CHAPTER 5
AIRCRAFT STRUCTURAL REPAIRS

Methods of repairing structural portions of an aircraft are numerous and varied, and no set of specific repair patterns has been found which will apply in all cases. Since design loads acting in various structural parts of an aircraft are not always available, the problem of repairing a damaged section must usually be solved by duplicating the original part in strength, kind of material, and dimensions. Some general rules concerning the selection of material and the forming of parts which may be applied universally by the airframe mechanic will be considered in this chapter.

The repairs discussed are typical of those used in aircraft maintenance and are included to introduce some of the operations involved. For exact information about specific repairs, consult the manufacturer's maintenance or service manuals.

BASIC PRINCIPLES OF SHEET METAL REPAIR

The first and one of the most important steps in repairing structural damage is "sizing up" the job and making an accurate estimate of what is to be done. This sizing up includes an estimate of the best type and shape of patch to use; the type, size, and number of rivets needed; and the strength, thickness, and kind of material required to make the repaired member no heavier (or only slightly heavier) and just as strong as the original. Also inspect the surrounding members for evidence of corrosion and load damage so that the required extent of the "cleanout" of the old damage can be estimated accurately. After completing the cleanout, first make the layout of the patch on paper, then transfer it to the sheet stock selected. Then, cut and chamfer the patch, form it so that it matches the contour of that particular area, and apply it.

Maintaining Original Strength

In making any repair, certain fundamental rules must be observed if the original strength of the structure is to be maintained. The patch plate should have a cross-sectional area equal to, or greater than, that of the original damaged section. If the member is subjected to compression or to bending loads, place the splice on the outside of the member to secure a higher resistance to such loads. If the splice cannot be placed on the outside of the member, use material that is stronger than the material used in the original member.

To reduce the possibility of cracks starting from the corners of cutouts, try to make cutouts either circular or oval in shape. Where it is necessary to use a rectangular cutout, make the radius of curvature at each corner no smaller than 1/2 in. Either replace buckled or bent members or reinforce them by attaching a splice over the affected area.

Be sure the material used in all replacements or reinforcements is similar to the material used in the original structure. If it is necessary to substitute an alloy weaker than the original, use material of a heavier gage to give equivalent cross-sectional strength. But never practice the reverse; that is, never substitute a lighter gage stronger material for the original. This apparent inconsistency is because one material can have greater tensile strength than another, but less compressive strength, or vice versa. As an example, the mechanical properties of alloys 2024–T and 2024–T80 are compared in the following paragraph.

If alloy 2024–T were substituted for alloy 2024–T80, the substitute material would have to be thicker unless the reduction in compressive strength was known to be acceptable. On the other hand, if 2024–T80 material were substitued for 2024–T stock, the substitute material would have to be thicker unless the reduction in tensile strength was known to be acceptable. Similarly, the buckling and torsional strength of many sheet-metal and tubular parts are dependent primarily upon the thickness rather than the allowable compressive and shear strengths.

When forming is necessary, be particularly careful, for heat-treated and cold-worked alloys will stand very little bending without cracking. Soft alloys, on the other hand, are easily formed but are not strong enough for primary structures. Strong alloys can be formed in their annealed condition and heat treated to develop their strength before assembling.

In some cases, if the annealed metal is not available, heat the metal, quench it according to regular heat-treating practices, and form it before age-hardening sets in. The forming should be completed in about half an hour after quenching, or the material will become too hard to work.

The size of rivets for any repair can be determined by referring to the rivets used by the manufacturer in the next parallel rivet row inboard on the wing, or forward on the fuselage. Another method of determining the size of rivets to be used is to multiply the thickness of the skin by three and use the next larger size rivet corresponding to that figure. For example, if the skin thickness is 0.040-in., multiply 0.040 by 3, which equals 0.120; use the next larger size rivet, 1/8 in. (0.125 in.).

All repairs made on structural parts of aircraft require a definite number of rivets on each side of the break to restore the original strength. This number varies according to the thickness of the material being repaired and the size of the damage. The number of rivets or bolts required can be determined by referring to a similar splice made by the manufacturer, or by using the following rivet formula:

$$\text{Number of rivets required on each side of the break} = \frac{L \times T \times 75{,}000}{S \text{ or } B}.$$

The number of rivets to be used on each side of the break is equal to the length of the break (L) times the thickness of the material (T) times 75,000, divided by the shear strength or bearing strength (S or B) of the material being repaired, whichever is the smaller of the two.

The length of the break is measured perpendicular to the direction of the general stress running through the damaged area.

The thickness of the material is the actual thickness of the piece of material being repaired and is measured in thousandths of an inch.

The 75,000 used in the formula is an assumed stress load value of 60,000 p.s.i. increased by a safety factor of 25%. It is a constant value.

Shear strength is taken from the charts shown in figure 5–1. It is the amount of force required to cut a rivet holding together two or more sheets of material. If the rivet is holding two parts, it is under single shear; if it is holding three sheets or parts, it is under double shear. To determine the shear strength, the diameter of the rivet to be used must be known. This is determined by multiplying the thickness of the material by three. For example, material thickness 0.040 multiplied by 3 equals 0.120; the rivet selected would be 1/8 in. (0.125 in.) in diameter.

Bearing strength is a value taken from the chart shown in figure 5–2 and is the amount of tension required to pull a rivet through the edge of two sheets riveted together, or to elongate the hole. The diameter of the rivet to be used and the thickness of material being riveted must be known to use the bearing strength chart. The diameter of the rivet would be the same as that used when determining the shear strength value. Thickness of material would be that of the material being repaired.

Example:

Using the formula, determine the number of 2117–T rivets needed to repair a break 2-1/4 in. long in material 0.040-in. thick:

$$\text{Number rivets per side} = \frac{L \times T \times 75{,}000}{S \text{ or } B}.$$

*Single-Shear Strength of Aluminum-Alloy Rivets (Pounds)									
Composition of Rivet (Alloy)	Ultimate Strength of Rivet Metal (Pounds Per Square Inch)	Diameter of Rivet (Inches)							
		1/16	3/32	1/8	5/32	3/16	1/4	5/16	3/8
2117 T	27,000	83	186	331	518	745	1,325	2,071	2,981
2017 T	30,000	92	206	368	573	828	1,472	2,300	3,313
2024 T	35,000	107	241	429	670	966	1,718	2,684	3,865
*Double-shear strength is found by multiplying the above values by 2.									

FIGURE 5–1. Single shear strength chart.

Thickness of Sheet (Inches)	Diameter of Rivet (Inches)							
	1/16	3/32	1/8	5/32	3/16	1/4	5/16	3/8
0.014	71	107	143	179	215	287	358	430
.016	82	123	164	204	246	328	410	492
.018	92	138	184	230	276	369	461	553
.020	102	153	205	256	307	410	412	615
.025	128	192	256	320	284	512	640	768
.032	164	245	328	409	492	656	820	984
.036	184	276	369	461	553	738	922	1,107
.040	205	307	410	512	615	820	1,025	1,230
.045	230	345	461	576	691	922	1,153	1,383
.051	261	391	522	653	784	1,045	1,306	1,568
.064		492	656	820	984	1,312	1,640	1,968
.072		553	738	922	1,107	1,476	1,845	2,214
.081		622	830	1,037	1,245	1,660	2,075	2,490
.091		699	932	1,167	1,398	1,864	2,330	2,796
.102		784	1,046	1,307	1,569	2,092	2,615	3,138
.125		961	1,281	1,602	1,922	2,563	3,203	3,844
.156		1,198	1,598	1,997	2,397	3,196	3,995	4,794
.188		1,445	1,927	2,409	2,891	3,854	4,818	5,781
.250		1,921	2,562	3,202	3,843	5,125	6,405	7,686
.313		2,405	3,208	4,009	4,811	6,417	7,568	9,623
.375		2,882	3,843	4,803	5,765	7,688	9,068	11,529
.500		3,842	5,124	6,404	7,686	10,250	12,090	15,372

FIGURE 5–2. Bearing strength chart (pounds).

Given:

L = 2-1/4 (2.25) in.
T = 0.040 in.
Size of rivet: 0.040 x 3 = 0.120, so rivet must be 1/8 in. or 0.125.
S = 331 (from the shear strength chart).
B = 410 (from the bearing strength chart).

(Use S to find number of rivets per side as it is smaller than B.)

Substituting in the formula:

$$\frac{2.25 \times 0.040 \times 75{,}000}{331} = \frac{6{,}750}{331} = 20.39 \text{ (or 21) rivets/side.}$$

Since any fraction must be considered as a whole number, the actual number of rivets required would be 21 for each side, or 42 rivets for the entire repair.

Maintaining Original Contour

Form all repairs in such a manner that they will fit the original contour perfectly. A smooth contour is especially desirable when making patches on the smooth external skin of high-speed aircraft.

Keeping Weight to a Minimum

Keep the weight of all repairs to a minimum. Make the size of the patches as small as practicable and use no more rivets than are necessary. In many cases, repairs disturb the original balance of the structure. The addition of excessive weight in each repair may unbalance the aircraft so much that it will require adjustment of the trim-and-balance tabs. In areas such as the spinner on the propeller, a repair will require application of balancing patches so that a perfect balance of the propeller assembly can be maintained.

GENERAL STRUCTURAL REPAIR

Aircraft structural members are designed to perform a specific function or to serve a definite purpose. The prime objective of aircraft repair is to restore damaged parts to their original condition. Very often, replacement is the only way in which this can be done effectively. When repair of a damaged part is possible, first study the part carefully so that its purpose or function is fully understood.

Strength may be the principal requirement in the repair of certain structures, while others may need entirely different qualities. For example, fuel tanks and floats must be protected against leakage; but cowlings, fairings, and similar parts must have such properties as neat appearance, streamlined shape, and accessibility. The function of any damaged part must be carefully determined so that the repair will meet the requirements.

INSPECTION OF DAMAGE

When visually inspecting damage, remember that there may be other kinds of damage than that caused by impact from foreign objects or collision. A rough landing may overload one of the landing gear, causing it to become sprung; this would be classified as load damage. During inspection and "sizing up of the repair job," consider how far the damage caused by the sprung shock strut extends to supporting structural members.

A shock occurring at one end of a member will be transmitted throughout its length; therefore, inspect closely all rivets, bolts, and attaching structures along the complete member for any evidence of damage. Make a close examination for rivets that have partially failed and for holes which have been elongated.

Another kind of damage to watch for is that caused by weathering or corrosion. This is known as corrosion damage. Corrosion damage of aluminum material is usually detected by the white crystalline deposits that form around loose rivets, scratches, or any portion of the structure that may be a natural spot for moisture to settle.

Definition of Defects

Types of damage and defects which may be observed on parts of this assembly are defined as follows:

Brinelling—Occurrence of shallow, spherical depressions in a surface, usually produced by a part having a small radius in contact with the surface under high load.

Burnishing—Polishing of one surface by sliding contact with a smooth, harder surface. Usually no displacement nor removal of metal.

Burr—A small, thin section of metal extending beyond a regular surface, usually located at a corner or on the edge of a bore or hole.

Corrosion—Loss of metal from the surface by chemical or electrochemical action. The corrosion products generally are easily removed by mechanical means. Iron rust is an example of corrosion.

Crack—A physical separation of two adjacent portions of metal, evidenced by a fine or thin line across the surface, caused by excessive stress at that point. It may extend inward from the surface from a few thousandths inch to completely through the section thickness.

Cut—Loss of metal, usually to an appreciable depth over a relatively long and narrow area, by mechanical means, as would occur with the use of a saw blade, chisel or sharp-edged stone striking a glancing blow.

Dent—Indentation in a metal surface produced by an object striking with force. The surface surrounding the indentation will usually be slightly upset.

Erosion—Loss of metal from the surface by mechanical action of foreign objects, such as grit or fine sand. The eroded area will be rough and may be lined in the direction in which the foreign material moved relative to the surface.

Chattering—Breakdown or deterioration of metal surface by vibratory or "chattering" action. Usually no loss of metal or cracking of surface but generally showing similar appearance.

Galling—Breakdown (or build-up) of metal surfaces due to excessive friction between two parts having relative motion. Particles of the softer metal are torn loose and "welded" to the harder.

Gouge—Grooves in, or breakdown of, a metal surface from contact with foreign material under heavy pressure. Usually indicates metal loss but may be largely displacement of material.

Inclusion—Presence of foreign or extraneous material wholly within a portion of metal. Such material is introduced during the manufacture of rod, bar or tubing by rolling or forging.

Nick—Local break or notch on edge. Usually displacement of metal rather than loss.

Pitting—Sharp, localized breakdown (small, deep cavity) of metal surface, usually with defined edges.

Scratch—Slight tear or break in metal surface from light, momentary contact by foreign material.

Score—Deeper (than scratch) tear or break in metal surface from contact under pressure. May show discoloration from temperature produced by friction.

Stain—A change in color, locally causing a noticeably different appearance from the surrounding area.

Upsetting—A displacement of material beyond the normal contour or surface (a local bulge or bump). Usually indicates no metal loss.

CLASSIFICATION OF DAMAGE

Damages may be grouped into four general classes. In many cases, the availability or lack of repair materials and time are the most important factors in determining whether a part should be repaired or replaced.

Negligible Damage

Damage which does not affect the structural integrity of the member involved, or damage which can be corrected by a simple procedure without placing flight restrictions on the aircraft, is classified as negligible damage. Small dents, scratches, cracks, or holes that can be repaired by smoothing, sanding, stop drilling, or hammering out, or otherwise repaired without the use of additional materials, fall in this classification.

Damage Repairable by Patching

Damage repairable by patching is any damage exceeding negligible damage limits which can be repaired by bridging the damaged area of a component with a material splice. The splice or patch material used in internal riveted and bolted repairs is normally the same type of material as the damaged part, but one gage heavier. In a patch repair, filler plates of the same gage and type of material as that in the damaged component may be used for bearing purposes or to return the damaged part to its original contour.

Damage Repairable by Insertion

Damage which can be repaired by cutting away the damaged section and replacing it with a like section, then securing the insertion with splices at each end is classified as damage repairable by insertion.

Damage Necessitating Replacement of Parts

Replacement of an entire part is considered when one or more of the following conditions exist:

(1) When a complicated part has been extensively damaged.

(2) When surrounding structure or inaccessibility makes repair impractical.

(3) When damaged part is relatively easy to replace.

(4) When forged or cast fittings are damaged beyond the negligible limits.

STRESSES IN STRUCTURAL MEMBERS

Forces acting on an aircraft, whether it is on the ground or in flight, cause pulling, pushing, or twisting within the various members of the aircraft structure. While the aircraft is on the ground, the weight of the wings, fuselage, engines, and empennage causes forces to act downward on the wing and stabilizer tips, along the spars and stringers, and on the bulkheads and formers. These forces are passed on from member to member causing bending, twisting, pulling, compression, and shearing.

As the aircraft takes off, most of the forces in the fuselage continue to act in the same direction; but because of the motion of the aircraft, they increase in intensity. The forces on the wingtips and the wing surfaces, however, reverse direction and instead of being downward forces of weight, they become upward forces of lift. The forces of lift are exerted first against the skin and stringers, then are passed on to the ribs, and finally are transmitted through the spars to be distributed through the fuselage.

The wings bend upward at their ends and may flutter slightly during flight. This wing bending cannot be ignored by the manufacturer in the original design and construction, and cannot be ignored during maintenance. It is surprising how an aircraft structure composed of structural members and skin rigidly riveted or bolted together, such as a wing, can bend or act so much like a leaf spring.

The five types of stresses (figure 5–3) in an aircraft are described as tension, compression, shear, bending, and torsion (or twisting). The first three are commonly called basic stresses, the last two, combination stresses. Stresses usually act in combinations rather than singly.

Tension

Tension (or tensile stress) is the force per unit area tending to stretch a structural member. The strength of a member in tension is determined on the basis of its gross area (or total area), but calculations involving tension must take into consideration the net area of the member. Net area is defined as the gross area minus that removed by drilling holes or by making other changes in the section. Placing rivets or bolts in holes makes no appreciable difference in added strength, as the rivets or bolts will not transfer tensional loads across holes in which they are inserted.

Compression

Compression (or compressive stress) is the force per unit area which tends to shorten (or compress) a structural member at any cross section. Under a compressive load, an undrilled member will be stronger than an identical member with holes drilled through it. However, if a plug of equivalent or stronger material is fitted tightly in a drilled member, it will transfer compressive loads across the hole, and the member will carry approximately as large a load as if the hole were not there. Thus, for compressive loads, the gross or total area may be used in determining the stress in a member if all holes are tightly plugged with equivalent or stronger material.

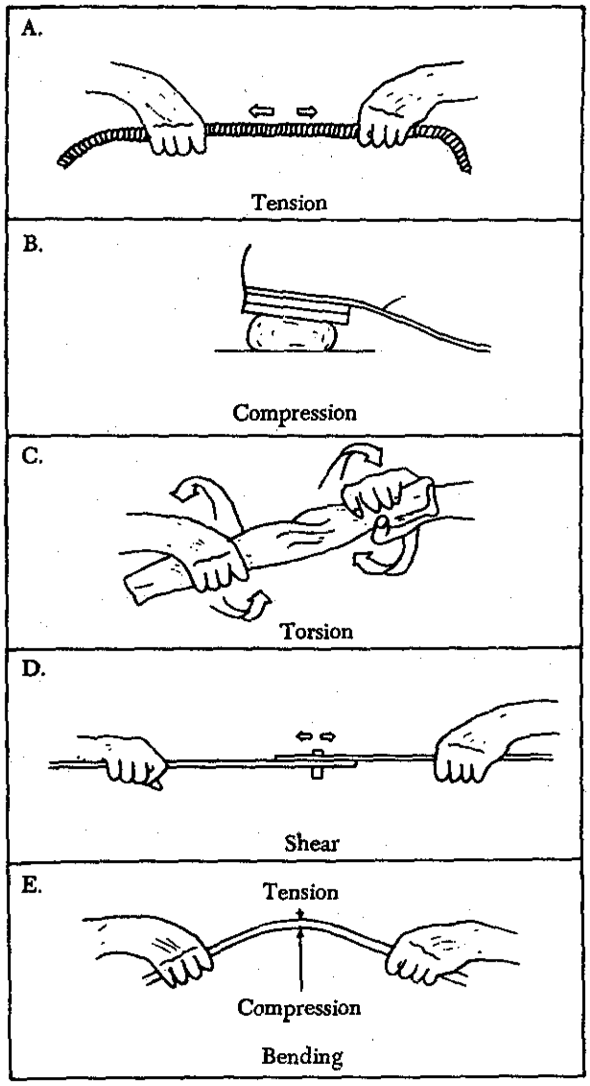


FIGURE 5–3. Five stresses acting on an aircraft.

Shear

Shear is the force per unit area which causes adjacent particles of material to slide past each other. The term "shear" is used because it is a sideways stress of the type that is put on a piece of paper or a sheet of metal when it is cut with a pair of shears. Shear stress concerns the aviation mechanic chiefly from the standpoint of rivet and bolt applications, particularly when attaching sheet stock, because if a rivet used in a shear application gives way, the riveted or bolted parts are pushed sideways.

Bending

Bending (or beam stress) is actually a combination of two forces acting upon a structural member at one or more points. In figure 5-3 note that the bending stress causes a tensile stress to act on the upper half of the beam and a compressive stress on the lower half. These stresses act oppositely on the two sides of the center line of the member, which is called the neutral axis. Since these forces acting in opposite directions are next to each other at the neutral axis, the greatest shear stress occurs along this line, and none exists at the extreme upper or lower surfaces of the beam.

Torsion

Torsion (or twisting stress) is the force which tends to twist a structural member. The stresses arising from this action are shear stresses caused by the rotation of adjacent planes past each other around a common reference axis at right angles to these planes. This action may be illustrated by a rod fixed solidly at one end and twisted by a weight placed on a lever arm at the other, producing the equivalent of two equal and opposite forces acting on the rod at some distance from each other. A shearing action is set up all along the rod, with the center line of the rod representing the neutral axis.

SPECIAL TOOLS AND DEVICES FOR SHEET METAL

The airframe mechanic does a lot of work with special tools and devices that have been developed to make his work faster, simpler, and better. These special tools and devices include dollies and stakes and various types of blocks and sandbags used as support in the bumping process.

Dollies and Stakes

Sheet metal is often formed or finished (planished) over variously shaped anvils called dollies and stakes. These are used for forming small, odd-shaped parts, or for putting on finishing touches for which a large machine may not be suited. Dollies are meant to be held in the hand, whereas stakes are designed to be supported by a flat cast iron bench plate fastened to the workbench (figure 5-4).

Most stakes have machined, polished surfaces which have been hardened. Do not use stakes to back up material when chiseling, or when using any similar cutting tool because this will deface the surface of the stake and make it useless for finish work.

V-Blocks

V-blocks made of hardwood are widely used in airframe metalwork for shrinking and stretching metal, particularly angles and flanges. The size of the block depends on the work being done and on personal preference. Although any type of hardwood is suitable, maple and ash are recommended for best results when working with aluminum alloys.

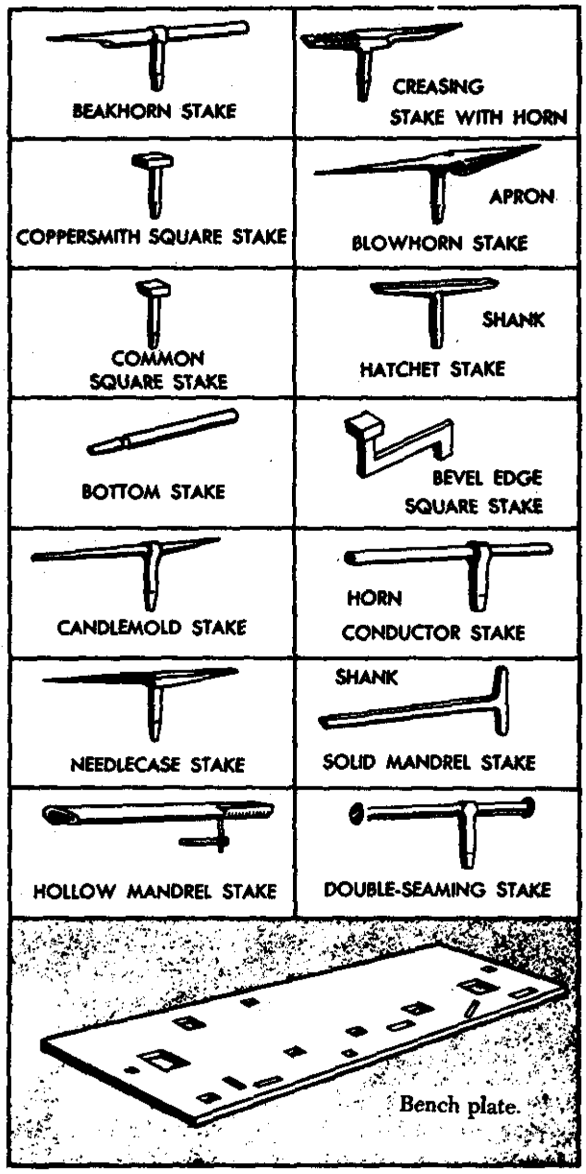


FIGURE 5-4. Bench plate and stakes.

Hardwood Form Blocks

Hardwood form blocks can be constructed to duplicate practically any aircraft structural or nonstructural part. The wooden block or form is shaped to the exact dimensions and contour of the part to be formed.

Shrinking Blocks

A shrinking block consists of two metal blocks and some device for clamping them together. One block forms the base, and the other is cut away to provide space where the crimped material can be hammered. The legs of the upper jaw clamp the material to the base block on each side of the crimp so that the material will not creep away but will remain stationary while the crimp is hammered flat (being shrunk). This type of crimping block is designed to be held in a bench vise.

Shrinking blocks can be made to fit any specific need. The basic form and principle remain the same, even though the blocks may vary considerably in size and shape.

Sandbags

A sandbag is generally used as a support during the bumping process. A serviceable bag can be made by sewing heavy canvas or soft leather to form a bag of the desired size, and filling it with sand which has been sifted through a fine mesh screen.

Before filling canvas bags with sand, use a brush to coat the inside of it with softened paraffin or beeswax, which forms a sealing layer and prevents the sand from working through the pores of the canvas.

Holding Devices

Vises and clamps are tools used for holding materials of various kinds on which some type of operation is being performed. The type of operation being performed and the type of metal being used determine the holding device to be used.

The most commonly used vises are shown in figure 5–5; the machinist's vise has flat jaws and usually a swivel base, whereas the utility bench vise has scored, removable jaws and an anvil-faced back jaw. This vise will hold heavier material than the machinist's vise and will also grip pipe or rod firmly. The back jaw can be used for an anvil if the work being done is light.

The carriage clamp, or C-clamp, as it is commonly called, is shaped like a large C and has three main parts: (1) The threaded screw, (2) the jaw, and (3) the swivel head. The swivel plate, which is at the bottom of the screw, prevents the end from turning directly against the material being clamped. Although C-clamps vary in size from 2 in. upward, their function is always that of clamping or holding

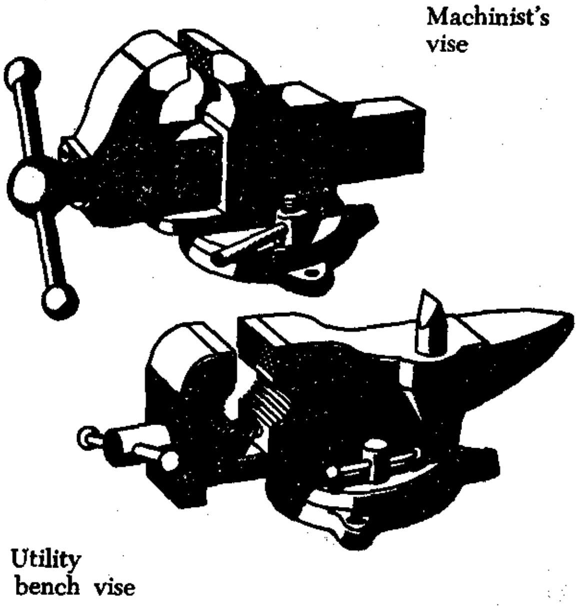


FIGURE 5–5. Vises.

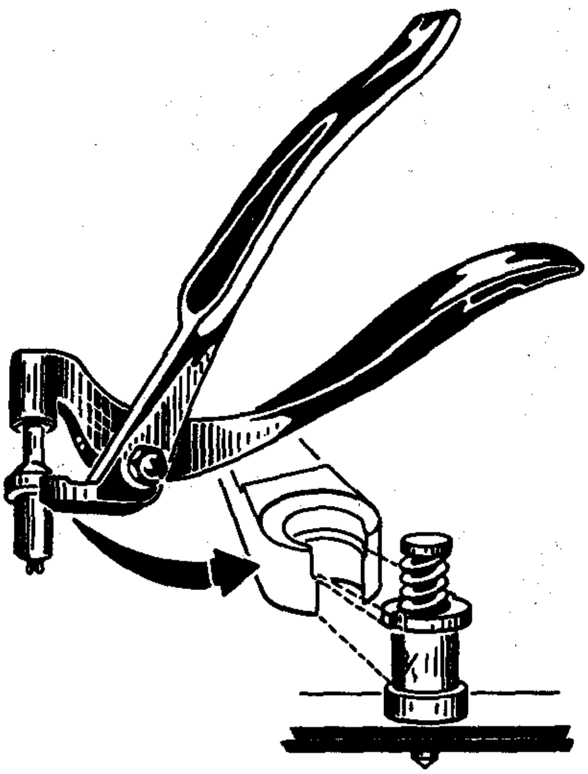


FIGURE 5–6. Cleco fastener.

The shape of the C-clamp allows it to span obstructions near the edge of a piece of work. The greatest limitation in the use of the carriage clamp is its tendency to spring out of shape. It should never be tightened more than hand-tight.

The most commonly used sheet-metal holder is the Cleco fastener (figure 5–6). It is used to keep drilled parts made from sheet stock pressed tightly together. Unless parts are held tightly together they will separate while being riveted.

This type of fastener is available in six different sizes: 3/32-, 1/8-, 5/32-, 3/16-, 1/4-, and 3/8-in. The size is stamped on the fastener. Special pliers are used to insert the fastener in a drilled hole. One pair of pliers will fit the six different sizes.

Sheet-metal screws are sometimes used as temporary holders. The metal sheets must be held tightly together before installing these screws, since the self-tapping action of the threads tends to force the sheets apart. Washers placed under the heads of the screws keep them from marring or scratching the metal.

