9 strategies for preventing part collisions on the press brake

A good plan and the right tooling can help end the struggle for metal manufacturers

By David Bishop | February 25, 2023 | Article | Bending/Folding | 0 Comments

Parts can collide with nearly every surface of the press brake. Having a plan and using the right tools can help prevent those collisions and make life easier in the brake department.

On the first day they use a new press brake, operators commit themselves to a battle that will endure for as many years as that press brake remains in service. Unfortunately, many will be unaware they have committed themselves to this struggle and will find out the hard way that they have not properly prepared for it.

This endless battle is against part collisions—formed parts colliding with virtually every surface on the machine, including the ram (upper beam), the bed (lower beam), the tooling, and potentially the backgauge.

Some of the most common part collisions occur during the bending of very deep U-shaped parts, as well as parts with long return flanges, that end up hitting the ram (see **Figure 1**).

The most common type of collision occurs when the part collides with the punch. This often happens with parts that have a long return flange or a complicated bend sequence, or with parts like door frames that have return flanges (see **Figure 2**) projected toward both the front and rear sides of the punch. **Figure 3** shows another typical example: a deep box colliding with the clamping system.

If you have spent a significant amount of time running a press brake, you've probably experienced all these collisions and countless more. Fortunately, there are strategies you can follow to maximize bending freedom while minimizing collisions that damage parts and, not least, the press brake itself.

1. Pay Attention to the Open Height

When purchasing a new press brake, always get a machine with as much open height as possible between the bottom of the ram and the top of the bed. The size of this work envelope will determine how much you can put into it, and it typically can't be changed (see **Figure 4**).

2. Use the Tallest Tools Possible

Press brakes with open heights of 20 in. (508 mm) have become commonplace, while machines with 25.590 in. (650 mm) of open height and even more are offered by multiple press brake manufacturers. These machines allow for the use of extremely tall punches, dies, punch extenders, and die-riser combinations that provide tremendous versatility when bending deep U-shaped parts, four-sided parts, and thick plate (see **Figure 5**).

Always invest in the tallest tooling combination that your machine's open height will allow. The taller a punch or die is, the more versatile it is. Nothing is worse than purchasing a short tool only to find out months or even years later that it won't form a part that you need to produce. In this case, you might need to purchase a taller version of the same punch, and you'll have wasted the money that you spent on the shorter punch.

3. Consider Low-profile Die Holders

If your press brake has a limited open height, it might not be able to form some of your parts because the required tooling is simply too tall. If your machine has a manual die holder, try changing to a low-profile die holder for additional open height.

If the machine has a manual crowning system, you can also replace it temporarily with a low-profile die holder. Unfortunately, in doing so, you will lose the ability to compensate for deflection that naturally occurs in the press brake during bending. This may make it necessary to manually shim the dies to achieve accurate bends.

