SECTION B

Hand Tools

Our ability to make and use tools has been directly responsible for all technical advancement. Prior to the development of advanced metalworking, natural materials such as stone, flint, and wood provided the only tool materials. When metals and metalworking techniques became better established, tool development advanced greatly, which led to the many fine tools of today. A study of tools must logically begin with those used by hand for hand operations. In this section you will be introduced to the basic complement of hand tools used in all branches of mechanical technology.

Work-holding devices were not developed in early times. Artisans in many Middle Eastern and Asian countries still preferred to use their feet instead of a vise to hold the workpiece. Machinists today tend to take the bench vise for granted, seldom realizing that they could hardly get along without it.

Arbor presses and hydraulic shop presses are useful and powerful shop tools. If they are used incorrectly, however, they can be hazardous to the operator, and workpieces can be ruined.

Noncutting tools such as screwdrivers, pliers, and wrenches should be properly identified. It is impossible to request a particular tool from the toolroom without knowing its correct name.

Cutting hand tools such as hacksaws, files, hand reamers, taps, and dies are important to a machinist. In this section you will also be introduced to the pedestal grinder and its important functions in the machine shop.

The units that follow in this section will instruct you in the identification, selection, use, and safety of these important hand tools and hand-operated machines.

CAUTION

Hand Tool Safety Tools described in this section are quite safe if they are used as they were designed to be used. For example, a screwdriver is not meant to be a chisel, and a file is not meant to be a pry bar. Wrenches should be the correct size for the nut or bolt head so they will not slip. Inch measure wrenches should not be used on metric fasteners. When a wrench slips, skinned knuckles are often the result. Hacksaws should be held by the handle, not the frame. Fingers wrapped around the frame tend to get mashed. Files should never be used without a handle because the tang can severely damage the hand or wrist. Safety precautions are noted throughout this section for using other equipment, such as presses and pedestal grinders.

UNIT ONE

Arbor and Shop Presses

The arbor press and the small shop press are common sights in most machine shops. It would be difficult indeed to get along without them. You will find these tools extremely useful when you know how to use them, but if you are not instructed in their use, they can be dangerous to you and destructive to the workpiece.

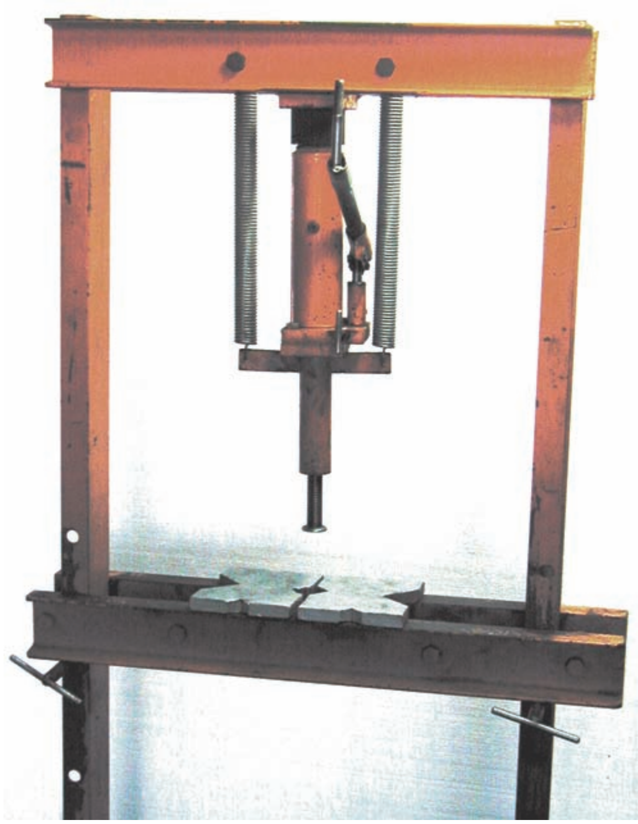

Figure B-1 Small hydraulic shop press.

OBJECTIVES

After completing this unit, you should be able to:

- Install and remove a bronze bushing using an arbor press.
- Press on and remove a ball bearing from a shaft on an arbor press using the correct tools.
- Press on and remove a ball bearing from a housing using an arbor press and correct tooling.
- Install and remove a mandrel using an arbor press.
- Install and remove a shaft with key in a hub using the arbor press.

TYPES OF PRESSES

The arbor press is an essential piece of equipment in the small machine shop. Without it a machinist would be forced to resort to the use of a hammer or sledge to make any forced fit, a process that could easily damage the part.

Two basic types of hand-powered arbor presses are manufactured and used: the hydraulic (Figure B-1) and the mechanical (Figure B-2). The lever gives a "feel" or a sense of pressure applied, which is not possible with the power-driven presses. This pressure sensitivity is needed when small delicate parts are being pressed so that a worker will know when to stop before collapsing the piece.

USES OF PRESSES

The major uses of the arbor press are installing and removing bushings, installing and removing ball and roller bearings (Figure B-3), pressing shafts into hubs (Figure B-4), pressing mandrels into workpieces, straightening and bending, and broaching keyseats. A **keyseat** is an axially located rectangular groove in a shaft or hub. Keyseats in shafts are cut in milling

Figure B-2 Simple ratchet floor-type arbor press (*Dake Corporation*).

Figure B-3 Roller bearing being removed from axle.

Figure B-4 Shaft being pressed into hub.

machines, but keyseats in hubs of gears, sprockets, or other driven members must be either broached or cut in a keyseater, a special type of vertical shaper.

PROCEDURES

Installing Bushings

A **bushing** is a short metal tube, machined inside and out to precision dimensions, and usually made to fit into a bore, or accurately machined hole. Many kinds of bushings are used for various purposes and are usually installed with an **interference fit** or press fit. This means that the bushing is slightly larger than the hole into which it is pressed. The amount of interference will be considered in greater detail in a later unit. There are many bushings made of various materials, including bronze and hardened steel, but they all have one thing in common: they must be lubricated with high-pressure lube before they are pressed into the bore. Oil is not used, as it will simply wipe off and cause the bushing to seize the bore. Seizing is the condition in which two unlubricated metals tend to weld together under pressure. In this case it may cause the bushing to be damaged beyond repair.

The bore should always have a strong chamfer, that is, an angled or beveled edge, since a sharp edge would cut into

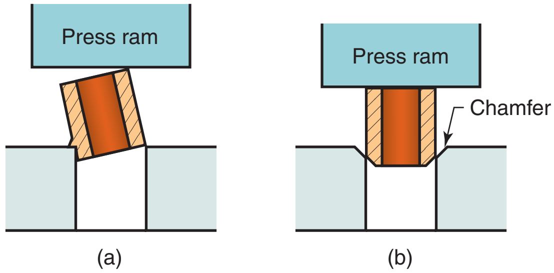

Figure B-5 (*a*) Bushing being pressed where bore is not chamfered and bushing is misaligned; (*b*) bushing being pressed into correctly chamfered hole in correct alignment.

the bushing and damage it (Figure B-5). The bushing should also have a long tapered chamfer or start so it will not dig in and enter misaligned. Bushings are prone to go in crooked, especially hardened steel bushings, if there is a sharp edge. Care should be taken to see that the bushing is straight entering the bore and that it continues into the bore in proper alignment. This should not be a problem if the tooling is right, that is, if the end of the press ram is square and if it is not loose and worn. The proper bolster plate should also be used under the part so that it cannot tilt out of alignment. Sometimes, special tooling is used to guide the bushing (Figure B-6). When the workpiece is resting on a solid bolster plate, only the pressure needed to force the bushing into place should be applied, especially if the bushing is longer than the bore length. Excessive pressure might distort and collapse the bushing and cause it to be undersized (Figure B-7). However, some bolster plates provide various size holes so that a bushing or part can extend through. In that case, the press ram can be brought to the point where it contacts the workpiece, and there is no danger of upsetting the bushing.

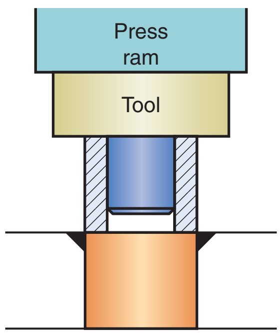

Figure B-6 Special tool to keep bushing square to press ram.

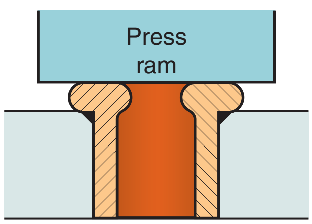

Figure B-7 Effect of excessive pressure on bushing that exceeds bore length.

Ball and Roller Bearings

Ball and roller bearings pose special problems when they are installed and removed by pressing. This is because the pressure must be applied directly against the race and not through the balls or rollers, since this could destroy the bearing. Frequently, when removing ball bearings from a shaft, the inner race is hidden by a shoulder and cannot be supported in the normal way. In this case, a special tool called a bearing puller is used (Figure B-8). On the inner and outer races, bearings may be installed by pressing on the race with a steel tube of the proper diameter. As with bushings, high-pressure lubricant should be used.

Sometimes there is no other way to remove an old ball bearing except by exerting pressure through the balls. When this is done, there is a real danger that the race may be violently shattered. In this case, a scatter shield must be used. A scatter shield is a heavy steel tube about 8 to 12 in. long set up to cover the work. The shield is placed around the bearing during pressing to keep shattered parts from injuring the operator. It is a good safety practice to always use a scatter shield when ball bearings are removed from a shaft by pressing. Safety glasses should be worn during all pressing operations.

Bores and Shafts

Holes in the hubs of gears, sprockets, and other machine parts are also frequently designed for a force fit. In these instances, there is usually a keyseat that needs to be aligned. A keyseat is a groove in which a key is placed. This key, in turn, also fits into a slot in the hub of a gear or pulley and secures the part against the shaft, keeping it from rotating. When pressing shafts with keys into hubs with keyseats, it is sometimes helpful to chamfer the leading edge of the key so that it will align

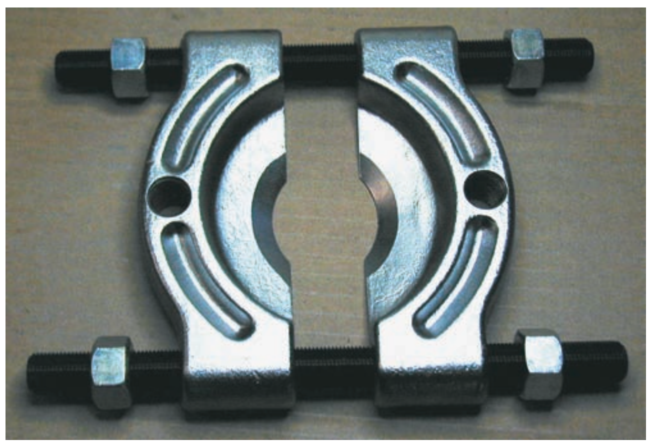

Figure B-8 Bearing puller, a special tool for supporting inner bearing race (see use in Figure B-3).

itself properly (Figure B-9). Seizing will occur in this operation, as with the installation of bushings, if high-pressure lubricant is not used (Figure B-10).

Figure B-9 Chamfer on key helps in alignment of parts being pressed together.

Figure B-10 This shaft had just been made by a machinist and was forced into an interference-fit bore for a press fit. No lubrication was used, and it immediately seized and welded to the bore, which was also ruined.

Mandrels

Mandrels, cylindrical pieces of steel with a slight taper, are pressed into bores in much the same way that shafts are pressed into hubs. There is one important difference, however; since the mandrel is tapered about .006 inch per foot, it can be installed only with the small end in first.

The large end of the mandrel may have a flat where the lathe dog screw can rest. The large end may also be determined by measuring with a micrometer or by trying the mandrel in the bore. The small end should start into the hole, but the large end should not. Apply lubricant and press the mandrel in until definite resistance is felt (Figure B-11).

Keyseat (Keyway) Broaching

The process of broaching is just one of the machining processes. Broaching is the process of cutting out shapes on the interior of a metal part. Broaching can be done on both internal and external surfaces. In keyseat broaching, a slot or groove is cut inside the bore through a hub or pulley so that a key can be retained.

Although many types of keyseating machines are in use in many machine shops, keyseat broaching is often done on

Figure B-11 Mandrel being lubricated and pressed into part for further machining.

Figure B-12 Broaching a keyway (*Asnuntuck Community College*).

arbor presses. Keyseats are only one type of cutting that can be done by the push-type procedure. Such internal shapes as a square or hexagon can also be cut by this method (Figure B-12). All that is needed for these procedures is the proper size arbor press and a set of keyseat broaches (Figure B-13), which are hardened cutters with stepped teeth so that each tooth cuts only a definite amount when pushed or pulled through a part. These are available in inch and metric dimensions.

The procedure for broaching keyseats (multiple-pass method) is as follows:

Step 1 Choose the bushing that fits the bore and the broach, and put it in place in the bore.

Step 2 Insert the correct-size broach into the bushing slot (Figure B-14).

Figure B-13 Keyseat (keyway) broach.

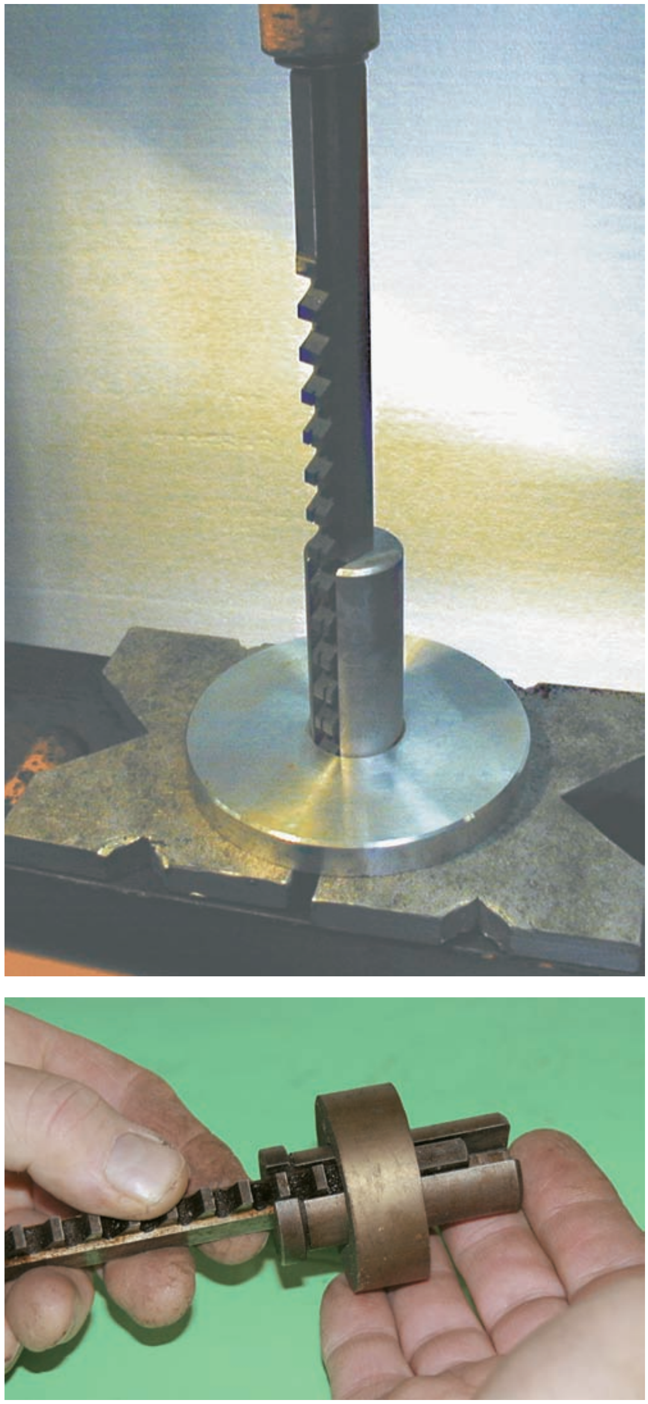

Figure B-14 Broach with guide bushing inserted into a hub.

Step 3 Place this assembly in the arbor press (Figure B-15).

Step 4 Lubricate.

Step 5 Push the broach through.

Step 6 Clean the broach.

Step 7 Place the second-pass shim in place.

Step 8 Insert broach.

Step 9 Lubricate.

Figure B-15 Broach with guide bushing placed in arbor press that is ready to be lubricated and have the first pass performed.

Step 10 Push the broach through.

Step 11 If more than one shim is needed to obtain the correct depth, repeat the procedure (Figure B-16).

Clean the tools and return them to their box, and deburr and clean the finished keyseat.

Production or single-pass broaching requires no shims or second-pass cuts, and with some types no bushings need be used (Figure B-17).

Two important things to remember when push broaching are alignment and lubrication. Misalignment, caused by a worn or loose ram, can cause the broach to hog (dig in) or break. Sometimes this can be avoided by facing the teeth of the broach toward the back of the press and permitting the bushing to protrude above the work to provide more support for the broach. After starting the cut, relieve the pressure to allow the broach to center itself. Repeat this procedure during each cut.

At least two or three teeth should be in contact with the work. If needed, stack two or more workpieces to lengthen the cut. The cut should never exceed the length of the standard bushing used with the broach. Never use a broach on material harder than Rockwell C35. You will study hardness testing later in this book. If it is suspected that a part is harder than mild steel, its hardness should be determined before any broaching is attempted.

Use a good high-pressure lubricant. Also apply a sulfur-base cutting oil to the teeth of the broach. Always lubricate the back of the keyseat broach to reduce friction, regardless of the material to be cut. Brass is usually broached dry, but bronzes cut better with oil or soluble oil. Cast iron is broached dry, and kerosene (solvent) or cutting oil is recommended for aluminum.

Bending and Straightening

A shaft to be straightened is placed between two nonprecision vee blocks—steel blocks with a vee-shaped groove running the length of the blocks that support a round workpiece. In the vee blocks, the shaft is rotated to detect runout, or the

Figure B-16 Shims in place behind broach that is ready to be lubricated and have the final cut made on part.

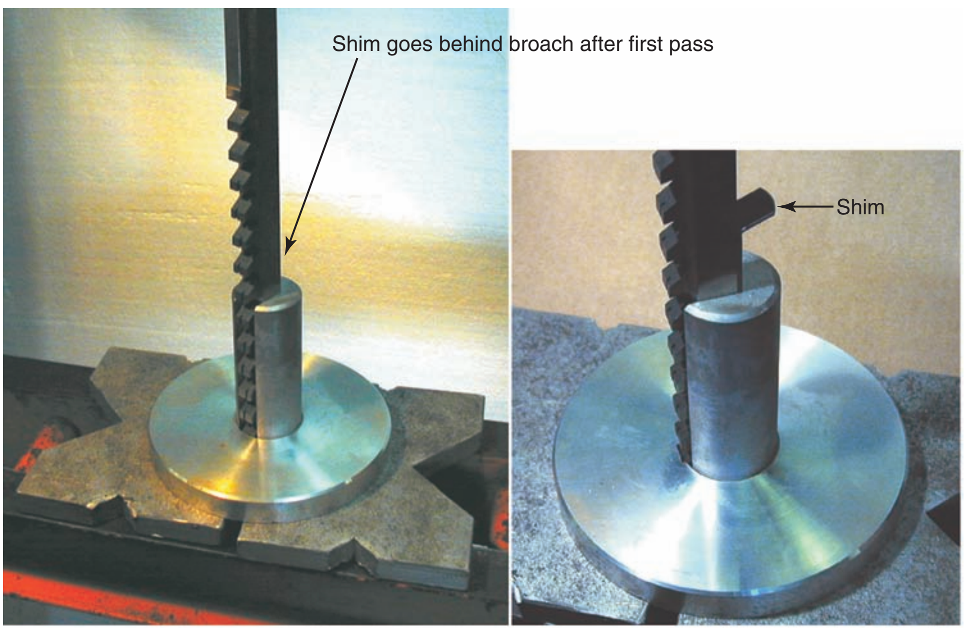

Figure B-17 Production push broaching without bushing or shims (*The duMont Company, LLC*).

