Sheet Metal Design Guide

♀323

This guide will walk you through the process of designing sheet metal parts using Autodesk Fusion 360 in direct modeling. This guide is intended to only cover the topics regarding sheet metal design and will not cover basic concepts such as extruding, sketching, or hole-making. Please reference other guides or the Fusion 360 tutorials to understand those concepts. Sheet metal tools exist in parametric modeling, but the method described here in direct modeling has shown to be faster overall for designing sheet metal assemblies.

Start with your high level design. The following high level design considerations are important for adequate sheet metal design.

- Flanges provide structure. Do not make long unsupported spans of sheet metal (~1 inch or longer) without introducing flanges to support them.

- Rivet hole locations must have clearance for the rivet gun to fit. This is made easy in the example design by placing all rivets either on the top or on the bottom of the assembly—note that there are no flanges facing sideways that will be riveted together.

- 1/2" (for FRC) and 3/8" (for FTC) hole spacing for rivet holes makes it easy to add new plates and features later if needed. Note this guide doesn't cover the creation of rivet holes beyond a basic description.

- 1/8" rivets use 0.128" holes, 3/16" rivets use 0.189" holes.

- Do not cut out any material within around 1/2" of the outside surface of the bend. If this area needs to be cut out for clearance reasons, cut out the entire bend. If this isn't followed, bends will be inconsistent as the tiny flange is difficult to bend properly.

- Be sure flanges are at least 3/4" or longer for ease of bending. 1/2" may be possible with added difficulty in fringe cases.

- Use polycarbonate or 5052 aluminum for bent parts. 6061 AL is for straight parts only.

To draw a basic bend, use the following process.

