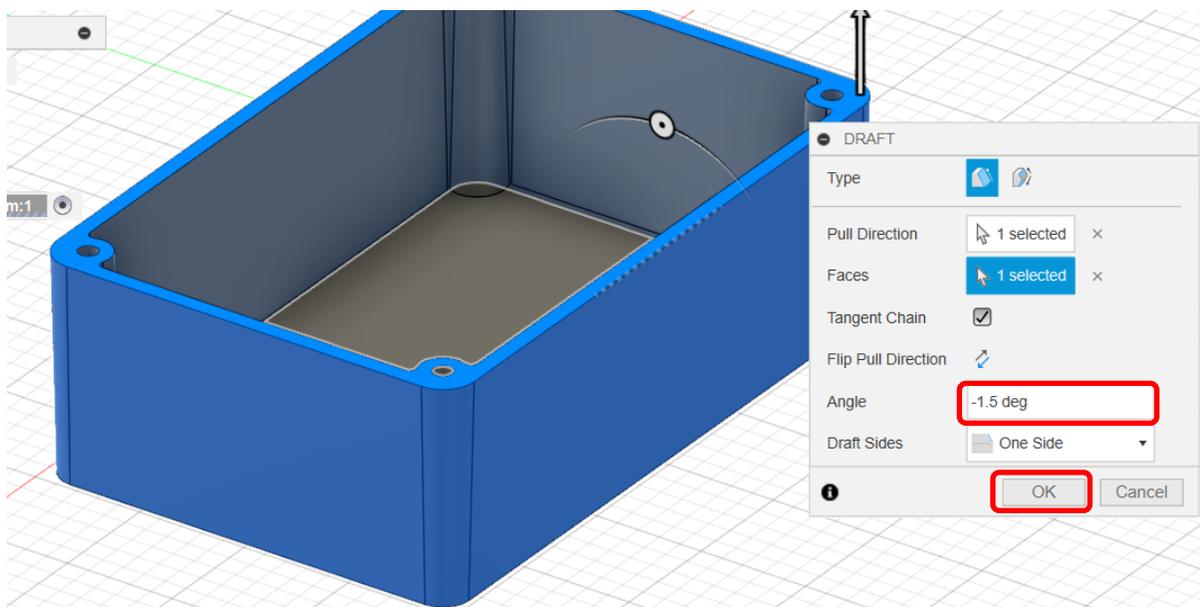
- enter **1.5** for **Angle** and click **OK**



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

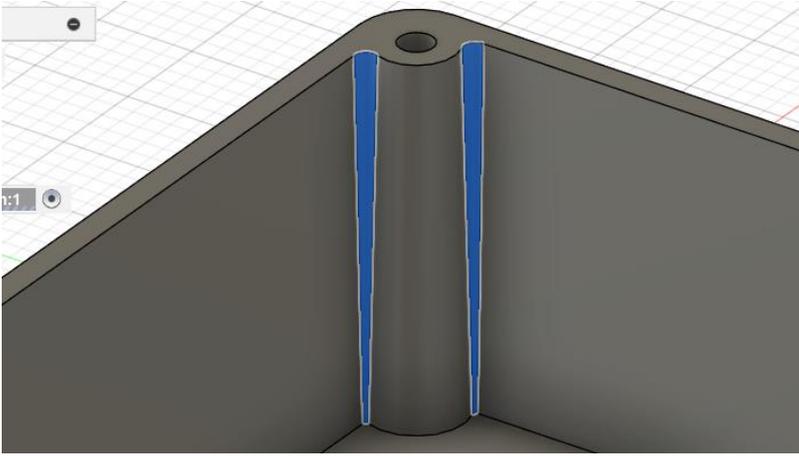


- enter **-1.5** (note the minus sign) for **Angle** and click **OK**

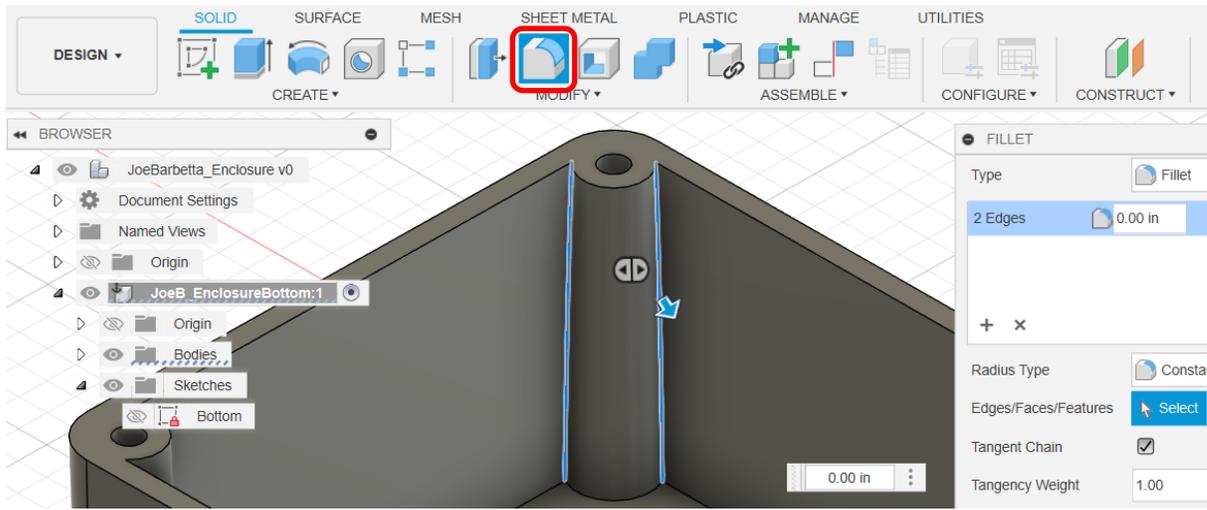


This is the result of applying

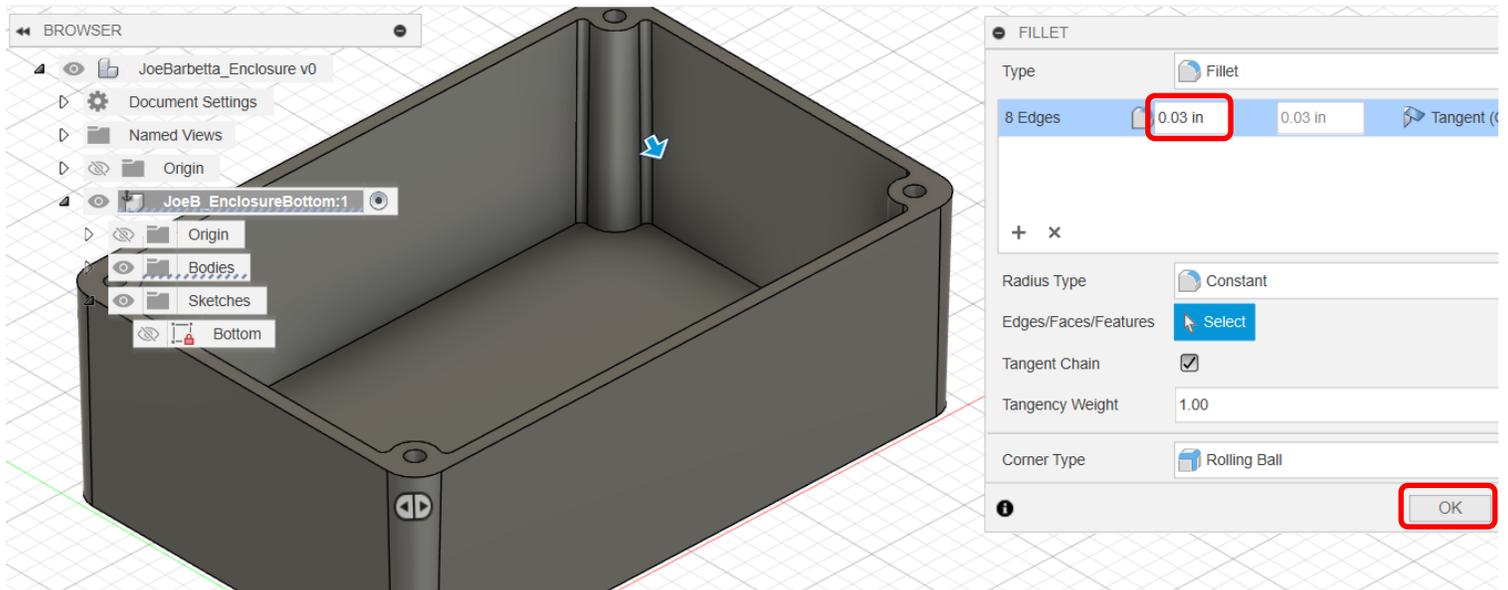
- zoom into an interior corner
- **hold the Shift key down** and select the **2 fillets** and press the **Delete** key
- perform the same operation at the **3 other interior corners**



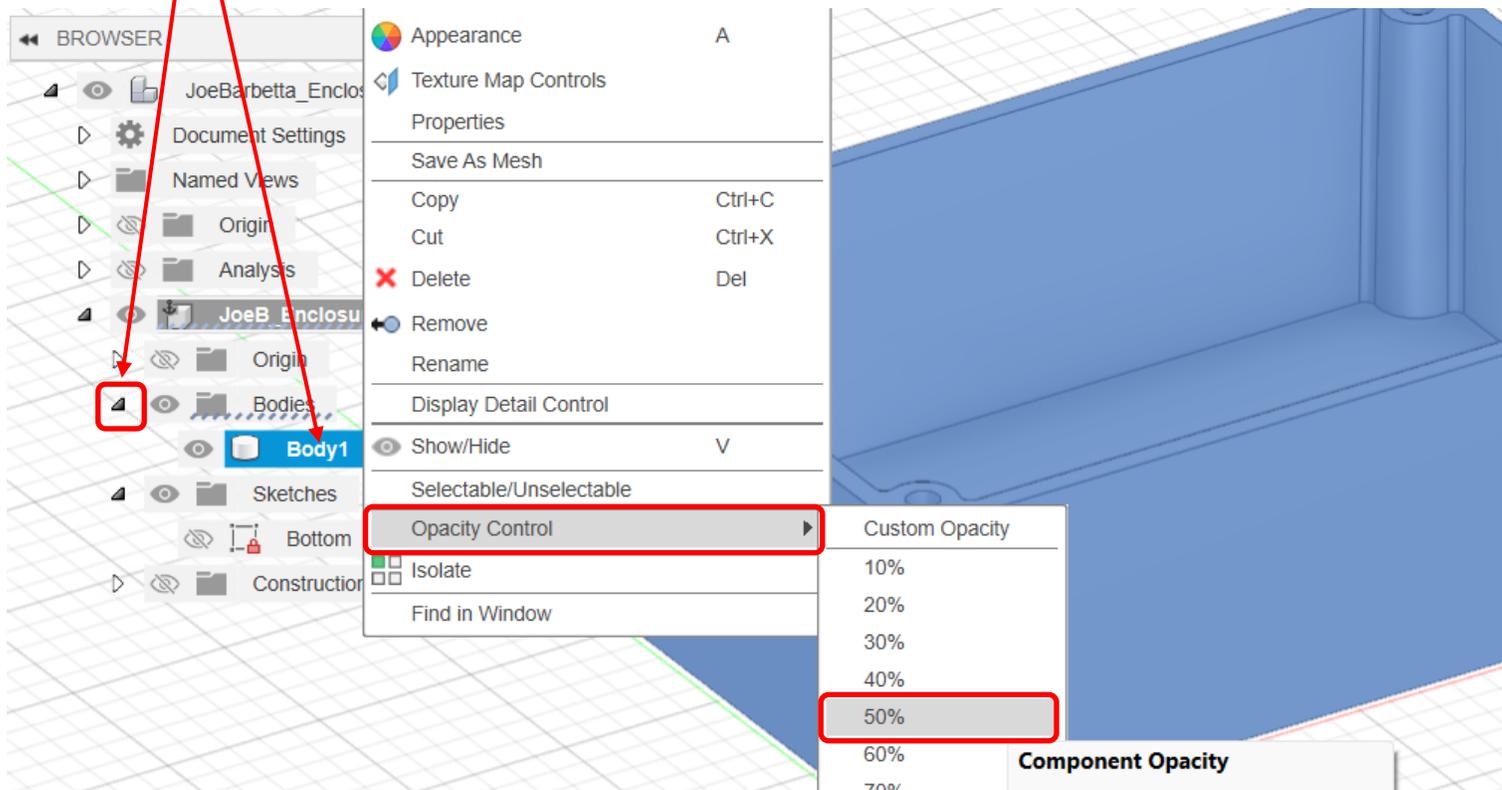
- select the **Fillet** tool
- as done previously, zoom into each interior corner to select **2 edges at each interior corner**



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

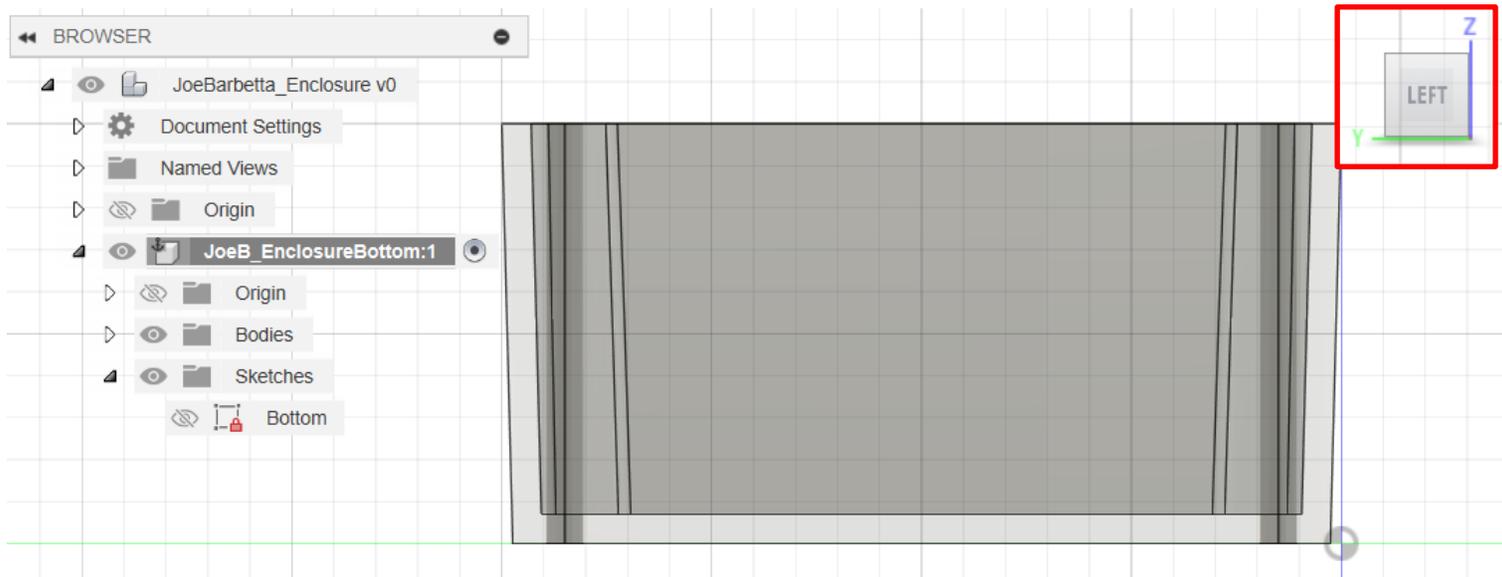


- click on the **arrow** for the **Bodies** folder to open it
- right-click on **Body1** and select **Opacity Control** and **50%**



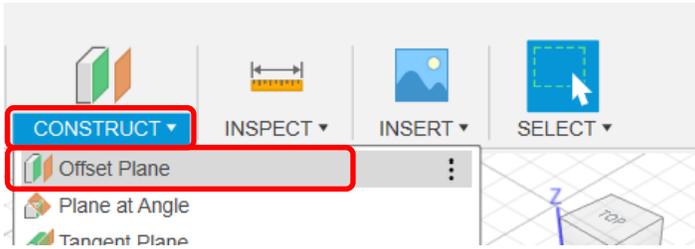
- click on the **LEFT** labeled face of the **View Cube**.

The angled walls show the draft.

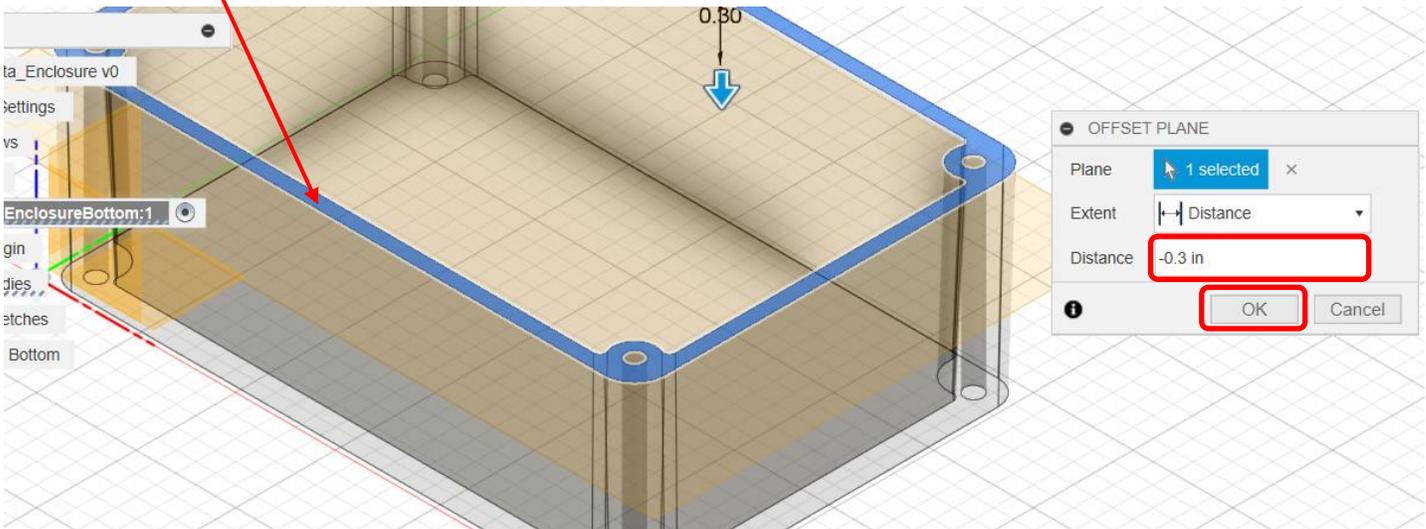


Adding Draft to Holes

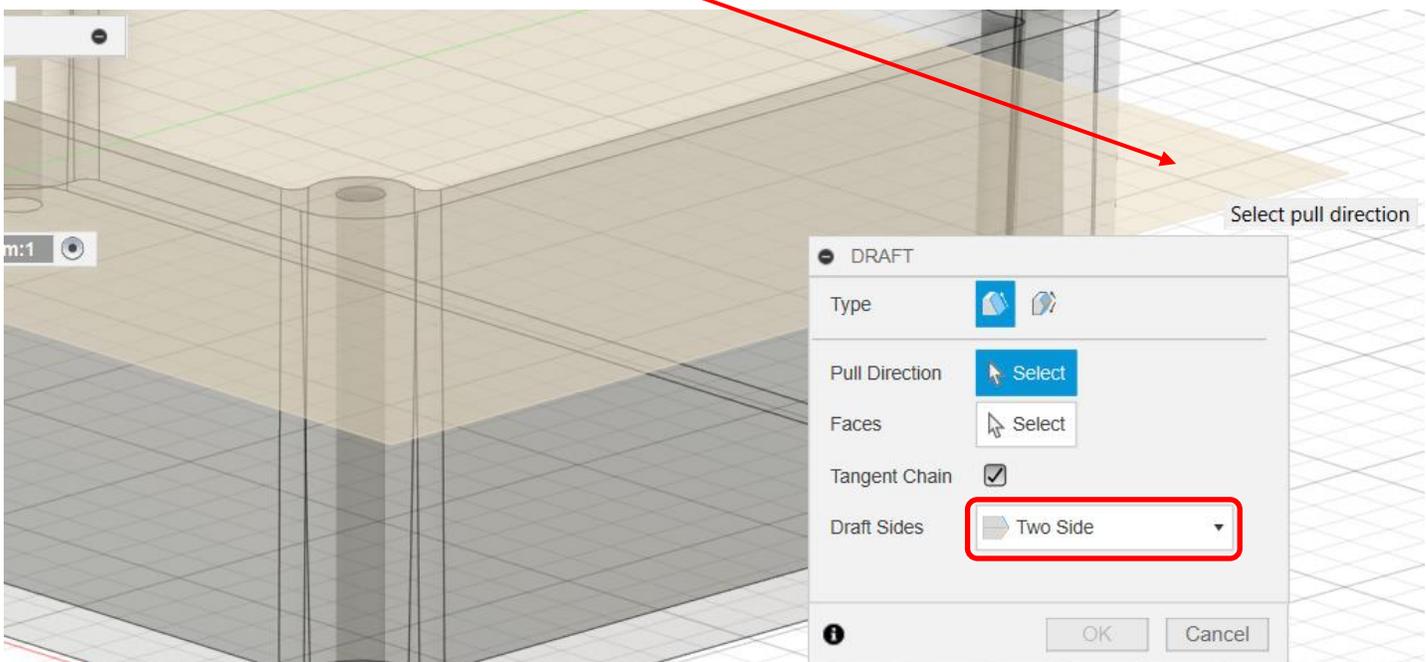
- from the **CONSTRUCT** menu select **Offset Plane**



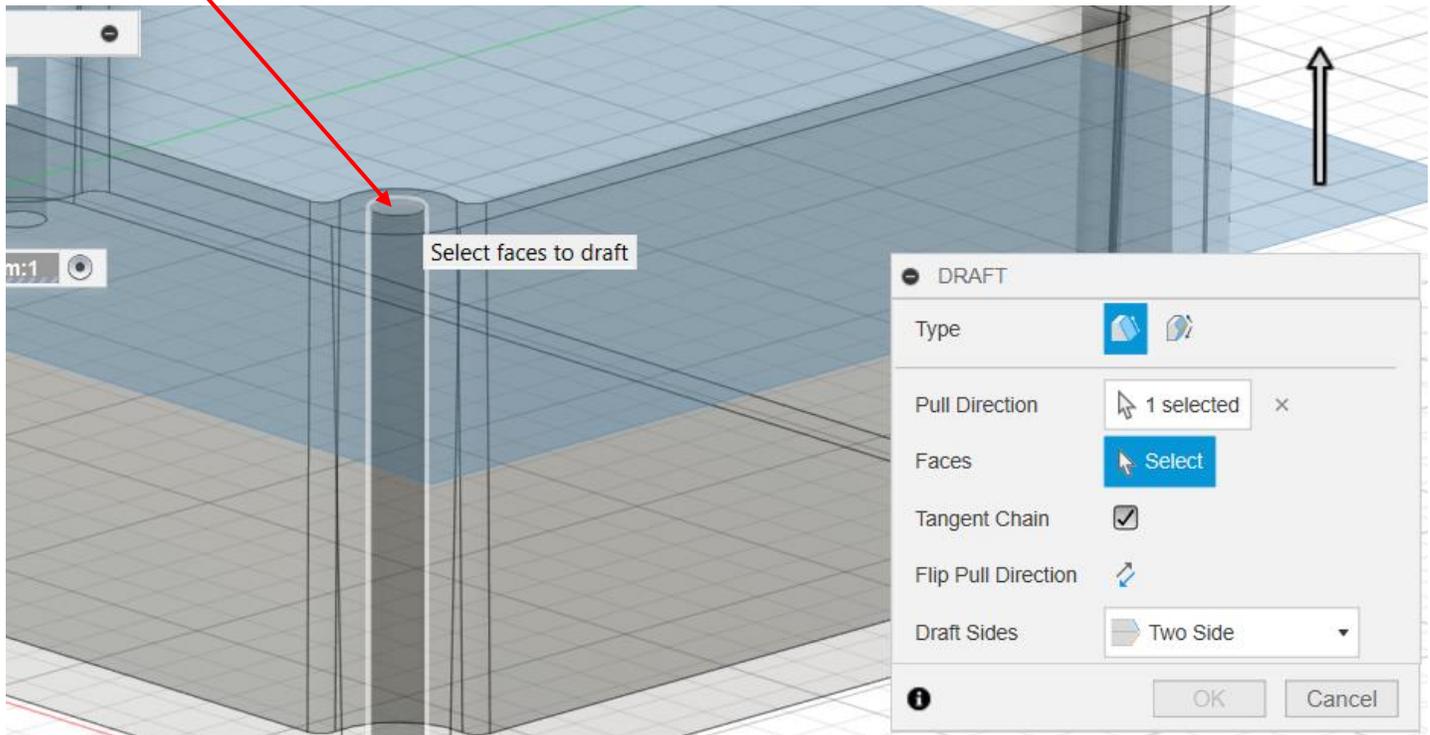
- click on the **top surface** and enter a value of **-0.3** (note the minus sign) for **Distance** and click **OK**



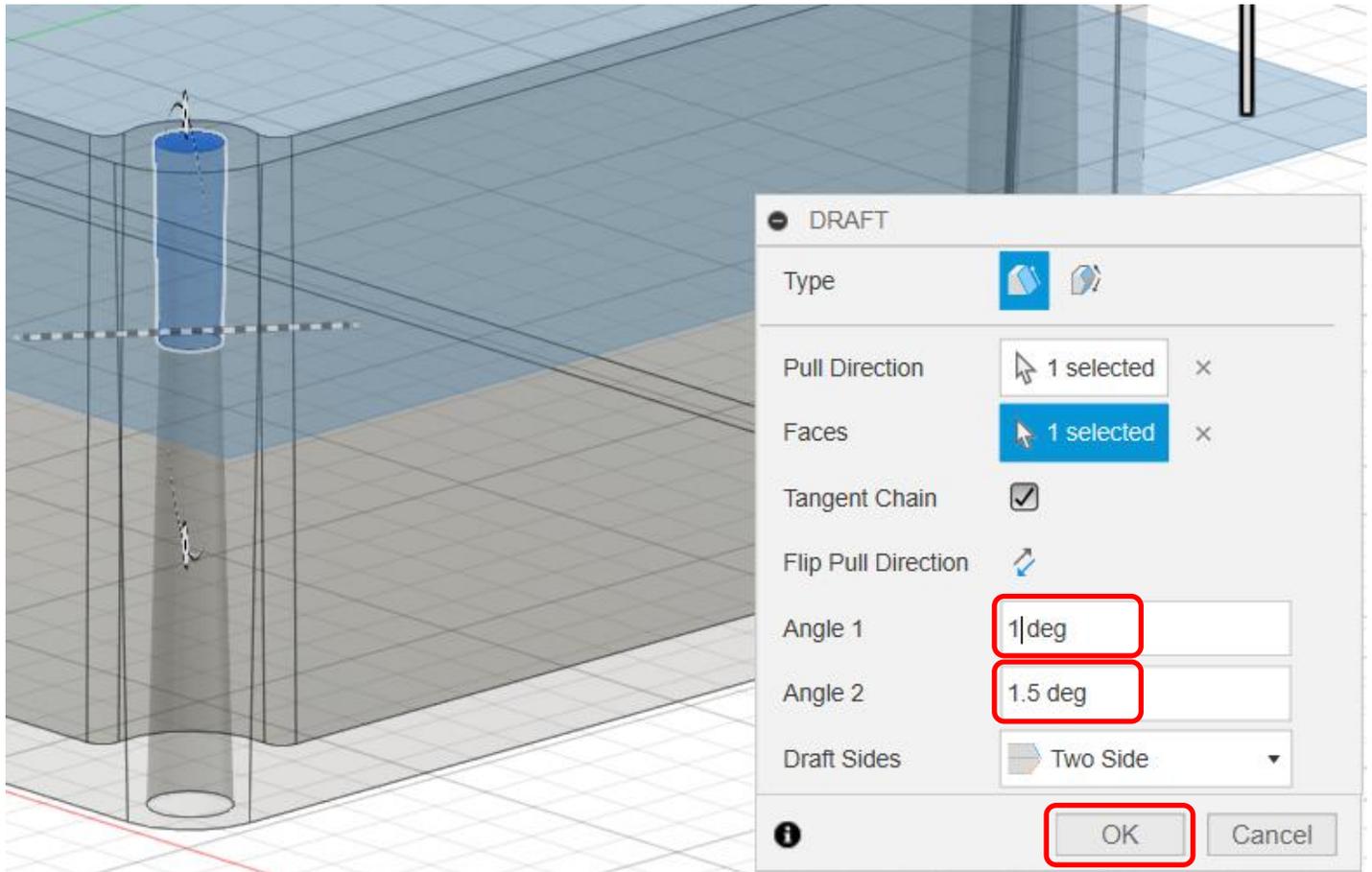
- click on the **Construction Plane** just created
- change the **Draft Sides** to **Two Sides**



- 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**

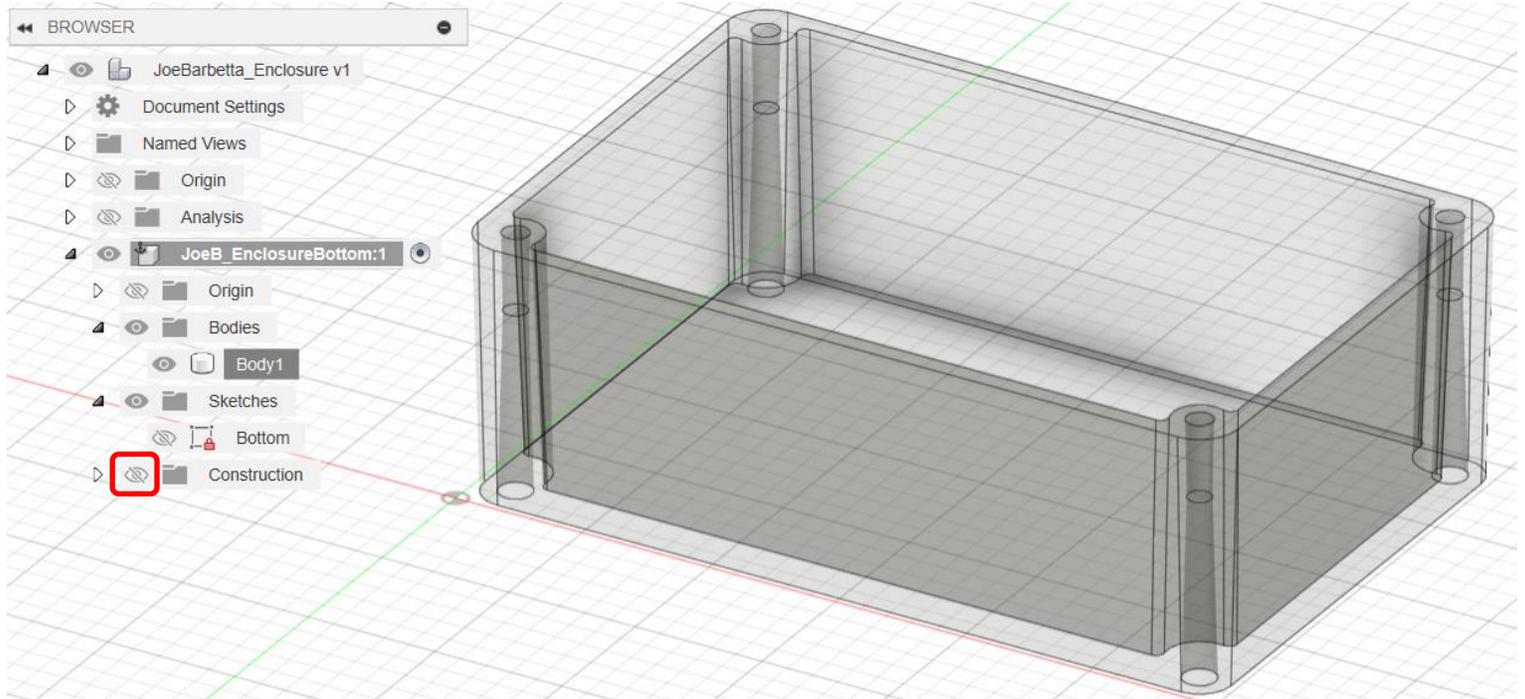


- perform the same operation on the **other 3 holes**. The Draft tool should remember the 2 angles each time, but verify this when drafting the other holes. You may have to reselect **Two Side** for **Draft Sides** each time.

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

- click the **eye** icon next to the **Construction** folder to hide the Construction plane

- right-click on **Body1** 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 assignment could be optimized further, however, for simplicity it is being limited to the bottom tapered hole.

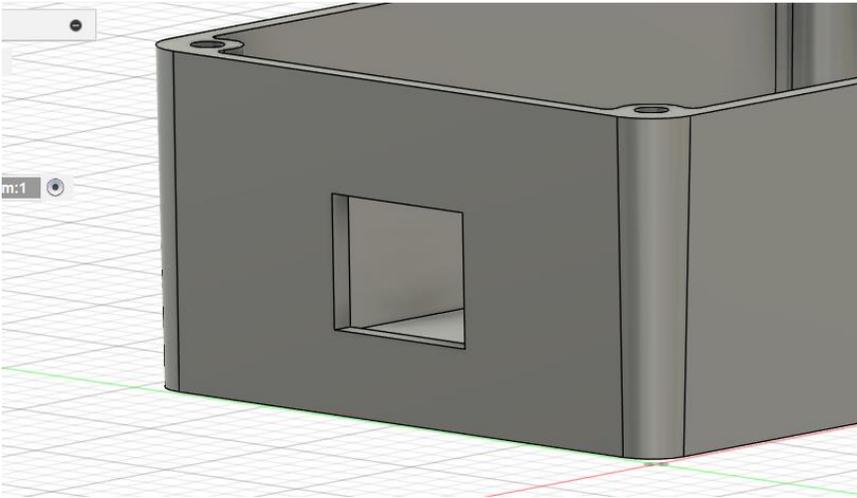
Sink marks

Adding Side Holes

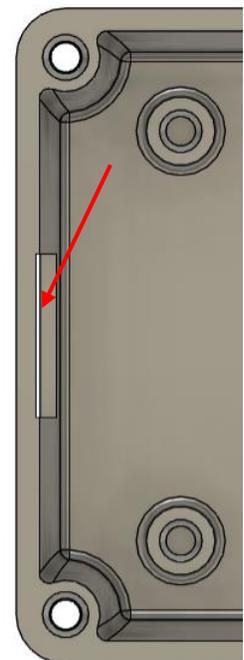
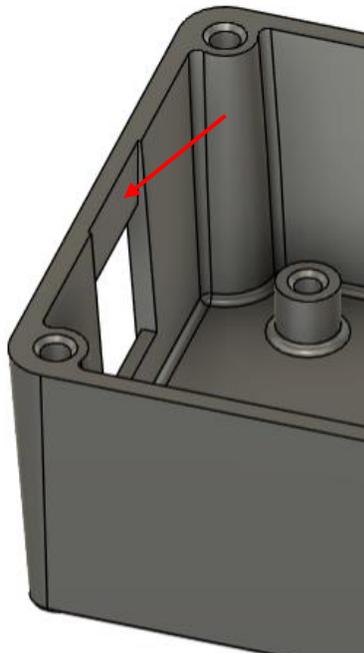
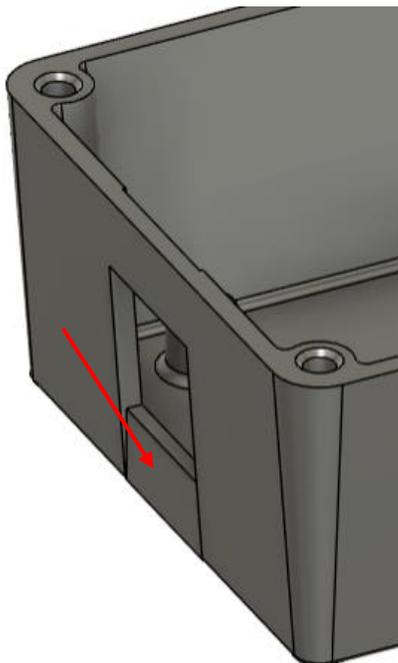
A common requirement for an enclosure are holes for connectors, for example a 0.500 x 0.500 in hole as shown below.

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. If 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.

