# Fusion 360 Modeling - Modeling from a Print

Fusion 360 Modeling - Modeling from a Print. Modeling Introduction. - YouTube

This tutorial creates a Solid model – as opposed to a Surface model, which is nothing more than a skin to define a shape. This tutorial is inches, and if your default measurement is in mm, you can change it by selecting the personalization icon in the upper right corner, followed by Preferences / Default Units / Design and enter "in".
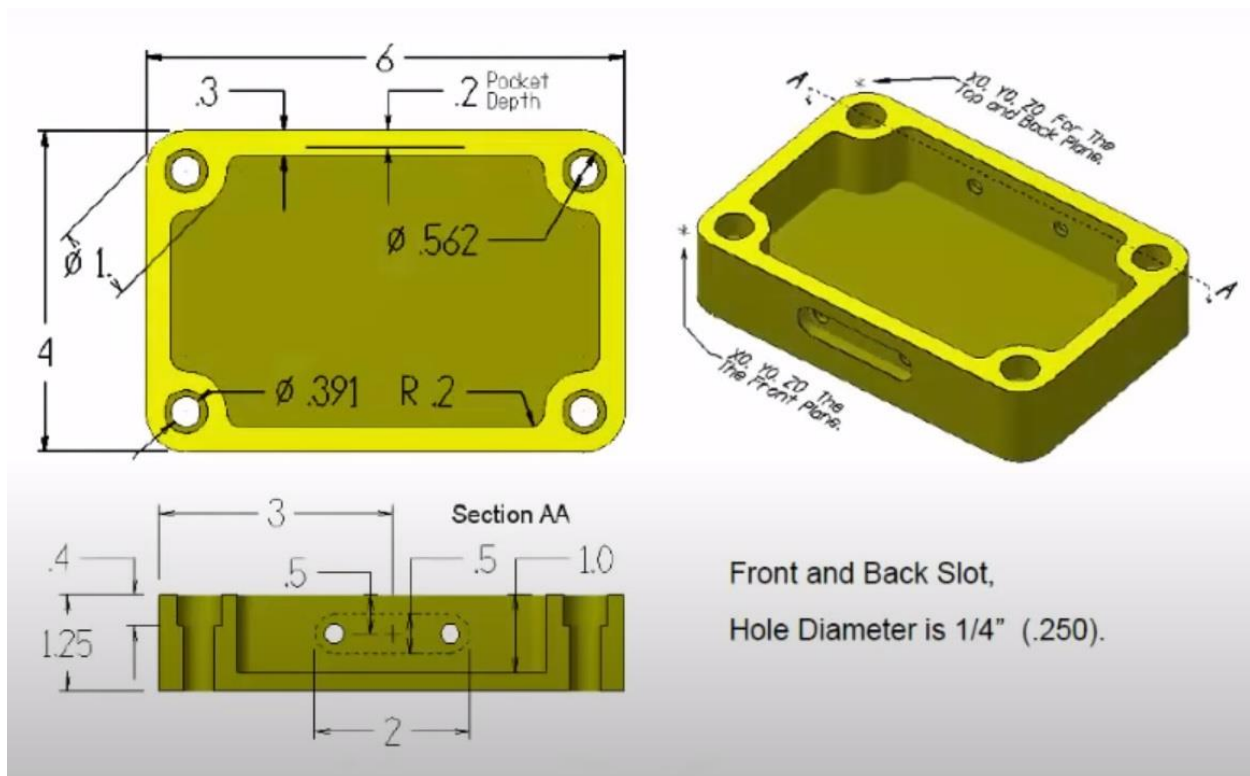
(Part Drawing)

1. **Create a New Sketch:** Orient it on the bottom plane to get a top-down view of the model. (X/Z plane)
2. **Create a Two-Point Rectangle:** Select the *Create / Rectangle* tool and place the first point on the origin, then drag outwards and down. Make it 6" wide and 4" deep. One way to build it is to begin drawing it out and rather than dimensioning it after creating the shape, enter 6" in the width entry field and then **_tab_** to the depth entry field and enter 4".
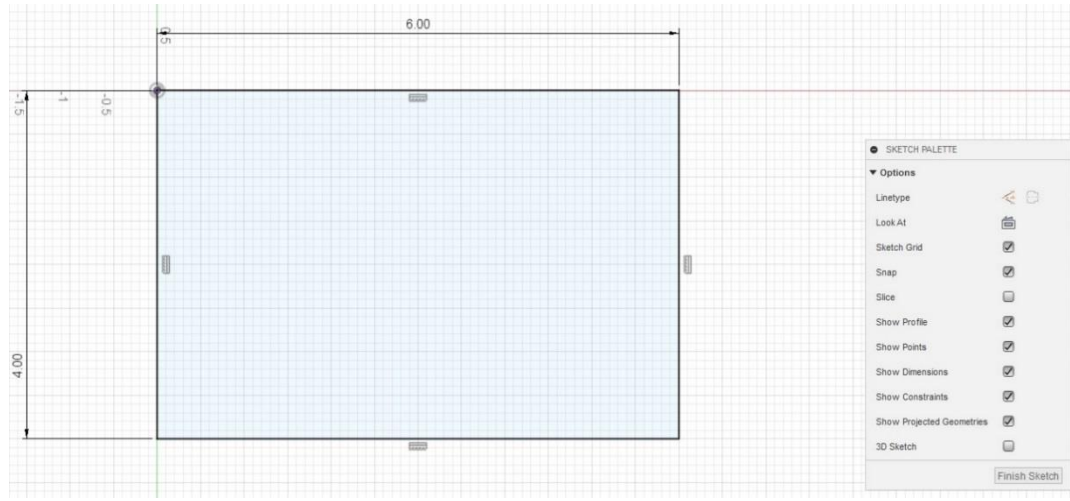
3. **Extrude the Rectangle:** Close the sketch by left-clicking on the "*Finish Sketch*" button on the *Sketch Pallet* panel. Select the *Perspective* view by left-clicking on the *Home* icon in the upper/right corner adjacent to the view cube.

