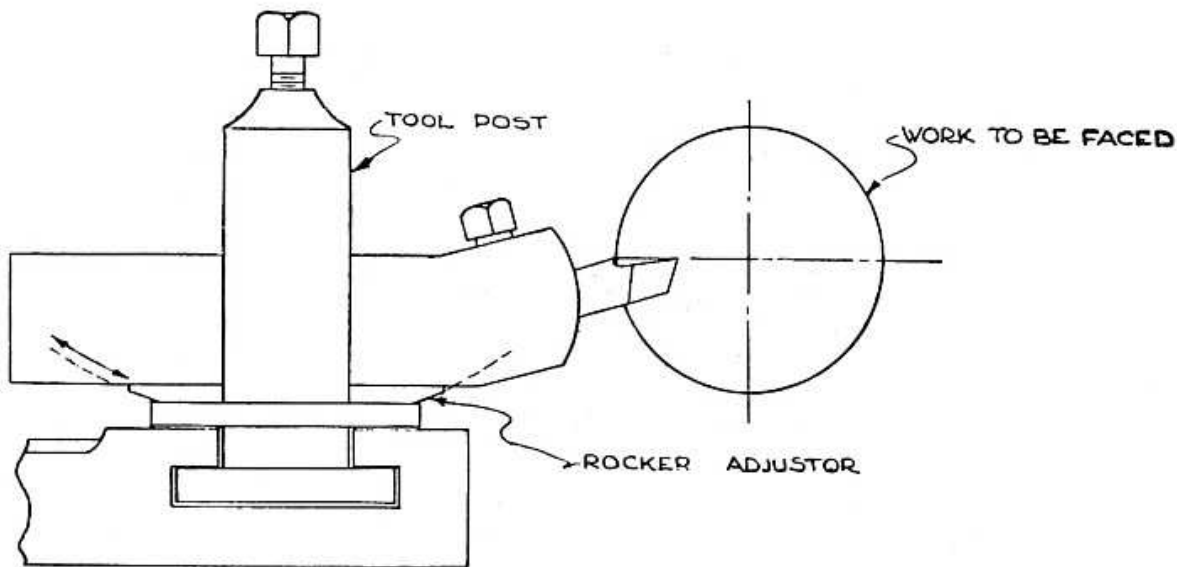## Section V

GENERAL LATHE OPERATIONS

FIGURE 34.—Tool bit position for facing.

22. General.—While most turning jobs differ in nature, many fundamental operations are carried on in the same manner for all classes of work. These may be considered as general operations and will be described in subsequent paragraphs.

23. Facing.—*a.* Square finishing of the ends of the work to a desired length is referred to as facing and may be done either in the lathe chuck or between centers.

b. In order to face a surface in a lathe, the cutting edge of the tool bit must be set at the center of the axis of rotation to obtain a smooth surface. The point of the tool bit may be raised or lowered by shifting the tool post rocker (fig. 34), which is used to support the

tool holder. After centering the work and clamping the tool holder in position, the tool bit is moved to the outer edge of the work by means of the cross feed screw. The lathe may then be started and the tool bit brought up carefully against the revolving work to the depth of cut desired. A clamp screw is provided on the carriage to hold it in position on the ways when taking a radial cut and should be locked during the facing operation. The tool bit is fed across the work either by hand or power with a feed approximately 0.010 inch per revolution. As the tool approaches center, its position relative to the work should be noted and, if necessary, readjusted to intersect the center exactly. When taking a heavy cut, it is best to start at the outer edge of the work and move toward the center as de-
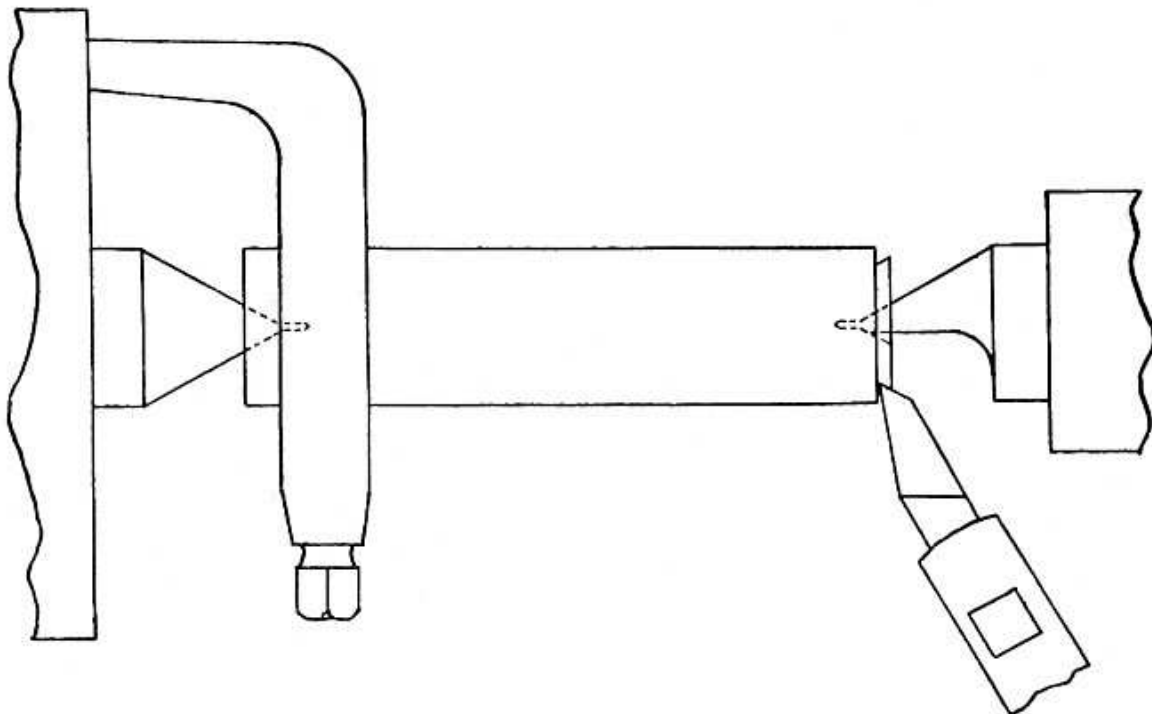

FIGURE 35.—Facing on centers.

scribed above, although, when taking a light cut the tool may be moved in either direction.

c. Facing between centers is usually accomplished by supporting the end of the work to be faced upon a half center and continuing as described above, except that in this operation the tool bit must be ground with a more pointed nose to allow it to cut clear to the center hole (fig. 35).

24. Centering work.—a. The drilling of center holes is generally accomplished by one of the following methods:

(1) By revolving the work to be drilled in the lathe chuck and feeding the center drill by means of the tailstock spindle.

(2) By drilling in the drill press.

(3) By having the center drill in the lathe headstock and holding the work by hand.

b. When holding the work to be drilled in the lathe chuck, its center should be spotted with a small hole by using a sharp pointed tool.

c. The proper size of center drill should be selected in order to make the center hole large enough to carry the load of machining, but not so large as to be unsightly or interfere in any manner with operations that follow. A table of center drill sizes for various diameters is given below:

Diameter of work (inches)	Diameter of combination drill and countersink	
	Drill	Countersink
½	0. 043	⅛
1	. 09375	³⁄₁₀
1½	. 128	³⁄₁₀
2	. 157	⁷⁄₁₆

d. When finished the center hole should be smooth and have a polished appearance. If it is rough or improperly drilled it will wear, causing the work to become loose and scoring the center. Figure 36
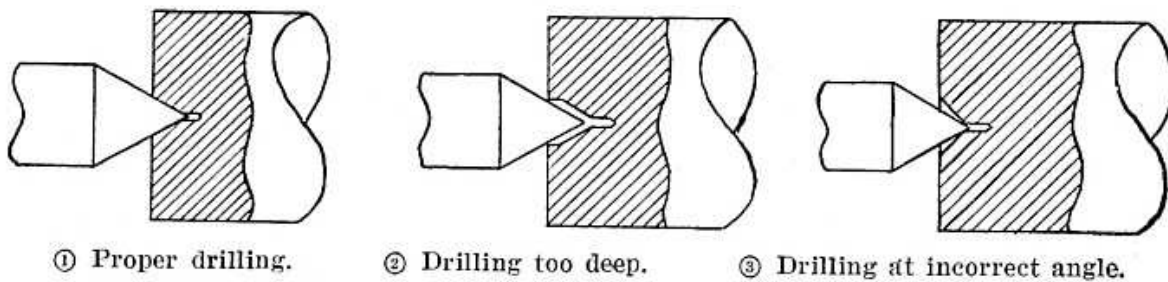

① Proper drilling.   ② Drilling too deep.   ③ Drilling at incorrect angle.
FIGURE 36.—Correct and incorrect drilling of center holes.

shows properly and improperly drilled center holes.

e. When drilling the center hole in the drill press or lathe, the center must be located by one of the methods shown in figure 37 and lightly center punched.

f. The four principal conditions which may cause the drill to break when drilling center holes are as follows:

(1) *Dull center drill.*—The drill must be sharp and in good condition.

(2) *Tailstock off center.*—The zero lines on the tailstock and cricket must be in line during any drilling operation.

(3) *Lathe speed too slow.*—The lathe must turn at the proper r. p. m. for the size of center drill used.

① By hermaphrodite caliper.   ② By center head.

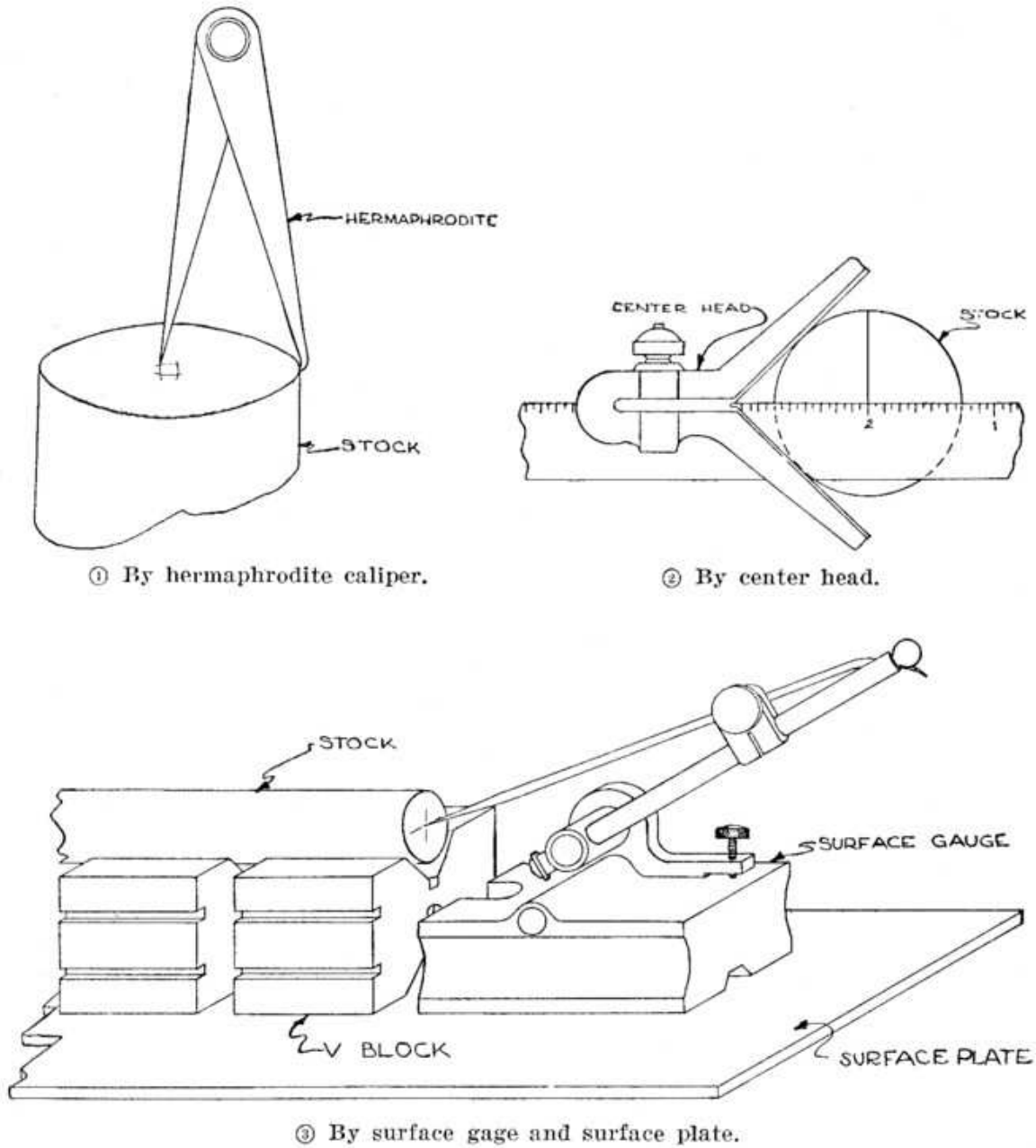

③ By surface gage and surface plate.

FIGURE 37.—Locating centers in round stock.

(4) *Insufficient cutting oil.*—The center drill must be well supplied with a good cutting oil and fed slowly.

25. Grinding centers.—The plain or standard lathe centers are ordinarily furnished with the lathe. Due to insufficient lubrication, misalinement, or normal wear, these centers require occasional regrinding which may be accomplished in the following manner (fig. 38):
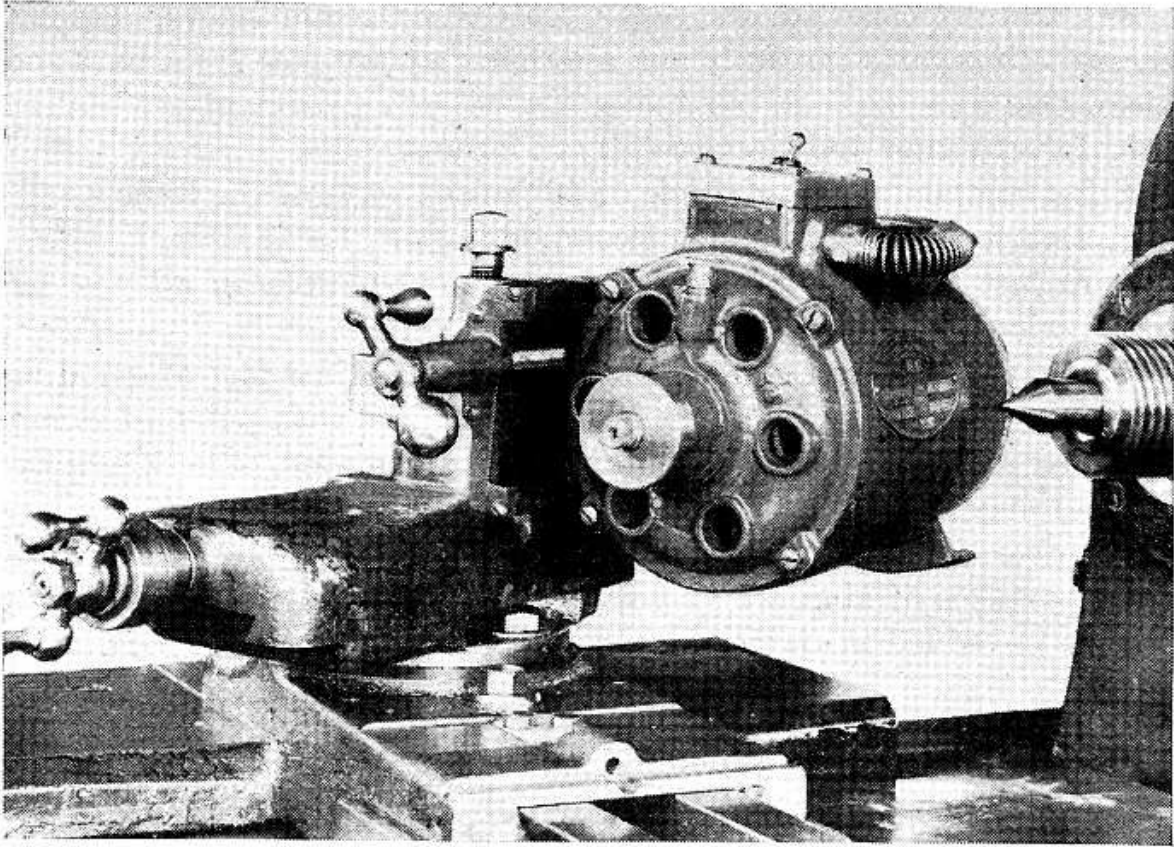
① Side view.
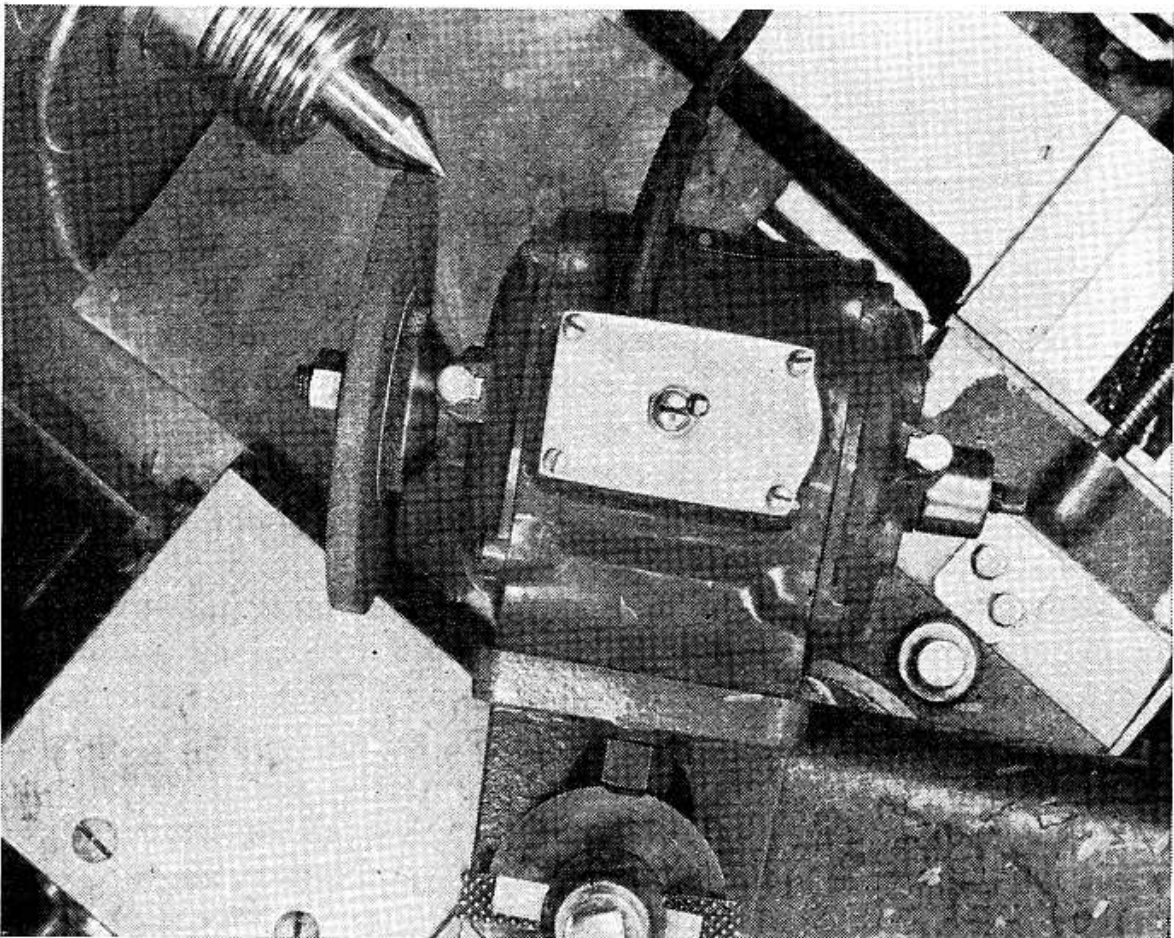
② Top view.

