Fundamentals of Press Brake Tooling
2nd Edition

Sample Chapter 3: The Basic 90° Bend
Press brake bending falls into two basic categories with several compromise options. The first is the foundation for all press brake work and is called air bending. The second type is called bottom bending.

**A) Air Bending**

Air bending is defined as three points of contact with the part to form a straight line angle (Fig. 3-1). The nose of the top or upper die forces the part to be formed into the vee shaped lower die. The included angle machined on both the upper and lower die must not allow any contact with the part except the nose of the upper die and corners of the vee opening in the lower die. When the upper die has penetrated deep enough into the lower die to produce the required angle (this is at the bottom of the forming stroke), the upper die is returned to the top of the stroke releasing the now formed part. When the part is released, the two legs of the newly formed part will spring back somewhat until the stresses in the formed part are balanced. If the material is simple cold rolled steel, it is common for the metal to open 2° to 4° from the angle actually made during the forming stroke.

The greater majority of press brake forming is making a simple 90° vee bend in a part. To allow for springback, the angle cut on the upper and lower dies will be machined to an angle less than 90°, normally between 75° and 85°. This allows the part to have only three points of contact with the tooling and no contact with the other surfaces.

The nose radius of the upper die should be equal to, or slighter less
than, the metal thickness that is being formed. The sharper the nose radius, the greater the
die wear. Special nose radii are often required for aluminum, high tensile material, or exotic
materials.

There are two simple rules of thumb that have been used for years to choose tooling that
will give the most consistent and accurate air bend when forming mild steel. The recommended
vee die openings found on air bend tonnage charts are based on these methods.

The first rule, developed in the 1920s to determine the best vee die opening, is to multiply
the material thickness by 8 and round the answer to the nearest simple fraction. For example,
16 gauge mild steel has a nominal thickness of 0.060". Multiply 0.060" × 8, and the answer is
0.48". To select the proper vee opening, the answer is rounded up to 0.5".

Press brake operators also found that when forming mild steel, the inside radius in the bent
material was a function of the vee die opening. Although the inside radius is a parabolic shape
rather than a true radius, it is common practice to measure this arc with a simple radius gage
that closely fits the formed part. Therefore, the second rule is that the expected inside radius
is 0.156 (5/32) times the vee die opening being used. If the vee die opening is greater than 12
times the vee opening, it becomes apparent that the inside radius is actually elliptical, and
any dimensional radius called for on a drawing is an estimate. If an attempt is made to form
a part using a vee opening less than 6 times the material thickness, the inside radius will not
be a radius since the material will try to form a theoretical inside radius of less than one metal
thickness—which is impractical to air bend.

Based on the above rules, a 0.5" vee opening (calculated for 16 gauge) × 0.156 will equal
an approximate 0.075" inside radius. Note that the rule, which applies mostly to mild steel
material, does not refer to the material thickness being used. If the first example of 16 gauge
mild steel recommends that a 0.5" vee opening be selected, the resultant 0.075" inside radius
will be slightly larger than the 0.060" material thickness. If 18 (0.048) gauge mild steel was
formed using the same 0.5" vee die opening, a similar 0.075" inside radius would be formed
into the thinner material. If 14 (0.075) gauge mild steel was formed over the same die, the
resultant inside radius would be very close to metal thickness. Therefore, for most of the

