Stretching

Stretching metal is achieved by hammering or rolling metal under pressure. For example, hammering a flat piece of metal causes the material in the hammered area to become thinner in that area. Since the amount of metal has not been decreased, the metal has been stretched. The stretching process thins, elongates, and curves sheet metal. It is critical to ensure the metal is not stretched too much, making it too thin, because sheet metal does not rebound easily. *[Figure 4-117]*

Stretching one portion of a piece of metal affects the surrounding material, especially in the case of formed and extruded angles. For example, hammering the metal in the horizontal flange of the angle strip over a metal block causes its length to increase (stretched), making that section longer than the section near the bend. To allow for this difference in length, the vertical flange, which tends to keep the material near the bend from stretching, would be forced to curve away from the greater length.

Shrinking

Shrinking metal is much more difficult than stretching it. During the shrinking process, metal is forced or compressed into a smaller area. This process is used when the length of a piece of metal, especially on the inside of a bend, is to be reduced. Sheet metal can be shrunk in by hammering on a V-block or by crimping and then using a shrinking block.

To curve the formed angle by the V-block method, place the angle on the V-block and gently hammer downward against the upper edge directly over the "V." While hammering, move the angle back and forth across the V-block to compress the material along the upper edge. Compression of the material along the upper edge of the vertical flange will cause the formed angle to take on a curved shape. The material in the horizontal flange will merely bend down at the center, and the length of that flange will remain the same. *[Figure 4-118]*

To make a sharp curve or a sharply bent flanged angle, crimping and a shrinking block can be used. In this process, crimps are placed in the one flange, and then by hammering the metal on a shrinking block, the crimps are driven, or shrunk, one at a time.

Cold shrinking requires the combination of a hard surface, such as wood or steel, and a soft mallet or hammer because a steel hammer over a hard surface stretches the metal, as opposed to shrinking it. The larger the mallet face is, the better.

Bumping

Bumping involves shaping or forming malleable metal by hammering or tapping—usually with a rubber, plastic, or rawhide mallet. During this process, the metal is supported by a dolly, a sandbag, or a die. Each contains a depression into which hammered portions of the metal can sink. Bumping can be done by hand or by machine.

Crimping

Crimping is folding, pleating, or corrugating a piece of sheet metal in a way that shortens it or turning down a flange on a seam. It is often used to make one end of a piece of stove pipe slightly smaller so that one section may be slipped into another. Crimping one side of a straight piece of angle iron with crimping pliers causes it to curve. *[Figure 4-119]*

Folding Sheet Metal

Folding sheet metal is to make a bend or crease in sheets, plates, or leaves. Folds are usually thought of as sharp, angular bends and are generally made on folding machines such as the box and pan brake discussed earlier in this chapter.

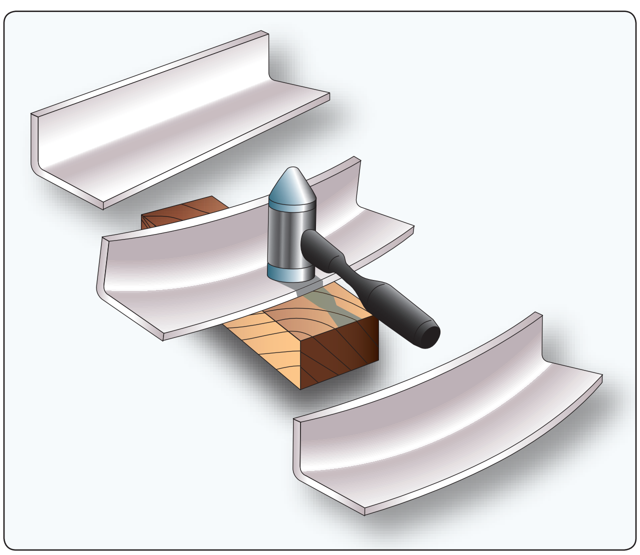

Figure 4-117. *Stretch forming metal.*

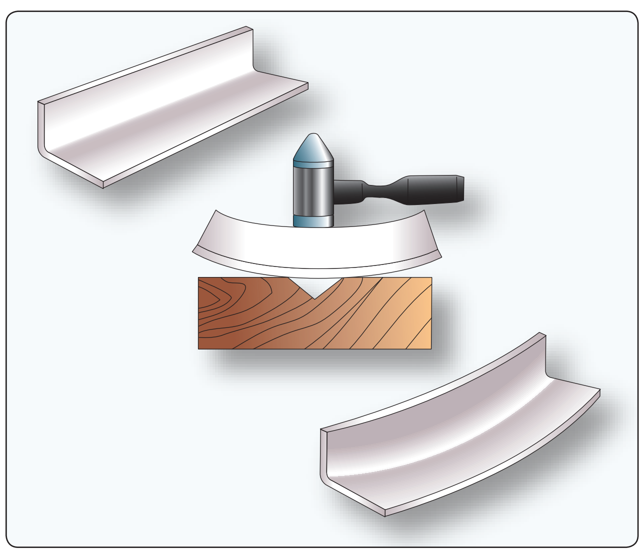

Figure 4-118. *Shrink forming metal.*

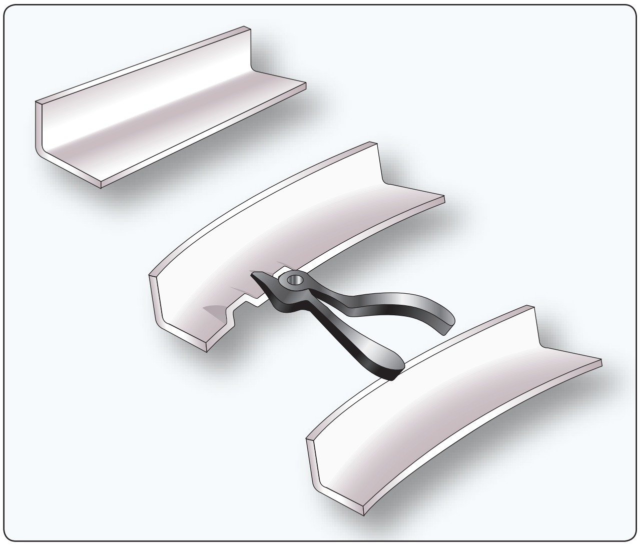

Figure 4-119. *Crimping metal.*

Layout and Forming

Terminology

The following terms are commonly used in sheet metal forming and flat pattern layout. Familiarity with these terms aids in understanding how bend calculations are used in a bending operation. *Figure 4-120* illustrates most of these terms.

Base measurement—the outside dimensions of a formed part. Base measurement is given on the drawing or blueprint or may be obtained from the original part.

Leg—the longer part of a formed angle.

Flange—the shorter part of a formed angle—the opposite of leg. If each side of the angle is the same length, then each is known as a leg.

Grain of the metal—natural grain of the material is formed as the sheet is rolled from molten ingot. Bend lines should be made to lie at a 90º angle to the grain of the metal if possible.

Bend allowance (BA)—refers to the curved section of metal within the bend (the portion of metal that is curved in bending). The bend allowance may be considered as being the length of the curved portion of the neutral line.

Bend radius—the arc is formed when sheet metal is bent. This arc is called the bend radius. The bend radius is measured from a radius center to the inside surface of the metal. The minimum bend radius depends on the temper, thickness, and type of material. Always use a Minimum Bend Radius Table to determine the minimum bend radius for the alloy that is going to be used. Minimum bend radius charts can be found in manufacturer's maintenance manuals.

Bend tangent line (BL)—the location at which the metal starts to bend and the line at which the metal stops curving. All the space between the band tangent lines is the bend allowance.

Neutral axis—an imaginary line that has the same length after bending as it had before bending. *[Figure 4-121]* After bending, the bend area is 10 to 15 percent thinner than before bending. This thinning of the bend area moves the neutral line of the metal in towards the radius center. For calculation purposes, it is often assumed that the neutral axis is located at the center of the material, although the neutral axis is not exactly in the center of the material. However, the amount of error incurred is so slight that, for most work, assuming it is at the center is satisfactory.

Mold line (ML)—an extension of the flat side of a part beyond the radius.

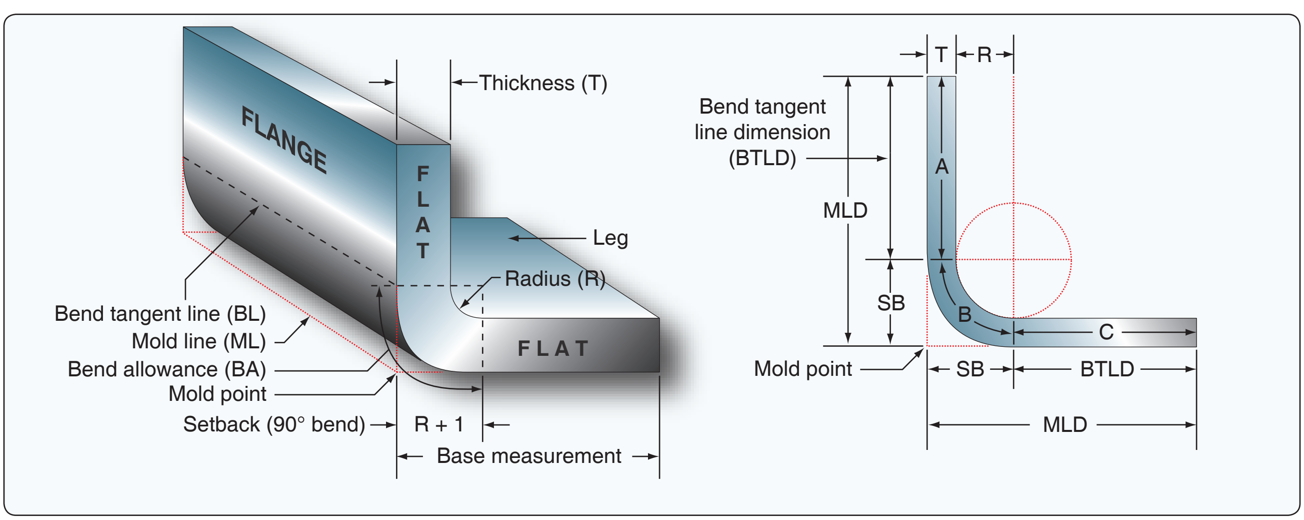

Figure 4-120. *Bend allowance terminology.*

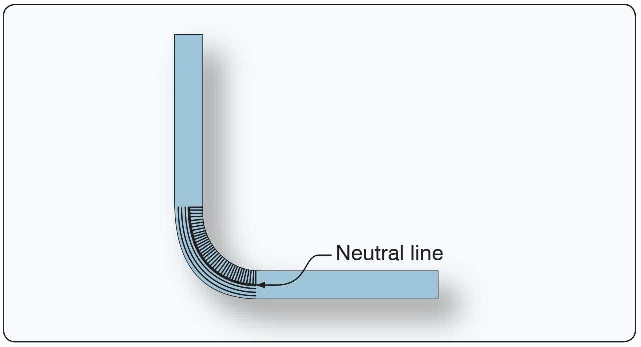

Figure 4-121. *Neutral line.*

Mold line dimension (MLD)—the dimension of a part made by the intersection of mold lines. It is the dimension the part would have if its corners had no radius.