FIGURE 38.—Grinding lathe centers with tool post grinder

a. Carefully clean spindle taper, center, and center sleeve. Inspect these mating parts carefully and see that they are free from burs and blemishes.

b. Insert center and sleeve solidly in lathe spindle.

c. Set compound rest at angle of 30° to the ways of the lathe.

d. Lubricate dovetail slide of compound.

e. Adjust tapered gib at side of compound to remove all possible lost motion without causing it to bind.

f. Mount a tool post grinder in tool post and adjust face of grinding wheel parallel to conical surface of center. See that point of center is alined with center hole in end of arbor of grinder. Check grinder for end play and adjust if necessary.

g. Cover carriage and ways with paper or cloth to protect them from particles of abrasive grit from grinding wheel as this grit will become imbedded in the metal and rapidly destroy accuracy of lathe.
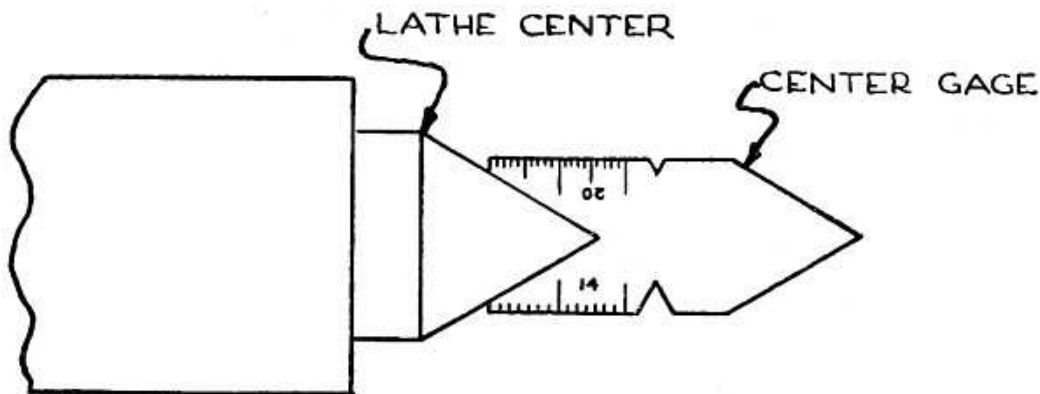

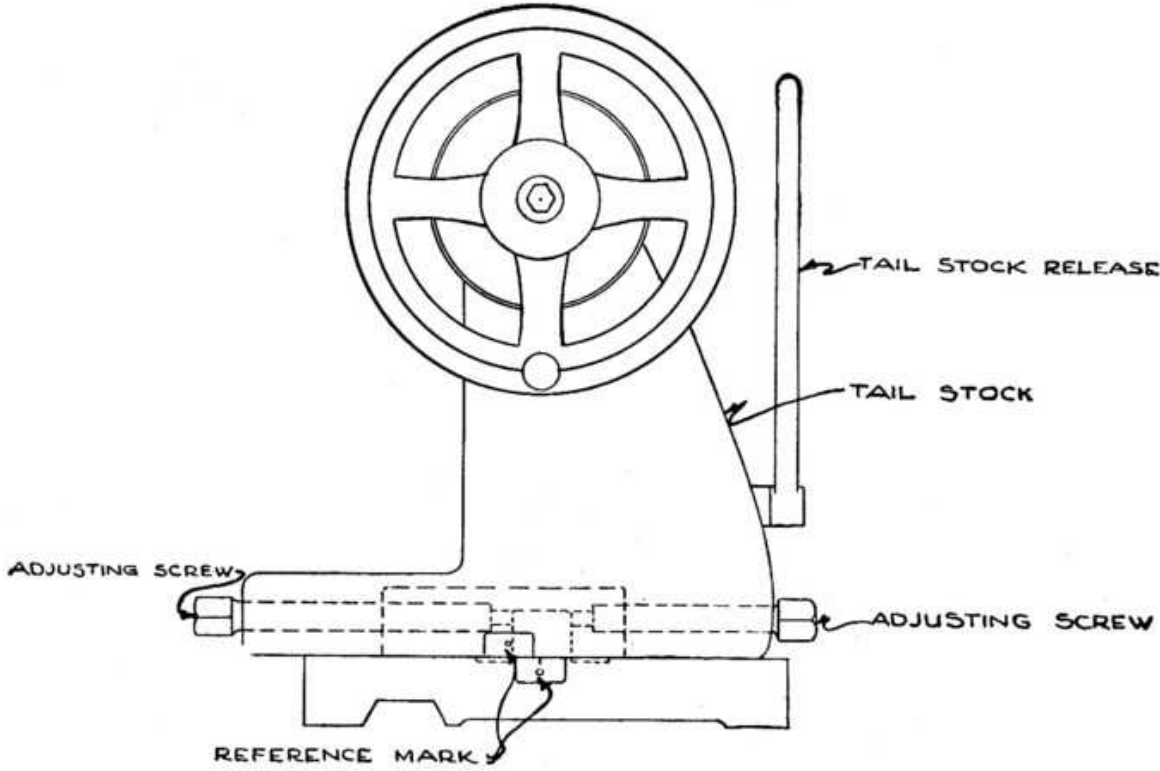FIGURE 39.—Testing 60° angle with center gage.

h. Set compound rest near center of its travel and move carriage and cross feed up until grinding wheel is within a few thousandths of an inch of the center. Lock carriage in this position to prevent movement during grinding operation.

i. Start lathe at a moderate speed (100 to 150 r. p. m.) and then start grinder, making sure that lathe and grinding wheel turn in the same direction of rotation.
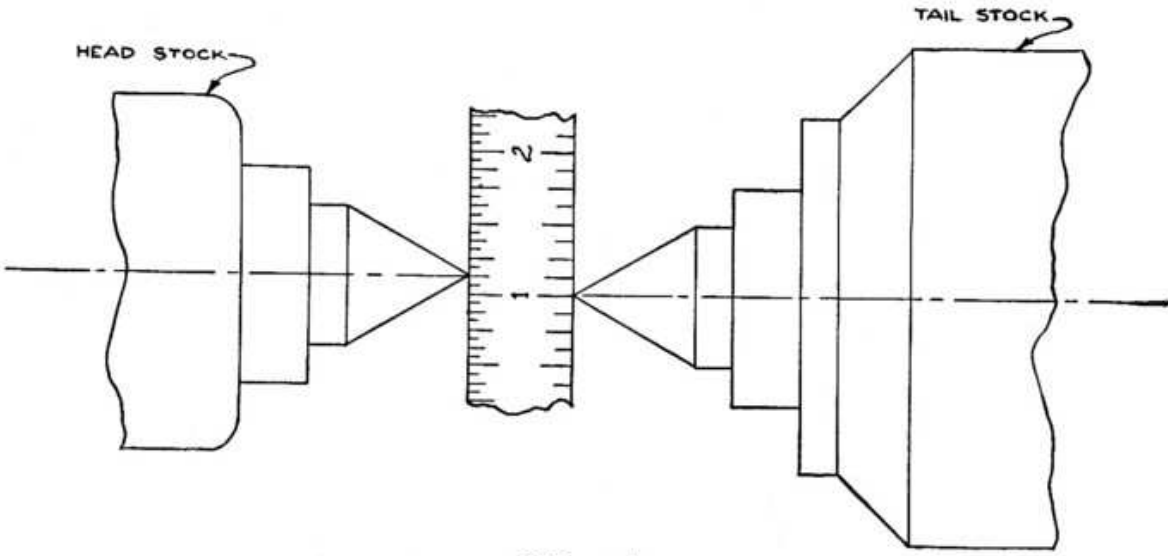
j. Move grinding wheel by means of cross feed until it just touches revolving center. Set depth of cut between 0.001 inch and 0.002 inch and carefully feed grinding wheel along face of center by hand feed, starting at small end of center and feeding toward headstock. Repeat operation until center is clean and smooth.

k. Using a 60° center gage (fig. 39), check angle of center. If it is not true, readjust compound accordingly and repeat above operations.

TAIL STOCK RELEASE
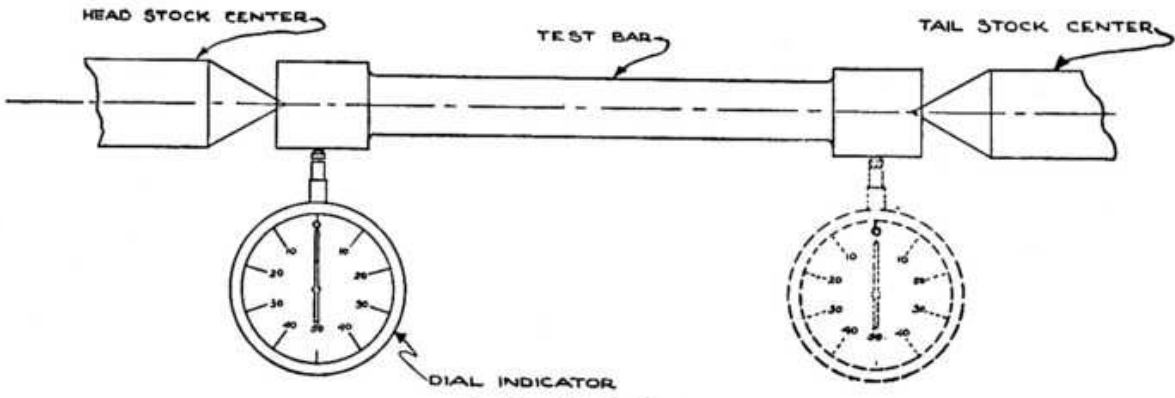
TAIL STOCK

ADJUSTING SCREW

ADJUSTING SCREW

REFERENCE MARK

① By reference marks.

HEAD STOCK

TAIL STOCK

② By rule.

HEAD STOCK CENTER

TEST BAR

TAIL STOCK CENTER

DIAL INDICATOR

③ By dial indicator.

FIGURE 40.—Alinement of centers.

l. Polish centers with fine emery cloth after they are ground, using a high spindle speed. The dead or tailstock center should be ground first and the live or headstock center last. This precaution will allow the live center to be left in place after grinding, giving it more accuracy.

26. Alining centers.—*a.* In order to turn a shaft straight and true between centers, it is necessary that the centers be in a plane parallel to the ways of the lathe. The tailstock may be moved laterally to accomplish this alinement by means of two adjusting screws after it has been released from the ways. At the rear of the tailstock on the cricket are two zero lines, and the centers are approximately alined when these lines coincide as shown in figure 40 ①. This approximate alinement may be checked by moving the tailstock up until the centers almost touch, and observing their relative positions as shown in figure 40 ②.

b. Positive alinement of centers may be obtained by placing a test bar between centers and bringing both ends of the bar to a zero reading by the use of an indicator clamped in the tool post as shown in figure 40 ③. It is necessary to have the tailstock clamped to the ways and the test bar properly adjusted between centers when taking the indicator readings.

c. Another method commonly employed by the mechanic for positive alinement is to take a light cut over the work held between centers. The work is then measured at each end with a micrometer and if the readings are found to differ the tailstock is adjusted accordingly. The procedure is then repeated until alinement is obtained.

27. Straight turning.—*a.* Straight turning may be performed upon work held in the chuck; however, the majority of work turned in the lathe is held between the lathe centers.

b. Centers must be in good condition and allowance should be made for expansion in the work so that it will not bind. When work is mounted and revolved between centers, there will be friction where the tailstock center bears in the work center hole. This bearing point must be lubricated at intervals, as the pressure of the tool bit against the work sets up considerable friction at this point. The ordinary machine oils do not have enough body to resist this pressure successfully, allowing metal-to-metal contact which will soon destroy the accuracy of the center and center hole. A good center lubricant can be made of white lead and machine oil mixed to about the consistency of thick cream.

c. For straight turning, the cutting edge of the tool bit should be slightly above center (fig. 41), as this position gives the tool bit a better

cutting action. The distance above center is governed by the diameter of the work and the end clearance of the tool bit. As a general rule, the point of contact is raised $\frac{3}{64}$ inch above center for each inch of work diameter.
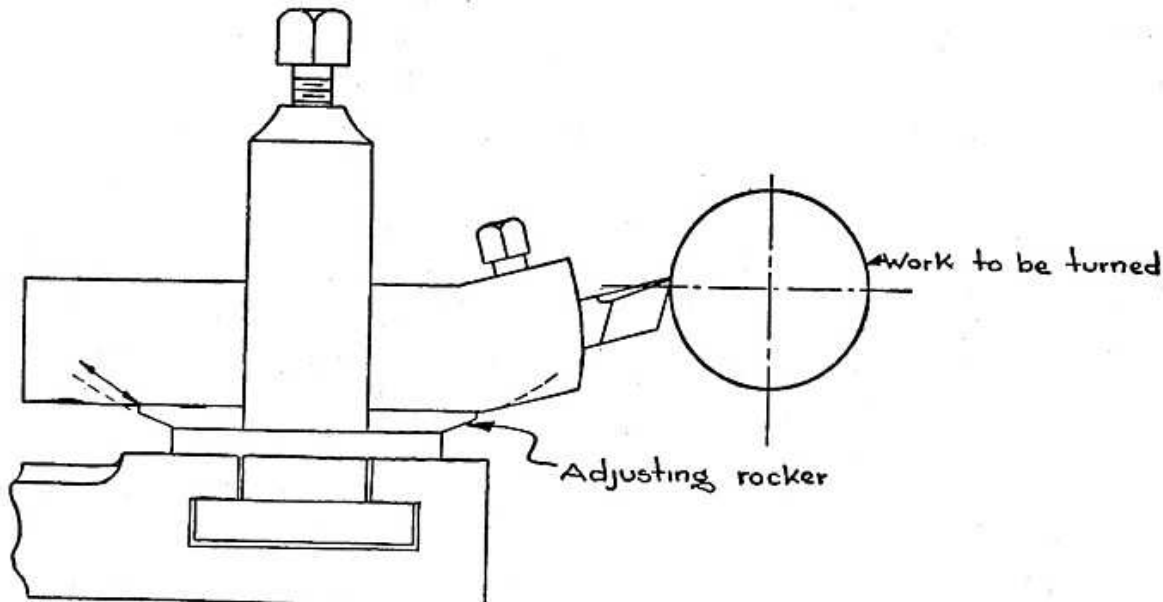

FIGURE 41.—Tool bit position for external work.

d. Centered work is usually driven by a dog of the bent tail type. Care must be taken to see that the tail of the dog does not wedge or bind in the drive plate slot as shown in figure 42 ②, thereby preventing the work from seating properly in the live center. Figure 42 ① illustrates a set-up using a dog of correct size. If the dog is clamped to a finished surface, some means should be provided to keep the set screw or dog from marring its finish. A piece of soft metal placed between the work and the dog (fig. 43) will generally give ample protection.

e. Care must be taken to avoid any condition that will allow the work to spring or vibrate during the turning operation. When the set-up is made, the tailstock spindle should be run out only as far as necessary. Rigidity may also be increased by clamping the tool bit and its holder as short as possible.

f. In general, for straight turning operations a roughing cut should first be made to between 0.010 inch and 0.020 inch of the finished diameter. The finish cuts may then be taken using a finishing tool and fine feed. The application of a suitable cutting lubricant is advisable where a high finish is desired.

28. Shoulder turning.—Shoulders in lathe work are usually of the filleted or square corner types.

a. Filleted shoulders, unless a special radius is designated, are cut with the round nose of the tool.

b. Square shoulders are cut with a shoulder tool, ground and set as illustrated in figure 44, after the work has been rough turned to within $\frac{1}{64}$ inch of its finished diameter and length. The following procedure applies only to the finishing operation:
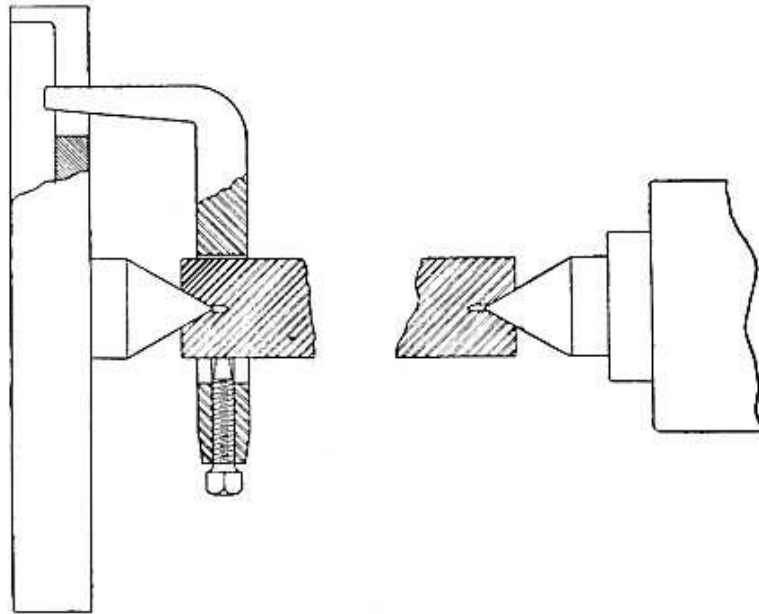
① Correct.
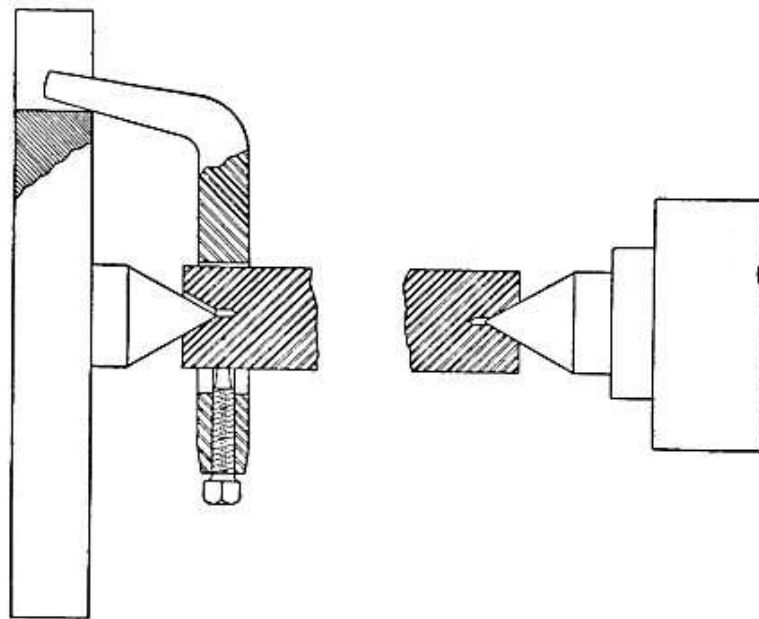
② Incorrect.

FIGURE 42.—Lathe dogs for turning between centers.

(1) With the compound set parallel to the ways, carefully cut out fillet left in corner by roughing tool.

(2) Using longitudinal feed, turn small diameter to finished size up to shoulder.

(3) Lock carriage and, using compound rest, feed tool amount needed to finish work to length.

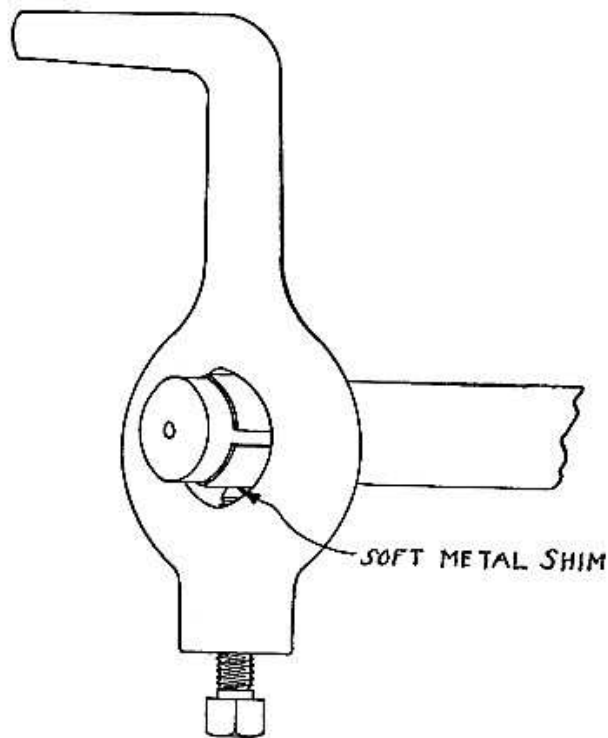(4) Engage cross slide power feed and allow it to carry tool away from shoulder.
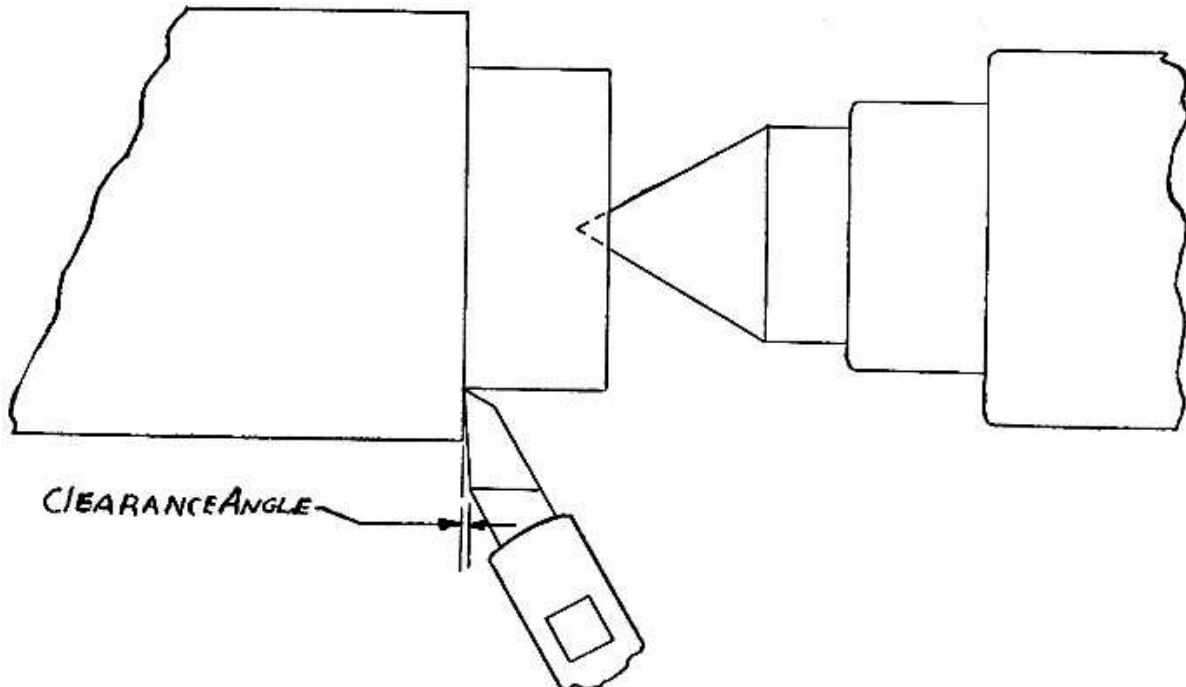

FIGURE 43.—Protecting work from lathe dog and set screw.

FIGURE 44.—Shoulder turning in lathe.

29. Taper turning.—*a.* Tapers may be turned on the lathe either by feeding at an angle with the compound rest, by setting over the tailstock center, or by means of the taper attachment. When turning tapers, the tool bit must be set at the height of the center at all times.