<table>
<thead>
<tr>
<th>Standard Gauge Number</th>
<th>Weight Pounds/Sq Ft.</th>
<th>Inch Equivalent</th>
<th>Tolerance Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>5.625</td>
<td>0.1345</td>
<td>0.1419-0.1271</td>
</tr>
<tr>
<td>11</td>
<td>5.000</td>
<td>0.1196</td>
<td>0.1270-0.1121</td>
</tr>
<tr>
<td>12</td>
<td>4.375</td>
<td>0.1046</td>
<td>0.1120-0.0972</td>
</tr>
<tr>
<td>14</td>
<td>3.125</td>
<td>0.0747</td>
<td>0.0821-0.0710</td>
</tr>
<tr>
<td>16</td>
<td>2.500</td>
<td>0.0598</td>
<td>0.0635-0.0568</td>
</tr>
<tr>
<td>18</td>
<td>2.000</td>
<td>0.0478</td>
<td>0.0508-0.0449</td>
</tr>
<tr>
<td>20</td>
<td>1.500</td>
<td>0.0359</td>
<td>0.0388-0.0344</td>
</tr>
<tr>
<td>22</td>
<td>1.250</td>
<td>0.0299</td>
<td>0.0313-0.0284</td>
</tr>
</tbody>
</table>

Fig. 3-2. Common steel gauge thicknesses used in most press brake work.
common gauge thicknesses normally used for press brake forming, a vee die opening of 6 times the metal thickness rounded to the next simple fraction will produce an inside radius close to one metal thickness. Consult the next section (B) describing forming tolerances to understand why the eight times metal thickness vee die opening remains the recommended and most used vee opening selection. See the chart of different gauges of mild steel showing the nominal thickness plus the possible tolerance range (Fig. 3-2).

It is also interesting to note that each gauge thickness has a weight in “pounds per square foot” (lb/ft²) that is a simple number. For example, 16 gauge is listed at 2.500 lb/ft². The “gauge” system for steel was established in the late 1880s to allow the steel companies to regulate their production. The width of the steel being rolled could be set, and the length of material rolled over a specific time period could be measured. To determine the weight per square foot, the thickness had to be determined. The steel industry devised a gauge system to facilitate calculation of the tonnage of the steel being processed. Refer to Fig. 3-2 which illustrates the comparative lb/ft² versus material thickness for the more popular gauges used in press brake work. The current gauge thickness of steel was standardized as a federal law passed by the U. S. Congress on March 3, 1893. The gauge system law is based on a steel density of 489.6 pounds per cubic foot (lb/ft³).

B) Air Bend Forming Tolerances (Angular Only)

Since mild steel may not be consistent from piece to piece, coil to coil, or heat to heat, angular variations must be expected. The material could change in chemistry, which affects the tensile and yield strength. The rolling of the material during the manufacturing process may cause thickness variations that affect the angular consistency.

Other variations result from worn tooling, press brakes that do not consistently repeat at the bottom of the stroke, or poor setup by the operator or setup person. Most of the angular variation encountered will be found to be material variations. If the press brake is properly maintained, it should repeat to the bottom of the stroke each time within an acceptable tolerance. Worn tooling, once it has been set up and shimmed to produce an acceptable part, does not change from part to part. If the operator is locating the part properly, and assisting the part upwards during the forming stroke as required, the part tolerance should not be affected. It should be noted that if a formed part is removed from the press brake with a correctly formed angle, and then dropped on the floor or thrown into a container, the formed angle can open up and be out of tolerance.

If only the standard gauge tolerances are considered, a simple sketch, showing a drawing of a part having some thickness that is formed into a 90° angle, can be used to determine tolerances. The part sketch should show an inside and outside radius of the part. The sketch should include three marks: one mark to show where the top die contacts the part on the inside of the bend, and two marks on the outside of the material to show where the part would contact the vee die corner radii.

The sketch illustrates a part of nominal gauge thickness as it would look at the bottom of the forming stroke with the appropriate tooling contact. Fig. 3-3 illustrates (by use of dotted lines) possible material variations within a gauge range. If the material is thicker, the outside surface is pushed further down into the vee die cavity, resulting in an angle overbend. If the material is thinner than nominal, the outside surface does not penetrate into the vee die sufficiently to make the proper angle. Thus the angle remains open. Since only the material thickness was changed, it becomes clearly evident that material variations will cause angular variations when using simple air bend dies. If the material thickness becomes thicker than the material
used for the original setup, an over bend angle can be expected. If the material thickness is thinner than the material used for the original setup, the bend angle will be open.