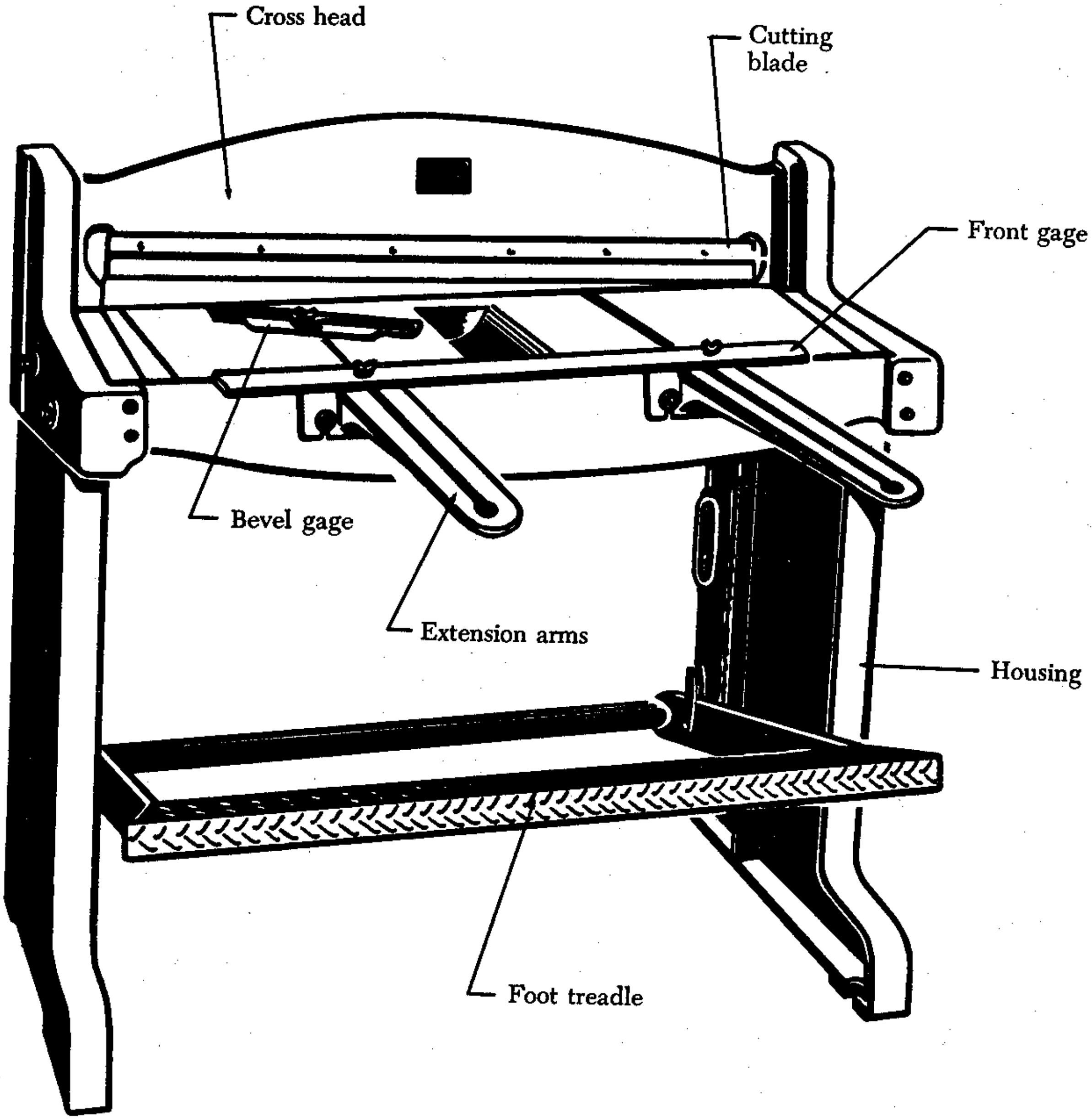


FIGURE 5–7. Squaring shears.

METALWORKING MACHINES

Without metalworking machines a job would be more difficult and tiresome, and the time required to finish a task would be much longer. Some of the machines used are discussed here; these include the powered and nonpowered metal-cutting machines, such as the various types of saws, powered and nonpowered shears, and nibblers. Also included is the forming equipment (both power driven and nonpowered), such as brakes and forming rolls, the bar folder, and shrinking and stretching machines.

Metal Cutting Manually Operated Tools—Lever Type

Squaring shears provide a convenient means of cutting and squaring metal. These shears consist of a stationary lower blade attached to a bed and a movable upper blade attached to a crosshead (figure 5–7). To make the cut, the upper blade is moved down by placing the foot on the treadle and pushing downward.

The shears are equipped with a spring which raises the blade and treadle when the foot is removed. A scale, graduated in fractions of an inch, is scribed on the bed. Two squaring fences, consisting of thick strips of metal and used for squaring metal sheets, are placed on the bed, one on the right side and one on the left. Each is placed so that it forms a 90° angle with the blades.

Three distinctly different operations can be performed on the squaring shears: (1) Cutting to a line, (2) squaring, and (3) multiple cutting to a specific size. When cutting to a line, the sheet is placed on the bed of the shears in front of the cutting blade with the cutting line directly even with the cutting edge of the bed. The sheet is cut by stepping on the treadle while the sheet is held securely in place by the holddown clamp.

Squaring requires several steps. First, one end of the sheet is squared with an edge (the squaring fence is usually used on the edge). Then the remaining edges are squared by holding one squared end of the sheet against the squaring fence and making the cut, one edge at a time, until all edges have been squared.

When several pieces must be cut to the same dimensions, use the gage which is on most squaring shears. The supporting rods are graduated in fractions of an inch, and the gage bar may be set at any point on the rods. Set the gage at the desired distance from the cutting blade of the shears and push each piece to be cut against the gage bar. All the pieces can then be cut to the same dimensions without measuring and marking each one separately.

Scroll shears (figure 5–8) are used for cutting irregular lines on the inside of a sheet without cutting through to the edge. The upper cutting blade is stationary while the lower blade is movable. The machine is operated by a handle connected to the lower blade.

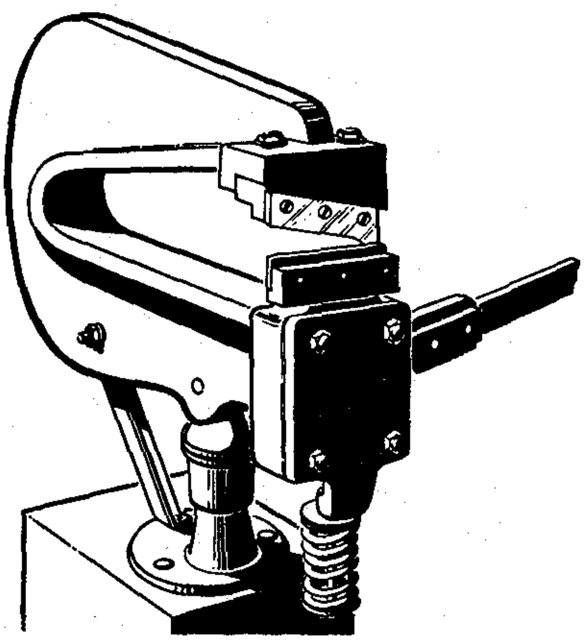


FIGURE 5–8. Scroll shears.

Throatless shears (figure 5–9) are best used to cut 10-gage mild carbon sheet metal and 12-gage stainless steel. The shear gets its name from its construction; it actually has no throat. There are no obstructions during cutting since the frame is throatless. A sheet of any length can be cut, and the metal can be turned in any direction to allow for cutting irregular shapes. The cutting blade (top blade) is operated by a hand lever.

The rotary punch (figure 5–10) is used in the airframe repair shop to punch holes in metal parts. This machine can be used for cutting radii in corners, for making washers, and for many other jobs where holes are required. The machine is composed of two cylindrical turrets, one mounted over the other and supported by the frame. Both turrets are synchronized so that they rotate together, and index pins assure correct alignment at all times. The index pins may be released from their locking position by

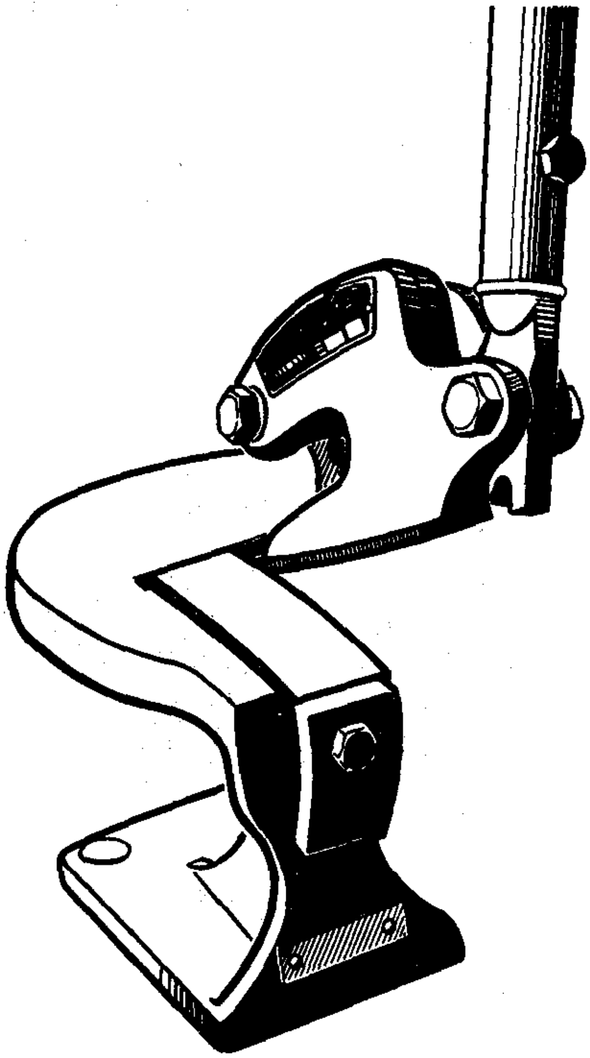


FIGURE 5–9. Throatless shears.

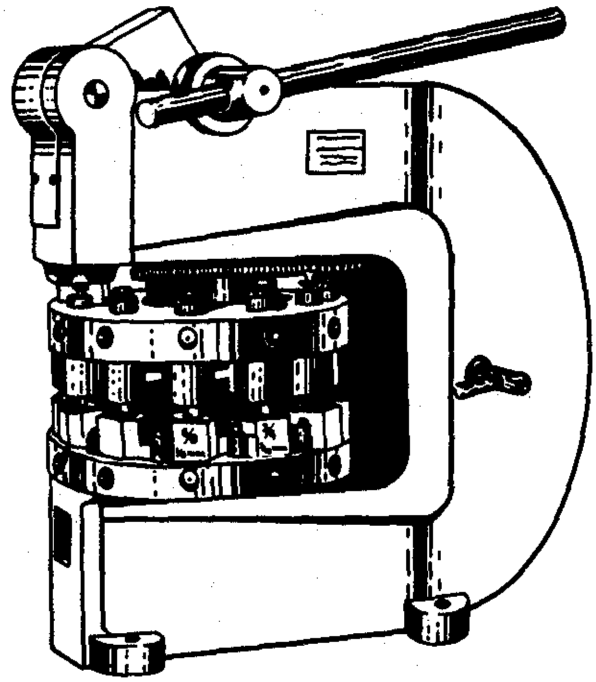


FIGURE 5–10. Rotary punch.

rotating a lever on the right side of the machine. This action withdraws the index pins from the tapered holes and allows an operator to turn the turrets to any size punch desired.

When rotating the turret to change punches, release the index lever when the desired die is within 1 in. of the ram, and continue to rotate the turret slowly until the top of the punch holder slides into the grooved end of the ram. The tapered index locking pins will then seat themselves in the holes provided and, at the same time, release the mechanical locking device, which prevents punching until the turrets are aligned.

To operate the machine, place the metal to be worked between the die and punch. Pull the lever on the top side of the machine toward you. This will actuate the pinion shaft, gear segment, toggle link, and the ram, forcing the punch through the metal. When the lever is returned to its original position, the metal is removed from the punch.

The diameter of the punch is stamped on the front of each die holder. Each punch has a point in its center which is placed in the centerpunch mark to punch the hole in the correct location.

Metal-Cutting Power-Operated Tools

The electrically operated portable circular-cutting Ketts saw (figure 5–11) uses blades of various diameters. The head of this saw can be turned to any desired angle, and is very handy for removing damaged sections on a stringer. Advantages of a Ketts saw are:

(1) The ability to cut metal up to 3/16 in. thick.

(2) No starting hole is required.

(3) A cut can be started anywhere on a sheet of metal.

(4) The capability of cutting an inside or outside radius.

To prevent grabbing, keep a firm grip on the saw handle at all times. Before installing a blade, it should be checked carefully for cracks. A cracked blade can fly apart and perhaps result in serious injury.

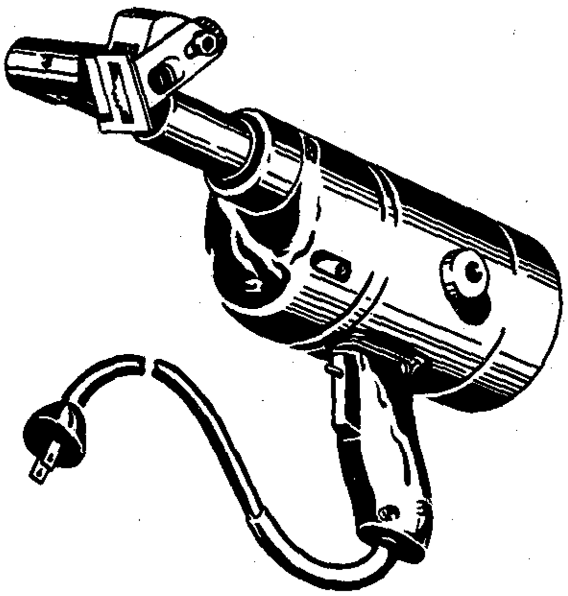


FIGURE 5–11. Ketts saw.

The portable, air powered reciprocating saw (figure 5–12) has a gun-type shape for balancing and ease of handling and operates most effectively at an air pressure of from 85 to 100 p.s.i. The reciprocating saw uses a standard hacksaw blade and can cut a 360° circle or a square or rectangular hole. This saw is easy to handle and safe to use.

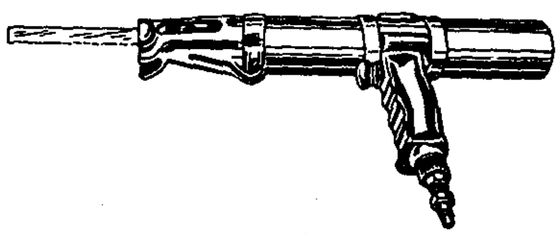


FIGURE 5–12. Reciprocating saw.

A reciprocating saw should be used in such a way that at least two teeth of the saw blade are cutting at all times. Avoid applying too much downward pressure on the saw handle because the blade may break.

Nibblers

Stationary and portable nibblers are used to cut metal by a high-speed blanking action. The cutting or blanking action is caused by the lower die moving up and down and meeting the upper stationary die. The shape of the lower die permits small pieces of metal approximately 1/16-in. wide to be cut out.

The cutting speed of the nibbler is controlled by the thickness of the metal being cut. Sheets of metal with a maximum thickness of 1/16 in. can be cut satisfactorily. Too much force applied to the metal during the cutting operation will clog the dies, causing the die to fail or the motor to overheat.

The spring-loaded screw on the base of the lower die should be adjusted to allow the metal to move freely between the dies. This adjustment must be sufficient to hold the material firmly enough to prevent irregular cuts. The dies may be shimmed for special cutting operations.

Portable Power Drills

One of the most common operations in airframe metalwork is that of drilling holes for rivets and bolts. This operation is not difficult, especially on light metal. Once the fundamentals of drills and their uses are learned, a small portable power drill is usually the most practical machine to use. However, there will be times when a drill press may prove to be the better machine for the job.

Some portable power drills will be encountered which are operated by electricity and others which are operated by compressed air. Some of the electrically operated drills work on either alternating or direct current, whereas others will operate on only one kind of current.

Portable power drills are available in various shapes and sizes to satisfy almost any requirement (figure 5–13). Pneumatic drills are recommended for use on projects around flammable materials where sparks from an electric drill might become a fire hazard.

When access to a place where a hole is to be drilled is difficult or impossible with a straight drill, various types of drill extensions and adapters are used. A straight extension can be made from an ordinary piece of drill rod. The twist drill is attached to the drill rod by shrink fit, brazing, or silver soldering. Angle adapters can be attached to either an electric or pneumatic drill when the location of the hole is inaccessible to a straight drill. Angle adapters have an extended shank fastened to the chuck of the drill. In use, the drill is held in one hand and the adapter in the other to prevent the adapter from spinning around the drill chuck.

A flexible extension can be used for drilling in places which are inaccessible to ordinary drills. Its flexibility permits drilling around obstructions with a minimum of effort.

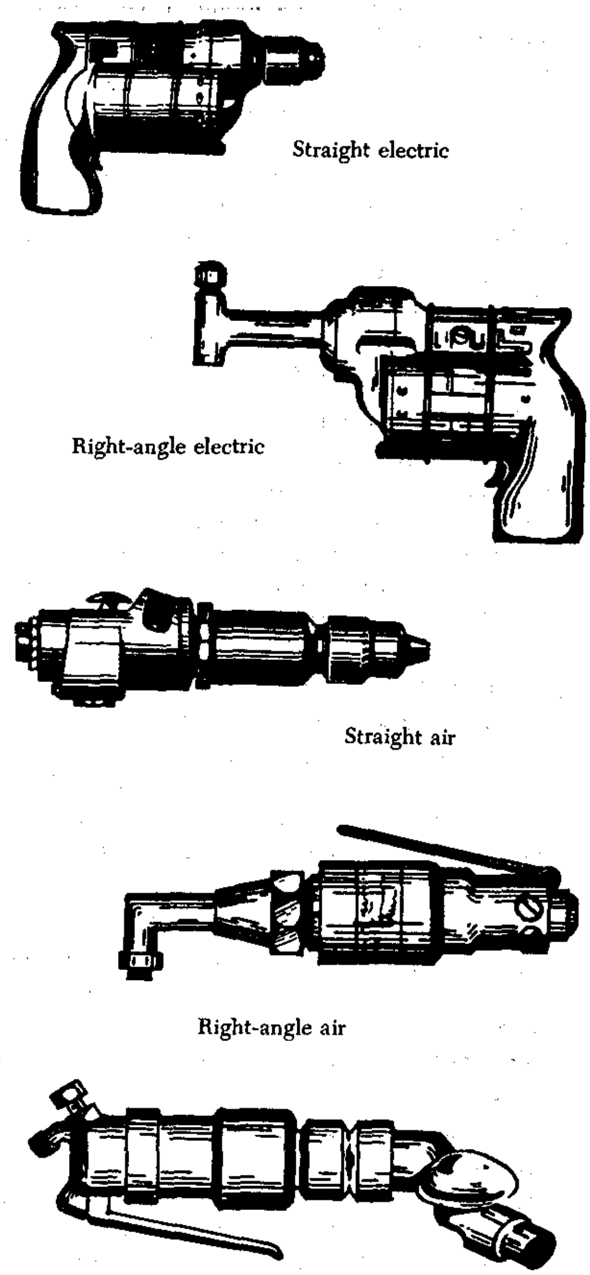


FIGURE 5–13. Portable power drills.

When using the portable power drill, hold it firmly with both hands. Before drilling, be sure to place a backup block of wood under the hole to be drilled to add support to the metal.

The twist drill should be inserted in the chuck and tested for trueness or vibration. This may be visibly checked by running the motor freely. A drill that wobbles or is slightly bent should not be used since such a condition will cause enlarged holes.

The drill should always be held at right angles to the work regardless of the position or curvatures. Tilting the drill at any time when drilling into or withdrawing from the material may cause elongation (egg shape) of the hole.

Always wear safety goggles while drilling.

When drilling through sheet metal, small burrs are formed around the edge of the hole. Burrs must be removed to allow rivets or bolts to fit snugly and to prevent scratching. Burrs may be removed with a bearing scraper, a countersink, or a twist drill larger than the hole. If a drill or countersink is used, it should be rotated by hand.

Drill Press

The drill press is a precision machine used for drilling holes that require a high degree of accuracy. It serves as an accurate means of locating and maintaining the direction of a hole that is to be drilled and provides the operator with a feed lever that makes the task of feeding the drill into the work an easy one.

A variety of drill presses are available; the most common type is the upright drill press (figure 5–14).

When using a drill press, the height of the drill press table is adjusted to accommodate the height of the part to be drilled. When the height of the part is greater than the distance between the drill and the table, the table is lowered. When the height of the part is less than the distance between the drill and the table, the table is raised.

After the table is properly adjusted, the part is placed on the table and the drill is brought down to aid in positioning the metal so that the hole to be drilled is directly beneath the point of the drill. The part is then clamped to the drill press table to prevent it from slipping during the drilling operation. Parts not properly clamped may bind on the drill and start spinning, causing the loss of fingers or hands or serious cuts on the operator's arms or body. Always make sure the part to be drilled is properly clamped to the drill press table before starting the drilling operation.

The degree of accuracy that it is possible to attain when using the drill press will depend to a certain extent on the condition of the spindle hole, sleeves, and drill shank. Therefore, special care must be exercised to keep these parts clean and free from nicks, dents, or warpage. Always be sure that the sleeve is securely pressed into the spindle hole. Never insert a broken drill in a sleeve or spindle hole. Be careful never to use the sleeve-clamping

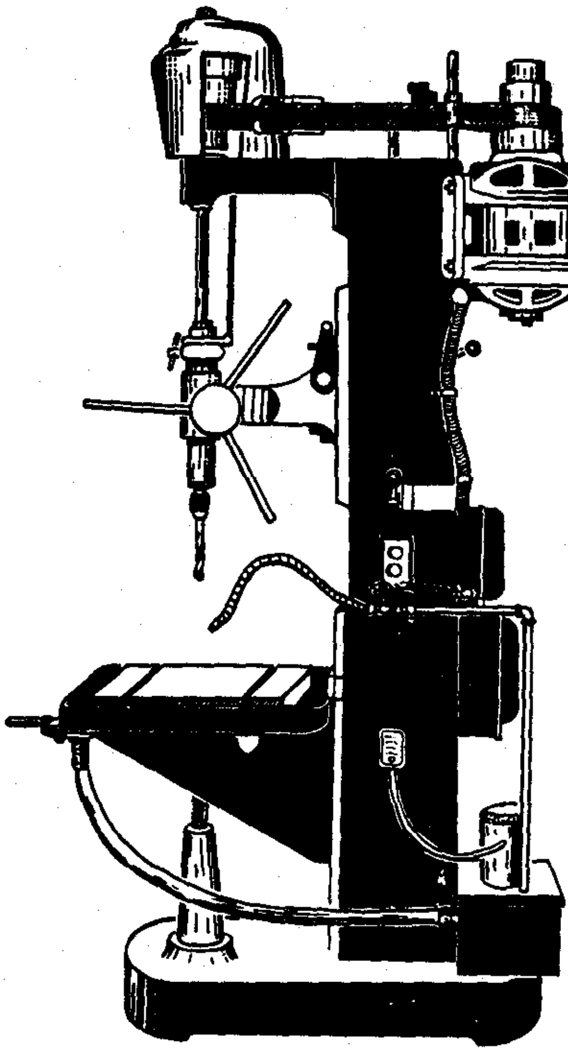


FIGURE 5–14. Drill press.

vise to remove a drill since this may cause the sleeve to warp.

Grinders

The term grinder applies to all forms of grinding machines. To be specific, it is a machine having an abrasive wheel which removes excess material while producing a suitable surface. There are many kinds of grinding machines, but only those which are helpful to the airframe mechanic will be discussed here.

Grinding Wheels

A grinding wheel is a cutting tool with a large number of cutting edges arranged so that when they become dull they break off and new cutting edges take their place.

Silicon carbide and aluminum oxide are the kinds of abrasives used in most grinding wheels. Silicon carbide is the cutting agent for grinding hard, brittle material, such as cast iron. It is also used in grinding aluminum, brass, bronze, and copper. Aluminum oxide is the cutting agent for grinding steel and other metals of high tensile strength.

The size of the abrasive particles used in grinding wheels is indicated by a number which corresponds to the number of meshes per linear inch in the screen through which the particles will pass. As an example, a number 30 abrasive will pass through a screen having 30 holes per linear inch, but will be retained by a smaller screen having more than 30 holes per linear inch.

The bond is the material which holds the abrasive particles together in forming the wheel. The kind and amount of bond used determines the hardness or softness of the wheel. The commonly used bonds are vitrified, silicate, resinoid, rubber, and shellac. Vitrified and silicate are the bonds used most frequently, vitrified bond being used in approximately three-fourths of all grinding wheels made. This bonding material forms a very uniform wheel and is not affected by oils, acids, water, heat, or cold. The silicate bond, however, is best suited for grinding edged tools.

Resinoid bonded wheels are better for heavy-duty grinding. Rubber bonded wheels are used where a high polish is required. Shellac bonded wheels are used for grinding materials where a buffed or burnished surface is needed.

A pedestal or floor type grinder usually has a grinding wheel on each end of a shaft which runs through an electric motor or a pulley operated by a belt. This grinder is used for sharpening tools and other general grinding jobs.

The wet grinder, although similar to the pedestal grinder, differs from it in that the wet grinder has a pump to supply a flow of water on a single grinding wheel. The water reduces the heat produced by material being ground against the wheel. It also washes away any bits of metal or abrasive removed during the grinding operation. The water returns to a tank and can be re-used.

A common type bench grinder found in most metalworking shops is shown in figure 5–15. This grinder can be used to dress mushroomed heads on chisels, and points on chisels, screwdrivers, and drills. It can be used for removing excess metal from work and smoothing metal surfaces.

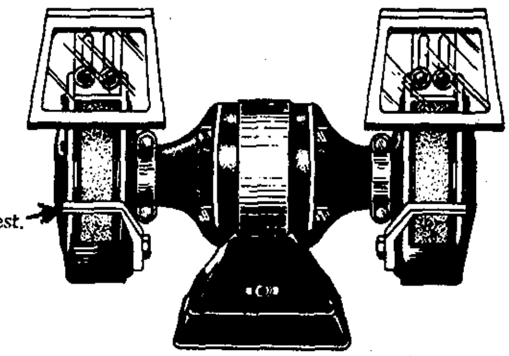


FIGURE 5–15. Bench grinder.

This type grinder is generally equipped with one medium-grain and one fine-grain abrasive wheel. The medium-grain wheel is usually used for rough grinding where a considerable quantity of material is to be removed or where a smooth finish is unimportant. The fine-grain wheel is usually used for sharpening tools and grinding to close limits because it removes metal more slowly, gives the work a smooth finish, and does not generate enough heat to anneal the edges of cutting tools. When it is necessary to make a deep cut on work or to remove a large amount of metal, it is usually good practice to grind with the medium-grain wheel first and then finish up with the fine-grain wheel.

The grinding wheels are removable, and the grinders are usually designed so that wire brushes, polishing wheels, or buffing wheels can be substituted for the abrasive wheels.

As a rule, it is not good practice to grind work on the side of an abrasive wheel. When an abrasive wheel becomes worn, its cutting efficiency is reduced because of a decrease in surface speed. When a wheel becomes worn in this manner, it should be discarded and a new one installed.

Before using a bench grinder, make sure the abrasive wheels are firmly held on the spindles by the flange nuts. If an abrasive wheel should come off or become loose, it could seriously injure the operator in addition to ruining the grinder.

Another hazard is loose tool rests. A loose tool rest could cause the tool or piece of work to be "grabbed" by the abrasive wheel and cause the operator's hand to come in contact with the wheel. If this should happen, severe wounds may result.

Always wear goggles when using a grinder, even if eyeshields are attached to the grinder. Goggles should fit firmly against your face and nose. This is the only way to protect your eyes from the fine pieces of steel. Goggles that do not fit properly should be exchanged for ones that do fit.

Be sure to check the abrasive wheel for cracks before using the grinder. A cracked abrasive wheel is likely to fly apart when turning at high speeds. Never use a grinder unless it is equipped with wheel guards.

FORMING MACHINES

Forming machines can be either hand operated or power driven. Small machines are usually hand operated, whereas the larger ones are power driven. Straight line machines include such equipment as the bar folder, cornice brake, and box and pan brake. Rotary machines include the slip roll former and combination machine. Power-driven machines are those that require a motor of some description for power. These include such equipment as the power-driven slip roll former, and power flanging machine.

Bar Folder

The bar folder (figure 5–16) is designed for use in making bends or folds along edges of sheets. This machine is best suited for folding small hems, flanges, seams, and edges to be wired. Most bar folders have a capacity for metal up to 22 gage in thickness and 42 inches in length.

Before using the bar folder, several adjustments must be made for thickness of material, width of fold, sharpness of fold, and angle of fold.

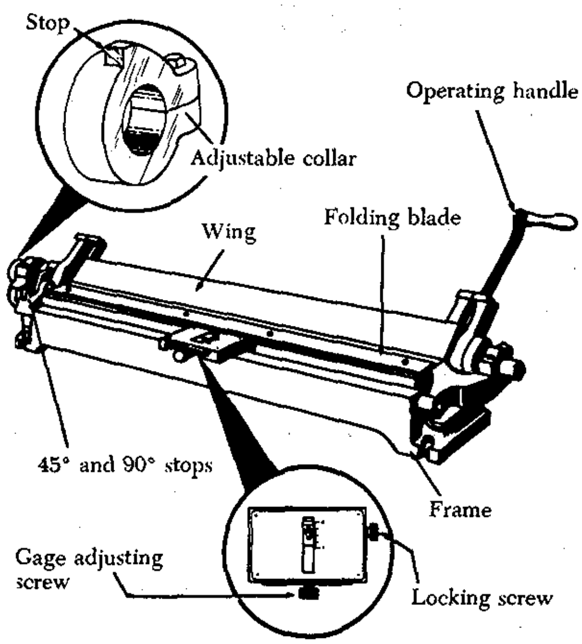


FIGURE 5–16. Bar folder.

