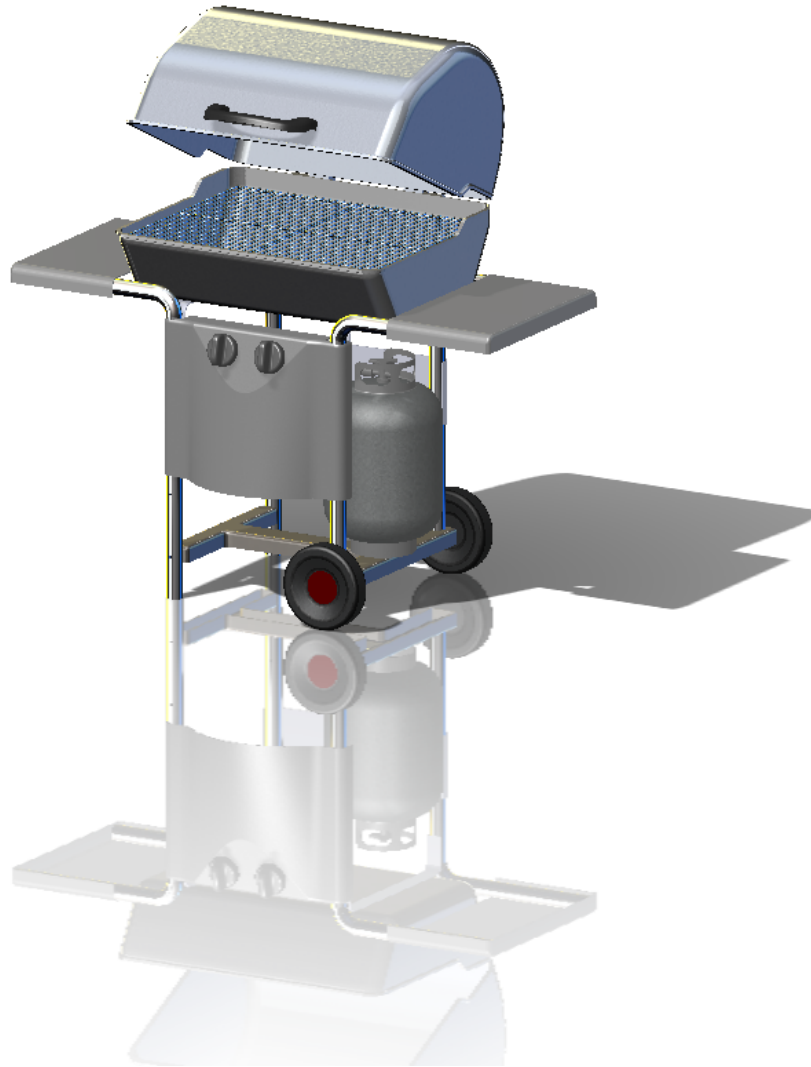


Gas Grill Project

(Cheat Sheet)



This document is meant as a guide to make the parts that are more challenging in the Gas Grill project from the *Beginner's Guide to SolidWorks* book; the simple parts are not included in this guide as they are easy enough to make from the part's detail drawings available either in PDF or eDrawings from the downloads section of our website.

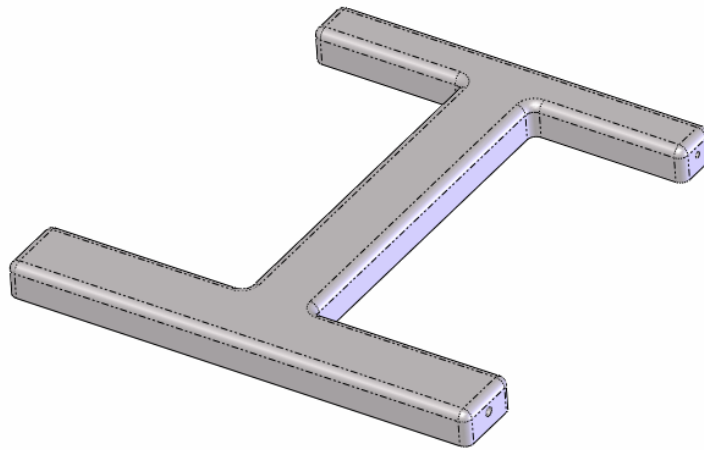
Disclaimer: These models and drawings are not meant for manufacturing or intended as a detailing or designing guide, but as a teaching aid to practice SolidWorks modeling (i.e. no drafts in plastic parts).

MechaniCAD Inc. authorizes you to print this guide as long as it is not changed or modified in any way, and credit is given to us when appropriate.

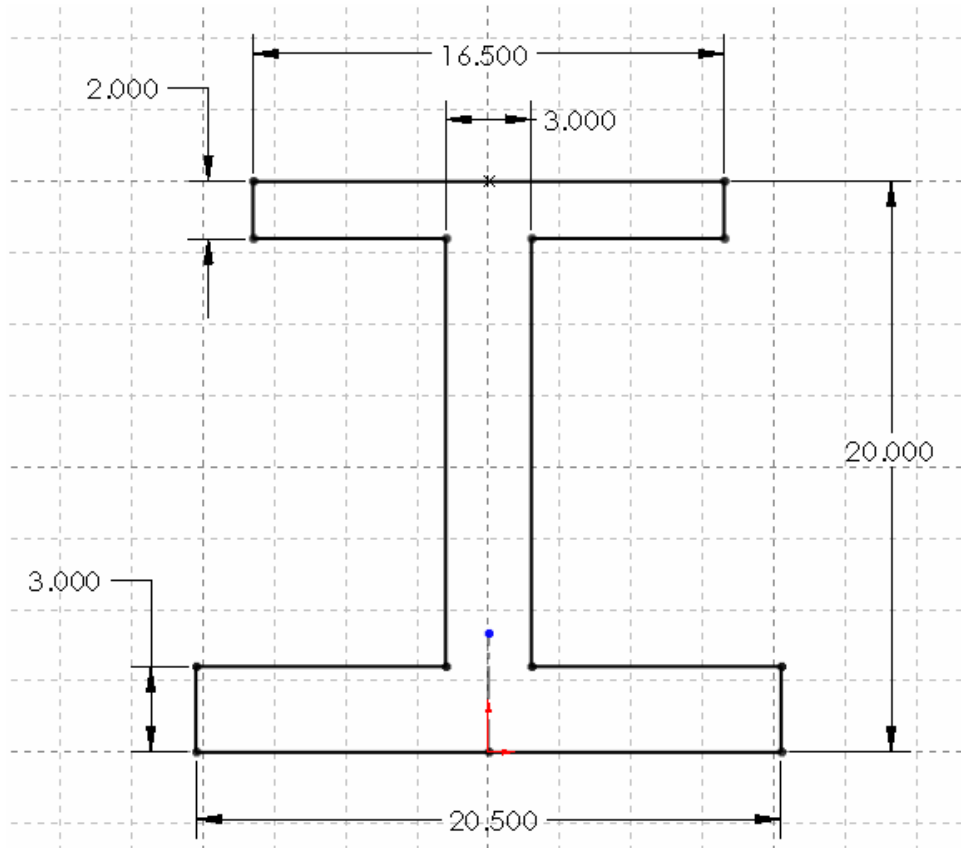
For comments or suggestions please contact us by email at:

areyes@mechanicad.com

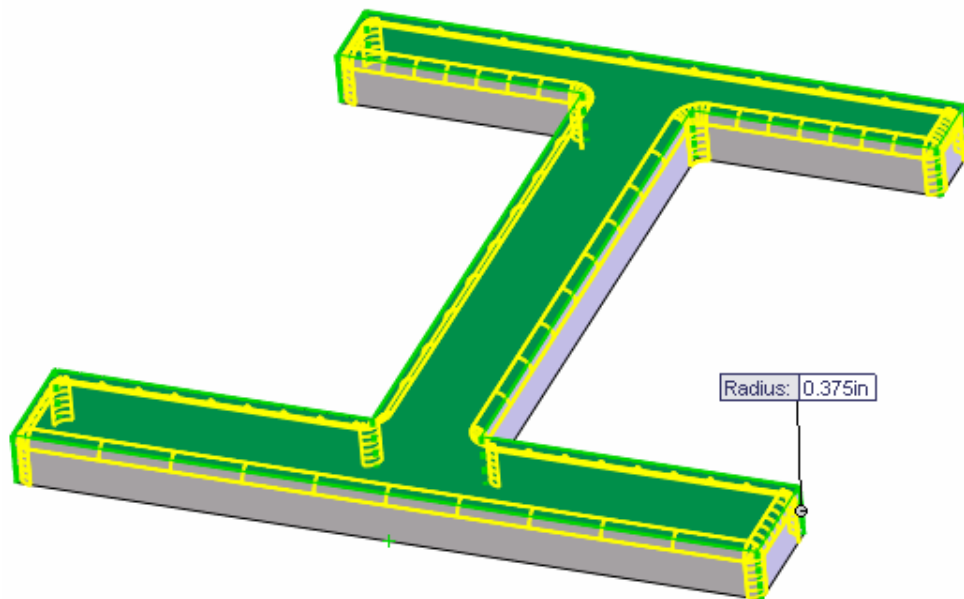
Base:



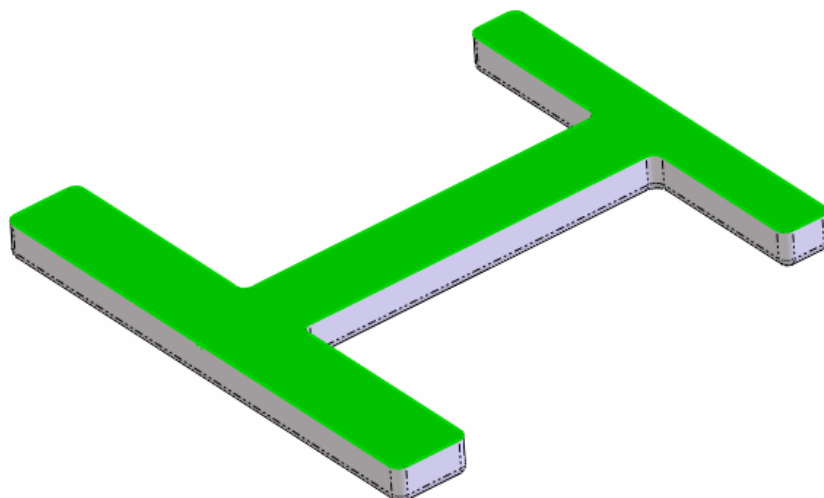
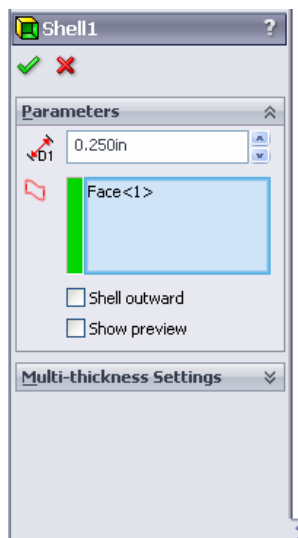
Sketch1 on Top Plane. Extrude 1.5"



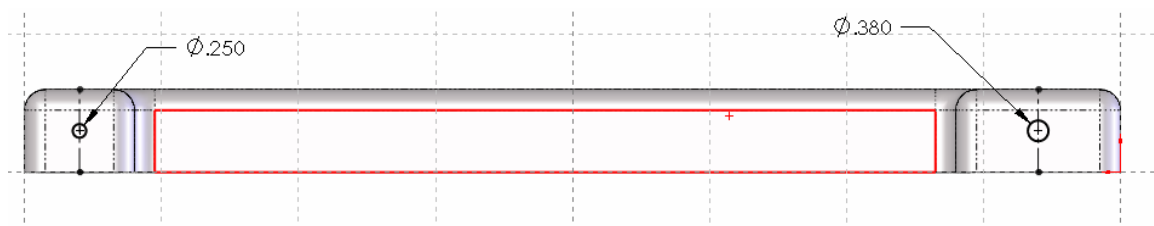
Fillet 3/8"



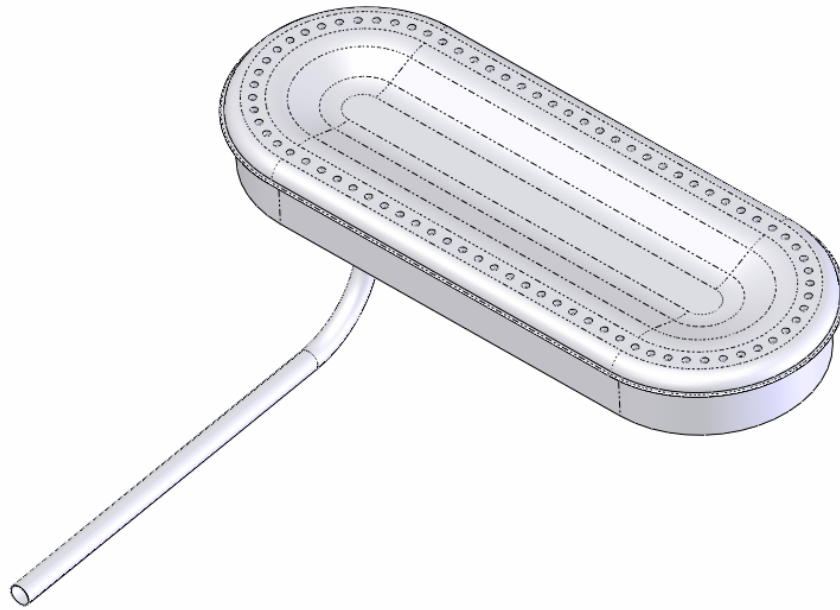
Shell 0.25", remove bottom face



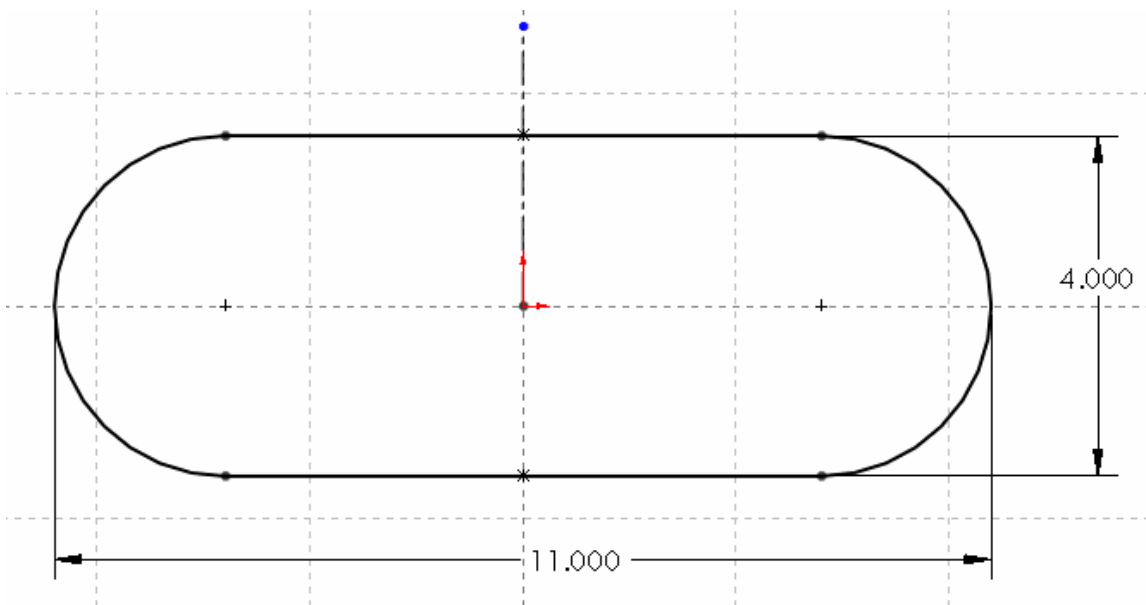
Sketch2 on Right Plane (Left View). Cut Through All both directions.



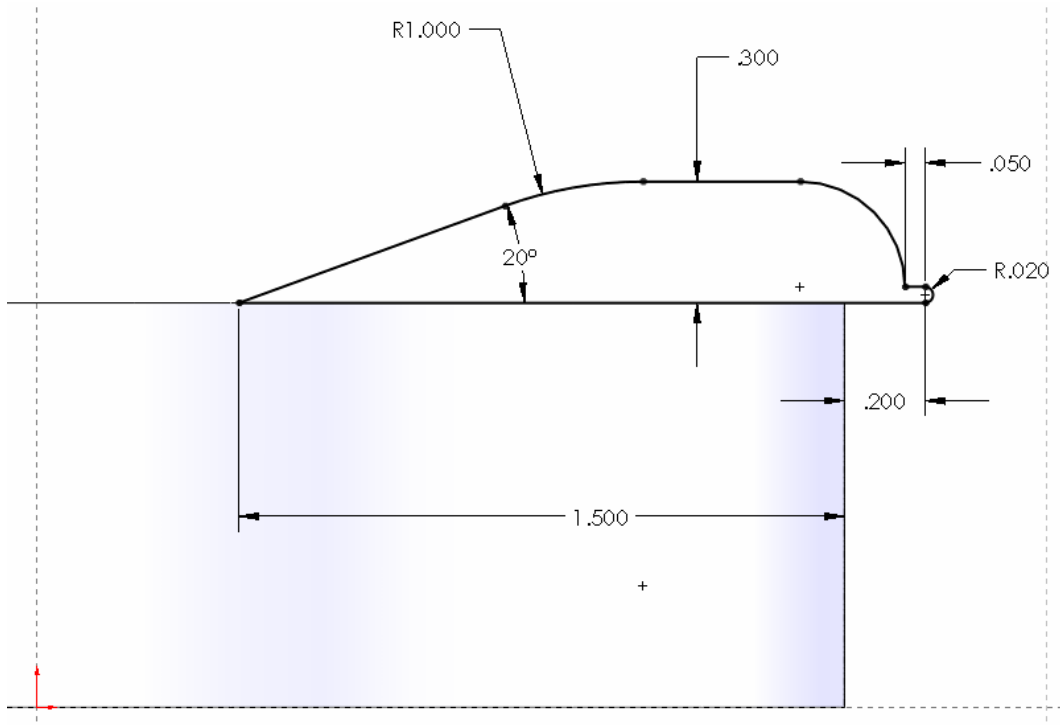
Burner



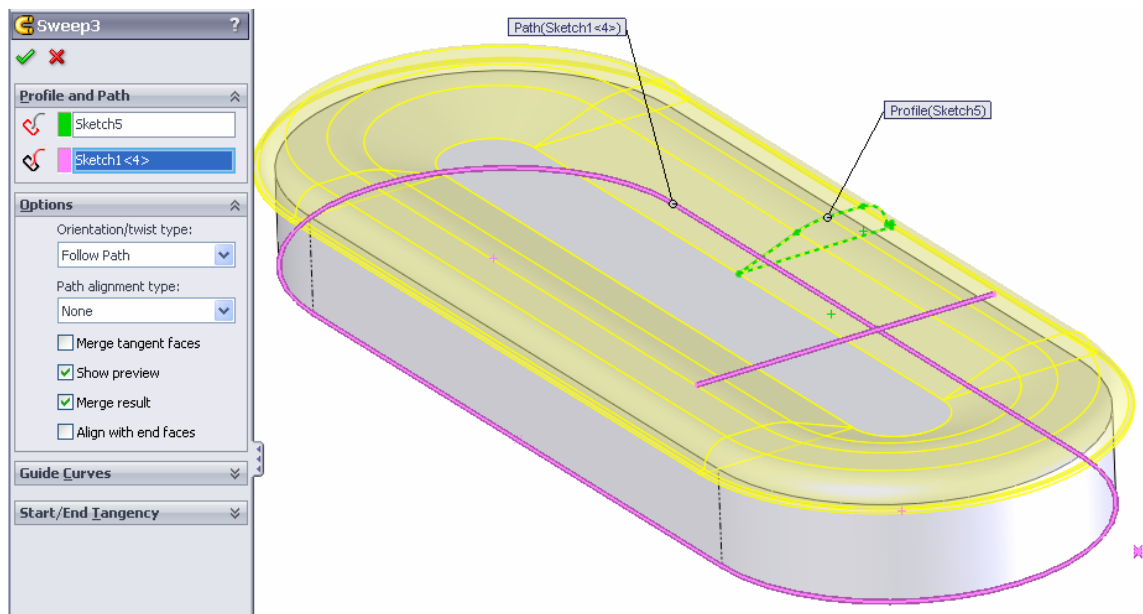
Sketch Top Plane. Extrude 1"



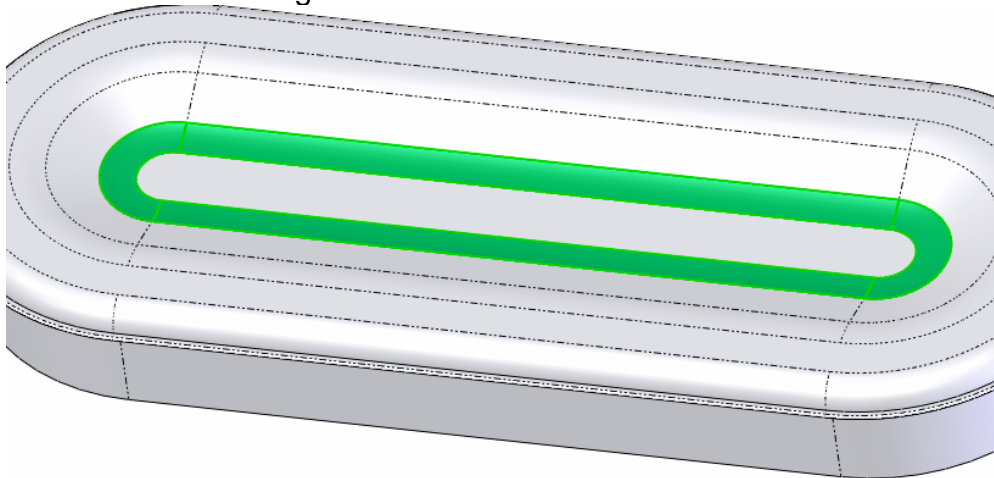
Sketch on the Right Plane (Right View). Exit Sketch. This is the sweep profile



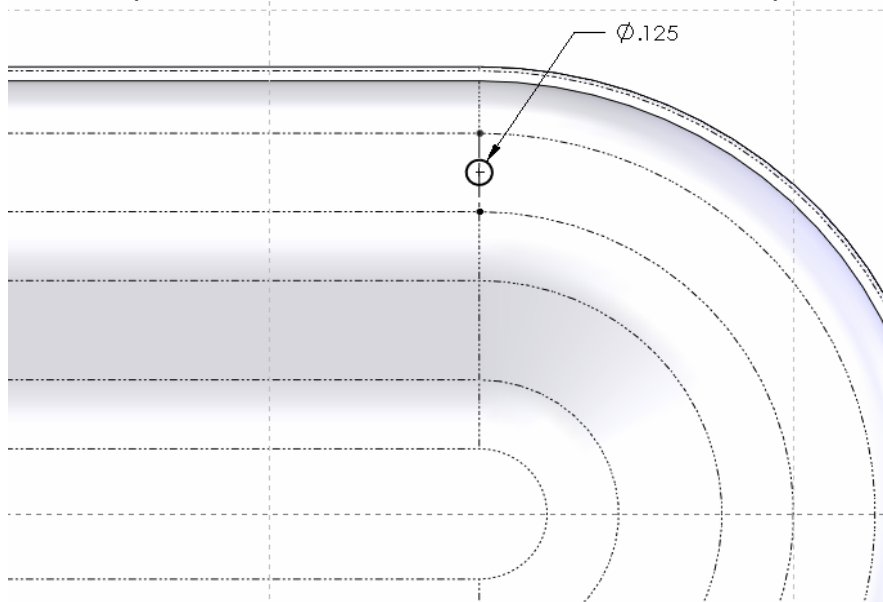
Make Sweep, reuse Sketch1 (yes, you can reuse a sketch!) from the first Extrude, select it from the fly-out feature manager as Path, select previous sketch for Profile.