Mold point—the point of intersection of the mold lines. The mold point would be the outside corner of the part if there were no radius.

K-Factor—the percentage of the material thickness where there is no stretching or compressing of the material, such as the neutral axis. This percentage has been calculated and is one of 179 numbers on the K chart corresponding to one of the angles between 0° and 180° to which metal can be bent. *[Figure 4-122]* Whenever metal is to be bent to any angle other than 90° (K-factor of 90° equal to 1), the corresponding K-factor number is selected from the chart and is multiplied by the sum of the radius (R) and the thickness (T) of the metal. The product is the amount of setback (see next paragraph) for the bend. If no K chart is available, the K-factor can be calculated with a calculator by using the following formula: the K value is the tangent of one-half the bend angle.

Setback (SB)—the distance the jaws of a brake must be setback from the mold line to form a bend. In a 90° bend, SB = R + T (radius of the bend plus thickness of the metal). The setback dimension must be determined prior to making the bend because setback is used in determining the location of the beginning bend tangent line. When a part has more than one bend, setback must be subtracted for each bend. The majority of bends in sheet metal are 90° bends. The K-factor must be used for all bends that are smaller or larger than 90°.

SB = K(R+T)

Sight line—also called the bend or brake line, it is the layout line on the metal being formed that is set even with the nose of the brake and serves as a guide in bending the work.

Flat—that portion of a part that is not included in the bend. It is equal to the base measurement (MLD) minus the setback.

Flat = MLD – SB

Closed angle—an angle that is less than 90° when measured between legs, or more than 90° when the amount of bend is measured.

Open angle—an angle that is more than 90° when measured between legs, or less than 90° when the amount of bend is measured.

Total developed width (TDW)—the width of material measured around the bends from edge to edge. Finding the TDW is necessary to determine the size of material to be cut. The TDW is less than the sum of mold line dimensions since the metal is bent on a radius and not to a square corner as mold line dimensions indicate.

Layout or Flat Pattern Development

To prevent any waste of material and to get a greater degree of accuracy in the finished part, it is wise to make a layout or flat pattern of a part before forming it. Construction of interchangeable structural and nonstructural parts is achieved by forming flat sheet stock to make channel, angle, zee, or hat section members. Before a sheet metal part is formed, make a flat pattern to show how much material is required in the bend areas, at what point the sheet must be inserted into the forming tool, or where bend lines are located. Bend lines must be determined to develop a flat pattern for sheet metal forming.

When forming straight angle bends, correct allowances must be made for setback and bend allowance. If shrinking or stretching processes are to be used, allowances must be made so that the part can be turned out with a minimum amount of forming.

Making Straight Line Bends

When forming straight bends, the thickness of the material, its alloy composition, and its temper condition must be considered. Generally speaking, the thinner the material is, the more sharply it can be bent (the smaller the radius of bend), and the softer the material is, the sharper the bend is. Other factors that must be considered when making straight line bends are bend allowance, setback, and brake or sight line.

The radius of bend of a sheet of material is the radius of the bend as measured on the inside of the curved material. The minimum radius of bend of a sheet of material is the sharpest curve, or bend, to which the sheet can be bent without critically weakening the metal at the bend. If the radius of bend is too small, stresses and strains weaken the metal and may result in cracking.

Degree	K	Degree	K	Degree	K	Degree	K	Degree	K
1	0.0087	37	0.3346	73	0.7399	109	1.401	145	3.171
2	0.0174	38	0.3443	74	0.7535	110	1.428	146	3.270
3	0.0261	39	0.3541	75	0.7673	111	1.455	147	3.375
4	0.0349	40	0.3639	76	0.7812	112	1.482	148	3.487
5	0.0436	41	0.3738	77	0.7954	113	1.510	149	3.605
6	0.0524	42	0.3838	78	0.8097	114	1.539	150	3.732
7	0.0611	43	0.3939	79	0.8243	115	1.569	151	3.866
8	0.0699	44	0.4040	80	0.8391	116	1.600	152	4.010
9	0.0787	45	0.4142	81	0.8540	117	1.631	153	4.165
10	0.0874	46	0.4244	82	0.8692	118	1.664	154	4.331
11	0.0963	47	0.4348	83	0.8847	119	1.697	155	4.510
12	0.1051	48	0.4452	84	0.9004	120	1.732	156	4.704
13	0.1139	49	0.4557	85	0.9163	121	1.767	157	4.915
14	0.1228	50	0.4663	86	0.9324	122	1.804	158	5.144
15	0.1316	51	0.4769	87	0.9489	123	1.841	159	5.399
16	0.1405	52	0.4877	88	0.9656	124	1.880	160	5.671
17	0.1494	53	0.4985	89	0.9827	125	1.921	161	5.975
18	0.1583	54	0.5095	90	1.000	126	1.962	162	6.313
19	0.1673	55	0.5205	91	1.017	127	2.005	163	6.691
20	0.1763	56	0.5317	92	1.035	128	2.050	164	7.115
21	0.1853	57	0.5429	93	1.053	129	2.096	165	7.595
22	0.1943	58	0.5543	94	1.072	130	2.144	166	8.144
23	0.2034	59	0.5657	95	1.091	131	2.194	167	8.776
24	0.2125	60	0.5773	96	1.110	132	2.246	168	9.514
25	0.2216	61	0.5890	97	1.130	133	2.299	169	10.38
26	0.2308	62	0.6008	98	1.150	134	2.355	170	11.43
27	0.2400	63	0.6128	99	1.170	135	2.414	171	12.70
28	0.2493	64	0.6248	100	1.191	136	2.475	172	14.30
29	0.2586	65	0.6370	101	1.213	137	2.538	173	16.35
30	0.2679	66	0.6494	102	1.234	138	2.605	174	19.08
31	0.2773	67	0.6618	103	1.257	139	2.674	175	22.90
32	0.2867	68	0.6745	104	1.279	140	2.747	176	26.63
33	0.2962	69	0.6872	105	1.303	141	2.823	177	38.18
34	0.3057	70	0.7002	106	1.327	142	2.904	178	57.29
35	0.3153	71	0.7132	107	1.351	143	2.988	179	114.59
36	0.3249	72	0.7265	108	1.376	144	3.077	180	Inf.

Figure 4-122. *K-factor.*

A minimum radius of bend is specified for each type of aircraft sheet metal. The minimum bend radius is affected by the kind of material, thickness of the material, and temper condition of the material. Annealed sheet can be bent to a radius approximately equal to its thickness. Stainless steel and 2024-T3 aluminum alloy require a fairly large bend radius.

Bending a U-Channel

To understand the process of making a sheet metal layout, the steps for determining the layout of a sample U-channel will be discussed. *[Figure 4-123]* When using bend

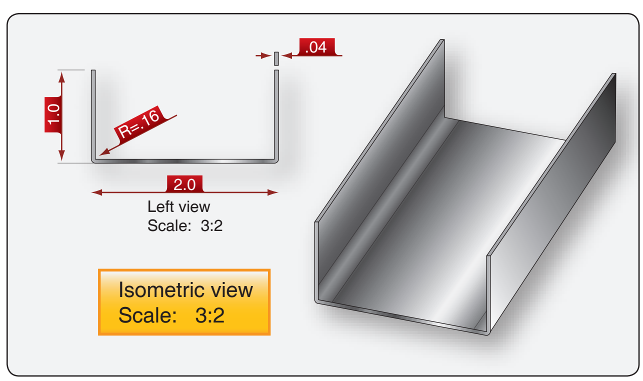

Figure 4-123. *U-channel example.*

allowance calculations, the following steps for finding the total developed length can be computed with formulas, charts, or computer-aided design (CAD) and computer-aided manufacturing (CAM) software packages. This channel is made of 0.040-inch 2024-T3 aluminum alloy.

Step 1: Determine the Correct Bend Radius

Minimum bend radius charts are found in manufacturers' maintenance manuals. A radius that is too sharp cracks the material during the bending process. Typically, the drawing indicates the radius to use, but it is a good practice to double check. For this layout example, use the minimum radius chart in *Figure 4-124* to choose the correct bend radius for the alloy, temper, and the metal thickness. For 0.040, 2024-T3 the minimum allowable radius is 0.16-inch or ⁵⁄₃₂-inch.

Step 2: Find the Setback

The setback can be calculated with a formula or can be found in a setback chart available in aircraft maintenance manuals or Source, Maintenance, and Recoverability books (SMRs). *[Figure 4-125]*

Using a Formula to Calculate the Setback

SB = setback

K = K-factor (K is 1 for 90° bends)

R = inside radius of the bend

T = material thickness

Since all of the angles in this example are 90° angles, the setback is calculated as follows:

SB = K(R+T) = 0.2 inches

NOTE: K = 1 for a 90° bend. For other than a 90° bend, use a K-factor chart.

Using a Setback Chart to Find the Setback

The setback chart is a quick way to find the setback and is useful for open and closed bends, because there is no need to calculate or find the K-factor. Several software packages and online calculators are available to calculate the setback. These programs are often used with CAD/CAM programs. *[Figure 4-125]*

CHART 204
MINIMUM BEND RADIUS FOR ALUMINUM ALLOYS

Thickness	5052-0 6061-0 5052-H32	7178-0 2024-0 5052-H34 6061-T4 7075-0	6061-T6	7075-T6	2024-T3 2024-T4	2024-T6
.012	.03	.03	.03	.03	.06	.06
.016	.03	.03	.03	.03	.09	.09
.020	.03	.03	.03	.12	.09	.09
.025	.03	.03	.06	.16	.12	.09
.032	.03	.03	.06	.19	.12	.12
.040	.06	.06	.09	.22	.16	.16
.050	.06	.06	.12	.25	.19	.19
.063	.06	.09	.16	.31	.22	.25
.071	.09	.12	.16	.38	.25	.31
.080	.09	.16	.19	.44	.31	.38
.090	.09	.19	.22	.50	.38	.44
.100	.12	.22	.25	.62	.44	.50
.125	.12	.25	.31	.88	.50	.62
.160	.16	.31	.44	1.25	.75	.75
.190	.19	.38	.56	1.38	1.00	1.00
.250	.31	.62	.75	2.00	1.25	1.25
.312	.44	1.25	1.38	2.50	1.50	1.50
.375	.44	1.38	1.50	2.50	1.88	1.88

Bend radius is designated to the inside of the bend. All dimensions are in inches.