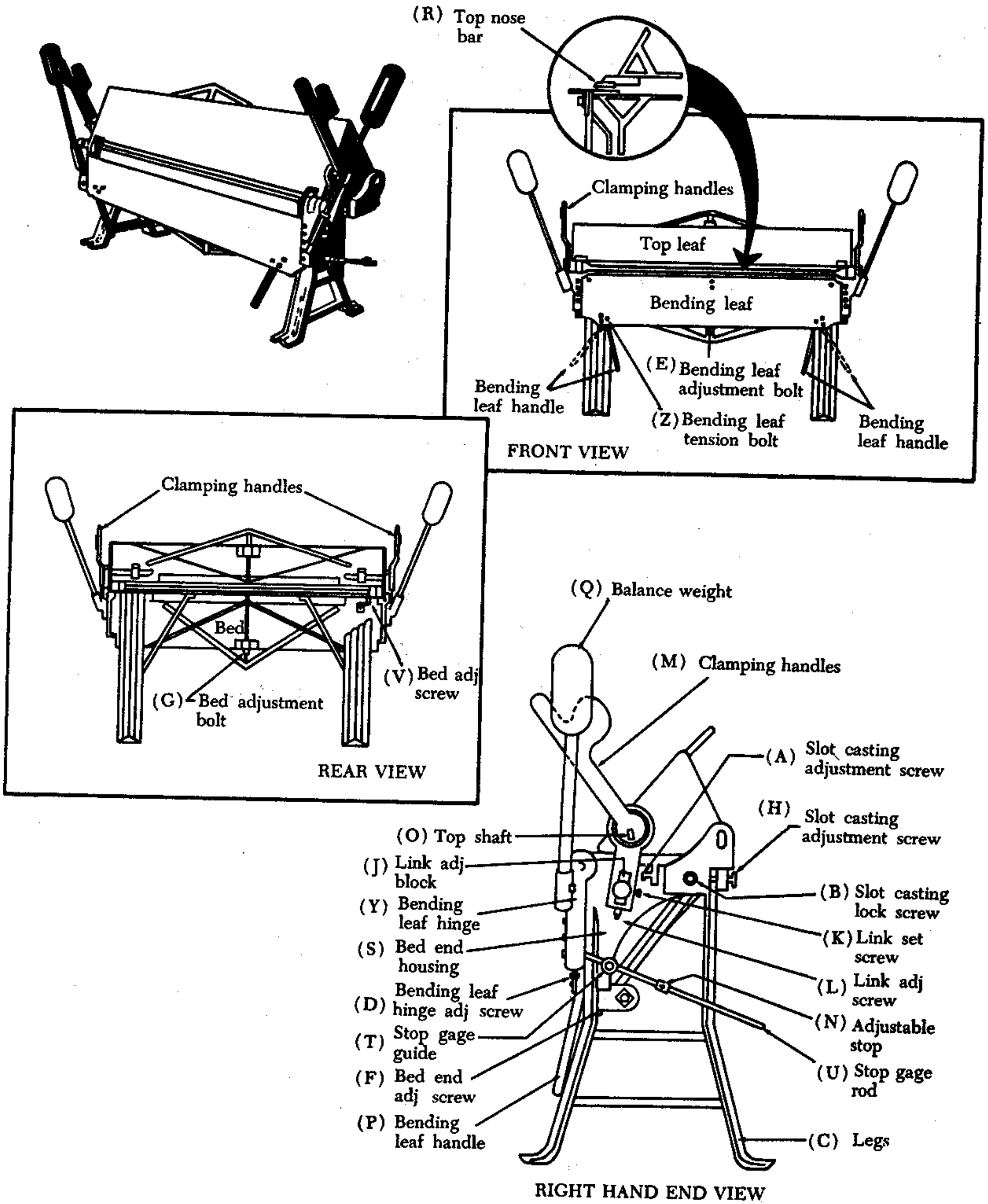


FIGURE 5–17. Cornice Brake.

The adjustment for thickness of material is made by adjusting the screws at each end of the folder. As this adjustment is made, place a piece of metal of the desired thickness in the folder and raise the operating handle until the small roller rests on the cam. Hold the folding blade in this position and adjust the setscrews so that the metal is clamped securely and evenly the full length of the folding blade. After the folder has been adjusted, test each end of the machine separately with a small piece of metal by actually folding it.

There are two positive stops on the folder, one for 45° folds or bends and the other for 90° folds or bends. An additional feature (a collar) is provided and can be adjusted to any degree of bend within the capacity of the machine.

For forming angles of 45° or 90°, the correct stop is moved into place. This will allow the handle to be moved forward to the correct angle. For forming other angles, the adjustable collar shown in the inset view of figure 5–16 is used. This is accomplished by loosening the setscrew and setting the stop at the desired angle. After setting the stop, tighten the setscrew and complete the bend.

To make the fold, adjust the machine correctly and then insert the metal. The metal goes between the folding blade and the jaw. Hold the metal firmly against the gage and pull the operating handle toward the body. As the handle is brought forward, the jaw automatically raises and holds the metal until the desired fold is made. When the handle is returned to its original position, the jaw and blade will return to their original positions and release the metal.

Cornice Brake

The cornice brake (fig. 5–17) has a much greater range of usefulness than the bar folder. Any bend formed on a bar folder can be made on the cornice brake. The bar folder can form a bend or edge only as wide as the depth of the jaws. In comparison, the cornice brake allows the sheet that is to be folded or formed to pass through the jaws from front to rear without obstruction.

In making ordinary bends with the cornice brake, the sheet is placed on the bed with the sight line (mark indicating line of bend) directly under the edge of the clamping bar. The clamping bar is then brought down to hold the sheet firmly in place. The stop at the right side of the brake is set for the proper angle or amount of bend, and the bending leaf is raised until it strikes the stop. If other bends are to be made, the clamping bar is lifted and the sheet is moved to the correct position for bending.

The bending capacity of a cornice brake is determined by the manufacturer. Standard capacities of this machine are from 12- to 22-gage sheet metal, and bending lengths are from 3 to 12 ft. The bending capacity of the brake is determined by the bending edge thickness of the various bending leaf bars.

Most metals have a tendency to return to their normal shape—a characteristic known as springback. If the cornice brake is set for a 90° bend, the metal bent will probably form an angle of about 87° to 88°. Therefore, if a bend of 90° is desired, set the cornice brake to bend an angle of about 93° to allow for springback.

Slip Roll Former

The slip roll former (figure 5–18) is manually operated and consists of three rolls, two housings, a base, and a handle. The handle turns the two front rolls through a system of gears enclosed in the housing.

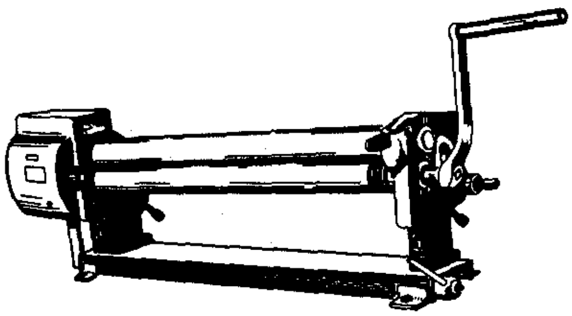


FIGURE 5–18. Slip roll former.

The front rolls serve as feeding or gripping rolls. The rear roll gives the proper curvature to the work. The front rolls are adjusted by two front adjusting screws on each end of the machine. The rear roll is adjusted by two screws at the rear of each housing. The front and rear rolls are grooved to permit forming of objects with wired edges. The upper roll is equipped with a release which permits easy removal of the metal after it has been formed.

When using the slip roll former, the lower front roll must be raised or lowered so that the sheet of metal can be inserted. If the object has a folded edge, there must be enough clearance between the rolls to prevent flattening the fold. If a metal requiring special care (such as aluminum) is being formed, the rolls must be clean and free of imperfections.

The rear roll must be adjusted to give the proper curvature to the part being formed. There are no gages that indicate settings for a specific diameter; therefore, trial-and-error settings must be used to obtain the desired curvature.

The metal should be inserted between the rolls from the front of the machine. Start the metal between the rolls by rotating the operating handle in a clockwise direction.

A starting edge is formed by holding the operating handle firmly with the right hand and raising the metal with the left hand. The bend of the starting edge is determined by the diameter of the part being formed. If the edge of the part is to be flat or nearly flat, a starting edge should not be formed.

Be sure that fingers or loose clothing are clear of the rolls before the actual forming operation is started. Rotate the operating handle until the metal is partly through the rolls and change the left hand from the front edge of the sheet to the upper edge of the sheet. Then roll the remainder of the sheet through the machine.

If the desired curvature is not obtained, return the metal to its starting position by rotating the handle counterclockwise. Raise or lower the rear roll and roll the metal through the rolls again. Repeat this procedure until the desired curvature is obtained, then release the upper roll and remove the metal.

If the part to be formed has a tapered shape, the rear roll should be set so that the rolls are closer together on one end than on the opposite end. The amount of this adjustment will have to be determined by experiment.

If the job being formed has a wired edge, the distance between the upper and lower rolls and the distance between the lower front roll and the rear roll should be slightly greater at the wired end than at the opposite end.

Forming Processes

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. Forming may be a very simple process, such as making one or two holes for attaching, or it may be exceedingly complex, requiring shapes with complex curvatures.

Parts are formed at the factory on large presses or by drop hammers equipped with dies of the correct shape. Every part is planned by factory engineers, who set up specifications for the materials to be used so that the finished part will have the correct temper when it leaves the machines. A layout for each part is prepared by factory draftsmen.

Forming processes used on the flight line and those practiced in the maintenance or repair shop are almost directly opposite in the method of procedure. They have much in common, however, and many of the facts and techniques learned in the one process can be applied to the other.

Forming is of major concern to the airframe mechanic and requires the best of his knowledge and skill. This is especially true since forming usually involves the use of extremely light-gage alloys of a delicate nature which 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 will deal with the procedures for forming aluminum or aluminum alloy parts.

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 have to be stopped and the metal annealed before the process can be continued or completed. Alloy 2024 in the "O" condition can be formed into almost any shape by the common forming operations, but it must be heat-treated afterward.

When forming, use hammers and mallets as sparingly as practicable, and make straight bends on bar folders or cornice brakes. Use rotary machines whenever possible. If a part fits poorly or not at all, do not straighten a bend or a curve and try to re-form it, discard the piece of metal and start with a new one.

When making layouts, be careful not to scratch aluminum or aluminum alloys. A pencil, if kept sharp, will be satisfactory for marking purposes. Scribers make scratches which induce fatigue failure; but they may be used if the marking lines fall outside the finished part, that is. if the scribed line will be part of the waste material. Keep bench tops covered with material hard enough to prevent chips and other foreign material from becoming imbedded in them. Be sure also to keep bench tops clean and free from chips, filings, and the like. For the protection of the metals being worked, keep vise jaws covered with soft metal jaw caps.

Stainless steel can be formed by any of the usual methods but requires considerably more skill than is required for forming aluminum or aluminum alloys. Since stainless steel work-hardens very readily, it requires frequent annealing during the forming operations. Always try to press out stainless steel parts in one operation. Use dies, if possible.

FORMING OPERATIONS AND TERMS

The methods used in forming operations include such sheetmetal work processes as shrinking, stretching, bumping, crimping, and folding.

Bumping

Shaping or forming malleable metal by hammering or pounding is called bumping. 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

Folding, pleating, or corrugating a piece of sheet metal in a way that shortens it is called crimping. Crimping is often used to make one end of a piece of stovepipe slightly smaller so that one section may be slipped into another. Turning down a flange on a seam is also called crimping. Crimping one side of a straight piece of angle iron with crimping pliers will cause it to curve, as shown in figure 5–19.

Stretching

Hammering a flat piece of metal in an area such as that indicated in figure 5–19 will cause the material in that area to become thinner. However, since the amount of metal will not have been decreased, it will cover a greater area because the metal will have been stretched.

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, as shown in figure 5–19, would cause its length to be increased (stretched); therefore, that section would become 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

During the shrinking process, material is forced or compressed into a smaller area. The shrinking 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 two ways: (1) By hammering on a V-block (figure 5–20), or (2) by crimping and then shrinking on a shrinking block.

To curve the formed angle by the V-block method, place the angle on the V-block and gently

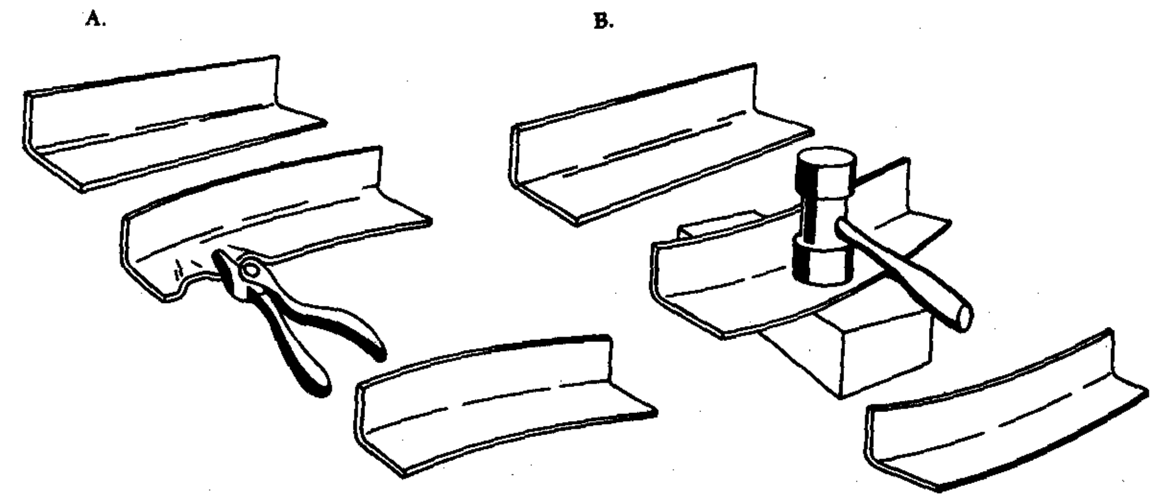


FIGURE 5–19. Crimping and stretching.

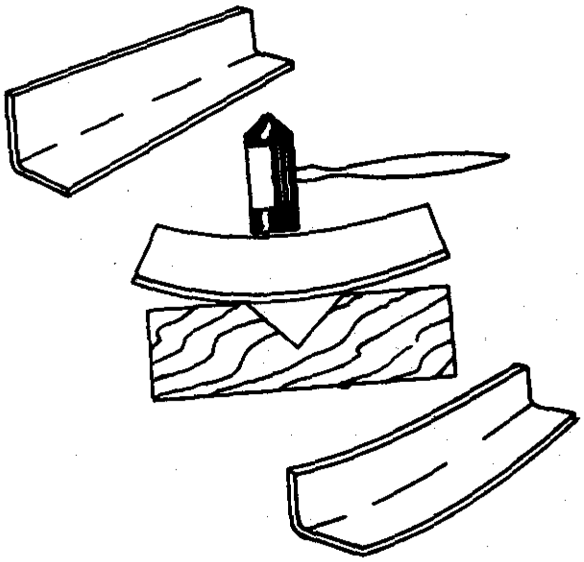


FIGURE 5–20. Shrinking on a V-block.

hammer downward against the upper edge directly over the "V" (figure 5–20). 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.

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 will be driven out (shrunk out) one at a time.

Folding

Making bends in sheets, plates, or leaves is called folding. Folds are usually thought of as sharp, angular bends; they are generally made on folding machines.

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, the sharper it can be bent (the smaller the radius of bend), and the softer the material, the sharper the bend. 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 will weaken the metal and may result in cracking.

A minimum radius of bend is specified for each type of aircraft sheet metal. The kind of material, thickness, and temper condition of the sheet are factors affecting it. Annealed sheet can be bent to a radius approximately equal to its thickness. Stainless steel and 2024–T aluminum alloy require a fairly large bend radius (see fig. 5–28).

Bend Allowance

When making a bend or fold in a sheet of metal, the bend allowance must be calculated. Bend allowance is the length of material required for the bend. This amount of metal must be added to the overall length of the layout pattern to assure adequate metal for the bend.

Bend allowance depends on four factors: (1) The degree of bend, (2) The radius of the bend, (3) The thickness of the metal, and (4) The 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 will be 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 will be greater, and the bend allowance will be greater. The degree of bend will affect the overall length of the metal, whereas the thickness influences the radius of bend.

Bending a strip 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 ($0.445 \times T$) from the inside of the radius of the bend (figure 5–21).

When bending metal to exact dimensions, the length of the neutral line must be determined so that sufficient material can be allowed 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 established. The bend allowance formula for a 90° bend is discussed in the following paragraphs.

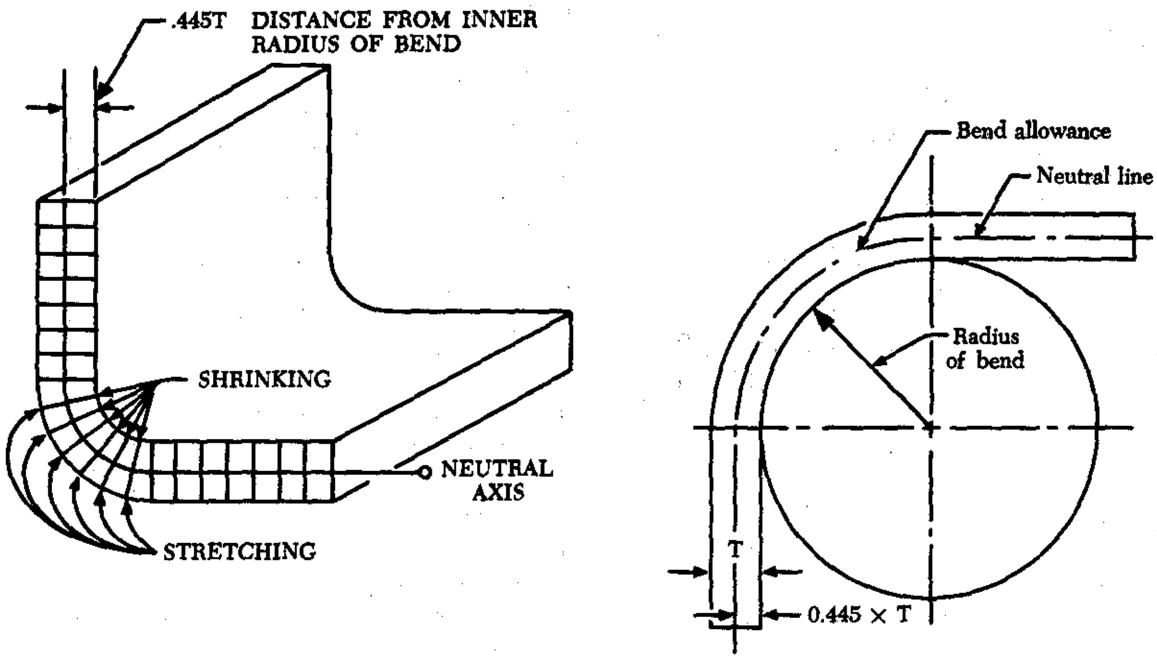


FIGURE 5-21. Neutral axis.

Method #1 Formula #1

To the radius of bend (R) add one-half the thickness of the metal, (½ T). This gives R+½ T, or the radius of the circle of approximately the neutral axis.

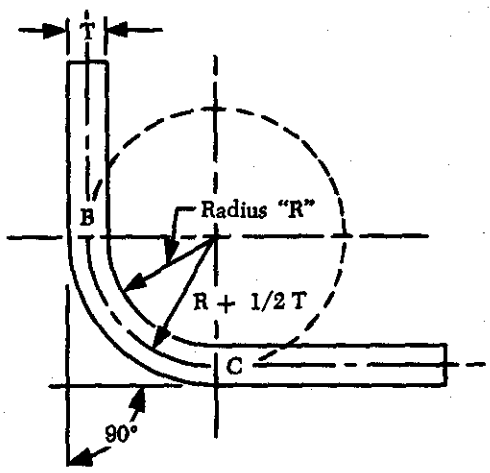


FIGURE 5-22. Bend allowance, 90° bend.

Compute the circumference of this circle by multiplying the radius of curvature of the neutral line (R+½ T in figure 5-22) by 2π:

$2\pi(R+\frac{1}{2}T)$.
Note: $\pi=3.1416$.

Since a 90° bend is a quarter of the circle, divide the circumference by 4. This gives:

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

Therefore, bend allowance for a 90° bend is

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

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

Bend allowance

$$= \frac{2X3.1416\ (0.250+1/2X0.051)}{4}$$

$$= \frac{6.2832\ (0.250+0.02555)}{4}$$

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

$=0.4323$.

Thus, if necessary, bend allowance or the length of material required for the bend is 0.4323, or 7/16 in.

The formula is slightly in error because actually the neutral line is not exactly in the center of the sheet being bent. (See figure 5-22.) However, the amount of error incurred in any given problem is so slight that, for most work, since the material used is thin, the formula is satisfactory.

Method #2 Formula #2

This formula uses two constant values which 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 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

$= (0.01743 \times R + 0.0078 \times T) \times N$

where:

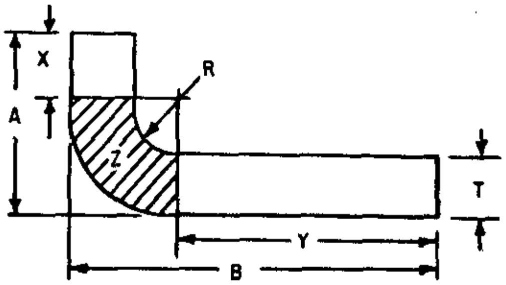
R = The desired bend radius,

T = Thickness of the material, and

N = Number of degrees of bend.

BA = Bend allowance

BA = $0.01743 \times 20°$



Bend allowance (Z) = (0.01743R + 0.0078T) x (No. of degrees of bend)

T = thickness of metal

R = radius of bend

Z = bend allowance

X = A - (R + T)

Y = B - (R + T)

Total developed length = X + Y + Z

FIGURE 5–23. Computing bend allowance.

Method #3 Use of 90° Bend Chart

Either formula may be used in the absence of a bend allowance chart. To determine bend allowance for any degree of bend by use of the chart (figure 5–24), find the allowance per degree for the number of degrees in the bend.

Radius of bend is given as a decimal fraction on the top line of the chart. Bend allowance is given directly below the radius figures. The top number in each case is the bend allowance for a 90° angle, whereas the lower placed number is for a 1° angle. Material thickness is given in the left column of the chart.

To find the bend allowance when the sheet thickness is 0.051 in., the radius of bend is ¼ in. (0.250-in.), and the bend is to be 90°. Reading across the top of the bend allowance chart, find the column for a radius of bend of 0.250 in. Now find the block in this column that is opposite the gage of 0.051 in the column at left. The upper number in the block is 0.428, the correct bend allowance in inches for a 90° bend.

Method #4 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. The lower number in this case is 0.004756. Therefore, if the bend is to be 120°, the total bend allowance in inches will be 120X0.004756, or 0.5707 in.

SETBACK

When bending a piece of sheet stock, it is necessary to know the starting and ending points of the bend so that the length of the "flat" of the stock can be determined. Two factors are important in determining this, the radius of bend and the thickness of the material.

In figure 5–27, note that *setback is the distance* from *the bend tangent line to the mold point.* The mold point is the point of intersection of the lines extending from the outside surfaces, whereas the bend tangent lines are the starting and end points of the bend. Also note that setback is the same for the vertical flat and the horizontal flat.

Another way to look at setback is this: If the mandrel in a cornice brake is adjusted to the edge of the bed, a piece of metal is inserted, and a 90° bend is to be made, when the bending leaf is raised to 90°, the metal will be cut due to the compressing action of the leaf. The mandrel must be "set back" from the edge of the bed one thickness of the metal for a 90° bend. This permits the metal to flow thereby forming a correct bend.

Calculating Setback, Formula #1

To calculate the setback for a 90° bend, merely add the inside radius of the bend to the thickness of the sheet stock, i.e.

Setback = R + T.

Example:

Calculate the setback for a 90° bend, if the

RADIUS	OF BEND IN INCHES													
GAGE	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 .000837	.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 5–24. Bend allowance chart.

material is 0.051-in. thick and the radius of bend is specified to be 1/8 in. (0.125).

$$\begin{aligned} \text{Setback} &= R+T. \\ &= 0.125+0.051 \\ &= 0.176 \text{ in.} \end{aligned}$$

Calculating Setback, Formula #2

To calculate setback for angles larger or smaller than 90°, consult standard setback charts (figure 5–25), or "K" chart, for a value called "K", and then substitute this value in the formula.

$$\text{Setback} = K \ (R+T).$$

The value of K varies with the number of degrees in the bend.

Example:

Calculate the setback for a 120° bend with a radius of bend of 0.125 in. in a sheet 0.032-in. thick.

$$\begin{aligned} \text{Setback} &= K \ (R+T). \\ &= 1.7320 \ (0.125+0.032) \\ &= 0.272 \text{ in.} \end{aligned}$$

Brake or Sight Line

The brake or sight line is the mark on a flat sheet which is set even with the nose of the radius bar of the cornice brake and serves as a guide in bending. The brake line can be located by measuring out one radius from the bend tangent line closest to the end which is to be inserted under the nose of the brake or against the radius form block. The nose of the brake or radius bar should fall directly over the brake or sight line as shown in figure 5–26.

Bend Allowance Terms

Familiarity with the following terms is necessary for an understanding of bend allowance and its application to an actual bending job. Figure 5–27 illustrates most of these terms.

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.

A	K	A	K	A	K
1°	.00873	61°	.58904	121°	1.7675
2°	.01745	62°	.60086	122°	1.8040
3°	.02618	63°	.61280	123°	1.8418
4°	.03492	64°	.62487	124°	1.8807
5°	.04366	65°	.63707	125°	1.9210
6°	.05241	66°	.64941	126°	1.9626
7°	.06116	67°	.66188	127°	2.0057
8°	.06993	68°	.67451	128°	2.0503
9°	.07870	69°	.68728	129°	2.0965
10°	.08749	70°	.70021	130°	2.1445
11°	.09629	71°	.71329	131°	2.1943
12°	.10510	72°	.72654	132°	2.2460
13°	.11393	73°	.73996	133°	2.2998
14°	.12278	74°	.75355	134°	2.3558
15°	.13165	75°	.76733	135°	2.4142
16°	.14054	76°	.78128	136°	2.4751
17°	.14945	77°	.79543	137°	2.5386
18°	.15838	78°	.80978	138°	2.6051
19°	.16734	79°	.82434	139°	2.6746
20°	.17633	80°	.83910	140°	2.7475
21°	.18534	81°	.85408	141°	2.8239
22°	.19438	82°	.86929	142°	2.9042
23°	.20345	83°	.88472	143°	2.9887
24°	.21256	84°	.90040	144°	3.0777
25°	.22169	85°	.91633	145°	3.1716
26°	.23087	86°	.93251	146°	3.2708
27°	.24008	87°	.80978	147°	3.3759
28°	.24933	88°	.96569	148°	3.4874
29°	.25862	89°	.98270	149°	3.6059
30°	.26795	90°	1.00000	150°	3.7320
31°	.27732	91°	1.0176	151°	3.8667
32°	.28674	92°	1.0355	152°	4.0108
33°	.29621	93°	1.0538	153°	4.1653
34°	.30573	94°	1.0724	154°	4.3315
35°	.31530	95°	1.0913	155°	4.5107
36°	.32492	96°	1.1106	156°	4.7046
37°	.33459	97°	1.1303	157°	4.9151
38°	.34433	98°	1.1504	158°	5.1455
39°	.35412	99°	1.1708	159°	5.3995
40°	.36397	100°	1.1917	160°	5.6713
41°	.37388	101°	1.2131	161°	5.9758
42°	.38386	102°	1.2349	162°	6.3137
43°	.39391	103°	1.2572	163°	6.6911
44°	.40403	104°	1.2799	164°	7.1154
45°	.41421	105°	1.3032	165°	7.5957
46°	.42447	106°	1.3270	166°	8.1443
47°	.43481	107°	1.3514	167°	8.7769
48°	.44523	108°	1.3764	168°	9.5144
49°	.45573	109°	1.4019	169°	10.385
50°	.46631	110°	1.4281	170°	11.430
51°	.47697	111°	1.4550	171°	12.706
52°	.48773	112°	1.4826	172°	14.301
53°	.49858	113°	1.5108	173°	16.350
54°	.50952	114°	1.5399	174°	19.081
55°	.52057	115°	1.5697	175°	22.904
56°	.53171	116°	1.6003	176°	26.636
57°	.54295	117°	1.6318	177°	38.188
58°	.55431	118°	1.6643	178°	57.290
59°	.56577	119°	1.6977	179°	114.590
60°	.57735	120°	1.7320	180°	Infinite

FIGURE 5–25A Setback (K) chart.

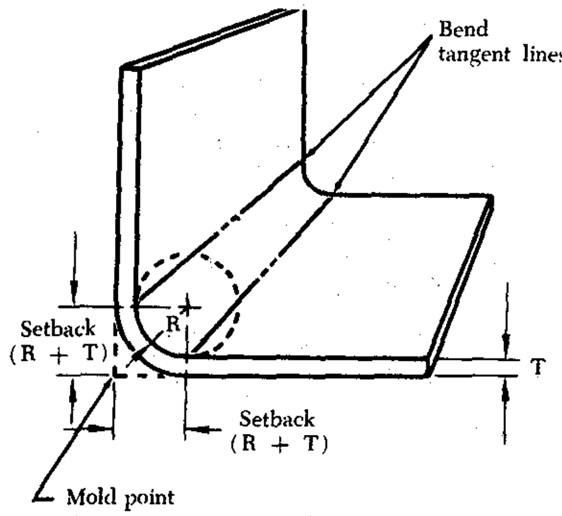


FIGURE 5–25B. Setback, 90° bend.

Mold Line (ML). The line formed by extending the outside surfaces of the leg and flange. (An imaginary point from which real base measurements are provided on drawings.)

Bend Tangent Line (BL). The line 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.

Bend Allowance (BA). The amount of material consumed in making a bend (figure 5–12).

