Demtool Sheetmetal Guidelines 2021

The updated Sheetmetal Guidelines 2021 document is ready to be released to customers.

It can be found in the root directory of S-drive: S:\FORMS\Demtool Sheet Metal Guidelines 2021.pdf

Please do not edit the Word document without consulting Doro.

Sheet Metal Guidelines 2021

This document addresses the best practices for cost-effective sheet metal design and fabrication and covers some common issues we see at Demtool. Following these steps will result in lower costs by reducing the amount of time we need to prepare your files and make necessary adjustments to provide more accurate and consistent parts.

Laser Cutting & Etching

For fastest quoting we require detailed PDF drawings and 3D solid models or native Solidworks files. Our high-end laser and forming software is based on Solidworks and will translate the 3D solid model and assign sheet metal properties and bending tools to provide the most accurate flat pattern to match our brake press tooling. File formats we can open include STEP, Parasolid X_T, IGES. Any critical final dimensions must be detailed in the PDF drawing. Excessive tolerances, unnecessary dimensions, and workarounds often increase manufacturing cost.
Orientation of the flat pattern is also important, especially when using material with a finished side, like #4 brushed stainless. Finished sides should always be drawn face up, or as if the finished side is facing towards us the viewer, not away. The grain direction is horizontal along the x-axis. 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 sheet metal the bottom side will always have more burr than the top and may acquire scuffs or scratches from the table bed slats. Therefore, we consider the top the good side for all materials. 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.
#4 brushed stainless material is cut with the good side face up with protective PVC plastic film and the bottom side also has a back pass which is #4 brushed finish but not the same quality as the top and not protected with film.
Where finishes have a grain direction, such as #4 brushed, the rotation of the profile in your drawing matters as well. Grains should always be oriented horizontally (along the x-axis). If the

part is formed, an isometric 3D view with annotations detailing which side is finished and in which direction the grain goes should also be included.
Laser etching will only be applied to the good side. We can add part numbers or text in any font if the location and details are provided in the PDF drawing. Any logos or artwork intended for laser etching must be clearly noted and preferably included in the solid model file.

Brake Press Forming & Typical Issues

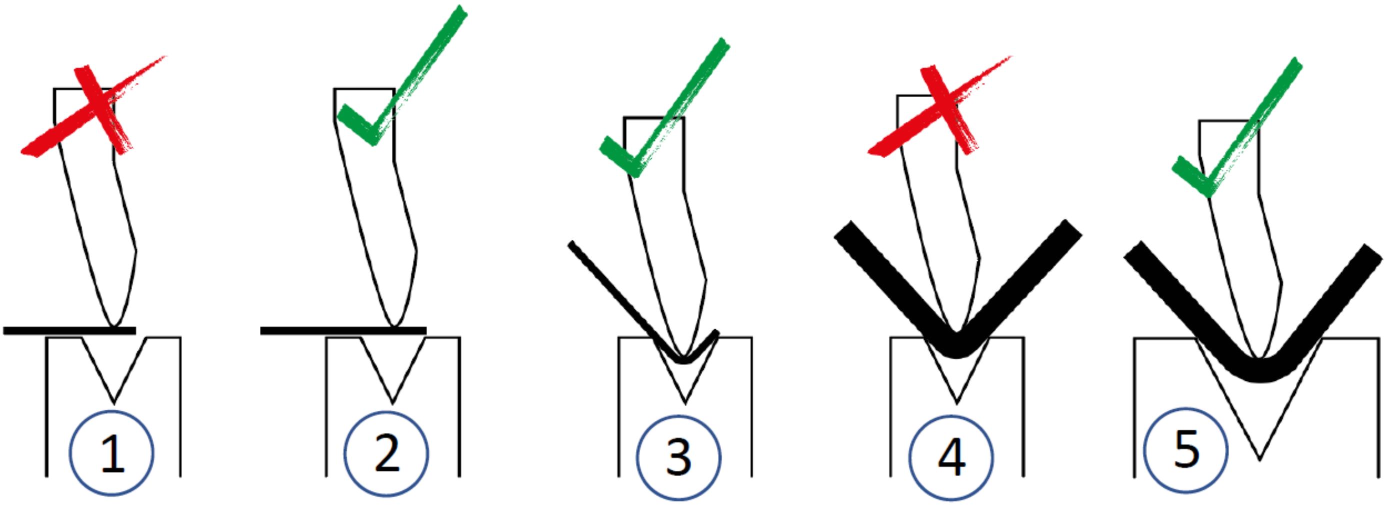
The most efficient and cost-effective way to quote parts is a 3D solid model file with inside bend radius equal to material thickness, this is the natural flow of material when bending metal.