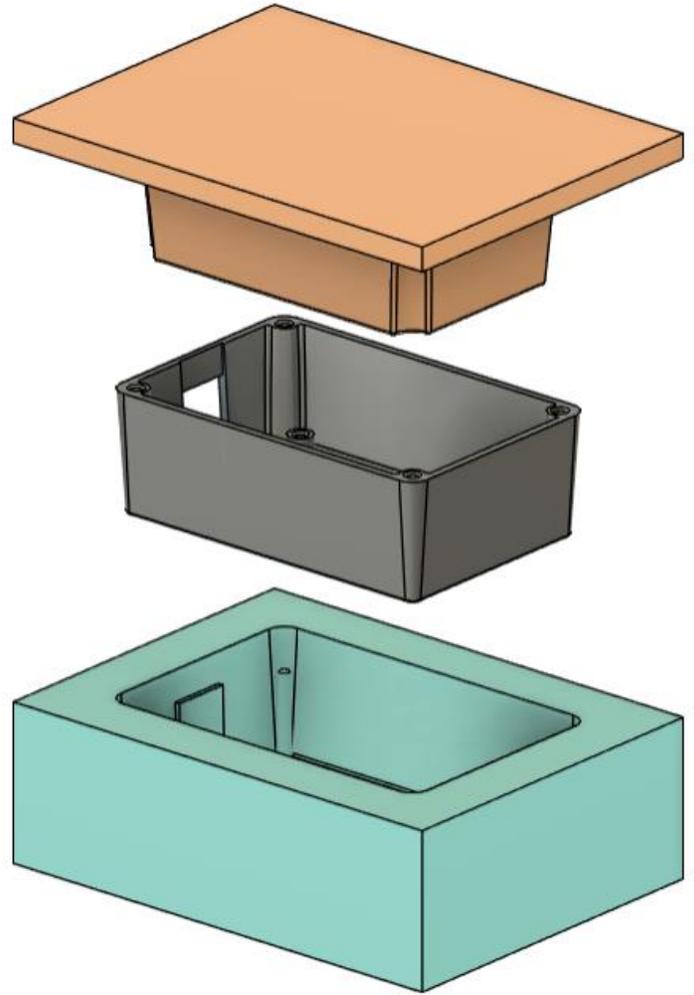
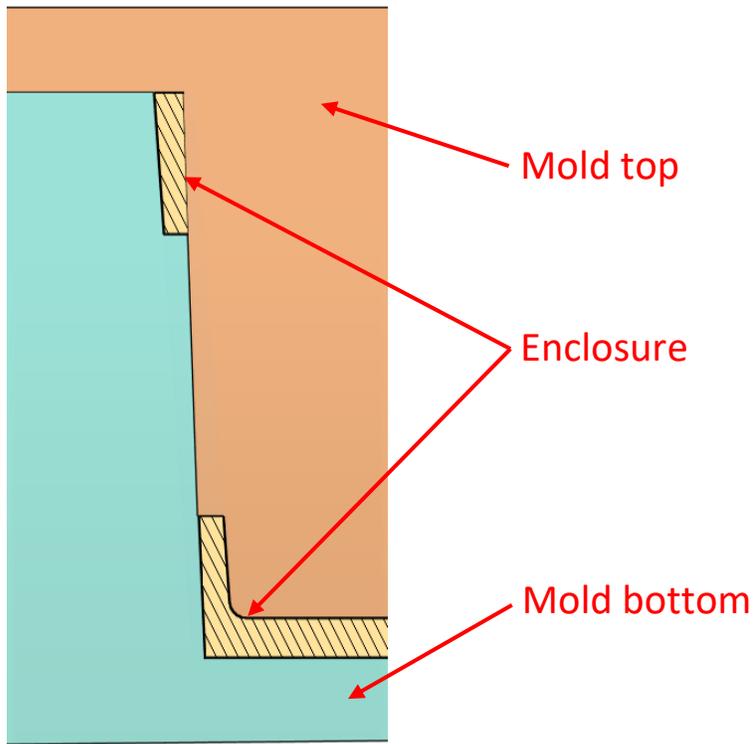


One method to avoid the need for side actions is to apply a **larger draft** on walls where there will be a side hole. The below pictures show a **4 degree draft**. One then removes material below the hole on the outside wall and also remove material above the hole on the interior wall. The 3rd picture 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.

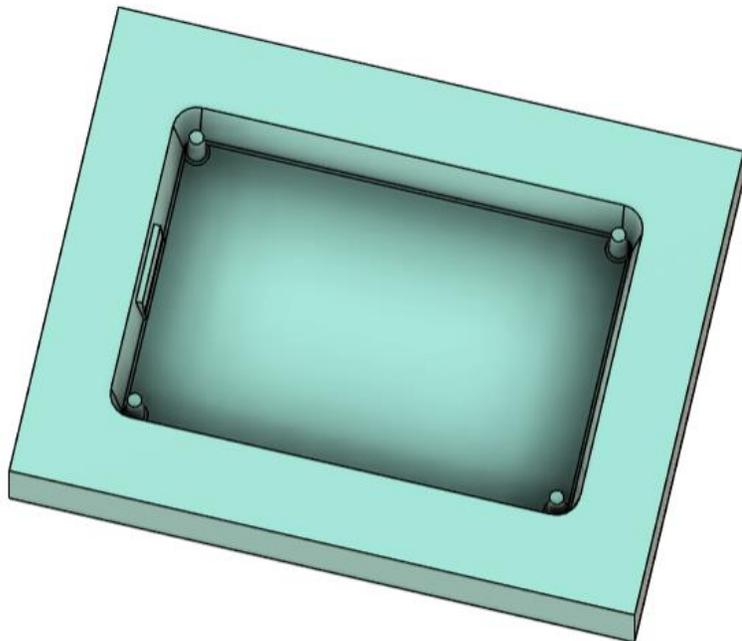


The mold halves shown here are simplified and were created to illustrate the shape around the cavity. A real mold would be larger and would include pins, gating, ejector pins, etc.

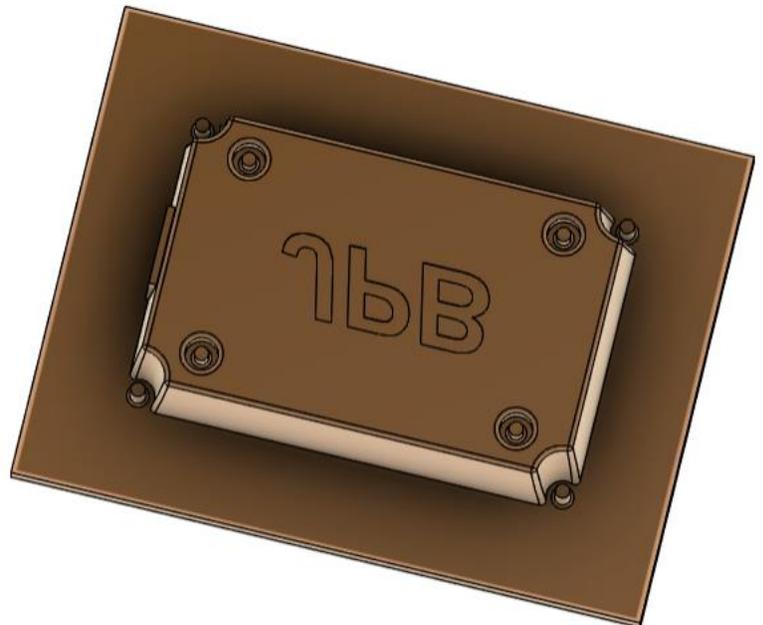
This is a cross-section of the hole and mold.

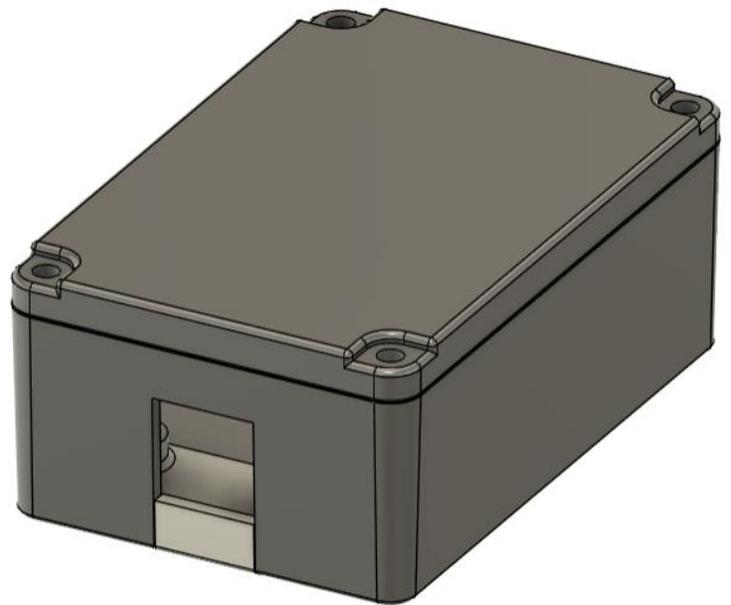
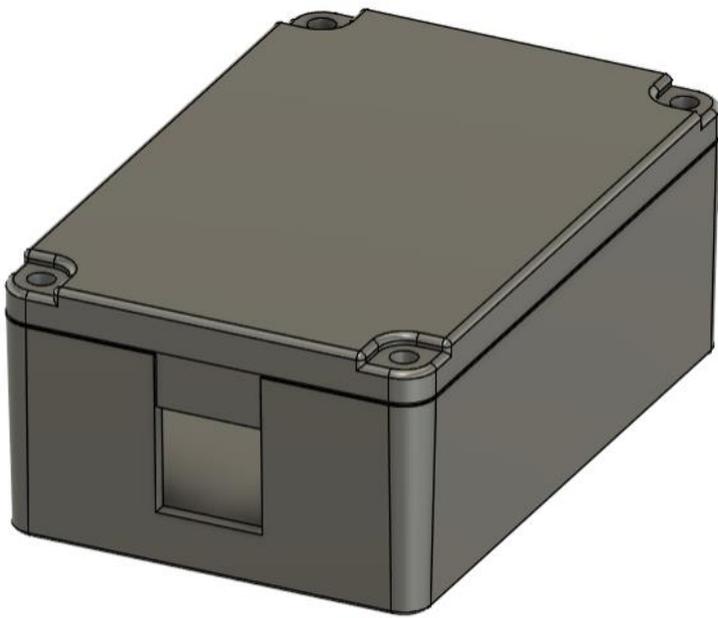


Mold bottom



Mold top

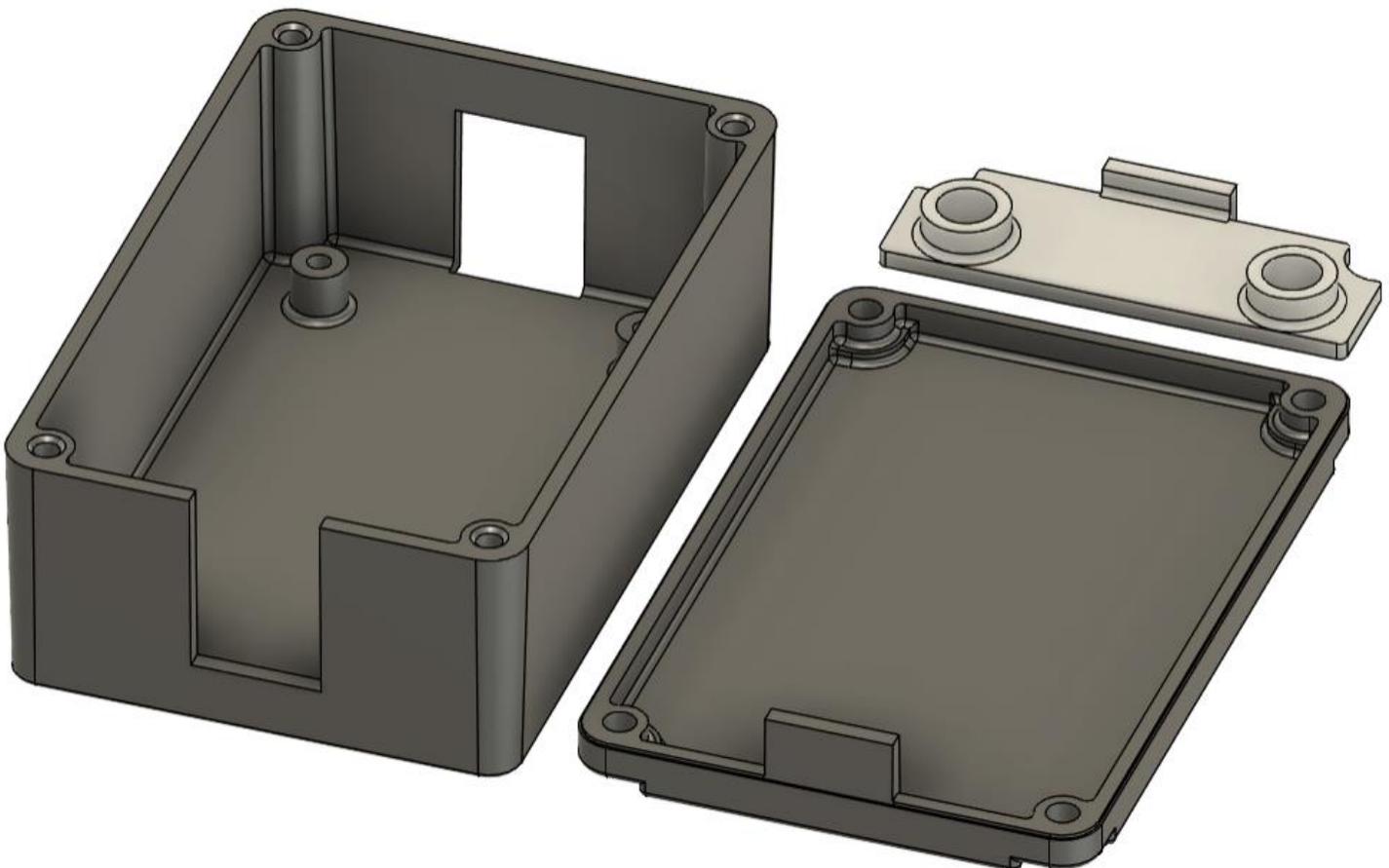
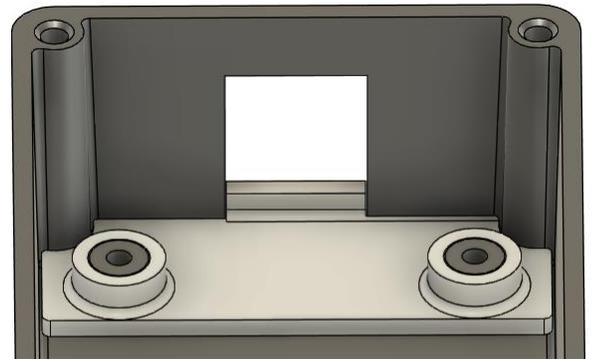




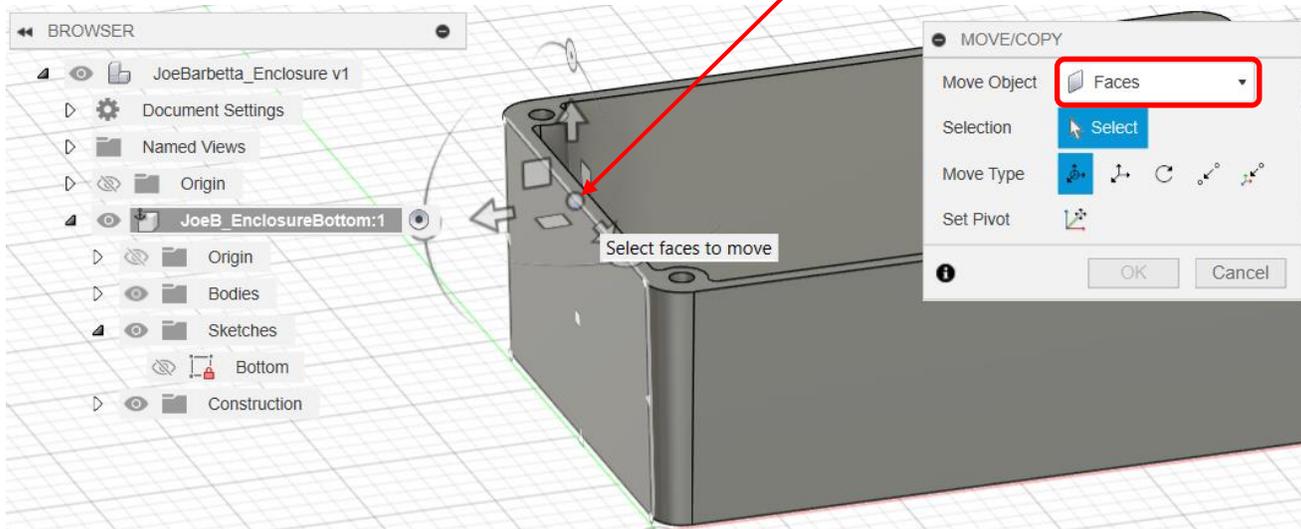
The above pictures shows the enclosure using 2 of the methods introduced.

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

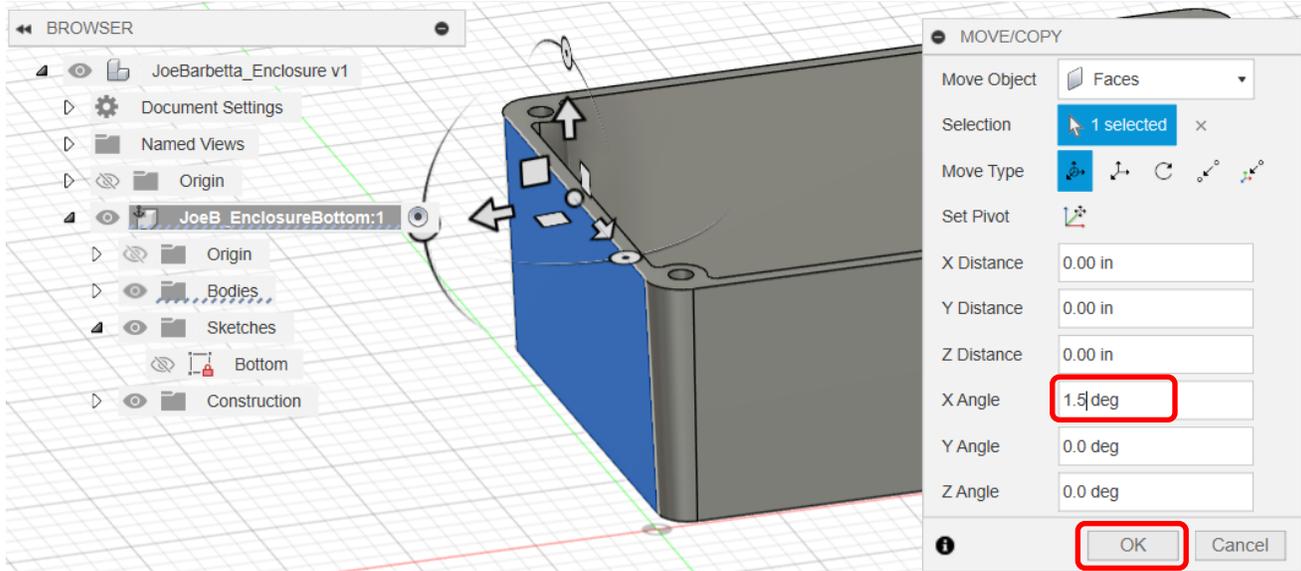
The right picture above using an extra insert part, shown on the right. It is common to have a **family mold**, which will produce both the bottom shell and top cover. This insert would just be another cavity in the mold.



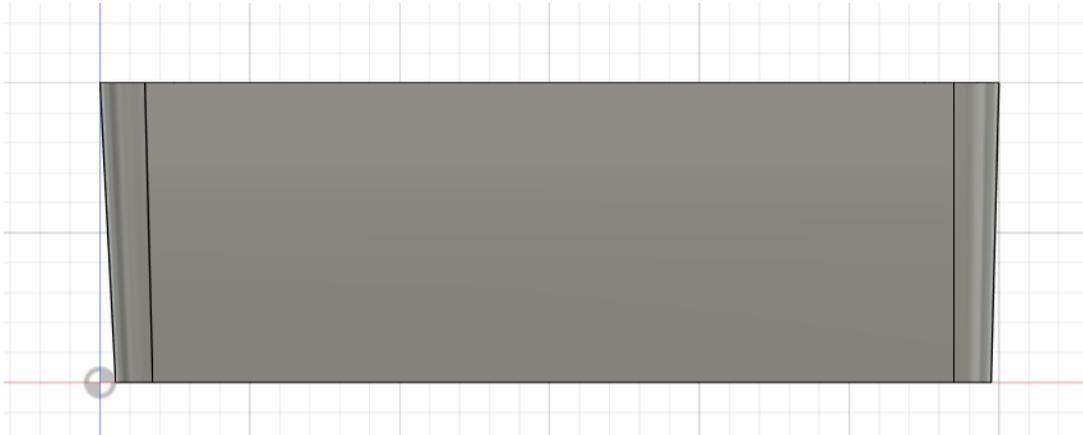
- rotate the view to access the **exterior surface of the same wall**
- select the **Move/Copy** tool again and ensure **Move Object** is set the **Faces**
- click on the exterior face and move the **pivot center up to the top edge**, as shown below



- set the **X Angle** to **1.5** (No minus sign). The **top edge should not move**, but the **bottom edge should over out slightly**. As mentioned previously, if a different Angle value was adjusted for the interior wall, the same should be adjusted here.
- click **OK**



A front view of the enclosure should show that the one wall now has a different angle.

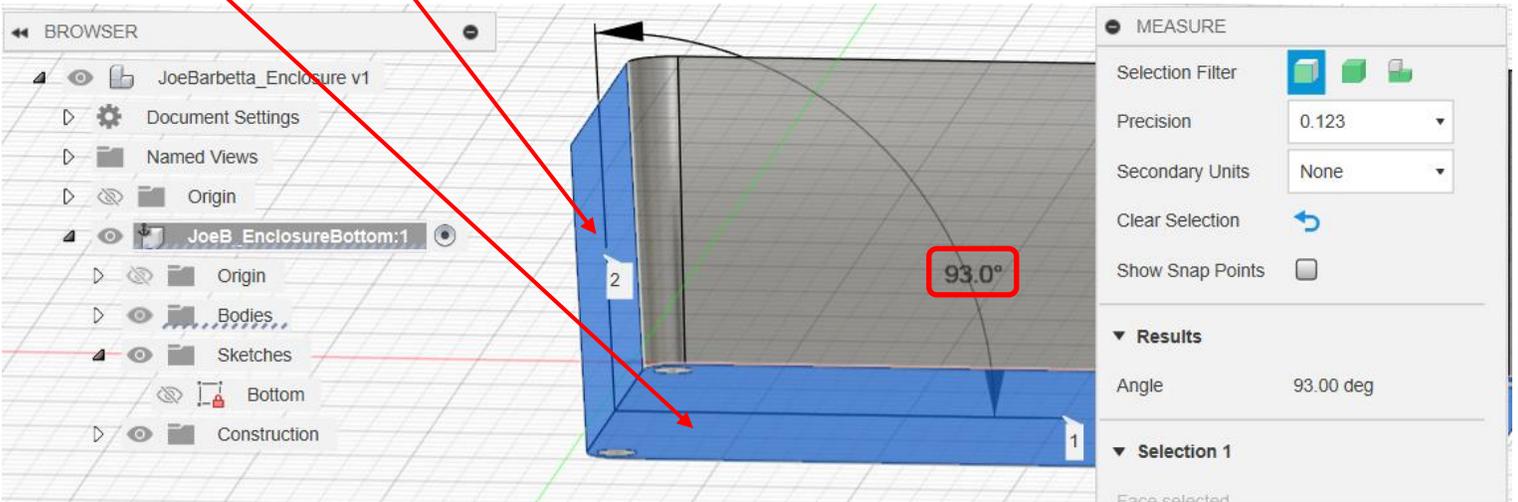


Creating the Side Hole – Measuring Draft

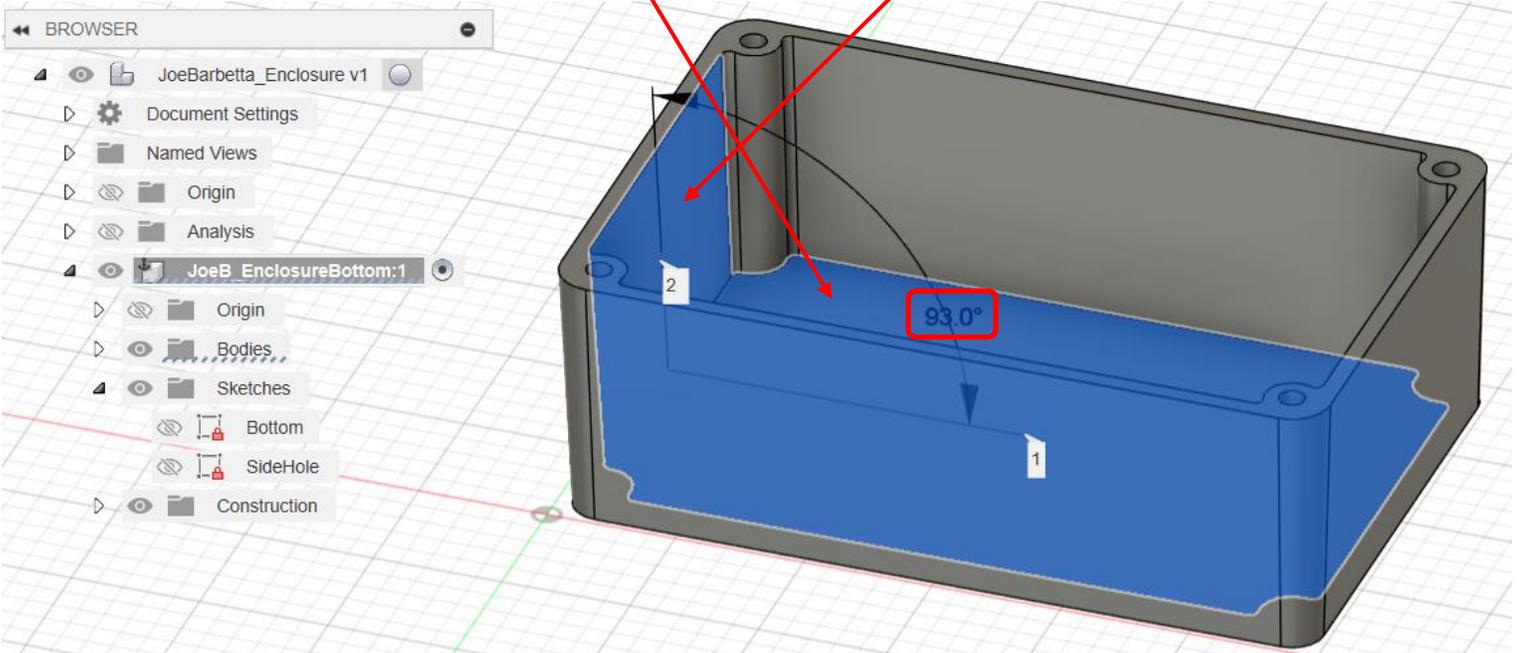
- from the top **INSPECT** menu, select **Measure**



- click on **bottom** and then **side wall** to view the angle. **93** indicates a **3 degree draft**, which is twice that of the other walls.
- click **Close** on the **MEASURE** window

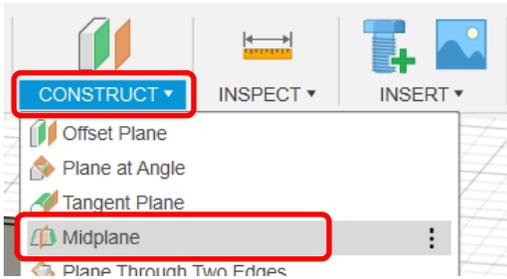


- select the **Measure** tool again and click on the **bottom** and then the same **side wall interior**. It should also indicate a 93 degree angle.

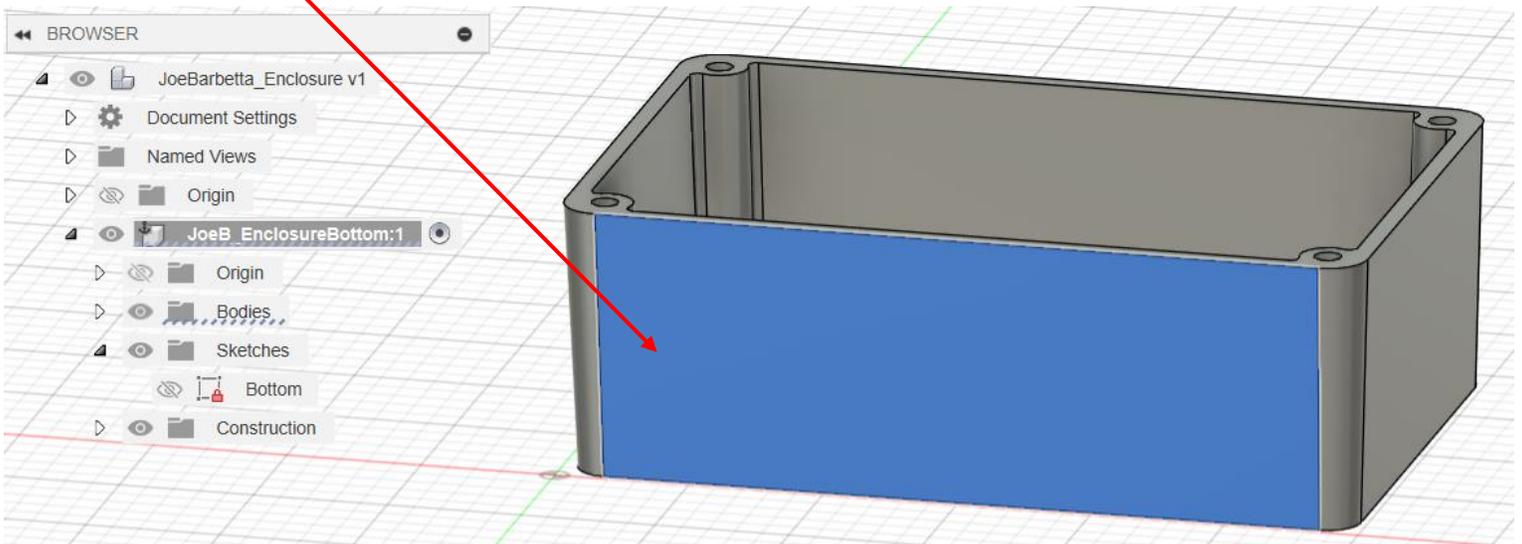


Creating the Side Hole – Creating a Construction Midplane

- from the top **CONSTRUCT** menu, select **Midplane**

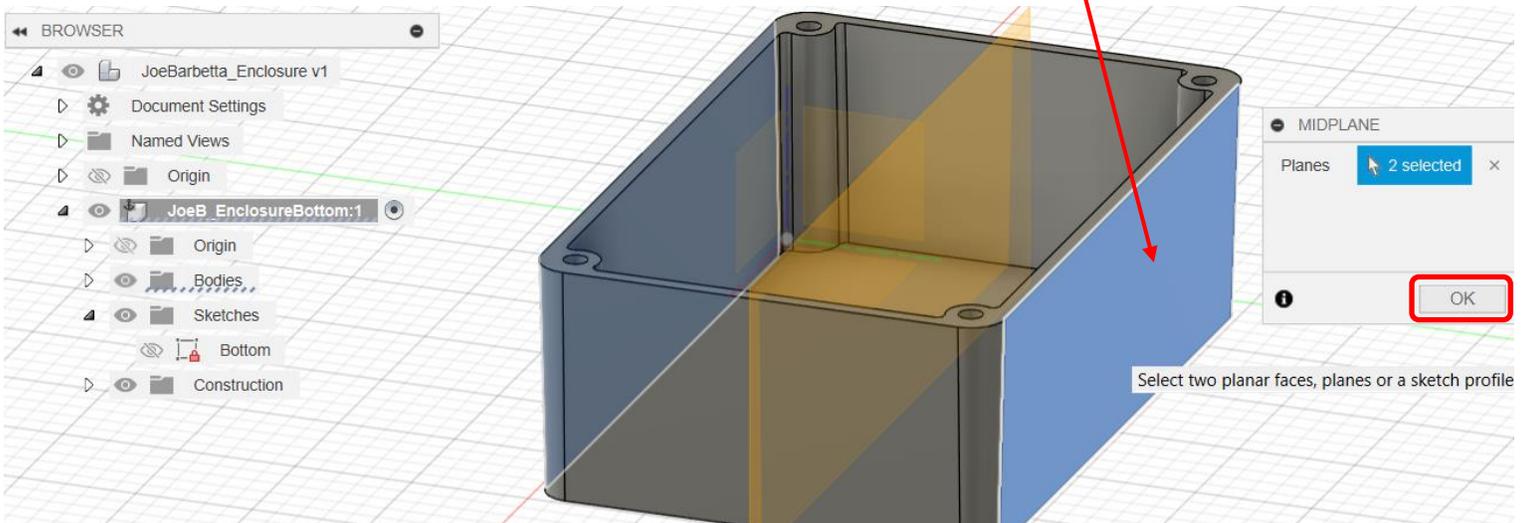


- click on an **exterior wall face** as shown



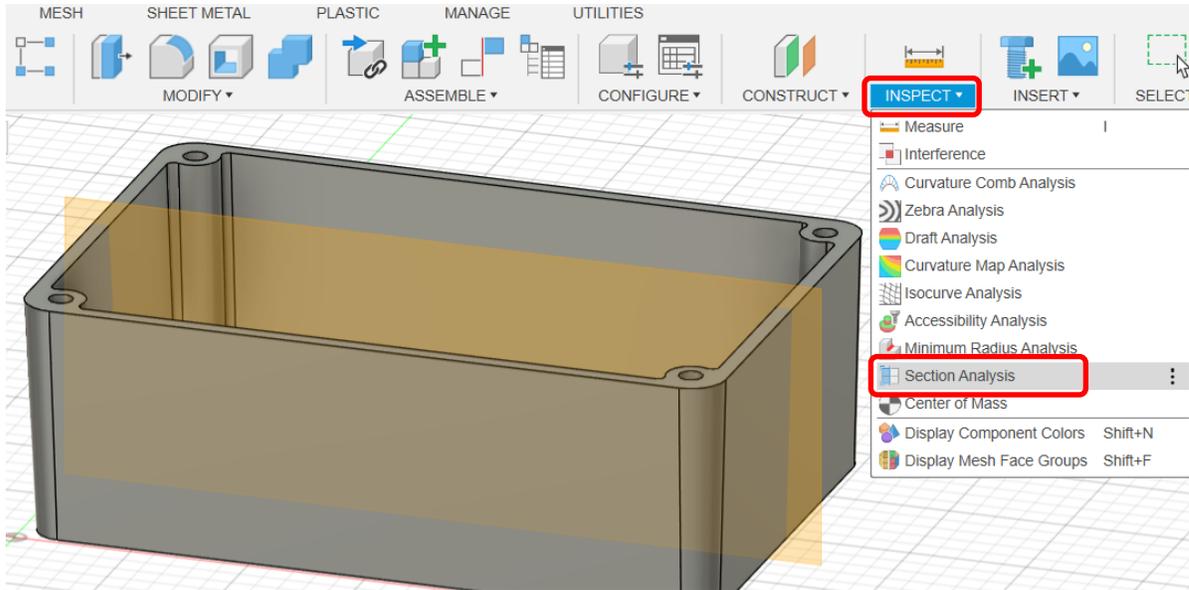
- adjust the view to access the exterior of the opposite wall, as shown below and **click on that face**, which should result in a plane showing in the center of the enclosure.

