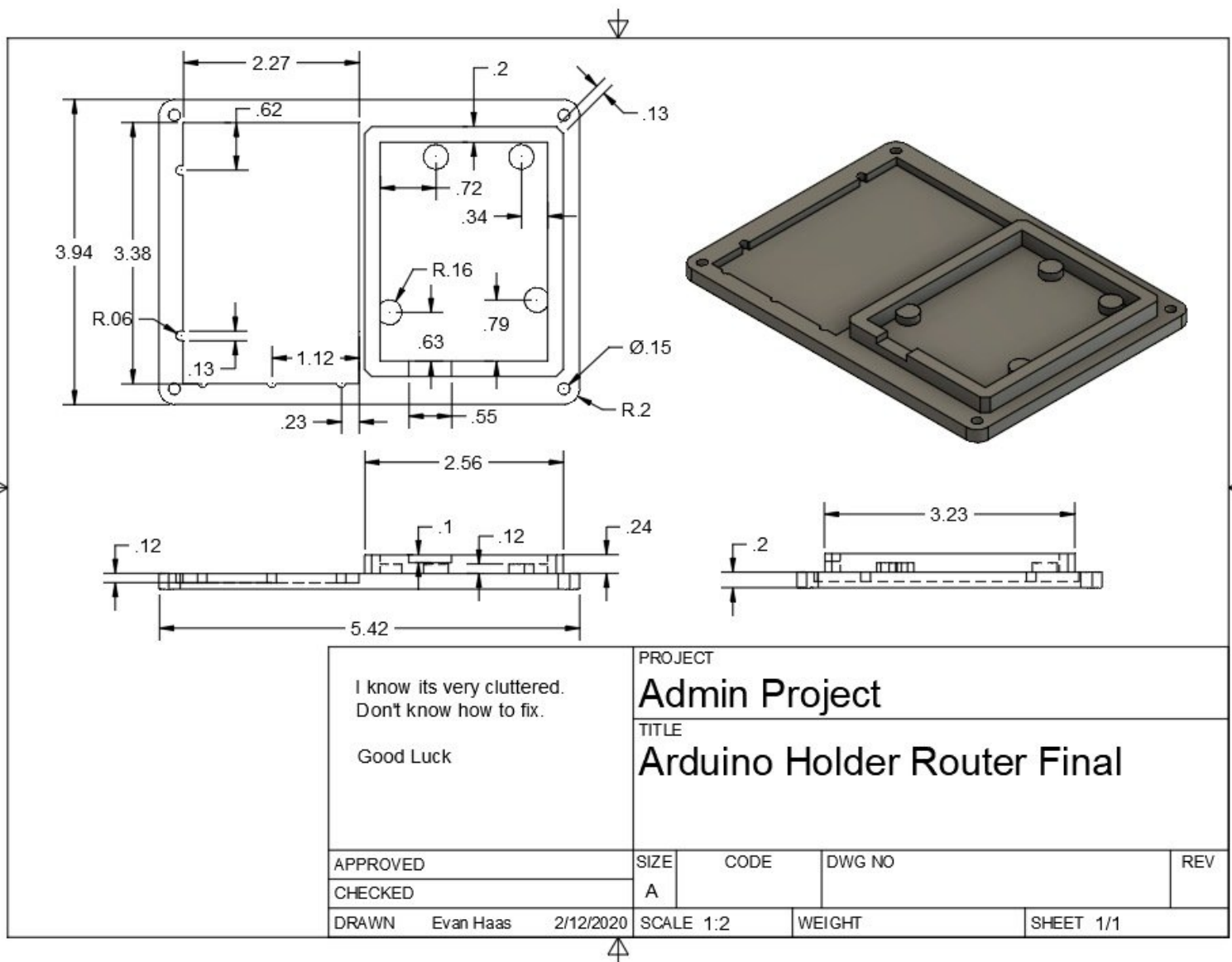




## Training Part 2.1 - CAD Arduino Holder

This will teach you how to CAD the Arduino Holder. It is encouraged to attempt CADing the part prior to following the tutorial.

Written By: Kenny Davis Jr





## TOOLS:

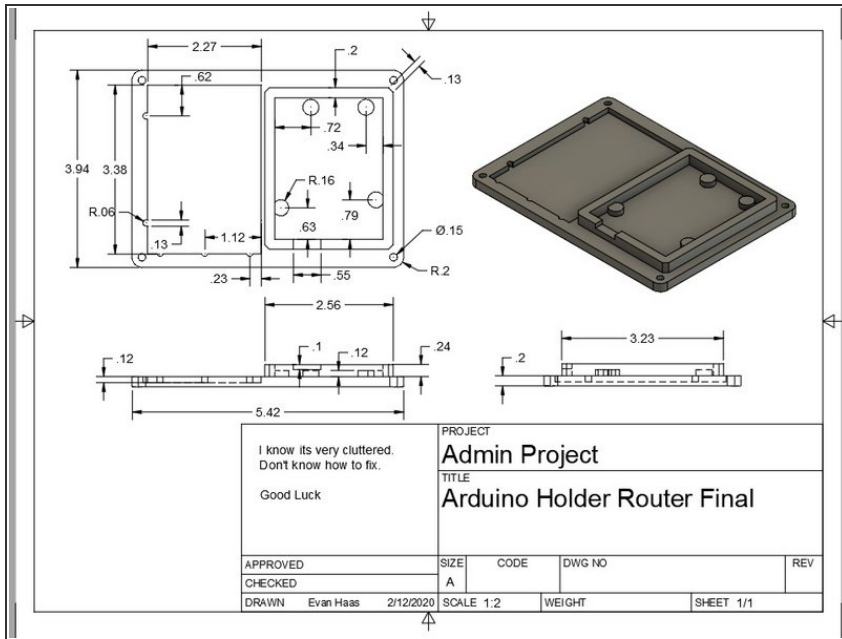
- [Desktop / Laptop Computer](#) (1)