Each gauge of material can be carefully sketched using a magnified scale, or using computer graphics that could measure angular variations that would not only show a 90° bend but also show their thicker and thinner tolerances as described above. It would be found that the average angular variation for gauge material would be about ±2°.

Practical experience has shown that a normal stack of material supplied to a press brake will not have the entire range of tolerance allowed on the tolerance chart. Some material variations can be anticipated, since to produce a coil of steel, in order to keep the strip tracking in a straight line, the center of the sheet is made slightly thicker than each edge. When the coil is cut or blanked to the material dimensions needed to make a particular part, some thickness difference will occur. How much, or in what direction, will not be known unless each part is measured and marked prior to making the required bends. In almost all cases, this is impractical both from a cost and time viewpoint.

Experience in working with sheet metal has proven that material variations in sheets of mild steel up to 10 gauge thick and as long as 10' will cause an actual angular variation of ±0.75° when air bending. Additional variation should be expected from the initial test part, which seemed to be acceptable, but may have had variation due to machine deflection, die wear, or machine repeatability. In sheet metal (10 gauge or thinner), surface hardness caused by the rolling operation in the manufacturing process, and chemistry changes in the material, all add some possibilities for variations. Because of the many other factors that must be considered, an additional ±0.75° must be added to the tolerance range. The total tolerance range is the addition of tolerances that are expected from probable material variations, plus the variations caused by all of the other unknown factors just listed. A realistic tolerance that should be considered when air bending 10 gauge or thinner mild steel up to 10' long is ±1.5°.

For plate, an additional degree is required, since the material variations are much greater. Tolerance for air bending material 7 gauge and thicker will be ±2.5° up to 1/2" thick plate. Heavier materials are often formed to an improved tolerance by using more than one stroke of the ram, and it is important to remember that any discussion of tolerance is based on using the recommended upper and lower dies.

To hold a consistent bend requires a vee die opening that allows the legs of the part to penetrate down into the vee die sufficiently to allow each leg or flange to have a flat distance
of 2.5 metal thicknesses past the outside radius of the part before contact with the corners of the vee die. The flat is needed to provide control of the bend angle. The recommended “8 times metal thickness” vee die opening provides a good flat to allow consistent parts to be formed within the tolerance range discussed. A smaller vee opening (e.g., 6 times metal thickness vee opening) will actually form a slightly smaller inside radius, but the flat from the outside radius to the contact with the vee die corners will also be reduced. This reduction of the flat surface results in additional angular variations in the part. A larger vee die opening will provide a greater flat, but also increases the size of the inside radius. The larger radius will result in more springback when the forming pressure is released, introducing more potential part variation.

The practical tolerance for air bending sheet metal up to 10 gauge thick, and 10' long, is ±1.5°. This variation is often felt to be more than can be accepted but, as with all tolerances, the maximum range possible does not normally occur in one part. A standard statistical bell-shaped curve should reflect the actual bend variations. This means the greater majority of parts will be formed with much less variation. Most production runs require only a few parts of each shape to be formed. With the availability of high-tech, computer access press brakes, air bending is regaining its popularity, which had dropped somewhat from the 1960s through the 1980s.

**C) Forming with Bottoming Dies**

To obtain better angular consistency, or to compensate for repeatability or deflection problems of the press brake, a forming method called **bottoming** may be selected (Fig. 3-4). Bottoming often creates problems for the press brake operator. The forming method has four different definitions depending on the tooling design and how it is used during the forming cycle. Any simple straight line forming where the formed part touches the sloping “vee” section, in addition to the corners of the vee opening, is no longer an air bend. It must be classified as some type of bottoming die because completion of the bend will require more force than would be required to make a similar air bend.