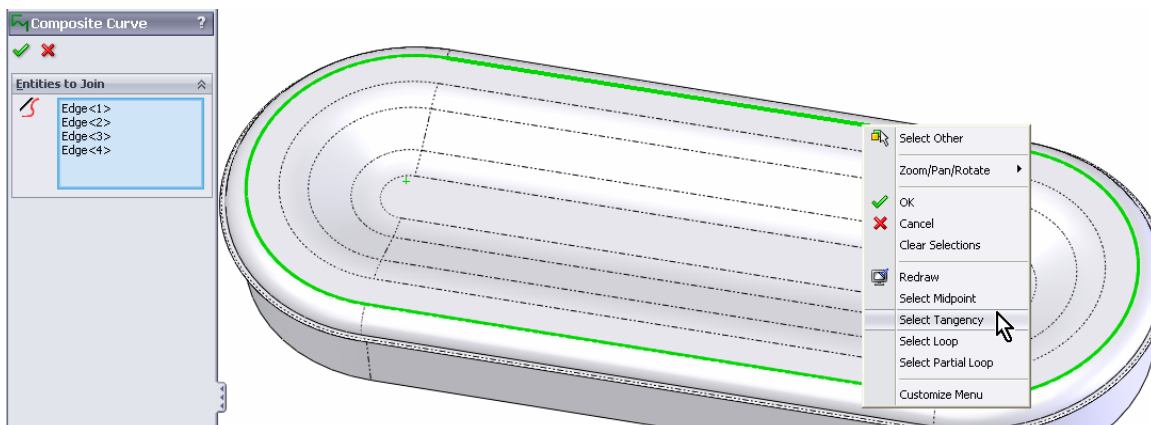
Add Fillet 1" on bottom edge.



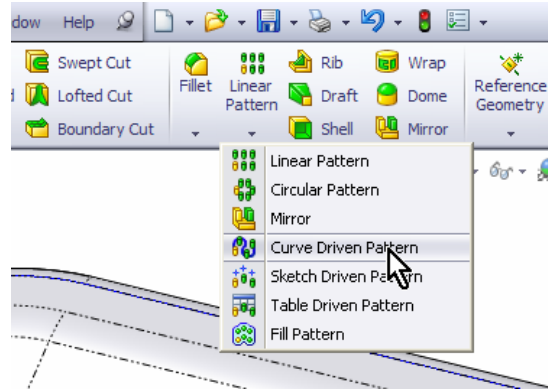
Make sketch on top most face. Use centerline, add circle at midpoint. Cut 0.1"



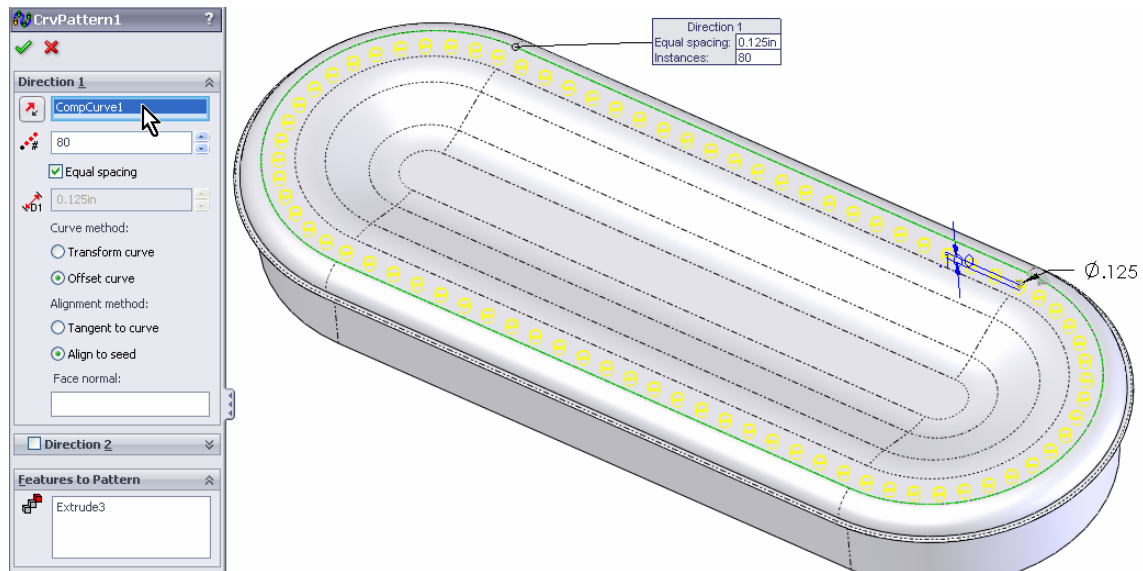
Make Composite Curve using edges from top face. Right-Mouse-Click on one edge, use "Select Tangency" to select all tangent edges at the same time.



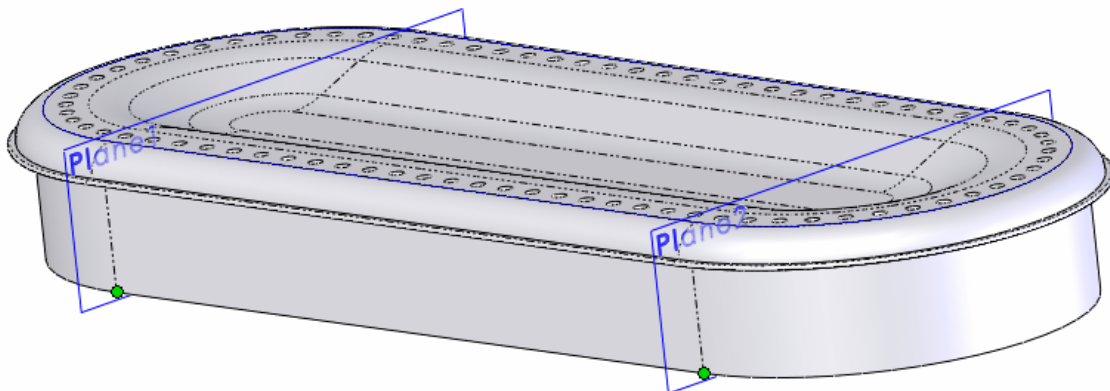
Use **Curve Driven Pattern** to pattern holes around the composite curve.



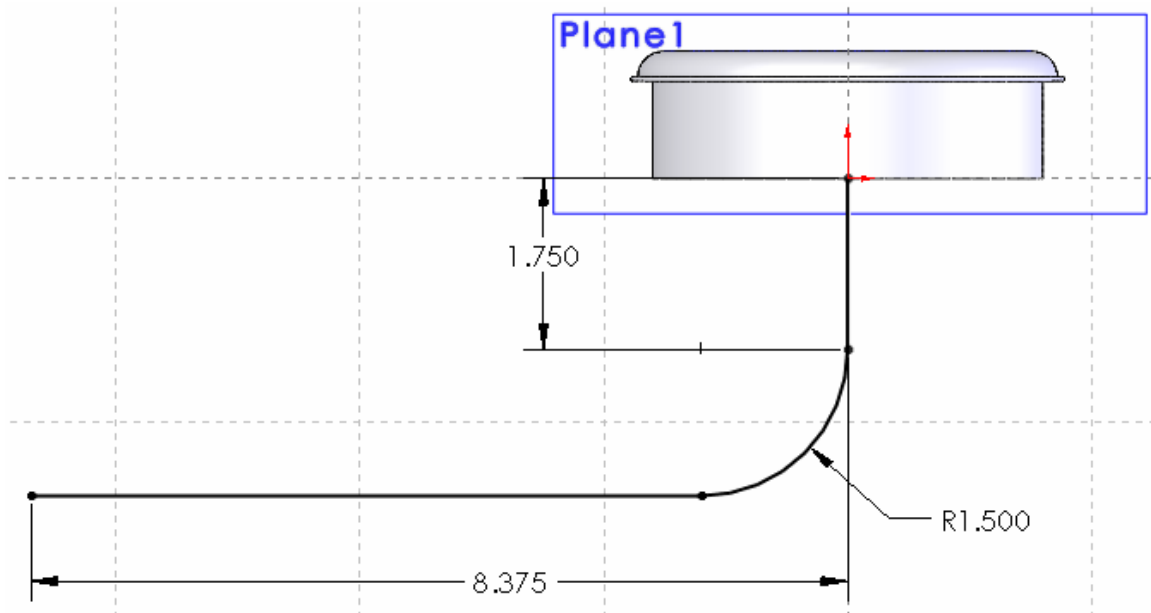
Select composite curve from fly out feature manager, not from the screen. Select the previous hole to pattern. Use "Offset Curve" and "Equal Spacing" options.



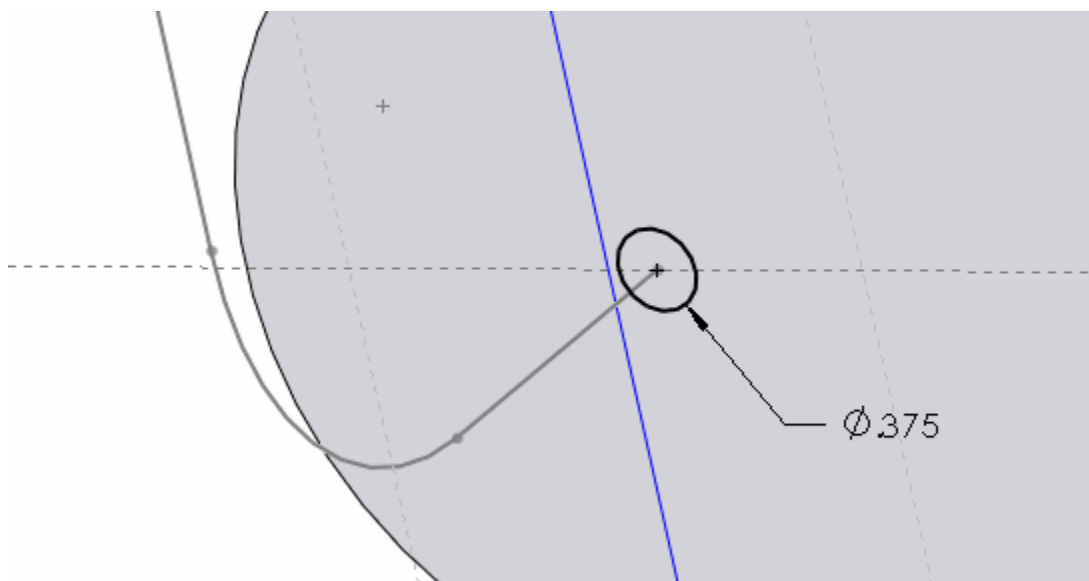
Make 2 auxiliary planes parallel to "Right" plane at the end points indicated.



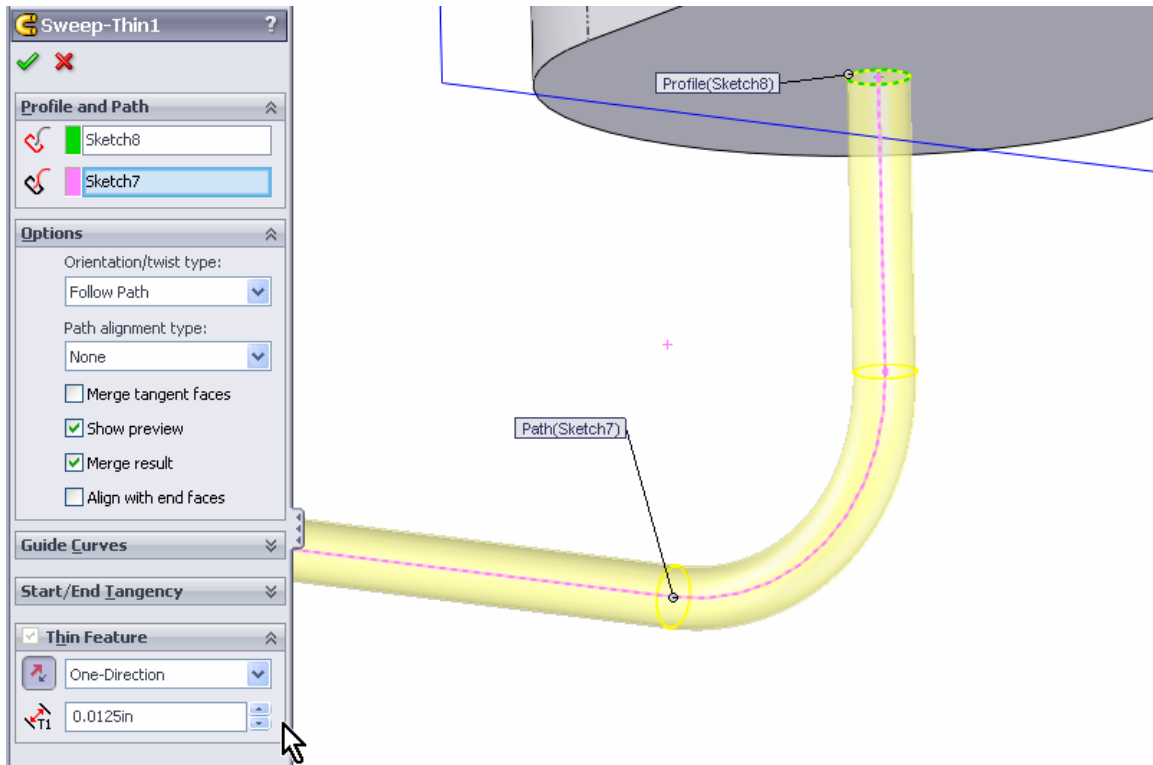
Make sketch on first aux. plane to be the path for a tube. Exit sketch.



Make the profile sketch on the bottom face coincident to the path sketch before.

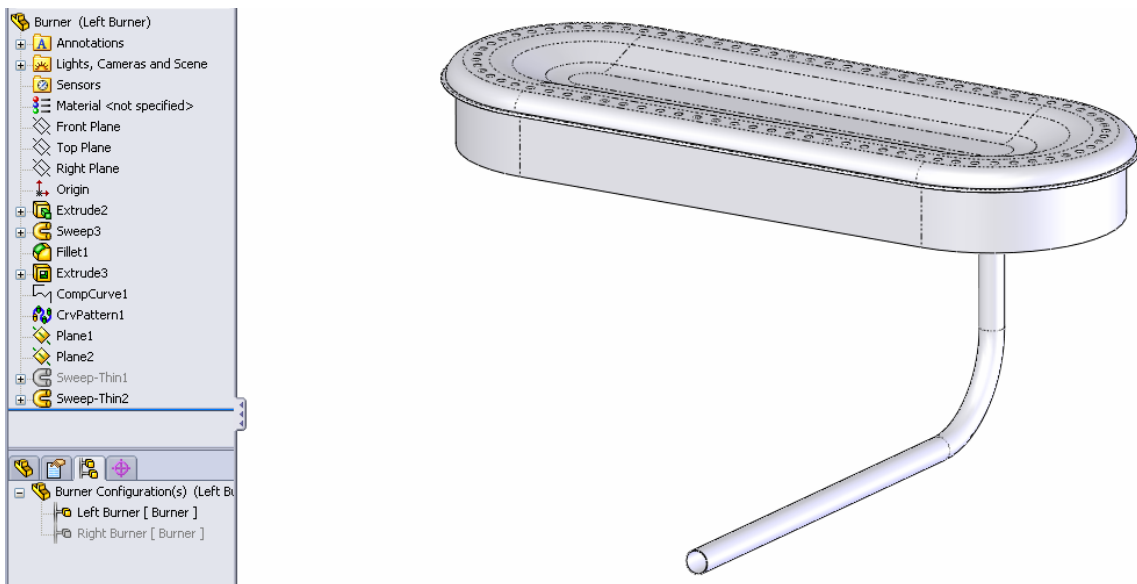


Make sweep using “Thin Feature” option 0.0125” thick.

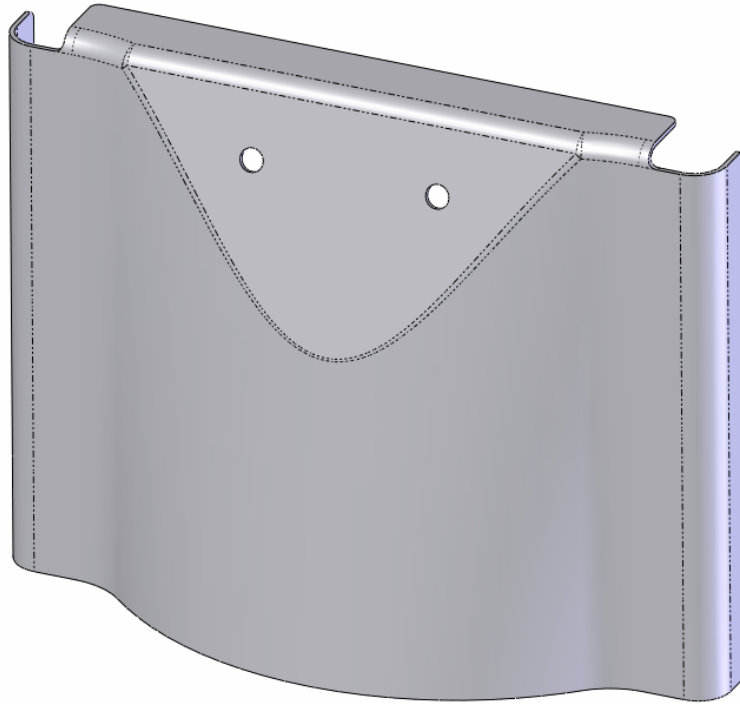


Repeat path and profile on second aux. plane to make second sweep.

Make 2 configurations “Left Burner” & “Right Burner”. Suppress left sweep for left burner, right sweep for right burner configuration.

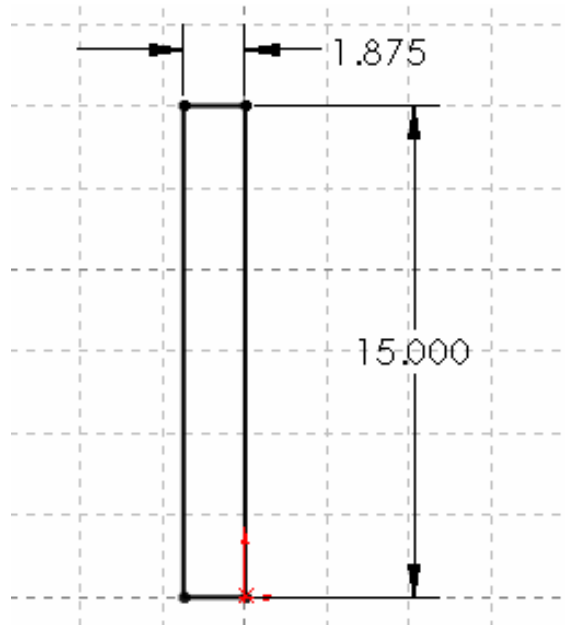


Front Cover

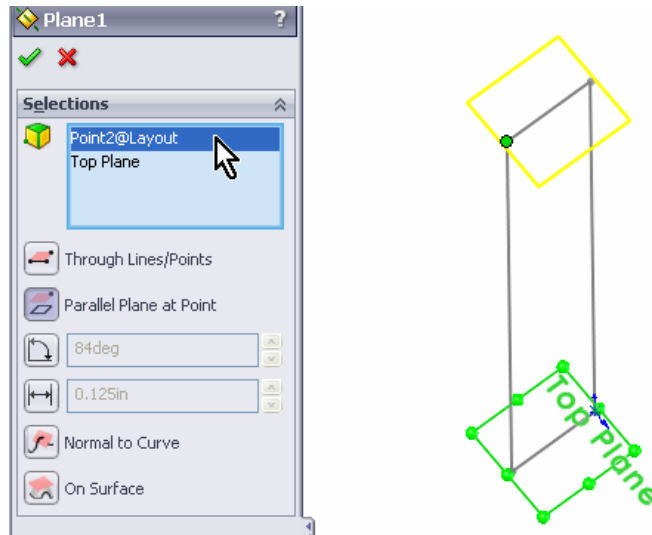


First feature is a Sweep using 3 guide curves.

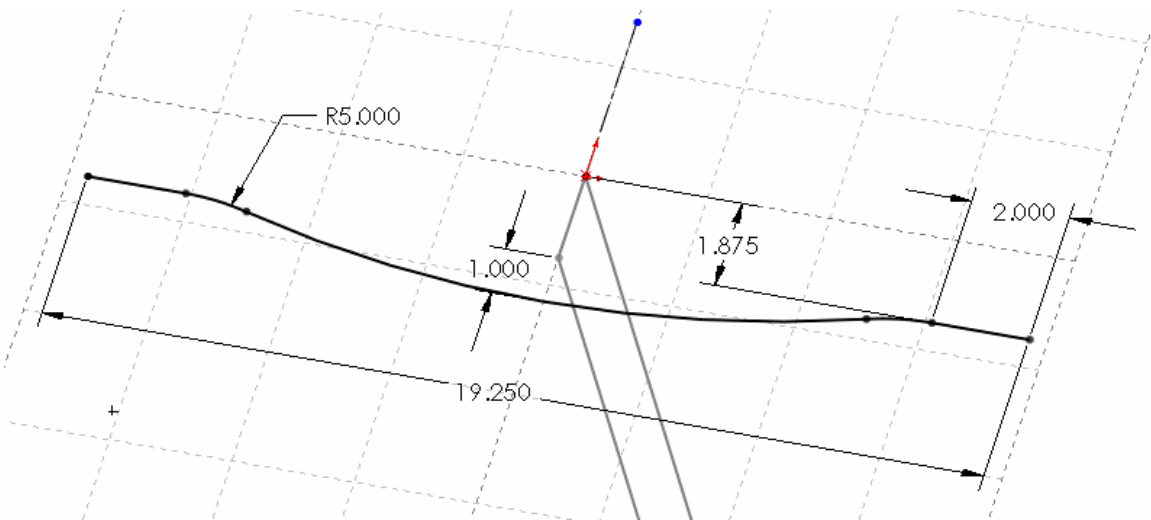
Make *layout* sketch on right plane. Exit Sketch. This is just to use as reference to create planes. Rename Layout.



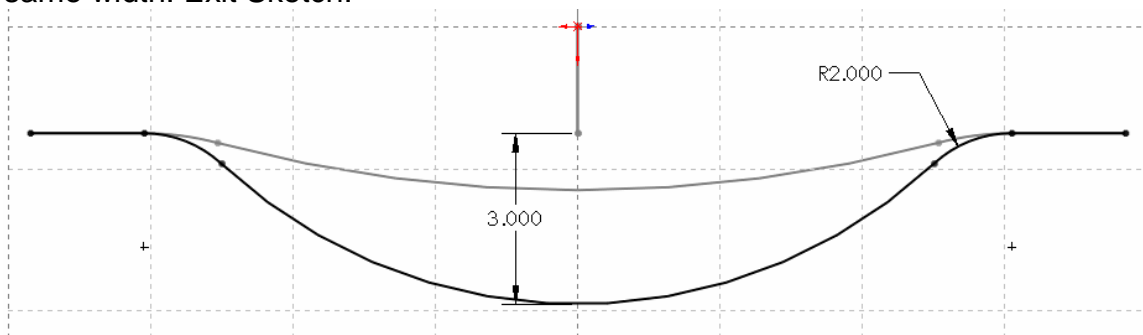
Create a plane parallel to Top at top of layout sketch.



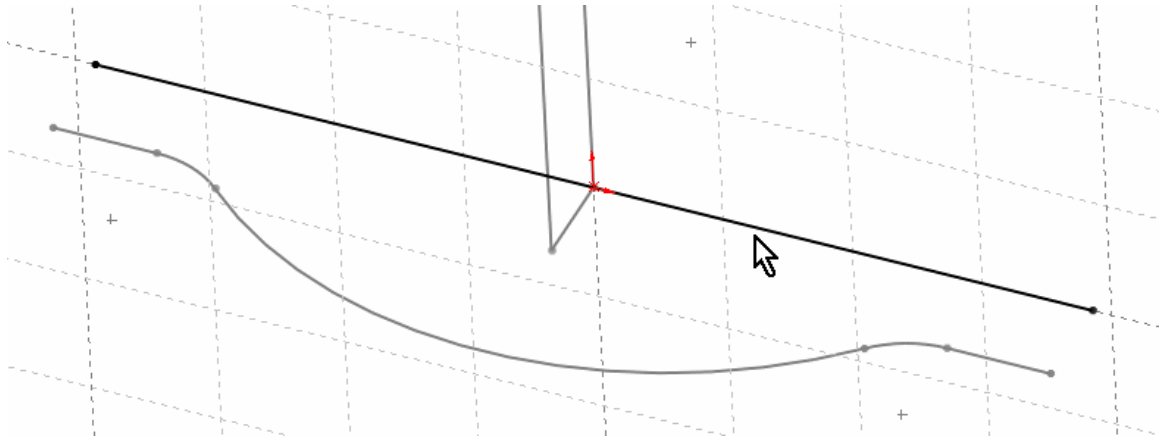
Make sketch on Plane1 as follows. This is a guide curve. Use the Layout sketch to locate. Must be symmetrical. Exit Sketch.



