CAD Training Part 1 B

This training will focus on making a simple assembly. This will be an accelerated version of the Lake Effect CAD Training. That document should also be read as it includes more practice including revolves.

This lesson will work on CADing the gearboxes we use on robots.

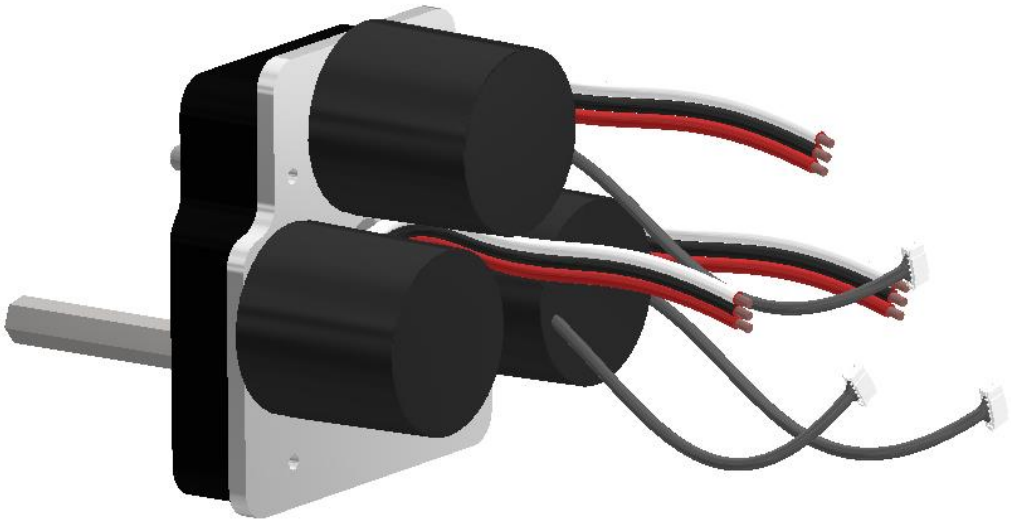

Figure 1: Gearbox assembly from turret bot CAD

B. Gearbox Cover

Now that we have the faceplate, we will be CADing the 3D printed cover that protects the gears from dust and metal shavings.

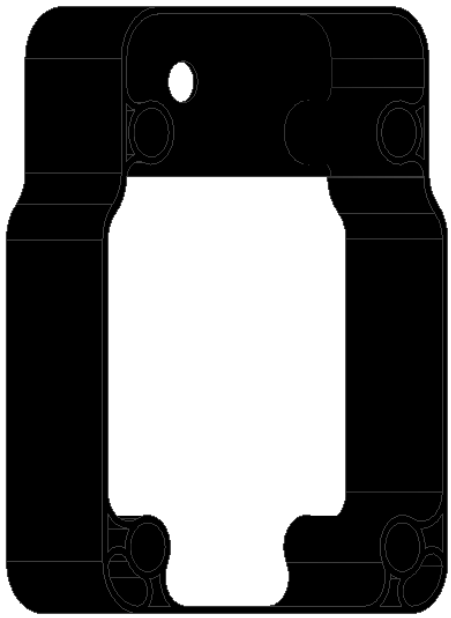

Figure 2

1. Outer Shell

Inner Lines

Create a new part and start a sketch on the XY plane. First, draw the perimeter of the cover with the dimensions from Figure 3. Once you have drawn the perimeter, add 4 circles for the mounting screws. The dashed lines are construction geometry, enabled in the right-click menu after drawing a line.