Radius (R). The radius of the bend—always to the inside of the metal being formed unless otherwise stated. (The minimum allowable radius for bending a given type and thickness of material should always be ascertained before proceeding with any bend allowance calculations.)

Setback (SB). The setback is the distance from the bend tangent line to the mold point. In a 90-degree bend SB=R+T (radius of the bend plus thickness of the metal). The setback dimension must be determined prior to making the bend as it (setback) is used in determining the location of the beginning bend tangent line (figure 5–27).

Bend Line (also called Brake or Sight Line). The layout line on the metal being formed which is set even with the nose of the brake and serves as a guide in bending the work. (Before forming a bend, it must be decided which end of the material can be most conveniently inserted in the brake. The bend line is then measured and marked off with a soft-lead pencil from the bend tangent line closest to the end which is to be placed under the brake. This measurement should be equal to the radius of the bend. The metal is then inserted

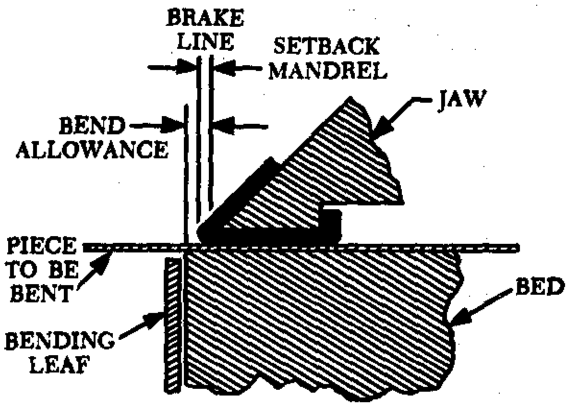


FIGURE 5-26. Setback – locating bend line in brake.

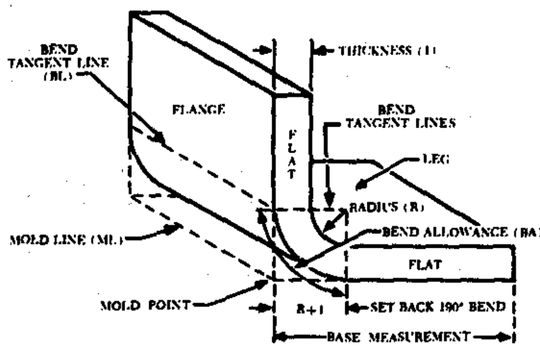


FIGURE 5-27. Bend allowance terms.

in the brake so that the nose of the brake will fall directly over the bend line, as shown in figure 5–26.)

Flat (short for flat portion). The flat portion or flat of a part is that portion not included in the bend. It is equal to the base measurement minus the setback.

Base Measurement. The outside dimensions of a formed part. Base measurement will be given on the drawing or blueprint, or may be obtained from the original part.

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.

"K" No. 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" No. of 1.0), the corresponding "K" No. is selected from the chart and is multiplied by the sum of the radius and the thickness of the metal. The product is the amount of setback for the bend.

MAKING LAYOUTS

It is wise to make a layout or pattern of the part before forming it to prevent any waste of material and to get a greater degree of accuracy in the finished part. Where straight angle bends are concerned, correct allowances must be made for setback and bend allowance. If the shrinking or stretching processes are to be used, allowances must

Designation	Gage							
	0.020	0.025	0.032	0.040	0.050	0.063	0.071	0.080
2024-O	1/32	1/16	1/16	1/16	1/16	3/32	1/8	1/8
2024-T4	1/16	1/16	3/32	3/32	1/8	5/32	7/32	1/4
5052-O	1/32	1/32	1/16	1/16	1/16	1/16	1/8	1/8
5052-H34	1/32	1/16	1/16	1/16	3/32	3/32	1/8	1/8
6061-O	1/32	1/32	1/32	1/16	1/16	1/16	3/32	3/32
6061-T4	1/32	1/32	1/32	1/16	1/16	3/32	5/32	5/32
6061-T6	1/16	1/16	1/16	3/32	3/32	1/8	3/16	3/16
7075-O	1/16	1/16	1/16	1/16	3/32	3/32	5/32	3/16
7075-W	3/32	1/32	1/8	5/32	3/16	1/4	9/32	5/16
7075-T6	1/8	1/8	1/8	3/16	1/4	5/16	3/8	7/16

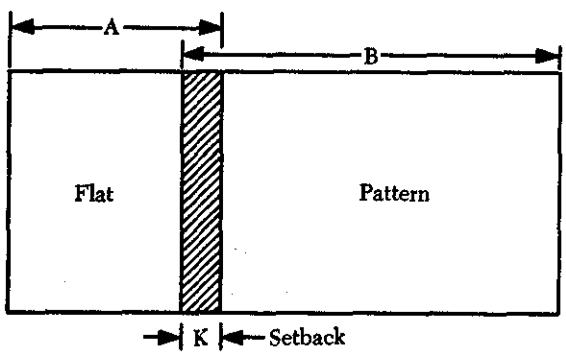
FIGURE 5–28. Minimum bend radii for aluminum alloys.

be made so that the part can be turned out with a minimum amount of forming.

The layout procedures can be put into three general groups: (1) Flat layout, (2) Duplication of pattern, and (3) Projection through a set of points. All three processes require a good working knowledge of arithmetic and geometry. This presentation will discuss only two processes, flat layout and duplication of pattern.

Referring to the "K" chart, figure 5–27, it is noted that the "K" value for 90° is equal to 1T (thickness of metal). Further observation will show that for an angle of less than 90° the setback is less than 1T, for an angle of more than 90° the setback is more than 1T.

The use of 1T setback in a bend of less than 90° (open angle) would result in the flange of the bend being too long. Conversely in an angle of over 90° with less than 1T setback the flange would be too short.



Developed Length of Pattern = A+B. To determine the developed Length of the flat pattern, deduct the "K" dimension from the sum of the dimensions A+B.

FIGURE 5–29. Setback development.

Flat Layout

Assume that it is necessary to lay out a flat pattern of a channel (figure 5–31) in which the **left-hand flat, A, is to be 1 in. high, the right-hand flat, C, is to be 1-1/4 in. high, and the distance between the outside surface of the two flats, B, is to be 2 in. The material is 0.051-in. thick, and the radius of bend is to be 3/16 in. (0.188). The angles are to be 90°. Proceed as follows:**

(1) Determine the setback to establish the distance of the flats.

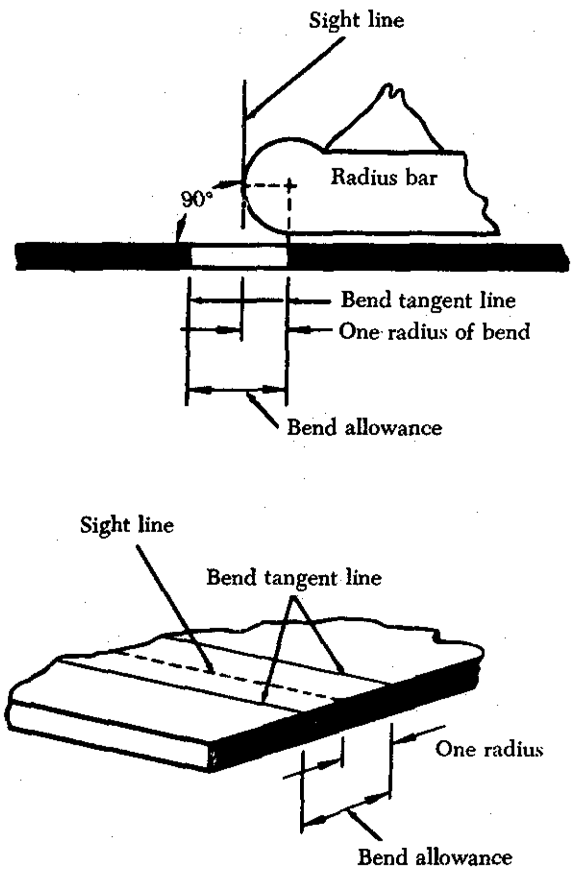


FIGURE 5–30. Brake or sight line.

(a) The setback for the first bend:

Setback = R + T

= 0.188 + 0.051

= 0.239.

(b) The first flat A is equal to the overall dimension less setback:

Flat A = 1.000 — 0.239

= 0.761 in.

(2) Calculate the bend allowance for the first bend by using the bend allowance chart (figure 5–24). (BA=0.3307 or 0.331.)

(3) Now lay off the second flat, B. **This is equal to the overall dimension less the setback at each end, or B minus two setbacks:** (See figure 5–31.)

Flat B = 2.00 — (0.239 + 0.239)

= 2.000 — .478

= 1.522 in.

(4) The bend allowance for the second bend is the same as that for the first bend (0.331). Mark off this distance. (See figure 5–31.)

(5) The third flat, C, is equal to the overall dimension less the setback. Lay off this distance. (See figure 5–31.)

Flat C = 1.250 — 0.239
= 1.011 in.

(6) Adding the measurements of flats A, B, and C, and both bend allowances, (0.761 + 0.331 + 1.522 + 0.331 + 1.011), the sum is 3.956, or approximately 4.00 inches. Totaling the three flats, A, B, and C, 1 in., 2 in., and 1–1/4 in., respectively, the sum is 4.250 in. of material length. This illustrates how setback and bend allowance affect material lengths in forming straight line bends. In this case, the reduction is approximately 1/4 in.

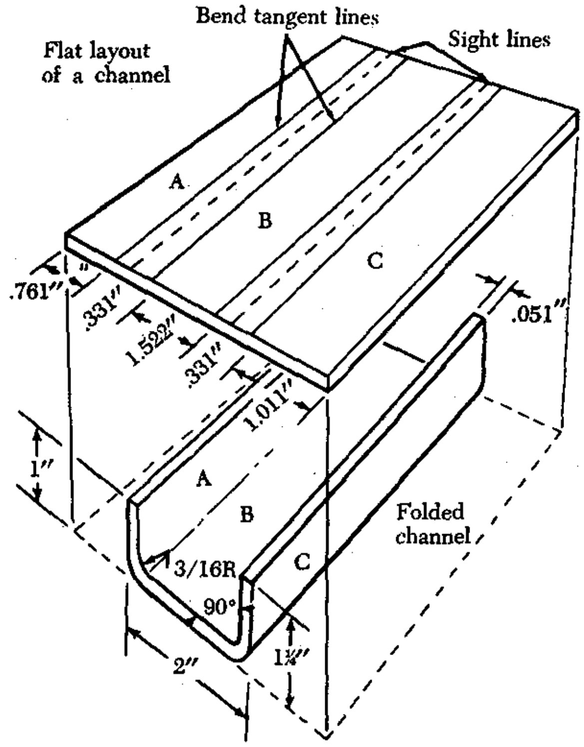


FIGURE 5–31. Flat layout of a channel.

After all measurements are calculated, cut the material and mark off the brake or sight lines as shown in figure 5–31.

Duplication of Pattern

When it is necessary to duplicate an aircraft part and blueprints are not available, take measurements directly from the original or from a duplicate part. In studying the following steps for laying out a part to be duplicated, refer to the illustrations in figure 5–32.

Draw a reference (datum) line, AB, on the sample part and a corresponding line on the template material (example 1, figure 5–32).

Next, with point A on the sample part as a center, draw an arc having a radius of approximately 1/2 in. and extending to the flanges (example 2, figure 5–32).

Draw similar arcs each with a radius 1/2 in. greater than the previous one until the entire part is marked. In case there is an extremely sharp curve in the object, decrease the distance between the arcs to increase the number of arcs. This procedure will increase the accuracy of the layout. An arc must pass through every corner of the part; one arc may pass through more than one corner (example 3, figure 5–32).

Locate the coordinate point on the layout by measuring on the part with dividers. Always measure the distance from the reference point to the beginning of the bend line on the flange of the part.

After locating all points, draw a line through them, using a French curve to ensure a smooth pattern (example 4, figure 5–32).

Allow for additional material for forming the flange and locate the inside bend tangent line by measuring, inside the sight line, a distance equal to the radius of bend of the part.

Using the intersection of the lines as a center, locate the required relief holes. Then cut out and form as necessary.

Relief Holes

Wherever two bends intersect, material must be removed to make room for the material contained in the flanges. Holes are therefore drilled at the intersection. These holes, called relief holes, prevent strains from being set up at the intersection of the

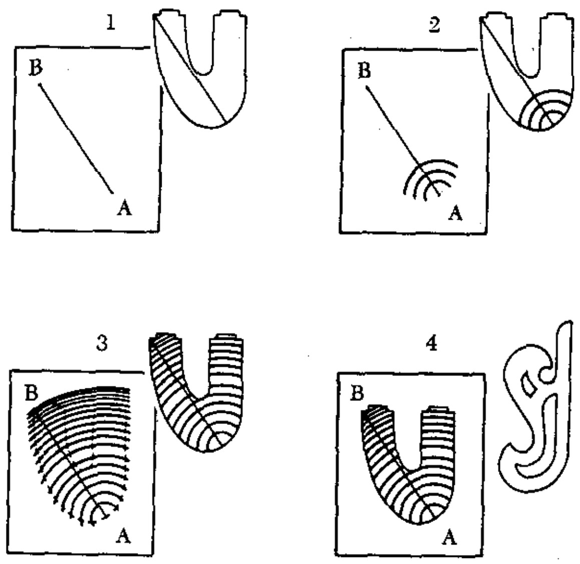


FIGURE 5–32. Duplicating a pattern.

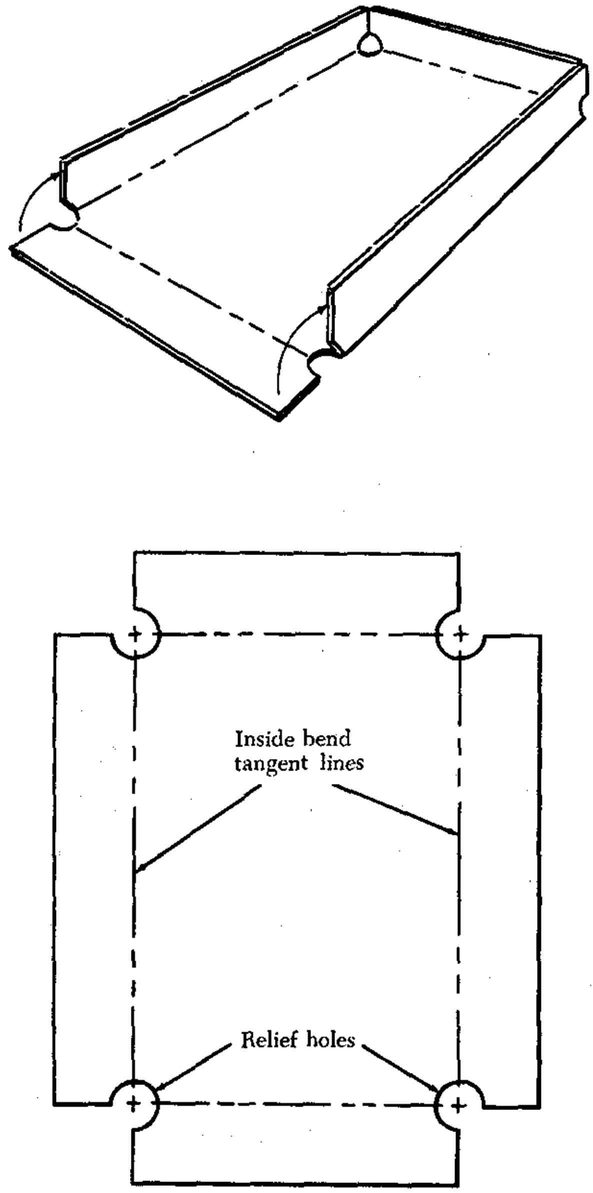


FIGURE 5–33. Locating relief holes.

inside bend tangent lines which would cause the metal to crack. Relief holes also provide a neatly trimmed corner from which excess material may be trimmed.

The size of relief holes varies with thickness of the material. They should be not less than 1/8 in. in diameter for aluminum alloy sheet stock up to and including 0.064-in. thick, or 3/16 in. for stock ranging from 0.072 in. to 0.128 in. in 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 (1/8 in.).

Relief holes must touch the intersection of the inside bend tangent lines. To allow for possible error in bending, make the relief holes so they will extend 1/32 to 1/16 in. behind the inside bend tangent lines. It is good practice to use the intersection of these lines as the center for the holes (figure 5–33). 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.

Lightening Holes

Lightening holes are cut in rib sections, fuselage frames, and other structural parts to decrease weight. To keep from weakening the member by removal of the material, flanges are often pressed around the holes to strengthen the area from which the material was removed.

Lightening holes should never be cut in any structural part unless authorized. The size of the lightening hole and the width of the flange formed around the hole are determined by design specifications. Margins of safety are considered in the specifications so that the weight of the part can be decreased and still retain the necessary strength.

Lightening holes may be cut by any one of the following methods:

(1) Punching out, if the correct size punch die is available.

(2) Cutting out with a fly cutter mounted on a drill.

(3) Scribing the circumference of a hole with dividers and drilling around the entire circumference with a small drill, allowing enough clearance to file smooth.

(4) Scribing the circumference of the hole with dividers, drilling the hole inside the circumference large enough to insert aviation snips, cutting out excess metal, and filing smooth.

Form the flange by using a flanging die, or hardwood or metal form blocks. Flanging dies consist of two matching parts, a female and a male die. For flanging soft metal, dies can be of hardwood, such as maple. For hard metal or for more permanent use, they should be made of steel. The pilot guide should be the same size as the hole to be flanged, and the shoulder should be the same width and angle as the desired flange.

When flanging lightening holes, place the material between the mating parts of the die and form it by hammering or squeezing the dies together in a vise or in an arbor press. The dies will work more smoothly if they are coated with light machine oil.

Note that in the two form blocks shown on the left side of figure 5-34, the hole in the upper block is the same size as the hole to be flanged and is chamfered to the width of the flange and the angle desired, whereas in the lower block, the hole is the same diameter as that of the flange. Either type may be used. When using the upper block, center the material to be flanged and hammer it with a stretching mallet, around and around, until the flange conforms to the chamfer. When using the lower block, center the lightening hole over the hole in the block, then stretch the edges, hammering the material into the hole, around and around, until the desired flange is obtained. Occasionally, the chamfer is formed with a cone-shaped male die used in conjunction with the form block with which the part was formed.

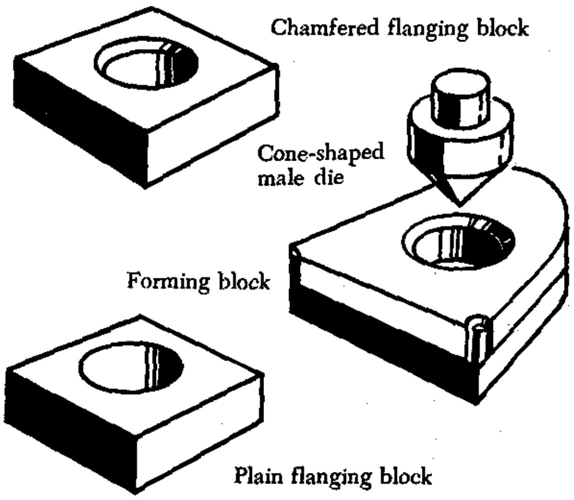


FIGURE 5-34. Flanging form blocks.

HAND FORMING

All forming revolves around the process of shrinking and stretching, and hand forming processes are no exception. If a formed or extruded angle is to be curved, either stretch one leg or shrink the other, whichever will make 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 lightening holes is often stretched to form a beveled reinforcing ridge around them.

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 by proceeding as explained in the following paragraphs.

After laying out and cutting a blank to size, clamp it rigidly along the bending line between two wooden blocks held in a vise. The wooden forming block should have one edge rounded for the desired radius of bend. It should also be curved slightly beyond the 90° point to allow for springback.

By tapping lightly with a rubber, plastic, or rawhide mallet, bend the metal protruding beyond the bending blocks to the desired angle. 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 forced down to the desired angle against the forming block. Allow for springback by driving the material slightly farther than the actual bend. If a large amount of metal extends beyond the bending blocks, maintain hand pressure against the protruding sheet to prevent "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 the one flange is usually preferred since this process requires only a V-block and a mallet and is easily accomplished.

In the stretching process, place the flange to be stretched in the groove of the V-block. Using a stretching mallet, strike the flange directly over the V portion with light, even blows while gradually forcing it downward into the V. Too heavy a blow will buckle the angle strip. Keep moving the angle strip across the V-block, but always strike the spot directly above the V. Form the curve gradually and evenly by moving the strip slowly back and forth, distributing the hammer blows at equal spaces on the flange.

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 will determine 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 will work-harden 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.

Curving an extruded or formed angle strip by shrinking may be accomplished by either of two methods, the V-block method or the shrinking block method. Of the two, the V-block is, in general, more satisfactory because it is faster, easier, and affects the metal less. However, very 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, as shown in figure 5–35. Hold it firmly so that it does not bounce when hammered, and strike the edge of the upper flange with light blows of a round, soft-faced mallet. Begin at one end of the angle strip and, working back and forth, strike light blows directly over the V-portion of the block. Strike the edge of the flange at a slight angle as this tends to keep the vertical flange from bending outward.

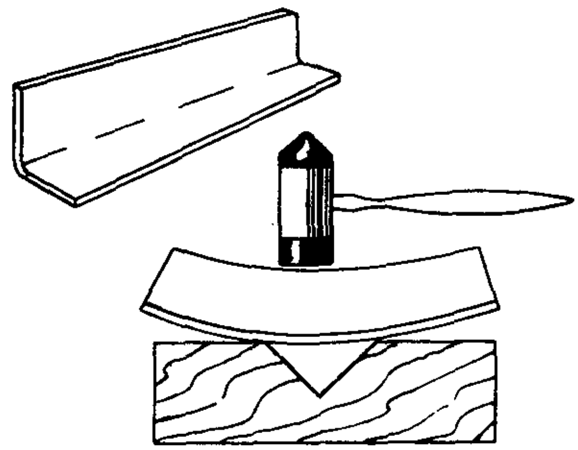


FIGURE 5–35. V-blocks.

Occasionally, check the curve for accuracy with the pattern. If a sharp curve is made, the angle (cross section of the formed angle) will close 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 jaws of the C-clamps should be covered with masking tape. If the angle has already closed, bring the flange back to the correct angle with a few blows of a mallet or with the aid of a small hardwood block. If any portion of the angle strip is curved too much, reduce it by reversing the angle on the V-block and hammering with a suitable mallet, as explained in the previous paragraph on stretching. After obtaining the proper curve, smooth the entire angle by planishing with a soft-faced mallet.

If the curve in a formed angle is to be quite sharp or if the flanges of the angle are rather broad, the shrinking block method is generally used. In this process, crimp the flange which is to form the inside of the curve.

When making a crimp, hold the crimping pliers so that the jaws are about 1/8 in. apart. By rotating the wrist back and forth, bring the upper jaw of the pliers into contact with the flange, first on one side and then on the other side of the lower jaw. Complete the crimp by working a raised portion into the flange, gradually increasing the twisting motion of the pliers. Do not make the crimp too large because it will be difficult to work out. The size of the crimp depends upon the thickness and softness of the material, but usually about 1/4 in. is sufficient. Place several crimps spaced evenly along the desired curve with enough space left between each crimp so that jaws of the shrinking block can easily be attached.

After completing the crimping, place the crimped flange in the shrinking block so that one crimp at a time is located between the jaws. Flatten each crimp with light blows of a soft-faced mallet, starting at the apex (the closed end) of the crimp and gradually working toward the edge of the flange. Check the curve of the angle with the pattern periodically during the forming process and again after all the crimps have been worked out. If it is necessary to increase the curve, add more crimps and repeat the process. Space the additional crimps between the original ones so that the metal will not become unduly work-hardened at any one point. If the curve needs to be increased or decreased slightly at any point, use the V-block.

After obtaining the desired curve, planish the angle strip over a stake or a wooden form.

Flanged Angles

The forming process for the following two flanged angles is slightly more complicated than that just discussed in that the bend is shorter (not gradually curved) and necessitates shrinking or stretching in a small or concentrated area. If the flange is to point toward the inside of the bend, the material must be shrunk. If it is to point toward the outside, it must be stretched.

In forming a flanged angle by shrinking, use wooden forming blocks similar to those shown in figure 5–36 and proceed as follows:

(1) Cut the metal to size, allowing for trimming after forming. Determine the bend allowance for a 90° bend and round the edge of the forming block accordingly.

(2) Clamp the material in the form blocks as shown in figure 5–36, and bend the exposed flange against the block. After bending, tap the blocks slightly. This induces a setting process in the bend.

(3) Using a soft-faced shrinking mallet, start hammering near the center and work the flange down gradually toward both ends. The flange will tend to buckle at the bend because the material is made to occupy less space. Work the material into several small buckles instead of one large one and work each buckle out gradually by hammering lightly and gradually compressing the material in each buckle. The use of a small hardwood wedge block, as shown in figure 5–36, will aid in working out the buckles.

(4) Planish the flange after it is flattened against the form block and remove small irregularities. If the form blocks are made of hardwood, use a metal planishing hammer. If the forms are made of metal, use a soft-faced mallet. Trim the excess material away and file and polish.

Forming by Stretching

To form a flanged angle by stretching, use the same forming blocks, wooden wedge block, and mallet as in the shrinking process and proceed as follows:

(1) Cut the material to size (allowing for trim), determine bend allowance for a 90° bend, and round off the edge of the block to conform to the desired radius of bend.

(2) Clamp the material in the form blocks as shown in figure 5–36.

(3) Using a soft-faced stretching mallet, start hammering near the ends and work the flange down smoothly and gradually to prevent cracking and splitting. Planish the flange and angle as described in the previous procedure, and trim and smooth the edges, if necessary.

Curved Flanged Parts

Curved flanged parts are usually hand formed. Of the types shown in figure 5–37, the one with relief holes is probably the simplest to form. It has a concave flange (the inside flange) and a convex flange (the outside flange).

The concave flange is formed by stretching, the convex flange by shrinking. Such parts may be formed with the aid of hardwood or metal forming blocks. These blocks are made in pairs similar to those used for straight angle bends and are identified in the same manner. They differ in that they are made specifically for the particular part to be formed, they fit each other exactly, and they conform to the actual dimensions and contour of the finished article.

The mating parts may be equipped with aligning pins to aid in lining up the blocks and holding the metal in place. The blocks may be held together by C-clamps or a vise. They also may be held together with bolts by drilling through both forms and the metal, provided the holes do not affect the strength of the finished part. The edges of the forming block are rounded to give the correct radius of bend to the part, and are undercut to allow for springback of the metal. The undercut is especially necessary if the material is hard or if the bend must be highly accurate.

Note the various types of forming represented in figure 5–37. In the plain nose rib, only one large convex flange is used; but, because of the great distance around the part and the likelihood of buckles in forming, it is rather difficult to form. The flange and the beaded portion of this rib provide sufficient strength to make this a very good type to use. In the type with relief holes, the concave flange gives difficulty in forming; however, the outside flange is broken up into smaller sections by relief holes (notches inserted to prevent strains in a

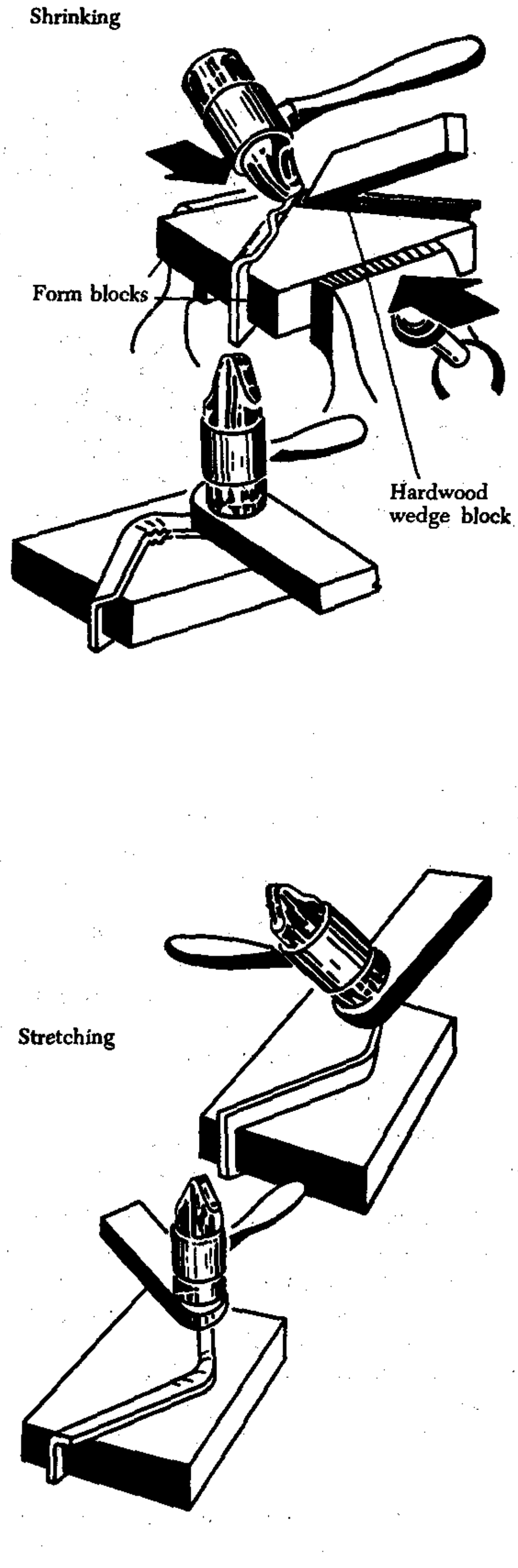


FIGURE 5–36. Forming a flanged angle.