(1) The compound rest can be swiveled and locked at any angle (fig. 45) and is ideal for cutting short tapers. When turning tapers by this method the tool must be slowly and steadily fed by hand if a good finish is to be obtained on the work. Tapers are generally expressed in thousandths of an inch per foot, and it is necessary to convert the figure into angular measurement when using the compound rest, since
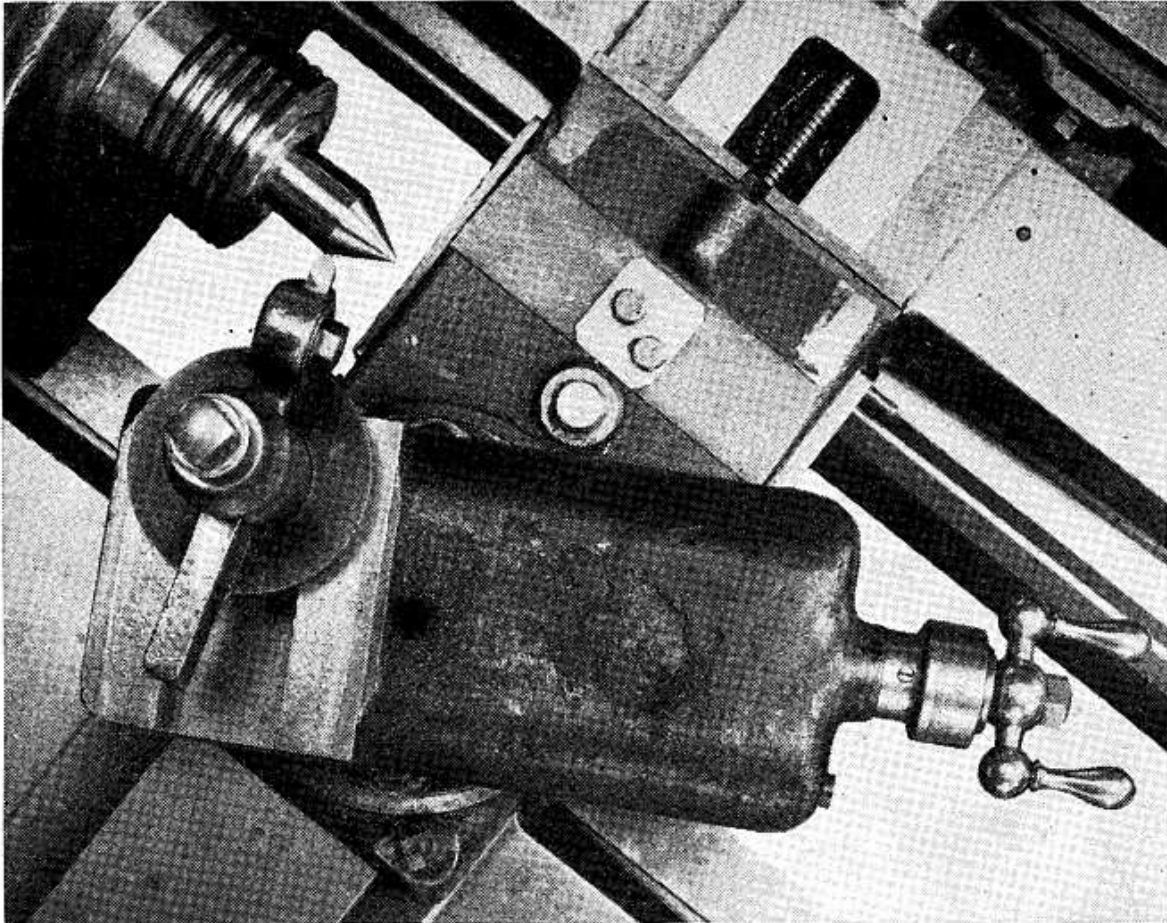

FIGURE 45.—Cutting taper with compound rest.

it is graduated in degrees. The taper is converted by means of the following formula:

$$\frac{\text{Taper per inch}}{2} = \text{tangent of angle.}$$

Two points that must be kept in mind when setting the compound rest for angular work are—

(a) Angular measurement in degrees is taken from a line parallel to the axis of the work or in alinement with the ways of the lathe.

(b) The angle to which the compound is set makes an included angle of twice that amount on the work.

(2) When turning taper by offsetting the tailstock center (fig. 46), it is first necessary to loosen the tailstock clamp. The tailstock slide may then be offset the required amount by means of the adjusting

screws located on either side of the cricket. Setting the tailstock slide toward the tool post results in a taper with the small diameter at the tailstock end of the work, while setting the tailstock slide away from the tool post puts the smaller diameter at the headstock end. In determining the proper offset for the tailstock slide it must be remembered that the amount of set-over varies with the over-all length of the piece being tapered. For example, the tailstock must be reset in order to cut the same taper on pieces of different lengths. The amount of offset is determined by the following formula:
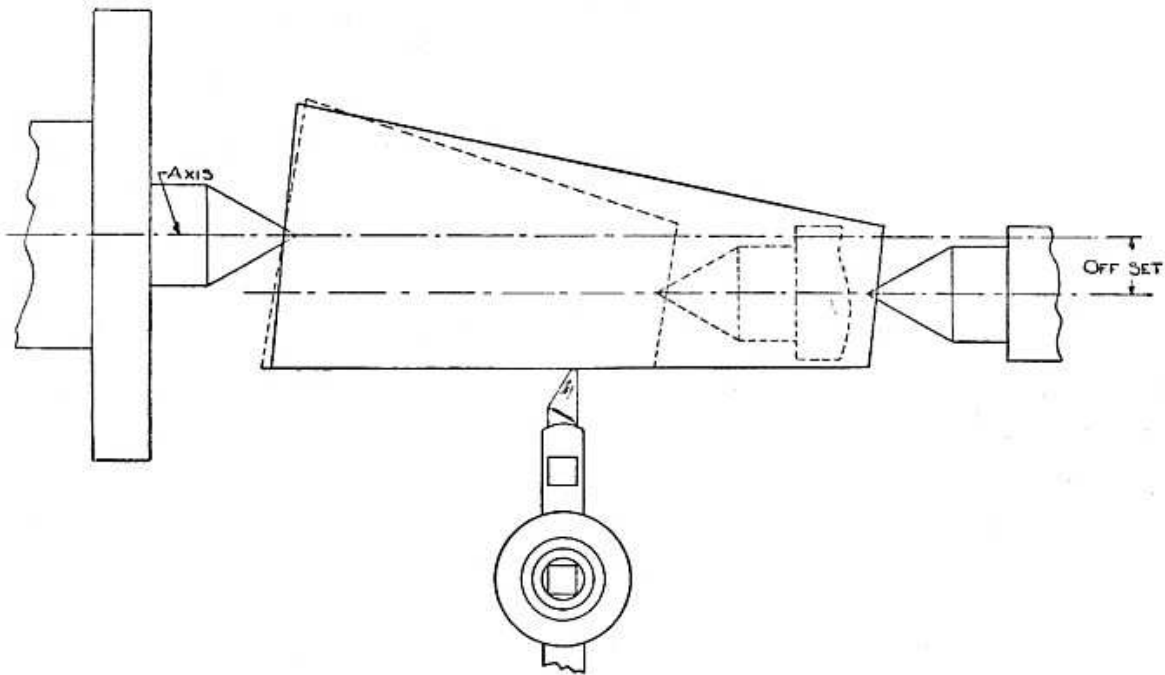

FIGURE 46.—Taper turning by offsetting center.

$$\frac{\text{Taper per inch}}{2} \times \text{length of work} = \text{set-over}.$$

(3) The taper attachment (fig. 47) is an attachment for the machining of all types of tapers, both internal and external. Its length of travel is considerably greater than that of the compound rest; however, the angle to which it may be set is limited. When in use the taper attachment is set to the angle desired by the graduated scale on the attachment bracket, which reads either in inches per foot or degrees of taper. A sliding guide block is set on the way of the taper attachment guide bar, which in turn is fastened to the lathe cross slide by a link and connecting screw. As the lathe carriage is moved, the guide block is caused to move along the guide bar which is set at the required angle to the axis of the lathe. This in turn causes the lathe cross slide to move back and forth at right angles to the axis of the lathe, according to the direction in which the carriage is traversed. The lathe cross slide

must be released from its feed screw during this operation so that it may be free to follow the taper set on the guide bar. Since the guide block must have a certain amount of clearance in its fit on the way of the guide bar (to give freedom of movement), there will be a slight amount of lost motion in the cross link when the travel of the carriage is reversed. This lost motion in the cross slide of the lathe must be compensated for by moving the carriage far enough past the end of
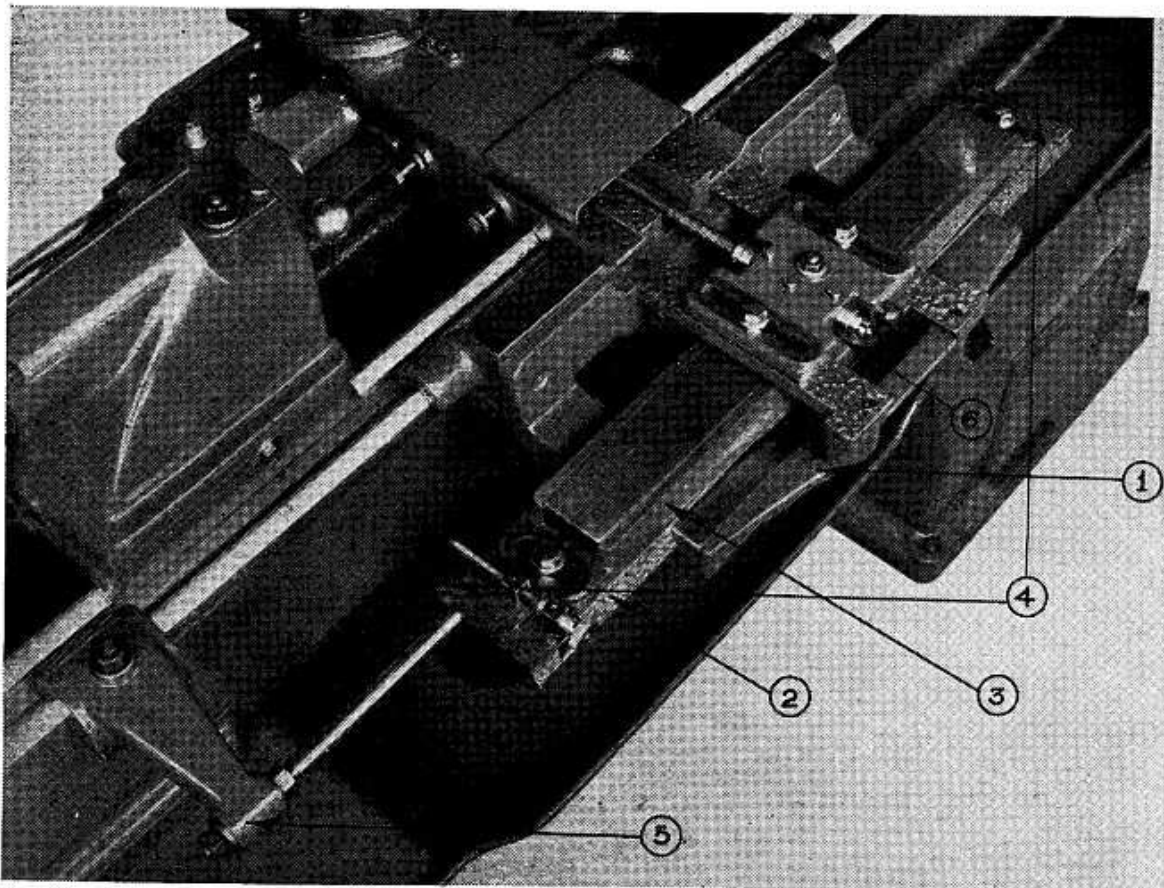
① Saddle.   ② Base.   ③ Guide bar.
④ Graduations.   ⑤ Clamp.   ⑥ Guide block.

FIGURE 47.—Lathe taper attachment.

the work so that when it is brought forward to cutting position the backlash will have been taken out, allowing the cross slide to follow the taper accurately. If the back lash of the guide bar and block is not eliminated, a portion of the work will be cut straight in place of tapered, and for this reason some lathe manufacturers have patented taper attachment backlash devices standardized on their machines.

b. Two methods of checking external tapers are—

(1) By drawing three light chalk lines along the length of the tapered portion of the work and then wringing the taper a part of a turn in a taper ring gage.

(2) By laying off equally spaced scriber lines around the tapered portion of the work and using a micrometer to determine differences in diameters between the marks.

30. Taper boring.—*a.* In general, the rules that are applicable to outside taper turning also apply to the boring of tapered holes. The cutting point of the tool is placed on center and, if the taper attachment is used, care must be exercised to eliminate the backlash of the slide fittings so that the hole will not be bored straight at the
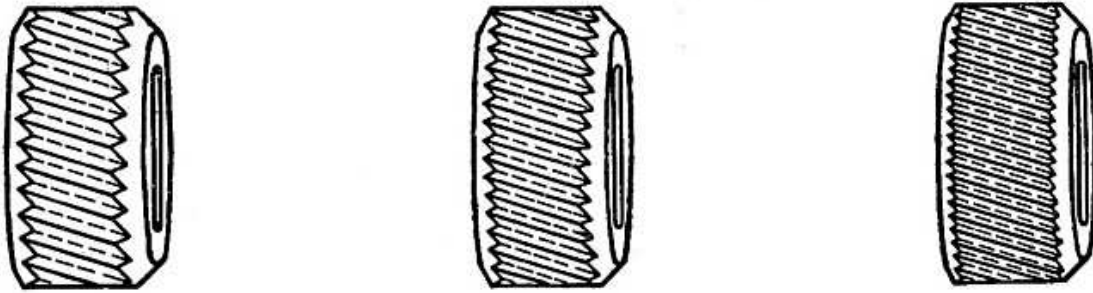
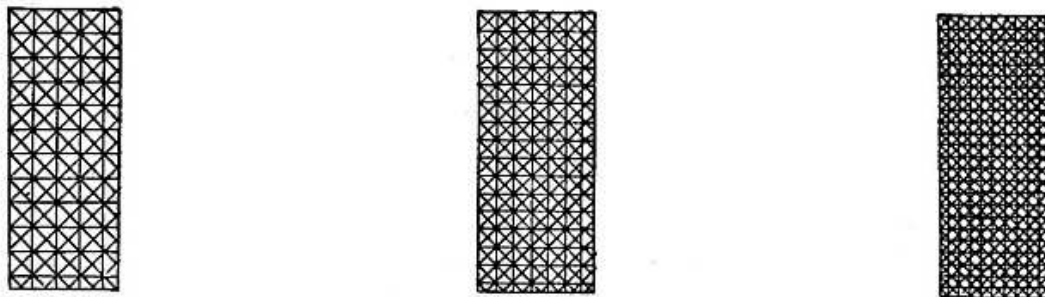COARSE KNURL   MEDIUM KNURL   FINE KNURL

FIGURE 48.—Knurl rolls and patterns.

start. The measurement of the size and taper of the hole is generally made with a taper plug gage by the cut and try method. After a cut or two has been taken, the bore is cleaned, the gage rubbed lightly with chalk, inserted in the hole, and twisted slightly to cause the chalk to show where the gage is bearing. Any necessary corrections may then be made and the boring continued until the taper is brought to size. A very light application of prussian blue to the gage will give better results than chalk for more accurate work.

b. When making a blind tapered hole, such as may be required in drill sockets, etc., it is best to drill the hole carefully to correct depth with a drill of the same size as specified for the small end of the hole. This gives the advantage of boring to the right size without the removal of metal at the extreme bottom of the bore, which is rather difficult, particularly in small, deep holes.

31. Knurling.—*a.* Knurling, as previously mentioned, is an operation of rolling depressions into the surface of the work by means of small wheels or rollers with helical teeth milled in their faces. Most knurling tools produce a diamond pattern (fig. 48), and are provided with interchangeable rollers for coarse, medium, and fine work.

b. For knurling, the lathe should be run at approximately one-half the normal cutting speed with the knurling tool pressed hard against the work. The longitudinal feed of the carriage may then be engaged and the knurling tool fed across the surface. A coarse power

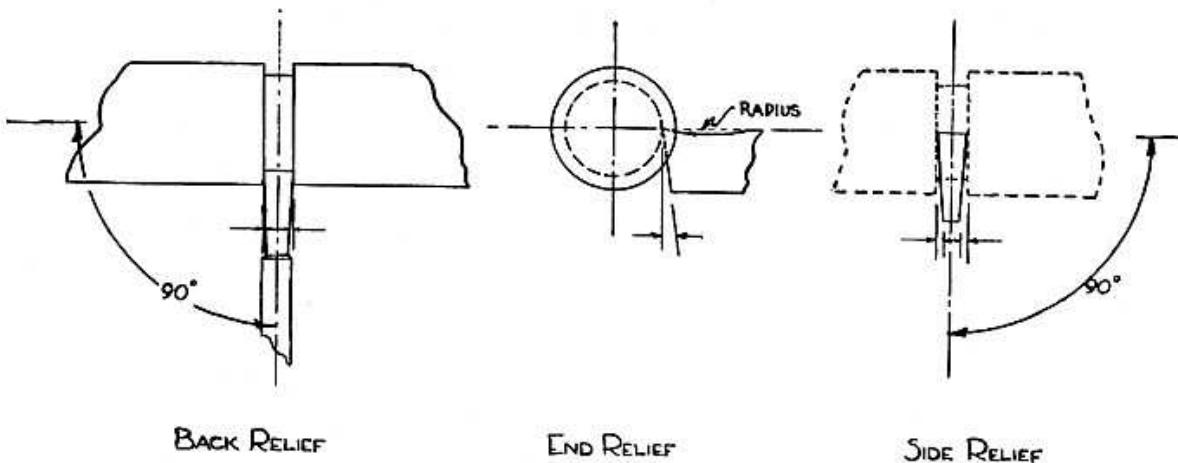BACK RELIEF   END RELIEF   SIDE RELIEF
FIGURE 49.—Parting tool positions.

feed is used, with plenty of oil when knurling any kind of material, and three to five passes over the work should be made to complete the knurl.

32. Parting.—*a.* Parting is accomplished in the lathe with a narrow blade tool bit held in a suitable holder. In this operation particular care must be exercised in the grinding and setting of the tool as well as in the adjustment of the machine.

b. The tool must be ground so that the cutting end has about 5° of relief, and better results will be obtained if the top of the cutter is tapered slightly toward the bottom of the blade. The cutting edge must be sharp, and the sides of the blade should be relieved about 1° to 2°. A slight concavity carried back from the cutting edge on the top of the tool will help the cutting action. This back rake may be produced by holding the tool upright on the grinding wheel and

allowing the curvature of the wheel to make the required concave cut. Figure 49 shows a correctly ground parting tool.

c. The cutting edge of the tool is set at center height with the sides of the blade parallel to the cross slide. The side relief of the tool should be checked after making the set-up to see that proper cutting clearance is provided. Work to be cut should be held in a chuck, preferably of the four-jawed type, with the point at which the parting is to take place as close as possible to the chuck jaws. Work that is too long to be held in this manner may be supported by the center rest but should never be placed between centers.

d. For successful parting, the lathe spindle bearings must fit snugly and the cross slide and compound rest gibs should be taken up fairly tight to avoid unnecessary lost motion. The cutting speed for parting should be comparable to turning speeds, and the feed should be sufficient to keep a thin chip coming from the work continuously. A power feed of approximately 0.002 inch per work revolution may be used for this operation, although it is best for the novice to use the hand feed, thereby retaining better control of the tool in case of emergency. If too much pressure is used on the cross feed the tool will gouge, and when not enough pressure is applied the tool will chatter.

e. Cast iron and brass may be parted dry, but all other materials, particularly ferrous metals, must be kept wet with cutting oil.

33. Filing and polishing.—*a.* The purpose of filing and polishing in the lathe is to remove toolmarks, improve the finish, or slightly reduce the dimension.

b. Mill files are generally considered best for lathe filing. The bastard cut file is used for roughing, and the second cut mill file for the finer class of work. Other types, such as the round, half round, and gulleting files, may also be used for such operations as filing radii, fillets, curves, etc.

c. The speed for filing ferrous metals should be four to five times the rough turning speed, while for nonferrous metals it should be only two to three times the roughing speed. If the speed is too high the file has a tendency to slide over the work, causing it to dull rapidly and glaze the work. If the speed is too slow, the work may be filed out-of-round.

d. In lathe filing, the file should be passed slowly over the revolving work so that the work will have made several revolutions before the stroke has been completed. The file is held at an angle of about 10° to the right and moved with a slow, sliding motion from left to right so that the teeth will have more of a shearing action. The motion of stroke and file angle should never be the opposite, as this

will cause chatter marks on the work. The pressure exerted on the file with the hands should be less than when filing at the bench. Since there are less teeth in contact with the work, the file must be cleaned frequently to avoid scratching. Since filing should be used for little more than to remove lathe toolmarks from work, only 0.002 inch to 0.005 inch should be left for the filing operation. Figure 50 illustrates the correct method of filing on the lathe.
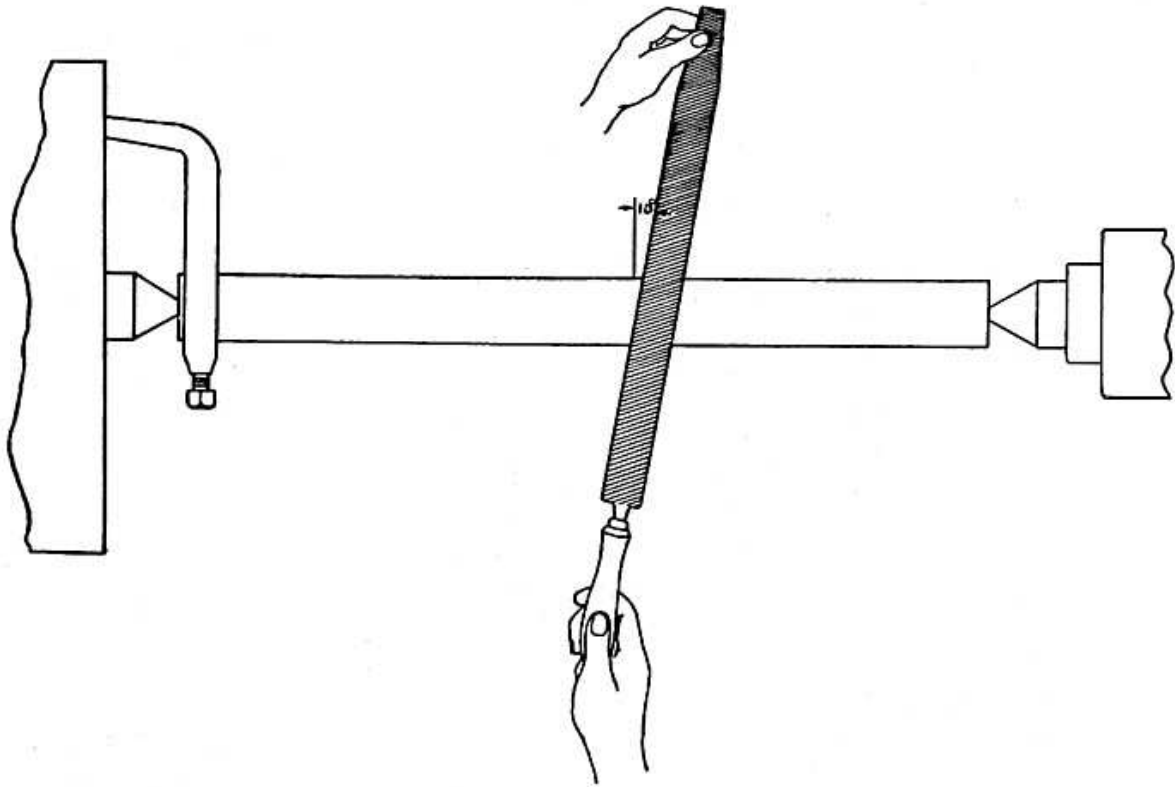

FIGURE 50.—Filing on lathe.

e. Polishing with either emery cloth or sandpaper is desirable to improve the surface after filing. Emery cloth is best for the ferrous metals, whereas sandpaper gives better results on nonferrous materials. The most effective speed for polishing with ordinary abrasives is approximately 5,000 f. p. m. Most lathes are not capable of speeds this high, particularly on small work, and it is therefore necessary to select as high a speed as conditions will permit. In most cases the cloth or paper is held directly in the hand and applied to the work, although it may be held or tacked over a stick of wood and used in the same manner as a file. Polishing clamps made of wood may also be used on plain round work such as shafting, etc.

f. Since polishing will slightly reduce the dimensions of the work, 0.00025 inch to 0.0005 inch should be allowed for this operation. The use of oil with the abrasive will give the work a dull satin finish, while dry polishing leaves a bright surface. Crocus cloth may be

used, following the emery cloth or sandpaper, to give a finish of higher luster.

g. The center adjustment and lubrication should be watched closely while polishing, as the high speed and friction of the cloth will cause rapid expansion of the work with resultant loss of center lubrication.

