CHAPTER 6

ENGINE LATHES

CHAPTER LEARNING OBJECTIVES

Upon completing this chapter, you should be able to do the following:

- *Describe and explain the use of engine lathes.*
- *Explain engine lathe setup.*
- *Describe and explain engine lathe operations.*
- *Describe and explain methods of taper turning.*
- *Describe and explain cutting screw threads on an engine lathe.*

There are several types of lathes installed in shipboard machine shops, including the engine lathe, horizontal turret lathe, vertical turret lathe, and several variations of the basic engine lathe, such as bench and gap lathes. All lathes, except the vertical turret type, have one thing in common for all usual machining operations—the workpiece is held and rotated around a horizontal axis while being formed to size and shape by a cutting tool. In a vertical turret lathe, the workpiece is rotated around a vertical axis.

Horizontal lathes, as well as many of their attachments, and their operation are described in this chapter.

ENGINE LATHE

An engine lathe similar to the one shown in figure 6-1 is found in every machine shop. It is used mainly for turning, boring, facing, and screw cutting, but it may also be used for drilling, reaming, knurling, grinding, spinning, and spring winding. The work held in an engine lathe can be rotated at any one of a number of different speeds. The cutting tool can be accurately controlled by hand or power for longitudinal feed and cross-feed. (Longitudinal feed is the movement of the cutting tool parallel to the axis of the lathe; cross-feed is the movement of the cutting tool perpendicular to the axis of the lathe.)

Lathe size is determined by various methods, depending upon the manufacturer. Generally, the size is determined by two measurements: (1) either the diameter of work it will swing over the bed or the diameter of work it will swing over the cross-slide and (2) either the length of the bed or the maximum distance between centers. For example, a 14-inch by 6-foot lathe has a bed that is 6 feet long and will swing work (over the bed) up to 14 inches in diameter.

Engine lathes range in size from small bench lathes with a swing of 9 inches to very large lathes for turning work of large diameters, such as low-pressure turbine rotors. A 16-inch swing lathe is a good, average size for general purposes and is usually the size installed in ships that have only one lathe.

To learn the operation of a lathe, you must be familiar with the names and functions of the principal parts. In studying the principal parts in detail, remember that lathes all provide the same general functions even though the design may differ among manufacturers. As you read the description of each part, find its location on the lathe pictured in figure 6-1. For specific details on a given lathe, refer to the manufacturer's technical manual for that machine.

BED AND WAYS

The bed is the base for the working parts of the lathe. The main feature of the bed is the ways, which are formed on its upper surface and run the full length of the bed. The tailstock and carriage slide on the

Deleted—No permission granted for electronic copy.

Figure 6-1.—Gear-head engine lathe.

ways in alignment with the headstock. The headstock is permanently bolted to the end at the operator's left.

Figure 6-2 shows the ways of a typical lathe. The inset shows the inverted V-shaped ways (1, 3, and 4) and the flat way (2). The ways are accurately machined parallel to the axis of the spindle and to each other. The V-ways are guides that allow the carriage and tailstock to move over them only in their longitudinal direction. The flat way, number 2, takes

most of the downward thrust. The carriage slides on the outboard V-ways (1 and 4), which, because they are parallel to way number 3, keep the carriage aligned with the headstock and the tailstock at all times—an absolute necessity if accurate lathe work is to be done. Some lathe beds have two V-ways and two flat ways, while others have four V-ways.

For a lathe to perform satisfactorily, the ways must be kept in good condition. A common fault of careless machinists is to use the bed as an anvil for driving arbors or as a shelf for hammers, wrenches, and chucks. Never allow anything to strike a hard blow on the ways or damage their finished surfaces in any way. Keep them clean and free of chips. Wipe them off daily with an oiled rag to help preserve their polished surface.

HEADSTOCK

The headstock carries the headstock spindle and the mechanism for driving it. The headstock is similar to an automobile transmission except that it has more gear-shift combinations and therefore has a greater number of speed changes. On some lathes the gears are shifted hydraulically and others are manual. Our examples show the manual type of gear shifters. A speed index plate, attached to the headstock, shows the lever positions for the different spindle speeds.

Deleted—No permission
granted for electronic copy.

Figure 6-2.—Rear view of lathe.

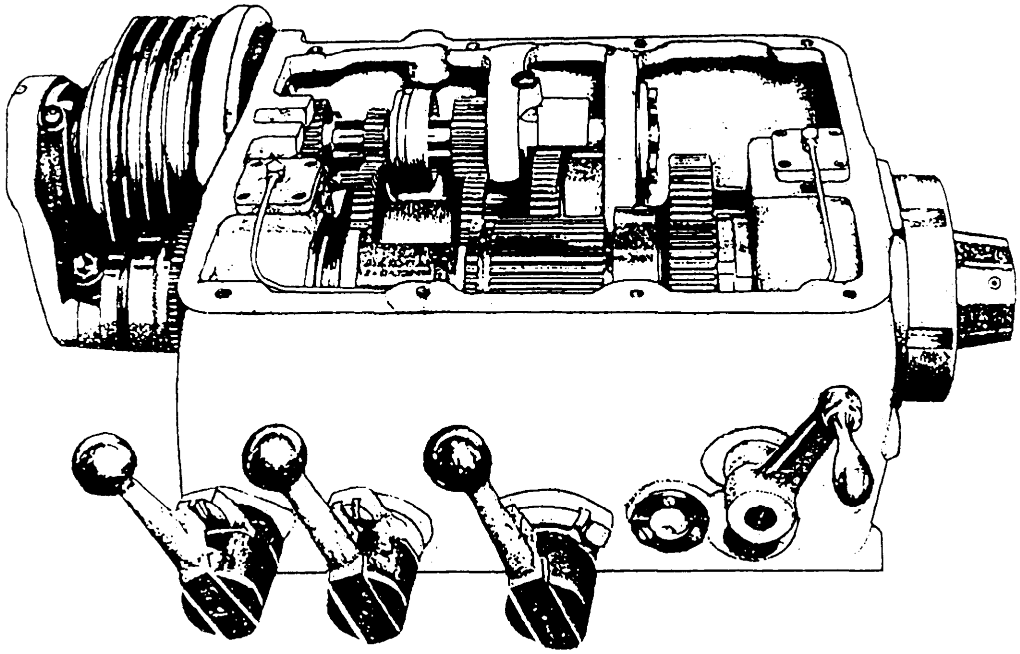


Figure 6-3.—Sliding gear-type headstock

Figure 6-4 shows this plate for the geared headstock in figure 6-3. Always stop the lathe when you shift gears to avoid damaging the gear teeth.

Figure 6-3 shows the interior of a typical geared headstock that has 16 different spindle speeds. The driving pulley at the left is driven at a constant speed by a motor located under the headstock. Various combinations of gears in the headstock transmit the power from the drive shaft to the spindle through an intermediate shaft. Use the speed-change levers to shift the sliding gears on the drive and intermediate shafts to line up the gears in different combinations. This produces the gear ratios you need to obtain the various spindle speeds. Note that the back gear lever has high and low speed positions for each combination of the other gears (fig. 6-4).

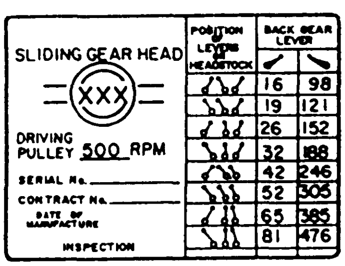


Figure 6-4.—Speed index plate.

The headstock casing is filled with oil to lubricate the gears and the shifting mechanism it contains. Parts not immersed in the oil are lubricated by either the splash produced by the revolving gears or by an oil pump. Be sure to keep the oil to the oil level indicated on the oil gauge.

The headstock spindle (fig. 6-5) is the main rotating element of the lathe and is directly connected to the work, which revolves with it. The spindle is supported in bearings at each end of the headstock through which it projects. The section of the spindle between the bearings carries the pulleys or gears that

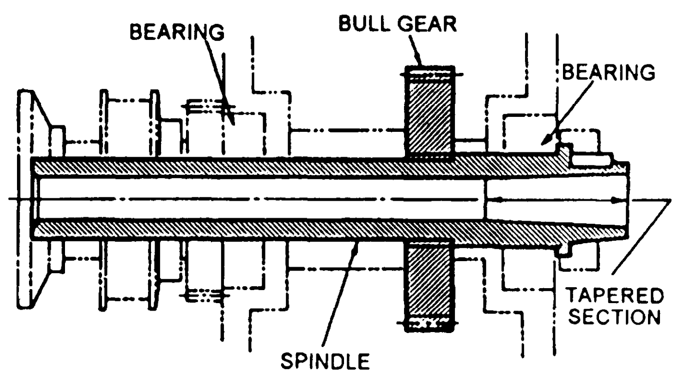


Figure 6-5.—Cross section of a headstock spindle.

turn the spindle. The nose of the spindle holds the driving plate, the faceplate, or a chuck. The spindle is hollow throughout its length so that bars or rods can be passed through it from the left and held in a chuck at the nose. The chuck end of the spindle is bored to a Morse taper to receive the live center. The hollow spindle also permits the use of the draw-in collet chuck, which is discussed later in this chapter. At the other end of the spindle is the gear by which the spindle drives the feed and screw-cutting mechanism through a gear train located on the left end of the lathe.

TAILSTOCK

The tailstock (fig. 6-6) may be used to hold the dead or ball bearing center or it can be used to hold tapered shank drills, reamers, and drill chucks. The tailstock moves on the ways along the length of the bed to accommodate work of varying lengths. It can be clamped in the desired position by the tailstock clamping nut (13).

The dead center (11) is held in a tapered hole (bored to a Morse taper) in the tailstock spindle (6). To move the spindle back and forth in the tailstock barrel for longitudinal adjustment, turn the handwheel (9) that turns the spindle-adjusting screw (7) in a tapped hole in the spindle at (8). The spindle is kept from revolving by a key (4) that fits a spline, or keyway (5), cut along the bottom of the spindle as shown. After making the final adjustment, use the binding clamp (10) to lock the spindle in place.

The tailstock body is made in two parts. The bottom, or base (1), is fitted to the ways; the top (2) can move laterally on its base. The lateral movement can be closely adjusted by setscrews. Zero marks inscribed on the base and top indicate the center position and provide a way to measure setover for taper turning. Setover of the tailstock for taper turning is described in a later chapter.

Deleted—No permission granted for electronic copy.

Figure 6-6.—Cross section of a tailstock.

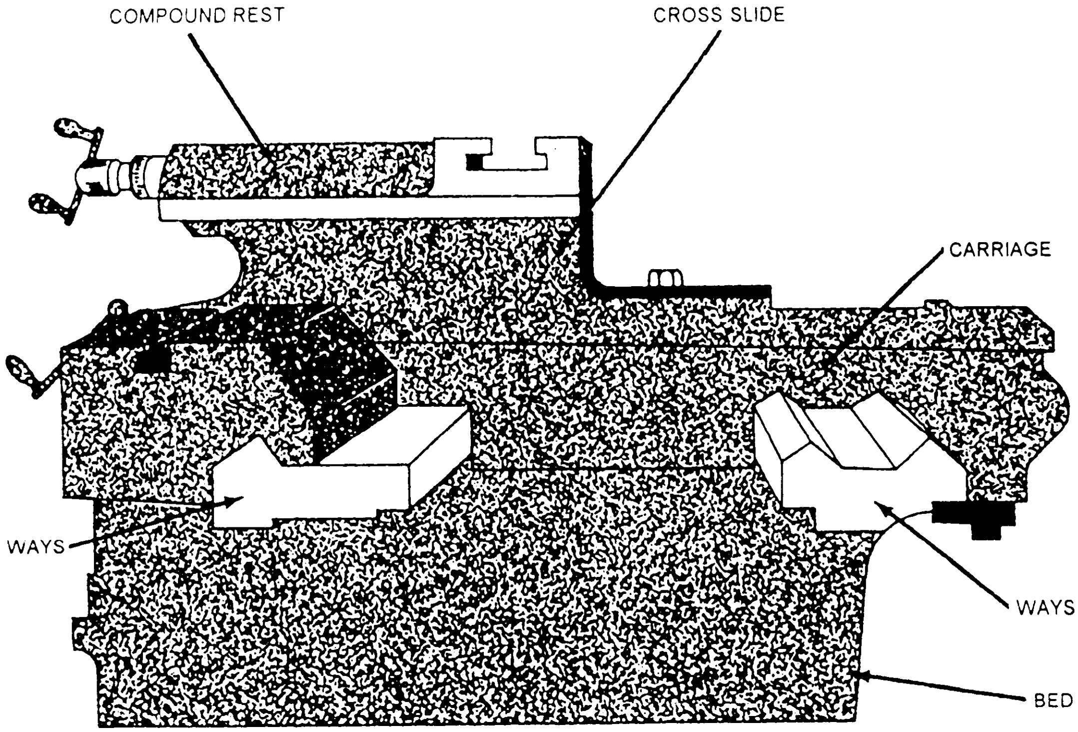


Figure 6-7.—Side view of a carriage mounted on the bed.

Before you insert a center or tooling into the spindle, carefully clean the tapered shank and wipe out the tapered hole of the spindle. After you put a drill or a reamer into the tapered hole of the spindle, be sure to tighten it in the spindle so that the tool will not revolve. If the drill or reamer is allowed to revolve, it will score the tapered hole and destroy its accuracy. The spindle of the tailstock is engraved with graduations that help in determining the depth of a cut when you drill or ream.

CARRIAGE

The carriage carries the cross-feed slide and the compound rest that, in turn, carries the cutting tool in the toolpost. The carriage slides on the ways along the bed (fig. 6-7).

Figure 6-8 shows a top view of the carriage. The wings of the H-shaped saddle contain the bearing surfaces, which are fitted to the V-ways of the bed. The crosspiece is machined to form a dovetail for the cross-feed slide. The cross-feed slide is closely fitted to the dovetail and has a tapered gib that fits between the carriage dovetail and the matching dovetail of the cross-feed slide. The gib permits small adjustments to remove any looseness between the two parts. The

Deleted—No permission granted for electronic copy.

Figure 6-8.—Carriage (top view).

slide is securely bolted to the cross-feed nut that moves back and forth when the cross-feed screw is turned by the handle. The micrometer dial on the cross-feed handle is graduated to permit accurate infeed. Depending on the manufacturer of the lathe, the dial may be graduated so that each division represents a 1 to 1 or a 2 to 1 ratio. The compound rest is mounted on top of the cross-feed slide.

The carriage has T-slots or tapped holes for clamping work for boring or milling. When the lathe is used in this manner, the carriage movement feeds the work to the cutting tool, which is revolved by the headstock spindle.

You can lock the carriage in any position on the bed by tightening the carriage clamp screw. Use the clamp screw only when doing such work as facing or cutting-off, for which longitudinal feed is not required. Normally, keep the carriage clamp in the released position. Always move the carriage by hand to be sure it is free before you apply the automatic feed.

APRON

The apron is attached to the front of the carriage. It contains the mechanism that controls the movement of the carriage for longitudinal feed and thread cutting and controls the lateral movement of the cross-slide. You should thoroughly understand the construction and operation of the apron before you attempt to operate the lathe.

In general, a lathe apron contains the following mechanical parts:

- A longitudinal feed handwheel for moving the carriage by hand along the bed. This handwheel turns a pinion that meshes with a rack gear secured to the lathe bed.
- Gear trains driven by the feed rod. These gear trains transmit power from the feed rod to move the carriage along the ways and to move the cross-slide across the ways, thus providing powered longitudinal feed and cross-feed.
- Friction clutches operated by knobs on the apron to engage or disengage the power-feed mechanism. (Some lathes have a separate clutch for longitudinal feed and cross-feed; others have a single clutch for both.) (**NOTE:** The power feeds are usually driven through a friction clutch to prevent damage to the gears if excessive strain is put on the feed mechanism. If clutches are not provided, there is some form of safety device that operates to disconnect the feed rod from its driving mechanism.)
- A selective feed lever or knob for engaging the longitudinal feed or cross-feed as desired.
- Half-nuts that engage and disengage the lead screw when the lathe is used to cut threads. They are opened or closed by a lever located on the right side of the apron. The half-nuts fit the thread of the lead screw, which turns in them like a bolt in a nut when they are clamped over it. The carriage is then moved by the thread of the lead screw instead of by the gears of the apron feed mechanisms. (The half-nuts are engaged only when the lathe is used to cut threads, at which time the feed mechanism must be disengaged. An interlocking device that prevents the half-nuts and the feed mechanism from engaging at the same time is usually provided as a safety feature.)

Aprons on lathes made by different manufacturers differ somewhat in construction and in the location of controlling levers and knobs. But, they all are designed to perform the same functions. The principal difference is in the arrangement of the gear trains for driving the automatic feeds. For example, in some aprons there are two separate gear trains with separate operating levers for longitudinal feed and cross feed. In others, both feeds are driven from the same driving gear on the feed rod through a common clutch, with one feed at a time connected to the drive by a selector lever. The apron shown in figure 6-9 is of the latter type.

Deleted—No permission granted for electronic copy.

Figure 6-9.—Rear view of a lathe apron.

Deleted—No permission granted for electronic copy.

Figure 6-10.—Quick-change gear box (rear view).

FEED ROD

The feed rod transmits power to the apron to drive the longitudinal feed and cross feed mechanisms. The feed rod is driven by the spindle through a train of gears, and the ratio of its speed to that of the spindle can be varied by changing gears to produce various rates of feed The rotating feed rod drives gears in the apron. These gears in turn drive the longitudinal feed and cross-feed mechanisms through friction clutches, as explained in the description of the apron.

Lathes that do not have a separate feed rod have a spline in the lead screw to serve the same purpose. The apron shown in figure 6-9 belongs to a lathe of this type and shows clearly how the worm that drives the feed mechanism is driven by the spline in the lead screw. If a separate feed rod were used, it would drive the feed worm in the same manner, that is, by means of a spline. The spline permits the worm, which is keyed to it, to slide freely along its length to conform with the movement of the carriage apron.

LEAD SCREW

The lead screw is used for thread cutting. Along its length are accurately cut Acme threads that engage the threads of the half-nuts in the apron when half-nuts are clamped over it. When the lead screw turns in the closed half-nuts, the carriage moves along the ways a distance equal to the lead of the thread in each revolution of the lead screw. Since the lead screw is connected to the spindle through a gear train (discussed later in the section on quick-change gear mechanism), the lead screw rotates with the spindle. Therefore, whenever the half-nuts are engaged, the longitudinal movement of the carriage is directly controlled by the spindle rotation. The cutting tool is moved a definite distance along the work for each revolution that the spindle makes.

The ratio of the threads per inch of the thread being cut and the thread of the lead screw is the same as the ratio of the speeds of the spindle and the lead screw. For example: If the lead screw and spindle turn at the same speed, the number of threads per inch being cut is the same as the number of threads per inch of the lead screw. If the spindle turns twice as fast as the lead screw, the number of threads being cut is twice the number of threads per inch of the lead screw.

You can cut any number of threads by merely changing gears in the connecting gear train to get the desired ratio of spindle and lead screw speeds.

Deleted—No permission granted for electronic copy.

Figure 6-11.—Quick-change gear box.

QUICK-CHANGE GEAR MECHANISM

To do away with the inconvenience and loss of time involved in removing and replacing change gears, most modern lathes have a self-contained change gear mechanism, commonly called the quick-change gear box. There are a number of types used on different lathes but they are all similar in principle.

The mechanism consists of a cone-shaped group of change gears. You can instantly connect any single gear to the gear train by moving a sliding tumbler gear controlled by a lever. The cone of gears is keyed to a shaft that drives the lead screw (or feed rod) directly or through an intermediate shaft. Each gear in the cluster has a different number of teeth and hence produces a different gear ratio when connected in the tram. The same thing happens as when the screw gear in the gear train is changed, described previously. Sliding gears also produce other changes in the gear tram to increase the number of different ratios you can get with the cone of change gears just described. All changes are made by shifting appropriate levers or knobs. An index plate or chart mounted on the gear box indicates the position for placing the levers to get the necessary gear ratio to cut the thread or produce the feed desired.

Figure 6-10 is the rear view of one type of gear box, showing the arrangement of gears. Splined shaft F turns with gear G, which is driven by the spindle through the main gear train on the end of the lathe. Shaft F in turn drives shaft H through tumbler gear T, which can be engaged with any one of the cluster of eight different size gears on shaft H by means of lever C. Shaft H drives shaft J through a double clutch gear, which takes the drive through one of three gears, depending on the position of lever B (right, center, or left). Shaft J drives the lead screw through gear L.

Either the lead screw or the feed rod can be connected to the final drive shaft of the gear box by engaging appropriate gears.

Twenty-four different gear ratios are provided by the quick-change gear box shown in figure 6-11. The

lower lever has eight positions, each of which places a different gear in the gear train and hence produces eight different gear ratios. The three positions of the upper level produce three different gear ratios for each of the 8 changes obtained with the lower lever, thus making 24 combinations in the box alone. You can double this range by using a sliding compound gear that provides a high- and low-gear ratio in the main gear train. This gives two ratios for every combination obtainable in the box, or 48 combinations in all.

The index chart on the gear box also shows the various rates of power longitudinal feed per spindle revolution that you can get by using the feed mechanism of the apron. For example, in figure 6-11, note that the finest longitudinal feed is 0.0030 inch per revolution of spindle, the next finest is 0.0032 inch, and so on. To arrange the gear box for power longitudinal feed, select the feed you wish to use and follow the same procedure explained for cutting screw threads, except that you engage the power feed lever instead of the half-nuts. Cross-feeds are not listed on the chart but you can determine them by multiplying the longitudinal feeds by 0.375, as noted on the index plate.

On a lathe with a separate feed rod, a feed-thread shifting lever located at the gear box connects the drive to the feed rod or the lead screw as desired. When the feed rod is engaged, the lead screw is disengaged and vice versa.

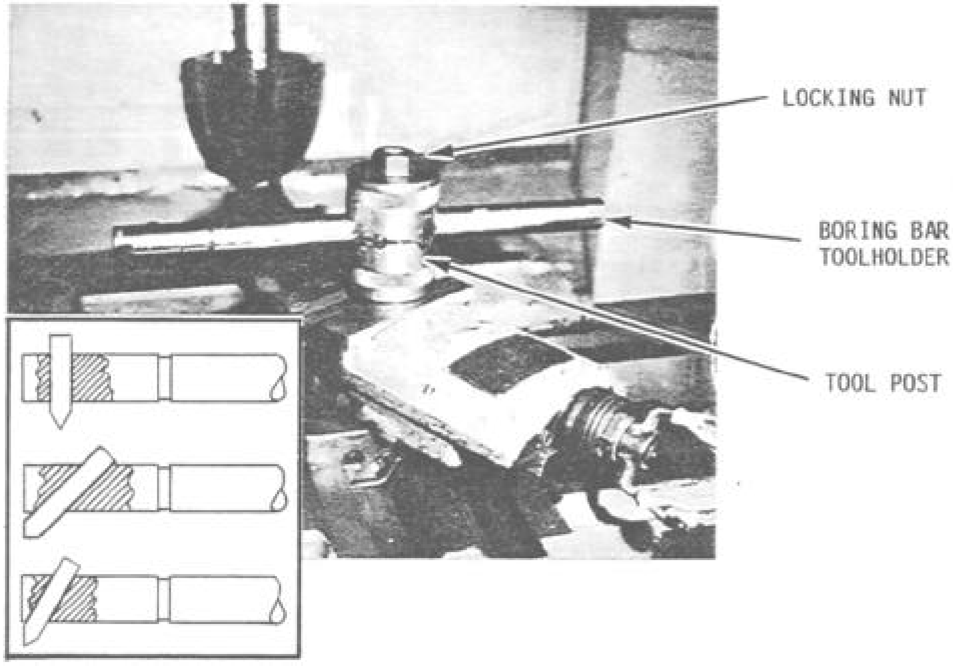
You can cut metric threads on some lathes that have an English lead screw (threads per inch machined on it) by transposing a set of gears in the lead screw to the spindle gear train that provides the correct conversion ratio. You can find information on this in handbooks for machinists, in the equipment technical manual, and through direct correspondence with the equipment manufacturer.

