Acute and Hem Bending

Acute Bend Angle Tools

Acute bend tools are used to achieve bend angles that are greater than 90° and are the first step in hemming or seaming. Acute tooling, comes in two basic angles: 30° and 45°. Figure 1 shows an example of an acute punch and die set.

Acute tooling can only be used for air forming; bottom bending and coining operations cannot be accomplished because the angle tool cannot withstand the side loading forces.

Die width selection for the acute bends is the same as any other. You will be air forming so the radius is based on the 20% rule, so you simply ignore the fact that these tools are not at 90° and continue the calculations as before.

True, the answers would be slightly different if the correct die angles are used in the calculations, but due to a limited selection of die widths, the angle makes little difference while making the math easy.

But, ultimately you have to pick between die width A and die width B.

The standard acute punches measure 43° and 28° and are mated to 45° and 30° dies.

One major thing that does change with die angle is the minimum flange length. An acute die requires a greater minimum flange length than a standard 90° V-die.

Figure 2 illustrates the differences in minimum flange length; both of the illustrations are of the same die width but with different die angles. Table 1 shows the minimum allowable die width to flange length for acute tooling.

Bend Radius

The selection of punch radii will require you to take several factors into consideration.

The acute bend actually produces a small flat across the inside of the bend rather than a true radius, figure 3; this flat is tends to be more pronounced in softer material.

Large and profound radii will produce the multi-breakage factor with a totally different effect. The distance across this flat is in direct proportion to the punch radius and can be expressed as:

The flat width = the punch radius times two.
Material Thickness | 1.6 | 2.0 | 2.3 | 2.6 | 3.0 | 3.5 | 4.0 | 4.5 | 5.0  
---|---|---|---|---|---|---|---|---|---
30° | 10 | 16 | 24 | 24 | 35 | 35 | - | - | -  
45° | 10 | 10 | 19 | 19 | - | - | 35 | 35 | 41  

Table 1

The rules of sharp, radius, and profound bends still apply with acute bend angles. For example, the flat of the sharp bend will still be solved from the 63% threshold, etc.

Aluminum adds one other aspect, cracking around the outside surface from circumferential strain. This cracking can be kept to a minimum if the flat across the inside is at least two times the material thickness.

The minimum allowable inside radius for an acute bend in an aluminum alloy can be expressed in the following way:

Minimum acute flat for aluminum = (Mt x 2)

Minimum acute punch radius for aluminum = minimum flat length x 2

Hemming and seaming

Hemming and seaming are basically the same process. The hem is primarily used for reinforcement, safety, and edge appearance, figure 4. A seam is a method of joining two pieces of sheet metal together, figure 5.

In the hemming process the bend is started with an acute tool set and then flattened in flattening dies, figure 6. This procedure is the same for seaming except the tonnage is increased slightly.

Before we can determine the tonnages required we must determine which of the four different styles of hems we will be using, starting with closed hem and Tear drop hem and open hems. Closed means “flattened completely” and “open hems” have various degrees of opening at the hem, figure 7.

The second two are the tear drop and rope hems, figure 7 shows examples of all four. A few points of note: a hems minimum flange is ten times the material thickness.

Closed hems should not be used if any chemical processes are to be preformed after forming, e.g., etch or chromate. This is because those chemicals remain in the gaps and can later leach out during welding or painting.

The open hem is becoming more popular for that reason. The open hem is created by placing a shim of material of the correct thickness; that of the gap; open hem, figure 7. The bend is flattened and the extra piece is removed; this does mean that the tonnage required will be the same as a closed hem. The differences between each is obvious; the reasons for each are customarily left up to the design engineer. However, as the operator, you need to notice the differences in tonnage.

*Note that these particular tonnages are tons per meter.*
<table>
<thead>
<tr>
<th>Material Thickness</th>
<th>Open Hem</th>
<th>Closed Hem</th>
</tr>
</thead>
<tbody>
<tr>
<td>.06</td>
<td>9</td>
<td>23</td>
</tr>
<tr>
<td>.08</td>
<td>12</td>
<td>32</td>
</tr>
<tr>
<td>1.0</td>
<td>15</td>
<td>40</td>
</tr>
<tr>
<td>1.2</td>
<td>17</td>
<td>50</td>
</tr>
<tr>
<td>1.6</td>
<td>24</td>
<td>63</td>
</tr>
<tr>
<td>2.0</td>
<td>30</td>
<td>80</td>
</tr>
<tr>
<td>2.6</td>
<td>55</td>
<td>90</td>
</tr>
<tr>
<td>3.2</td>
<td>70</td>
<td>100</td>
</tr>
<tr>
<td>4.5</td>
<td>105</td>
<td>200</td>
</tr>
</tbody>
</table>

Table 2

Hemming Tonnage in Metric Tons

The last thing that needs to be answered is how to find the flat length of the workpiece, the “bend deduction”. The formula for the bend deductions, outside setbacks, and bend allowances cease to be of value when the bend angle exceeds $174^\circ$.

The blank size is computed through the use of the “hem allowance” (Ha). In materials with a thickness less than .080, the hem allowance factor is 43% of the material thickness.

Take the following example:

Material thickness = .059 with a Hem length = 1.000

Times 43% = .43 Hem allowance = -.025

Hem allowance = .025 Flat blank length = .975

Should the occasion arise that requires an extremely tight tolerance, a pre-test of the material should be made to find the absolute value of the hem allowance.