- Draw the profile as two lines (for an L profile) or three lines (for a C profile) with your desired flange lengths. Optionally, you can use the sketch fillet tool to add the outer bend radius at this step. In this guide the bend radii will be added as fillets later.
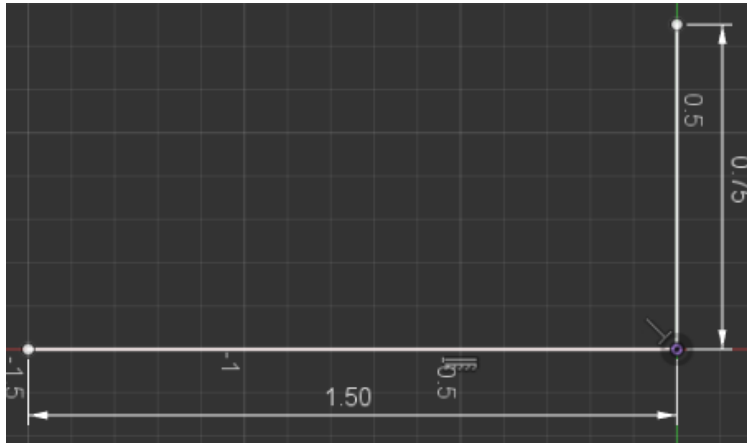
- Offset the lines by the material thickness, usually 0.0625" or 0.125", then close the ends with two additional lines (shown in blue here).
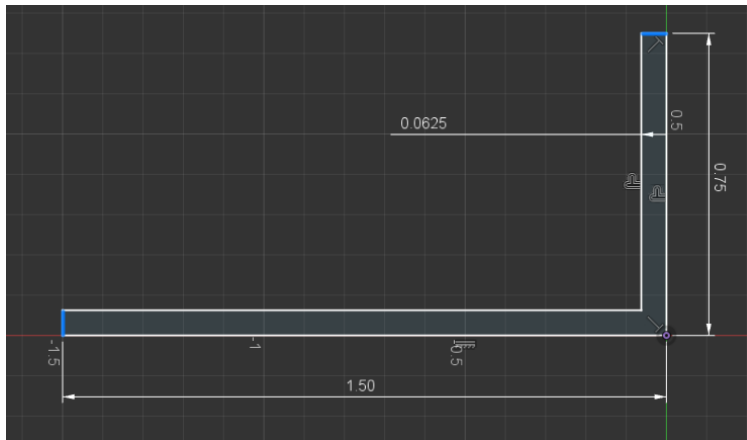
- Extrude to the desired length, then add fillets for the outer and inner bend radii. If the radii were added in the sketch phase, they will not be needed here. 0.375" corner radii have been added to the part to remove sharp corners.
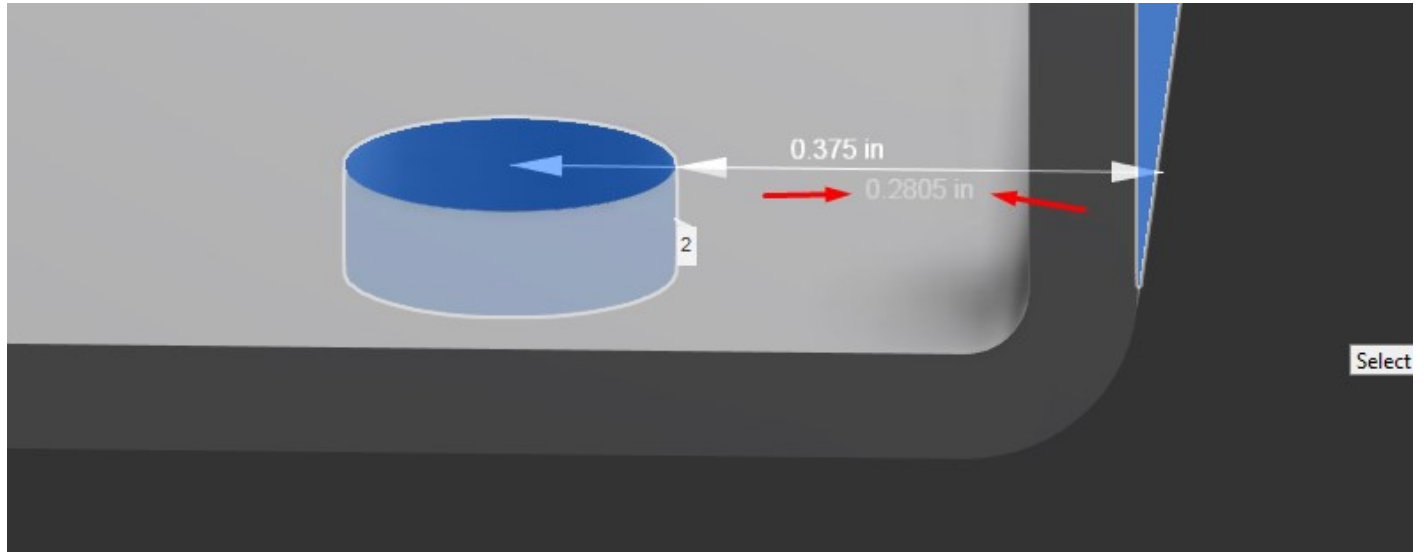 1/16": 0.100" outer radius, 0.0375" inner radius
  1/8": TBD
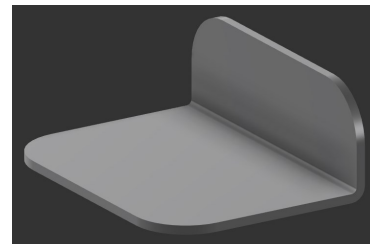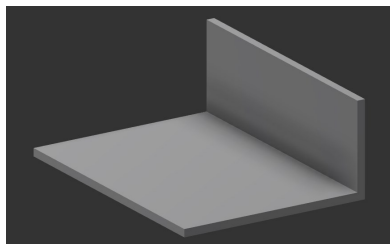
Sheet Metal Design Guide

♀*323*

When adding rivet holes or other holes, be sure the features are not closer than 1/4" to the outside edge of the material. In this case, 0.2805" is greater than 0.250" so the hole is acceptable.

If features need to be closer, like a pocket for clearance remove the bend in that area as shown in the final part render below. This ensures there won't be an awkward bend in that area. Notice there are no fillets where the bend was removed. The tool will add a 3 mm fillet when machining. Adding a fillet in the design will make the model difficult to convert to a flat plate for machining. If a radius larger than 3 mm is needed, lift the face up and add the fillet making sure the entire fillet is on one flange and doesn't extend into the bend.