CAUTION

Bending and straightening are frequently done on hydraulic shop presses. Mechanical arbor presses are not usually used for this purpose. There is a definite safety hazard in this type of operation, as a poor setup can allow pieces under pressure to fly suddenly out of the press. Brittle materials such as cast iron or hardened steel bearing races can suddenly break under pressure and explode into fragments.

amount of bend in the shaft. The rotation is measured on a dial indicator, a device capable of detecting small mechanical movements, and read from a calibrated dial. The high point is found and marked on the shaft (Figure B-18). After the indicator is removed, a soft metal pad such as copper is placed between the shaft and the ram and pressure is applied (Figure B-19). The shaft should be bent back to a straight position and then slightly beyond that point. The pressure is then removed and the dial indicator is again put in position. The shaft is rotated as before, and the position of the mark noted, as well as the amount of runout. If improvement has been found, continue the process; but if the first mark is opposite the new high point, too much pressure has been applied. Repeat the same steps, applying less pressure on the opposite side.

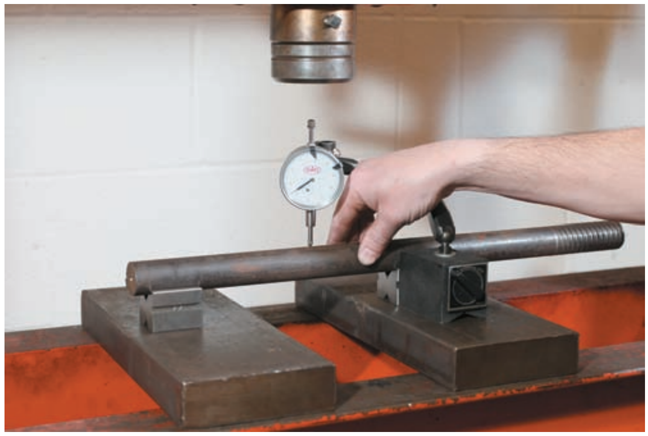

Figure B-18 Part being indicated for runout prior to straightening.

Figure B-19 Pressure being applied to straighten shaft.

Other straightening jobs on flat stock and other shapes are done in a similar fashion. Frequently, two or more bends will be found that may be opposite or not in the same direction. This condition is best corrected by straightening one bend at a time and checking with a straightedge and feeler gage. Special shop press tooling is sometimes used for simple bending jobs in the shop.

SELF-TEST

1. Why is it important to know how to use the arbor press properly and how to set up pressing operations correctly?
2. What kinds of arbor presses are made? What makes them different from large commercial presses?
3. List several uses of the arbor press.
4. A newly machined steel shaft with an interference fit is pressed into the bore of a steel gear. The result is a shaft ruined beyond repair; the bore of the gear is also badly damaged. What has happened? What caused this failure?

5. The ram of an arbor press is loose in its guide and the pushing end is rounded off. What kind of problems could be caused by this?
6. When a bushing is pushed into a bore that is located over a hole in the bolster plate of a press, how much pressure should you apply to install the bushing: 30 tons, 10 tons, or just enough to seat the bushing into the bore?
7. When pressing a shaft from the inner race of a ball bearing, where should the bearing be supported on the bolster plate of the press?
8. What difference is there in the way a press fit is obtained between mandrels and ordinary shafts?
9. Prior to installing a bushing with the arbor press, what two important steps must be taken?
10. Name five ways to avoid tool breakage and other problems when using push broaches for making keyseats in the arbor press.

INTERNET REFERENCES

Information on shop presses

www.dakecorp.com

www.buffalohydraulic.com

UNIT TWO

Work-Holding and Hand Tools

The bench vise is a basic but very necessary tool in the shop. With proper care and use, this work-holding tool will give many years of faithful service. Hand tools are essential in all the mechanical trades. This unit will help you learn the names and uses of most of the noncutting tools used by machinists.

OBJECTIVES

After completing this unit, you should be able to:

- Identify various types of vises, their uses, and their maintenance.
- Identify the proper tool for a given job.
- Determine the correct use of a selected tool.

TYPES OF VISES

Vises of various types are used by machinists when doing hand or bench work. They should be mounted in such a way that a long workpiece can be held in a vertical position extending alongside the bench (Figure B-20). Some bench vises have a solid base, and others have a swivel base (Figure B-21). The machinist's bench vise is measured by the width of the jaws (Figure B-22).

Toolmakers often use small vises that pivot on a ball and socket for holding delicate work. Handheld vises, called *pin vises,* are made for holding very small or delicate parts.

Most bench vises have hardened insert jaws that are serrated for greater gripping power. These crisscross serrations are sharp and will dig into finished workpieces enough to mar them beyond repair. Soft jaws (Figure B-23) made of copper, other soft metals, or wood, are used to protect a finished surface on a workpiece. These soft jaws are made to slip over the vise jaws. Some vises used for sheet metal work have smooth, deep jaws.

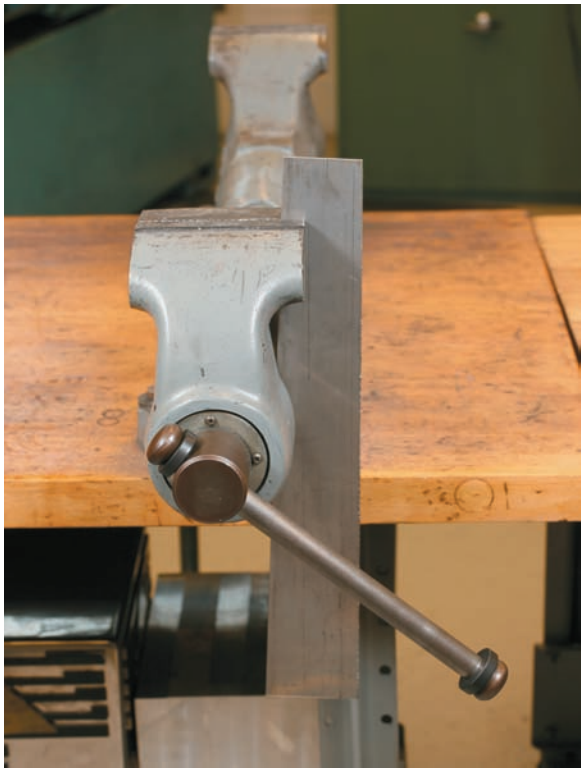

Figure B-20 When long work is clamped in the vise vertically, it should clear the workbench.

USES OF VISES

Vises are used to hold work for filing, hacksawing, chiseling, and bending light metal. They are also used for holding work when assembling and disassembling parts.

Vises should be placed on the workbench at the correct working height for the individual. The top of the vise jaws should be at elbow height. Poor work is produced when the vise is mounted too high or too low. A variety of vise heights should be provided in the shop or skids made available to stand on.

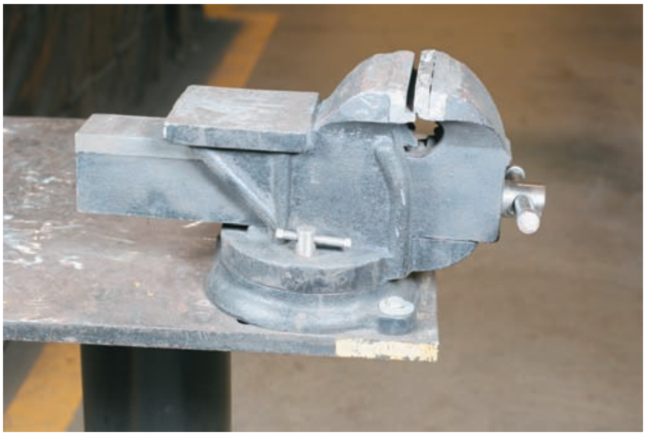

Figure B-21 Swivel-base bench vise.

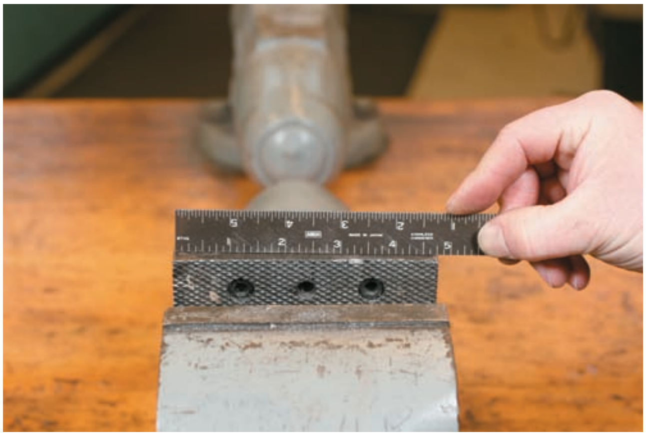

Figure B-22 How to measure a vise.

Figure B-23 View of the soft jaws placed on the vise.

CARE OF VISES

Like any other tool, vises have limitations. “Cheater” bars or pipes should not be used on the handle to tighten the vise. Heat from a torch should not be applied to work held in the jaws, as the hardened insert jaws will then become softened. There is usually one vise in a shop reserved for heating and bending.

Heavy hammering should not be done on a bench vise. The force of bending should be against the fixed jaw rather than the movable jaw of the vise. Bending light flat stock or small round stock in the jaws is permissible if a light hammer is used. The movable jaw slide bar (Figure B-24) should never be hammered on, as it is usually made of thin cast iron and can be cracked quite easily. An anvil is often provided behind the solid jaw for the purpose of light hammering.

Bench vises should occasionally be taken apart so that the screw, nut, and thrust collars can be cleaned and lubricated (Figure B-25). The screw and nut should be cleaned in solvent. A heavy grease should be packed on the screw and thrust collars before reassembly.

Figure B-24 Never hammer on the slide bar of a vise. This may crack or distort it.

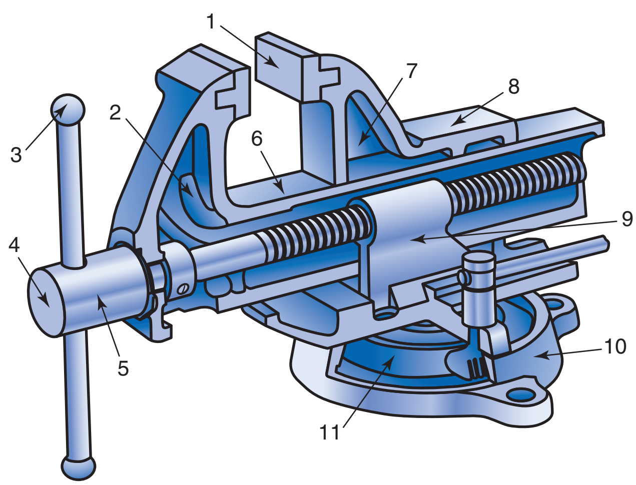

Figure B-25 Cutaway view of a vise: (1) replaceable hardened tool steel faces pinned to jaw; (2) malleable iron front jaw; (3) steel handle with ball ends; (4) cold-rolled steel screw; (5) bronze thrust bearing; (6) front jaw beam; (7) malleable iron back jaw body; (8) anvil; (9) nut, mounted in back jaw keyseat for precise alignment; (10) malleable iron swivel base; (11) steel tapered gear and lock bolt.

CLAMPS

C-clamps are used to hold workpieces on machines such as drill presses, as well as to clamp parts together. The size of the clamp is determined by the largest opening of its jaws. Heavy-duty C-clamps (Figure B-26) are used by machinists to hold heavy parts such as steel plates together for drilling or other machining operations. The clamp shown in the top view (Figure B-27) has a shielded screw. The clamp screw is protected by a sheet metal cover. Thus the screw is protected from dirt and damage. Parallel clamps (Figure B-28) are used to hold small parts. Since they do not have as much holding power as C-clamps, this usually limits the use of parallel clamps to delicate work. Precision measuring setups are usually held in place with parallel clamps.

Figure B-26 Heavy-duty C-clamp.

Figure B-27 Two types of C-clamps (*Courtesy of Wilton Corporation*).

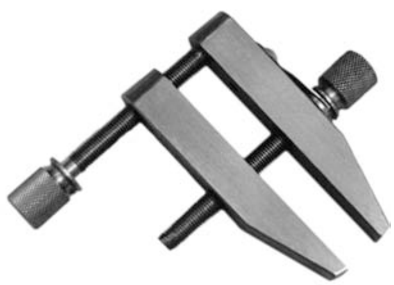

Figure B-28 Single-size parallel clamps.

PLIERS

Pliers come in several shapes and with several types of jaw action. Simple combination or slip joint pliers (Figure B-29) will do most jobs for which you need pliers. The slip joint allows the jaws to expand to grasp a larger size workpiece. They are measured by overall length and are made in 5-, 6-, 8-, and 10-in. sizes.

Interlocking joint pliers (Figure B-30), or water pump pliers, were made to tighten packing gland nuts on water pumps on cars and trucks but are useful for a variety of jobs. Pliers should never be used as a substitute for a wrench, as the nut or bolt head will be permanently deformed by the serrations in the plier jaws, and the wrench will no longer fit properly. Round nose pliers (Figure B-31)

Figure B-29 Slip joint or combination pliers.

Figure B-30 Interlocking joint or water pump pliers.

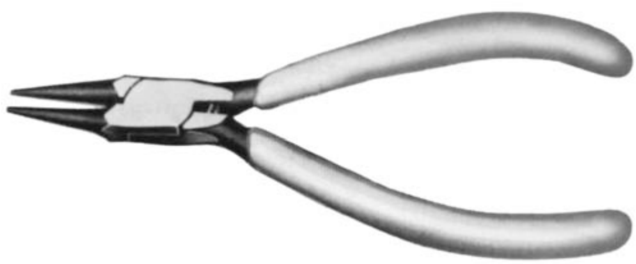

Figure B-31 Round nose or wire looper pliers (*Courtesy Snap-on Tools Corporation*).

are used to make loops in wire and to shape light metal. Needlenose pliers are used for holding small delicate workpieces in tight spots. They are available in both straight (Figure B-32) and bent nose (Figure B-33) types. Linemen's pliers (Figure B-34) can be used for wire cutting and bending. Some types have wire stripping grooves and insulated handles. Diagonal cutters (Figure B-35) are used only for wire cutting.

Figure B-32 Needlenose pliers, straight.

Figure B-33 Needlenose pliers, bent.

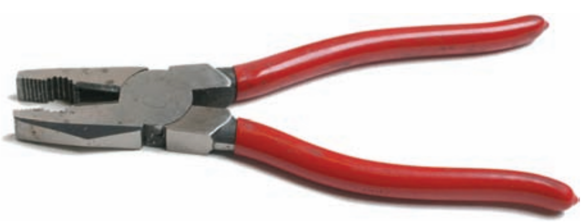

Figure B-34 Side cutting pliers (*Courtesy Snap-on Tools Corporation*).

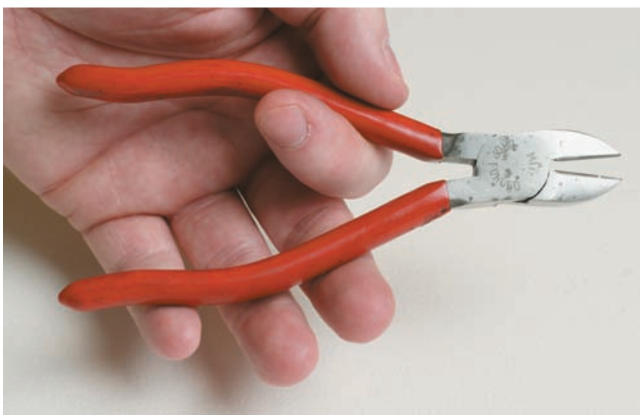

Figure B-35 Diagonal cutters.

Figure B-36 Vise grip wrench.

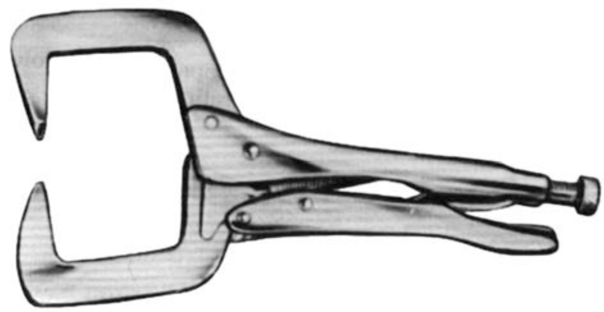

Figure B-37 Vise grip C-clamp (*Courtesy Snap-on Tools Corporation*).

Figure B-38 Maul.

The lever-jawed locking wrench has an unusually high gripping power. The screw in the handle adjusts the lever action to the work size (Figure B-36). They are made with special jaws for various uses such as the C-clamp types used in welding (Figure B-37).

HAMMERS

Hammers are classified as either hard or soft. Hard hammers have steel heads such as blacksmith types or mauls made for heavy hammering (Figure B-38). The ball peen

Figure B-39 Ball peen hammer.

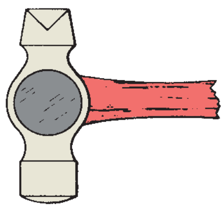

Figure B-40 Straight peen hammer.

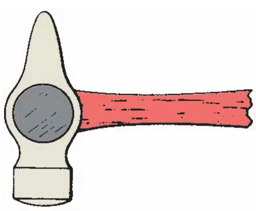

Figure B-41 Cross peen hammer.

Figure B-42 Plastic hammer.

Figure B-43 Lead hammer.

hammer (Figure B-39) is the one most frequently used by machinists. It has a rounded surface on one end of the head, which is used for upsetting or riveting metal, and a hardened striking surface on the other. Two hammers should never be struck together on the face, as pieces could break off. Hammers are specified according to the weight of the head. Ball peen hammers range from 2 oz to 3 lb. Those under 10 oz are used for layout work. Two other shop hammers are the straight peen (Figure B-40) and the cross peen (Figure B-41).

Soft hammers are made of plastic (Figure B-42), brass, copper, lead (Figure B-43), or rawhide and are used to position workpieces that have finishes that would be damaged by a hard hammer. A dead blow hammer is sometimes used in place of a lead hammer because, like the lead hammer, the dead blow hammer does not have a tendency to rebound. When a hammer bounces away from a workpiece, the work will not remain in place but will move slightly. The movable jaw on most machine tool vises tends to move slightly upward when tightened against the workpiece. Thus, the workpiece is moved upward and out of position. The machinist must then use a dead blow hammer or lead hammer to reposition it.

WRENCHES

A large variety of wrenches are made for different uses such as turning capscrews, bolts, and nuts. The adjustable wrench, commonly called a *crescent* wrench (Figure B-44), is a general-purpose tool and will not suit every job, especially those requiring work in close quarters. The wrench should be rotated toward the movable jaw and should fit the nut or bolt tightly. The size of the wrench is determined by its overall length in inches.

Open end wrenches (Figure B-45) are best suited to square-headed bolts, and usually fit two sizes, one on each end. The ends of this type of wrench are angled so they can be used in close quarters. Box wrenches (Figure B-46) are also double ended and offset to clear the user's hand. The box completely surrounds the nut or bolt and usually has 12 points so that the wrench can be reset after rotating only

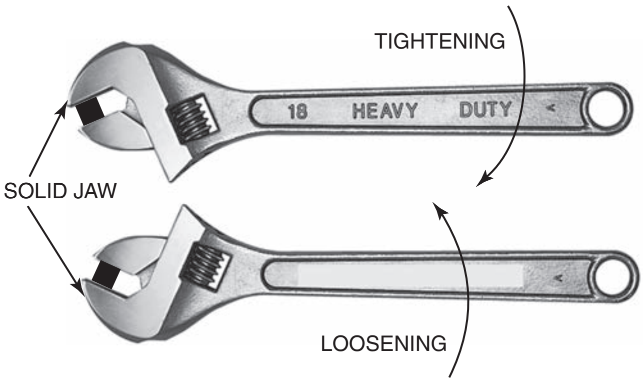

Figure B-44 Adjustable wrench showing the correct direction of pull. The movable jaw should always face the direction of rotation.

Figure B-45 Open end wrench.

Figure B-46 Box wrench.

a partial turn. Mostly used on hex-headed bolts, these wrenches have the advantage of precise fit. Combination and open end wrenches are made with a box at one end and an open end at the other (Figure B-47).

Socket wrenches are similar to box wrenches in that they also surround the bolt or nut and usually are made with 12 points contacting the six-sided nut. Sockets are made to be detached from various types of drive handles (Figure B-48).