Figure 4-124. *Minimum bend radius (from the Raytheon Aircraft Structural Inspection and Repair Manual).*

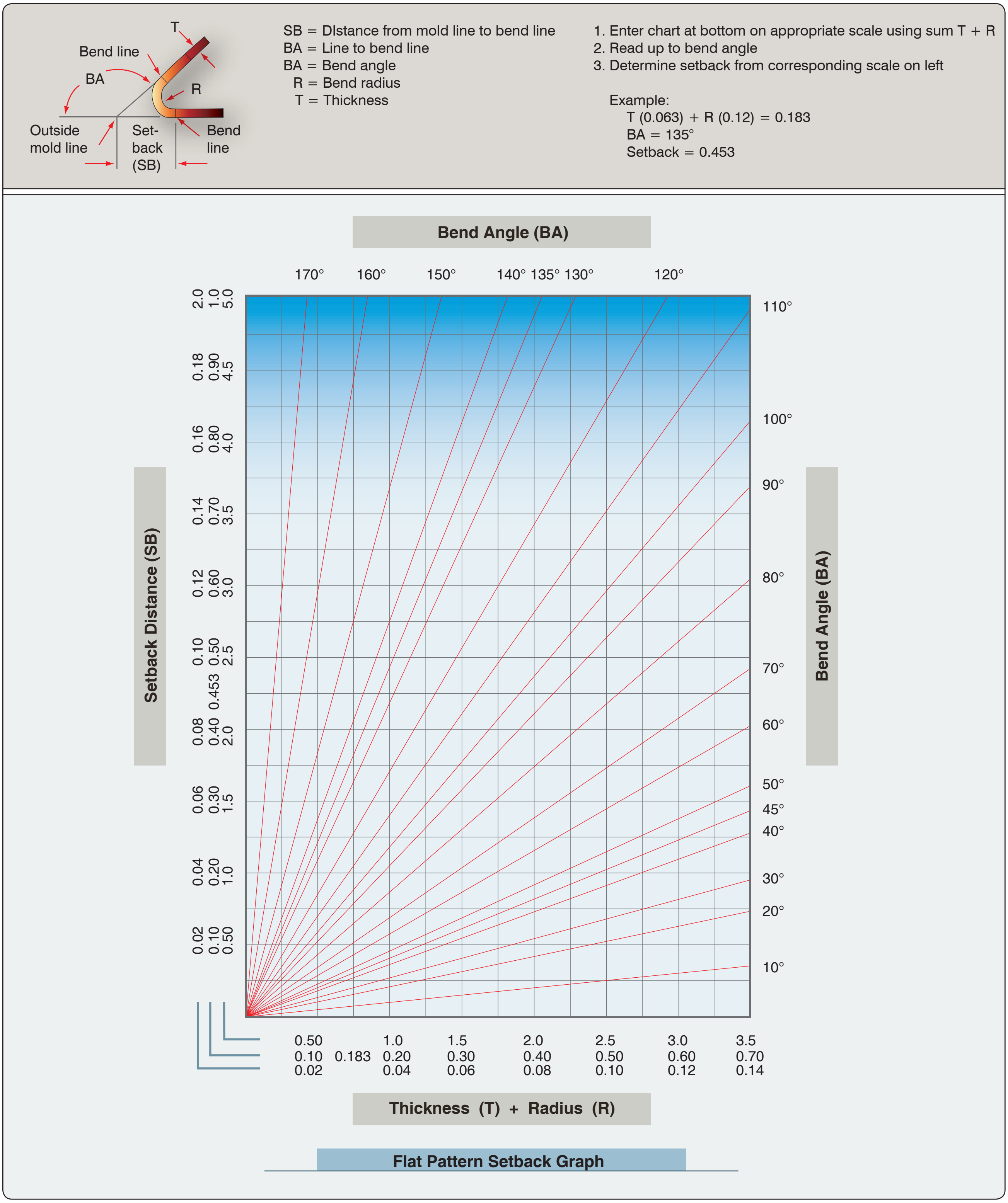

Figure 4-125. *Setback chart.*

- Enter chart at the bottom on the appropriate scale with the sum of the radius and material thickness.
- Read up to the bend angle.
- Find the setback from corresponding scale on the left.

Example:

- Material thickness is 0.063-inch.
- Bend angle is 135°.
- R + T = 0.183-inch.

Find 0.183 at the bottom of the graph. It is found in the middle scale.

- Read up to a bend angle of 135°.
- Locate the setback at the left hand side of the graph in the middle scale (0.435-inch). *[Figure 4-125]*

Step 3: Find the Length of the Flat Line Dimension

The flat line dimension can be found using the formula:

Flat = MLD – SB

MLD = mold line dimension

SB = setback

The flats, or flat portions of the U-channel, are equal to the mold line dimension minus the setback for each of the sides, and the mold line length minus two setbacks for the center flat. Two setbacks need to be subtracted from the center flat because this flat has a bend on either side.

The flat dimension for the sample U-channel is calculated in the following manner:

Flat dimension = MLD – SB

Flat 1 = 1.00-inch – 0.2-inch = 0.8-inch

Flat 2 = 2.00-inch – (2 × 0.2-inch) = 1.6-inch

Flat 3 = 1.00-inch – 0.2-inch = 0.8-inch

Step 4: Find the Bend Allowance

When making a bend or fold in a piece of metal, the bend allowance or length of material required for the bend must be calculated. Bend allowance depends on four factors: degree of bend, radius of the bend, thickness of the metal, and type of metal used.

The radius of the bend is generally proportional to the thickness of the material. Furthermore, the sharper the radius of bend, the less the material that is needed for the bend. The type of material is also important. If the material is soft, it can be bent very sharply; but if it is hard, the radius of bend is greater, and the bend allowance is greater. The degree of bend affects the overall length of the metal, whereas the thickness influences the radius of bend.

Bending a piece of metal compresses the material on the inside of the curve and stretches the material on the outside of the curve. However, at some distance between these two extremes lies a space which is not affected by either force. This is known as the neutral line or neutral axis and occurs at a distance approximately 0.445 times the metal thickness (0.445 × T) from the inside of the radius of the bend. *[Figure 4-126]*

The length of this neutral axis must be determined so that sufficient material can be provided for the bend. This is called the bend allowance. This amount must be added to the overall length of the layout pattern to ensure adequate material for the bend. To save time in calculation of the bend allowance, formulas and charts for various angles, radii of bends, material thicknesses, and other factors have been developed.

Formula 1: Bend Allowance for a 90° Bend

To the radius of bend (R) add ½ the thickness of the metal (½T). This gives R + ½T, or the radius of the circle of the neutral axis. *[Figure 4-127]* Compute the circumference of this circle by multiplying the radius of the neutral line (R + ½T) by 2π (NOTE: $\pi = 3.1416$): $2\pi\,(R + \tfrac{1}{2}T)$. Since a 90° bend is a quarter of the circle, divide the circumference by 4. This gives:

$$\frac{2\pi\,(R + \tfrac{1}{2}T)}{4}$$

This is the bend allowance for a 90° bend. To use the formula for a 90° bend having a radius of ¼ inch for material 0.051-inch thick, substitute in the formula as follows.

$$\text{Bend allowance} = \frac{(2 \times 3.1416)(0.250 + \tfrac{1}{2}(0.051))}{4}$$

$$= \frac{6.2832(0.250 + 0.0255)}{4}$$

$$= \frac{6.2832(0.2755)}{4}$$

$$= 0.4327$$

The bend allowance, or the length of material required for the bend, is 0.4327 or 7⁄16-inch.

Formula 2: Bend Allowance for a 90° Bend

This formula uses two constant values that have evolved over a period of years as being the relationship of the degrees in the bend to the thickness of the metal when determining the bend

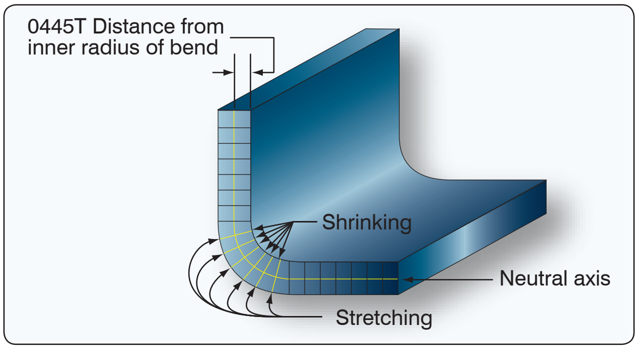

Figure 4-126. *Neutral axis and stresses resulting from bending.*

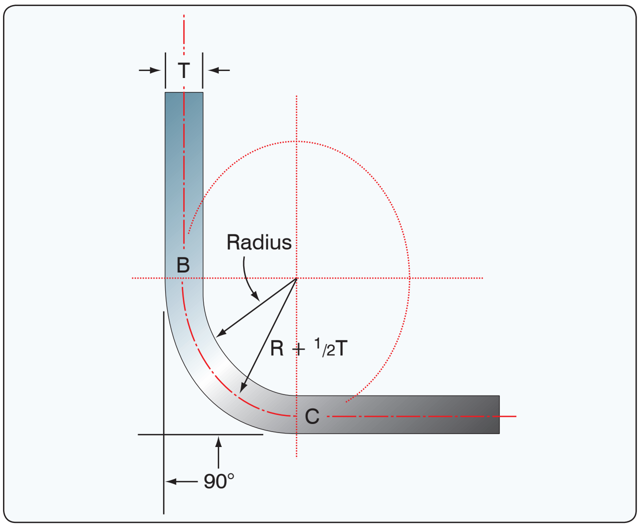

Figure 4-127. *Bend allowance for a 90° bend.*

allowance for a particular application. By experimentation with actual bends in metals, aircraft engineers have found that accurate bending results could be obtained by using the following formula for any degree of bend from 1° to 180°.

