Fundamentals of metal-forming

Applications engineer says having a precision flat pattern is 'imperative' when using a press brake

BY RICK WEBER

HOW important is having a precision flat pattern when using a press brake?

Mike Bates, applications engineer for SigmaTEK Systems, would go as far as calling it "imperative."

In a SigmaTEK webinar, "Fundamentals of Metal-Forming," he said having a perfect flat pattern for any job is one of the main keys to precision bending.

"I've seen entire days of production lost in the shop—with just trying to figure out, with the tools they had, with the part they needed to bend, with the tolerances they had—with going in and trying to fudge those numbers so they could come out with a functional part," Bates said. "When it comes down to it, the flat pattern has to be right. For every thousands of an inch you're over, you're adding issues to the brake. They're going to have to go in and adjust dies, move bends, change angles. You can lose a lot of time very quickly by not having a correct flat pattern. You could end up losing material, all because the flat pattern was not right to start with.

"If the flat is not correct, the operator is in store for a very rough day. They will have to toy with the angles and lengths to try to keep the part in tolerance. This also often occurs when you do not have the exact tooling setup for the part."

The mathematics of metal-forming requires an understanding of the K-Factor.

"When you're performing a bend, there are changes that happen," he said. "Along the bottom of the part, you have overall expansion where it's stretched around that punch. There are compression forces on the top side of it from where the initial contact is happening. You have compression on the top, expansion on the bottom. The key factor is that neutral line.

"The K-Factor allows you to find that spot so you can determine what your bend allowance or reduction is and how much material you need to cover this bend. The K-Factor is a ratio of location of the neutral line to the material thickness as defined by t/T where "t" equals the location of the neutral line and "T" equals material thickness. The K-Factor formulation does not take the forming stresses into account but is simply a geometric calculation of the location of the neutral line after the forces are applied and is thus the roll-up of all the unknown (error) factors for a given setup. The K-Factor depends on many factors, including the material, the type of bending operation—coining, bottoming, air-bending, etcetera—and the tools, and is typically between 0.3 to 0.5.

"That is the roughest thing to figure out. Once you have your K-Factor and tool, you can kind of predict what's going to happen. Everything else becomes a lot more static."

Bates said bend allowance is "the length of the arc through the bend area at the neutral axis.

"That's going to be the length of that material—how much material I need to allow," he explained. "So if I have an eight-inch bend with an eight-inch flange, how much additional do I need to add to perform the bend at ¼ inch?"

The bend deduction is "the amount of material you will have to remove from the total length of your flanges in order to arrive at the flat pattern."

Saving money

The setup speed also is critical.

He provided an example:

"You are running one press brake, using three shifts, with a burden rate of $40 per hour. A normal setup takes 15 minutes. If you are doing eight setups per shift per day, that's 12 hours lost per day at $40 per hour, for a total of $240 per day. At five days a week, that's $1200 per week, and $62,400 per year wasted on setups.

"That's not making you any money," he said. "That's money lost. If you could drop the setup time to 7½ minutes, that's an overall saving per year of over $31,200 per machine."

Gentlemen, this is a press brake

Bates broke down the discussion to the very basics. He defined a press brake as a large industrial machine that uses large amounts of force to bend materials—aluminum, stainless steel, sheet metal of any kind. It utilizes changeable tooling in a punch-and-die system, with machine sizes ranging from 36 inches to hundreds of feet and forces ranging from two tons up into the thousands.

Machine components include: a machine frame that will be permanent and bulky; an upper beam that is actually moving up and down, performing functions; an upper and lower tool holder; an upper and lower tool; lower beam; left and right fingers that allow parts to be supported; and, at the top, hydraulic cylinders that drive the press itself.

He said most of the machines these days have four or more axes.

There are three ways to bend:

- *Air bending.* "This is the most modern and widely used in manufacturing processes. It uses the three contact points to determine the size of the bend. The rule of thumb is to use the bottom die that's five to eight times the material thickness. Best-case scenario, for a one-quarter inch part, is you'd want to use a 1½ -inch bottom to fit this."
- *Bottom bending.* "It forces the material all the way into the die. This isn't done as widely. There are some parts that just require it, but it's rougher on tooling and an outdated way of forming. It's bad for the machine, bad for tooling, and it's not supported by most offline programming software."
- *Coining.* "Bottom bending and coining often are considered to be the same process, but they are not. Unlike bottoming, coining actually penetrates and thins the material. It uses five to ten times the amount of force to perform the same bend as air bending. So you're actually stamping that radius into the material. This is a pretty rudimentary style of forming. Coining is the oldest method and, for the most part, is no longer practiced because of the extreme tonnages it requires. It's extremely tough on the tool. I've heard it compared to the difference between just opening the door and walking into a room and walking through the wall. It's not something you want to put your machine through on daily basis. And it's not supported by most offline programming software."

The types of bending operations:

- *Bump bending.* "This is used to break up a large radius bend into smaller forms. So if you only have tooling to perform an eight-inch bend, you could bump that over a four-inch bend and hit it roughly five times, and that would allow that to look like an eight-inch bend.
- *Offset Bending.* "It forms two very close forms in one pass of the machine. It's also known as joggle bending. It's pretty commonly used in the industry."
- *Pre bending.* "The first pass of a multi-process part, normally a 30-degree angle."
- *Hemming.* "When a material is folded over onto itself."
- *Flattening.* "Large dies used to straighten materials."

He said crowning is a process used to compensate for a lack of bend or "displacement" in the center of a part.

"With force being focused from the outside of the machine, the force may lower as you get closer to the center of the machine," he said. "By adding a slight hump or raise in the lower tool, this helps to equalize the load on the part, holding the bend angle across the rail. In the past, this has been done by shimming the center with strips of paper, metal, or plastic. Most of the newer machines today come with crowning built in." ■