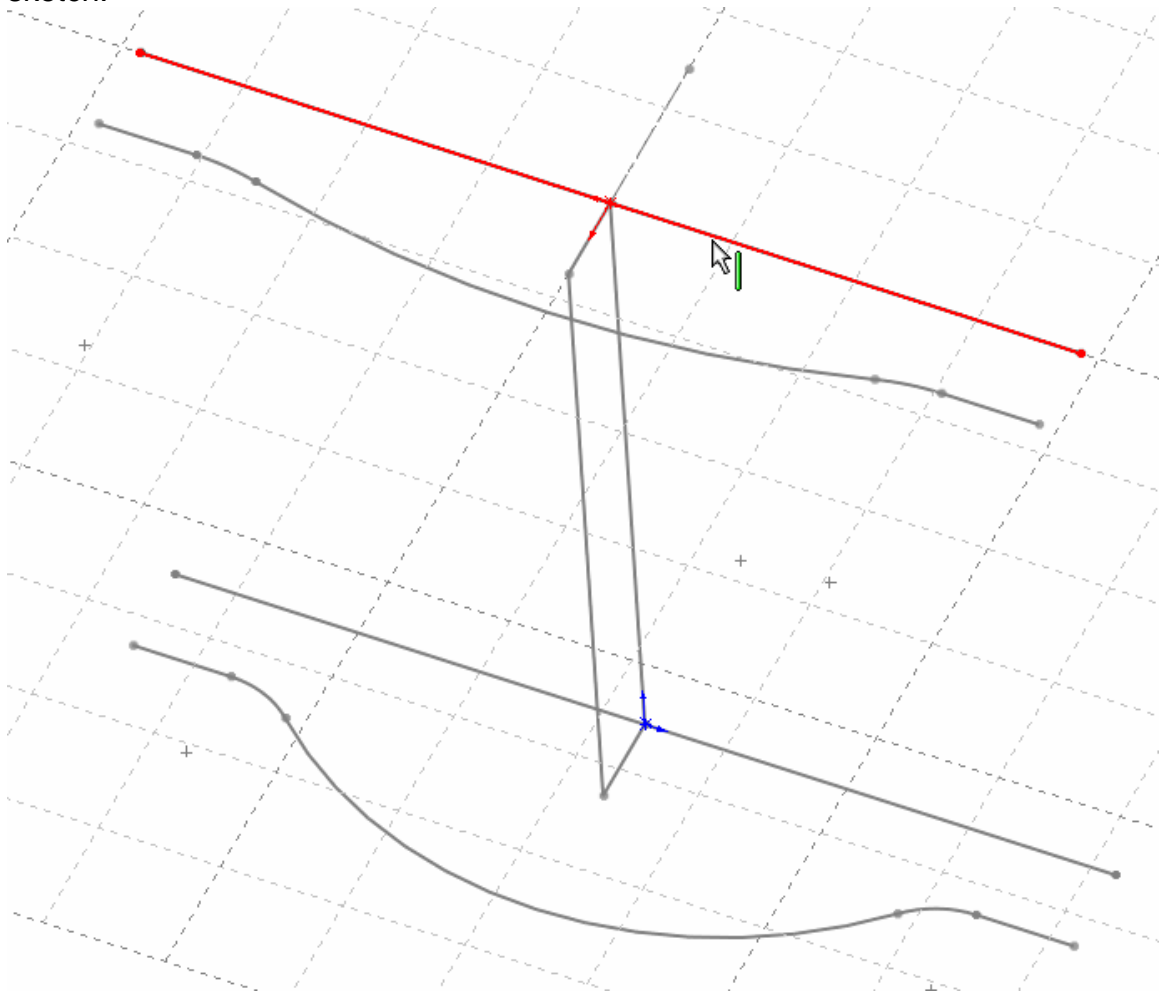
Make sketch on "Top Plane". Add coincident relations to first guide curve to be same width. Exit Sketch.



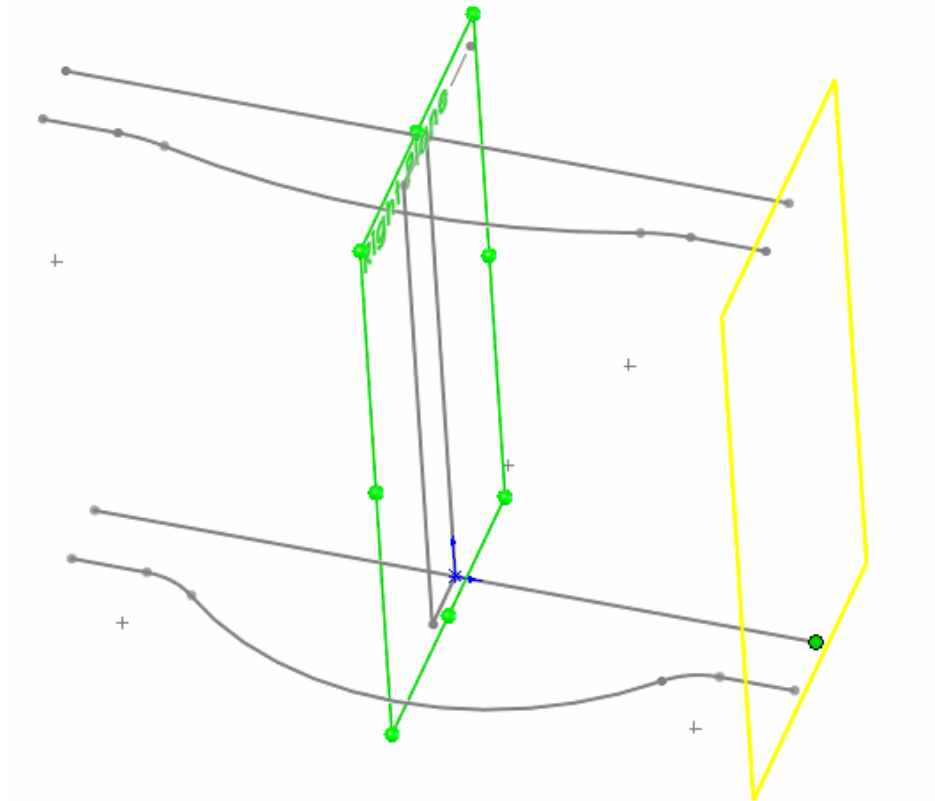
Make sketch on Top Plane, this is the path. Make same width as guides using relations, no dimensions needed. Exit sketch.



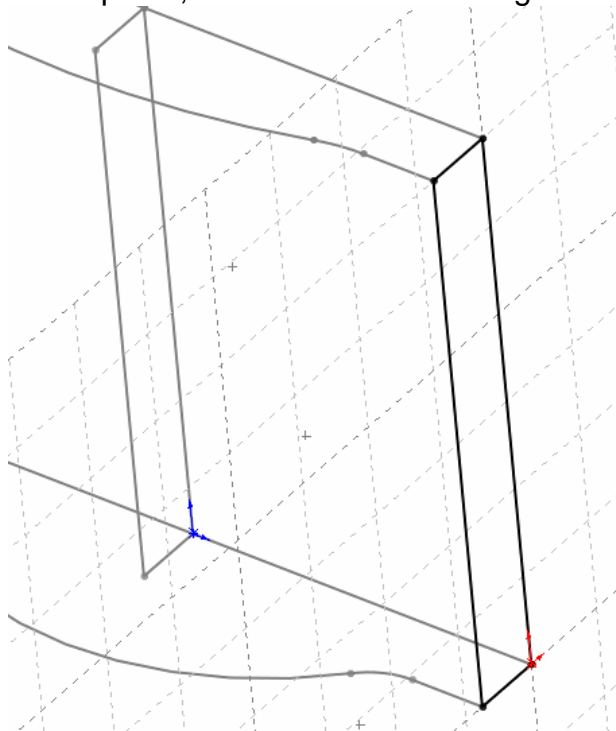
Make sketch on Plane1, same as path sketch. This is the 3rd guide curve. Exit sketch.



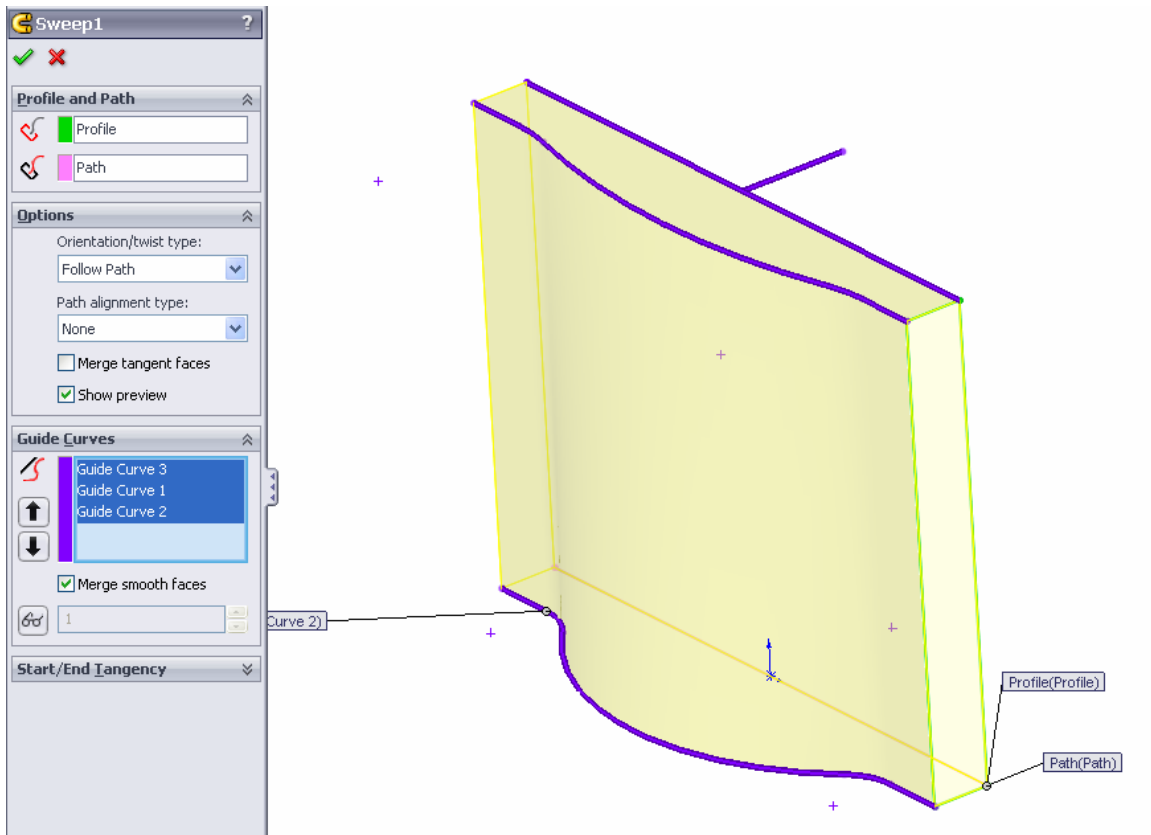
Make parallel plane at end of path.



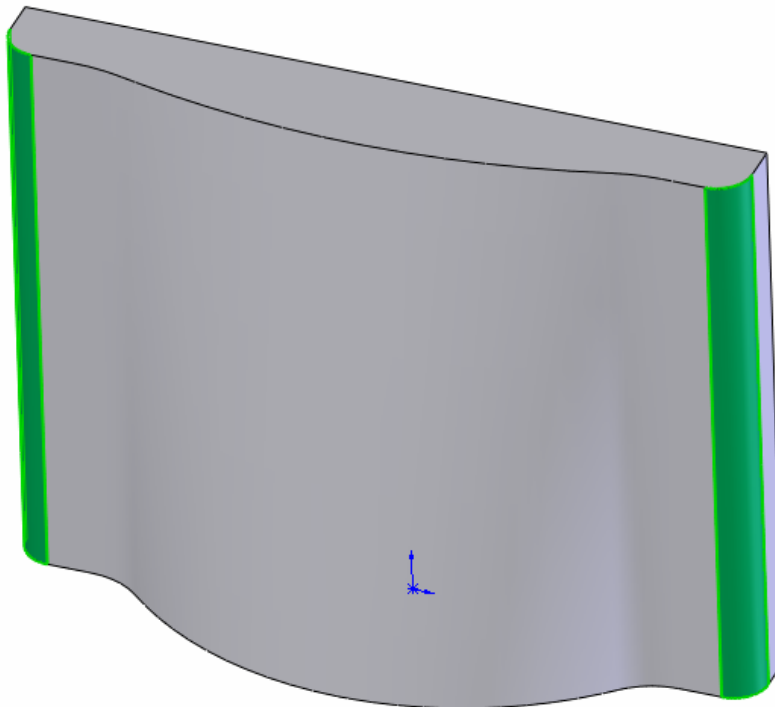
Make Profile path on this plane, make coincident to all guides and path.



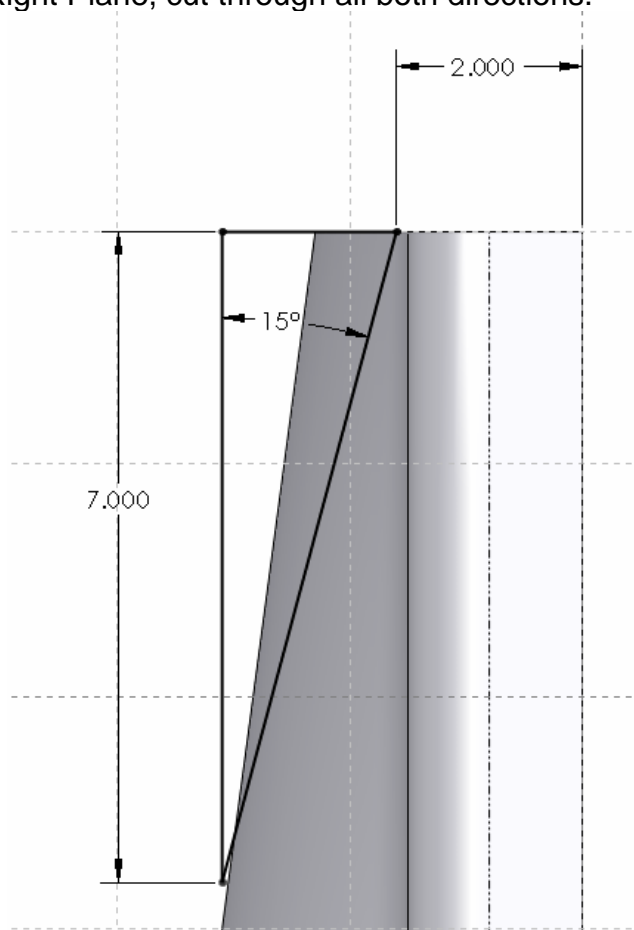
Make sweep using all 3 guide curves.



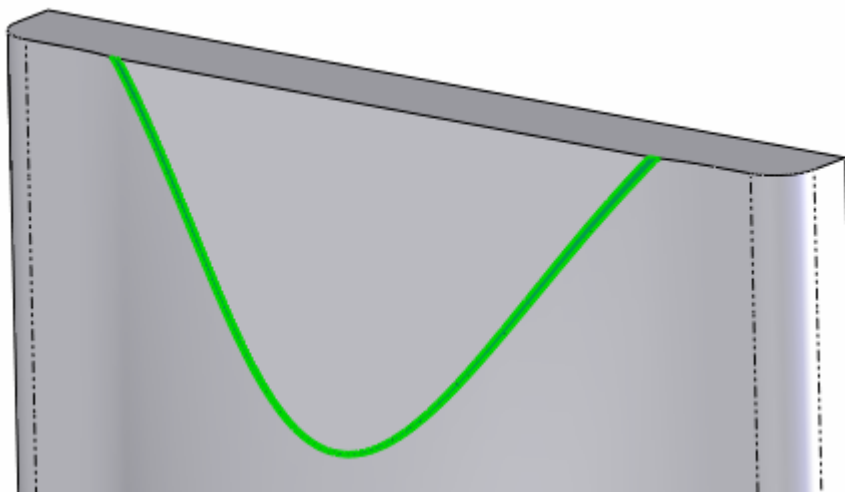
Fillet corners 0.875"



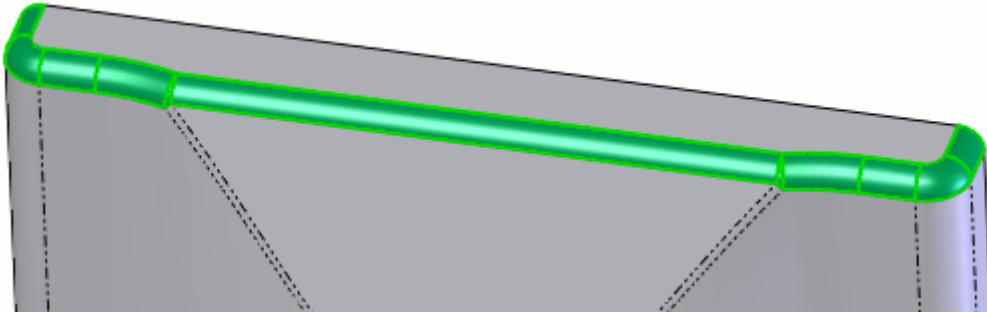
Make sketch on Right Plane, cut through all both directions.



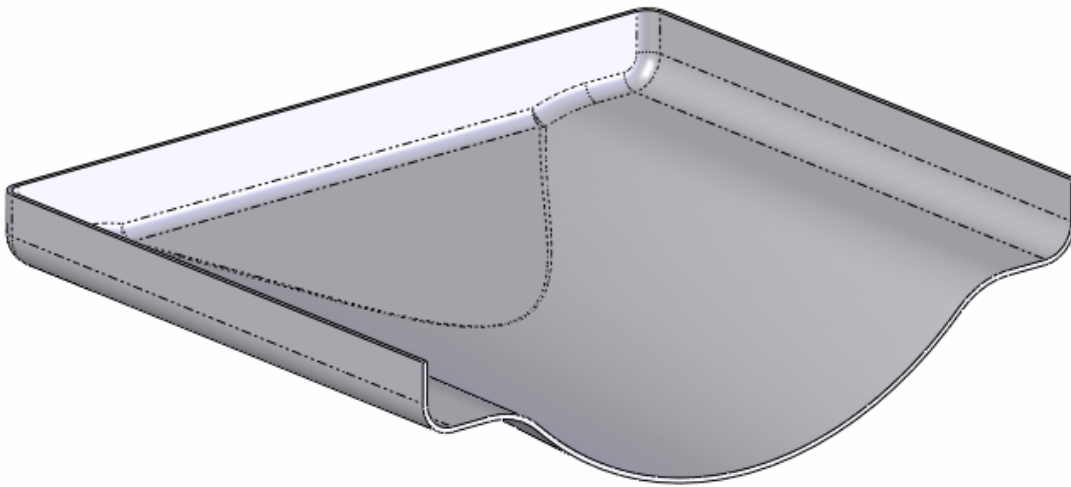
Round edges 0.5"



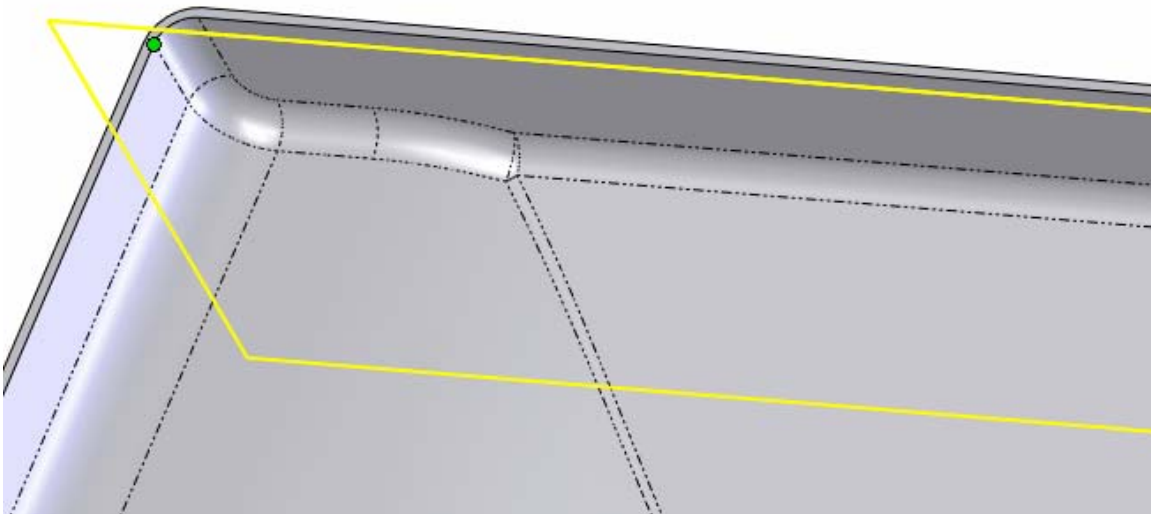
Round edges 0.5"



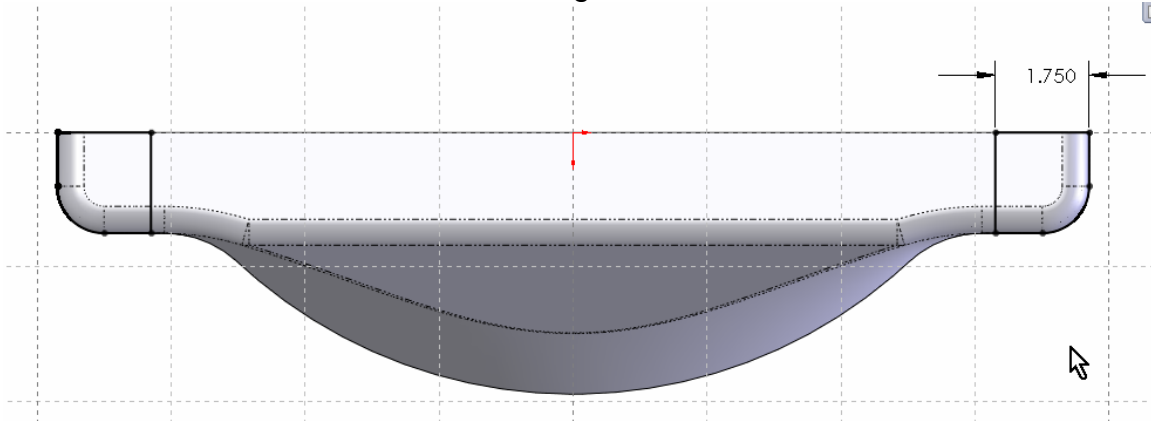
Shell part 0.125"



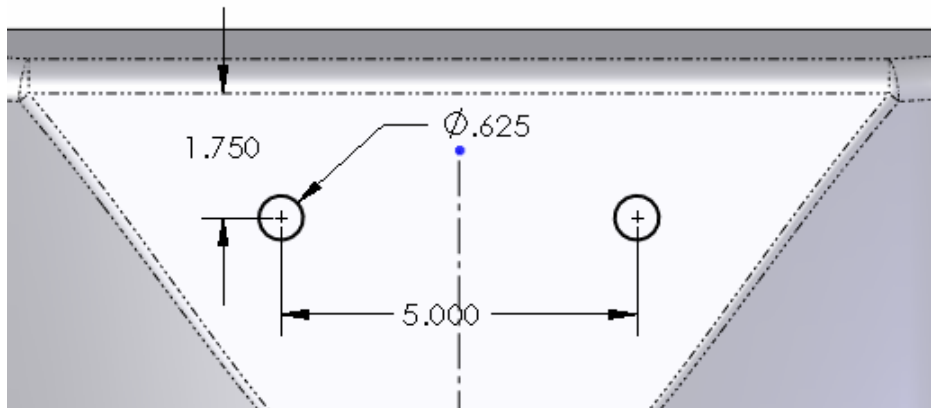
Make parallel plane at vertex.



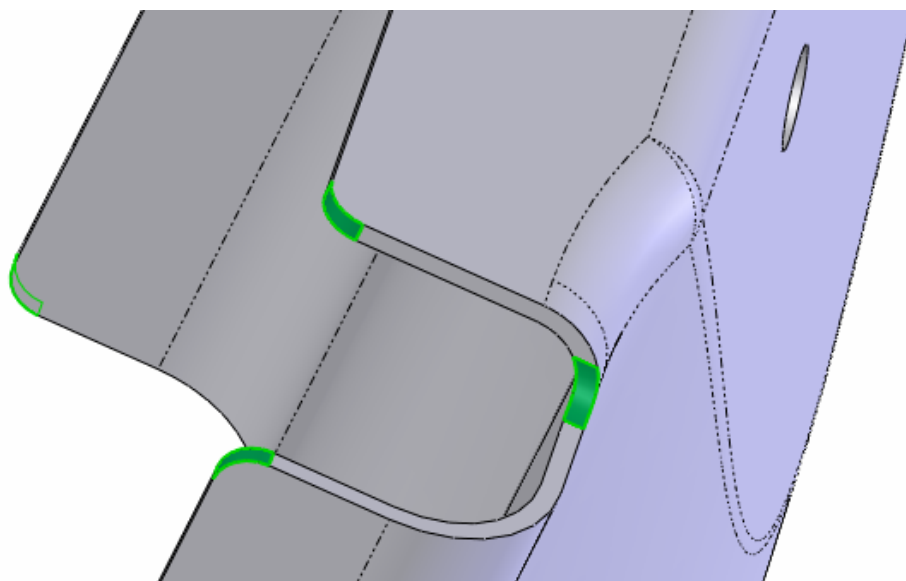
Make sketch on this plane using convert entities and dimension as shown. Cut through all



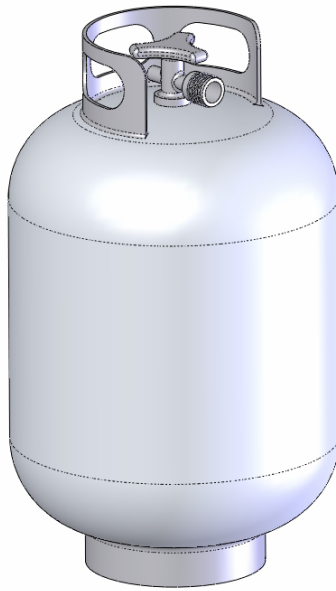
Make sketch on front flat face, cut through all (Knob cutouts).



