EML2322L – MAE Design and Manufacturing Laboratory

Sheetmetal Design

This document explains how to design sheetmetal parts for the course design project. The top priority for part design is function; above all else, the part must function properly without failing. Beyond that, ***good*** designers are concerned with designing parts that can be made quickly and cost effectively; this means keeping parts simple, material use low and manufacturing processes to a minimum. The following points discuss these topics.

Objectives

1. **Function.** Most low stress (and many high stress) components requiring moderate stiffness can be created from sheetmetal. Sheetmetal is particularly effective for parts that function as containers, chutes and gates, but can effectively be used for mounting brackets as well.

2. **Attachment method.** Sheetmetal parts are typically welded (permanent), riveted (semi-permanent), or connected via fasteners (removable) to other parts. Since sheetmetal parts are typically too thin to be threaded (i.e. too thin to achieve the required 5 threads of engagement), attachment holes should be designed as thru holes.

3. **Mechanical properties.** Sheetmetal parts are typically lighter than their billet machined counterparts. In general, simple sheetmetal parts require looser tolerances and are less stiff. Sheetmetal parts requiring welding should be created from steel, because thin aluminum is much more difficult to weld (because of its higher thermal conductivity).

4. **Manufacturing properties.** Sheetmetal parts are typically cheaper and faster to produce than their billet machined counterparts.

Sheetmetal Design Tips

1. **Understand the lab's sheetmetal capabilities.** As with any design, it is important to understand the manufacturing processes available while in the design phase. Consequently, it is helpful to watch the course sheetmetal demonstration video, review the available tooling presented in this document, and receive TA training on the sheetmetal fabrication equipment in lab.

2. **General shape.** Use simple shapes such as straight cuts, bends, and punched holes. Whenever possible, avoid internal cuts (cuts not starting on an edge), curved cuts, and close-fit holes.

3. **Keep it simple.** Complex sheetmetal parts are difficult or impossible to manufacture using tooling available in the lab. Typically, complex sheetmetal parts can be broken down into multiple simple parts that can then be attached together to create an equivalent complex part.

4. **Material specification.** Choose a material and gauge that is appropriate for the part: steel is cheaper, stiffer, and excellent for welding; aluminum is more expensive, less stiff, and not as weldable; and a thicker gauge results in greater cost, stiffness, and manufacturing time. 16 gauge sheetmetal is the thickest material that may be used in the lab's sheetmetal forming machines. When deciding on the material type and gauge, it is instructional to handle the

example sheetmetal pieces available in the lab. Manufacturing time and thus part cost are typically proportional to material strength; knowing this, ***always select the weakest material that satisfies the design constraints*** (i.e. weldability, stiffness, etc.) to ensure the cheapest material and lowest part cost.

5. **Surface finish & appearance.** Sheetmetal parts are low tolerance. Therefore, components interfacing with sheetmetal parts must not require a high degree of flatness at the interface. ***This means sheetmetal parts do not require shiny machined surfaces to function properly, so the exterior surfaces should purposely be noted as "not finished" on the part drawing.*** Also, none of the objectives listed above mention appearance, so the dull stock finish is appropriate for the sheetmetal parts used in this course and reduces manufacturing time.

6. **Clearance holes versus line fits for fasteners. If using ¼'' fasteners to attach motor mounts to a frame, never use line fits (i.e. 0.250''), but rather, clearance fits.** A fastener needs clearance to pass thru a part without threads. In this case, specify Ø0.257" or Ø0.266" holes since these are the industry standard sizes listed in the tap and clearance drill chart for clearance holes.

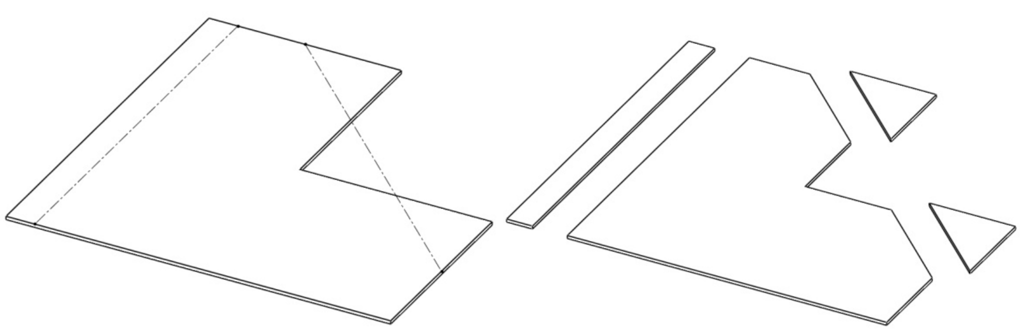
7. **Tolerances.** Sheetmetal parts produced in industry are made using high precision CNC lasers and bending equipment. However, the lab's capabilities are more modest and traditional, meaning the tolerances are liberal, typically on the order of ± 0.060" (i.e. 1/16"). Sheetmetal features are generally drawn using a tape measure, ruler and a marker, and cut by aligning the layout lines by eye, so design sheetmetal parts to have large feature tolerances.