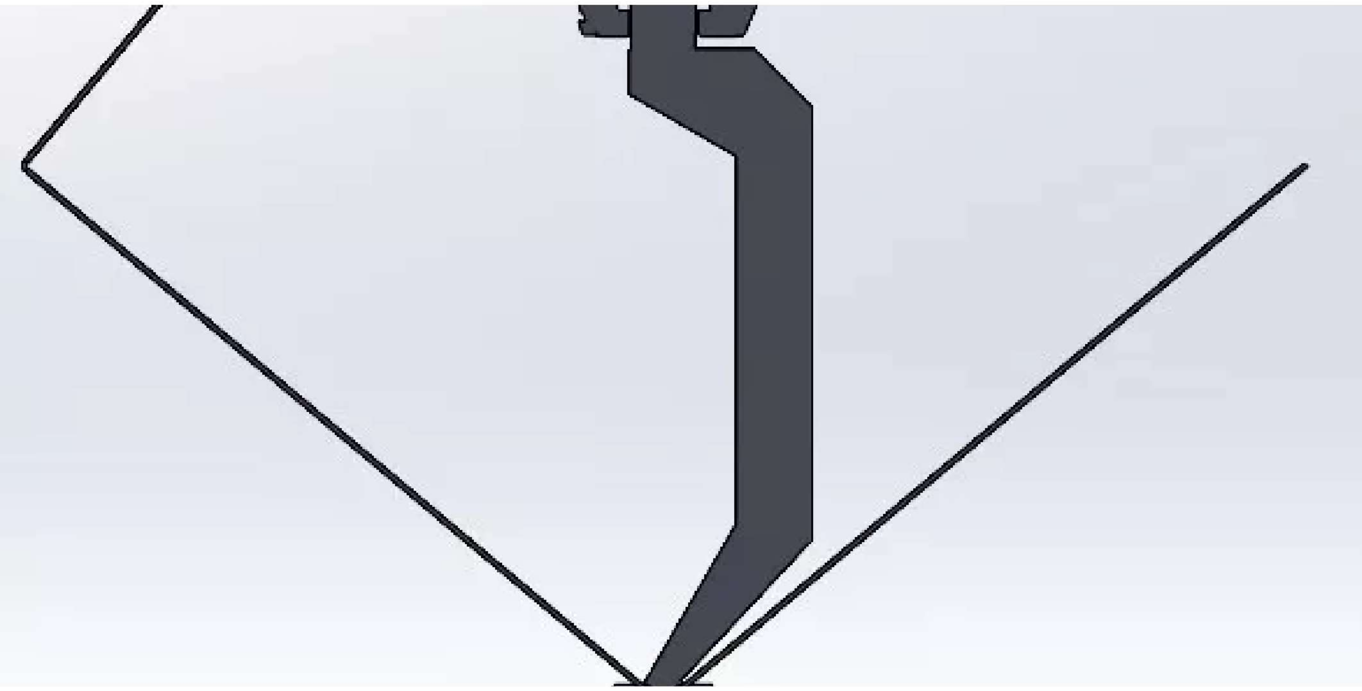


FIGURE 1. A deep U-shaped part collides with the ram.

reload your crowning system for the next job.

4. Allow for Plenty of Working Space

Be sure to leave plenty of working space between the punches and the dies. This should allow for easy part manipulation during the bending sequence and easy part removal when finished.

When possible, maintain at least 4 in. (101 mm) of working space between the punch and die when bending small to mid-sized parts made of light-gauge materials, and 6 in. (152 mm) or more when bending boxes and four-sided parts. Of course, parts with complicated bend sequences and those made of thick plate may require even more.

5. Have Enough Stroke Length

Stroke length should be equal to at least 50% of the machine's open-height specification. Of course, the clamping system, punches, dies, and crowning system or die holder (whichever applies) will consume a large percentage of the open height when installed.

When bending materials between 22 and 10 ga., have enough stroke length for the punch to reach the top of the die holder when all of the components except the dies are installed. This ensures sufficient stroke length to reach the bottom of the V opening on any die. It's also good practice not to run the drive components out to the end of the stroke.

Note that this rule isn't practical when bending thick plate over large V-die openings. But again, when working with material between 22 and 10 ga., it can be a good practice to follow.

6. Consider Punch Extenders

If you are bending four-sided parts like boxes and pans, be very careful when selecting the punch. You want to prevent the sides of the parts from sweeping up toward the ram or the clamping system when you bend the third and fourth side (see **Figure 6**).

Because of the extreme tooling height requirements to form these parts, you might find you don't have a punch tall enough to bend them. In these cases, you might need a special, extremely tall punch made specifically for the job. Another, usually less expensive, option is to use a punch extender combined with a standard punch. Punch extenders can have more value as well, since they can be used to form other parts in the future.