Pipe wrenches (Figure B-49), as the name implies, are used for holding and turning pipe. These wrenches have

Figure B-47 Combination wrench.

Figure B-48 Socket wrench set.

sharp serrated teeth and will damage any finished part on which they are used. Strap wrenches (Figure B-50) are used for extremely large parts or to avoid marring the surface of tubular parts.

Spanner wrenches come in several basic types, including face and hook. Face types are sometimes called pin spanners (Figure B-51). Spanners are made in fixed sizes or adjustable types (Figures B-52 to B-54).

Socket head wrenches (Figure B-55) are six-sided bars having a 90-degree bend near one end. They are used with socket head capscrews and socket setscrews.

Torque wrenches (Figure B-56) are widely used by machinists and mechanics to provide the correct amount of tightening torque on a screw or nut. A dial reads in English measure (inch-pounds and foot-pounds) or in metric measure (kilogram-centimeters and newton-meters).

The hand tap wrench (Figure B-57) is used for medium-sized and large taps. The T-handle tap wrench (Figure B-58) is used for small taps $\frac{1}{4}$ in. and under, as its more sensitive "feel" results in less tap breakage.

SCREWDRIVERS

The two types of screwdrivers that are most used are the standard (Figure B-59) and Phillips (Figure B-60). Both types are made in various sizes and in several styles: straight, shank, and offset (Figures B-61 and B-62). It is important to use the right width blade when installing or removing screws (Figure B-63). The shape of the tip is also important. If the tip is badly worn or incorrectly ground, it will tend to jump out of the slot. Never use a screwdriver as a chisel or pry bar. Keep a screwdriver in proper shape by using it only on the screws for which it was meant.

Figure B-49 Pipe wrenches, external and internal.

Figure B-50 Strap wrench.

Figure B-51 Fixed face spanner.

Figure B-52 Adjustable face spanner.

Figure B-53 Hook spanner.

Figure B-54 Adjustable hook spanner.

Figure B-55 Socket head wrench.

Figure B-56 Dial and click–type torque wrench.

Figure B-57 Hand tap wrench.

Figure B-58 T-handle tap wrench.

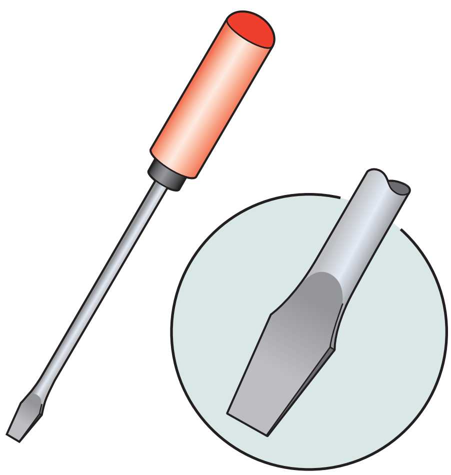

Figure B-59 Screwdriver, standard.

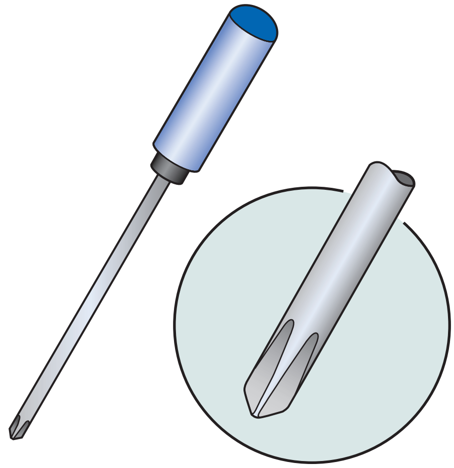

Figure B-60 Screwdriver, Phillips.

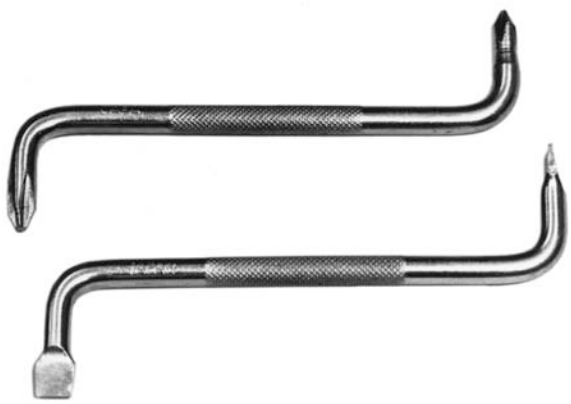

Figure B-61 Standard and Phillips offset screwdrivers.

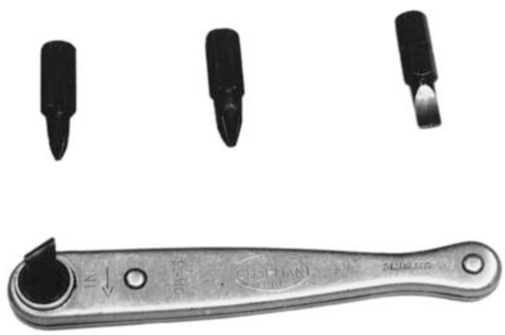

Figure B-62 Ratchet offset screwdriver with interchangeable points.

CHISELS AND PUNCHES

Chisels and punches (Figure B-64) are useful tools for machinists. The tool at the top of the illustration is a pin punch, used to drive out straight, taper, and roll pins. The drift punch below it is used as a starting punch for driving out pins. In the middle is a center punch that makes a starting point for drilling. The two bottom tools are cold chisels. Cold chisels are made in many shapes and are useful for cutting off rivet heads and welds.

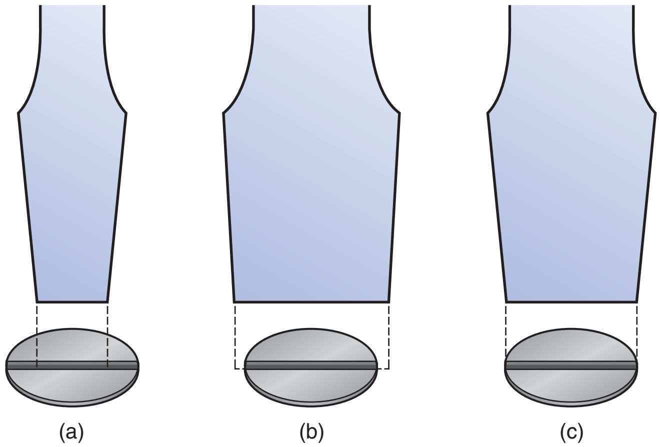

Figure B-63 Width of a screwdriver blade: (*a*) too narrow; (*b*) too wide; (*c*) correct width.

Figure B-64 Common chisels and punches used by machinists. Top to bottom: pin punch, drift punch, center punch, small flat chisel, and large flat chisel.

CAUTION

Here are safety hints for using wrenches:

1. Make sure that the wrench you select fits properly. If it is a loose fit, it may round off the corners of the nut or bolt head.
2. Pull on a wrench instead of pushing to avoid injury.
3. Never use a wrench on moving machinery.
4. Do not hammer on a wrench or extend the handle for additional leverage. Use a larger wrench.

PROFESSIONAL PRACTICE

The way a worker maintains his or her hand tools reveals the kind of machinist he or she is. Dirty, greasy, or misused tools carelessly thrown into a drawer are difficult to find or use the next time around. After a hand tool is used, it should be wiped clean with a shop towel and stored neatly in the proper place. If the tool was drawn from a tool room, the attendant may not accept a dirty tool.

SELF-TEST

1. What clamping position should be considered when mounting a vise on a workbench?
2. Name two types of bench vises.
3. How is the machinist's bench vise measured for size?
4. Explain two characteristics of the insert jaws on vises.
5. How can a finished surface be protected in a vise?
6. Name three things that should never be done to a vise.
7. How should a vise be lubricated?
8. A 4-in. machinist bench vise has jaws 4 in. wide. True or false?
9. What is the purpose of soft jaws?
10. Parallel clamps are used for heavy-duty clamping work, and C-clamps are used for holding precision setups. True or false?
11. To remove a nut or bolt, slip joint or water pump pliers make a good substitute for a wrench when a wrench is not handy. True or false?
12. What advantage does the lever-jawed wrench offer over other similar tools such as pliers?
13. Would you use a 3-lb ball peen hammer for layout work? If not, what size do you think is right?
14. Some objects should never be struck with a hard hammer—a finished machine surface or the end of a shaft, for instance. What could you use to avoid damage?
15. A machine has a capscrew that needs to be tightened and released quite often. Which wrench would be best to use in this case: an adjustable or box wrench? Why?
16. Why should pipe wrenches never be used on bolts, nuts, or shafts?
17. What are two important things to remember about standard screwdrivers that will help you avoid problems in their use?

INTERNET REFERENCES

http://wiltontool.com
http://armstrongtool.com

UNIT THREE

Hacksaws

The hacksaw is one of the more frequently used hand tools. The hand hacksaw is a relatively simple tool to use, but the facts and rules presented in this unit will help you improve your use of the hacksaw.

OBJECTIVE

After completing this unit, you should be able to:

- Identify, select, and use hand hacksaws.

HACKSAW DESIGN

The hacksaw consists of three parts: the frame, the handle, and the saw blade (Figure B-65). Frames are either the solid or adjustable type. The solid frame can be used with only one length of saw blade. The adjustable frame can be used with hacksaw blades from 8 to 12 in. in length. The blade can be mounted to cut in line with the frame or at a right angle to the frame (Figures B-66 and B-67). By turning the blade at right angles to the frame, you can continue a cut that is deeper than the capacity of the frame. If the blade is left in line with the frame, the frame will eventually hit the workpiece and limit the depth of cut.

Most hacksaw blades are made from high-speed steel and in standard lengths of 8, 10, and 12 in. Blade length is the distance between the centers of the holes at each end.

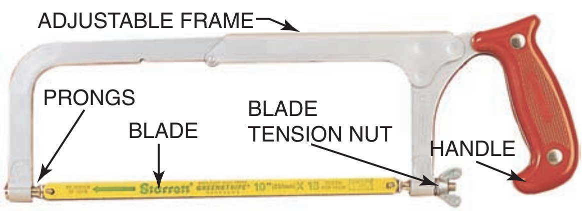

Figure B-65 Parts of a hacksaw.

Figure B-66 Straight sawing with a hacksaw.

Hand hacksaw blades are generally $\frac{1}{2}$ in. wide and .025 in. thick. The *kerf,* or cut, produced by the hacksaw is wider than the .025-in. thickness of the blade because of the set of the teeth (Figure B-68).

The *set* refers to the bending of teeth outward from the blade itself. Two kinds of sets are found on hand hacksaw blades. The first is the straight or alternate set (Figure B-69), in which one tooth is bent to the right and the next tooth to the left for the length of the blade. The second kind of set is the wavy set, in which a number of teeth are gradually bent to the right and then to the left (Figure B-70). A wavy set is found on most fine-tooth hacksaw blades.

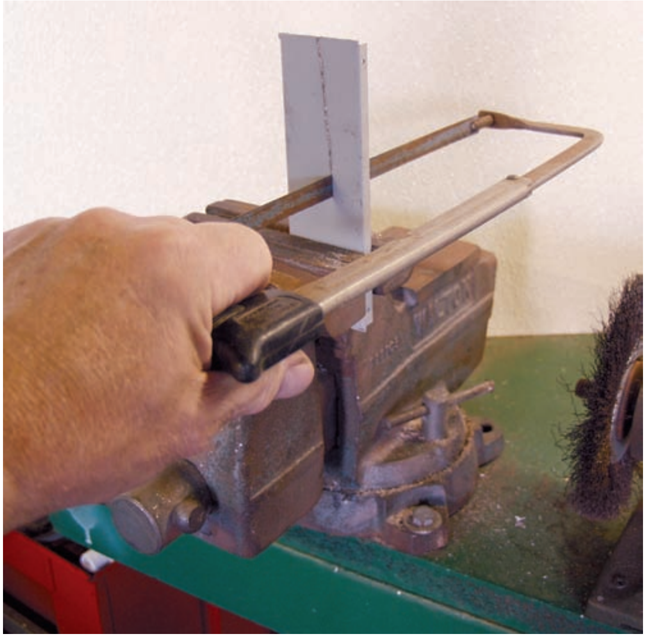

Figure B-67 Sawing with the blade set at 90 degrees to the frame.

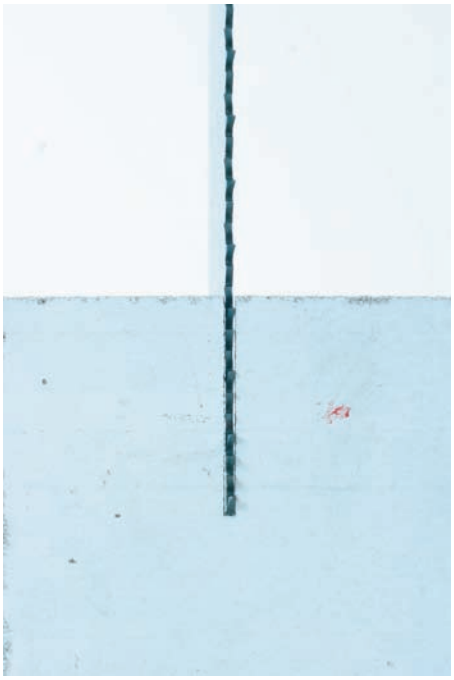

Figure B-68 The kerf is wider than the blade because of the set of the teeth.

The spacing of the teeth on a hand hacksaw blade is called the *pitch* and is expressed in teeth per inch of length (Figure B-71). Standard pitches are 14, 18, 24, and 32 teeth per inch, with the 18-pitch blade used as a general-purpose blade.

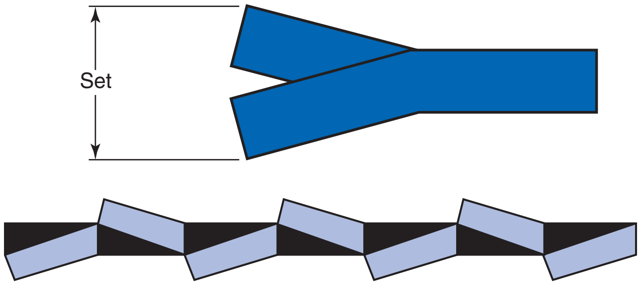

Figure B-69 Straight (alternate) set.

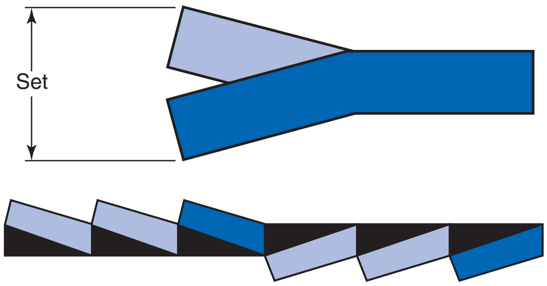

Figure B-70 Wavy set.

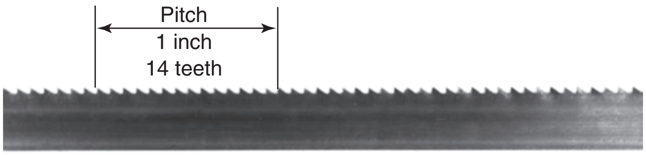

Figure B-71 The pitch of the blade is expressed as the number of teeth per inch.

HANDSAW USE

The hardness and thickness of a workpiece determine to a great extent which pitch blade to use. As a rule, you should use a coarse-tooth blade on soft materials, to have sufficient clearance for the chips, and a fine-tooth blade on harder materials. But you should also have at least three teeth cutting at any time, which may require a fine-tooth blade on soft materials with thin cross sections.

Hand hacksaw blades fall into two categories: soft-backed or flexible blades and all-hard blades. On the flexible blades only the teeth are hardened, the back being tough and flexible. The flexible blade is less likely to break when used in places that are difficult to get at, such as in cutting off bolts on machinery. The all-hard blade is, as the name implies, hard and very brittle, and should be used only where the workpiece can be rigidly supported, as in a vise. On an all-hard blade even a slight twisting motion may break the blade. All-hard blades, in the hands of a skilled person, will cut true, straight lines and give long service.

The blades are mounted in the frame with the teeth pointing away from the handle so that the hacksaw cuts only on the forward stroke. No cutting pressure should be applied to the blade on the return stroke as this tends to dull the teeth. The sawing speed with the hacksaw should be from 40 to 60 strokes per minute. To get the maximum performance from a blade, make long, slow, steady strokes using the full length of the blade. Sufficient pressure should be maintained on the forward stroke to keep the teeth cutting. Teeth on a saw blade will dull rapidly if too little or too much pressure is put on the saw. The teeth will dull also if too fast a cutting stroke is used; a speed in excess of 60 strokes a minute will dull the blade because friction will overheat the teeth.

The saw blade may break if it is too loose in the frame or if the workpiece slips in the vise while sawing. Too much pressure may also cause the blade to break. A badly worn blade, one which the set has been worn down, will cut too narrow a kerf, which will cause binding and perhaps breakage of the blade. When this happens and a new blade is used to finish the cut, turn the workpiece over and start with the new blade from the opposite side and make a cut to meet the first one (Figure B-72). The set on the new blade is wider than the old kerf. Forcing the new blade into an old cut will immediately ruin it by wearing the set down.

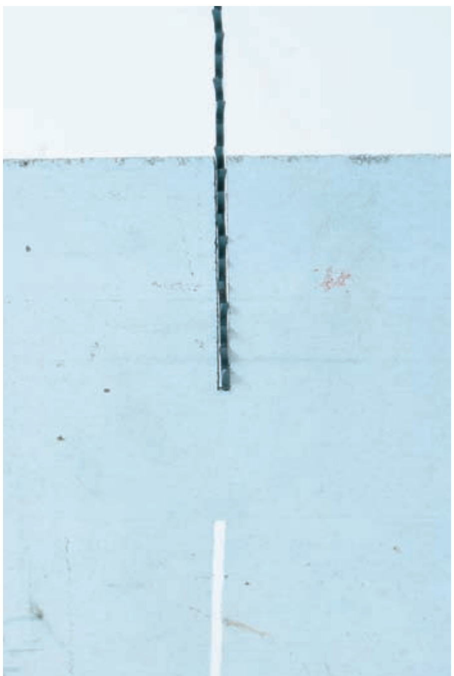

Figure B-72 A new blade must be started on the opposite side of the work, not in the same kerf made by the old blade.

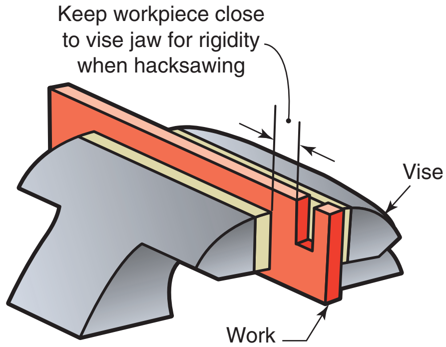

Figure B-73 The workpiece is being sawed close to the vise to present vibration and chatter.

A cut on a workpiece should be started with only light cutting pressure, with the thumb or fingers on one hand acting as a guide for the blade. Sometimes it helps to start a blade in a small vee-notch filed into the workpiece. When a workpiece is supported in a vise, make sure that the cutting is done close to the vise jaws for a rigid setup free of chatter (Figure B-73). Work should be positioned in a vise so that the saw cut is vertical. This makes it easier for the saw to follow a straight line. At the end of a saw cut, just before the pieces are completely parted, reduce the cutting pressure or you may be caught off balance when the pieces come apart and cut your hands on the sharp edges of the workpiece. To saw thin material, sandwich it between two pieces of wood for a straight cut. Avoid bending the saw blades, because they are likely to break, and when they do, they usually shatter in all directions and could injure you or others nearby.

SELF-TEST

1. What is the kerf?
2. What is the set on a saw blade?
3. What is the pitch of the hacksaw blade?
4. What determines the selection of a saw blade for a job?
5. Hand hacksaw blades fall into two basic categories. What are they?
6. What speed should be used in hand hacksawing?
7. Give four causes that make saw blades dull.
8. Give two reasons why hacksaw blades break.
9. A new hacksaw blade should not be used in a cut started with a blade that has been used. Why?
10. What dangers exist when a hacksaw blade breaks while it is being used?

