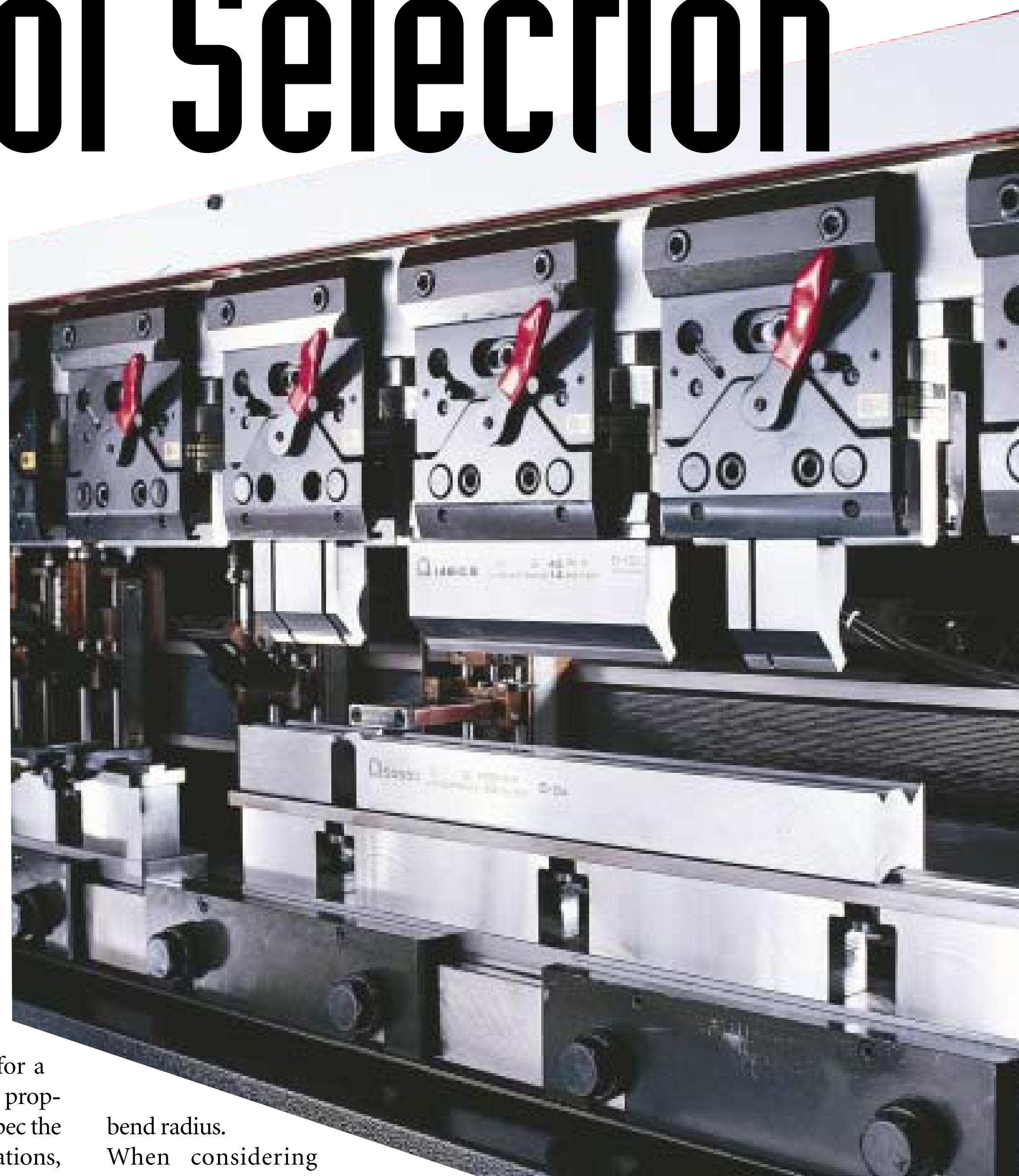
Press-Brake Tool Selection



Start from the bottom and work your way up—spec the V opening, find a compatible punch, then fine-tune to avoid marking and cracking, and optimize throughput.

BY KEVIN COYLE

When trying to solve a challenging metal-fabrication problem, it's good advice to start at the beginning to find a solution. But when it comes to selecting press-brake tooling, the beginning isn't always easy to find. A rule of thumb: Start from the bottom and work your way to the top.