1. Bend radius too small or large

The inside bend radius is determined by the bottom die tool and following typical guidelines will be equal to the material thickness being used. If a smaller radius is critical it can sometimes be achieved with a narrower die, but not without added complexity and time to ensure accuracy. When the inside bend radius is larger than the material thickness, it also adds complexity and time. Thick material cannot have a smaller radius because the tonnage required to bend the part would destroy the narrow die.

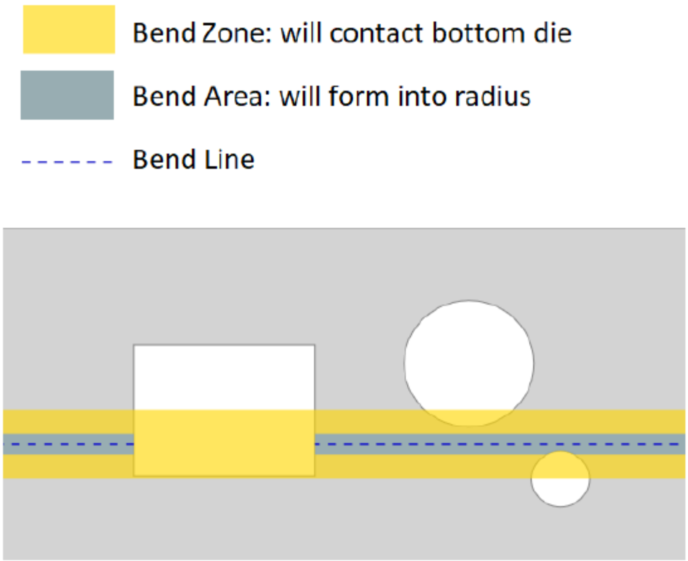
2. Bend flanges too short

The simplified formula dictating the shortest flange length is 4x the material thickness from the bend line. Shorter can sometimes be achieved with a narrower die but avoid if possible. See diagram below: 1: Flange too short, 2 & 3: Good, 4: Bend rad too tight for tooling, 5: Bend rad correct using larger die

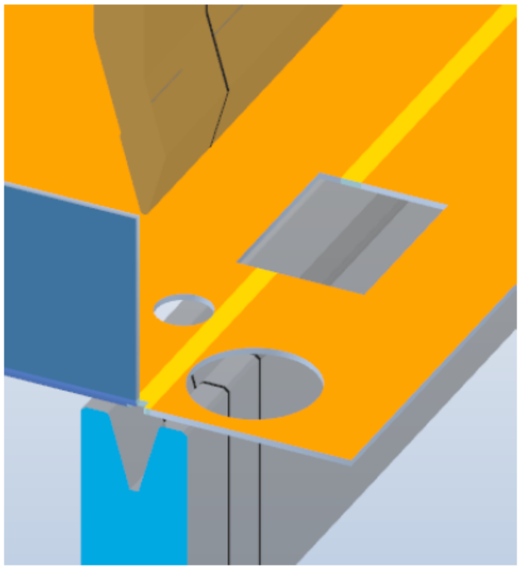


3. Holes or other cut-outs in bend zone

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 is required, a relief slit can often be cut along the bend line, beside the hole or slot and welded again afterward to achieve the form without distortion. Another option is to machine the hole after forming.



When holes and features fall into the bend zone they are not supported by the die



Some of these features are not supported by the shoulder of the die and will deform (the yellow line is the bend area)