*Fusion 360 Installed*

- [Computer Mouse](#) (1)

*\*Highly Recommended\**

## Step 1 — CAD File



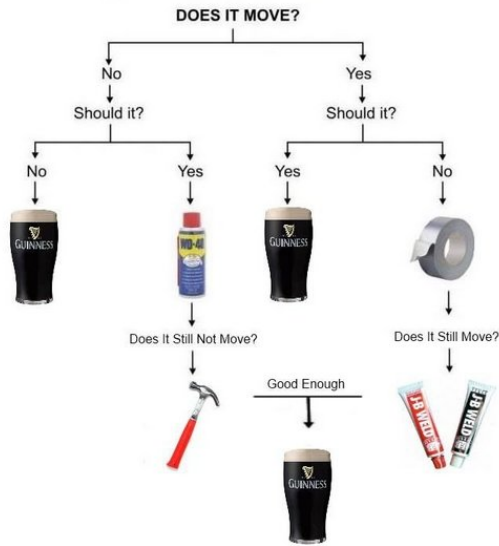
- Here is a link to download the PDF drawing
- <https://drive.google.com/file/d/1lykuGNM...>

**NOTE\* THIS DOZUKI CAN BE COMPLETED MANY WAYS**

- 1) If you are already familiar with Fusion 360, then use the drawing to complete the part
- 2) If you are competent, but still not sure about some feature, just skip to the Step with the feature you need help understanding
- 3) If you are not sure about how to start, follow this dozuki through its entirety, and it will walk you through start to finish

## Step 2 — Thinking through the process

### Engineering Flowchart

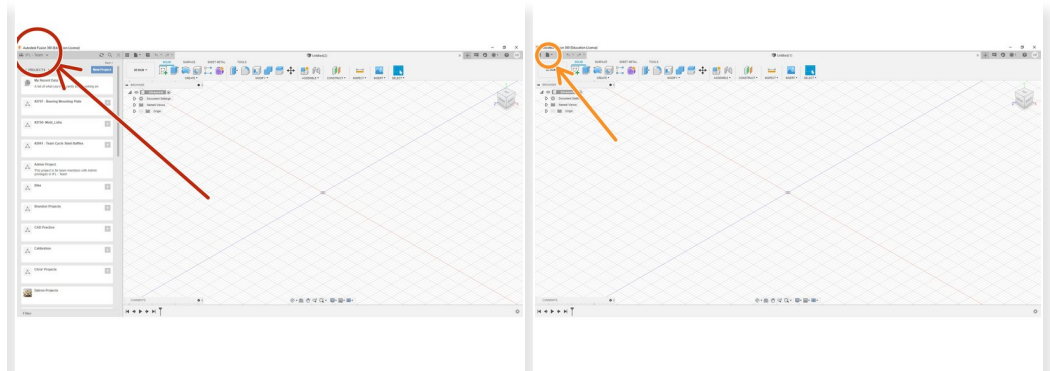


⚠ This step is not necessary and can be skipped

- When we begin drawing a CAD file, it is a good practice to observe the part
- From here we began by creating 2D shapes.
- We then use these 2D shapes to create 3D objects in space.

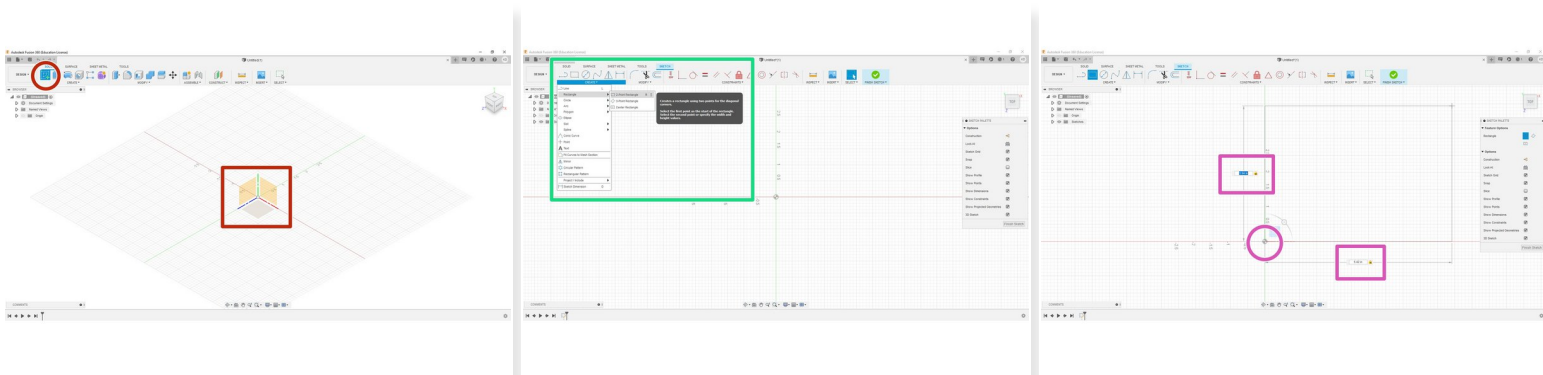
⚠ The picture is a highly detailed flow chart of the engineering design process.