Files

Files are often used to put the finishing touches on a machined workpiece, either to remove burrs or sharp edges or as a final fitting operation. Intricate parts or shapes are often produced entirely by skilled workers using files. In this unit you are introduced to the types and uses of files in metalworking.

OBJECTIVE

After completing this unit, you should be able to:

- Identify eight common files and some of their uses.

TYPES OF FILES

Files are tools that anyone in metalwork will use. Often, through lack of knowledge, these tools are misused. Files are made in many different lengths ranging from 4 to 18 in. (Figure B-74). Files are manufactured in many different shapes and are used for many specific purposes. Figure B-75 shows the parts of a file. When a file is measured, the length is taken from the heel to the point, with the tang excluded. Most files are made from high-carbon steel and are heat-treated to the correct hardness range. They are manufactured in four different cuts: single, double, curved tooth, and rasp. The single cut, double cut, and curved tooth are commonly encountered in machine shops. Rasps are generally used with wood. Curved tooth files will give excellent results with soft materials such as aluminum, brass, plastic, or lead.

Files also vary in their coarseness: rough, coarse, bastard, second cut, smooth, and dead smooth. The files most often used are the bastard, second cut, and smooth grades. Different sizes of files within the same coarseness designation will have varying sizes of teeth (Figure B-76): the longer the file, the coarser the teeth. For maximum metal removal a double-cut file is used. If the emphasis is on a smooth finish, a single-cut file is recommended.

The face of most files is slightly convex because they are made thicker in the middle than on the ends. Because of this curvature only some of the teeth are cutting at any one time, which makes them penetrate better. If the face were flat, it would be difficult to obtain an even surface because of the tendency to rock a file while filing. Some of this curvature is

Figure B-74 Files are made in several different lengths.

Figure B-75 Parts of a file.

Figure B-76 These two files are both bastard cut, but since they are of different lengths, they have different coarsenesses.

also offset by the pressure applied to make the file cut. New files do not cut as well as slightly used ones, since on new files some teeth are longer than most of the others and leave scratches on a workpiece.

Files are either blunt or tapered (Figure B-77). A blunt file has the same cross-sectional area from heel to point, whereas a tapered file narrows toward the point. Files fall into five basic categories: mill and saw files, machinists' files, Swiss pattern files, curved tooth files, and rasps. Machinists', mill, and saw files are classified as American pattern files. Mill files (Figure B-78) were originally designed to sharpen large saws in lumber mills, but now they are used for draw filing, filing on a lathe (Figure B-79), or filing a finish on a workpiece. Mill files are single cut and work well on brass and bronze. Mill files are slightly thinner than an equal-sized flat file, a machinist's file (Figure B-80) that is usually double cut. Double-cut files are used when fast cutting is needed. The finish produced is relatively rough.

Figure B-77 Blunt and tapered file shapes.

Figure B-78 Mill file.

Figure B-79 The lathe file has a longer angle on the teeth to clear the chips when filing on the lathe.

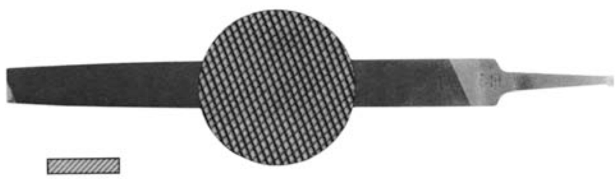

Figure B-80 The flat file is usually a double-cut file.

Pillar files (Figure B-81) have a narrower but thicker cross section than flat files. Pillar files are parallel in width and taper slightly in thickness. They also have one or two safe edges that allow filing into a corner without damaging the shoulder. Square files (Figure B-82) usually are double cut and are used to file in keyseats, slots, or holes.

If a thin file is needed with a rectangular cross section, a warding file (Figure B-83) is used. This file is often used by locksmiths when filing notches into locks and keys. Another file that will fit into narrow slots is a knife file (Figure B-84). The included angle between the two faces of this file is approximately 10 degrees.

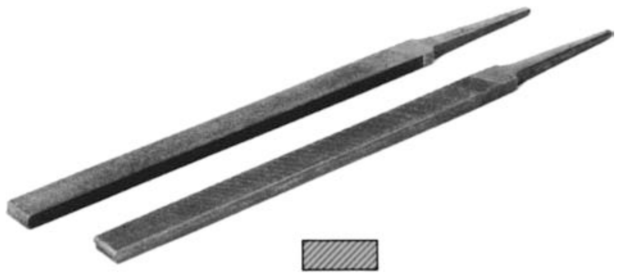

Figure B-81 Two pillar files.

Figure B-82 Square file.

Figure B-83 Warding file.

Figure B-84 Knife file.

Three-square files (Figure B-85), also called three-cornered files, are triangular in shape with the faces at 60-degree angles to each other. These files are used for filing internal angles between 60 and 90 degrees as well as to make sharp corners in square holes. Half-round files (Figure B-86) are available to file large internal curves. Half-round files, because of their tapered construction, can be used to file many different radii. Round files (Figure B-87) are used to file small radii or to enlarge holes. These files are available in many diameter sizes.

Swiss pattern files (Figure B-88) are manufactured to much closer tolerances than American pattern files but are made in the same shapes. Swiss pattern files are more slender, as they taper to finer points and their teeth extend to the extreme edges. Swiss pattern files range in length from 3 to 10 in., and their coarseness is indicated by numbers from 00 (coarse) to 6 (fine). Swiss pattern files are made with tangs to be used with file handles or as needle files with round or square handles that are part of the files. Another type of Swiss pattern file is the die sinkers' riffler (Figure B-89). These files are double-ended with cutting surfaces on either end. Swiss pattern files are used primarily by tool and die makers, mold makers, and other workers engaged in precision filing on delicate instruments.

Curved tooth files (Figure B-90) cut very freely and remove material rapidly. The teeth on curved tooth files are all of equal height, and the gullets or valleys between teeth are deep and provide sufficient room for the filings to curl and drop free. Curved tooth files are manufactured in three grades of cut—standard, fine, and smooth—and in lengths from 8 to 14 in. These files are made as rigid tang types for use with a file handle, or as rigid or flexible blade types used with special handles. Curved tooth file shapes are flat, half-round, pillar, and square.

The bastard cut file (Figure B-91) has a safe edge that is smooth. Flat filing may be done up to the shoulders of the workpiece without fear of damage. Files of other cuts and coarseness are also available with safe edges on one or both sides.

Thread files (Figure B-92) are used to clean up and reshape damaged threads. They are square in cross section and have eight different thread pitches on each file. The thread file of the correct pitch is most effectively used when held or stroked against the thread while it is rotating in a lathe. A thread can be repaired, however, even when it cannot be turned in a lathe.

Figure B-85 Three-square files are used for filing angles between 60 degrees and 90 degrees.

Figure B-86 Half-round files are used for internal curves.

Figure B-87 Round files are used to file a small radius or to enlarge a hole.

Figure B-88 Set of Swiss pattern files. Since these small files are very delicate and can be broken quite easily, great care must be exercised in their use.

Figure B-89 Die sinker's rifflers.

Figure B-90 Curved tooth files are used on soft metals.

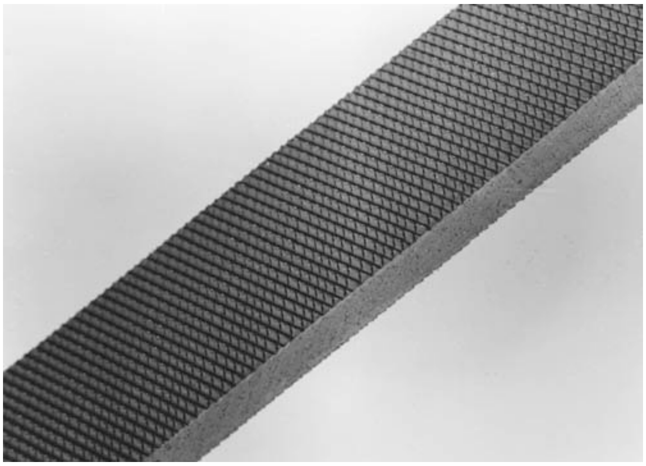

Figure B-91 A file with a safe edge will not cut into shoulders or corners when filing is being done.

Figure B-92 Thread files.

CARE AND USE OF FILES

Files do an efficient job of cutting only while they are sharp. Files and their teeth are hard and brittle. Do not use a file as a hammer or as a pry bar. When a file breaks, particles will fly quite a distance at high speed and may cause an injury. Files should be stored so that they are not in contact with any other file. The same applies to files on a workbench. Do not let files lie on top of one another because one file will break teeth on the other (Figure B-93). Teeth on files will also break if too much pressure is put on them while filing. On the other hand, if not enough pressure is applied while filing, the file only rubs the workpiece and dulls the teeth. A dull file can be identified by its shiny, smooth teeth and by the way it slides over the work without cutting. Dulling of teeth is also caused by filing hard materials or by filing too fast. A good filing speed is 40 to 50 strokes per minute, but remember that the harder the material, the slower the strokes should be; the softer the material, the coarser the file should be.

Too much pressure on a new file may cause **pinning,** that is, filings wedged in the teeth; the result is deep scratches on the work surface. If the pins cannot be removed with a file card (Figure B-94), try a piece of brass, copper, or mild steel, and push it through the teeth. Do not use a scriber or other hard object for this operation. A file will not pin as much if some blackboard chalk is applied to the face (Figure B-95). *Never* use a file without

Figure B-93 Files should be kept neatly arranged so that they will not strike one another and damage the cutting edges.

Figure B-94 Using a file card to clean a file.

Figure B-95 Using chalk on the file to help reduce pinning.

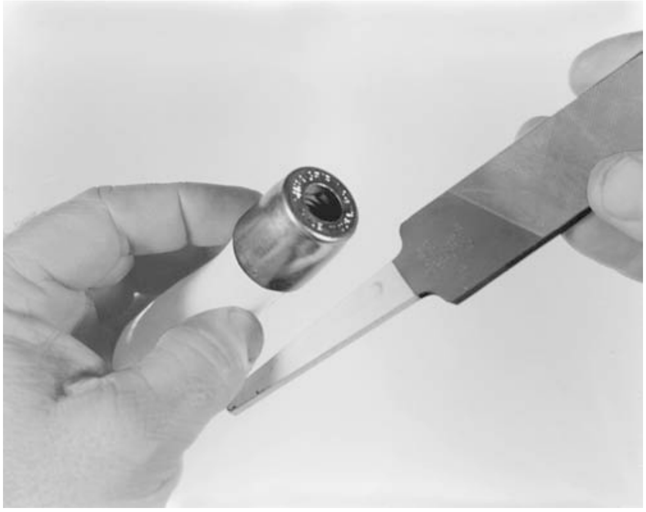

Figure B-96 A file should never be used without a file handle. This style of handle is designed to screw on rather than be driven on the tang.

a file handle, or the pointed tang may cause serious hand or wrist injury (Figure B-96).

Many filing operations are performed with the workpiece held in a vise. Clamp the workpiece securely, but remember to protect it from the serrated vise jaws with a soft piece of material such as copper, brass, wood, or paper. The workpiece should extend out of the vise so that the file clears the vise jaws by $\frac{1}{8}$ to $\frac{1}{4}$ in. Since a file cuts only on the forward stroke, no pressure should be applied on the return stroke. Letting the file drag over the workpiece on the return stroke helps release the small chips so that they can fall from the file. However, this can also dull the file and scratch the part, so do it cautiously.

Use a stroke as long as possible; this will make the file wear out evenly instead of just in the middle. To file a flat surface, change the direction of the strokes frequently to produce a crosshatch pattern (Figure B-97). By using a straightedge steel rule to test for flatness, you can easily determine where the high spots are that have to be filed away. It is best to make flatness checks often, because if any part is filed below a given layout line, the rest of the workpiece may have to be brought down just as far.

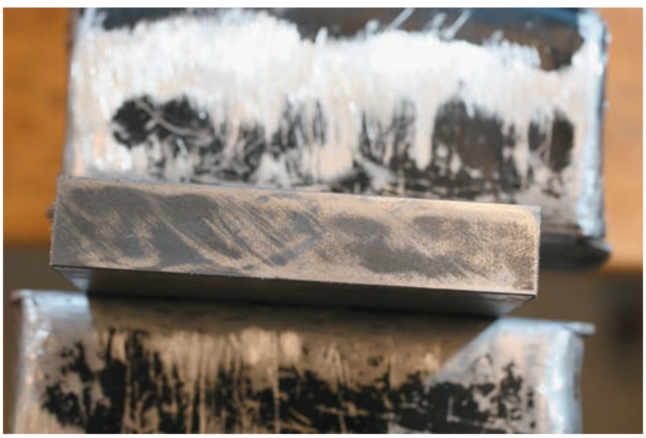

Figure B-97 The crosshatch pattern shows that this piece has been filed from two directions, resulting in a flatter surface.

Figure B-98 shows how a file should be held to file a flat surface. A smooth finish is usually obtained by draw filing (Figure B-99), whereby a single-cut file is held with both hands and drawn back and forth on a workpiece. The file should not be pushed over the ends of the workpiece, as this would leave rounded edges. To get a smooth finish, it sometimes helps to hold the file as shown in Figure B-100, making only short strokes. The pressure is applied by a few fingers and does not extend over the ends of the workpiece. When a round file or half-round file is used, the forward stroke should also include a clockwise rotation for deeper cuts and a smoother finish. People who are filing tend to run their hands or fingers over a newly filed surface. This deposits a thin coat of skin oil on the surface. When filing is resumed, the file will not cut for several strokes but will only slip over the surface, causing the file to dull more quickly.

Figure B-98 Proper filing position.

Figure B-99 Draw filing.

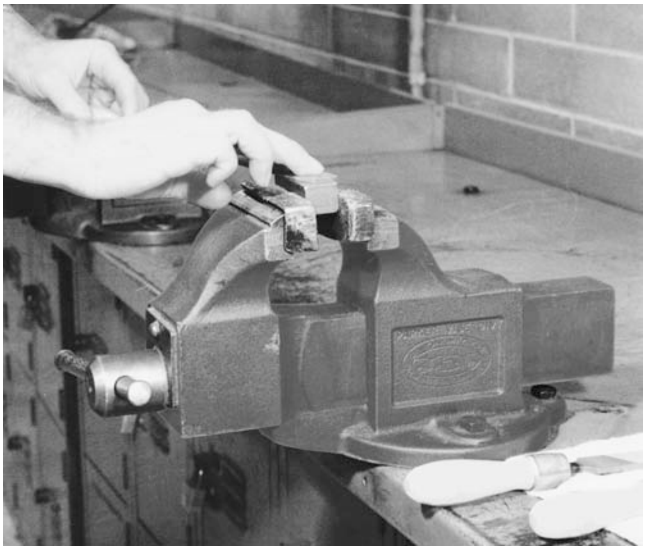

Figure B-100 Use this procedure to correct high spots on curvatures on the workpiece. Apply pressure with short strokes only where cutting is needed.

SELF-TEST

1. How is a file identified?
2. What are the four different cuts found on files?
3. Name four coarseness designations for files.
4. Which of the two kinds of files—single cut or double cut—is designed to remove more material?
5. Why are the faces of most files slightly convex?
6. What difference is there between a blunt and a tapered file?
7. What difference exists between a mill file and an equal-sized flat file?
8. What is a warding file?
9. An American pattern file differs in what way from a Swiss pattern file?
10. What are the coarseness designations for needle files?
11. Why should files be stored so they do not touch each other?
12. What happens if too much pressure is applied when filing?
13. What causes a file to get dull?
14. Why should a handle be used on a file?
15. Why should workpieces be measured often?
16. What happens when a surface being filed is touched with the hand or fingers?
17. How does the hardness of a workpiece affect the selection of a file?
18. How can rounded edges be avoided when a workpiece is draw filed?
19. Should pressure be applied to a file on the return stroke?
20. Why is a round file rotated while it is being used?

UNIT FIVE

Hand Reamers

Holes produced by drilling are seldom accurate in size and often have rough surfaces. A reamer is used to finish a hole to an exact dimension with a smooth finish. Hand reamers are often used to finish a previously drilled hole to an exact dimension and a smooth surface. When parts of machine tools are aligned and fastened with capscrews or bolts, the final operation is often hand reaming a hole in which a dowel pin is placed to maintain the alignment. Hand reamers are designed to remove only a small amount of material from a hole, usually from .001 to .005 in. These tools are made from high-carbon or high-speed steel. This unit describes commonly used hand reamers and how they are used.

OBJECTIVES

After completing this unit, you should be able to:

- Identify at least five types of hand reamers.
- Hand ream a hole to a specified size.

FEATURES OF HAND REAMERS

Figure B-101 shows the major features of the most common design of hand reamer. Another design is available with a pilot ahead of the starting taper (see *Machinery's Handbook* for details). The square on the end of the shank permits the clamping of a tap wrench or T-handle wrench to provide the driving torque for reaming. The diameter of this square is between .004 and .008 in. smaller than the reamer size, and the shank of the reamer is between .001 and .006 in. smaller, to guide the reamer and permit it to pass through a reamed hole without marring it. It is important that these tools not be put into a drill chuck, because a burred shank can ruin a reamed hole as the shank is passed through it.

Hand reamers have a long starting taper that is usually as long as the diameter of the reamer, but may be as long as one-third of the fluted body. This starting taper is usually slight and may not be apparent at a casual glance. Hand reamers do their cutting on this tapered portion. The gentle taper and length of the taper help to start the reamer straight and keep it aligned in the hole.

Details of the cutting end of the hand reamer are shown in Figure B-102. The full diameter or actual size of the hand reamer is measured where the starting taper ends and the margin of the land appears. The diameter of the reamer should be measured only at this junction, as the hand reamer is generally back tapered or reduced in outside diameter by about .0005 to .001 in. per inch of length toward the shank. This back tapering is done to reduce tool contact with the workpiece. When hand reamers become dull, they are resharpened at the starting taper, using a tool and cutter grinder.

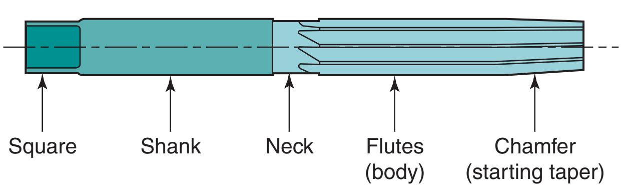

Figure B-101 Major features of the hand reamer.

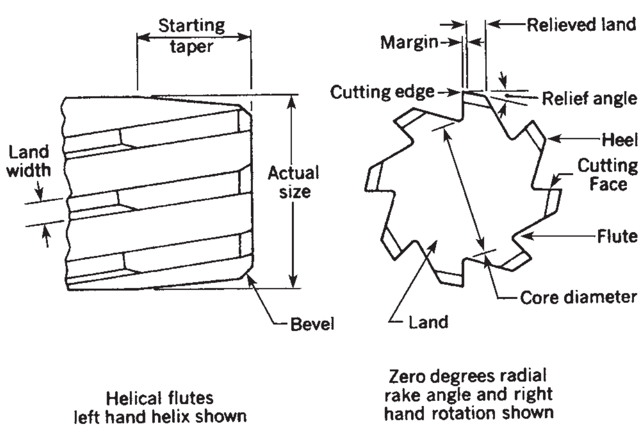

Figure B-102 Functional details of the hand reamer (*Besly Cutting Tools, Inc.*).

The hand reamer functions much like a scraper rather than an aggressive cutting tool like most drills and machine reamers. For this reason hand reamers typically have zero or negative radial rake on the cutting face rather than the positive radial rake characteristic of most machine reamers (see Section H, Unit 6). The right-hand cut with a left-hand helix is considered standard for hand reamers. The left-hand helix produces a negative axial rake for the tool, which contributes to a smooth cutting action.

Most reamers, hand or machine types, have staggered spacing on teeth, which means that the flutes or body channels are not precisely uniformly spaced. The difference is very small, only a degree or two, but it tends to reduce chatter by reducing harmonic effects between cutting edges. Harmonic chatter is especially a problem with adjustable hand reamers, which often leave a tooth pattern in the work.