4. K-Factors wrong

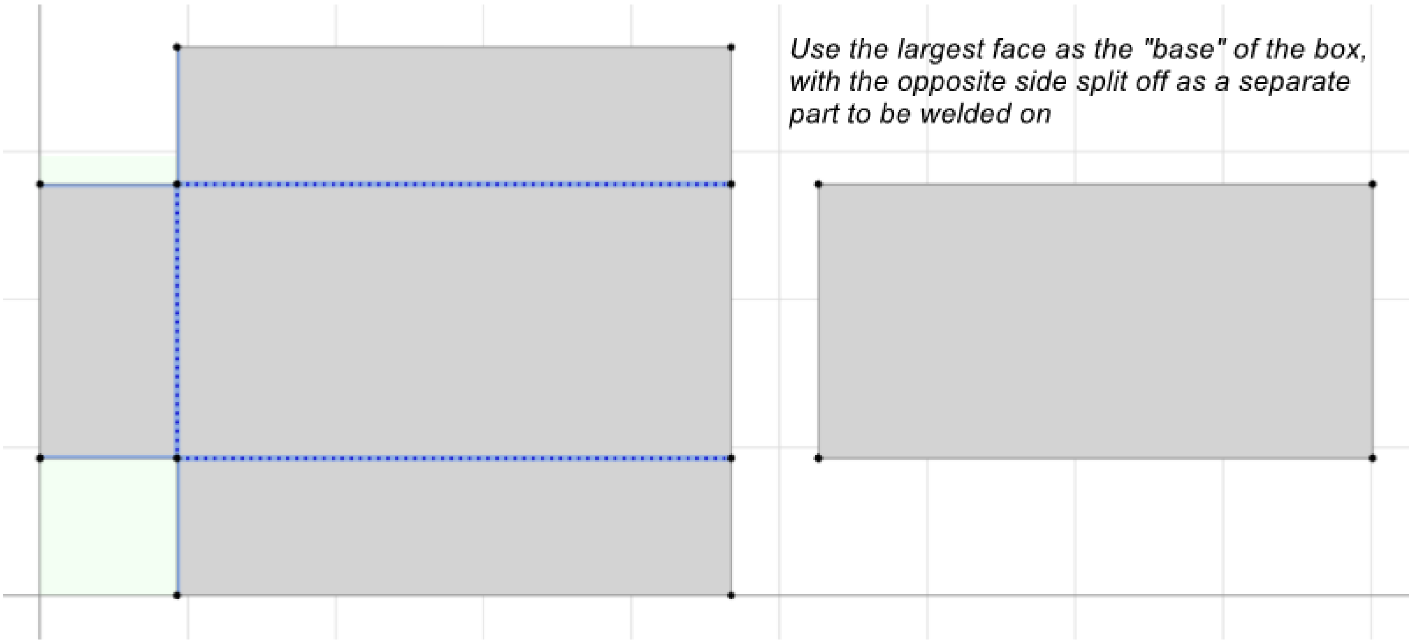
When modeling sheet metal in 3D use K-factor = 0.45 on formed parts, this matches the tooling that will be used to form your parts.

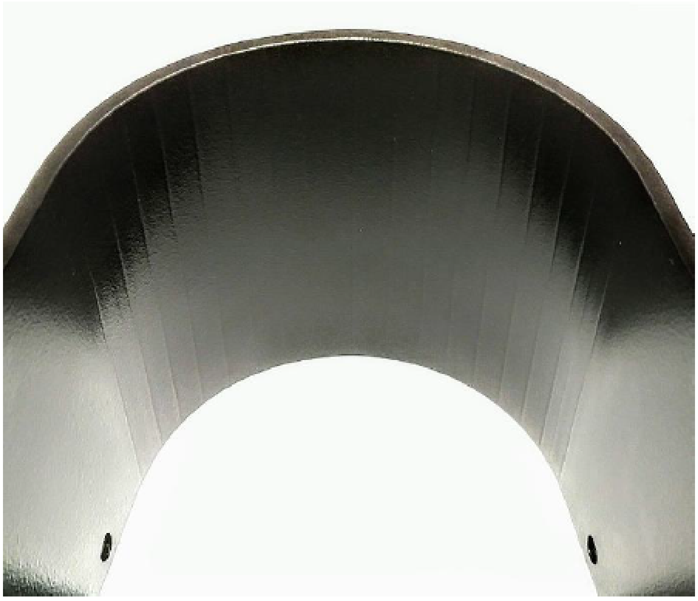
5. Parts or flanges too large or collide with the press

Some parts simply cannot be formed as one-piece and will need to be split into multiple pieces and welded together. Typically, we see this on large 4-sided boxes or narrow & deep U-channels. A good rule-of-thumb is that if the height of the flanges of a U-channel are deeper than the width between them, the part likely cannot be formed. The one exception is smaller parts where we can sometimes use special "gooseneck" tooling to form the U-channel. Parts can sometimes remain one-piece by using an auxiliary bend and pre-bending down the middle and flattening after all bends are complete, this is called “back bending” and it does leave tooling marks on both sides. For boxes specifically, it is ideal to make the largest face the "base" of the box and split off the opposite face as a separate piece to be welded on.

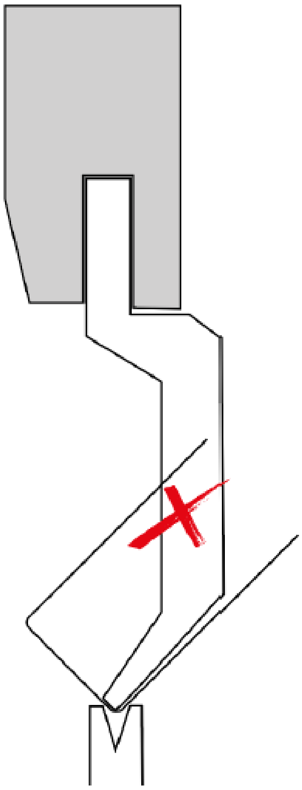
6. Part design demands large-radius bend

Sometimes a part is designed with a very large bend radius for functional or aesthetic reasons. To achieve this we can roll the part or, make custom tooling or use “bump forming” which is a series of small bends spaced close together. These bends will produce visible lines or steps, not a smooth surface, however an additional sanding process can help reduce this and leave a cleaner, smoother surface.