To determine the required height of the punch (or punch and punch extender combination) to bend a four-sided part, multiply the depth of the part by 1.7. Note that this rule does not apply to press brakes equipped with a heavy-duty clamping system. Measured from front to back, such systems are much wider than standard clamping systems and, therefore, create a much wider interference zone. If you have a heavy-duty clamping system and foresee a potential collision, contact your press brake tooling supplier to find out what solutions are available.

7. Consider Taller Dies and Die Risers

Before bending, always consider the location of the bends on your parts. For example, do they have any tabs or offset bends that are far into the sheet from the edges, thus creating a long down-flange? If so, you will need a tall die to prevent the down-flange from colliding with the die holder or the bed of the machine (see **Figure 7**).

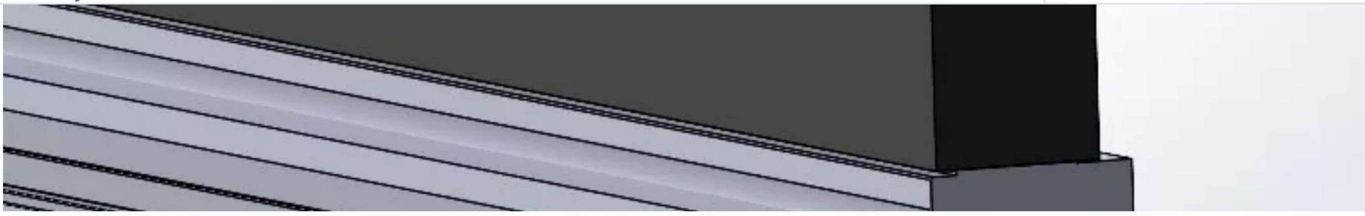


FIGURE 2. A return flange collides with the relief area on a gooseneck punch.

Nearly all press brake tooling manufacturers offer taller dies, die risers, or standard die holders that will enable you to raise the height of your dies farther above the bed of the machine. Before purchasing these items, however, verify that the backgauge on your press brake can be elevated to the higher location of the top of the die.

8. Simulate Your Bends

If you program your parts offline before sending the job to the press brake, you have a very powerful tool at your disposal. Bend simulations expose parts that will cause collisions and offer alternatives before you experience collisions at the press brake.

Of course, you may be able to perform the same simulations on your press brake's controller. Doing so, though, will tie up the machine when it could be bending parts.

9. Access Tooling Data Digitally

One factor contributing to part collisions is the press brake tooling manufacturers' primary focus on critical tooling dimensions that affect bending accuracy. These include the working height, the tip radii, and the tip angle on the punches, along with the working height, the width of the V opening, the V-opening angle, and the V-opening shoulder radii on the dies. Certain areas of the tool, particularly those that do not engage the material during bending, were (and some still are) machined to more open tolerances.