8. **Order of operations.** The order of forming operations is important for sheetmetal parts. Holes and cuts must be created prior to bending. The bending sequence can typically be performed in only one or two ways. Be familiar with the available tooling and processes and ***do not design parts that cannot be fabricated using the available tooling*** in lab.

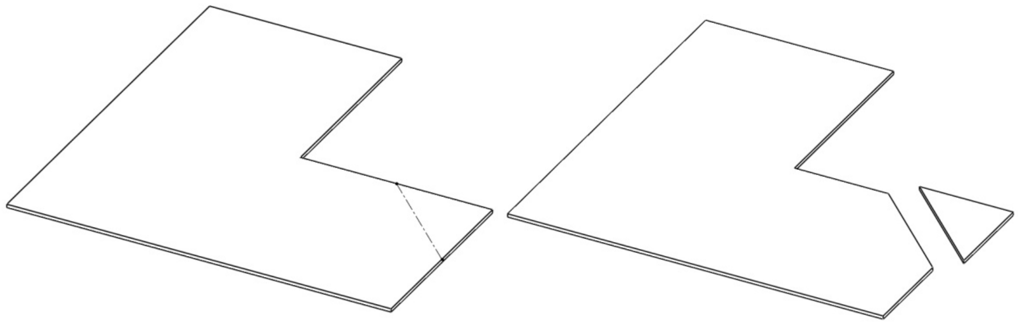
Available Tooling in EML2322L

CUTTING PROCESSES:

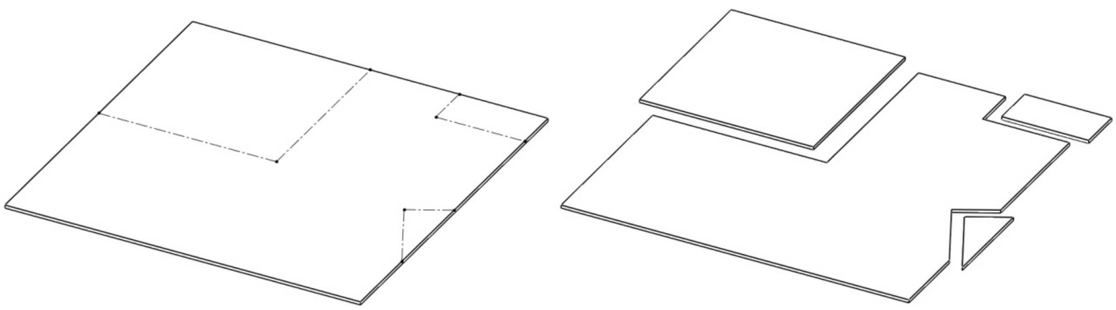
1. **Hydraulic shear.** Shears completely across the part.



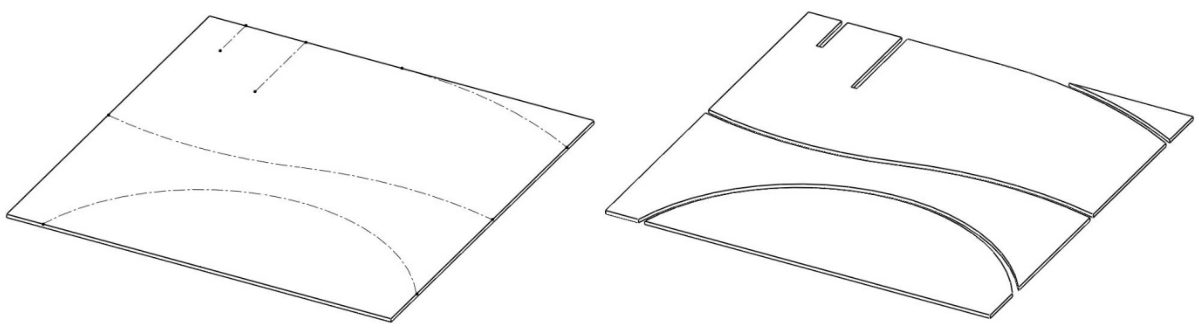
2. **Hand shear.** Partially shears a part.