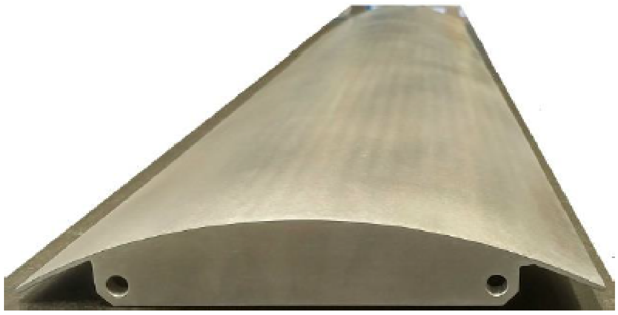

Use the largest face as the "base" of the box, with the opposite side split off as a separate part to be welded on



Large radius bend bump-formed, leaving visible bend lines inside and outside – the outside of this part has been smoothed out to leave a cleaner appearance – parts are assessed on a per-case basis



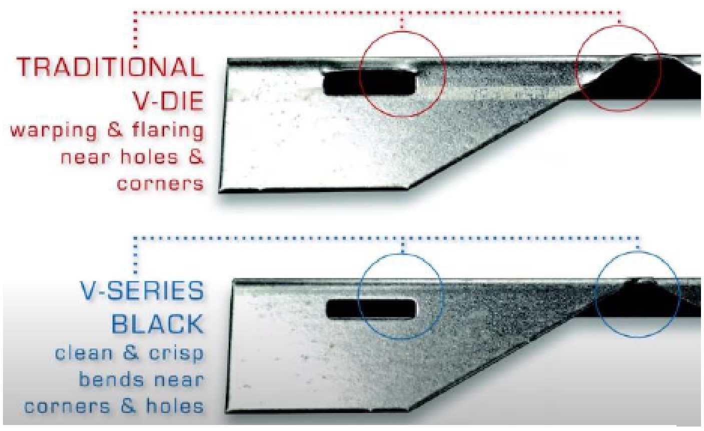
When parts are taller than they are wide it can create problems. For small parts we use gooseneck tooling. Larger parts like above need to be either 2-pcs welded or use a "back bend"



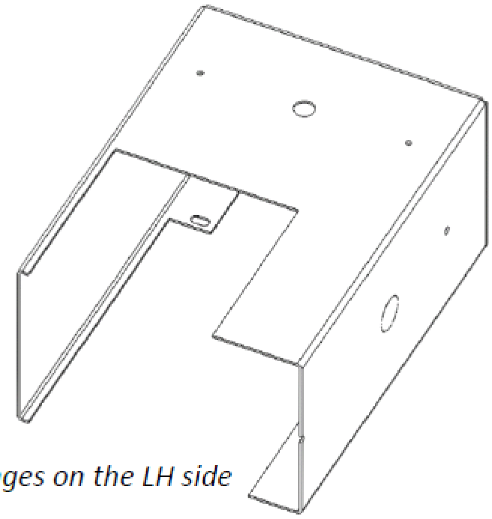
Bump-formed part with visible bend lines

Brake Press Exceptions

Traditional V-die tooling has limitations explained above regarding distortion of features that are within the bend area. Demtool has invested in special tooling that increases our ability to deal with these types of situations. Called "V-Series Black Roller Dies" this tooling allows much shorter flanges and reduces the width of the bend zone, allowing features to be closer to bends and almost eliminates distortion of edges and features. There are still limitations with this tooling: material cannot be too thick and exceed the tonnage rating of the tool, and flanges must meet certain minimum length requirements depending on the specific tool used. Parts are assessed on per-case basis.