Hand reamers are made with straight flutes (Figure B-103) or with helical flutes (Figure B-104). Most hand reamers are manufactured with a right-hand cut, which means they will cut when rotated in a clockwise direction. Helical or spiral fluted reamers are available with a right-hand helix or a left-hand helix. Helical flute reamers are especially useful when reaming a hole having keyseats or grooves cut into it, as the helical flutes tend to bridge the gaps and reduce binding or chattering.

Hand reamers for cylindrical holes are made as solid (Figure B-103 and Figure B-104) or expansion types (Figure B-105). Expansion reamers are designed for use where it is necessary to enlarge a hole slightly for proper fit, such as in maintenance applications. These reamers have an adjusting screw that allows limited expansion to an exact size. The maximum expansion of these reamers is approximately .006 in. for diameters up to $\frac{1}{2}$ in., .010 in. for diameters between $\frac{1}{2}$ and 1 in., and .012 in. for diameters between 1 and $1\frac{1}{2}$ in. These tools are frequently broken by attempts to expand them beyond these limits.

Helical flute expansion reamers are especially adapted for the reaming of bushings or holes having a keyseat or straight grooves because of their bridging and shearing cutting action.

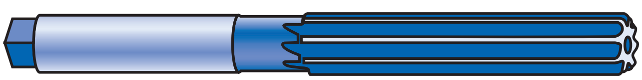

Figure B-103 Straight flute hand reamer.

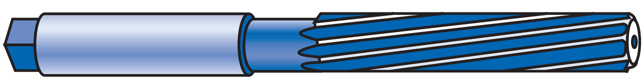

Figure B-104 Helical flute hand reamer.

Figure B-105 Straight flute expansion hand reamer.

Expansion reamers have a slightly undersized pilot on the end that guides the reamer and helps to keep it in alignment.

The adjustable hand reamer (Figure B-106) is different from the expansion reamer in that it has inserted blades. These cutting blades fit into tapered slots in the body of the reamer and are held in place by two locking nuts. The blades have a taper corresponding to the taper of the slots that keeps them parallel at any setting. Adjustments in reamer size are made by loosening one nut while tightening the other. Adjustable hand reamers are available in diameters from $\frac{1}{4}$ to 3 in. The adjustment range varies from $\frac{1}{32}$ in. on the smaller-diameter reamers to $\frac{5}{16}$ in. on the larger size reamers. Only a small amount of material should be removed at one time, as too large a cut will usually cause chatter.

Taper pin reamers (Figures B-107 and B-108) are used for reaming holes for standard taper pins used in the assembly of machine tools and other parts. Taper pin reamers have a taper of $\frac{1}{4}$ in. per foot of length and are manufactured in 18 different sizes numbered from 8/0 to 0 and on up to size 10. The smallest size, number 8/0, has a large-end diameter of .0514 in., and the largest reamer, a number 10, has a large-end diameter of .7216 in. The sizes of these reamers are designed to allow the small end of each reamer to enter a hole reamed by the next smaller size reamer. Like with other hand reamers, the helical flute reamer will cut with more shearing action and less chattering, especially on interrupted cuts.

Morse taper socket reamers are designed to produce holes for American Standard Morse taper shank tools. These reamers are available as roughing reamers (Figure B-109) and as finishing reamers (Figure B-110). The roughing reamer has

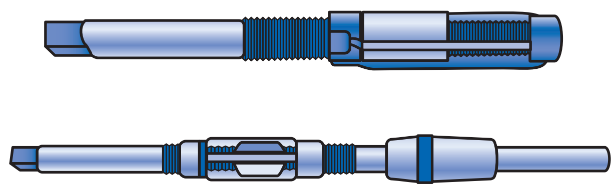

Figure B-106 Adjustable hand reamers. The lower reamer is equipped with a pilot and tapered guide bushing for reaming in alignment with a second hole.

Figure B-107 Straight flute taper pin hand reamer.

Figure B-108 Spiral flute taper pin hand reamer.

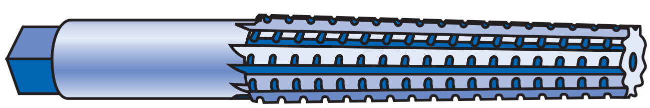

Figure B-109 Morse taper socket roughing reamer.

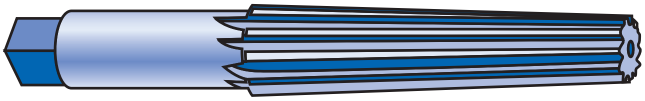

Figure B-110 Morse taper socket finishing reamer.

notches ground at intervals along the cutting edges. These notches act as chip breakers and make the tool more efficient at the expense of fine finish. The finishing reamer is used to impart the final size and finish to the socket. Morse taper socket reamers are made in sizes from No. 0, with a large-end diameter of .356 in., to No. 5, with a large-end diameter of 1.8005 in. There are two larger Morse tapers, but they are typically sized by boring rather than reaming.

USING HAND REAMERS

A hand reamer should be turned with a tap wrench or T-handle wrench rather than with an adjustable wrench. The use of a single-end wrench makes it almost impossible to apply torque without disturbing the alignment of the reamer with the hole. A hand reamer should be rotated slowly and evenly, allowing the reamer to align itself with the hole to be reamed. Use a tap wrench large enough to give a steady torque and to prevent vibration and chatter. Use a steady and large feed; feeds up to one-quarter of the reamer diameter per revolution can be used. Small and lightweight workpieces can be reamed by fastening the reamer vertically in a bench vise and rotating the work over the reamer by hand (Figure B-111).

In all hand reaming with solid, expansion, or adjustable reamers, never rotate the reamer backward to remove it from the hole, as this will dull it rapidly. If possible, pass the reamer through the hole and remove it from the far side without stopping the forward rotation. If this is not possible, it should be withdrawn while maintaining the forward rotation.

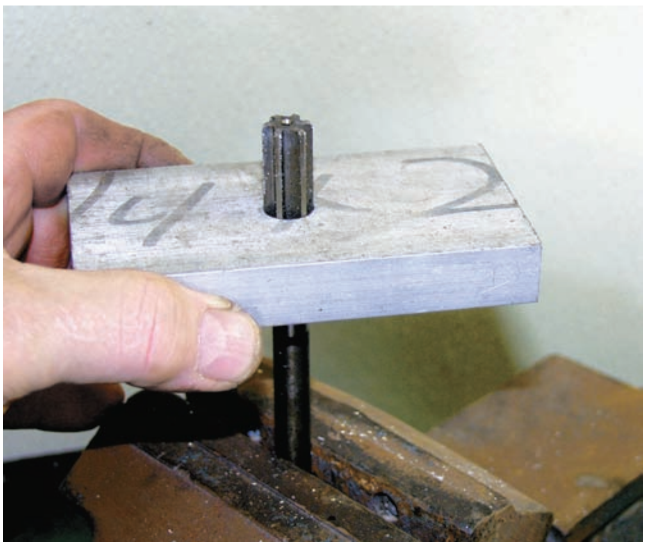

Figure B-111 Hand reaming a small workpiece with the reamer held in a vise.

The preferred stock allowance for hand reaming is between .001 and .005 in. Reaming more material than this would make it very difficult to force the reamer through the workpiece. Reaming too little, on the other hand, results in excessive tool wear because it forces the reamer to work in the zone of material work-hardened during the drilling operation. This stock allowance does not apply to taper reamers, for which a hole has to be drilled at least as large as the small diameter of the reamer. The hole size for a taper pin is determined by the taper pin number and its length. These data can be found in machinist handbooks.

Since cylindrical hand reaming is restricted to small stock allowances, it is most important that you be able to drill a hole of predictable size and of a surface finish that will assure a finished cleanup cut by the reamer. It is a good idea to drill a test hole in a piece of scrap of similar composition and carefully measure both for size and for an enlarged or bell-mouth entrance. You may find it necessary to drill a slightly smaller hole before drilling the correct reaming size to assure a more accurate hole size. Carefully spot drill the location before drilling the hole in your actual workpiece. The hole should then be lightly chamfered with a countersinking tool to remove burrs and to promote better reamer alignment.

The use of a cutting fluid also improves the cutting action and the surface finish when reaming most metals. Exceptions are cast iron and brass, which should be reamed dry.

When a hand reamer is started it should be checked for squareness on two sides of the reamer, 90 degrees apart. Another way to ensure alignment of the reamer with the drilled holes is to use the drill press as a reaming fixture. Put a piece of cylindrical stock with a 60-degree center in the drill chuck (Figure B-112) and use it to guide and follow the

Figure B-112 Using the drill press as a reaming fixture.

squared end of the reamer as you turn the tool with the tap wrench. Be sure to plan ahead so that you can drill, countersink, and ream the hole without moving the table or head of the drill press between operations.

On deep holes, or especially on holes reamed with taper reamers, it becomes necessary to remove the chips frequently from the reamer flutes to prevent clogging. Remove these chips with a brush to avoid cutting your hands.

Reamers should be stored so they do not contact one another to avoid burrs on the tools that can damage a hole being reamed. They should be kept in their original shipping tubes or set up in a tool stand. Always check reamers for burrs or for pickup of previous material before you use them. Otherwise, the reamed hole can be oversized or marred with a rough finish.

SELF-TEST

1. How is a hand reamer identified?
2. What is the purpose of a starting taper on a reamer?
3. What is the advantage of a spiral flute reamer over a straight flute reamer?
4. How does the shank diameter of a hand reamer compare with the diameter measured over the margins?
5. When are expansion reamers used?
6. What is the difference between an expansion and an adjustable reamer?
7. What is the purpose of cutting fluid used while reaming?
8. Why should reamers not be rotated backward?
9. How much reaming allowance is left for hand reaming?
10. If you were repairing the lathe tailstock taper, you would use a ________ reamer.

INTERNET REFERENCE

Information on reamers

http://www.icscuttingtools.com

UNIT SIX

Identification and Uses of Taps

Most internal threads produced today are made with taps. These taps are available in a variety of styles, each one designed to perform a specific type of tapping operation efficiently. This unit will help you identify and select taps for threading operations.

OBJECTIVES

After completing this unit, you should be able to:

- Identify common taps.
- Select taps for specific applications.

IDENTIFYING COMMON TAP FEATURES

Taps are used to cut internal threads in holes. This process is called tapping. Tap features are illustrated in Figures B-113 and B-114. The active cutting part of the tap is the *chamfer*, which is produced by grinding away the tooth form at an angle, with relief back of the cutting edge, so that the cutting action is distributed progressively over a number of teeth. The fluted portion of the tap provides space for chips to accumulate and for the passage of cutting fluids. Two-, three-, and four-flute taps are common.

The *major diameter* (Figure B-113) is the outside diameter of the tool as measured over the thread crests at the first full thread behind the chamfer. This is the largest diameter of the cutting portion of the tap, as most taps are back tapered or reduced slightly in thread diameter toward the shank. This back taper reduces the amount of tool contact with the thread during the tapping process, hence making the tap easier to turn.

Taps are made from either high-carbon steel or high-speed steel and have a hardness of about Rockwell C63. High-speed steel taps are far more common in manufacturing plants than carbon steel taps. High-speed steel taps typically are ground after heat treatment to ensure accurate thread geometry.

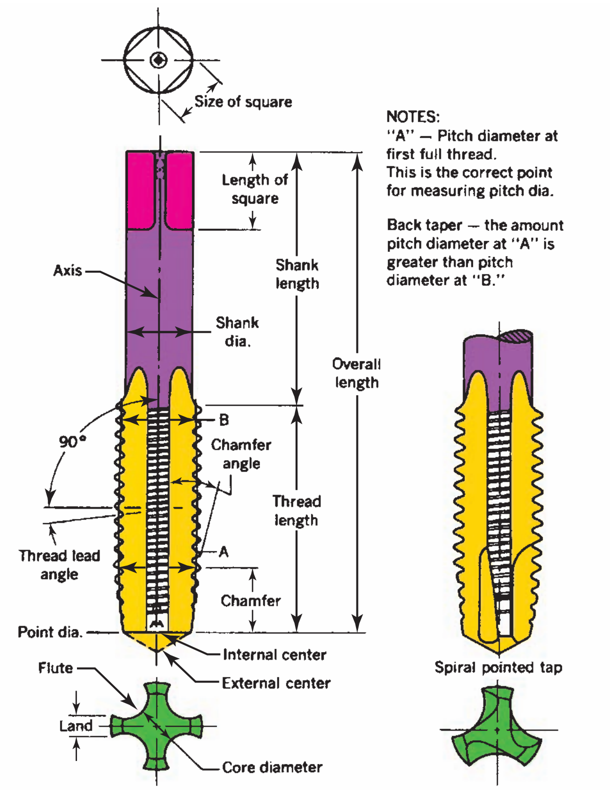

Figure B-113 General tap terms (*Besly Cutting Tools, Inc.*).

Another identifying characteristic of taps is the amount of chamfer at the cutting end of a tap (Figure B-114). A set consists of three taps—taper, plug, and bottoming taps—which are identical except for the number of chamfered threads. The taper tap is useful in starting a tapped thread square with the part. The most commonly used tap, both in hand and machine tapping, is a plug tap. Bottoming taps are used to produce threads that extend almost to the bottom of

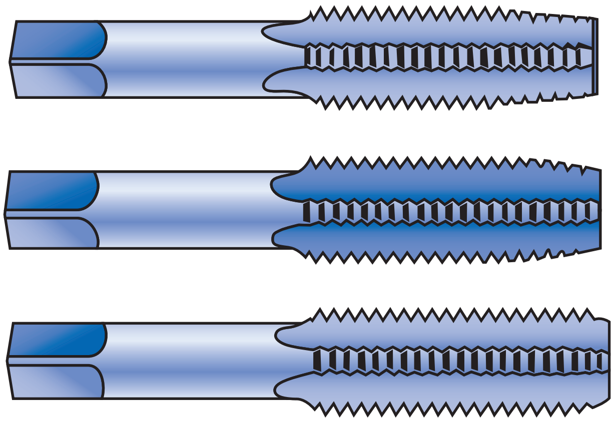

Figure B-114 Chamfer designations for cutting taps. Top to bottom: starting tap, plug tap, and bottoming tap.

a blind hole. A *blind hole* is one that is not drilled entirely through a part.

Serial taps are also made in sets of three taps for any given size of tap. Each of these taps has one, two, or three rings cut on the shank near the square. The No. 1 tap has smaller major and pitch diameters and is used for rough cutting the thread. The No. 2 tap cuts the thread slightly deeper, and the No. 3 tap finishes it to size. Serial taps are used when tapping tough metals by hand. Another tap used for tough metal such as stainless steel is the interrupted thread tap (Figure B-115). This tap has alternate teeth removed to reduce tapping friction.

Figure B-116 shows the identifying markings of a tap, where $\frac{5}{8}$ in. is the nominal size, 11 is the number of threads per inch, and NC refers to the standardized National Coarse thread series. G is the symbol used for ground taps. H3 identifies the tolerance range of the tap. HS means that the tap material is high-speed steel. Left-handed taps will also be identified by an LH or left-hand marking on the shank. More information on taps may be found in *Machinery's Handbook.*

OTHER KINDS AND USES OF TAPS

Spiral pointed taps (Figure B-117), often called *gun taps,* are especially useful for machine tapping of through holes or blind holes with sufficient chip room below the threads.

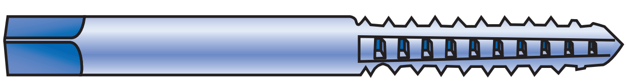

Figure B-115 Interrupted thread tap.

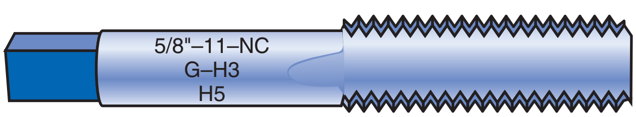

Figure B-116 Identifying marking on a tap.

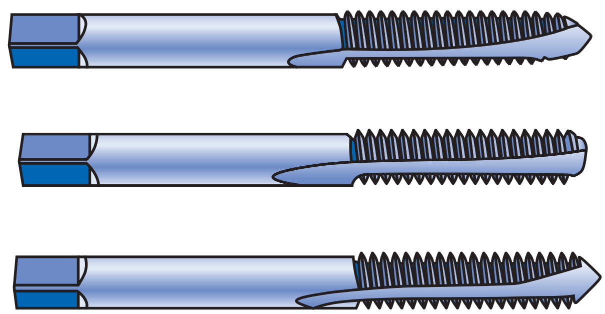

Figure B-117 Set of spiral pointed (or gun) taps.

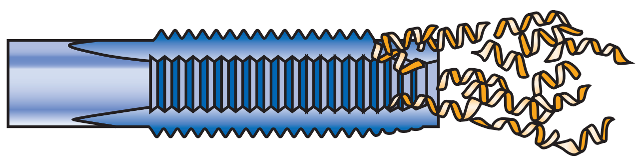

Figure B-118 Cutting action of spiral pointed taps.

When the spiral point is turned, the chips are forced ahead of the tap (Figure B-118). Since the chips are pushed ahead of the tap, the problems caused by clogged flutes, especially breakage of taps, are eliminated if it is a through hole. If a spiral pointed tap is used to tap a blind hole, sufficient hole depth is necessary to accommodate the chips that are pushed ahead of the tap. Also, since they are not needed for chip disposal, the flutes of spiral pointed taps can be made shallower, thus increasing the strength of the tap.

Spiral pointed taps can be operated at higher speeds and require less torque to drive than ordinary hand taps. Figure B-119 shows the design of the cutting edges. The cutting edges (*a*) at the point of the tap are ground at an angle (*b*) to the axis. Fluteless spiral pointed taps (Figure B-120) are recommended for production tapping of through holes in sections no thicker than the tap diameter. This type of tap is strong and rigid, which reduces tap breakage caused by misalignment. Fluteless spiral point taps give excellent results when tapping soft materials or sheet metal.

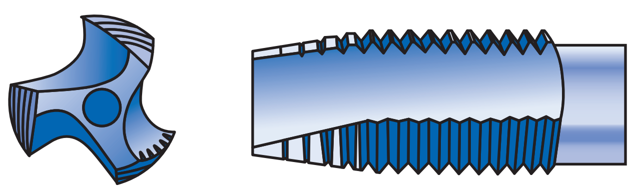

Figure B-119 Detail of spiral pointed tap.

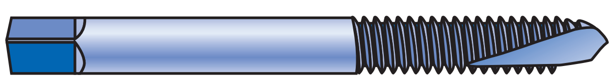

Figure B-120 Fluteless spiral pointed tap for thin materials.

Spiral fluted taps are made with helical flutes instead of straight flutes (Figure B-121), which draw the chips out of the hole. This kind of tap is also used when tapping a hole that has a keyseat or spline, as the helical lands of the tap will bridge the interruptions. Spiral fluted taps are recommended for tapping deep blind holes in ductile materials such as aluminum, magnesium, brass, copper, and die-cast metals. Fast spiral fluted taps (Figure B-122) are similar to regular spiral fluted taps, but the faster spiral flutes increase the chip lifting action and permit the spanning of comparably wider spaces.

Thread-forming taps (Figure B-123) are fluteless and do not cut threads in the same manner as conventional taps. They are forming tools, and their action can be compared with external thread rolling. On ductile materials such as aluminum, brass, copper, die castings, lead, and leaded steels, these taps give excellent results. Thread-forming taps are held and driven just like conventional taps, but because they do not cut the threads, no chips are produced. Problems of chip congestion and removal often associated with the tapping of blind holes are eliminated. Figure B-124 shows how the thread-forming tap displaces metal. The crests of the thread at the minor diameter may not be flat but will be slightly concave because of the flow of the displaced metal. Threads produced in this manner have improved surface finish and increased strength because of the cold working of the metal. The size of the hole to be tapped must be closely controlled, since too large a hole will result in a poor thread form, and too small a hole will result in the breaking of the tap.

A tapered pipe tap (Figure B-125) is used to tap holes with a taper of $\frac{3}{4}$ in. per foot for pipes with a matching thread and to produce a leakproof fit. The nominal size of a pipe tap is that of the pipe fitting and not the actual size of the tap. When taper pipe threads are tapped, every tooth of the tap engaged with the work is cutting until the rotation is stopped. This takes much more torque than does the tapping of a straight thread in which only the chamfered end and the first full thread are actually cutting. Straight pipe taps (Figure B-126) are used for tapping holes or couplings to fit taper-threaded pipe and to secure a tight joint when a sealer is used.