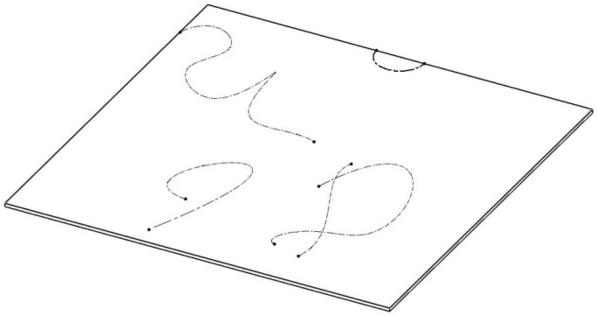
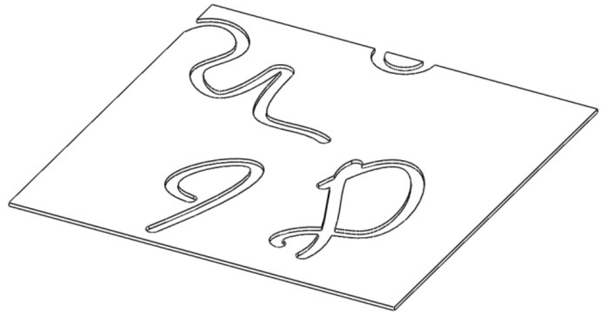
3. **Corner shear.** Shears a 90° angle into part from outside edges.



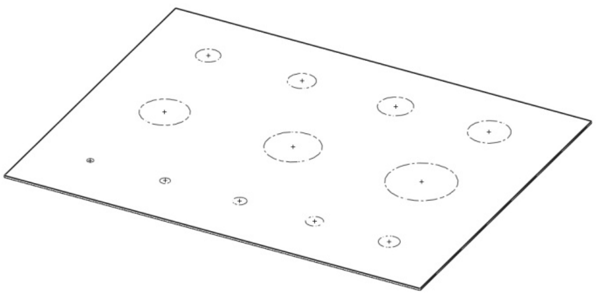
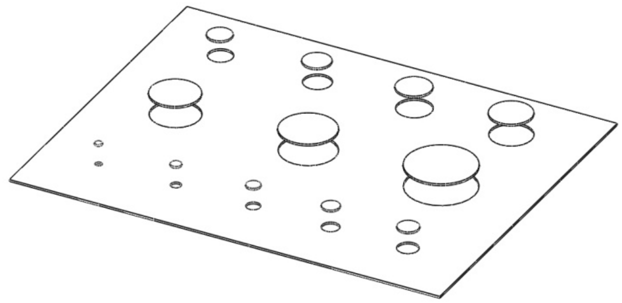
4. **DoALL bandsaw.** Creates cut from edge of part. The cut is not required to be straight, however, it cannot have a radius tighter than 4". If possible, use other sheetmetal cutting tools since this process is slow and has low precision. The width of the cut is ~1/16".



5. **Manual plasma cutter.** Creates cut in part. The cut can be complex and made internally or from an outside edge. However, plasma cutting results in hard slag that takes time to remove. Since the lab's plasma cutter is manually operated (as opposed to CNC), tolerances typically range from ±1/8" to ±1/4". The width of the cut is also between 1/8" and 1/4". The plasma cutting process only works with electrically conductive materials.