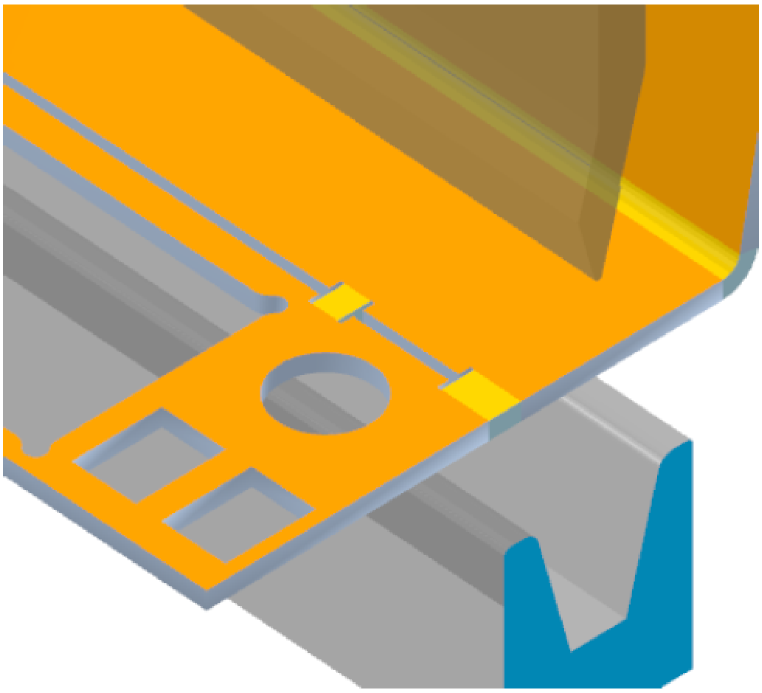


Special bottom dies help eliminate warping and flaring and allow for very short flange lengths



Very short flanges on the LH side

When features must be distortion-free but there are no tooling options available we can add laser-cut incisions on the bend lines. The incisions allow the rest of the material to bend and not the material beside the features. Common practice is to then weld the incisions closed and grind and sand the welds to blend-in with the rest of the part and make it appear seamless when complete.



Incisions added to the laser program on the bend line, also called relief cuts, allow the material to move instead of being forced to form, keeping holes and features distortion-free

Forming Tolerances

Critical dimensions & features must be clear on PDF drawings and non-critical dimensions should be marked as refence by showing them in parentheses. Formed parts cannot be held to machining tolerances and instead should be designed to accommodate general fabrication tolerances. Parts with multiple bends are very difficult to hold tight tolerances and expectations must be realistic. Excessive tolerances and unnecessary dimensions often increase manufacturing time – consider the end use of the part when applying tight tolerances, or if drawings cannot be altered please communicate that looser tolerances are acceptable.

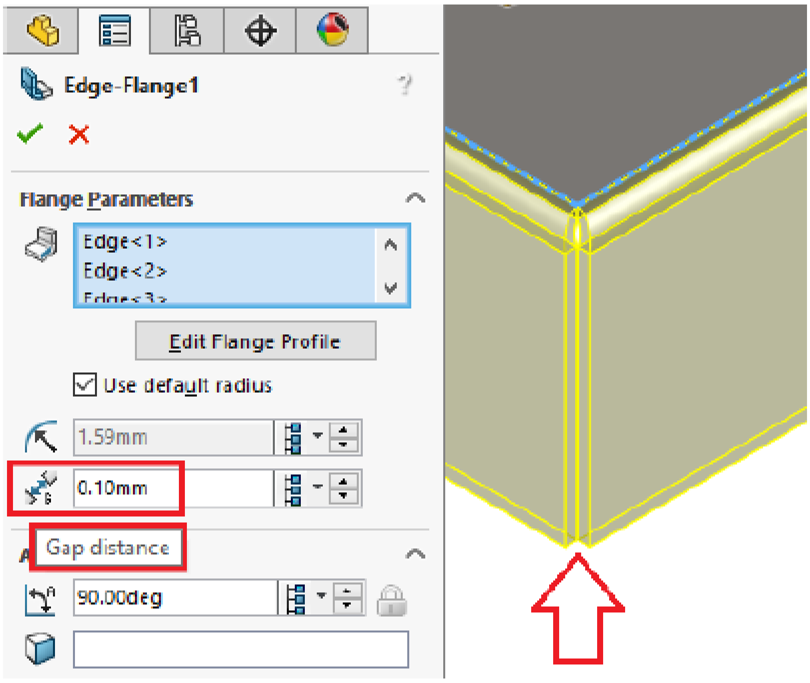
For one-off and low volume production a general tolerance of ±0.020” (.5mm) per bend should be achievable. For larger volume orders we can spend extra time and cut sample parts and dial-in the form programing ahead of production to achieve tighter tolerances. When forming any checker-plate material the tread pattern randomly changes the thickness of the material in the bend zone and an allowance for ±0.125” (3mm) should be accounted for.