A pulley tap (Figure B-127) is used to tap set screw and oilcup holes in the hubs of pulleys. The long shank also permits tapping in places that might be inaccessible for regular hand taps. When used for tapping pulleys, these taps are

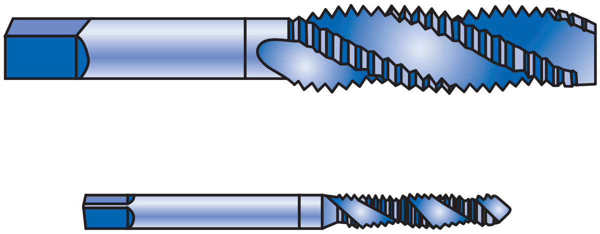

Figure B-121 Spiral fluted taps—regular spiral.

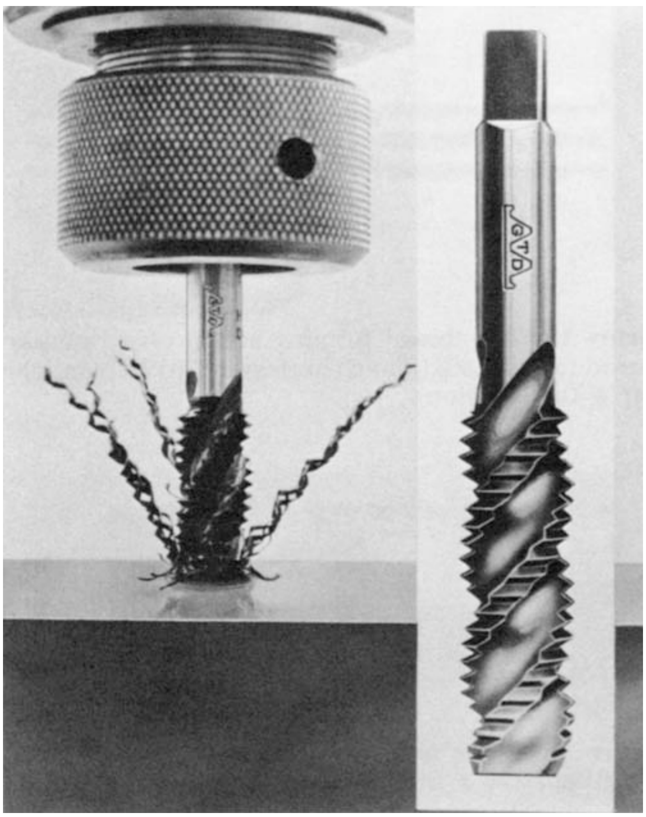

Figure B-122 Spiral fluted tap—fast spiral. The action of the tap lifts the chips out of the hole to prevent binding.

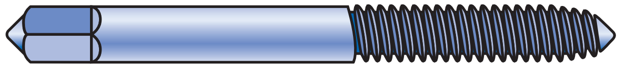

Figure B-123 Fluteless thread-forming tap.

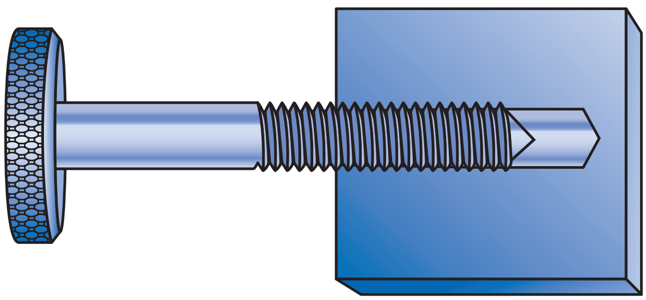

Figure B-124 The thread-forming action of a fluteless thread-forming tap.

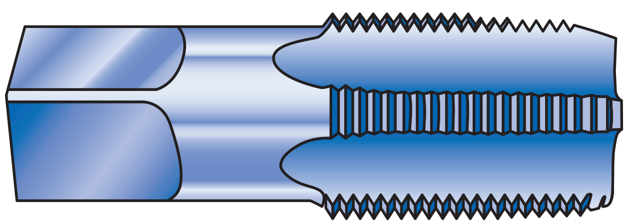

Figure B-125 Taper pipe tap.

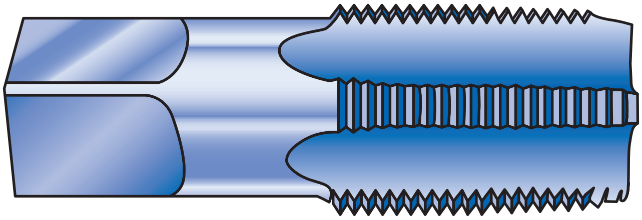

Figure B-126 Straight pipe tap.

Figure B-127 Pulley tap.

inserted through holes in the rims, which are slightly larger than the shanks of the taps. These holes serve to guide the taps and assure proper alignment with the holes to be tapped.

Nut taps (Figure B-128) differ from pulley taps in that their shank diameters are smaller than the root diameter of the thread. The smaller shank diameter makes the tapping of deep holes possible. Nut taps are used when small quantities of nuts are made or when nuts have to be made from tough materials such as some stainless steels or similar alloys.

Figure B-129 shows Acme taps for roughing and finishing. Acme threads are used to provide accurate movement—for example, in lead screws on machine tools—and for applying pressure in various mechanisms. On some Acme taps the roughing and finishing operation is performed with one tap (Figure B-130). The length of this tap usually requires a through hole.

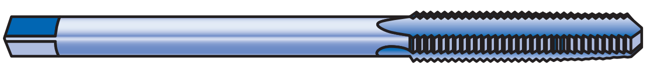

Figure B-128 Nut tap.

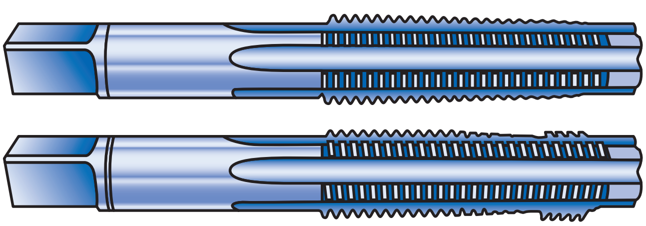

Figure B-129 Set of Acme thread taps. The upper tap is used for roughing, the lower tap for finishing.

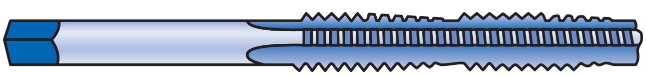

Figure B-130 Tandem Acme tap designed to rough and finish cut the thread in one pass.

Table B-1 Recommended Tap Rake Angles

0–5 Degrees	8–12 Degrees	16–20 Degrees
Bakelite	Bronze	Aluminum and alloys
Plastics	Hard rubber	Zinc die castings
Cast iron	Cast steel	Copper
Brass	Carbon steel	Magnesium
Hard rubber	Alloy steel	
	Stainless steel	

RAKE AND HOOK ANGLES ON CUTTING EDGES

When selecting a tap for the most efficient cutting, the cutting face geometry will be an important factor. It should vary depending on the material to be tapped. Cutting face geometry is expressed in terms of rake and hook (Figure B-131). The *rake* of a tap is the angle between a line through the flat cutting face and a radial line from the center of the tool to the tooth tip. The rake can be negative, neutral, or positive. *Hook angle*, on the other hand, relates to the concavity of the cutting face. It is defined by the intersection of the radial line with the tangent line through the tooth tip (tangential hook) or by the average angle of the tooth face from crest to root (chordal). Unlike the rake angle, the hook angle cannot be negative. Table B-1 gives the rake angle recommendations for workpiece materials. In general, the softer or more ductile the material, the greater the rake angle. Harder and more brittle materials call for reduced rake angles.

REDUCING FRICTION IN TAPPING

As discussed earlier in this unit, a tap is usually back tapered along the thread to relieve the friction between the tool and the workpiece. Another form of relief is often applied to taps with

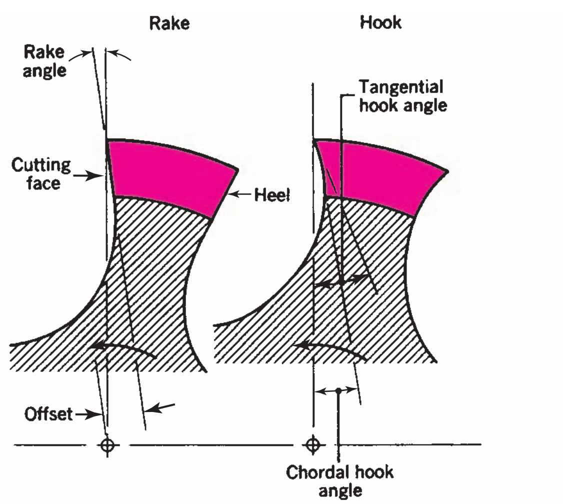

Figure B-131 Rake and hook angles on cutting taps (*Besly Cutting Tools, Inc.*).

the same results. When the fully threaded portion of the tap is cylindrical (other than back taper), it is called a *concentric thread* (Figure B-132). If the pitch diameter of the fully threaded portion of the tap is brought uniformly closer to the axis of the tap as measured from face to back (heel), it has *eccentric relief*. This means less tool contact with the workpiece and less friction. A third form of friction relief combines the concentric thread and the eccentric thread relief and is termed *con-eccentric*. The concentric margin gives substantial guidance, and the relief following the margin reduces friction. Relief is also provided behind the chamfer of the tap to provide radial clearance for the cutting edge. Relief may also be provided in the form of a channel that runs lengthwise down the center of the land (Figure B-133), termed a *concave groove land relief*.

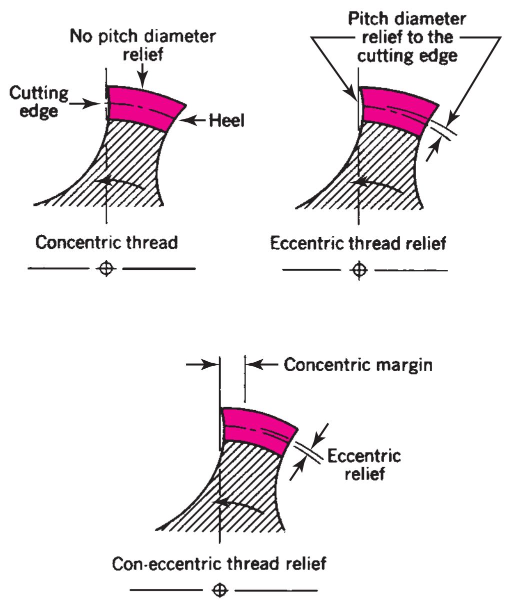

Figure B-132 Pitch diameter relief forms on taps (*Besly Cutting Tools, Inc.*).

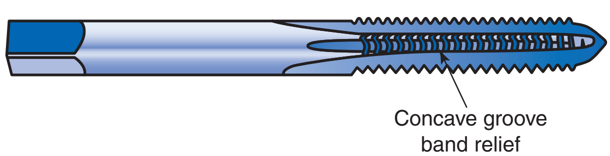

Figure B-133 Tap with concave groove land relief.

Other steps may also be taken to reduce friction and to increase tap life. Surface treatment of taps is often an answer if poor thread forming or tap breakage is caused by chips adhering to the flutes or welding to the cutting faces. These treatments generally improve the wear life of taps by increasing their abrasion resistance.

Different kinds of surface treatments are used by tap manufacturers. Liquid nitride produces a hard, shallow surface on high-speed steel tools when these tools are immersed in cyanide salts at closely controlled temperatures. Oxide finishes are usually applied in steam tempering furnaces and can be identified by their bluish black color. The oxide acts as a solid lubricant. It also holds liquid lubricant at the cutting edges during a tapping operation. Oxide treatments prevent chips from welding to the tool and reduce friction between the tool and the work. Chrome plating is an effective treatment for taps used on nonferrous metals and some soft steels. The chromium deposit is shallow and often referred to as *flash chrome plating*. Titanium nitride is another effective coating for extending the working life of taps.

SELF-TEST

1. What difference exists between a set of taps and serial taps?
2. Where is a spiral pointed tap used?
3. When is a fluteless spiral pointed tap used?
4. When is a spiral fluted tap used?
5. How are thread-forming taps different from conventional taps?
6. How are taper pipe taps identified?
7. Why are finishing and roughing Acme taps used?
8. Why are the rake angles varied on taps for different materials?
9. Name at least three methods used by tap *manufacturers* to reduce friction between the tap and the workpiece material.
10. What is the advantage of a fluteless tap?

INTERNET REFERENCES

Information on taps

www.e-taps.com

www.osgtool.com

http://www.google.com/threadingtaps

http://www.directindustry.com

UNIT SEVEN

Tapping Procedures

Today's mass production of consumer goods depends to a large extent on the efficient and secure assembly of parts using threaded fasteners. It takes skill to produce usable tapped holes, so a worker in the metal trades must have an understanding of the factors that affect the tapping of a hole, such as the work material and its cutting speed, the proper cutting fluid, and the size and condition of the hole. A good machinist can analyze a tapping operation, determine whether it is satisfactory, and usually find a solution if it is not. In this unit you will learn about common tapping procedures.

OBJECTIVES

After completing this unit, you should be able to:

- Select the correct tap drill for a specific percentage of thread.
- Determine the cutting speed for a given work material–tool combination.
- Select the correct cutting fluid for tapping.
- Tap holes by hand or with a drill press.
- Identify and correct common tapping problems.

TAP USE

Taps are used to cut internal threads in holes. The actual cutting process is called *tapping* and can be performed by hand or with a machine. A tap wrench (Figure B-134) or a T-handle tap wrench (Figure B-135) attached to the tap is used to provide driving torque while hand tapping. To obtain a greater accuracy in hand tapping, a hand tapper (Figure B-136) is used. This fixture acts as a guide for the tap to ensure that it stays in alignment and cuts concentric threads.

Holes can also be tapped in a drill press that has a spindle reverse switch, which is often foot operated for convenience. Drill presses without reversing switches can be used for tapping with a tapping attachment (Figure B-137). Some of these tapping attachments have an internal friction clutch, so that downward pressure on the tap turns the tap forward and feeds it into the work. Releasing downward pressure will automatically reverse the tap and back it out of the workpiece. Some tapping attachments have lead screws that provide tap feed rates equal to the lead of the tap. Most of these attachments also have an adjustment to

Figure B-134 Tap wrench.

Figure B-135 T-handle tap wrench.

Figure B-136 Hand tapper (*Courtesy of Enco*).

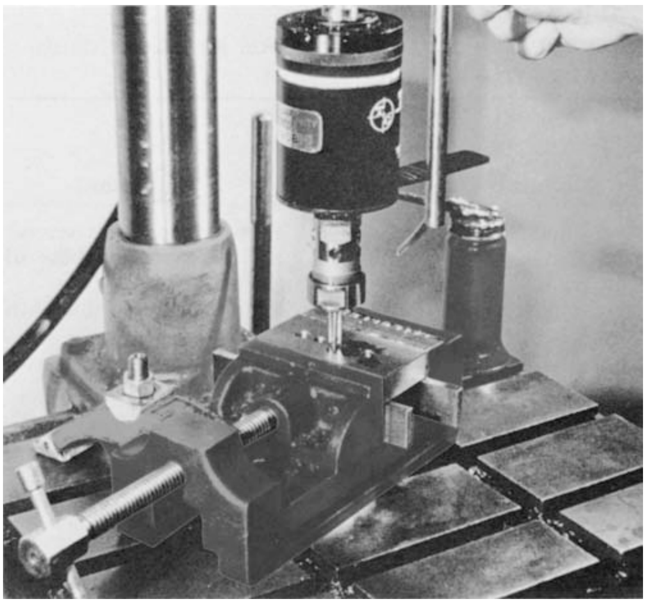

Figure B-137 Drill press tapping attachment.

limit the torque to match the size of the tap, which eliminates most tap breakage.

THREAD PERCENTAGE AND HOLE STRENGTH

The strength of the thread in a tapped hole depends largely on the workpiece material, the percentage of full thread depth used, and the length of the thread. The workpiece material is usually selected by the designer, but the machinist can often control the percentage of thread produced and the depth of the thread. The percentage of thread produced is dependent on the diameter of the drilled hole. Tap drill charts generally give tap drill sizes to produce 75 percent thread. (See Appendix Tables 3 and 4 for tap drill charts.)

An example will illustrate the relationships between the percentage of thread, torque required to drive the tap, and resulting thread strength. An increase in thread depth from 60 percent to 72 percent in AISI 1020 steel requires twice the torque to drive the tap, but it increases the strength of the thread by only 5 percent. The practical limit seems to be 75 percent of full thread, since a greater percentage of thread does not increase the strength of the threaded hole in most materials.

In some difficult-to-machine materials such as titanium alloys, high-tensile steels, and some stainless steels, 50 percent to 60 percent thread depth will give sufficient strength to the tapped hole. Threaded assemblies are usually designed so that the bolt breaks before the threaded hole strips. Common practice is to have a bolt engage a tapped hole by 1 to $1\frac{1}{2}$ times its diameter.

DRILLING THE RIGHT HOLE SIZE

The condition of the drilled hole affects the quality of the thread produced, as an out-of-round hole leads to an out-of-round thread. Bellmouthed holes will produce bellmouth threads. When an exact hole size is needed, the hole should be reamed before tapping. This is especially important for large-diameter taps and when fine pitch threads are used. The size of the hole to be drilled is usually obtained from tap drill charts, which usually show a 75 percent thread depth. If a thread depth other than 75 percent is wanted, use the following formula to determine the proper hole size: For the American National Unified form:

$$\text{Outside diameter of thread} - \frac{.01266 \times \text{\% of thread depth}}{\text{number of threads per inch (TPI)}} = \text{hole diameter}$$

For example, to calculate the hole size for a 1-in. 12-thread fastener with a 70 percent thread depth:

$$1.0 - \frac{.01266 \times 70\%}{12} = .926$$

This formula will work for any thread system.

SPEEDS FOR TAPPING

When a thread is tapped by hand with a tap wrench, speed is not a consideration at all, but when a tapping machine or attachment is used, speed is very important. The quality of the thread produced also depends on the speed at which a tap is operated. The selection of the best speed for tapping is limited, unlike the varying speeds and feeds possible with other cutting tools, because the feed per revolution is fixed by the lead of the thread. Excessive speed develops high temperatures that cause rapid wear of the tap's cutting edge. Dull taps produce rough or torn and off-size threads. High cutting speeds prevent adequate lubrication at the cutting edges and often create a problem of chip disposal.

When selecting the best speed for tapping, you should consider not only the material being tapped, but also the size of the hole, the kind of tap holder being used, and the lubricant being used. Table B-2 gives guidelines in selecting a speed and a lubricant for materials when using high-speed steel taps.

These cutting speeds in feet per minute have to be translated into rpm to be useful. For example, calculate the rpm when tapping a $\frac{3}{8}$-24 UNF hole in free-machining steel. The cutting speed chart gives a cutting speed between 60 and 80 ft/min. Use the lower figure; you can increase the speed once you see how the material taps. The formula for calculating rpm is

$$\frac{\text{cutting speed (CS)} \times 4}{\text{diameter } (D)} \quad \text{or} \quad \frac{60 \times 4}{3/8} = 640 \text{ rpm}$$

Lubrication is one of the most important factors in a tapping operation. Cutting fluids used when tapping serve as coolants but are more important as lubricants. It is important to select the correct lubricant because using the wrong lubricant may give results that are worse than if no lubricant were used. For lubricants to be effective, they should be applied in sufficient quantity to the actual cutting area in the hole. (See Section F for more information on cutting speeds and cutting fluids.)