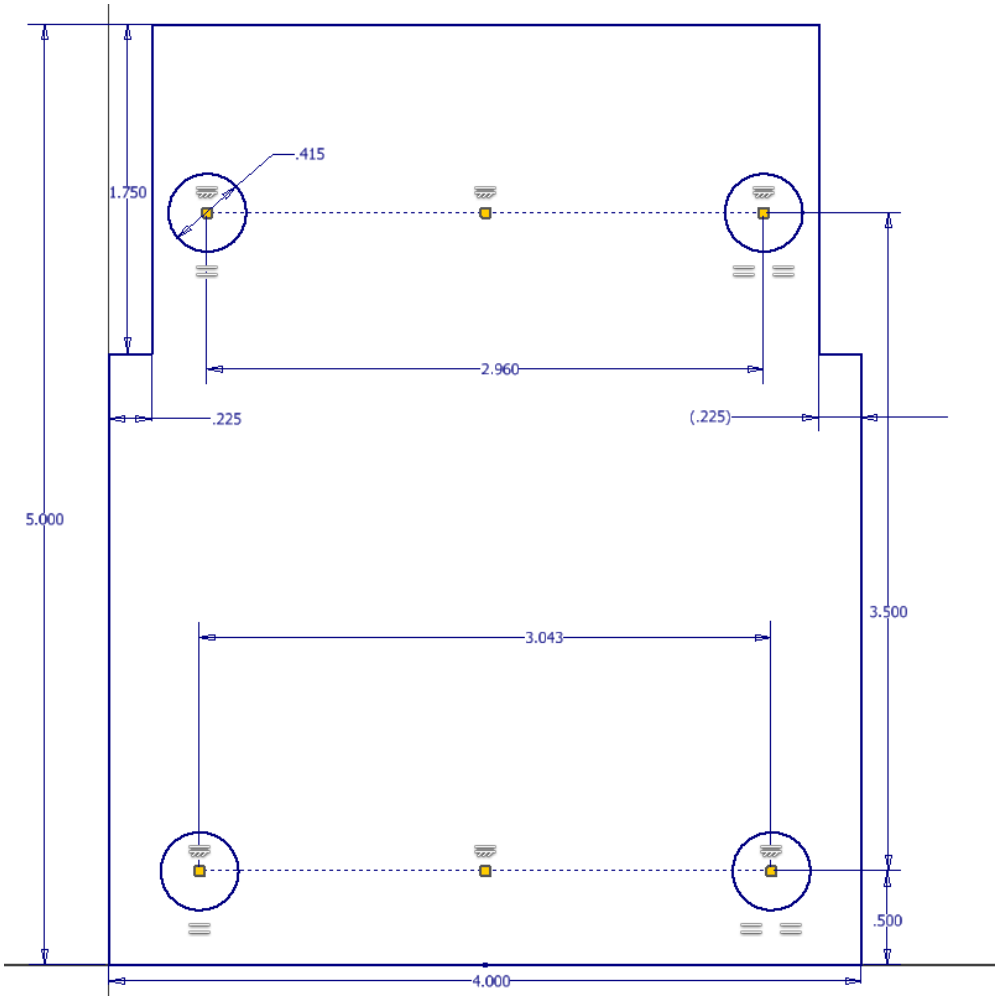

Figure 3

Outer Lines

Once you have drawn the perimeter, you will have to add an outer line to create an extrudable area. The easiest way to constrain this is to dimension the offset once, then reference that dimension elsewhere. This allows for changing the entire offset at once. The dimension name is found in the edit box, and can be used elsewhere for reference as seen in Figure 4. The circles will also have an outer with a diameter of $0.415 + \mathrm{d10} * 2$ (sub in your dimension number for d10).

Figure 4

Extrude + Fillet

Once the outer perimeters have been added, extrude the areas by 1.15”.