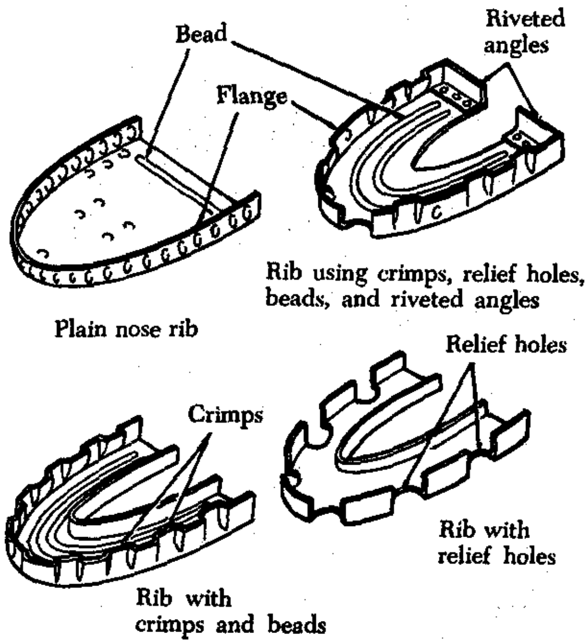


FIGURE 5-37. Nose ribs.

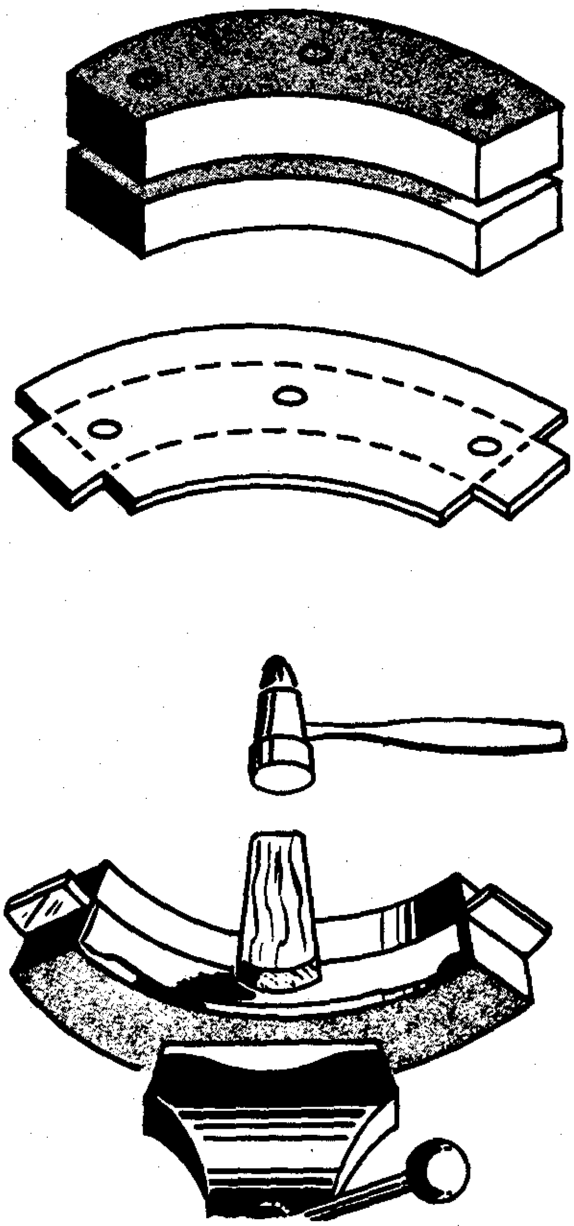


FIGURE 5-38. Forming a concave curve.

bend). In the type with crimps and beads, note that crimps are inserted at equally spaced intervals. The crimps are placed to absorb material and cause curving, while also giving strength to the part.

In the other nose rib illustrated, note that a combination of the four common forming methods is applied. They are crimping, beading, putting in relief holes, and using a formed angle riveted on at each end. The beads and the formed angles supply strength to the part.

The major steps in forming a curved flange part are explained in the following paragraphs.

Cut the material to size (allowing for trim), locate and drill holes for alignment pins, and remove all burrs (jagged edges). Place the material between the wooden blocks. Clamp blocks tightly in a vise so that the material will not move or shift. Clamp the work as closely as possible to the particular area being hammered to prevent strain on the form blocks and to keep the metal from slipping (figure 5-38).

Bend the flange on the concave curve first. This practice may keep the flange from splitting open or cracking when the metal is stretched. (Should this occur, a new piece will have to be made.) Using a soft mallet or wooden wedge block, start hammering at a point a short distance away from the beginning of the concave bend and continue toward the center of the bend. This procedure permits some of the excess metal along the tapered portion of the flange to be worked into the curve where it will be needed. Continue hammering until the metal is gradually worked down over the entire flange, flush with the form block.

Starting at the center of the curve and working toward both ends, hammer the convex flange down over the form (figure 5–39). Strike the metal with glancing blows, at an angle of approximately 30° off perpendicular, and with a motion that will tend to pull the part away from the block.

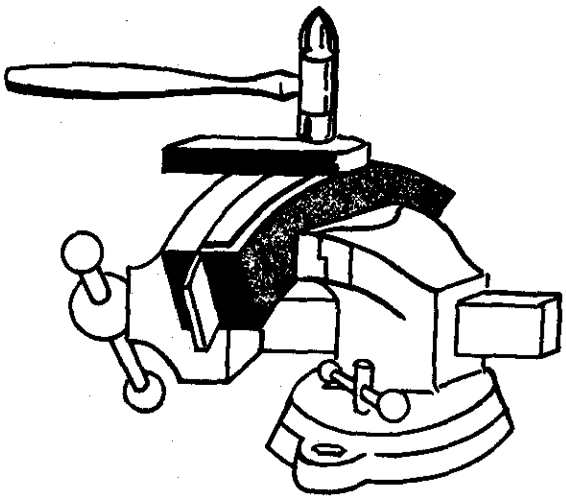


FIGURE 5–39. Forming a convex curve.

Stretch the metal around the radius bend and remove the buckles gradually by hammering on a wedge block.

While working the metal down over the form, keep the edges of the flange as nearly perpendicular to the block as possible. The wedge block helps keep the edge of the metal perpendicular to the block, lessens the possibility of buckles and of splitting or cracking the metal, and aids in removing buckles.

Finally, trim the flanges of excess metal, planish, remove burrs, round the corners (if any), and check the part for accuracy.

Bumping

Bumping on a form block or female die and bumping on a sandbag are the two common types practiced. In either method only one form is required, a wooden block, lead die, or sandbag. A good example of a part made by the block or die type of bumping is the "blister" or streamlined cover plate. Wing fillets constitute a good example of parts that are usually formed by bumping on a sandbag.

The lead die or the wooden block designed for bumping must have the same dimensions and contour as the outside of the blister. To provide sufficient bumping weight, and to give sufficient bearing surface for fastening the metal, the block or die should be at least 1 in. larger in all dimensions than the form requires.

When forming the wooden block, hollow it out with saws, chisels, gouges, files, and rasps. Smooth and finish it with sandpaper. Make the inside of the form as smooth as possible, because any slight irregularity will show up on the finished part. Prepare several templates (patterns of the cross section), such as those shown with the form block for the blister in figure 5–40, so that the form can be checked for accuracy.

Shape the contour of the form at points 2, 3, and 4. Shape the areas between the template check points to conform to the remaining contour and to template 4. Shaping of the form block requires particular care because the more nearly accurate it is, the less time it will take to produce a smooth, finished part.

Correct clamping of the material to the form block is an important part of the block-forming operation. Several methods are possible. For parts such as the blister, one of the best means of clamping the material is to use a full metal cutout or steel holddown plate as shown in figure 5–40.

In this process, place the holddown plate directly over the material to be formed and clamp it in position with bolts or C-clamps. Tighten the C-clamps or bolts just tight enough to hold the material flat against the face of the form block, but not so tight that the metal cannot be drawn into the form. If the material is not held flat against the face of the form, it will bend up or buckle away from the block. If it is not permitted to slip into the concave depression a little, the blister portion will become very thin in places.

Holddown plates should be of heavy steel, 1/8 in. for small forms and 1/4 in. or heavier for large forms.

If the material for making an all-metal holddown plate is not available, use a hardwood cutout. Make the cutout and use it in the same manner as the

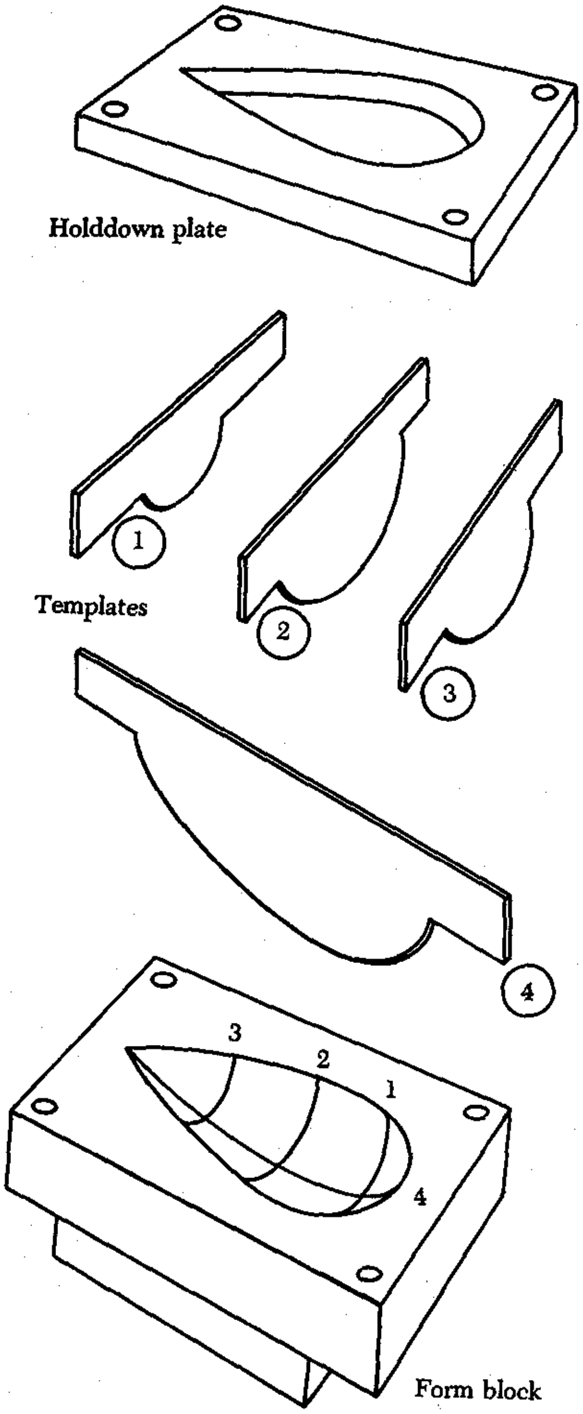


FIGURE 5-40. Form blocks and templates.

steel plate, but take greater precautions to make sure that the material is held as desired.

Pieced form clamps can be used if an all-metal holddown plate or hardwood cutout is not available or if a full cutout cannot be used. Be careful to clamp them properly and locate them so that they align with the edge of the form. If they are not aligned accurately, the material will bulge.

After preparing and checking the form, perform the bumping process according to the following general steps:

(1) Cut a metal blank to size, allowing an extra 1/2 to 1 in. to permit "drawing."
(2) Apply a thin coat of light oil to the block and to the aluminum to prevent galling (scraping on rough spots).
(3) Clamp the material between the block and steel plate, as previously described, so that it will be firmly supported yet able to slip a little toward the inside of the form.
(4) Clamp the bumping block in a bench vise. With a soft-faced mallet or with a hardwood drive block and suitable mallet, start the bumping near the edges of the form.
(5) With light blows of the mallet, work the material down gradually from the edges. Remember that the object of the bumping process is to work the material into shape by stretching it, rather than by forcing it into the form with heavy blows. Always start bumping near the edge of the form; never start near the center of the blister.
(6) Smooth the work as much as possible before removing it from the form. This can be done by rubbing the work with the rounded end of a maple block or with the round end of a stretching mallet.
(7) Remove the blister from the bumping block and trim it, leaving a 1/2-in. flange.
(8) Finally, drill the rivet holes, chamfer the edges 45°, and clean and polish the part.

Bumping on a sandbag is one of the most difficult types of sheet-metal hand forming, because there is no exact form block to serve as a guide. In this type of forming operation, a depression is made into a sandbag to take the shape of the hammered portion of the metal. The depression or pit has a tendency to shift from the hammering. This necessitates readjusting from time to time during the bumping process. The degree of shifting depends largely on the contour or shape of the piece being formed, and whether glancing blows must be struck to stretch, draw, or shrink the metal.

When forming by this method, prepare a contour template or some sort of a pattern to serve as a working guide and to ensure accuracy of the finished part. Make the pattern from ordinary kraft or similar paper, folding it over the part to be duplicated. Cut the paper cover at the points where it would have to be stretched to fit, and attach additional pieces of paper with masking tape to cover the exposed portions. After completely covering the part, trim the pattern to exact size.

Open the pattern and spread it out on the metal from which the part is to be formed. Although the pattern will not lie flat, it will give a fairly accurate idea of the approximate shape of the metal to be cut, and the pieced-in sections will indicate where the metal is to be stretched. When the pattern has been placed on the material, outline the part and the portions to be stretched using a pencil. Add at least 1 in. of excess metal when cutting the material to size. Trim off the excess metal after bumping the part into shape.

If the part to be formed is radially symmetrical, it will be fairly easy to shape since a simple contour template can be used as a working guide, making a pattern to indicate the portions of unequal stretching unnecessary. However, the procedure for bumping sheet metal parts on a sandbag follows certain basic rules which can be applied to any part, regardless of its contour or shape.

(1) Lay out and cut the contour template. This can be made of sheet metal, medium-heavy cardboard, or thin plywood.

(2) Determine the amount of metal needed, lay it out, and cut it to size, allowing at least 1/2 in. excess.

(3) Place a sandbag on a solid foundation capable of supporting heavy blows and, with the aid of a smooth-faced mallet, make a pit in the bag. Analyze the part to determine the correct radius of the pit for the forming operation. The pit will change with the hammering it receives and must be re-adjusted occasionally.

(4) Select a soft round-faced or bell-shaped mallet having a contour slightly smaller than the contour desired on the sheet-metal part. Holding one edge of the metal in the left hand, place the portion to be bumped near the edge of the pit on the sandbag. Strike the metal with light glancing blows, about 1/2 to 1 in. from the edge.

(5) Continue bumping toward the center, revolving the metal and working gradually inward until the desired shape is obtained. Shape the entire part as a unit.

(6) At frequent intervals during the bumping process, check the part for accuracy of shape by applying the template. If wrinkles are formed, work them out before they become too large.

(7) Finally, with a suitable stake and planishing hammer, or with a hand dolly and planishing hammer, remove small dents and hammer marks.

(8) With a scribe, mark around the outside of the object. Trim the edge and file until it is smooth. Clean and polish the part.

Joggling

A joggle is an offset formed on an angle strip to allow clearance for a sheet or an extrusion. Joggles are often found at the intersection of stringers and formers. One of these members, usually the former, has the flange joggled to fit flush over the flange of the stringer. The amount of offset is usually small; therefore, the depth of the joggle is generally specified in thousandths of an inch. The thickness of the material to be cleared governs the depth of the joggle. In determining the necessary length of the joggle, it is common practice to allow an extra 1/16 in. to give enough added clearance to assure a fit between the joggled, overlapped part.

There are a number of different methods by which joggles can be formed. If the joggle is to be made on a straight flange or flat piece of metal, form it on a cornice brake by inserting and bending up along the line of the joggle. Hold a piece of metal of the correct thickness to give the desired offset under the bent-up portion, and pound the flange down while the metal is still in the same position in the brake.

Where a joggle is necessary on a curved flange, forming blocks or dies made of hardwood, steel, or aluminum alloy may be used. If the die is to be used only a few times, hardwood is satisfactory as it is easily worked. If a number of similar joggles are to be produced, then use steel or aluminum alloy dies. Dies of aluminum alloy are preferred since they are easier to fabricate than those of steel and will wear about as long. These dies are sufficiently soft and resilient to permit forming aluminum alloy parts on them without marring, and nicks and scratches are easily removed from their surfaces.

When using joggling dies for the first time, test them for accuracy on a piece of waste stock. In this way you will avoid the possibility of ruining already fabricated parts. Always keep the surfaces of the blocks free from dirt, filings, and the like, so that the work will not be marred.

Working Stainless Steel

When working with stainless steel, make sure that the metal does not become unduly scratched or marred. Also take special precautions when shearing, punching, or drilling this metal. It takes about twice as much pressure to shear or punch stainless steel as it does mild steel. Keep the shear or punch and die adjusted very closely. Too much clearance will permit the metal to be drawn over the edge of the die and cause it to become work-hardened, resulting in excessive strain on the machine.

When drilling stainless steel, use a high-speed drill ground to an included angle of 140°. Some special drills have an offset point, whereas others have a chip curler in the flutes. When using an ordinary twist drill, grind its point to a stubbier angle than the standard drill point. Keep the drill speed about one-half that required for drilling mild steel, but never exceed 750 r.p.m. Keep a uniform pressure on the drill so the feed is constant at all times. Drill the material on a backing plate, such as cast iron, which is hard enough to permit the drill to cut all the way through the stock without pushing the metal away from the drill point. Spot the drill before turning on the power and also make sure that when the power is turned on, pressure is being exerted.

To avoid overheating, dip the drill in water after drilling each hole. When it is necessary to drill several deep holes in stainless steel, use a liquid coolant. A compound made up of 1 lb. of sulfur added to 1 gal. of lard oil will serve the purpose. Apply the coolant to the material immediately upon starting the drill. High-speed portable hand drills have a tendency to burn the drill points and excessively work-harden the material at the point of contact; thus high-speed portable hand drills should not be used because of the temperatures developed. A drill press adjustable to speeds under 750 r.p.m. is recommended.

Working Magnesium

Magnesium, in the pure state, does not have sufficient strength to be used for structural purposes; but, as an alloy, it has a high strength-to-weight ratio. Its strength is not affected by subzero temperatures, and this increases its adaptability to aircraft use. The nonmagnetic property of magnesium alloys makes it valuable for instrument cases and parts.

While magnesium alloys can usually be fabricated by methods similar to those used on other metals, it must be remembered that many of the details of shop practice cannot be applied. Magnesium alloys are difficult to fabricate at room temperature; therefore, operations other than the most simple ones must be performed at high temperatures. This requires preheating of the metal, or dies, or both.

Magnesium alloy sheets may be cut by blade shears, blanking dies, routers, or saws. Hand or circular saws are usually used for cutting extrusions to length. Conventional shears and nibblers should never be used for cutting magnesium alloy sheet because they produce a rough, cracked edge.

Shearing and blanking of magnesium alloys require close tool tolerances. A maximum clearance of from 3 to 5% of the sheet thickness is recommended. The top blade of the shears should be ground with an included angle of from 45° to 60°. The shear angle on a punch should be from 2° to 3°, with a 1° clearance angle on the die. For blanking, the shear angle on the die should be from 2° to 3° with a 1° clearance angle on the punch. Hold-down pressures should be used when possible. Cold shearing should not be accomplished on hard-rolled sheet thicker than 0.064 in. or annealed sheet thicker than 1/8 in. Shaving is used to smooth the rough, flaky edges of magnesium sheet which has been sheared. This operation consists of removing approximately 1/32 in. by a second shearing.

Hot shearing is sometimes used to obtain an improved sheared edge. This is necessary for heavy sheet and plate stock. Annealed sheet may be heated to 600° F., but hard-rolled sheet must be held under 400° F., depending on the alloy used. Thermal expansion makes it necessary to allow for shrinkage after cooling, which entails adding a small amount of material to the cold metal dimensions before fabrication.

Sawing is the only method used in cutting plate stock more than 1/2 in.-thick. Bandsaw raker-set blades of 4- to 6-tooth pitch are recommended for cutting plate stock or heavy extrusions. Small and medium extrusions are more easily cut on a circular cutoff saw having six teeth per inch. Sheet stock can be cut on bandsaws having raker-set or straight-set teeth with an 8-tooth pitch. Bandsaws should be equipped with nonsparking blade guides to eliminate the danger of sparks igniting the magnesium alloy filings.

Cold-working most magnesium alloys at room temperature is very limited because they work-harden very rapidly and do not lend themselves to any severe cold-forming. Some simple bending operations may be performed on sheet material, but the radius of bend must be at least seven times the thickness of the sheet for soft material and 12 times the thickness of the sheet for hard material. A radius of two or three times the thickness of the sheet can be used if the material is heated for the forming operation.

Wrought magnesium alloys tend to crack after they are cold-worked. Therefore, the best results are obtained if the metal is heated to 450° F. before any forming operations are attempted. Parts formed at the lower temperature range are stronger because the higher temperature range has an annealing effect on the metal.

There are some disadvantages to hot-working. First, heating the dies and the material is expensive and troublesome. Second, there are problems in lubricating and handling materials at these temperatures. However, there are some advantages to hot-working magnesium in that it is more easily formed when hot than are other metals and springback is reduced, resulting in greater dimensional accuracy.

When heating magnesium and its alloys, watch the temperature carefully as ths metal is easily burned. Overheating also causes small molten pools to form within the metal. In either case, the metal is ruined. To prevent burning, magnesium must be protected with a sulfur dioxide atmosphere while being heated.

Proper bending around a short radius requires the removal of sharp corners and burrs near the bend line. Layouts should be made with a carpenter's soft pencil because any marring of the surface may result in fatigue cracks.

It is permissible to heat small pieces of magnesium with a blowtorch, provided proper precautions are exercised. It must be remembered that magnesium will ignite when it is heated to a temperature near its boiling point in the presence of oxygen.

Press or leaf brakes can be used for making bends with short radii. Die and rubber methods should be used where bends are to be made at right angles, which complicate the use of a brake. Roll forming may be accomplished cold on equipment designed for forming aluminum. The most common method of forming and shallow drawing magnesium is an operation in which a rubber pad is used as the female die. This rubber pad is held in an inverted steel pan which is lowered by a hydraulic press ram. The press exerts pressure on the metal and bends it to the shape of the male die.

The machining characteristics of magnesium alloys are excellent, making possible the use of maximum speeds of the machine tools with heavy cuts and high feed rates. Power requirements for machining magnesium alloys are about one-sixth of those for mild steel.

Filings, shavings, and chips from machining operations should be kept in covered metal containers because of the danger of combustion. To repeat a previous reminder, in case of a magnesium fire, do not try to extinguish it with water. The oxygen in the water supports the combustion and increases the intensity of the fire. Dry powder (sodium bicarbonate) is the recommended extinguishing agent for magnesium fires.

RIVET LAYOUT

Rivet layout consists of determining (1) the number of rivets required; (2) the size and style of rivet to use; (3) its material, temper condition, and strength; (4) the size of the rivet holes; (5) distance of the rivet holes and rivets from the edges of the patch; and (6) the spacing of the rivets throughout the repair. Since distances are measured in terms of rivet diameters, application of the measurements is simple once the correct rivet diameter is determined.

Single-row, two-row, and three-row layouts designed for small repair jobs are discussed in this section. More complicated layouts for large repairs, which require the application of rivet formulas, are discussed later in this chapter.

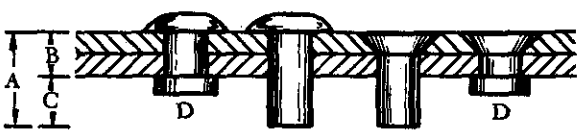
The type of head, size, and strength required in a rivet are governed by such factors as the kind of forces present at the point riveted, the kind and thickness of the material to be riveted, and location of the riveted part on the aircraft.

The type of head required for a particular job is determined by its installation location. Where a smooth aerodynamic surface is required, countersunk head rivets should be used. Universal head rivets may be used in most other locations. If extra strength is required and clearance permits, roundhead rivets may be used; if the necessary clearance is not available, flathead rivets may be used.

The size (or diameter) of the selected rivet shank should correspond in general to the thickness of the material being riveted. If too large a rivet is used in a thin material, the force necessary to drive the rivet properly will cause an undesirable bulging around the rivet head. On the other hand, if too small a rivet diameter is selected for thick material the shear strength of the rivet will not be great enough to carry the load of the joint. As a general rule, the rivet diameter should be not less than three times the thickness of the thicker sheet. Rivets most commonly chosen in the assembly and repair of aircraft range from 3/32-in. to 3/8-in. diameter. Ordinarily, rivets smaller than 3/32-in. diameter are never used on any structural parts which carry stresses.

When rivets are to pass completely through tubular members, select a rivet diameter equivalent to at least one-eighth the outside diameter of the tube. If one tube "sleeves" or fits over another, take the outside diameter of the outside tube and use one-eighth of that distance as the minimum rivet diameter. A good practice is to calculate the minimum rivet diameter and then use the next larger size rivet.

When determining the total length of a rivet for installation, the combined thickness of the materials to be joined must be known. This measurement is known as grip length (B of figure 5–41). The total length of the rivet (A of figure 5–41) should be equal to grip length plus the amount of rivet shank necessary to form a proper shop head. The length of rivet required to form a shop head is 1-1/2 times the diameter of the rivet shank (C of figure 5–41).



A – Total rivet length

B – Grip length

C – Amount of rivet length needed for proper shop head ($1\frac{1}{2} \times$ rivet dia.)

D – Installed rivets

FIGURE 5–41. Determining length of rivet.

Using figure 5–41 and the above information, the formula A = B + C was developed. (A, total rivet length; B, grip length; C, material needed to form a shop head.)

Properly installed rivets are shown in D of figure 5–41. Note carefully the method used to measure total rivet lengths for countersunk rivets and the other types of heads.

Whenever possible, select rivets of the same alloy number as the material being riveted. For example, use 1100 and 3003 rivets on parts fabricated from 1100 and 3003 alloys, and 2117–T and 2017–T rivets on parts fabricated from 2017 and 2024 alloys.

The 2117–T rivet is usually used for general repair work, since it requires no heat treatment, is fairly soft and strong, and is highly corrosion resistant when used with most types of alloys. The 2024–T rivet is the strongest of the aluminum alloy rivets and is used in highly stressed parts. However, it must be soft when driven. Never replace 2024–T rivets with 2117–T rivets.

The type of rivet head to select for a particular repair job can be determined by referring to the type used within the surrounding area by the manufacturer. A general rule to follow on a flush-riveted aircraft is to apply flush rivets on the upper surface of the wing and stabilizers, on the lower leading edge back to the spar, and on the fuselage back to the high point of the wing. Use universal head rivets in all other surface areas.

In general, try to make the spacing of the rivets on a repair conform to that used by the manufacturer in the area surrounding the damage. Aside from this fundamental rule, there is no specific set of rules which governs spacing of rivets in all cases. However, there are certain minimum requirements which must be observed.

The edge distance, or distance from the center of the first rivet to the edge of the sheet, should be not less than two rivet diameters nor more than four. The recommended edge distance is about two and one-half rivet diameters. If rivets are placed too close to the edge of the sheet, the sheet is likely to crack or pull away from the rivets; and if they are spaced too far from the edge, the sheet is apt to turn up at the edges.

Rivet pitch is the distance between the centers of adjacent rivets in the same row. The smallest allowable rivet pitch is three rivet diameters. The average rivet pitch usually ranges from six to eight rivet diameters, although rivet pitch may range from four to 10 rivet diameters. Transverse pitch is the perpendicular distance between rivet rows; it is usually equal to 75% of the rivet pitch. The smallest allowable transverse pitch is two and one-half rivet diameters.

When splicing a damaged tube and the rivets pass completely through the tube, space the rivets four to seven rivet diameters apart if adjacent rivets are at right angles to each other, and space them five to seven rivet diameters apart if the rivets are in line (parallel to each other). The first rivet on each side of the joint should be not less than two and one-half rivet diameters from the end of the sleeve.

The general rules of rivet spacing, as applied to straight-row layout, are quite simple. In a single-row layout, first determine the edge distance at each end of the row then lay off the rivet pitch (distance between rivets) as shown in figure 5–42. In the two-row layout, lay off the first row as just described, place the second row a distance equal to the transverse pitch from the first row, and then lay off rivet spots in the second row so that they fall midway between those in the first row. In the three-row layout, first lay off the first and third rows, then determine the second row rivet spots by using a straightedge. (See figure 5–42.)

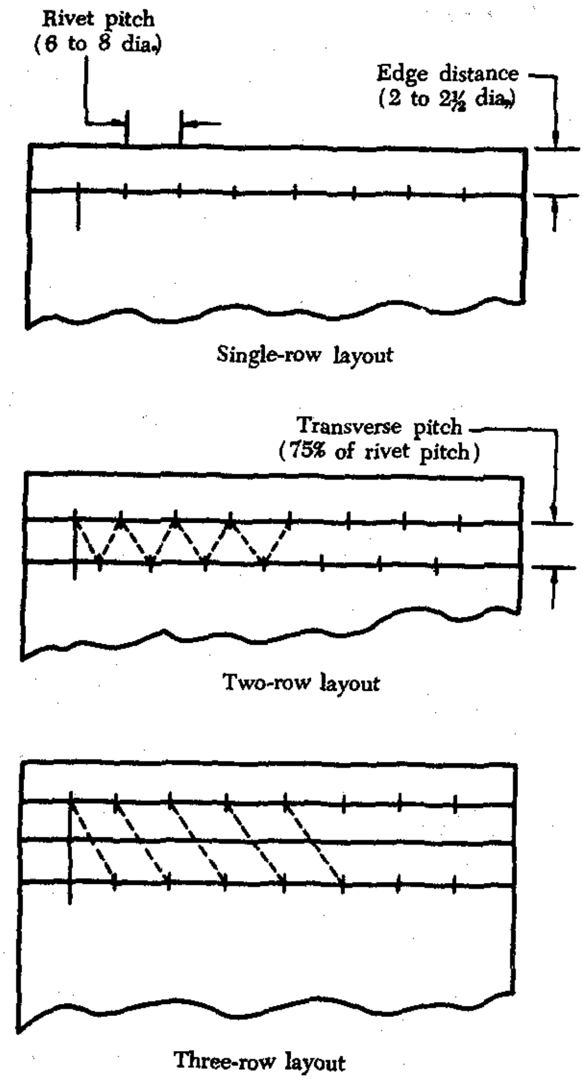


Figure 5-42. Rivet spacing.