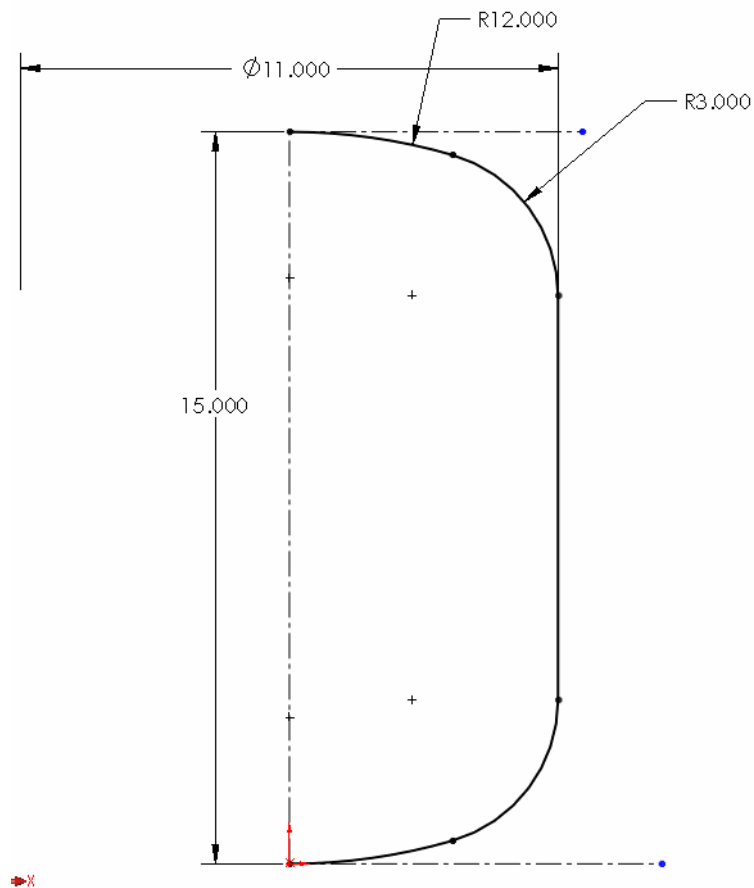
Round corners 0.25"



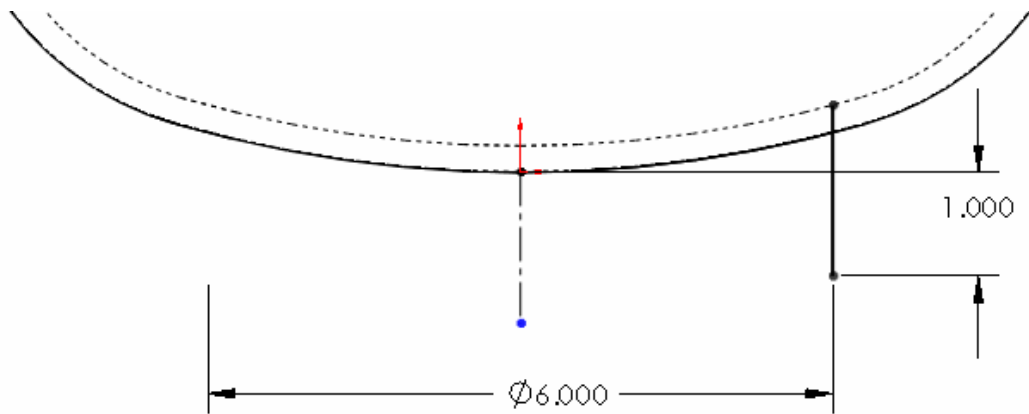
Gas Tank



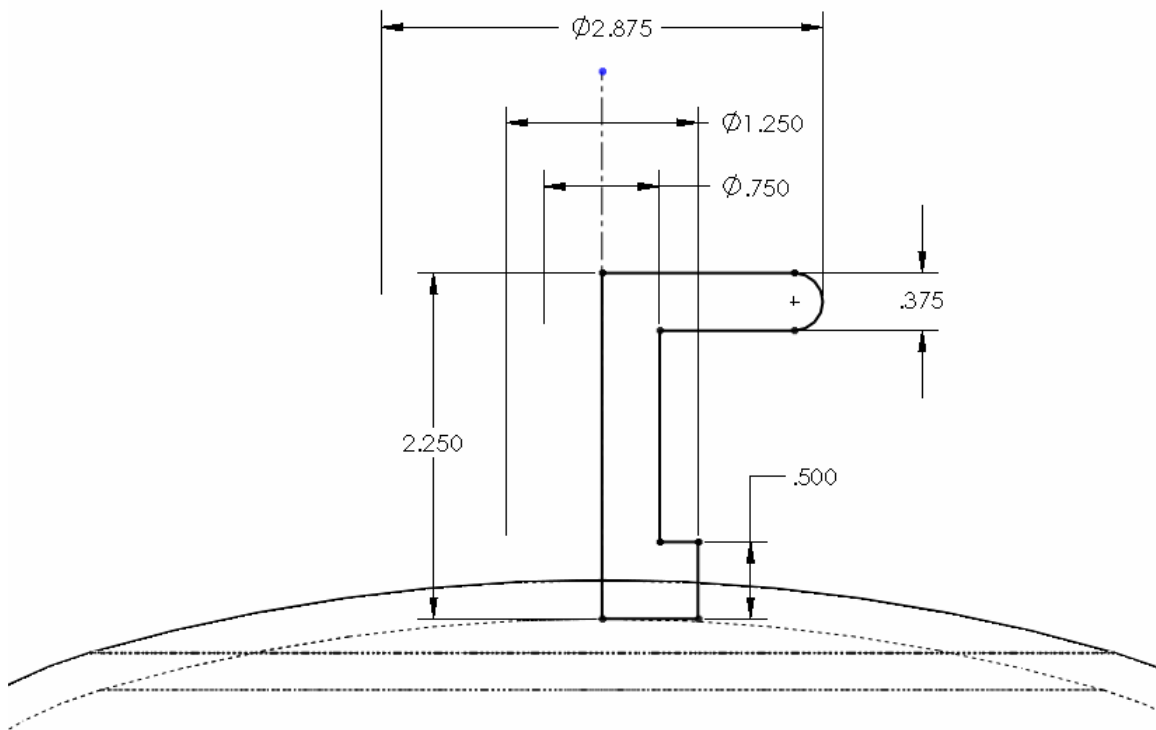
Sketch on Front Plane. Make revolved boss using thin feature option 0.250" thick



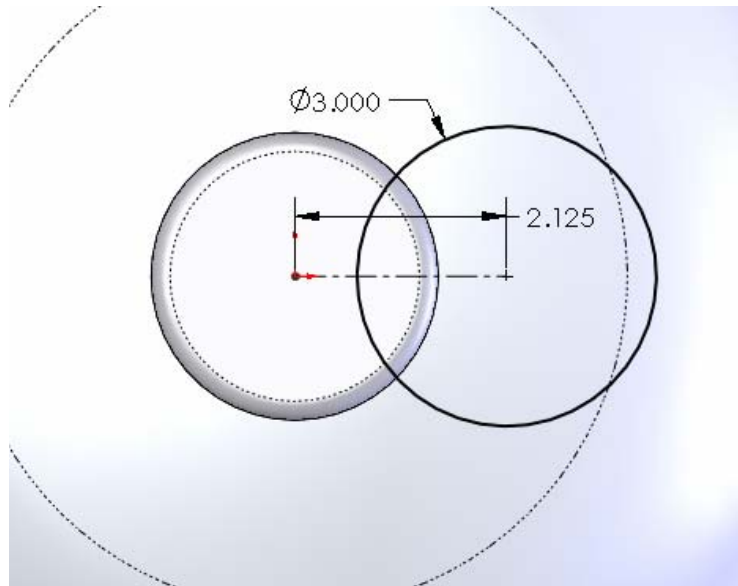
Sketch in the Front Plane, Revolved boss thin feature 0.125" thick to make base of tank. Make sure to touch the inside of the tank, the 6" diameter is internal dimension.



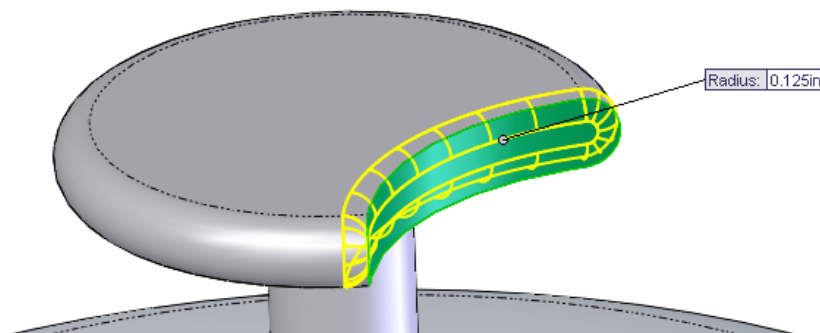
Sketch in the Front Plane, revolved boss to make valve body



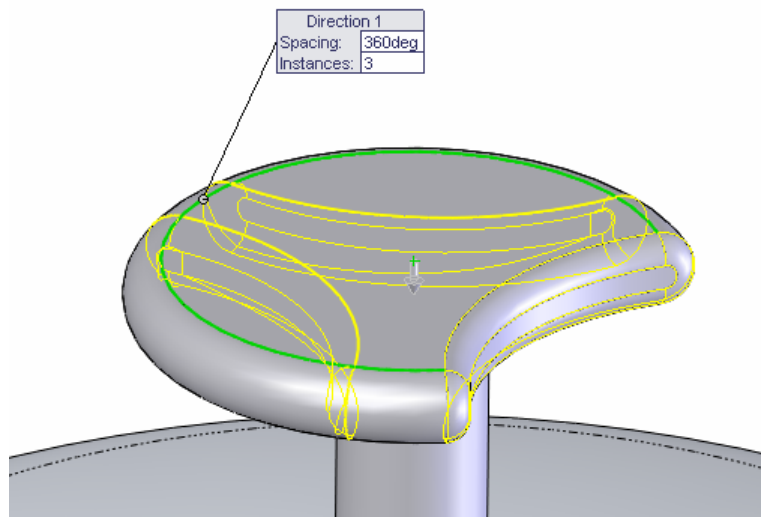
Sketch on top of valve to make cut in valve. Cut using Up to Next option.



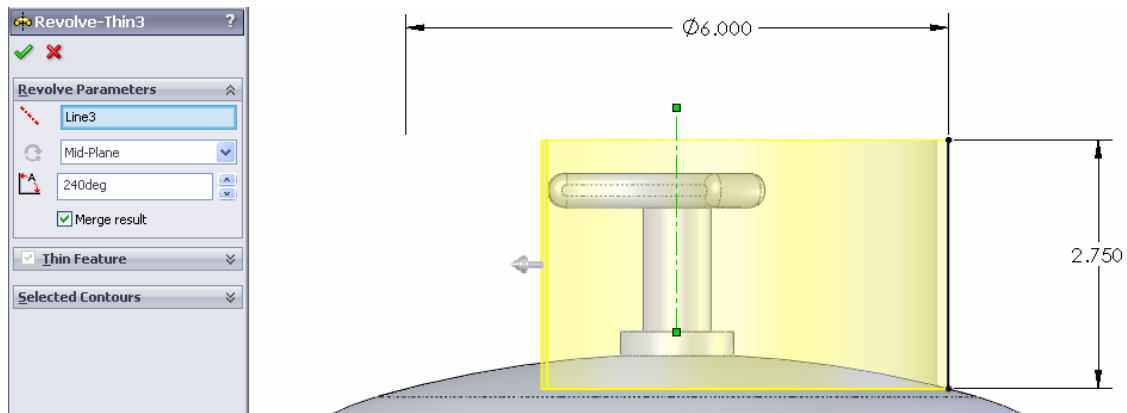
Fillet face 0.125"



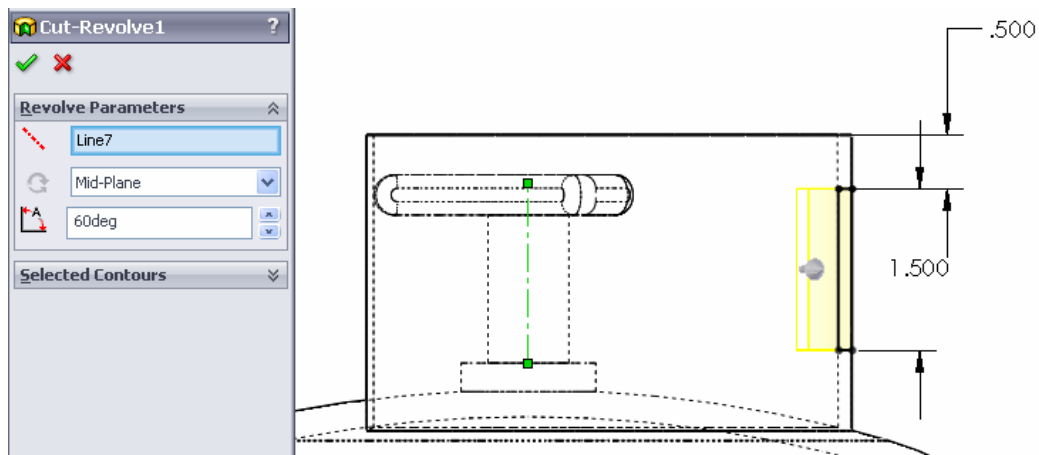
Pattern cut and fillet 3 times.



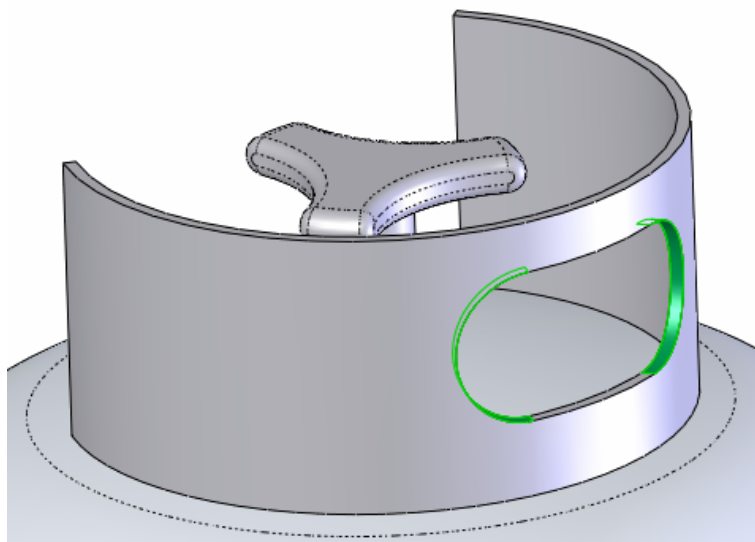
Sketch on Front Plane, revolved boss thin feature. 6" diam. is outside dimension. Revolve 240°, mid-plane.



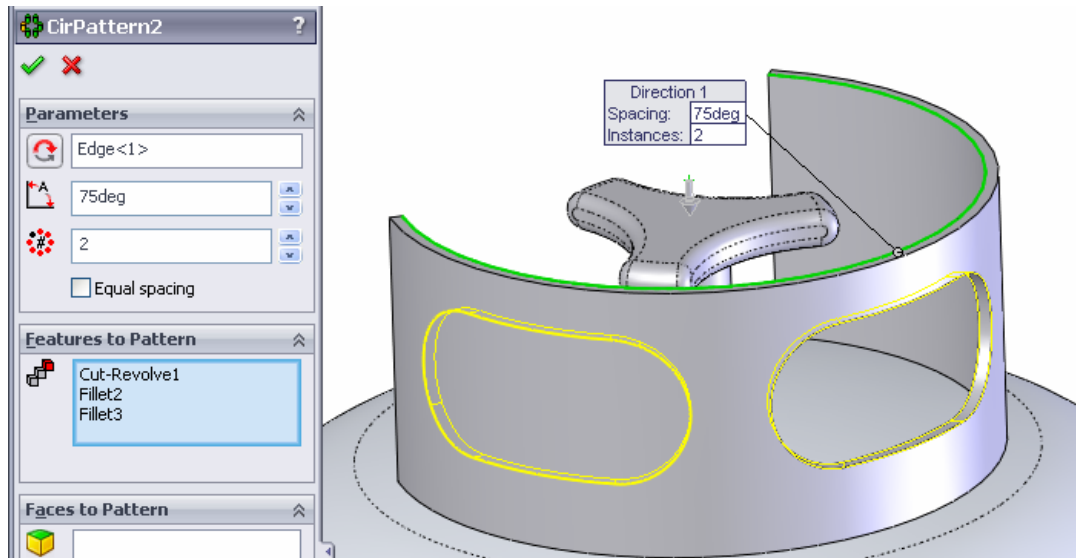
Sketch on Front Plane, cut revolve 60° mid-plane option.



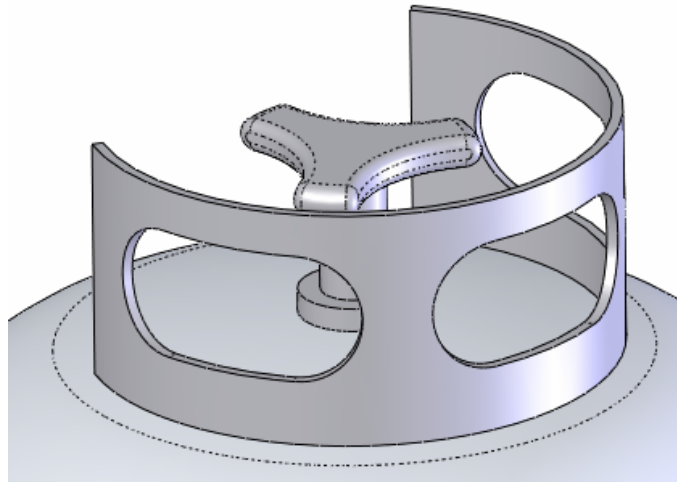
Add full round fillets to round edges.



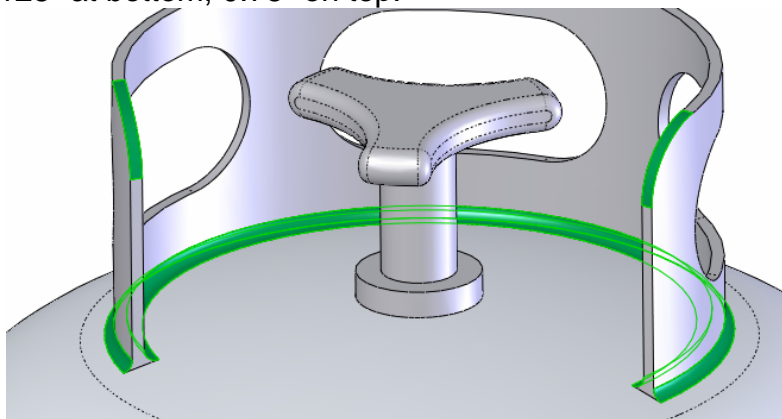
Circular Pattern with previous cut and fillets, 2 instances 75° spacing.



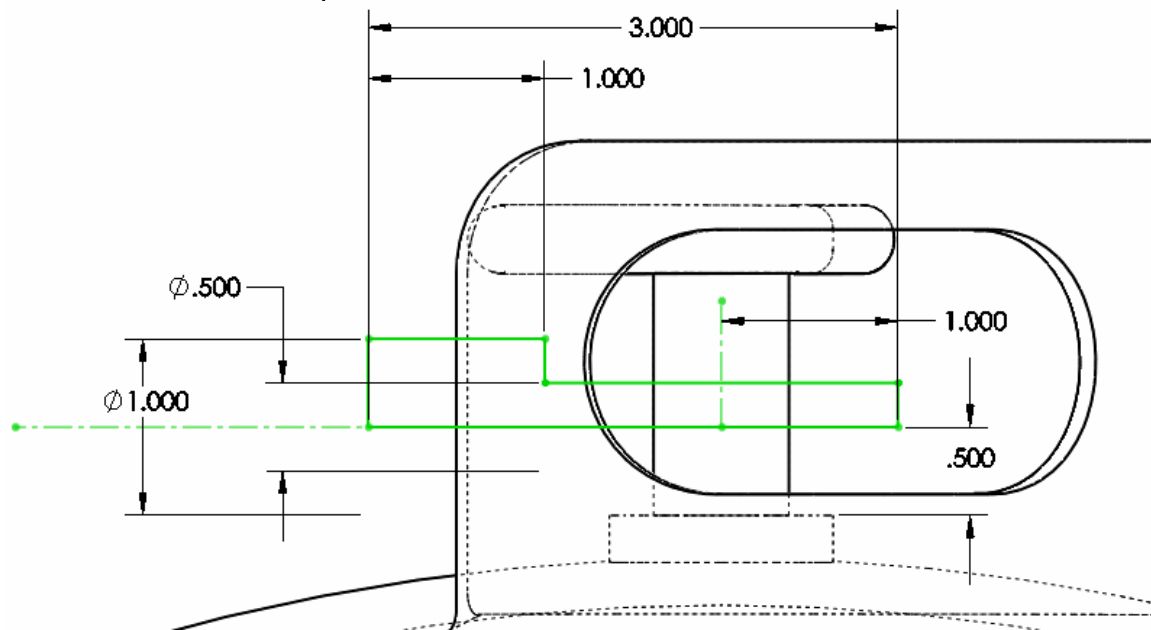
Repeat in other direction to have 3 cutouts complete.



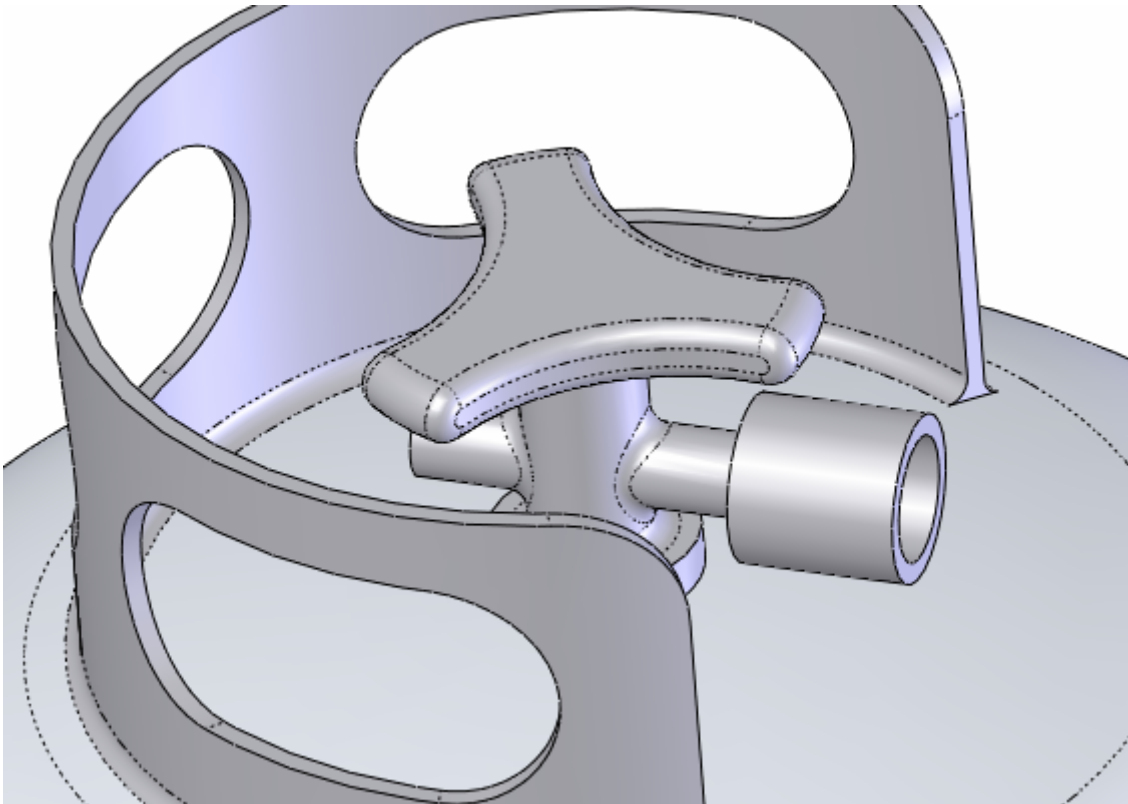
Add fillets 0.125" at bottom, 0.75" on top.