COMPOUND REST

The compound rest provides a rigid, adjustable mounting for the cutting tool. The compound rest assembly has the following principal parts (fig. 6-12):

Deleted—No permission granted for electronic copy.

Figure 6-12.-Compound rest.



28.299

Figure 6-13.—Castle-type toolpost and toolholder.

- The compound rest swivel (2), which can be swung around to any desired angle and clamped in position. It is graduated over an arc of 90° on each side of its center position for ease in setting to the angle you select. This feature is used in machining short, steep tapers such as the angle on bevel gears, valve disks, and lathe centers.

- The compound rest top, or topslide (3), is mounted as shown on the swivel section (2) on a dovetailed slide. It is moved along the slide by the compound rest feed screw turning in the nut (4), operated by the handle (5), in a manner similar to the cross feed described previously (fig. 6-8). This provides for feeding at any angle (determined by the angular setting of the swivel section), while the cross-slide feed provides only for feeding at a right angle to the axis of the lathe. The graduated collar on the compound rest feed screw reads in thousandths of an inch for fine adjustment in regulating the depth of cut.

ATTACHMENTS AND ACCESSORIES

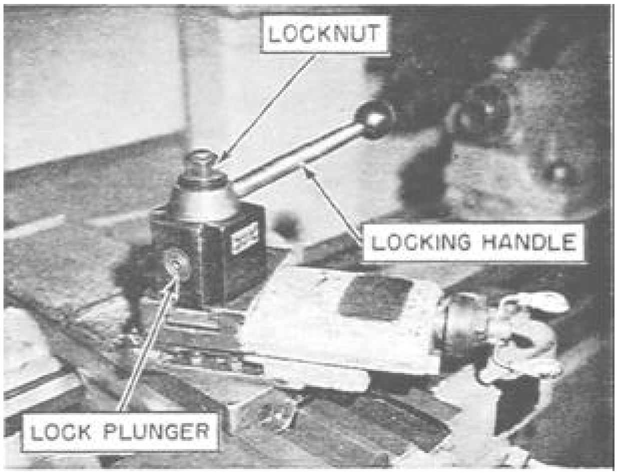
Accessories are the tools and equipment used in routine lathe machining operations. Attachments are special fixtures that may be secured to the lathe to extend the versatility of the lathe to include taper-cutting, milling, and grinding. Some of the common accessories and attachments used on lathes are described in the following paragraphs.

TOOLPOSTS

Three popular types—standard, castle, and quick change—are discussed in the following paragraphs. The sole purpose of the toolpost is to provide a rigid support for the toolholder.

The standard toolpost is mounted in the T-slot of the compound rest top as shown in figure 6-12. A toolholder (13) is inserted in the slot in the toolpost and rests on the toolpost wedge (11) and the toolpost ring (12). By tightening the setscrew (10), you clamp the whole unit firmly in place with the tool in the desired position.

The castle-type toolpost (fig. 6-13) is used with boring bar-type toolholders. It mounts in the T-slot and the toolholder (boring bar) passes through it and the holddown bolt. By tightening the locking nut, you clamp the entire unit firmly in place. Various size holes through the toolpost allow the use of assorted diameter boring bars.



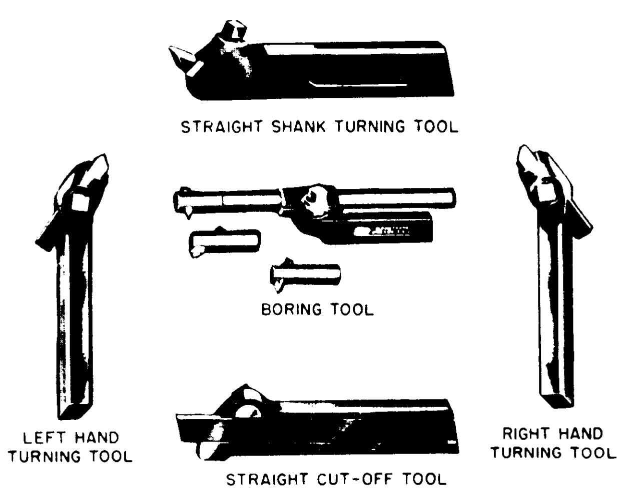
28.300

Figure 6-14.—Quick-change toolpost.

The quick-change toolpost (fig. 6-14) is available in most Navy machine shops. It mounts in the T-slots and is tightened in place by the locknut, which clamps the toolpost firmly in place. Special-type toolholders are used in conjunction with this type of toolpost and are held in place by a locking plunger, which is operated by the toolholder locking handle. Some toolposts have a sliding gib to lock the toolholder. With this type of toolpost, only the toolholders are changed, allowing the toolpost to remain firmly in place.

TOOLHOLDERS

Lathe toolholders are designed to be used with the various types of toolposts. Only the three most



28.67

Figure 6-15.—Standard lathe toolholders.

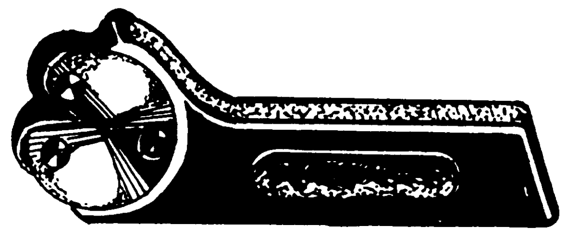


Figure 6-16.—Knurling tool.

commonly used types-standard, boring bar, and quick change-are discussed in this chapter. The toolholder holds the cutting tool (tool bit) in a rigid and stable position. Toolholders are generally made of a softer material than the cutting tool. They are large in size and help to carry the heat generated by the cutting action away from the point of the cutting tool.

Standard toolholders were discussed briefly in chapter 5 of this manual. However, there are more types (figs. 6-15 and 6-16) than those discussed in chapter 5.

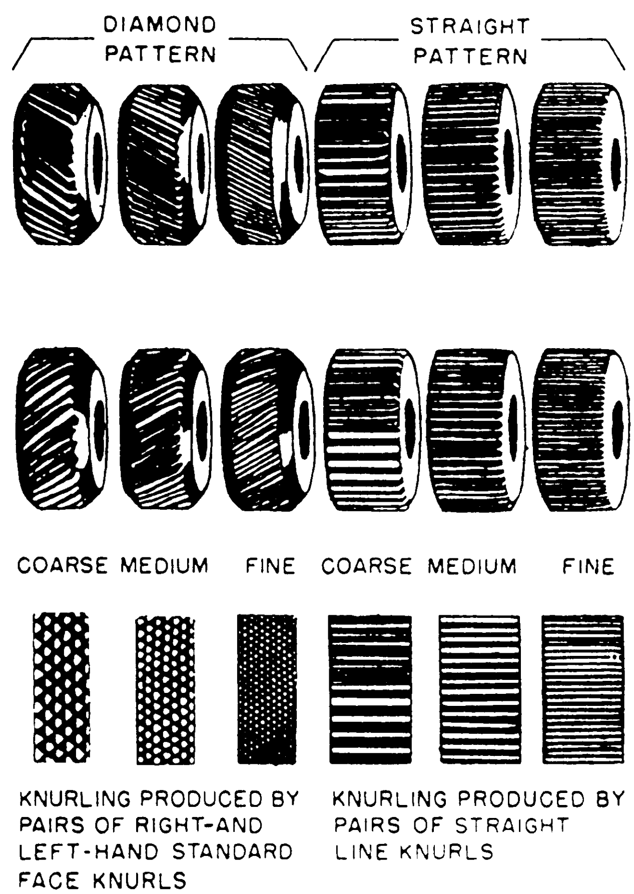


Figure 6-17.—Types of knurling rollers.

A knurling tool (fig. 6-16) forms a pattern on the work by being fed into the work as it revolves. The purpose of knurling is to give a roughened surface on round metal parts, like knobs, to give a better grip for handling. The knurled roller comes in a wide variety of patterns. (See fig. 6-17.)

The boring bar toolholder is nothing more than a piece of round stock with a screw-on cap (fig. 6-13). The caps are available with square holes broached through them at various angles and sizes. When the proper size tool bit is inserted into the cap and the cap is screwed on to the threaded end of the piece of round stock, the entire unit becomes a very rigid boring tool, which is used with the castle-type toolpost.

The quick-change toolholder, which is the most widely used toolholder (fig. 6-18), is mounted on the toolpost by sliding it from above and downward over the dovetails. This toolholdcr has a height adjusting ring to allow you to set the proper height before locking it in place. The quick-change toolholder comes in a wide range of styles. A few of these styles are shown in figure 6-19.

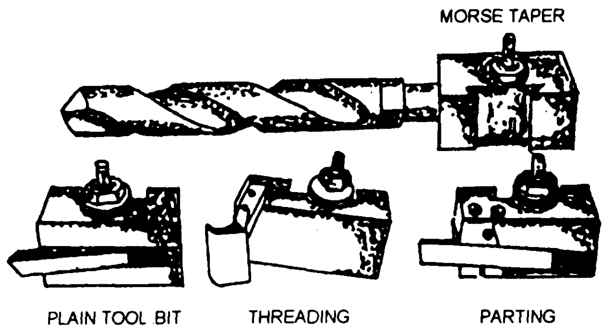


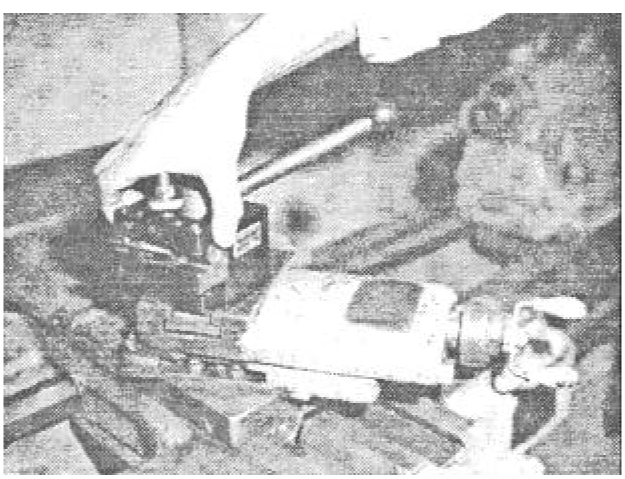
Figure 6-19.—Quick-change toolholder.

LATHE CHUCKS

The lathe chuck is a device for holding lathe work. It is mounted on the nose of the spindle. The work is held by jaws that can be moved in radial slots toward the center to clamp down on the sides of the work. These jaws are moved in and out by screws turned by a chuck wrench applied to the sockets located at the outer ends of the slots.

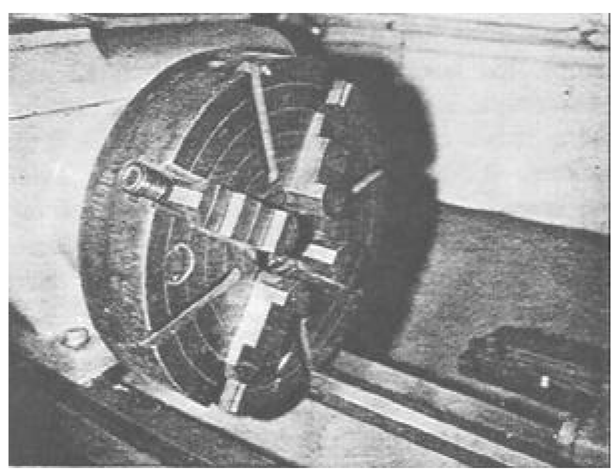
The four-jaw independent lathe chuck, figure 6-20, is the most practical for general work. The four jaws are adjusted one at a time, making it possible to hold work of various shapes and to adjust the center of the work to coincide with the axial center of the spindle.

There are several different styles of jaws for four-jaw chucks. You can remove some of the chuck jaws by turning the adjusting screw and then re-inserting them in the opposite direction. Some chucks have two sets of jaws, one set being the reverse of the other. Another style has jaws that are bolted onto a slide by two socket-head bolts. On this style you can reverse the jaws by removing the bolts, reversing the jaws, and re-inserting the bolts. You can make special jaws for this style chuck in the shop and machine them to fit a particular size outside or inside diameter.



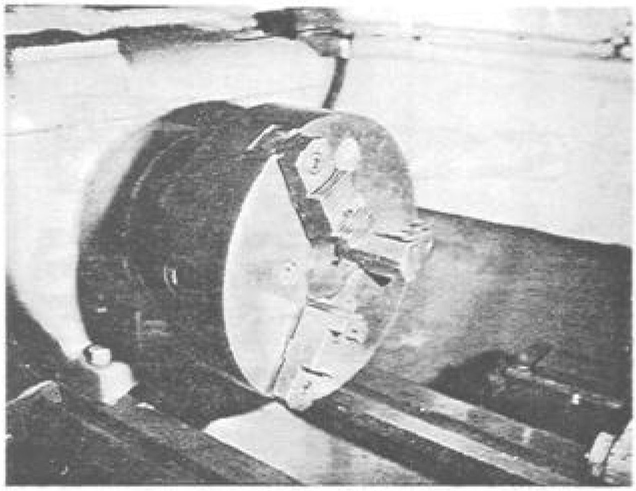
28.302

Figure 6-18.—Quick-change toolpost and toolholder.



28.304

Figure 6-20.—Four-jaw independent chuck.



28.305

Figure 6-21.—Three-jaw universal chuck.

The three-jaw universal or scroll chuck (fig. 6-21) can be used only for holding round or hexagonal work. All three jaws move in and out together in one operation. They move simultaneously to bring the work on center automatically. This chuck is easier to operate than the four-jaw type, but when its parts become worn you cannot rely on its accuracy in centering. Proper lubrication and constant care in use are necessary to ensure reliability. The same styles of jaws available for the four-jaw chuck are also available for the three-jaw chuck.

Combination chucks are universal chucks that have independent movement of each jaw in addition to the universal movement.

Figure 6-3 shows the usual means provided for attaching chucks and faceplates to a lathe. The tapered nose spindle (fig. 6-3) is usually found on lathes that have a swing greater than 12 inches. Matching internal tapers and keyways in chucks for these lathes ensure accurate alignment and radial locking. A free-turning, internally threaded collar on the spindle screws onto a boss on the back of the chuck to secure the chuck to the spindle nose. On small lathes, chucks are screwed directly onto the threaded spindle nose.

The draw-in collet chuck is used to hold small work for machining. It is the most accurate type of chuck and is intended for precision work.

Figure 6-22 shows the five parts of the collet chuck assembled in place in the lathe spindle. The collet, which holds the work, is a split cylinder with an outside taper that fits into the tapered closing sleeve and screws into the threaded end of the'hollow drawbar that passes through the hollow spindle. When the handwheel, which is attached by threads to the outside of the drawbar, is turned clockwise, the drawbar pulls the collet into the tapered sleeve, thereby decreasing the diameter of the hole in the collet. As the collet is closed around the work, the work is centered accurately and is held firmly by the chuck.

Collets are made with hole sizes ranging from 1/64 inch up, in MS-inch steps. The best results are obtained when the diameter of the work is exactly the same size as the dimension stamped on the collet.

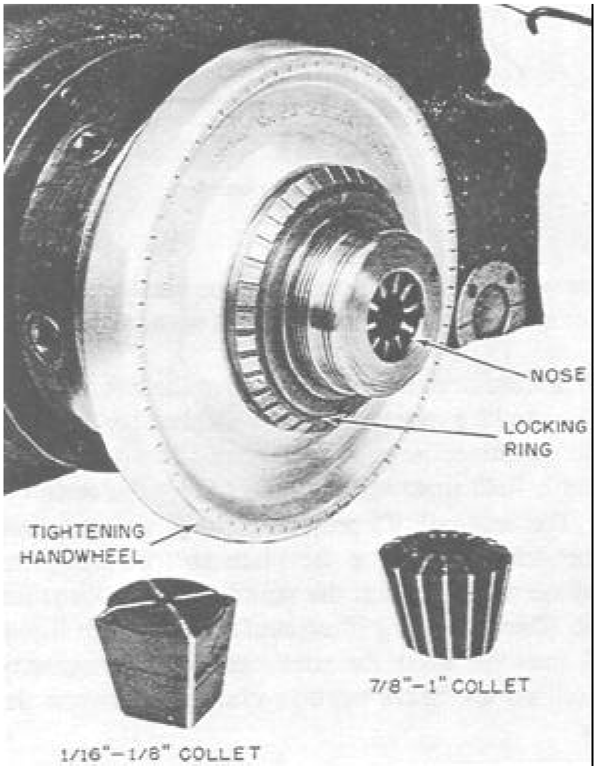
To ensure accuracy of the work when using the draw-in collet chuck, be sure that the contact surfaces of the collet and the closing sleeve are free of chips and dirt. (**NOTE:** The standard collet has a round hole, but special collets for square and hexagonal shapes are available.)

The rubber collet chuck (fig. 6-23) is designed to hold any bar stock from 1/16 inch up to 1 3/8 inch. It is different from the draw-in type of collet previously mentioned in that tht bar stock does not have to be exact in size.

The rubber flex collet consists of rubber and hardened steel plates. The nose of the chuck has external threads, and, by rotating the handwheel (fig. 6-23), you compress the collet around the bar. This exerts equal pressure from all sides and enables you to align the stock very accurately. The locking ring, when pressed in, gives a safe lock that prevents the collet from coming loose when the machine is in operation.

Deleted—No permission granted for electronic copy.

Figure 6-22.—Draw-in collet chuck assembled.



28.306
Figure 6-23.—Rubber flex collet chuck.

Deleted—No permission granted for electronic copy.

Figure 6-24.—Drill chuck.

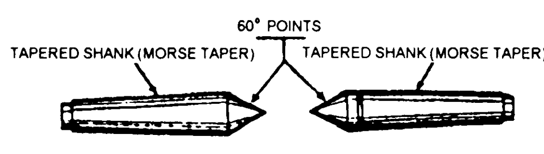
Drill chucks are used to hold center drills, straight shank drills, reamers, taps, and small rods. The drill chuck is mounted on a tapered shank or arbor that fits the Morse taper hole in either the headstock or tailstock spindle. Figure 6-24 shows the three-jaw type. A revolving sleeve operated by a key opens or closes the three jaws simultaneously to clamp and center the drill in the chuck.

Faceplates are used for holding work that cannot be swung on centers or in a chuck because of its shape or dimensions. The T-slots and other openings on the surface of the faceplate provide convenient anchor points for bolts and clamps used to secure the work to the faceplate. The faceplate is mounted on the nose of the spindle.

The driving plate is similar to a small faceplate and is used primarily for driving work that is held between centers. A radial slot receives the bent tail of a lathe dog clamped to the work to transmit rotary motion to the work.

LATHE CENTERS

The lathe centers shown in figure 6-25 provide a means for holding the work between points so it can be turned accurately on its axis. The headstock spindle center is called the **LIVE** center because it revolves with the work. The tailstock center is called the **DEAD** center because it does not turn. Both live and dead centers have shanks turned to a Morse taper to fit the tapered holes in the spindles; both have points finished to an angle of 60°. They differ only in that the dead center is hardened and tempered to resist the wearing effect of the work revolving on it. The live center revolves with the work and is usually left soft. The dead center and live center must **NEVER**



28.93
Figure 6-25.—Lathe centers.

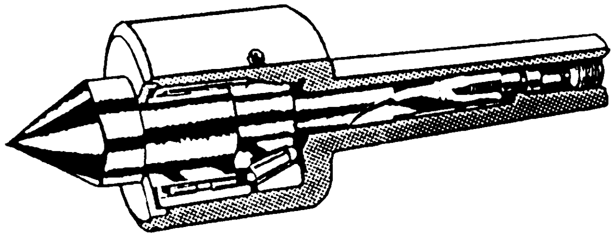


Figure 6-26.—Cutaway showing the construction of a hall bearing center.

be interchanged. A dead center requires a lubricant between it and the center hole to prevent seizing and burning of the center. (**NOTE:** There is a groove around the hardened tail center to distinguish it from the live center.)

The centers fit snugly in the tapered holes of the headstock and tailstock spindles. If chips, dirt, or burrs prevent a perfect fit in the spindles, the centers will not run true.

To remove the headstock center, insert a brass rod through the spindle hole and tap the center to jar it loose; you can then pull it out with your hand. To remove the tailstock center, run the spindle back as far as it will go by, turning the handwheel to the left. When the end of the tailstock screw bumps the back of the center, it will force the center out of the tapered hole. (See fig. 6-6.)

The ball bearing center shown in figure 6-26 is the most commonly used center for working between centers. As your work expands from the heat generated from machining, you should occasionally feel the area that houses the ball bearings to make sure they are not overheating. If the bearings become too warm, you will need to decrease the pressure of the center against the work.

For machining hollow cylinders, such as pipe, use a bull-nosed center called a pipe center. Figure 6-27 shows its construction. The taper shank (A) fits into the head and tail spindles in the same manner as the lathe centers. The conical disk (B) revolves freely on the collared end. Different size disks are supplied to accommodate various ranges of pipe sizes.

LATHE DOGS

Lathe dogs are used with a driving plate or faceplate to drive work being machined on centers

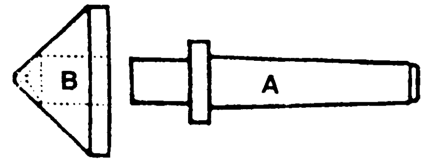


Figure 6-27.—Pipe center.

whenever the frictional contact alone between the live center and the work is not sufficient to drive the work.

The common lathe dog shown at the left in figure 6-28 is used for round work or work that has a regular section (square, hexagon, octagon). The piece to be turned is held firmly in the hole (A) by the setscrew (B). The bent tail (C) projects through a slot or hole in the driving plate or faceplate so that when the faceplate revolves with the spindle, it also turns the work. The clamp dog illustrated at the right in figure 6-28 may be used for rectangular or irregularly shaped work. Such work is clamped between the jaws.

CENTER REST

The center rest, also called the steady rest, is used for the following purposes:

Deleted—No permission granted for electronic copy.

Figure 6-28.—Lathe dogs.

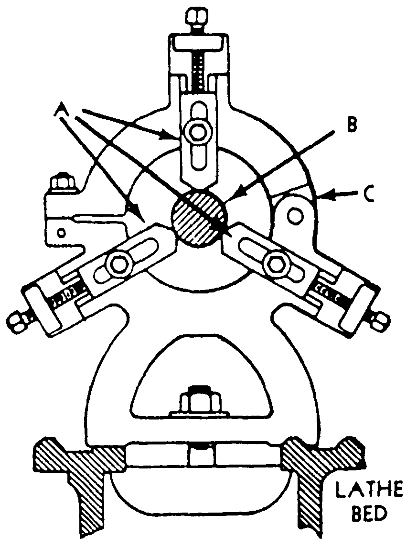


Figure 6-29.—Center rest.