Table B-2 Recommended Cutting Speeds and Lubricants for Machine Tapping

Material	Speeds (ft/min)	Lubricant
Aluminum	90–100	Kerosene and light base oil
Brass	90–100	Soluble oil or light base oil
Cast iron	70–80	Dry or soluble oil
Magnesium	20–50	Light base oil diluted with kerosene
Phosphor bronze	30–60	Mineral oil or light base oil
Plastics	50–70	Dry or air jet
Steels		
Low carbon	40–60	Sulfur-base oil
High carbon	25–35	Sulfur-base oil
Free machining	60–80	Soluble oil
Molybdenum	10–35	Sulfur-base oil
Stainless	10–35	Sulfur-base oil

SOLVING TAP PROBLEMS

In Table B-3, common tapping problems are presented with some possible solutions. Occasionally, it becomes necessary to remove a broken tap from a hole. If a part of the broken tap extends out of the workpiece, removal is relatively easy with a pair of pliers. If the tap breaks flush with or below the surface of the workpiece, a tap extractor can be used (Figure B-138). Before trying to remove a broken tap, remove the chips in the flutes. A jet of compressed air or cutting fluid can be used for this.

CAUTION

Always stand aside when cleaning out holes with compressed air, as chips and particles tend to fly out at high velocity.

When the chips are packed so tightly in the flutes or the tap is jammed in the work so that a tap extractor cannot be used, the tap may be broken up with a pin punch and removed piece by piece. If the tap is made from carbon steel and cannot be pin punched, the tap can be annealed so it becomes possible to drill it out.

On high-speed steel taps it may be necessary to use an electrical discharge machine (EDM), sometimes called a *tap disintegrator,* to remove the broken tap. These machines erode away material from extremely hard workpieces while they are immersed in a fluid. The shape of the hole conforms precisely to that of the electrode.

TAPPING PROCEDURE, HAND TAPPING

Step 1 Determine the size of the thread to be tapped and select the tap.

Step 2 Select the proper tap drill with the aid of a tap drill chart. Choose a taper tap for hand tapping; or if a drill press or tapping machine is to be used for alignment, use a plug tap.

Step 3 Fasten the workpiece securely in a drill press vise. Calculate the correct rpm for the drill used:

$$\text{rpm} = \frac{\text{CS} \times 4}{D}$$

Figure B-138 Tap extractor.

Table B-3 Common Tapping Problems and Possible Solutions

Causes of Tap Breakage	Solution
Tap hitting bottom of hole or bottoming on packed chips	Drill hole deeper. Eject chips with air pressure. (*Caution:* Stand aside when you do this and always wear safety glasses.) Use spiral fluted taps to pull chips out of hole. Use a thread-forming tap.
Chips are packing in flutes	Use a tap style with more flute space. Tap to a lesser depth or use a smaller percentage of threads. Select a tap that will eject chips forward (spiral point) or backward (spiral fluted).
Hard materials or hard spots	Anneal the workpiece. Reduce cutting speed. Use longer chamfers on tap. Use taps with more flutes.
Inadequate lubricant	Use the correct lubricant and apply a sufficient amount of it under pressure at the cutting zone.
Tapping too fast	Reduce cutting speed.
Excessive wear	
Abrasive materials	Improve lubrication. Use surface-treated taps. Check the alignment of tap and hole to be tapped.
Chips clogging flutes	
Insufficient lubrication	Use a better lubricant and apply it with pressure at the cutting zone.
Excessive speed	Reduce cutting speed.
Wrong-style tap	Use a more-free-cutting tap such as a spiral pointed tap, spiral fluted tap, interrupted thread tap, or surface-treated taps.
Torn or rough threads	
Dull tap	Resharpen.
Chip congestion	Use a tap with more chip room. Use lesser percentage of thread. Drill a deeper hole. Use a tap that will eject chips.
Inadequate lubrication and chips clogging flutes	Correct as suggested previously.
Hole improperly prepared	Torn areas on the surface of the drilled, bored, or cast hole will be shown in the minor diameter of the tapped thread.
Undersize threads	
Pitch diameter of tap too small	Use a tap with a large pitch diameter.
Excessive speed	Reduce tapping speed.
Thin-wall material	Use a tap that cuts as freely as possible. Improve lubrication. Hold the workpiece so that it cannot expand while it is being tapped. Use an oversize tap.
Dull tap	Resharpen.
Oversize or bellmouth threads	
Loose spindle or worn holder	Replace or repair spindle or holder.
Misalignment	Align spindle, fixture, and work.
Tap oversized	Use a smaller-pitch-diameter tap.
Dull tap	Resharpen.
Chips packed in flutes	Use a tap with deeper flutes, spiral flutes, or spiral points.
Buildup on cutting edges of tap	Use correct lubricant and tapping speed.

Drill the hole using the recommended coolant. Check the hole size.

Step 4 Countersink the hole entrance to a diameter slightly larger than the major diameter of the threads (Figure B-139). This allows the tap to be started more easily, and it protects the start of the threads from damage.

Step 5 Mount the workpiece in a bench vise so that the hole is in a vertical position.

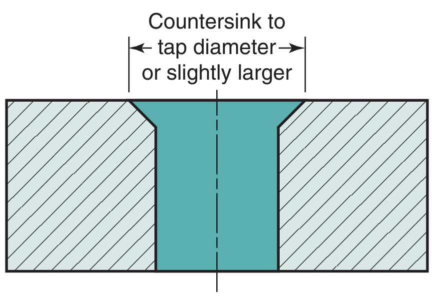

Figure B-139 Preparing the workpiece.

Step 6 Tighten the tap in the tap wrench.

Step 7 Cup your hand over the center of the wrench (Figure B-140) and place the tap in the hole in a vertical position. Start the tap by turning two or three turns in a clockwise direction for a right-hand thread. At the same time, keep a steady pressure downward on the tap. When the tap is started, it may be turned as shown in Figure B-141.

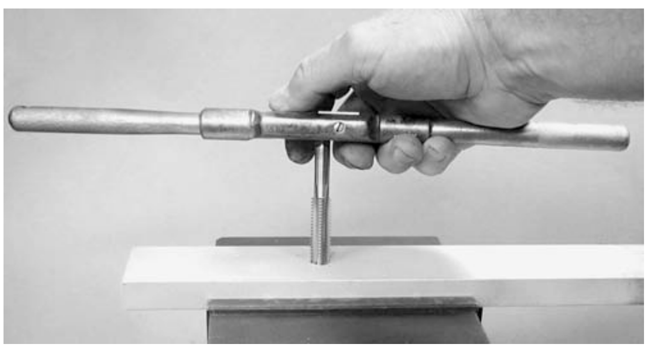

Figure B-140 Starting the tap.

Figure B-141 Tapping a thread by hand.

Step 8 After the tap is started for several turns, remove the tap wrench without disturbing the tap. Place the blade of a square against the solid shank of the tap to check for squareness (Figure B-142). Check from two positions 90 degrees apart. If the tap is not square with the work, it will ruin the thread and possibly break in the hole if you continue tapping. Back the tap out of the hole and restart.

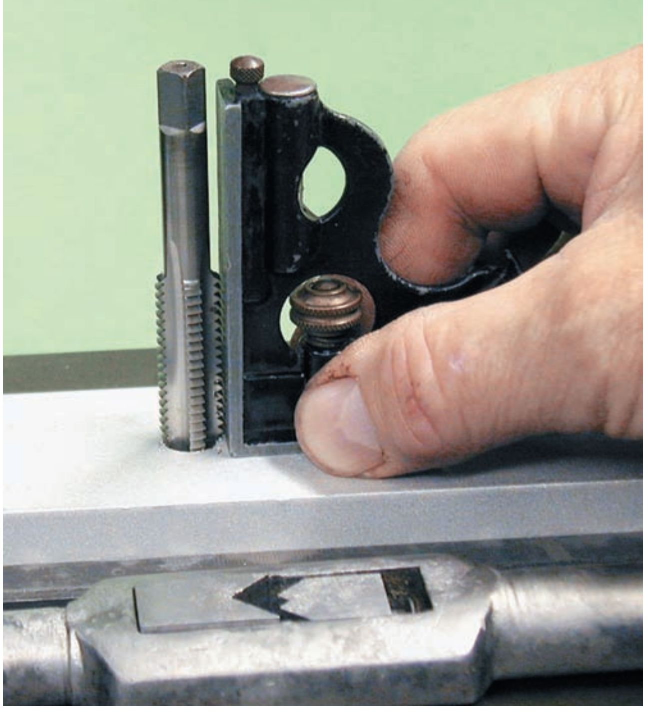

Figure B-142 Checking the tap for squareness.

Step 9 Use the correct cutting oil on the tap when cutting threads.

Step 10 Turn the tap clockwise one-quarter to one-half turn and then turn it back three-quarters of a turn to break the chip. Do this with a steady motion to avoid breaking the tap.

Step 11 When tapping a blind hole, use the taps in the order starting, plug, and then bottoming. Remove the chips from the hole before using the bottoming tap, and be careful not to hit the bottom of the hole with the tap.

Step 12 Figure B-143 shows a 60-degree point center chucked in a drill press to align a tap squarely with the previously drilled hole. Only very slight follow-up pressure should be applied to the tap. Too much downward pressure will cut a loose, oversize thread.

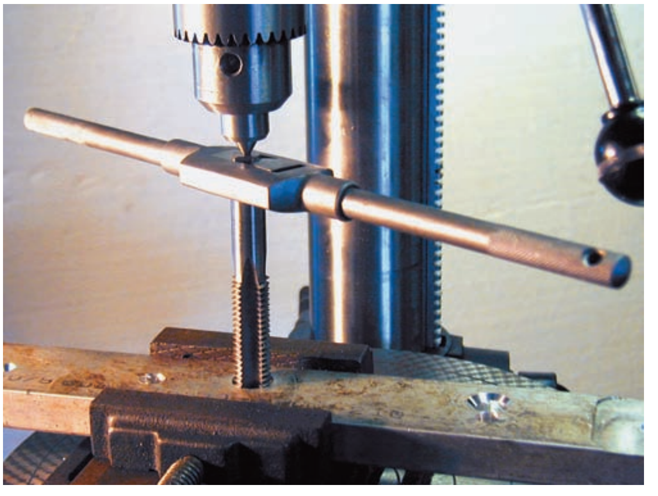

Figure B-143 Using the drill press as a tapping fixture.

SELF-TEST

1. What kind of tools are used to drive taps when hand tapping?
2. What is a hand tapper?
3. What is a tapping attachment?
4. Which three factors affect the strength of a tapped hole?
5. How deep should the usable threads be in a tapped hole?
6. When should tap drill holes be reamed?
7. What causes taps to break while tapping?
8. What causes rough and torn threads?
9. What causes oversized threads in a hole?
10. Give three methods of removing broken taps from holes.

UNIT EIGHT

Thread-Cutting Dies and Their Uses

A die is used to cut external threads on the surface of a bolt or rod. Many machine parts and mechanical assemblies are held together with threaded fasteners, most of which are mass-produced. If necessary, the threaded portion of a bolt may be extended with a die toward the head, but this should be done with unhardened bolts, as cutting heat-treated bolts will dull the die. In this unit you will be introduced to some thread-cutting dies and their uses.

OBJECTIVES

After completing this unit, you should be able to:

- Identify dies used for hand threading.
- Select and prepare a rod for threading.
- Cut threads with a die.

Dies are used to cut external threads on round materials. Some dies are made from carbon steel, but most are made from high-speed steel. Dies are identified by the markings on the face as to the size of thread, number of threads per inch, and form of thread, such as NC, UNF, or other standard designations (Figure B-144).

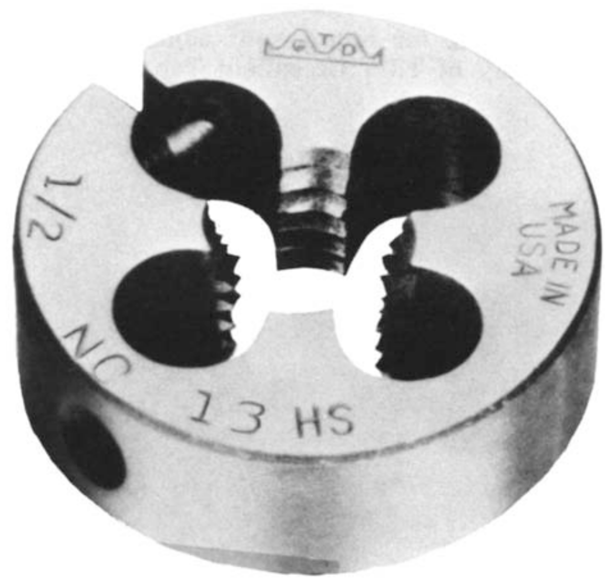

Figure B-144 Markings on a die. (Example shown is a round split adjustable die.)

COMMON TYPES OF HAND THREADING DIES

The die shown in Figure B-144 is an example of a round split adjustable die, also called a *button die*. These dies are made in all standardized thread sizes up to $1\frac{1}{2}$-in. thread diameters and $\frac{1}{2}$-in. pipe threads. The outside diameters of these dies vary from $\frac{5}{8}$ to 3 in.

Adjustments on these dies are made by turning a fine-pitch screw that forces the sides of the die apart or allows them to spring together. The range of adjustment of round split adjustable dies is very small, allowing only for a loose or tight fit on a threaded part. Adjustments made to obtain threads several thousandths of an inch oversize will result in poor die performance, because the heel of the cutting edge will drag on the threads. Excessive expansion may cause the die to break.

Some round split adjustable dies do not have the built-in adjusting screw. Adjustments are then made with the three screws in the diestock (Figure B-145). Two of these screws on opposite sides of the diestock hold the die in the diestock and also provide closing pressure. The third screw engages the split in the die and provides opening pressure. These dies are used in a diestock for hand threading or in a machine holder for machine threading.

Another type of threading die is the two-piece die, whose halves (Figure B-146) are called *blanks*. The blanks are assembled in a collet consisting of a cap and the guide (Figure B-147). The normal position of the blanks in the collet is indicated by witness marks (Figure B-148). The adjusting

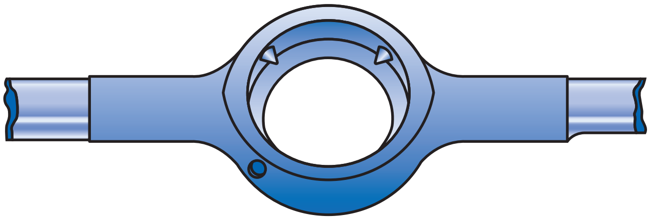

Figure B-145 Diestock for round split adjustable dies.

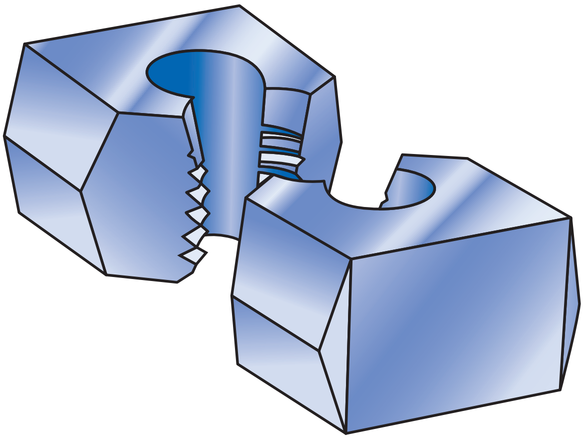

Figure B-146 Die halves for two-piece die.

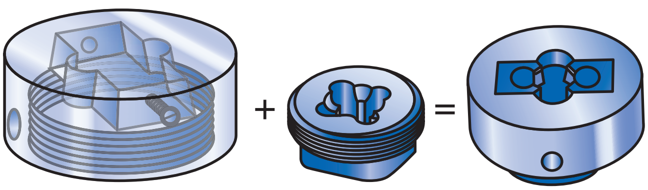

Figure B-147 Components of a split adjustable die collet.

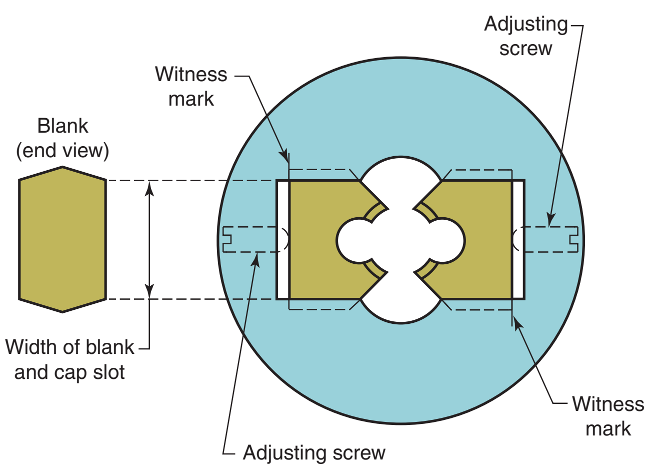

Figure B-148 Setting the die position to the witness marks on the die and collet assembly.

screws allow for precise control of the cut thread size. The blanks are inserted in the cap with the tapered threads toward the guide. Each of the two die halves is stamped with a serial number. Make sure that the halves you select have the same numbers. The guide used in the collet serves as an aid in starting and holding the dies square with the work being threaded. Each thread size uses a guide of the same nominal or indicated size. Collets are held securely in diestocks (Figure B-149) by a knurled setscrew that seats in a dimple in the cap.

Hexagon rethreading dies (Figure B-150) are used to recut slightly damaged or rusty threads. Rethreading dies are driven with a wrench large enough to fit the die. Solid square dies (Figure B-151) have the same uses as hexagon rethreading dies. All the die types discussed previously are also available in pipe thread sizes. Square dies are used to cut new threads and have sufficient chip clearance for this purpose.

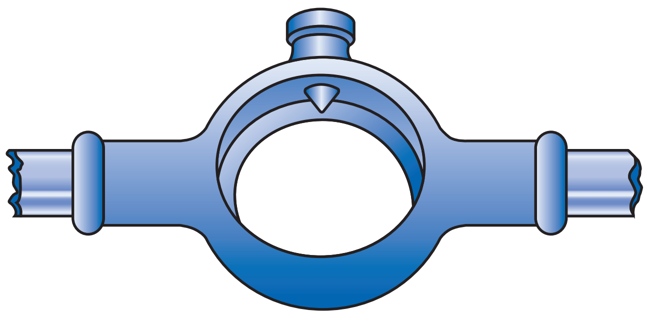

Figure B-149 Diestock for adjustable die and collet assembly.

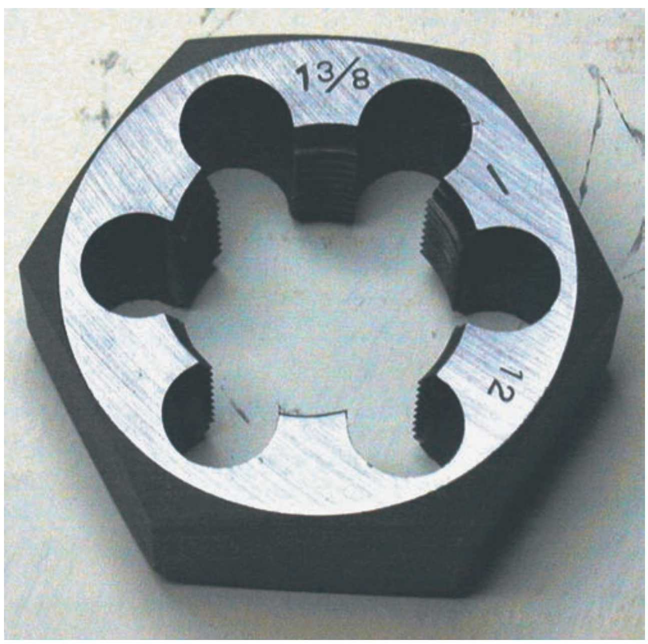

Figure B-150 Hexagon rethreading die.

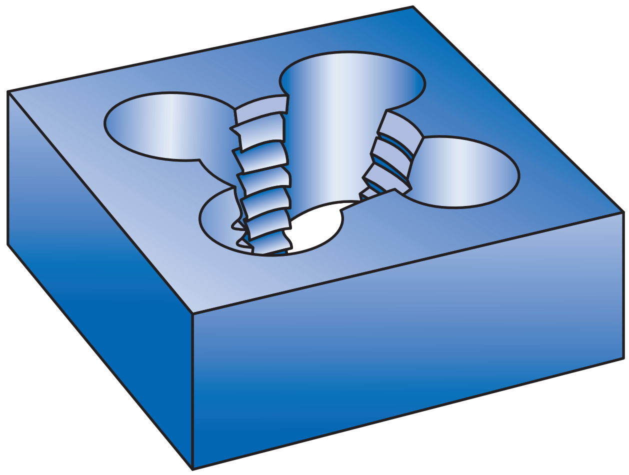

Figure B-151 Solid square die.