Make sketch on Front plane. Make revolved boss.

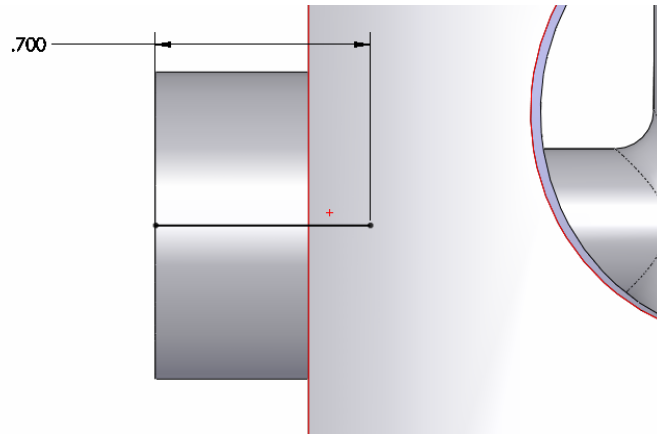


Make inside cut 0.75" diameter and 0.75" deep. Add .125" fillets.

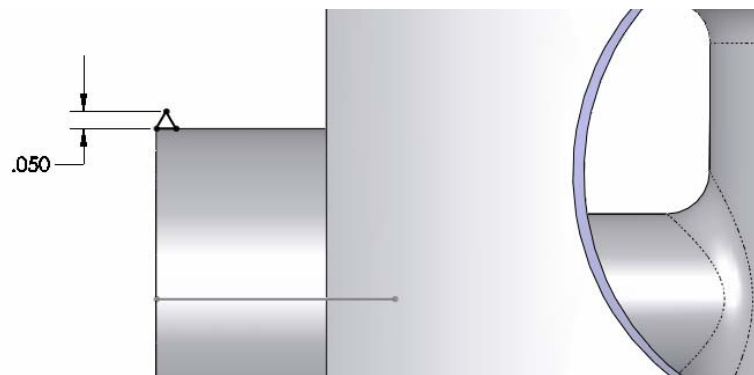


Make Thread:

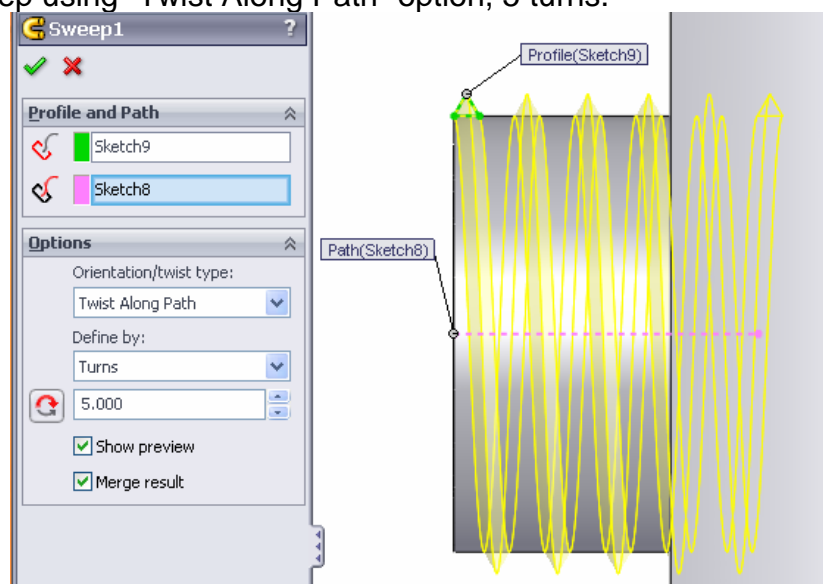
Add sketch for path on Front Plane. Exit sketch.



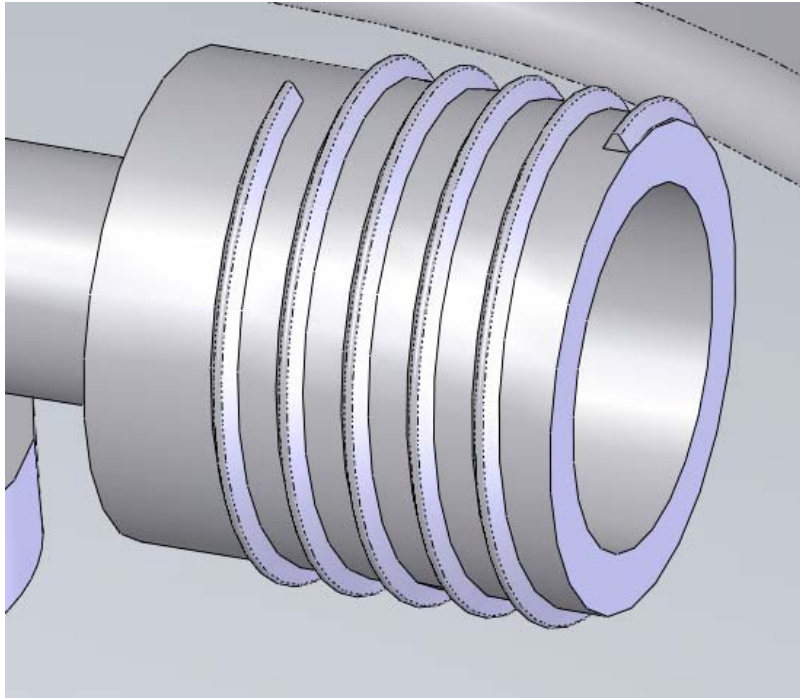
Add sketch on Front Plane for the Profile. (This is not supposed to be a true thread profile, it is meant for cosmetic purposes only)



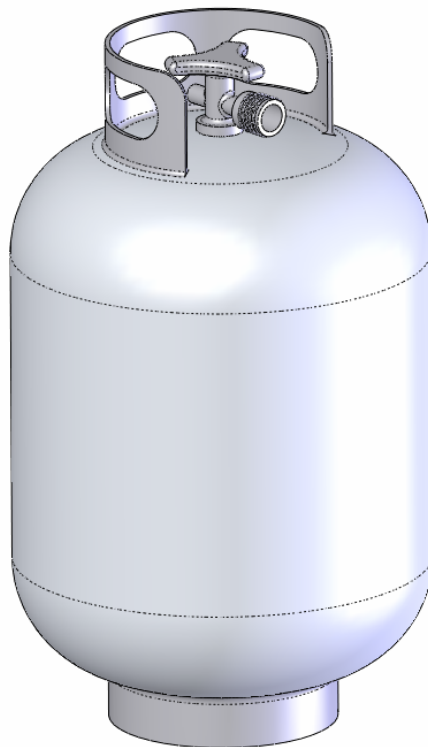
Make Sweep using "Twist Along Path" option, 5 turns.



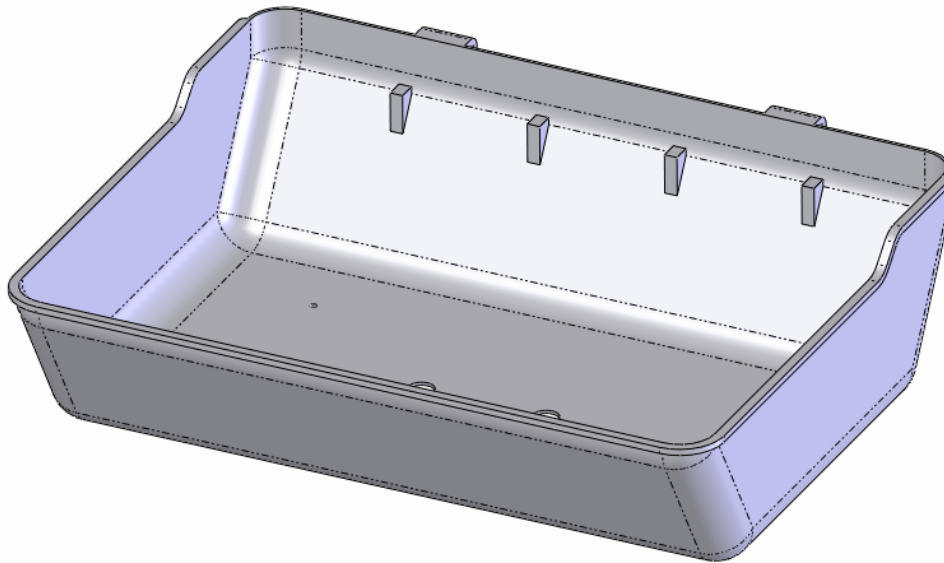
Add a 0.1" fillet to finish the thread.



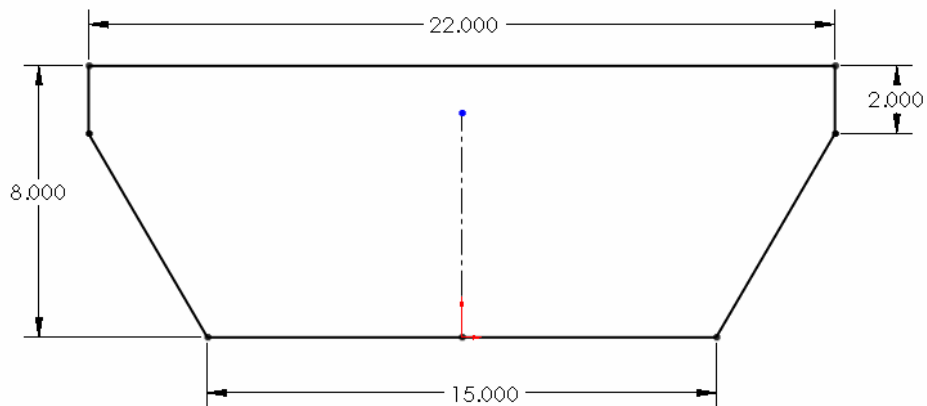
Finished gas tank.



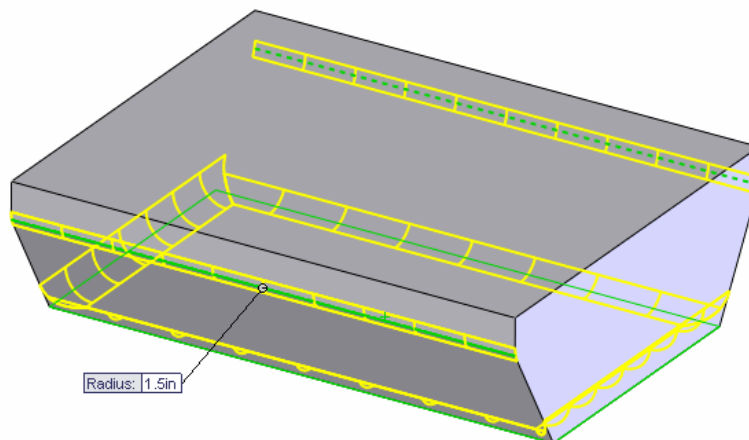
Grill Bottom



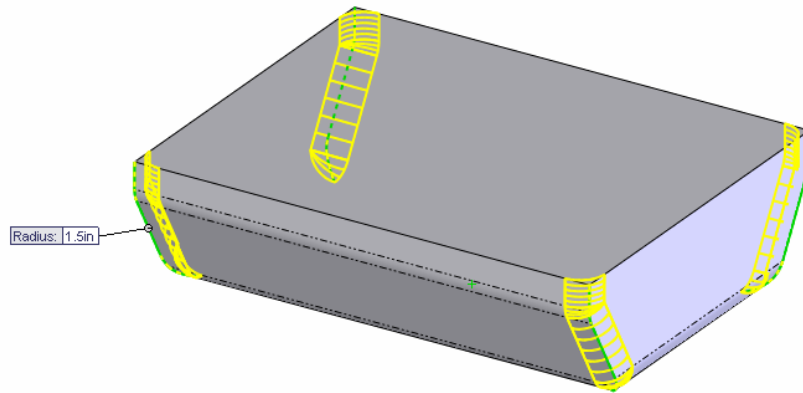
Sketch on the Right Plane, extrude using mid plane option 28"



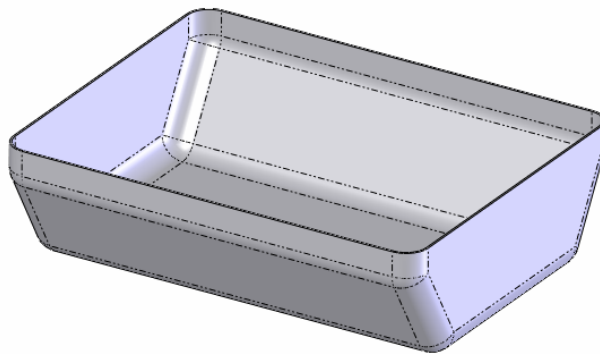
Add 1.5" fillets



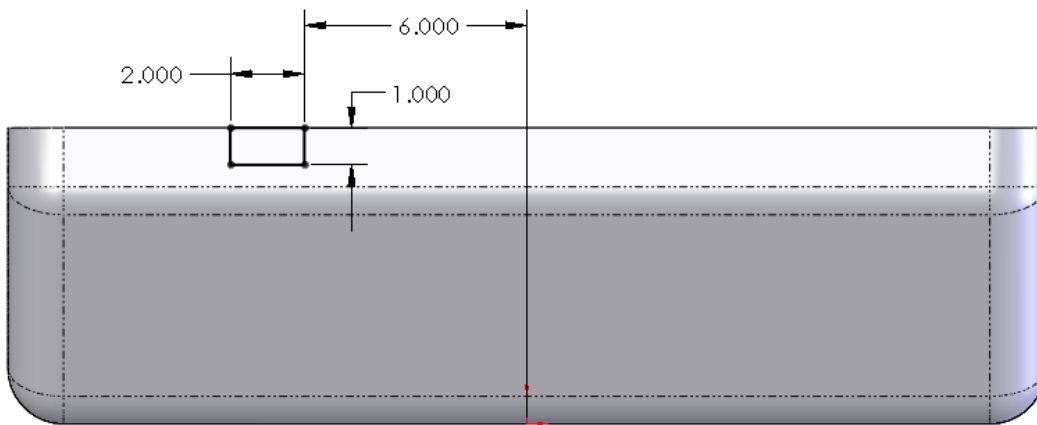
Add corner fillets 1.5"



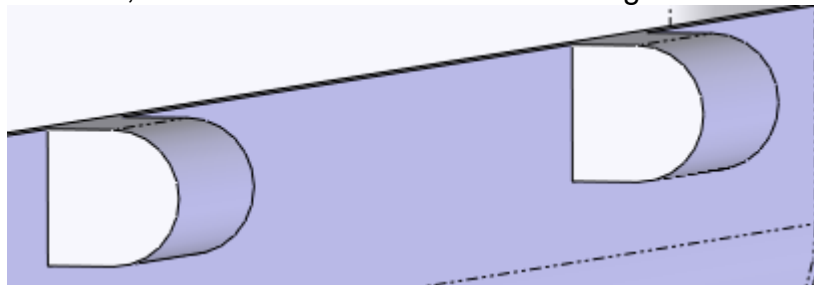
Shell part 0.125"



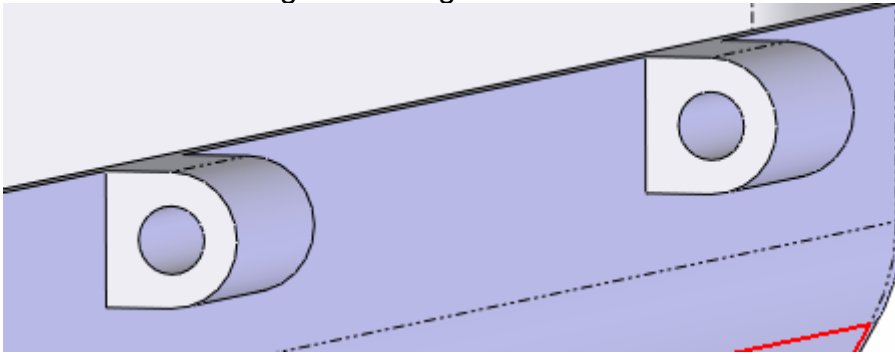
Add boss for hinge on back face. Sketch as shown, extrude 1"



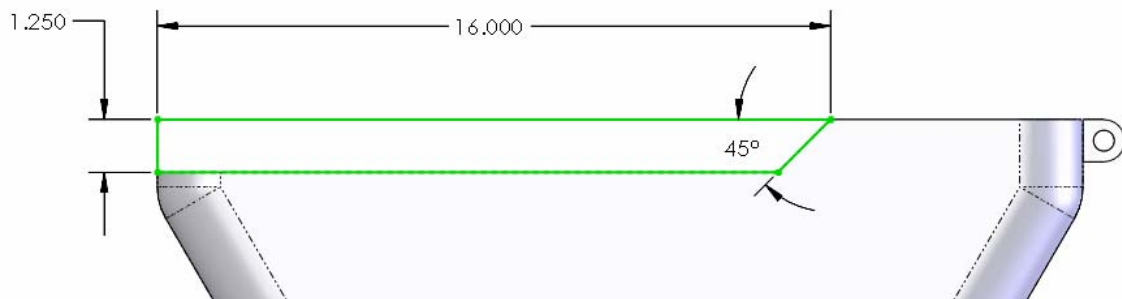
Make full round fillet, and mirror boss and fillet about Right Plane.



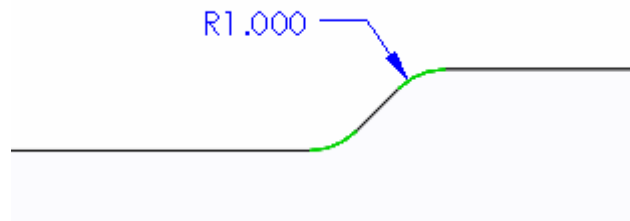
Make 0.5" cut through both hinge bosses.



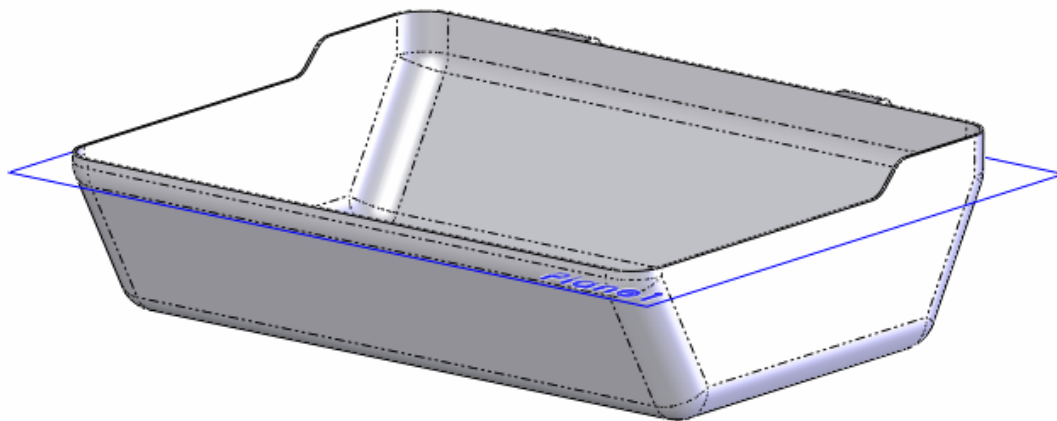
Make through all cut on the side.



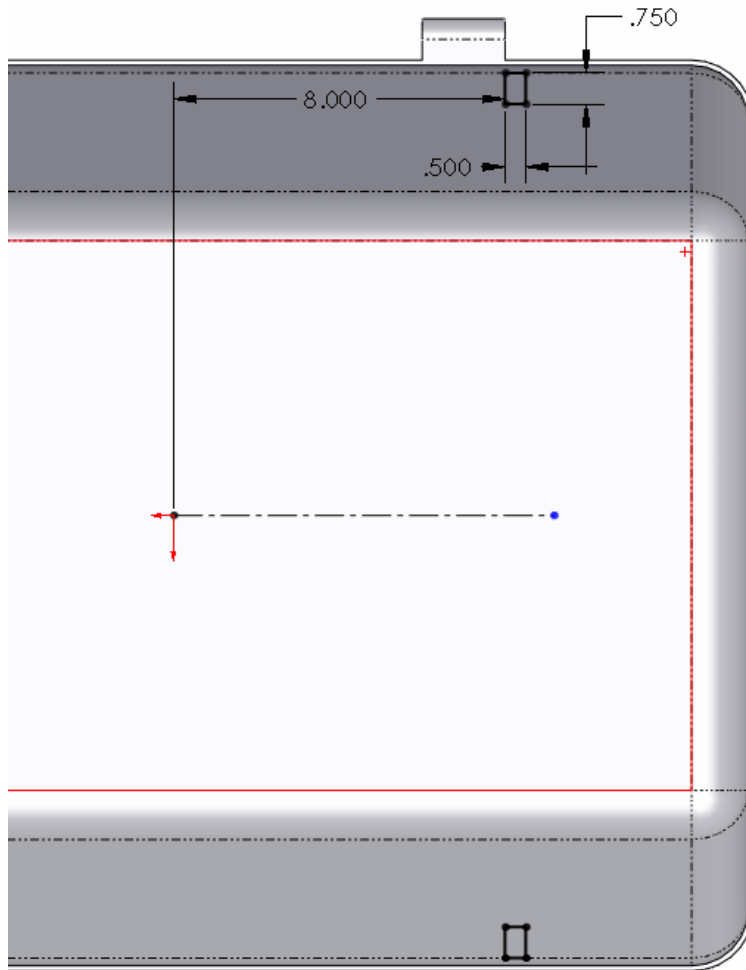
Fillet edges 1" to round corners.



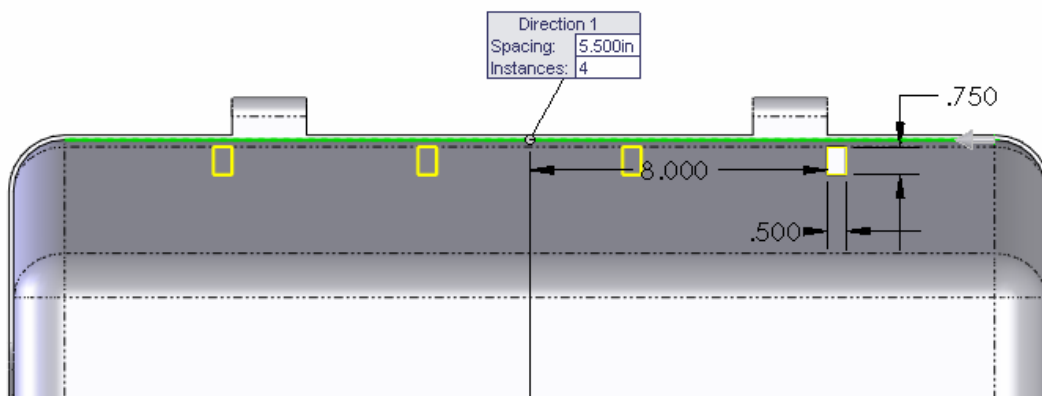
Add plane 6" above "Top Plane".



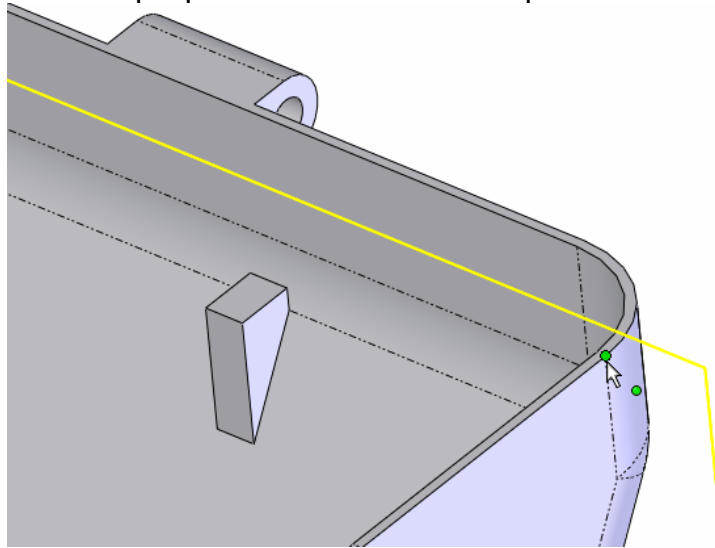
Make grill supports on this plane. Extrude Up to next into part. Make symmetrical.