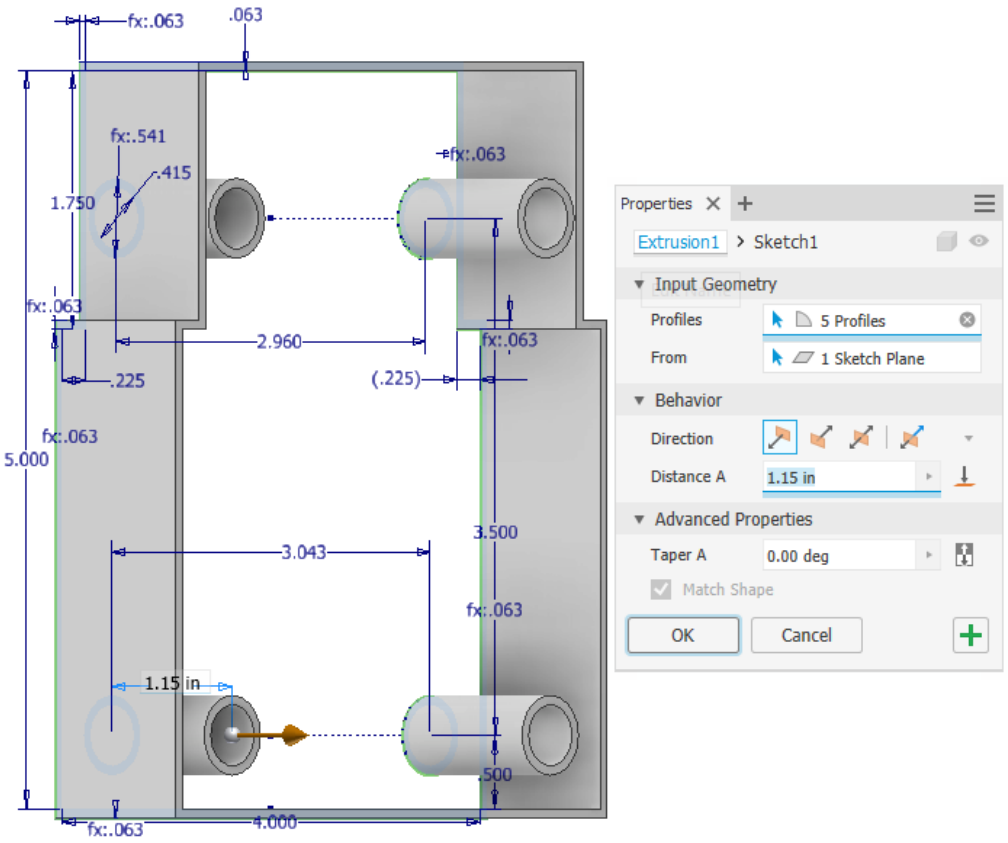

Figure 5

Finally, fillet the corners for aesthetic and structural purposes. This will be done in 3 steps, the first with a radius of 0.375”, the second with a radius of $0.375 + \mathrm{d10}$, and the third with $0.375 + \mathrm{d10}$ (again, sub d10 with your dimension).

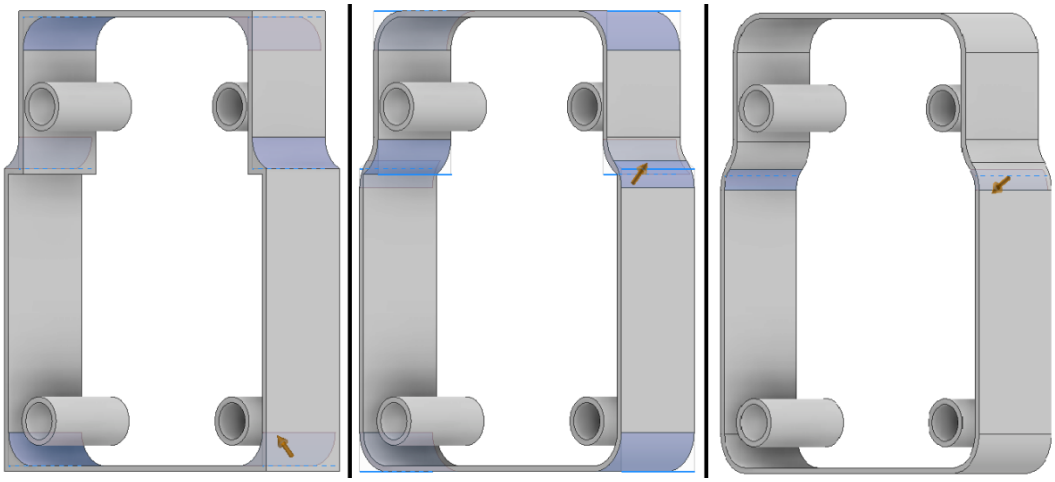

Figure 6

2. Hole Supports

As you may have noticed, the screw holes are floating. This will be fixed by adding supports between them and the outer frame. They will be curves as that has a good balance between material and strength.