**1) True Bottoming**

The upper and lower dies are machined so that the forming surfaces have the same angle as the angle of the part that is to be formed. If a 90° angle is required, the upper and lower die surfaces are machined to a 90° angle symmetrical around the centerline. The radius of the tip or nose of the upper die is machined with a one metal thickness radius, or to the closest simple fraction. The tooling for machining radii is often limited to specific fractions, and then converted to corresponding decimal dimensions. It is common practice, since most bottoming work is preformed using materials 14 gauge or thinner, to select die bars of the same width for the upper and lower dies.

Often the vee opening selected is the same 8 times metal thickness vee die opening recommended for an air bend die. Some operators, however, are more comfortable with

![Fig. 3-4. Bottoming illustrating the material contacting the sides of the vee die opening.](image-url)
the vee die opening being 6 times metal thickness. This opening causes the material to initially form to an inside radius of approximately one metal thickness. When material is formed, either using the air bend method or with bottoming type tools, as the part is forced into the vee opening, an inside radius is formed into the metal. Although called a radius, it is actually some type of “parabolic” shape. This is very important to know since it helps to explain what happens to the legs of the part during a forming cycle using bottoming dies.

During the forming cycle, several functions occur that can affect the quality of the final angle. The nose radius of the upper die is machined with a true radius. The inside radius formed on the inside of the part is an elliptical shape due to the part being air bent as it travels into the die cavity. The elliptical shape will be slightly larger than the radius machined on the die. When the outside legs of the part strike the sloping sides of the vee die opening, several conditions may result. Depending on the position of the top die at the bottom of the stroke, and the amount of force or tonnage striking the part, the operator may find, as shown in Fig. 3-5, one of the following.

Stage 1) The inside radius of the part will follow the 0.156 times the vee opening rule, as in air bending.

Stage 2) If the stroke pushed the part down to the bottom of the vee die using only the force required to air bend the part, the formed angle would spring open, probably 2° to 4°, when the upper die returns to the top of the stroke.

Stage 3) If the forming stroke had been lowered slightly so that the tonnage at the bottom of the stroke built up to about 1.5 to 2 times the normal air bend tonnage, then the pressure was released as the ram returned to the top of the stroke, the resultant angle will be overbent by several degrees. The overbent angle will be very consistent in tolerance but will not be the desired final angle.

Stage 4) If the bottom of the stroke ram setting is increased so that the tonnage at the bottom of the stroke builds up to 3 to 5 times the tonnage required for a simple air bend, the corners of the upper die will force the overbent legs of the part back to the desired angle, normally 90°.

The obvious question is: “Why does the part overbend to an angle less than 90° when the die angle apparently should limit the flange motion?” The answer is fairly simple. Take one hand and hold it up in front of you. Keep your four fingers together and open your thumb to form an angle between your thumb and forefinger. Notice the large elliptical shape that your skin makes between the thumb and forefinger. Take the forefinger of the other hand and start to press it down into the center of the elliptical area between the thumb and forefinger. Immediately, your thumb and forefinger will start to move together, reducing the size of the original angle you had made. The same phenomenon occurs when a bottoming operation is used. The upper die radius is a true radius. The shape formed in the material when it is pushed down into the vee die is somewhat elliptical. At the bottom of the stroke, as tonnage is built up, the part will overbend just like your fingers did. The flanges will overbend until they touch the corners of the top die. If the pressure is released at that time, the flanges may spring back. If the part were struck hard enough that the area contacted by the upper die exceeded the yield point of the material, springback would be eliminated. If released from the forming pressure at that time, the part may still be in an overbent condition. It will remain there until the upper die is set lower to allow the corners of the upper die to wedge the flanges open to an acceptable 90° angle. This requires a great deal of tonnage. The sharper the nose radius of the upper, the greater the amount of overbending.

True bottoming will produce a good consistent angle and an inside radius of one metal thickness. However, as pointed out, the forming tonnage required will be 3 to 5 times the
Chapter 3: The Basic 90° Bend

Stage 1 — Part forms the same as if is being “air bent.” The inside radius being formed will follow the \( 0.156 \times \) the vee opening rule.