- click **OK**

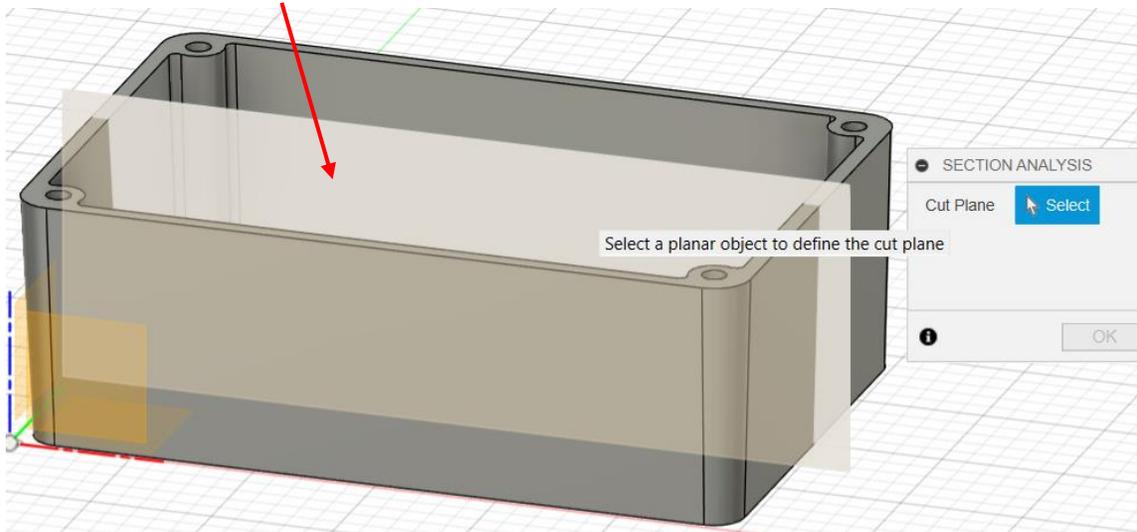


Creating the Side Hole – Creating a Section Analysis

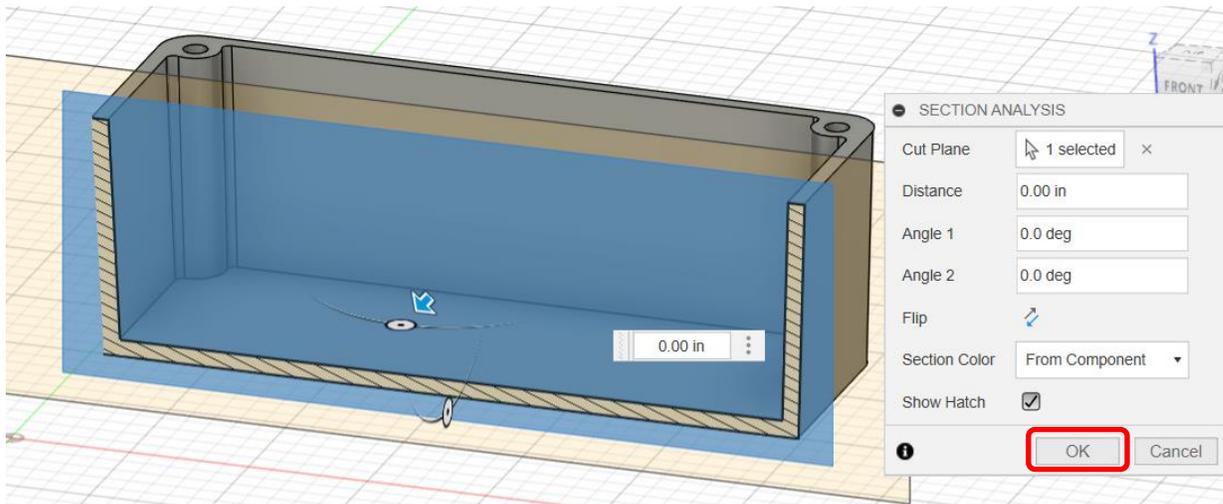
- from the **INSPECT** menu select **Section Analysis**



- click on the **Construction Plane**

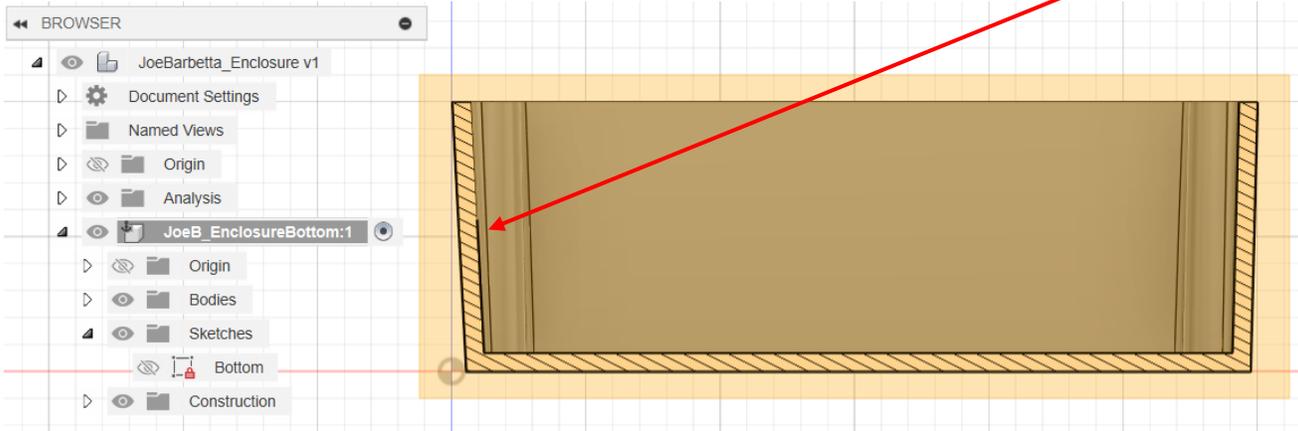


- click **OK**

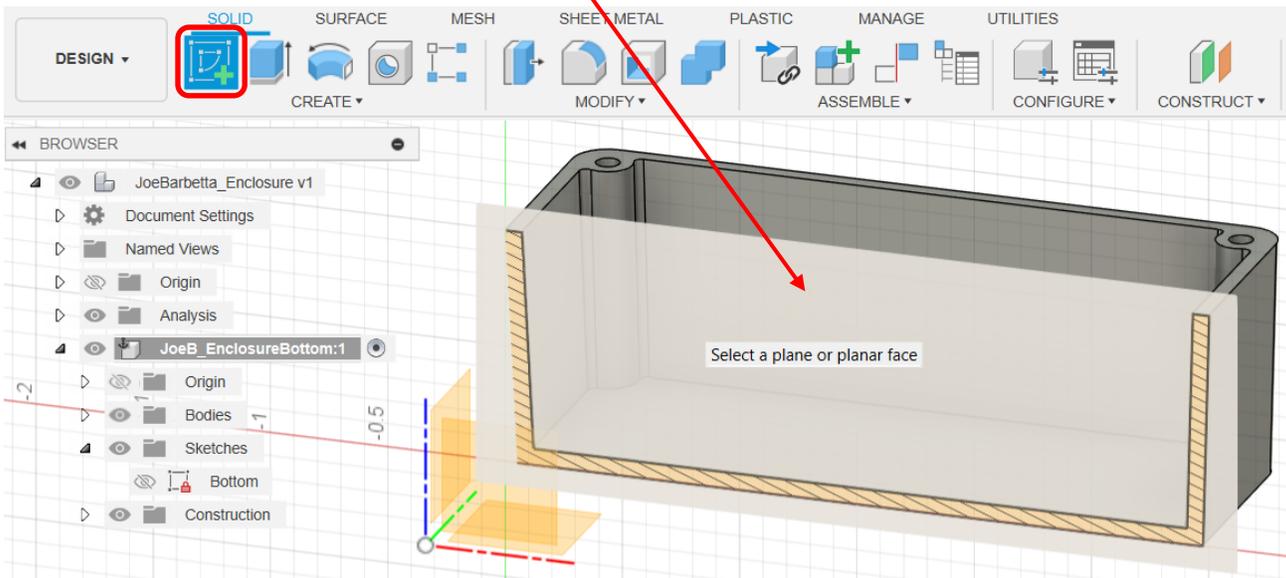


Creating the Side Hole – Creating a Cross-section Sketch

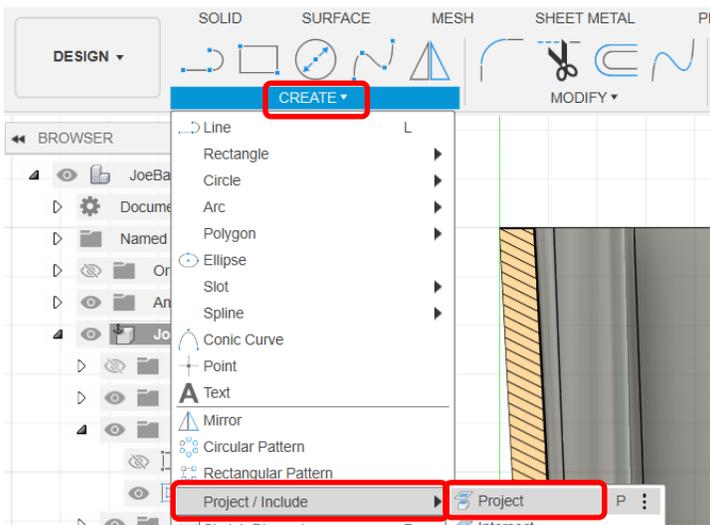
Note that when looking at the **Construction Plane** before creating a Sketch, the wall with the **larger draft angle** should be on the left.



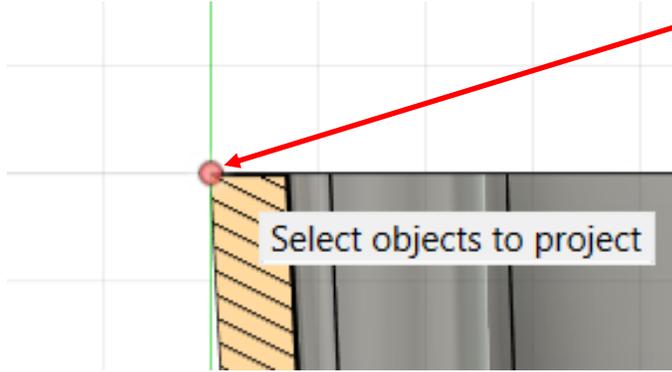
- select **Create Sketch** and click on the **Construction Plane**



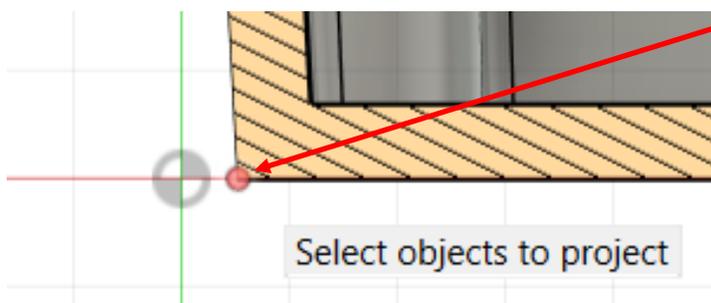
- from the **CREATE** menu, select **Project / Include** and **Project**



- zoom into the top left of the cross-section and click on the **top left point**

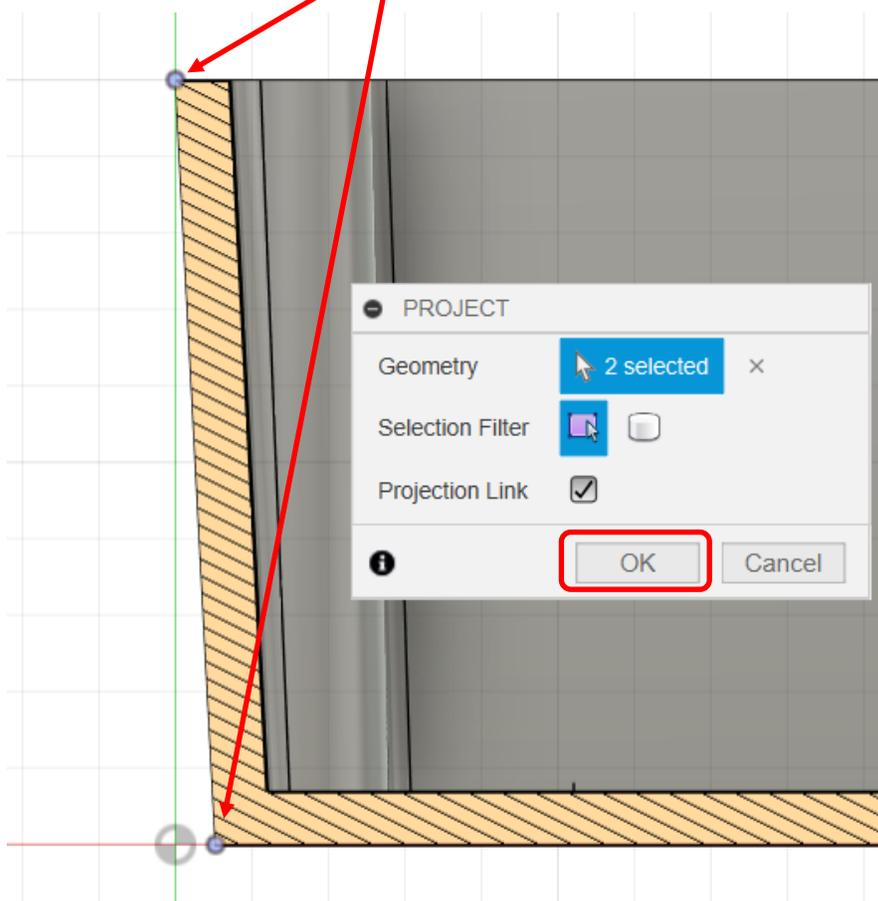


- zoom into the bottom left of the cross-section and click on the **bottom left point**

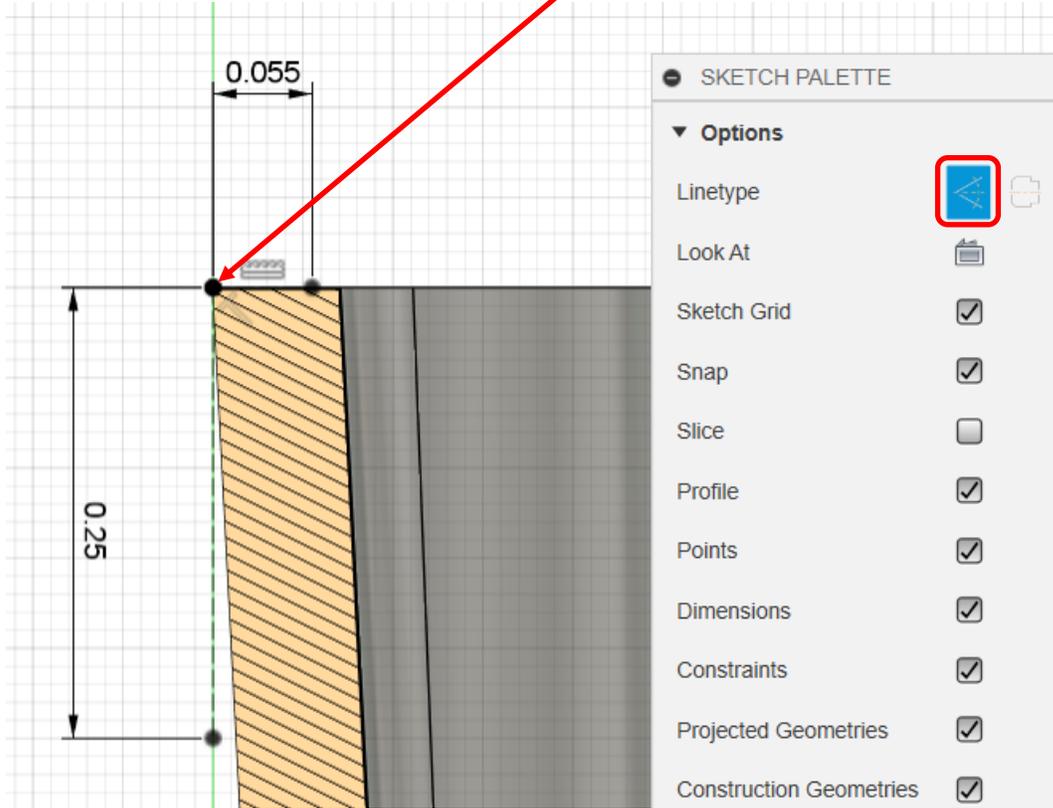


Note the top and bottom **Projection Dots** that were just created.

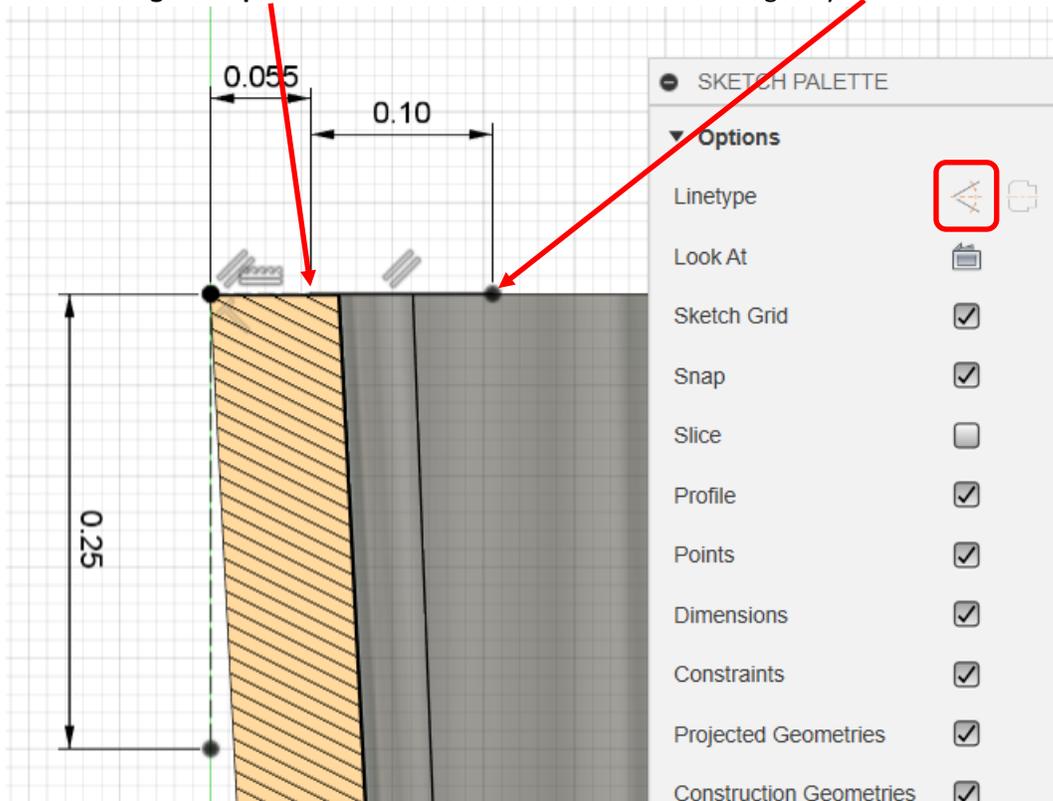
- click **OK**



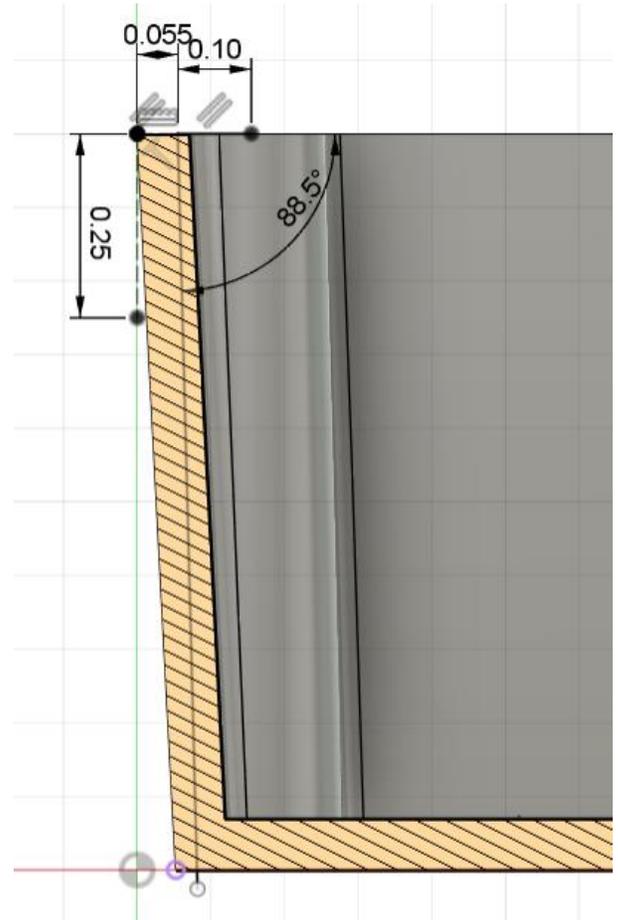
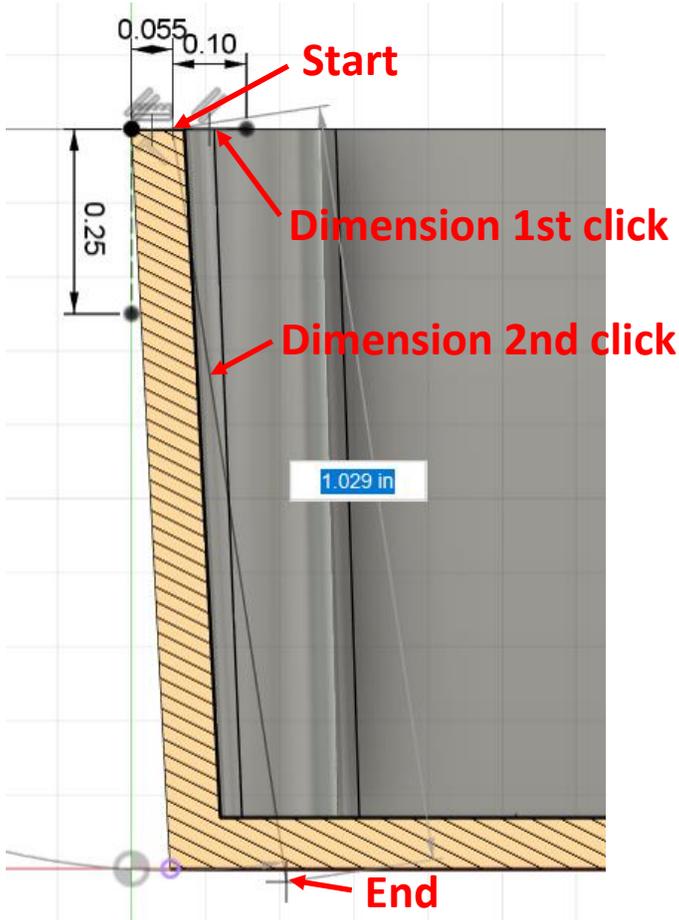
- click on the **Construction** line icon to highlight it blue
- select the **Line** tool and create a line from the **top left point** to the right by **0.055**
- select the **Line** tool and create a line from the same **top left point** down by **0.25**



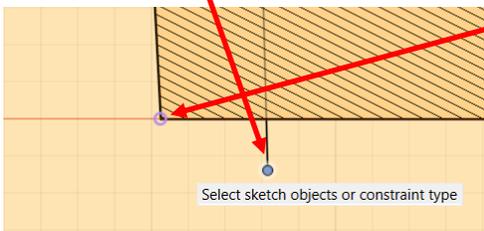
- click on the **Construction** line icon to remove the highlighting
- from the **right endpoint** of the **0.055** line create a line to the right by **0.10**



- start a line at the **right endpoint of the 0.055 line**
- extend the line down and slightly to the right to a position **just slightly under the enclosure** as shown. The position is not critical and it is ok if a value other than 1.029 shows.
- from the bottom of the **CREATE** menu, select **Sketch Dimension**
- click on the **top horizontal line** and then the **diagonal line** and enter **88.5**. The result is that of the right picture.



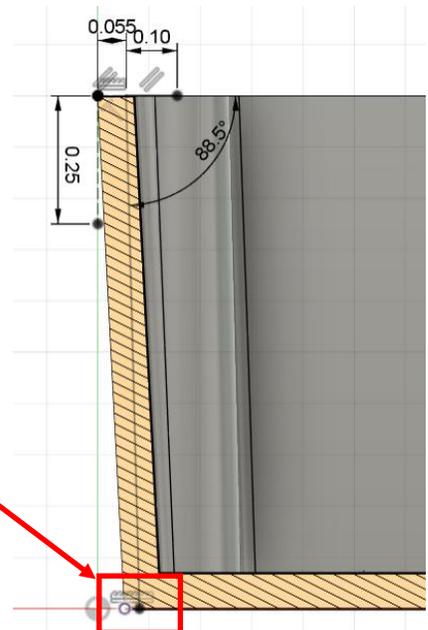
- from the **CONSTRAINTS** menu, select **Horizontal / Vertical**
- click on the **bottom endpoint** of the line just created and the **bottom corner** of the enclosure and press the **Esc** key



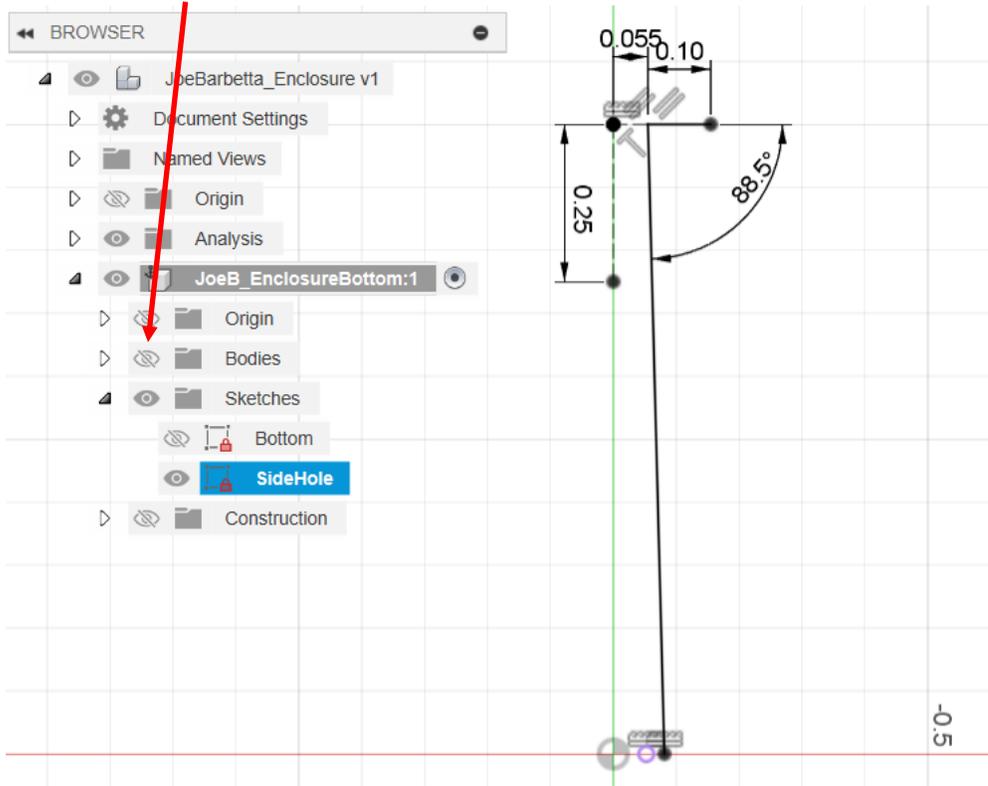
Zoomed in view of bottom



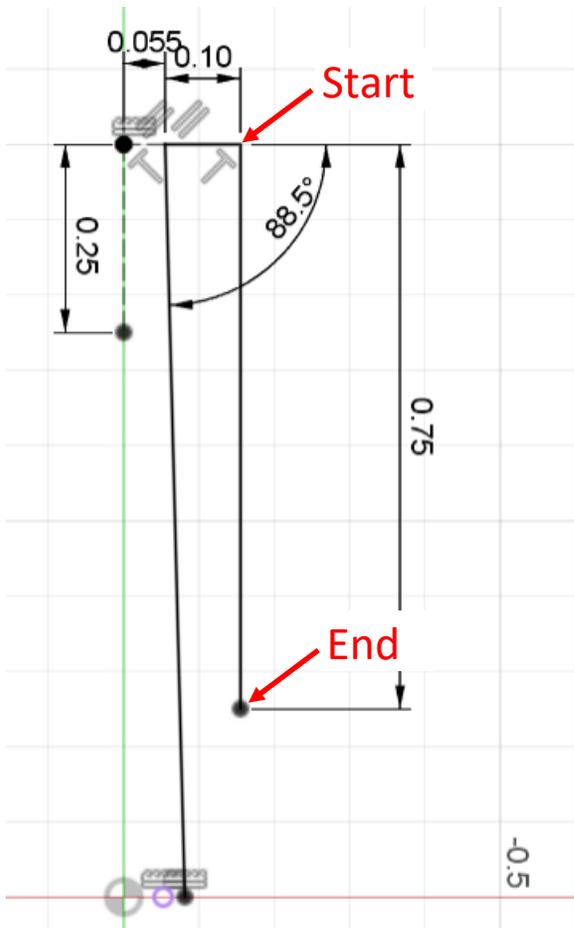
After Constraints applied



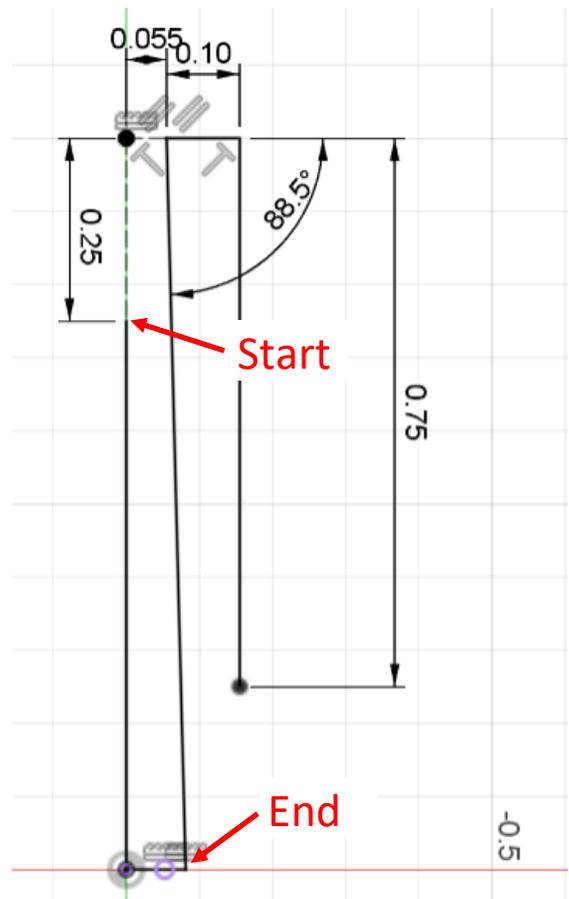
- click on the **eye** icon to hide the enclosure bottom. This will make Sketch lines more visible.



- use the **Line** tool again to start a line at the **end of the 0.10 line** and downward with a value of **0.75**

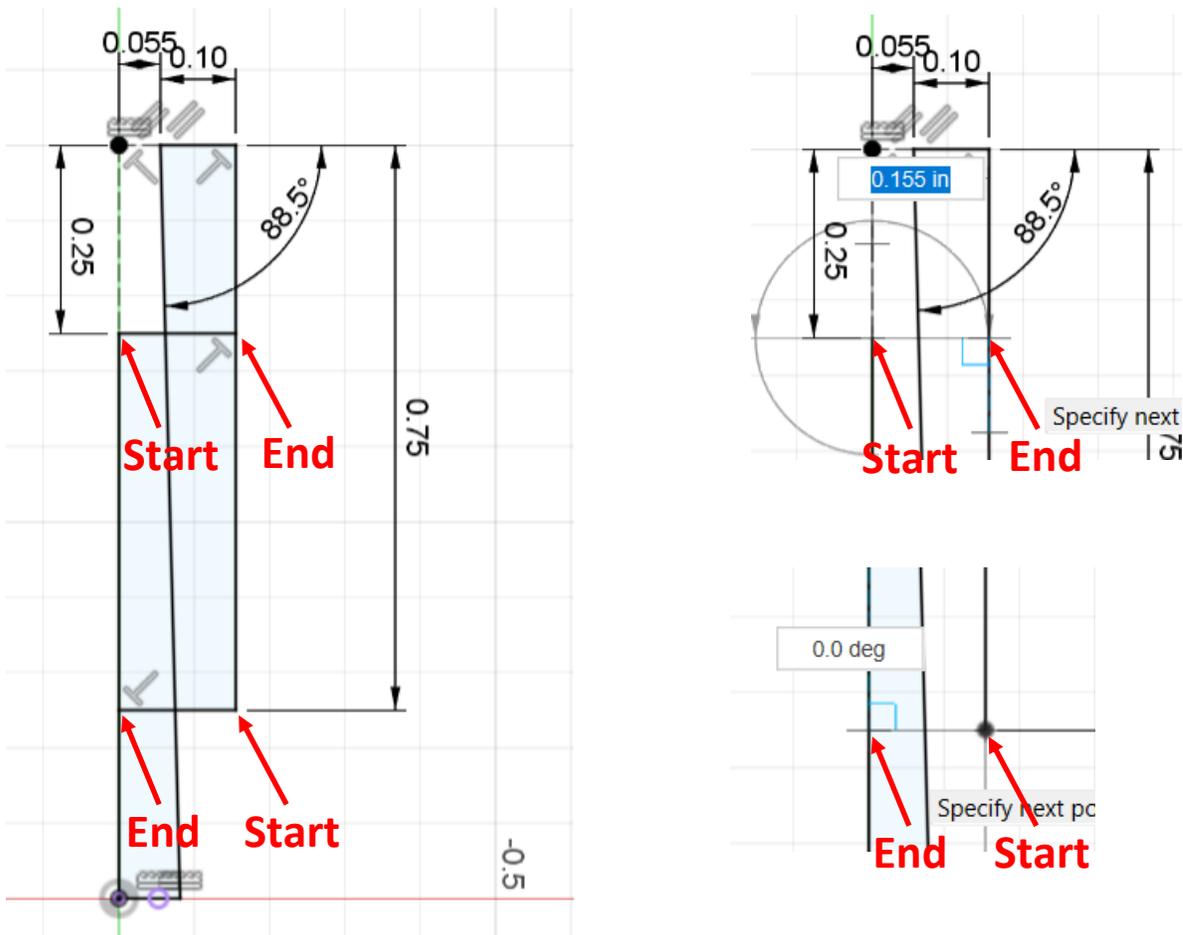


- use the **Line** tool again to start a line at the **end of the 0.25 line** and downward to the Origin and then to the right to meet the diagonal line

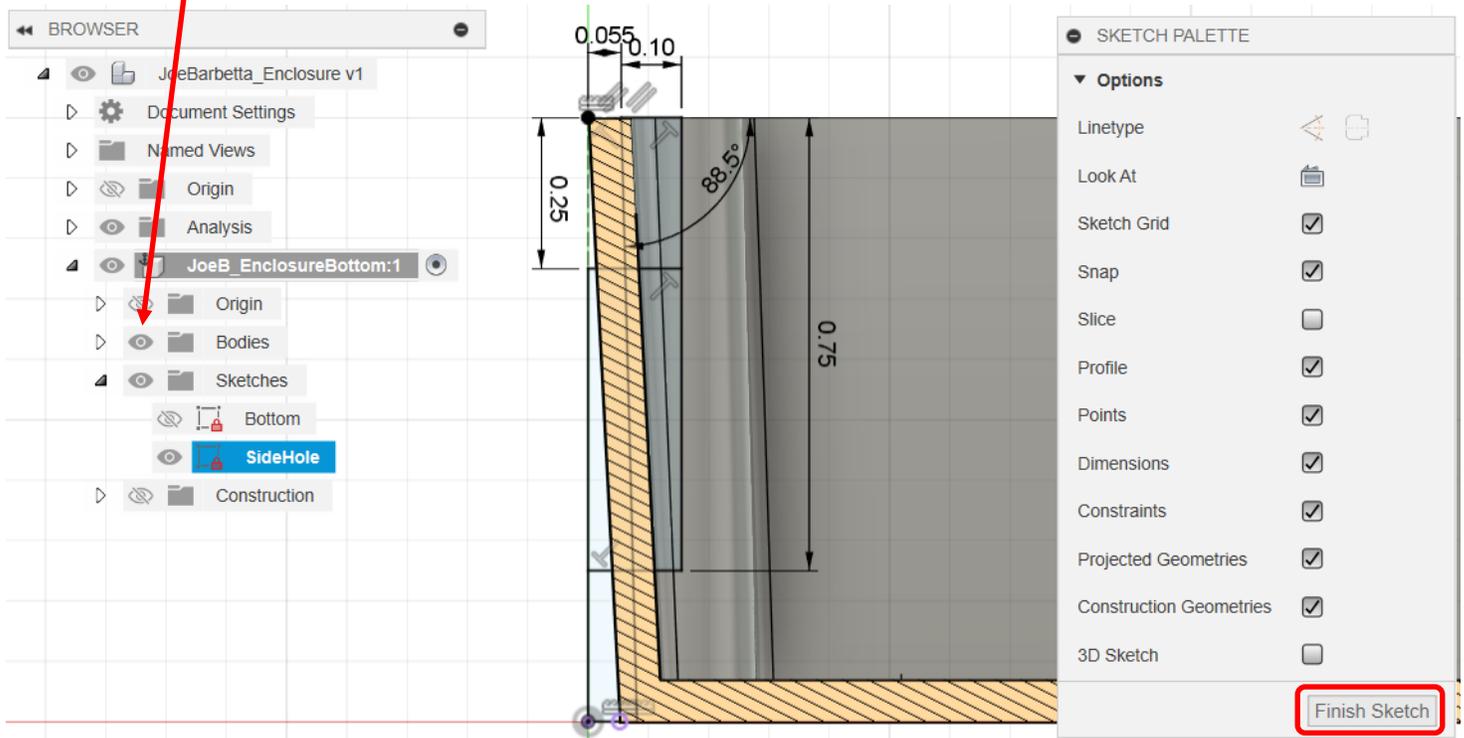


- create a line from the **bottom of the 0.25 line** to the **0.75 line**. As shown on the right, position the end until the blue right-angle symbol shows. After clicking the symbol will become a gray perpendicular symbol.