Select the *Extrude* tool (*Create/Extrude*). Extrude the rectangle 1.25".
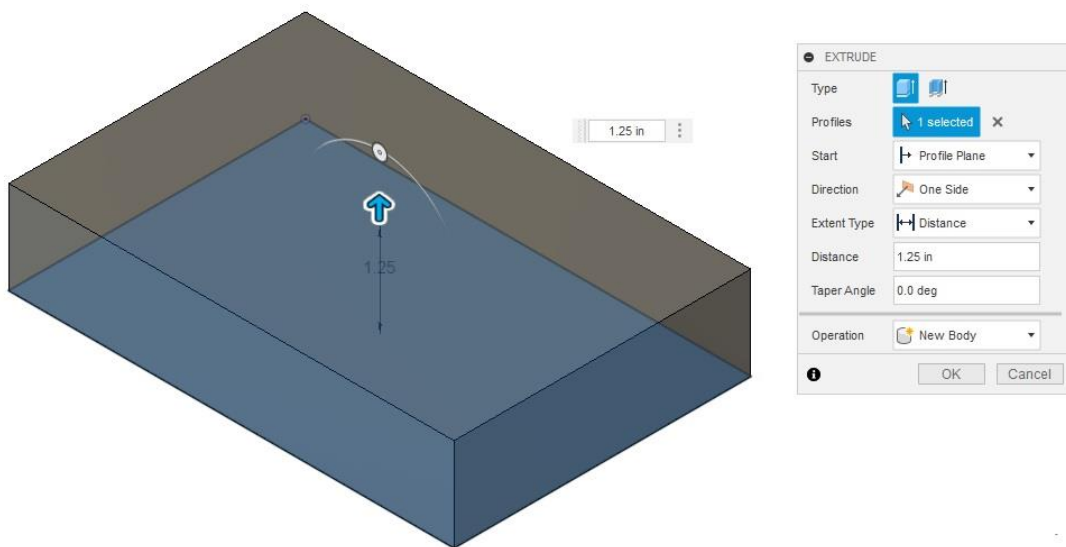

(Extruded Rectangle)

4. **Round all Corners:** Select the *Fillet* tool (*Modify/Fillet*) to make each corner .5".
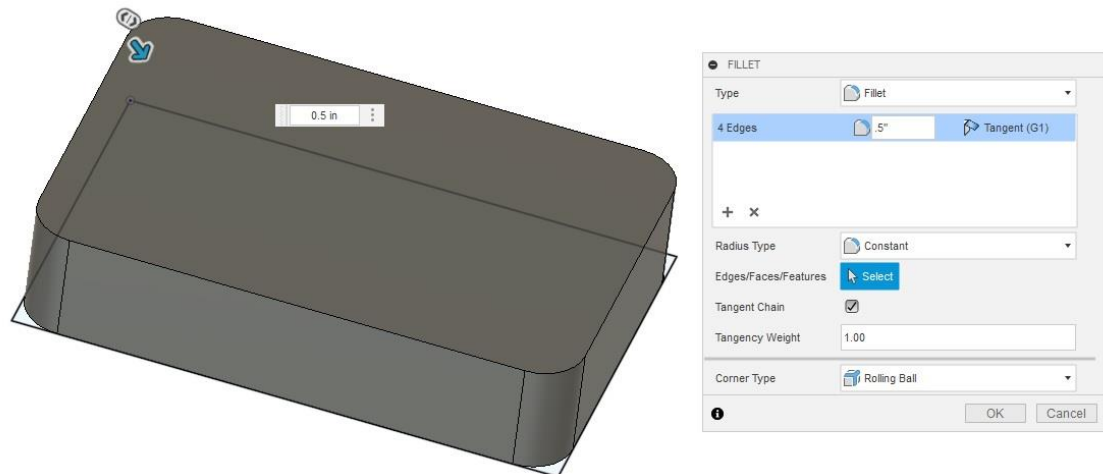
(Extruding the Four Corners)

5. **Begin Creating the Pocket for the Inside Profile**: Create a *New Sketch* on the same plane as the rectangle, but instead of selecting one of the three planes as before, *select the top surface of the rectangle*, since that is the surface that will be inset. The view will change to the *Top* view.
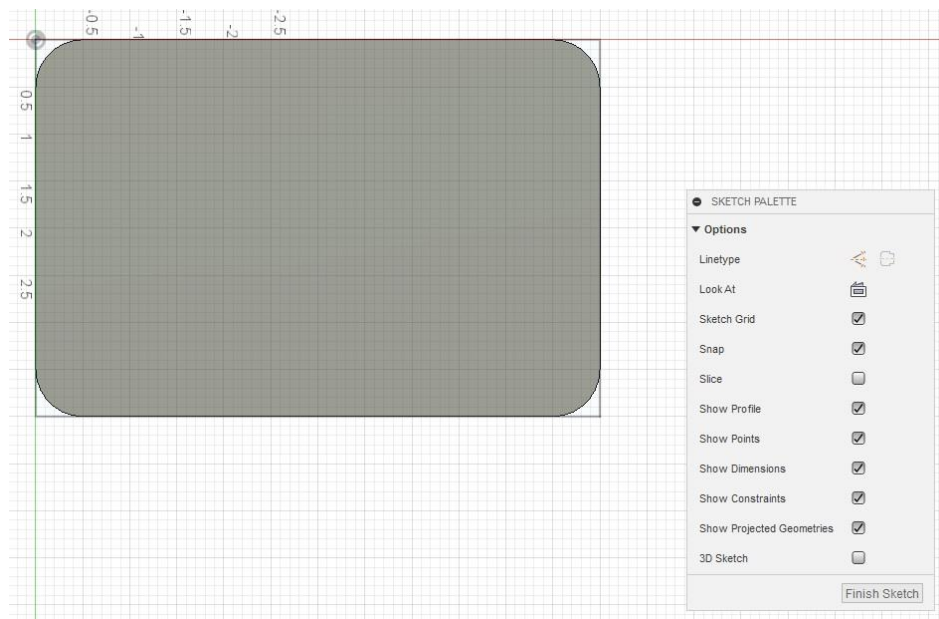

(Top View Orientation)

Create the Outer Wall Thickness: Switch to the *Perspective* view again. The outer wall thickness needs to be .3". Select the *Offset* tool (*Modify/Offset)* and left-click on the outer edge of the top surface. Keep

Select Chain checked in the *Offset* panel which selects the entire perimeter, when one edge is selected. Enter **-.3**" for the offset.

Tip: Note that four points are placed in each corner indicating the center of the radius. These will be used as reference points in the next step.
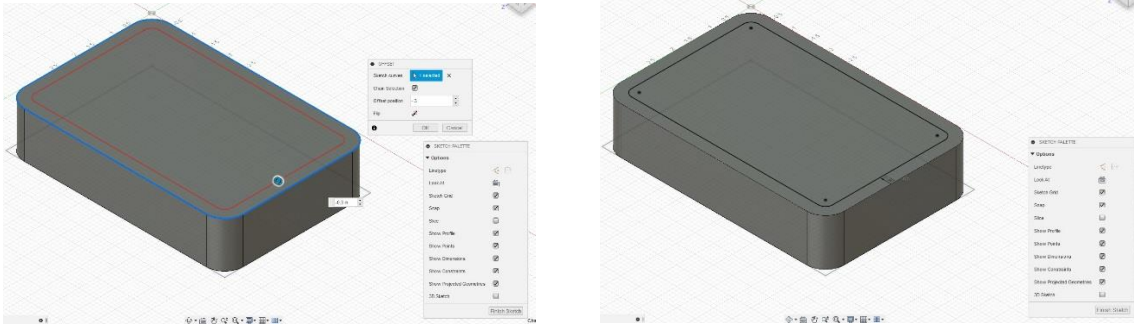

(Outer Wall Thickness Defined)