Stage 2 — Part contacts the bottom of the vee opening and the forming tonnage builds up to slightly more than the normal air bend tonnage. If the ram is reversed at this point, the part will spring open to an angle greater than the die angle.

Stage 3 — The ram travels farther down into the lower dies and the forming tonnage is increased. The material overbends slightly until the inside legs of the part touch the corners of the upper die. If the pressure is released at this point, the part may spring back to an angle close to the desired angle or it may stay overbent. If it remains overbent, the angle will be very consistent but it will simply be the wrong shape.

Stage 4 — The ram is lowered to allow the corners of the upper die to push the flanges of the formed angle back to the angle of the upper and lower die. At the same time, the inside radius reforms the inside radius that has been formed in the part to the same radius that is machined on the nose of the upper die.

Fig. 3-5. Stages that occur when bottoming a simple 90 degree bend.
tonnage needed to form the same angle by using the air bend method. Since the forming tonnage becomes so high, often requiring a much larger press brake, most bottoming work is limited to 14 gauge or thinner material. All parts, prior to selecting the forming process, should be reviewed to determine if sufficient tonnage is available to properly form the part.

2) Bottoming with Springback

A skilled press brake operator may often be able to form a variety of parts using the overbending function that occurs in a bottoming forming cycle as described previously (Fig. 3-6). The operator must carefully adjust the forming cycle stroke to allow the angle to overbend, but not be “set.” When the ram moves back to the top of the stroke, the formed angle will spring back to the required shape. This method only requires about 1.5 times the normal air bend tonnage, and may provide an angular accuracy slightly better than air bend tolerances. The disadvantage is that, if the part is hit too hard, the angle will stay overbent. Then, only bottoming tonnage will allow the upper die to push the legs back to 90°.

This forming method requires a great deal of operator skill to obtain good parts consistently (ref. Fig. 3-5, Stages 2 and 3). Many users of small tonnage press brakes attempt to use this method, even using sharp nose upper dies, in an effort to form their parts. Often the operator will rehit overbent parts several times in an effort to square the legs of a 90° bend angle.

If bottoming with springback forming is done with an upper die that has a nose radius smaller than metal thickness, the upper die will produce a crease or groove in the inside surface of the radius. This crease will occur when the top die contacts the material and pressure is built up to start the bending of the material into the vee opening. Some people will mistake this crease as a sharp inside radius. The actual part shape is the normal inside radius with a crease in the center.

There are a number of companies selling what is called “high precision” press brake tooling (often associated with the European style tooling discussed in Chapter 21) that promotes 88° angles on their dies. This falls into the “bottoming with springback” concept. This type of die is not designed to work with “programmable angle” press brake options available in many new high-tech machines, since they are programmed to work only with true air bend dies. The 88° dies do not fall into this category since they require that the material actually touch the sides of the lower die to reduce some of the springback.

3) Coining

Some designers of parts believe that the inside radius of the part should be smaller than metal thickness. The
only way this can be done is to force a small radius on the upper die (smaller than one metal thickness) into the inside radius that has been formed into the metal during the air bend portion of the forming stroke. The sharp nose radius on the upper die pushes down into the part at the bottom of the stroke and reforms the inside into a smaller radius. When solid metal is displaced or changed in shape, it is like the flat surfaces of a metal disc being reformed into a new shape, such as a penny, dime, or nickel. In this case, the displacement of the metal creates the new desired part, which is called a coin. When the upper die displaces the metal in the inside radius of the part, the forming method is called **coining**. The force required to displace the metal of the inside radius of a part to a \( \frac{1}{2} \) metal inside radius will range from 5 to 10 times the tonnage required to air bend that material using the recommended vee die opening (Fig. 3-7).