Bend allowance = (0.01743R + 0.0078T)N where:

R = the desired bend radius

T = the thickness of the metal

N = number of degrees of bend

To use this formula for a 90° bend having a radius of .16-inch for material 0.040-inch thick, substitute in the formula as follows:

$$\text{Bend allowance} = (0.01743 \times 0.16) + (0.0078 \times 0.040) \times 90 = 0.27 \text{ inches}$$

Use of Bend Allowance Chart for a 90° Bend

In *Figure 4-128,* the radius of bend is shown on the top line, and the metal thickness is shown on the left hand column. The upper number in each cell is the bend allowance for a 90° bend. The lower number in the cell is the bend allowance per 1° of bend. To determine the bend allowance for a 90° bend, simply use the top number in the chart.

Example: The material thickness of the U-channel is 0.040-inch and the bend radius is 0.16-inch.

Reading across the top of the bend allowance chart, find the column for a radius of bend of .156-inch. Now, find the block in this column that is opposite the material thickness (gauge) of 0.040 in the column at the left. The upper number in the cell is (0.273), the correct bend allowance in inches for a 90° bends.

Several bend allowance calculation programs are available online. Just enter the material thickness, radius, and degree of bend and the computer program calculates the bend allowance.

Use of Chart for Other Than a 90° Bend

If the bend is to be other than 90°, use the lower number in the block (the bend allowance for 1°) and compute the bend allowance.

Example:
The L-bracket shown in *Figure 4-129* is made from 2024-T3 aluminum alloy and the bend is 60° from flat. Note that the bend angle in the figure indicates 120°, but that is the number of degrees between the two flanges and not the bend angle from flat. To find the correct bend angle, use the following formula:

Bend Angle = 180° – Angle between flanges

The actual bend is 60°. To find the correct bend radius for a 60° bend of material 0.040-inches thick, use the following procedure.

1. Go to the left side of the table and find 0.040-inch.
2. Go to the right and locate the bend radius of 0.16-inch (0.156-inch).
3. Note the bottom number in the block (0.003034).
4. Multiply this number by the bend angle:
   $0.003034 \times 60 = 0.18204$

Metal Thickness	RADIUS OF BEND, IN INCHES													
	1/32 .031	1/16 .063	3/32 .094	1/8 .125	5/32 .156	3/16 .188	7/32 .219	1/4 .250	9/32 .281	5/16 .313	11/32 .344	3/8 .375	7/16 .438	1/2 .500
.020	.062 .000693	.113 .001251	.161 .001792	.210 .002333	.259 .002874	.309 .003433	.358 .003974	.406 .004515	.455 .005056	.505 .005614	.554 .006155	.603 .006695	.702 .007795	.799 .008877
.025	.066 .000736	.116 .001294	.165 .001835	.214 .002376	.263 .002917	.313 .003476	.362 .004017	.410 .004558	.459 .005098	.509 .005657	.558 .006198	.607 .006739	.705 .007838	.803 .008920
.028	.068 .000759	.119 .001318	.167 .001859	.216 .002400	.265 .002941	.315 .003499	.364 .004040	.412 .004581	.461 .005122	.511 .005680	.560 .006221	.609 .006762	.708 .007862	.805 .007862
.032	.071 .000787	.121 .001345	.170 .001886	.218 .002427	.267 .002968	.317 .003526	.366 .004067	.415 .004608	.463 .005149	.514 .005708	.562 .006249	.611 .006789	.710 .007889	.807 .008971
.038	.075 .00837	.126 .001396	.174 .001937	.223 .002478	.272 .003019	.322 .003577	.371 .004118	.419 .004659	.468 .005200	.518 .005758	.567 .006299	.616 .006840	.715 .007940	.812 .009021
.040	.077 .000853	.127 .001411	.176 .001952	.224 .002493	.273 .003034	.323 .003593	.372 .004134	.421 .004675	.469 .005215	.520 .005774	.568 .006315	.617 .006856	.716 .007955	.813 .009037
.051		.134 .001413	.183 .002034	.232 .002575	.280 .003116	.331 .003675	.379 .004215	.428 .004756	.477 .005297	.527 .005855	.576 .006397	.624 .006934	.723 .008037	.821 .009119
.064		.144 .001595	.192 .002136	.241 .002676	.290 .003218	.340 .003776	.389 .004317	.437 .004858	.486 .005399	.536 .005957	.585 .006498	.634 .007039	.732 .008138	.830 .009220
.072			.198 .002202	.247 .002743	.296 .003284	.436 .003842	.394 .004283	.443 .004924	.492 .005465	.542 .006023	.591 .006564	.639 .007105	.738 .008205	836 .009287
.078			.202 .002249	.251 .002790	.300 .003331	.350 .003889	.399 .004430	.447 .004963	.496 .005512	.546 .006070	.595 .006611	.644 .007152	.745 .008252	.840 .009333
.081			.204 .002272	.253 .002813	.302 .003354	.352 .003912	.401 .004453	.449 .004969	.498 .005535	.548 .006094	.598 .006635	.646 .007176	.745 .008275	.842 .009357
.091			.212 .002350	.260 .002891	.309 .003432	.359 .003990	.408 .004531	.456 .005072	.505 .005613	.555 .006172	.604 .006713	.653 .007254	.752 .008353	.849 .009435
.094			.214 .002374	.262 .002914	.311 .003455	.361 .004014	.410 .004555	.459 .005096	.507 .005637	.558 .006195	.606 .006736	.655 .007277	.754 .008376	.851 .009458
.102				.268 .002977	.317 .003518	.367 .004076	.416 .004617	.464 .005158	.513 .005699	.563 .006257	.612 .006798	.661 .007339	.760 .008439	.857 .009521
.109				.273 .003031	.321 .003572	.372 .004131	.420 .004672	.469 .005213	.518 .005754	.568 .006312	.617 .006853	.665 .008394	.764 .008493	.862 .009575
.125				.284 .003156	.333 .003697	.383 .004256	.432 .004797	.480 .005338	.529 .005678	.579 .006437	.628 .006978	.677 .007519	.776 .008618	.873 .009700
.156					.355 .003939	.405 .004497	.453 .005038	.502 .005579	.551 .006120	.601 .006679	.650 .007220	.698 .007761	.797 .008860	.895 .009942
.188						.417 .004747	.476 .005288	.525 .005829	.573 .006370	.624 .006928	.672 .007469	.721 .008010	.820 .009109	.917 .010191
.250								.568 .006313	.617 .006853	.667 .007412	.716 .007953	.764 .008494	.863 .009593	.961 .010675

Figure 4-128. *Bend allowance.*

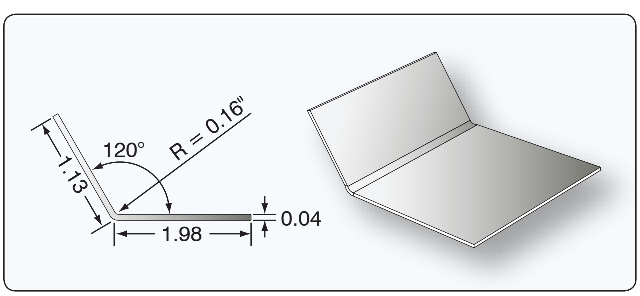

Figure 4-129. *Bend allowance for bends less than 90°.*

Step 5: Find the Total Developed Width of the Material

The total developed width (TDW) can be calculated when the dimensions of the flats and the bend allowance are found. The following formula is used to calculate TDW:

TDW = Flats + (bend allowance × number of bends)

For the U-channel example, this gives:

TDW = Flat 1 + Flat 2 + Flat 3 + (2 × BA)

TDW = 0.8 + 1.6 + 0.8 + (2 × 0.27)

TDW = 3.74-inches

Note that the amount of metal needed to make the channel is less than the dimensions of the outside of the channel (total of mold line dimensions is 4 inches). This is because the metal follows the radius of the bend rather than going from mold line to mold line. It is good practice to check that the calculated TDW is smaller than the total mold line dimensions. If the calculated TDW is larger than the mold line dimensions, the math was incorrect.

Step 6: Flat Pattern Lay Out

After a flat pattern layout of all relevant information is made, the material can be cut to the correct size, and the bend tangent lines can be drawn on the material. *[Figure 4-130]*

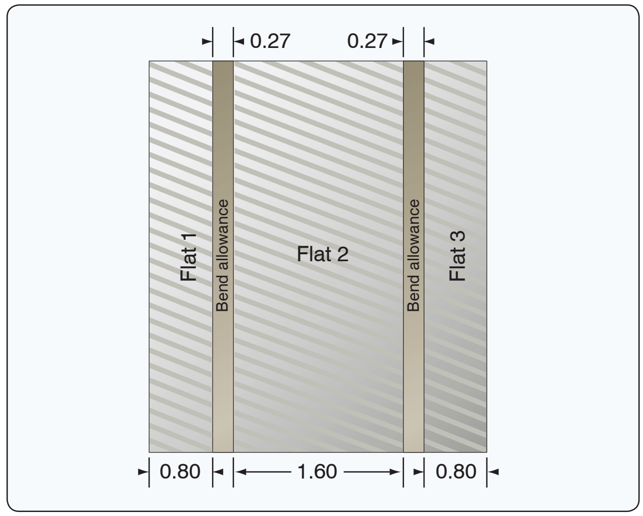

Figure 4-130. *Flat pattern layout.*

Step 7: Draw the Sight Lines on the Flat Pattern

The pattern laid out in *Figure 4-130* is complete, except for a sight line that needs to be drawn to help position the bend tangent line directly at the point where the bend should start. Draw a line inside the bend allowance area that is one bend radius away from the bend tangent line that is placed under the brake nose bar. Put the metal in the brake under the clamp and adjust the position of the metal until the sight line is directly below the edge of the radius bar. *[Figure 4-131]* Now, clamp the brake on the metal and raise the leaf to make the bend. The bend begins exactly on the bend tangent line.

NOTE: A common mistake is to draw the sight line in the middle of the bend allowance area, instead of one radius away from the bend tangent line that is placed under the brake nose bar.