6. **<u>Create the Inside Corner Rounded Edges</u>**: Return to the *Top* view and select the *Create / Circle* tool (choosing the *Center Diameter Circle* from the options list) and left-click one of the corner points to anchor it. Drag the circle out from the center and use 1" for the diameter in the tool panel. Repeat for each corner. The anchor points ensure the circles are constrained to the outer wall thickness profile and the circle boundaries will turn black.
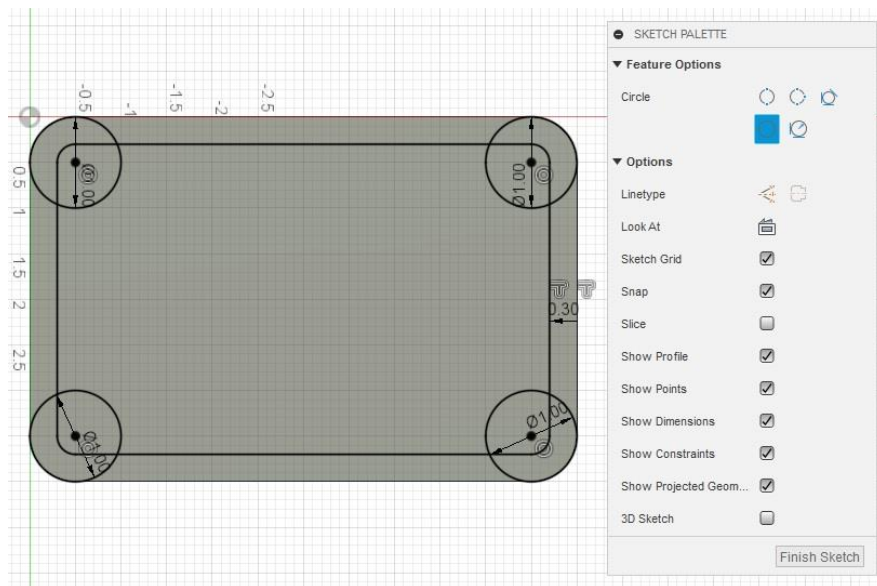

(Inside Corner Radius Created)

7. **Cut the Pocket**: Because all of the drawing lines are black, and constrained to one another, Fusion connects the outer circles and inner rectangle together. Close the sketch.

Using the ***Perspective*** view, select the center shape and using the ***Extrude*** tool *(Create / Extrude)*, making the extrusion Distance -1".
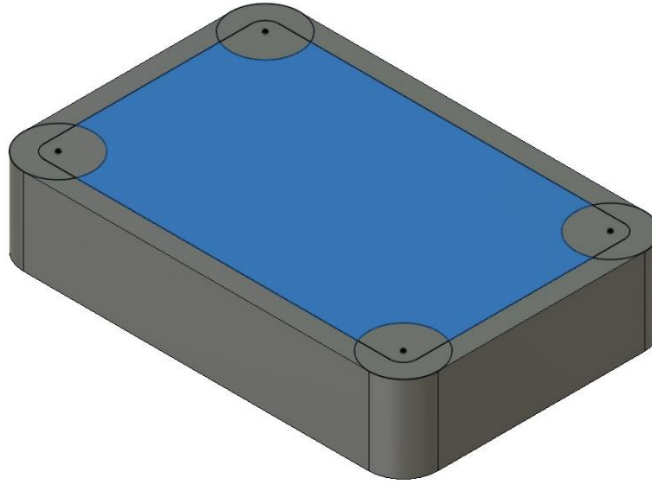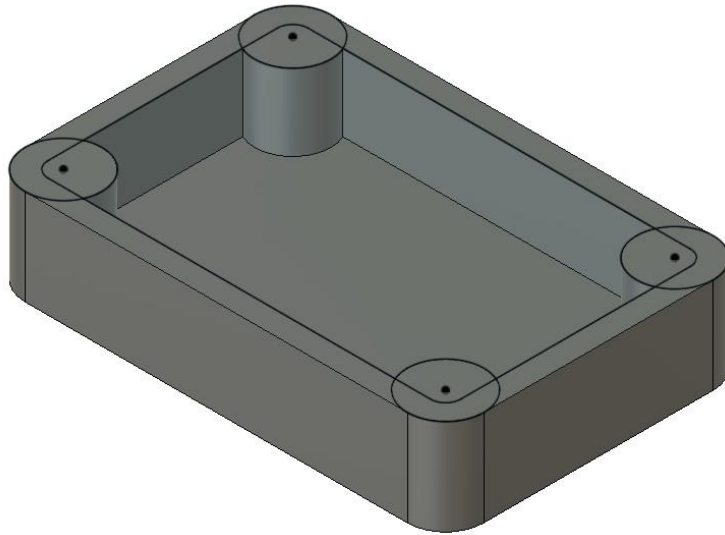

(Center Shape Selected to Create the Pocket)
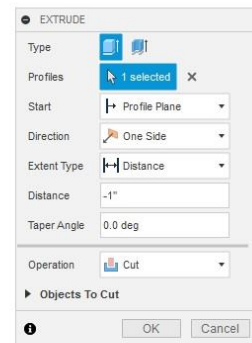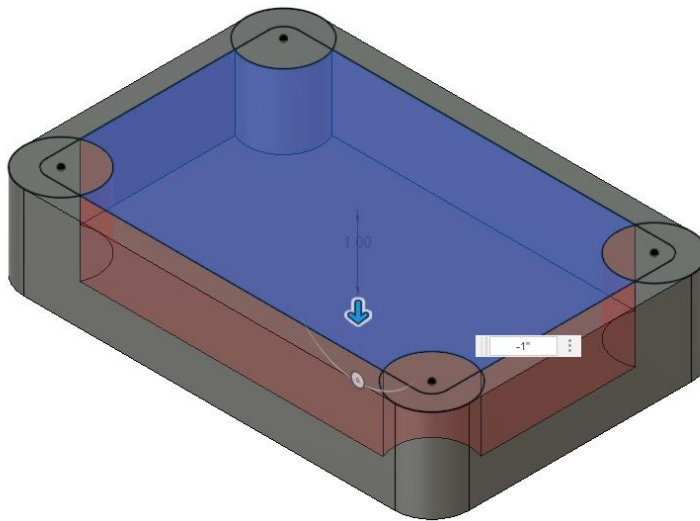

(Pocket Negative-Extruded)