Weldments – Closed Corners

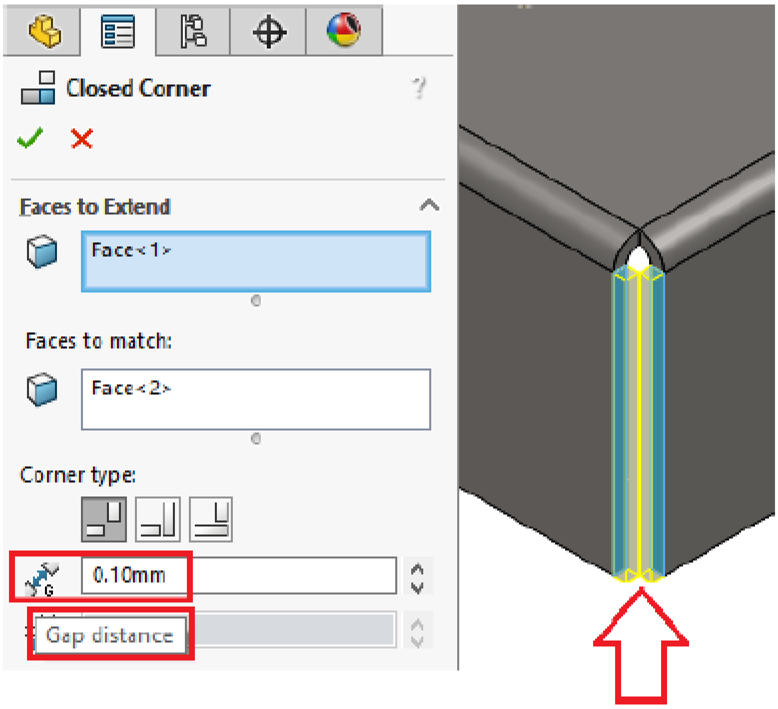
Often, we see boxes or pans which require all 4 corners to be welded but they are left wide open, meaning there is a sizable gap between the edge flanges after they are formed. The corners should use the minimum gap possible to make nicer welds. A quick fix for this in 2D CAD is to move the welded edges 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” or 3mm).

If corners don’t need welding, and the design dictates, you may still use open corners. However, it’s especially important to keep them closed when welding thin material, as the risk of warping is much more severe due to the extra heat needed to fill the gap with weld. In Solidworks there are options to control the style of closed corner and the gap distance between the edges of the flanges. Use a very small, almost zero gap distance for perfectly closed corners for welding.