Using a J-Chart To Calculate Total Developed Width

The J-chart, often found in the SRM, can be used to determine bend deduction or setback and the TDW of a flat pattern layout when the inside bend radius, bend angle, and material thickness are known. *[Figure 4-132]* While not as accurate as the traditional layout method, the J-chart provides sufficient information for most applications. The J-chart does not require difficult calculations or memorized formulas because the required information can be found in the repair drawing or can be measured with simple measuring tools.

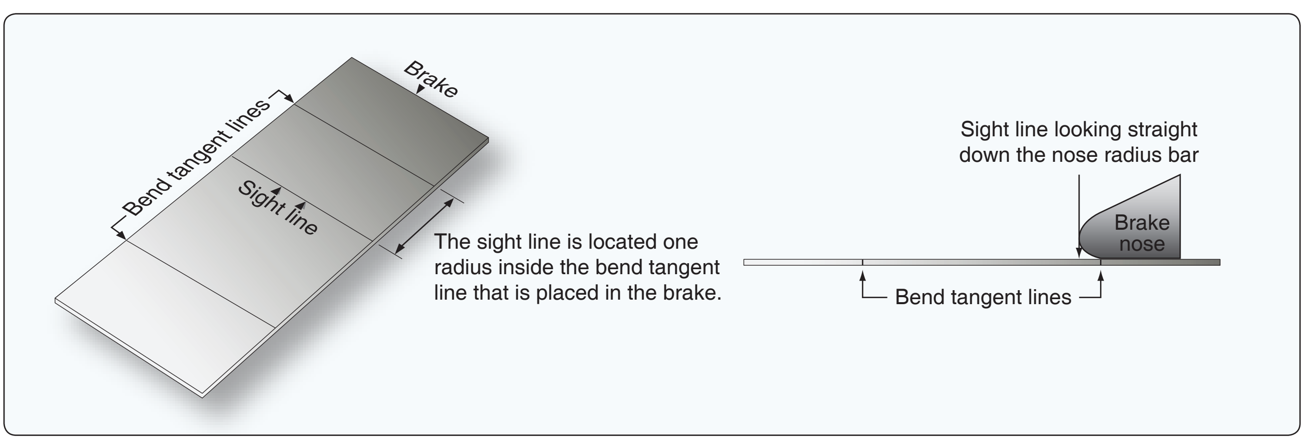

Figure 4-131. *Sight line.*

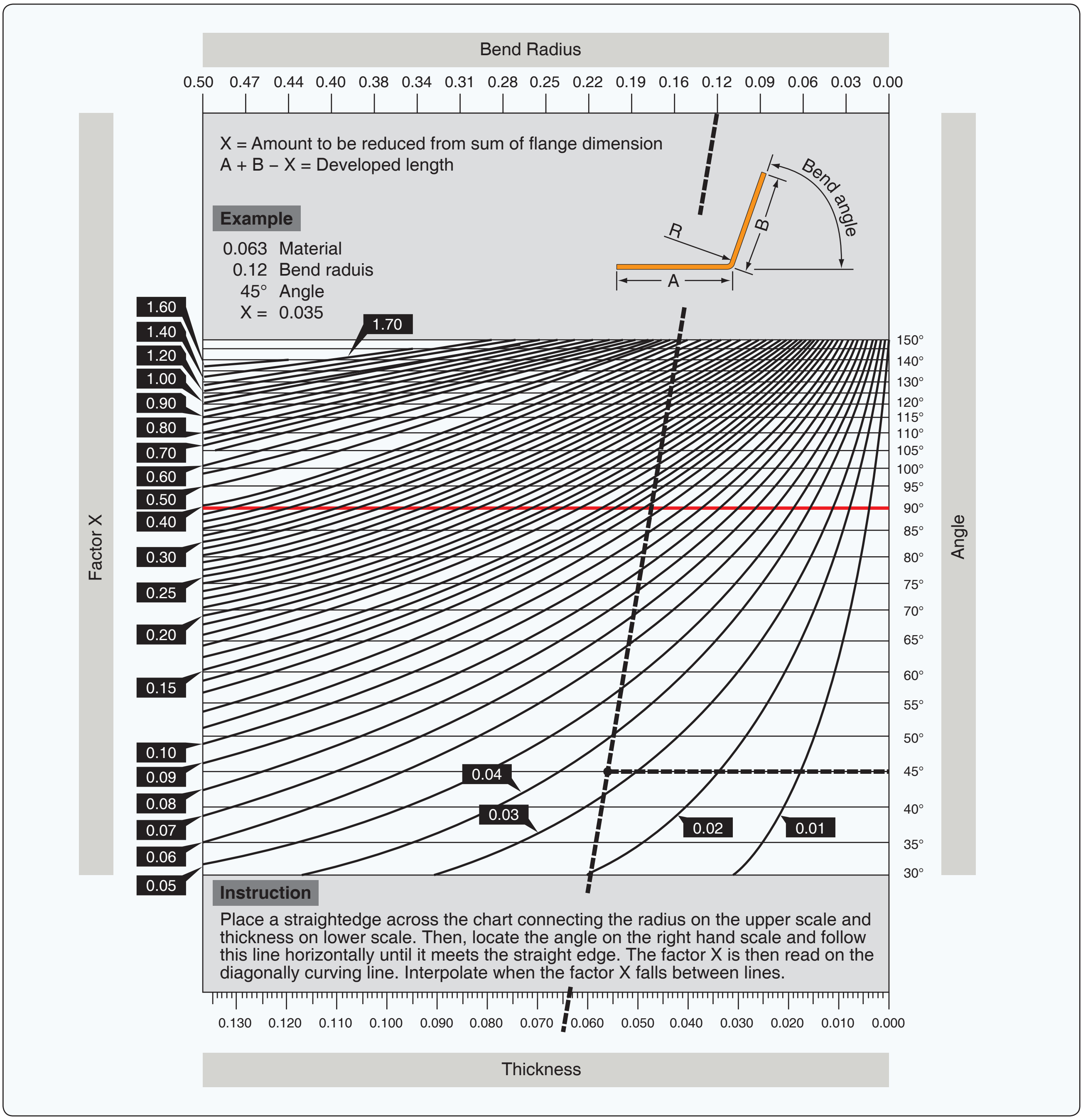

Figure 4-132. *J chart.*

When using the J-chart, it is helpful to know whether the angle is open (greater than 90°) or closed (less than 90°) because the lower half of the J-chart is for open angles and the upper half is for closed angles.

How To Find the Total Developed Width Using a J-Chart

- Place a straightedge across the chart and connect the bend radius on the top scale with the material thickness on the bottom scale. *[Figure 4-132]*
- Locate the angle on the right hand scale and follow this line horizontally until it meets the straight edge.
- The factor X (bend deduction) is then read on the diagonally curving line.
- Interpolate when the X factor falls between lines.
- Add up the mold line dimensions and subtract the X factor to find the TDW.

Example 1

Bend radius = 0.22-inch

Material thickness = 0.063-inch

Bend angle = 90º

ML 1 = 2.00/ML 2 = 2.00

Use a straightedge to connect the bend radius (0.22-inch) at the top of the graph with the material thickness at the bottom (0.063-inch). Locate the 90° angle on the right hand scale and follow this line horizontally until it meets the straightedge. Follow the curved line to the left and find 0.17 at the left side. The X factor in the drawing is 0.17-inch. *[Figure 4-133]*

Total developed width =
(Mold line 1 + Mold line 2) – X factor

Total developed width = (2 + 2) – .17 = 3.83-inches

Example 2

Bend radius = 0.25-inch

Material thickness = 0.050-inch

Bend angle = 45º

ML 1 = 2.00/ML 2 = 2.00

Figure 4-134 illustrates a 135° angle, but this is the angle between the two legs. The actual bend from flat position is 45° (180 – 135 = 45). Use a straightedge to connect the bend radius (0.25-inch) at the top of the graph with the material thickness at the bottom (.050-inch). Locate the 45° angle on the right hand scale and follow this line horizontally until it meets the straight edge. Follow the curved line to the left and find 0.035 at the left side. The X factor in the drawing is 0.035 inch.

Total developed width =
(Mold line 1 + Mold line 2) – X factor

Total developed width = (2 + 2) – .035 = 3.965-inch

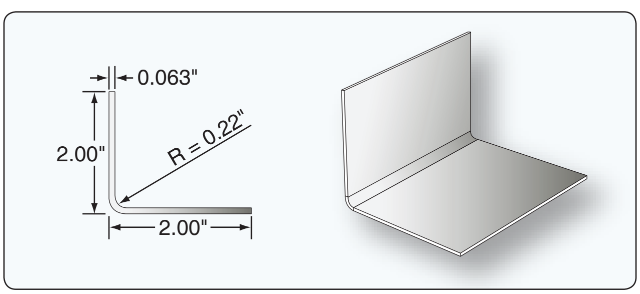

Figure 4-133. *Example 1 of J chart.*

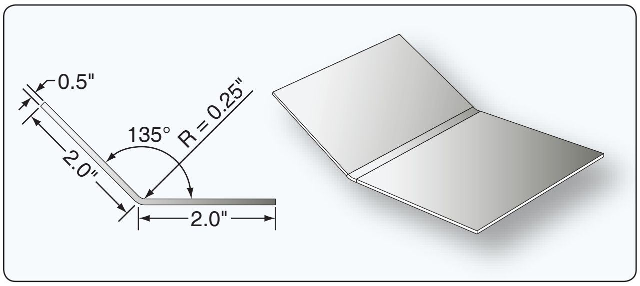

Figure 4-134. *Example 2 of J chart.*

Using a Sheet Metal Brake to Fold Metal

The brake set up for box and pan brakes and cornice brakes is identical. *[Figure 4-135]* A proper set up of the sheet metal brake is necessary because accurate bending of sheet metal depends on the thickness and temper of the material to be formed and the required radius of the part. Any time a different thickness of sheet metal needs to be formed or when a different radius is required to form the part, the operator needs to adjust the sheet metal brake before the brake is used to form the part. For this example, an L-channel made from 2024 –T3 aluminum alloy that is 0.032-inch thick will be bent.

Step 1: Adjustment of Bend Radius

The bend radius necessary to bend a part can be found in the part drawings, but if it is not mentioned in the drawing, consult the SRM for a minimum bend radius chart. This chart lists the smallest radius allowable for each thickness and temper of metal that is normally used. To bend tighter than this radius would jeopardize the integrity of the part. Stresses left in the area of the bend may cause it to fail while in service, even if it does not crack while bending it.