RIVET INSTALLATION

The various tools needed in the normal course of driving and upsetting rivets include drills, reamers, rivet cutters or nippers, bucking bars, riveting hammers, draw sets, dimpling dies or other types of countersinking equipment, rivet guns, and squeeze riveters. Self-tapping screws, C-clamps, and fasteners are riveting accessories commonly used to hold sheets together when riveting.

Several of these tools were discussed earlier in this chapter. Other tools and equipment needed in the installation of rivets are discussed in the following paragraphs.

Hole Duplicators

When sections of skin are replaced with new sections, the holes in the replacement sheet or in the patch must be drilled to match existing holes in the structure. These holes can be located with a hole duplicator. The peg on the bottom leg of the duplicator fits into the existing rivet hole. The hole in the new part is made by drilling through the bushing on the top leg. If the duplicator is properly made, holes drilled in this manner will be in perfect alignment. A separate duplicator must be used for each diameter of rivet.

Rivet Cutters

In cases where rivets of the required length are unavailable, rivet cutters can be used to cut rivets to the desired length. When using the rotary rivet cutter, insert the rivet in the correct hole, place the required number of shims under the rivet head, and squeeze as though it were a pair of pliers. Rotation of the disks will cut the rivet to give the right length, which is determined by the number of shims inserted under the head. When using a large rivet cutter, place it in a vise, insert the rivet in the proper hole, and cut by pulling the handle, thus shearing off the rivet. If regular rivet cutters are not available, diagonal cutting pliers can be used as a substitute cutter.

Bucking Bars

A bucking bar is a tool which is held against the shank end of a rivet while the shop head is being formed. Most bucking bars are made of alloy bar stock, but those made of better grades of steel last longer and require less reconditioning. Bucking bars are made in a number of different shapes and sizes to facilitate rivet bucking in all places where rivets are used. Some of the various bucking bars are shown in figure 5–43.

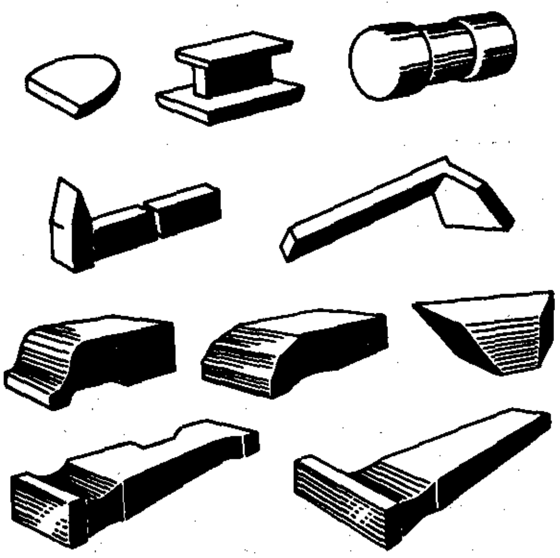


FIGURE 5–43. Bucking bars.

The bars must be kept clean, smooth, and well polished. Their edges should be slightly rounded to prevent marring the material surrounding the riveting operation.

Hand Rivet and Draw Sets

A hand rivet set is a tool equipped with a die for driving a particular type rivet. Rivet sets are available to fit every size and shape of rivet head. The ordinary set is made of 1/2-in. carbon tool steel about 6 in. long and is knurled to prevent slipping in the hand. Only the face of the set is hardened and polished.

Sets for round and brazier head rivets are recessed (or cupped) to fit the rivet head. In selecting the correct set, be sure that it will provide the proper clearance between the set and the sides of the rivet head and between the surfaces of the metal and the set. Flush or flat sets are used for countersunk and flathead rivets. To seat flush rivets properly, be sure that the flush sets are at least 1 in. in diameter.

Special draw sets are used to "draw up" the sheets to eliminate any opening between them before the rivet is bucked. Each draw set has a hole 1/32 in. larger than the diameter of the rivet shank for which it is made. Occasionally, the draw set and rivet header are incorporated into one tool. The header part consists of a hole sufficiently shallow so that the set will expand the rivet and head it when struck with a hammer.

Countersinks

The countersink is a tool which cuts a cone-shaped depression around the rivet hole to allow the rivet to set flush with the surface of the skin. Countersinks are made with various angles to correspond to the various angles of the countersunk rivet heads.

Special stop countersinks are available. Stop countersinks are adjustable to any desired depth, and the cutters are interchangeable so that holes of various countersunk angles can be made. Some stop countersinks have a micrometer set arrangement, in increments of 0.001 in., for adjusting the cutting depths.

Dimpling Dies

The process of making an indentation or a dimple around a rivet hole so that the top of the head of a countersunk rivet will be flush with the surface of the metal is called dimpling. Dimpling is done with a male and female die, or forms, often called punch and die set. The male die has a guide the size of the rivet hole and is beveled to correspond to the degree of countersink of the rivet head. The female die has a hole into which the male guide fits, and is beveled to a corresponding degree of countersink.

When dimpling, rest the female die on a solid surface then place the material to be dimpled on the female die. Insert the male die in the hole to be dimpled and with a hammer strike the male die until the dimple is formed. Two or three solid hammer blows should be sufficient. A separate set of dies is necessary for each size of rivet and shape of rivet head.

An alternate method is to use a countersunk head rivet instead of the regular male punch die, and a draw set instead of the female die, and hammer the rivet until the dimple is formed.

Dimpling dies for light work can be used in portable pneumatic or hand squeezers. If the dies are used with a squeezer, they must, of course, be adjusted accurately to the thickness of the sheet being dimpled.

Pneumatic Rivet Guns

The most common upsetting tool used in airframe repair work is the slow-hitting pneumatic hammer called a rivet gun. Pneumatic guns are available in various sizes and shapes (figure 5–44). The capacity of each gun, as recommended by the manufacturer, is usually stamped on the barrel; pneumatic guns operate on air pressures of from 90 to 100 p.s.i.

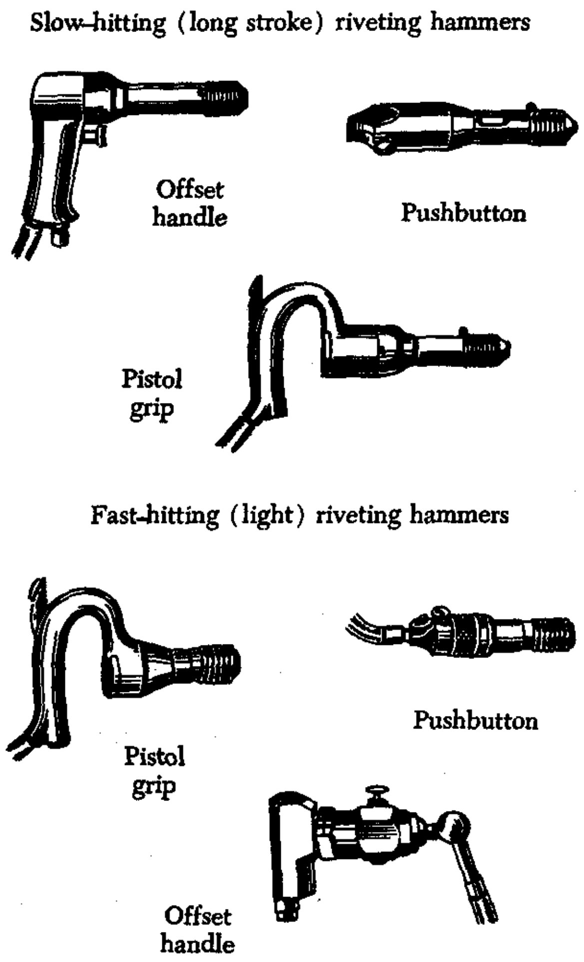


FIGURE 5–44. Types of rivet guns.

Pneumatic guns are used in conjunction with interchangeable rivet sets. Each set is designed to fit the type of rivet and location of the work. The shank of the set is designed to fit into the rivet gun. Force to buck the rivet is supplied by an air-driven hammer inside the barrel of the gun (figure 5–45). The sets are made of high-grade carbon tool steel and are heat treated to give them strength and wear resistance.

Some precautions to be observed when using a rivet gun are:

(1) Never point a rivet gun at anyone at any time. A rivet gun should be used for one purpose only—to drive or install rivets.

(2) Never depress the trigger mechanism unless the set is held tightly against a block of wood or a rivet.

(3) Always disconnect the air hose from the rivet gun when it will not be in use for any appreciable length of time.

Squeeze Riveters

The squeeze method of riveting is limited since it can be used only over the edges of sheets or assemblies where conditions permit, and where the reach of the squeeze riveter is deep enough. There are three types of rivet squeezers—hand, pneumatic, and pneudraulic. They are basically alike except that in the hand rivet squeezer, compression is supplied by hand pressure; in the pneumatic rivet squeezer, by air pressure; and in the pneudraulic, by a combination of air and hydraulic pressure. One jaw is stationary and serves as a bucking bar, the other jaw is movable and does the upsetting. Riveting with a squeezer is a quick method and requires only one operator.

Squeeze riveters are usually equipped with either a C-yoke or an alligator yoke. Yokes are available in various sizes to accommodate any size of rivet. The working capacity of a yoke is measured by its gap and its reach. The gap is the distance between the movable jaw and the stationary jaw; the reach is the inside length of the throat measured from the center of the end sets.

End sets for squeeze riveters serve the same purpose as rivet sets for pneumatic rivet guns and are available with the same type heads. They are interchangeable to suit any type of rivet head. One part of each set is inserted in the stationary jaw, while the other part is placed in the movable jaws. The manufactured head end set is placed on the stationary jaw whenever possible. However, during some operations, it may be necessary to reverse the end sets, placing the manufactured head end set on the movable jaw.

PREPARATION OF RIVET HOLES

It is very important that the rivet hole be of the correct size and shape and free from burrs. If the hole is too small, the protective coating will be scratched from the rivet when the rivet is driven

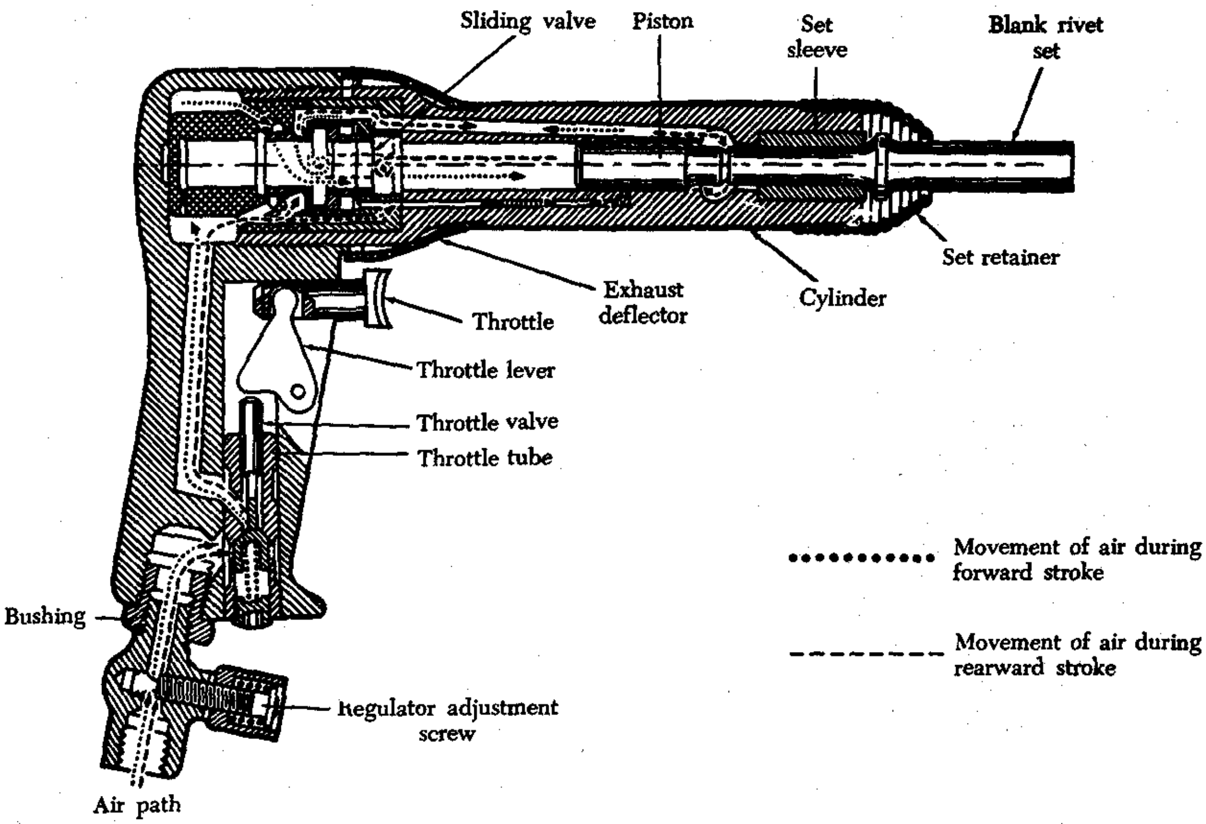


FIGURE 5–45. Rivet gun nomenclature.

through the hole. If the hole is too large, the rivet will not fill the hole completely. When it is bucked, the joint will not develop its full strength, and structural failure may occur at that spot.

If countersinking is required, consider the thickness of the metal and adopt the countersinking method recommended for that thickness. If dimpling is required, keep hammer blows or dimpling pressures to a minimum so that no undue work-hardening occurs in the surrounding area.

Drilling

To make a rivet hole of the correct size, first drill a hole slightly undersize. This is known as predrilling, and the hole is called a pilot hole. Ream the pilot hole with a twist drill of the correct size to get the required dimension. Pilot and reaming drill sizes are shown in figure 5–46. The recommended clearance for rivet holes is from 0.002 to 0.004 in.

When drilling hard metals the twist drill should have an included angle of 118° and should be operated at low speeds; but for soft metals, use a twist drill with an included angle of 90° and it should be operated at higher speeds. Thin sheets of aluminum alloy are drilled with greater accuracy by a drill having an included angle of 118° because the large angle of the drill has less tendency to tear or elongate the hole.

Center punch locations for rivet holes before beginning the actual drilling. The center punch mark

Rivet Diameter	Pilot Size	Ream Size
3/32	3/32 (.0937)*	40 (.098)
1/8	1/8 (.125)	30 (.1285)
5/32	5/32 (.1562)	21 (.159)
3/16	3/16 (.1875)	11 (.191)
1/4	1/4 (.250)	F (.257)
5/16	5/16 (.3125)	O (.316)
3/8	3/8 (.375)	V (.377)

*Note that ream size exceeds the maximum tolerance of .004 inch. This is permissible only if the next larger drill size happens to be so much larger than the tolerance of .004 inch.

FIGURE 5–46. Pilot and reaming twist drill sizes.

acts as a guide and lets the drill grip or bite into the metal with greater ease. Make the center punch mark large enough to prevent the drill from slipping out of position, but punch lightly enough not to dent the surrounding material. Hold a hard, smooth, wooden backing block securely in position behind the hole locations when drilling.

Drilling is usually done with a hand drill or with a light power drill. Hold the power drill firmly with both hands. Extend the index and middle fingers of the left hand against the metal to act as a guide in starting a hole, and as a snubber or brake when the drill goes through the material. Before beginning to drill, always test the inserted twist drill for trueness and vibration by spinning the hand drill or running the motor freely and watching the drill end. If the drill wobbles, it may be because of burrs on its shank or because the drill is bent or incorrectly chucked. A drill that wobbles or is slightly bent must not be used because it causes enlarged holes.

Always hold the drill at right angles to the work, regardless of the position of the hole or the curvature of the material. Use an angle drill or drill extensions and adapters when access is difficult with a straight drill. Never tip the drill sideways when drilling or when withdrawing from the material because this causes elongation of the hole.

When holes are drilled through sheet metal, small burrs are formed around the edge of the hole. This is especially true when using a hand drill since the drill speed is slow and there is a tendency to apply more pressure per drill revolution. Remove all burrs with a burr remover before riveting.

Countersinking and Dimpling

An improperly made countersink reduces the strength of a flush-riveted joint and may even cause failure of the sheet or the rivet head. The two methods of countersinking commonly used for flush riveting in aircraft construction and repair are the machine or drill countersinking, and dimpling or press countersinking. The proper method for any particular application depends on the thickness of the parts to be riveted, the height and angle of the countersunk head, the tools available, and accessibility.

As a general rule, use the drill countersink method when the thickness of the material is greater than the thickness of the rivet head, and use the dimpling method on thinner material. Figure 5–47 illustrates general rules for countersinking. Note in figure 5–47A that the material is quite thick and the head of the countersunk rivet extends only about halfway through the upper layer of metal. Countersinking will leave plenty of material for gripping.

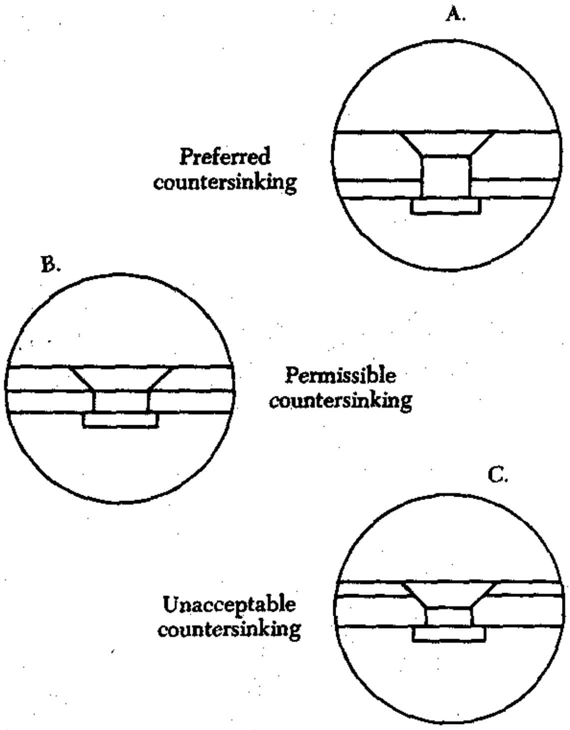


FIGURE 5–47. Countersinking.

In figure 5–47B, the countersunk head reaches completely through the upper layer. This condition is permissible but should be avoided.

In figure 5–47C, the head extends well into the second layer of material. This indicates that the material is thin and that most of it would be ground away by drill countersinking; therefore, dimpling is preferred. Dimpling will work best if the material is not over 0.040-in. thick.

Machine or drill countersinking is accomplished by a suitable cutting tool machined to the desired angle. The edge of the hole is cut away so that the countersunk rivet head fits snugly into the recess. The resulting recess is referred to as the "well" or "nest."

During the process of machine countersinking, first drill the original rivet hole to the exact rivet size, as recommended in the table in figure 5–46. The limits within which the head of the rivet may extend either above or below the surface of the metal are close, 0.006 in. in most cases. Therefore, perform the countersinking accurately, using equipment which is capable of producing results within the specified tolerance.

Hold the countersinking tool firmly at right angles to the material. Do not tip it. Tipping elongates the well and prevents the countersunk rivet head from fitting properly. Oversized rivet holes, undersized countersink pilots (in the case of the stop countersink), chattering caused by improper use of the countersink or by a countersink in poor condition, and a countersink not running true in the chuck of the drill are some of the causes of elongated wells.

Press countersinking or dimpling can be accomplished by either of two methods. Male and female die sets can be used, or using the rivet as the male die and the draw die as the female die is acceptable. In either case, the metal immediately surrounding the rivet hole is pressed to the proper shape to fit the rivet head. The depression thus formed, as in machine countersinking, is known as the "well" or "nest."

The rivet must fit the well snugly to obtain maximum strength. The number of sheets which can be dimpled simultaneously is limited by the capacity of the equipment used. The dimpling process may be accomplished by the use of hand tools, by dies placed in a pneumatic squeeze or single shot riveter, or by using a pneumatic riveting hammer.

Dimpling dies are made to correspond to any size and degree of countersunk rivet head available. The dies are usually numbered, and the correct combination of punch and die to use is indicated on charts specified by the manufacturer. Both male and female dies are machined accurately and have highly polished surfaces. The male die or punch is cone shaped to conform to the rivet head and has a small concentric pilot shaft that fits into the rivet hole and female die. The female die has a corresponding degree of countersink into which the male guide fits.

When dimpling a hole, rest the female die on some solid surface, place the material on the female die, insert the male die in the hole to be dimpled, and then hammer the male die. Strike with several solid blows until the dimple is formed.

In some cases, the face of the male die is convex to allow for springback in the metal. Dies of this type are used to advantage when the sheet to be dimpled is curved. Some dies have flat faces and are principally used for flat work. Dimpling dies are usually made so that their included angle is 5° less than that of the rivet. This arrangement allows for springback of the metal.

In die dimpling, the pilot hole of the female die should be smaller than the diameter of the rivet to be used. Therefore, the rivet hole must be reamed to the exact diameter after the dimpling operation has been completed so that the rivet fits snugly.

When using a countersink rivet as the male dimpling die, place the female die in the usual position and back it with a bucking bar. Place the rivet of the required type into the hole and strike the rivet with a pneumatic riveting hammer. This method of countersinking is often called "coin pressing." It should be used only when the regular male die is broken or not available.

Coin pressing has a distinct disadvantage in that the rivet hole must be drilled to correct rivet size before the dimpling operation is accomplished. Since the metal stretches during the dimpling operation, the hole becomes enlarged and the rivet must be swelled slightly before driving to produce a close fit. Because the rivet head will cause slight distortions in the recess, and these are characteristic only to that particular rivet head, it is wise to drive the same rivet that was used as the male die during the dimpling process. Do not substitute another rivet, either of the same size or a size larger.

Thermo-Dimpling

This type of dimpling consists of two processes, radius dimpling and coin dimpling. The major difference between radius and coin dimpling is in the construction of the female die. In radius dimpling a solid female die is used. Coin dimpling uses a sliding ram female die (figure 5–48) that makes this process superior.

During the coin dimpling process, the metal is coined (made to flow) into the contours of the dies so that the dimple assumes the true shape of the die. The pressure exerted by the coining ram prevents the metal from compressing and thereby assures uniform cross sectional thickness of the sides of the dimple and a true conical shape.

Coin dimpling offers several advantages. It improves the configuration of the dimple, produces a more satisfactory aerodynamic skin surface, eliminates radial and circumferential cracking, ensures a stronger and safer joint, and allows identical dies to be used for both skin and understructure dimpling.

The material being used is a very important factor to consider in any dimpling operation. Materials such as corrosion-resistant steel, magnesium,

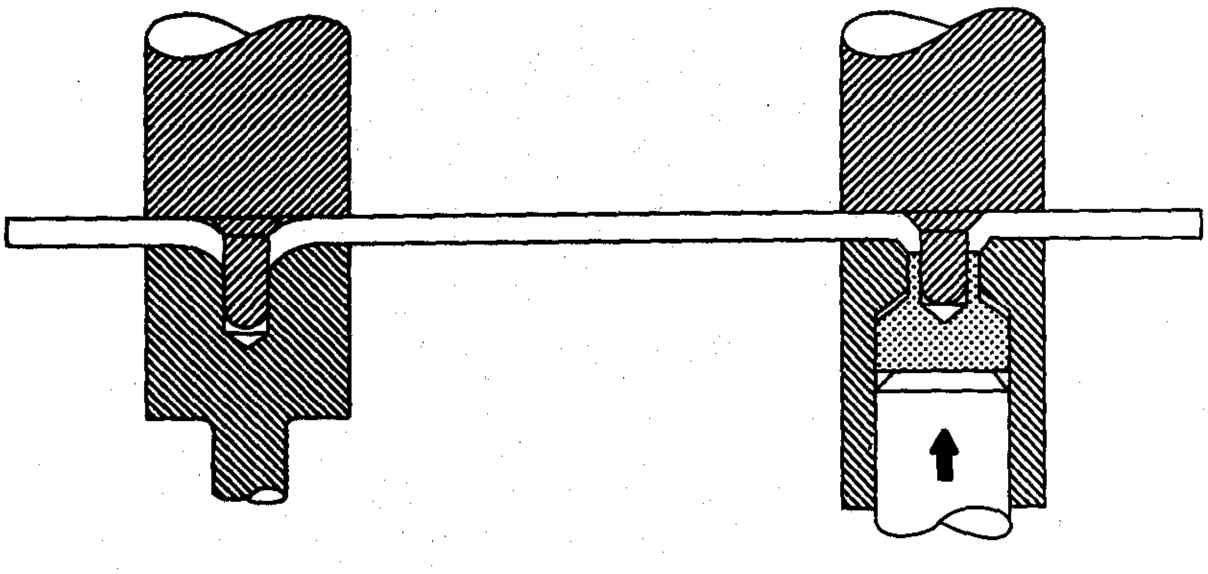


FIGURE 5–48. Radius and coin dimpling dies.

and titanium each present different dimpling problems.

The 2024–T aluminum alloy can be satisfactorily coin dimpled either hot or cold. However, cracking in the vicinity of the dimple may result from cold dimpling because of hard spots in the metal. Hot dimpling will prevent such cracking.

The 7075–T6 and 2024–T81 aluminum alloys are always hot dimpled. Magnesium alloys also must be hot dimpled because, like 7075–T6, they have low formability qualities. Titanium is another metal that must be hot dimpled because it is tough and resists forming. The same temperature and dwell time used to hot dimple 7075–T6 is used for titanium.

Corrosion-resistant steel is cold dimpled because the temperature range of the heating unit is not high enough to affect dimpling.

The coin ram dimpling dies are designed with a number of built-in features. The faces of both the male and female dies are dished (the male concave and the female convex) at an angle of 2° on the pilot. This facilitates removal of the metal after the dimple has been made.

The female dimpling set has two parts: (1) The body, which is merely a counterpart of the male die; and (2) the coining ram, which extends up through the center of the conical recess of the body. In forming a dimple, the metal is forced down into the female die by the male die. The metal first contacts the coining ram, and this supports the metal as it is forced down into the conical recess. When the two dies close to the point where the forces of both are squeezing the material, the coining ram forces the metal back into the sharp corners of the dies.

When cold dimpling, the dies are used alone. When hot dimpling, a strap or block heater is slipped over either or both dies and connected to an electric current.

The dies should be kept clean at all times and in good working order. It is advisable to clean them regularly with steel wool. Special precautions must be taken when the dies are in the machine. If the machine is operated with the dies in place but without material between them, the male die will enlarge and ruin the coining ram.

When possible, coin dimpling should be performed on stationary equipment and before the assembly of parts. However, many instances arise in which dimpling must be done after parts are assembled to other structures. In such cases, dimpling operations are performed by portable squeeze dimplers. Most squeezers may be used either for cold dimpling or, combined with a junction box, for hot dimpling.

There are dimpling applications in which it is not possible to accommodate any squeezer- or yoke-type equipment. Under these circumstances, it is necessary to use a pneumatic hammer and a bucking bar type of tool to hold the dimpling dies.

DRIVING RIVETS

The methods of driving solid shank rivets can be classified into two types, depending on whether the riveting equipment is portable or stationary. Since stationary riveting equipment is seldom used in airframe repair work, only portable equipment that is used in hand, pneumatic, or squeezer methods is discussed here.

Before driving any rivets, be sure that all holes line up perfectly, all shavings and burrs have been removed, and that the parts to be riveted are securely fastened together.

Two men, a "gunner" and a "bucker," usually work as a team when installing rivets. However, on some jobs the riveter holds a bucking bar with one hand and operates a riveting gun with the other. When team riveting, an efficient signal system can be employed to develop the necessary teamwork. The code usually consists of tapping the bucking bar against the work; one tap may mean "not fully seated, hit it again"; two taps may mean "good rivet"; three taps may mean "bad rivet, remove and drive another"; and so on.

Bucking

Selection of the appropriate bucking bar is one of the most important factors in bucking rivets. If the bar does not have the correct shape, it will deform the rivet head; if the bar is too light, it will not give the necessary bucking weight, and the material may become bulged toward the shop head; and, if the bar is too heavy, its weight and the bucking force may cause the material to bulge away from the shop head. Weights of bucking bars range from a few ounces to 8 or 10 lbs., depending upon the nature of the work. Recommended weights of bucking bars to be used with various rivet sizes are given in figure 5–49.

Rivet Diameter (In Inches)	Approx. Weight (In Pounds)
3/32	2 to 3
1/8	3 to 4
5/32	3 to 4½
3/16	4 to 5
1/4	5 to 6½

FIGURE 5–49. Recommended bucking bar weights.