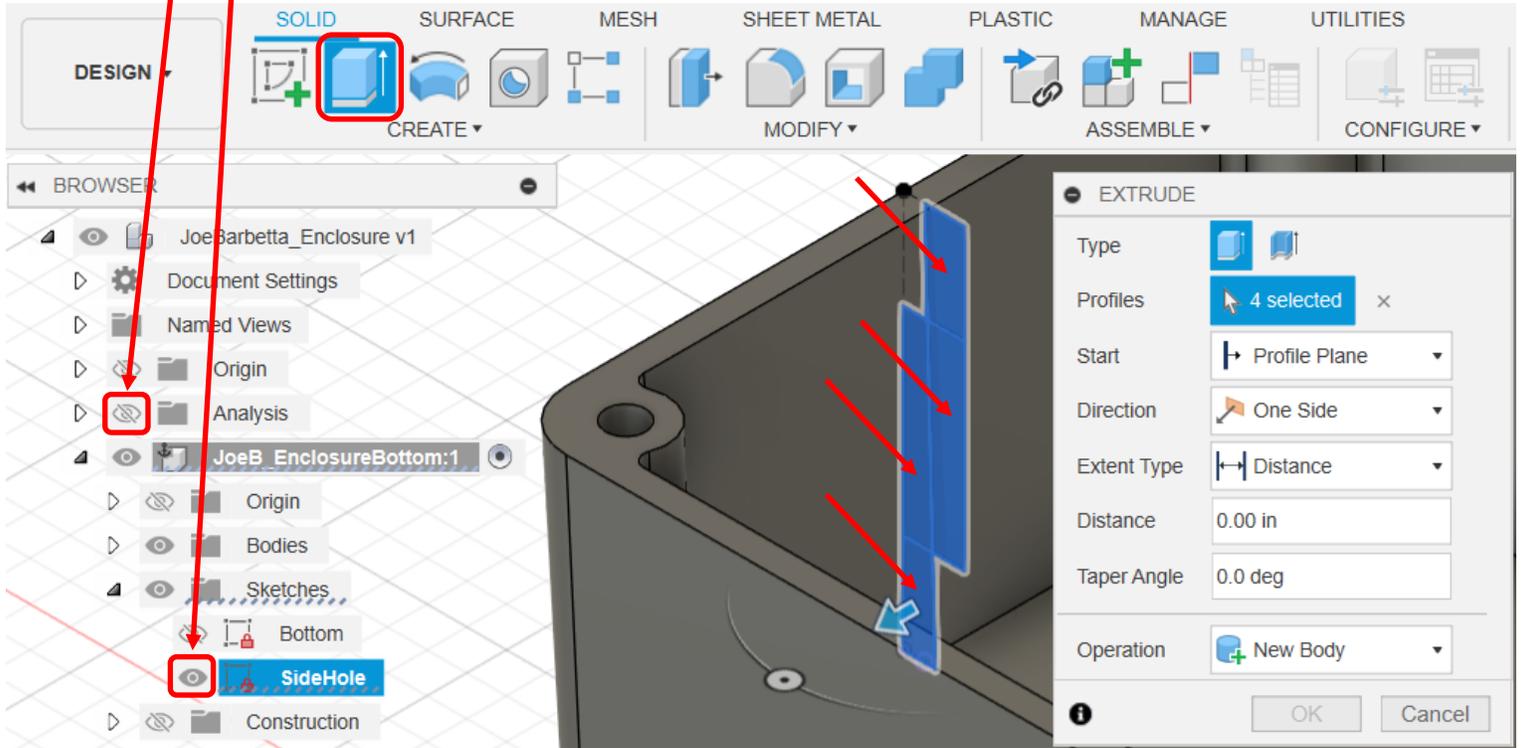
- create a line from the **bottom of the 0.75 line** to the **left vertical line**. As shown on the right, position the end until the blue right-angle symbol shows. After clicking the symbol will become a gray perpendicular symbol.



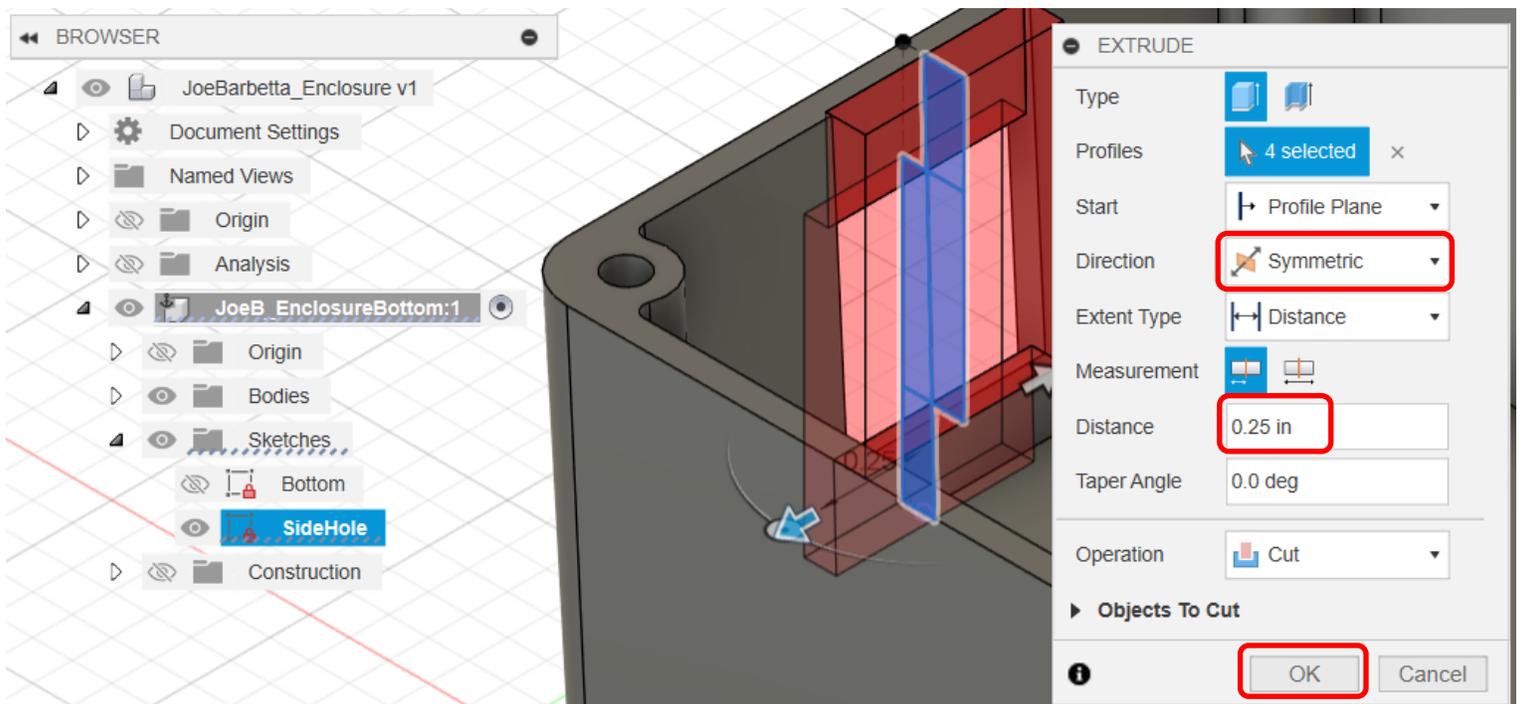
- click on the **eye** icon for **Bodies** to show the enclosure again and click **Finish Sketch**



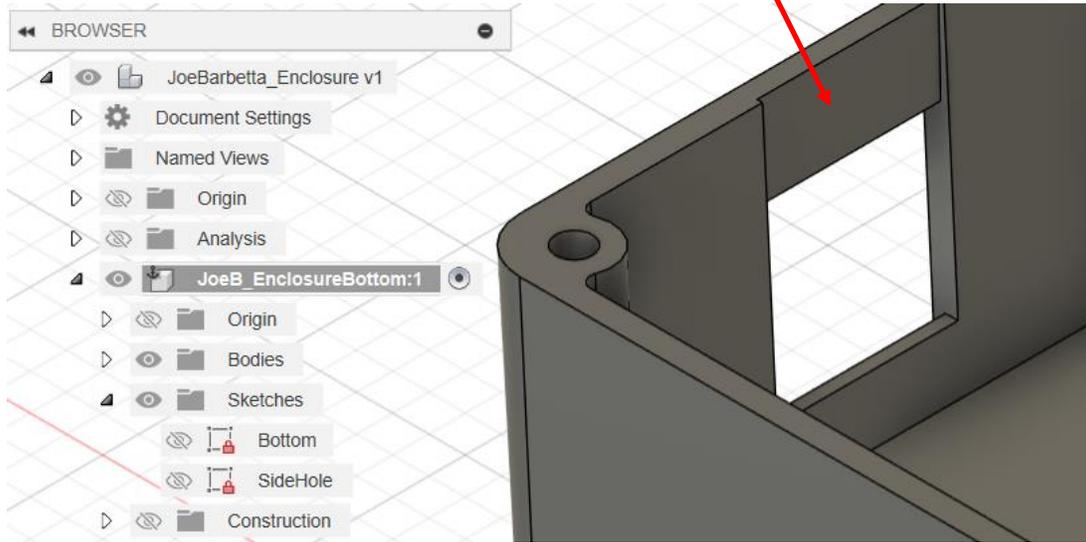
- click on the **eye** icon for **Analysis** to disable the cross-section view
- ensure that the **eye** icon for the **SideHole** Sketch is **active**
- select the **Extrude** tool and click on the **4 regions** to select them



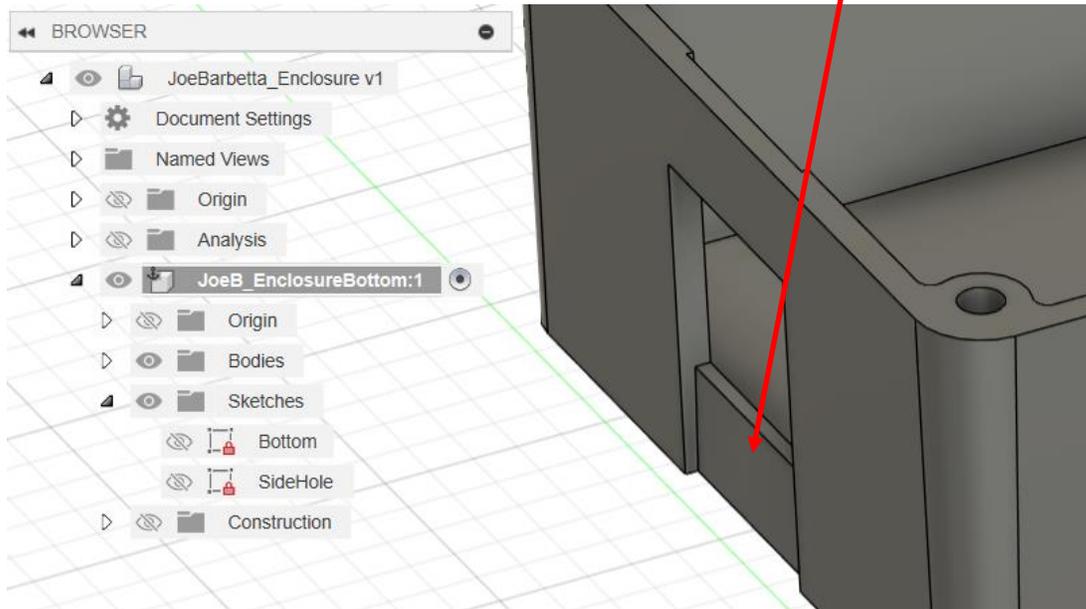
- select **Symmetric** for **Direction**
- enter **0.25** for **Distance**
- click **OK**



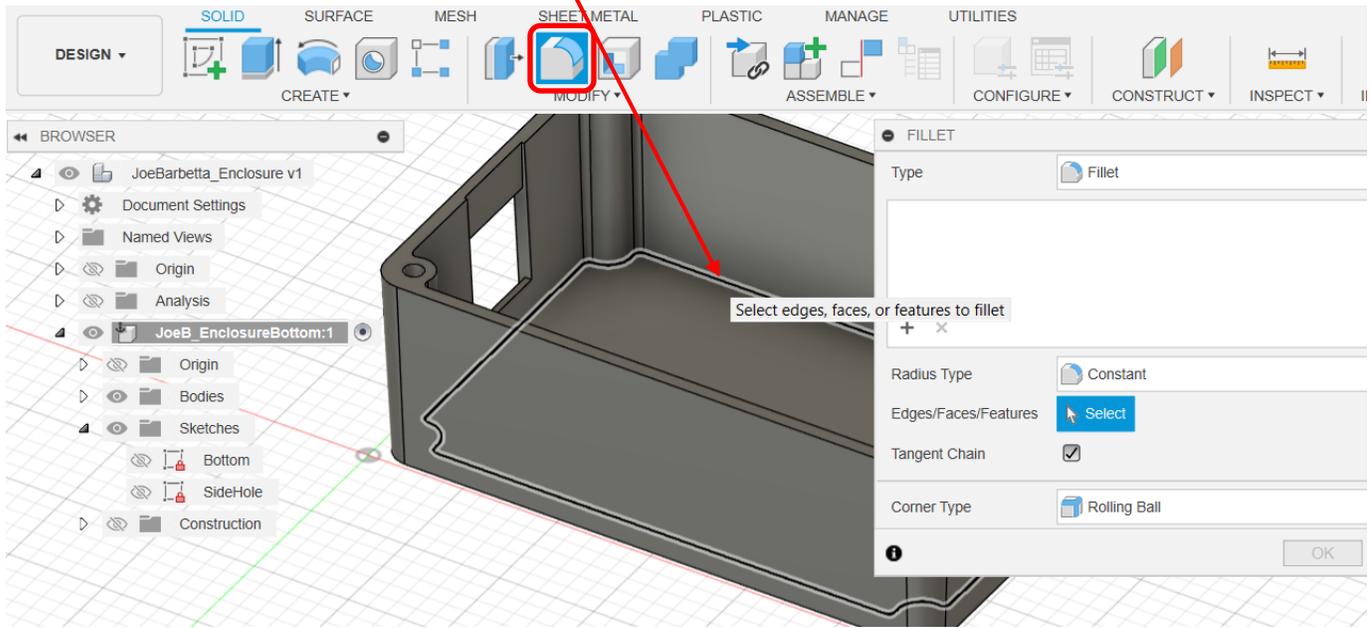
This is the result of the hole looking at the interior wall. The region above the hole is inset to accommodate part of the mold.



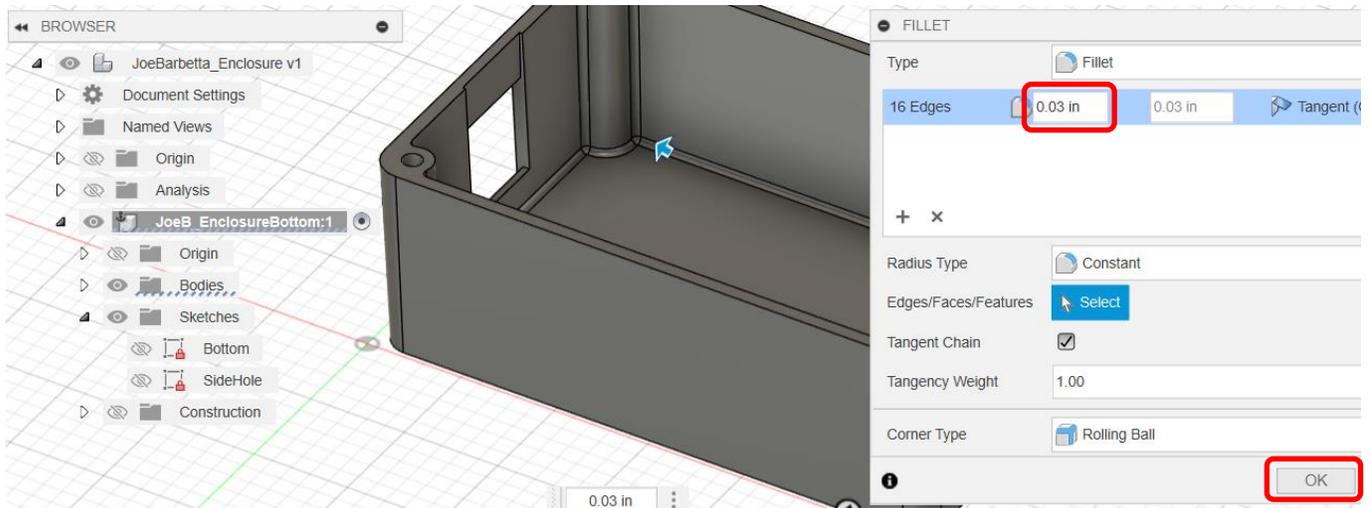
- rotate the view to check the hole looking at the exterior wall. The region below the hole is inset to accommodate part of the mold.



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

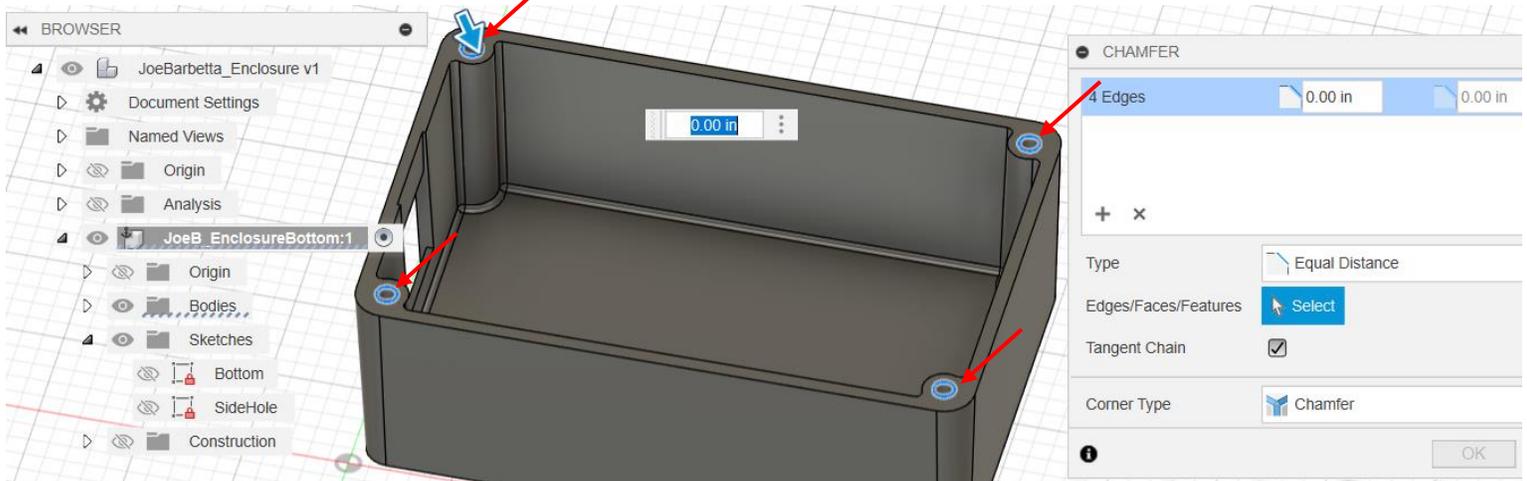


- enter a value of **0.03** and click **OK**

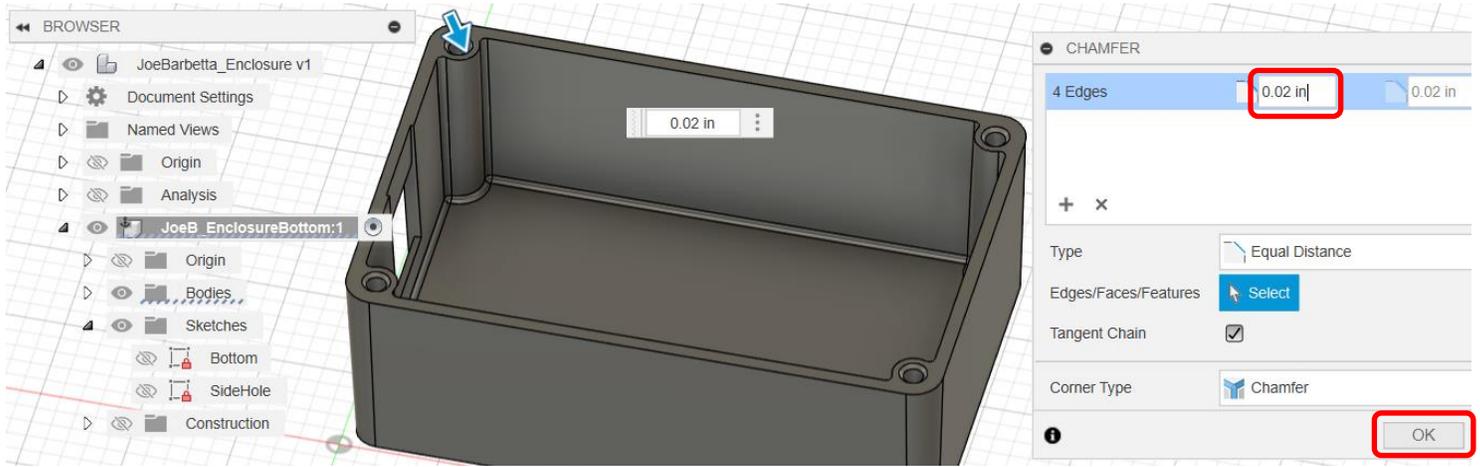


- from the **MODIFY** menu, select the **Chamfer** tool

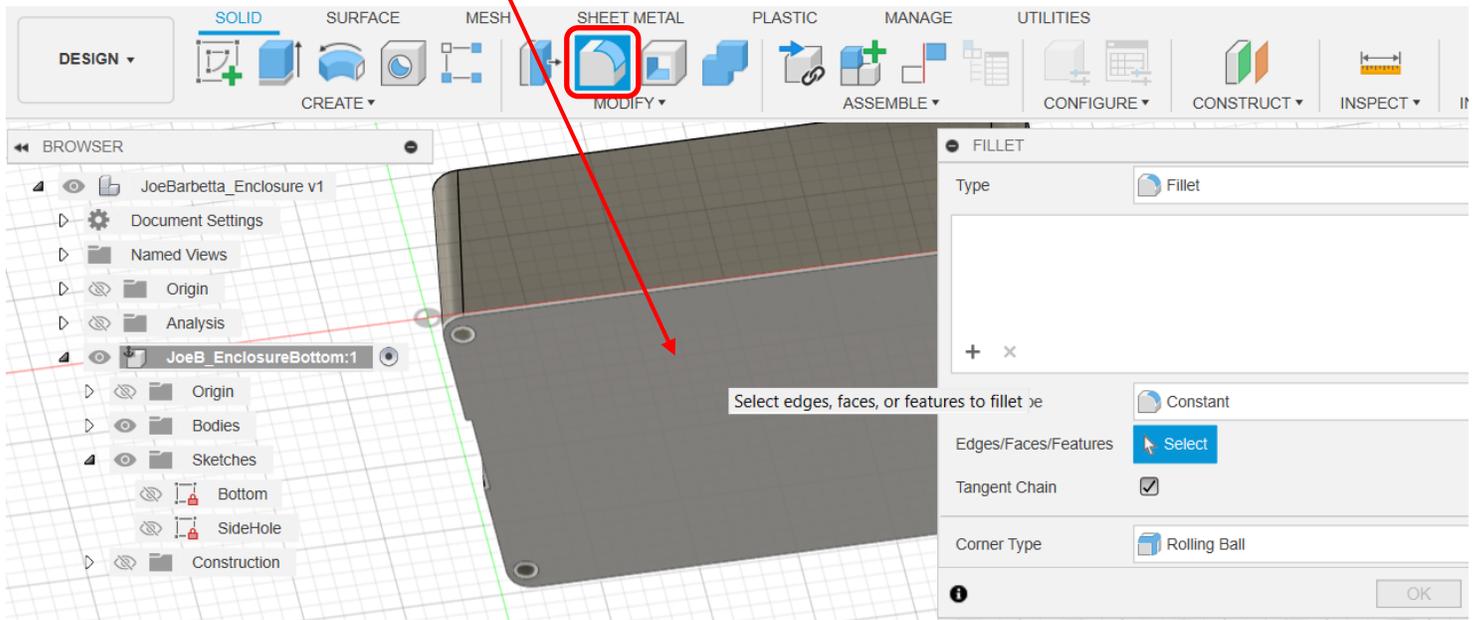
- click on the **edge of each of the 4 holes**



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

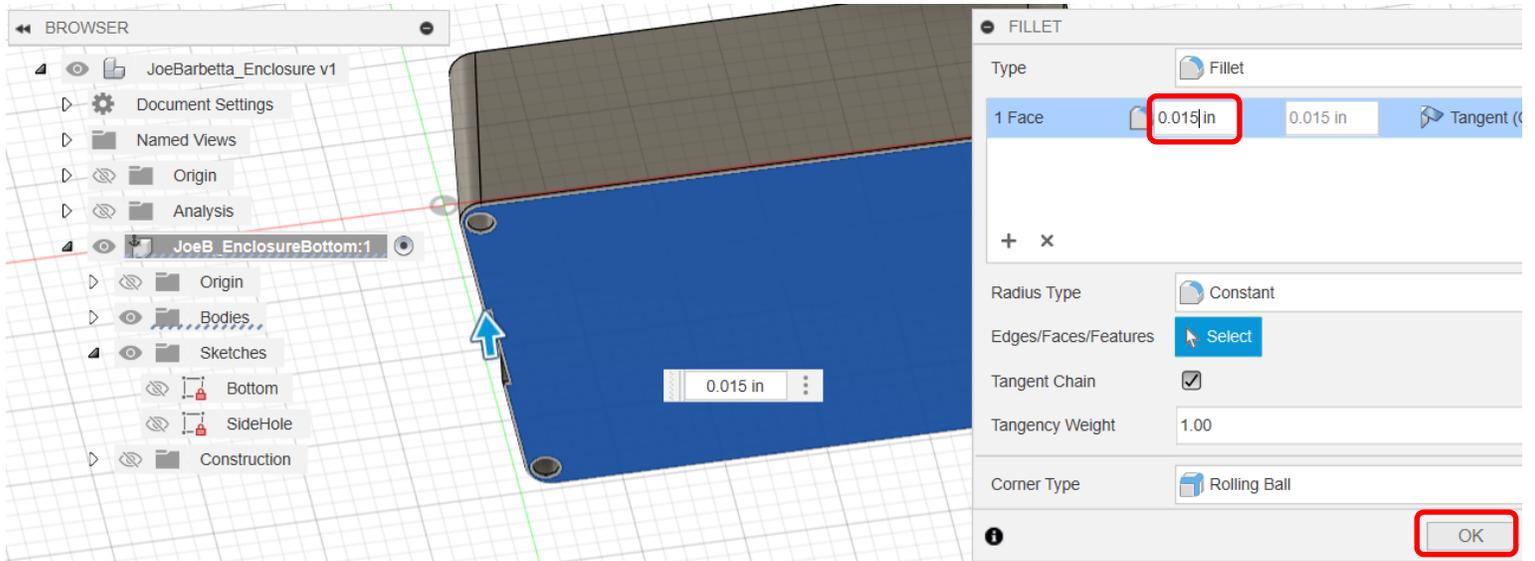


- select the **Fillet** tool and click on the **bottom of the enclosure**



- enter a value of **0.015** and click **OK**

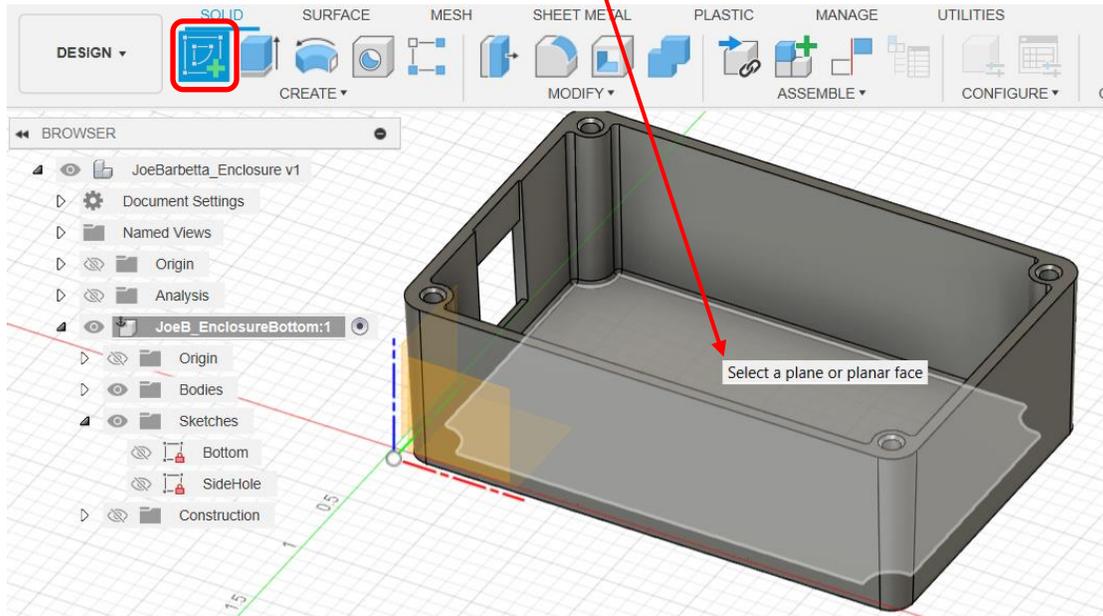
Note that **when a surface is selected, all of the features on the surface are filleted.**



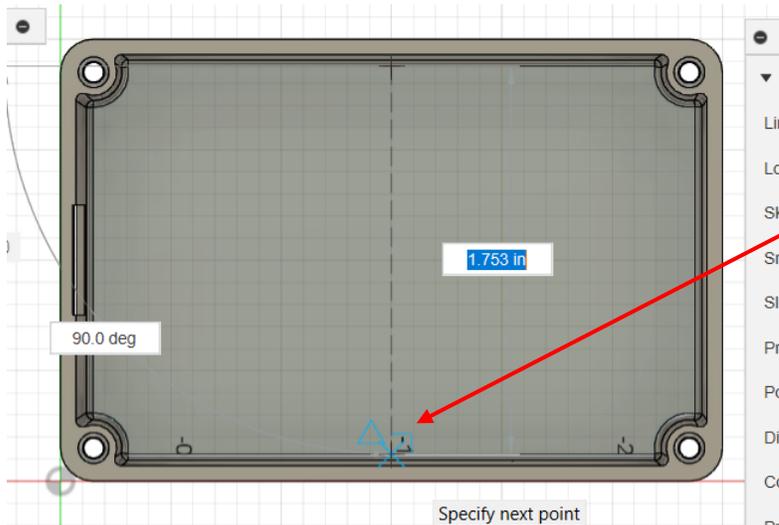
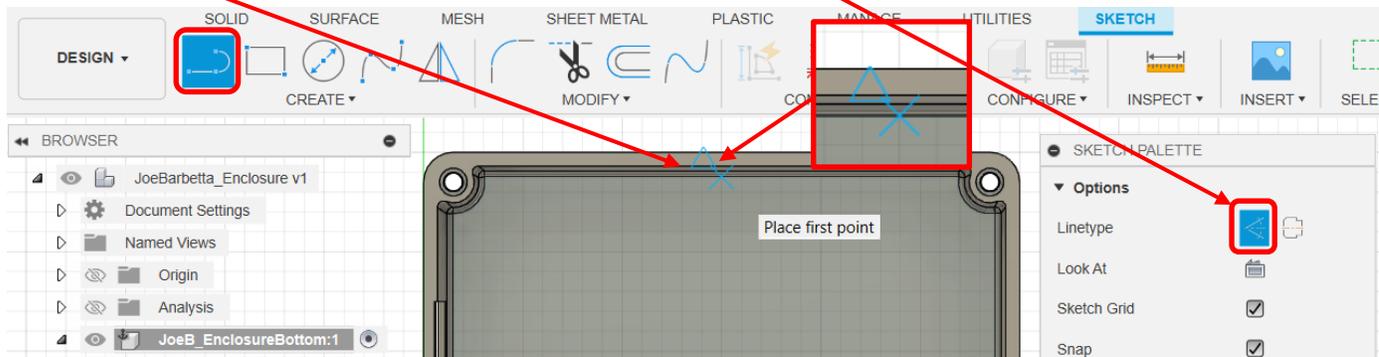
Adding Bosses

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

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

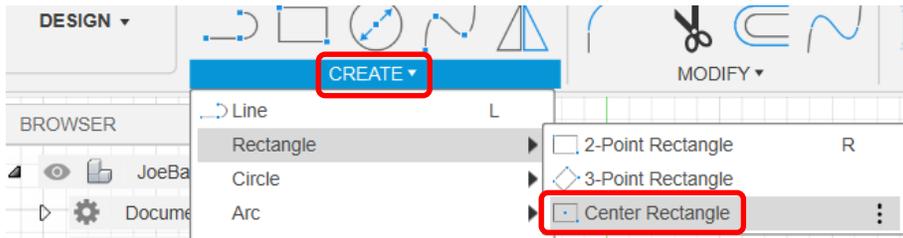


- click on the **Construction line icon** to highlight it blue
- select the **Line** tool and move the mouse over the center of the edge shown
- when a **blue triangle** appears, which indicates the center of the edge, **click on that point**

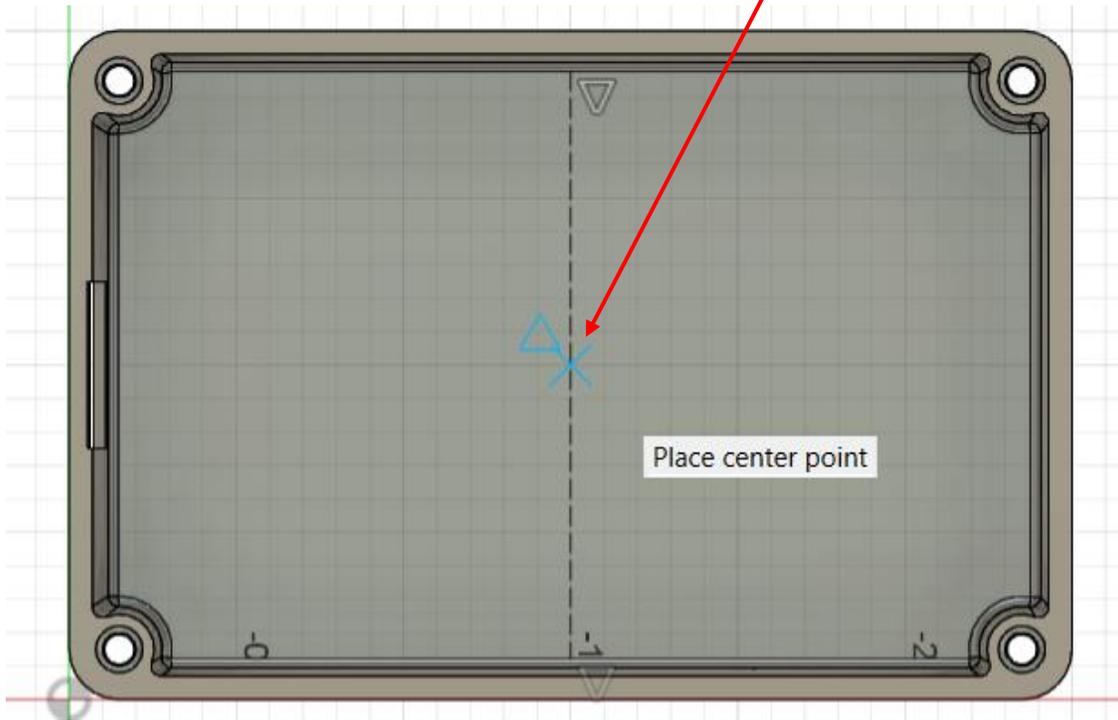


- extend the line downward to the opposite edge
- when a **blue x and triangle** appear, click on that point