- To provide an intermediate support or rest for long slender bars or shafts being machined between centers. It prevents them from springing due to cutting pressure or sagging as a result of their otherwise unsupported weight.

- To support and provide a center bearing for one end of work, such as a spindle, being bored or drilled from the end when it is too long to be supported by the chuck alone. The center rest, kept aligned by the ways, can be clamped at any desired position along the bed, as illustrated in figure 6-29. It is important that the jaws (A) be carefully adjusted to allow the work (B) to turn freely and at the same time keep it accurately centered on the axis of the lathe. The top half of the frame is hinged at C to make it easier to place the center rest in position without removing the work from the centers or changing the position of the jaws.

FOLLOWER REST

The follower rest is used to back up work of small diameter to keep it from springing under the pressure of cutting. This rest gets its name because it follows the cutting tool along the work As shown in figure 6-30, it is attached directly to the saddle by bolts (B). The adjustable jaws bear directly on the finished diameter of the work opposite and above the cutting tool.

TAPER ATTACHMENT

The taper attachment, illustrated in figure 6-31, is used for turning and boring tapers. It is bolted to the back of the carriage saddle. In operation, it is connected to the cross-slide so that it moves the cross slide laterally as the carriage moves longitudinally, thereby causing the cutting tool to move at an angle to the axis of the work to produce a taper.

Deleted—No permission granted for electronic copy.

Figure 6-30.—Follower rest.

Deleted—No permission granted for electronic copy.

Figure 6-31.—A taper attachment.

Deleted—No permission granted for electronic copy.

Figure 6-32.—Thread dial indicator.

The angle of the desired taper is set on the guide bar of the attachment, and the guide bar support is clamped to the lathe bed.

Since the cross-slide is connected to a shoe that slides on the guide bar, the tool follows along a line that is parallel to the guide bar and hence at an angle to the work axis corresponding to the desired taper.

The operation and application of the taper attachment will be explained further later in this chapter.

THREAD DIAL INDICATOR

The thread dial indicator, shown in figure 6-32, lets you quickly return the carriage to the beginning of the thread to set up successive cuts. This eliminates the necessity of reversing the lathe and waiting for the carriage to follow the thread back to its beginning. The dial, which is geared to the lead screw, indicates when to clamp the half-nuts on the lead screw for the next cut.

The threading dial consists of a worm wheel that is attached to the lower end of a shaft and meshed with the lead screw. The dial is located on the upper end of the shaft. As the lead screw revolves, the dial turns. The graduations on the dial indicate points at which the half-nuts may be engaged. When the threading dial is not being used, it should be disengaged from the lead screw to prevent unnecessary wear to the worm wheel.

CARRIAGE STOP

You can attach the carriage stop to the bed at any point where you want to stop the carriage. The carriage stop is used principally in turning, facing, or boring duplicate parts; it eliminates the need for repeated measurements of the same dimension. To operate the carriage stop, set the stop at the point where you want to stop the feed. Just before the carriage reaches this point, shut off the automatic feed and carefully run the carriage up against the stop.

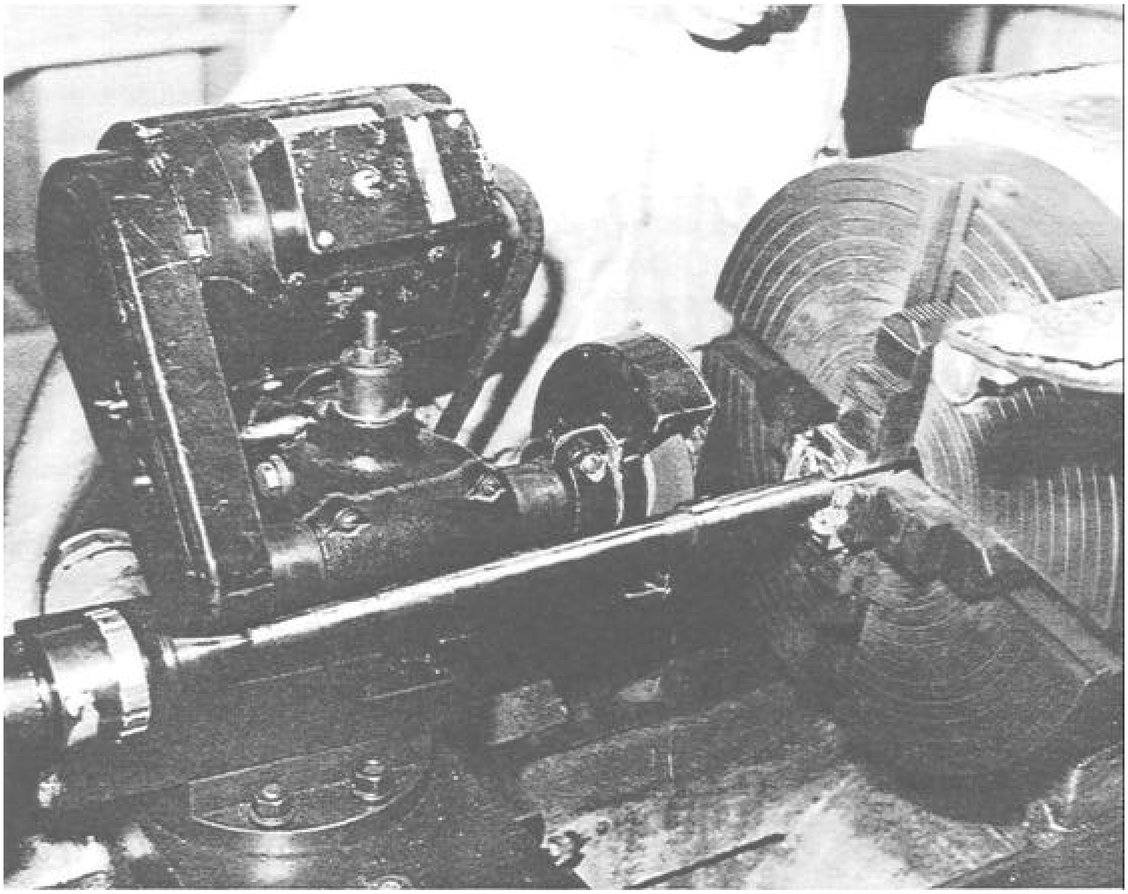
Carriage stops are provided with or without micrometer adjustment. Figure 6-33 shows a micrometer carriage stop. Clamp it on the ways in the approximate position required and then adjust it to the exact setting using the micrometer adjustment. (**NOTE:** Do not confuse this stop with the automatic carriage stop that automatically stops the carriage by disengaging the feed or stopping the lathe.)

GRINDING ATTACHMENT

The grinding attachment, illustrated in figure 6-34, is a portable grinder with a base that fits on the compound rest in the same manner as the toolpost. Like the cutting tool, the grinding attachment can be fed to the work at any angle. It is used for grinding hard-faced valve disks and seats, for grinding lathe centers, and for all kinds of cylindrical grinding. For internal grinding, small wheels are used on special quills (extensions) screwed onto the grinder shaft.

Deleted—No permission granted for electronic copy.

Figure 6-33.—Micrometer carriage stop.



28.101

Figure 6-34.—Grinder mounted on a compound rest.

MILLING ATTACHMENT

The milling attachment adapts the lathe to perform milling operations on small work, such as cutting keyways, slotting screwheads, machining flats, and milling dovetails. Figure 6-35 illustrates the setup for milling a dovetail.

The milling cutter is held in an arbor driven by the lathe spindle. The work is held in a vise on the milling attachment. The milling attachment is mounted on the cross-slide and therefore its movement can be controlled by the longitudinal feed and cross feed of the lathe. The depth of the cut is regulated by the longitudinal feed while the length of the cut is regulated by the cross feed. Vertical motion is controlled by the adjusting screw at the top of the attachment. The vise can be set at any angle in a horizontal or vertical plane. A milling attachment is unnecessary in shops equipped with milling machines.

Deleted—No permission granted for electronic copy.

Figure 6-35.—Milling attachment.

TRACING ATTACHMENTS

A tracing attachment for a lathe is useful whenever you have to make several parts that are identical in design. A tracer is a hydraulically actuated attachment that carries the cutting tool on a path identical to the shape and dimensions of a pattern or template of the part to be made. The major parts of the attachment are a hydraulic power unit, a tracer valve to which the stylus that follows the template is attached, a cylinder and slide assembly that holds the cutting tool and moves in or out on the command of the tracer valve hydraulic pressure output, and a template rail assembly that holds the template of the part to be made. There are several different manufacturers of tracers, and each tracer has a slightly different design and varying operating features. Tracers can be used for turning, facing, and boring and are capable of maintaining a dimensional tolerance equal to that of the lathe being used. Templates for the tracer can be made from either flat steel or aluminum plate or from round bar stock. It is important that the template be exactly like the finished part you require. Any scratch, dent, or mismachined dimension will be reproduced on the parts to be made.

OTHER TYPES OF ENGINE LATHES

The type of engine lathe that has been described in this chapter is the general-purpose, screw-cutting, precision lathe that is universally used in the machine shops of ships in the Navy. Repair ships also carry other types. A short description of some other types follows.

A bench lathe (fig. 6-36) is a small engine lathe mounted on a bench. Such lathes are sometimes used in the toolroom of repair ships.

The gap (extension) lathe shown in figure 6-37 has a removable bed piece shown on the deck in front of the lathe. This piece can be removed from the lathe bed to create a gap into which work of larger diameter may be swung. Some gap lathes are designed so that the ways can be moved longitudinally to create the gap.

LATHE SETUP

Before an aircraft is permitted to take off, the pilot and crew must go through a check-out procedure to determine whether the engines, controls, and safety features are in proper operating condition. The same applies to an engine lathe. Part of this procedure is to make sure the lathe is set up properly. Before starting a lathe machining operation, always make sure the machine is set up for the job you are doing. If the work is mounted between centers, check the alignment of the dead center with the live center and make any required changes. Ensure that the toolholder and the cutting tool are set at the proper height and angle. Check the workholding accessory to make sure the workpiece is held securely. Use the center rest or follower rest to support long workpieces.

LATHE SAFETY

In machine operations, there is one sequence of events that you must always follow. **SAFETY FIRST, ACCURACY SECOND, AND SPEED**

Deleted—No permission granted for electronic copy.

Figure 6-36.—A bench lathe.

LAST. As with any shop equipment you must observe all posted safety precautions. Review your equipment operators manual for safety precautions and any chapters of *Navy Occupational Safety and Health (NAVOSH) Program Manual for Forces Afloat,* OPNAV Instruction 5100.19B, which pertain to the equipment you will be operating.

ALIGNING AND LEVELING

Alignment and levelness must be correct to ensure proper operation of all engine lathes. They must be aligned and leveled when installed and must be checked periodically. Since the information provided here is only a brief overview, you should follow the machines' manufacturers instructions.

The three basic methods of aligning and leveling lathes are spirit method, optical, and test bar. The optical method requires special equipment. If your ship or shore station has an optical shop, they may be able to optically align your lathes or there are civilian companies that provide this service. Since Machinery Repairman do not optically align lathes, we will only discuss the spirit level and test bar methods.

You should remember that the spirit level method will work well at shore stations, but it will produce only approximate levelness aboard ship. The test bar method is the most accurate for shipboard use.

The leveling of lathes actually refers to the removal of the twist in the bed. This twist results from setting the machine on an uneven foundation. The machine is leveled by adjusting one or more legs to remove the twist in the bed so that it is straight and parallel with the spindle.

The Spirit Level Method

The leveling, or untwisting, of a lathe requires the use of a very accurate level. An ordinary carpenter's or machinist's level is not sufficiently accurate for leveling lathes. A sensitive, graduated-tube spirit level reading to 10 seconds of arc per graduation (0.0006 inch per foot) is required. The level should be adjustable and should have both a short base and a long tube.

The procedure for leveling engine lathes is as follows:

1. Loosen the lag screws that hold the right end legs to the deck. Do not loosen the lag screws on the left end (headstock) legs.

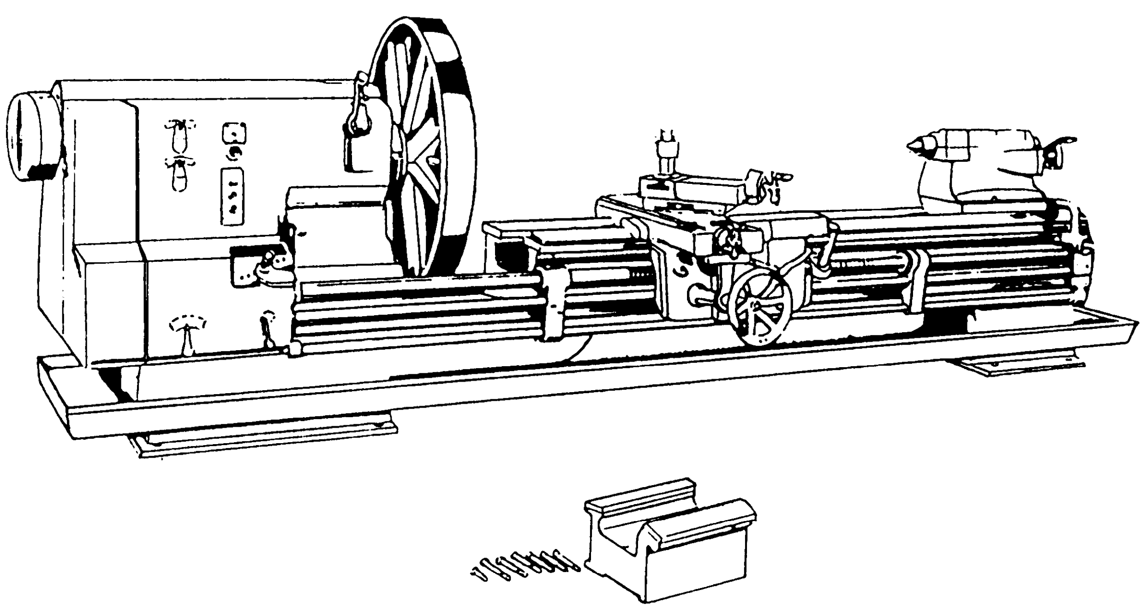


Figure 6-37.—A gap lathe.

2. Place the level across the bed at a right angle to the center line of the bed near the headstock (as shown in view A fig. 6-38). Adjust the level until the bubble is in the center and allow at least 30 seconds for the bubble to come to rest.

3. Without changing its adjustment, move the level to the other end of the ways and place it again at a right angle to the center line of the bed (as shown in view B, fig. 6-38). Then, by adjusting the leveling screws on the right-hand legs, bring the bubble to center. If the machine does not have leveling screws, use steel shims under the legs.

4. Repeat steps 2 and 3 until the difference in the bubble readings at the two positions is less than one division.

5. Tighten the lag screws and repeat steps 2 and 3 as a final check.

Deleted—No permission granted for electronic copy.

Figure 6-38.—A. Placing the level at right angles to the center line of the back of the headstock. B. placing the level at right angles to the center line of the bed at the outer end of the ways.

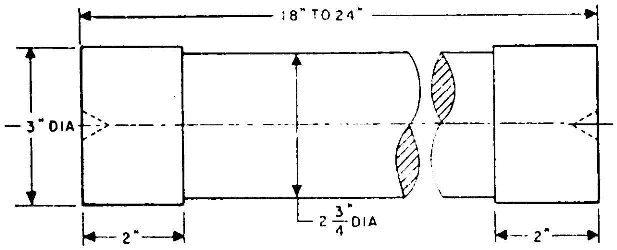


Figure 6-39.—Lathe alignment test bar.

The Test Bar Method

The first step in the test bar method is to make the test bar. A test bar may be any metal bar, 3 inches in diameter and approximately 18 to 24 inches long. (See fig. 6-39 for a sample). Next take a light cut on each end of the test bar **WITHOUT CHANGING THE TOOL SETTING.** Measure the diameter of each end. A difference in diameter indicates a misalignment. Adjust the machine leveling screws and repeat the procedure until a cut on both ends of the test bar results in the same diameter.

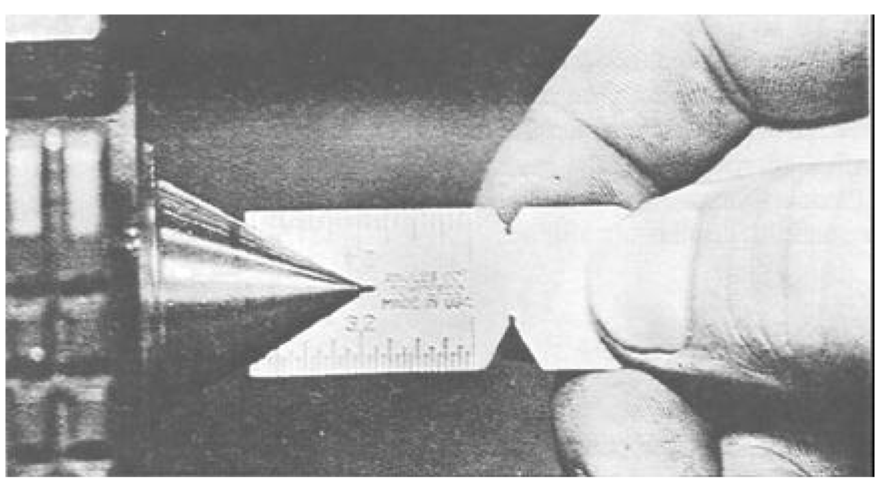
PREPARING THE CENTERS

The first step in preparing the centers is to see that they are accurately mounted in the headstock and tailstock spindles. The centers and the tapered holes in which they are fitted must be perfectly clean. Chips and dirt left on the contact surfaces will impair accuracy by preventing a perfect fit of the bearing surfaces. Be sure that there are no burrs in the spindle hole. If you find burrs, remove them by carefully scraping or reaming the surface with a Morse taper reamer. Burrs will produce the same inaccuracies as chips and dirt.

Center points must be accurately finished to an included angle of 60°. Figure 6-40 shows the method of checking the angle with a center gauge. The large notch of the center gauge is intended for this particular purpose. If the test shows that the point is not perfect, true the point in the lathe by taking a cut over the point with the compound rest set at 30°. To true a hardened tail center, either anneal it and then machine it or grind it if a grinding attachment is available.

Aligning and Testing

To turn a shaft straight and true between centers, be sure the centers are in the same plane parallel to the ways of the lathe. You can align the centers by releasing the tailstock from the ways and then moving the tailstock laterally with two adjusting screws. At the rear of the tailstock are two zero lines, and the centers are approximately aligned when these lines coincide. To check the approximate alignment, move the tailstock up until the centers almost touch and observe their relative positions as shown in



28.105

Figure 6-40.—Checking the center point with a center gauge.

Deleted—No permission granted for electronic copy.

Figure 6-41.—Aligning lathe centers.

figure 6-41. To produce very accurate work, especially if it is long, use the following procedure to determine and correct errors in alignment not otherwise detected.

Mount the work to be turned, or a piece of stock of similar length, on the centers. With a turning tool in the toolpost, take a small cut to a depth of a few thousandths of an inch at the headstock end of the work. Then, remove the work from the centers to allow the carriage to be run back to the tailstock without withdrawing the tool. Do not touch the tool setting. Replace the work in the centers, and with the tool set at the previous depth take another cut coming in from the tailstock end. Compare the diameters of these cuts with a micrometer. If the diameters are exactly the same, the centers are in perfect alignment. If they are different, adjust the tailstock in the direction required by using the setover adjusting screws. Repeat the test and adjustment until a cut at each end produces equal diameters.

Another method you can use to check for positive alignment of lathe centers is to take a light cut over the work held between the centers. Then, measure the work at each end with a micrometer. If the readings differ, adjust the tailstock to remove the difference. Repeat the procedure until the centers are aligned.

Truing Centers

To machine or true a lathe center, remove the faceplate from the spindle. Then, insert the live center into the spindle and set the compound rest at an angle of 30° with the axis of the spindle, as shown in figure 6-42. If you are using a three- or four-jaw chuck, secure the material that you are using to manufacture your center from in the chuck and proceed from this point. Place a round-nose tool in the toolpost and set the cutting edge of the tool at the exact center point of the lathe center. Machine a light cut on the center point and test the point with a center gauge. All lathe centers, regardless of their size, are finished to an included angle of 60°.

If you must regrind a dead center for the tailstock, it is best to do it using a cylindrical grinder. The cylindrical grinder will be covered in chapter 10.

SETTING THE TOOLHOLDER AND CUTTING TOOL

The first requirement for setting the tool is to have it rigidly mounted on the toolpost holder. Be sure the tool sits squarely in the toolpost and that the setscrew is tight. Reduce overhang as much as possible to prevent the tool from springing during cutting. If the tool has too much spring, the point of the tool will catch in the work, causing chatter and damaging both the tool and the work. The relative distances of A and B in figure 6-43 show the correct overhang for the tool bit and the holder. When a quick-change toolholder is used, tool overhang should not exceed twice the width of the cutting tool, or of the shank, when you use a carbide insert cutting tool.

The point of the tool must be correctly positioned on the work. When you are using a high-speed cutting tool to straight turn steel, cast iron, and other relatively hard metals, set the point about 5° above center. A rule measurement of approximately 3/64

Deleted—No permission granted for electronic copy.

Figure 6-42.—Machining a lathe center.

Deleted—No permission granted for electronic copy.

Figure 6-43.—Tool overhang.

inch times the diameter of the workpiece equals 5°. The point of a high-speed steel cutting tool being used to cut aluminum should be set slightly more than 5° above center, while the points of tools used to cut copper, brass, and other soft metals should be set exactly on center. The point of cast alloy (stellite and so on), carbide, and ceramic cutting tools should be placed exactly on center regardless of the material being cut. The tool point should be placed on center for threading, turning tapers, parting (cutting-off), or boring.

You can adjust the height of the tool in the toolholder illustrated in figure 6-43 by moving the half-moon wedge beneath the toolholder in or out as required. The quick-change toolholder has an adjusting screw to stop the tool at the correct height. Some square turret toolholders require a shim beneath the tool to adjust the height.

There are several methods you can use to set a tool on center. You can place a dead center in the tailstock and align the point of the tool with the point of the center. The tailstock spindle on many lathes has a line on the side that represents the center. You can also place a 6-inch rule against the workpiece in a vertical position and move the cross-slide in until the tool lightly touches the rule and holds it in place. Look at the rule from the side to determine if the height of the tool is correct. The rule will be straight up and down when the tool is exactly on center and will be at an angle when the tool is either high or low.

METHODS OF HOLDING THE WORK

You cannot perform accurate work if the work is improperly mounted. The following are requirements for proper mounting:

- The work center line must be accurately centered along the axis of the lathe spindle.
- The work must be held rigidly while being turned.
- The work must not be sprung out of shape by the holding device.
- The work must be adequately supported against any sagging caused by its own weight and against springing caused by the action of the cutting tool.

There are four general methods of holding work in the lathe: (1) between centers, (2) on a mandrel, (3) in a chuck, and (4) on a faceplate. Work may also be clamped to the carriage for boring and milling; the boring bar or milling cutter is held and driven by the headstock spindle.

Holding Work Between Centers

To machine a workpiece between centers, drill center holes in each end to receive the lathe centers. Secure a lathe dog to the workpiece and then mount the work between the live and dead centers of the lathe.