34. Radius turning and boring.—*a.* The forming of curves, radii, and other irregular shapes, both internal and external, is generally accomplished either by free hand guiding of the cutting tool or by the use of specially ground forming tools.

(1) A considerable amount of manual dexterity is required for free hand guiding of a tool to form an outline that will be smooth of surface and sufficiently near the proper shape and dimensions. The sizing of such shapes may be greatly assisted by the repeated use of calipers at various dimensional points. Templates may also be used to advantage in the production of irregular outlines when applied in the following manner:

(*a*) Scribe outline of work on a piece of tin.

(*b*) Clamp this tin template to ways of lathe and attach an improvised movable pointer to lathe saddle.

(*c*) Set tool bit at place desired on work and bend pointer to corresponding place on template.

(*d*) With the work revolving at the proper speed, manually operate both cross and longitudinal feeds so as to keep pointer following lay-out reproducing its shape on the work.

(*e*) File and polish to improve finish and remove any small irregularities that may be present.

(2) Specially ground form tools are particularly adaptable for accurately producing small, irregular shapes, although some difficulty will probably be encountered with their use since the bearing surface of such a tool is rather broad. More pressure and machine power is needed to make it cut than is necessary for the ordinary cutting tool, and there is a considerable tendency for it to tear the work.

b. The foregoing methods must not be considered accurate. When accuracy of shape is necessary the tool travel should be controlled by some type of mechanical generating agent. For the forming of small radii and curves of simple nature, these may be composed of swivel-type tool holders. For larger radii, link mechanisms may be attached to the machine to give the desired results.

35. Spring winding on lathe.—Springs may be easily and quickly wound in the lathe, on mandrels of appropriate size, using annealed spring wire. When the spring winding mandrel diameter is larger than $\frac{3}{8}$ inch, the mandrel is mounted in the lathe chuck. Mandrels less than $\frac{3}{8}$ inch in diameter are mounted in a drill chuck,

held in the lathe headstock. In fastening the wire, one chuck jaw is loosened and the wire placed between the chuck jaw and the mandrel. It is good practice to support the outer end of the mandrel with the tailstock center to prevent distortion. The wire is placed in a lathe boring tool holder, together with a piece of brass, having a notch filed in it and tightened just enough to prevent slippage. Tension or compression springs of different pitches may be wound in this manner by selectively gearing the longitudinal carriage feed. (See fig. 51.)
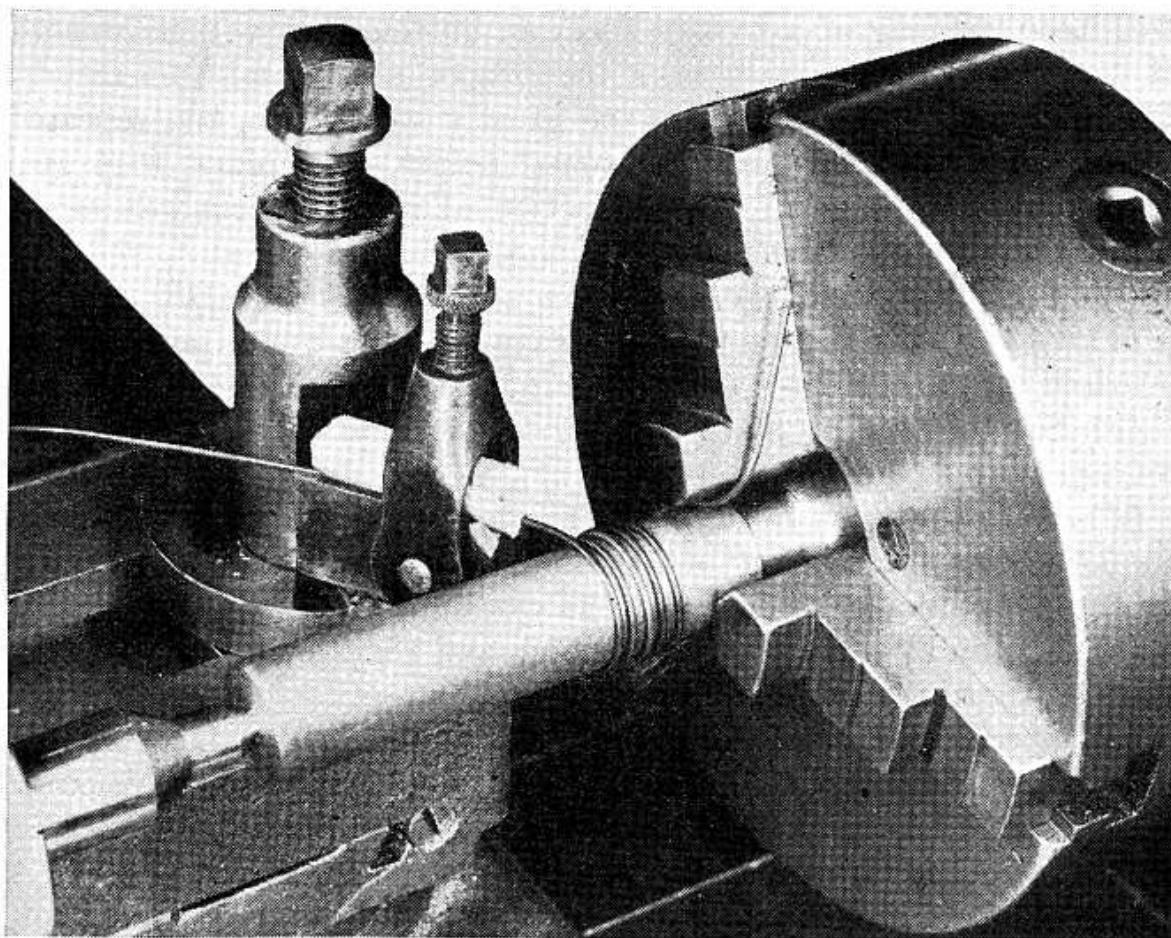
FIGURE 51.—Spring winding on lathe.

SECTION VI

DRILLING, REAMING, AND BORING

36. Drilling.—*a.* Drilling is ordinarily done in the lathe by holding and revolving the work in the lathe chuck, with the drill suitably held in the tailstock and fed by means of the tailstock hand

screw. Drilling may also be done by holding and revolving the drill in a chuck or in the headstock spindle taper with the work held in some attachment, such as a drill pad or crotch center.

(1) Straight shank drills are usually held in a drill chuck which is placed in the tailstock. This method may also be used for holding center drills and other small, straight shank tools.

(2) Taper shank drills may be held directly in the tailstock spindle so long as they are a good fit. In this case, some precautionary measure should be taken to prevent any turning of the drill with resultant damage to the taper in the tailstock spindle. A drill held in this manner, by means of a lathe dog, is illustrated in figure 52 ①.

(3) A drill holder may be used for holding the larger size drills in the lathe as shown in figure 52 ②. This device is composed of a steel socket, bored to fit the drill shank and having an arm projecting at right angles to its body. A large center hole is provided at the heel of the socket to fit the lathe tailstock center, and the arm is allowed to rest on the carriage to keep the drill from revolving. While in operation, the holder should be held tightly to the center by the pressure of the projecting arm against the tool post. Under no circumstances should the tailstock be withdrawn or loosened while the lathe is running. To withdraw the drill, it is necessary to stop the lathe completely.
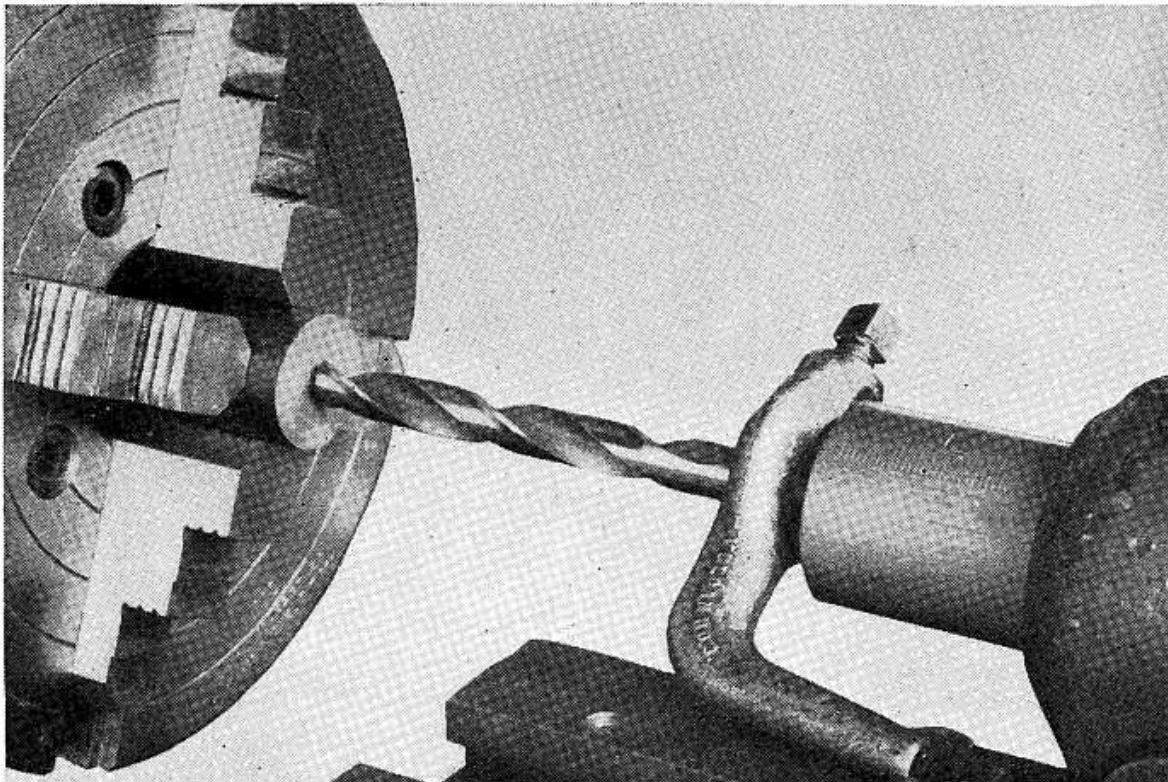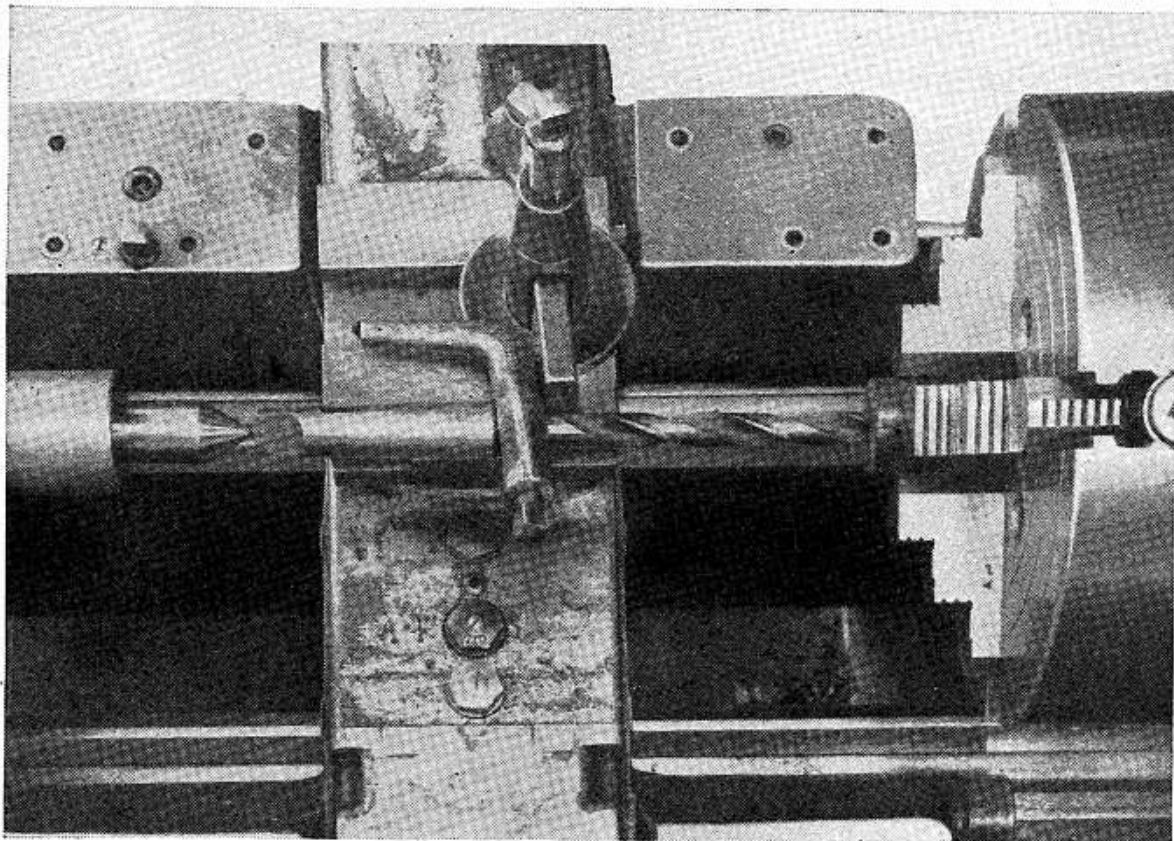

① In tailstock spindle.
Figure 52.—Method of holding drill in lathe.
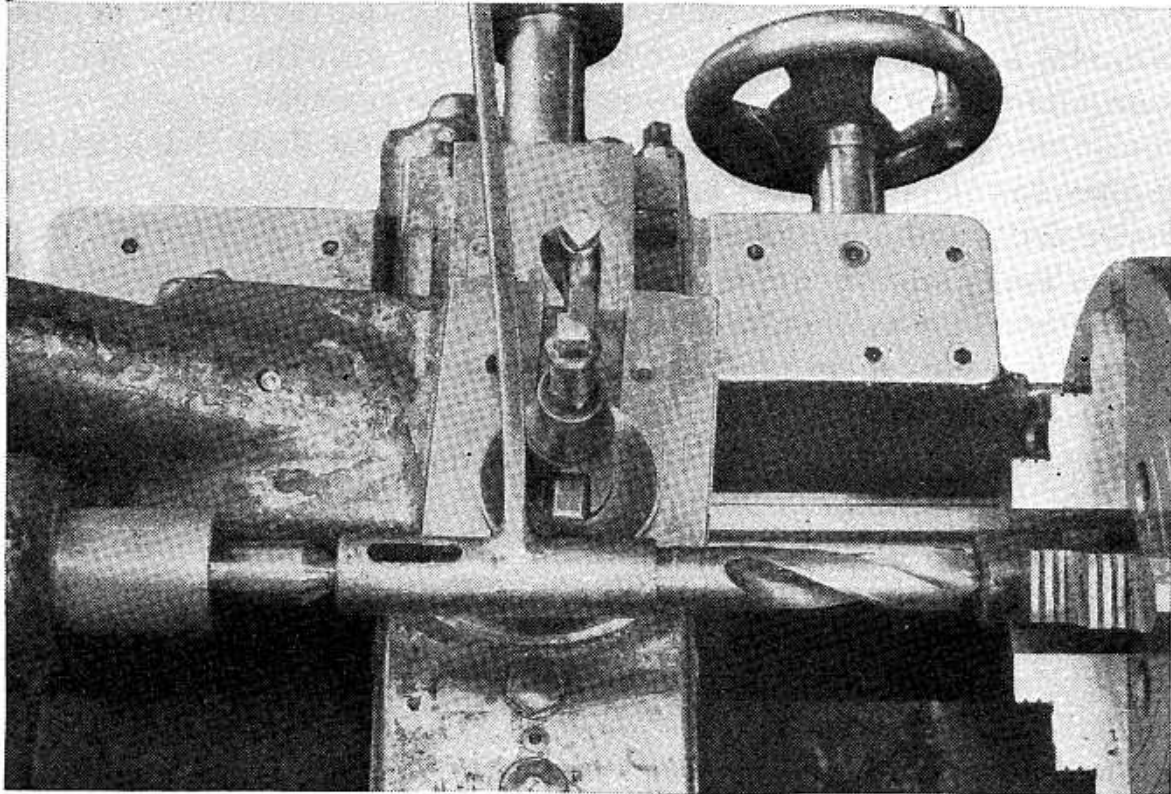
② In drill holder.

③ On tailstock center.

FIGURE 52.—Method of holding drill in lathe—Continued.

(4) A lathe dog applied as shown in figure 52 ③ may also be used to keep the drill from turning. When this method is used, the center hole in the tang of the drill is mounted on the tailstock center with the lathe dog clamped to the neck of the drill and the tail of the dog allowed to bear on the compound rest. During the drilling operation, the drill should be held back with the tool post to keep it on the tailstock center.

(5) A crotch center, having a **V**-groove on the holding face, is used for cross drilling round work, such as pin holes in shafts, oil holes in bushings, etc. The drill is held in the headstock spindle and the work fed by hand with the tailstock hand wheel (fig. 53).
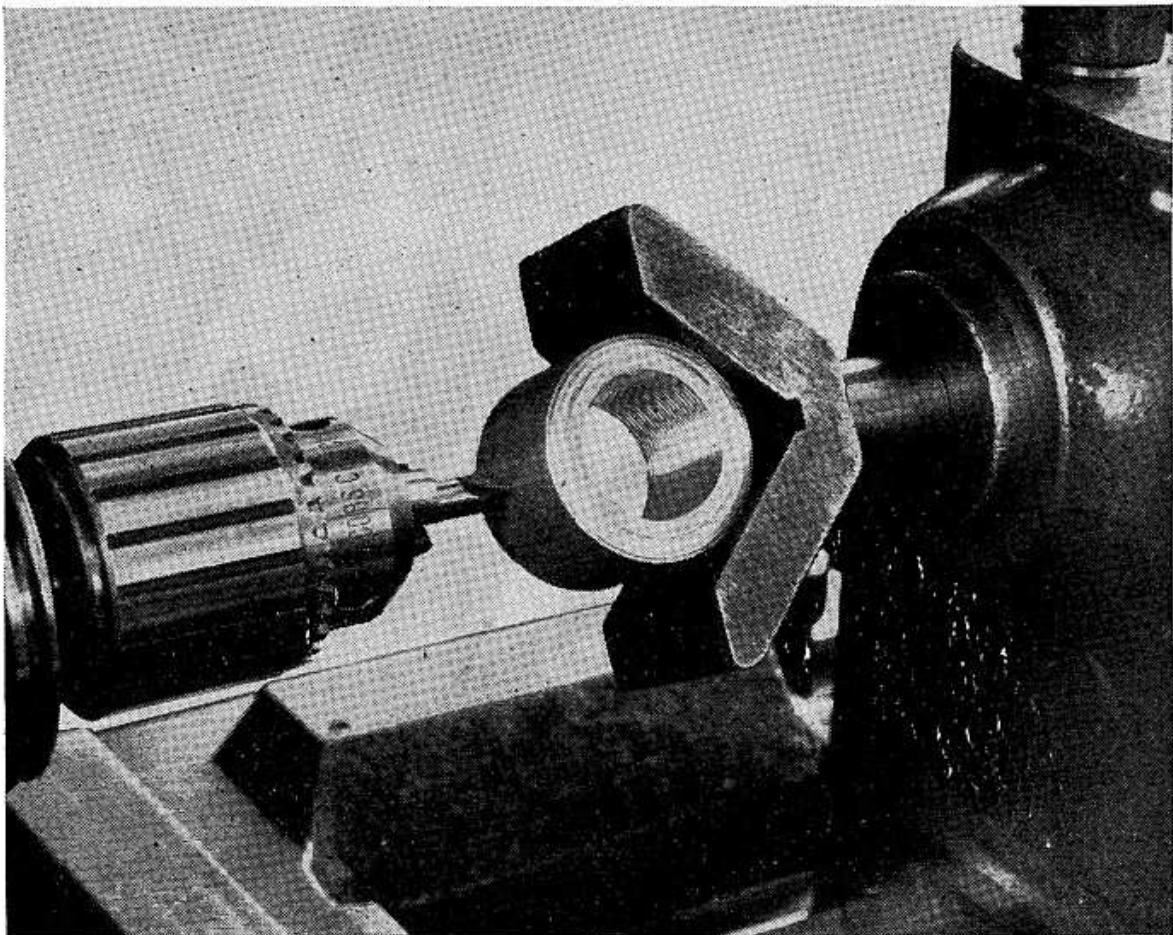
FIGURE 53.—Work being held in crotch center.