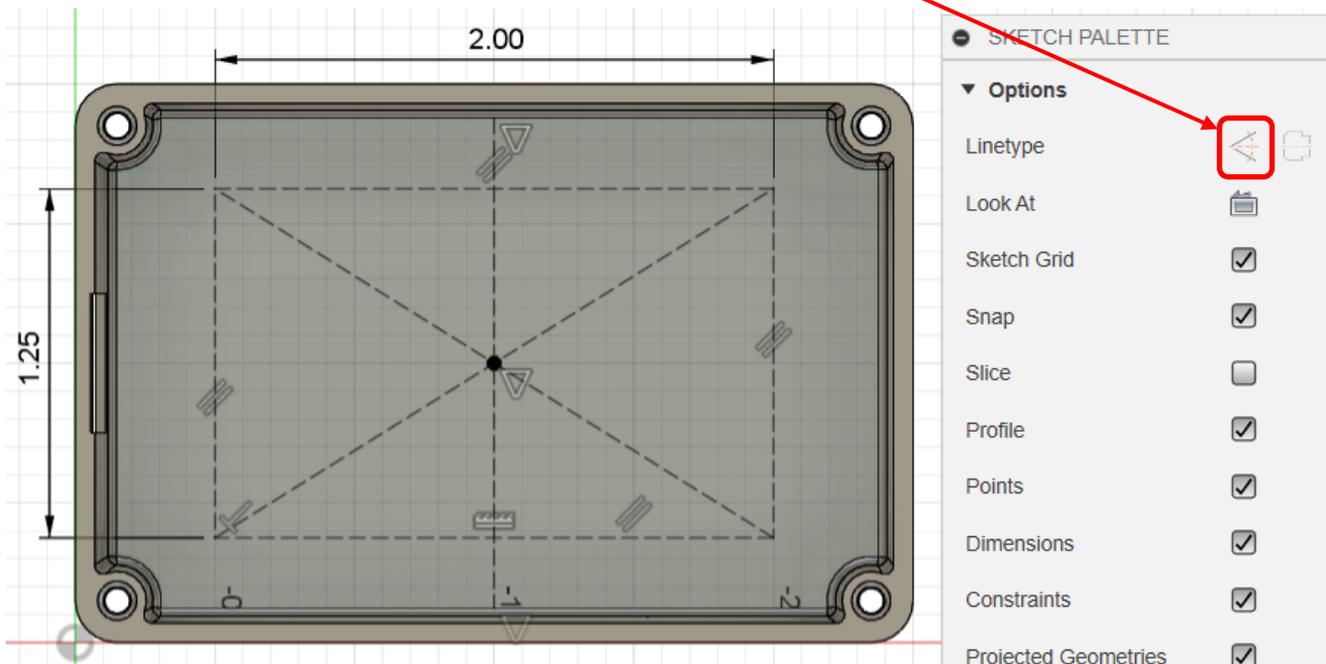
- from the **CREATE** menu, select **Center Rectangle**



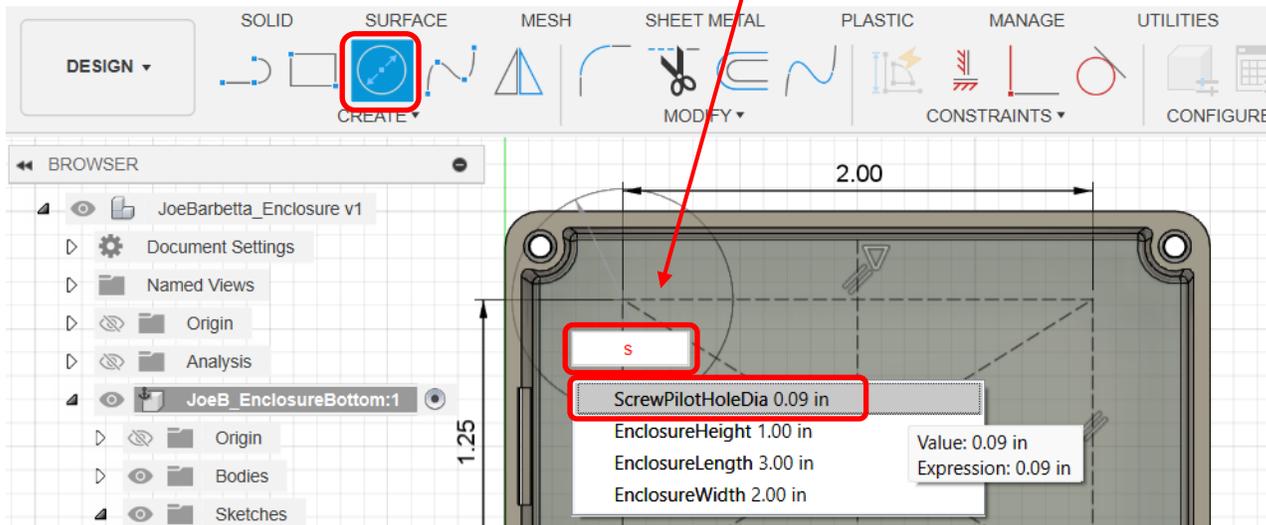
- move the mouse over the **dashed line** and **click when the blue triangle** appears



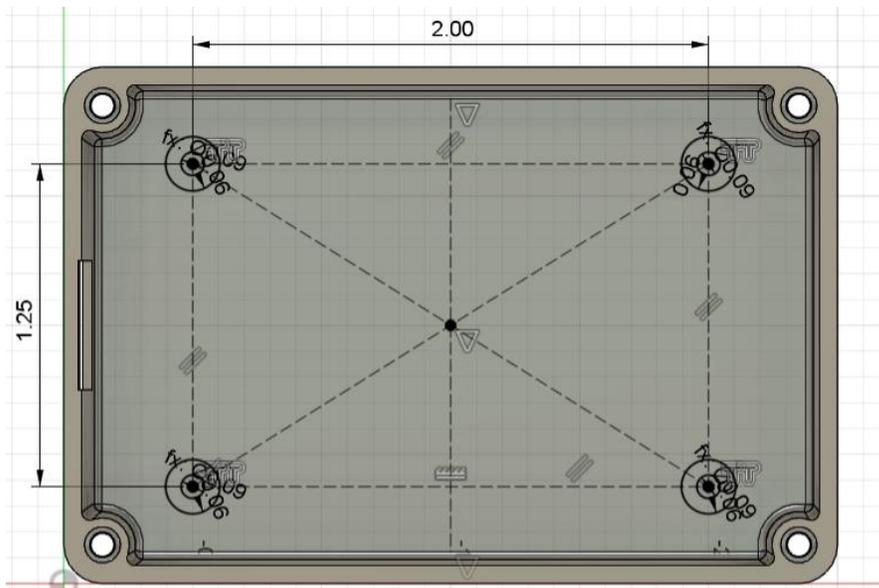
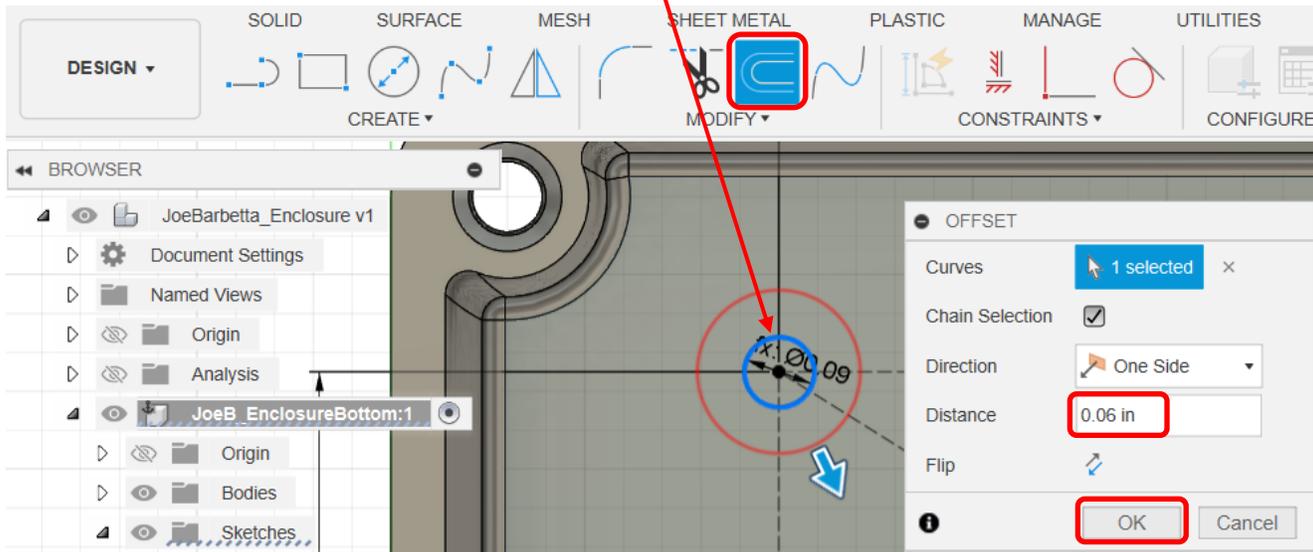
- extend the **rectangle outward** and enter values of **1.25** and **2.00** as shown
- click on the **Construction line icon** again to **remove the blue highlighting**



- 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

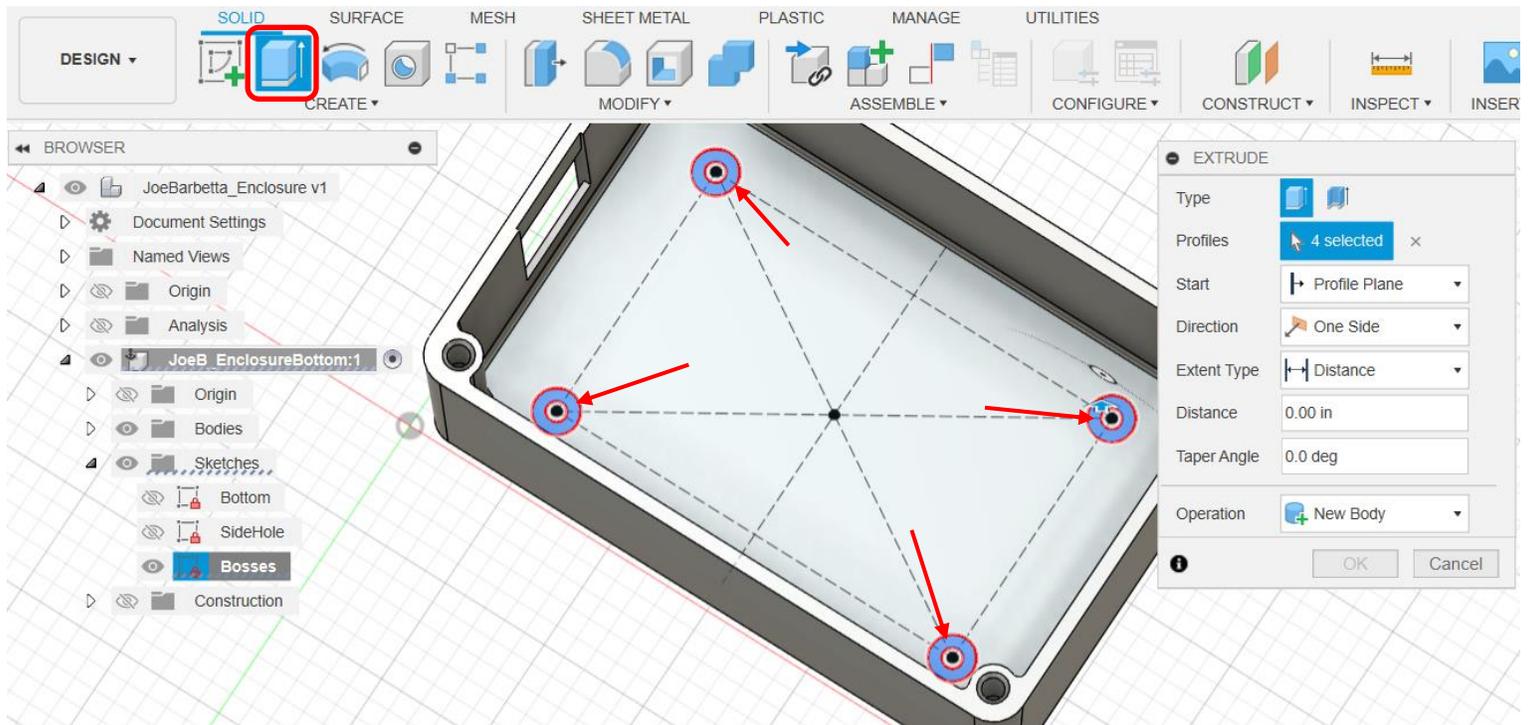


- select the **Offset** tool and click on the **circle just created**
- **extend the circle outward**, enter a **Distance** of **0.06**, and click **OK**

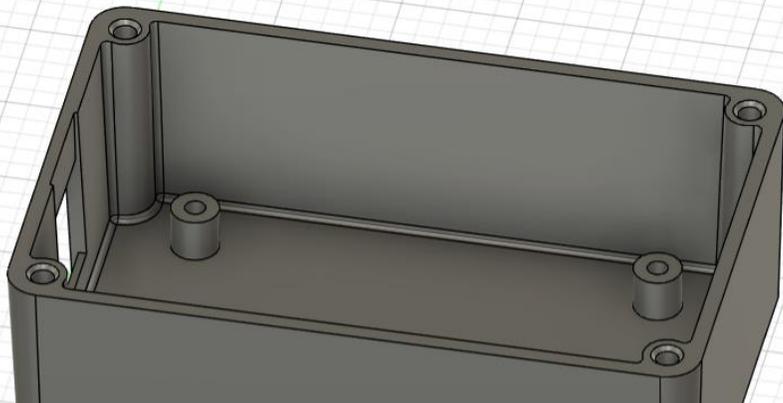
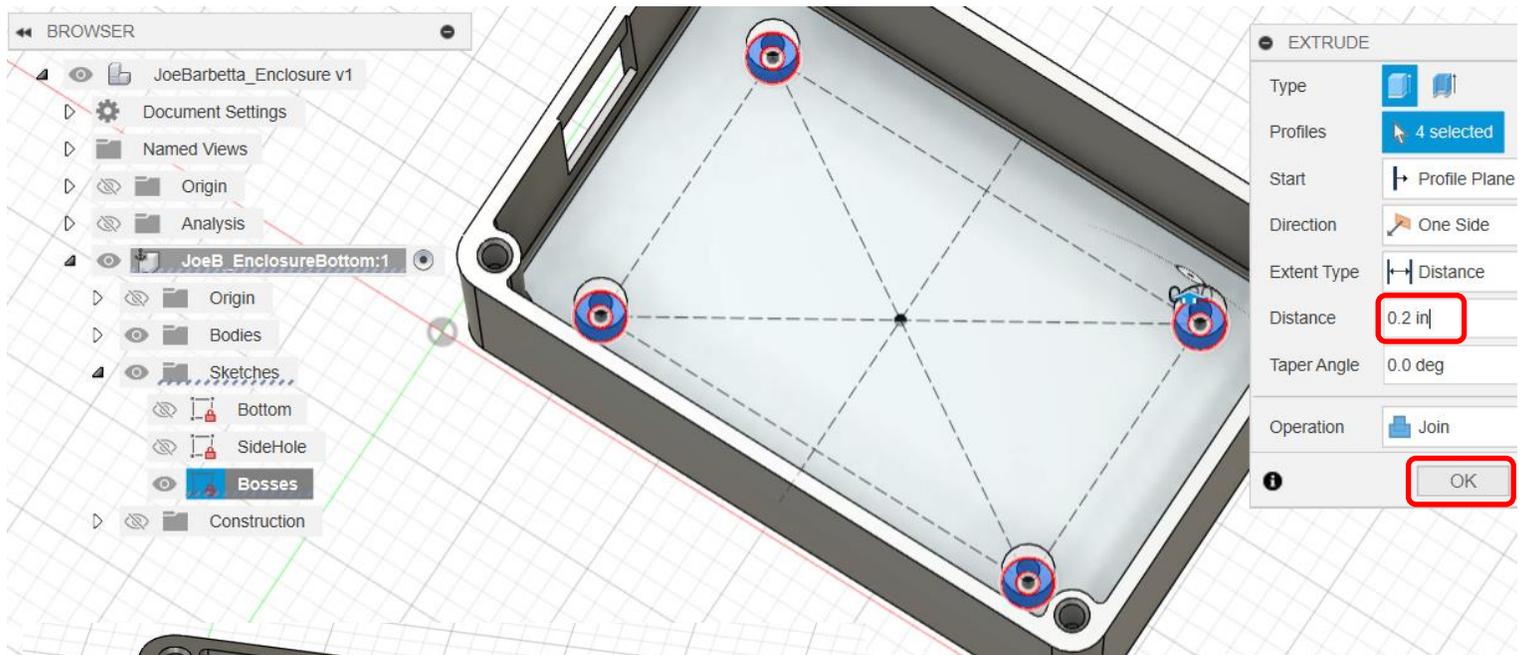


- perform the **same circle and offset operations** at the **other 3 corners**
- rename the Sketch to **Bosses**
- click **Finish Sketch**

- 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**.

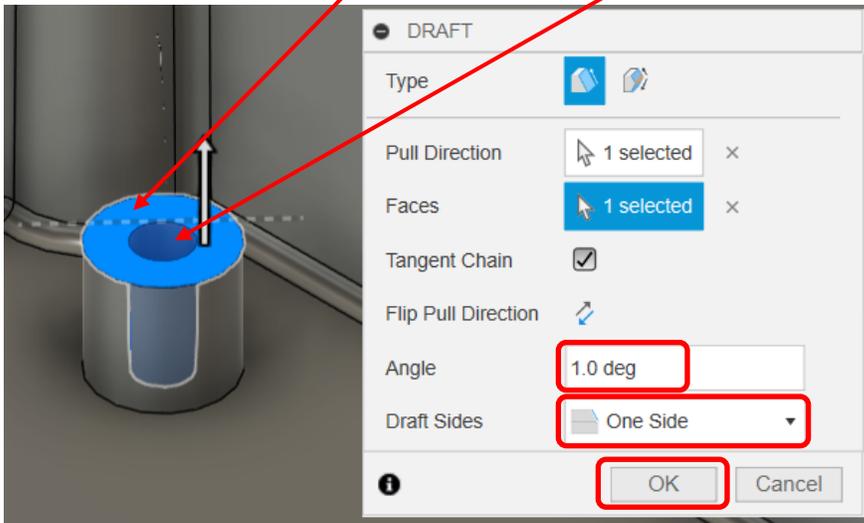


- enter **0.20** for Distance and click **OK**

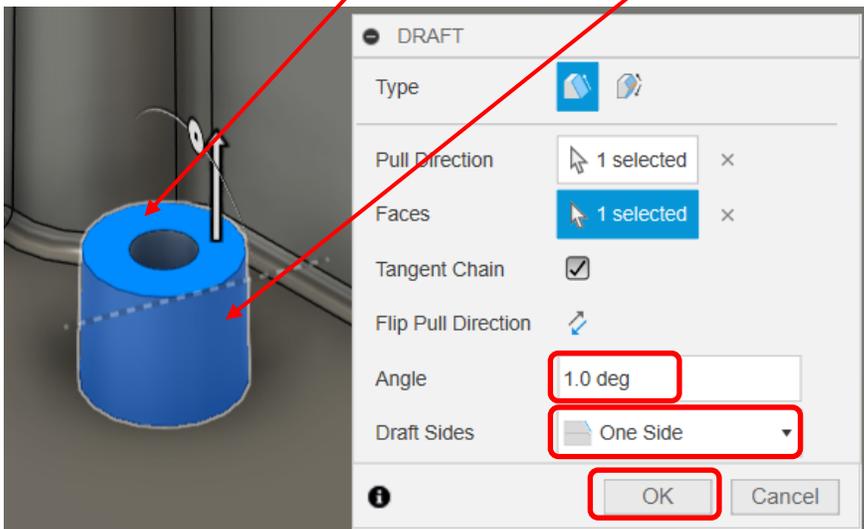


This is the result of the extrude.
The other 2 bosses are obscured by the wall.

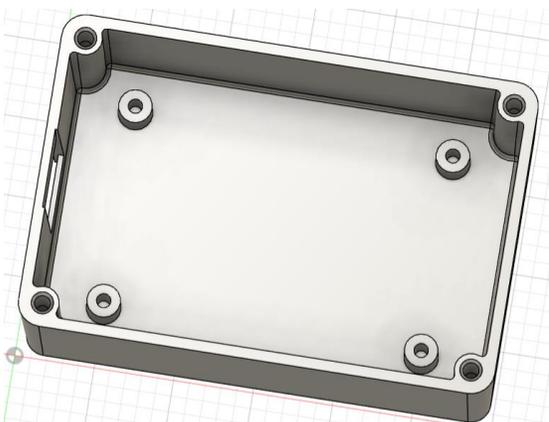
- zoom into one of the bosses
- from the **MODIFY** menu, select **Draft**
- click on the **top surface of the boss** and then on the **interior of the hole**
- ensure that **Draft Sides** is set to **One Side**, enter a value of **1.0** for **Angle**, and click **OK**



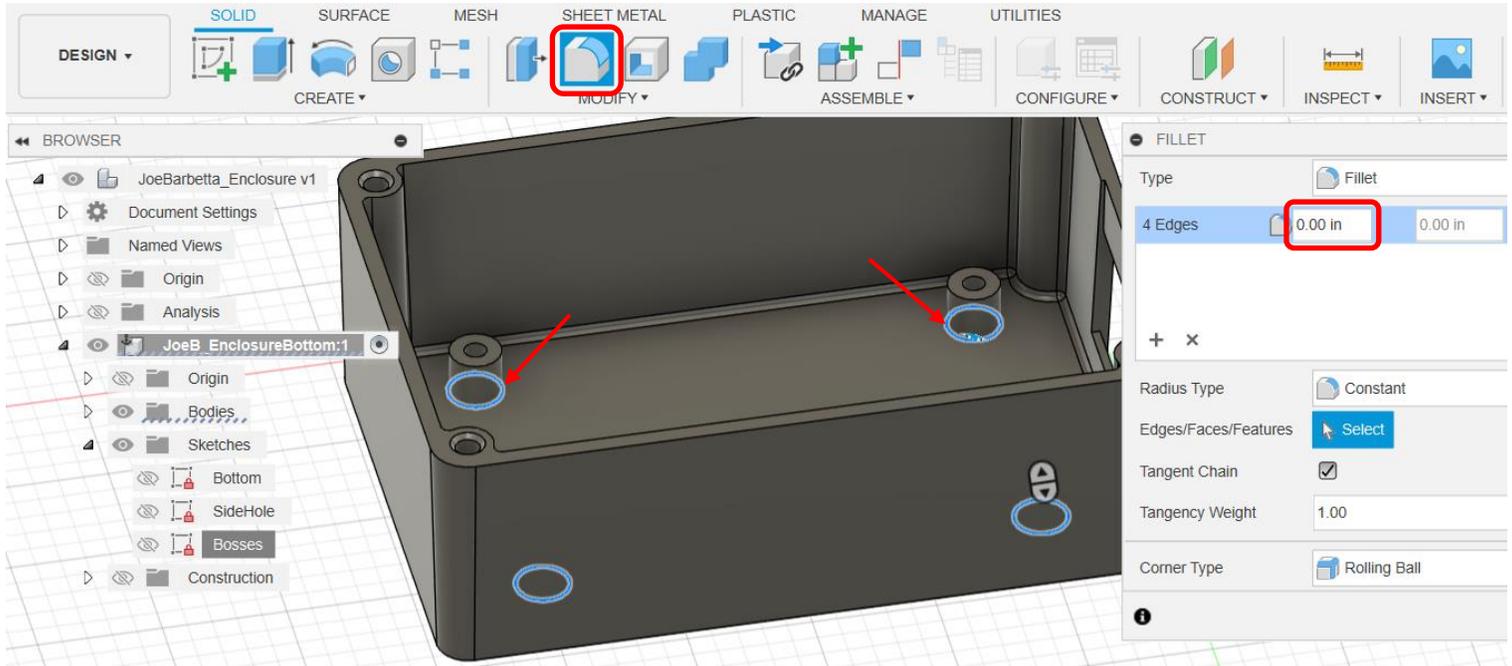
- select the **Draft** tool again
- click on the **top surface of the boss** and then on the **Exterior of the hole**
- ensure that **Draft Sides** is set to **One Side**, enter a value of **1.0** for **Angle**, and click **OK**



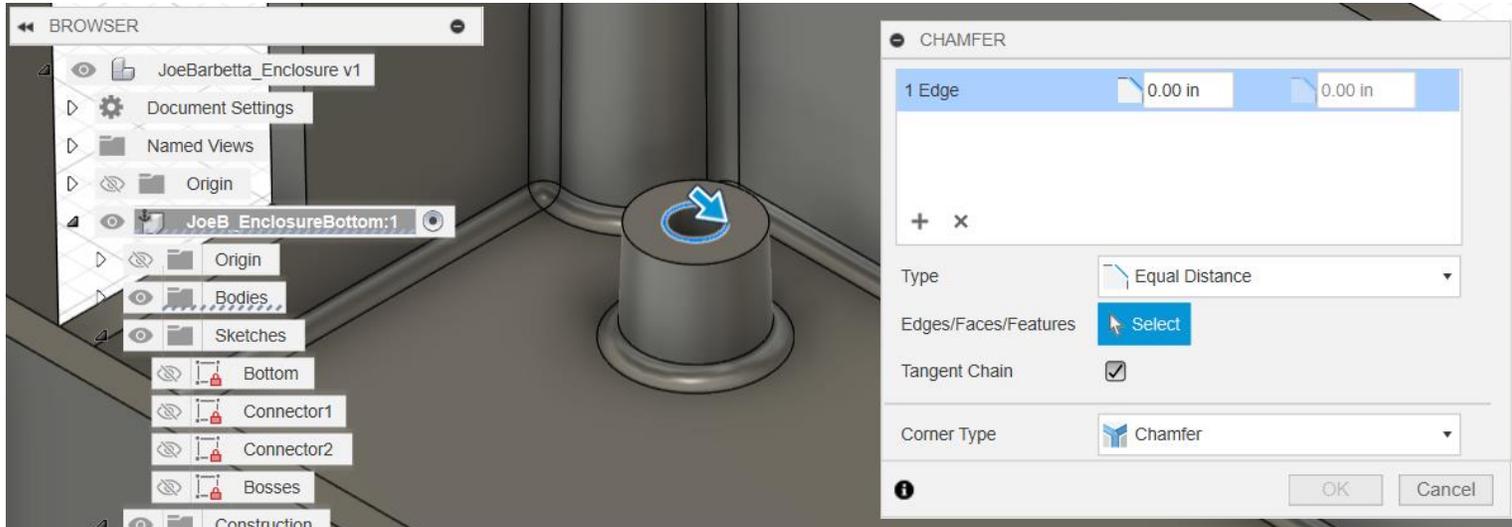
- perform the above **2 Draft operations** on the **other 3 bosses**



- 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**

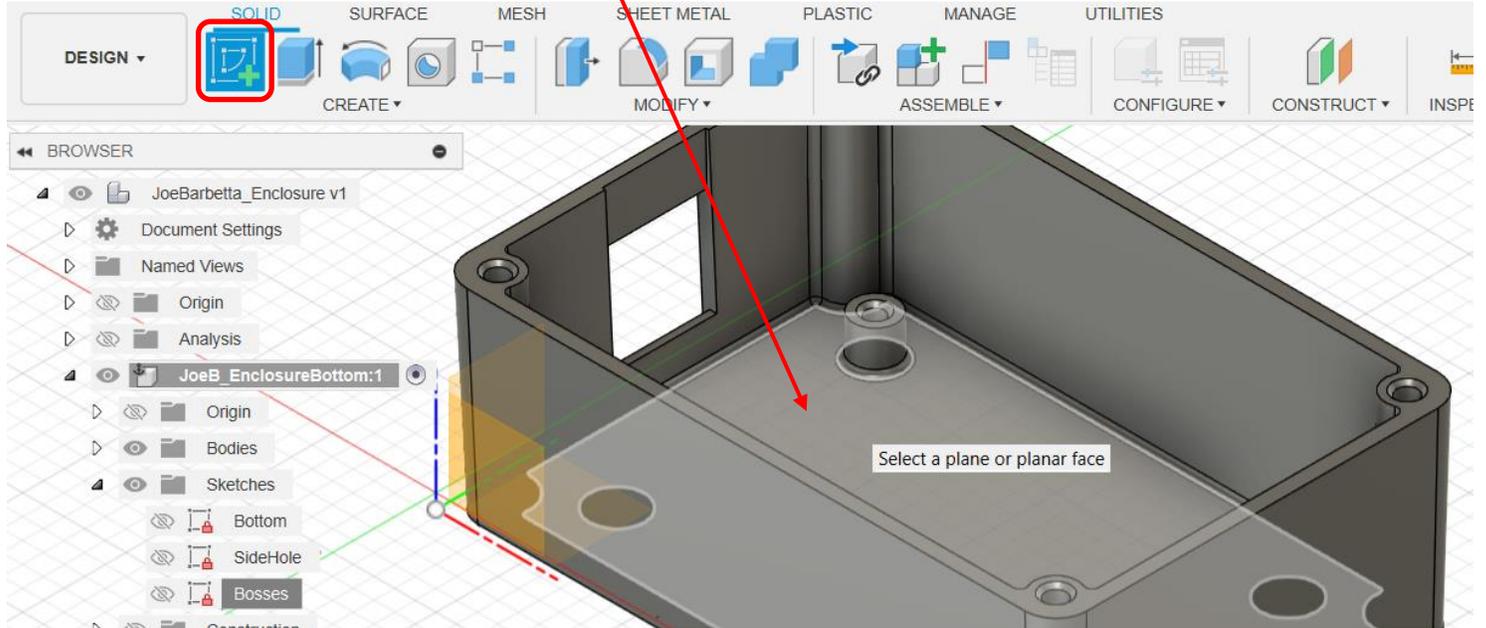


- from the **MODIFY** menu, select the **Chamfer** tool
- click on the **hole edge of each boss**, enter a value of **0.02**, and click **OK**

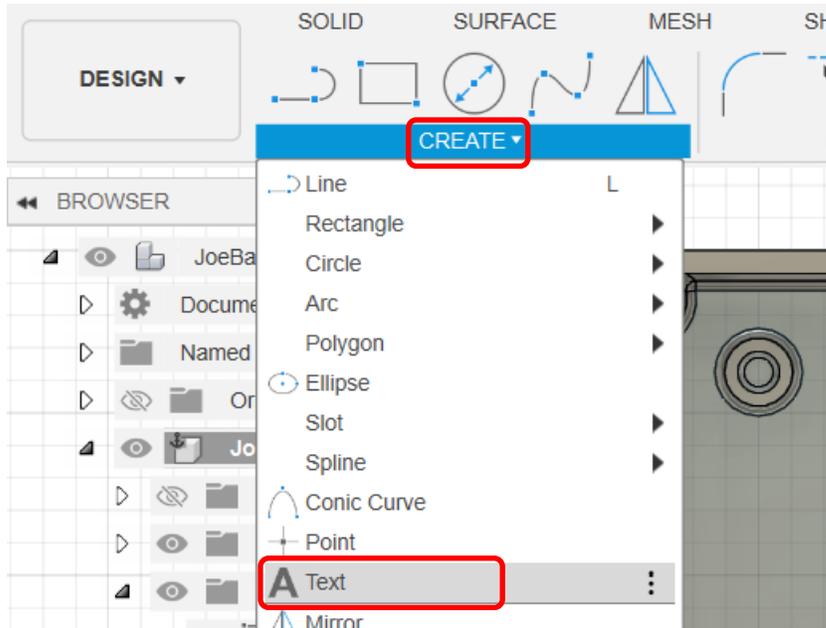


Adding Text

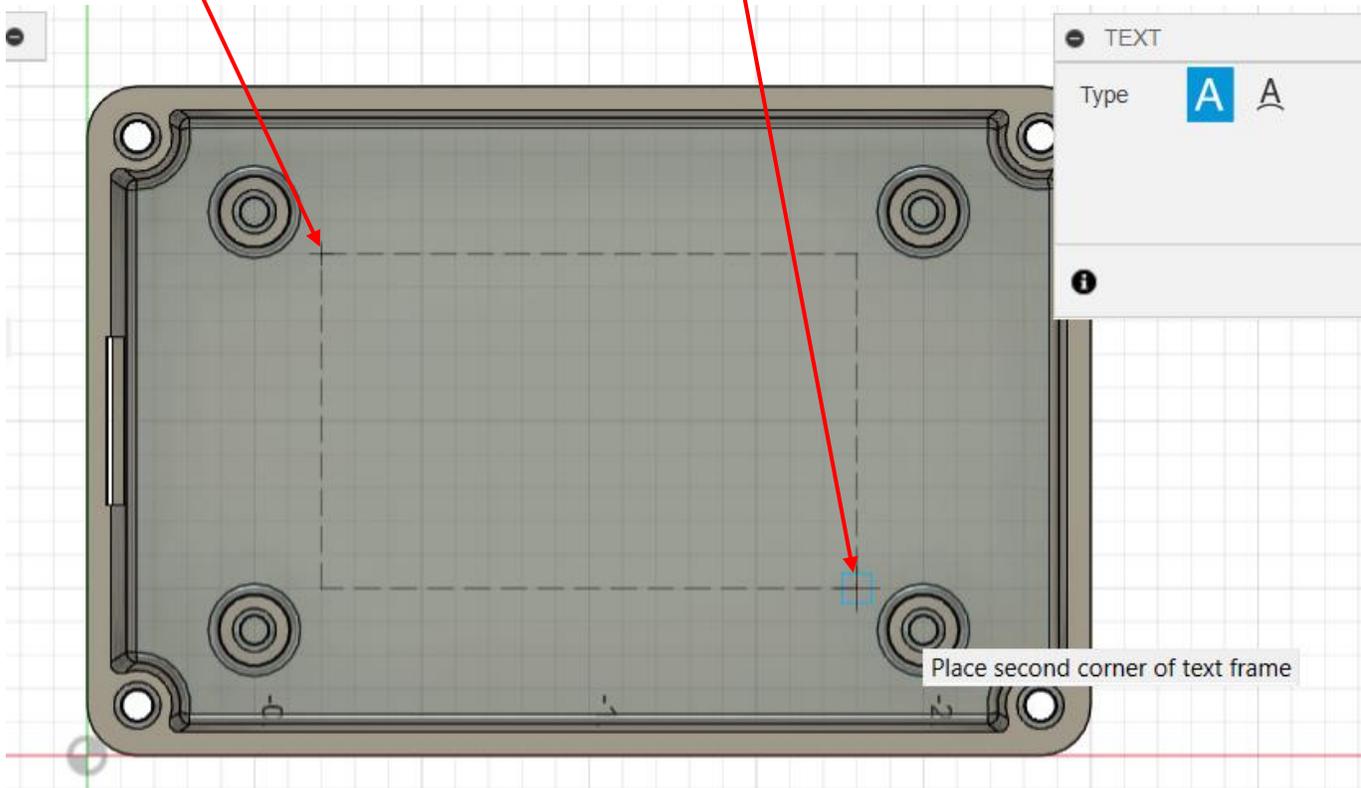
- select **Create Sketch** and click on the **interior bottom surface**
- rename the **Sketch** to **Text**