## Step 3 — Startup Fusion 360



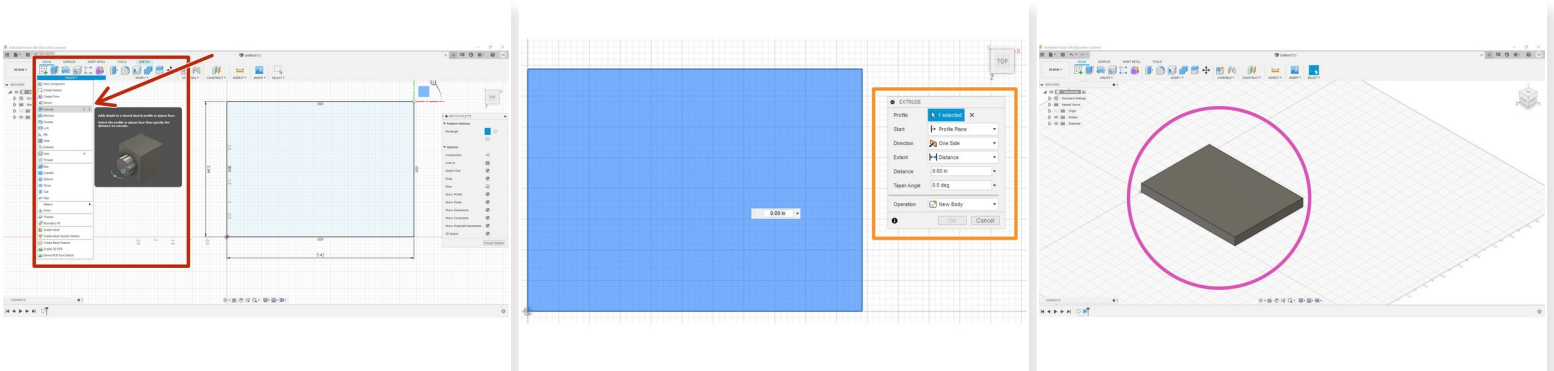
- Start the program Fusion 360
- Select your name on the top right of the screen
- Make sure the "IFL - Team" is selected
- Select "IFL - Team"
- ① If "IFL - Team" is not available, contact either David Kriesberg or Kenny Davis Jr to be added
- Began by clicking the file button (circled in orange), and select "New Design"
- ① This can also be accomplished with Cntl+N

## Step 4 — Creating the Body Rectangular Outline



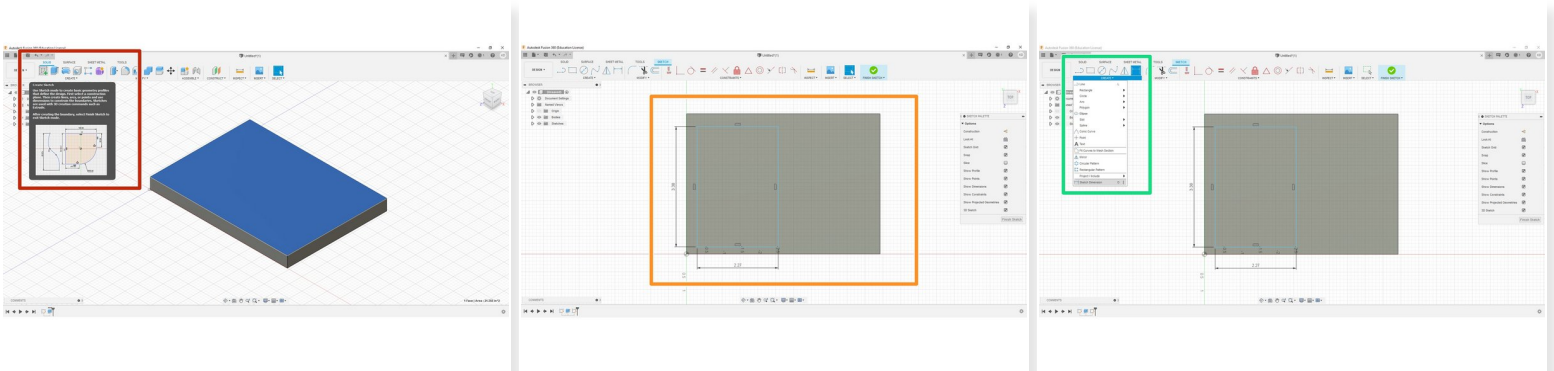
- As with every CAD file, we will begin by creating the most basic shape
- Looking at the CAD drawing, we see that a RECTANGLE is the most basic shape, so keeping that in mind, we will start a new sketch
  - ① Remember for every CAD model, we begin in 2D, and then move to 3D. This is why we are starting with a rectangle, and then we will "bring it into 3D"
- Click the "Create Sketch" button (red circle), then, select the XY or "top" plane (red rectangle)
- Maneuver the view to a plan view, this can be accomplished by selecting "Top" on the view cube
- In the "sketch" tab, click the drop down menu named "create," and click "rectangle" and then "2-point rectangle"
- First click in the origin (pink circle), then type in the dimensions (pink rectangles) you found on the drawing (3.94 x 5.42) and hit enter
  - ① Type in one dimension number, then TABing will take you to the other number