(6) In an emergency, the lathe can be used as a drill press by placing the drill pad center in the tailstock spindle and drilling the work (fig. 54).

b. A drill, when used in the lathe, should always be preceded by a center drill or some other means of centering, such as spotting with the lathe tool. Large sizes of drills should be preceded by a lead drill, the diameter of which must not be greater than the thickness of the web of the drill to be used.

c. When drilling in the lathe, the correct speed usually seems too fast, due to the fact that the chuck, being so much larger than the drill, influences the operator's judgment. It is therefore advisable to refer to a suitable table to obtain the proper speed at which various materials may be drilled.

d. It is absolutely necessary to have the head and tailstock in proper alinement during all drilling and reaming operations to produce a true hole and prevent drill or reamer breakage.
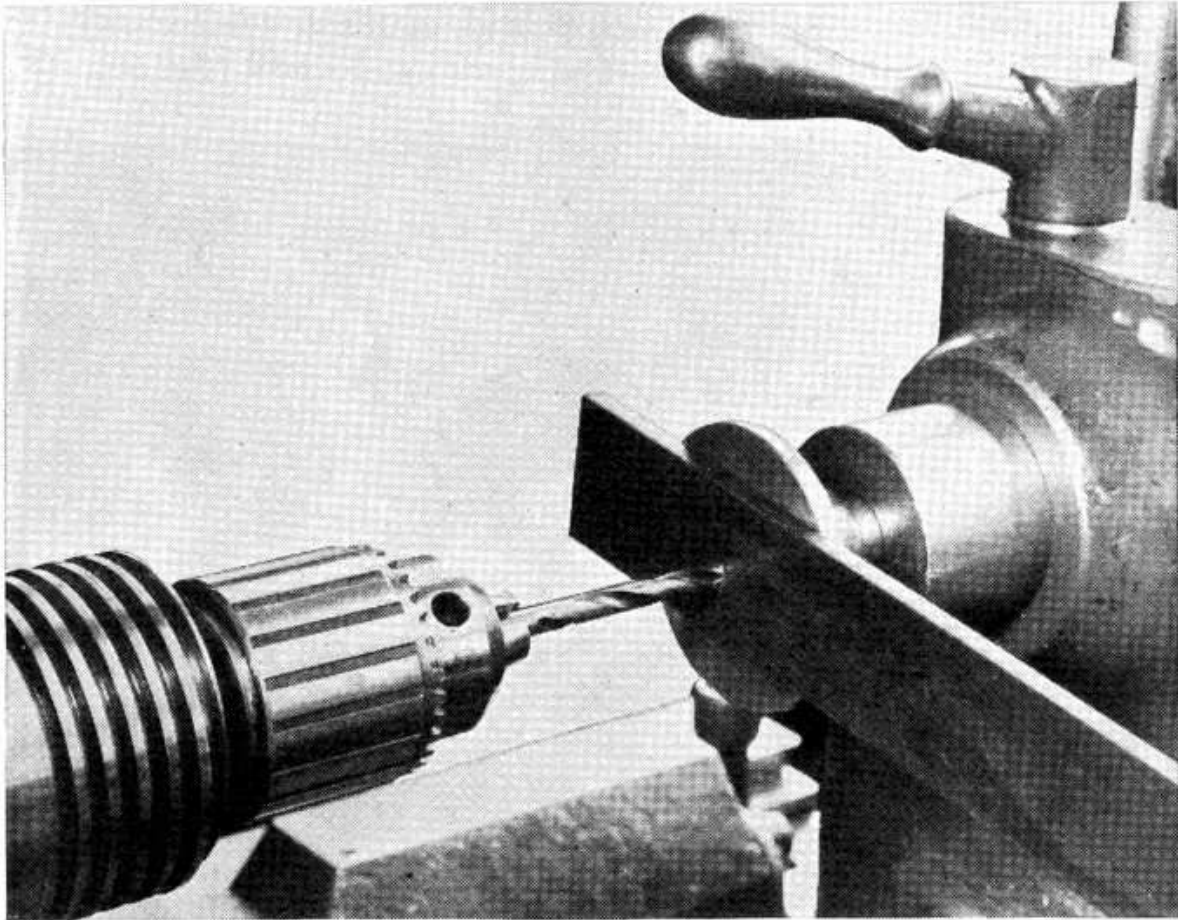

FIGURE 54.—Drilling work held on pad center.

37. Reaming.—*a.* Reamers are used to finish a number of holes quickly and accurately to the same diameter. Machine reamers are frequently used in the lathe and may be held in the same manner as drills. A typical application of the machine reamers is illustrated in figure 55.

(1) Machine reamer speeds are approximately one-half that used for drilling.

(2) When a hole is to be machine reamed, it is drilled or bored to within 0.005 inch or 0.010 inch of the finished size so that it will only be necessary for the reamer to remove the tool marks.

b. Hand reamers are used for various types of work in the lathe, and when it is necessary to finish a bore in this manner the following rules must be observed:

(1) Lock lathe spindle by placing it in low gear.

(2) Have hole within 0.005 inch of required finish size.

(3) Support reamer with tailstock center.

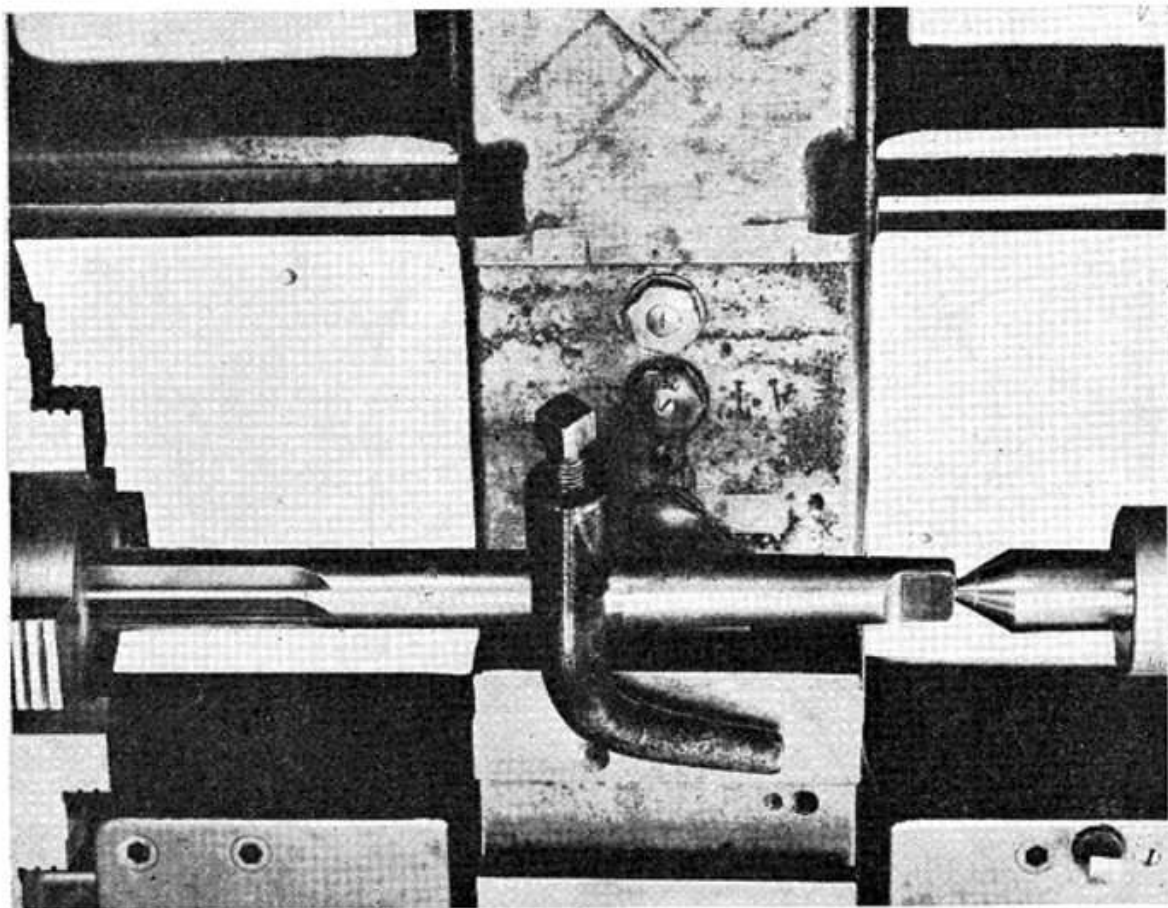(4) Revolve reamer with a wrench and feed simultaneously with tailstock hand wheel.

FIGURE 55.—Reaming on lathe.

(5) Never turn reamer backward.

38. Boring.—*a.* The purpose of boring is to enlarge holes that have previously been drilled or cored. For this operation, the tool bit is attached to or forged on the end of the boring bar which is clamped in a holder and securely held on the compound. The boring tool is fed in the same manner as a turning tool with the exception that the depth of the boring cut is increased by moving the carriage in the opposite direction. Commercially manufactured boring bars should be used in preference to the hand forged types whenever possible, since better results can ordinarily be obtained with them. The boring bar should always be as large as the hole will

permit and clamped as short as possible to prevent unnecessary spring.

b. For straight boring, best results may be obtained if the cutting point of the tool bit is set 5° above center, although in small holes where the bar has very little room this angle must usually be reduced in order to keep the bar from rubbing in the bore. Care should be exercised in setting the tool bit and boring bar so that the top of the bit will have a positive rake of approximately 10°, measured tangential to the radius of the bore. This position (fig. 56) gives a
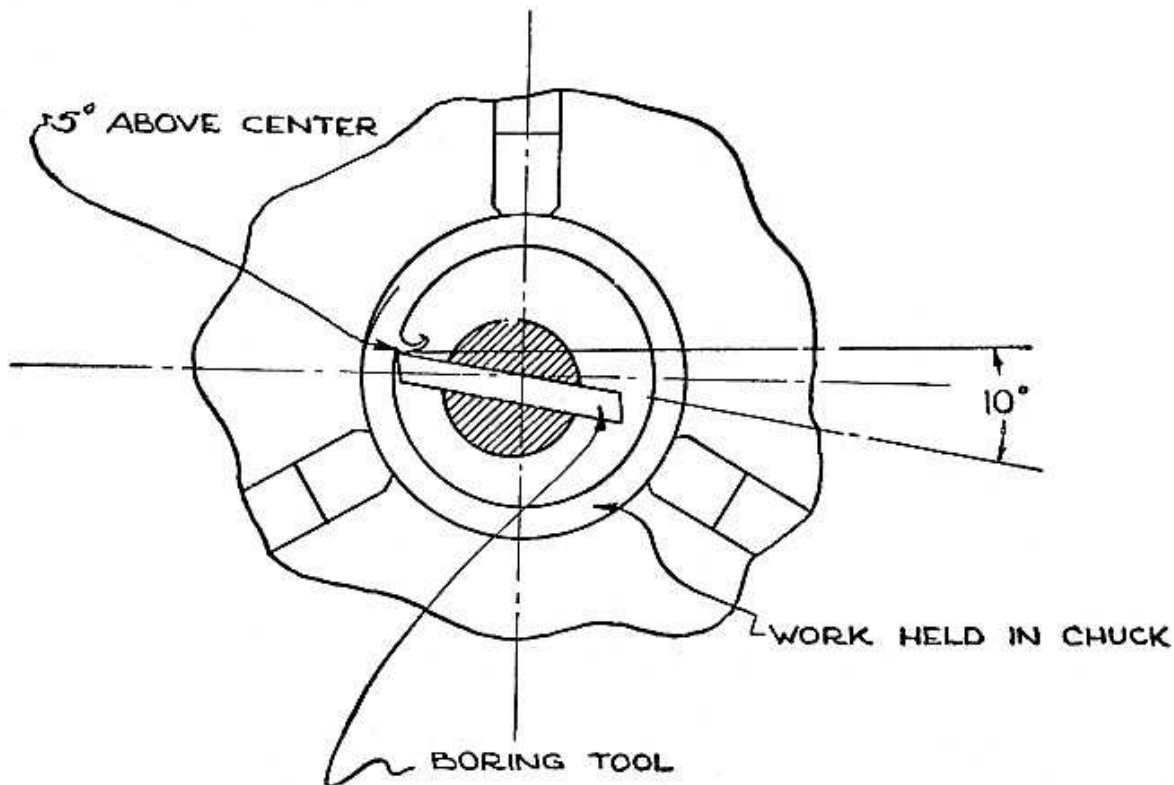
FIGURE 56.—Boring tool position.

freer cutting action than would be possible if the tool and bar were set to a point where the tool bit top would have a negative rake in relation to the tangent.

(1) When boring tapered holes, the tool bit must be set at center height with the same amount of rake as used for straight boring.

(2) By reducing the speed to approximately one-half the normal cutting speed, a much better finish may be obtained.

(3) Due to the spring of the tool, there is a general tendency to bore bell-mouthed holes (larger at the beginning than at the end). In order to prevent this error, a light finishing cut with a sharp tool should be taken with the tool feeding into the bore. At the completion of this cut, the feed should be reversed and the tool allowed to travel out with the same setting.

c. Boring tools are ground similarly to left hand turning tools (fig. 57). The end relief, however, must be ground in relation to the radius of the bore so that the heel of the tool will not rub the surface of the work.
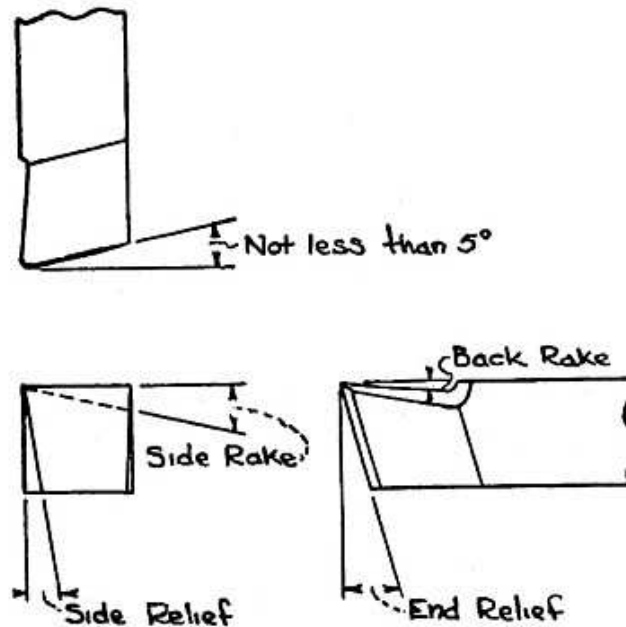

FIGURE 57.—Boring tool nomenclature.

SECTION VII

THREADS AND THREADING

39. General.—*a.* A helical projection of uniform section on the external or internal surface of a cylinder or cone is known as a screw thread. Screw threads are used both as fastening devices and to transmit motion. In general, the V and American fine and coarse threads are used to fasten parts, while the square and Acme threads are used to transmit motion.

b. The common terms relating to screw threads may be defined as follows:

(1) *Pitch.*—The distance from a given point on one thread to a similar point on an adjacent thread measured along the axis.

(2) *Lead.*—The distance the screw thread advances along the axis in one turn. On a single threaded screw, the lead and the pitch are identical; on a double thread the lead is twice the pitch, etc.

(3) *Major diameter.*—The largest diameter of the thread of the screw or nut.

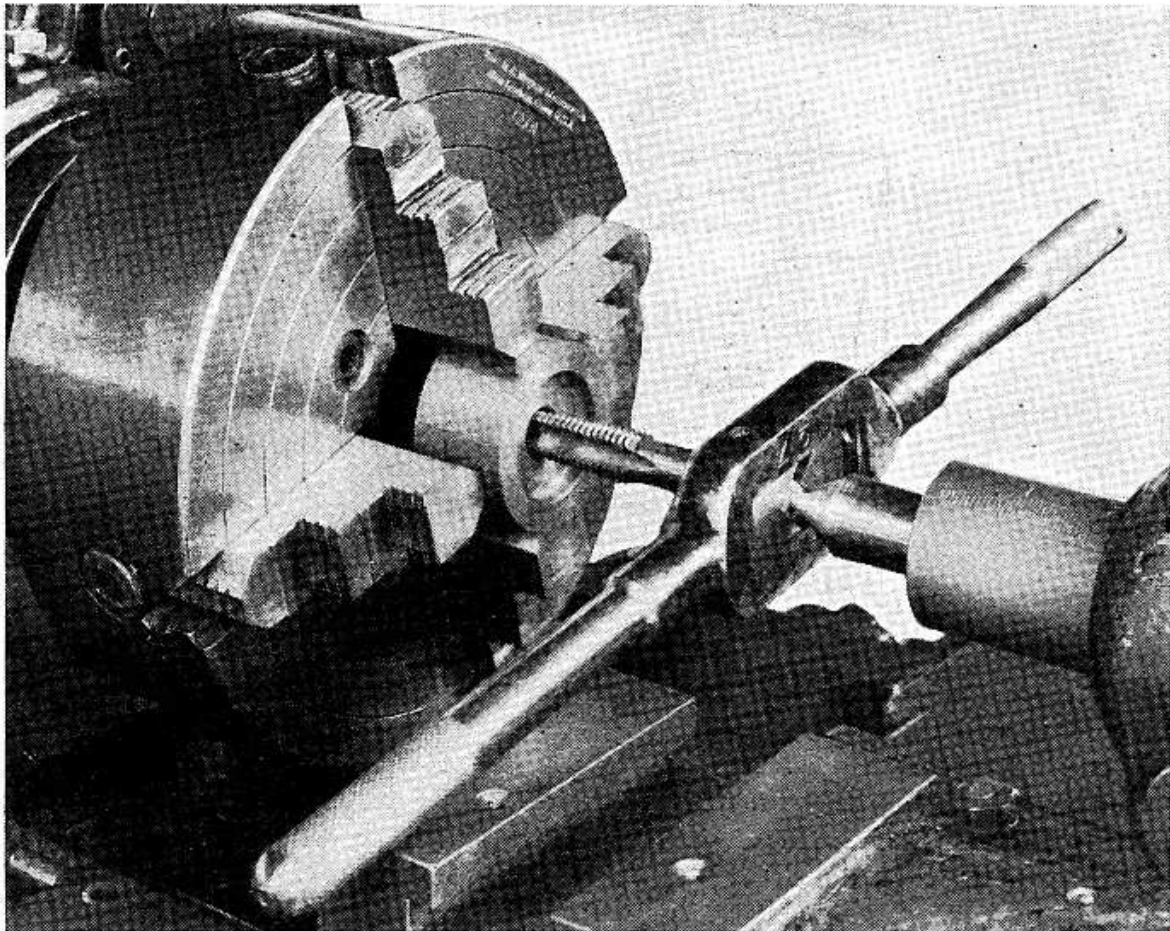(4) *Minor diameter.*—The smallest diameter of the thread of the screw or nut.

FIGURE 58.—Tapping on lathe.

(5) *Pitch diameter.*—The diameter of an imaginary cylinder, the surface of which would pass through the threads at such a point as to make equal the width of the threads and the width of the spaces cut by the surface of the cylinder.

c. Threads may be cut on the lathe either by the use of taps and dies or by means of the lathe thread-cutting mechanism.

40. Tapping threads.—Threads may be tapped in the lathe by either of the following methods:

a. With the spindle of the lathe locked, and the work held in the chuck, the tap is placed in the work and the tailstock center run

against its countersunk end. The tap may then be turned by means of a hand wrench while pressure is applied to it with the tailstock hand wheel. This method is recommended for large taps and for material that does not machine easily.

b. When tapping free-machining materials, the tap may be held as illustrated in figure 58 while the lathe is operated at very slow speed. It is important that a constant pressure be maintained on the tap by means of the tailstock hand wheel to prevent damage to both tool and work.
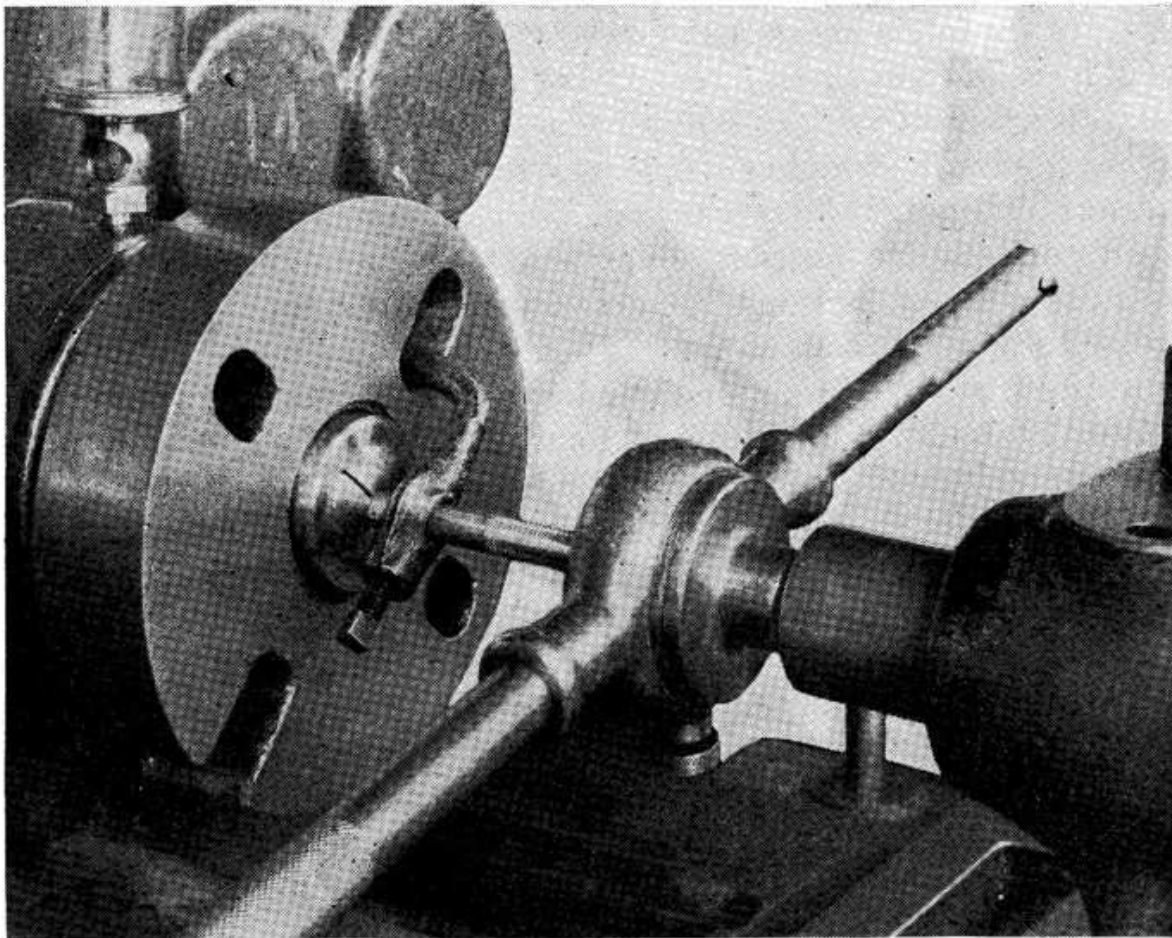
FIGURE 59.—Die cutting threads on lathe.