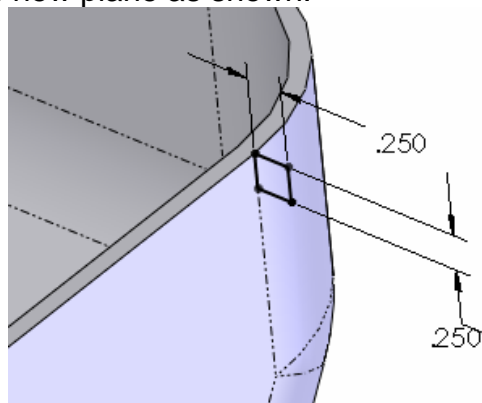
Pattern extrusions 5.5", total 4 instances.



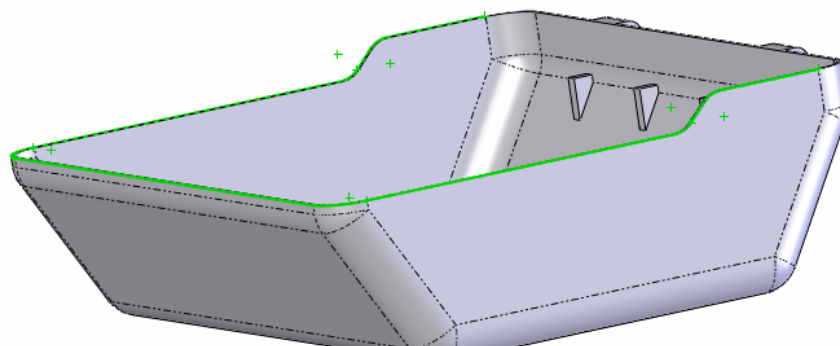
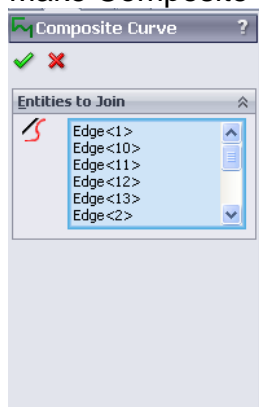
Make plane for sweep's profile. Parallel to Front plane at vertex.



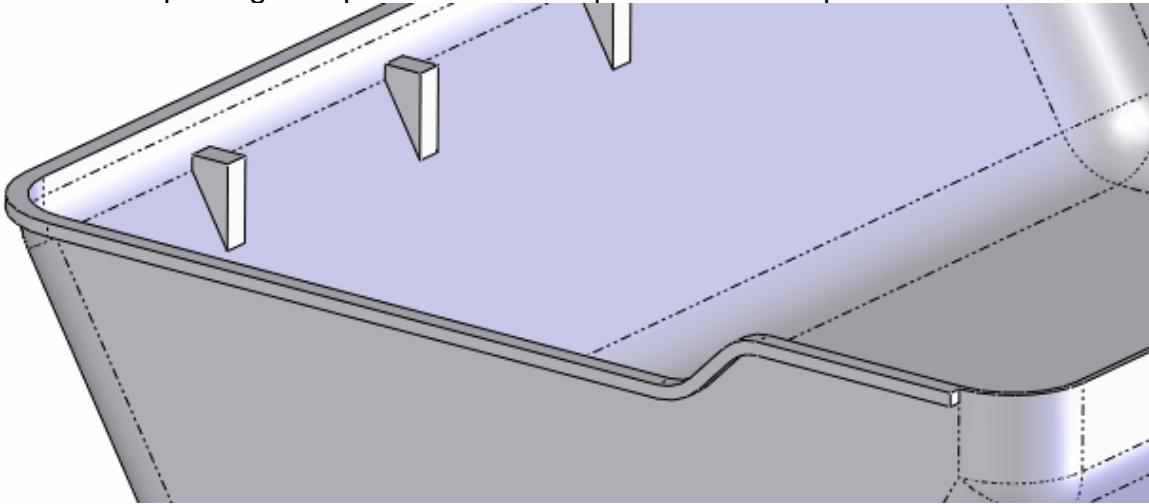
Make profile in this new plane as shown.



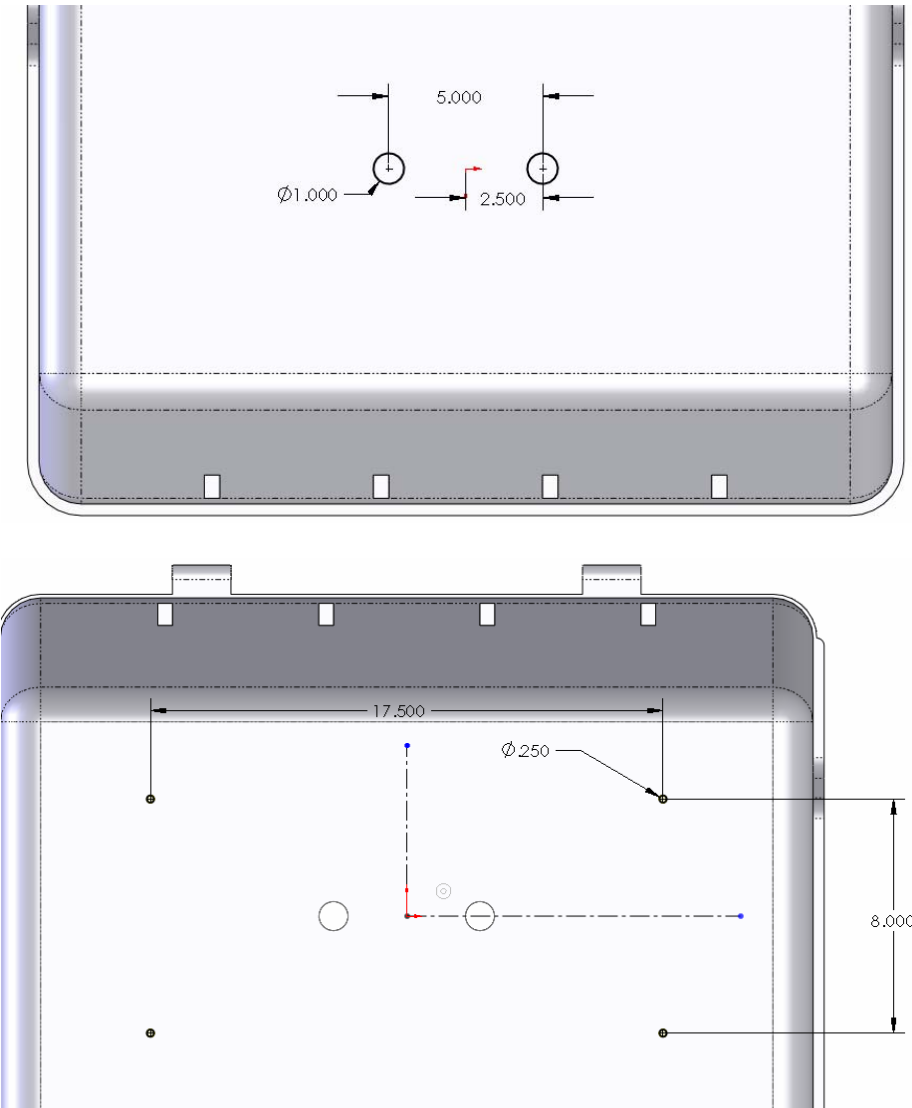
Make Composite curve using the edges indicated for path. (Outside edges)



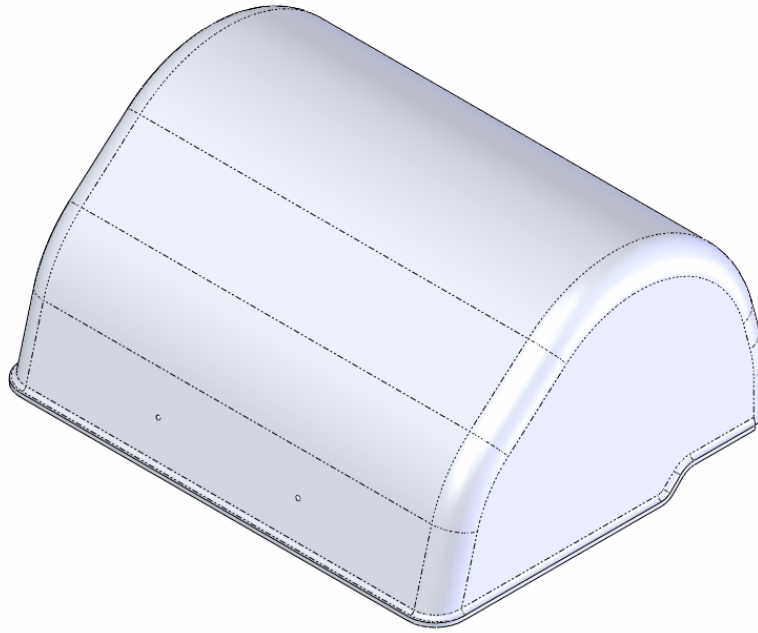
Make sweep using Composite Curve and profile to finish lip.



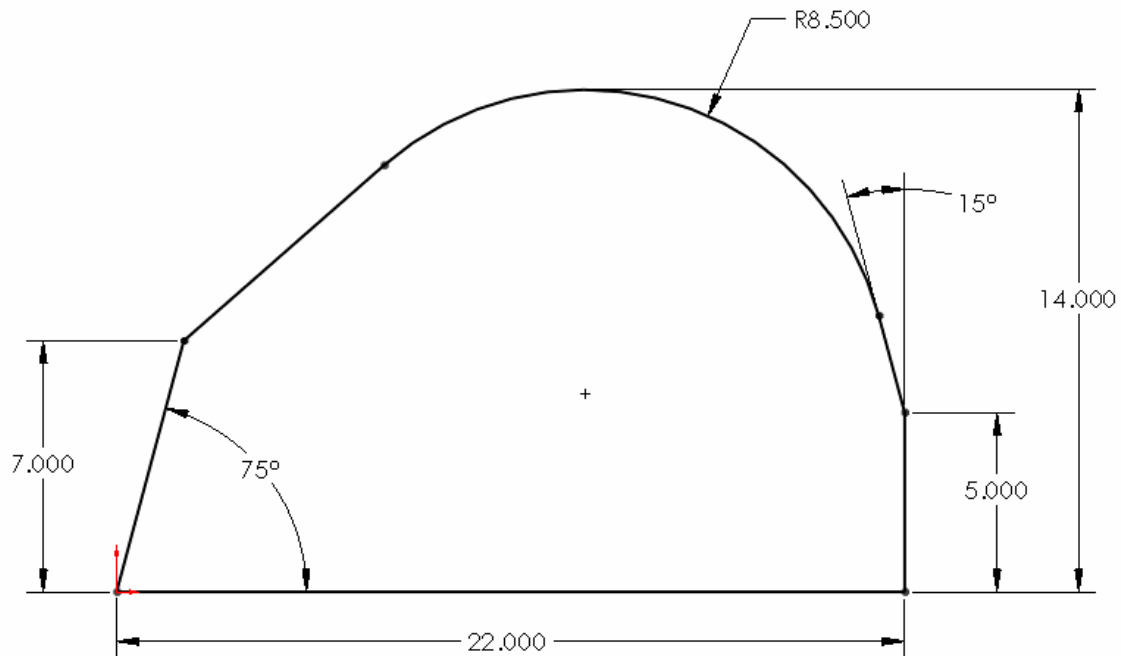
Add cuts on the bottom for screws and burners. Add fillets as needed.



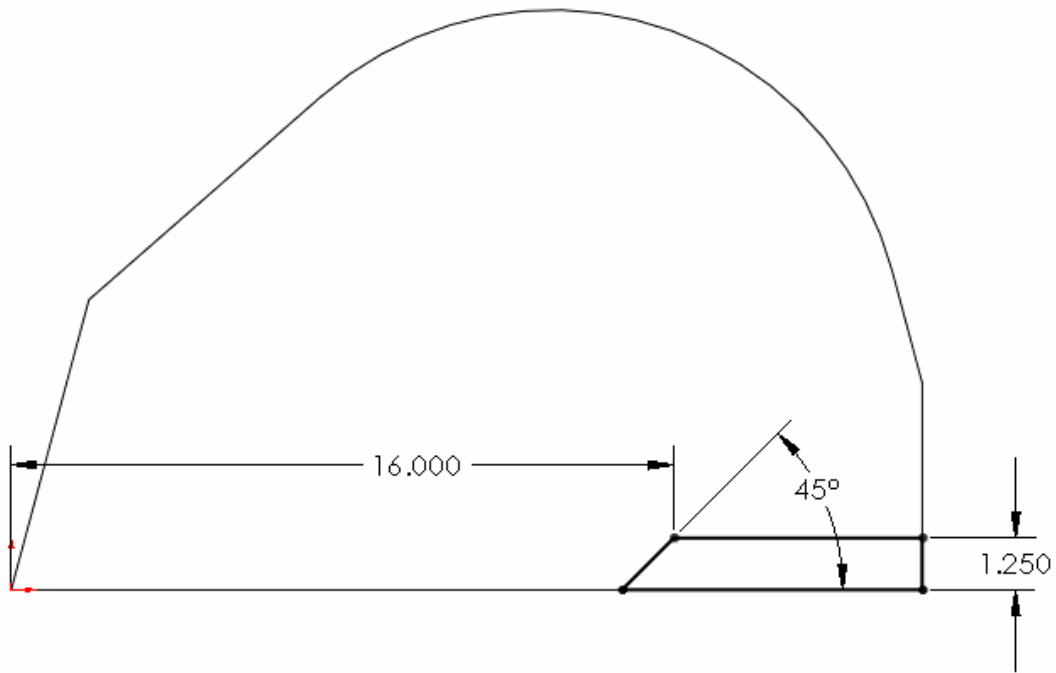
Grill Cover



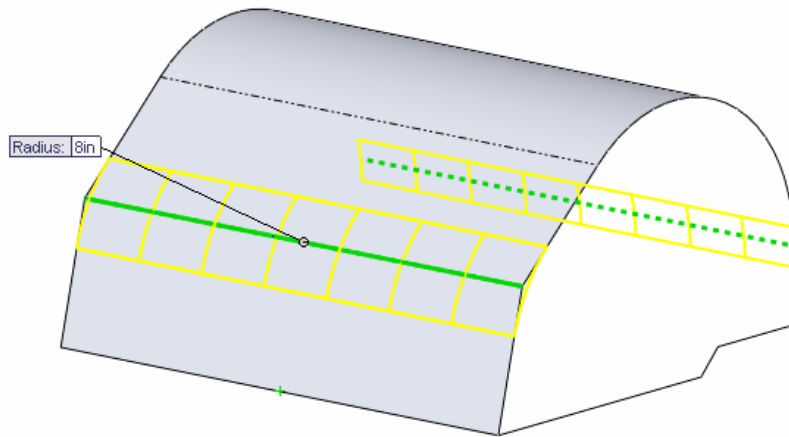
Sketch on Right Plane. Extrude 28" mid plane.



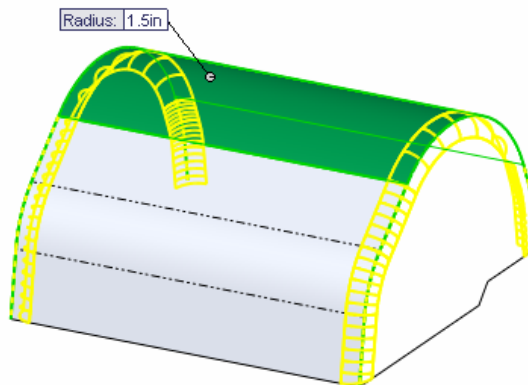
Sketch on Right Plane, cut through all both directions.



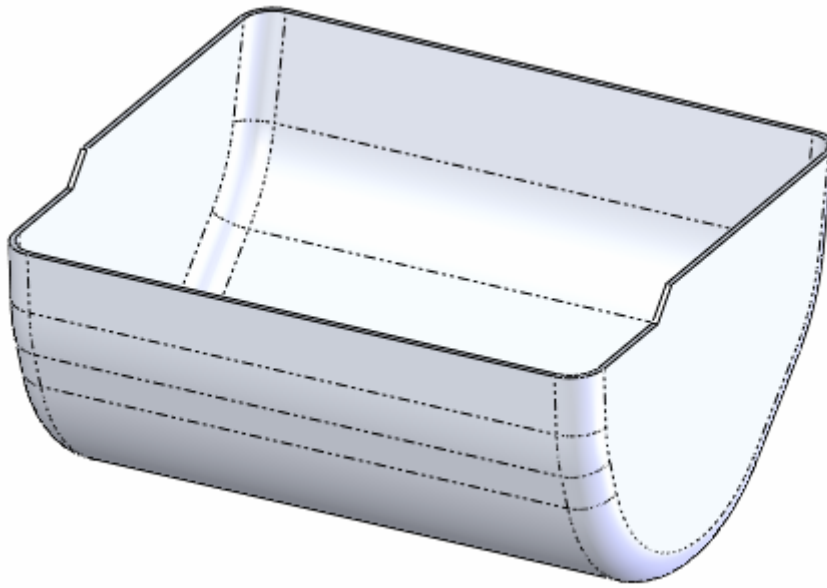
Add 8" fillets



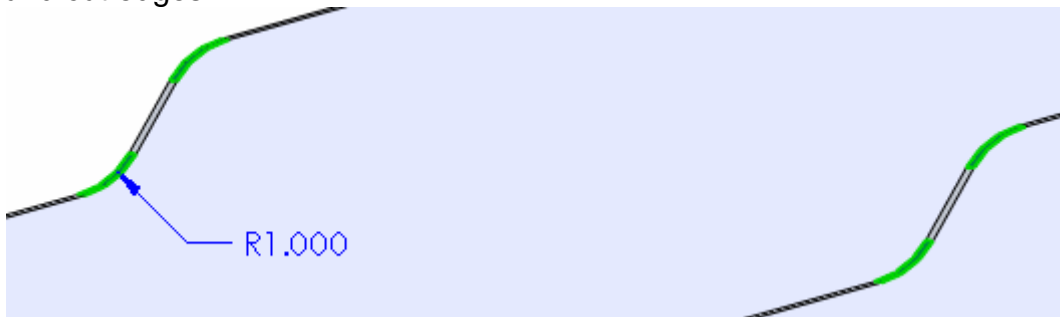
Add 1.5" fillet to the sides (select top face to round both sides)



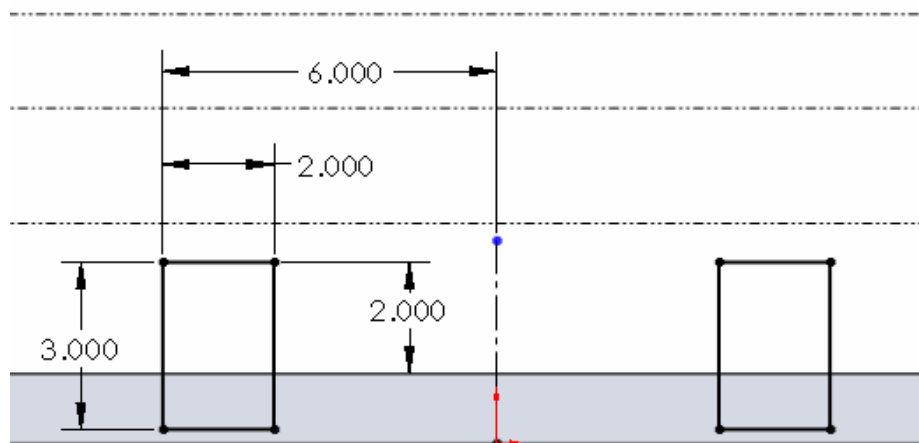
Shell 0.25" removing 3 bottom faces.



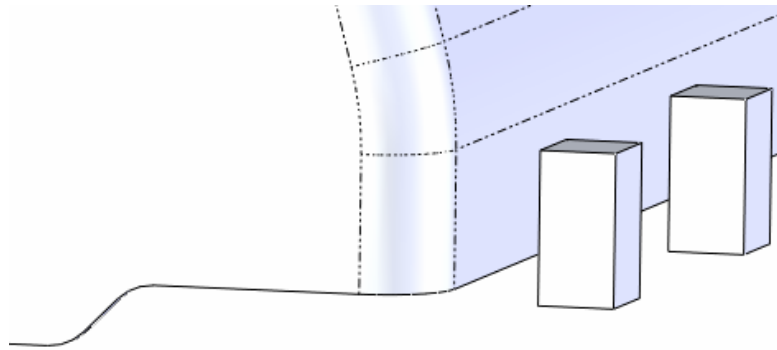
Round cut edges 1"



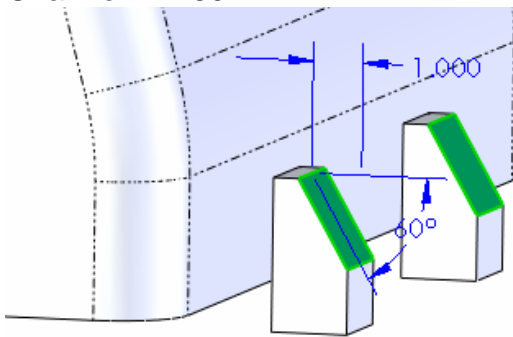
On the back face add sketch for hinges.



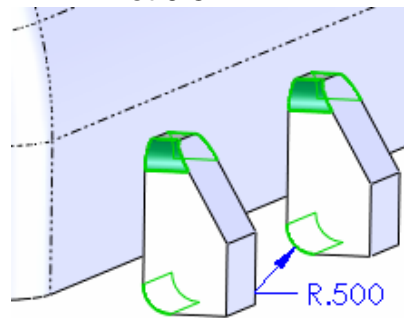
Extrude 1.5"



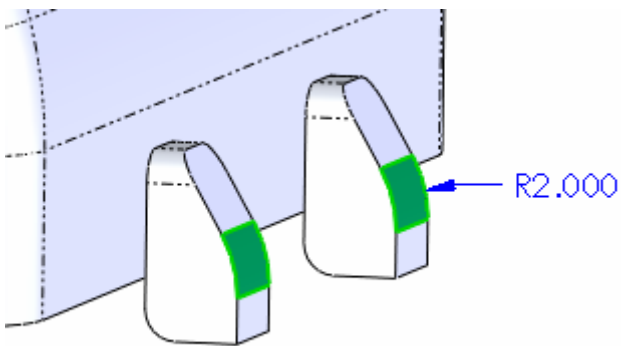
Chamfer 1"x 60°



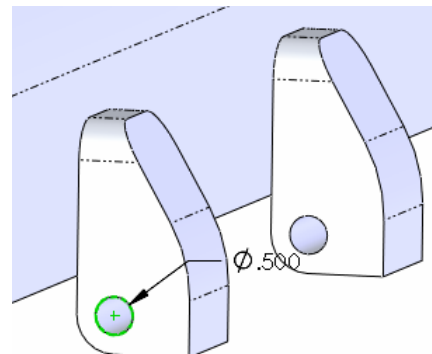
Fillet 0.5"



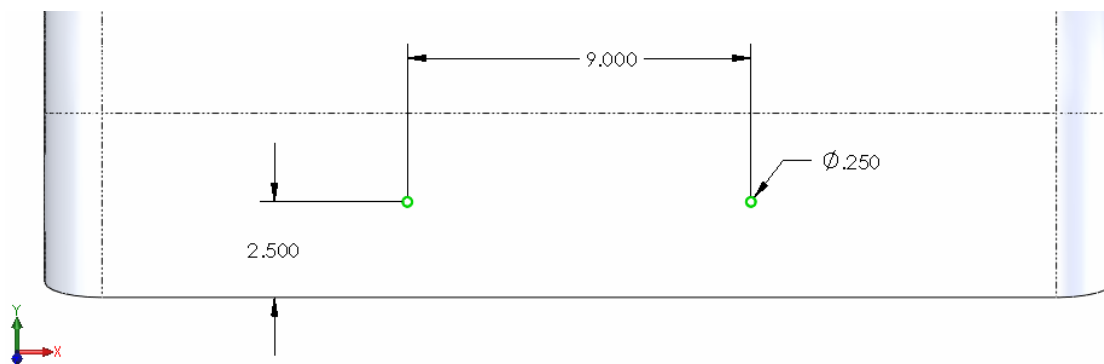
Fillet 2"



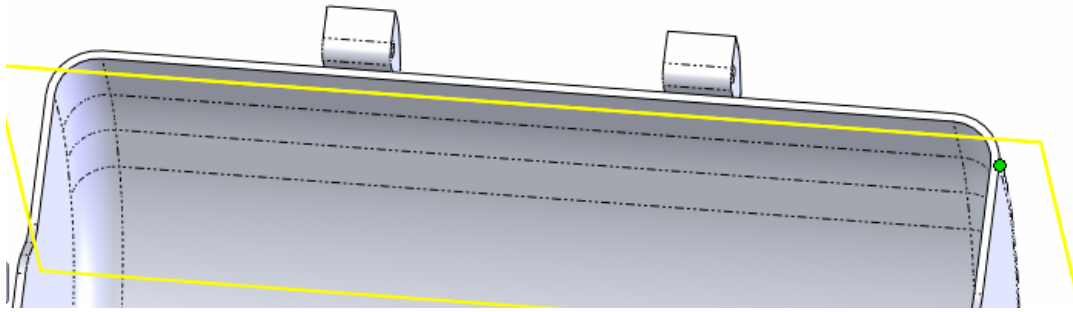
Make hole



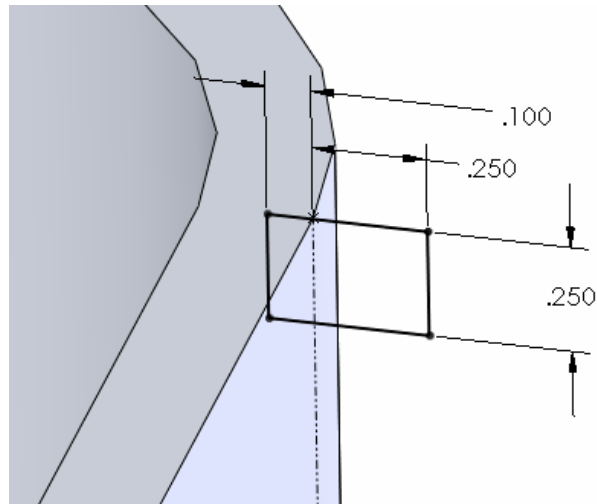
Make holes for handle in front face



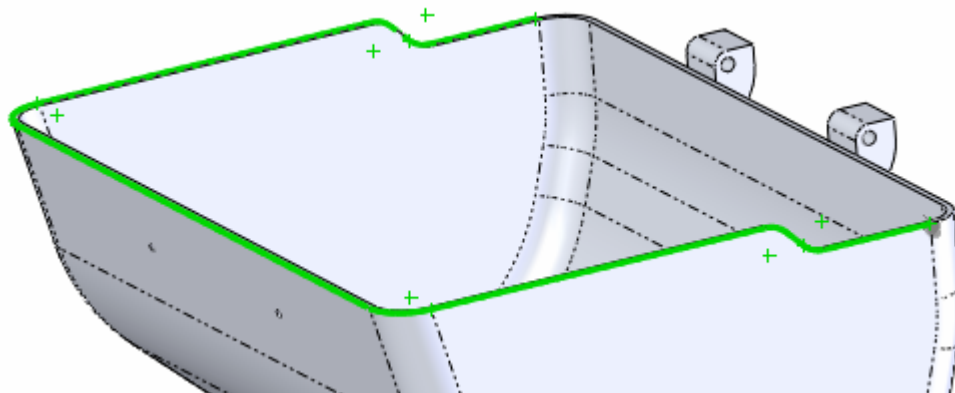
Make plane parallel to Front Plane at vertex.