CENTERING THE WORK.—To center drill round stock such as drill-rod or cold-rolled steel, secure the work to the head spindle in a universal chuck or a draw-in collet chuck. If the work is too long and too large to be passed through the spindle, use a center rest to support one end. It is good shop practice to first take a light finishing cut across the face of the end of the stock to be center drilled. This will provide a smooth and even surface and will help prevent the center drill from "wandering" or breaking. The centering tool is held in a drill chuck in the tailstock spindle and fed to the work by the tailstock handwhecl (fig. 6-44).

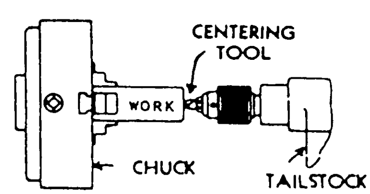


Figure 6-44.—Drilling center hole.

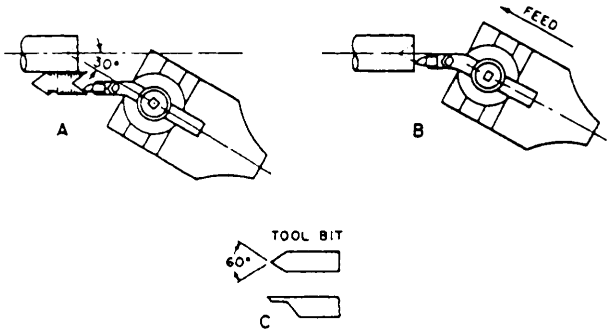


Figure 6-45.—Boring center hole.

If you must center a piece very accurately, bore the tapered center hole after you center drill to correct any run-out of the drill. You can do this by grinding a tool bit to fit a center gauge at a 60° angle. Then, with the toolholder held in the toolpost, set the compound rest at 30° with the line of center as shown in figure 6-45. Set the tool exactly on the center for height and adjust the tool to the proper angle with the center gauge as shown at A. Feed the tool as shown at B to correct any runout of the center. The tool bit should be relieved under the cutting edge as shown at C to prevent the tool from dragging or rubbing in the hole.

For center drilling a workpiece, the combined drill and countersink is the most practical tool. Combined drills and countersinks vary in size and the drill points also vary. Sometimes a drill point on one end will be 1/8 inch in diameter and the drill point on the opposite end will be 3/16 inch in diameter. The angle of the center drill is always 60° so that the countersunk hole will fit the angle of the lathe center point.

In center drilling, use a drop or two of oil on the drill. Feed the drill slowly and carefully to prevent breaking the tip. Use extreme care when the work is heavy, because it is then more difficult to "feel" the proper feed of the work on the center drill.

If the center drill breaks in countersinking and part of the broken drill remains in the work, you must remove the broken part. Sometimes you can jar it loose, or you may have to drive it out by using a chisel. But it may stick so hard that you cannot easily remove it. If so, anneal the broken part of the drill and drill it out.

The importance of having proper center holes in the work and a correct angle on the point of the lathe centers cannot be overemphasized. To do an accurate job between centers on the lathe, you must countersink holes of the proper size and depth and be sure the points of the lathe centers are true and accurate.

Deleted—No permission granted for electronic copy.

Figure 6-46.—Examples of center holes.

Figure 6-46 shows correct and incorrect countersinking for work to be machined on centers. In example A, the correctly countersunk hole is deep enough so that the point of the lathe centers does not come in contact with the bottom of the hole.

In example B of figure 6-46, the countersunk hole is too deep, causing only the outer edge of the work to rest on the lathe center. Work cannot be machined on centers countersunk in this manner.

Example C shows a piece of work that has been countersunk with a tool having too large an angle. This work rests on the point of the lathe center only. It is evident that this work will soon destroy the end of the lathe center, thus making it impossible to do an accurate job.

MOUNTING THE WORK.—Figure 6-47 shows correct and incorrect methods of mounting work between centers. In the correct example, the driving dog is attached to the work and rigidly held by the setscrew. The tail of the dog rests in the slot of the drive plate and extends beyond the base of the slot so that the work rests firmly on both the headstock center and tailstock center.

In the incorrect example, the tail of the dog rests on the bottom of the slot on the faceplate at A, thereby pulling the work away from the center points, as shown at B and C, causing the work to revolve eccentrically.

Deleted—No permission granted for electronic copy.

Figure 6-47.—Examples of work mounted between centers.

When you mount work between centers for machining, there should be no end play between the work and the dead center. However, if the work is held too tightly by the tail center, when the work begins revolving it will heat the center point and destroy both the center and the work. To prevent overheating, lubricate the tail center with a heavy oil or a lubricant specially made for this purpose. If you are using a ball bearing center, no lubricant is necessary.

Holding Work on a Mandrel

Many parts, such as bushings, gears, collars, and pulleys, require all the finished external surfaces to run true with the hole that extends through them. That is, the outside diameter must be true with the inside diameter or bore.

A mandrel is simply a round piece of steel of convenient length that has been centered and turned true with the centers. Commercial mandrels are made of tool steel, hardened and ground with a slight taper (usually 0.0005 inch per inch). On sizes up to 1 inch the small end is usually one-half of one thousandth of an inch under the standard size of the mandrel, while on larger sizes this dimension is usually one thousandth of an inch under standard. This taper allows the standard hole in the work to vary according to the usual shop practice, and still provides the necessary fit to drive the work when the mandrel is pressed into the hole. However, the taper is not great enough to distort the hole in the work. The countersunk centers of the mandrel are lapped for accuracy, while the ends are turned smaller than the body of the mandrel and are provided with flats, which give a driving surface for the lathe dog.

General practice is to finish the hole to a standard size, within the limit of the accuracy desired. Thus, a 3/4-inch standard hole will have a finished dimension of from 0.7505 to 0.7495 inch, or a tolerance of one-half of one thousandth of an inch above or below the true standard size of exactly 0.750 inch. First, drill or bore the hole to within a few thousandths of an inch of the finished size; then remove the remainder of the material with a machine reamer.

Press the piece on a mandrel tightly enough so the work will not slip while it is machined and clamp a dog on the mandrel, which is mounted between centers. since the mandrel surface runs true with respect to the lathe axis, the turned surfaces of the work on the mandrel will be true with respect to the hole in the piece.

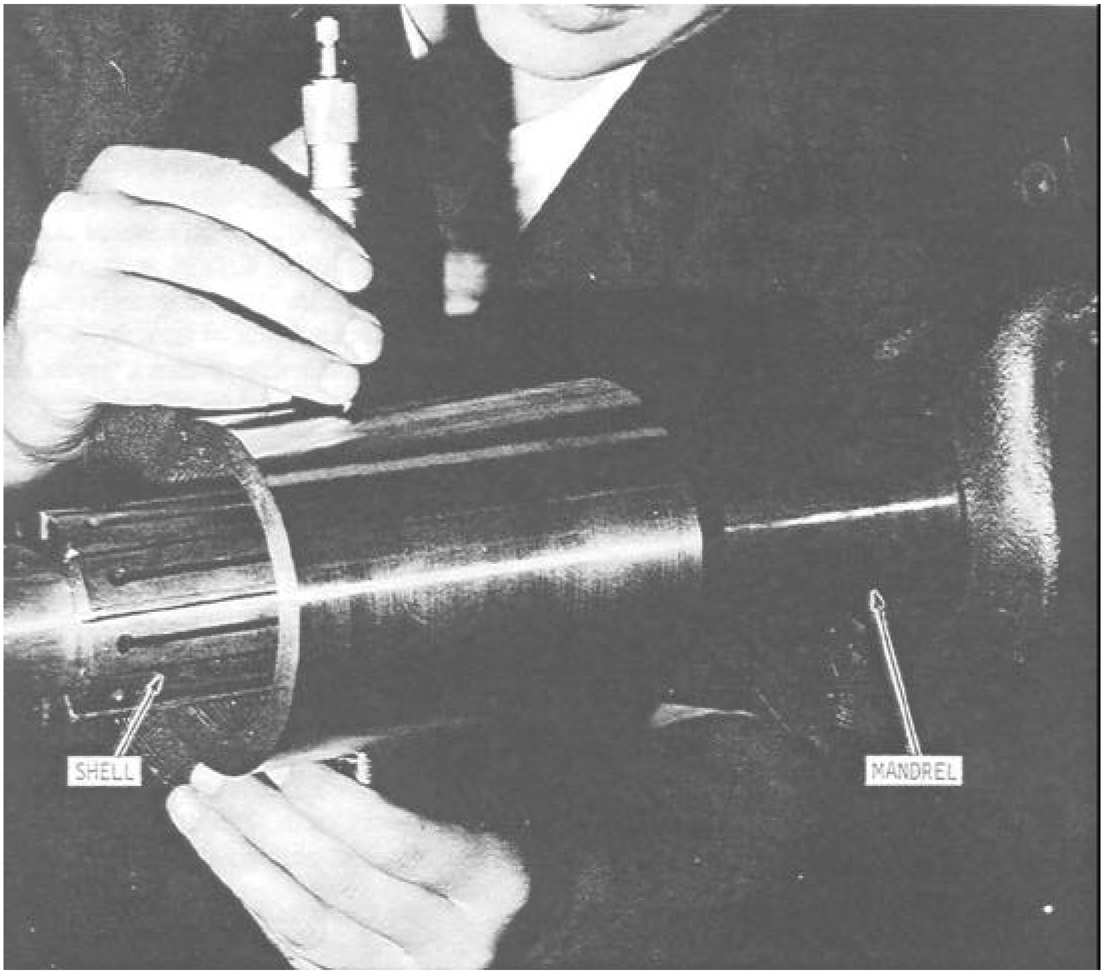
The size of the mandrel is always marked on the large end to avoid error and for convenience in placing work on it. The work is driven or pressed on from the small end and removed the same way.

When the hole in the work is not standard size, or if no standard mandrel is available, make a soft mandrel to fit the particular piece to be machined.

Use a few drops of oil to lubricate the surface of the mandrel before pressing it into the work, because clean metallic surfaces gall or stick when pressed together. If you do not use lubricant, you will not be able to drive the mandrel out without ruining the work.

Whenever you machine work on a mandrel, be sure the lathe centers are true and accurately aligned; otherwise, the finished turned surface will not be true. Before turning accurate work, test the mandrel on centers before placing any work on it. The best test for runout is one made with a dial indicator. Mount the indicator on the toolpost so the point of the indicator just touches the mandrel. As the mandrel is turned slowly between centers, any runout will be registered on the indicator dial.

If runout is indicated and you cannot correct it by adjusting the tailstock, the mandrel itself is at fault (assuming that the lathe centers are true) and cannot be used. The countersunk holes may have been damaged, or the mandrel may have been bent by careless handling. Be sure you always protect the



28.116

Figure 6-48.—A split-shell expansion mandrel.

ends of the mandrel when you press or drive it into the work. A piece of work mounted on a mandrel must have a tighter press fit to the mandrel for roughing cuts than for finishing cuts. Thick-walled work can be left on the mandrel for the finishing cut, but thin-walled work should be removed from the mandrel after the roughing cut and lightly reloaded on the mandrel before the finish cut is taken.

In addition to the standard lathe mandrel just described, there are expansion mandrels, gang mandrels, and eccentric mandrels.

An **EXPANSION** mandrel is used to hold work that is reamed or bored to nonstandard size. Figure 6-48 shows an expansion mandrel composed of two parts: a tapered pin that has a taper of approximately 1/16 inch for each inch of length and an outer split shell that is tapered to fit the pin. The split shell is placed in the work and the tapered pin is forced into the shell, causing it to expand until it holds the work properly.

A **GANG** mandrel (fig. 6-49) is used for holding several duplicate pieces such as gear blanks. The pieces are held tightly against a shoulder by a nut at the tailstock end.

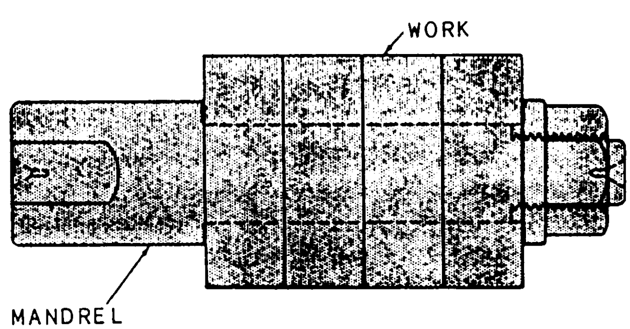


Figure 6-49.—Gang mandrel.

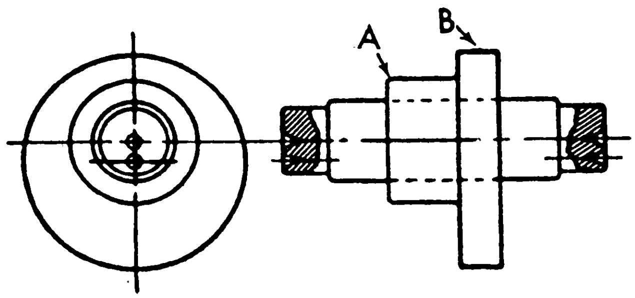


Figure 6-50.—Work on an eccentric mandrel.

An **ECCENTRIC** mandrel has two sets of countersunk holes, one pair of which is off-center an amount equal to the eccentricity of the work to be machined. Figure 6-50 illustrates its application: A is to be machined concentric with the hole in the work, while B is to be machined eccentric to it.

Holding Work In Chucks

The independent chuck and universal chuck are used more often than other workholding devices in lathe operations. A universal chuck is used for holding relatively true cylindrical work when accurate concentricity of the machined surface and holding power of the chuck are secondary to the time required to do the job. An independent chuck is used when the work is irregular in shape, must be accurately centered, or must be held securely for heavy feeds and depth of cut.

FOUR-JAW INDEPENDENT CHUCK.—Figure 6-51 shows a rough casting mounted in a four-jaw independent lathe chuck on the spindle of the lathe. Before truing the work, determine which part you wish to turn true. To mount a rough casting in the chuck, proceed as follows:

1. Adjust the chuck jaws to receive the casting. Each jaw should be concentric with the ring marks indicated on the face of the chuck. If there are no ring marks, set the jaws equally distant from the circumference of the chuck body.

2. Fasten the work in the chuck by turning the adjusting screw on jaw No. 1 and jaw No. 3, a pair of jaws which are opposite each other. Next tighten jaws No. 2 and No. 4 (opposite each other).

3. At this stage the work should be held in the jaws just tightly enough so it will not fall out of the chuck while being trued.

4. Revolve the spindle slowly, and with a piece of chalk mark the high spot (A in fig. 6-51) on the work while it is revolving. Steady your hand on the toolpost while holding the chalk.

5. Stop the spindle. Locate the high spot on the work and adjust the jaws in the proper direction to true the work by releasing the jaw opposite the chalk mark and tightening the one nearest the tank.

6. Sometimes the high spot on the work will be located between adjacent jaws. When it is, loosen the two opposite jaws and tighten the jaws adjacent to the high spot.

7. When the work is running true in the chuck, tighten the jaws gradually, working the jaws in pairs as described previously, until all four jaws clamp the work tightly. Be sure that the back of the work rests flat against the inside face of the chuck, or against the faces of the jaw stops (B in figure 6-51).

Use the same procedure to clamp semi-finished or finished pieces in the chuck, except center these pieces more accurately in the chuck. If the runout tolerance is very small, use a dial indicator to determine the runout.

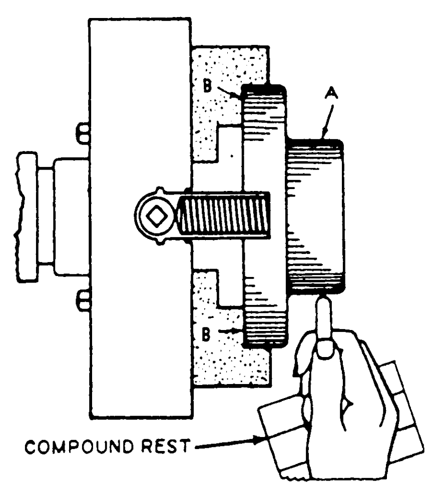


Figure 6-51.—Work mounted in a four-jaw independent chuck.

Deleted—No permission granted for electronic copy.

Figure 6-52.—Centering work with a dial indicator.

Figure 6-52 illustrates the use of a dial test indicator in centering work that has a hole bored in its center. As the work is revolved, the high spot is indicated on the dial of the instrument to a thousandth of an inch. The jaws of the chuck are adjusted on the work until the indicator hand registers no deviation as the work is revolved.

When the work consists of a number of duplicate parts that are to be tightened in the chuck, release two adjacent jaws and remove the work. Place another piece in the chuck and retighten the two jaws just released.

Each jaw of a lathe chuck, whether an independent or a universal chuck, has a number stamped on it to correspond to a similar number on the chuck. When you remove a chuck jaw for any reason, always put it back into the proper slot.

When the work to be chucked is frail or light, tighten the jaw carefully so the work will not bend, break, or spring.

To mount rings or cylindrical disks on a chuck, expand the chuck jaws against the inside of the workpiece. (See fig. 6-53.)

Regardless of how you mount the workpiece, NEVER leave the chuck wrench in the chuck while the chuck is on the lathe spindle. If the lathe should be started, the wrench could fly off the chuck and injure you or a bystander.

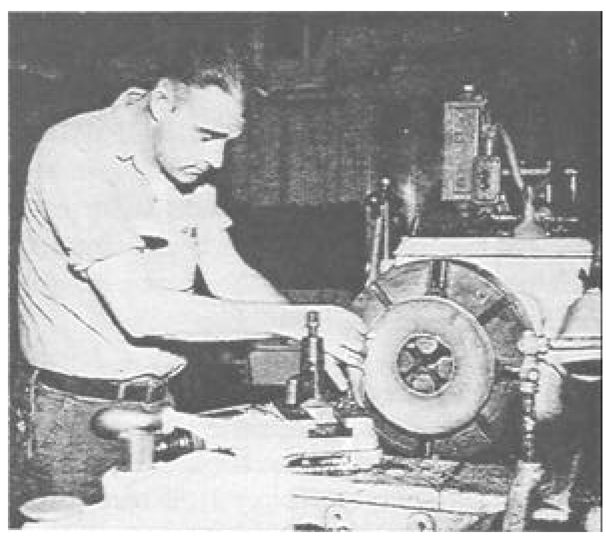
THREE-JAW UNIVERSAL CHUCK.—A three-jaw universal, or scroll, chuck allows all jaws to move together or apart in unison. A universal chuck will center almost exactly at the first clamping, but after a period of use it may develop inaccuracies of from 0.002 to 0.010 inch in centering the work, requiring the runout of the work to be corrected. Sometimes you can make the correction by inserting a piece of paper or thin shim stock between the jaw and the work on the **HIGH SIDE.**

When you chuck thin sections, be careful not to clamp the work too tightly, since the diameter of the piece will be machined while the piece is distorted. Then, when you release the pressure of the jaws after finishing the cut, there will be as many high spots as there are jaws, and the turned surface will not be true.

DRAW-IN-COLLET CHUCK.—A draw-in collet chuck is used for very fine accurate work of small diameter. Long work can be passed through the hollow drawbar, and short work can be placed directly into the collet from the front. TIghten the collet on the work by rotating the drawbar handwheel to the right. This draws the collet into the tapered closing sleeve. Turn the handle to the left to release the collet.

You will get the most accurate results when the diameter of the work is the same as the dimension stamped on the collet. The actual diameter of the work may vary from the collet dimension by 0.001 inch. However, if the work diameter varies more than this, the accuracy of the finished work will be affected. Most draw-in collet chuck sets are sized in 1/64-inch increments to allow you to select a collet within the required tolerances.

RUBBER FLEX COLLET CHUCK.—A rubber flex collet chuck is basically the same as the draw-in collet, except that the size of the stock held is not as critical. The rubber collets are graduated in



28.121

Figure 6-53.—Work held from inside by a four-jaw independent chuck.

Deleted—No permission granted for electronic copy.

Figure 6-54.—Eccentric machining of work mounted on a faceplate.

1/16-inch steps and will tighten down with accuracy on any size within the 1/16-inch range.

CARE OF CHUCKS.—To preserve a chuck's accuracy, handle it carefully and keep it clean. Never force a chuck jaw by using a pipe as an extension on the chuck wrench.

Before mounting a chuck, remove the live center and fill the hole with a rag to prevent chips and dirt from getting into the tapered hole of the spindle.

Clean and oil the threads of the chuck and the spindle nose. Dirt or chips on the threads will not allow the chuck to seat properly against the spindle shoulder and will prevent the chuck from running true. Since there are a number of different ways that chucks mount to machines, you must refer to your operators manual for mounting instructions.

When you mount or remove a heavy chuck, lay a board across the bedways to protect them and to help support the chuck as you put it on or take it off. Most larger chucks are drilled and tapped to accept a pad eye for lifting with a chainfall.

The procedures for mounting and removing faceplates are the same as for mounting and removing chucks.

Holding Work On a Faceplate

A faceplate is used for mounting work that cannot be chucked or turned between centers because of its peculiar shape. A faceplate is also used when holes are to be accurately machined in flat work, as in figure 6-54, or when large and irregularly shaped work is to be faced on the lathe.

Deleted—No permission granted for electronic copy.

Figure 6-55.—Work damped to an angle plate.

Work is secured to the faceplate by bolts, clamps, or any suitable clamping means. The holes and slots in the faceplate are used to anchor the holding bolts. Angle plates may be used to locate the work at the desired angle, as shown in figure 6-55. (Note the counterweight added for balance.)

For work to be mounted accurately on a faceplate, the surface of the work in contact with the faceplate must be accurate. Check the accuracy with a dial indicator. If you find runout, reface the surface of the work that is in contact with the faceplate. It is good practice to place a piece of paper between the work and the faceplate to keep the work from slipping.

Before securely clamping the work, move it about on the surface of the faceplate until the point to be machined is centered accurately over the axis of the lathe. Suppose you wish to bore a hole, the center of which has been laid out and marked with a prick punch. First, clamp the work to the approximate position on the faceplate. Then, slide the tailstock up to where the dead center just touches the work. Note, the dead center should have a sharp, true point. Now revolve the work slowly and, if the work is off center, the point of the dead center will scribe a circle on the work. If the work is on center, the point of the dead center will coincide with the prick punch mark.

Deleted—No permission granted for electronic copy.

Figure 6-56.—Work mounted on a carriage for boring.

Holding Work On the Carriage

If a piece of work is too large or bulky to swing conveniently in a chuck or on a faceplate, you can bolt it to the carriage or the cross slide and machine it with a cutter mounted on the spindle. Figure 6-56 shows a piece of work being machined by a fly cutter mounted in a boring bar that is held between centers and driven by a lathe dog.

Using the Center Rest and Follower Rest

Long slender work often requires support between its ends while it is turned; otherwise the work would spring away from the tool and chatter. The center rest is used to support such work so it can be turned accurately at a faster feed and cutting speed than would be possible without the center rest. (See fig. 6-57).

Place the center rest where it will give the greatest support to the piece to be turned. This is usually at about the middle of its length.

Ensure that the center point between the jaws of the center rest coincides exactly with the axis of the lathe spindle. To do this, place a short piece of stock in a chuck and machine it to the diameter of the workpiece to be supported. Without removing the stock from the chuck, clamp the center rest on the ways to the lathe and adjust the jaws to the machined surface. Without changing the jaw settings, slide the center rest into position to support the workpiece. Remove the stock used for setting the center rest and set the workpiece in place. Use a dial indicator to true

Deleted—No permission granted for electronic copy.

Figure 6-57.—Use of a center rest to support work between centers

the workpiece at the chuck. Figure 6-58 shows how a chuck and center rest are used to machine the end of a workpiece.