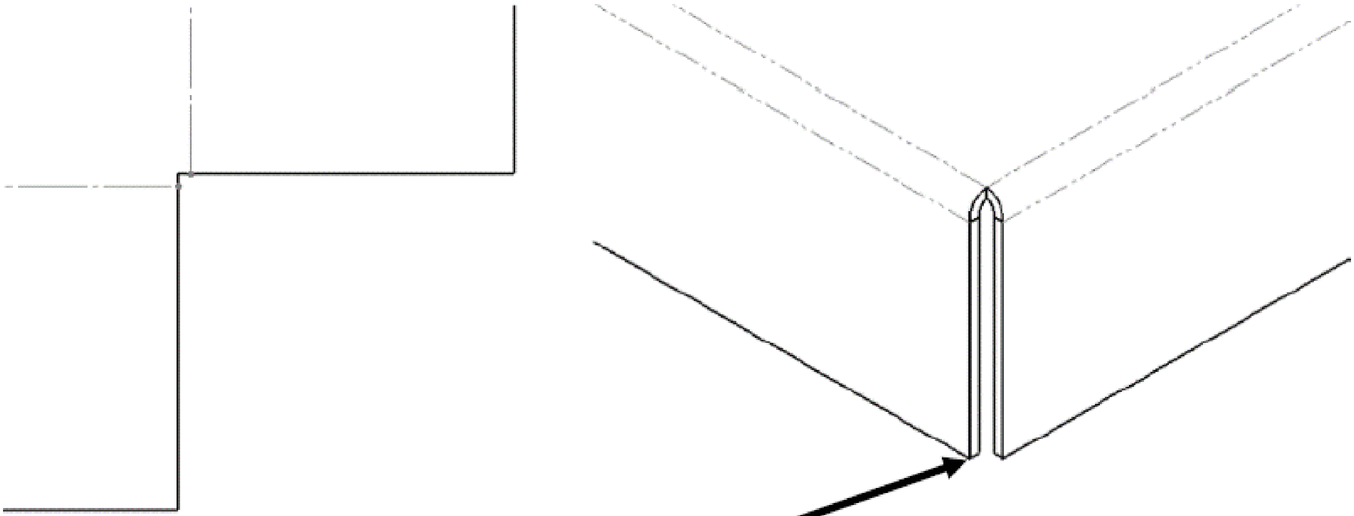


Edit flange features to set gap

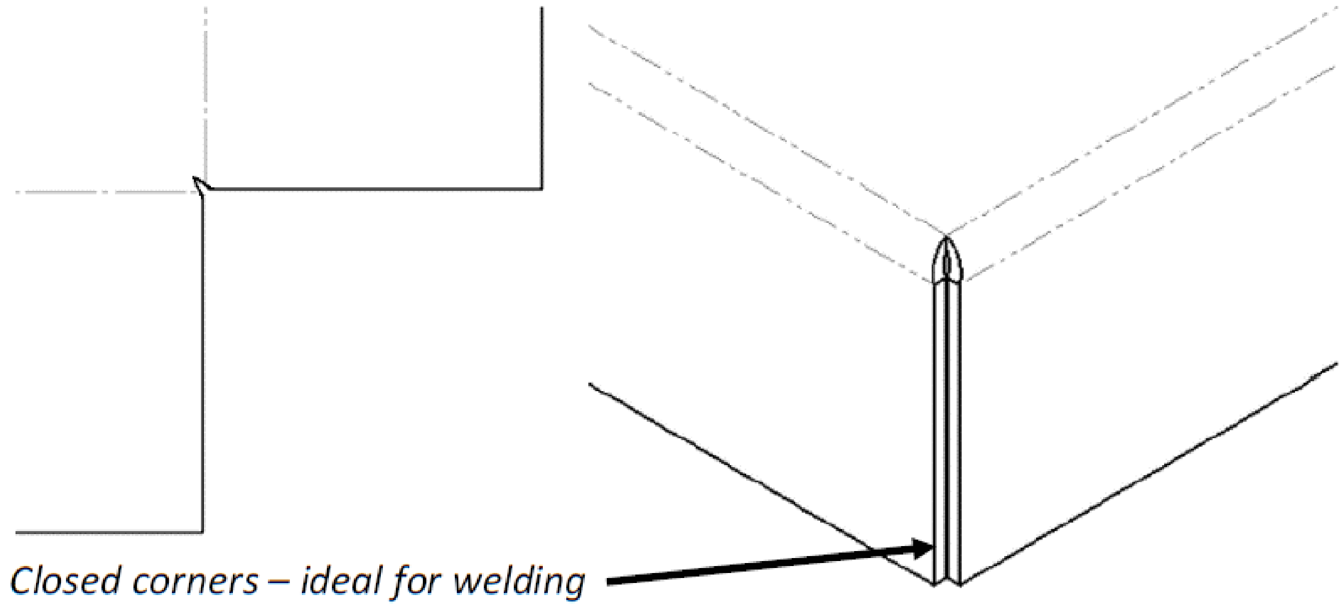


Add Closed Corner feature to flanges

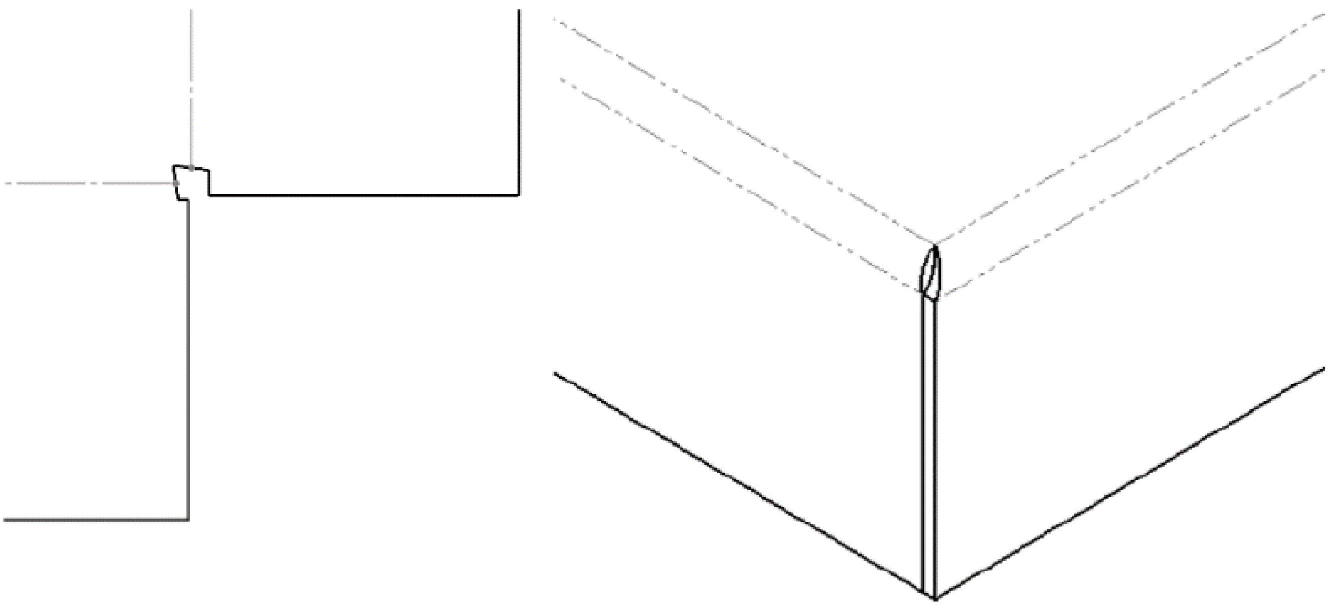
[**Closed Corners – Examples – Note Bend Lines and Edges in Flat Patterns**]()