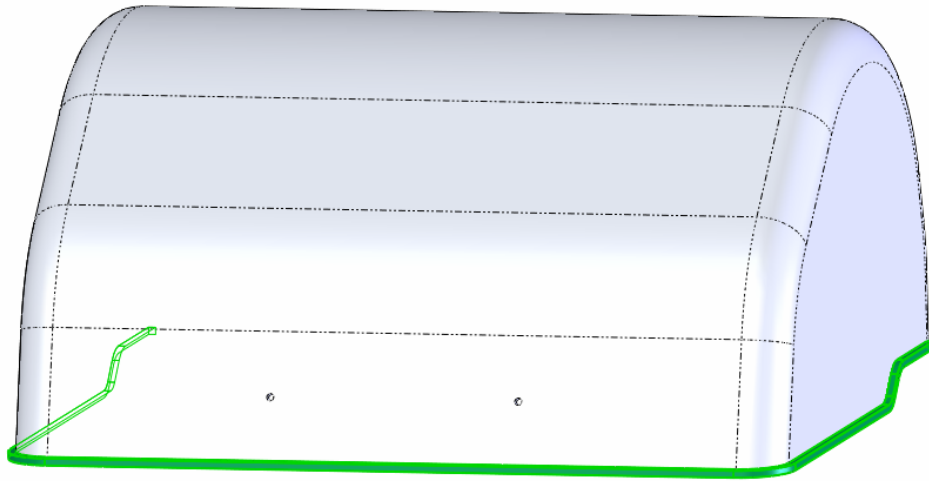
Draw the profile for the sweep. This will be the lip as in the grill bottom. Notice the profile is *inside* the part. This is because in the front of the cover the sweep would not merge with the main body. When doing sweeps like this its safer to 'bite' into the part with the profile to make sure it will merge with the rest of the part. Use a point to locate the sketch.



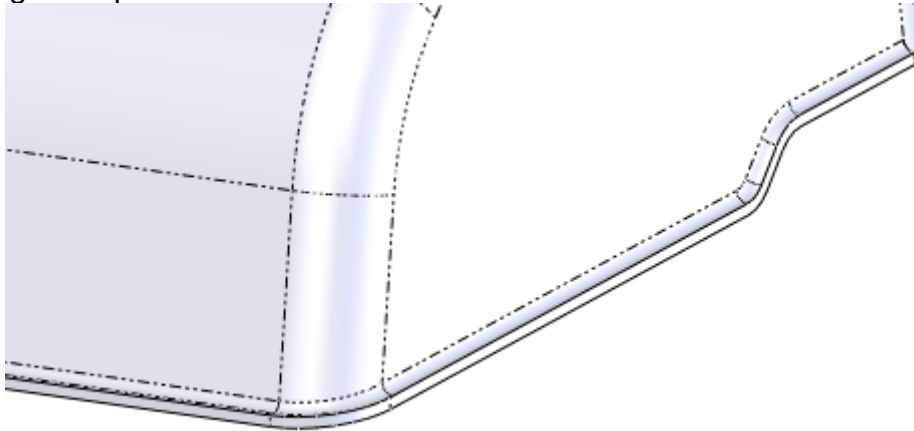
Make a composite curve using the edges as in the grill bottom.



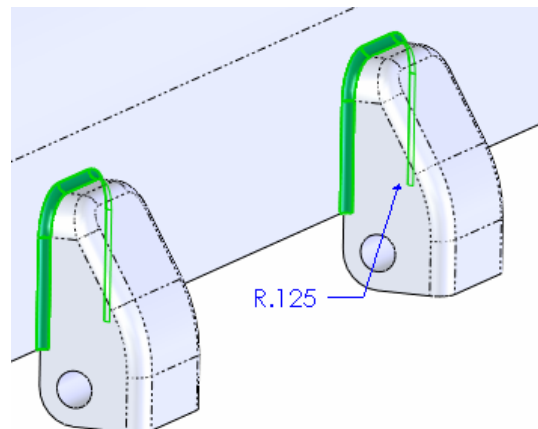
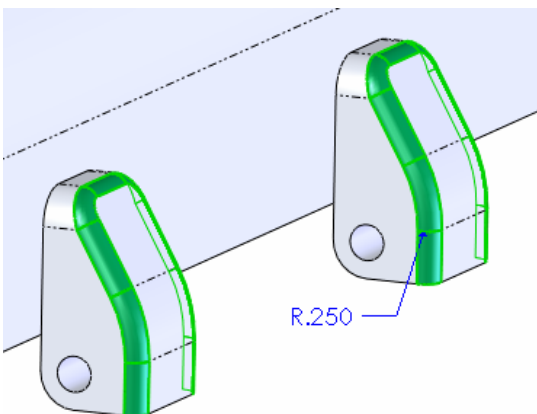
Make a sweep using the profile and composite curve



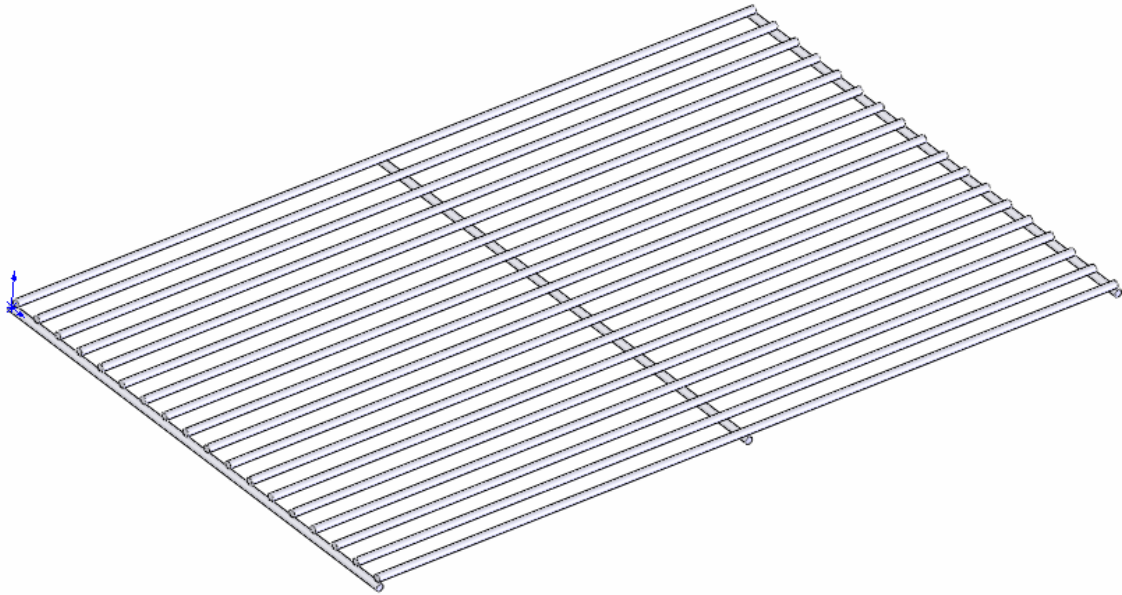
Fillet edge of Lip 0.25"



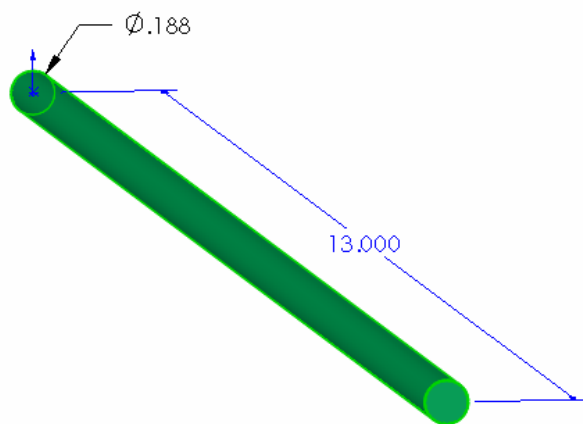
Add fillets



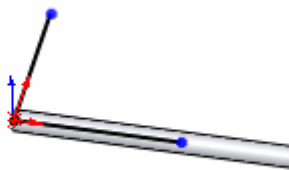
Grill



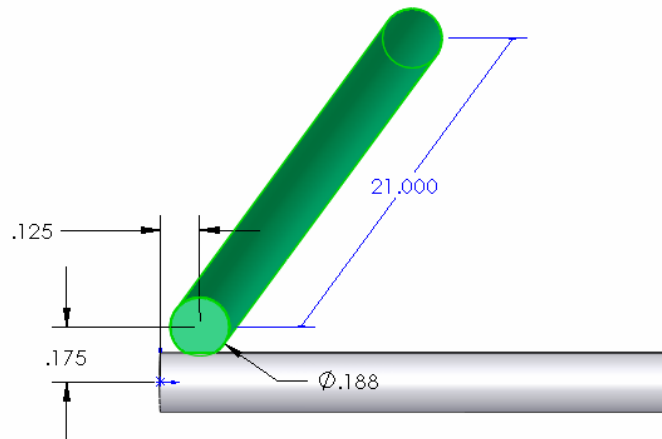
Make sketch on Right Plane and extrude



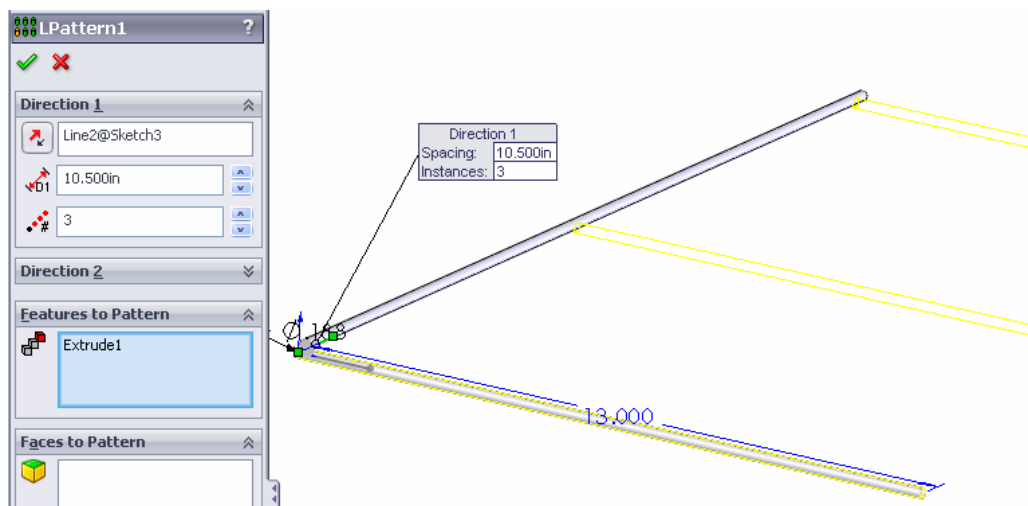
Add a sketch to use for direction on linear patterns. Exit sketch.



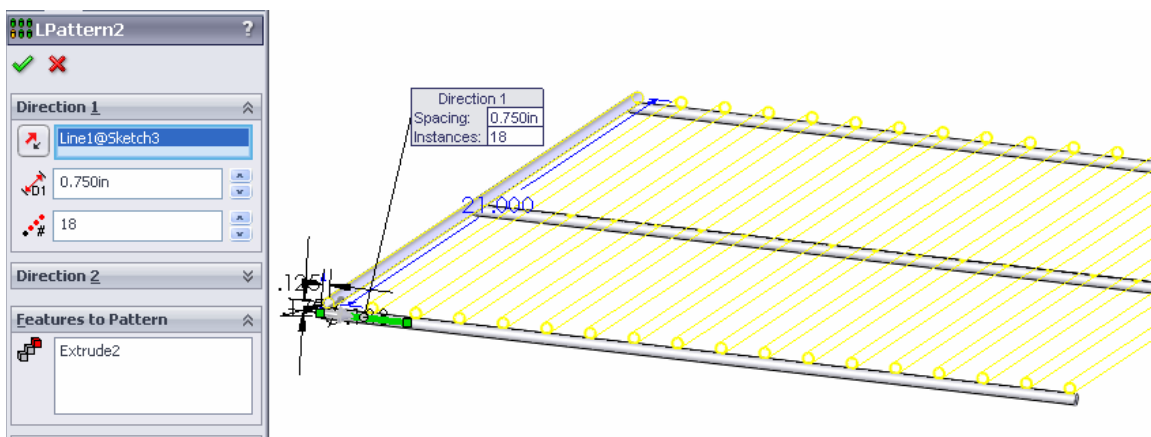
Make sketch on Front Plane. Extrude 21"



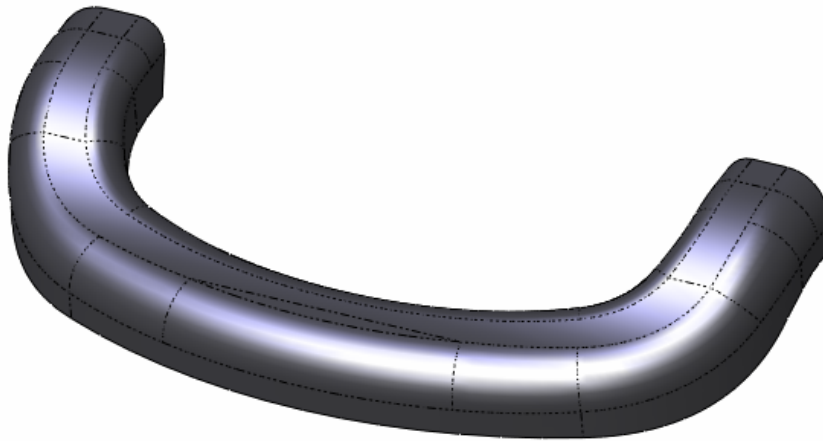
Make linear pattern of first extrusion. Use sketch for direction.



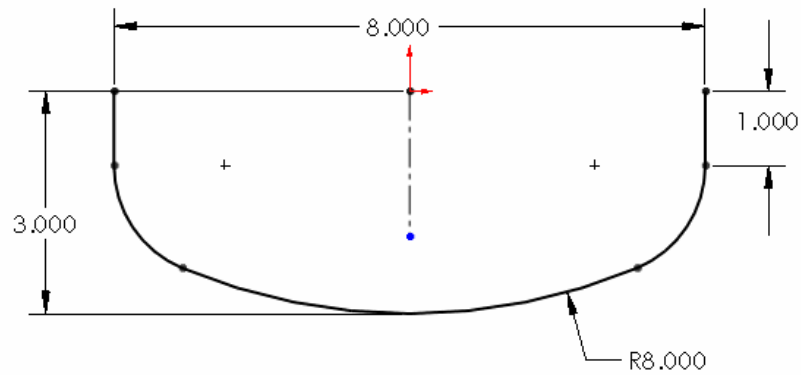
Make second linear pattern using second extrusion. Use sketch for direction.



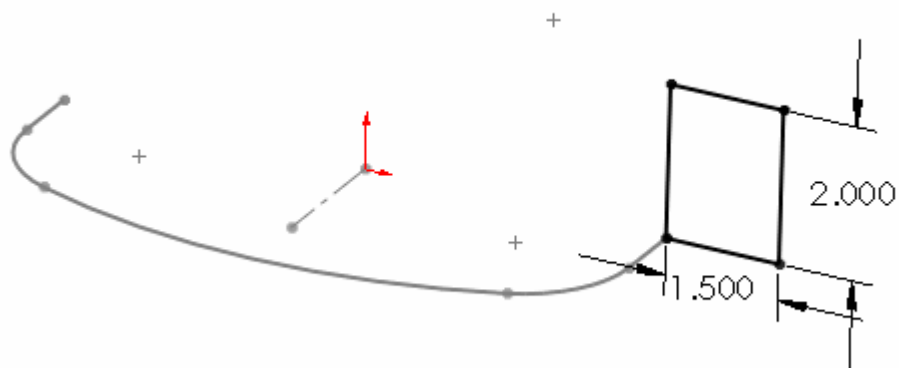
Handle



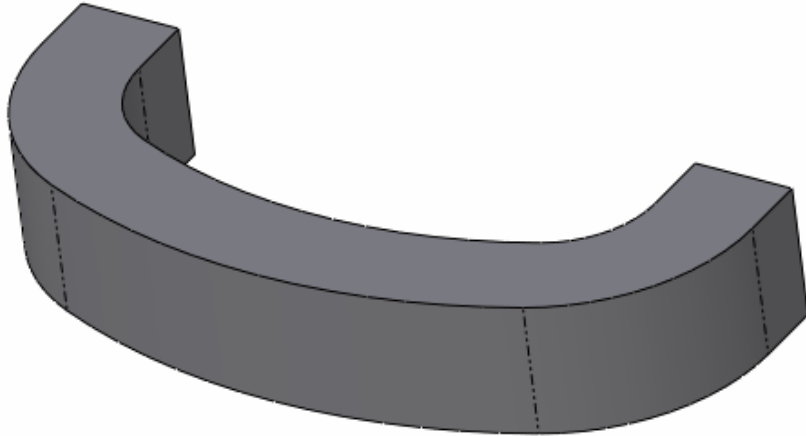
Make sketch on Top Plane for path.



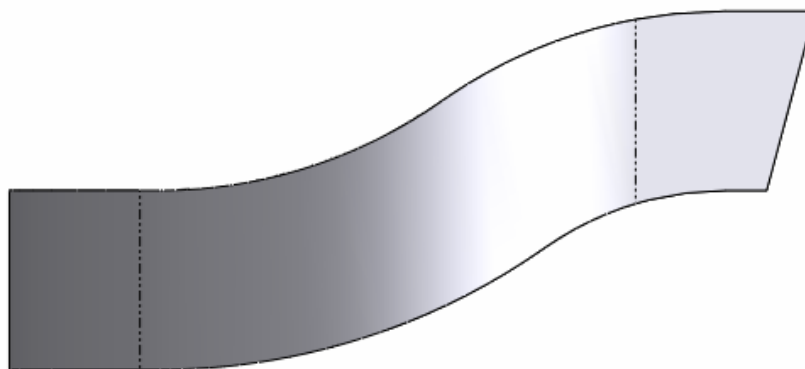
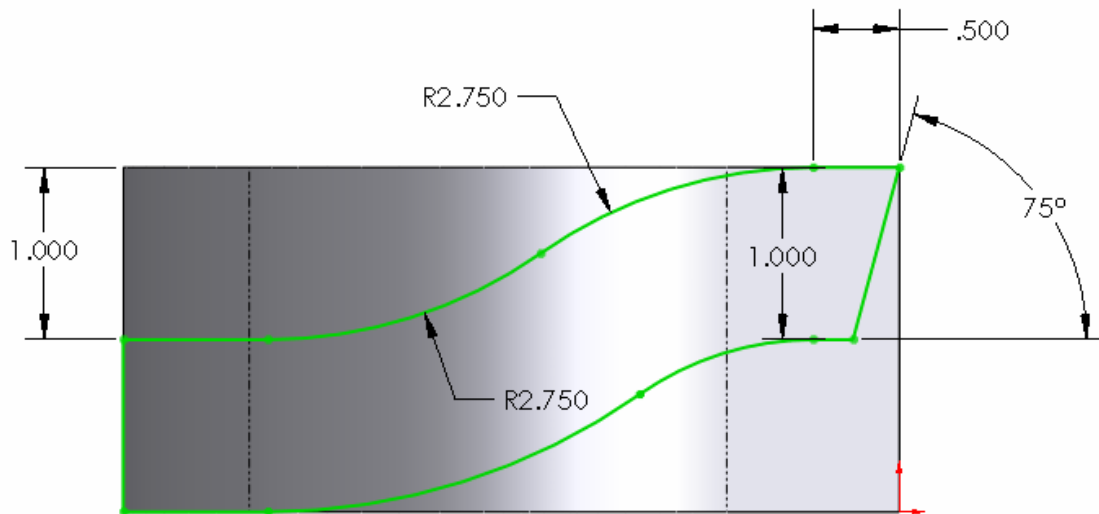
Make sketch on Front Plane for profile.



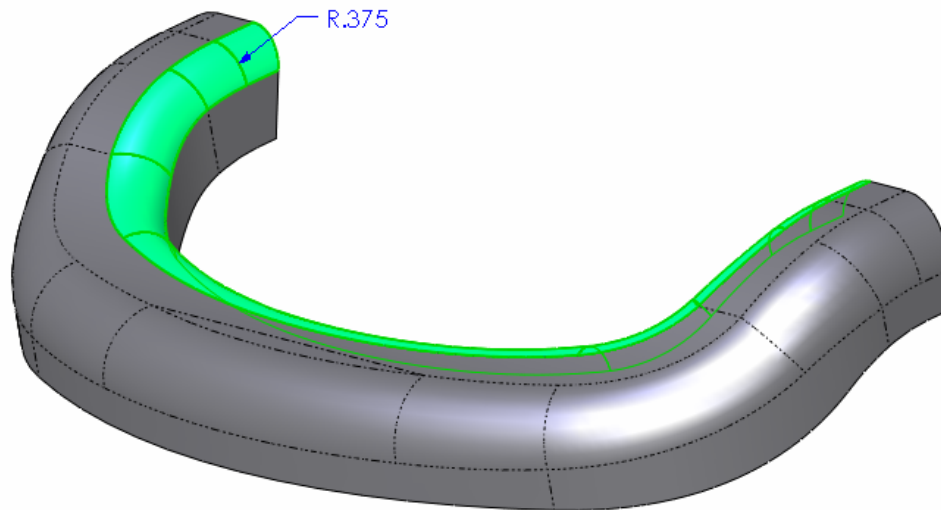
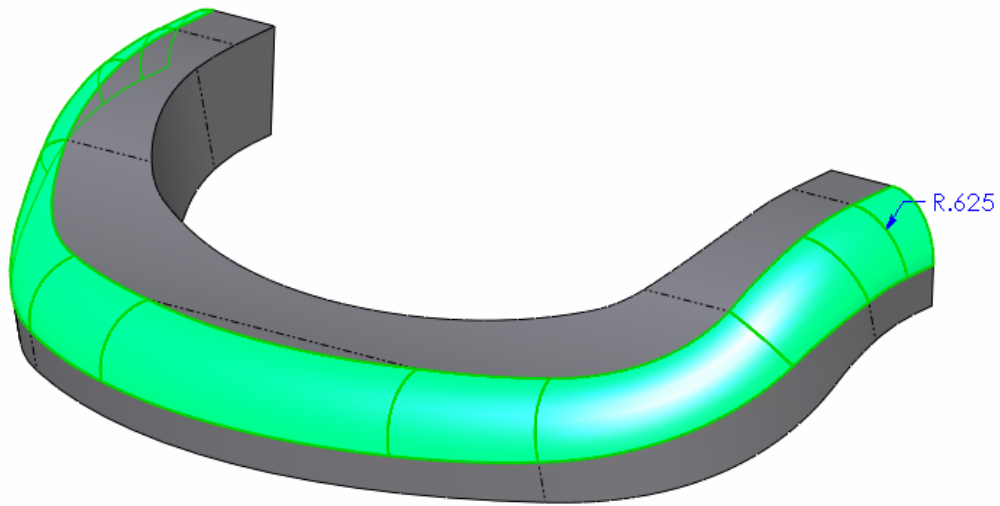
Make sweep.