The follower rest differs from the center rest in that it moves with the carriage and provides support against the forces of the cut. To use the tool, turn a "spot" to the desired finish diameter and about 5/8 to 3/4 inch wide on the workpiece. Then, adjust the jaws of the follower rest against the area you just machined. The follower rest will move with the cutting tool and support the point being machined.

The follower rest (fig. 6-59) is indispensable for chasing threads on long screws, as it allows the cutting of a screw with a uniform pitch diameter. Without the follower rest, the screw would be inaccurate because it would spring away from the tool.

Deleted—No permission granted for electronic copy.

Figure 6-58.—Work mounted in a chuck and center rest.

Use a sufficient amount of grease, oil or other available lubricant on the jaws of the center rest and follower rest to prevent "seizing" and scoring the workpiece. Check the jaws frequently to see that they do not become hot. The jaws may expand slightly if they get hot and push the work out of alignment (when the follower rest is used) or bind (when the center rest is used). If you are using a center rest or follower rest that is equipped with ball bearings, no lubrication is necessary.

ENGINE LATHE OPERATIONS

Up to this point, you have studied the preliminary steps leading up to performing machine work on the lathe. You have learned how to mount the work and the tool, and which tools are used for various purposes. The next step is to learn how to use the lathe. We will now cover some of the basic operations that you will accomplish on the engine lathe.

Remember that accuracy is the prime requisite of a good machine job; so before you start, be sure that the centers are true and properly aligned, that the work is mounted properly, and that the cutting tools are correctly ground and sharpened.

As we cover the various operations that you will perform on a lathe, remember the tooling used in your shop may be different from what is pictured in this manual. You may use different toolholders or cutting tools, but the operations are basically the same. Always remember that you are only limited by your imagination as to the various tooling and setup combinations you can use.

Deleted—No permission granted for electronic copy.

Figure 6-59.—Follower rest supporting screw while thread is being cut.

PLANNING THE JOB

It is important for you to study the blueprint of the part to be manufactured before you begin machining. Check over the dimensions and note the points or surfaces from which they are laid out. Plan the steps of your work in advance to determine the best way to proceed. Check the overall dimensions and be sure the stock you intend to use is large enough for the job. For example, small design features, such as collars on pump shafts or valve stems, will require that you use stock of much larger diameter than that required for the main features of the workpiece.

CUTTING SPEEDS AND FEEDS

Cutting speed is the rate at which the surface of the work passes the point of the cutting tool. It is expressed in feet per minute (fpm).

To find the cutting speed, multiply the diameter of the work (DIA) in inches times 3.1416 times the number of revolutions per minute (rpm) and divide by 12.

$$CS = \frac{DIA \times 3.1416 \times rpm}{12}$$

The result is the peripheral or cutting speed in fpm. For example, a 2-inch diameter part turning at 100 rpm will produce a cutting speed of

$$\frac{2 \times 3.1416 \times 100}{12} = 52.36\ fpm$$

If you have selected a recommended cutting speed from a chart for a specific type of metal, you will need to figure what rpm is required to obtain the recommended cutting speed. Use the following formula:

$$rpm = \frac{CS \times 12}{DIA \times 3.1416}$$

Recommended cutting speeds in fpm for turning carbon and alloy steels using either high-speed steel or carbide tool bits can be found in the cutting speeds section of the current edition of the *Machinery's Handbook.*

FEED is the amount the tool advances in each revolution of the work. It is usually expressed in thousandths of an inch per revolution of the spindle. The index plate on the quick-change gear box indicates the setup for obtaining the feed desired. The amount of feed to use is best determined from experience.

Cutting speeds and tool feeds are determined by various considerations: the hardness and toughness of the metal being cut; the quality, shape, and sharpness of the cutting tool; the depth of the cut; the tendency of the work to spring away from the tool; and the rigidity and power of the lathe. Since conditions vary, it is good practice to find out what the tool and work will stand, and then select the most practical and efficient speed and feed consistent with the finish desired.

If the cutting speed is too slow, the job takes longer than necessary and the work produced is often unsatisfactory because of a poor finish. On the other hand, if the speed is too fast the tool edge will dull quickly and will require frequent regrinding. The cutting speeds possible are greatly affected by the use of a suitable cutting coolant. For example, steel that can be rough turned dry at 60 rpm can be turned at about 80 rpm when flooded coolant.

When **ROUGHING** parts down to size, use the greatest depth of cut and feed per revolution that the work, the machine, and the tool will stand at the highest practical speed. On many pieces, when tool failure is the limiting factor in the size of the roughing cut, it is usually possible to reduce the speed slightly and increase the feed to a point that the metal removed is much greater. This will prolong tool life. Consider an example of when the depth of cut is 1/4 inch, the feed is 20 thousandths of an inch per revolution, and the speed is 80 fpm. If the tool will not permit additional feed at this speed, you can usually drop the speed to 60 fpm and increase the feed to about 40 thousandths of an inch per revolution without having tool trouble. The speed is therefore reduced 25 percent, but the feed is increased 100 percent. The actual time required to complete the work is less with the second setup.

On the **FINISH TURNING OPERATION,** a very light cut is taken since most of the stock has been removed on the roughing cut. A fine feed can usually be used, making it possible to run a high surface speed. A 50 percent increase in speed over the roughing speed is commonly used. In particular cases, the finishing speed may be twice the roughing speed. In any event, run the work as fast as the tool will withstand to obtain the maximum speed in this operation. Use a sharp tool to finish turning.

COOLANTS

A coolant serves two main purposes: (1) It cools the tool by absorbing a portion of the heat and reduces the friction between the tool and the metal being cut. (2) It keeps the cutting edge of the tool flushed clean. A coolant generally allows you to use a higher cutting speed, heavier feeds, and depths of cut than if you performed the machining operation dry. The life of the cutting tool is also prolonged by coolants. The most common coolants used are soluble oil and synthetic coolants. Refer to the manufacturers' recommendations for proper mixing rates.

The various operations used and materials machined on a lathe may cause problems in the selection of the proper coolant. A possible solution is to select a coolant that is suitable for the majority of the materials you plan to work with.

CHATTER

A symptom of improper lathe operation is known as "chatter." Chatter is vibration in either the tool or the work. The finished work surface will appear to have a grooved or lined finish instead of the smooth surface that is expected. The vibration is set up by a weakness in the work, work support, tool, or tool support and is perhaps the most elusive thing you will find in the entire field of machine work. As a general rule, strengthening the various parts of the tool support train will help. It is also advisable to support the work with a center rest or follower rest.

Begin your search for the cause of the chatter by making sure that the surface speed is not excessive. Since excessive speed is probably the most frequent cause of chatter, reduce the speed and see if the chatter stops. You may also increase the feed, particularly if you are taking a rough cut and the finish is not important. Another adjustment you can try is to reduce the lead angle of the tool (the angle formed between the surface of the work and the side cutting edge of the tool). You may do this by positioning the tool closer and perpendicular to the work.

If none of these actions work, examine the lathe and its adjustments. Gibs may be loose or bearings may be worn after a long period of heavy service. If the machine is in perfect condition, the fault may be in

the tool or the tool setup. Check to be sure the tool has been properly sharpened to a point or as near to a point as the specific finish will permit. Reduce the overhang of the tool as much as possible and recheck the gib and bearing adjustments. Finally, be sure that the work is properly supported and that the cutting speed is not too high.

FACING

Facing is the machining of the end surfaces and shoulders of a workpiece. In addition to squaring the ends of the work, facing will let you accurately cut the work to length. Generally, in facing the workpiece you will need to take only light cuts since the work has already been cut to approximate length or rough machined to the shoulder.

Figure 6-60 shows how to face a cylindrical piece. Place the work on centers and install a dog. Using a right-hand side tool, take one or two light cuts from the center outward to true the work.

If both ends of the work must be faced, reverse the piece so the dog drives the end just faced. Use a steel ruler to lay out the required length, measuring from the faced end to the end to be faced. After you ensure that there is no burr on the finished end to cause an inaccurate measurement, mark off the desired dimension with a scribe and face the second end.

Figure 6-61 shows the facing of a shoulder having a fillet corner. First, take a finish cut on the outside of the smaller diameter section. Next, machine the fillet with a light cut by manipulating the apron handwheel and the cross-feed handle in unison to produce a smooth rounded surface. Finally, use the tool to face from the fillet to the outside diameter of the work.

In facing large surfaces, lock the carriage in position since only cross feed is required to traverse the tool across the work. With the compound rest set at 90° (parallel to the axis of the lathe), use the micrometer collar to feed the tool to the proper depth of cut in the face. For greater accuracy in getting a given size when flnishing a face, set the compound rest at 30°. In this position, 0.001-inch movement of the compound rest will move the tool exactly 0.0005-inch in a direction parallel to the axis of the lathe. (In a 30° - 60° right triangle, the length of the side opposite the 30° angle is equal to one-half of the length of the hypotenuse.)

TURNING

Turning is the machining of excess stock from the periphery of the workpiece to reduce the diameter. Bear in mind that the diameter of the work being turned is reduced by the amount equal to twice the depth of the cut; thus, to reduce the diameter of a piece by 1/4 inch, you must remove 1/8 inch of metal from the surface.

To remove large amounts of stock in most lathe machining, you will take a series of roughing cuts to remove most of the excess stock and then a finishing cut to accurately “size” the workpiece.

Deleted—No permission granted for electronic copy.

Figure 6-60.—Right-hand side tool.

Deleted—No permission granted for electronic copy.

Figure 6-61.—Facing a shoulder.

Deleted—No permission granted for electronic copy.

Figure 6-62.—Rough turning.

Rough Turning

Figure 6-62 illustrates a lathe taking a heavy cut. This is called rough turning. When a great deal of stock is to be removed, you should take heavy cuts in order to complete the job in the least possible time.

Be sure to select the proper tool for taking a heavy chip. The speed of the work and the amount of feed of the tool should be as great as the tool will stand.

When taking a roughing cut on steel, cast iron, or any other metal that has a scale on its surface, be sure to set the tool deeply enough to get under the scale in the first cut. If you do not, the scale on the metal will dull the point of the tool.

Rough machine the work to almost the finished size; then be very careful in taking measurements on the rough surface.

Often the heat produced during rough turning will expand the workpiece, and the lubricant will flow out of the live center hole. This will result in both the center and the center hole becoming worn. Always check the center carefully and adjust as needed during rough turning operations. If you are using a ball bearing center, feel the area where the bearings are located and ensure the center is not too warm.

Figure 6-63 shows the position of the tool for taking a heavy chip on large work. Set the tool so that if anything causes it to change position during the machining operation, the tool will move away from the work, thus preventing damage to the work. Also, setting the tool in this position may prevent chatter.

Finish Turning

When you have rough turned the work to within about 1/32 inch of the finished size, take a finishing cut. A fine feed, the proper coolant and a keen-edged tool are necessary to produce a smooth finish. Measure carefully to be sure you are machining the work to the proper dimension. Stop the lathe whenever you take any measurements.

If you must finish the work to extremely close tolerances, wait until the piece is cool before taking the finish cut. If the piece has expanded slightly because of the heat generated by turning and you turn it to size while it is hot, the piece will be undersize after it has cooled and contracted.

If you plan to finish the work on a cylindrical grinder, leave the stock slightly oversize to allow for the metal the grinder will remove.

Perhaps the most difficult operation for a beginner in machine work is taking accurate measurements. So much depends on the accuracy of the work that you should make every effort to become proficient in using measuring instruments. You will develop a certain “feel” through experience. Do not be discouraged if your first efforts do not produce perfect results. Practice taking measurements on pieces of known dimensions. You will acquire the skill if you are persistent.

Turning to a Shoulder

A time-saving procedure for machining a shoulder is illustrated in figure 6-64. First, locate and

Deleted—No permission granted for electronic copy.

Figure 6-63.—Position of tool for heavy cut.

Deleted—No permission granted for electronic copy.

Figure 6-64.—Machining to a shoulder.

scribe the exact location of the shoulder on the work. Next, use a parting tool to machine a groove 1/32 inch from the scribe line toward the smaller finish diameter end and 1/32 inch larger than the smaller finish diameter. Then, take heavy cuts up to the shoulder made by the parting tool. Finally, take a finish cut from the small end to the should scribe line. This procedure eliminates detailed measuring and speeds up production.

PARTING AND GROOVING

One of the methods of cutting off a piece of stock while it is held in a lathe is a process called parting. This process uses a specially shaped tool with a cutting edge similar to that of a square nose tool. The parting tool is fed into the rotating work, perpendicular to its axis, cutting a progressively deeper groove as the work rotates. When the cutting edge of the tool gets to the center of the work being parted, the work drops off as if it were sawed off. Parting is used to cut off parts that have already been machined in the lathe or to cut tubing and bar stock to required lengths.

Parting tools can be the inserted blade type or can be ground from a standard tool blank. They may also be brazed on carbide or carbide inserts. Figure 6-65 shows two basic types of parting tools. For the tool to have maximum strength, the length of the cutting portion of the blade that extends from the holder should be only slightly greater than half the diameter of the work to be parted. The end cutting edge of the tool must feed directly toward the center of the workpiece. To ensure this, place a center in the tailstock and align the parting tool vertically with the tip of the center. The chuck should hold the work to

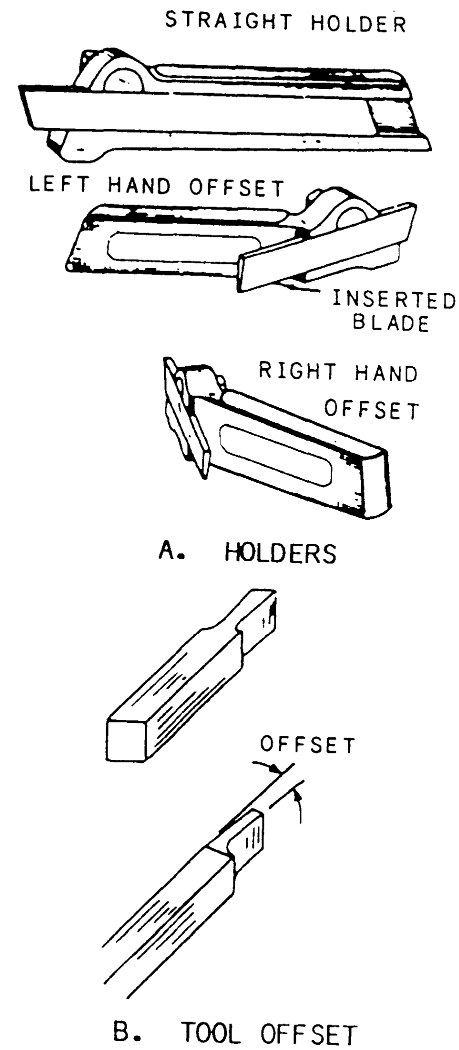


Figure 6-65.—Parting tools.

be parted with the point at which the parting is to occur as close as possible to the chuck jaws. Always make the parting cut at a right angle to the centerline of the work. Feed the tool into the revolving work with the cross-slide until the tool completely separates the work.

Cutting speeds for parting are usually slower than turning speeds. You should use a feed that will keep a thin chip coming from the work. If chatter occurs, decrease the speed and increase the feed slightly. If the tool tends to gouge or dig in, decrease the feed.

Grooves are machined in shafts to provide for tool runout in threading to a shoulder, to allow clearance for assembly of parts, to provide lubricating channels, or to provide a seating surface for seals and O-rings.

Square, round, and "V" grooves and the tools that are used to produce them are shown in figure 6-66.

The grooving tool is a type of forming tool. It is ground without side rake or back rake and is set to the work at center height with a minimum of overhang. The side and end relief angles are generally somewhat less than for turning tools. When you machine a groove, reduce the spindle speed to prevent the chatter that often develops at high speeds because of the greater amount of tool contact with the work.

DRILLING AND REAMING

Drilling operations performed in a lathe differ very little from drilling operations performed in a drilling machine. For best results, start the drilling operation by drilling a center hole in the work, using a combination center drill and countersink. The combination countersink-center drill is held in a drill chuck that is mounted in the tailstock spindle. After you have center drilled the work, replace the drill chuck with a taper shank drill. (**NOTE: BEFORE** you insert any tool into the tailstock spindle, inspect the shank of the tool for burrs. If the shank is burred, remove the burrs with a handstone.) Feed the drill into the work by using the tailstock handwheel. Use a coolant/lubricant whenever possible and maintain sufficient pressure on the drill to prevent chatter, but not enough to overheat the drill.

If the hole is quite long, back the drill out occasionally to clear the flutes of metal chips. Large diameter holes may require you to drill a pilot hole first. This is done with a drill that is smaller than the finished diameter of the hole. After you have drilled the pilot hole to the proper depth, enlarge the hole with the finish drill. If you plan to drill the hole completely through the work, slow down the feed as the drill nears the hole exit. This will produce a smoother exit hole by causing the drill to take a finer cut as it exits the hole.

If the twist drill is not ground correctly, the drilled hole will be either excessively oversized or out of round. Check the drill for the correct angle, clearance, cutting edge lengths and straightness before setting it up for drilling. It is almost impossible to drill a hole exactly the same size as the drill regardless of the care taken in ensuring an accurately ground drill and the proper selection of speeds and feeds. For this reason, any job that requires close tolerances or a good finish on the hole should be reamed or bored to the correct size.

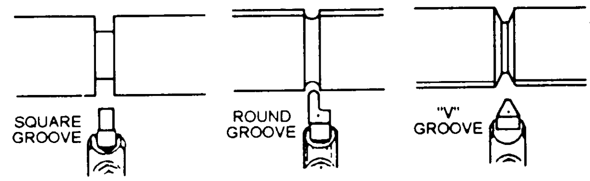


Figure 6-66.—Three common types of grooves.

If the job requires that the hole be reamed, it is good practice to first take a cleanup cut through the hole with a boring tool. This will true up the hole for the reaming operation. Be sure to leave about 1/64 inch for reaming. The machine reamer has a taper shank and is held in and fed by the tailstock. To avoid overheating the reamer, set the work speed at about half that used for the drilling operation. During the reaming operation, keep the reamer well lubricated. This will keep the reamer cool and also flush the chips from the flutes. Do not feed the reamer too fast; it may tear the surface of the hole and ruin the work.

BORING

Boring is the machining of holes or any interior cylindrical surface. The piece to be bored must have a drilled or core hole, and the hole must be large enough to insert the tool. The boring process merely enlarges the hole to the desired size or shape. The advantage of boring is that you get a perfectly true round hole.

Work to be bored may be held in a chuck, bolted to the faceplate, or bolted to the carriage. Long pieces must be supported at the free end of a center rest.

When the boring tool is fed into the hole in work being rotated on a chuck or faceplate, the process is called single point boring. It is the same as turning except that the cutting chip is taken from the inside. The cutting edge of the boring tool resembles that of a turning tool. Boring tools may be the solid forged type or the inserted cutter bit type.

When the work to be bored is clamped to the top of the carriage, a boring bar is held between centers and driven by a dog. The work is fed to the tool by the automatic longitudinal feed of the carriage. Three types of boring bars are shown in figure 6-67. Note the countersunk center holes at the ends to fit the lathe centers.

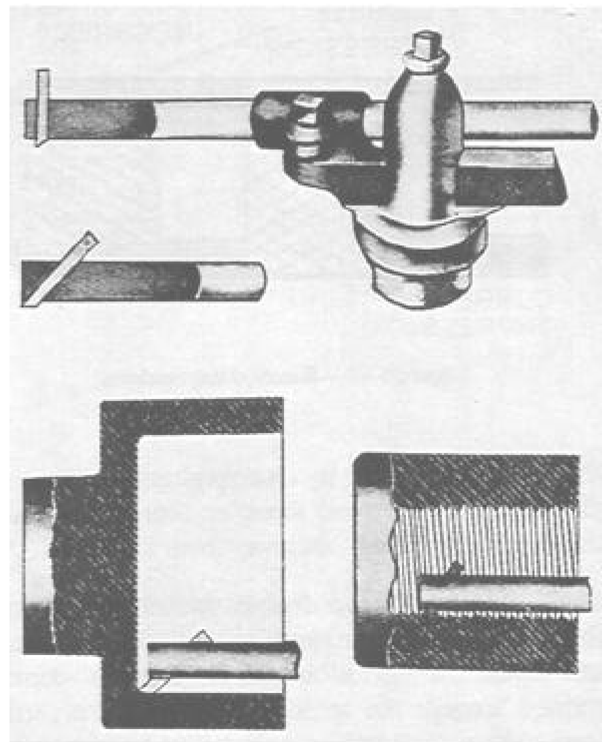
Part A of figure 6-67 shows a boring bar fitted with a fly cutter held by a headless setscrew. The

other setscrew, bearing on the end of the cutter, is for adjusting the cutter to the work.

Part B of figure 6-67 shows a boring bar fitted with a two-edge cutter held by a taper key. This is more of a finishing or sizing cutter, as it cuts on both sides and is used for production work.

The boring bar shown in part C of figure 6-67 is fitted with a cast iron head to adapt it for boring work of large diameter. The head is fitted with a fly cutter similar to the one shown in part A. The setscrew with the tapered point adjusts the cutter to the work.

Figure 6-68 shows a common type of boring bar holder and applications of the boring bar for boring and internal threading. When threading is to be done in a blind hole, it sometimes becomes necessary to undercut or relieve the bottom of the hole. This will enable mating parts to be screwed all the way to the shoulder and make the threading operation much easier to do.



28.135

Figure 6-68.—Application of boring bar holder.

KNURLING

Knurling is the process of rolling or squeezing impressions into the work with hardened steel rollers that have teeth milled into their faces. Examples of the various knurling patterns are shown in figure 6-17. Knurling provides a gripping surface on the work; it is also used for decoration. Knurling increases the diameter of the workpiece slightly when the metal is raised by the forming action of the knurl rollers.

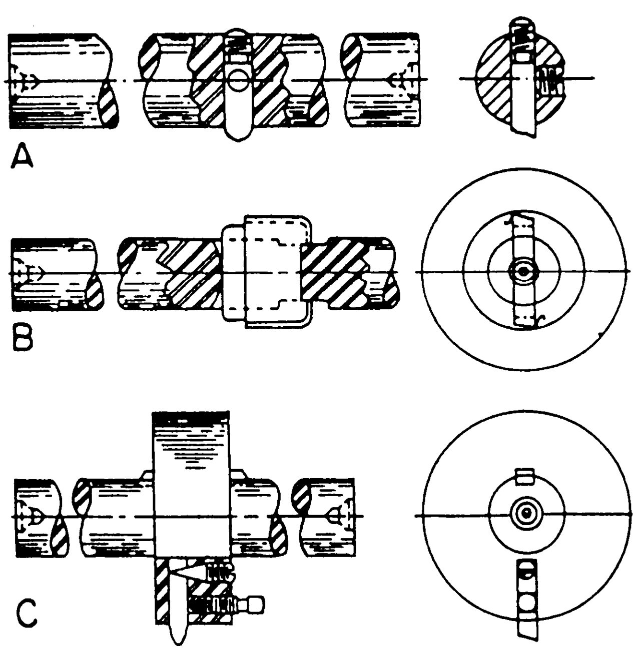


Figure 6-67.—Various boring bars.

The knurling tool (fig. 6-16) is set up so the faces of the rollers are parallel to the surface of the work and with the upper and lower rollers equally spaced above and below the work axis or center line. The spindle speed should be about half the roughing speed for the type of metal being machined. The feed should be between 0.015 inch and 0.025 inch per revolution. The work should be rigidly mounted in the tailstock to help offset the pressure exerted by the knurling operation.