Figure 4-135. *Brake radius nosepiece adjustment.*

The brake radius bars of a sheet metal brake can be replaced with another brake radius bar with a different diameter. *[Figure 4-136]* For example, a 0.032-inch 2024-T3 L channel needs to be bent with a radius of ⅛-inch and a radius bar with a ⅛-inch radius must be installed. If different brake radius bars are not available, and the installed brake radius bar is smaller than required for the part, it is necessary to bend some nose radius shims. *[Figure 4-137]*

If the radius is so small that it tends to crack annealed aluminum, mild steel is a good choice of material. Experimentation with a small piece of scrap material is necessary to manufacture a thickness that increases the radius to precisely 1⁄16-inch or ⅛-inch. Use radius and fillet gauges to check this dimension. From this point on, each additional shim is added to the radius before it. *[Figure 4-138]*

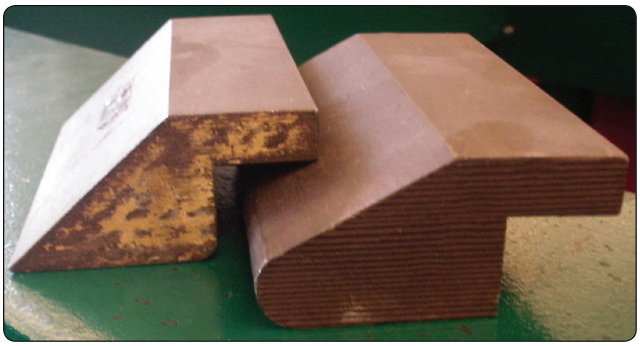

Figure 4-136. *Interchangeable brake radius bars.*

Example: If the original nose was 1⁄16-inch and a piece of .063-inch material (1⁄16-inch) was bent around it, the new outside radius is ⅛-inch. If another .063-inch layer (1⁄16-inch) is added, it is now a 3⁄16-inch radius. If a piece of .032-inch (1⁄32-inch) instead of .063-inch material (1⁄16-inch) is bent around the ⅛-inch radius, a 5⁄32-inch radius results.

Step 2: Adjusting Clamping Pressure

The next step is setting clamping pressure. Slide a piece of the material with the same thickness as the part to be bent under the brake radius piece. Pull the clamping lever toward the operator to test the pressure. This is an over center type clamp and, when properly set, will not feel springy or spongy when pulled to its fully clamped position. The operator must be able to pull this lever over center with a firm pull and have it bump its limiting stops. On some brakes, this adjustment has to be made on both sides of the brake.

Place test strips on the table 3 inches from each end and one in the center between the bed and the clamp, adjust clamp pressure until it is tight enough to prevent the work pieces from slipping while bending. The clamping pressure can be adjusted with the clamping pressure nut. *[Figure 4-139]*

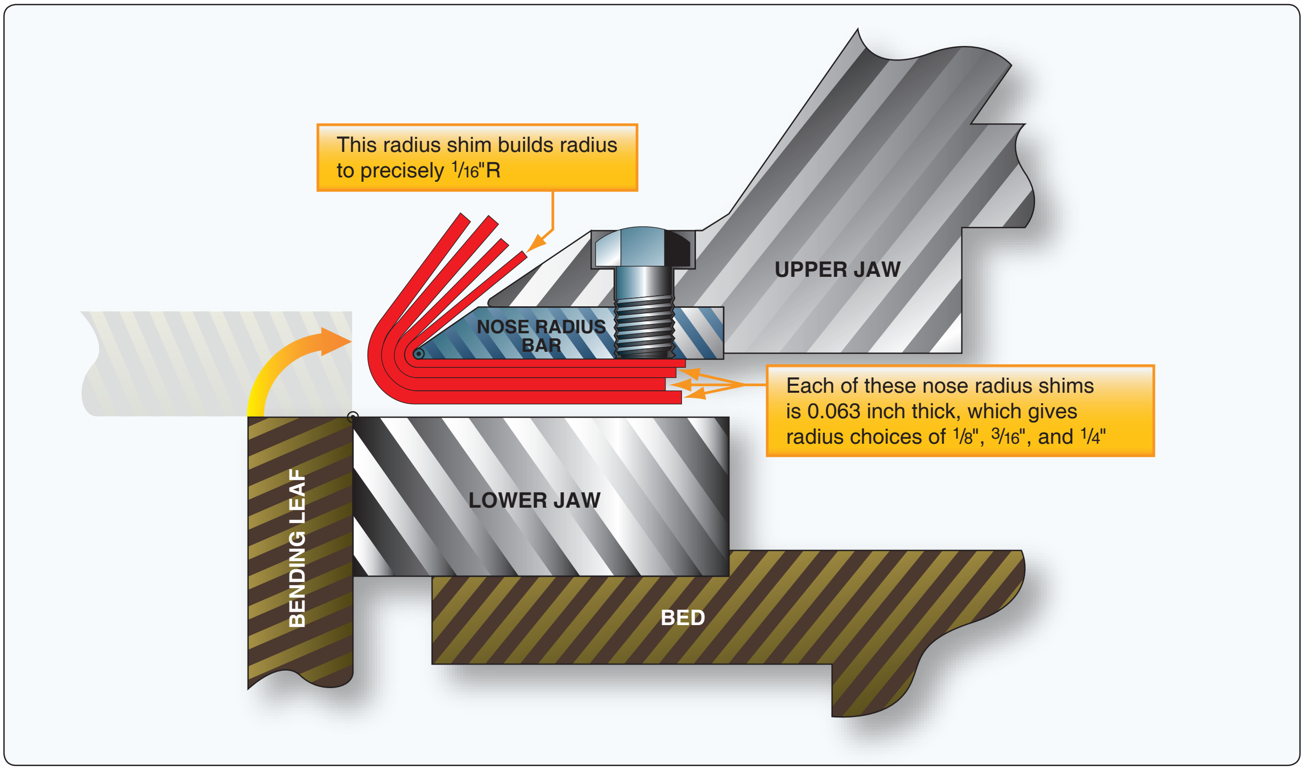

Figure 4-137. *Nose radius shims may be used when the brake radius bar is smaller than required.*

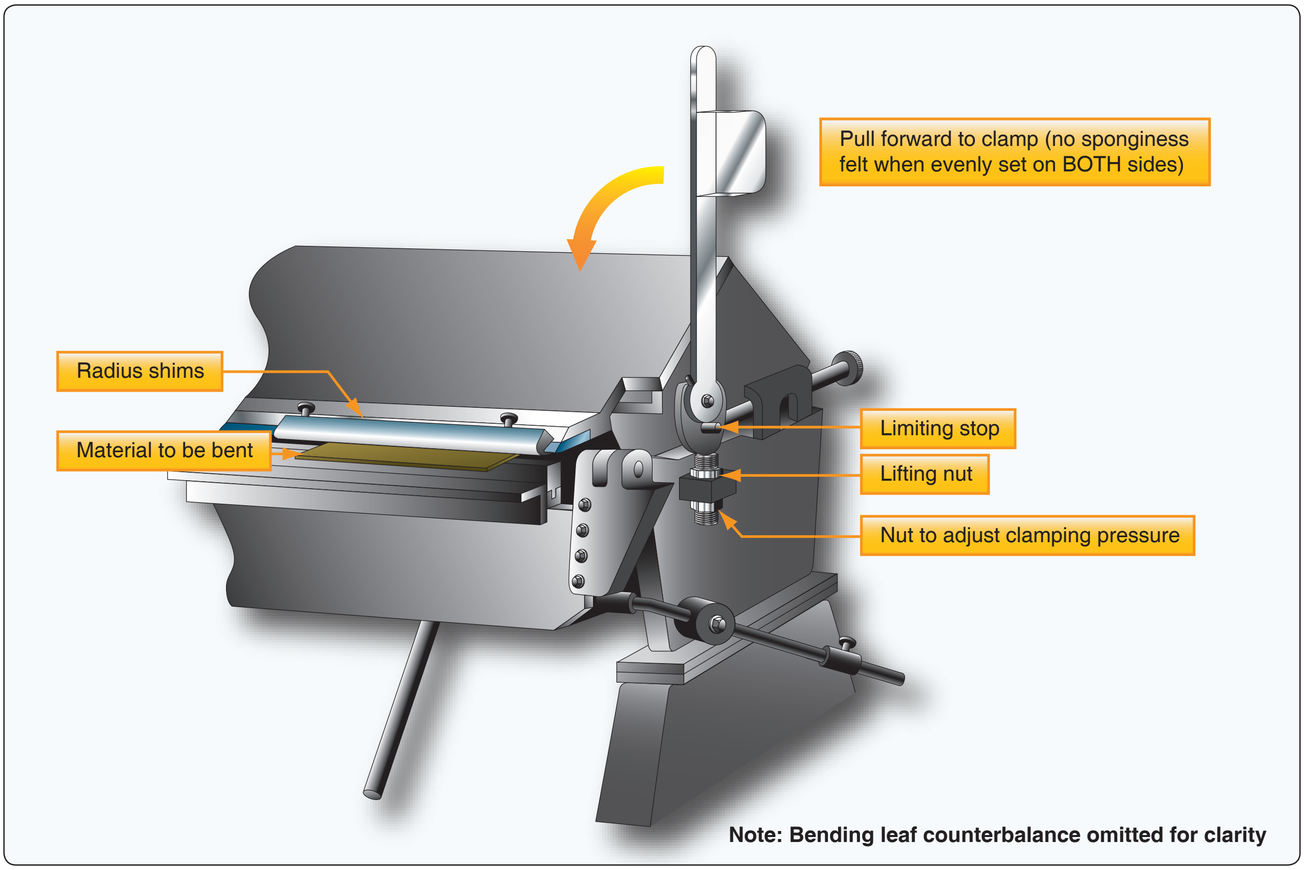

Figure 4-138. *General brake overview including radius shims.*

Step 3: Adjusting the Nose Gap

Adjust the nose gap by turning the large brake nose gap adjustment knobs at the rear of the upper jaw to achieve its proper alignment. *[Figure 4-140]* The perfect setting is obtained when the bending leaf is held up to the angle of the finished bend and there is one material thickness between the bending leaf and the nose radius piece. Using a piece of material the thickness of the part to be bent as a feeler gauge can help achieve a high degree of accuracy. *[Figures 4-140* and *4-141]* It is essential this nose gap be perfect, even across the length of the part to be bent. Check by clamping two test strips between the bed and the clamp 3 inches from each end of the brake. *[Figure 4-142]* Bend 90° *[Figure 4-143]*, remove test strips, and place one on top of