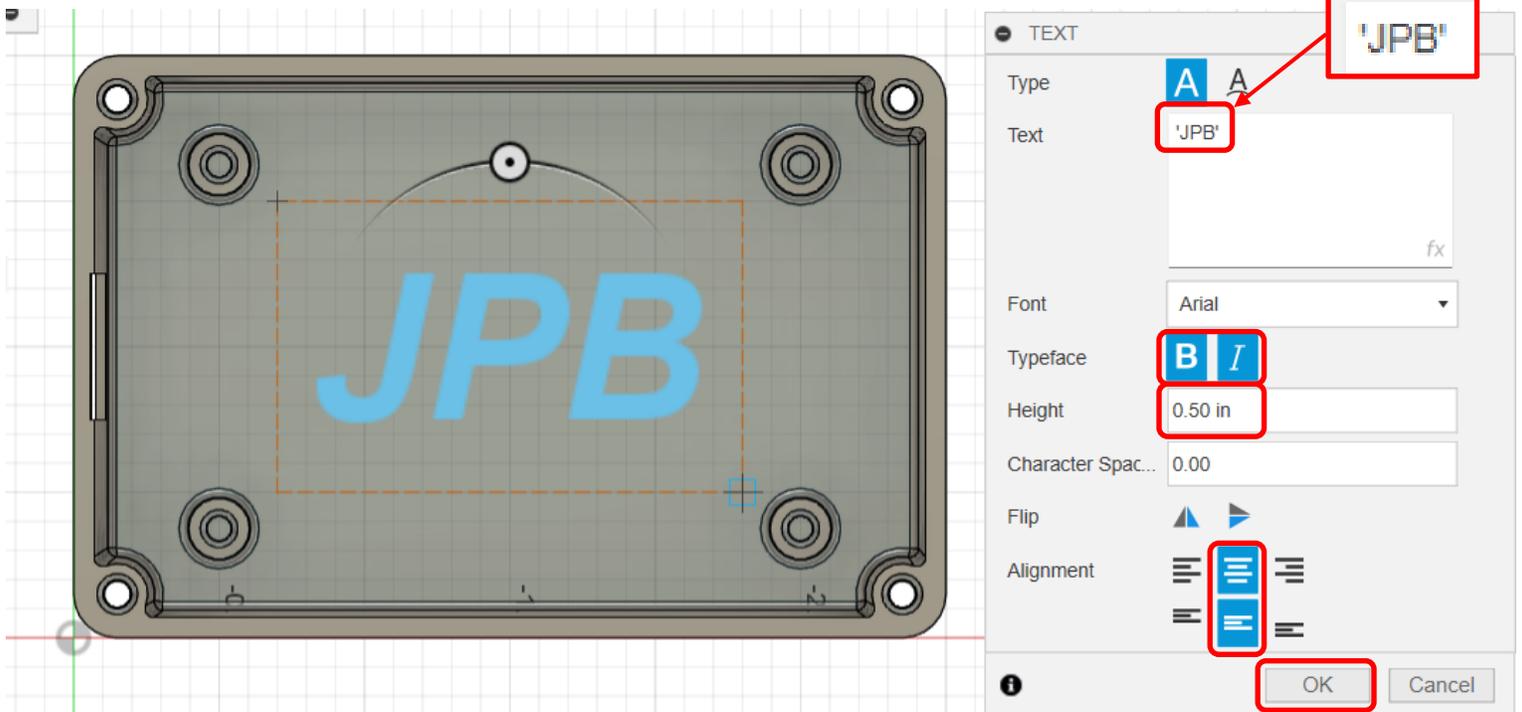
- from the **CREATE** menu, select **Text**
- if a message window appears regarding **Parametric Text**, click is **OK** button



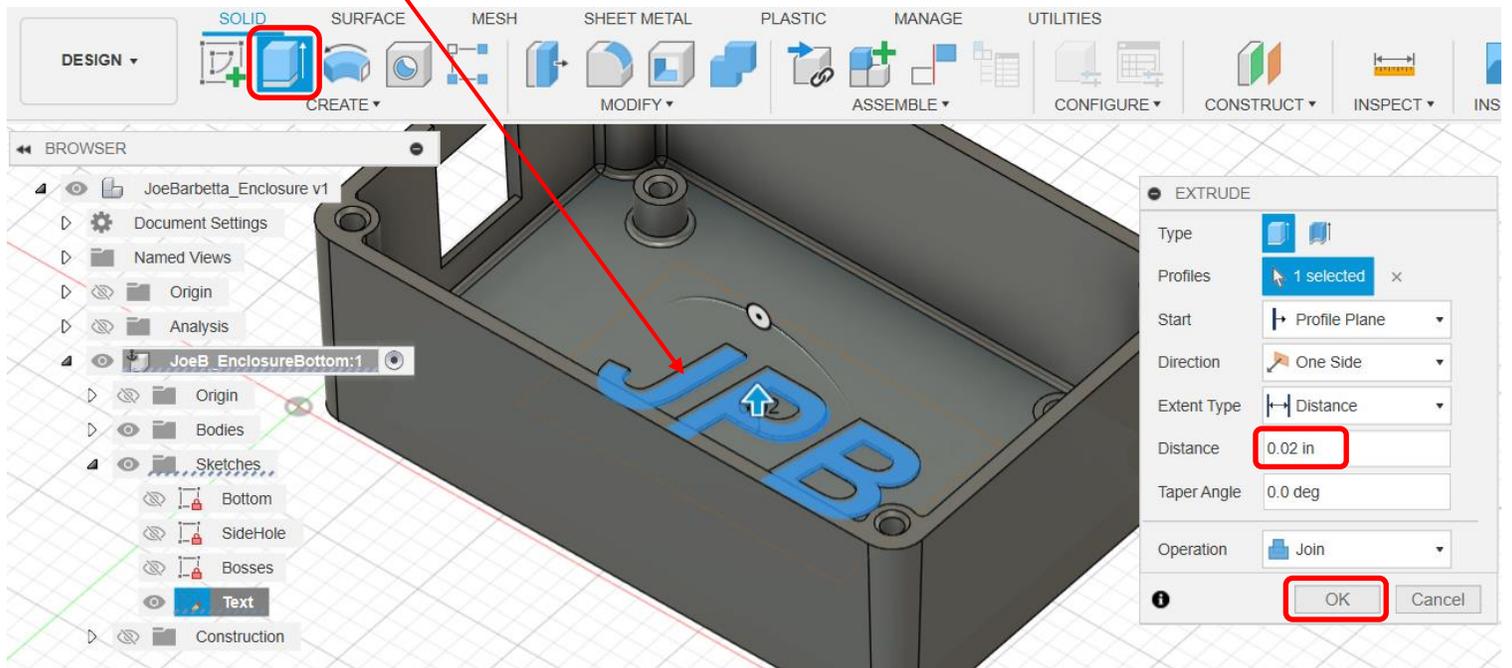
- if a message window appears regarding **Parametric Text**, click is **OK** button
- click on a **point near the top left boss**. These point locations are not critical.
- extend the rectangle down and to the right and click on a **point near the bottom right boss**



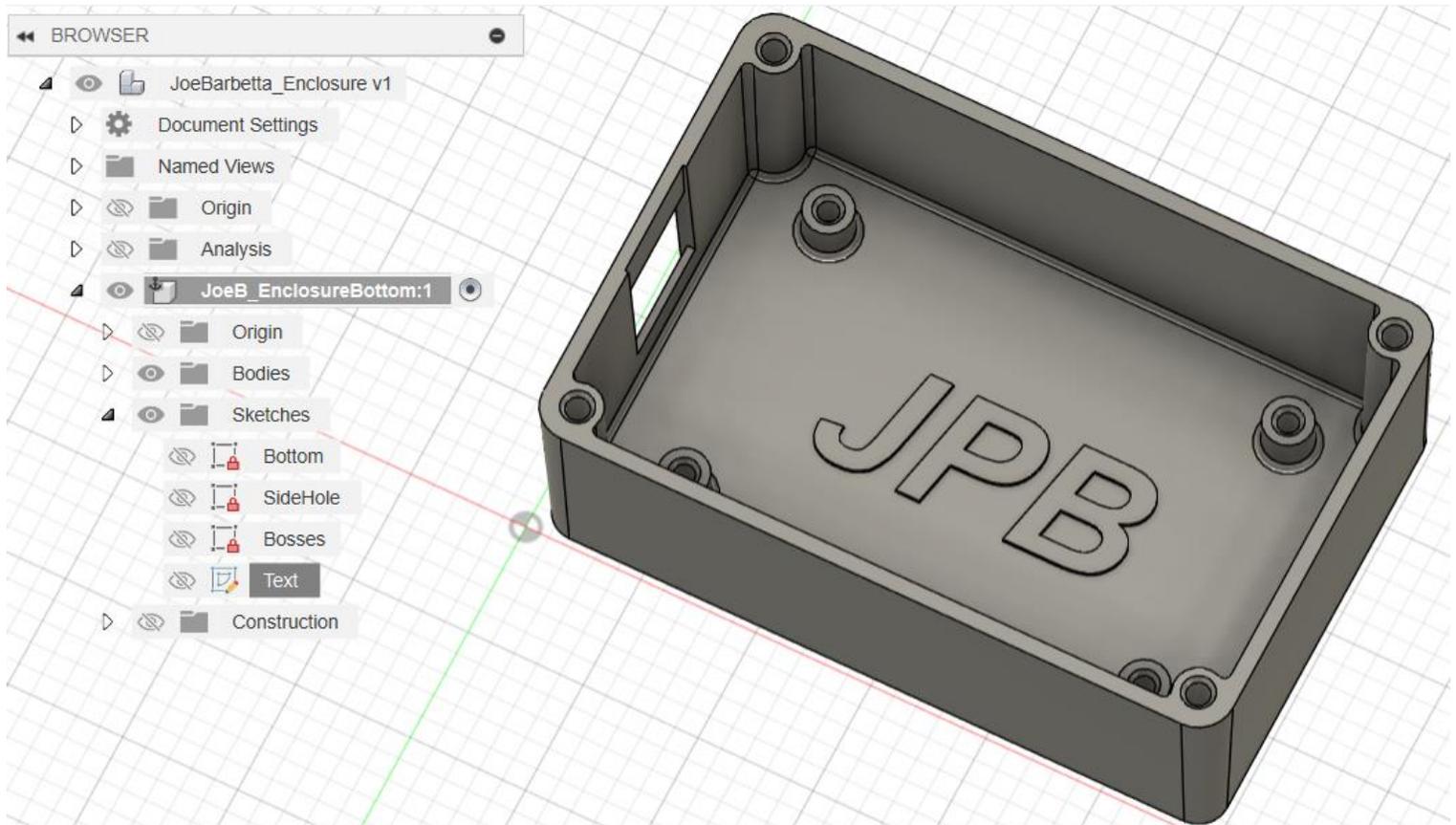
- in the Text box enter your **3 initials preceded by and followed by a single quote**
- click on the **Bold** and **Italic** icons to highlight them
- enter **0.5** for **Height**
- select the **2 center Alignment options**
- click **OK** and **Finish Sketch**



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



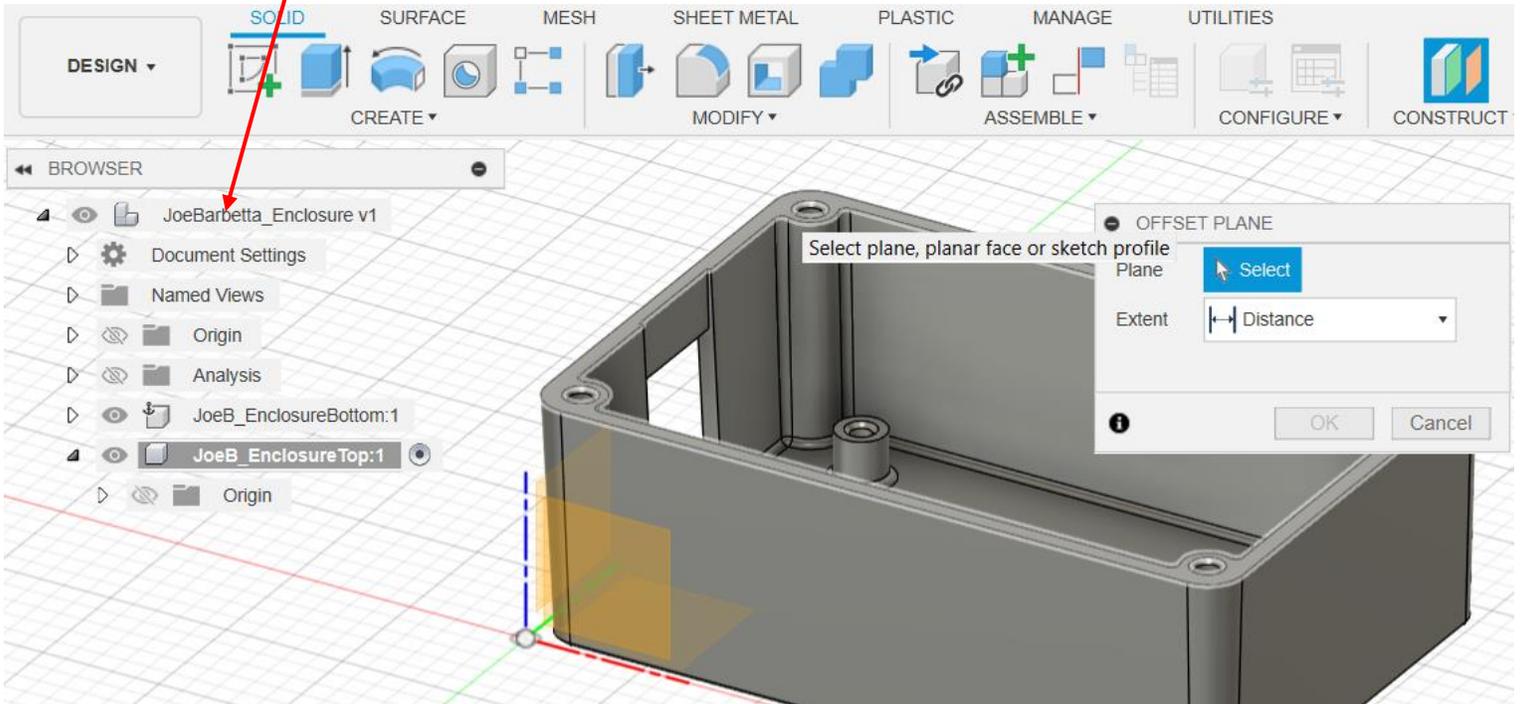
- admire your enclosure
- yell "**Dude, we need a cover!**"



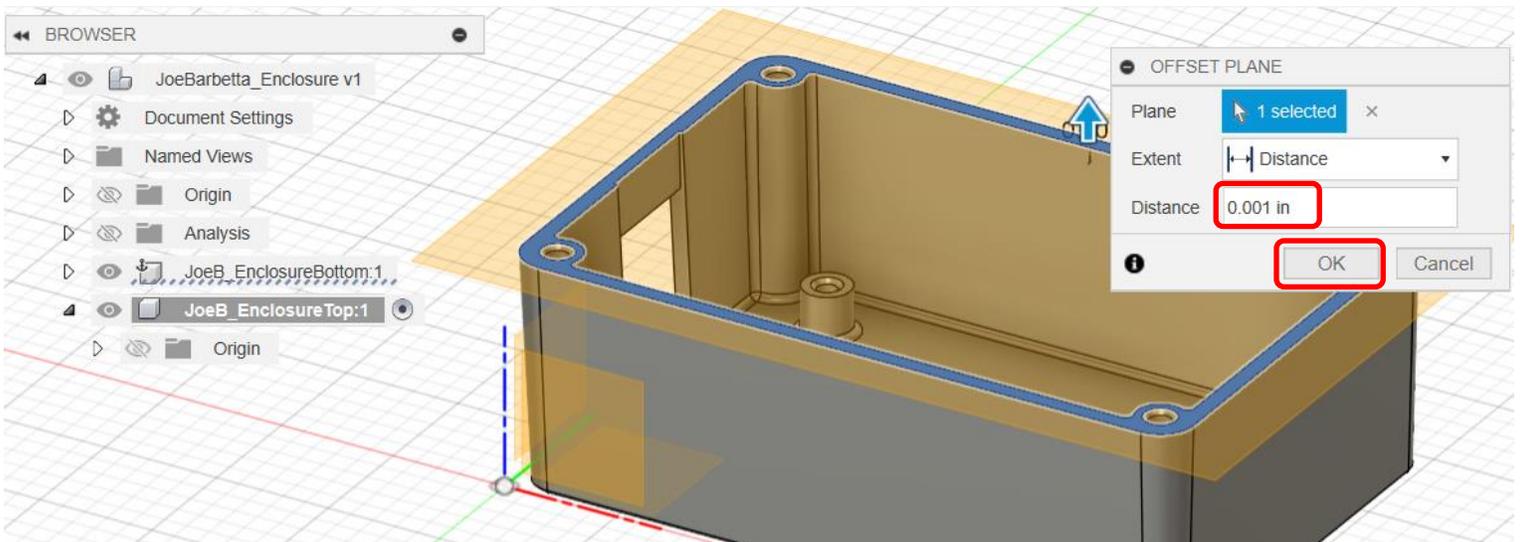
Creating the Enclosure Cover

- right-click on the **Project Name** and select **New Component** and name is using your **first name, last name initial, followed by “_EnclosureTop”**, e.g. “JoeB_EnclosureTop”

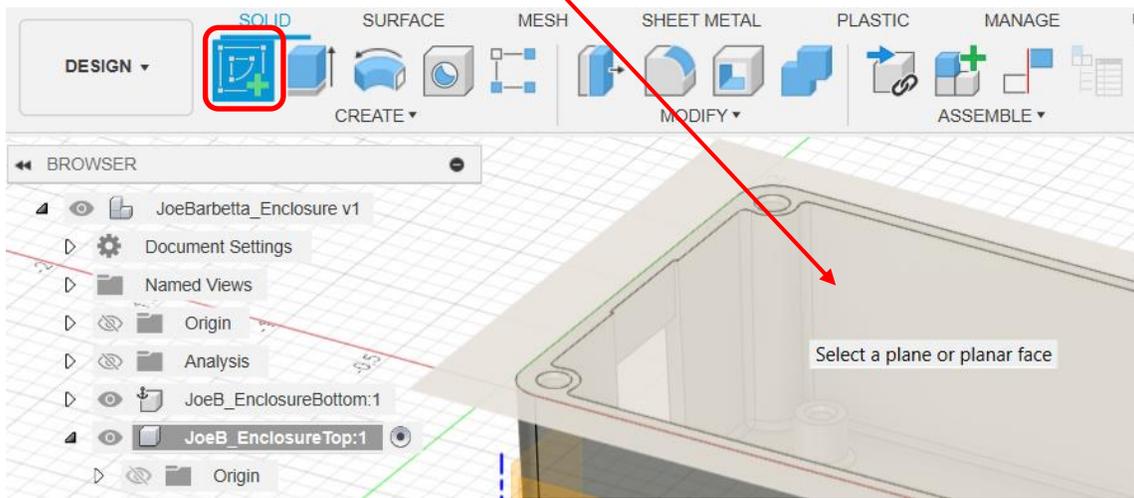
- from the **CONSTRUCT** menu, select **Offset Plane** and click on the top surface of the enclosure bottom



- enter **0.001** for Distance and click **OK**. The Offset Plane and small distance allows the Sketch to not automatically contain elements of the enclosure bottom.



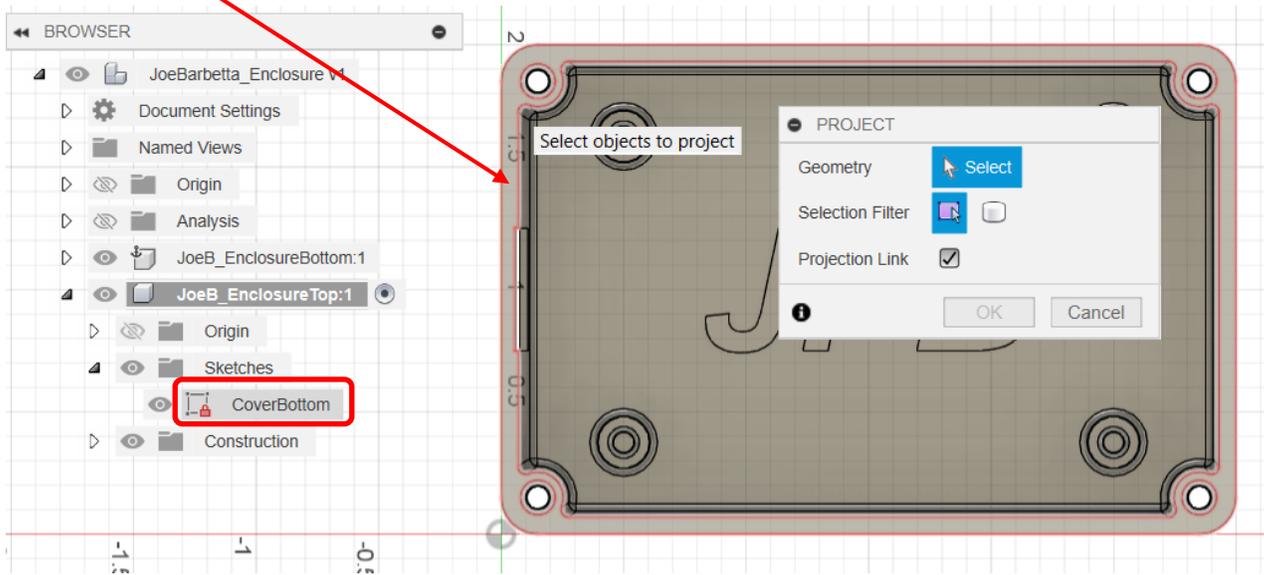
- select **Create Sketch** and click on the **Construction Plane**



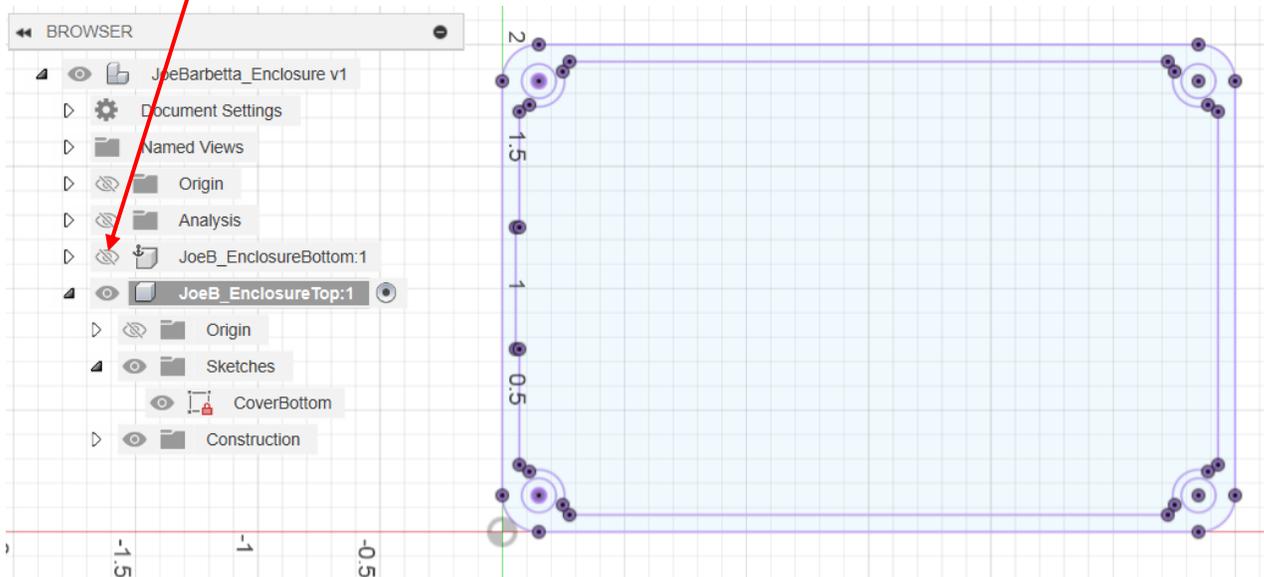
- right-click on the new **Sketch**, select **Rename**, and change the name to **CoverBottom**

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

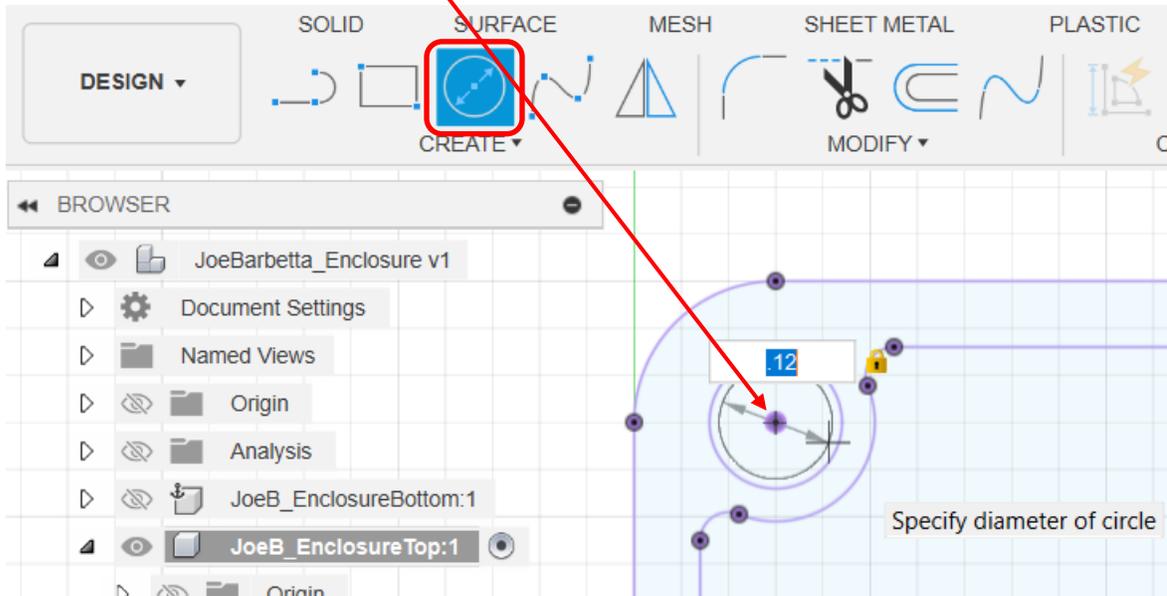
- click on the **top of the enclosure wall**, which should result in the red lines and circles as shown



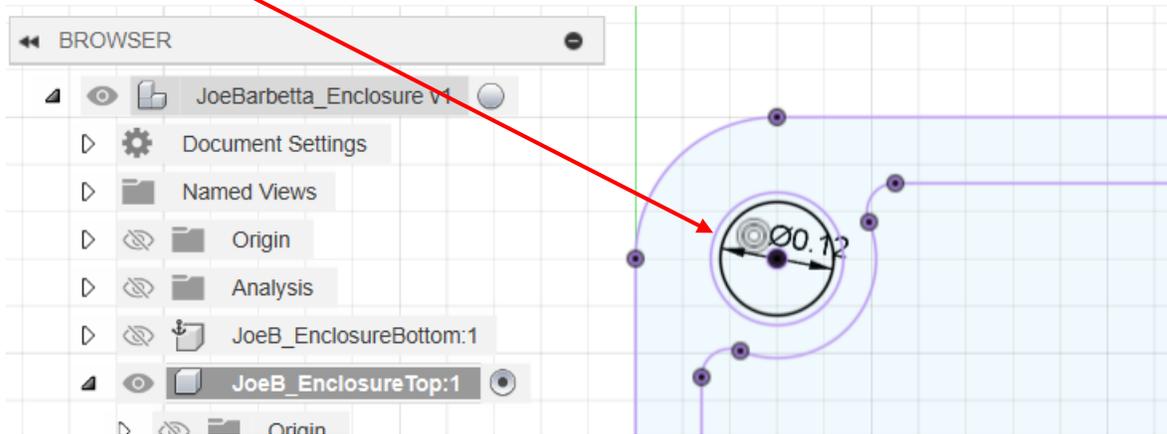
- click on the **eye** icon for the **EnclosureBottom** Component to hide it



- zoom into the top left corner of the Sketch
- select the **Center Diameter Circle** tool
- click on the **center of the violet circle** and enter a value of **0.12**

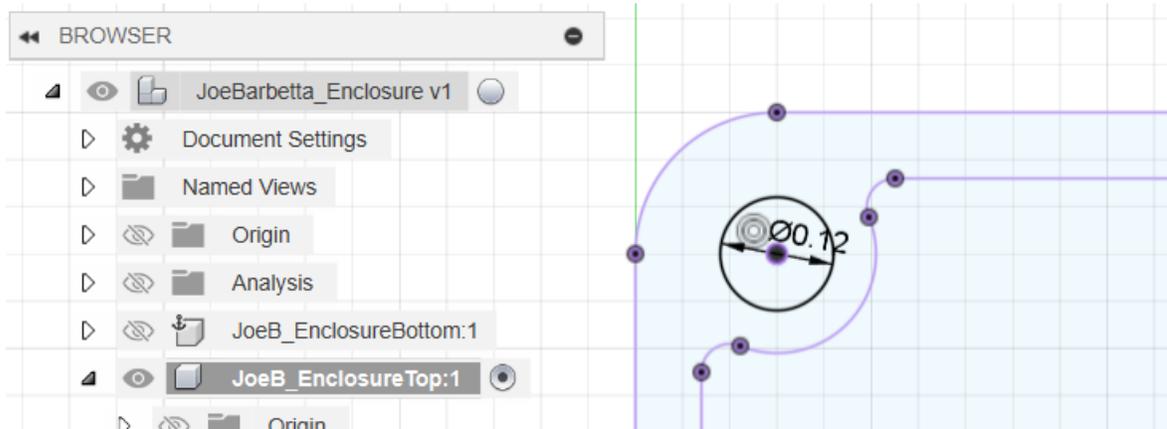


- click on the **violet circle** and press the **Delete** key

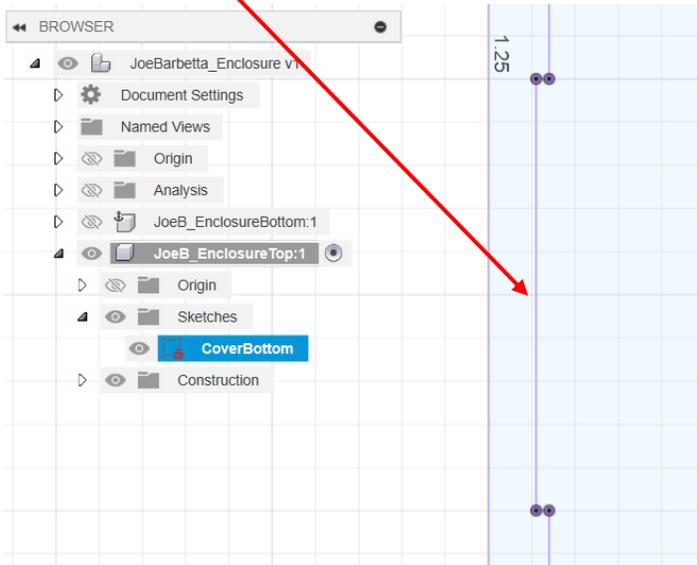


This should be the result.

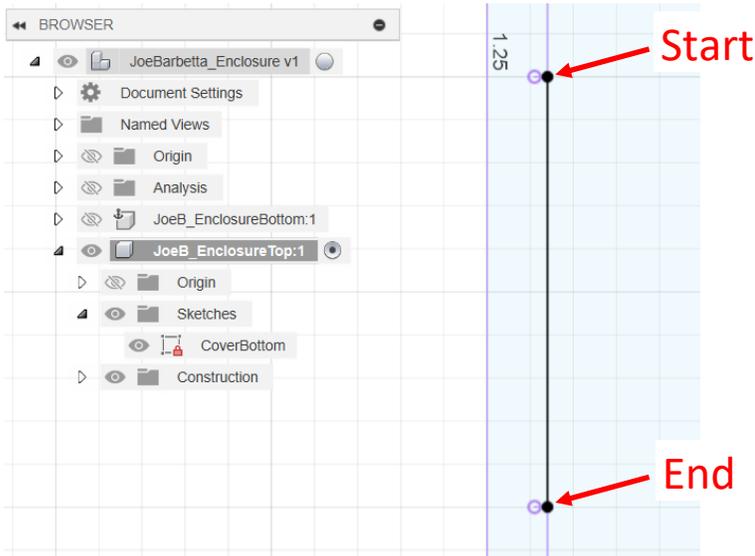
- zoom into the **other 3 corners** and **perform the same operation** to add the 0.12 circle and delete the violet circle



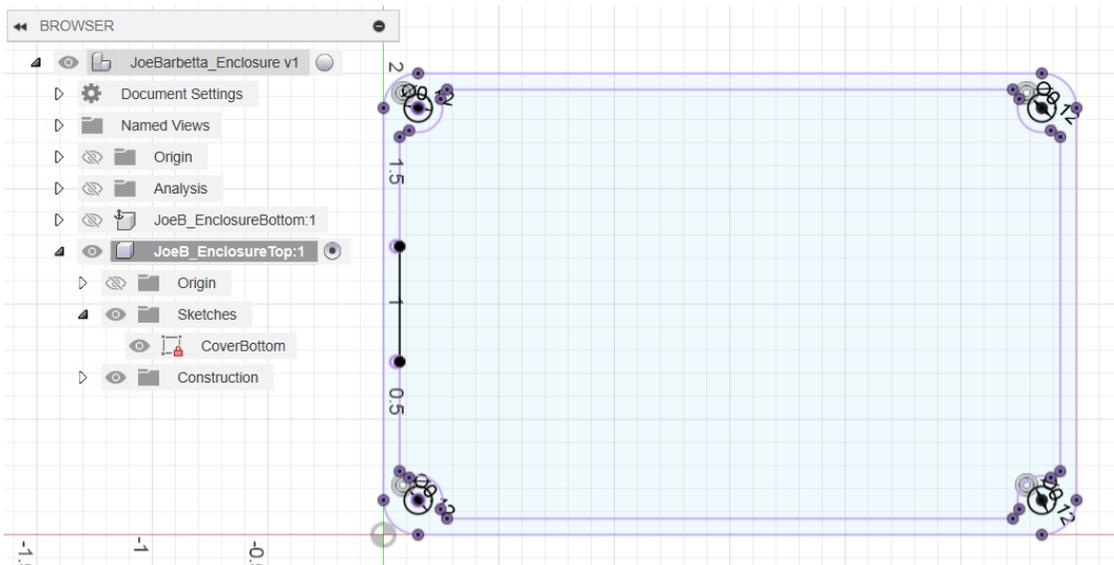
- zoom into the side of the **Sketch** with the offset line segment
- click on the **violet line** and press the **Delete** key



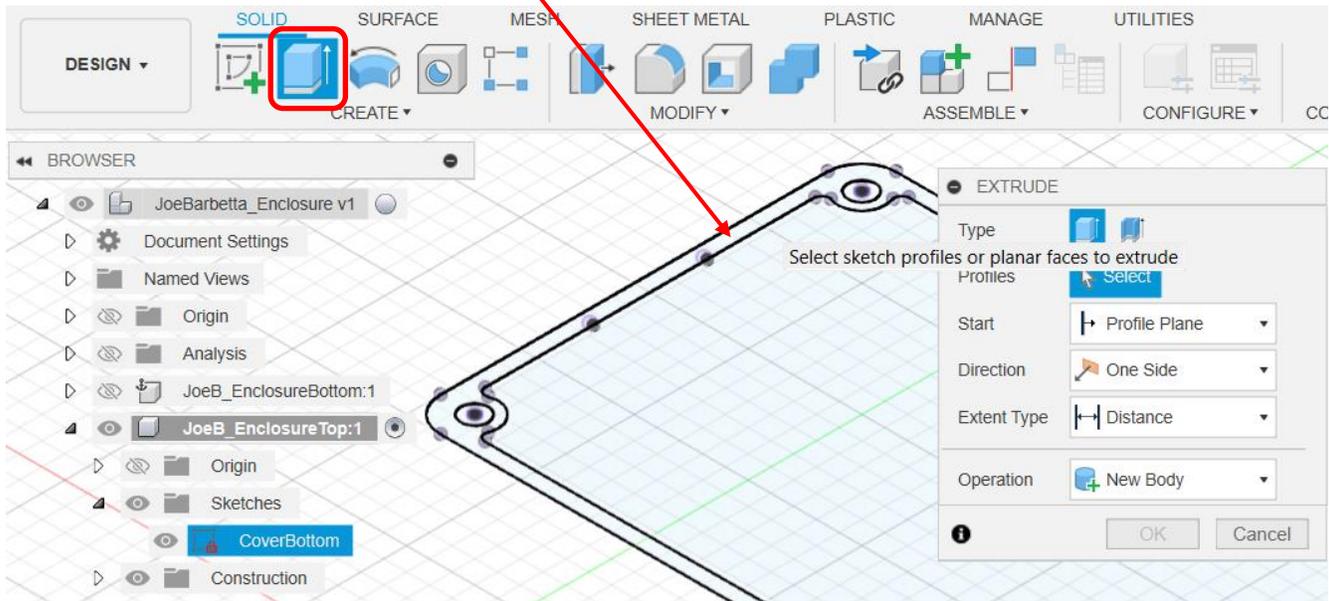
- select the **Line** tool and **create a line** to complete the violet line segment