41. Die cutting threads.—*a.* When cutting threads with the die, the work is usually held in the headstock and the die set on the work with the stock resting upon the compound. The threads are cut (fig. 59) by causing the lathe to revolve at a slow speed while pressure is applied by means of the tailstock hand wheel.

b. The advantage of tapping and die cutting threads in the lathe is that the thread will generally be more accurate in relationship to its axis than when done on the bench.

42. Lathe thread-cutting mechanism.—*a.* In preparing the lathe for thread cutting, the selecting levers on the quick change gear

box are set to correspond to the number of threads per inch desired. Generally there are two or more levers on the box along with a chart giving the position of these levers for the various threads to be cut. In some of the older types of lathes no quick-change gear box is incorporated for gear changes. In this case the gears are separate from the lathe, and proper combinations must be selected and attached by the operator whenever it is desired to cut a thread. Selection is made by referring to a gear table attached to the lathe. In the absence of such a table the operator must calculate the proper gears to be used to produce the desired results.

b. In the following explanation of lathe gearing, the primary points will be considered first. These consist of the gear on the lathe spindle and the gear on the lead screw. Since both are fixed in position, another gear on a swinging arm is used to make the connection between them. This gear is known as the idler, and its size is optional as long as it performs its function, which is to transfer the motion from the spindle to the lead screw. The gear on the spindle is the driving gear, while the gear attached to the lead screw is the driven gear. The gears furnished with the lathe for thread cutting will generally range from 16 to 120 teeth by increments of four. Included in the set are a few odd gears for cutting such pitches as 11½, 13, 27, etc.

(1) In order to select gears for a certain thread, a fraction must be taken, the numerator of which represents the number of threads per inch of the lead screw, while the denominator represents the number of threads per inch desired. The numerator will then equal the number of teeth in the driving gear, and the denominator, the number of teeth in the driven gear. For example, if the pitch of the lead screw is 4 and the thread to be cut is 12, then $\frac{4}{12}$ would be the fraction—4 representing the driving gear and 12 the driven gear. However, since each number is smaller than the number of teeth of any gear in the range, it is necessary to increase the numbers without changing the value of the fraction. This may be done by multiplying both the numerator and denominator of the fraction by the same number:

$$\tfrac{4}{12} \times \tfrac{4}{4} = \tfrac{16}{48}.$$

Thus, change gears of 16 and 48 teeth will be required to cut 12 threads per inch. The 16-tooth gear is the driver and is placed on the spindle, while the 48-tooth gear is the driven and is placed on the lead screw. (See fig. 60.)

(2) On some lathes the driving gear is not placed directly on the lathe spindle but on a stud that is geared down from the spindle. In cases where such additional gearing produces a change of ratio, this

change must be considered when determining the proper gears to be used.

   c. When figuring the gear sizes for cutting extremely fine or coarse threads, the number of teeth required for the larger gear will often exceed the number available if simple gearing is used. In a case of this kind it is necessary to make use of compound gearing (fig. 61).
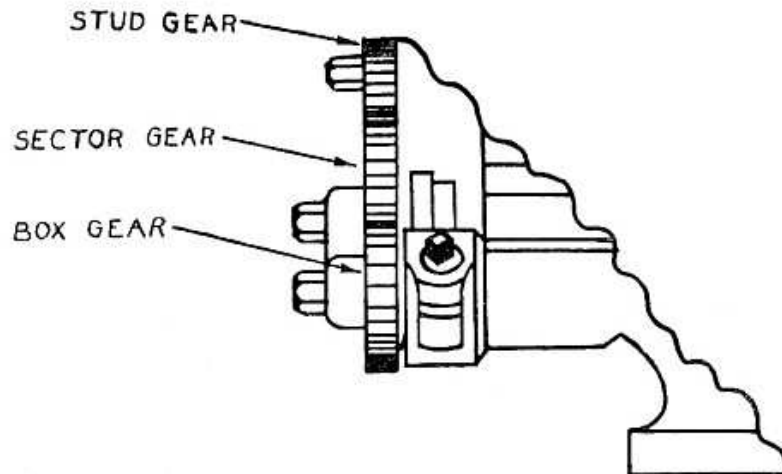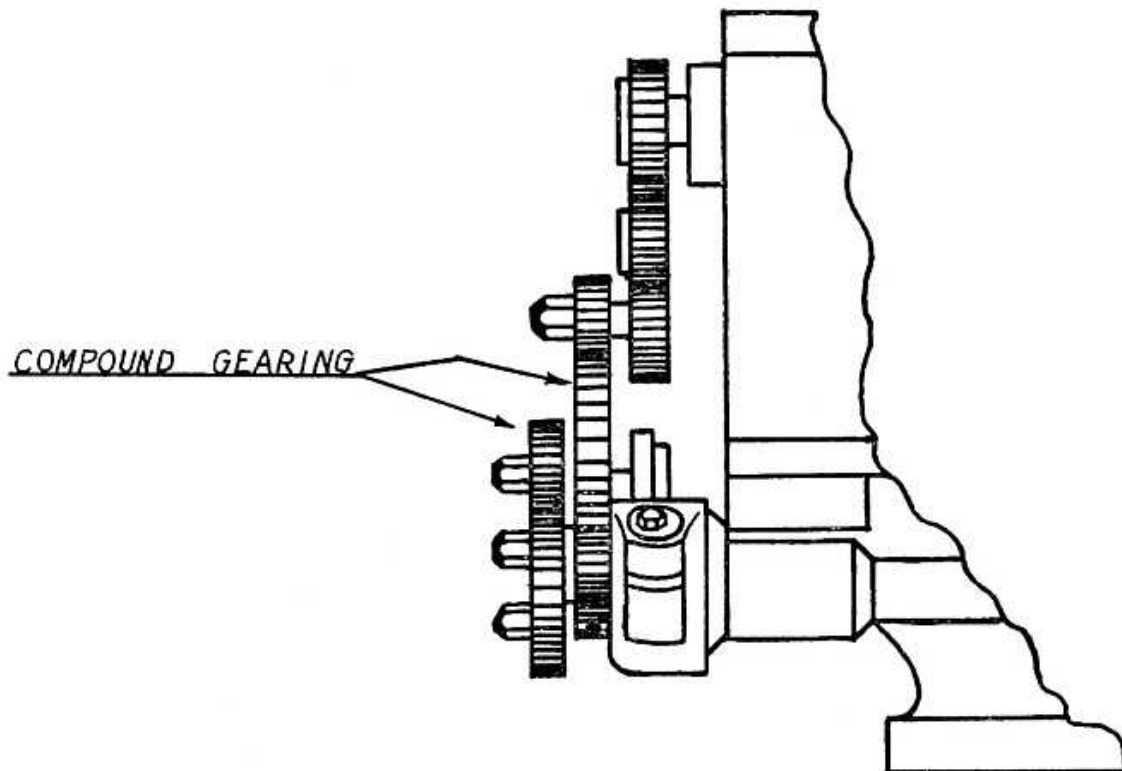

FIGURE 60.—Simple gearing.

FIGURE 61.—Compound gearing.

   (1) When setting compound gearing in a lathe, the place that is ordinarily occupied by the idler gear is used to hold two gears of different sizes that are keyed together by a quill. One of the gears is meshed with the lathe spindle and the other with the lead screw.

Instead of being idler gears, the one that engages with the spindle is a driven gear while the other is a driving gear.

(2) As an example in the use of compound gearing, let it be assumed that 48 threads per inch are to be cut, using a lead screw pitch of 4. As previously explained, 4/48 will be the fraction to use for figuring the gearing. The smallest number must be raised until it equals at least the number of teeth of the smallest gear. Therefore, as 16 is usually the smallest number of teeth available, the numerator and denominator are both multiplied by 4:

$$\frac{4}{48} \times \frac{4}{4} = \frac{16}{172}.$$

In simple gearing the driving gear would have 16 teeth and the driven gear 172 teeth; however, 172 teeth is far greater than the available tooth range. For use in compound gearing the fraction 4/48 is changed to any two components whose product is equal to 4/48; for example:

$$\frac{1}{3} \times \frac{4}{16} = \frac{4}{48}$$

As gears corresponding to these components are not available, it is necessary to raise their numerators and denominators as follows:

$$\frac{1}{3} \times \frac{16}{16} = \frac{16}{48} \text{ and } \frac{4}{16} \times \frac{6}{6} = \frac{24}{96}.$$

Thus, 16 and 24 represent the number of teeth of the driving gears, while 48 and 96 represent the number of teeth of the driven gears. In this connection, it must be remembered that the lathe spindle is the driving member and the lead screw the driven member, while the gear that meshes with the spindle on the compound stud is a driven gear and the one that meshes with the lead screw is a driving gear.

d. When cutting metric threads, a pair of transposing gears is used on the compound stud. One of these gears has 50 teeth while the other has 127 teeth. In the English system, threads are specified in threads per inch, whereas, in the metric system they are specified in threads per centimeter. 50/127 of an inch is equal to 1 cm.; therefore, if the lathe is geared for a certain number of threads per inch in simple gearing and the transposing gears of 50 and 127 teeth are placed in compound in the gear train, instead of cutting a certain number of threads per inch the same number of threads per centimeter will be cut.

e. When a thread pitch must be cut that is not given on the thread index plate, it is necessary to change the sector gear. The following

formula will allow the operator to determine the number of teeth required in the sector replacement gear:

$$\frac{A \times B}{C} = X$$

Where: A = number of teeth in stud gear.

B = number of threads per inch to be cut.

C = nearest number to B on index plate.

X = number of teeth in replacement sector gear.

As an example in the use of this formula, let it be assumed that 27 threads per inch are to be cut on a lathe having a 48 tooth stud gear and a 48 tooth sector:

$$\frac{48 \times 27}{24} = \frac{1296}{24} = 54$$

With the feed box handle in 24 thread position and the 54 tooth gear on the sector, the lathe will cut 27 threads per inch.
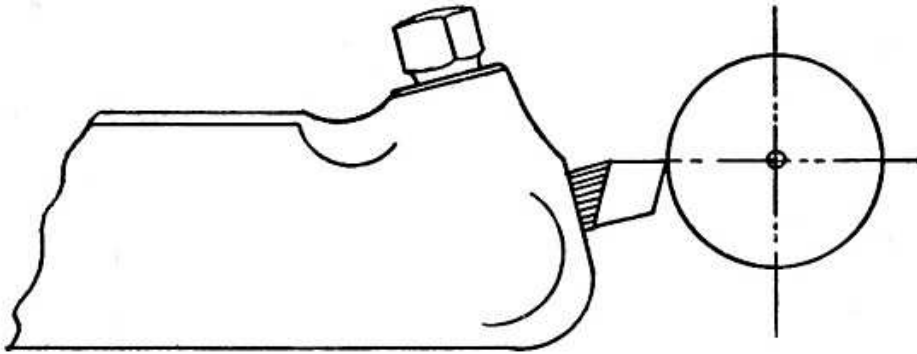
FIGURE 62.—Position of threading tool.

f. In any thread-cutting operation it is necessary to place the feed reversing lever in neutral in order to close the split nut on the lead screw to cause the carriage to travel at threading speed. Since the lead screw can be caused to revolve either clockwise or counterclockwise by means of a reversing gear or lever, it is also necessary to see that the lead screw turns in the proper direction.

43. Cutting 60° threads.—*a.* In cutting all 60° threads the compound rest is carefully set at a 30° angle and its slides adjusted, as any lost motion at this point may cause considerable trouble during the threading operation. If the work is to be held between centers, the centers must be clean, freshly oiled, and adjusted so there is no end play. Unless otherwise specified, the end of the thread should be chamfered by cutting either a 45° angle or a radius larger than the depth of the thread.

(1) Much depends on the careful preparation of the cutting tool for thread cutting. It must be sharp, smooth, and properly shaped to obtain good results. It is often considered good practice to grind

two tools, one for roughing out the thread and the other for finishing. The roughing tool should be made with top rake, while the finishing tool should be flat at the top so that this surface is horizontal in both directions when set to the work (fig. 62) Particular attention must be given to the end and side relief, especially that on the leading side of the tool bit. Since the thread is cut around the shaft at an angle, the leading edge of the tool bit must have more relief than the trailing edge to prevent interference.

(2) The 60° center gage (fig. 63) is used for gaging the angle of the tool bit for cutting the U. S. Standard, National coarse and fine, metric, sharp **V** and all other threads of this angle. Since a tool will tend to cut a groove slightly larger than its own angle, it will be found that if the tool angle is made slightly less than the thread angle desired the finished groove will be more accurate than if the tool were made to correspond exactly to the gage. It is therefore necessary to make the roughing tool angle ½° to ¼° less than the
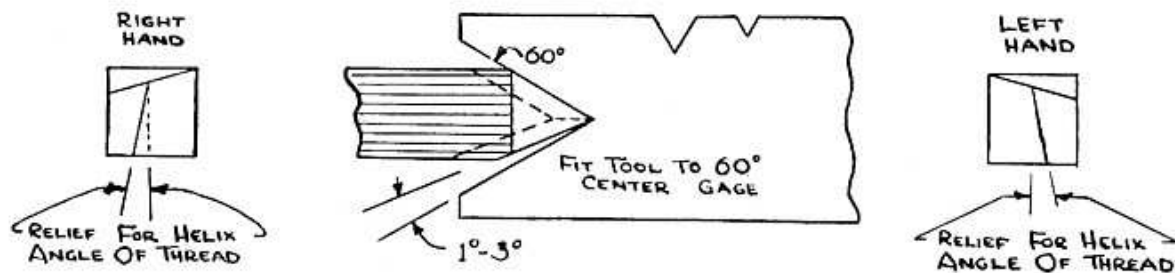
FIGURE 63.—Grinding 60° threading tool.

gage angle, while the finishing tool bit angle should be 3° to 4° less than the gage angle.

(3) The walls of all 60° threads are straight lines, and since the tool bit edges are used to make these thread walls they must be straight also. Should a tool bit be used with irregular sides, this irregularity will be transmitted to the thread, seriously reducing its efficiency.

(4) In order that the truncation at the bottom of the finished thread will not interfere with its mating member, the point of the tool bit should be truncated ½ to ⅔ the amount required by the thread.

(5) Since the compound rest is set at an angle of 30° from parallel to the ways of the lathe and the feed for each cut is made with the compound, the tool will cut only on one side of the thread groove as it is advanced for each successive cut. This enables the tool to have side rake, allowing faster cutting and producing better finishes. Furthermore, if a tool is fed so that both sides cut at the same time, there may be a gouging or tearing action which must be avoided as much as possible in threading. It will also be best to decrease the amount of feed as the tool goes deeper into the work to reduce further

this tearing tendency. At any time that a bur is seen forming at the top of the thread, a file should be used to break it down. Figure 64 illustrates the proper method of feeding a 60° tool bit.

b. The following general procedure should be used in the cutting of a 60° external thread:

(1) Mount roughing tool bit in lathe at center height. Use center gage to square it to surface of work or to some other surface parallel to ways.

(2) Advance tool to surface of work and set thread cutting stop. Set micrometer dial on cross feed to zero, with handle setting in such a position that it may be readily and rapidly disengaged (backed off) by operator.
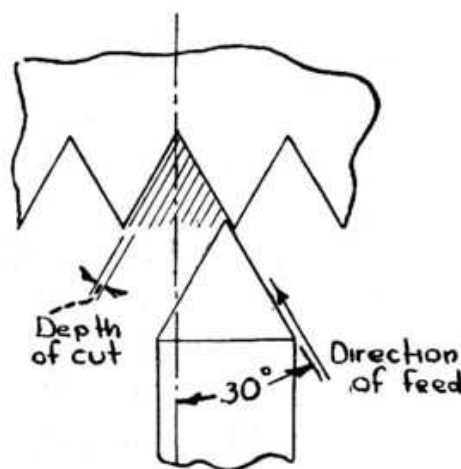
FIGURE 64.—Action of threading tool.

(3) By use of compound, advance tool to take a cut of about 0.003 inch, leaving cross feed at its zero setting. With tool clearing end of work and lathe revolving at proper cutting speed, close split nut on lead screw, which will cause tool to advance over work to make first cut for thread.

(4) At point in work where thread is to stop, back tool out of work by quick turn of cross feed screw and at same time disengage split nut from lead screw. Return tool point to its starting position, reset cross feed dial at zero, and advance compound for another cut.

(5) Repeat this process until thread is roughed out to proper depth for finishing.

(6) Remove roughing tool and insert finishing tool in holder. Carefully set tool bit to center height and check for squareness with work.

(7) Engage lead screw and allow tool to travel along work a short distance away from threaded path. Stop lathe, keeping lead screw

engaged, and by moving cross feed and compound rest cranks bring point of tool bit into groove.

(8) Using a piece of white paper underneath work to provide good visibility, position tool in such a manner that it clears front side of groove between 0.005 inch and 0.010 inch when it bears on thread back (tool bit and work must be thoroughly cleaned before making this adjustment)

(9) Reset cross feed dial to zero without disturbing tool and recheck to make sure that tool will return to same setting. Using fine cuts, bring thread to its finished size.

c. A final threading operation that should be used on all precision work consists of finishing the front and back sides of thread in the following manner:

(1) To finish the back side of the thread, the tool is advanced for each cut by the cross feed screw. Several cuts will be necessary to finish one side of the thread, and even though the marks of the roughing tool have been removed, cuts must be taken until the point of the tool reaches the truncation left by the roughing operation. If the side of the thread does not have the proper finish at that time, the tool will have to be reset and the operation repeated. Any additional cuts attempted after the bottom of the thread is reached will interfere at the front side of the V, tearing the surface.

(2) Next in order will be the finishing of the front side of the thread. This is accomplished with the compound rest at 30°, and the tool set with the gage contacting the front side of the bit. The tool may then be shifted to catch the thread, leaving a clearance of 0.002 inch to 0.005 inch at the back of the V. The cross feed dial is reset to zero and the tool fed into the front side of the thread by means of the compound rest. Since the tool is now advanced by the compound, as many cuts as are necessary may be taken to bring the thread to size without danger of tearing the metal.

(3) For best results the last few finish cuts should be very thin (.0005 inch or less, if possible). A cut of this depth will produce a mirror-like finish, provided all other conditions are right. The tool must at all times take a positive chip from the work. If, due to dullness or too light a cut, the tool rides over the work, the surface will be hardened to some extent, and on the next pass the tool may take a heavy cut and gouge the work.

(4) For very fine threads and for threads requiring only an ordinary degree of finish, the finishing tool may be omitted and the thread finished to size with the roughing tool.

d. Internal threading is accomplished in the same manner as the cutting of external threads with the following exceptions:

(1) An undercut is necessary. The major diameter of the thread desired should be placed where the thread runs out for easy backing-off of the tool.

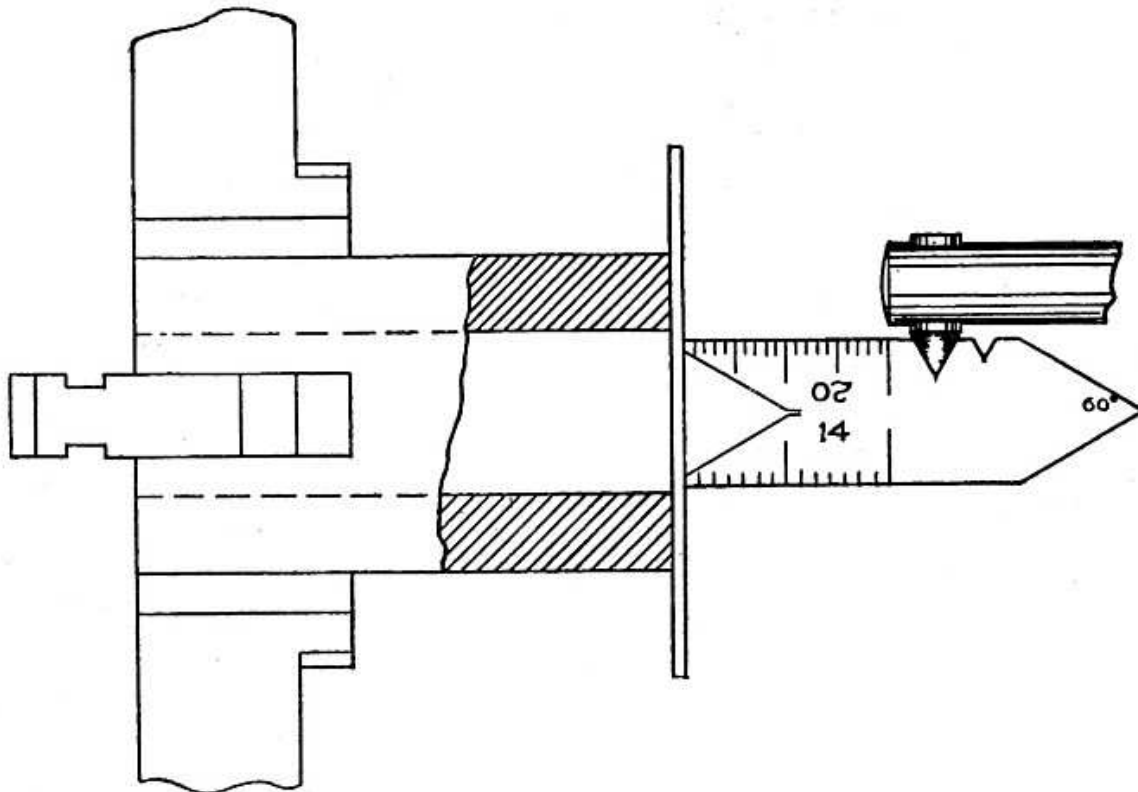(2) The tool must be set in a reverse position.

FIGURE 65.—Setting internal threading tool.

(3) The tool is advanced by moving the compound toward the operator.

(4) Lighter cuts are taken than for external threading due to the spring of the internal tool.

(5) More end relief is required to prevent the heel of the tool from rubbing than when cutting external threads.

(6) Sufficient clearance must be allowed between the tool and the inside diameter of the hole to permit the backing out of the tool when the end of the cut has been reached.