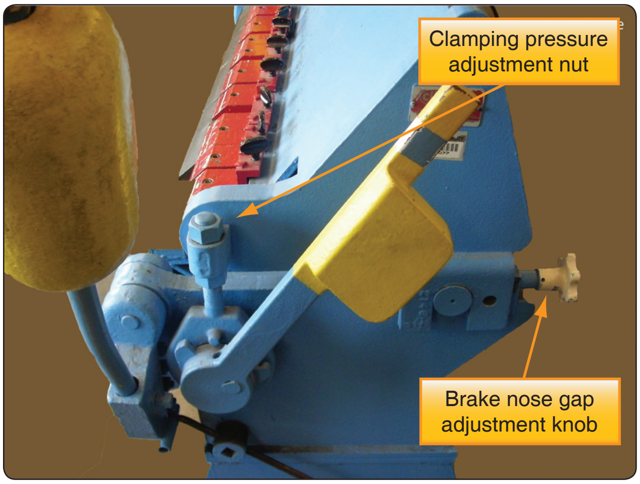

Figure 4-139. *Adjust clamping pressure with the clamping pressure nut.*

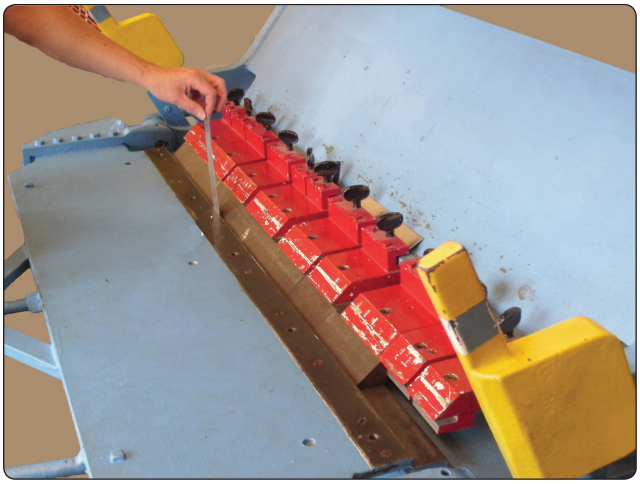

Figure 4-140. *Brake nose gap adjustment with piece of material same thickness as part to be formed.*

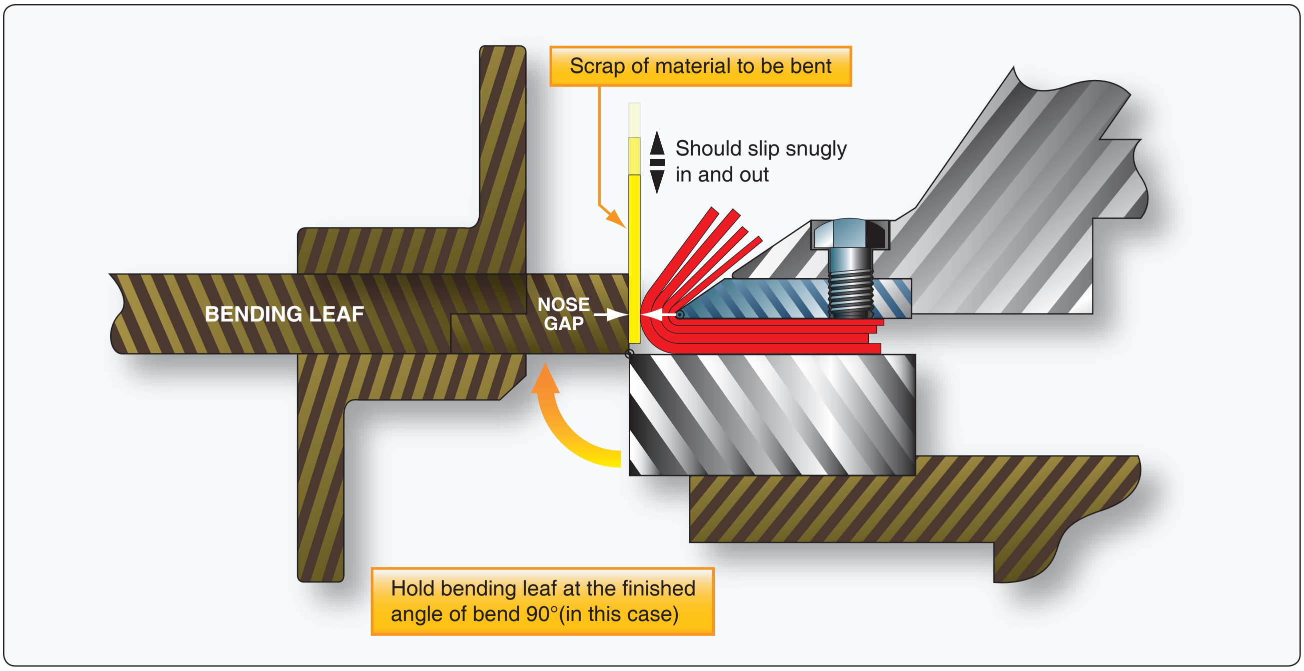

Figure 4-141. *Profile illustration of brake nose gap adjustment.*

Figure 4-142. *Brake alignment with two test strips 3-inches from each end.*

Figure 4-143. *Brake alignment with two test strips bent at 90°.*

the other; they should match. *[Figure 4-144]* If they do not match, adjust the end with the sharper bend back slightly.

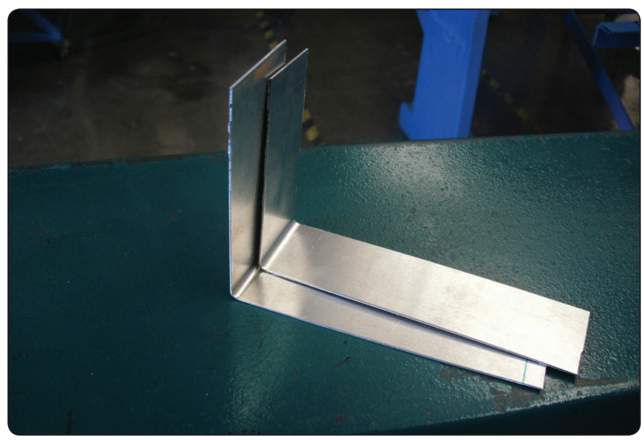

Figure 4-144. *Brake alignment by comparing test strips.*

Folding a Box

A box can be formed the same way as the U-channel described on in the previous paragraphs, but when a sheet metal part has intersecting bend radii, it is necessary to remove material to make room for the material contained in the flanges. This is done by drilling or punching holes at the intersection of the inside bend tangent lines. These holes, called relief holes and whose diameter is approximately twice the bend radius, relieve stresses in the metal as it is bent and prevent the metal from tearing. Relief holes also provide a neatly trimmed corner from which excess material may be trimmed.

The larger and smoother the relief hole is, the less likely it will be that a crack will form in the corner. Generally, the radius of the relief hole is specified on the drawing. A box and pan brake, also called a finger brake, is used to bend the box. Two opposite sides of the box are bent first. Then, the fingers of the brake are adjusted so the folded-up sides ride up in the cracks between the fingers when the leaf is raised to bend the other two sides.

The size of relief holes varies with thickness of the material. They should be no less than ⅛-inch in diameter for aluminum alloy sheet stock up to and including 0.064-inch thick, or 3⁄160-inch in diameter for stock ranging from 0.072-inch to 0.128-inch thickness. The most common method of determining the diameter of a relief hole is to use the radius of bend for this dimension, provided it is not less than the minimum allowance (⅛-inch).

Relief Hole Location

Relief holes must touch the intersection of the inside bend tangent lines. To allow for possible error in bending, make the relief holes extend 1⁄32-inch to 1⁄16-inch behind the inside bend tangent lines. It is good practice to use the intersection of these lines as the center for the holes. The line on the inside of the curve is cut at an angle toward the relief holes to allow for the stretching of the inside flange.

The positioning of the relief hole is important. *[Figure 4-145]* It should be located so its outer perimeter touches the intersection of the inside bend tangent lines. This keeps any material from interfering with the bend allowance area of the other bend. If these bend allowance areas intersected with each other, there would be substantial compressive stresses that would accumulate in that corner while bending. This could cause the part to crack while bending.

Layout Method

Lay out the basic part using traditional layout procedures. This determines the width of the flats and the bend allowance. It is the intersection of the inside bend tangent lines that index the bend relief hole position. Bisect these intersected lines and move outward the distance of the radius of the hole on this line. This is the center of the hole. Drill at this point and finish by trimming off the remainder of the corner material. The trim out is often tangent to the radius and perpendicular to the edge. *[Figure 4-146]* This leaves an open corner. If the corner must be closed, or a slightly longer flange is necessary, then trim out accordingly. If the corner is to be welded, it is necessary to have touching flanges at the corners. The length of the flange should be one material thickness shorter than the finished length of the part so only the insides of the flanges touch.

Open and Closed Bends

Open and closed bends present unique problems that require more calculations than 90° bends. In the following 45° and a 135° bend examples, the material is 0.050-inch thick and the bend radius is 3⁄16-inch.