## Step 5 — Creating the Extruded Body



- Now that we have our rectangle in 2D, we will simply extrude it out
- Begin by going to the "solid" tab, and dropping down the "create" button, and select extrude from the option
- This will then prompt an "extrusion command center" as shown by the orange square, there isn't much to do here except type in our thickness and enter
- ★ For any questions on what any of the options mean, ask any IFL staffers or slack Kenny Davis Jr
- When finished, it should look like the pink circle

## Step 6 — Creating Breadboard Cutout Part 1

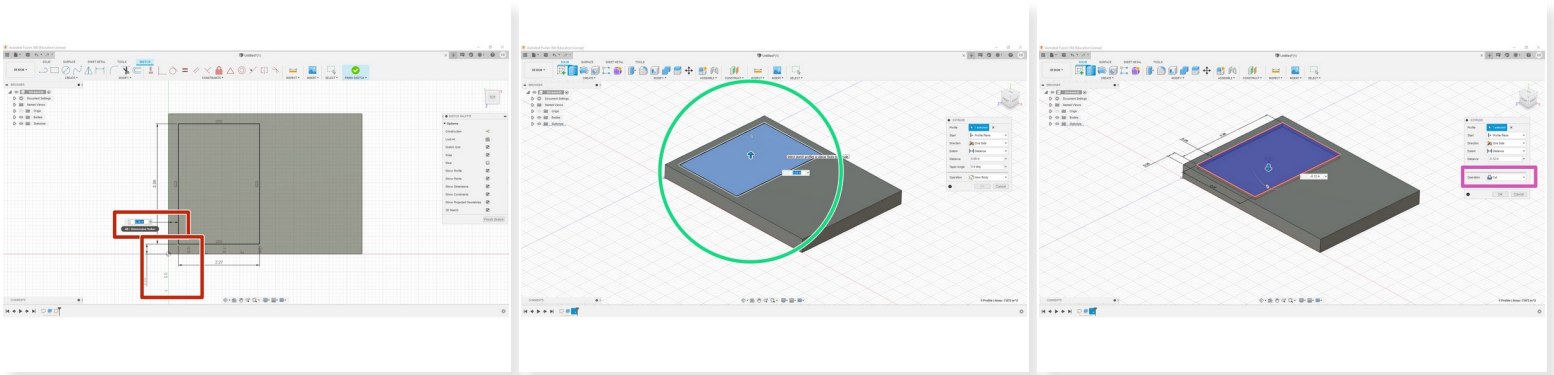


- We will begin by creating a square on the top, and extrude cutting through
- Begin by selecting the top surface, and then selecting "create new sketch (red rectangle)"
- Change the view to the "top" view, and draw a rectangle (look at step 4, the green sub-bullet) with the dimensions from the drawing (2.27 x 3.38)
  - ① Location can be approximate
- Next, we are going to define where it is with reference to the outer body
- Select the dimension tool (green square) by going to "create" and then clicking "dimension all the way at the bottom"

