Press brake punches

The Styles of Press Brake Punches

Figure 1 shows the double shoulder American style punch, the single shoulder Precision Ground European style and the New Standard.

The greatest difference between these styles of tool mounting is the way the power flows from the ram and into the work. As you can clearly see for the American Planed style tool, the power flows through two shoulders; for European Precision Ground, the power comes through a single shoulder; and the New Standard, the power flows directly through the head of the tool.

Of the three the planed tooling is the most difficult to work on a precision level because no tools, cut from a larger tool are alike. Making centering and mixing and matching tools almost impossible. Both the European and New Standard are precision ground and very precise, making setups easier and quicker.

Even if you are using what is referred to as a Precision Planed tool, you will still find yourself at the mercy of the way the tool is mounted into the press.

The Types of Punches

There are three different types of standard press brake punches: the straight punch, the gooseneck punch and the acute punch acute. These are by no means the only kinds of tooling available for press brakes, but these are the three basic ones, figure 2.

The first, and arguably the most important factor in the selection process is the angle of the punch. The angle of the punch is called an included angle. What this means is that a 90° punch will measure with a protractor as 90° and a 88° punch will measure with a protractor at 92°. Knowing the punch angle will allow you to select a tool that allows for springback and/or other factors in the selection process.

In order to compensate for springback we must allow an angular clearance at least equal to the amount of springback you are expecting to encounter. A piece of soft aluminum with a known springback of 2°, would require an angular punch and die clearance of 2°, one degree on each side.

This type of tooling combination could not “coin” a bend that wasn’t a sharp bend. Figure 3 demonstrates this idea of clearance. The punch
and die set would allow the workpiece to be brought up to 92° and then released to “springback” to 90° using air forming. It would also allow the workpiece to be brought up to 92° and then forced back to 90° as in bottom bending.

The included angle of the punch **MUST** be equal to or greater than the included die angle. In figure 3, you see an example of how mismatched tooling could damage the tooling as well as ruin the workpiece by back-bending at the point that the pinch is occurring.

Consider the radius of the punch and all of its implications to the forming process. A punch radius can be either metric or decimal, depending on the tooling manufacturer.

The difference may at first glance seem insignificant: a 1/32 is equal to .032 and a metric .8 equals .030. Using a piece of material measuring .036 in combination with the above mentioned radii, a difference of only .001 is found in the calculated bend deduction.

But, with a punch radius of 1/8 (.125 inch) and a 3.0 metric (.118 inch); the difference is now .003 inches per bend. After several bends this difference can become substantial. For example, after six bends the error could be .018 inches.

It’s true that in the general scheme of things this is a very nominal amount of error, but, it is error that you will need to deal with at some point.

A **sharp** bend relationship can be used to your advantage here, but remember, a sharp bend is a function of the material and not that of the punch.

A punch radius of less than 63% of the material thickness should be considered “**sharp**” by definition. For example, with a material thickness of .100-inch, the bend would turn sharp at .063 inches.

Any punch radius equal to or less than .63 (63%) would be sharp. A punch radius of .062, .032, .015 will produce the same bend radius and bend deduction in an air form, figure 4.

If the engineer were to call for a .032-inch bend radius in .100-inch thick material, all of the above listed **punch radius** would be valid. The closer your punch tip’s radius actually comes to the 63% threshold, the less that you will fight bend angle issues caused by variances in material thickness or grain direction, hardness, etc.

On the other hand, if you need to use an extra small die width, the pinching that can occur as a result the small “ditch” caused by a sharp tool may allow you to achieve a bend angle in a bottom bend. Increasingly, the smaller the radius becomes in relation to material thickness, the greater the pinching becomes.

Either way, the bend deduction will remain the same if it is calculated at 63% and air formed.

**Punch Mounting**

There are three styles of punches and three different punch mountings: American Planed with duel shoulders, European Style Precision Ground with a single shoulder, and the New Standard that mounts directly into the ram, where the power flows through the head of the tool.
**Tonnages**

One of the most critical aspects of press brake forming is tooling selection. What are the tools capable of? What kinds of loads can they withstand? Some manufacturers of press brake tooling give a maximum allowable tonnage per foot for their tooling, others do not.

For this section we will be using a tonnage limit of 30-tons per foot. **Note that this is not the only load limit; it is manufacture-specific** and the only person that can tell you the load limits of your press is the manufacturer.

Many times I have written articles about how to develop the tonnage requirements for a particular situation and followed that explanation with the statement,

**Never exceed the manufacturer’s maximum allowable tonnage**

Only to be asked later: “My tooling isn’t rated, how do I know if I’m using too much tonnage?”

Exactly how much tonnage-per-foot a given tool will take is dependent on material it is made of, tool geometry, and treatments to the tool. Even for the best of engineers this is a difficult call. Take a standard straight press brake punch, where there body of the tool is at least two inches wide, figure 5. It can be safely said that a straight punch will withstand far more tonnage-per-foot than the press brake with a 30-ton limit can withstand.

Before continuing this discussion, we need to define two terms that relate to press brakes. The first is deflection which is the deformation of the ram and bed of the press brake that naturally occurs under a load, figure 6. When the pressure is removed the deflection that is present under load, disappears. In other words, deflection is normal and to be expected.