Always hold the face of the bucking bar at right angles to the rivet shank. Failure to do this will cause the rivet shank to bend with the first blows of the rivet gun, and will cause the material to become marred with the final blows. The bucker must hold the bucking bar in place until the rivet is completely driven. If the bucking bar is removed while the gun is in operation, the rivet set may be driven through the material. Do not bear down too heavily on the shank of the rivet. Allow the weight of the bucking bar to do most of the work. The hands merely guide the bar and supply the necessary tension and rebound action.

Allow the bucking bar to vibrate in unison with the gun set. This process is called coordinated bucking. Coordinated bucking can be developed through pressure and stiffness applied at the wrists; with experience, a high degree of deftness can be obtained.

Lack of proper vibrating action, the use of a bucking bar that is too light or too heavy, and failure to hold the bucking bar at right angles to the rivet can all cause defective rivet heads. A rivet going "clubhead" (malforming) can be corrected by rapidly moving the bucking bar across the rivet head in a direction opposite that of clubhead travel. This corrective action can be accomplished only while the gun is in action and the rivet is partly driven. If a rivet shank bends at the beginning of the bucking operation, place the bar in the corrective position only long enough to straighten the shank.

Hand Driving

Under certain conditions, it may be necessary to rivet by hand driving. Either of two methods can be used depending upon the location and accessibility of the work. In the one method, the manufactured head end of the rivet is driven with a hand set and hammer, the shank end is bucked with a suitable bucking bar. In the other method, the shank end of the rivet is driven with a hand set and a hammer, and the manufactured head is bucked with a hand set held in a vise or a bottle bar (a special bucking bar recessed to hold a rivet set). This method is known as reverse riveting. It is commonly used in hand riveting but is not considered good practice in pneumatic riveting.

When using either of the described methods, keep hammer strokes to a minimum. Too much hammering will change the crystalline structure of the rivet or the material around it, causing the joint to lose some of its strength. Hold the bucking bar and rivet set square with the rivet at all times. Misuse of the rivet set and bucking bar will result in marring or scratching the rivet head or material, and may cause undue corrosion. This, in turn, will weaken the structure of the aircraft.

The diameter of a correctly formed shop head should be one and one-half times the diameter of the rivet shank, and the height should be about one-half the diameter.

Pneumatic Driving

The procedure for pneumatic riveting is practically the same as for hand riveting. Preparation of the sheet, selection of rivets, and drilling of rivet holes are the same. In hand riveting, however, the pressure for bucking the rivet is applied using a hand set and hammer. In pneumatic riveting, the pressure is applied with a set and an air-driven hammer or gun.

To get good riveting results with a pneumatic rivet gun, follow these basic pointers:

(1) Select the right type and size of rivet gun and the correct rivet set for the size of rivet to be driven. Install the rivet set firmly, as shown in figure 5–50.

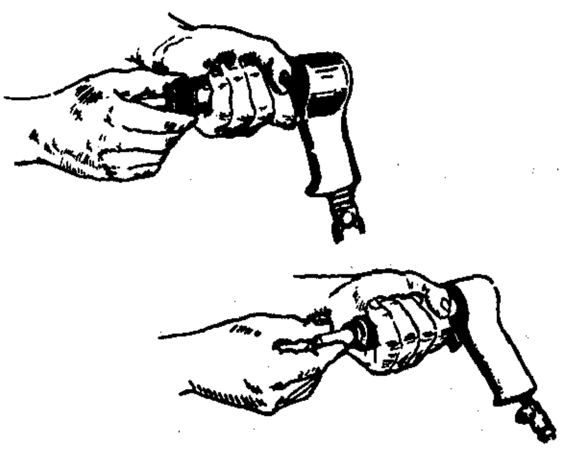


FIGURE 5–50. Installing rivet set.

(2) Adjust the speed of the riveting gun (vibrations per minute). Always press set firmly against a block of wood before pressing the trigger. Never operate the gun without resistance against the set because the vibrating action may cause the retaining spring to break, allowing the gun set to fly out of the gun. Also, free vibration may flare or mushroom the gun end of the set, causing it to bind in the barrel of the gun.

(3) Hold the rivet set at right angles to the work to prevent damage to the rivet head or the surrounding material as shown in figure 5–51. Upset the rivet with a medium burst from the rivet gun.

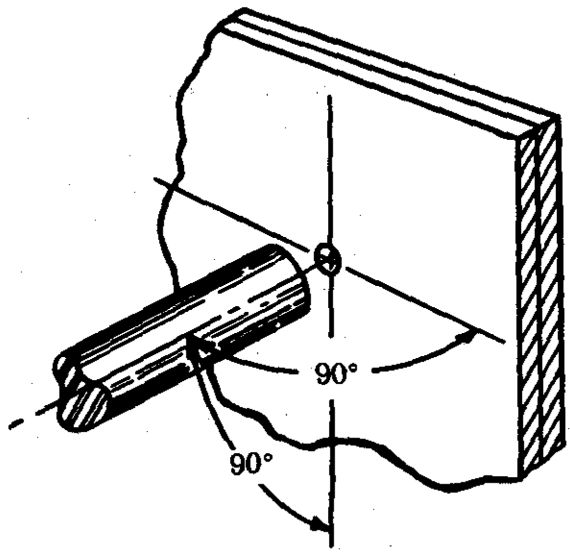


FIGURE 5–51. Position of the set.

(4) Remove the bucking bar and check the shop head of the rivet. It should be one and one-half times the diameter of the rivet in width and one-half times the rivet diameter in height. If the rivet needs further driving, repeat the necessary procedures to complete the job.

A small piece of adhesive tape applied to the cupped end of the rivet set often corrects an unsatisfactory cupped condition, which occasionally gives trouble in forming uniformly shaped rivet heads.

Squeeze Riveting

The squeeze method of driving a rivet produces the most uniform and balanced type of shop head. Each rivet is upset in a single operation; all rivets are headed over with uniform pressure; all heads are formed alike; and each rivet shank is sufficiently and uniformly expanded to completely fill each rivet hole. Squeeze riveters come equipped with pairs of end sets, each pair being designed for a particular job. Once the correct end set is selected and the squeezer adjusted for a particular application, all the rivets will be driven uniformly, thus providing an efficient method of riveting.

Portable squeezers are particularly suited for riveting large assemblies where the tool must be moved in relation to the work. They are not too heavy and can easily be operated by one person. The preparation of the material for riveting with the squeeze riveter is the same as for hand or pneumatic riveting. For better results when using the squeeze riveter, observe these rules:

(1) Carefully select and insert suitable end sets to match the rivet being used. The importance of using the right end sets cannot be overemphasized. It is impossible to buck the rivet properly unless the correct pairs are used. Always be sure that the air is shut off or the squeezer is disconnected when inserting end sets.

(2) Adjust the squeezer cylinder pressure to obtain the correct pressure for the diameter of the rivet being used. Most squeezers are equipped with a blowoff valve to regulate the cylinder pressure. This unit governs the amount of air pressure allowed in the cylinder.

(3) Carefully regulate the gap to conform to the length of the rivet being used. Some squeezers are equipped with a gap regulator which controls the stroke of the plunger of a C-yoke squeezer, or the movement of the movable jaw of an alligator-yoke squeezer. For squeezers not equipped with a gap regulator, the gap can be adjusted by placing metal shims under the end sets of both jaws, or by using end sets of different lengths. On some types of squeeze riveters, the end set on the stationary jaw is held in place by an Allen screw, which allows regulation of the gap.

(4) Before using the squeezer on the work, test the cylinder pressure and gap for accuracy of adjustment on a piece of scrap material. The scrap material must be the same thickness as the material being used, and the rivets the same length and diameter.

(5) If the parts to be riveted are small and easily handled, mount the squeeze riveter in a bench vise or in a special clamp, and hold the part to be riveted in your hand.

Microshaving

Sometimes it is necessary to use a microshaver when making a repair involving the use of countersunk rivets. If the smoothness of the material (such as skin) requires that all countersunk rivets be driven within a specific tolerance, a microshaver is used. This tool has a cutter, stop, and two legs or stabilizers, as shown in figure 5–52.

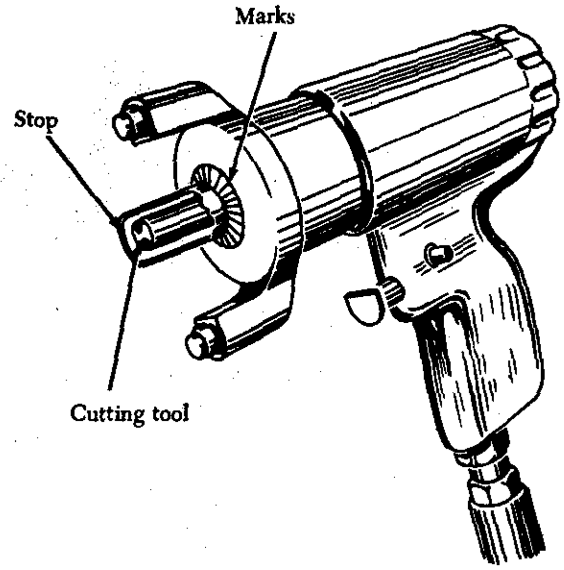


FIGURE 5–52. Microshaver.

The cutting portion of the microshaver is located inside the stop. The depth of cut can be adjusted by pulling outward on the stop and turning it in either direction (clockwise for deeper cuts). The marks on the stop permit adjustments of 0.001 in.

If the microshaver is adjusted and held correctly, it will cut the head of a countersunk rivet to within 0.002 in. without damaging the surrounding material. Adjustments should always be made on scrap material. When correctly adjusted, the shaver will leave a small round dot about the size of a pinhead on the microshaved rivet.

RIVET FAILURES

Generally speaking, the design of riveted joints is based on the theory that the total joint strength is simply the sum of the individual strengths of a whole group of rivets. It is then obvious that, if any one rivet fails, its load must immediately be carried by others of the group; if they are unable to carry this added load, progressive joint failure then occurs. Stress concentrations will usually cause one rivet to fail first; and careful analysis of such a rivet in a joint will indicate that it has been too highly loaded, with the possibility that neighboring rivets may have partially failed.

Shear Failure

Shear failure is perhaps the most common of rivet failures. It is simply a breakdown of the rivet shank by forces acting along the plane of two adjacent sheets, causing a slipping action which may be severe enough to cut the rivet shank in two. If the shank becomes loaded beyond the yield point of the material and remains overloaded, a permanent shift is established in the sheets and the rivet shank may become joggled.

Bearing Failure

If the rivet is excessively strong in shear, bearing failure occurs in the sheet at the edge of the rivet hole. The application of large rivets in thin sheets brings about such a failure. In that case, the sheet is locally crushed or buckled, and the buckling destroys the rigidity of the joint. Vibrations, set up by engine operation or by air currents in flight, may cause the buckled portion to flutter and the material to break off close to the rivet head. If buckling occurs at the end of the sheet, a tear-out may result. In either case, replacement of the sheet is necessary.

Head Failure

Head failure may result from complex loadings occuring at a joint, causing stresses of tension to be applied to the rivet head. The head may fail by shearing through the area corresponding to the rivet shank, or, in thicker sheets, it may fail through a prying action which causes failure of the head itself. Any visible head distortion is cause for replacement. This latter type of head failure is especially common in blind rivets.

Rivet Inspection

To obtain high structural efficiency in the manufacture and repair of aircraft, an inspection must be made of all rivets before the part is put in service. This inspection consists of examining both the shop and manufactured heads and the surrounding skin and structural parts for deformities. A scale or rivet gage can be used to check the condition of the upset rivet head to see that it conforms to the proper requirements. Deformities in the manufactured head can be detected by the trained eye alone. However, on flush rivets, a straightedge can be used as shown in figure 5–53.

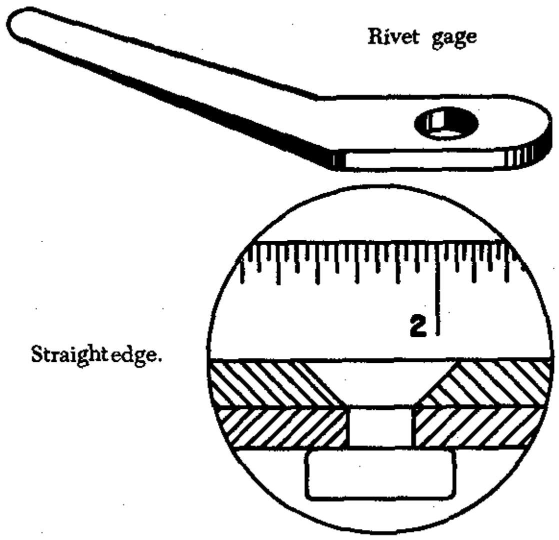


FIGURE 5–53. Tools used to gage rivets.

Some common causes of unsatisfactory riveting are improper bucking, rivet set slipping off or being held at the wrong angle, and rivet holes or rivets of the wrong size. Additional causes for unsatisfactory riveting are countersunk rivets not flush with the well; work not properly fastened together during riveting; the presence of burrs, rivets too hard, too much or too little driving; and rivets out of line.

Occasionally, during an aircraft structural repair, it is wise to examine adjacent parts to determine the true condition of neighboring rivets. In doing so, it may be necessary to remove the paint. The presence of chipped or cracked paint around the heads may indicate shifted or loose rivets. Look for tipped or loose rivet heads. If the heads are tipped or if rivets are loose, they will show up in groups of several consecutive rivets and will probably be tipped in the same direction. If heads which appear to be tipped are not in groups and are not tipped in the same direction, tipping may have occurred during some previous installation.

Inspect rivets known to have been critically loaded, but which show no visible distortion, by drilling off the head and carefully punching out the shank. If, upon examination, the shank appears joggled and the holes in the sheet misaligned, the rivet has failed in shear. In that case, try to determine what is causing the shearing stress and take the necessary corrective action. Flush rivets that show head slippage within the countersink or dimple, indicating either sheet bearing failure or rivet shear failure, must be removed for inspection and replacement.

Joggles in removed rivet shanks indicate partial shear failure. Replace these rivets with the next larger size. Also, if the rivet holes show elongation, replace the rivets with the next larger size. Sheet failures (such as tear-outs, cracks between rivets, and the like) usually indicate damaged rivets, and the complete repair of the joint may require replacement of the rivets with the next larger size.

The general practice of replacing a rivet with the next larger size (1/32 in. greater diameter) is necessary to obtain the proper joint strength of rivet and sheet when the original rivet hole is enlarged. If the rivet in an elongated hole is replaced by a rivet of the same size, its ability to carry its share of the shear load is impaired and joint weakness results.

REMOVING RIVETS

When removing a rivet for replacement, be very careful so that the rivet hole will retain its original size and shape and replacement with a larger size rivet will not be necessary. If the rivet is not removed properly, the strength of the joint may be weakened and the replacement of rivets made more difficult.

When removing a rivet, work on the manufactured head. It is more symmetrical about the shank than the shop head, and there will be less chance of damaging the rivet hole or the material around it. To remove rivets, use hand tools, a power drill, or a combination of both. The preferred method is to drill through the rivet head and drive out the remainder of the rivet with a drift punch. First, file a flat area on the head of any round or brazier head rivet, and center punch the flat surface for drilling. On thin metal, back up the rivet on the upset head when center punching to avoid depressing the metal. The dimple in 2117–T rivets usually eliminates the necessity of filing and center punching the rivet head.

Select a drill one size smaller than the rivet shank and drill out the rivet head. When using a power drill, set the drill on the rivet and rotate the chuck several revolutions by hand before turning on the power. This procedure helps the drill cut a good starting spot and eliminates the chance of the drill slipping off and tracking across the metal. Drill the rivet to the depth of its head, while holding the drill at a 90° angle. Be careful not to drill too deep because the rivet shank will turn with the drill and cause a tear. The rivet head will often break away and climb the drill, which is a good signal to withdraw the drill. If the rivet head does not come loose of its own accord, insert a drift punch into the hole and twist slightly to either side until the head comes off.

Drive out the shank of the rivet with a drift punch slightly smaller than the diameter of the shank. On thin metal or unsupported structures, support the sheet with a bucking bar while driving out the shank. If the shank is exceptionally tight after the rivet head is removed, drill the rivet about two-thirds of the way through the thickness of the material and then drive out the remainder of the rivet with a drift punch.

The procedure for the removal of flush rivets is the same as that just described except that no filing is necessary. Be very careful to avoid elongation of the dimpled or the countersunk holes. The rivet head should be drilled to approximately one-half the thickness of the top sheet.

SPECIAL RIVETS

There are many places on an aircraft where access to both sides of a riveted structure or structural part is impossible, or where limited space will not permit the use of a bucking bar. Too, in the attachment of many nonstructural parts, such as aircraft interior furnishings, flooring, deicing boots, and the like, the full strength of solid shank rivets is not necessary.

For use in such places, special rivets have been designed which can be bucked from the front. They are sometimes lighter than solid shank rivets, yet amply strong for their intended use. These rivets are manufactured by several corporations and have unique characteristics that require special installation tools, special installation procedures, and special removal procedures. Because these rivets are often inserted in locations where one head (usually the shop head) cannot be seen, they are also called blind rivets.

The various types of mechanically expanded rivets, their fabrication, composition, uses, selection, and identification were discussed in Chapter 6, Hardware, Materials, and Processes, in the Airframe and Powerplant Mechanics General Handbook, AC 65–9A. The installation techniques will be covered in this section.

Installation Tools

The tools used to install self-plugging (friction lock) rivets depend upon the manufacturer of the rivet being installed. Each company has designed special tools which should always be used to ensure satisfactory results with its product. Hand tools as well as pneumatic tools are available.

After selection or determination of the rivet to be used in any installation, the proper size twist drill must be determined. Generally, manufacturers recommend the following finish drill sizes for the common shank diameters (figure 5–54).

Be very careful when drilling the material. Hold the drill at right angles to the work at all times to keep from drilling an elongated hole. The blind rivet will not expand as much as a solid shank rivet. If the hole is too large or elongated, the shank will not properly fill the drilled hole. Common hand or pneumatic powered drills can be used to drill the holes. Some manufacturers recommend predrilling the holes; others do not.

Equipment used to pull the stem of the rivet, as previously stated, will depend upon the manufacturer of the rivet. Both manually operated and power-operated guns are manufactured for this purpose. Nomenclature for various tools and assemblies available depends upon the manufacturer. Application and use of the equipment is basically the same. Whether the equipment is called hand tool, air tool, hand gun, or pneumatic gun (figure 5–55), all of these are used with but one goal, the proper installation of a rivet.

The choice of installation tools is influenced by several factors: The quantity of rivets to be installed, the availability of an air supply, the accessibility of the work, and the size and type of rivet to be installed. In addition to a hand or power riveter, it is necessary to select the correct "pulling head" to complete the installation tool.

Selection of the proper pulling head is of primary importance since it compensates for the variables of head style and diameter. Since your selection will depend on the rivets to be installed, you should consult the applicable manufacturer's literature.

SELF-PLUGGING (FRICTION LOCK) RIVETS

Self-plugging (friction lock) rivets are fabricated in two common head styles: (1) A protruding head similar to the AN470 or universal head, and (2) a 100° countersunk head. Other head styles are available from some manufacturers.

The stem of the self-plugging (friction lock) rivet may have a knot or knob on the upper portion, or it may have a serrated portion as shown in figure 5–56.

The sequence of steps to follow in the installation of self-plugging (friction lock) rivets is basically the same as that for solid shank rivets, but the methods and equipment vary. The following steps are typical of any installation:

(1) Select the rivet to be installed—determined by thickness of material to be riveted, strength desired in assembly, and location of installation (protruding or countersunk head).

(2) Drill the hole(s)—determine size of twist drill to be used, do not elongate rivet hole, remove burrs, and use a stop countersink if necessary.

CHERRYLOCK			
Rivet Diam.	Drill Size	Minimum	[illegible]
3/32	#40	.097	.100
1/8	#30	.129	.132
5/32	#20	.160	.164
3/16	#10	.192	.196
1/4	F	.256	.261
BULBED CHERRYLOCK			
1/8	#27	.143	.146
5/32	#16	.176	.180
3/16	#5	.205	.209

COUNTERSINKING DIMENSIONS

	100° MS20426 HEAD		100° NAS1097 HEAD		90° UNIS[illegible]	
Rivet Diam.	C Max.	C Min.	C Max.	C Min.	C Max.	C Min.
3/32	.182	.176	—	—	—	—
1/8	.228	.222	.195	.189	.173	.167
5/32	.289	.283	.246	.240	.216	.210
3/16	.356	.350	.302	.296	.258	.252
1/4	.479	.473	.395	.389	—	—

FIGURE 5–54. Cherry rivet installation data.

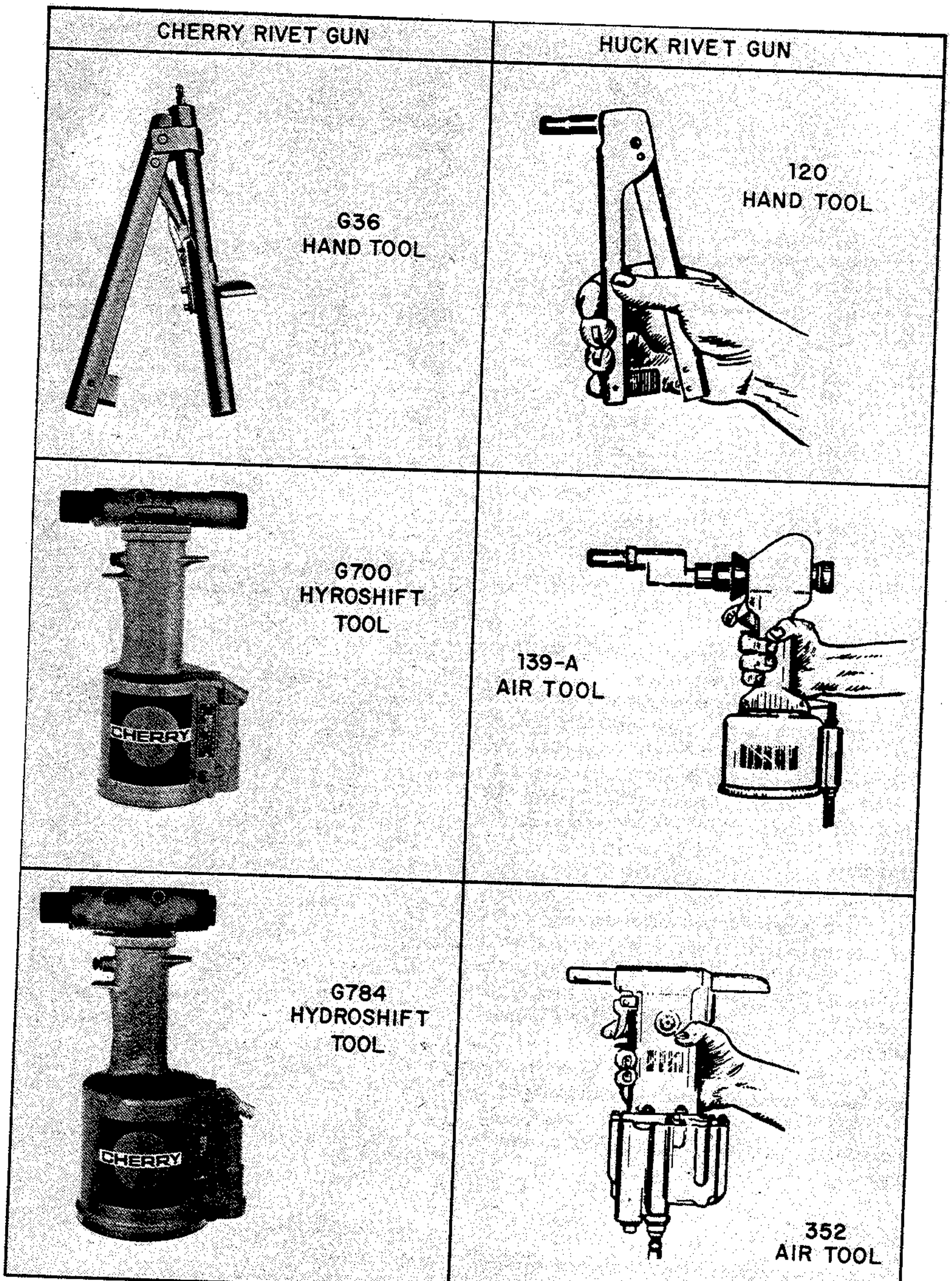


FIGURE 5-55. Self-plugging (friction lock) rivet guns.

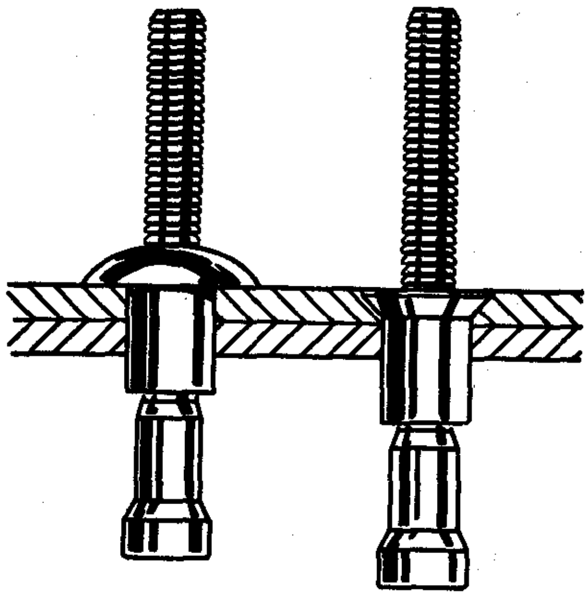


FIGURE 5-56. Self-plugging (friction lock) rivets.

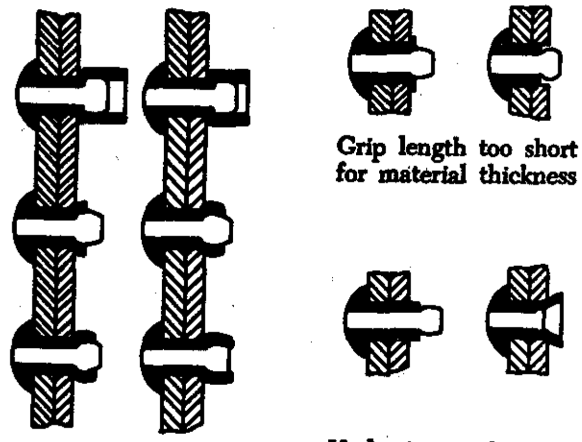


FIGURE 5-57. Inspection of self-plugging (friction lock) rivets.

(3) Install the rivet—make certain the rivet head is seated firmly, position the selected tool on the rivet stem, pull rivet stem until the stem snaps, apply approximately 15 lbs. of pressure to the end of the stem, and trim the stem flush with the rivet head. If aerodynamic smoothness is a factor, the stem can be shaved with a rivet shaver.

Inspection

The inspection of installed self-plugging (friction lock) rivets is very limited. Often the only inspection that can be made is on the head of the rivet. It should fit tightly against the metal. The stem of the rivet should be trimmed flush with the head of the rivet whether it is a protruding head or a countersunk head.

If you can see the shop head side of the installed rivet, inspect it for the requirements illustrated in figure 5-57. When the rivet head is considered unsatisfactory, remove the rivet and install another in its place.

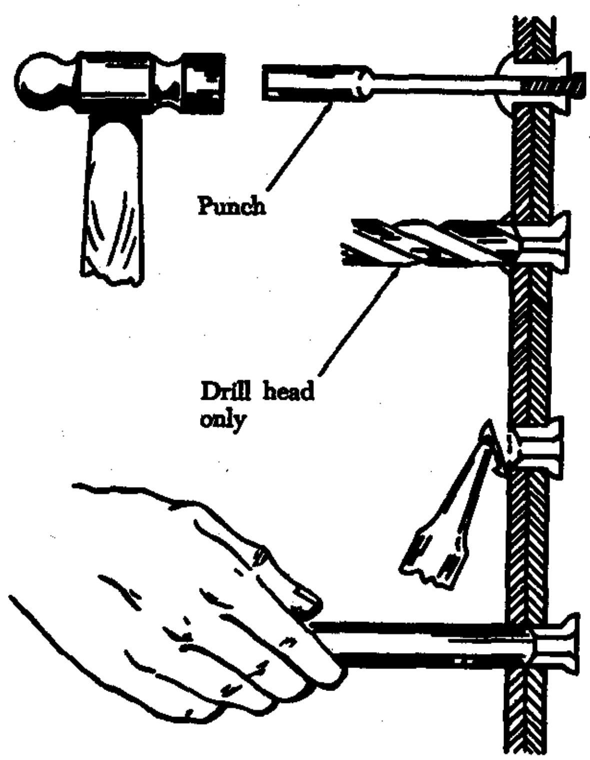


FIGURE 5-58. Removal of self-plugging (friction lock) rivets.

Removal Procedures

Self-plugging (friction lock) rivets are removed in the same manner as solid shank rivets except for the preliminary step of driving out the stem (figure 5-58). The following steps should be used in their proper sequence:

(1) Punch out the rivet stem with a pin punch.

(2) Drill out the rivet head, using a drill the same size as the rivet shank.
(3) Pry off the weakened rivet head with a pin punch.
(4) Push out the remainder of the rivet shank with a punch. If the shank will not push out, drill the shank, taking care not to enlarge the hole in the material.

SELF-PLUGGING (MECHANICAL LOCK) RIVETS

Self-plugging, mechanical lock rivets are similar to self-plugging, friction lock rivets, except for the manner in which they are retained in the material. This type of rivet has a positive mechanical locking collar that resists vibration that would cause the friction lock rivets to loosen and possibly fall out (figure 5–59). Also the mechanical locking type rivet stem breaks off flush with the head and usually does not require further stem trimming when properly installed. Self-plugging, mechanical lock rivets display all the strength characteristics of solid shank rivets and in almost all cases can be substituted rivet for rivet.