 Continue on to part 2

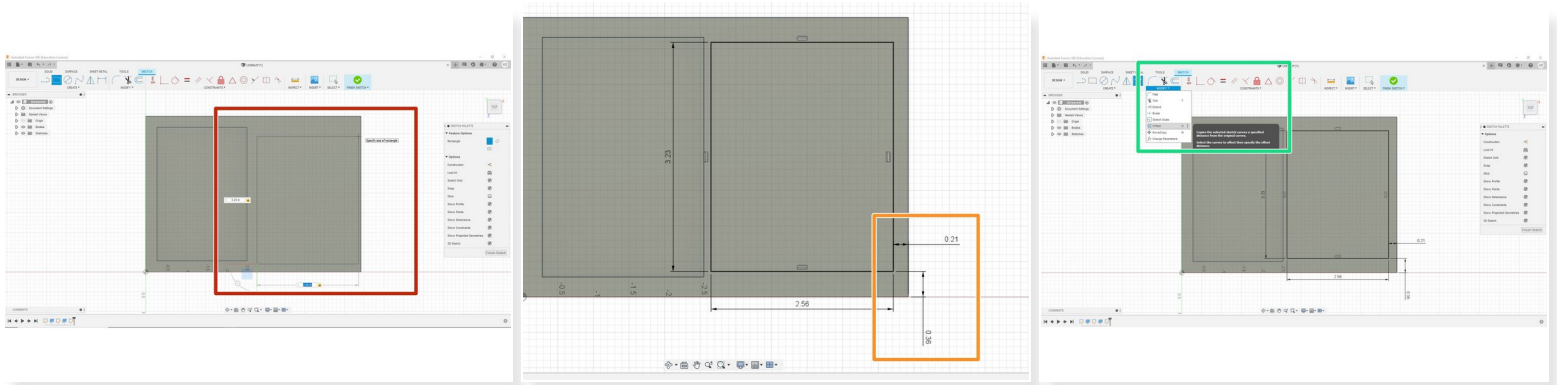


## Step 7 — Creating Breadboard Cutout Part 2



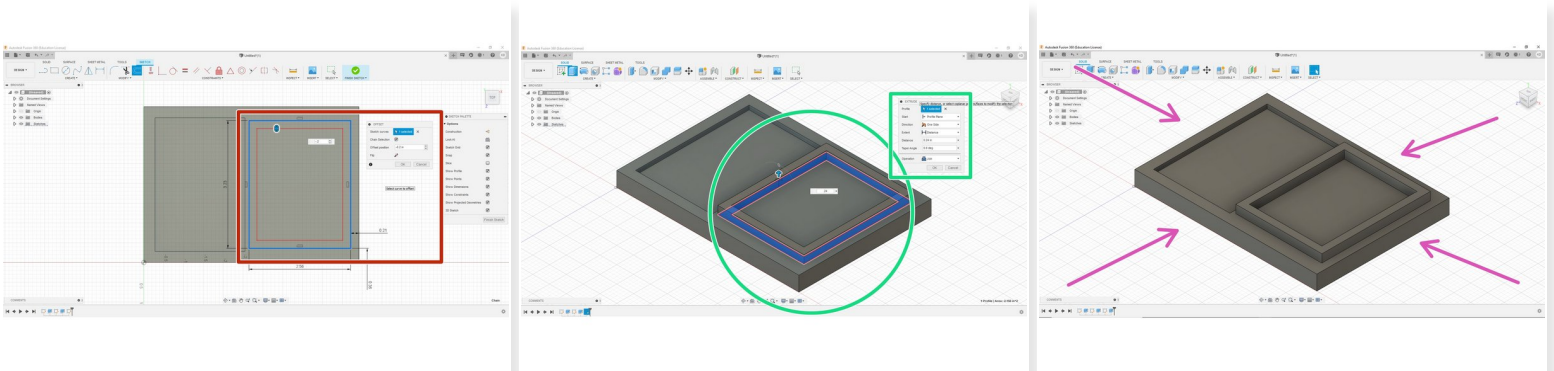
- Dimension from the outer edge to the rectangle both .28" (red squares)
- Then, go to the "solids" and select "extrude"
- For the body we are extruding, be sure sure to select the inside of the rectangle we drew in step 6
- Now input .12 in but make sure that the operation is set to "cut" (pink square), and hit enter to see the new cut feature

## Step 8 — Creating Arduino Walls Part 1



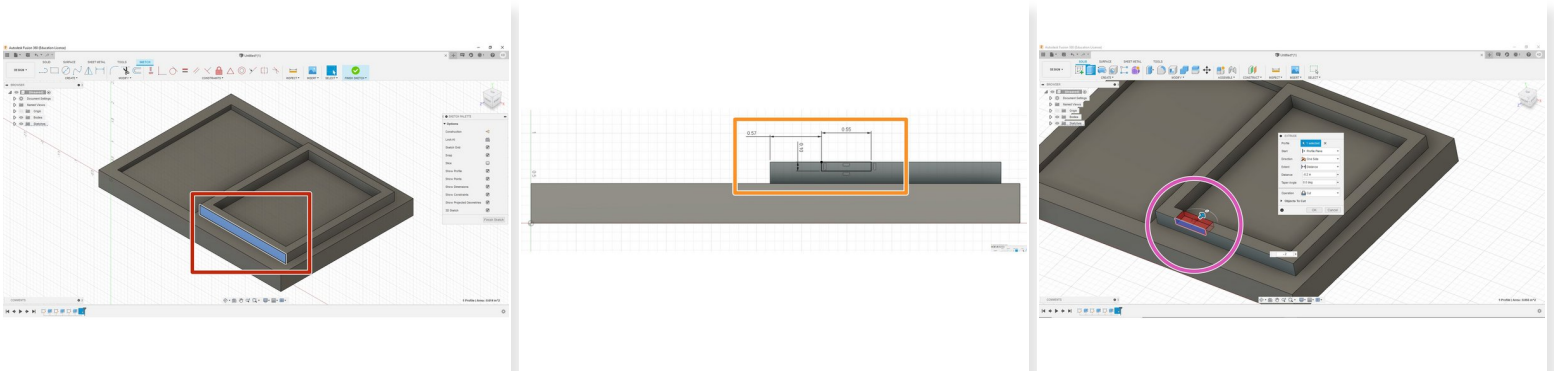
- Let's first begin by selecting our top surface again and drawing 1 rectangle (2.56 x 3.23)(red rectangle)
- Now let's dimension off our rectangle, select the dimension tool, and offset .36X and -.21Y (orange rectangle)
- Finally, in order to create a uniform wall thickness, let us select our offset tool by going "Modify" in the "Sketch" tab, from here drop down the menu and select "Offset"