Press Brakes are designed for center line loading, working in the center of the press. The power of the ram is being delivered through the sides of the ram while the force of the bend is in the center. On machines that allow off center loading, you can work directly under a hydraulic cylinder and not have any deflection present, but you only have half of the press brake’s tonnage available.

The second term is ram upset which occurs when the maximum deflection press brakes ram has been exceeded. When this happens, the ram and bed stay permanently deflected (bent); and once “upset”, both the ram and the bed are difficult and expensive to repair.
The maximum tonnage that a press brake is safely capable of using could be stated as a rule of thumb “Never apply full machine tonnage over a length less than 60% of the distance between the side frames” figure 7.

For example, regardless of the machine’s full tonnage rating, if the bed is ten feet between the side frames, a full load could only be uniformly applied to forming lengths of six feet or greater.

To use full tonnage over less area would ensure that your press brake would suffer some degree of upset in the bed and ram. The tooling could also be overloaded.

Considering all the different variables: material, geometry, etc., it is very difficult to predict the maximum allowable tonnage capacity for a given tool.

Using a 250-ton press brake with 14-feet between the side frames and applying the 60% minimum-working, the maximum tonnage per foot is calculated in the following way:

60% of 14 feet gives a value of 8.4 feet. Divide the maximum tonnage of 250-tons by 8.4 feet to arrive at 29.76 tons per foot. Expressed as maximum tonnage per foot, round up to 30 tons per foot, 27.22 tons metric.

The formulas for converting metric tons to US tons and back are:

\[
\text{Metric tons} \times 1.102 = \text{U.S. tons} \\
\text{U.S. tons} / 1.102 = \text{Metric tons}
\]

Maximum machine tonnage capacity can be expressed as:
\[
\frac{\text{Tonnage}}{\text{(Distance between the side frames} \times 60\%)}
\]

Gooseneck Punches

While there are variations, standard throat depths begin where 20% of the total tool body past center has been removed, figure 8; leaving only 30% of the possible tool body available to withstand a tonnage load.

A 70% (20% past center) throat depth will produce a 30% drop of tonnage capacity from that of a straight tool. It also follows that for any additional increase in throat depth, there will be a corresponding decrease in tool capacity.

Values in table 1 are from an Amada catalog; the maximum load limit for the smaller machines is 30-tons per foot. The listed tooling tonnage values reflect this fact. This limit is not universal, the load limit of any press brake is related to the thickness, width, height and make-up of the ram.

Only the manufacturer of your press brake and press brake tooling can tell you the load limit of your press an its tooling, figure 8.

It should also be noted that American planed tooling is not rated for tonnage limits like much of the precision ground tools are. Making the load limit of you tools the maximum tons per foot load limit of the press brake.
### Table 1

#### Die Tonnage

As for the maximum tonnage of a die, the same principles applies, the load limit of the die or the load limit of the press brake, whichever is greater. The maximum tonnage per foot would be 30 tons per foot under maximum load, half that tonnage per foot for a sectionalized set. Acute tooling maximums should also be halved to 15 tons or less to avoid splitting the die or damaging the press brake.

Tonnage can be computed quickly and accurately. As we already know, charts can be inaccurate, whether for bend deductions or tonnage, but just like calculating bend deductions, the tonnage can be accurately predicted almost every time. The tonnage per inch can be expressed as:

\[
\text{Tons per inch} = \left( \frac{575 \times M \times t^2}{\text{die width}} \right) / 12
\]

Note the **575** is one of those time tested constants used in mild cold rolled steel calculations.

The formula solves the pressure required to bend a one inch piece of mild cold rolled steel in the given die width that you have selected. Multiply this value by the number of inches in a given bend and that answer will be the total tonnage required for the bend.

Now that you know how to calculate the tonnage, you need to apply it. Does the required tonnage exceed the capabilities of the tool to withstand the force? You will need to find the tooling fact sheet and find the tonnage-per-inch for the tools to find out.

The maximum tooling tonnage per inch times the number of inches to be formed, equals total allowable tonnage for that particular tool.

\[
\text{TOTAL ALLOWABLE TONNAGE} \quad \text{minus} \quad \text{TOTAL REQUIRED TONNAGE} \quad \text{equals} \quad \text{EXCESS TOOLING CAPABILITY}
\]

\[
\text{NEVER EXCEED ALLOWABLE TOOLING TONNAGE!}
\]

Because of the variances in the tensile strengths between materials the formula for tonnage just given is incomplete. The basic formula is based in the tensile strength of mild cold rolled steel, 60k tensile.
**tensile strength** is defined as the ability of a material to bear weight without breaking or being pulled apart under a smooth load and not sudden impact. Look at different tensile and yield strengths in table 2. By no means a complete list it does include the some of the most common type of materials found in today's sheet metal shop.

