Before a part is attached to the aircraft during either manufacture or repair, it has to be shaped to fit into place. This shaping process is called forming. It may be a simple process, such as making one or two holes for attaching. Or, it may be a complex process, such as making shapes with complex curvatures. Forming, which tends to change the shape or contour of a flat sheet or extruded shape, is accomplished by either stretching or shrinking the material in a certain area to produce curves, flanges, and various irregular shapes. Since the operation involves altering the shape of the stock material, the amount of shrinking and stretching almost entirely depends on the type of material used. Fully annealed (heated and cooled) material can withstand considerably more stretching and shrinking and can be formed at a much smaller bend radius than when it is in any of the tempered conditions.

When aircraft parts are formed at the factory, they are made on large presses or by drop hammers equipped with dies of the correct shape. Factory engineers, who designate specifications for the materials to be used to ensure the finished part has the correct temper when it leaves the machines, plan every part. Factory draftsmen prepare a layout for each part. (Figure 14-1)

Forming processes used on the flight line and those practiced in the maintenance or repair shop cannot duplicate a manufacturer's resources, but similar techniques of factory metal working can be applied in the handcrafting of repair parts.

Forming usually involves the use of extremely light-gauge alloys of a delicate nature that can be readily made useless by coarse and careless workmanship. A formed part may seem outwardly perfect, yet a wrong step in the forming procedure may leave the part in a strained condition. Such a defect may hasten fatigue or may cause sudden structural failure.

Of all the aircraft metals, pure aluminum is the most easily formed. In aluminum alloys, ease of forming varies with the temper condition. Since modern aircraft are constructed chiefly of aluminum and aluminum alloys, this section deals with the procedures for forming aluminum or aluminum alloy parts with a brief discussion of working with stainless steel, magnesium, and titanium.

Most parts can be formed without annealing the metal, but if extensive forming operations, such as deep draws (large folds) or complex curves, are planned, the metal should be in the dead soft or annealed condition. During the forming of some complex parts, operations may need to be stopped and the metal annealed before the process can be continued or completed. For example, alloy 2024 in the "0" condition can be formed into almost any shape by the common forming operations, but it must be heat treated afterward.

**FORMING OPERATIONS AND TERMS**

Forming requires either stretching or shrinking the metal, or sometimes doing both. Other processes used to form metal include bumping, crimping, and folding.

**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 14-2)
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 14-3)

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 by 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 14-4)

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.
**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 14-5* 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 90° 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 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 14-6)*

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.
- **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.
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. (Figure 14-7)

- 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.

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. When using bend 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. (Figure 14-8)

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 14-9 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 14-10)

**USING A FORMULA TO CALCULATE THE SETBACK**

\[ SB = K (R + T) \]

\[ SB = 1 (.16 + .40) = .56 \text{ inches} \]

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 14-10)

- 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°.
- \( R + T = 0.183 \text{ -inch} \)

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