Because the extrusion value is negative and moves into the solid rectangle, Fusion assumes (correctly) this is a ***Cut*** operation in the tool panel.
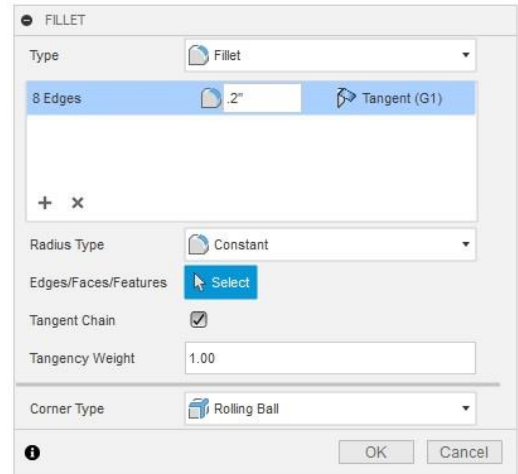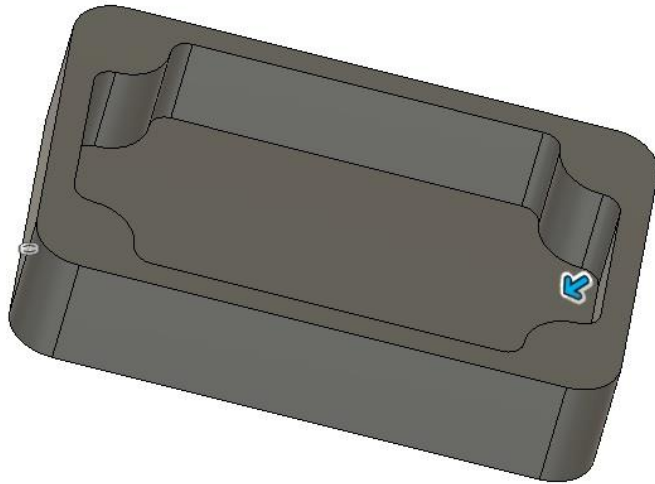
(Completed Pocket)

Add an Inside Fillet to the Corners: Select the *Fillet* tool followed by each of the eight inside corner edges and apply a .2" fillet in the tool panel entry field.

Tip: To make it easier to select the inside edges in this step, turn the Sketch visibility *off* by left-clicking on the appropriate sketch in the browser.
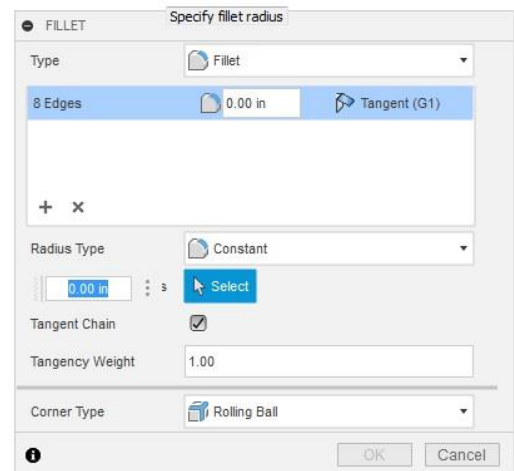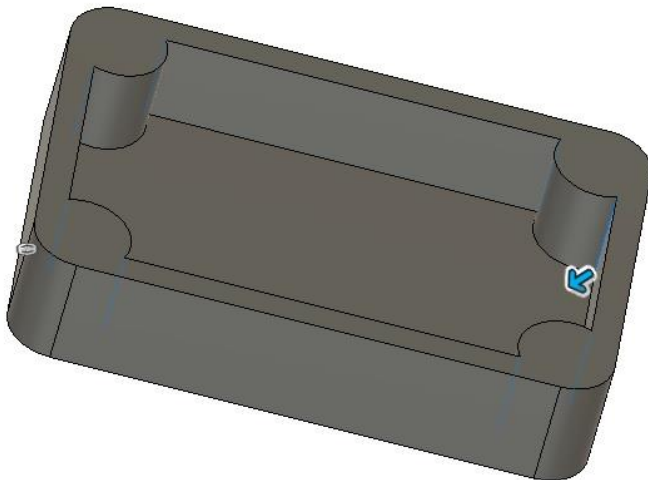

(Inside Edges Selected)

(Fillet Applied to Inside Edges)

8. **Create Reference Points to Reference the Corner Counter-Bored Holes**:
Create a *New Sketch* and select the top face of the rectangle to establish a
sketch plane. Select the *Create / Point* tool and position the cursor over the
center area of any corner. When the cursor is aligned with the center, a blue
circle appears. Left-click to place a point and repeat for the remaining three
corners.

Close the sketch.
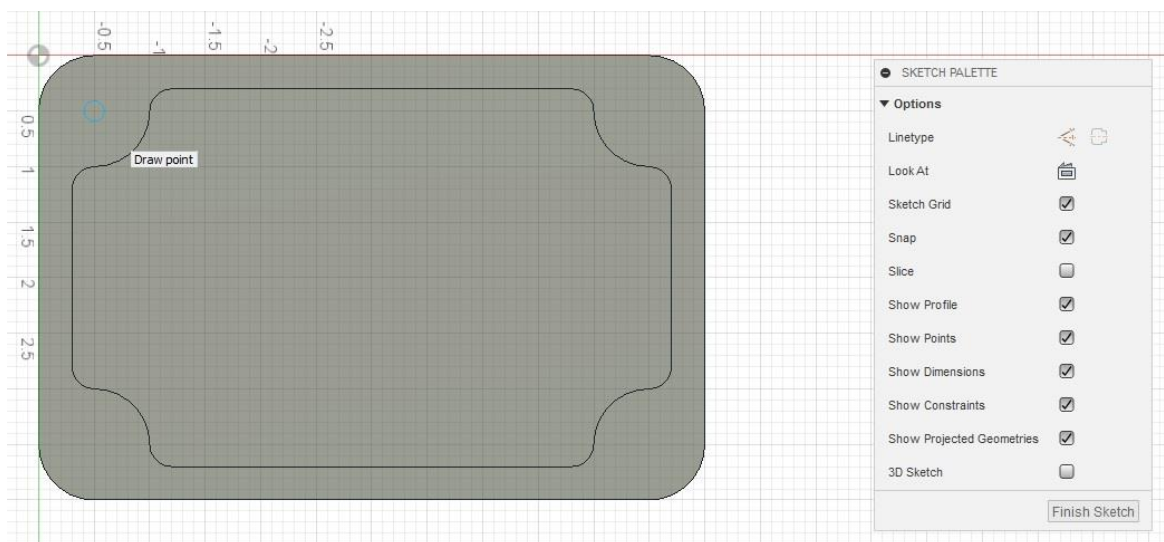
(First Corner Center Point Identified)
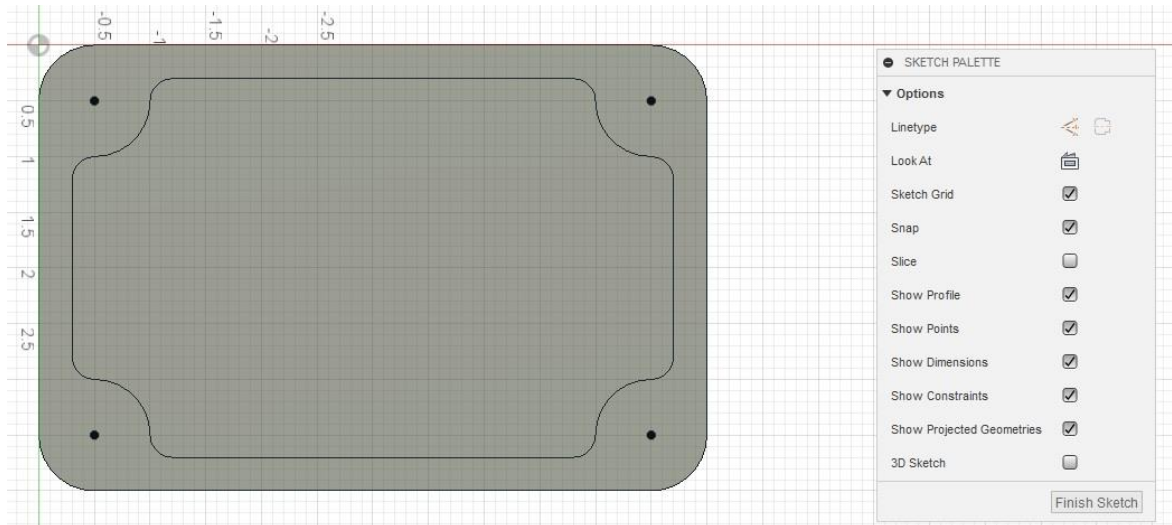
(Four Corner Points Established)