## Step 9 — Creating Arduino Walls Part 2



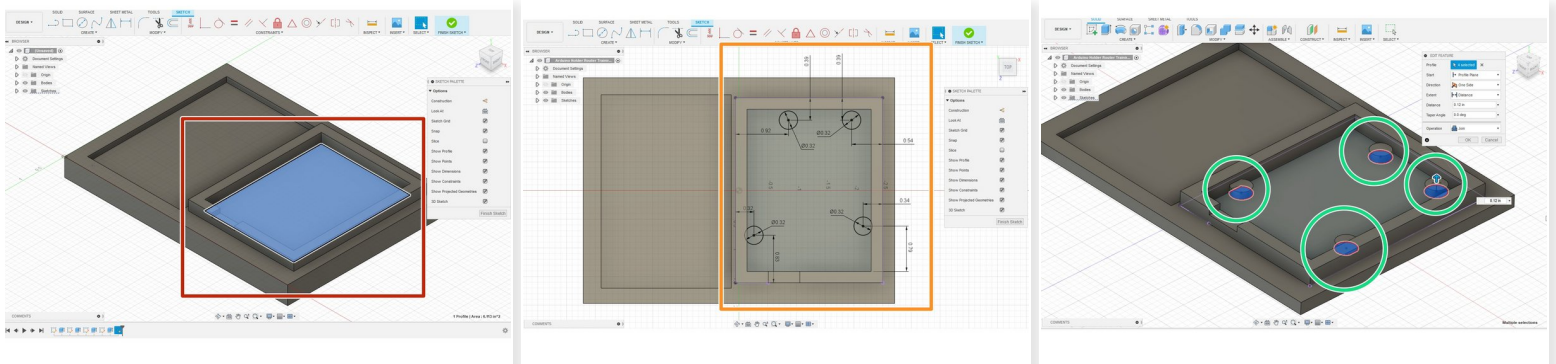
- Now with the offset tool selected, we can select the rectangle we just drew, and type our thickness of  $-.2$ "
- ❗ The reason the value is negative is simply because the offset would go on the outside if it were positive, if you are unsure what I mean, type in  $.2$ " and see what happens
- With the outline of the arduino wall, we can now extrude, go ahead and select extrude, then extrude out the wall region we just offset by  $.24$ "
- When finished, the part should now look like the last figure

## Step 10 — Creating Power Adapter Cutout



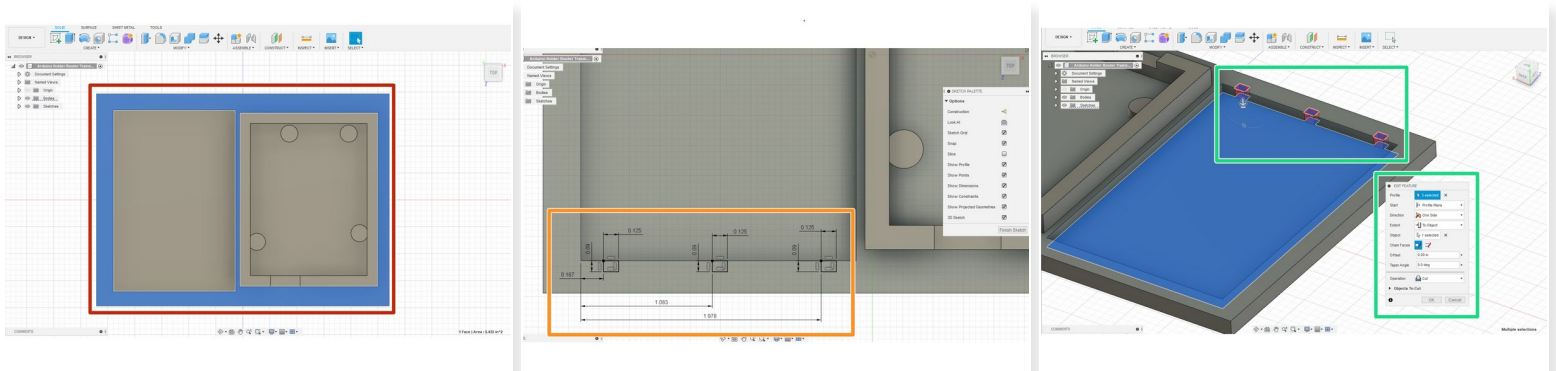
- Begin by creating a sketch on the right facing wall we just created, and go into plan view
- Next create 2 point rectangle, and make sure to select the top line first as shown by the arrow, and input the dimensions .55x and .12
  - ⓘ Selecting the top line will create a relationship where the point of the rectangle is "hooked" to the top, and this is exactly what we need
- Dimension .57" from the outside of the wall
- Now simply select extrude, and cut away -.2" (the thickness of the wall)
- ⓘ Remember cutting away requires a negative distance

