Demtool Inc.

1611 Snyder's Road East
Petersburg, Ontario
N0B 2H0

(519) 634-8001

(519) 634-1021

info@demtool.com

www.demtool.com

Sheet Metal Guidelines

This document attempts to address the most common issues and best practices for sheet metal design and fabrication at *Demtool*. These are the steps we take to prepare your design for production. Following these steps will result in lower costs and better parts.

Laser Cutting & Etching

Drawings should be provided as flattened patterns in DXF (ASCII/2000) format. DXFs should avoid scaling and be authored 1:1 in either inch or millimeter units. Dimension annotations are also useful to include for confirming scale.

Only cut profiles should reside on ***Layer 0*** and be completely closed (continuous) meaning no open gaps. Overlapping profiles should also be avoided. Block entities should be exploded or avoided altogether to ensure all elements and not just parent blocks reside on *Layer 0*.

Orientation of the flat pattern is also important to have correct, especially when the part uses material with a finished side, like #4 brushed or #8 mirrored stainless. Materials with finished sides should always be drawn face up, or as if the finished side is facing towards us the viewer, not away. This is because these finishes come with a protective film which is always faced up on the cutting table.

The opposite holds true for texturized surfaces like checker plate/diamond tread or rigidized sheet. Whereas a finished (protected) surface should be oriented face up, textured surfaces like these are cut face down.

When laser cutting any sheet metal the bottom side will always have a little more burr than the top, and may acquire scuffs or scratches from the table bed slats. This is why we consider the top the **good side**. In parts without a specific finished surface side, designers may still wish to imply which side is *good* by orienting your flat patterns up or down accordingly.

In the case of finishes with a grain direction, such as *#4 brushed stainless,* the rotation of the profile in your DXF matters as well. Grain directions should always be oriented **horizontally**. An isometric 3D view with annotation showing which side is finished (or polished/good/etc.) and in which direction the grain is intended to go (when applicable) should also be included to confirm these details.

Any profiles intended for etching should reside on a separate layer named **ETCH** (not *Layer 0*). Etched profiles are always marked on the *good* side.

Bend lines *must* be included and dimensioned in the flat pattern, and should reside on separate layers (not *Layer 0*). When a sheet metal part is modeled in 3D, bend lines and blank lengths are calculated very precisely by the software. When exporting flattened DXF patterns from 3D CAD, it is advantageous to include the precise bend lines for use during the forming process.

Brake Press Forming

Any critical formed dimensions must be illustrated in accompanying formed views. Excessive tolerances and unnecessary dimensions will increase the cost of the part.

Typical problems we often encounter when reviewing formed sheet metal designs include:

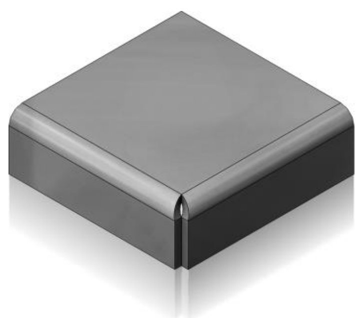
1. **Bend radius too small.** Inside bend radii should be equal to or larger than the material thickness being used. If a smaller radius is crucial it can sometimes be achieved, but not without excessive strain on our tooling. We understand that design constrains may require this sometimes.
2. **Bend radius too large.** The case where the inside bend radius is larger than material thickness adds a degree of difficulty as well. Tooling selection, bump forming or custom tooling are options for achieving this, but add extra cost and time to production. Whenever possible keep inside bend radius equal to material thickness.
3. **Bend flanges too short.** The simplified formula dictating the shortest flange length is *4x the material thickness from the bend line*. Shorter can sometimes be achieved but it's avoided if possible.
4. **Holes or other cut-outs in bend zone.** Putting holes too close to a bend line will result in a flared distortion of the hole after forming. When designing hole locations use the above *4x material thickness from bend line* rule. If a closer proximity must be used, sometimes a relief slit can be cut along the bend line, beside the hole or slot and welded back up if needed after bending to achieve the form without distortion. Another option might be to machine the hole or feature after forming. Obviously these types of workarounds add cost to the part.

Weldments & Assemblies

Often we see boxes or pans which require all 4 corners to be welded but they are left wide open, meaning there is a sizable gap between the edge flanges after they are formed. The corners should use the minimum gap possible to make nicer welds. A quick fix for this in 2D CAD is to move the welded edges to be collinear with the adjacent bend line.

Overlap style corners are also acceptable, but result in a less-attractive welded edge. Gaps should still be kept to a minimum. Corner reliefs are necessary in materials thicker than 11ga (1/8”).

Open corner, gap too large



Open corner, minimal gap

Closed corner, overlap style

Assembled and/or welded parts must include drawings for each component as well as the assembly and any sub-assemblies used. A bill of materials can also be helpful and is usually worth including. All welding and assembly requirements should be detailed in the drawings.

If the assembly is complex it is advantageous to include a 3D model in SolidWorks or STEP format. These can be used to inspect fitment or mating features in closer detail.

When including sheet metal designs from SolidWorks make sure your model flattens properly before sending.

Machined Holes & Countersinks

Holes are best drawn at mid tolerance. This means the mid point between the range of acceptable deviation. For example, if a hole is specified at 4mm diameter +/- 0.2mm then 4.2-3.8mm would be acceptable and the hole should be drawn at 4.0mm (being the mid point of this range). But, if that hole was 4mm diameter +0.3 /-0.1 then the accepted range would be between 4.3 and 3.9mm, therefore the hole should be drawn at 4.1mm.

Our general tolerance on the laser is **+/-0.005”**. If more precision is needed, we can create a custom calibration for the material and part as required.

Holes intended for tapping in up to 1/2” thick steel should be drawn at the exact tap drill size. In thicker plate the material around the laser cut hole becomes much more hardened by heat, putting added strain on tap bits. Workarounds involve laser cutting pilot holes, and drilling them out before tapping, or by machining the cut features after forming. This is usually determined by us on a per-case basis.

When countersinking holes, it’s important to minimize the chance of a burr on the opposite side by enlarging the through hole. A general rule of 0.030” material thickness is left intact under the countersink to avoid this burr. See the diagram below for further explanation.

It is also helpful to know the specific fastener being used when countersinking so we can properly set the depth of the head. Annotating countersink drawings with fastener details is the best way to communicate this.