To select press-brake tooling for a bending application, start with the proper bottom-die opening, and then spec the top punch. For air-bending operations, several parameters must be considered when specifying the V opening. These include base-material thickness, flange length of the part and the desired inside bend radius.

When considering material thickness, the V opening should be six to 10 times the material thickness—the multiplying factor depends on the actual thickness of the material. When a specific inside radius is required, use a bending-force chart (next page) to determine the optimum V openings for the application. Bending-force charts,

Tooling that mounts on a press brake without the use of wrenches or other tools will minimize setup time. These quick-change tools use a punch- and die-mounting system that eliminates the need for tool centering before each production run, eliminating a time-consuming setup step.

Kevin Coyle is tooling product manager, Amada America, Inc., Buena Park, CA; 714/739-2111.

AIR BENDING FORCE CHART

Die Opening Selection Formula

t					
t	GAUGE	12 OR LESS	11-5/16"	3/8" - 1/2"	5/8" UP
	INCHES	.105 OR LESS	.120 - .313	.375 - .500	.625 AND UP
"V" SIZE		6 X *t*	8 X *t*	10 X *t*	12 X *t*

The above formulas are for reference only. For more detailed information refer to the chart below.

If the material thickness and inside bend radius are known, the following can be obtained from the chart below:

1. Pressure required for bending the material for 1 foot.
2. Opening of the die to be used.
3. Minimum bendable flange length.

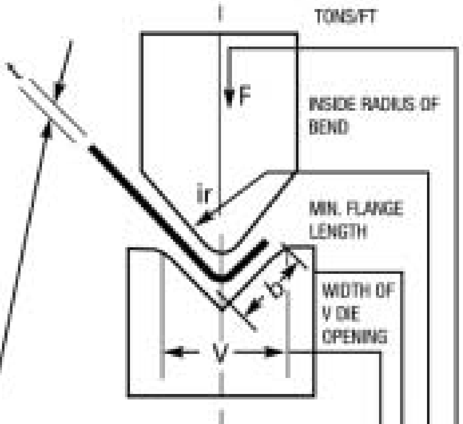
t Material thickness (tensile strength: 56892-71115 lbs/in^2)

F Tons per 1 foot

ir Inside bend radius

b Minimum flange length

v Die opening



THICKNESS

MILD STEEL

t GAUGE	t DEC.																						
		4	6	7	8	10	12	14	16	18	20	25	32	40	50	63	80	100	125	160	200	250	mm (v)
		.156	.250	.281	.313	.375	.500	.563	.625	.688	.750	1.000	1.250	1.500	2.000	2.500	3.000	4.000	5.000	6.000	8.000	10.000	inches (v)
		.125	.188	.203	.219	.281	.344	.406	.438	.531	.563	.688	.875	1.125	1.375	1.750	2.188	2.188	3.500	4.500	5.500	6.875	inches (b)
		.031	.031	.047	.047	.063	.078	.094	.109	.125	.141	.156	.203	.250	.313	.406	.516	.625	.750	1.001	1.313	1.625	inches (ir)
20	0.036	5.4	3.6	3.0	2.5	2.0	1.7																
18	0.048		6.8	5.8	4.8	3.7	2.7	2.4	2.0														
16	0.060				7.8	6.2	5.0	4.2	3.5	3.1	2.7												
14	0.075					11.0	8.2	7.0	5.6	4.8	4.1	3.1											
12	0.105						15.0	13.0	11.0	9.4	7.4	5.5	4.0										
11	0.120								16.0	13.0	10.0	7.3	5.0	3.8									
10	0.135										12.0	9.0	6.2	4.7	3.5								
.188	0.188											24.0	15.0	11.0	7.5	5.7							
.250	0.250												30.0	20.0	14.0	10.5	8.5						
.313	0.313													36.0	25.0	18.0	13.0	10.0					
.375	0.375														38.0	28.0	20.0	15.0	11.0				
.500	0.500															52.0	39.0	30.0	22.0	16.0			
.625	0.625																70.0	52.0	38.0	27.0	20.0	15.0	
.750	0.750																		66.0	43.0	32.0	22.0	
1.000	1.000																			90.0	61.0	44.0	
1.250	1.250																				102.0	70.0	

F

TONS/FT

Force required per linear foot to bend mild steel with air bend dies. (Measured in tons.)