The actual knurling operation is simple if you follow a few basic rules. The first step is to make sure that the rollers in the knurling tool turn freely and are free of chips and imbedded metal between the cutting edges. During the knurling process, apply an ample supply of oil at the point of contact to flush away chips and provide lubrication. Position the carriage so that one-third to one-half of the face of the rollers extends beyond the end of the work. This eliminates part of the pressure required to start the knurl impression. Force the knurling rollers into contact with the work. Engage the spindle clutch. Check the knurl to see if the rollers have tracked properly, as

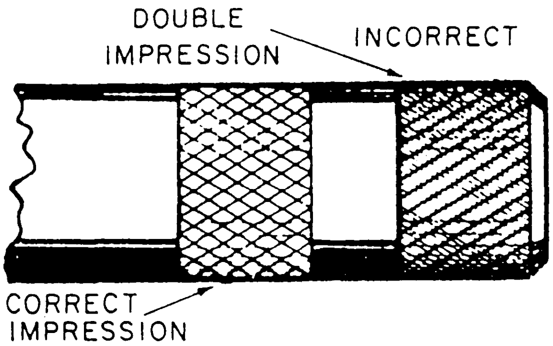


Figure 6-69.—Knurled impressions.

shown in figure 6-69, by disengaging the clutch after the work has revolved three or four times and by backing the knurling tool away from the work.

If the knurls have double tracked, as shown in figure 6-69, move the knurling tool to a new location and repeat the operation. If the knurl is correctly formed, engage the spindle clutch and the carriage feed. Move the knurling rollers into contact with the correctly formed knurled impressions. The rollers will align themselves with the impressions. Allow the knurling tool to feed to within about 1/32 inch of the end of the surface to be knurled. Disengage the carriage feed and with the work revolving, feed the carriage by hand to extend the knurl to the end of the surface. Force the knurling tool slightly deeper into the work, reverse the direction of feed and engage the carriage feed. Allow the knurling tool to feed until the opposite end of the knurled surface is reached. Never allow the knurls to feed off the surface.

Repeat the knurling operation until the diamond impressions converge to a point. Passes made after the correct shape is obtained will result in stripping away the points of the knurl. Clean the knurl with a brush and remove any burrs or sharp edges with a file. When knurling, do not let the work rotate while the

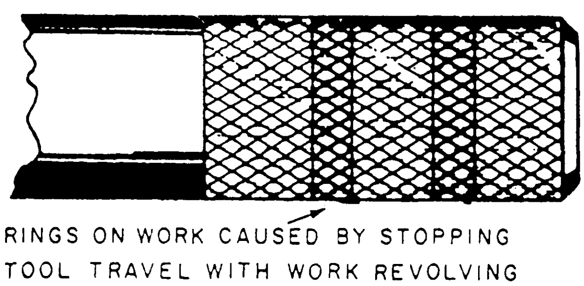


Figure 6-70.—Rings on a knurled surface.

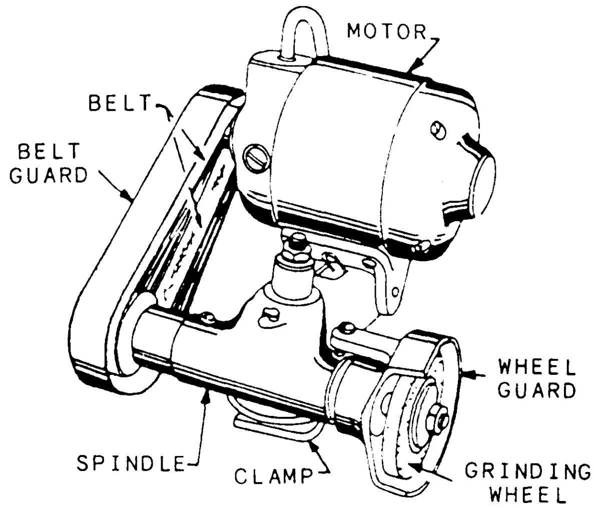


Figure 6-71.—Toolpost grinder.

tool is in contact with it if the feed is disengaged. This will cause rings to be formed on the surface, as shown in figure 6-70.

SETTING UP THE TOOLPOST GRINDER

The toolpost grinder is a portable grinding machine that can be mounted on the compound rest of a lathe in place of the toolpost. It can be used to machine work that is too hard to cut by ordinary means or to machine work that requires a very fine finish. Figure 6-71 shows a typical toolpost grinder.

The grinder must be set on center, as shown in figure 6-72. The centering holes located on the spindle shaft are used for this purpose. The grinding wheel takes the place of a lathe cutting tool; it can perform most of the same operations as a cutting tool. Cylindrical, tapered, and internal surfaces can be ground with the toolpost grinder. Very small grinding

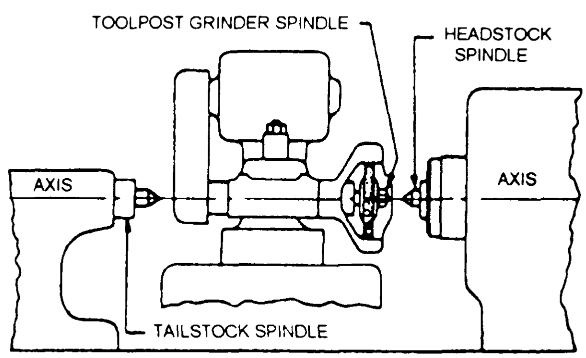


Figure 6-72.—Mounting the grinder at center height.

wheels are mounted on tapered shafts, known as quills, to grind internal surfaces.

The grinding wheel speed is changed by using various sizes of pulleys on the motor and spindle shafts. An instruction plate on the grinder gives both the diameter of the pulleys required to obtain a given speed and the maximum safe speed for grinding wheels of various diameters. Grinding wheels are safe for operation at a speed just below the highest recommended speed. A higher than recommended speed may cause the wheel to disintegrate. For this reason, wheel guards are furnished with the toolpost grinder to protect against injury.

Always check the pulley combinations given on the instruction plate of the grinder when you mount a wheel. Be sure that the combination is not reversed, because this may cause the wheel to run at a speed far in excess of that recommended. During all grinding operations, wear goggles to protect your eyes from flying abrasive material.

Before you use the grinder, dress and true the wheel with a diamond wheel dresser. The dresser is held in a holder that is clamped to the chuck or faceplate of the lathe. Set the point of the diamond at center height and at a 10° to 15° angle in the direction of the grinding wheel rotation, as shown in figure 6-73. The 10° to 15° angle prevents the diamond from gouging the wheel. Lock the lathe spindle by placing the spindle speed control lever in the low rpm position. (**NOTE:** The lathe spindle does not revolve when you are dressing the grinding wheel.)

Bring the grinding wheel into contact with the diamond dresser by carefully feeding the cross-slide in by hand. Move the wheel slowly by hand back and forth over the point of the diamond, taking a maximum cut of 0.0002 inch. Move the carriage if the face of the wheel is parallel to the ways of the lathe. Move the compound rest if the face of the wheel is at an angle. Make the final depth of cut of 0.0001 inch with a slow, even feed to obtain a good wheel finish. Remove the diamond dresser holder as soon as you finish dressing the wheel and adjust the grinder to begin the grinding operation.

Rotate the work at a fairly low speed during the grinding operation. The recommended surface speed is 60 to 100 fpm. The depth of cut depends upon the hardness of the work, the type of grinding wheel, and the desired finish. Avoid taking grinding cuts deeper than 0.002 inch until you gain experience. Use a fairly low rate of feed. You will soon be able to judge whether the feed should be increased or decreased. Never stop the work or the grinding wheel while they are in contact with each other.

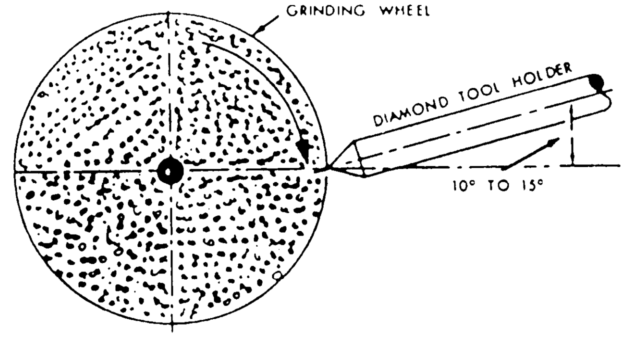
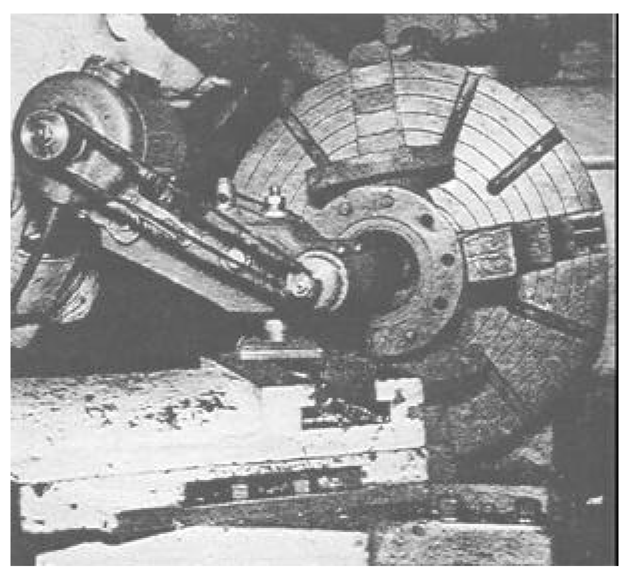


Figure 6-73.—Position of the diamond dresser.

Figure 6-74 illustrates refacing the seat of a high-pressure steam valve that has a hard, Stellite-faced surface. The refacing must be done with a toolpost grinder. Be sure that all inside diameters run true before starting the machine work. Spindle speed of the lathe should be about 40 rpm or less. Too high a speed will cause the grinding wheel to vibrate. Set the compound rest to correspond with the valve seat angle. Use the cross-slide hand feed or the micrometer stop on the carriage for controlling the depth of cut; use the compound rest for traversing the grinding wheel across the work surface. Remember, whenever you grind on a lathe, always place a cloth across the ways of the bed and over any other



28.136

Figure 6-74.—Refacing seat of high-pressure steam valve.

machined surfaces that could become contaminated from grinding dust.

TAPERS

Taper is the gradual decrease in the diameter of thickness of a piece of work toward one end. To find the amount of taper in any given length of work, subtract the size of the small end from the size of the large end. Taper is usually expressed as the amount of taper per foot of length, or as an angle. The following examples explain how to determine taper per foot of length.

EXAMPLE 1: Find the taper per foot of a piece of work 2 inches long: Diameter of the small end is 1 inch; diameter of the large end is 2 inches.

The amount of the taper is 2 inches minus 1 inch, which equals 1 inch. The length of the taper is given as 2 inches. Therefore, the taper is 1 inch in 2 inches of length. In 12 inches of length it would be 6 inches. (See fig. 6-75).

EXAMPLE 2: Find the taper per foot of a piece 6 inches long. Diameter of the small end is 1 inch; diameter of the large end is 2 inches.

The amount of taper is the same as in example 1; that is, 1 inch. (See fig. 6-75.) However, the length of this taper is 6 inches; hence the taper per foot is 1 inch × 12/6 = 2 inches per foot.

From the foregoing, you can see that the length of a tapered piece is very important in computing the

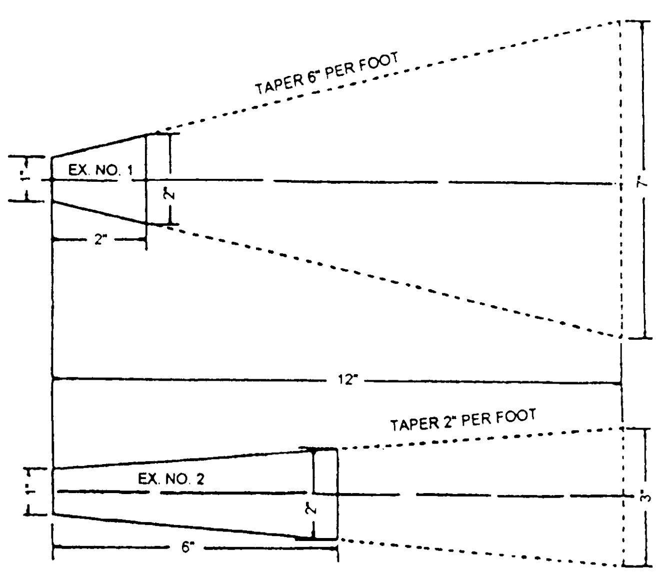


Figure 6-75.—Tapers.

taper. If you bear this in mind when machining tapers, you will not go wrong. Use the formula:

$$TPF = TPI \times 12$$

where:

TPF = TAPER PER FOOT

TPI = TAPER PER INCH

Other formulas used in figuring tapers are as follows:

$$TPI = \frac{T}{L}$$

where:

TPI = TAPER PER INCH

T = TAPER (Difference between large and small diameters, expressed in inches)

L = LENGTH of taper, expressed in inches

$$T = \frac{L \times TPF}{12} \text{ and } T = TPI \times L \text{ (in inches)}$$

$$TPI = \frac{TPF}{12}$$

Tapers are frequently cut by setting the angle of the taper on the appropriate lathe attachment. There are two angles associated with a taper-the included angle and the angle with the center line. The included angle is the angle between the two angled sides of the taper. The angle with the center line is the angle between the center line and either of the angled sides. Since the taper is turned about a center line, the angle between one side and the center line is always equal to the angle between the other side and the center line. Therefore, the included angle is always twice the angle with the center line. The importance of this relationship will be shown later in this chapter.

There are several well-known tapers that are used as standards for machines on which they are used. These standards make it possible to make or get parts to fit the machine in question without detailed measuring and fitting. By designating the name and number of the standard taper being used, you can immediately find the length, the diameter of the small and large ends, the taper per foot, and all other

pertinent measurements in appropriate tables found in the current edition of the *Machinery's Handbook.*

There are three standard tapers with which you should be familiar: (1) the **MORSE TAPER** (approximately 5/8 inch per foot) used for the tapered holes in lathe and drill press spindles and the attachments that fit them, such as lathe centers, drill shanks, and so on; (2) the **BROWN & SHARPE TAPER** (1/2 inch per foot, except No. 10, which is 0.5161 inch per foot) used for milling machine spindle shanks; and (3) the **JARNO TAPER** (0.600 inch per foot) used by some manufacturers because of the ease with which its dimensions can be determined:

$$\text{Diameter of large end} = \frac{\text{taper number}}{8}$$

$$\text{Diameter of small end} = \frac{\text{taper number}}{10}$$

$$\text{Length of taper} = \frac{\text{taper number}}{2}$$

Two additional tapers that are considered standard are the tapered pin and pipe thread tapers. Tapered pins have a taper of 1/4 inch per foot, while tapered pipe threads have a taper of 3/4 inch per foot.

Methods of Turning Tapers

In ordinary straight turning, the cutting tool moves along a line parallel to the axis of the work, causing the finished job to be the same diameter throughout. If, however, in cutting, the tool moves at an angle to the axis of the work, a taper will be produced. Therefore, to turn a taper, you must either mount the work in the lathe so the axis on which it turns is at an angle to the axis of the lathe, or cause the cutting tool to move at an angle to the axis of the lathe.

There are three methods in common use for turning tapers:

- **SET OVER THE TAILSTOCK,** which moves the dead center away from the axis of the lathe and causes work supported between centers to be at an angle with the axis of the lathe.
- **USE THE COMPOUND REST** set at an angle, which causes the cutting tool to be fed at the desired angle to the axis of the lathe.
- **USE THE TAPER ATTACHMENT,** which also causes the cutting tool to move at an angle to the axis of the lathe.

In the first method, the cutting tool is fed by the longitudinal feed parallel to the lathe axis, but a taper is produced because the work axis is at an angle. In the second and third methods, the work axis coincides with the lathe axis, but a taper is produced because the cutting tool moves at an angle.

SETTING OVER the TAILSTOCK.—As stated earlier in this chapter, you can move the tailstock top sideways on its base by using the adjusting screws. In straight turning you use these adjusting screws to align the dead center with the tail center by moving the tailstock to bring it on the center line of the spindle axis. For taper turning, you deliberately move the tailstock off center, and the amount you move it determines the taper produced. You can approximate the amount of setover by using the zero lines inscribed on the base and top of the tailstock as shown in figure 6-76. For final adjustment use a dial indicator to measure the distance you move the tailstock.

In turning a taper by this method, the distance between centers is of utmost importance. To illustrate, figure 6-77 shows two very different tapers

Deleted—No permission granted for electronic copy.

Figure 6-76.—Tailstock setover lines for taper turning.

Deleted—No permission granted for electronic copy.

Figure 6-77.—Setover of tailstock showing importance of considering length of work.

produced by the same amount of setover of the tailstock, because for one taper the length of the work between centers is greater than for the other. THE CLOSER THE DEAD CENTER IS To THE LIVE CENTER, THE STEEPER WILL BE THE TAPER PRODUCED. Suppose you want to turn a taper on the full length of a piece 12 inches long with one end having a diameter of 3 inches, and the other end a diameter of 2 inches. The small end is to be 1 inch smaller than the large end; so you set the tailstock over one-half of this amount or 1/2 inch in this example. Thus, at one end the cutting tool will be 1/2 inch closer to the center of the work than at the other end; so the diameter of the finished job will be 2 × 1/2 or 1 inch less at the small end. Since the piece is 12 inches long, you have produced a taper of 1 inch per foot. Now, if you wish to produce a taper of 1 inch per foot on a piece only 6 inches long, the small end will be only 1/2 inch less in diameter than the larger end, so you should set over the tailstock 1/4 inch or one-half of the distance used for the 12-inch length.

By now you can see that the setover is proportional to the length between centers. Setover is computed by using the following formula:

$$S = \frac{T}{2} \times \frac{L}{12}$$

where:

S = setover in inches

T = taper per foot in inches

L = length of taper in inches

$\frac{L}{12}$ = *length in feet of taper*

Remember that L is the length of the work from the live center to the dead or ball bearing center. If the work is on a mandrel, L is the length of the mandrel between centers. You cannot use the setover tailstock method for steep tapers because 2the setover would be too great and the work would not be properly supported by the lathe centers. The bearing surface becomes less and less satisfactory as the setover is increased. Do not exceed 0.250-inch setover since your center hole and your tailstock center will not align properly.

After turning a taper by the tailstock setover method, do not forget to realign the centers for straight turning of your next job.

USING the COMPOUND REST. —The compound rest is generally used for short, steep tapers. Set it at the angle the taper will make with the center line (that is, half of the included angle of the taper. Then, feed the tool to the work at this angle by using the compound rest feed screw. The length of taper you can machine is short because the travel of the compound rest is limited.

One example of using the compound rest for taper work is the truing of a lathe center. Other examples are refacing an angle type valve disk and machining the face of a bevel gear. Such jobs are often referred to as working to an angle rather than as taper work.

The graduations marked on the compound rest provide a quick means for setting it to the angle desired. When the compound rest is set at zero, the cutting tool is perpendicular to the lathe axis. When the compound rest is set at 90° on either side of zero, the cutting tool is parallel to the lathe axis.

To set up the compound rest for taper turning, first determine the angle to be cut, measured from the center line. This angle is half of the included angle of the taper you plan to cut. Then, set the compound rest to the complement of the angle to be cut (90° minus the angle to be cut). For example, to machine a 50° included angle (25° angle with the center line), set the compound rest at 90°–25°, or 65°.

When you must set the compound rest very accurately, to a fraction of a degree for example, run the carriage up to the faceplate and set the compound rest with a vernier bevel protractor set to the required angle. Hold the blade of the protractor on the flat surface of the faceplate and hold the base of the protractor against the finished side of the compound rest.

USING THE TAPER ATTACHMENT.—For turning and boring long tapers with accuracy, the taper attachment is indispensable. It is especially useful in duplicating work; you can turn and bore identical tapers with one setting of the taper guide bar. Set the guide bar at an angle to the lathe that corresponds to the desired taper. The tool cross slide will be moved laterally by a shoe, which slides on the guide bar as the carriage moves longitudinally. The cutting tool will move along a line parallel to the guide bar. The taper produced will have the same angular measurement as that set on the guide bar. The

Deleted—No permission granted for electronic copy.

Figure 6-78.—End view of taper guide bar.

guide bar is graduated in degrees at one end and in inches per foot of taper at the other end to provide for rapid setting. Figure 6-78 is a view of the end that is graduated in inches per foot of taper.

When you prepare to use the taper attachment, run the carriage up to the approximate position of the work to be turned. Set the tool on line with the center of the lathe. Then, bolt or clamp the holding bracket to the ways of the bed (the attachment itself is bolted to the back of the carriage saddle) and tighten the clamp (C, fig. 6-79). The taper guide bar now controls the lateral movement of the cross slide. Set the guide bar for the taper desired; the attachment is ready for operation. To make the final adjustment of the tool for size, use the compound rest feed screw, since the cross-feed screw is inoperative.

There will be a certain amount of lost motion or backlash when the tool first starts to feed along the work. This is caused by looseness between the

Deleted—No permission granted for electronic copy.

Figure 6-79.—Turning a taper using taper attachment.

cross-feed screw and the cross-slide nut. If the backlash is not eliminated, a straight portion will be turned on the work. You can remove the backlash by moving the carriage and tool slightly past the start of the cut and then returning the carriage and tool to the start of the cut.

Methods of Boring Tapers

Taper boring is usually done with either the compound rest or the taper attachment. The rules that apply to outside taper turning also apply to the boring of taper holes. Begin by drilling the hole to the correct depth with a drill of the same size as the specified small diameter of the taper. This gives you the advantage of boring to the right size without having to remove metal at the bottom of the bore, which is rather difficult, particularly in small, deep holes.

For turning and boring tapers, set the tool cutting edge exactly at the center of the work. That is, set the point of the cutting edge even with the height of the lathe centers; otherwise, the taper may be inaccurate.

Cut the hole and measure its size and taper using a taper plug gauge and the "cut and try" method.

1. After you have taken one or two cuts, clean the bore.

2. Rub the gauge lightly with chalk (or prussian blue if the taper must be highly accurate).

3. Insert the gauge into the hole and turn it SLIGHTLY so the chalk (or prussian blue) rubs from the gauge onto the surface of the hole. If the workpiece is to be mounted on a spindle, use the tapered end of the spindle instead of a gauge to test the taper.

4. Areas that do not touch the gauge will be shown by a lack of chalk (or prussian blue).

5. Continue making minor corrections until all, or an acceptable portion, of the hole's surface touches the gauge. Be sure the taper diameter is correct before you turn the taper to its finish diameter.

Figure 6-80 shows a Morse standard taper plug and a taper socket gauge. They not only give the proper taper, but also show the proper distance that the taper should enter the spindle.

SCREW THREADS

Much of the machine work performed by a Machinery Repairman includes the use of screw threads. The thread forms you will be working with most are V-form threads, Acme threads, and square threads. Each of these thread forms is used for specific purposes. V-form threads are commonly used on fastening devices, such as bolts and nuts, as well as on machine parts. Acme screw threads are generally used for transmitting motion, such as between the lead screw and lathe carriage. Square threads are used to increase mechanical advantage and to provide good clamping ability as in the screw jack or vise screw. Each of these screw forms is discussed more fully later in the chapter. We will also discuss cutting threads on an engine lathe.