(7) It is necessary to calculate the minor diameter of the thread by means of the following formula:

$$\text{Minor diameter} = \text{major diameter} - \frac{1.026}{\text{number of threads}}$$

e. It is always best to check the pitch of an external or internal thread after the first light cut has been taken over the work. This may be done by using a screw pitch gage or by counting the number of threads per inch with a scale or rule (fig. 66).
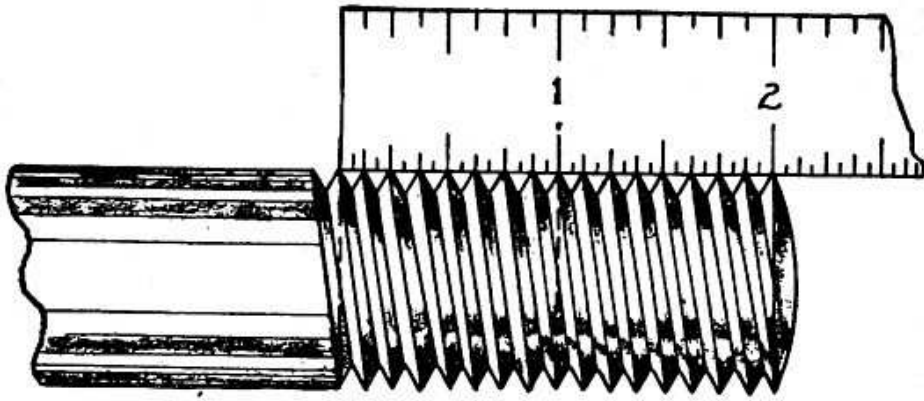
FIGURE 66.—Counting number of threads per inch with rule.

f. For all thread-cutting operations the work and tool must be well lubricated with a good grade of cutting oil. This is important as it does much to help keep the tool from tearing the work.
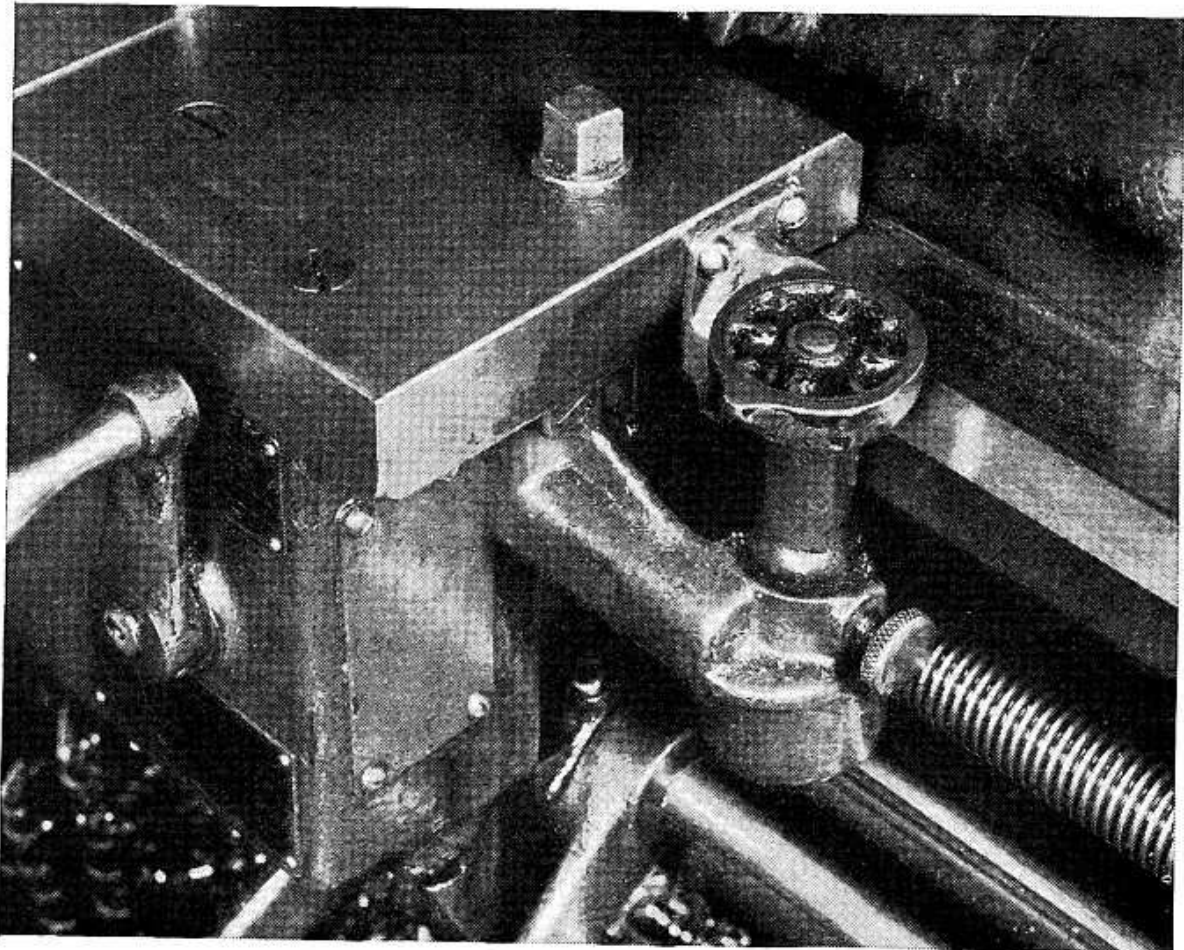
FIGURE 67.—Thread-chasing dial.

g. The thread-chasing dial (fig. 67) is of great assistance when cutting threads in the lathe, especially when the thread to be cut is not a common division or multiple of the lead screw pitch. The dial is attached to the carriage and driven by means of the lead screw. An instruction plate is generally attached to the apron of the lathe, giving

information as to the use of the thread-chasing dial for various threads, and should be carefully observed before starting the threading operation. Without the use of this dial, trouble will be experienced in getting the tool point to retrace the path already cut. Some types of lathes are not equipped with thread-chasing dials but have a reversing lever to take its place. When cutting threads with this type of lathe, it is only necessary to withdraw the tool at the end of the cut and reverse the direction of motion of the lead screw.

h. Accurate measurement is necessary during all thread cutting operations. Several methods used for measuring 60° threads are outlined as follows:

(1) *Thread calipers.*—The calipers have the ends of their legs dressed to fit into the thread form. They are set to a screw or bolt of the size required and this size transferred to the work being done.

(2) *Thread micrometers.*—The micrometers are made with the spindle pointed and the anvil notched so as to bear on the side walls of the thread being measured. The point of the spindle is blunt to prevent interference with the bottom of the thread groove, and the amount of this bluntness of truncation limits the micrometer to the measurement of a definite range of thread sizes. Thread micrometers are therefore manufactured in different size ranges, such as 1 inch: 8 to 13 threads, 14 to 20 threads, 22 to 30 threads; 2 inches: 14 to 20 threads, 22 to 30 threads, etc. As the thread micrometer measures the pitch diameter of the thread, the micrometer reading for a screw is obtained by sub-
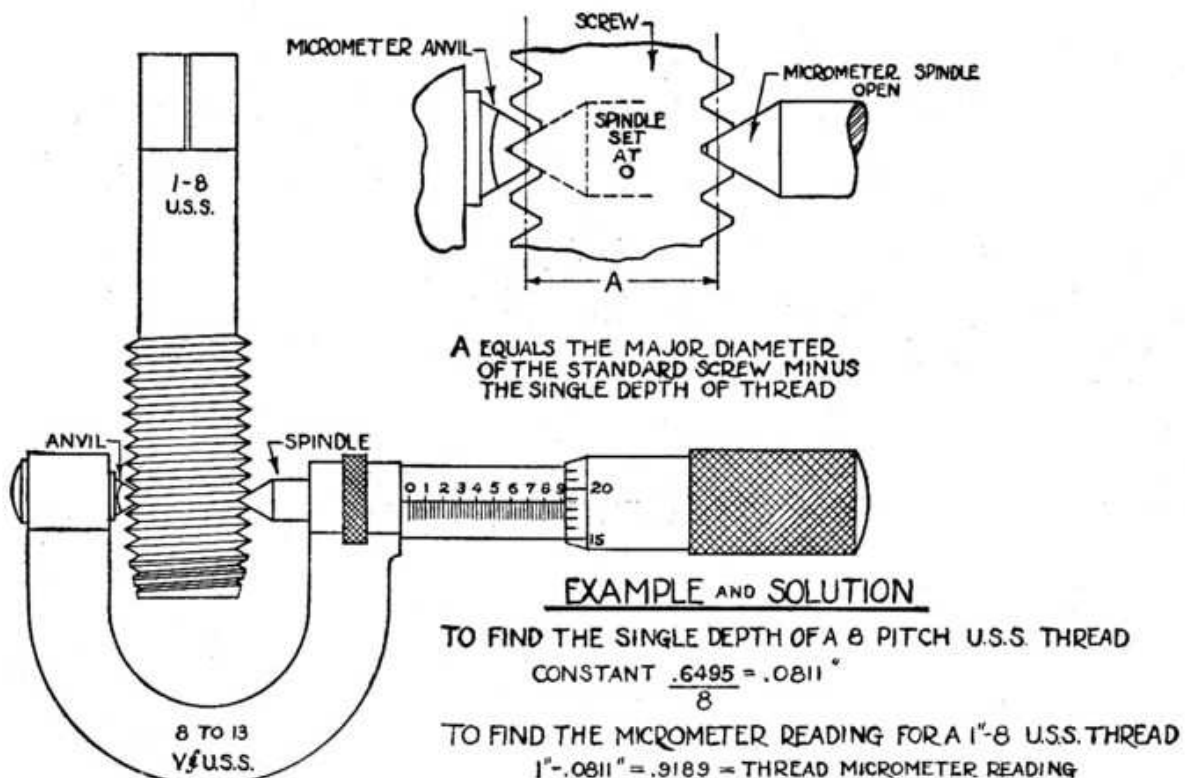
FIGURE 68.—Measuring threads using thread micrometer.

tracting the single depth of the thread from its outside diameter. (See fig. 68.)

(3) *Saybolt or three wire system.*—This method allows the use of an ordinary micrometer, and measurement is accomplished by placing three wires (all of the same size) into the thread groove (fig. 69). The
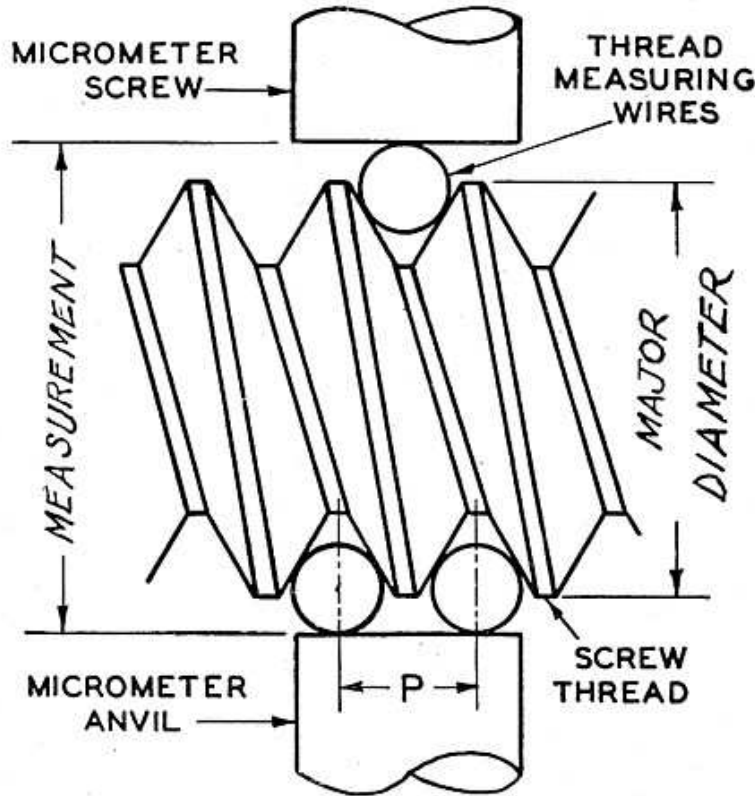
FIGURE 69.—Measurement of threads by three-wire system.

size of the wires should be such that they will bear on the side walls of the thread at the pitch diameter when the thread is finished. Knowing the outside diameter of the screw, the wire sizes, and pitch of the thread, the measurement over the wires can then be computed by means of the following formula:

$$D = OD - 1.515\,P + 3\,W$$

Where: D = Measurement over the wires.

OD = Outside diameter of screws being measured.

W = Wire diameter, $\dfrac{.577}{\text{Number of threads}}$

P = Pitch of thread being cut.

Example: Three wire calculation for ¾ inch by 10 threads.

$$D = 0.750 - (1.515 \times .10) + (3 \times .057)$$
$$D = 0.772$$

(4) *Optical comparator.*—By this method an enlarged outline of the thread being cut is projected on a screen for visual comparison with a perfect thread outline of the same size.

44. Pipe threads.—Pipe threads are cut in a manner similar to ordinary 60° threads, with the exception that a taper attachment must be used to give the thread a taper of ¾ inch per foot (fig. 70).
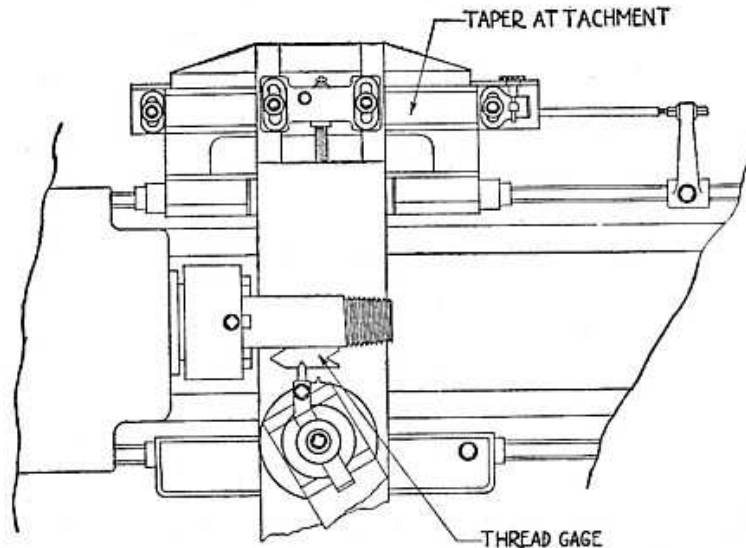
FIGURE 70.—Cutting tapered pipe thread.

Pipe threads are measured by means of a gage, and their specifications may be calculated by the following formulas:

$$\text{Pipe thread truncation} = 0.033 \times P$$
$$\text{Length of pipe thread} = 0.80 \; OD + 6.8 \times P$$

45. Square threads.—*a.* Square threads are used for feed screws, jack screws, vise screws, etc.

b. For square threading, the compound rest is set parallel to the axis of the machine. A tool bit, ground from 0.002 inch to 0.008 inch undersize, is used for the roughing cut while finishing is accomplished with tool bits having left and right rake. The front and back sides of the thread are finished by moving the compound rest parallel to the axis. (See fig. 71.)

c. Both the depth and the width of all standard square threads are equal to one-half the pitch (fig. 72 ①). Square threads are measured by means of a micrometer and single wire or drill of the proper size (fig. 72 ②).

46. Acme threads.—*a.* Acme threads are used for adjusting screws and feed screws on various machines. These threads have an included angle of 29° (fig. 73 ①).
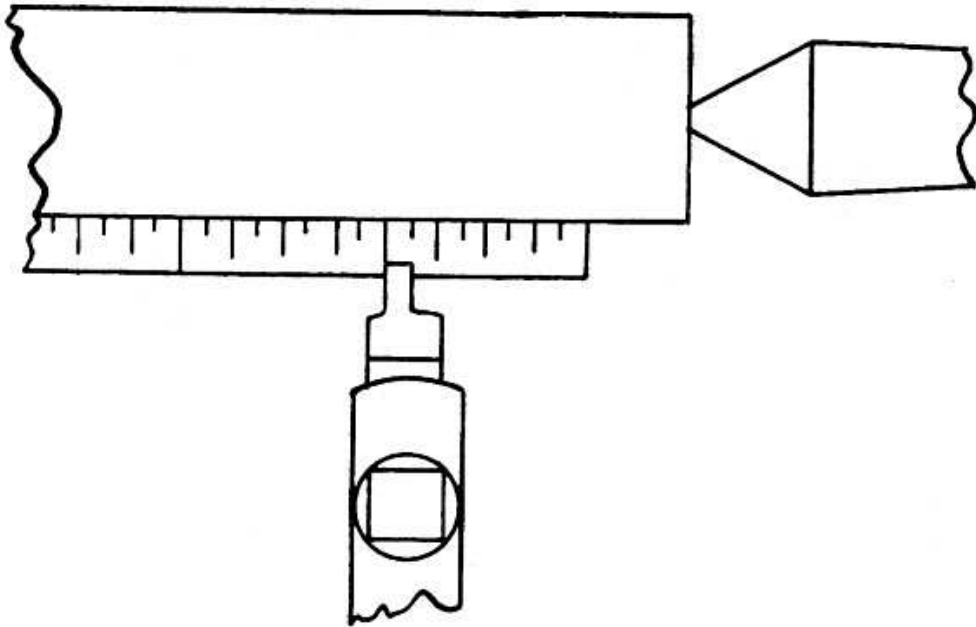
① Correct setting of tool to work.

TOP

FRONT   SIDE
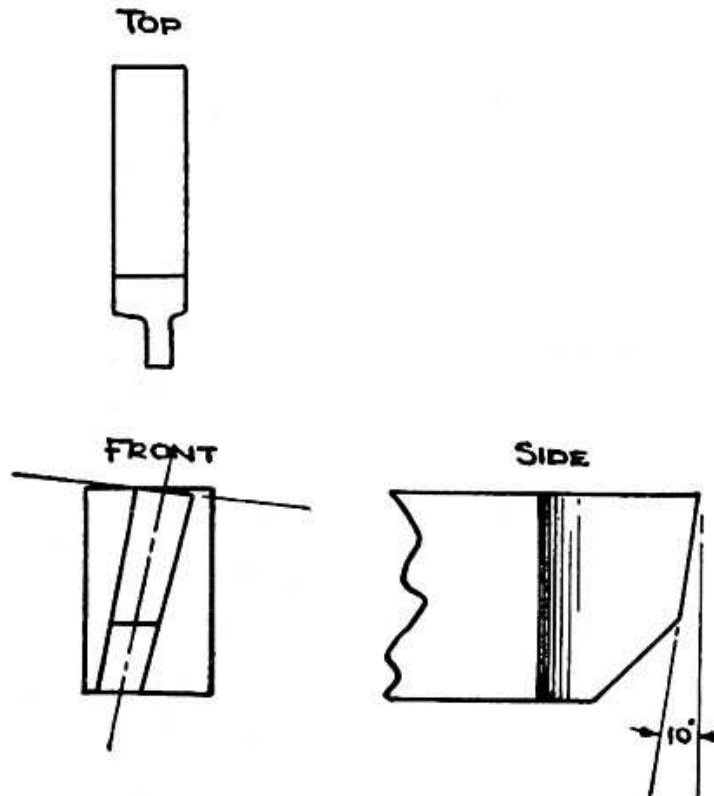
② Properly ground tool bit.

FIGURE 71.—Square thread tool.

b. Acme threads are cut with the compound rest set parallel to the axis of the machine. The general cutting procedure is the same as for square threads and the tool is set as illustrated in figure 73 ②.

c. The depth of the Acme thread is equal to one-half the pitch plus 0.01 inch and is measured as shown in figure 73 ③, using a single wire or drill of the proper size.

① Form. ② Measurement.
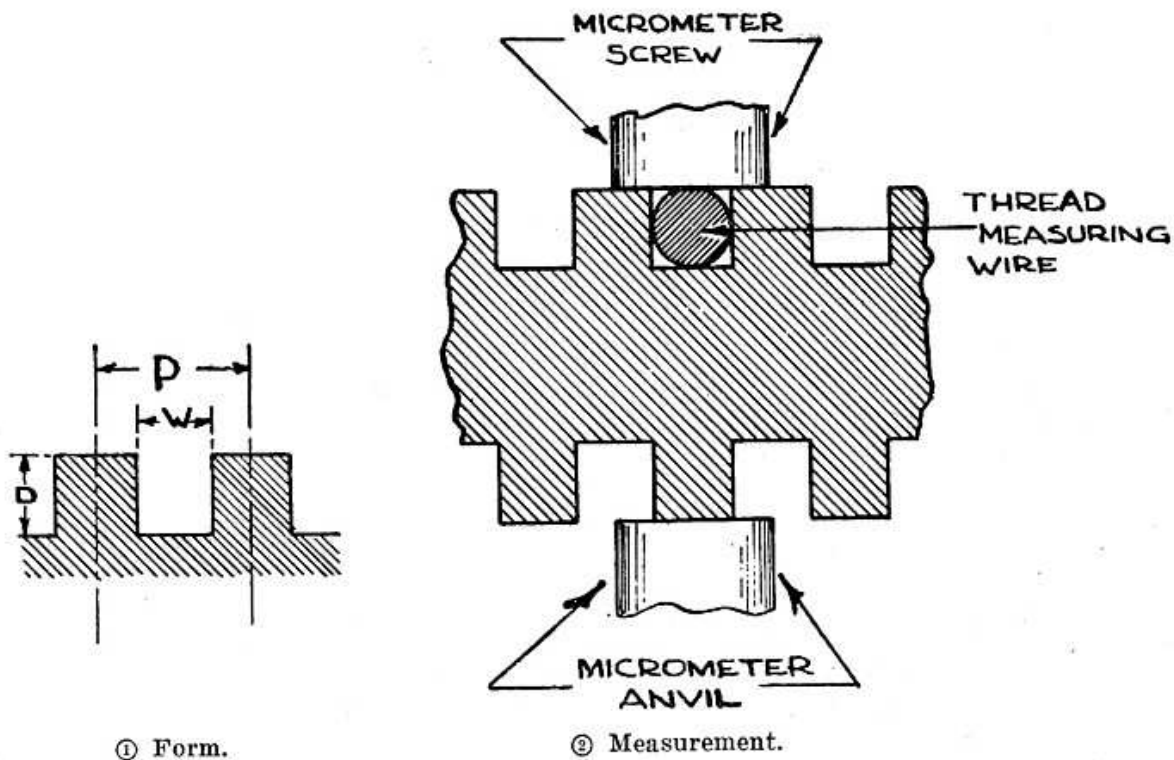
FIGURE 72.—Square threads.

d. Both Acme and square threads are usually cut with a relatively coarse pitch. Due to the large helix angle produced by this wide thread spacing, careful attention must be given to the relief of the tool bit. In order to check accurately the amount of relief necessary to prevent binding of the bit in the groove, a gage may be made in the following manner:

(1) Using a small rectangular piece of sheet metal, scribe a line at 90° to one of its sides and on it lay off a distance equal to the circumference of the thread to be cut at its minor diameter.