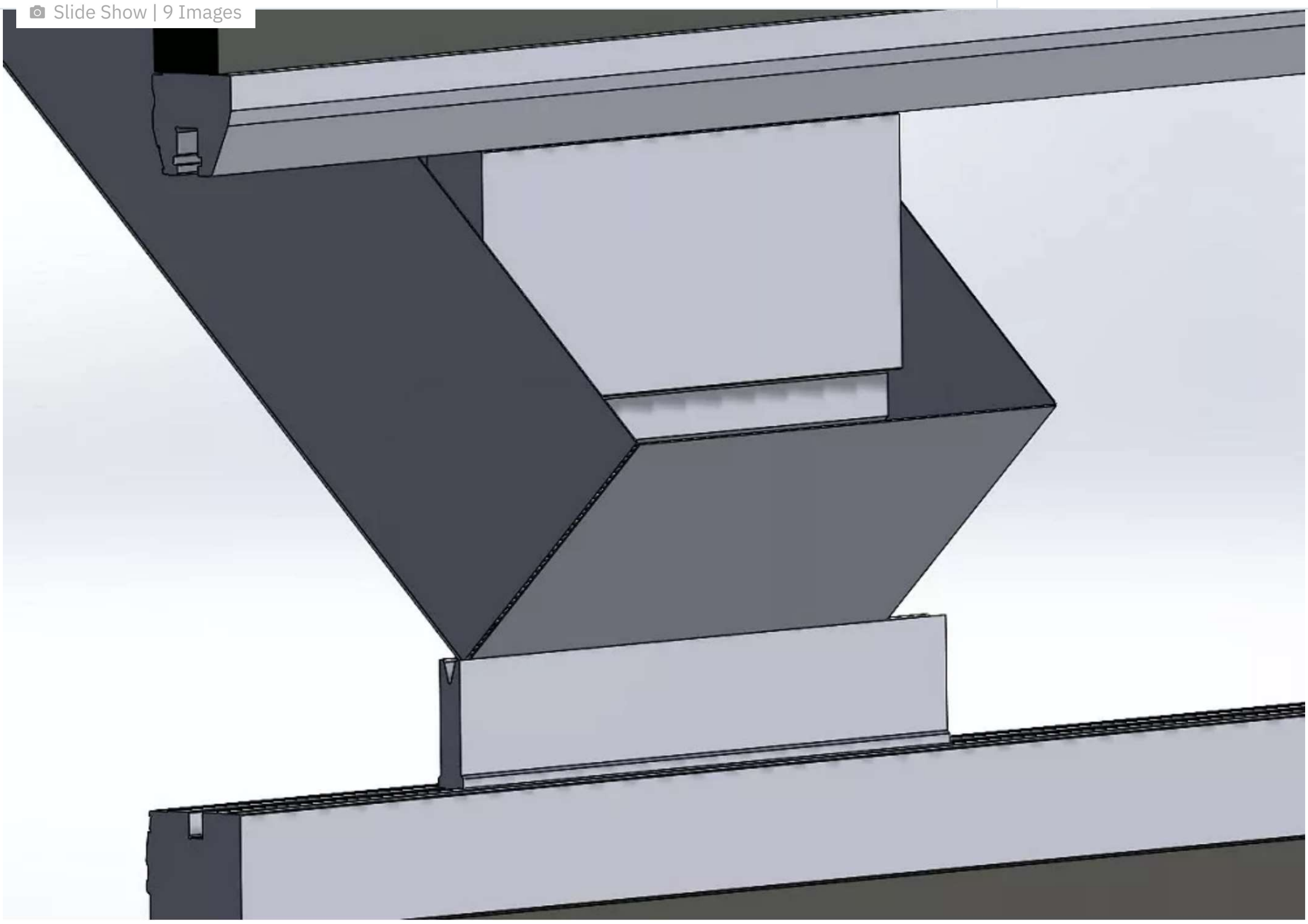
Of course, additional machining adds cost, and end users often are satisfied to make small angle corrections by hand when light-gauge parts experience a minor collision with tooling. However, with the ever-increasing demand for higher-quality parts and greater productivity, many press brake tooling manufacturers have had to tighten up the tolerances in these areas to eliminate the mismatches from one tool to another.

More recently, a new technology referred to as smart tooling has emerged in which each tool and the container it is shipped in are equipped with a DM code. This makes it possible to scan individual tools or all tools at the same time using a mobile phone (see **Figure 8**). After scanning, you have immediate access to each tool's exact digital twin and its complete specifications.

The data files can be uploaded to the cloud and then to your press brake control. Toolholders equipped with a Bluetooth module also can transfer the data from the tools directly to your press brake control. This technology helps improve bending simulation, as all collision avoidance calculations are based on accurate tool data, not assumptions. It also has numerous application possibilities for tooling inventory control and management.

Smart tooling can be an important piece of the collision-avoidance puzzle. Combine it with bend simulation and the right tooling and accessories, and your struggle with part collisions could finally come to an end.

Slide Show | 9 Images



YOU MAY ALSO LIKE

The rules of press brake tool selection

The (really) basic basics of press brake tooling