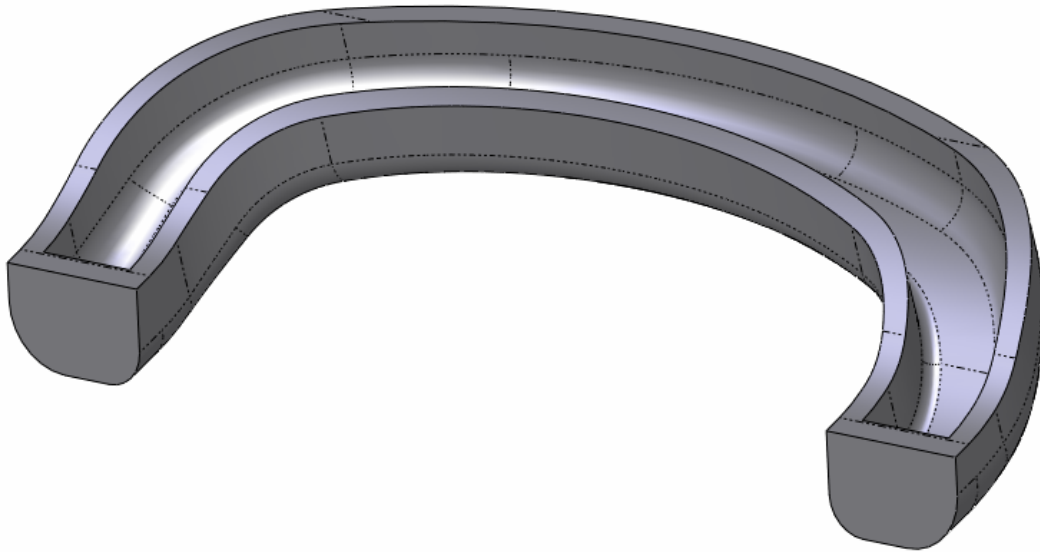
Make a cut using the “Flip side to cut” option. Tip: Draw the top line and offset it, trim and add the lines on the sides to close it.



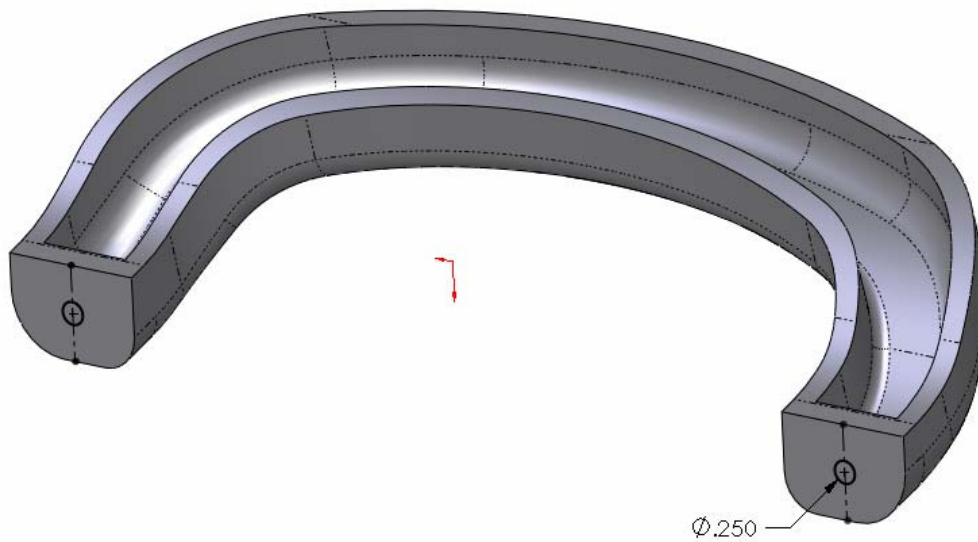
Add fillets



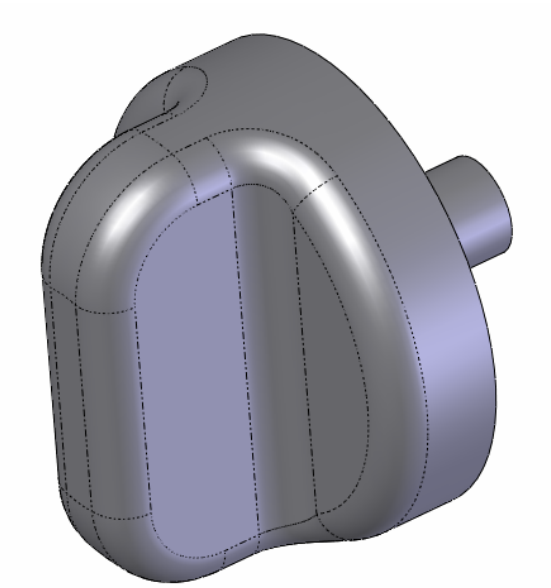
Shell part 0.25"



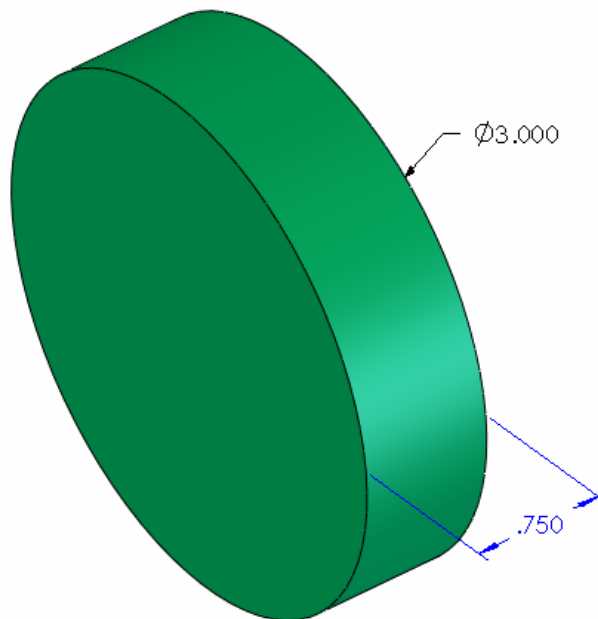
Add holes in center of faces for screws. Cut "Up to Next"



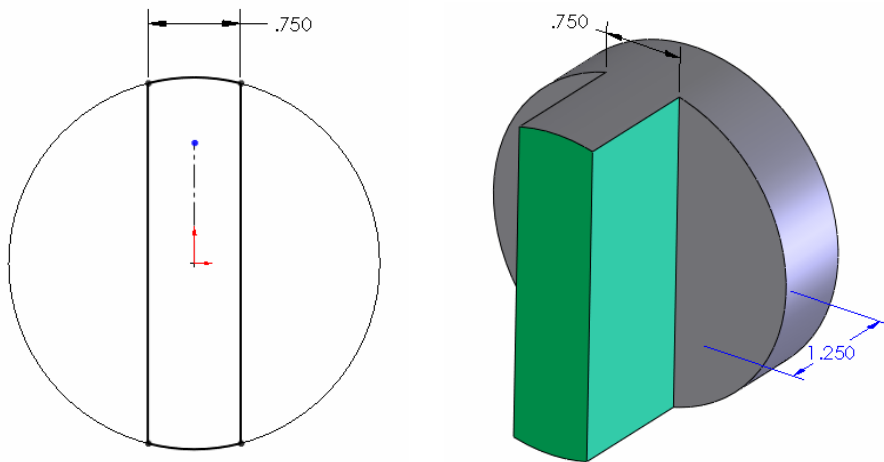
Knob



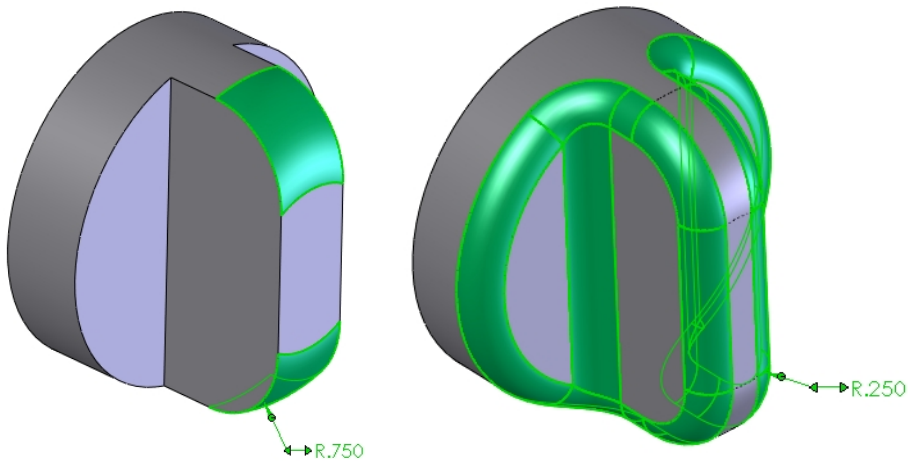
Make sketch on Front Plane and extrude



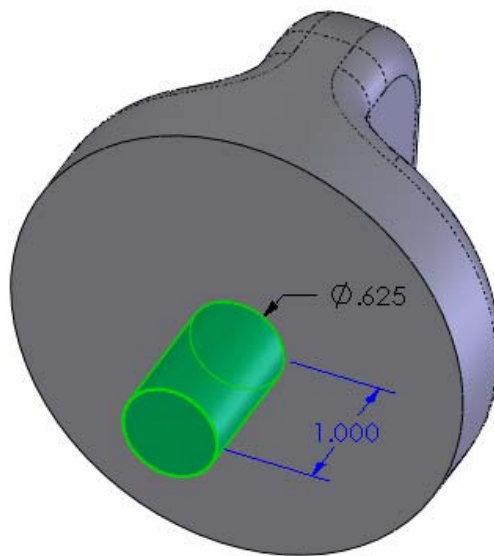
Make sketch on front face and extrude 0.75"



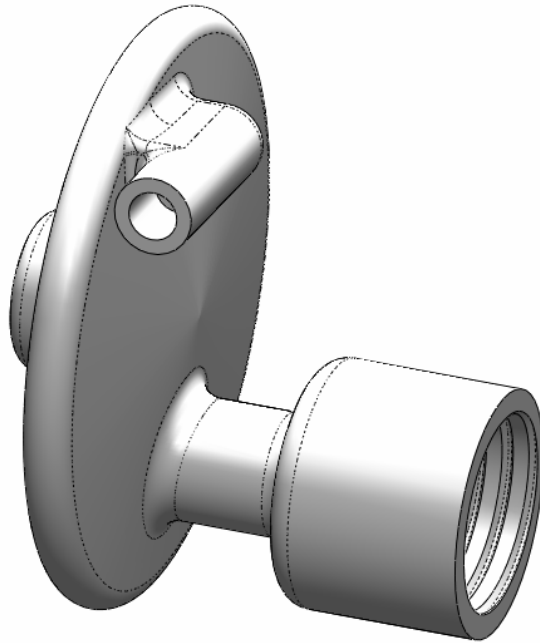
Add Fillets



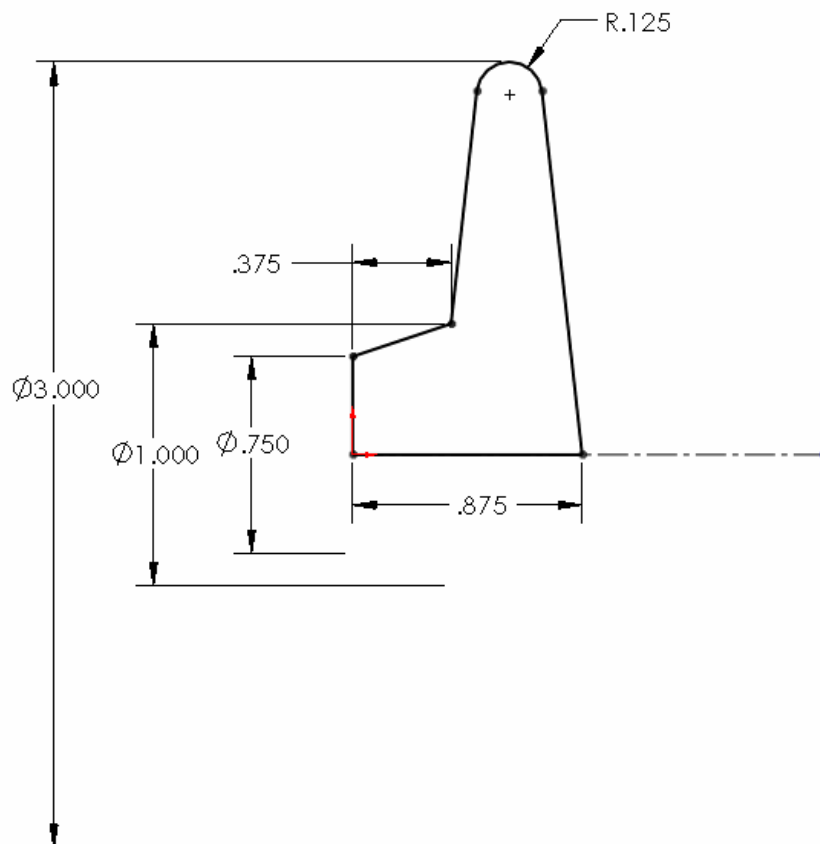
Add extrusion on the back.



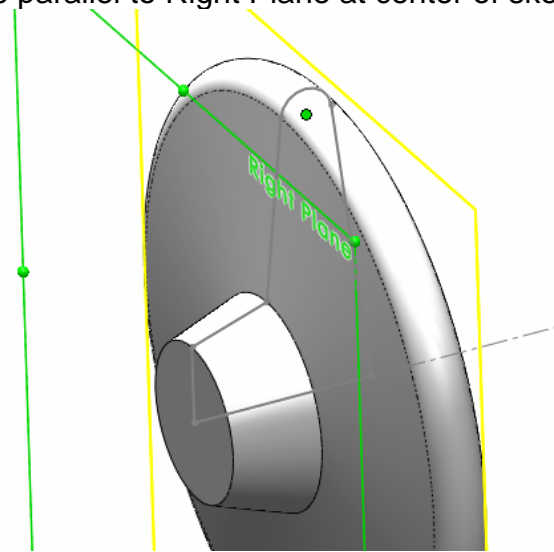
Gas Regulator



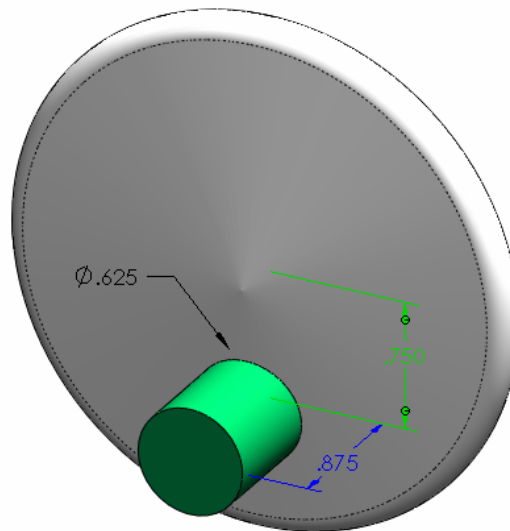
Sketch on Front Plane. Make revolved boss.



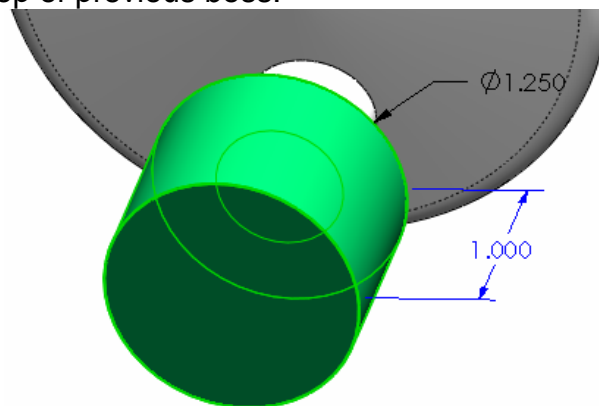
Make auxiliary plane parallel to Right Plane at center of sketch top arc.



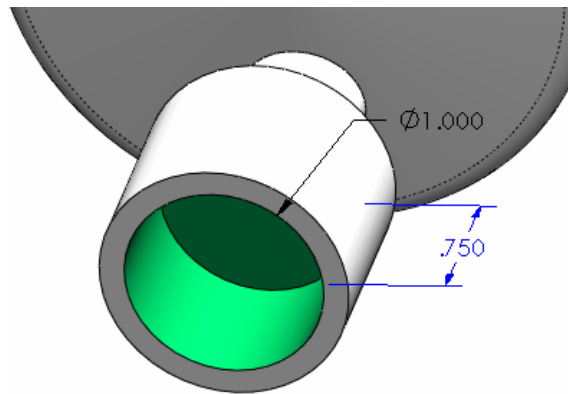
Make sketch on auxiliary plane, extrude 0.875"



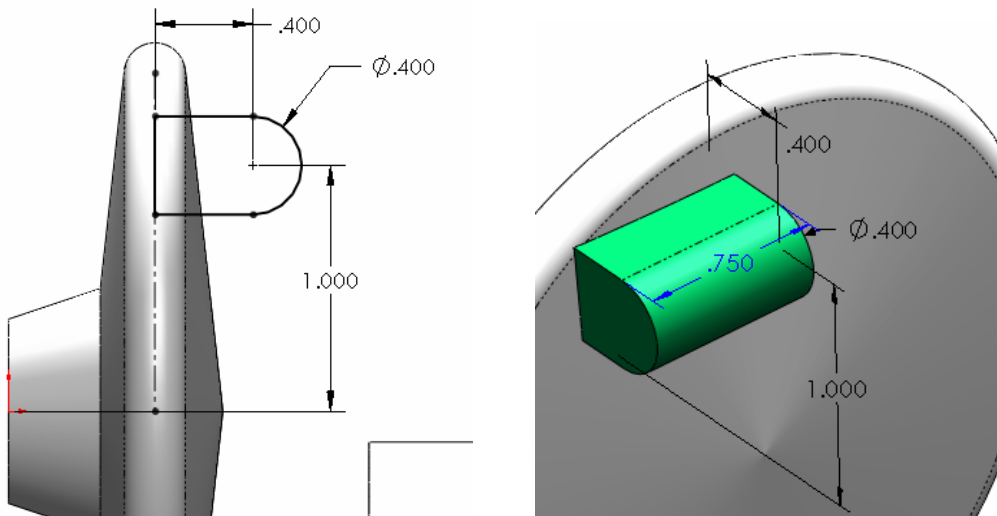
Add extrusion on top of previous boss.



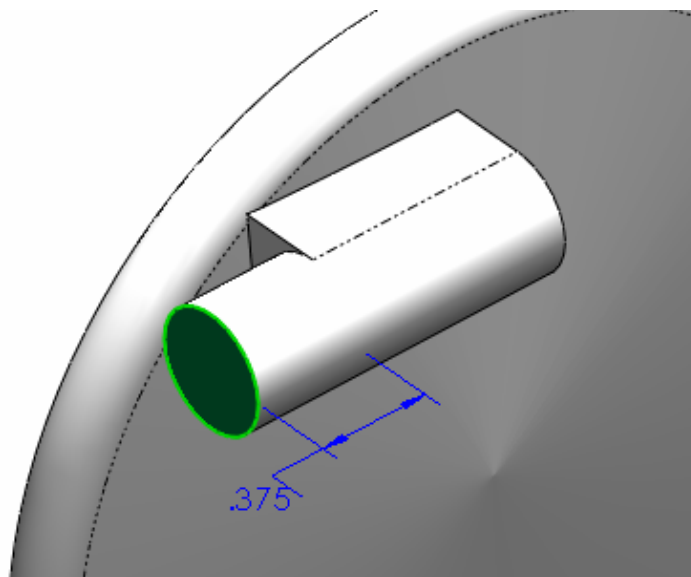
Make cut inside



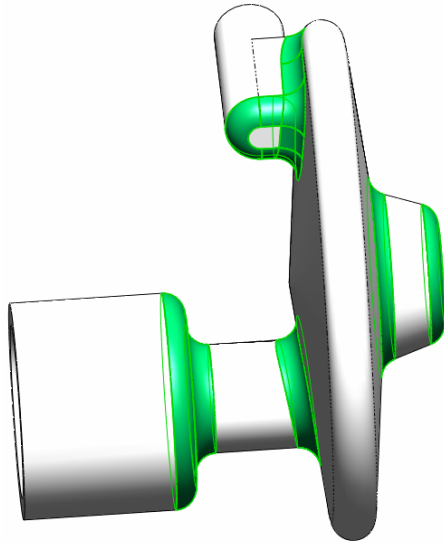
Make extrusion in Front Plane. Extrude 0.75"



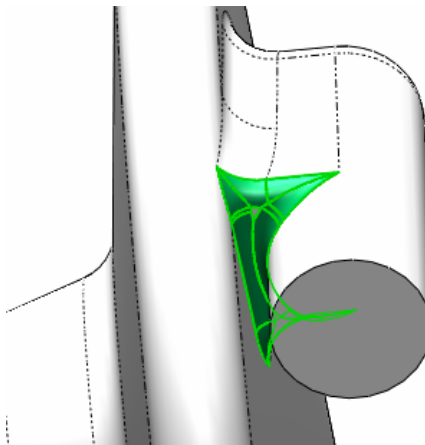
Make 0.375" long boss on top of previous boss.



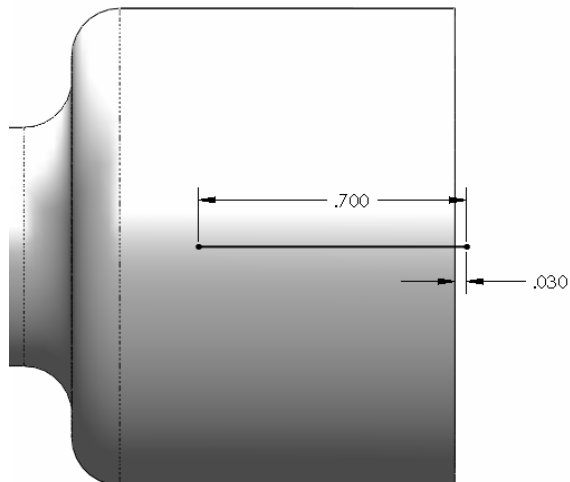
Add 0.125" fillet



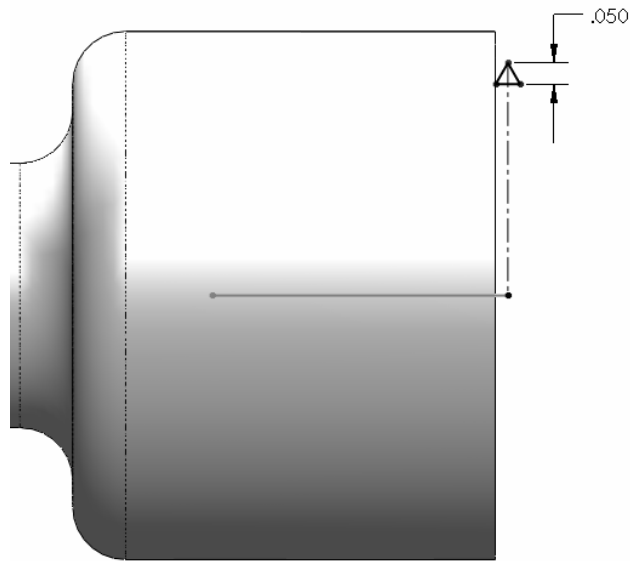
Add 0.50" fillet



Make thread using Cut-Sweep.
Make path sketch on Front Plane. Exit sketch.



Make Profile sketch in Front Plane. Start outside to make a smooth thread start.



Make Cut-Sweep using Twist Along Path option with 5 turns. Reverse direction if needed.