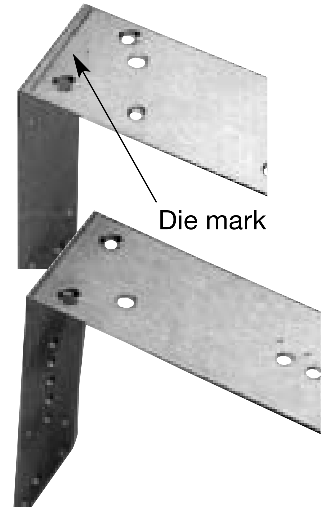
published by press-brake machine and tooling manufacturers, provide data such as inside radius produced, required tonnage, smallest bendable flange and the V opening of the die.

Charting Flange Length and Bend Radius

When forming a part with a relatively short flange length, a bending-force chart assists in determining the V opening. This chart lists the smallest flange each V opening can form. By comparing this to the required flange length, a proper V opening can be determined. If the flange is extremely short, the necessary V opening may cause the required tonnage to exceed the capacity of the tool selected. You must not overlook the maximum allowable tonnage of the die selected, to avoid an unsafe situation at the press brake.

If the part being formed requires a specific inside bend radius, the bending-force chart can provide the V opening as well. For air bending, the inside radius typically equals the material thickness. By locating the desired radius on the chart, a corresponding V opening can be identified that will produce that inside radius. Again, verify that the required tonnage, for the V opening selected and the material thickness being formed, does not exceed the allowable tonnage of the tool being used, and that the press brake can produce the tonnage necessary to form the desired part length.

The high tonnages involved in forming sheetmetal can leave impressions from the tooling on the material (top). Reduce or eliminate this marking (bottom) by using a bottom die with increased shoulder radii, or laying a polyurethane sheet or similar protective material over the bottom die.



Time to Select a Punch

With the V opening specified based on the desired part parameters, the engineer then can select a compatible punch, starting with specifying punch-tip radius. Most European-style punches come with a variety of tip radii. To form mild and stainless steels, select a punch-tip radius roughly half the material thickness being formed. Since the V opening will determine the formed inside bend radius, punch-tip radius should not exceed the material thickness, or else the inside bend radius will increase. When bending soft aluminum, it may be necessary to use a punch-tip radius equal to the desired inside radius, as the material will tend to form to the punch tip.

The angle of the punch tip also will affect formed-part dimensions. Most European-style punches come with 88- and 90-deg. punch-tip angles. The 88-deg. punch tip can be used to overcome springback, which most commonly occurs when forming in stainless steel. Generally, forming stainless steel to 90 deg., the bend will spring open 1 or 2 deg. because of the material's hardness. Overbending 1 to 2 deg. will overcome this springback and allow the part to spring open to the desired 90 deg. This flexibility in the punch-tip angle allows for lower-tonnage air bending while producing the desired bend design.

Eliminate Cracking and Unsightly Die Marks

Common tooling-related defects related to press-brake forming include die marks on the material, part cracking and inconsistent bend angles. These defects can be eliminated through some aspect of tool selection.

Customers often demand parts with no marking. But, due to the high tonnages involved in forming sheetmetal, impressions from the tooling on the material are inevitable. These can be very noticeable when forming aluminum or prepainted materials. One way to reduce this marking: Use a bottom die with increased shoulder radii. This allows the material to roll over the shoulders of the die, with less chance for the die to dig into the material and leave a mark. For additional protection from die marks, lay a polyurethane sheet or similar protective material over the bottom die, between it and the base material.

When base-material cracking occurs, increasing the V opening of the die generally eliminates the problem. If cracking persists, try increasing the punch-tip radius.

With the proper V die and punch selected, the style of the tooling can help increase overall production. Look for press-brake tooling that can be mounted on the press brake without the use of wrenches or other tools, to minimize setup time. These quick-change tools allow an operator to change the press brake over for the next job quickly, preserving time for part production. Such tools use a punch- and die-mounting system that eliminates the need for tool centering before each production run. The mounting system maintains a consistent tooling centerline, eliminating a time-consuming setup step.

Production also can be improved with the use of specialty tooling designed to form complicated bends with fewer hits. This type of tooling finds use with relatively large production quantities, where a shop can justify the added tooling cost. Common bends produced with this type of tooling include offsets or joggles, hemmed flanges and narrow channels. **MF**