Press brake dies

Die Selection

Bottom bending and coining "stamp" the inside bend radius into the part. In air forming, the inside bend radius is "floated" as a percentage of the die opening. Once we determine the radius, we can calculate the required bend deductions.

But how do we find the perfect Die Opening—one that will establish the radius we want to achieve regardless of the method of forming? You may ask, "Why should I care? If I'm bottoming or coining, I am going to force the punch, material, and die together and stamp the radius. In this case, the die opening will not make much difference in the establishment of the bend radius. And in an air form, I can just calculate the inside radius by using the 20% rule, establishing the inside radius based on a percentage of the die opening and material type.

“So again, why then should I care?”

To answer that question, let's first look at what happens when you bend a piece of sheet metal in any of the standard die shapes: V-die, acute V-die or channel dies, as shown in figure 1.

The Drawing Process

Have you ever looked at the back side of a bend and noticed the die markings on both sides of the workpiece? These marks are caused by the drawing process—that is, the dragging of material over the top edges of the die and into the die space.

This is not the same type of drawing found in hydro-forming or deep drawing/glossary stamping; no real stretching of the material is occurring.

Besides the die marking, there may be something else occurring in some materials. The bending process could be creating a secondary radius that is completely unique from the radius that you are trying to attain. This secondary radius has a direct relationship to the die opening.

The larger the die opening is in relation to the material thickness, the greater this secondary radius can become, figure 3.

You will not see this radius once the bend is completed. But this secondary radius can and sometimes does manifest itself as bend angle, making the bend appear to have a greater bend angle than is actually being produced in the sheet metal itself. If occurring, this effect can work for you or against you, depending on how it is applied. Knowing that this extra radius is possible and how to work with it can make or break a part.

This effect can be a particularly big issue if you are bottom bending. If you are bottoming in a large die opening and this extra radius is occurring, it can manifest itself as extra bend angle, and this may create a problem. In bottoming, the bend is brought up to the desired bend angle plus a degree or two for springback, just like air forming. But rather
than releasing the material as in an air form, bottom bending continues down with the tool into the die space until the desired depth is achieved, forcing the over-bent material back to the die angle, which in bottoming is usually 90°. This effect is known as springforward, as opposed to springback.

Because the bottoming process over-bends and then forces the material back to the set angle of the die, when the estimated depth of penetration is reached, the part is removed and checked, and we find that the bend is still over-bent. And because we first over-bent the part and are now forcing the material back open, we may think, “I need to hit the part harder to force the material to the set angle.”

There is a big difference between an optimal die opening that is geometrically perfect and the actual die you are going to use. Still, you want to get as close as possible to perfect for numerous reasons—like the one just described. To find the best working die opening, without just guessing, you will need a starting point.

You will find that best starting point by making a couple of minor assumptions about the bend. The first is that the bend has 90° of bend angle, regardless of what the bend angle really is. This will give a good starting number and simplify the calculations, allowing you to work with a 45° right triangle.

**The Perfect Die Opening**

When it comes to any kind of machine, you generally do not want to overuse or underuse it. You get the most value at half of the maximum working value. That being said, isn’t the combination of die, punch, and material really a “machine”? Of course it is.

So, let’s look at the die and try to determine how we use it to half its working value.

So what is half the working value of a die? Under perfect conditions, that point occurs halfway down the die face, as shown in figure 4. This is the point where we want the material to separate from one side of the die, at the tangent point on the bend, and rejoin the die on the opposite face.

In a perfect die opening, this halfway point down the die face is exactly twice the bend’s outside setback (OSSB). The OSSB is the distance from the tangent of the radius and the leg (flat) to the apex of the bend. In this situation, we find that we have an OSSB from the tangent point down to the bend apex, and an OSSB from that tangent point up to the top of the die. You are holding the part geometrically perfect in the die opening.

This makes finding the optimal die opening easy. Again, we’re assuming that the bend angle is 90°, whether it is or isn’t. This means we’re working with a right-angle triangle, and also makes the OSSB equivalent to the outside bend radius. If you know the outside bend radius, there’s no need to calculate the OSSB. Double the outside bend radius, and you have the hypotenuse of a 45° right triangle. From there, it is a simple process to determine the die opening, as shown in figures 5 and 6.

All this can be simplified to the following equation:

**Geometrically perfect die opening = (Outside radius × 0.7071) × 4.**
If we apply this to the example in figure 5, we would have:

\[(0.150 \times 0.7071) \times 4 = 0.424 \text{ in.}\]

To deal with springback, we increase the multiplier a little (and, hence, the die opening). In material thicknesses less than 0.125 in., a realistic working multiplier is 4.85, as in:

\[(\text{outside radius} \times 0.7071) \times 4.85.\]

In material between 0.125 and 0.250 in., the multiplier is 5.85 in., because of tonnage requirements and the increased springback of thicker materials.

There Is No Perfect Die, So Now What?