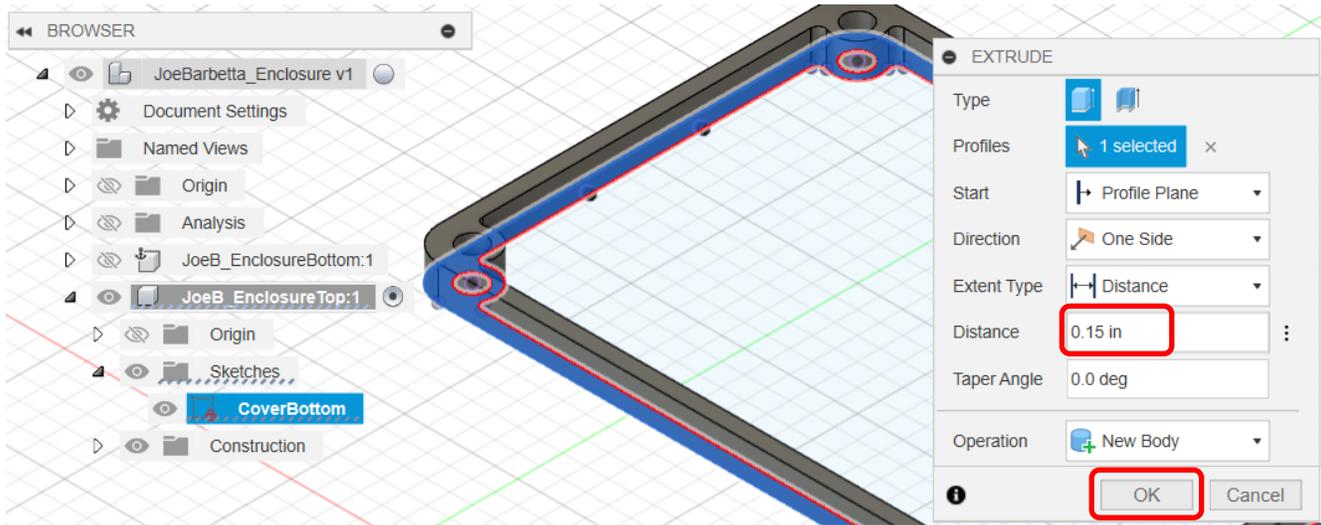
- click **Finish Sketch**



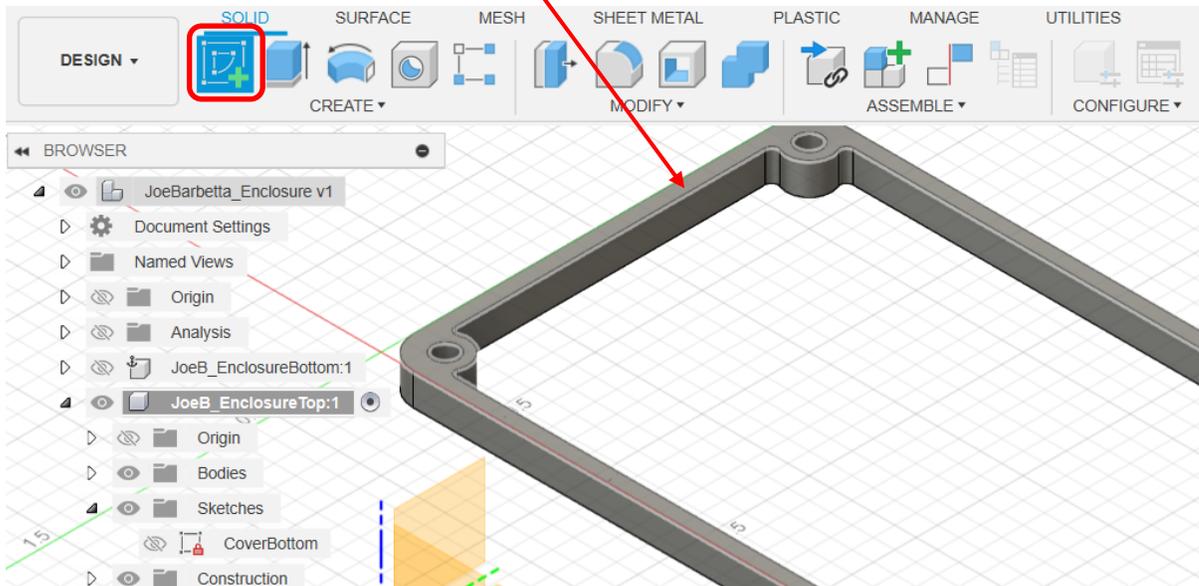
- select **Create Sketch** and click on the **outer region**



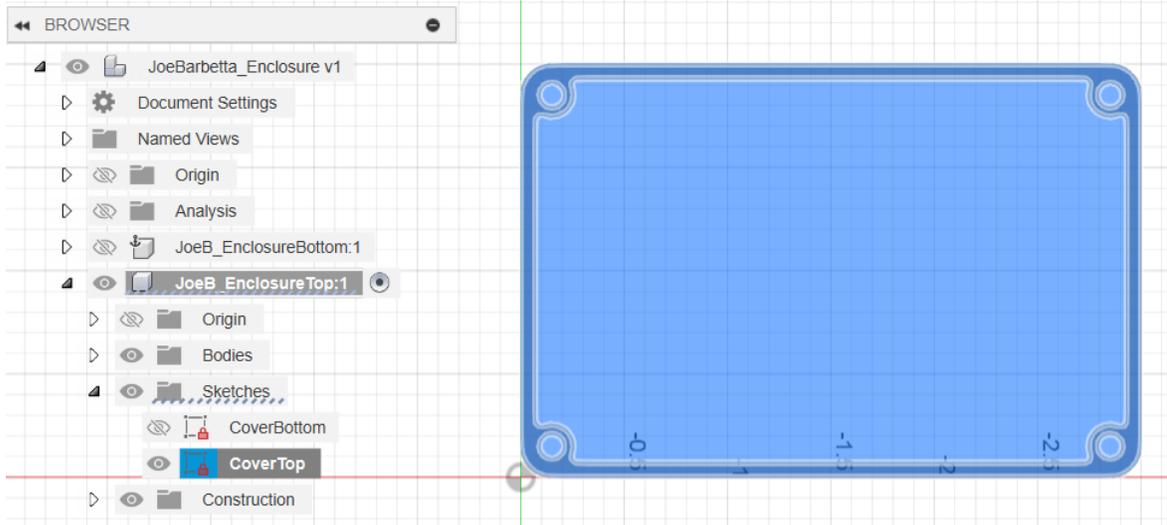
- enter a value of **0.15** for **Distance** and click **OK**



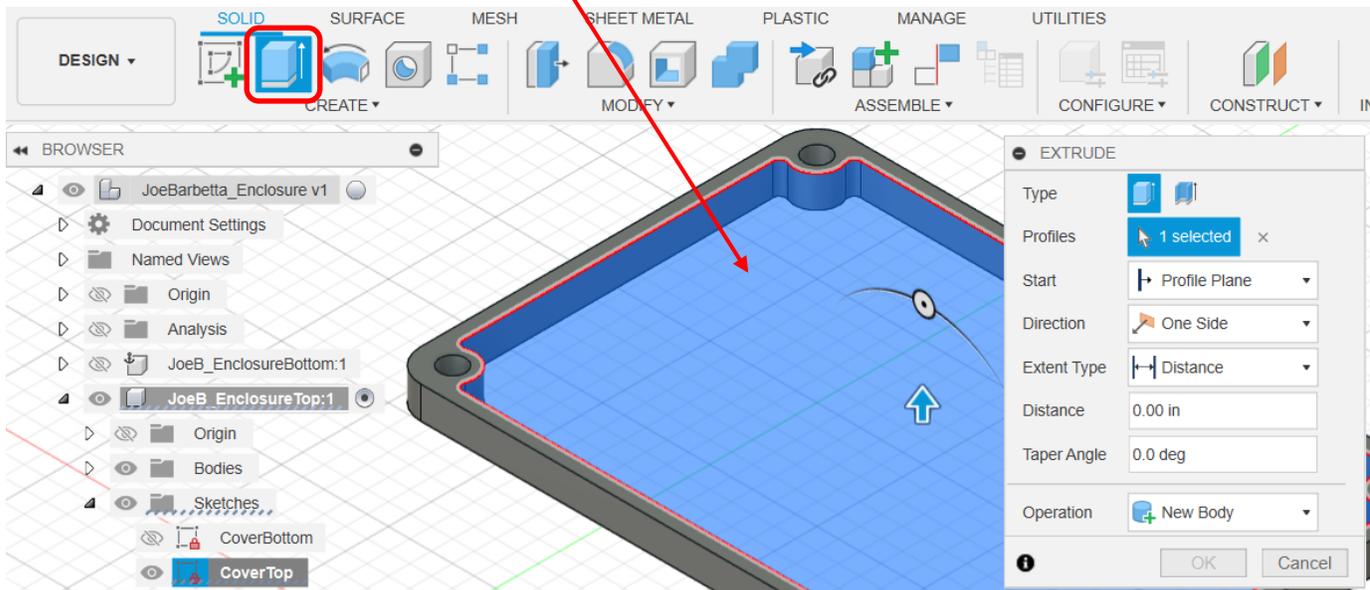
- select **Create Sketch** and click on the **top of the body just created**



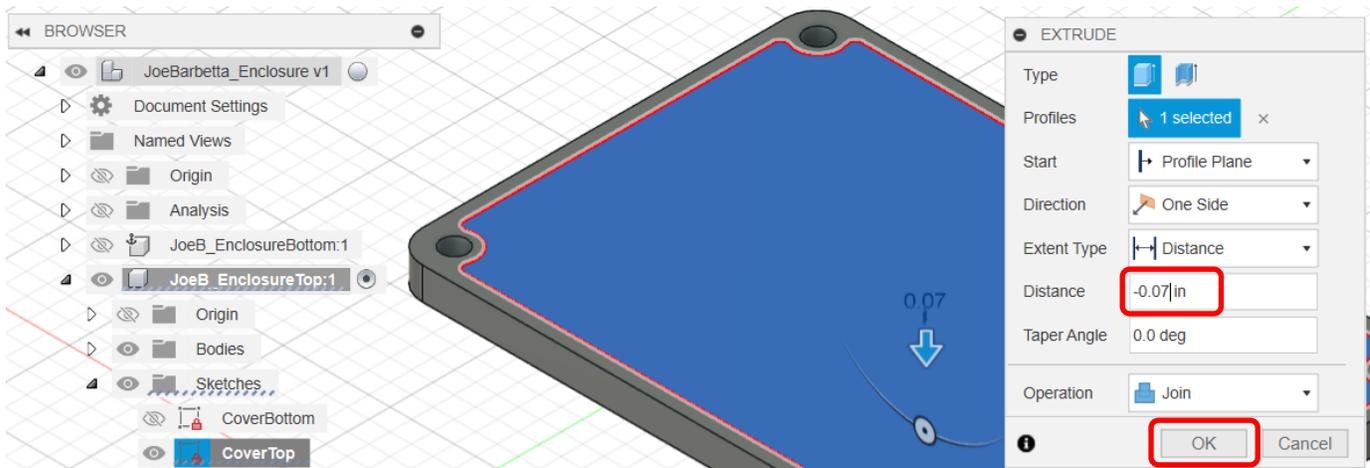
- click on **Finish Sketch**



- select the **Extrude** tool and click on the **center region**

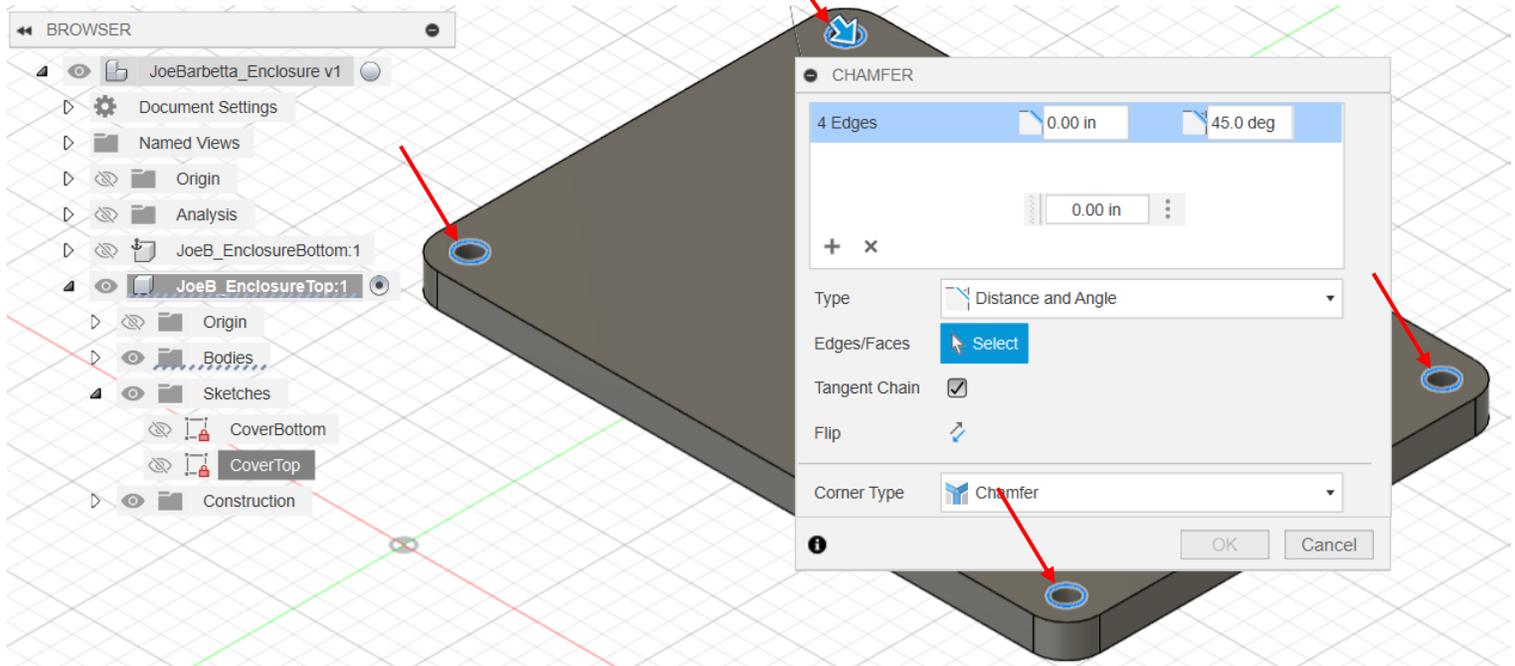


- enter **-0.07** (note the minus sign) and click **OK**

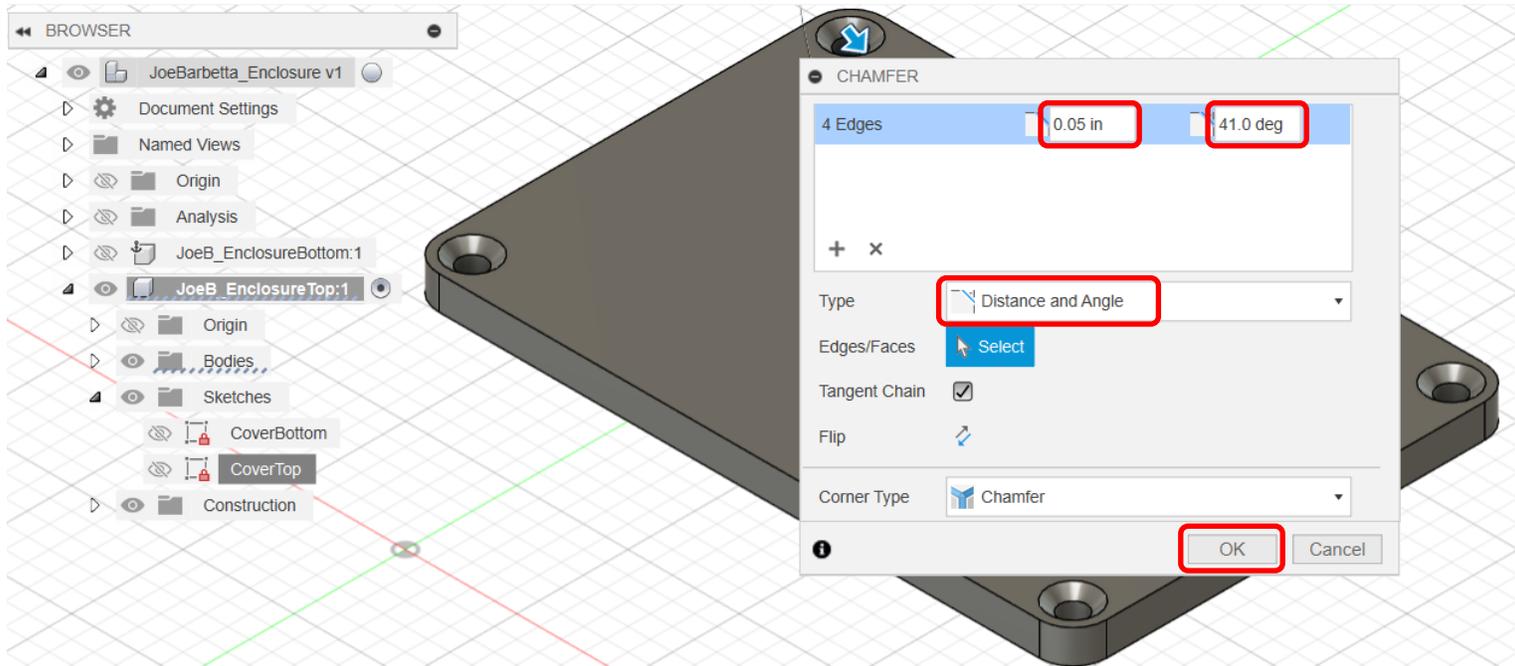


Enclosure Cover – Creating Chamfers for Flat-Head Screws

- from the **MODIFY** menu, select **Chamfer**
- click on the **edge of the 4 circles**

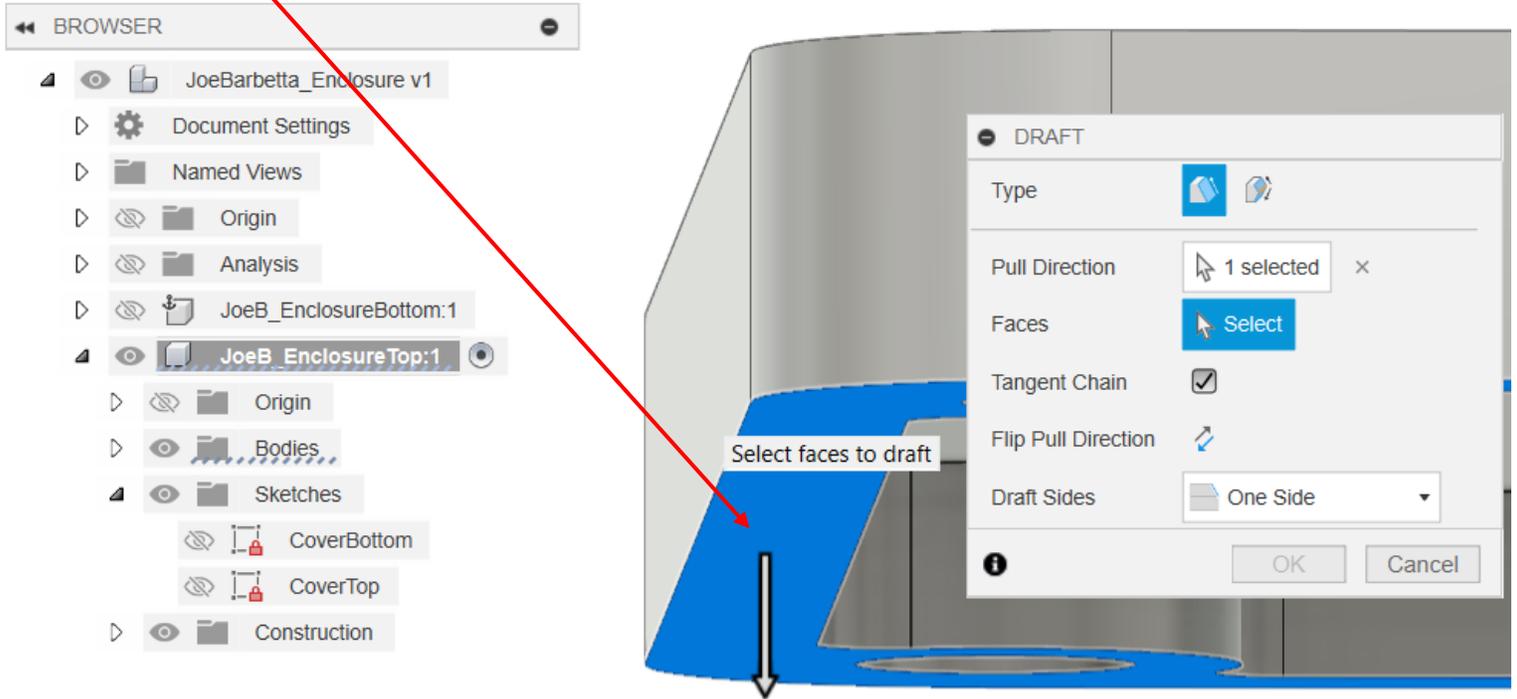


- for **Type** select **Distance and Angle**
- enter **0.05** and **41** and click **OK**

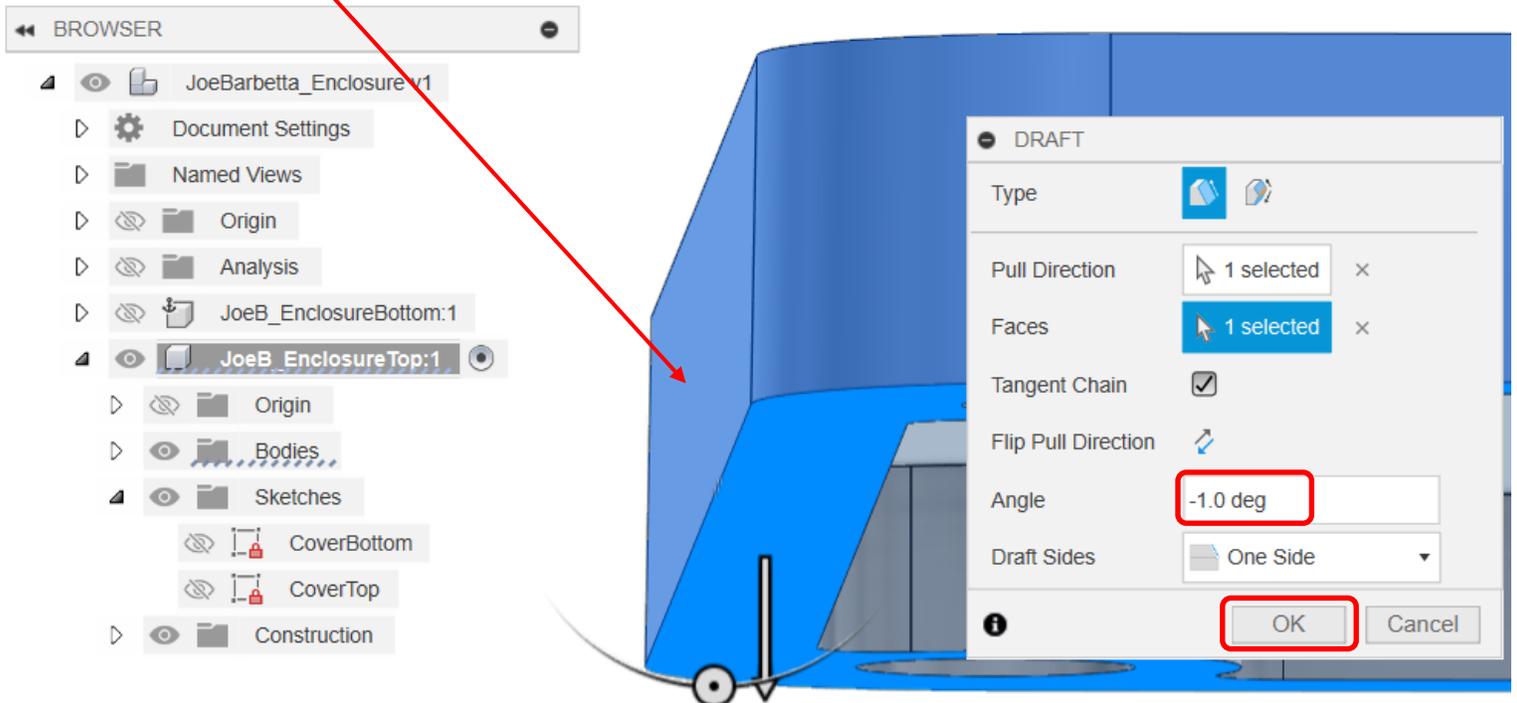


Enclosure Cover – Adding Draft Angles

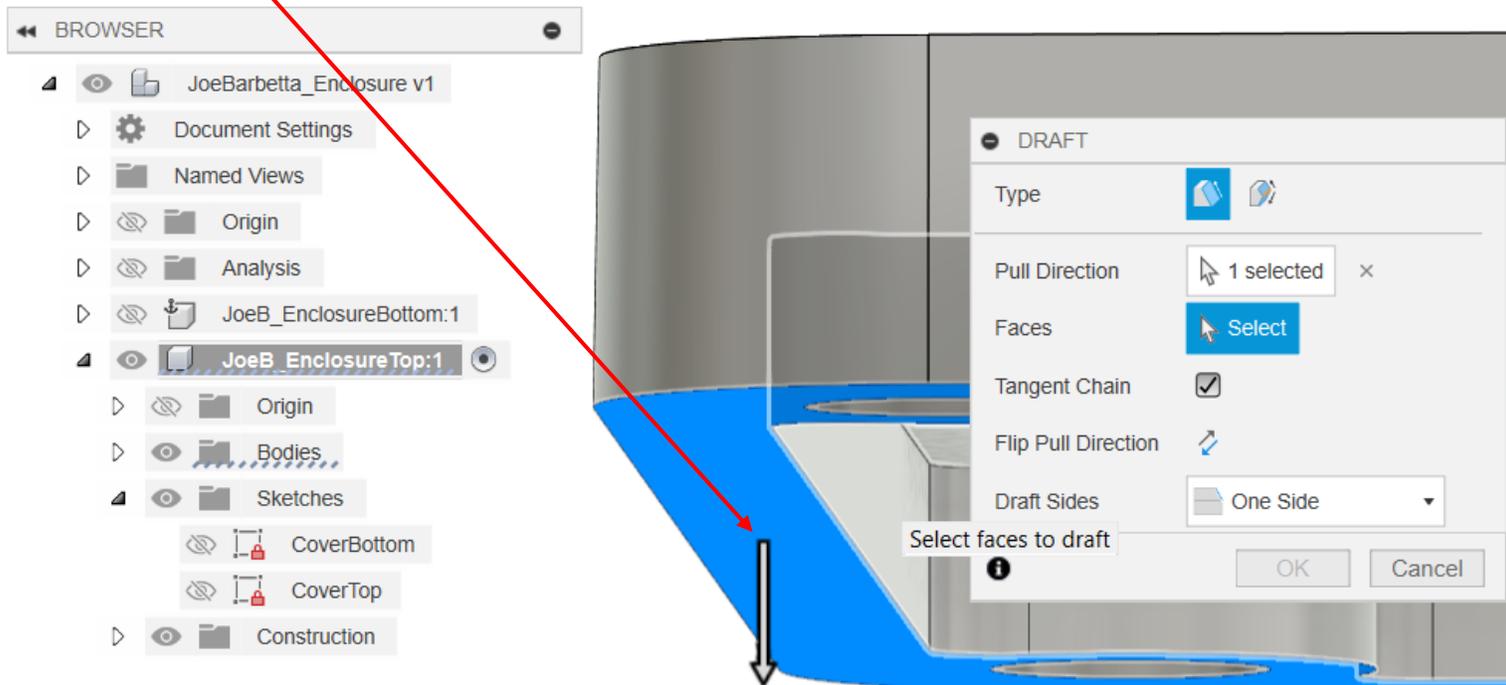
- zoom in to the **bottom of the cover rim**, as shown below
- from the **MODIFY** menu, select **Draft**
- click on the **rim bottom surface**



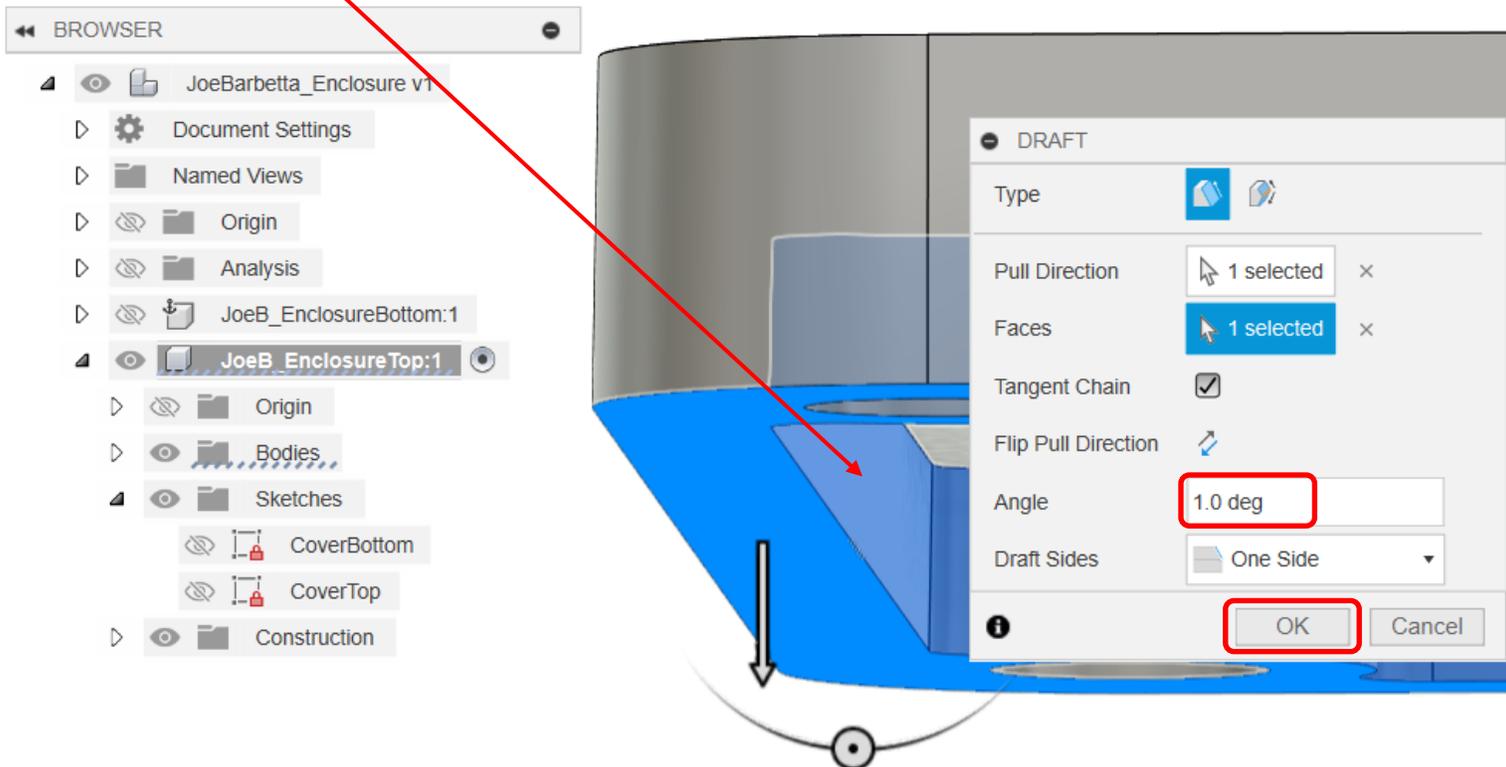
- click on **outer side surface**
- for **Angle** enter **-1.0** (note the minus sign) and click **OK**



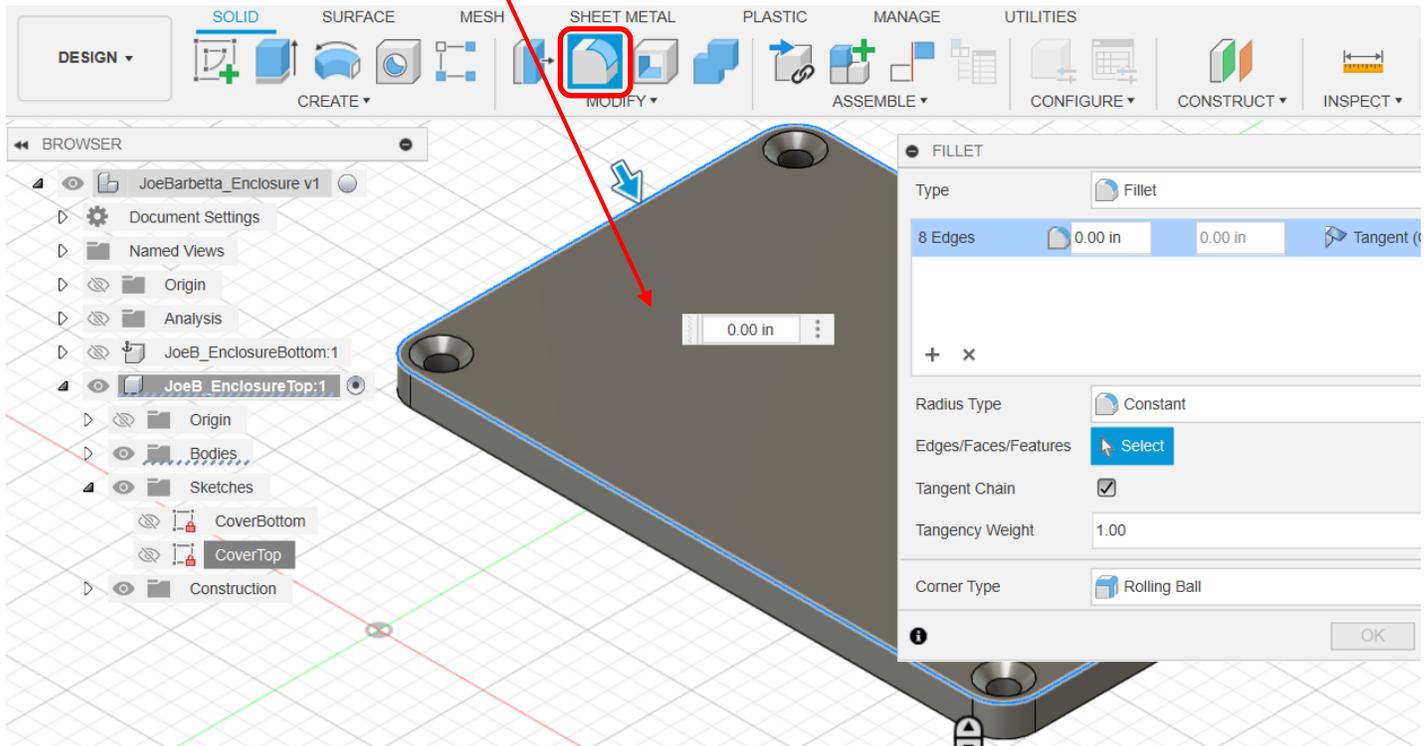
- adjust the view to access the interior wall of the rim, as shown below
- from the **MODIFY** menu, select **Draft**
- click on the **rim bottom surface**