10. **Create the Counter Bored Holes**: Select the *Create / Hole* tool and left-click on each of the corner center points. In the *Hole* panel, enter the following:

Hole Type: Counter Bore (icon)

Counter Bore outer Diameter: .562"

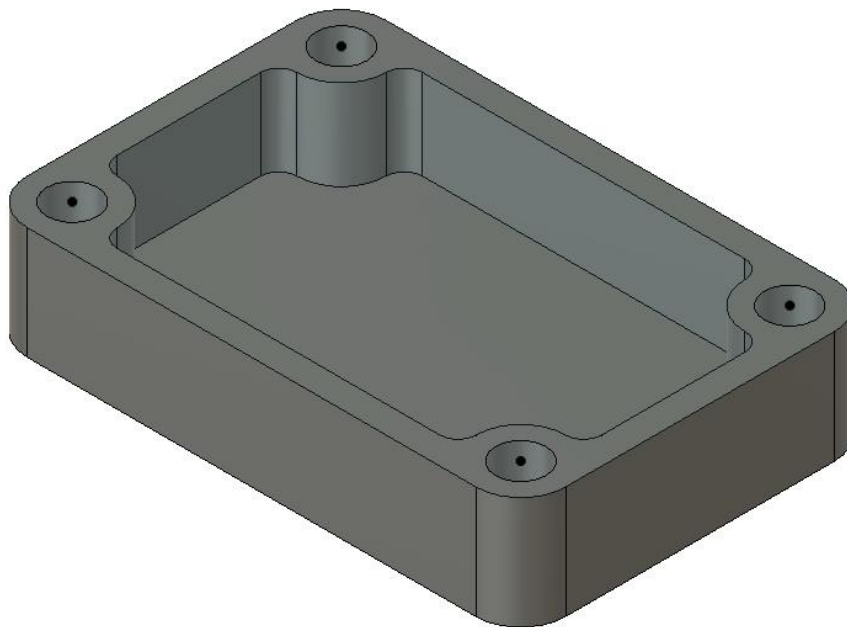**Counter Bore Depth**: .4"

Counter Bore Drill Hole: .391"

For the overall depth, from the *Extent* menu, select *All*. This ensures if the dimensions change in a later revision and the bottom thickness changes, the center thru-hole follows the change, rather than entering a specific depth which locks the part dimensions.
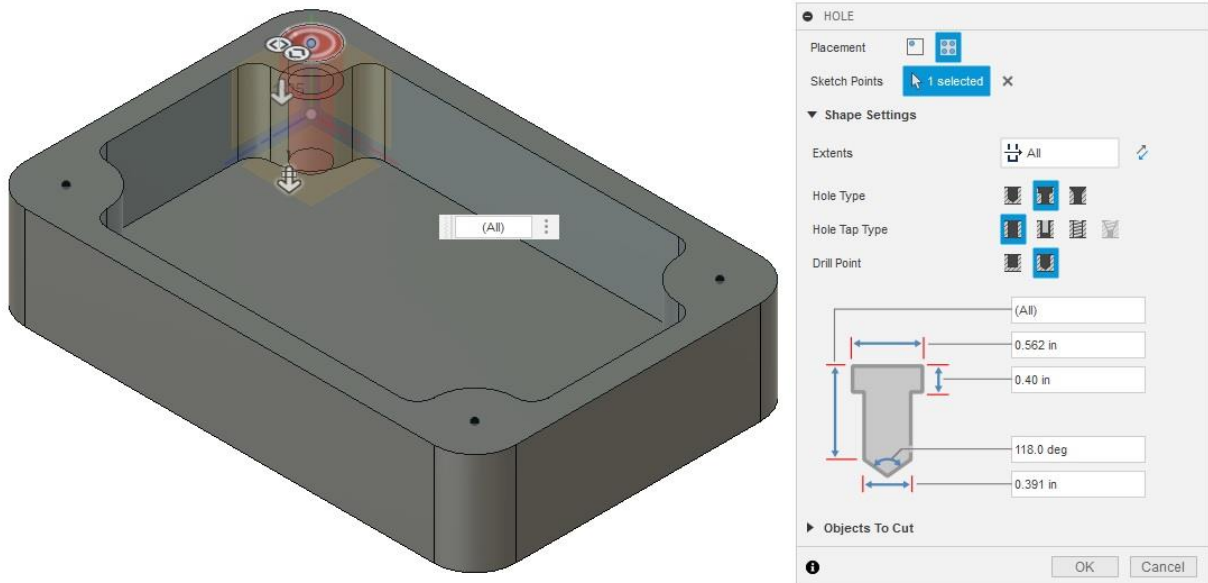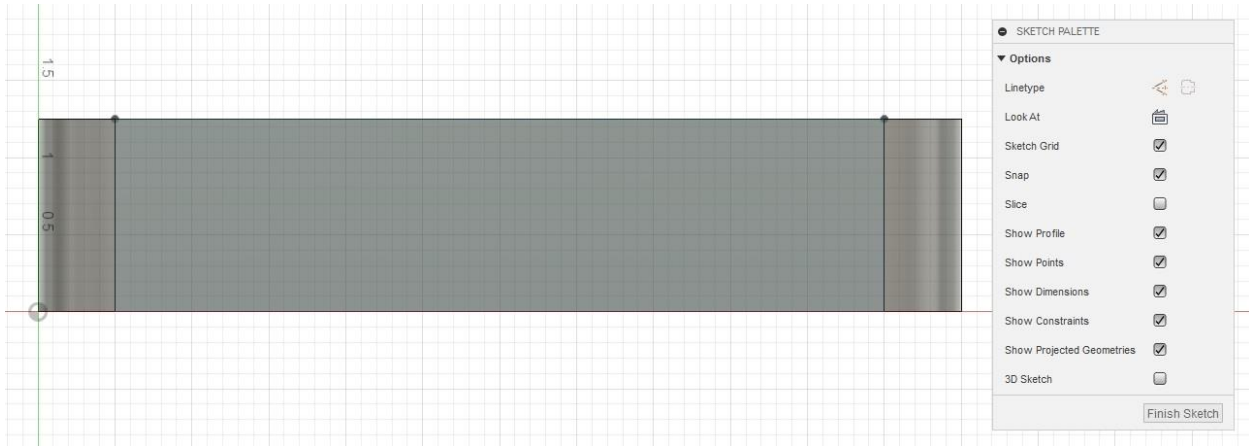
(First Counter Bored Hole Created)

(Remaining Counter Bored Holes Finished)

9. **Create the First Connector Slot**: Remaining in the ***Perspective*** view, create a ***New Sketch*** and select the ***Front*** face of the part to establish a sketch plane.


