Offset tooling

There are two sub-groups of offset tooling:

“Up-spring” or “Up-sweep” and the “Horizontal”. The upspring tool is used to place two bends close together, bends that are normally too close for conventional forming methods.

The horizontal offset shown in figure 1 is designed to offset the material, one material thickness.

To determine the appropriate offset tool for a given job, subtract the material thickness from the outside dimension. The resulting number is the measured depth of the offset as measured from face to face.

\[
\text{156 Offset - .059 Material thickness = .097 Required offset depth}
\]

In the example just given, the required offset dimension of .097 is the “A” dimension shown in figure 2, from the outside surface to the inside surface of the part. The .097-inches in this example is the depth of the off-set tool as measured between the two faces of the die.

On a standard die opening the measurement is taken between the two top corners of the die. Figure 3 is incorrect in the way a standard die is checked, and figure 4 is the correct way to measure the tool offset.

Determining the radius

For an offset tool is done in the same manner as a bottomed or coined bend in standard tooling; radius of the tool is stamped into the material; but, this is only when the tool is used to its full extent.

Before bottom is reached there are plenty of air formed angles available. If you are not bottoming out the tool set, the radius is found using the 20% rule. This is only true for the up-spring tool; the horizontal tool is always an air form.

Again, the up-spring style of tool is a bottoming tool, designed to stamp the angle and radius. Usually but not always 90° the tool was meant to stamp the angle and the dimension of the offset into the material.

For many years these tools only came in 90° unless custom made. But overtime started to make the transition to air forming tools and are now available with die angles appropriate for air forming.

There are variations in tool angle to allow for springback when forming.

Measuring the tool dimension for a horizontal offset is made the same way, from die face to die face, figure 5.
Finding the Bend Deduction

While it is possible to calculate your way through an offset bend: flange dimensions minus bed deductions; and you lose a material thickness. But because the part is constrained on the center dimension, calculating your way through can be difficult. The material is forced to dimension, .097-inches in our example, the material between the bends cannot elongate normally. That means the elongation needs to go elsewhere, and that elsewhere are the two flanges exiting the tool set.

It quickly becomes clear that a test bend is the best and quickest option for determining the flat dimension of the blank.

One other point of note, a 90° offset tool can cause the part to take on a “Z” shape, the over-bending that occurs just before being forced back to 90° by the bottoming process.

The Horizontal Offset

The horizontal tool is not a bottoming tool, it is used primarily for the “stepping” of a material one material thickness. Here the angle and radius are normally unimportant and clearance between the tool faces is the main overriding factor, figure 6.

The maximum bend angle that can be safely achieved with the horizontal tool is approximately 70°. Above 70° the horizontal offset tool will begin to act like a shear, cutting the material rather than bending it.

Another important factor to consider is side thrust. Side thrust occurs in both types of offset, it is most pronounced in the upspring style, figure 7. In a standard die set the thrust is applied equally to all surfaces causing the tooling to “sit” still during the forming process.

But the offset tool wants to “push out” to the side in both directions and there will be times this side thrust will get out of hand which could injure you, damage the tooling and/or the product. Should side thrust become a problem, a thrust plate can be attached to both ends of the tooling to reduce the side thrust, figure 8.

Air forming requires much less tonnage, but the final bend will take on more of a “Z” shape rather than a true 90° offset. Again, this makes the calculating of the flat blank very difficult without test bending a sample piece first, due to forming method and tool angle variations.

Offset Installation

In looking closely at the tool set, there is no front or back to the tool; it can be installed facing either direction. It would be a natural instinct to install the tooling so that the workpiece would arc upward during the forming operation, in the same way the part would move using a standard V-die set.

Please pay close attention to the way in which this type of tool is installed into the press brake. The best practice is to install this type of tooling is to form down in the front.

The reason is to keep the backstops out of the die space, figure 9.
It may be true that the majority of your offset bends are placed far enough that entering the die space is not a problem.

But it only takes placing the stop in the work space once to ruin the part, the tool and the backstops.

Again, best practice is to face the tool so that the part moves down in front of the press brake; figures 10 and 11 illustrate this concept.

**Variable Offset Bend Angles (offset tooling)**

Many times a blueprint will call for a bend angle other than 90°. For example, a 45° bend angle and a .250 called inside dimension is an illustration in figure 12.

Now there are two different ways that a bend of this nature can be made. First, with a standard punch and die set, bending the workpiece to the desired angle and dimension using two separate hits. The second is by using an up-spring off-set tool. In using the up-spring tool the bend angle is controlled through the depth of penetration and the size of the die set.

Penetrate a .500-inch offset die 50% and you can create a 45° bend at a .250-inches.

Figure 13 shows this principal at work, regardless of the angle or dimension. All of these offset dimensions could be easily produced in the same .500 offset tool.

As you might have guessed by now, this process can be expressed in a mathematical formula; but again we need to take a few liberties with the inputs.

Nonetheless the following is a list of the variables and the formulas do work, down to about 30° of bend angle. With bend angles less than 30° the tool width/dimension/angle relationships become realistically unworkable.

The formulas and variables given here should give you a valid dimensional value for the ram setting or depth of penetration. These are measured up, from dead bottom of stroke where the die faces are touching without the material being present.

**The Variables**

\[ A \] = The required inside dimension  
\[ BD \] = The degree of bend angle  
\[ B \] = The actual measured tool dimension  
\[ Rp \] = The radius of the punch  
\[ Dp \] = Developed penetration  
\[ Di \] = The difference + / _  
\[ Od \] = Optimum die width

**The Formulas**

Optimum tool dimension = \( \left( \frac{90}{BD} \right) \times A \)  
Developed penetration = \( \left( \frac{B \times \text{sine} \ BD}{2} \right) + Mt \)  
Difference +/- = \( \frac{B - Od}{2} \)
Actual machine input depth = Dp + Di + Rp – 0.03

These formulas are another example of the applications of right angle trigonometry in precision sheet metal forming.

If a 30° offset bend with a dimension of 0.165 on the “c” or Opposite side of the triangle is achieved, both “a” (the Hypotenuse) and “b” (the Adjacent) side dimensions will be achieved.

Follow the progression of the angle and the dimension as it reaches 0.500 at 90°. As the material is being pulled down into the die space, side “a” will increase in direct proportion to the depth of penetration, side “c”. The bend angle also changes in direct relation to the change in penetration.

If the calculated die width is unavailable, you may have to give a little on either the bend angle or the dimension to produce a good part. That is the reason for the “actual measured tool dimension” in our formulas. If possible it is better to cheat the bend angle instead of the dimension because it is much harder to measure.

**Variable Offsets** *(standard press brake tooling)*

The second way that we can produce a variable angle offset is by using a standard punch and die set; it is accomplished by stepping the bend out toward you, figure 14.

First find the inside bend line:

Step #1 = side “a” + flange dimension – bend deduction

Second finding the bend line of the second bend.

Step #2 = flange dimension – bend deduction / 2

**offset**

Offset Tooling Application
Courtesy of Asma LLC