Lower Holes

Create a new sketch on the font face of the cover and zoom into the lower left hole. To create the supports, do the following:

1. Draw an arc which connects the hole to the shell
2. Add a tangential constraint to the arc for both the hole and shell
3. Draw a second arc, and constrain its midpoint to that of the first arc
4. Set the radius of the inner arc to 0.126”
5. Dimension the distance between the arcs equal to the thickness dimension from earlier
6. Repeat 1-5 on the other side of the hole with a radius of 0.25”
7. Repeat 1-3 for both shapes, and use the Equal constraint to set sizes

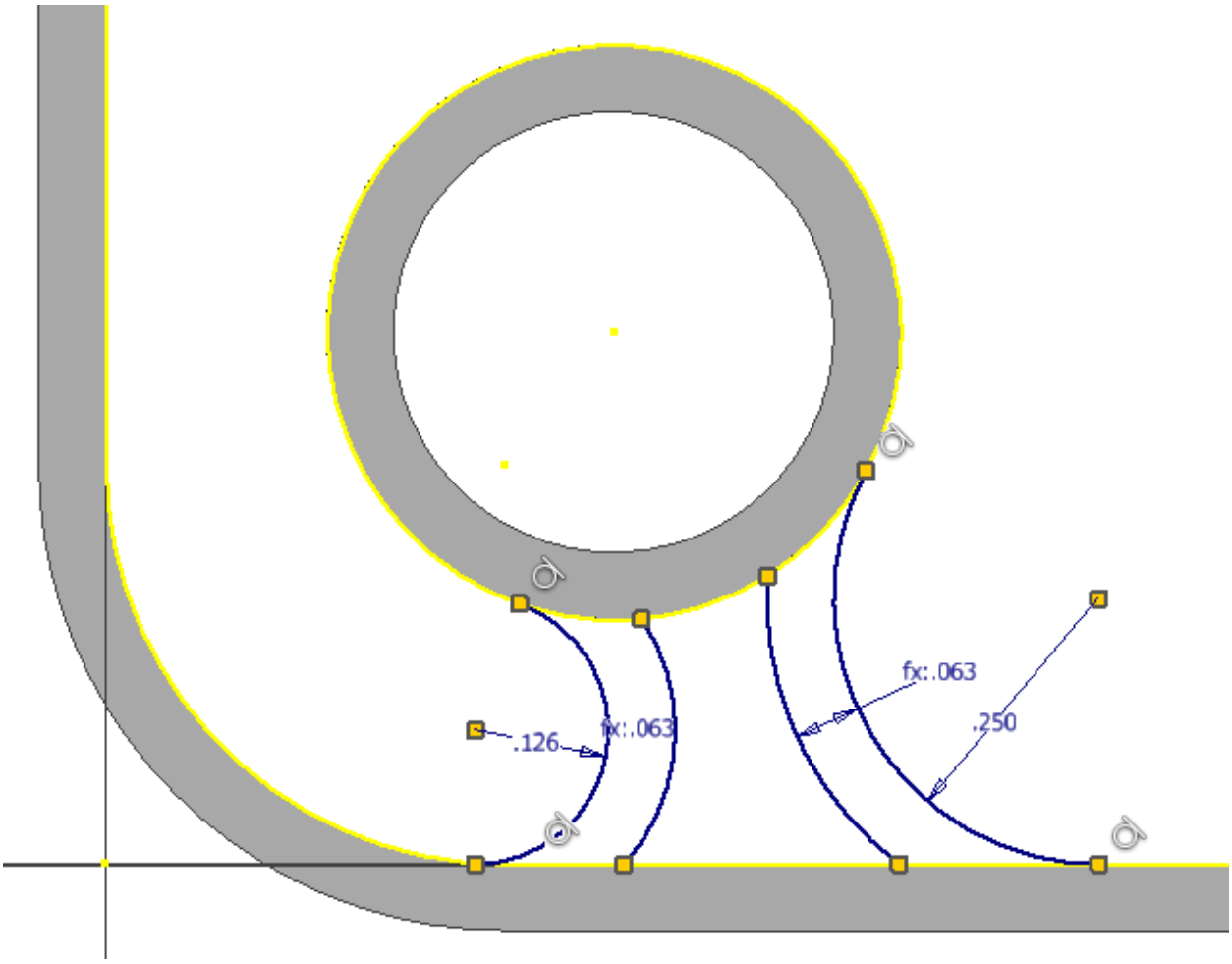

Figure 7

Now that you have done the left side, create a construction line at the midpoint of the base and use the mirror tool to create the right side. Select the arcs as the content, and the construction line as the Mirror Line Be sure that the shell walls are also included in the mirror.

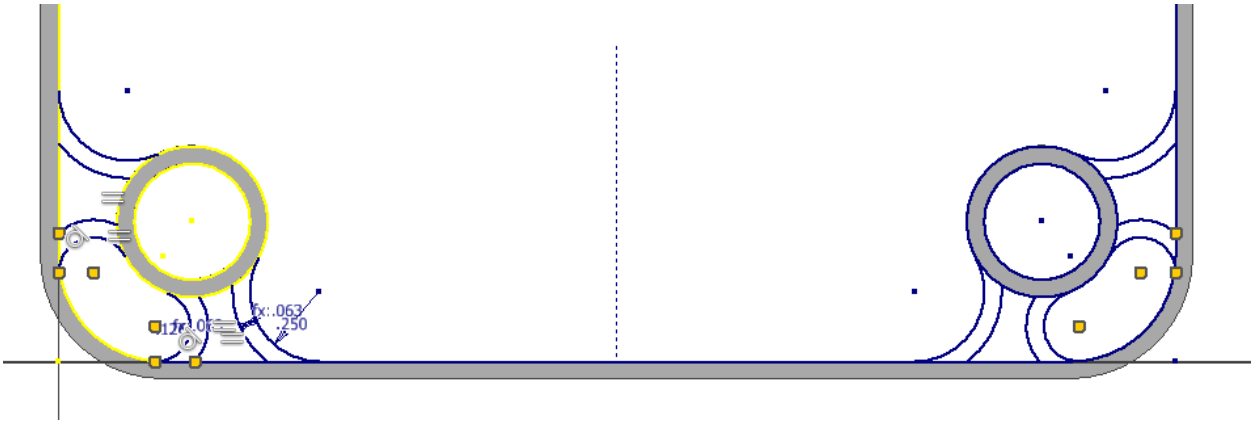

Figure 8

Now simply extrude the areas to the back of the shell.

Upper holes

The upper holes are made much like the lower ones, but with radii of 0.15” and 0.125”.