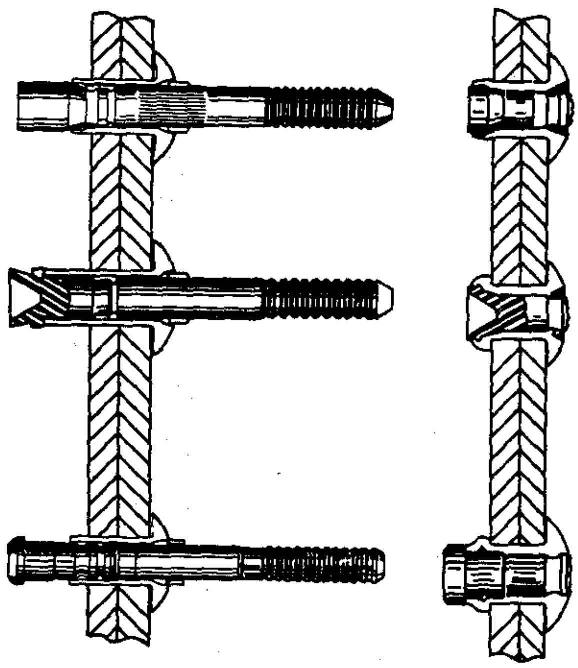


FIGURE 5–59. Self-plugging, mechanical lock rivets.

Huck Rivets

Self-plugging, mechanical lock rivets require special driving assemblies. It is best to use tools manufactured by the company that produces the rivet.

The Huck CKL rivet is installed by using the model CP350 blind rivet tool. The nose of the tool includes: (1) A set of chuck jaws which fit the serrated grooves on the rivet stem and pull it through the rivet shank to drive the rivet; (2) an outer anvil which bears against the outer portion of the manufactured head during the driving operation; and (3) an inner anvil which advances automatically to drive the locking collar into position after the blind head is formed (figure 5–60).

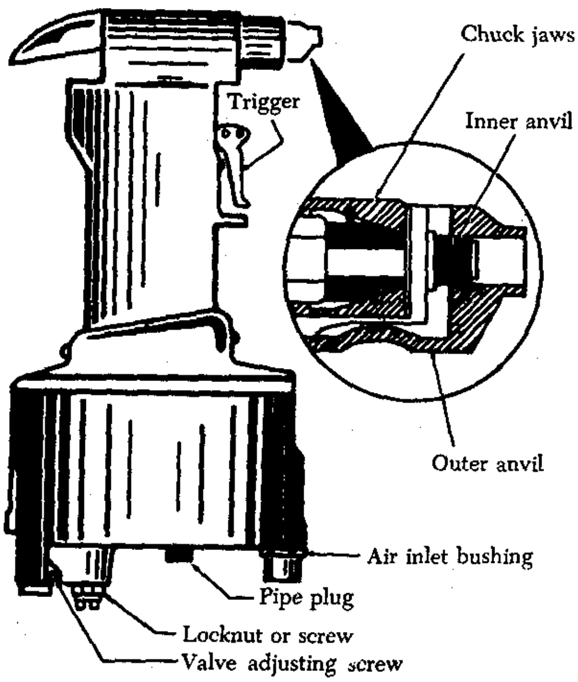


FIGURE 5–60. Huck model CP350 rivet pull tool.

A change in rivet diameter requires a change in chuck jaws, outer anvil, and inner thrust bearing, and an adjustment of the shift operating pressure. Adjustment procedures are specified by the manufacturer.

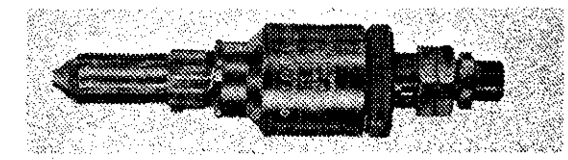
Cherrylock Rivets

Cherrylock rivets are installed with a hydroshift or mechanical shift tooling system. The hydroshift system is a newer design and when available should be used in place of the mechanical system.

Cherrylock Mechanical Tooling

Most existing Cherry riveters, either hand or power operated, may be used to install Cherrylock rivets when equipped with the proper mechanical pulling head.

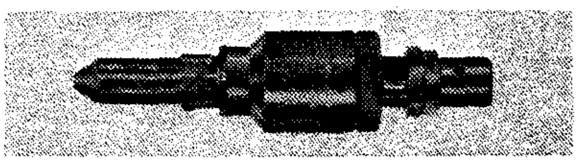
Cherrylock mechanical pulling heads are of two types: the H615 (figure 5–61) and H640 (figure 5–62) series. They differ only in their method of attachment to the riveter. The H615 series is for the smaller screw-on type tools and the H640 is for the larger clip-on type. Both pulling heads will install bulbed and wiredraw Cherrylock rivets.



RIVET DIAMETER	PULLING HEAD NUMBER
1/8″	H615-4C { Universal Head Countersunk Head H615-4S Uni-Sink Head
5/32″	H615-5C { Universal Head Countersunk Head H615-5S Uni-Sink Head
3/16″	H615-6C { Universal Head Countersunk Head H615-6S Uni-Sink Head

Figure 5–61. H615 Series pulling head tool.

A separate pulling head is required to install each diameter Cherrylock rivet. Separate pulling heads are recommended for universal and countersunk head rivets but countersunk pulling heads may be used for both styles.



RIVET DIAMETER	PULLING HEAD NUMBER
1/8″	H640-4C { Universal Head Countersunk Head H640-4S Uni-Sink Head
5/32″	H640-5C { Universal Head Countersunk Head H640-5S Uni-Sink Head
3/16″	H640-6C { Universal Head Countersunk Head H640-6S Uni-Sink Head
1/4″	H640-8C { Universal Head Countersunk Head

Figure 5–62. H640 Series pulling head tool.

RIVET DIAMETER	PULLING HEAD NUMBER	DIMENSIONS A	DIMENSIONS B
3/32″	H681-3C { Universal Head	.188	.346
	Countersunk Head	.163	.331
1/8″	H681-4C { Universal Head	.250	.359
	Countersunk Head	.208	.341
	H681-B166-4 Uni-Sink Head	.250	.359
5/32″	H681-5C { Universal Head	.313	.377
	Countersunk Head	.269	.352
	H681-B166-5 Uni-Sink Head	.313	.377
3/16″	H681-6C { Universal Head	.375	.419
	Countersunk Head	.335	.385
	H681-B166-6 Uni-Sink Head	.375	.419
1/4″	H681-8C { Universal Head	.500	.452
	Countersunk Head	.458	.398

Figure 5–63. H681 Series pulling head tool.

Cherrylock Hydroshift Tooling

The hydroshift tooling system is an advanced design in which the sequence of operations necessary to install the rivet is accomplished hydraulically within the hydroshift tool rather than by means of a mechanical pulling head.

Cherrylock hydroshift pulling heads are of one type only, the H681 (figure 5–63).

A separate H681 pulling head is required to install each diameter Cherrylock rivet. Separate pulling heads are recommended for universal and countersunk head rivets but countersunk pulling heads may be used for both styles.

Hydroshift riveters are factory adjusted to break the rivet stem flush and set the collar properly. Fine adjustments to the shift point setting can be made by the operator. This adjustment determines the flushness of the break of the rivet stem (figures 5–64 and 5–65).

Installation Procedures

Procedures for installing self-plugging (mechanical lock) rivets are basically the same as those used for installing the friction lock type of rivets. Precautions to be observed are:

(1) Be sure the correct grip range is selected.

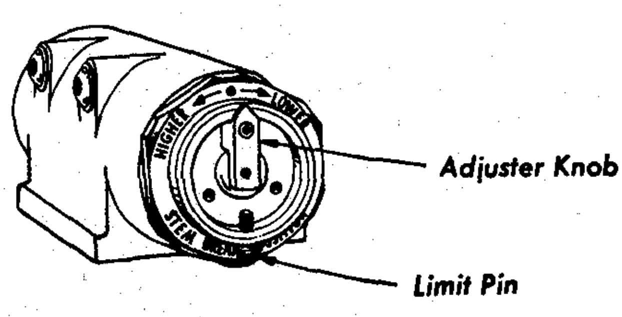


Figure 5–64. H681 pulling head adjuster.

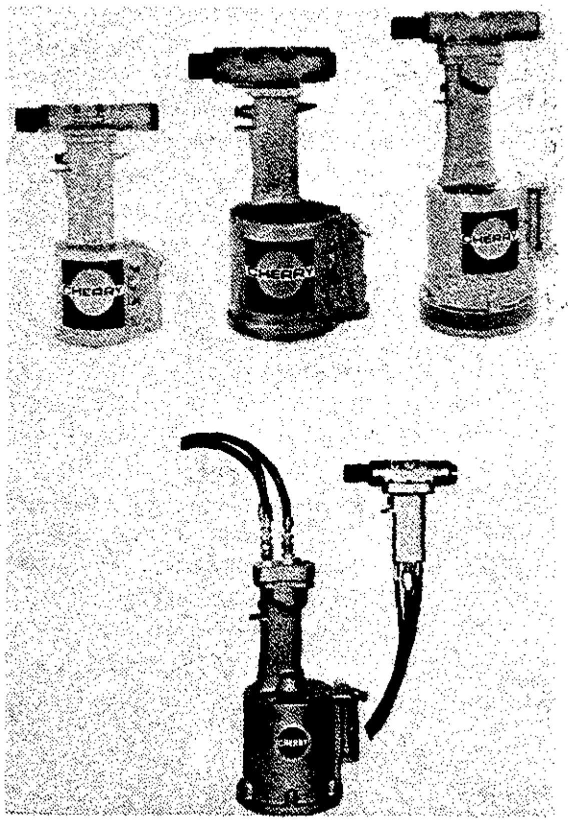


FIGURE 5-65. Hydroshift pulling tool.

(2) Always use the correct nose assembly or pulling tool for the diameter rivet selected. (For the CKL rivet, check the tool air pressure for the correct setting.)

(3) When inserting the rivet in the tool and the material, hold a slight pressure against the head of the rivet.

(4) Determine that the rivet is completely driven before lifting the tool from the rivet head. (The stem should snap.)

(5) Check each rivet after the driving sequence has been completed for proper stem breakage. (The rivet stem should snap off even with the head of the rivet.)

Inspection

Visual inspection of the seating of the pin in the manufactured head is the most reliable and simplest means of inspection for mechanical lock rivets. If the proper grip range has been used and the locking collar and broken end of the stem are approximately flush with the manufactured head, the rivet has been properly upset and the lock formed. Insufficient grip length is indicated by the stem breaking below the surface of the manufactured head. Excessive grip length is indicated by the stem breaking

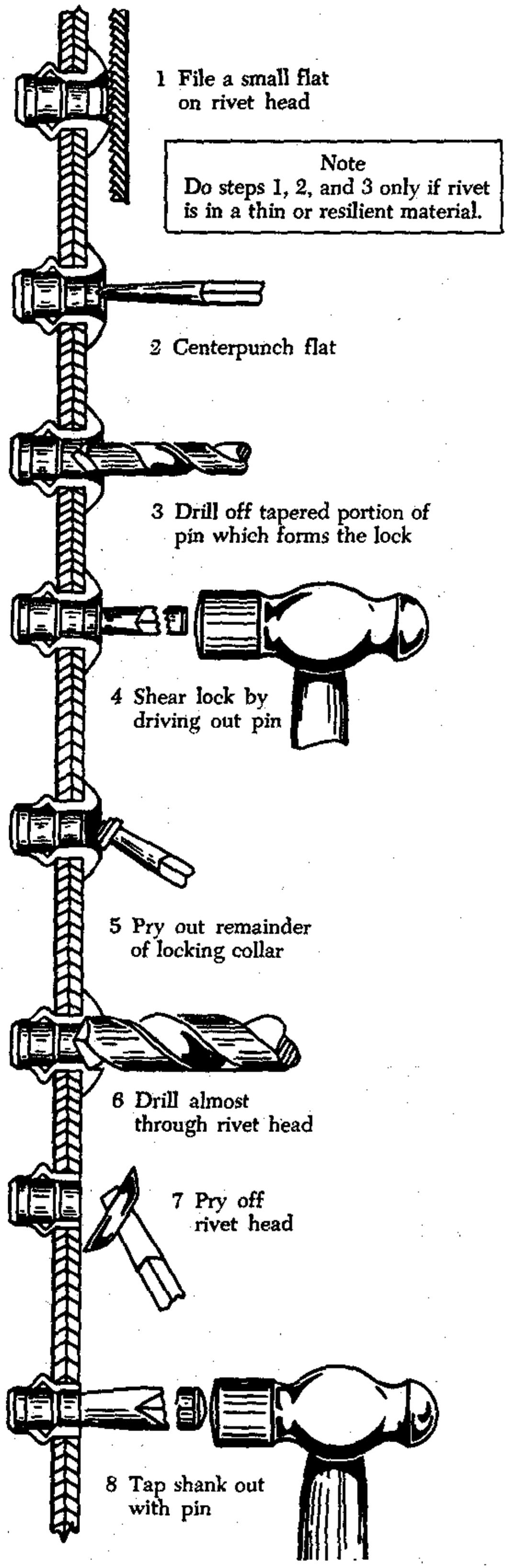


FIGURE 5-66. Rivet removal.

off well above the manufactured head. In either case, the locking collar might not be seated properly, thus forming an unsatisfactory lock.

Removal Procedures

The mechanical lock rivet can easily be removed by following the procedures illustrated in figure 5–66.

PULL-THRU RIVETS

This type of blind mechanically expanded rivet is used as a tacking rivet to attach assemblies to hollow tubes, and as a grommet. It differs from the two previously discussed rivets in that the stem pulls completely through the sleeve of the rivet during installation. Pull-thru rivets are structurally weak because of the hollow center after installation is completed. Methods and procedures for installation, inspection, and removal are not discussed here because of the limited use for this type rivet in the airframe field. Figure 5–67 illustrates a typical pull-thru rivet before and after installation.

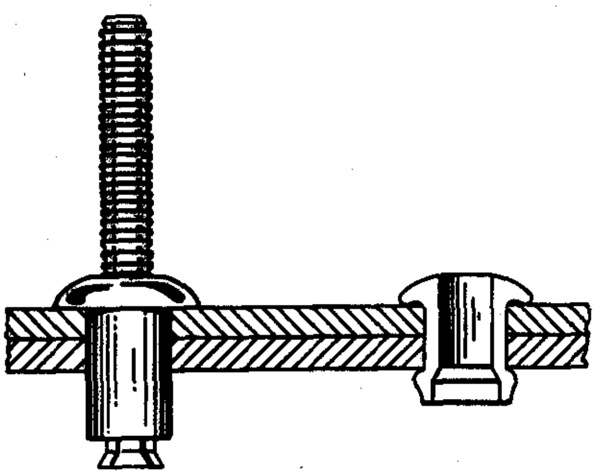


FIGURE 5–67. Pull-thru rivet.

RIVNUTS

Rivnut is the trade name of a hollow blind rivet made of 6053 aluminum alloy, counterbored and threaded on the inside. Rivnuts are installed by one person using a special tool which heads the rivet on the blind side of the material (figure 5–68). The Rivnut is threaded on the mandrel of the heading tool and inserted in the rivet hole. The heading tool is held at right angles to the material; the handle is squeezed, and the mandrel crank is turned clockwise after each stroke. Continue squeezing the handle and turning the mandrel crank of the heading tool until you feel a solid resistance, indicating that the rivet is set.

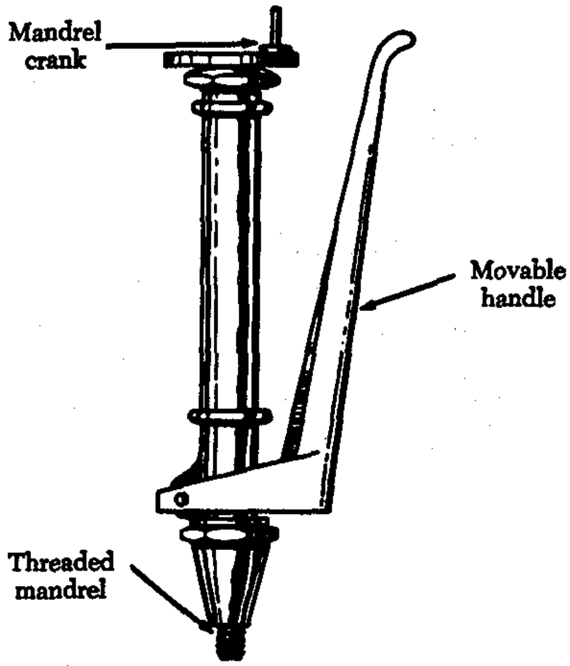


FIGURE 5–68. Rivnut heading tool.

All Rivnuts, except the thin head (0.048 in.) countersunk type, are available with or without small projections (keys) attached to the head to keep the Rivnut from turning. Keyed Rivnuts are used for service as a nut plate, whereas those without keys are used for straight blind riveting repairs where no torque loads are imposed. A keyway cutter is needed when installing Rivnuts which have keys (figure 5–69).

Tools used in the installation of Rivnuts include the hand-operated heading tools, the pneumatic power Rivnut driver, and the keyway cutter. All heading tools have a threaded mandrel onto which the Rivnut is threaded until the head of the Rivnut is against the anvil of the heading tool.

Hand-operated heading tools are made in three types: (1) Straight, (2) 45°, and (3) 90°. The pneumatic power driving tools are made in two types: (1) Lever throttle and (2) offset handle. With the power tool, the threading, upsetting, and withdrawal or unthreading, are accomplished by compressed air through the manipulation of finger-

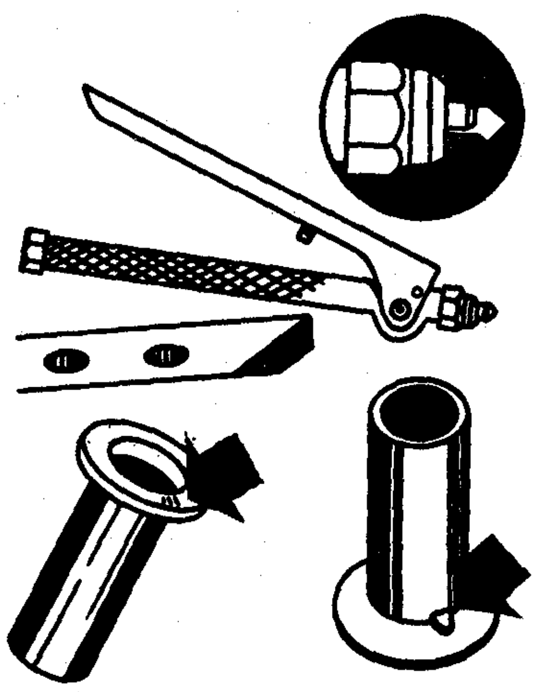


FIGURE 5–69. Keyed Rivnut and keyway cutter.

tip controls. The keyway cutter is for cutting keyways only. In some instances, the keyway cutter cannot be used because the material may be too thick. If such is the case, use a small round file to form the keyway.

The important factors to be considered in selecting Rivnuts are grip range, style of head, condition of Rivnut end, and the presence or absence of a key.

Proper grip length is the most important of these conditions. The grip range of a Rivnut can be determined from its number. For example, a 6–45 has a maximum grip of 0.45 in. Note the procedure to follow when determining the grip range. The total thickness of the sheets shown in figure 5–70 is 0.052 in. By referring to the Rivnut data chart in figure 5–70, we see that 6–75 is the grip length to choose since the maximum grip length of the preceding size (6–45) is only 0.045 in. and would be too short. The grip length of the 6–75 Rivnut actually ranges from 0.045 in., the maximum length of the preceding size (6–45), to 0.075 in., which is the maximum grip length of the 6–75 Rivnut.

The objective when installing this type of rivet is to produce an ideal bulge on the blind side of the work without distorting any of the threads inside the Rivnut. In other words, be sure the bulge takes place between the first thread of the rivet and the lower edge of the riveted material. The space between the ideal bulge and the upper thread, where the gripping takes place, is considered the grip range.

When selecting head style, apply the same rules as for solid shank rivet application. Select key-type Rivnuts whenever screws are to be inserted, and use closed-end Rivnuts only in special places, such as sealed compartments of floats or pressurized compartments.

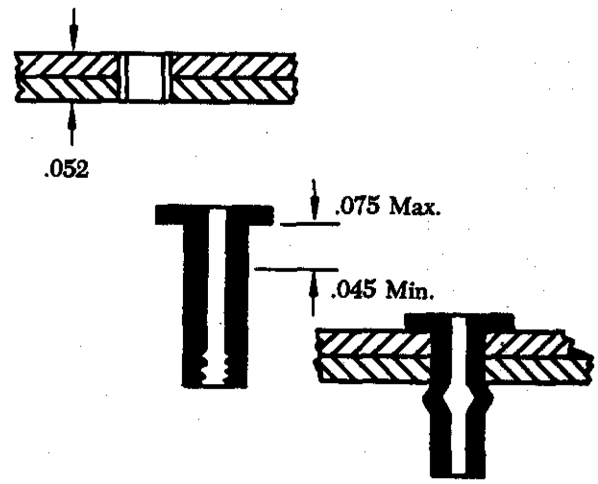
Drilling the holes for Rivnuts requires the same precision as for solid shank rivets. The shank of the Rivnut must fit snugly in the hole. To obtain the best results for a flathead installation, first drill a pilot hole smaller than the shank diameter of the Rivnut and then ream it to the correct size.

If keyed Rivnuts are used, cut the keyway after the hole has been reamed. In cutting the keyway, hold the keyway cutter so that it makes a 90° angle with the work. Also, cut the keyway on the side of the hole away from the edge of the sheet, especially when the Rivnut is used on the outside row. Operate the keyway cutter by inserting it in the hole and squeezing the handles.

The use of flush Rivnuts is limited. For metal which has a thickness greater than the minimum grip length of the first rivet of a series, use the machine countersink; for metal thinner than the minimum grip length of the first rivet, use the dimpling process. Do not use the countersunk Rivnut unless the metal is thick enough for machine countersinking, or unless the underside is accessible for the dimpling operation.

For a countersunk Rivnut the sheets to be joined can usually be machine countersunk. This method is preferred because the bearing surface in a dimpled hole in one sheet of average gage will normally occupy the entire gripping surface of the Rivnut, thus limiting its grip range to that of an anchored nut only.

When installing Rivnuts, among the things to check is the threaded mandrel of the heading tool to see that it is free from burrs and chips from the previous installation. Then screw the Rivnut on the mandrel until the head touches the anvil. Insert the Rivnut in the hole (with the key positioned in the keyway, if a key is used) and hold the heading tool at right angles to the work. Press the head of the



Flat—0.32 Head Thickness		
6-45	6-75	6-100
8-45	8-75	8-100
10-45	10-75	10-100
6B45	6B75	6B100
8B45	8B75	8B100
10B45	10B75	10B100
6K45	6K75	6K100
8K45	8K75	8K100
10K45	10K75	10K100
6KB45	6KB75	6KB100
8KB45	8KB75	8KB100
10KB45	10KB75	10KB100
100°—0.48 Head Thickness		
6-91	6-121	6-146
8-91	8-121	8-146
10-91	10-121	10-146
6B91	6B121	6B146
8B91	8B121	8B146
10B91	10B121	10B146
100°—0.63 Head Thickness		
6-106	6-136	6-161
8-106	8-136	8-161
10-106	10-136	10-161
6B106	6B136	6B161
8B106	8B136	8B161
10B106	10B136	10B161
6K106	6K136	6K161
8K106	8K136	8K161
10K106	10K136	10K161
6KB106	6KB136	6KB161
8KB106	8KB136	8KB161
10KB106	10KB136	10KB161

FIGURE 5–70. Determining Rivnut grip length.

Rivnut tightly against the sheet while slowly squeezing the handles of the heading tool together until the Rivnut starts to head over. Then release the handle, and screw the stud further into the rivnut. This prevents stripping the threads of the Rivnut before it is properly headed. Again squeeze the handles together until the Rivnut heading is complete. Now remove the stud of the heading tool from the Rivnut by turning the crank counterclockwise.

The action of the heading tool draws the Rivnut against the anvil, causing a bulge to form in the counterbored portion of the Rivnut on the inaccessible side of the work. This bulge is comparable to the shop head on an ordinary solid shank rivet. The amount of squeeze required to head the Rivnut properly is best determined by practice. Avoid stripping the thread in the Rivnut.

The installation of a Rivnut is incomplete unless it is plugged either with one of the plugs designed for that purpose or with a screw used for attaching purposes. A Rivnut does not develop its full strength when left hollow.

Three types of screw plugs can be used: (1) The 100° countersunk screw plug, (2) the headless screw plug, and (3) the thin ovalhead screw plug. The 100° countersunk and the headless screw plugs have either a Phillips or a Reed and Prince recess. The oval head either has a common screwdriver slot, a Phillips, or a Reed and Prince recess. All screw plugs are made of high tensile strength SAE steel and are cadmium plated.

The same tools are used for installation of the splined Rivnut as for installation of the standard types, but the pullup stud of the heading tool must be adjusted to accommodate the longer shank.

DILL LOK-SKRUS AND LOK-RIVETS

Dill Lok-Skru and Lok-Rivet are trade names for internally threaded rivets (two piece). They are used for blind attachment of such accessories as fairings, fillets, access door covers, door and window frames, floor panels, and the like. Lok-Skrus and Lok-Rivets are similar to the Rivnut in appearance and application. Lok-Skrus and Lok-Rivets, however, come in three parts and require more clearance on the blind side than the Rivnut to accommodate the barrel.

Special hand- and air-operated power tools are required for installation of Lok-Skrus. An interchangeable barrel blade fits into the blade handle and is held in place by a set screw. The barrel blade has a flattened portion which fits into a slot in the end of the Lok-Skru barrel. The head driver has projections which fit into recesses in the Lok-Skru head. Head drivers and blades are interchangeable for use with various sizes and styles of Lok-Skrus.

The drilling procedure for Lok-Skrus is identical to that for common solid shank rivets. To install the Lok-Skru, insert the Lok-Skru tool so that the blade extends through the barrel slot and the driver sets firmly in the head slot. Insert the fastener in the drilled hole. Fit the ratchet handle assembly together and adjust the pawl lever for proper ratchet direction. Hold the ratchet handle stationary and turn the barrel blade handle to the left until the barrel is drawn firmly against the sheet on the opposite side. Press the tool firmly against the Lok-Skru to hold the tool blade and driver in the slots.

Stop turning the barrel handle when the Lok-Skru barrel has been drawn against the sheet. Finally, tighten by an additional quarter turn or less on the ratchet handle, drawing the head into the sheet. This time, hold the blade handle stationary while turning the ratchet handle. Test the tightness of the installation with an ordinary 8-in. screwdriver which has been ground round on the end. Attachments are made by using the attaching screw and a regular screwdriver.

DEUTSCH RIVETS

The Deutsch rivet is a high-strength blind rivet with a minimum shear strength of 75,000 p.s.i. and can be installed by one man. This rivet consists of two parts, a stainless steel sleeve and a hardened steel drive pin. The pin and sleeve are coated with a lubricant and a corrosion inhibitor.

A Deutsch rivet may be driven with an ordinary hammer or a pneumatic rivet gun and a flathead set. Seat the rivet in the previously drilled hole and then drive the pin into the sleeve. If the Deutsch rivet is driven into a tight hole, a hollow drift punch should be used to seat the rivet against the material. The punch should clear the drive pin and rest against the head of the rivet to prevent premature expansion of the sleeve and head.

The driving action causes the pin to exert pressure against the sleeve and forces the sides of the sleeve out. This stretching forms a shop head on the end of the rivet and provides a positive fastening action for the fastener. The ridge on the top of the rivet head locks the pin into the rivet as the last few blows are struck.

The head of the Deutsch rivet should never be shaved or milled. Milling or shaving will destroy the locking action of the ring on top of the rivet head.

Another feature of the Deutsch rivet is that it can be installed without going all the way through the second piece of material. However, this type of installation is not recommended unless the second piece is very thick.

One of the main restrictions to the use of the Deutsch rivet is that no bucking tool is used to take up the shock of driving. The structure where installation is made must be heavy and solid enough to support the driving forces.

If a Deutsch rivet that extends through the material is to be removed, use the same procedures used to remove a solid shank rivet. The head can be drilled off, and the pin can be driven out with a drift punch slightly smaller than the diameter of the drive pin. To drive the sleeve out of the material, use a drive punch slightly smaller than the diameter of the sleeve.

If the rivet does not extend through the material, drill out the drive pin to approximately one-half its depth. Then tap the hole and finish drilling out the remainder of the pin. Next, insert a screw through a spacer and tighten the screw into the sleeve. Continue tightening the screw until the sleeve is removed.

HI-SHEAR RIVETS

Hi-Shear pin rivets are essentially threadless bolts. The pin is headed at one end and is grooved about the circumference at the other. A metal collar is swaged onto the grooved end, effecting a firm tight fit.

The proper length rivet may be determined by part number or by trial. Part numbers for pin rivets can be interpreted to give the diameter and grip length of the individual rivets. A typical part number and an explanation of the terms are discussed in Chapter 6, Hardware, Materials, and Processes, in the Airframe and Powerplant Mechanics General Handbook, AC 65–9A.

To determine correct grip length by trial, insert the correct diameter rivet in the hole. The straight portion of the shank should not extend more than $\frac{1}{16}$ in. through the material, insert the correct diameter rivet in the hole. The straight portion of the shank should not extend more than $\frac{1}{16}$ in. through the material. Place a collar over the grooved end of the rivet. Check the position of the collar.