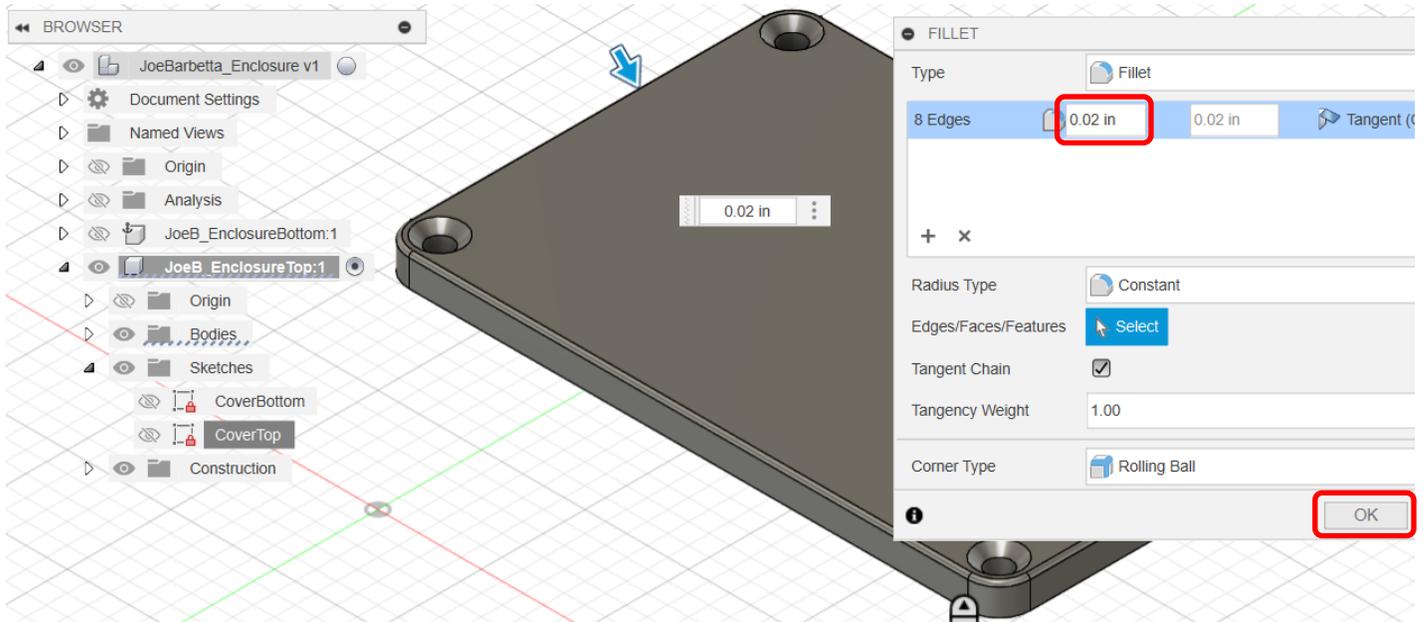
- click on **rim inner surface**
- for **Angle** enter **1.0** (positive value) and click **OK**



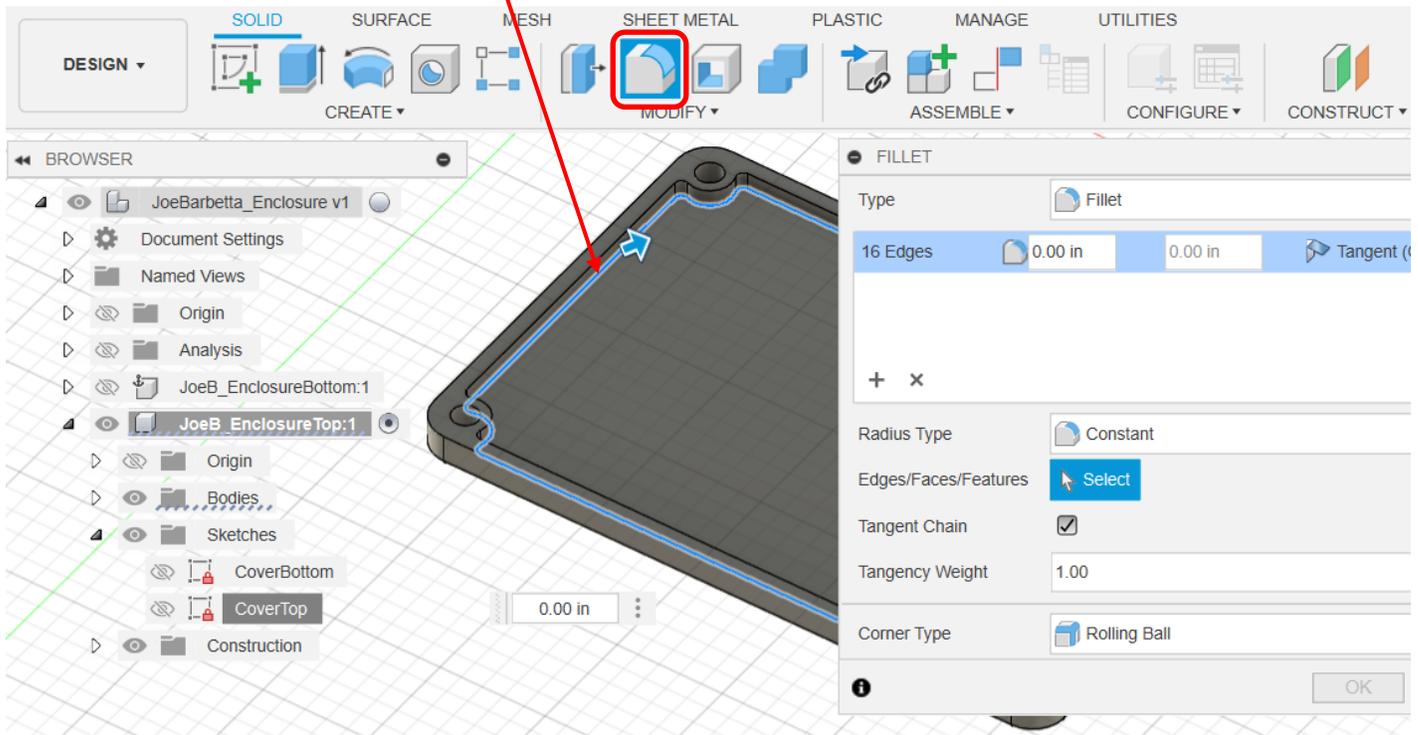
- select the **Fillet** tool and click on the **top edge** of the cover



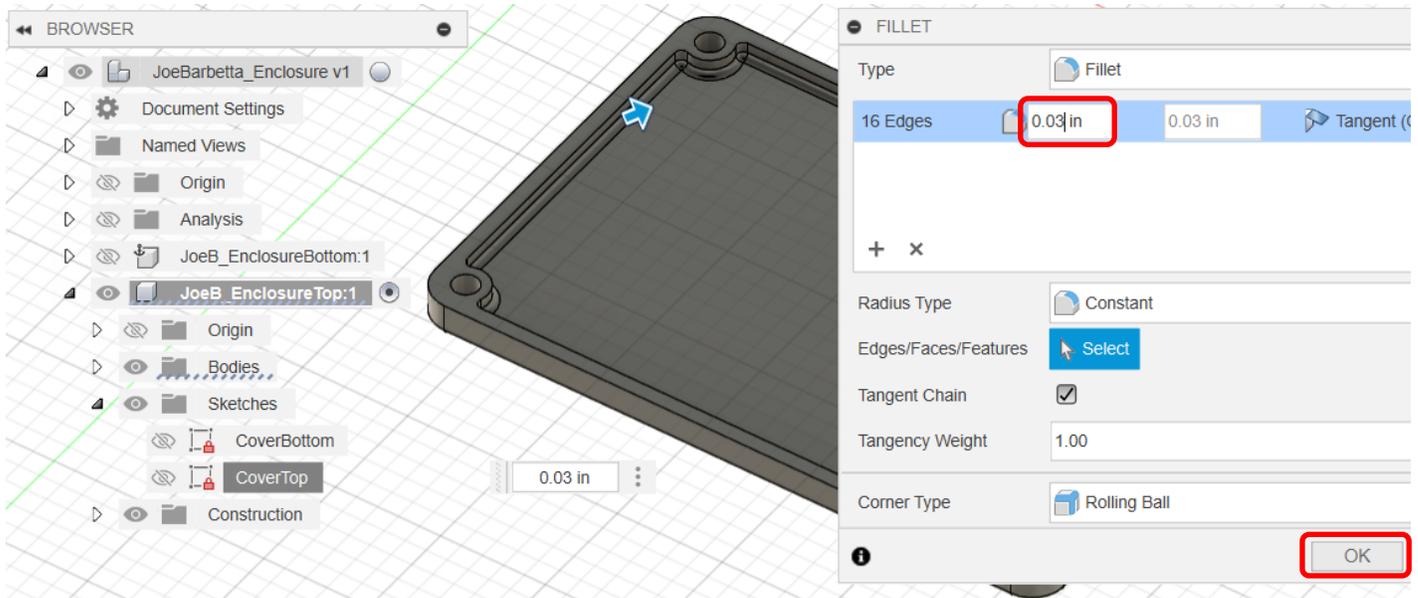
- enter **0.02** and click **OK**



- adjust the view to access the underside of the cover
- select the **Fillet** tool and click on the **inner edge** of the cover

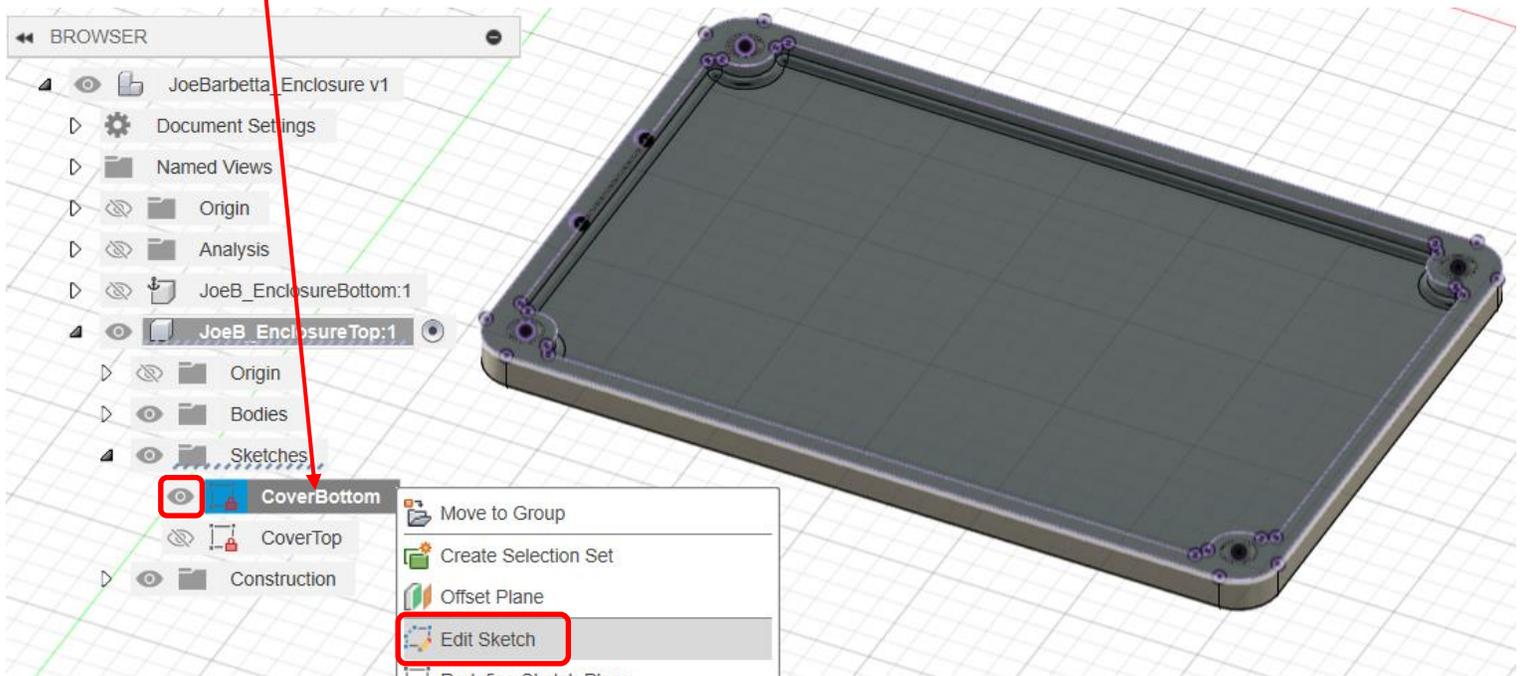


- enter **0.03** and click **OK**

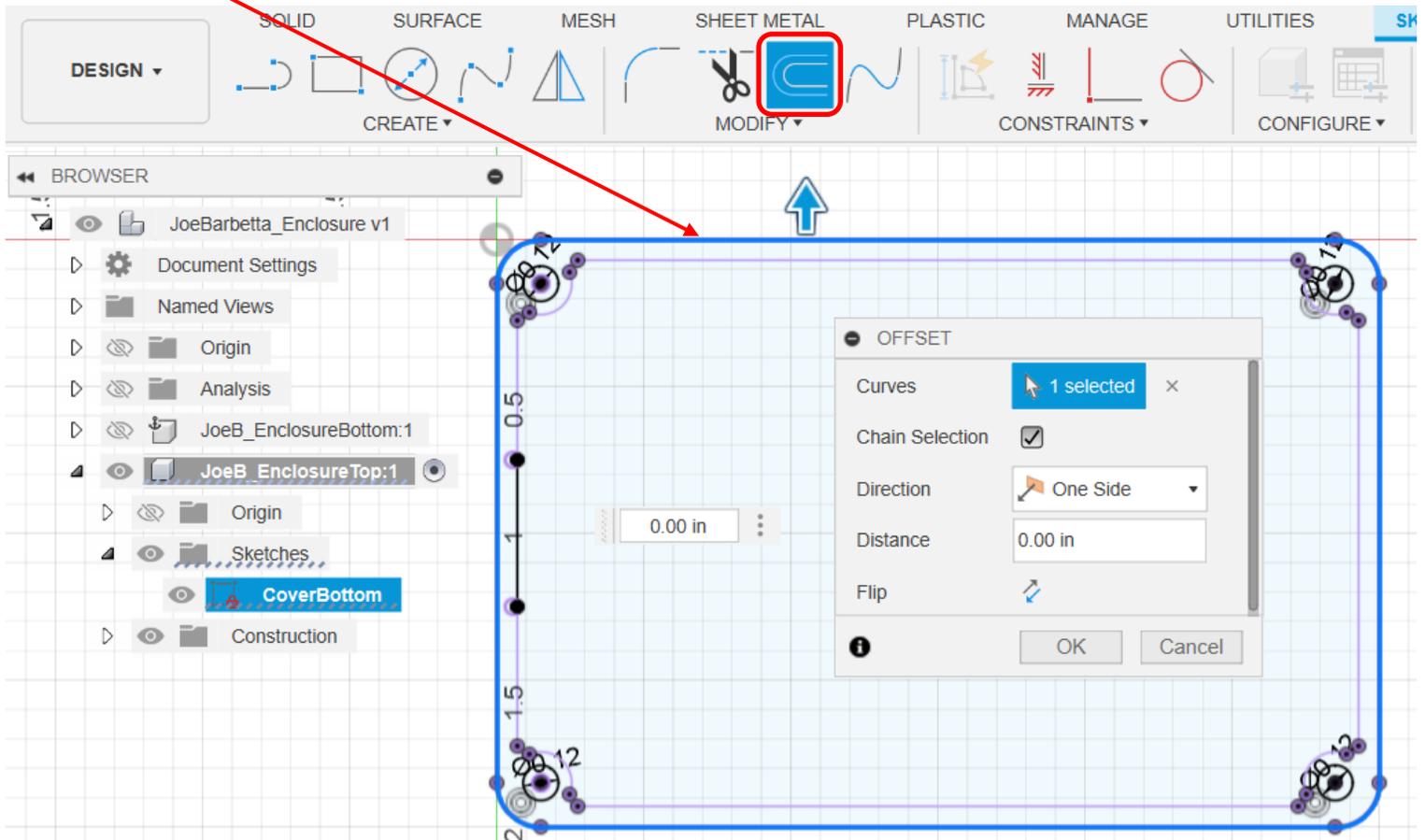


Enclosure Cover – Refining the Cover Interface

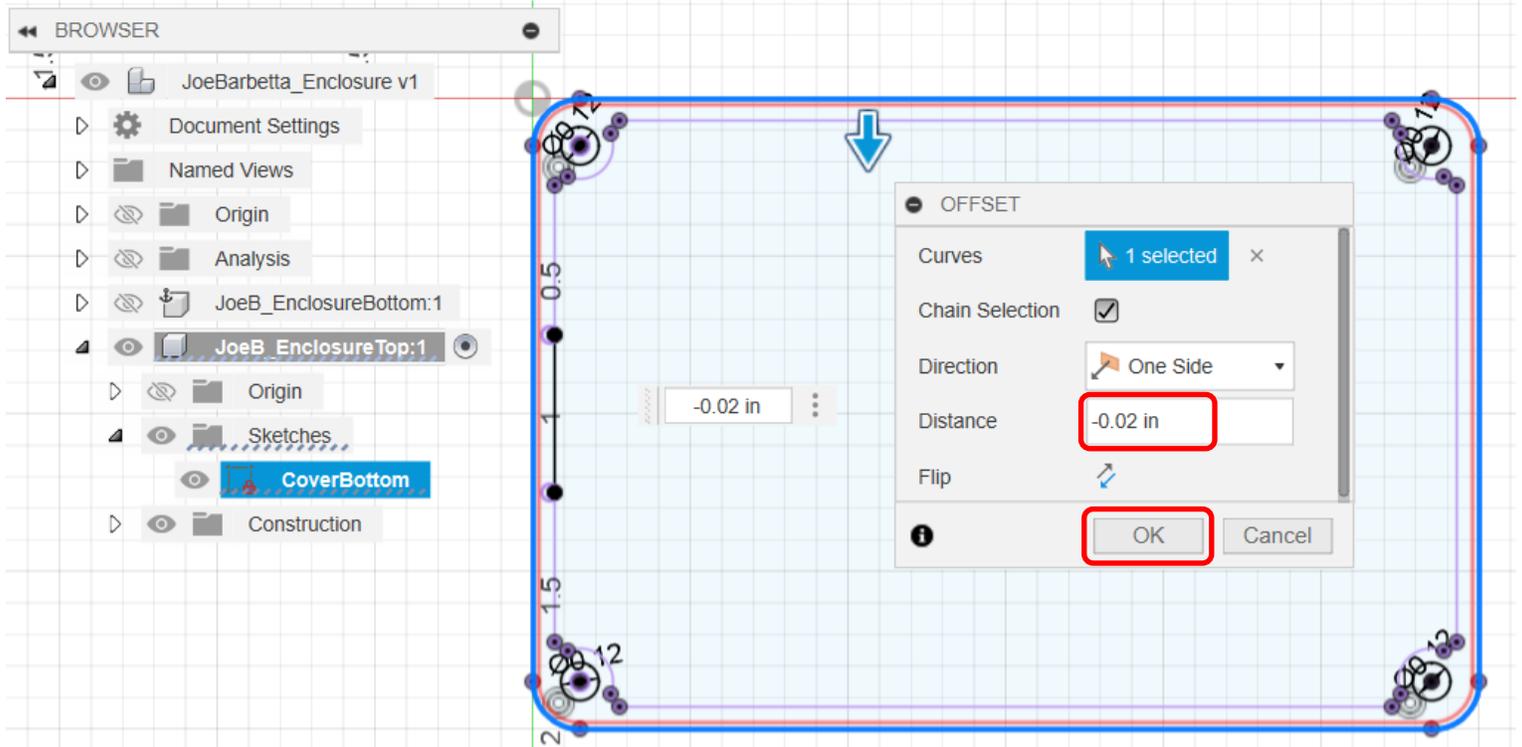
- click on the **eye** icon for the **CoverBottom** Sketch to make it visible
- right-click on **CoverBottom** and select **Edit Sketch**



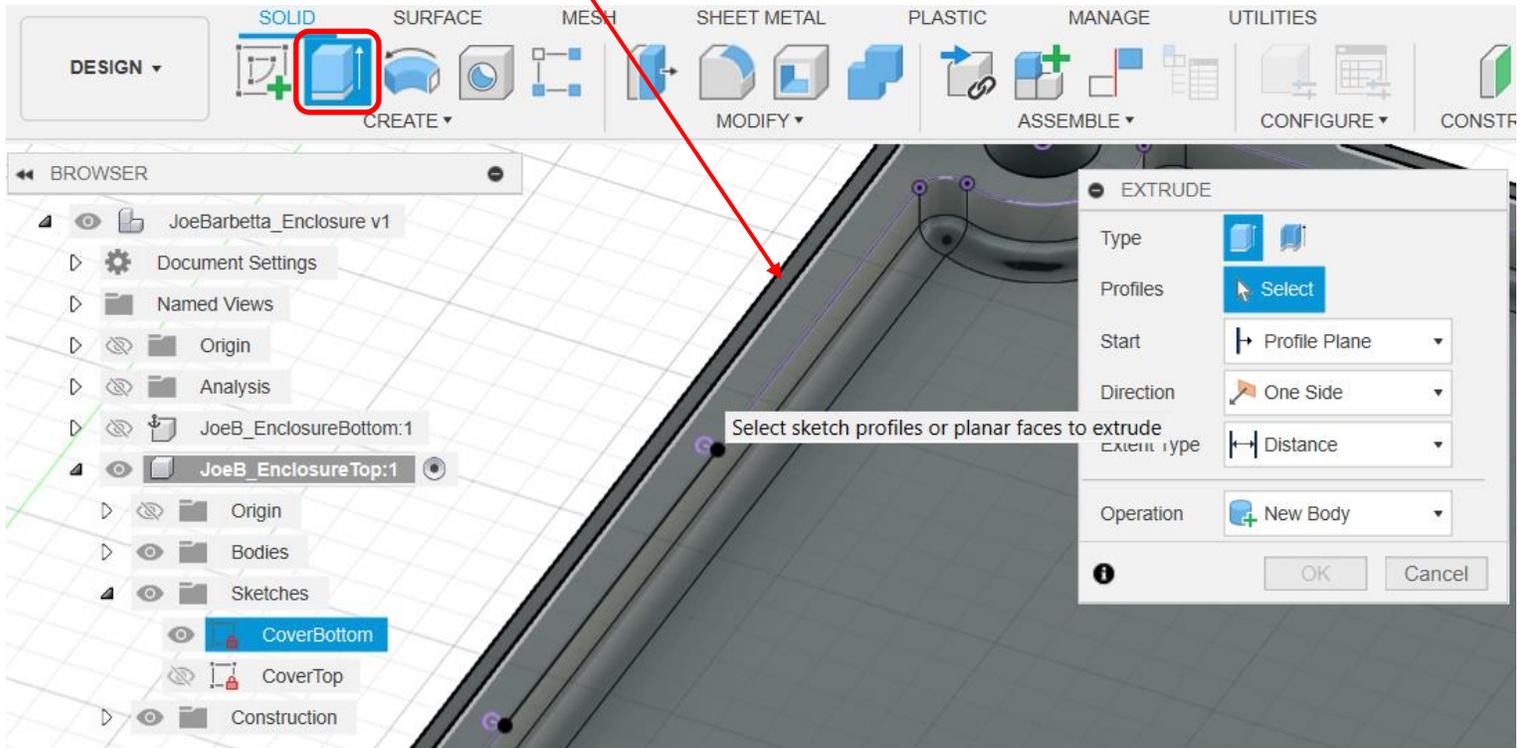
- select the **Offset** tool
- click on the **outer line** to highlight it



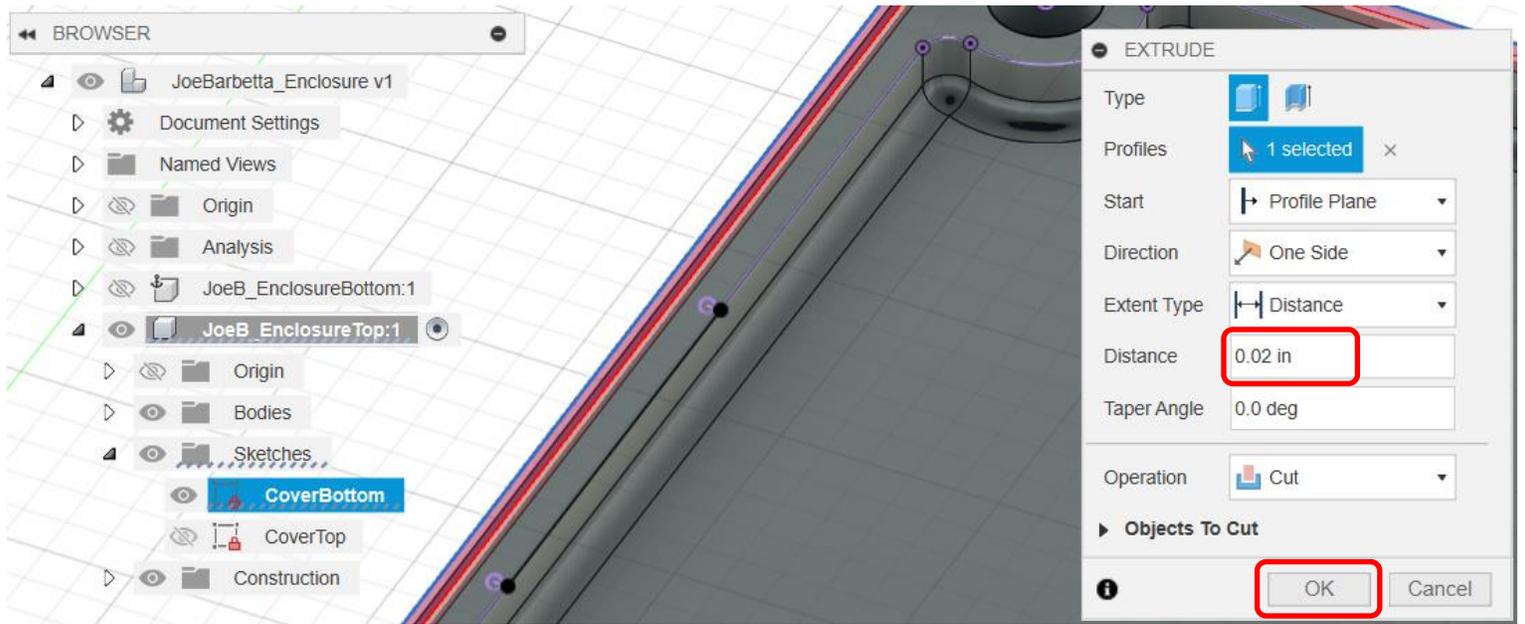
- for Distance enter **-0.02** (not the minus sign), which should result in an inner red line, and click **OK** and **Finish Sketch**



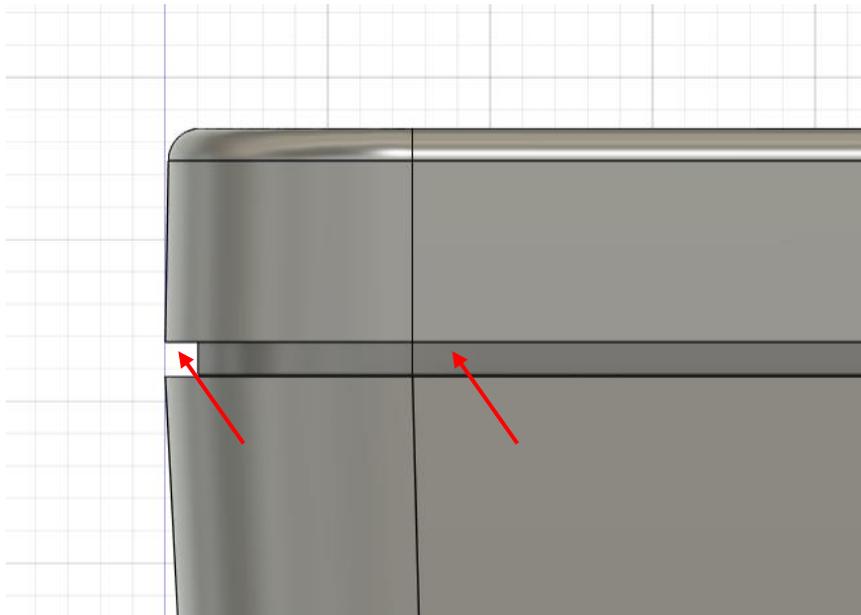
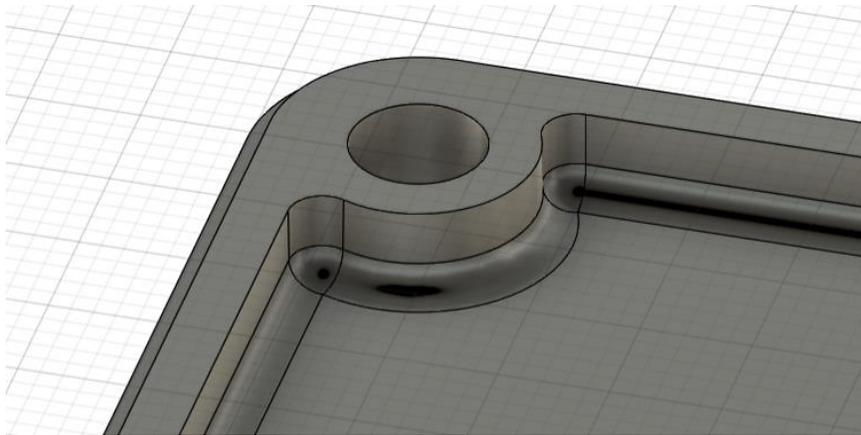
- select the **Extrude** tool and click on the **thin outer region**



- for **Distance** enter **0.02** and click **OK**



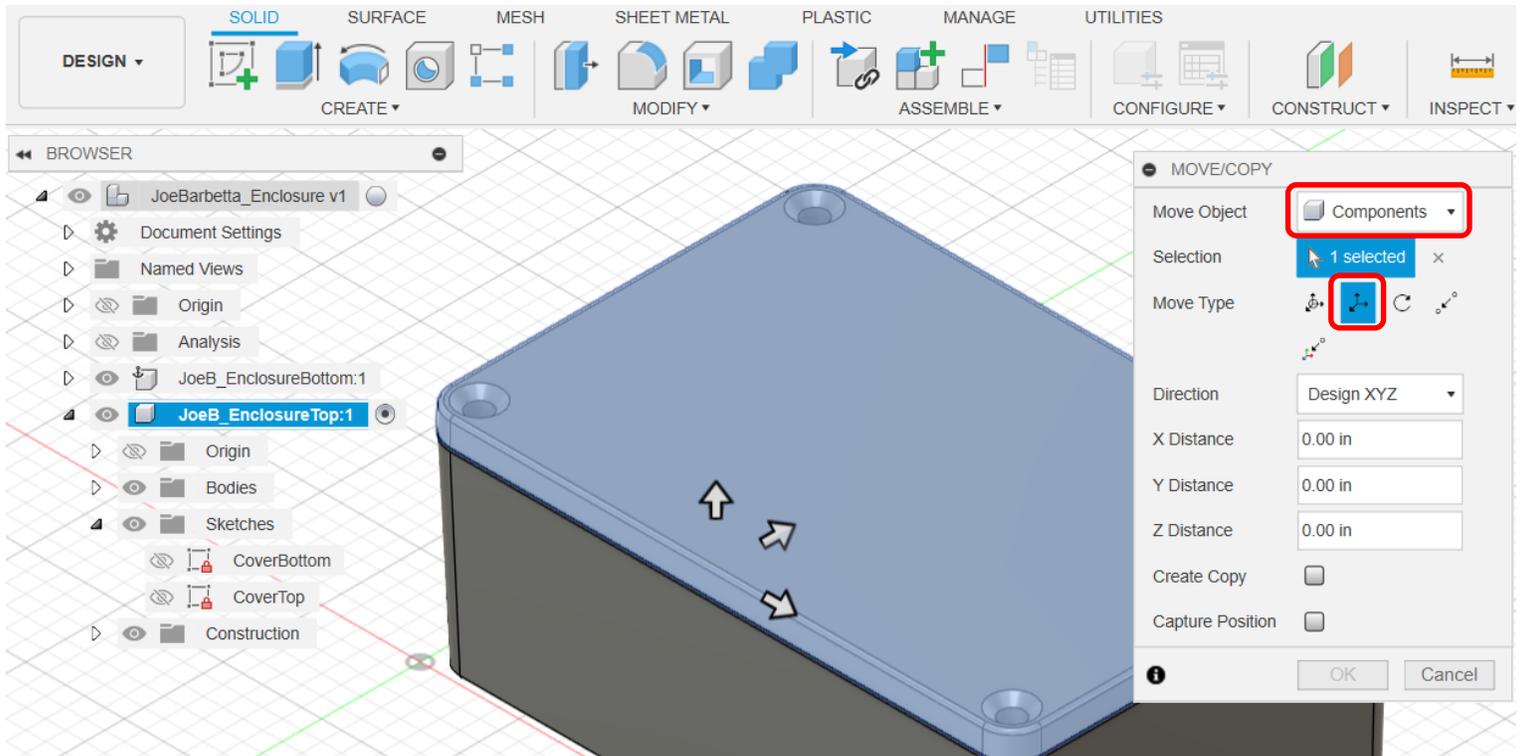
This is a close-up of the result.



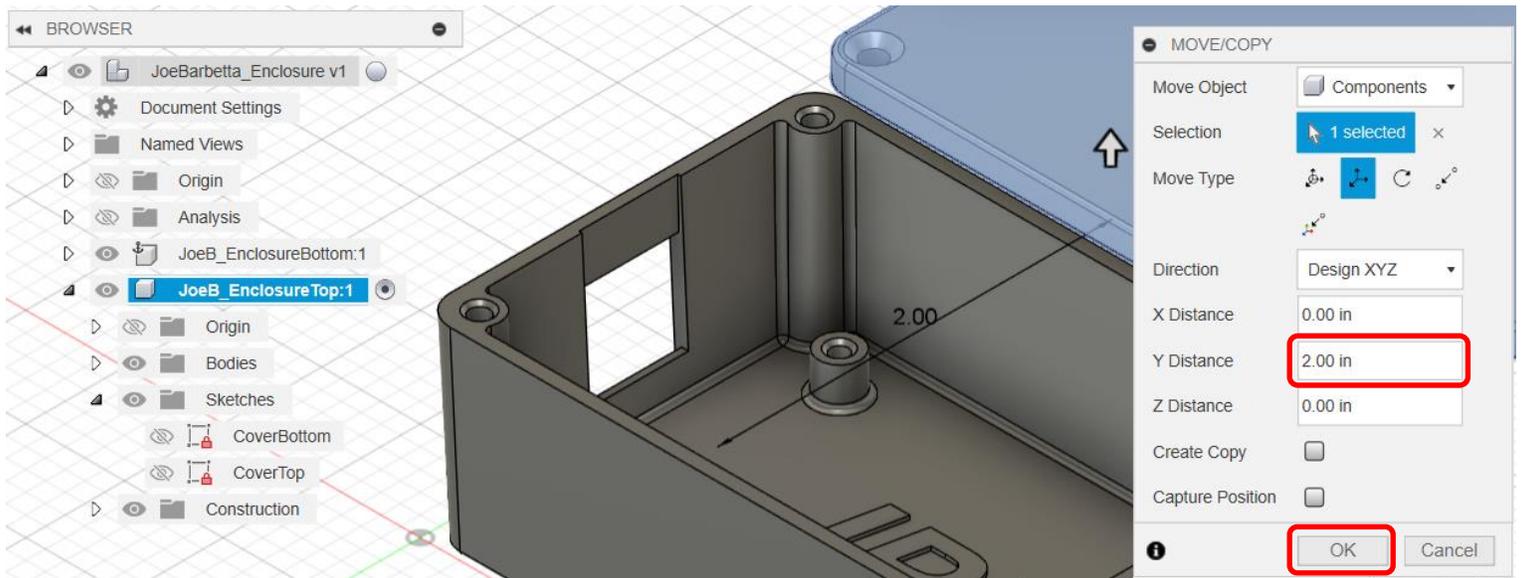
This is a side view of the cover and enclosure bottom.

This channel can add to the aesthetics because it can help hide slight mismatches of the bottom and cover walls.

- from the **MODIFY** menu, select **Move/Copy**
- click on the **Cover**
- ensure that **Move Object** is set to **Components** and the **Translation** icon is selected for **Move Type**



- set **Y Distance** to **2.00** and click **OK**
- If the **Cover** moves in a different direction, the enclosure may have been created on different axes. **X Distance** or **Z Distance** may have to be used instead.



- click on the **circle** for the **Project Name** to activate it
- click on **arrows** to ensure that **each Component is open** and that **all Sketch Names are visible**
- adjust the view to be similar to that below to allow your **initials**, the **side hole**, **2 bottom bosses**, and the **cover** to be visible
- take a screenshot for the submission
- at home print out the screenshot to hang on your refrigerator