## Step 11 — Creating Arduino Screw Holes



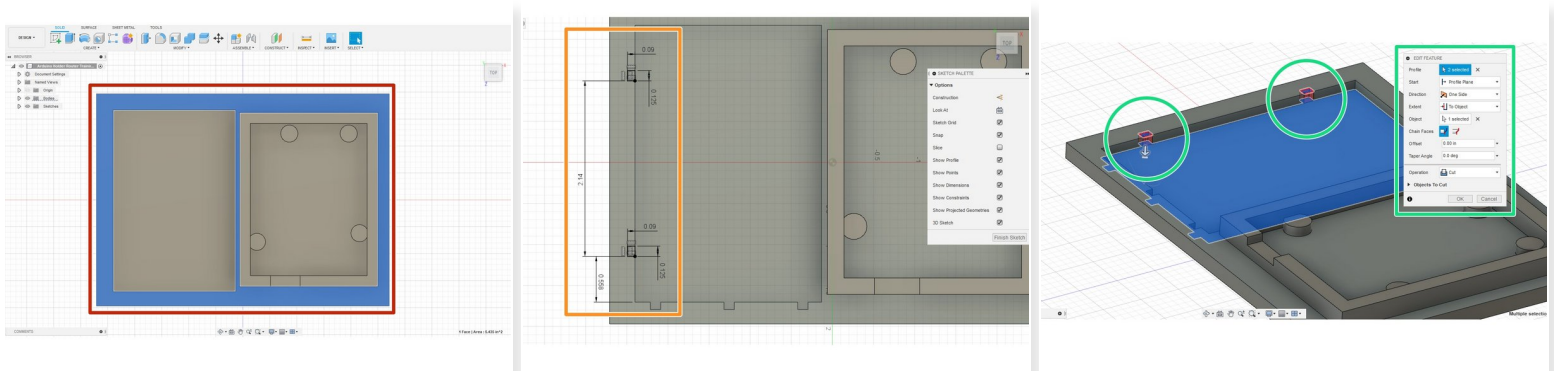
- Begin by selecting the surface where the Arduino goes, then start a new drawing.
- Draw the sketch given in the second photo. All of the distances and lengths are given.
- ❗ Be sure not to leave any blue lines, which means the sketch has undefined properties.
- When finished, simply select the four circles and "extrude" shown by the green circles. Make is ".12" tall.

## Step 12 — Creating Breadboard Link Holes (Bottom)



- Begin by selecting the surface in blue, as shown by the red rectangle.
- Create a new sketch, and draw 3 rectangles as shown in the second picture. All dimensions are given.
- Finally, extrude down to the bottom of the breadboard surface. This will cut down to the surface requested, as shown by the red cut away.

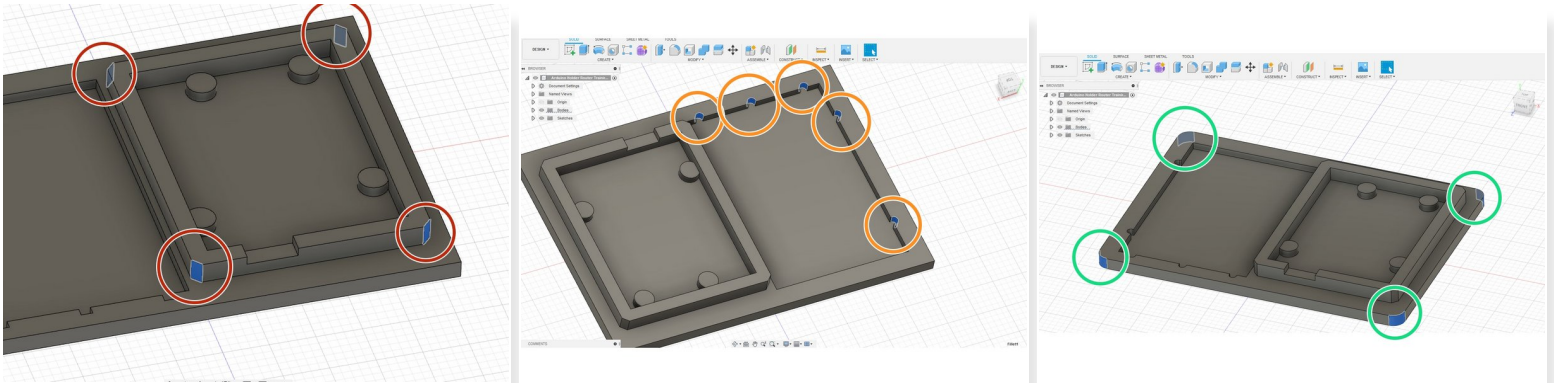
## Step 13 — Creating Breadboard Link Holes (Left Side)



- Again chose the middle surface as shown by the red rectangle.
- Use all the dimensions as given by the second picture.
- Extrude down to the bottom surface.
- ❗ All of the steps are a repeat of the previous step.