There are several terms used in describing screw threads and screw thread systems that you must know before you can calculate and machine screw threads. Figure 6-81 illustrates some of the following terms:

EXTERNAL THREAD: A thread on the outside surface of a cylinder.

INTERNAL THREAD: A thread on the inside surface of a hollow cylinder.

RIGHT-HAND THREAD: A thread that, when viewed axially, winds in a clockwise and receding direction.

LEFT-HAND THREAD: A thread that, when viewed axially, winds in a counterclockwise and receding direction.

LEAD: The distance a threaded part moves axially in a fixed mating part in one complete revolution.

PITCH: The distance between corresponding points on adjacent threads.

Deleted—No permission granted for electronic copy.

Figure 6-80.—Morse taper socket gauge and plug gauge.

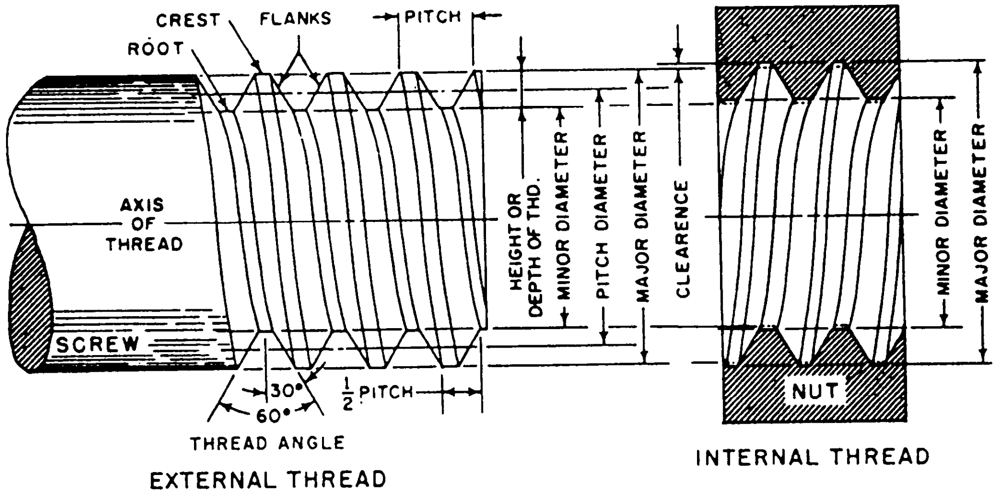


Figure 6-81.—Screw thread nomenclature.

SINGLE THREAD: A single (single start) thread whose lead equals the pitch.

MULTIPLE THREAD: A multiple (multiple start) thread whose lead equals the pitch multiplied by the number of starts.

CLASS OF THREADS: A group of threads designed for a certain type of fit. Classes of threads are distinguished from each other by the amount of tolerance and allowance specified.

THREAD FORM: The view of a thread along the thread axis for a length of one pitch.

FLANK: The side of the thread.

CREST: The top of the thread (bounded by the major diameter on external threads; by the minor diameter on internal threads).

ROOT: The bottom of the thread (bounded by the minor diameter on external threads; by the major diameter on internal threads).

THREAD ANGLE: The angle formed by adjacent flanks of a thread.

PITCH DIAMETER: The diameter of an imaginary cylinder that is concentric with the thread axis and whose periphery passes through the thread profile at the point where the widths of the thread and the thread groove are equal. The pitch diameter is the diameter that is measured when the thread is machined to size. A change in pitch diameter changes the fit between the thread being machined and the mating thread.

NOMINAL SIZE: The size that is used for identification. For example, the nominal size of a 1/2-20 thread is 1/2 inch, but its actual size is slightly smaller to provide clearance.

ACTUAL SIZE: The measured size.

BASIC SIZE: The theoretical size. The basic size is changed to provide the desired clearance or fit.

MAJOR DIAMETER: The diameter of an imaginary cylinder that passes through the crests of an external thread or the roots of an internal thread.

MINOR DIAMETER: The diameter of an imaginary cylinder that passes through the roots of an external thread or the crests of an internal thread.

HEIGHT OF THREAD: The distance from the crest to the root of a thread measured along a perpendicular to the axis of the threaded piece (also called straight depth of thread).

SLANT DEPTH: The distance from the crest to the root of a thread measured along the angle forming the side of the thread.

ALLOWANCE: An intentional difference between the maximum material limits of mating parts. It is the minimum clearance (positive allowance) or maximum interference (negative allowance) between such parts.

TOLERANCE: The total permissible variation of a size. The tolerance is the difference between the limits of size.

THREAD FORM SERIES: Threads are made in many different shapes, sizes, and accuracies.

When special threads are required by the product designer, he will specify in detail all the thread characteristics and their tolerances for production information. When a standard thread is selected, however, the designer needs only to specify size, number of threads per inch, designation of the standard series and class of fit. With these specifications, all other information necessary for production can be obtained from the established standard, as published. The abbreviated designations for the different series are as follows:

Abbreviation	Full Title of Standard Series
UNC	Unified coarse thread series
UNF	Unified fine thread series
UNEF	Unified extra fine thread series
NC	American National coarse thread series
NF	American National fine thread series
NEF	American National extra-fine thread series
UN	Unified constant pitch series including 4, 6, 8, 12, 16, 20, 28, and 32 threads per inch
NA	American National Acme thread series
NPT	American National tapered pipe thread series
NPS	American National straight pipe thread series
NH	American National hose coupling thread series
NS	American National form thread-special pitch
N BUTT	National buttress thread

THREAD DESIGNATION: A thread is designated by nominal size, number of threads per inch, series symbol, and class symbol, in that order. For example, the designation 1/4-20 UNC-3A specifies a thread with the following characteristics:

Nominal thread diameter = 1/4 inch

Number of threads per inch = 20

Series (unified coarse) = UNC

Class = 3

External thread = A

Unless the designation LH (left hand) follows the class designation, the thread is assumed to be a right-hand thread. An example of the designation for a left-hand thread is 1/4-20 UNC-3A-LH.

V-FORM THREADS

The three forms of V-threads that you must know how to machine are the V-sharp, the American National and the American Standard unified. All of these threads have a 60° included angle between their sides. The V-sharp thread has a greater depth than the others and the crest and root of this thread have little or no flat. The external American Standard unified thread has slightly less depth than the external American National thread but is otherwise similar. The American Standard unified thread is actually a modification of the American National thread. This modification was made so that the unified series of threads, which permits interchangeability of standard threaded fastening devices manufactured in the United States, Canada, and the United Kingdom, could be included in the threading system used in the United States.

To cut a V-form screw thread, you need to know (1) the pitch of the thread, (2) the straight depth of the thread, (3) the slant depth of the thread, and (4) the width of the flat at the root of the thread. The pitch of a thread is the basis for calculating all other dimensions and is equal to 1 divided by the number of threads per inch. The tap drill size is equal to the thread size minus the pitch, or the thread size minus ONE divided by the number of threads per inch.

$$\textit{Tap drill size} = \textit{Thread size} - \frac{1}{n}$$

When you feed the thread cutting tool into the workpiece, use the slant depth to determine how far to feed the tool into the work. The point of the threading tool must have a flat equal to the width of the flat at the root of the thread (external or internal thread, as applicable). If the flat at the point of the tool is too wide, the resulting thread will be too thin. If the flat is too narrow, the thread will be too thick.

The following formulas will provide the information you need for cutting V-form threads:

1. V-sharp thread

   Pitch = $\frac{1}{n}$ or 1 ÷ number of threads per inch

   Straight Depth of thread = 0.866 × pitch

2. American National Thread

   Pitch = 1 ÷ number of threads per inch

   Straight depth of external thread = 0.64952 × pitch or 064952p

   Straight depth of internal thread = 0.541266 × pitch or 0.541266p

   Width of flat at point of tool for external and internal threads = 0.125 × pitch or 0.125p

   Slant depth of external thread = 0.750 × pitch or 0.750p

   Slant depth of internal thread = 0.625 × pitch or 0.625p

3. American Standard Unified

   Pitch = 1 ÷ number of threads per inch or $\frac{1}{n}$

   Straight depth of external thread = 0.61343 inch × pitch or 0.61343p

   Straight depth of internal thread = 0.54127 inch × pitch or 0.54127p

   Width of flat at root of external thread = 0.125 inch × pitch or 0.125p

   Width of flat at crest of external thread = 0.125 inch × pitch or 0.125p

   Double height of external thread = 1.22687 inch × pitch or 1.22687p

   Double height of internal thread = 1.08253 inch × pitch or 1.08253p

   Slant depth of external thread = 0.708 × pitch or 0.708p

   Slant depth of internal thread = 0.625 × pitch or 0.625p

NOTE: MULTIPLYING the constant by the pitch, as in the preceding formulas, produces the same result as if you divide the constant by the number of threads per inch.

To produce the correct thread profile, you must use a tool accurately ground to the correct angle and contour. Also, you must set the cutting tool in the correct position. Figure 6-82 shows how a tool must be ground and set. Remember, if you are using a

Deleted—No permission granted for electronic copy.

Figure 6-82.—Threading tool setup for V-form threads.

Deleted—No permission granted for electronic copy.

Figure 6-83.—Acme thread and formulas for cutting.

quick-change toolholder you will not need to grind the top of the tool. If carbide inserts are being used, the thread form is already ground.

Grind the point of the tool to an angle of 60°, as shown in A of figure 6-82. Use a center gauge or a thread tool gauge for grinding the tool to the exact angle required. The top of the tool is usually ground flat, with no side rake or back rake. However, for cutting threads in steel, side rake is sometimes used.

Set the threading tool square with the work, as shown in B and C of figure 6-82. Use the center gauge to adjust the point of the threading tool; if you carefully set the tool, a perfect thread will result. If you do not set the threading tool perfectly square with the work, the angle of the thread will be incorrect.

For cutting external threads, place the top of the threading tool exactly on center as shown in D of figure 6-82. Note that the top of the tool is ground flat and is in exact alignment with the lathe center. This is necessary to obtain the correct angle of the thread.

The size of the threading tool for cutting an internal thread is important. The tool head must be small enough to be backed out of the thread and still leave enough clearance to be drawn from the threaded hole without injuring the thread. However, the boring bar that holds the threading tool for internal threading should be both as large as possible in diameter and as short as possible to keep it from springing away from the work during cutting.

OTHER FORMS OF THREADS

In the following section, other forms of screw threads are illustrated with pertinent information on cutting these threads.

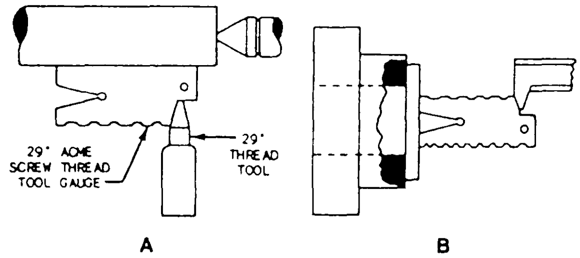


Figure 6-84.—Use of Acme thread tool gauge.

The Acme Screw Thread

The Acme screw thread is used on valve stems, the lead screw of a lathe, and other places that require a strong thread. The top and bottom of the threads have an included angle of 29° (fig. 6-83).

Parts A and B of figure 6-84 show the method of setting an Acme threading tool for cutting an external and internal Acme thread, respectively. Note that a 29° Acme thread gauge is used in the same manner as the center gauge was used for V-form screw threads. Adjust the cutting edge of the tool to line it up exactly with the beveled edge of the gauge. The notches in the Acme thread gauge let you grind the squared front edge of the tool bit accurately according to the pitch of the thread to be machined.

In cutting an Acme thread, be sure the clearance is 0.010 inch between the top of the thread of the screw and the bottom of the thread of the nut in which it fits.

The Square Thread

The square thread (fig. 6-85) is used when heavy threads are required, such as in jack screws, press screws, and feed screws. It is used for much the same

Deleted—No permission granted for electronic copy.

Figure 6-85.—Square thread and formulas.

purpose as the Acme thread, which is used in many places where the square thread was formerly used. The disadvantage of square threads is that the straight sides do not allow sideplay adjustment.

The cutting edge width of the tool for cutting square screw threads is exactly one-half the pitch, but the width of the edge of the tool for threading nuts is from 0.001 to 0.003 inch larger. This permits a sliding fit on the screw.

Set the threading tool for cutting square threads square with the work.

Be sure the clearance between the top of the screw thread and the bottom of the nut thread is about 0.005 to 0.008 inch for each inch of thread diameter.

Buttress Thread

On a buttress thread (fig. 6-86) the load resisting side is nearly perpendicular to the thread axis and is called the pressure flank. The American Standard form of the buttress thread has a 7° angle on the pressure flank; other forms have 0°, 3°, or 5°. However, the American Standard form is most often used, and the formulas in this section apply to this form. The buttress thread can be designed to either push or pull against the internal thread of the mating part into which it is screwed. The direction of the thrust will determine the way you grind your tool for machining the thread. An example of the designation symbols for an American Standard buttress thread form is as follows:

6 – 10 (←*N BUTT*–2)

where

6 = *basic major diameter of* 6.000 *inches*

10 = 10 *threads per inch*

(← = *internal member to push against external member)*

N BUTT = National Buttress Form

2 = *class of fit*

NOTE: A symbol such as "←(" indicates that the internal member is to pull against the external member.

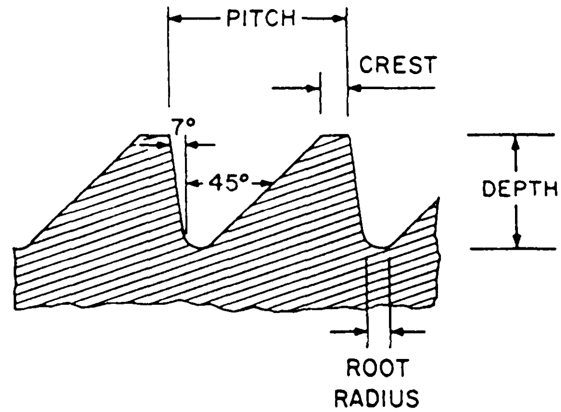


Figure 6-86.—Buttress thread.

The formulas for the basic dimensions of the American Standard buttress external thread are as follows:

$$\text{Pitch} = \frac{1}{n}$$

Width of flat at crest = 0.1631 × pitch

Root radius = 0.0714 × pitch

Depth of thread = 0.6627 × pitch

The classes of fit are 1 = free, 2 = medium, 3 = close. The specific dimensions involved concern the tolerance of the pitch diameter and the major diameter and vary according to the nominal or basic size. Consult a handbook for specific information on the dimensions for the various classes of fit.

Pipe Threads

American National Standard pipe threads are similar to the unified threads in that both have an included angle of 60° and a flat on the crest and the root of the thread. Pipe threads can be either tapered or straight, depending on the intended use of the threaded part. A description of the two types is given in the following paragraphs.

Tapered pipe threads are used to provide a pressure-tight joint when the internal and external mating parts are assembled correctly. Depending on the closeness of the fit of the mating parts, you may need to use a sealing tape or a sealer (pipe compound) to prevent leakage at the joint. The taper of the threads is 3/4 inch per foot. Machine and thread the section of pipe at this angle. The hole for the internal threads should be slightly larger than the minor diameter of the small end of the externally threaded part.

An example of a pipe thread is shown here.

NPT 1/4-18

where

NPT = tapered pipe thread

1/4 = *inside diameter of the pipe in inches*

18 = *threads per inch*

Figure 6-87 shows the typical taper pipe thread.

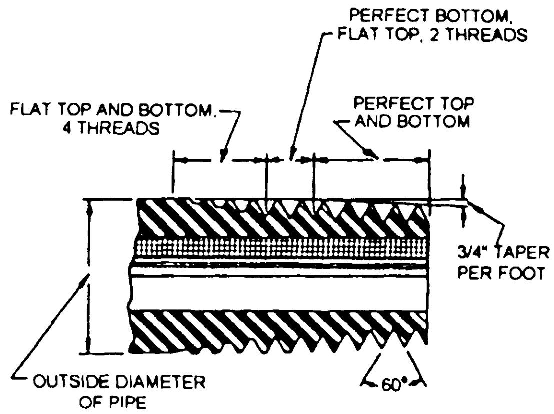


Figure 6-87.—Typical taper pipe thread.

Straight pipe threads are similar in form to tapered pipe threads except that they are not tapered. The same nominal outside diameter and thread dimensions apply. Straight pipe threads are used for joining components mechanically and are not satisfactory for high-pressure applications. Sometimes a straight pipe thread is used with a tapered pipe thread to form a low-pressure seal in a vibration-free environment.

CLASSES OF THREADS

Classes of fit for threads are determined by the amount of tolerance and allowance allowed for each particular class. The tolerance (amount that a thread may vary from the basic dimension) decreases as the class number increases. For example, a class 1 thread has more tolerance than a class 3 thread. The pitch diameter of the thread is the most important thread element in controlling the class of fit. The major diameter for an external thread and the minor diameter or bore size for an internal thread are also important, however, since they control the crest and root clearances more than the actual fit of the thread. A brief description of the different classes of fit follows:

- Classes 1A and 1B: Class 1A (external) and class 1B (internal) threads are used where quick and easy assembly is necessary and where a liberal allowance is required to permit ready assembly, even with slightly bruised or dirty threads.

- Classes 2A and 2B: Class 2A (external) and class 2B (internal) threads are the most commonly used threads for general applications, including production of bolts, screws, nuts, and similar threaded fasteners.

- Classes 3A and 3B: Class 3A (external) and class 3B (internal) threads are used where closeness of fit and accuracy of lead and angle of thread are important. These threads require consistency that is available only through high quality production methods combined with a very efficient system of gauging and inspection.

- Class 5: Class 5 threads are interference-fit threads in which the external threaded member is larger than the internally threaded member when both members are in the free state and which, when assembled, become the same size and develop a holding torque through elastic compression, plastic movement of the material, or both. There are a number of different thread designations within class 5. They distinguish between external and internal threads and the types of material the external thread will be driven into. This information may be found in the interference-fit threads section of *Machinery's Handbook.*

MEASURING SCREW THREADS

Thread measurement is needed to ensure that the thread and its mating part will fit properly. It is important that you know the various measuring methods and the calculations that are used to determine the dimensions of threads.

The use of a mating part to estimate and check the needed thread is common practice when average accuracy is required. The thread is simply machined until the thread and the mating part will assemble. A snug fit is usually desired with very little, if any, play between the parts.

You will sometimes be required to machine threads that need a specific class of fit, or you may not have the mating part to use as a gauge. In these cases, you must measure the thread to make sure you get the required fit.

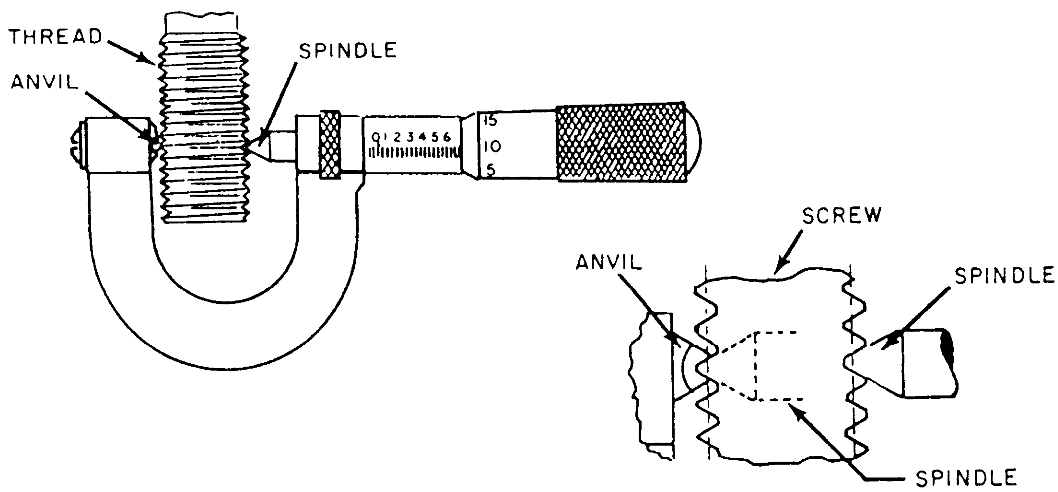


Figure 6-88.—Measuring threads with a thread micrometer.

An explanation of the various methods normally available to you is given in the following paragraphs.

Thread micrometers are used to measure the pitch diameter of threads. They are graduated and read in the same manner as ordinary micrometers. However, the anvil and spindle are ground to the shape of a thread, as shown in figure 6-88. Thread micrometers come in the same size ranges as ordinary micrometers: 0 to 1 inch, 1 to 2 inches, and so on. In addition, they are available in various pitch ranges. The number of threads per inch must be within the pitch range of the thread.

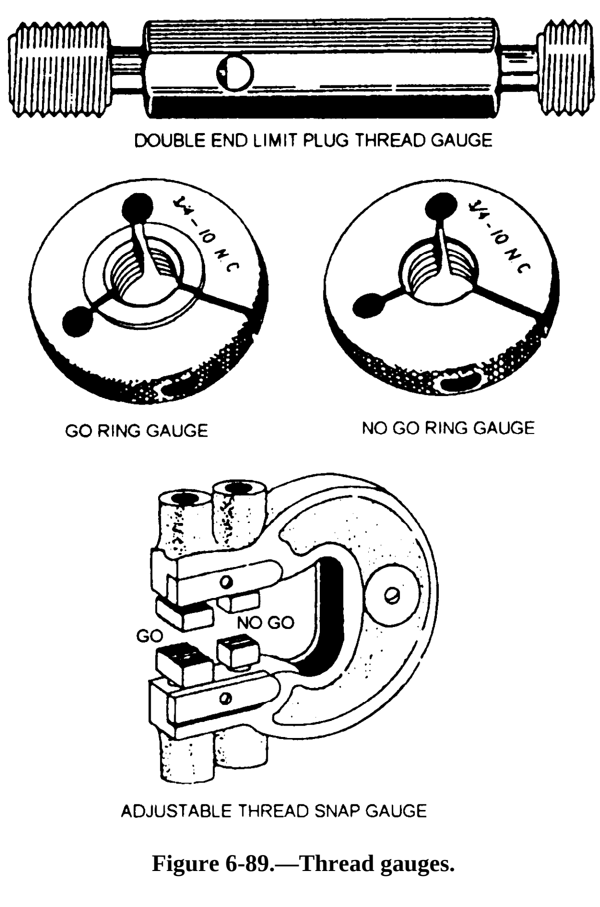


Figure 6-89.—Thread gauges.

Go and no-go gauges, such as those shown in figure 6-89, are often used to check threaded parts. The thread should fit the “go” portion of the gauge, but should not screw into or onto the “no-go” portion. Ring and plug gauges are available for the various sizes and classes of fit of thread. They are probably the most accurate method of checking threads because they envelop the total thread form, and in effect, check not only the pitch diameter and the major and minor diameters, but also the lead of the thread.

The pitch diameter of a thread can be accurately measured by an ordinary micrometer and three wires, as shown in figure 6-90.

