Making tools

The Form Gusset

Sheet metal gussets, or “Stiffeners”, used to strengthen a bend. Gussets are formed with a specially made tool.

There are at least two ways that a gusset tool can be created. One is by drilling a hole across the die opening and inserting a pin of the appropriate diameter. This method is commonly found in American Planed Style Tooling. The second way, common with precision ground tooling, is to use spacers and a slightly taller sheet metal “Gussetting Shim” set on edge creating a ridge slightly lower than the top of the die, figure 1.

In both methods the punch is split, left and right of the gussetting Tool with enough clearance to allow the material to flow rather than cut the material being formed. The spacers are typically thin, approximately .020 to .040 inches thick, one on either side of the gusset; keeping the die sections away from the gusseting tool; figure 2.

If the spacer is too thin the gussetting tooling will tear through the workpiece and weaken the part. If there is too much clearance the gussets will not function as a stiffener and in some cases will actually weaken the workpiece.

The sheet metal is bent as usual between the punch and die except in that area where the “gusseting tool” is present. The spacers separate the gusseting tool from the rest of the die; this provides a relief area for the sheet metal “flow” from the gusset to the area being formed normally by the punch and die set. Though not shimmed the punch will also need to be spaced apart, with the punch edges lining up directly with the die edges. This will prevent any serious damage to the tooling, parts and/or the operator. Figure 3 shows a 3-D view of a gusseting tool assembly.

To produce the flowing effect, small pieces of shim are added to either side of the gusset itself. Be sure to place a thin strip of stainless steel along the top of the bed or rail, between the dies and gusset assembly; it’s what protects the bed or rail from damage.

This will keep the gusset from imbedding itself into the bolster, rail or bed during the forming process.

Make sure this shim runs the entire length of the tooling set in order to maintain a common origin plane.

Carrying a Bend

There will be times when the exact length of the tooling will be unavailable.

For example, let’s say you had a workpiece that measures 6.087 (154.61 mm) along the length of the bend. The
closest die set is too long at eight inches, so multiple pieces of tooling will be needed instead. In this case the total die length only adds up to 6.000 inches (203.20 mm). This means that .087 (2.2 mm) will need to be compensated for through the use of spacing.

While it will not apply to all circumstances, it is generally best to carry the bend somewhere in the center rather than out to the outside of the bend. When the ends are not supported, they will tend to “blow out”, figure 4. The severity of this will vary according to the punch radius and the die width and the amount carried.

When this excess bend length is spread out down the length of the die both ends are supported, and corners will mate nicely, and the bend will be carried without a blowout.

The full length of the bend should have a smooth outside surface; however, some distortion may occur in the center if the span becomes too great, figure 5.

Forming Wraps

Many times a workpiece may have a hole or feature that lies within the area described by the mold lines or outside setback (OSSB).

Within this area, distortion of the feature will occur and will worsen as the feature approaches the bend line. This distortion can at least partially, if not totally, be stopped through the use of forming wraps, Also known as: wipes, double form strips, backups, etc.

The selection of the correct material thickness for the wraps to be used is critical if this process is to be done correctly. For this to work, place the outside surface of the workpiece at 0.044 of the combined total of the workpiece and the wrap, figure 6; the material plus 30% for the wrap thickness.

This will put the outside surface of the workpiece close as to the neutral axis as possible, where it will encounter the least amount of natural material deformation. As you may recall, 0.044 is the commonly used value for the neutral axis; the area within the bend where no change in the material occurs during forming. Of course this is not a solid piece of material so some distortion is still going to occur.

Material thickness = .046

Wrap thickness = .06

The total for this example is .096, 0.044 of .096 is .042, this would be as close to perfect as you could possibly hope for, with the material thickness being .046, figure 7.

The die width for this type of bending process should be selected on the basis of the combined total of wrap and material thickness.

Custom Punch and Dies

Time was, when a unique job came into the shop, something that required a custom tool to form, that print would first be sent to the machine shop. Once the tooling was completed, a test bend would be
made and the tool would go back to the machine shop for adjustments to the design. This might happen more than once, driving the cost of the tooling up.

There is an easier and less expensive method. Cut the tooling profile that you think is correct into a sheet of .250-inch hot rolled steel (HRPO). Cut enough pieces to laminate the tool to length. Now make a test bend; if it doesn’t work, you can quickly make adjustments and create a new tool. This process is much less time consuming and expensive than a machine shop, figure 8.

Figure 9 shows homemade tooling, made for producing a large radius in a narrow strip. If the tool was built in the same manner as figure 8, it would be very heavy. By spacing out the “body” parts of the tool and welding to the nose of the tool, the weight will be substantially reduced.

But, be aware of the number of square inches or millimeters that are interacting with the press brake or the tool spacers. Make sure you are not exceeding any tonnage per foot rules.

Punch or dies produced by this method require, that the tool pieces be cut on a laser or waterjet; punching is not an option due to the scalloping on the edges. Dies can also be created using this method, but may require the use of some kind of urethane sheet to remove any scratching from the finished part.

The tool shown in figure 9 was designed with spacing to reduce its overall weight. Although this tool has reduced weight, under a heavy load, it could have caused some serious spot damage to the press brake if there wasn’t enough contact area.

No less important is the punch angle of the tool: 90, 88, 87 degrees etcetera, included. It’s imperative that the complementary angle of the punch be equal to or less than the included angle of the V die. Failure to achieve this ratio will cause myriad of problems, such as exploding the V-die or damaging the workpiece.

V Dies

You must select the die design based on the intended task and materials. Are you going to be producing a small radius in thin material? Will you be producing an offset bend (joggle), or will you be working with thick materials and large bend radii?

The same rules that apply to selecting tools from a catalog apply to designing your V-die. The general rule is:

- V-dies less than 1/2 in. (16 mm) wide should have a 90° included radius.
- V-dies from 1/2 to 1 in. (16 to 32 mm) wide should have 88° radius.
V dies more than 1 in. (32 mm) wide, should be of a relieved style, Figure 10, with decreasing included angles: 85°, 78°, 73°, and so on as the width gets greater.

The **relieved dies** and angle parameters help to compensate for the increased **springback** produced by corresponding increases in material thickness and/or bend radius. The relationship between the punch angle and die angle do not matter. You can place a 90° punch into a 73° die because there is no interaction between the punch and die.

**Quick to Build**

Once the tool profiles are complete, you are ready to write the code, which generally is a fairly easy task. After you’ve written the code, it’s time to head for the laser. A punch press could be used to make a tool, but the final results will be less than stellar both in the quality of the tool and the parts it produces.

Within the code be sure to include one or more holes that will be used for pinning together all of the individual pieces of the tools. Once the tool pieces are cut and deburred, pin the pieces together using a piece of ready rod or doweling; then weld the pieces into solid tools. If you find that your newly produced tool and parts achieve the desired results, you can reproduce the tool with confidence.

With just a little forethought and a laser, you can build custom press brake tooling quickly, easily, and cheaply.