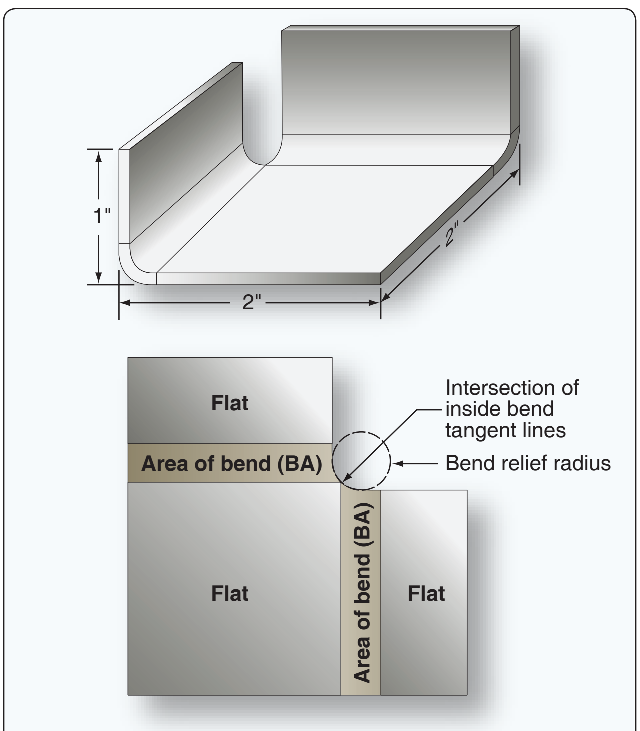

Figure 4-145. *Relief hole location.*

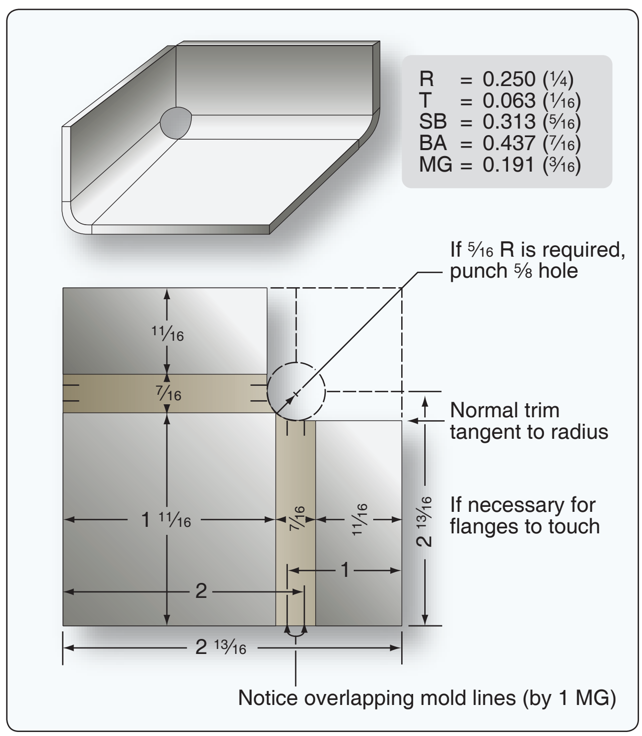

Figure 4-146. *Relief hole layout.*

Open End Bend (Less Than 90°)

Figure 4-147 shows an example for a 45° bend.

1. Look up K-factor in K chart. K-factor for 45° is 0.41421-inch.
2. Calculate setback.

   SB = K(R + T)

   SB = 0.41421-inch(0.1875-inch + 0.050-inch) = 0.098-inch
3. Calculate bend allowance for 45°. Look up bend allowance for 1° of bend in the bend allowance chart and multiply this by 45.

   0.003675-inch × 45 = 0.165-inch
4. Calculate flats.

   Flat = Mold line dimension – SB

   Flat 1 = .77-inch – 0.098-inch = 0.672-inch

   Flat 2 = 1.52-inch – 0.098-inch = 1.422-inch
5. Calculate TDW

   TDW = Flats + Bend allowance

   TDW = 0.672-inch + 1.422-inch + 0.165-inch = 2.259-inch.

Observe that the brake reference line is still located one radius from the bend tangent line.

Closed End Bend (More Than 90°)

Figure 4-148 shows an example of a 135° bend.

1. Look up K-factor in K chart. K-factor for 135° is 2.4142-inch.
2. Calculate SB.

   SB = K(R + T)

   SB = 2.4142-inch(0.1875-inch + 0.050-inch) = 0.57-inch
3. Calculate bend allowance for 135°. Look up bend allowance for 1° of bend in the bend allowance chart and multiply this by 135.

   0.003675-inch × 135 = 0.496-inch

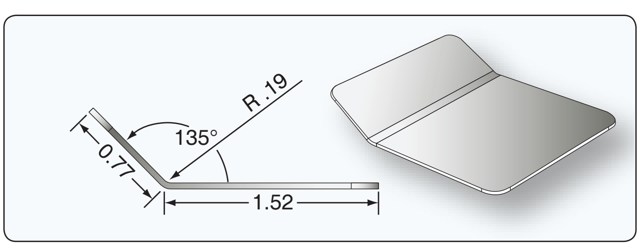

Figure 4-147. *Open bend.*

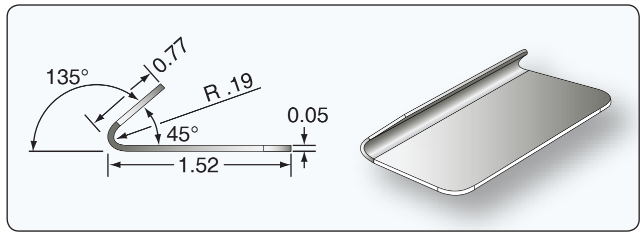

Figure 4-148. *Closed bend.*

4. Calculate flats.

   Flat = Mold line dimension – SB

   Flat 1 = 0.77-inch – 0.57-inch = 0.20-inch

   Flat 2 = 1.52-inch – 0.57-inch = 0.95-inch
5. Calculate TDW.

   TDW = Flats + Bend allowance

   TDW = 0.20-inch + 0.95-inch + 0.496-inch = 1.65-inch

It is obvious from both examples that a closed bend has a smaller TDW than an open-end bend and the material length needs to be adjusted accordingly.

Hand Forming

All hand forming revolves around the processes of stretching and shrinking metal. As discussed earlier, stretching means to lengthen or increase a particular area of metal while shrinking means to reduce an area. Several methods of stretching and shrinking may be used, depending on the size, shape, and contour of the part being formed.

For example, if a formed or extruded angle is to be curved, either stretch one leg or shrink the other, whichever makes the part fit. In bumping, the material is stretched in the bulge to make it balloon, and in joggling, the material is stretched between the joggles. Material in the edge of lightning holes is often stretched to form a beveled reinforcing ridge around them. The following paragraphs discuss some of these techniques.

Straight Line Bends

The cornice brake and bar folder are ordinarily used to make straight bends. Whenever such machines are not available, comparatively short sections can be bent by hand with the aid of wooden or metal bending blocks.

After a blank has been laid out and cut to size, clamp it along the bend line between two wooden forming blocks held in a vise. The wooden forming blocks should have one edge rounded as needed for the desired radius of bend. It should also be curved slightly beyond 90° to allow for spring-back.

Bend the metal that protrudes beyond the bending block to the desired angle by tapping lightly with a rubber, plastic, or rawhide mallet. Start tapping at one end and work back and forth along the edge to make a gradual and even bend. Continue this process until the protruding metal is bent to the desired angle against the forming block. Allow for spring-back by driving the material slightly farther than the actual bend. If a large amount of metal extends beyond the forming blocks, maintain hand pressure against the protruding sheet to prevent it from bouncing. Remove any irregularities by holding a straight block of hardwood edgewise against the bend and striking it with heavy blows of a mallet or hammer. If the amount of metal protruding beyond the bending blocks is small, make the entire bend by using the hardwood block and hammer.

Formed or Extruded Angles

Both formed and extruded types of angles can be curved (not bent sharply) by stretching or shrinking either of the flanges. Curving by stretching one flange is usually preferred since the process requires only a V-block and a mallet and is easily accomplished.

Stretching with V-Block Method

In the stretching method, place the flange to be stretched in the groove of the V-block. *[Figure 4-149]* (If the flange is to be shrunk, place the flange across the V-block.) Using a round, soft-faced mallet, strike the flange directly over the V portion with light, even blows while gradually forcing it downward into the V.

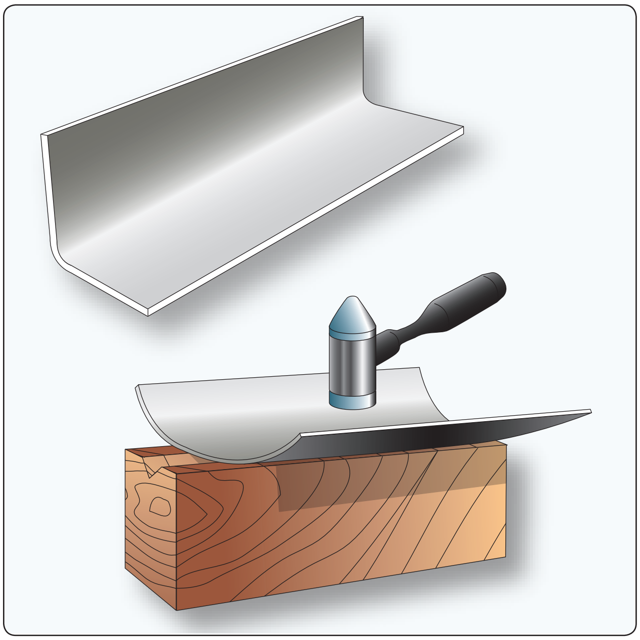

Figure 4-149. *V-block forming.*

Begin at one end of the flange and form the curve gradually and evenly by moving the strip slowly back and forth, distributing the hammer blows at equal spaces on the flange. Hold the strip firmly to keep it from bouncing when hammered. An overly heavy blow buckles the metal, so keep moving the flange across the V-block, but always lightly strike the spot directly above the V.

Lay out a full-sized, accurate pattern on a sheet of paper or plywood and periodically check the accuracy of the curve. Comparing the angle with the pattern determines exactly how the curve is progressing and just where it needs to be increased or decreased. It is better to get the curve to conform roughly to the desired shape before attempting to finish any one portion, because the finishing or smoothing of the angle may cause some other portion of the angle to change shape. If any part of the angle strip is curved too much, reduce the curve by reversing the angle strip on the V-block, placing the bottom flange up, and striking it with light blows of the mallet.

Try to form the curve with a minimum amount of hammering, for excessive hammering work hardens the metal. Work-hardening can be recognized by a lack of bending response or by springiness in the metal. It can be recognized very readily by an experienced worker. In some cases, the part may have to be annealed during the curving operation. If so, be sure to heat treat the part again before installing it on the aircraft.

Shrinking With V-Block and Shrinking Block Methods

Curving an extruded or formed angle strip by shrinking may be accomplished by either the previously discussed V-block method or the shrinking block method. While the V-block is more satisfactory because it is faster, easier, and affects the metal less, good results can be obtained by the shrinking block method.

In the V-block method, place one flange of the angle strip flat on the V-block with the other flange extending upward. Using the process outlined in the stretching paragraphs, begin at one end of the angle strip and work back and forth making light blows. Strike the edge of the flange at a slight angle to keep the vertical flange from bending outward.

Occasionally, check the curve for accuracy with the pattern. If a sharp curve is made, the angle (cross-section of the formed angle) closes slightly. To avoid such closing of the angle, clamp the angle strip to a hardwood board with the hammered flange facing upward using small C-clamps. The