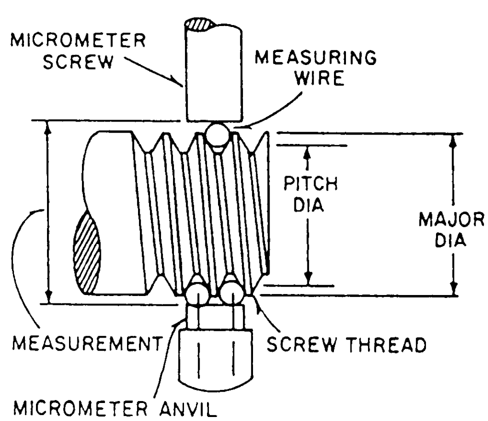


Figure 6-90.—Measuring threads using three wires.

The wire size you should use to measure the pitch diameter depends on the number of threads per inch. You will obtain the most accurate results when you use the **best wire size.** The best size is not always available, but you will get satisfactory results if you use wire diameters within a given range. Use a wire size as close as possible to the best wire size. To determine the wire sizes, use these formulas:

Best wire size = 0.57735 inch × pitch

Smallest wire size = 0.56 inch × pitch

Largest wire size = 0.90 inch × pitch

For example, the diameter of the best wire for measuring a thread that has 10 threads per inch is 0.0577 inch, but you could use any size between 0.056 inch and 0.090 inch.

NOTE: The wires should be fairly hard and uniform in diameter. All three wires must be the same size. You can use the shanks of drill bits as substitutes for the wires.

Use the following formulas to determine what the measurement over the wires should be for a given pitch diameter.

Measurement = pitch diameter - (0.86603 × pitch) + (3 × wire diameter)

$$M = PD - (0.86603 \times xP) + (3 \times W)$$

Use the actual size of the wires in the formula, not the calculated size.

Example: What should the measurement be over the wires for a 3/4-10 UNC-2A thread? First, determine the required pitch diameter for a class 2A 3/4-10 UNC thread. You can find this information in charts in several handbooks for machinists. The limits of the pitch diameter for this particular thread size and class are between 0.6832 and 0.6773 inch. Use the maximum size (0.6832 inch) for this example. Next, calculate the pitch for 10 threads per inch. The formula, "one divided by the number of threads per inch", will give you pitch $= \frac{1}{n}$. For 10 TPI, the pitch is 0.100 inch. As previously stated, the best wire size for measuring 10 TPI is 0.0577 inch, so assume that you have this wire size available. Now make the calculation. The data collected so far arr:

Thread = 3/4-10 UNC – 2A

Pitch diameter (PD) = 0.6832 in.

Pitch (P) = 0.100 in.

Wire size (W) = 0.0577 in.

The standard formula for the measurement over the wires was $M = PD - (0.86603 \times P) + (3 \times W)$. Enter the collected data in the correct positions of the formula:

$$M = 0.6832\ in. - (0.86603\ in \times 0.100\ in.) + (3 \times 0.0577\ in.)$$

$$M = 0.6832\ in. - 0.086603\ in. + 0.1731\ in.$$

$$M = 0.769697\ in.$$

The measurement over the wires should be 0.769697 in. or when rounded to four decimal places, 0.7697 in.

As mentioned in the beginning of the section on classes of threads, the major diameter is a factor also considered in each different class of fit. The basic or nominal major diameter is seldom the size actually machined on the outside diameter of the part to be threaded. The actual size is smaller than the basic size. In the case of the 3/4 - 10 UNC - 2A thread, the basic size is 0.750 in.; however, the size that the outside diameter should be machined to is between 0.7482 and 0.7353 in.

CUTTING SCREW THREADS ON THE LATHE

Screw threads are cut on the lathe by connecting the headstock spindle of the lathe with the lead screw through a series of gears to get a positive carriage feed. The lead screw is driven at the required speed in relation to the headstock spindle speed. You can arrange the gearing between the headstock spindle and lead screw so that you can cut any desired pitch. For example, if the lead screw has 8 threads per inch and you arrange the gears so the headstock spindle revolves four times while the lead screw revolves once, the thread you cut will be four times as fine as the thread on the lead screw, or 32 threads per inch. With the quick-change gear box, you can quickly and easily make the proper gearing arrangement by

Deleted—No permission granted for electronic copy.

Figure 6-91.—Compound rest set at 29 1/2°.

placing the levers as indicated on the index plate for the thread desired.

When you have the lathe set up to control the carriage movement for cutting the desired thread pitch, your next consideration is shaping the thread. Grind the cutting tool to the shape required for the form of the thread to be cut, that is-V-form, Acme, square, and so on.

Mounting Work in the Lathe

When you mount work between lathe centers for cutting screw threads, be sure the lathe dog is securely attached before you start to cut the thread. If the dog should slip, the thread will be ruined. Do not remove the lathe dog from the work until you have completed the thread. If you must remove the work from the lathe before the thread is completed, be sure to replace the lathe dog in the same slot of the driving plate.

When you thread work in the lathe chuck, be sure the chuck jaws are tight and the work is well supported. Never remove the work from the chuck until the thread is finished.

When you thread long slender shafts, use a follower rest. You must use the center rest to support one end of long work that is to be threaded on the inside.

Position of Compound Rest for Cutting Screw Threads

Ordinarily on threads of fine lead, you feed the tool straight into the work in successive cuts. For coarse threads, it is better to set the compound rest at one-half of the included angle of the thread and feed in along the side of the thread. For the last few finishing cuts, you should feed the tool straight in with the cross-feed of the lathe to make a smooth, even finish on both sides of the thread.

In cutting V-form threads and when maximum production is desired, it is customary to place the compound rest of the lathe at an angle of 29 1/2°, as shown in part A of figure 6-91. When you set the compound rest in this position and use the compound rest screw to adjust the depth of cut, you remove most of the metal by using the left side of the threading tool (B of fig. 6-91). This permits the chip to curl out of the way better than if you feed the tool straight in, and keeps the thread from tearing. Since the angle on the side of the threading tool is 30°, the right side of the tool will shave the thread smooth and produce a better finish; although it does not remove enough metal to interfere with the main chip, which is taken by the left side of the tool.

Using the Thread-Cutting Stop

Because of the lost motion caused by the play necessary for smooth operation of the change gears, lead screw, half-nuts, and so forth, you must withdraw the thread-cutting tool quickly at the end of each cut. If you do not withdraw the tool quickly, the point of the tool will dig into the thread and may break off.

To reset the tool accurately for each successive cut and to regulate the depth of the chip, use the thread-cutting stop.

First, set the point of the tool so that it just touches the work, then lock the thread-cutting stop by turning the thread-cutting stop screw (A of fig. 6-92) until the shoulder is tight against the stop (B of fig. 6-92). When you are ready to take the first cut, run the tool rest back by turning the cross-feed screw to the left several times, and move the tool to the point where the thread is to start. Then, turn the cross-feed screw to the right until the thread-cutting stop screw strikes the thread-cutting stop. The tool is now in the original position. By turning the compound rest feed screw in 0.002 inch or 0.003 inch, you will have the tool in a position to take the first cut.

For each successive cut after returning the carriage to its starting point, you can reset the tool accurately to its previous position. Turn the cross-feed screw to the right until the shoulder of the screw (A) strikes the stop (B). Then, you can regulate the depth of the next cut by adjusting the compound rest feed screw as it was for the first chip.

Be sure to use a coolant or lubricant while cutting the threads. This will help to cool the work and to wash the chips away.

For cutting an internal thread, set the adjustable thread-cutting stop with the head of the adjusting screw on the inside of the stop. Withdraw the tool by moving it toward the center or axis of the lathe.

You can use the micrometer collar on the cross-feed screw in place of the thread-cutting stop, if you desire. To do this, first bring the point of the threading tool up so that it just touches the work; then adjust the micrometer collar on the cross-feed screw to zero. Make all adjustments for obtaining the desired depth of cut with the compound rest screw. Withdraw the tool at the end of each cut by turning the cross-feed screw to the right one turn, stopping at zero. You can then adjust the compound rest feed screw for any desired depth.

Deleted—No permission granted for electronic copy.

Figure 6-92.—Adjustable thread-cutting stop mounted on carriage saddle (clamped to dovetail).

Engaging the Thread Feed Mechanism

When cutting threads on a lathe, clamp the half-nuts over the lead screw to engage the threading feed and release the half-nut lever at the end of the cut by means of the threading lever. Use the threading dial to determine when to engage the half-nuts so the cutting tool will follow the same path during each cut. When an index mark on the threading dial aligns with the witness mark on its housing, engage the half-nuts. For some thread pitches you can engage the half-nuts only when certain index marks are aligned with the witness mark. On most lathes you can engage the half-nuts as follows:

- For all even-numbered threads per inch, close the half-nuts at any line on the dial.
- For all odd-numbered threads per inch, close the half-nuts at any numbered line on the dial.
- For all threads involving one-half of a thread in each inch, such a 11 1/2, close the half-nuts at any odd-numbered line.

Cutting the Thread

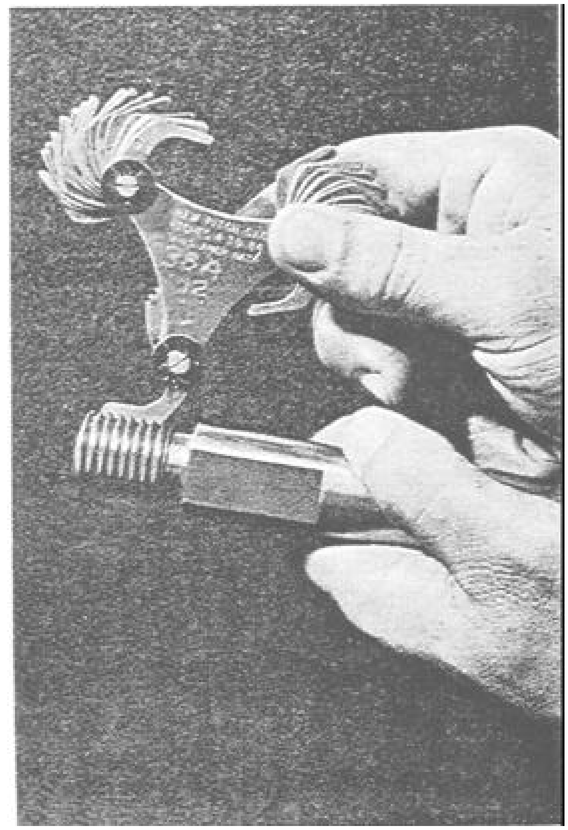
After setting up the lathe, as explained previously, take a very light trial cut just deep enough to scribe a line on the surface of the work, as shown in view A of figure 6-93. The purpose of this trial cut is to be sure that the lathe is arranged for cutting the desired pitch of thread.

To check the number of threads per inch, place a rule against the work, as shown in view B of figure 6-93, so that the end of the rule rests on the point of a thread or on one of the scribed lines. Count the scribed lines between the end of the rule and the first inch mark. This will give the number of threads per inch.

It is quite difficult to accurately count fine pitches of screw threads. A screw pitch gauge, used as illustrated in figure 6-94, is very convenient for checking the finer screw threads. The gauge consists of a number of sheet metal plates in which are cut the exact forms of threads of the various pitches; each plate is stamped with a number indicating the number of threads per inch for which it is to be used.

If the thread-cutting tool needs resharpening or gets out of alignment or if you are chasing the threads on a previously threaded piece, you must reset the tool so it will follow the original thread groove. To reset the tool, you may (1) use the compound rest feed screw and cross-feed screw to jockey the tool to the proper position, (2) disengage the change gears and turn the spindle until the tool is positioned properly, or (3) loosen the lathe dog (if used) and turn the work until the tool is in the proper position in the thread groove. Regardless of which method you use, you will usually have to reset the micrometer collars on the cross-feed screw and the compound rest screw.

Before adjusting the tool in the groove, use the appropriate thread gauge to set the tool square with the workpiece. Then, with the tool a few thousandths of an inch away from the workpiece, start the machine and engage the threading mechanism. When the tool has moved to a position near the groove into which you plan to put the tool, such as that shown by the solid tool in figure 6-95, stop the lathe without disengaging the thread mechanism.



28.153

Figure 6-44—Strew pitch gauge.

To reset the cutting tool into the groove, you will probably use the compound rest and cross-feed positioning method. By adjusting the compound rest slide forward or backward, you can move the tool laterally to the axis of the work as well as toward or away from the work. When the point of the tool coincides with the original thread groove (phantom view of the tool in fig. 6-95), use the cross-feed screw to bring the tool point directly into the groove. When you get a good fit between the cutting tool and the thread groove, set the micrometer collar on the cross-feed screw on zero and set the micrometer

Deleted—No permission granted for electronic copy.

Figure 6-93.—The first cut.

Deleted—No permission granted for electronic copy.

Figure 6-95.—Tool must be reset to original groove.

Deleted—No permission
granted for electronic copy.

Figure 6-96.—Finishing the end of a threaded piece.

collar on the compound rest feed screw to the depth of cut previously taken.

NOTE: Be sure that the thread mechanism is engaged and the tool is set square with the work before adjusting the position of the tool along the axis of the workpiece.

If it is inconvenient to use the compound rest for readjusting the threading tool, loosen the lathe dog (if used); turn the work so that the threading tool will match the groove, and tighten the lathe dog. If possible, however, avoid doing this.

Another method, which is sometimes used, is to disengage the reverse gears or the change gears; turn the headstock spindle until the point of the threading tool enters the groove in the work, and then reengage the gears.

The end of a thread may be finished by any one of several methods. The 45° chamfer on the end of a thread, as shown in view A of figure 6-96, is commonly used for bolts and capscrews. For machined parts and special screws, the end is often finished by rounding it with a forming tool, as shown in view B of figure 6-96.

LEFT-HAND SCREW THREADS

A left-hand screw (fig. 6-97) turns counter-clockwise when advancing (looking at the head of the screw), or just the opposite to a right-hand screw.

Deleted—No permission
granted for electronic copy.

Figure 6-97.—A left-hand screw thread.

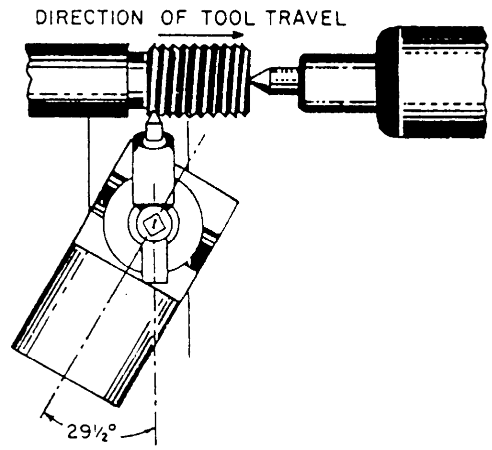


Figure 6-98.—Setup for left-hand external threads.

Left-hand threads are used for the cross-feed screws of lathes, the left-hand end of axles, one end of a turnbuckle, or wherever an opposite thread is desired.

The directions for cutting a left-hand thread on a lathe are the same as those for cutting a right-hand thread, except that you swivel the compound rest to the left instead of to the right. Figure 6-98 shows the correct position for the compound rest. The direction of travel for the tool differs from a right-hand thread in that it moves toward the tailstock as the thread is being cut.

Before starting to cut a left-hand thread, it is good practice, if feasible, to cut a neck or groove into the workpiece. (See fig. 6-97). Such a groove enables you to run the tool in for each pass, as you do for a right-hand thread.

Make the final check for both diameter and pitch of the thread, whether right-hand or left-hand, with the nut that is to be used, or with a ring thread gauge if one is available. The nut should fit snugly without play or shake, but it should not bind on the thread at any point.

MULTIPLE SCREW THREADS

A multiple thread, as shown in figure 6-99, is a combination of two or more threads, parallel to each other, progressing around the surface into which they are cut. If a single thread is thought of as taking the form of a helix, that is of a string or cord wrapped around a cylinder, a multiple thread may be thought of as several cords lying side by side and wrapped around a cylinder. There may be any number of threads, and they start at equally spaced intervals around the cylinder. Multiple threads are used when

rapid movement of the nut or other attached parts is desired and when weakening of the thread must be avoided. A single thread having the same lead as a multiple thread would be very deep compared to the multiple thread. The depth of the thread is calculated according to the pitch of the thread.

The tool selected for cutting multiple threads has the same shape as that of the thread to be cut and is similar to the tool used for cutting a single thread except that greater side clearance is necessary. The helix angle of the thread increases as the number of threads increases. The general method for cutting multiple threads is about the same as for single screw threads, except that the lathe gearing must be based on the lead of the thread (number of single threads per inch), and not the pitch, as shown in figure 6-99. Provisions must also be made to obtain the correct spacing of the different thread grooves. You can get the proper spacing by using the thread-chasing dial, setting the compound rest parallel to the ways, using a faceplate, or using the change gear box mechanism.

The use of the thread-chasing dial (fig. 6-100) is the most desirable method for cutting 60° multiple threads. With each setting for depth of cut with the compound, you can take successive cuts on each of the multiple threads so that you can use thread micrometers.

To determine the possibility of using the thread-chasing dial, first find out if the lathe can be geared to cut a thread identical to one of the multiple threads. For example, if you want to cut 10 threads per inch, double threaded, divide the number of threads per inch (10) by the multiple (2) to get the number of single threads per inch (5). Then, gear the lathe for 5 threads per inch.

To use the thread-chasing dial on a specific machine, refer to instructions usually found attached

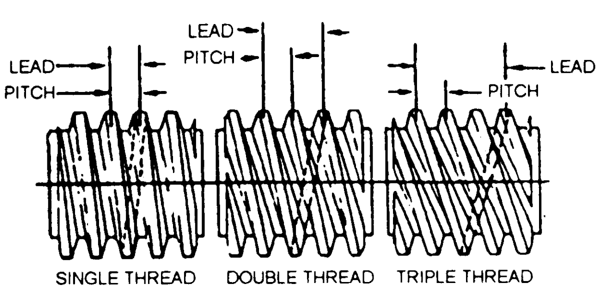


Figure 6-99.—Comparison of single- and multiple-lead threads.

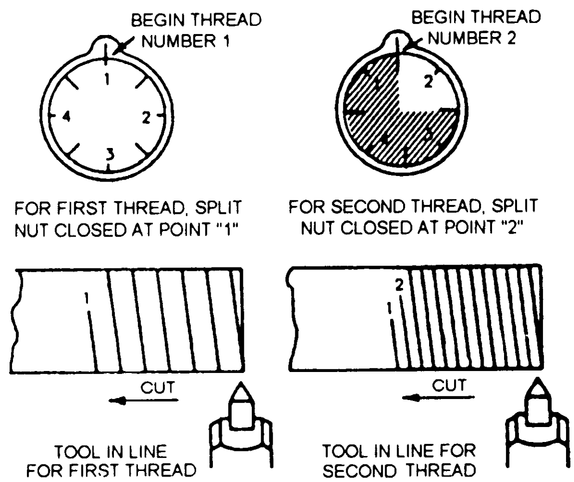


Figure 6-100.—Cutting multiple threads using the thread-chasing dial.

to the lathe apron. To cut 5 threads per inch, on most lathes, engage the half-nut at any numbered line on the dial, such as points 1 and 2 shown in figure 6-100. The second groove of a double thread lies in the middle of the flat surface between the grooves of the first thread. Engage the half-nut to begin cutting the second thread when an unnumbered line passes the index mark, as shown in figure 6-100. To ensure that you cut each thread to the same depth, engage the half-nut first at one of the numbered positions and cut in the first groove. Then, engage the half-nut at an unnumbered position so that alternate cuts bring both thread grooves down to size together. To cut a multiple thread with an even number of threads, first use the thread-chasing dial to cut the first thread. Then, use one of the other multiple thread-cutting procedures to cut the second thread.

Cutting of multiple threads by positioning the compound rest parallel to the ways should be limited to square and Acme threads. To use this method, set the compound rest parallel to the ways of the lathe and cut the first thread to the finished size. Then, feed the compound rest and tool forward, parallel to the thread axis a distance equal to the pitch of the thread and cut the next thread.

The faceplate method of cutting multiple threads involves changing the position of the work between centers for each groove of the multiple thread. One method is to cut the first thread groove in the conventional manner. Then, remove the work from between centers and replace it between centers so the tail of the dog is in another slot of the drive plate, as

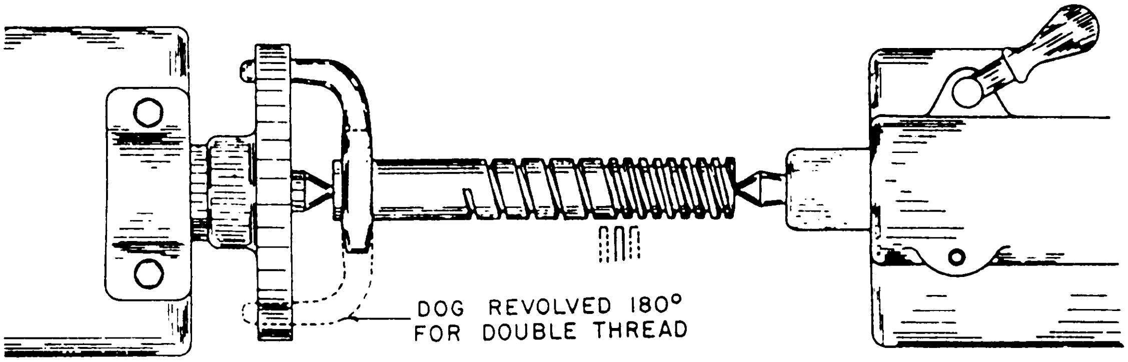


Figure 6-101.—Use of faceplate.

shown in figure 6-101. Two slots are necessary for a double thread, three slots for a triple thread, and so on. The number of multiples you can cut by this method depends on the number of equally spaced slots there are in the drive plate. There are special drive or index plates available, so that you can accurately cut a wide range of multiples by this method.

Another method of cutting multiple threads is to disengage either the stud gear or the spindle gear from the gear train in the end of the lathe after you cut a thread groove. Then, turn the work and the spindle the required part of a revolution, and reengage the gears for cutting the next thread. If you are to cut a double thread on a lathe that has a 40-tooth gear on the spindle, cut the first thread groove in the ordinary manner. Then, mark one of the teeth on the spindle gear that meshes with the next driven gear. Carry the mark onto the driven gear, in this case the reversing gear. Also, mark the tooth diametrically opposite the marked spindle gear tooth (the 20th tooth of the 40-tooth gear). Count the tooth next to the marked tooth as tooth number one. Then disengage the gears by placing the tumbler (reversing) gears in the neutral position, turn the spindle one-half revolution or 20 teeth on the spindle gear, and reengage the gear train. You may index the stud gear as well as the spindle gear. If the ratio between the spindle and stud gears is not 1 to 1, you will have to give the stud gear a proportional turn, depending upon the gearing ratio. The method of indexing the stud or spindle gears is possible only when you can evenly divide the number of teeth in the gear indexed by the multiple desired. Some lathes have a sliding sector gear that you can readily insert into or remove from the gear train by shifting a lever. Graduations on the end of the spindle show when to disengage and to reengage the sector gear for cutting various multiples.

THREADS ON TAPERED WORK

Use the taper attachment when you cut a thread on tapered work. If your lathe does not have a taper attachment, cut the thread on tapered work by setting over the tailstock The setup is the same as for turning tapers.

Part A of figure 6-102 shows the method of setting the threading tool with the thread gauge when you use the taper attachment. Part B of figure 6-102 shows the same operation for using the tailstock setover method.

Note that in both methods illustrated in figure 6-102, you set the threading tool square with the axis by placing the center gauge on the straight part of the work, NOT on the tapered section. This is very important.

Deleted—No permission granted for electronic copy.

Figure 6-102.—Cutting thread on tapered work.