There is a mistaken belief that a sharper inside radius made by coining will result in a smaller outside radius. This thinking can be disproved on the drawing board. A part, using the gauge thickness in question, should be drawn to a magnified scale showing the material at a typical 90° angle. The inside radius should be drawn to the same estimated radius that would be formed if the recommended vee die had been used. A line along the inside of each flange should be extended to illustrate a sharp, or 0", inside radius. The small area now shown by the two straight lines at 90° and the curved line of the inside radius illustrates the amount of material that would be displaced if a sharp corner was actually made in the part.

The displaced material can only dissipate into the outside radius. If the small amount of material in the sharp inside corner is measured and incorporated into the outside radius of the part, the actual outside radius might be several thousandths of an inch smaller than originally formed. Tests formed by The Cincinnati Shaper Company in the 1960s found that hitting parts in 16 gauge and 10 gauge mild steel up to 100 tons per foot (100 tons/ft) only changed the outside radius of the formed part 0.008". The resultant tonnage also caused the part shape to backbend from excess pressure at each corner of the vee die opening, providing a totally unacceptable formed final angle.

**4) Bottoming Using Angles Other Than 90°**

For many parts, there is a need for bottoming type accuracy, but the press brake does not have the available tonnage to form the part with true bottoming dies. The tonnage needed to bring the part to a consistent “overbent” position is only about 1.5 to 2 times the charted air bend tonnage for that gauge of mild steel. Once the part reaches a set overbent angle, the angle along the length of the bend line will be very consistent. If the part is one that will be repeatedly formed, it may be a good idea to have a special set of vee dies cut with an angle greater than 90°. This will allow the material to be somewhat “bottomed” at the lower tonnage. Instead of forming to an unwanted overbent angle of 88°, if the dies were machined to an angle of 92°, the formed part overbends 2°, resulting in the desired 90° bend.

Some materials will spring back unless hit at a tonnage greater than the available press
brake capacity. This is often true when stainless is to be formed. Stainless is often formed using bottoming dies, resulting in springback to an angle 2° to 3° greater than desired after the pressure is released. When inspected, the angle will be very consistent along the bend line. If the die is made having an 87° or 88° included angle, instead of 90°, the operator will be able to make an acceptable 90° bend angle using the bottoming with springback concept.

The dies that have been cut to a special angle are not general purpose dies. The operator must learn to use them in order to obtain good angles. They will solve a tonnage limitation problem and provide good consistency. They will demand that the tons/ft tonnage needed for the longest part must also be held if shorter lengths of the same part must also be made. If the 92° dies used to correct the part “overbend” problem for long parts were used with shorter length parts, but were formed at a tonnage normally needed for true bottoming, the resultant part angle would probably have a 92° (or whatever angle that was machined on the die) angle along the bend line. The same logic would prevail if a short piece of stainless was truly bottomed using the 88° dies—the final angle might be the 88° machined on the dies. This method is a good reminder that hydraulic press brakes have tonnage limitations. They cannot be overloaded. When a mechanical press brake was used, the operator often thought: “if the angle is not correct, hit it harder!” This logic caused many overloads, along with high repair bills.

5) Bottoming Tolerances

True bottoming or coining tolerances will cut the normal tolerances expected from air bending in half. Instead of the ±1.5° specified for air bending 10 gauge and thinner up to 10' long using the recommended vee die opening, a bottoming (or if the material is coined) tolerance of ±0.75° variation can be attained. To hold tighter tolerances, a great deal of operator inspection will be required with time allowed to measure and rehit some of the bends. The optimum tolerance is ±0.5°. If enough time is spent on each part, and if the material specifications are closely held, some parts have been held to the equivalent of machining tolerances. If this is required, allow sufficient time for a great deal of handwork by a skilled operator, since this will approach “craftsman”-type work.

“Bottoming with springback” tolerances will vary between air bend and bottoming tolerances. Due to the many possible die and material combinations, an acceptable tolerance range that can be expected in a typical production run cannot be provided.