(Front Face View in New Sketch)

From the ***Create*** menu, select the ***Slot*** tool, choosing the ***Overall Slot*** type from the options dropdown menu, which only requires two points to create the slot profile. The length is **2"** and the height is **.5"** roughly centered on the ***Front*** face. (The slot will be accurately positioned in the next steps and is offset in the following figure to better illustrate this.) The first point, followed by moving the cursor to the right, specifies the length of the slot. After left-clicking the second point to define the length, the next point below it defines the end radius / slot width. It is easiest to ***Dimension*** while drawing the slot to ensure the proper measurements are defined (unless using parameters).
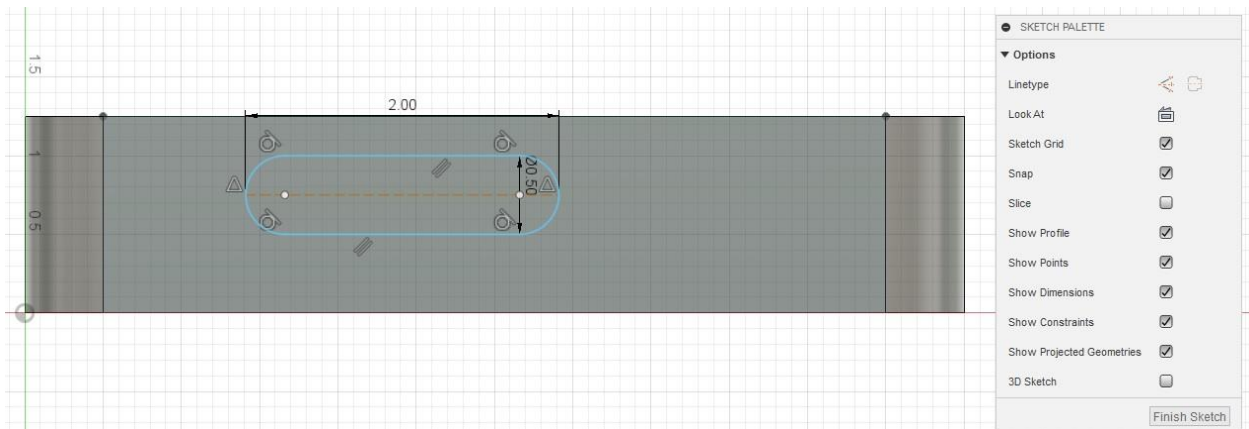
(Slot Placed with Defined Measurements)

Note that the slot edges are blue – meaning the shape is floating parallel to the front face –not constrained (connected) to the face. There are no reference points to anchor it to. Position-wise, it needs to be centered and moved upwards so the slot centerline is .5" from the top edge, using the **Dimension** tool.

Press "**D**" and select the dashed **centerline** of the slot, then select the **top edge** of the wall before moving the cursor to the side of the part to drag the dimension entry field away from the part. Enter **.5"** to position the centerline in relation to the upper edge of the part. The horizontal slot lines turn black, signifying they are constrained / locked to the top edge.
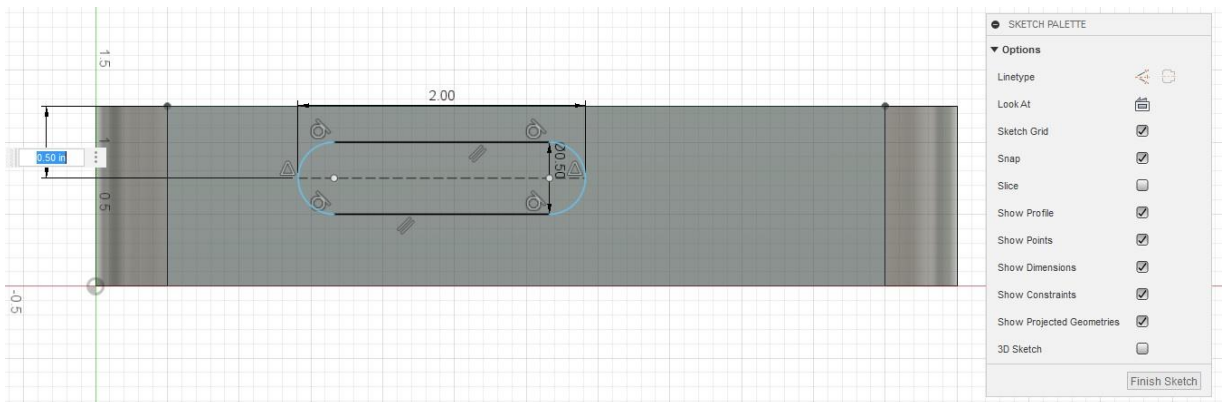
(Centerline Position to the Upper Edge Dimensioned)

10. **Constrain a Point to the Middle of the Centerline**: Press the **Escape** key to drop the **Dimension** tool and select the **Create /Point** tool. Move the cursor across the slot centerline towards the middle until a **blue** triangle appears, then left-click to place a reference point on the centerline. This creates a point constrained to the middle of the centerline. Press **Escape** to drop the **Point** tool.
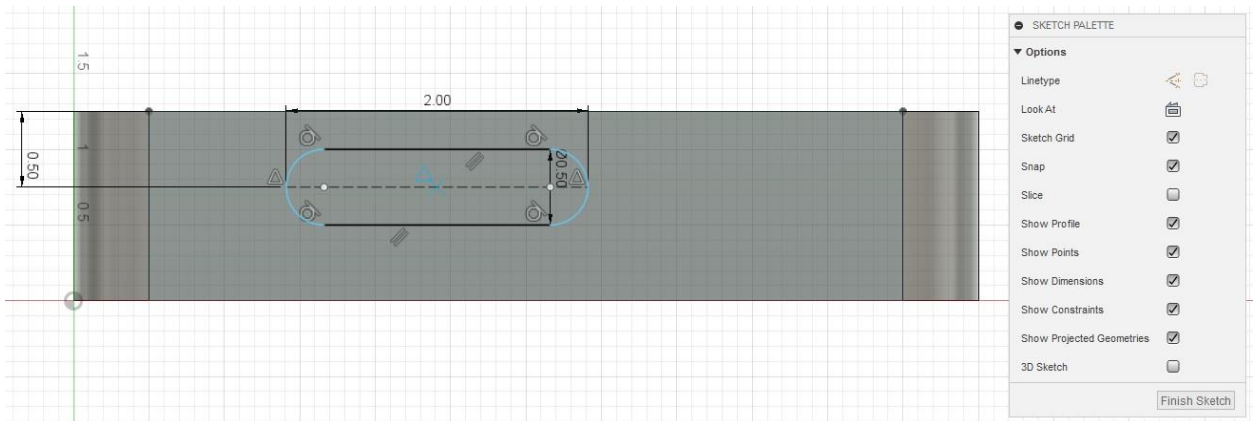
(Centerline Midpoint Blue Triangle Indicator)

11. **Center the Slot on the Face**: Use the *Dimension* tool to dimension the center point to the left edge by left-clicking on each. Enter **3"** in the *Dimension* numeric entry field, which centers the slot on the Front face.