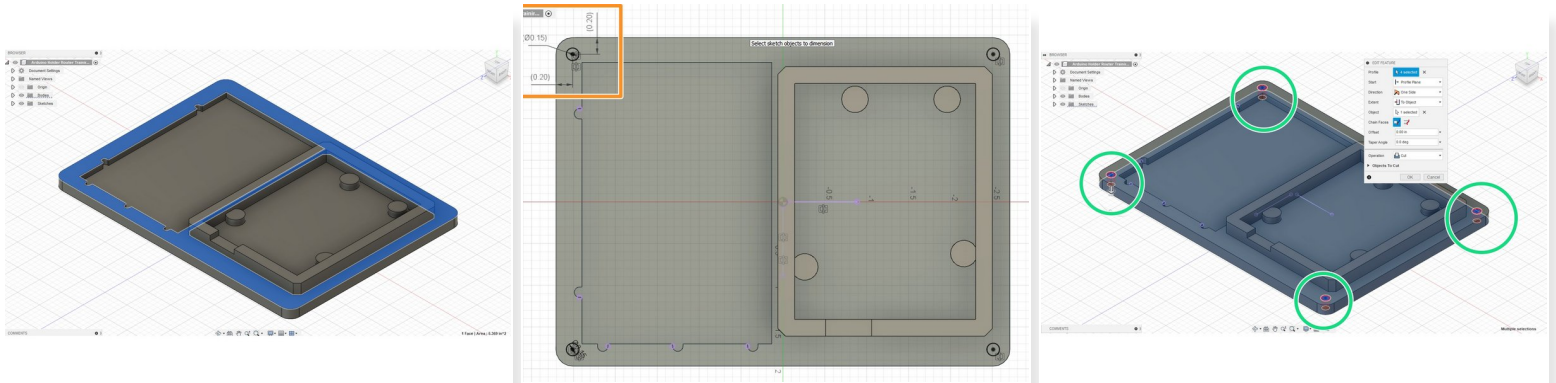
## Step 14 — Creating all Chamfers and Radii



- Begin with the chamfers on the Arduino older. Select all the corners in the red circle and select "chamfer."
- The chamfer should be .09"
- Next, select the "radius" tool and select the inside corners of all 5 rectangles we drew for the breadboard cutout.
- The radius should be .0625"
- Lastly, select all four corners of the part, shown in the green circles.
- Radius to .2"

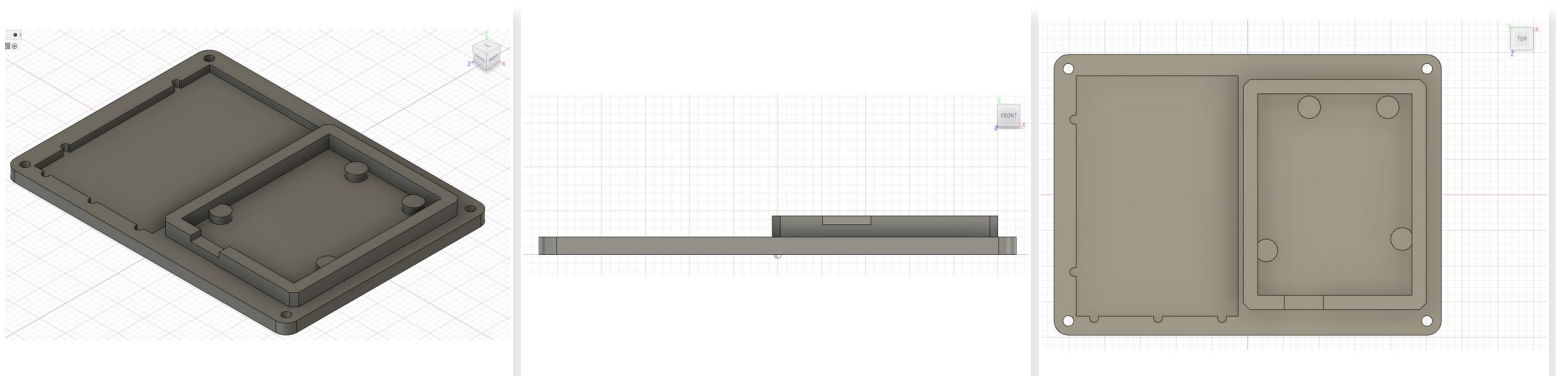


## Step 15 — Creating Outer Securing Holes



- Began by selecting the middle surface again. Create a sketch off here.
- Draw 4 circles at .15" in diameter, and .2" off the walls, as shown by the orange circles.
  - ❗ If more advanced, draw one circle and mirror it over, but this is not necessary.
- Lastly, extrude all 4 circles "through all."

## Step 16 — Final Regards



- ❗ Here are 3 different views of what the finished part should look like.