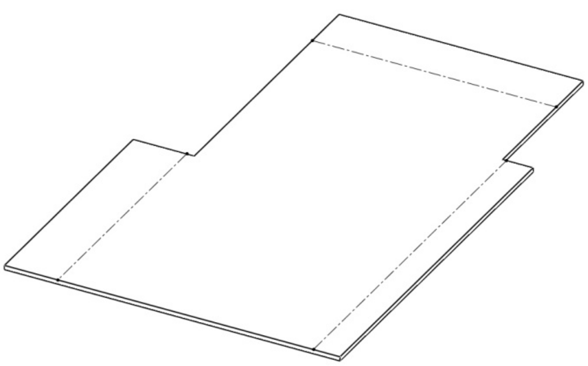
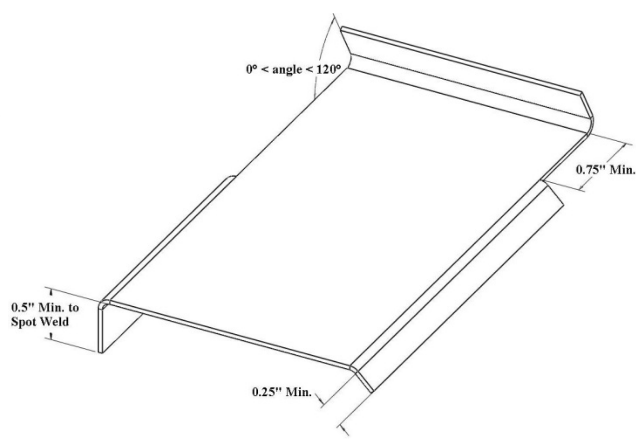



6. **Punch press.** Creates holes in flat sheetmetal. Available hole sizes are: $\frac{3}{16}, \frac{1}{4}, \frac{5}{16}, \frac{3}{8}, \frac{7}{16}, \frac{1}{2}, \frac{5}{8}, \frac{3}{4}, \frac{7}{8}, 1,$ and $1\frac{1}{4}$ inches. Do not call for line fits! Punched holes can be easily finished with a hand drill. For example, if you want a ¼-20 UNC fastener to pass through the sheetmetal, specify a free fit hole of Ø0.266" as noted in the tap and clearance drill chart. Punch the hole to ¼" and then hand drill the hole to final size.

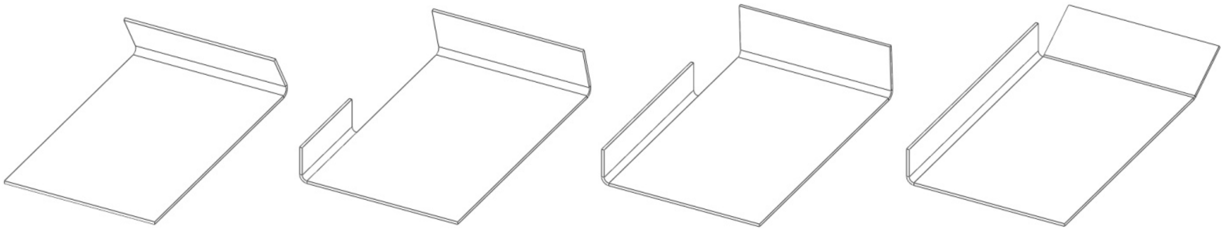



FORMING PROCESSES:

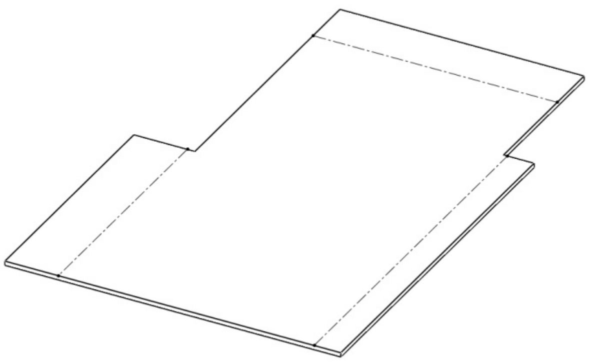
7. **Beam brake.** Creates a bend across a part. Bend angles can range from 0° to ~120°. Note the following:
    a. The flange (i.e. the bent section) length must be at least ¼" (½" if spot welding).
    b. There must be at least ¾" between the bend line and flanges in opposite directions.

