

Why should you care about inside bend radii?

By Steve D. Benson, Contributing Writer
January 29, 2004

Operators, designers, and engineers, why should you care about the inside bend radius if the customer doesn't? Because, ultimately, just how easy or difficult it is to produce a part depends on decisions made during the design stage. Misunderstanding terminology, process capabilities, or production methods can lead to mistakes that can make production more difficult. The most common mistake is incorrectly calculating and achieving the correct minimum inside bend radius.

A minimum bend radius is a function of the material and has little or nothing to do with the press brake punch tip. A minimum bend radius for one material thickness is not the same for another material thickness. In cold-rolled mild steel, the minimum bend radius is 63 percent of the material thickness. For example, a piece of 12-gauge (0.104-in., 2.64-mm) material turns sharp at 0.065 in. (1.66 mm), making any punch radius less than that value, by definition, a sharp bend, such as 0.062 in. (1/16 in., 1.58 mm), 0.032 in. (1/32 in., 0.82 mm), or 0.015 in. (1/64 in., 0.38 mm). Radius bends occur between the 63 percent threshold and a bend radius no greater than 10 times that of the material thickness.

Can a press brake operator achieve the desired bend radius on-the-fly? Well, maybe, maybe not. I suppose that depends on the operator's skill level.

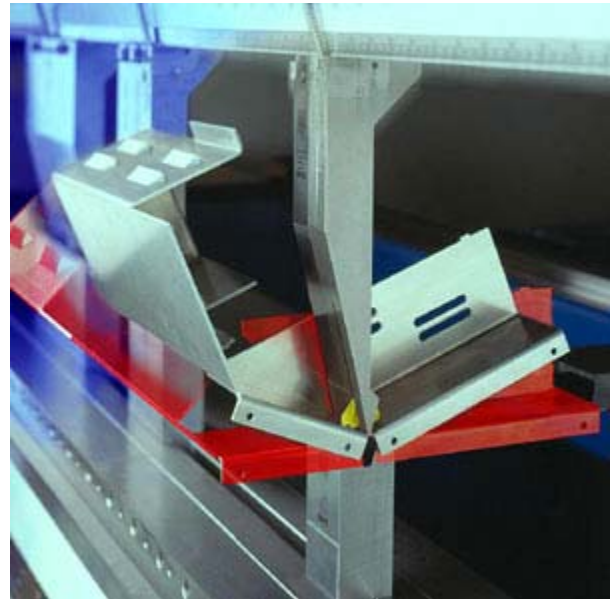


Photo courtesy of TRUMPF Inc.

Bending Basics

Before we continue, a discussion of bending principles is in order. First, we must establish a common language about forming methods, bend types, and their subsequent mathematical effects.

During any forming process, material elongation occurs. Elongation is a bit of a misnomer, because the material actually changes shape, which may or may not include elongation. The part of the material that normally would square into the corner must go somewhere else as a result of the radius effect. This elongation commonly is called a bend deduction (BD) or K factor—mathematically equivalent terms.

How many times have you referred to a bend deduction or K factor chart to find a bend deduction value only to discover later that it wasn't correct for the finished part? What happened? Was the chart wrong? Was the problem operator error? Was engineering asking for something that couldn't be done in practice? Any or all of these factors could have caused the error.

Chart Confusion

Let's look at examples from five BD charts. Sixteen-gauge cold-rolled steel (0.059 in., 1.50 mm) with a 1/32-in. (0.032-in., 0.83- mm) inside bend radius has the following values:

1/32-in. radius in 0.060-in. material

Chart #1 N/A
Chart #2 0.063
Chart #3 0.083
Chart #4 0.097
Chart #5 0.102

1/16-in. radius in 0.060-in. material

Chart #1 0.106
Chart #2 0.108
Chart #3 0.110
Chart #4 0.132
Chart #5 0.136

The difference between the high and the low values for one 1/32-in. single bend radius is 0.039 in. Multiplying 1/32 times three bends gives 0.117 in.

The difference between the high and low values for one 1/16-in. bend radius is 0.030—0.090 for three bends.

Are the charts wrong? No. Every value from every chart is valid for an achieved radius under a given set of circumstances. The problem lies in the fact that the radius called for rarely, if ever, matches the achieved radius. Most test samples confirm the discrepancy.

Correct Bend Deduction

Empirical-formula charts do not take into account all the variables. They just assume a 1/32-in. (0.83-mm) radius in 14-ga. material, rather than the natural minimum radius of the material of 0.046 in. (1.18 mm), 63 percent of the material thickness. This changes the value of the calculations. If the 0.032-in. (0.83-mm) value is used in the calculations, the resulting BD will be incorrect, because 0.046 in. (1.18 mm)—the achievable radius—produces an incorrect bend deduction value, which, in turn, makes the corresponding flat blank layout incorrect.

BD or K factor is developed in the following manner:

$$\begin{aligned}BD &= (2 \times \text{OSSB}) - \text{BA} \\ \text{OSSB} &= [\text{Tangent } (\text{D} / 2)] \times (\text{Rp} + \text{Mt}) \\ \text{BA} &= [(0.017453 \times \text{Rp}) + (0.0078 \times \text{Mt})] \times \text{D}\end{aligned}$$

Where:

BD = Bend deduction
OSSB = Outside setback
BA = Bend Allowance
Rp = Punch tip radius (can be interchanged with inside radius)
Mt = Material thickness
D = Degree of bend angle

These formulas are valid, but they do not take into account whether the bend to be produced is a sharp or radius bend.

Options

Picture a sheet of 0.100-in.-thick material (2.54 mm). By multiplying the material thickness by 0.63 (63 percent), the natural minimum radius can be determined—in this case, 0.063 in. (1.57 mm). Within the 63 percent range of this particular material, press brake tool radii of 0.062 in. (1.57 mm), 0.032 in. (0.83 mm), and 0.015 in. (0.41 mm) are valid sharp-bend tools.

No matter how far into the sharp range you go, it is the natural radius that will be produced, not the 1/16-in., 1/32-in., or 1/64-in. punch tip radius. The only thing that will change throughout the range is the severity of the ditch at the bottom of the bend.

Any of the punches listed could produce the part. Why? Because, for example, if the part required a 1/16-in. radius in 0.100-in.-thick material, the bend, by definition, would be a sharp bend. An experienced operator or setup person could use any tool radius equal to or less than the 63 percent sharp-bend threshold to achieve the desired bend. This has the effect of tripling the tooling options. As an operator, you might find that the perfect gooseneck punch is available only with a 1/32-in.-radius tip. Not a problem, since a sharp bend is a sharp bend, regardless of the tip used to achieve it.

Caveat

Of course, as everyone knows, there is no free lunch. The sharper the punch radius is in relationship to the material thickness, the greater the effect on angular error, and concurrently, dimensional error. In other words, any variances in material thickness, hardness, and grain direction exponentially increase with each decrease in punch radius.

By understanding and addressing what is happening at the point of bending, you can avoid many forming problems, including those associated with flat blank layouts (CAD or not) and the effects of a sharp bend on the finished part. By doing so, you'll have a healthier bottom line.



Steve D. Benson is the president and owner of Advanced Sheet Metal Applications© (ASMA), 398 Rose St. N.E., Salem, OR 97301-4468, 503-399-7514, steve@asmachronicle.com. ASMA's home page can be found at www.asmachronicle.com. He also is a member of the Fabricators & Manufacturers Association Intl.®, (FMA), Sheet Metal Council. Books and software authored by Benson also are available from the [FMA Bookstore](#).