Note that the slot outline is now black, signifying the slot is constrained / locked to the front face. Close the sketch.
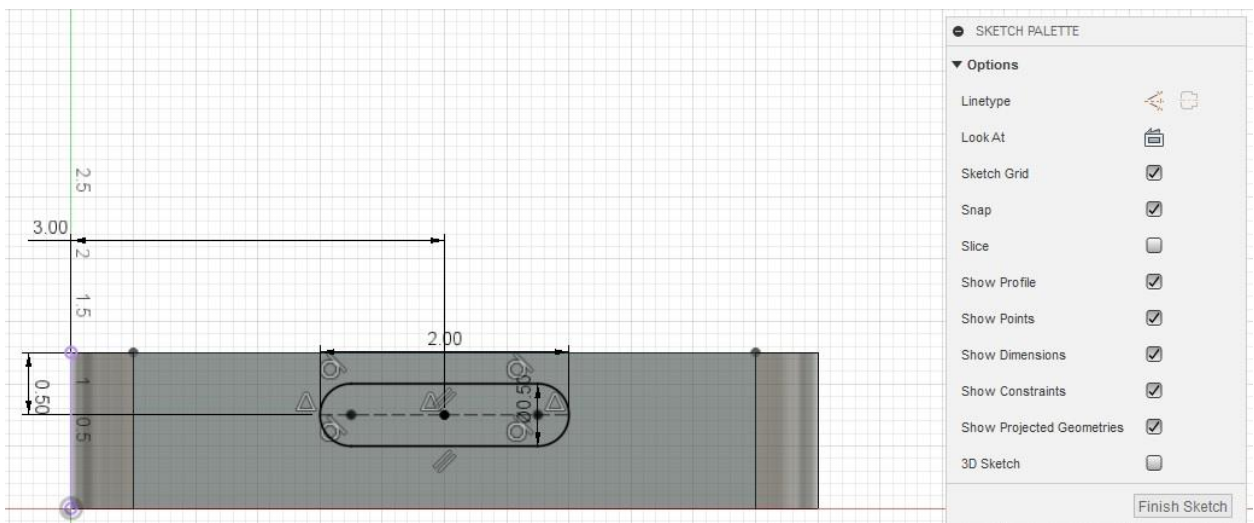
(Slot Centered on the Front Face Using the Dimension Tool)

12. **Extrude the Slot into the Face**: Switch to the *Perspective* view. Select the *Extrude* tool and highlight the shape of the slot to select it for extrusion. Enter -.2" to cut the slot into the front face. Turn off the slot sketch visibility to ensure the proper sketch plane can be selected in the next step.
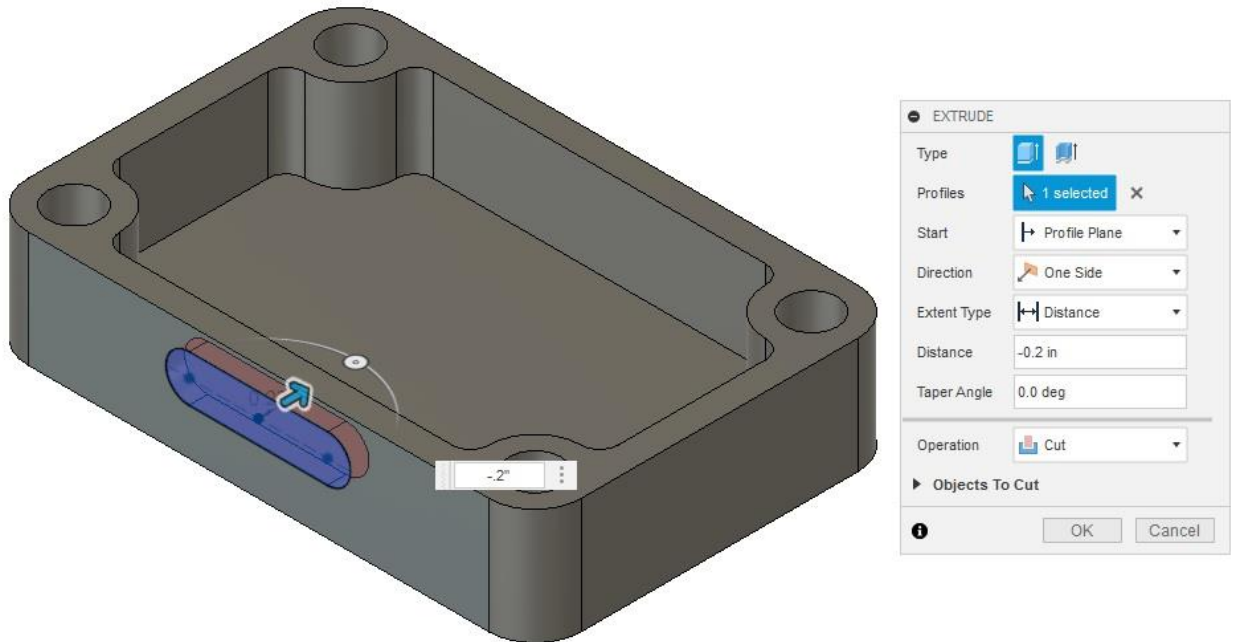
(Slot Cut into the Front Face)

13. **Cut Holes into the Slot Face**: While in the *Perspective* view, create a *New Sketch*, and select the inset slot face as the reference plane. (It may help to disable the slot face sketch visibility to aid in selecting the recessed slot face. The view will change to the *Front* face. Enable the slot sketch visibility for the next step.

Select the *Circle* tool and on one side of the slot, using the opposing end points on the centerline for the origins, left-click and drag out to .25" followed by a second circle on the other side of the slot.