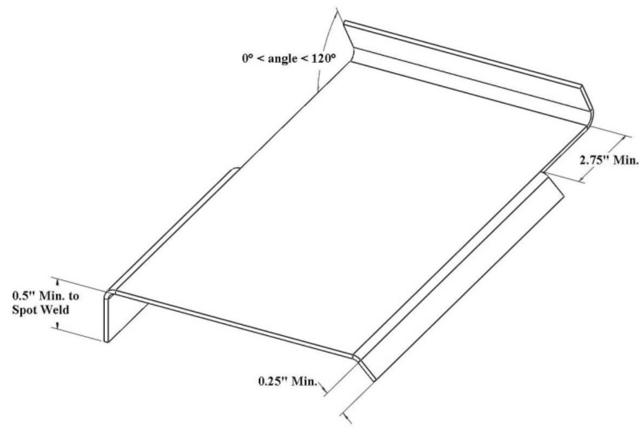



c. Interference between existing flanges (in the same direction as the desired bend) and the beam brake reduces the maximum bend angle to as little as 20°.

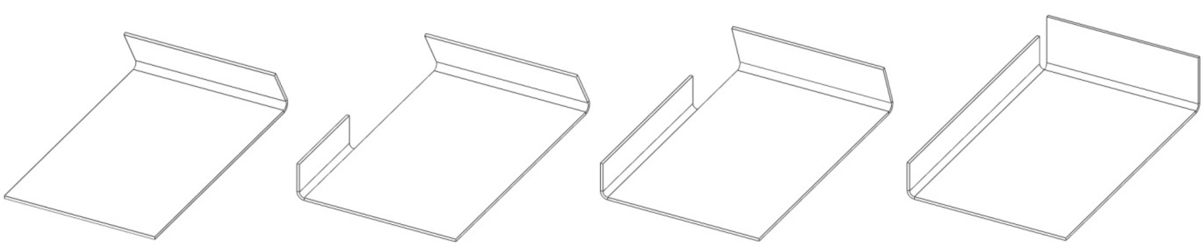


8. **<u>Finger brake.</u>** Creates a bend across a part such that interference between previous bends and machine is mitigated. Bend angles can range from 0° to ~120°. Note the following:
   a. The flange (i.e. the bent section) length must be at least ¼" (½" if spot welding).
   b. There must be at least 2-¾" between the bend line and flanges in opposite directions.



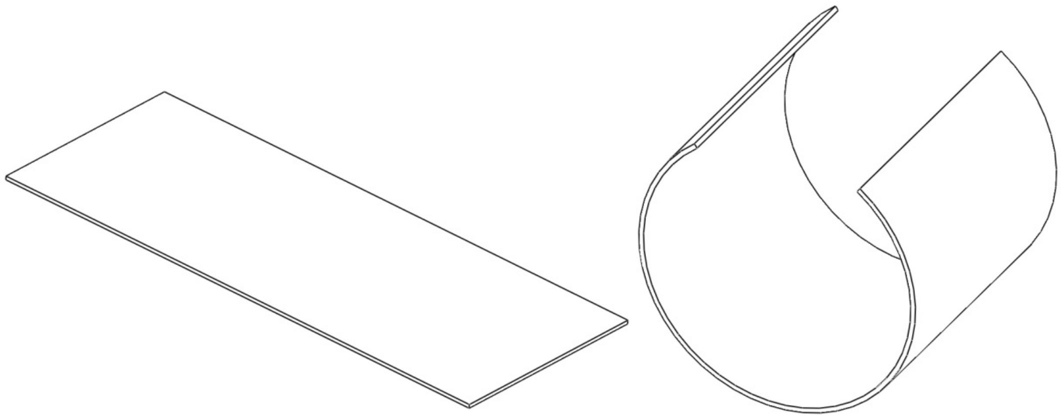


   c. Interference between existing flanges (in the same direction as the desired bend) and finger brake may reduce the maximum bend angle depending on the geometry of the existing flanges.
      i. If existing flanges are shorter than 3-½", the bend angle may be reduced to 90°.