HAND THREADING PROCEDURES

Threading of a rod should always be started with the leading or throat side of the die. This side is identified by the chamfer on the first two or three threads and also by the size markings. The chamfer distributes the cutting load over a number of threads, which produces better threads and less chance of chipping the cutting edges of the die. Cutting oil or other threading fluids are important in obtaining quality threads and maintaining long die life. Once a cut is started with a die, it will tend to follow its own lead, but uneven pressure on the die stock will make the die cut variable helix angles or "drunken" threads.

Threads cut by hand often show a considerable accumulated lead error. The *lead* of a screw thread is the distance a nut will move on the screw if it is turned one full revolution. This problem results because the dies are relatively thin compared with the diameter of thread they cut. Only a few threads in the die can act as a guide on the already cut threads. This error usually does not cause problems when standard or thin nuts are used on the threaded part. However, when an item with a long internal thread is assembled with a threaded rod, it usually tightens and then locks, not because the thread depth is insufficient, but because there is a lead error. This lead error can be as much as one-fourth of a thread in 1 in. of length.

The outside diameter of the material to be threaded should not be more than the nominal size of the thread and preferably a few thousandths of an inch (.002 to .005 in.) undersized. After a few full threads are cut, the die should be removed so that the thread can be tested with a nut or thread ring gage. A thread ring gage set usually consists of two gages, a *go* and a *no go* gage. As the names imply, a go gage should screw on the thread, whereas the no go gage will not go more than $1\frac{1}{2}$ turns on a thread of the correct size. Do not assume that the die will cut the correct size thread; always check by gaging or assembling. Adjustable dies should be spread open for the first cut and set progressively smaller for each pass after checking the thread size.

It is important that a die be started squarely on the rod to be threaded. A lathe can be used as a fixture for cutting threads with a die (Figure B-152). The rod is fastened in a lathe chuck for rotation, while the die is held square because it is supported by the face of the tailstock spindle. The carriage or the compound rest prevents the diestock from turning while the chuck is rotated by hand. As the die advances, the tailstock spindle is also advanced to stay in contact with the die. Do not force the die with the tailstock spindle, or a loose thread may result. A die may be used to finish to size a long thread that has been rough threaded on the lathe.

It is always good practice to chamfer the end of a workpiece before starting a die (Figure B-153). The chamfer on the end of a rod can be made by grinding on a pedestal grinder, by filing, or with a lathe. This will help in starting the cut and will also leave a finished thread end. When cutting threads

Figure B-152 Threading a rod with a hand die in a lathe.

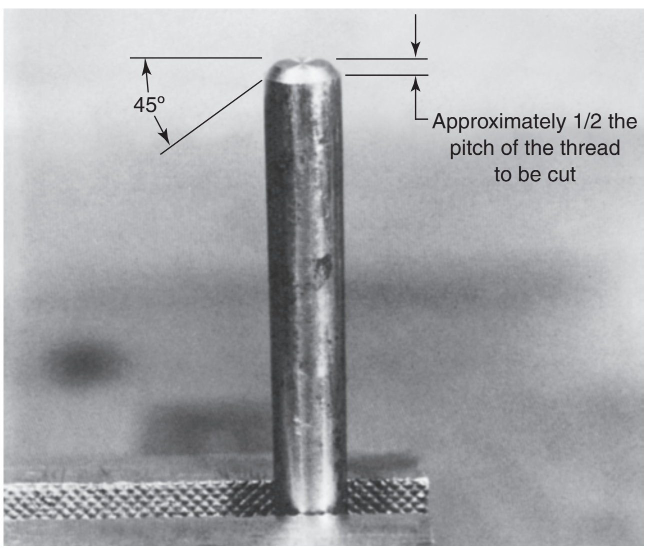

Figure B-153 Chamfer workpiece before using die.

with a hand die, reverse the rotation of the die after each full turn forward to break the chips into short pieces that will fall out of the die. Chips jammed in the clearance holes will tear the thread.

THREADING PROCEDURE, THREADING DIES

Step 1 Select the workpiece to be threaded and measure its diameter. Then, chamfer the end, either on a grinder or with a file. The chamfer should be at least as deep as the thread to be cut.

Step 2 Select the correct die and mount it in a diestock.

Step 3 Mount the workpiece in a bench vise. Short workpieces are mounted vertically and long pieces usually are held horizontally.

Step 4 To start the thread, place the die over the workpiece. Holding the diestock with one hand (Figure B-154), apply downward pressure and turn the die.

Step 5 When the cut has started, apply cutting fluid to the workpiece and die, and start turning the diestock with both hands (Figure B-155). After each complete revolution forward, reverse the die one-half turn to break the chips.

Step 6 Check to see that the thread is started square, using a machinist's square. Make any necessary corrections by applying slight downward pressure on the high side while turning.

Figure B-154 Start the die with one hand.

Figure B-155 Use both hands to turn the threading die.

Step 7 When several turns of the thread have been completed, check the fit of the thread with a nut, thread ring gage, thread micrometer, or the mating part. If the thread fit is incorrect, adjust the die with the adjustment screws and take another cut with the adjusted die. Continue making adjustments until the proper fit is achieved.

Step 8 Continue threading to the required thread length. To cut threads close to a shoulder, invert the die after the normal threading operation and cut the last two or three threads with the side of the die that has less chamfer.

SELF-TEST

1. What is a die?
2. What tool is used to drive a die?
3. How much adjustment is possible with a round split adjustable die?
4. What is the purpose of the guide in a two-piece adjustable die collet?
5. What are important points to watch when assembling two-piece dies in a collet?
6. Where are hexagon rethreading dies used?
7. Why do dies have a chamfer on the cutting end?
8. Why are cutting fluids used?
9. What diameter should a rod be before being threaded?
10. Why should a rod be chamfered before being threaded?

INTERNET REFERENCES

Information on threading dies

http://www.tapsdiesandreamers.com

http://en.wikipedia.org/Wiki/Taps_and_dies

UNIT NINE

Off-Hand Grinding

Although it is a machine tool, the pedestal grinder is used for many hand grinding operations, especially sharpening and shaping drills and tool bits. In this unit you will study the setup, use, and safety aspects of this important machine.

OBJECTIVE

After completing this unit, you should be able to:

- Describe the setup, use, and safety of the pedestal grinder.

OFF-HAND GRINDING ON PEDESTAL GRINDERS

The pedestal grinder is really a machine tool. However, since the workpiece is handheld, it is more logical to discuss this machine in conjunction with hand tools. Furthermore, you must be familiar with the pedestal grinder, as you will be using it early in your study of machine tool practices.

The pedestal grinder gets its name from the floor stand or pedestal that supports the motor and abrasive wheels. The pedestal grinder is a common machine tool that you will use almost daily in the machine shop. This grinding machine is used for general-purpose, off-hand grinding in which the workpiece is handheld and applied to the rapidly rotating abrasive wheel. One of the primary functions of the pedestal grinder is shaping and sharpening tool bits and drills in machine shop work. Pedestal grinders are often modified for use with rotary wire brushes or buffing wheels.

Large, heavy-duty pedestal grinders are sometimes found in machine shops. These grinders are used for rough grinding (snagging) welds, castings, and general rough work. These machines are generally set up in a separate location from the tool grinders. Rough grinding of metal parts should never be done on tool grinders because it causes the wheels to become rounded, grooved, uneven, and out-of-round. In that condition they are useless for tool grinding.

Setup of the Pedestal Grinder

The pedestal grinder in your shop stands ready for use most of the time. If it becomes necessary to replace a worn wheel, the side of the guard must be removed and the tool rest moved out of the way. A piece of wood may be used to prevent the wheel from rotating so that the spindle nut can be turned and removed (Figure B-156). Remember that the left side of the spindle has left-handed threads, and the right side has right-handed threads.

A new wheel should be ring tested to determine whether there are any cracks or imperfections (Figure B-157). Gently tap the wheel near its rim with a screwdriver handle or a piece of wood and listen for a clear ringing sound like that of a bell. A clear ring indicates a sound wheel that is safe to use; if a dull thud is heard, the wheel may be cracked and should not be used. Check that the flanges and the spindle are clean before mounting the wheel. Be sure that the center hole in the wheel is the correct size for the grinder spindle. If you must use a bushing, be sure that it is the correct size and installed properly (Figure B-158). Place a clean, undamaged

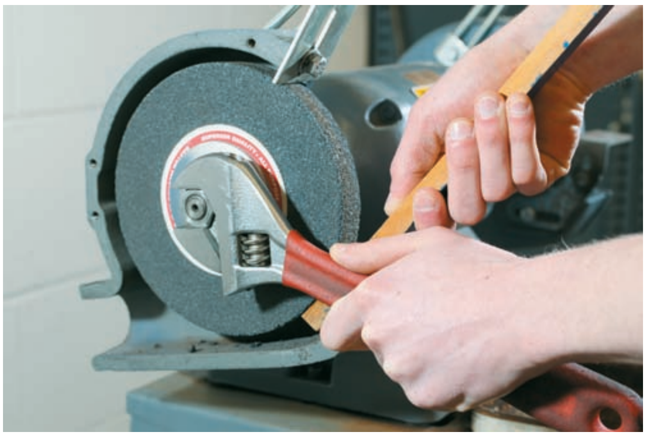

Figure B-156 Using a piece of wood to hold the wheel while removing the spindle nut.

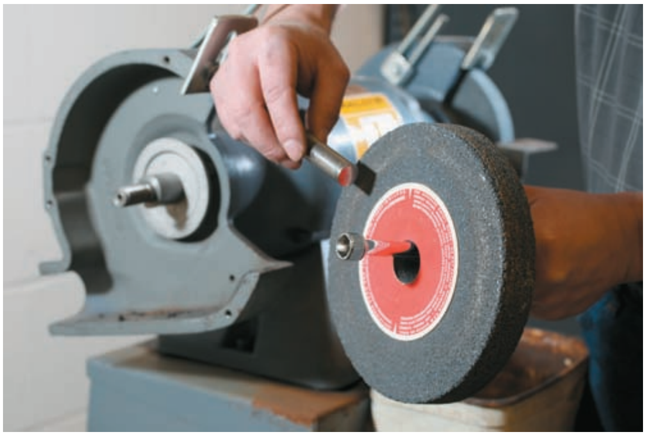

Figure B-157 The ring test is made before mounting the wheel.

Figure B-158 The wheel is mounted with the proper bushing in place.

blotter on each side between wheel and flanges. Tighten the spindle nut just enough to hold the wheel firmly. Excessive tightening will break the wheel.

After you have replaced the guard and cover plate, bring the tool rest up to the wheel so that there is $\frac{1}{16}$-to $\frac{1}{8}$-in. clearance between the rest and wheel (Figure B-159). If there is excessive space between the tool rest and the wheel, a small workpiece, such as a tool bit that is being ground, may flip up and catch between the wheel and tool rest.

CAUTION

Your finger may be caught between the workpiece and the grinding wheel, resulting in a serious injury, if the tool rest is not adjusted properly.

The clearance between the tool rest and wheel should never exceed $\frac{1}{8}$ in. The spark guard, located on the upper side of the wheel guard (Figure B-160), should be adjusted to within $\frac{1}{16}$ in. of the wheel. This protects the operator if the wheel should shatter.

Dressing the Grinding Wheel

Stand aside out of the line of the rotation of the grinding wheel and turn on the grinder. Let the wheel run idle for a full minute. A new wheel does not always run exactly true and therefore must be dressed. A Desmond dresser (Figure B-161) may be used to sharpen and to some extent true the face of the wheel. Pedestal grinder wheels often become grooved, out-of-round, glazed, or misshapen and therefore must be frequently dressed to obtain proper grinding results.

The grinding wheel dresser should be used so that the notch on the lower side is hooked behind the work rest. However, the dresser is often used in the manner shown in Figure B-161. When the tool rest extends along the side of the wheel, it is impossible to use the dresser properly.

Using the Pedestal Grinder

Bring the workpiece into contact with the wheel gently, without bumping. Grind only on the face of the wheel. The workpiece will heat from friction during the grinding operation. It may become too hot to hold in just a few seconds. To prevent

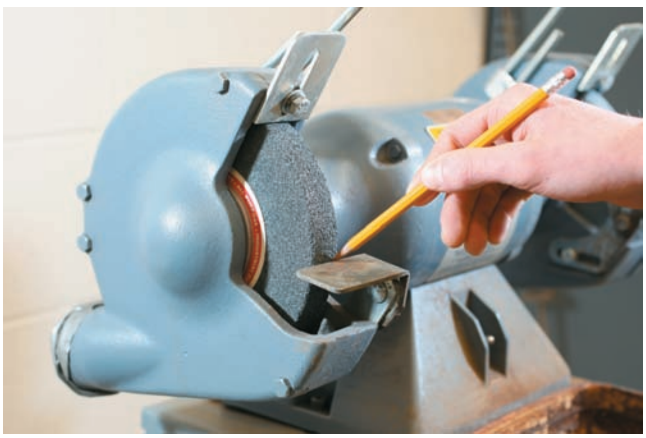

Figure B-159 The tool rest is adjusted.

Figure B-160 The spark guard is adjusted.

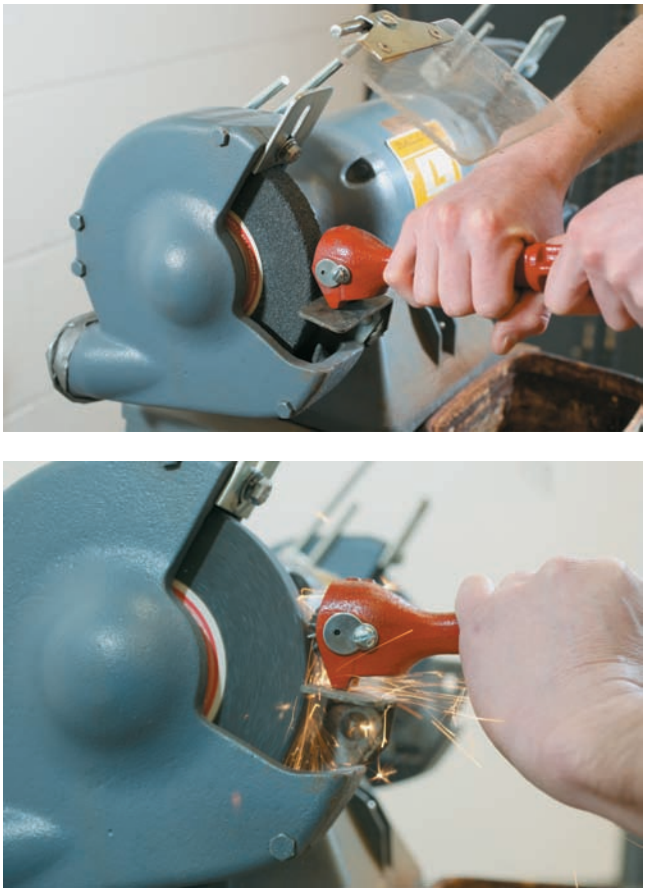

Figure B-161 The wheel is being dressed.

this, frequently cool the workpiece in the water pot attached to the grinder. Be especially careful when grinding drills and tool bits so that they do not become overheated. Excessive heat may permanently affect tool steel metallurgical properties.

Screwdrivers are probably the most misused of all tools, so they are often twisted or misshapen so much that they will no longer fit a screw slot or will damage the slot if they are used (Figure B-162). Screwdrivers can be ground flat on their sides, but a better method is to hollow grind them on a pedestal grinder (Figure B-163). The end of the tool should then be squared (Figure B-164) and given the proper thickness to fit the screw slot.

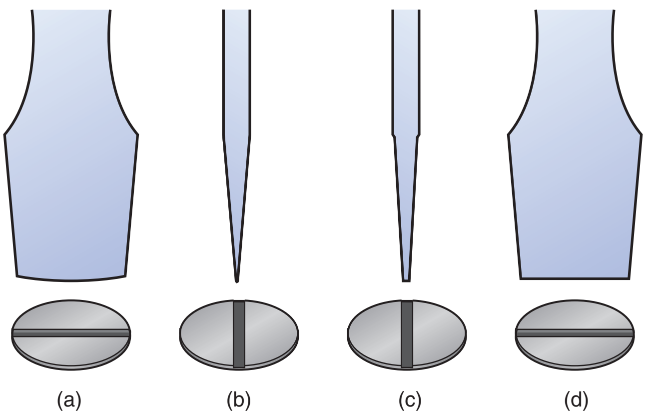

Figure B-162 The proper shape of the end of a screwdriver blade. Blades (*a*) and (*b*) are badly worn; blades (*c*) and (*d*) are ground correctly.

Center punches should also be hollow ground (Figure B-165) when they are sharpened. They should be evenly rotated while the point is ground to the correct angle. Flat cold chisels should also be hollow ground (Figure B-166) when they are sharpened.

Layout tools, such as spring dividers and scribers, should be kept sharp by honing on a fine, flat stone in the same way a knife is sharpened. However, when it becomes necessary to reshape a layout tool, exercise extreme care to avoid overheating the thin point. A fine-grit wheel should be used for sharpening most cutting tools, and they should be cooled frequently in water. Woodcutting tools are almost always made of plain carbon steel, which loses its hardness when overheated on the grinder. If one of these tools becomes blue-colored from the heat of grinding, it has become too soft for cutting purposes and must either be carefully ground back past the softened edge or rehardened and tempered.

Figure B-163 A screwdriver being hollow ground on the periphery of the wheel.

Figure B-164 The end of the blade being squared on the wheel.

Figure B-165 This punch is being correctly sharpened to produce a 90-degree angle for use as a center punch. It must be rotated while being ground. A sharper angle of 60 degrees may be ground to produce a prick punch.

Figure B-166 Flat chisel being ground to produce a 60-degree angle.

Safety Checkpoints on the Pedestal Grinder

Nonferrous metals such as aluminum and brass should never be ground on the aluminum oxide wheels found on most pedestal grinders. These metals fill the voids or spaces between the abrasive particles in the grinding wheel so that more pressure is needed to accomplish the desired grinding. This additional pressure sometimes causes the wheel to break or shatter. Pieces of grinding wheel may be thrown out of the machine at extreme velocities. Always use silicon carbide abrasive wheels for grinding nonferrous metals. Excessive pressure should never be used in any grinding operation. If this seems to be necessary, it means that the improper grit or grade of abrasive is being used, or the wheel is glazed and needs to be dressed. Always use the correct abrasive grit and grade for the particular grinding that you are doing. (For grinding wheel selection see Section L.)

Figure B-167 The force and speed of this action was such that the operator's head was jerked suddenly into the guard. Note that the cast aluminum guard was shattered as a result of the impact.

CAUTION

Always wear appropriate eye protection when dressing wheels or grinding on the pedestal grinder. Be sure that grinding wheels are rated at the proper speed for the grinder you are using. The safety shields, wheel guards, and spark guard must be kept in place at all times while grinding. The tool rest must be adjusted and the setting corrected as the diameter of the wheel decreases from use. Grinding wheels and rotary wire brushes may catch loose clothing or long hair (Figure B-167). Long hair should be contained in an industrial-type hairnet. Wire wheels often throw out small pieces of wire at high velocities.

SELF-TEST

1. What is the primary function of the pedestal grinder in a machine shop?
2. Why should a tool grinder never be used for rough grinding metal?

3. When a wheel needs to be reshaped, sharpened, and to some extent trued, what tool is usually used on a pedestal grinder?
4. When sharpening layout tools and reshaping screwdrivers, what is the most important concern?
5. Name at least three safety factors to remember when using the pedestal grinder.
6. How far from the wheel should the work rest be placed?
7. Why do you allow the grinder to run a bit just after changing a wheel?
8. What is the purpose of the wheel blotter?
9. What is the primary safety consideration when selecting a grinding wheel?
10. What does the wheel ring test do?

INTERNET REFERENCES

Information on off-hand grinding process and equipment

http://www.shef.ac.uk/safety

http://books.google.com