Tip: When creating the second circle, instead of entering the .25" diameter, left-click on the first circle's dimension. This creates a linked parameter, for the two circle diameters, (labeled as "fx") if changes are required later.
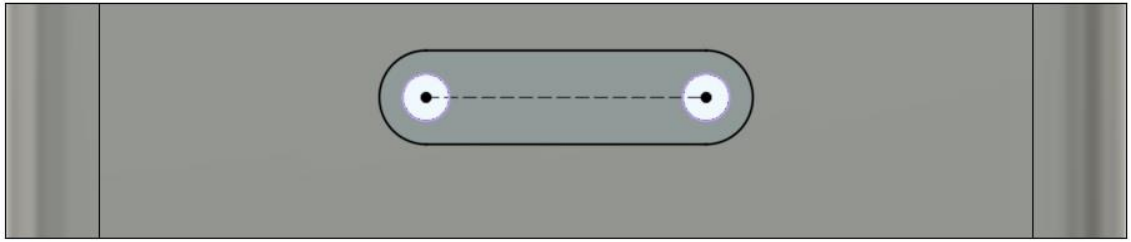
Close the sketch.
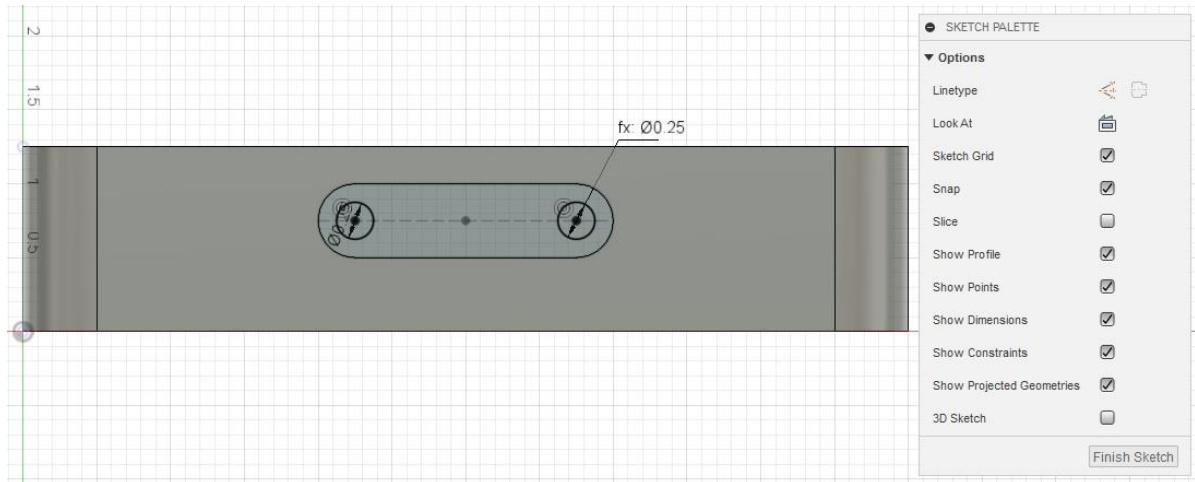
(Slot Holes Dimensioned)

14. **Extrude the Circles Through the Front and Back Walls**: Ensure the sketch visibility for the slot is disabled to be able to select the two circles. Select the *Extrude* tool, then select both circles. Drag inwards and in the tool panel, select "*All*" for the *Extent* type to apply the cut all the way through the opposite wall.
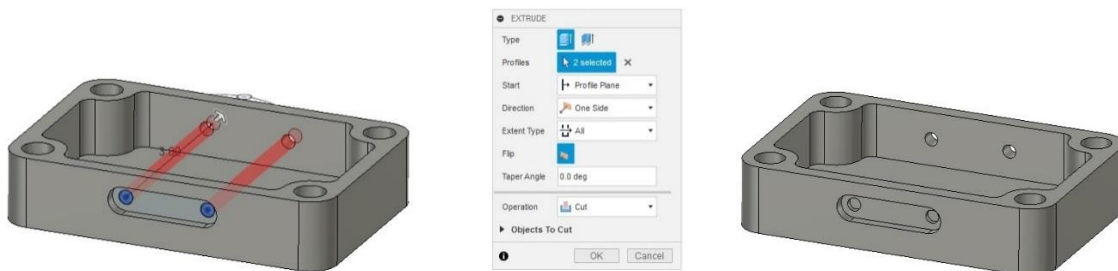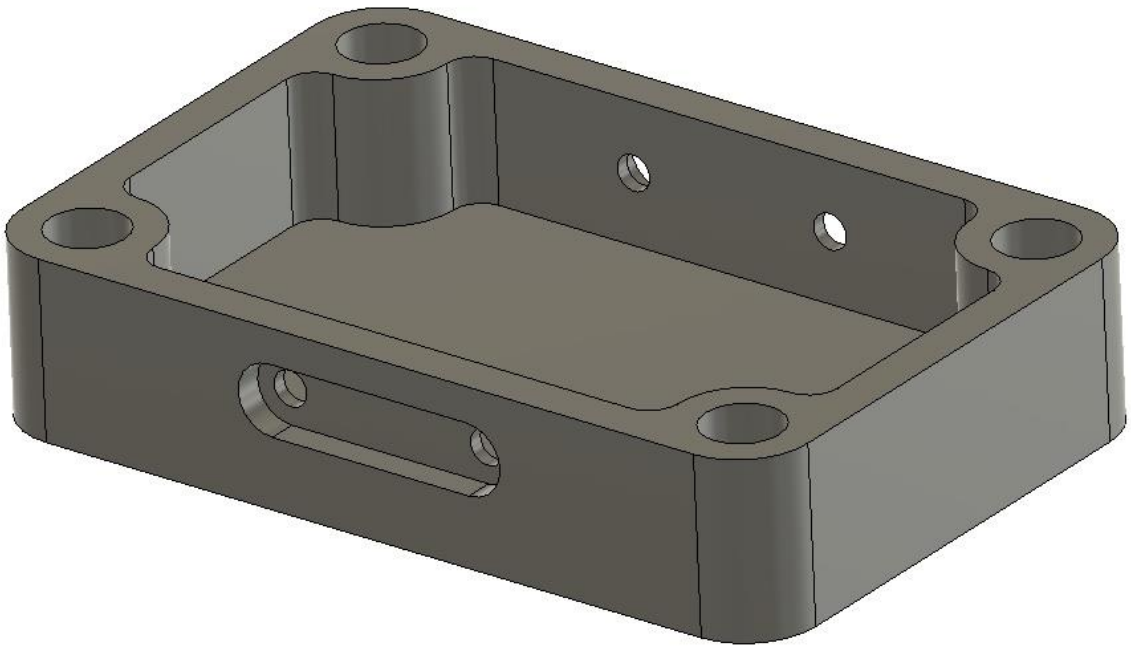

15. **Create a Second Slot on the Back Face**: Create a *New Sketch*, selecting the *Back* face as the sketch plane. Open the *Slot* tool, but this time, choose the *Center to Center* option, which uses the centers of the two previously-cut holes as references to position the new slot. Left-click in the middle of the left side circle, (a blue circle will appear indicating the cursor is on the center of the hole) followed by left-clicking in the middle of the right side circle. Left-click outside the circle to dimension the width of the slot .5". The new slot profile is black, since it is constrained in length and height to the two holes. Close the sketch.

(Back Face Slot Defined)

16. **Extrude the Back Face Slot Inwards**: Select the *Extrude* tool, then highlight the slot profile. Extrude it inwards by -.2". Disable the sketch visibility.

(Second Slot Extruded Inward)