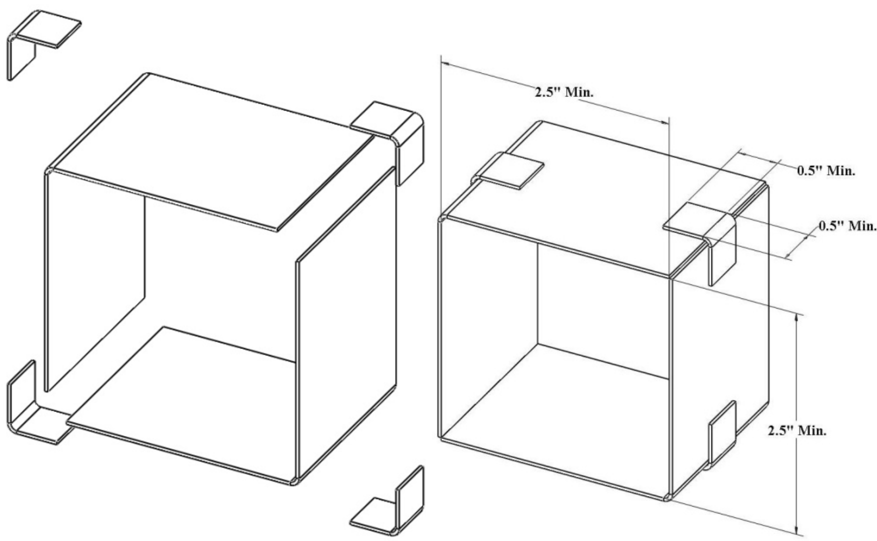
      ii. If existing flanges are longer than 3-1/2", the finger brake functions like the beam brake shown above.

9. **Roller.** Creates a radius in a flat sheet.

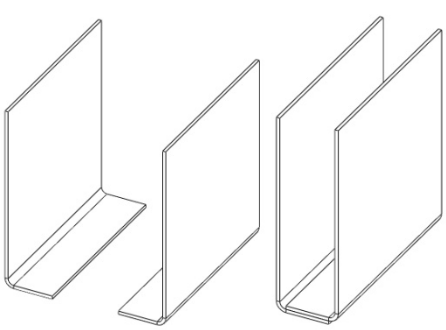


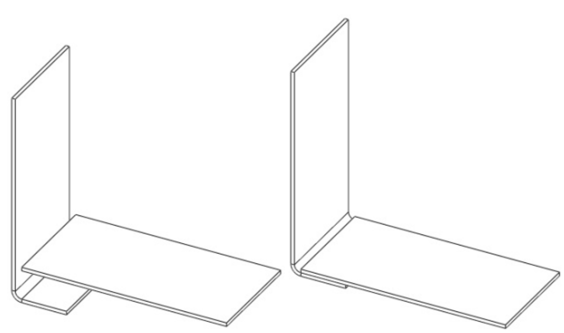
JOINING PROCESSES / METHODS:

10. **Spot welder.** Quickly and permanently joins similar metals (i.e. steel to steel). The laboratory's **spot welder** requires a minimum of 2.5" of open space above and below the spot weld location. Additionally, there must be ½" of material overlap for the weld joint.
    a. Using L-brackets



    b. With part overlap

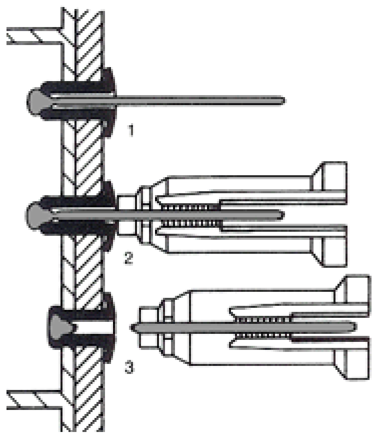




11. **MIG welder.** Permanently joins similar (i.e. steel to steel) metals. MIG welding is more versatile than spot welding; however, it takes significantly more time and skill. When possible, design to use the spot welder rather than the MIG welder, but realize MIG welding is still an efficient way to permanently attach sheetmetal pieces. *As with the other sheetmetal processes, the TAs can further illustrate these points or discuss particular design and application questions.*

12. **Fasteners.** Design with clearance holes for fasteners. If considering tapped holes, remember fasteners require 5 threads of engagement to function with full strength. Although sheetmetal in general refers to any metal with a thickness less than ¼", sheetmetal (in lab) is at most 0.060" thick (due to equipment limitations). Achieving 5 threads of engagement would therefore require a screw with 84 threads per inch. Since no fastener on the tap chart has 84+ threads per inch, do not design threaded holes in sheetmetal. If the need truly exists for fasteners to thread into sheetmetal, investigate rivet nuts. For more fastener information, review the Fastener Lecture Notes.