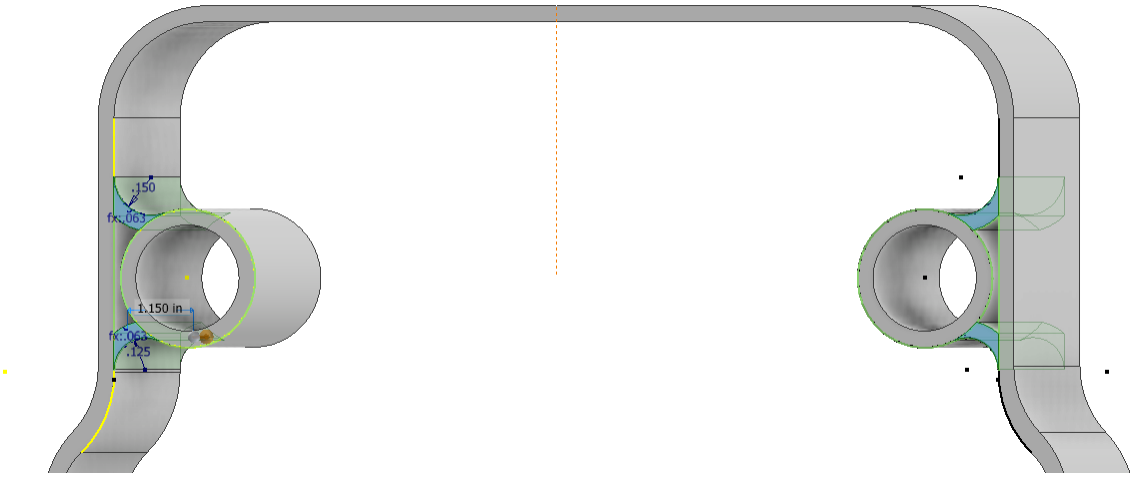

Figure 9

3. Finishing Touches

Solid backing

Start a new sketch on the back face and create a line between where the upper holes connect to the shell and extrude it 0.05” into the shell.

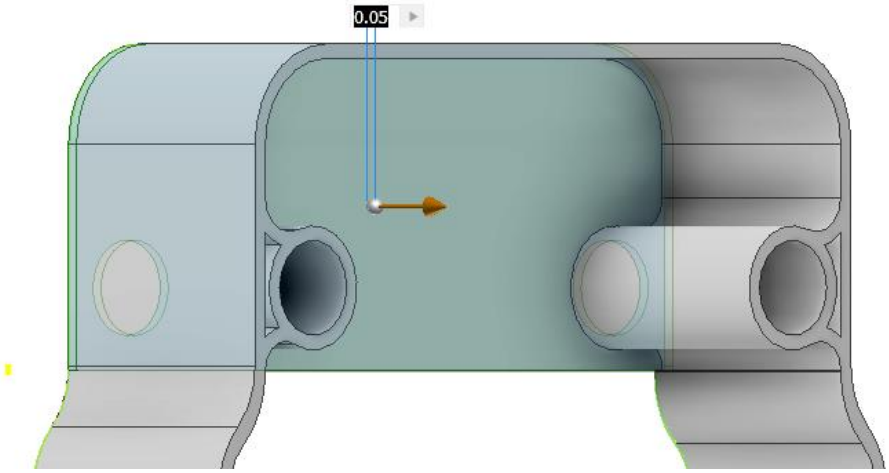

Figure 10

Finally, add a 0.355” hole in the middle and cut it out.

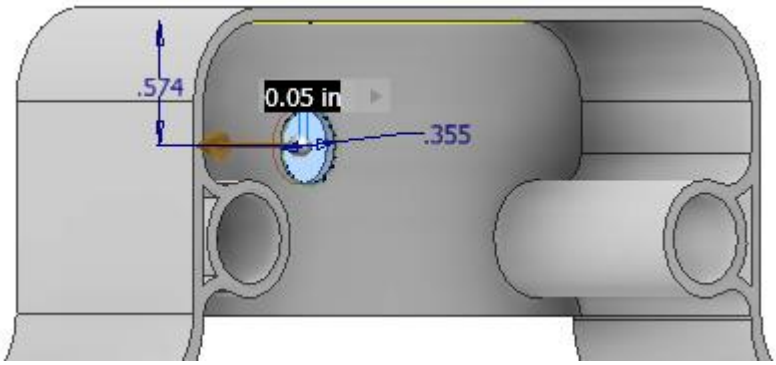

Figure 11

Material

Set the material to “ABS Plastic” and the appearance to “Smooth Black”.

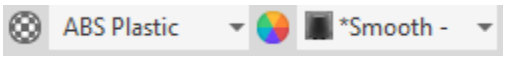

Figure 12