Open corners – not suitable for welding



Closed corners – ideal for welding



Overlapping corners – suitable for welding but not ideal

Weldments

Whenever possible weld symbols and notes should be included on drawings. Grinding and finishing requirements should also be included. Watertight welds will be leak-tested using a dye-penetrant method. Stainless steel weldments can be cleaned by mechanical and chemical methods, such as Scotchbrite or Surfox electrochemical “TIG brush”. Welds on #4 will be re-grained to match the surrounding surfaces. Welds on 2b stainless will be orbital sanded and blended.

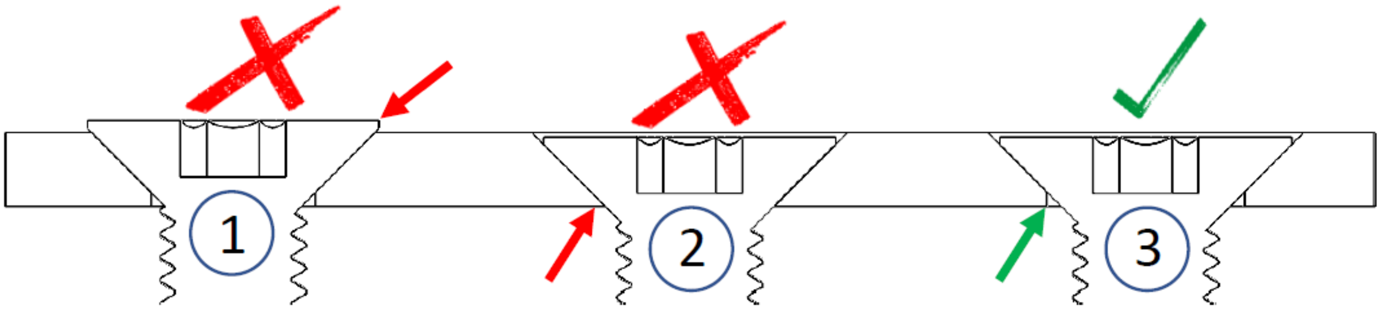
Machined Holes & Countersinks

Our general tolerance on the laser is approximately ±0.005” (0.13mm) but this varies with material thickness. If more precision is needed, we can create a custom calibration for the material and part as required.

Holes intended for tapping should be drawn at the exact tap drill size. Small holes in thick material may have to be machined and are assessed on a per-case basis. Holes that require tight diameter tolerances must be noted on the drawing with tolerances for reaming.

When countersinking holes, it’s important to minimize the chance of a burr on the opposite side by enlarging the through hole so the countersink does not cut thru to the opposite side. A general rule of 0.030” (0.75mm) 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. Note that standard counter-sink sizes in some CAD programs like Solidworks may not be large enough to bury the head of the fastener flush or below the surface of the part.



1: countersink size not large enough to bury the bolt head below the surface of the part

2: countersink enlarged, head now flush, but counter-sink cuts to the bottom side of the material and will push the thin material out creating a sharp secondary burr

3: countersink and thru hole both enlarged, head flush and a short flat section eliminates risk of burr

Note: some fastener/material thickness combinations could have the head of the fastener protrude below and the mating part would require a larger counter sink in the thread to ensure the fastener is seated.

PEM Clinch Fasteners & Holes

Best practice for self-clinching fasteners is to follow the guidelines provide by Penn Engineering (pemnet.com) Draw holes exactly to size and dimension with the tolerance outlined for each specific PEM. It is critical to clearly indicate which side the fastener protrudes. Parts are assessed on a per-case basis and may have the PEMs installed in the flat pattern or after forming, depending on geometry and style/size of PEMs. When selecting PEM nuts, choose the longest shank-code that falls within the material thickness restraints. Follow Penn guidelines for distance to edge of features and remember that PEM holes too close to bend lines will distort and would require incisions. There are stainless PEMs for use in aluminum & steel sheet, and special hardened stainless PEMs specifically for stainless sheet.

Revision #11
Created Thu, Feb 4, 2021 7:40 PM by Andrew Kittmer
Updated Thu, Oct 21, 2021 11:42 AM by Andrew Kittmer