It is rare that an optimal die width is actually available. For instance, if you find that the perfect die opening is 0.424 in., the closest available die may be either one with a die opening of 0.395 in. or another with an opening of 0.472 in. For the final selection, the closer we can get to perfect the better, but this may not always be practical. In this example, 0.395 in. is closer, but it may be too small from a tonnage perspective. You always need to check and make sure that you do not over-tonnage your machine or your tooling. The formula for tonnage per inch is:

\[\text{Tonnage per inch} = \frac{(575 \times (\text{material thickness})^2}{\text{die width} / 12) \times \text{material factor} \times \text{method factor}}\]

**Material factors**

- 60,000 PSI tensile CRS, 1.0
- Copper, 0.5
- H series aluminum, 0.5
- T6 aluminum, 1.28
- 304 stainless, 1.4

**Method factors**

- Air forming, 1.0
- Bottom bending, 5.0+
- Coining, 10+

You then multiply this value by the length of the bend to find the total tonnage required for the bend. If the tonnage is acceptable, you can use the closest die opening. If it’s not acceptable, you must use a larger die opening.

This die-selection method is valid for all three methods of bending–coining, bottoming, and air forming–and will, for the most part, keep the relationships between the die width and outside bend radius regardless of whether the material is thick with a small radius, thin material with a large radius, or if there is a one-to-one relationship between material thickness and the radius.

In the past, the general rule of thumb has been that eight times the material thickness is a perfect die opening—which it is, but only in a one-to-one relationship between the inside bend radius and the material thickness; for instance, a 0.062-in. radius in 0.062-in.-thick material. But what if the material is 0.036 in. with a 3-in. radius? Will eight times the material work? No, eight times the material thickness will not produce a valid or acceptable die opening for this large of a radius.
Benefits of a Perfect Die Opening

There are several other reasons for finding the perfect die opening. First, it keeps the material and inside radius relationships the same and, by doing so, the technician will be able to make adjustments to the controller in a more consistent manner over time.

For example, if you are working with a die opening that’s as close to perfect as possible, you may find it consistently takes, hypothetically speaking, 0.005 in. to achieve one degree of angular change. Using a larger-than-necessary die, it may take 0.020-in. to achieve the same angular change.

This means that the punch will need to go farther down into the V to effect a degree of angular change.

An optimal working die opening also helps when you use shims or an angular compensation device to create an angular change. When you use an optimal working die opening, it will require only a small amount of movement to bring about a degree of angular change, and you’ll only need light crowning.

But, if it takes more movement to create angular change (for example, 0.020 in.), then you will need more shimming. If you need to shim the larger die or use an angle compensation device, it may not work so well. You may exceed the ability of the compensation device or, worse yet, create the need to place so many shims under the die that you can end up putting a reverse crown into the die itself, figure 7.

On the other hand, you may want to use a larger die opening from time to time just to stabilize a specific forming project. For instance, you may want to account for variations in material properties. If the material thickness varies plus or minus 0.010-in. and it takes 0.020-in. of ram movement to cause a degree of angular change, then the angular error will be only one-half of a degree. However, the technician cannot make the decision to use a larger die on his or her own. This is because the part would need to be designed for the larger radius produced by air forming in a larger opening.

There are many more examples of why using a die opening as close to perfect as possible is worth the time and effort. Nonetheless, by staying as close as possible to perfect, your consistency will improve, your setup and run times will get shorter, and the risks of running into problems will be greatly reduced, figure 8.

Die Angle

Forming tools come in a standard variety of die angles. Measured as an included angle, the punch is read as a complementary angle. The die angles are changed to help compensate for springback. It is a standard practice that as the die width increases the die angle decreases. From die widths of .157 to .472, the die angle is 90°, for die widths from .472 to .984, its 88°; from .984 to 1.500 it is 85°, and from 1.500 and up it is 78°; etc.

Die Radius

On the top outside edges of the press brake die there is a radius which ranges from .015 to about .125. As you might imagine, the sharper this radius is, the more likely it is that die marks will be left behind. Die marks or gouging is the tendency to scrape the material’s surface as it is dragged down into the die space.

There is good reason for both types of edges, whether sharp or a radius. For example, a die with a sharp corner will
allow you to “catch” the edge of the workpiece when the flange is very small and the radius'ed edge would allow the workpiece to slip into the die space. On the other hand, a radius die edge will allow the workpiece to slide more freely through the forming process. This also helps to maintain a consistent bend angle. There is a point where the die radius gets large enough that die marks will no longer occur. That radius is expressed as follows:

**Mark free die radius = 1.7 * the material thickness.**

The Pinch Point

The pinch point is where the punch tip comes into contact with the material, clamping the material down against the top of the die. Pinch points are commonly used in conjunction with other controller applications such as retracting the backgauge. This allows the backgauge and stops to move out of the way before they interfere with a previous bend, etc.

Visualize the pinch point as the die width divided by two (one half of the die width is equal to the depth), add the material thickness, and then subtract .010 to ensure a tight grip of the material. The formula for finding the pinch point is:

**Pinch point = (die width / 2) + Mt – .010**

Hole distortion also has a direct relevance on the selection of die width. As a hole or feature moves closer to the bend line the greater the distortion will become. Any feature that lies inside the area described by the outside setback (OSSB) will distort, with or without a wrap, figure 9.

The least amount of distortion occurs when an optimum die width is used. The fact that as the die width increases, the inside radius will increase; as the inside radius increases, the outside setback will increase. The greater the outside setback, the greater the area of distortion.

**Press Brake Dies**

Press Brake Dies

*Courtesy of Asma LLC*