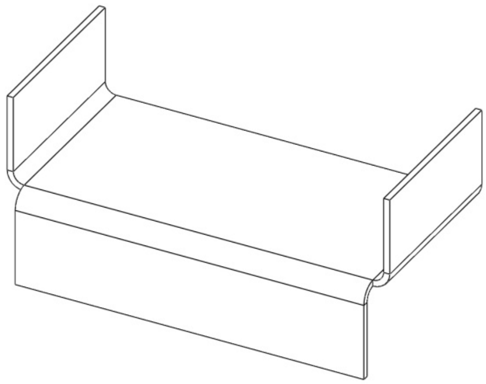
13. **Rivets.** Semi-permanently join two parts. For a good rivet design guide, see http://www.hansonrivet.com/w13.htm.



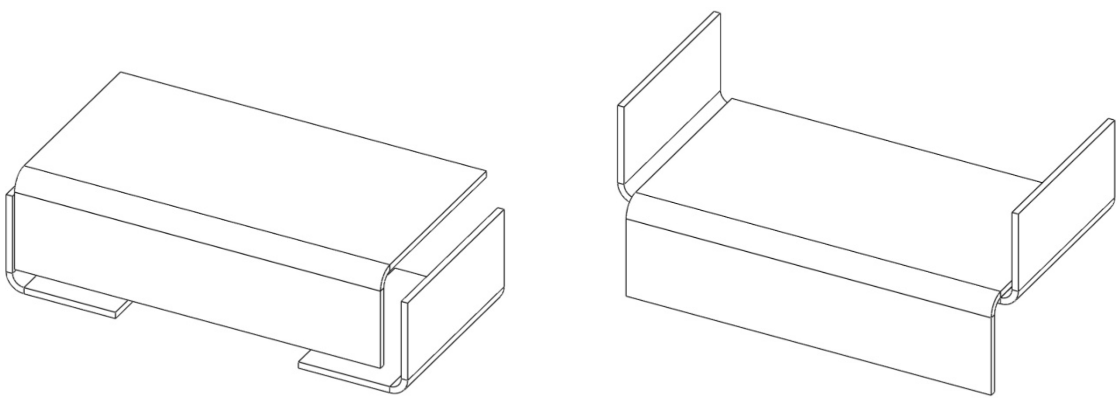
From http://www.hansonrivet.com/w13.htm

Sheetmetal Simplification Example 1:

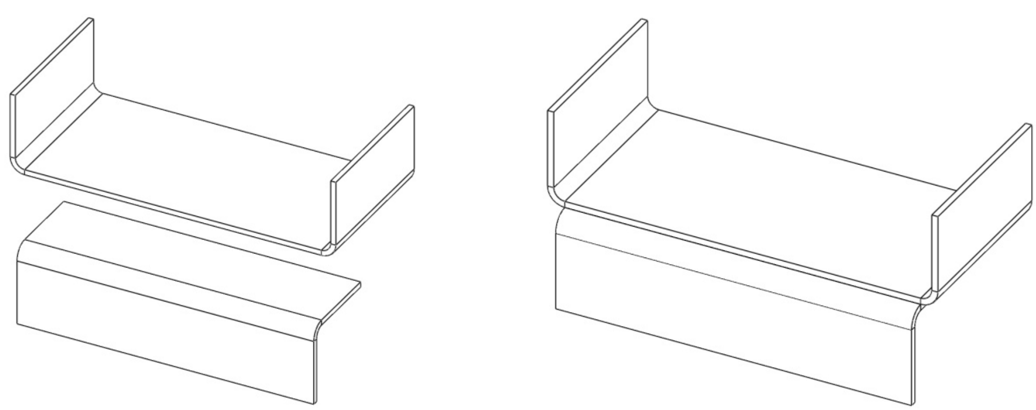
Impossible – Single part with opposing adjacent bends cannot be created on the lab's sheetmetal forming tools.



Simplification A – Spot weld 3 simple pieces together.

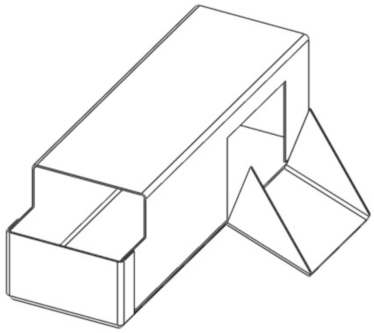


Simplification B – Spot weld 2 simple pieces together.



Sheetmetal Simplification Example 2:

Impossible – Single, complex part with bends that cannot be created on the lab’s sheetmetal forming tools.



Simplification – Spot weld 4 simple pieces together.