<table>
<thead>
<tr>
<th>Type of material</th>
<th>Tensile strength</th>
<th>Yield</th>
<th>Stock</th>
</tr>
</thead>
<tbody>
<tr>
<td>3003 -0.</td>
<td>8</td>
<td>3</td>
<td>annealed</td>
</tr>
<tr>
<td>3003</td>
<td>14.5</td>
<td>13.5</td>
<td>H-18</td>
</tr>
<tr>
<td>220</td>
<td>24</td>
<td>13</td>
<td>T4</td>
</tr>
<tr>
<td>AISI-SAE 1020</td>
<td>27.5</td>
<td>15</td>
<td>hot rolled</td>
</tr>
<tr>
<td>AISI-SAE 1020</td>
<td>45</td>
<td>31</td>
<td>hardened</td>
</tr>
<tr>
<td>AISI-SAE 1035</td>
<td>36</td>
<td>19.5</td>
<td>hot rolled</td>
</tr>
<tr>
<td>AISI-SAE 1035</td>
<td>33.5</td>
<td>40</td>
<td>cold rolled</td>
</tr>
<tr>
<td>AISI-SAE 1045</td>
<td>41</td>
<td>22.5</td>
<td>hot rolled</td>
</tr>
<tr>
<td>AISI-SAE 1045</td>
<td>45.5</td>
<td>38.5</td>
<td>cold drawn</td>
</tr>
<tr>
<td>AISI-SAE 1120</td>
<td>34.5</td>
<td>29</td>
<td>cold drawn</td>
</tr>
<tr>
<td>AISI-SAE 4140</td>
<td>54</td>
<td>47.5</td>
<td>tempered</td>
</tr>
<tr>
<td>B-cryilium copper</td>
<td>25</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ASTM B194</td>
<td>55</td>
<td>52</td>
<td>cold rolled</td>
</tr>
<tr>
<td>ASTM B194</td>
<td>35</td>
<td>16</td>
<td>annealed</td>
</tr>
<tr>
<td>Type 304</td>
<td>42.5 to 92.5</td>
<td>17.5 to 80</td>
<td>annealed</td>
</tr>
<tr>
<td>Type 304</td>
<td>42.5 to 92.5</td>
<td>17.5 to 80</td>
<td>cold rolled</td>
</tr>
<tr>
<td>Type 316</td>
<td>45.5 to 75</td>
<td>15 to 60</td>
<td>annealed</td>
</tr>
<tr>
<td>Type 431</td>
<td>60 to 97.5</td>
<td>42.5 to 75</td>
<td>annealed</td>
</tr>
</tbody>
</table>

Table 2

When discussing factors that are to be used in the formulas we need a base line reference. For this we use 60K tensile, AISI-1035 and assign it a factor value of one.

*AISI-1035 is the most common type of cold rolled steel used.*

The factors or multipliers for several materials are listed below. For any material type that has not been given a factor value here, a comparison of tensile strengths will allow you to make an educated guess of factor value required.

- 304 Stainless = 1.4 to 6
- Aluminum 6061 T6 = 1.28
- Cold rolled steel = 1.00
- Aluminum 5052 H32 = .50
Using the formula:

\[ \text{Tons per inch} = \left( \frac{575 \times M t^2}{\text{die width}} \right) / 12 \times \text{the material factor.} \]

\[ \text{Material type} = 304 \text{ stainless steel.} \]
\[ \text{Material thickness} = .050 \]
\[ \text{Punch radius} = .030 \]
\[ \text{Die width} = .236 \]
\[ \text{Bend length} = 6.375 \]

\[ \text{Tons per inch} = \left( \frac{575 \times .050^2}{.236} \right) / 12 = .507 \text{ tons per inch.} \]
\[ \text{Tonnage per length} = .507 \times 6.375 \text{ (bend length)} = 3.235 \text{ tons.} \]
\[ \text{Total tonnage} = 3.235 \times 1.4 \text{ (material factor)} = 4.530 \text{ tons.} \]

Solving for the allowable tooling tonnage begins by looking into the tooling manufacturer’s data sheets. Each manufacture should publish a list containing each piece of tooling the company produces. This list should have a tool reference number and the allowable tonnage per foot the tool will handle.

Taking that tonnage number and dividing it by 12 gives you the allowable tonnage-per-inch the tool will handle. Now, multiply that value by the total bend length. The answer you get is the maximum allowable tonnage over the bend length.

Make sure this value is greater than the required tonnage value.

Everything involving pressure up to now has been rooted around a 90° bend in a standard die. The maximum required tonnage doesn’t happen all at once; it builds up along a curve, figure 9. Under close observation you will notice that 80% of the total tonnage is developed in the first 20° of bend.

Even with a small bend angle the pressure on the tooling and equipment can become great.

**Pressure Flow**

When mounting the punch into the press brake special attention needs to be paid to how the power flows through the tool. The two forms of press brake tooling shown here are European, and Standard American, figure 10.

Standard American tooling is completely reversible as far as power flow is concerned, but it can lose its center in the die. It also loses the relationship from the backgauge to the bend line, figure 10.

European style tooling is completely reversible as to centers, but it can be installed wrong; figure 11 shows this type of tooling installed incorrectly. Notice how the pressure is flowing past the ram and onto the mounting bolts, setting up a very dangerous situation.