(2) At one end of this line and at right angles to it, lay off a distance equal to the pitch of the thread to be cut.

(3) Connect the ends of these two lines, and the angle of this third side is the angle of the required thread at its minor diameter.

(4) Cut off the sheet metal along this last line and gage the tool bit (fig. 74). The tool bit is considered correctly relieved when its side clears the gage from 3° to 6°.

47. Multiple threads.—*a.* It is sometimes necessary to cut a screw that will travel a considerable distance in one revolution. If a single pitch coarse thread were cut on the screw, the minor diameter of the thread would be so deep that the screw would be seriously weakened. In a case of this kind, the multiple-pitch thread may be used to good advantage. As an example in multiple thread cutting let it be assumed that an eight-pitch double thread is desired.
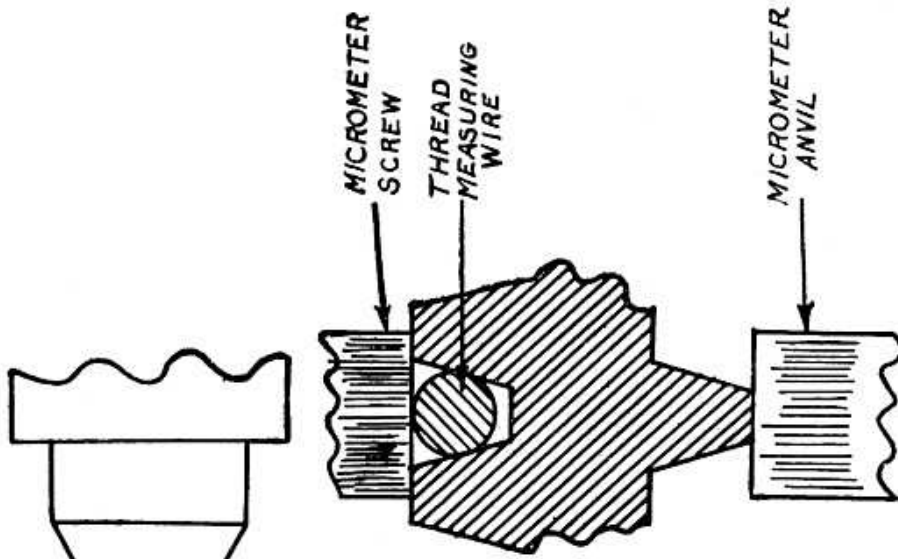
③ Measuring thread.
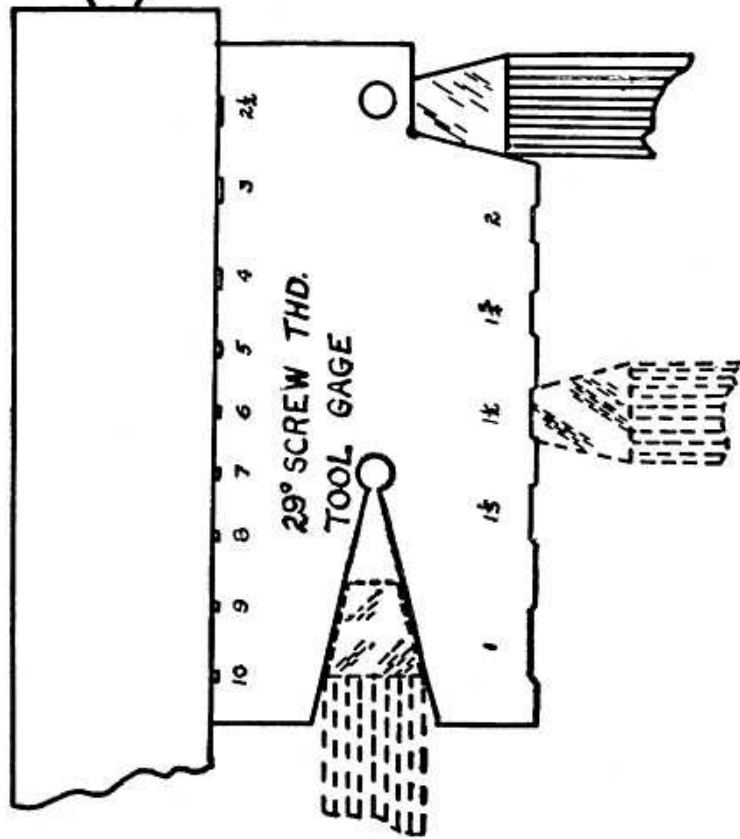
② Setting thread tool.
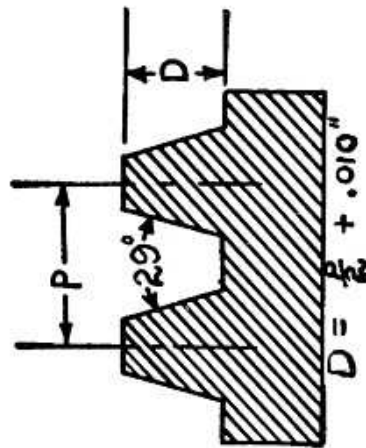
FIGURE 73.—Acme thread.

① Thread form.

The lathe is set to cut four threads per inch and the thread is cut to the depth of a standard eight pitch thread. Another thread of like depth is then cut on the screw, halfway between the spirals of the first thread, resulting in a thread of eight pitch but having ¼-inch lead. The triple pitch thread requires three spiral grooves to be cut around the screw in a similar manner, while the quadruple thread requires four grooves, etc.
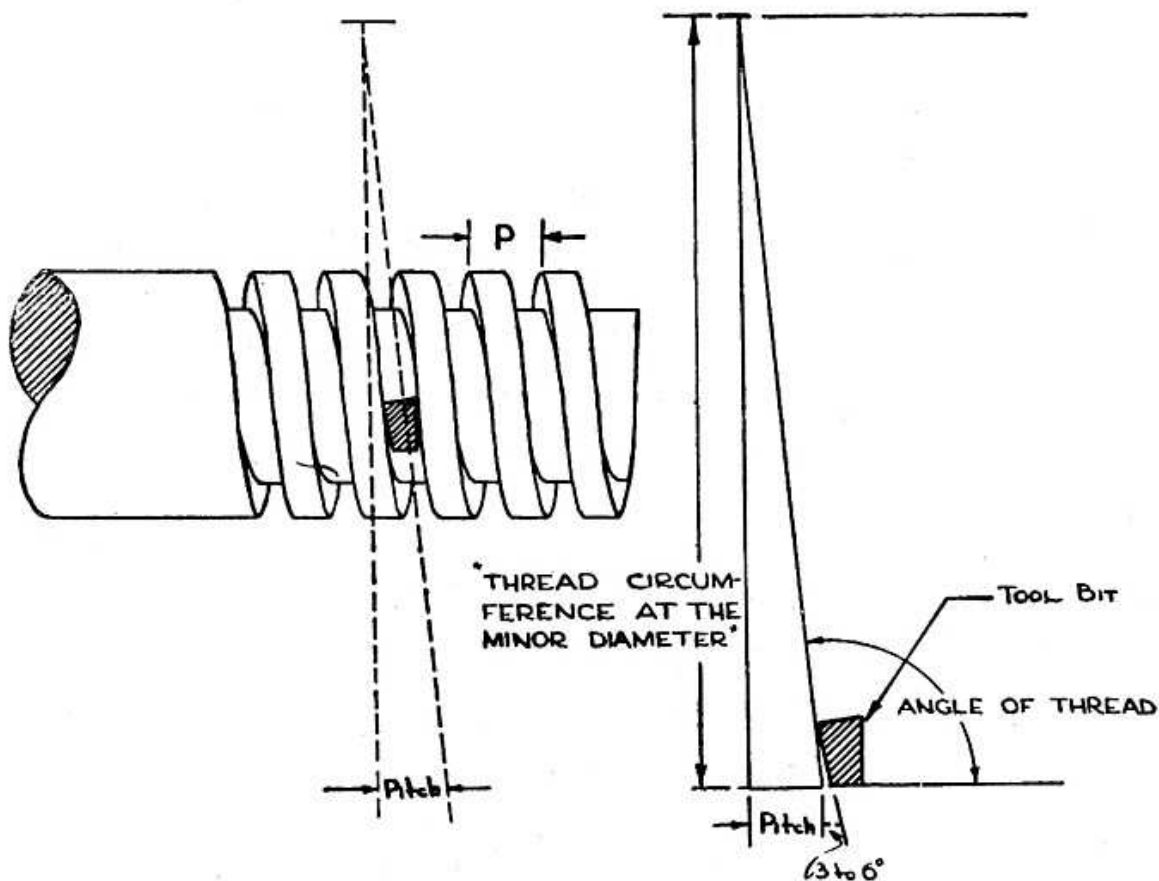
FIGURE 74.—Calculation for thread helix angle.

b. One of the major requirements of multiple threads is the equal spacing of the grooves and may be accomplished in any of the following ways:

(1) By the use of the thread chasing dial.

(2) By setting the compound rest parallel to the ways of the lathe, so that after the first pitch has been cut the micrometer dial of the compound may be moved the exact number of thousandths for the next pitch.

(3) By the use of a driving plate having the proper number of accurately spaced slots to accommodate the tail of the driving dog. After one pitch is cut, the tail of the dog is simply shifted to the next slot, etc.

(4) By counting the number of teeth in the gear on the sector arm at the head of the lathe. After one pitch is cut, the gear is disengaged and turned the number of teeth necessary to bring the tool point in place for the next pitch. This method is only possible when the number of teeth on the gear is evenly divisible by the thread to be cut.

48. Left-hand threads.—A left-hand screw thread is one that is advanced by counterclockwise rotation with respect to the head of the screw. Left-hand threads are cut in much the same way as the right-hand type except in the reverse order. The proper procedure is outlined as follows:

a. Reverse direction of rotation of lead screw causing carriage to move toward tailstock.
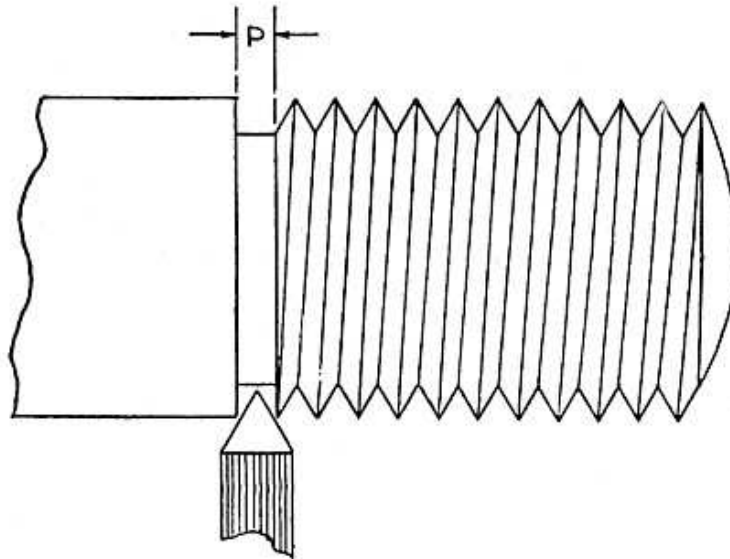
FIGURE 75.—Undercut for left hand threads.

b. Set compound rest in reverse direction to that used for right-hand threads.

c. Relieve tool bit for left-hand threads.

d. Undercut stock to minor diameter (fig. 75). This undercut should be of a width equal to the pitch of the thread and is provided for easy starting of the tool bit.

e. Proceed as in cutting right-hand threads.

SECTION VIII

TYPES OF FITS

49. Types of fits.—*a. General.*—In the fitting of the various parts of any mechanism, certain differences in dimensions of the mating members must be made so that they may operate properly.

b. Description.—(1) *Running or revolving.*—In this fit the shaft or revolving member is slightly smaller than its bearing to provide space for lubrication. This clearance is also necessary to avoid any danger of seizing or galling due to expansion. Running fits vary from 0.0002 inch to 0.015 inch, depending on the size, and class of work being fitted. The lathe headstock spindle with its bearing is a good example of this type of fit.

(2) *Sliding.*—In this class of fit the line of motion is at right angles to that of the running fit, and the clearance can be made somewhat smaller. It is considered a good practice to machine members which require this class of fit so that the tool marks will be in the direction of travel. If this is not possible, the work should be draw-filed and polished so that the lines are parallel to the motion. Good examples of the sliding fit are the lathe tailstock spindle in its bearing and the lathe carriage on the ways.

(3) *Wringing.*—In this fit the clearance between the male and female members is so slight that the parts have practically a metal-to-metal contact. A fit of this kind is difficult to produce in the lathe due to the close tolerances required. A good example of the wringing fit is to be found in the standard ring and plug gages. These gages, instead of being sized in the lathe, are hardened, ground, and lapped. The fit derives its name from the action necessary to bring the parts together. For example, it is not easy to insert the plug into the ring with a straight push, but if this push is accompanied by a slight rotary or wringing motion the gages will go together readily.

(4) *Tight.*—In lathe work this is understood to mean a fit so close as to make hand assembly of the members difficult yet not so close as to require driving or other forcing by mechanical means. This type of fit is used for gears, pulleys, etc., that are secured to shafts by keys, set screws, or taper pins, so that they may be easily mounted and dismounted when the retaining members are removed.

(5) *Driving.*—The internal member in this case must be distinctly larger than the external member, although the difference must not be so great that the parts cannot be put together by hammer blows. It is a good policy to taper the receiving ends of such members slightly so that they may start and line up easily. Assemblies with this class of fit are generally considered permanent although they may be dissembled. They are generally used for the placement of bushings in

wheels, pulleys, sleeves, etc., as well as for the assembly of permanent parts where load and service requirements are not severe and other holding methods are impractical.

(6) *Force.*—This fit is comparable to the driving fit but has a greater difference in sizes. This difference makes it necessary to use some mechanical means to force the members together. Forcing presses of various types are used to accomplish assemblies with this class of fit. In both the drive and force fit it is possible that the members may gall when put together. To prevent this a heavy lubricant of some kind should be applied before assembly.

(7) *Shrink.*—In this fit the outer member is made slightly smaller than the inner member, then expanded by heat, slipped over the inner member, and allowed to cool. Shrink fits are used where maximum grip is required between the parts and are found valuable in work where an intensity of stress will be applied to the united parts.

(8) *Expanding.*—There are times when a shrink fit is desired between certain members but the nature of the work will not allow the outer member to be heated. In this case the inner member is cooled to a low temperature by use of solidified carbon dioxide ($-110°$ F.) or liquid air ($-220°$ F.). This causes the member to shrink in size. While in this contracted condition it is assembled and allowed to expand.

(9) *Thread.*—The National Screw Thread Commission has established five distinct classes of screw thread fits known as: class I (loose fit), class 2 (free fit), class 3 (medium fit), class 4 (close fit), and class 5 (wrench fit).

50. Allowance.—It will be noted throughout the different fits that certain intentional differences in the sizes of mating members must be made to obtain the result desired. This intentional difference is termed "allowance." Before an allowance can be specified certain information must be available, such as the kind of fit, class of fit, kind of metal, mass of metal, length of bearing, and finish of surface.

51. Tolerance.—The term "tolerance" is used to denote the variation permitted in the dimensions of a machined part and may be defined as the difference between the maximum and minimum limits of a dimension. It is generally specified as being plus or minus a certain number of thousandths or ten-thousandths and is written immediately following the dimension. The greater the tolerance the more rapidly the work can be produced, although the constant demand is for smaller limits in all machine work. Charts giving allowance and tolerance for the various fits may be found in various handbooks on machine work.

SECTION IX

GEAR BLANK CALCULATIONS

52. General.—Although most gear-cutting operations are accomplished on the milling machine, the gear blanks must be turned to size on the lathe. For this reason, a description of the various types of gears is given in the paragraphs following, along with the calculations necessary to bring the blanks to their required shape and size.

53. Spur gears.—*a.* Spur gears may be considered as wheels or cylinders rolling against each other and provided with teeth to prevent them from slipping. Gears of this type are used to transmit motion between parallel shafts.

b. The size ratio of these cylinders equals the ratio of the actual gearing, and the point at which the cylinders bear upon each other is known as the pitch line or pitch circle of the gears. The diameter of this pitch circle is known as the pitch diameter. (See fig. 76.)
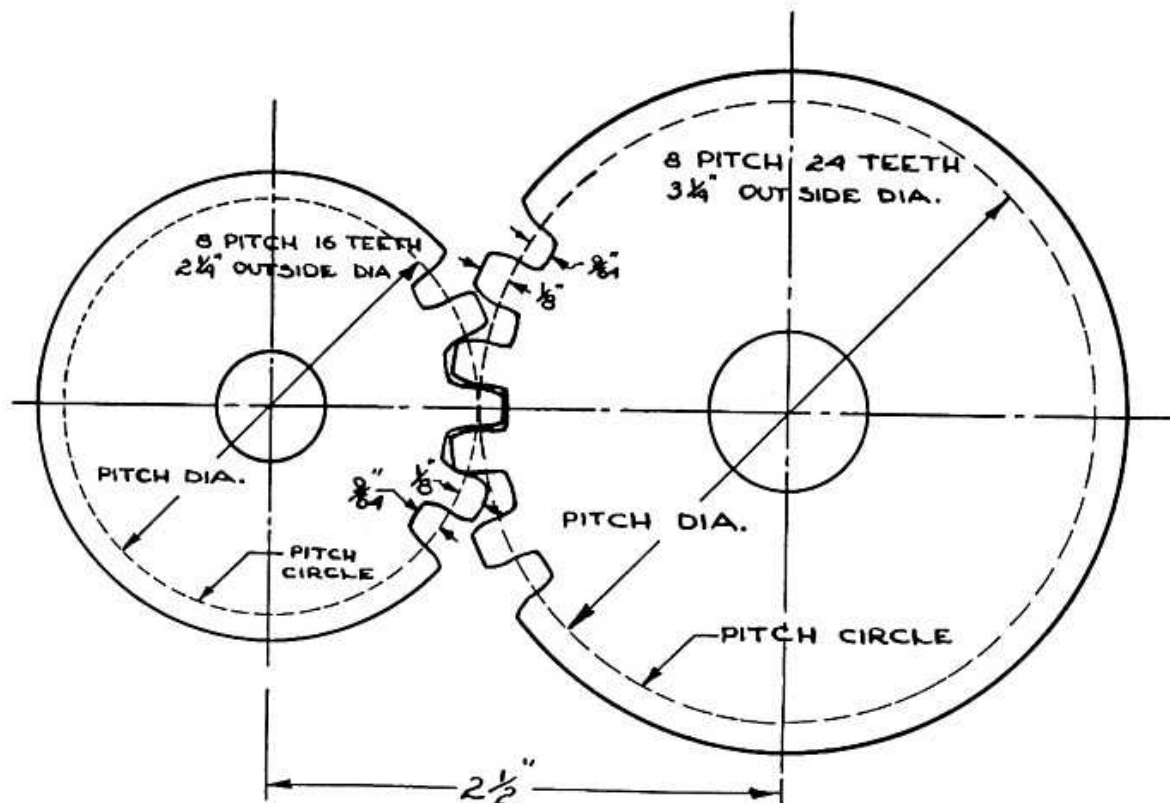
FIGURE 76.—Spur gear.

c. The size of gear teeth is commonly measured by what is known as diametral pitch. This is the number of teeth contained in a pitch diameter of 1 inch. For example, if there are 12 teeth in this measurement, the diametral pitch is 12; if there are 8 teeth the diametral pitch is 8, etc. In figure 76, there are 16 teeth of 8 diametral pitch in the small gear, making a pitch diameter of 2 inches, and 24 teeth of the same diametral pitch in the large gear making its pitch diameter 3 inches. From this calculation it may be readily seen that the ratio of the gears illustrated is 2 to 3 or 1 to 1.5. The following formula may be applied to any spur gear:

$$\text{Pitch diameter} = \frac{\text{Number of teeth}}{\text{diametral pitch}}$$

d. By referring to figure 76, it may be noted that a certain portion of the gear tooth extends beyond the pitch circle. This portion is known as the addendum. The height of the addendum is always equal to one divided by the diametral pitch. In order to obtain the outside diameter of a spur gear blank, two addenda must be added to the pitch diameter; therefore:

$$\text{Outside diameter} = \text{pitch diameter} + 2 \text{ addenda}$$
$$= \frac{\text{number of teeth} + 2}{\text{diametral pitch}}$$

54. Helical gears.—*a.* The teeth of helical gears are cut across the outer rim of a gear blank similarly to screw threads, and the angle of cut is referred to as the helix angle. Helical gears may be used to connect either parallel shafts or shafts set at any nonintersecting angle. They have greater strength than spur gears for a given size as there are more teeth in mesh at any one time. For this same reason, they are also more quiet and smooth in operation.

b. The same general rules that have been given for plain spur gears are also applicable to helical gears, although as the angle of spiral increases the pitch diameter also increases. The pitch diameter of helical gears may be calculated by means of the following formula:

$$\text{Pitch diameter} = \text{secant of helix angle} \times \frac{\text{number of teeth}}{\text{normal diametral pitch}}$$

c. As in spur gearing, the outside diameter of a helical gear is obtained by adding two addenda to the pitch diameter.

55. Bevel gears.—*a.* Gears of this type are used to transmit power when shafts are not parallel. They can be made to set at any angle, although the 90° position is the one most commonly used. Right angle bevel gears that have the same number of teeth are called miter gears.

b. Primarily, bevel gears may be considered as friction cones rolling against each other (fig. 77). The pitch diameter of the gears will be the major diameter of these friction cones, while the pitch circle is represented by the major circumference of each cone. Referring to figure 77, the lines *BC* and *CD* are, therefore, the pitch diameters of cones *ABC* and *ACD*, respectively. Also, points *B*, *C*, and *D* are located on the pitch circles of the cones. As the ratio of the pitch diameters gives the ratio of the bevel gears, the diameters of 2 inches and 3 inches give a ratio of 2 to 3 or 1 to 1.5. It should be noted at this time that the cone angles of a pair of bevel gears are such that if extended, they would terminate in a common apex, point *A*, figure 77.
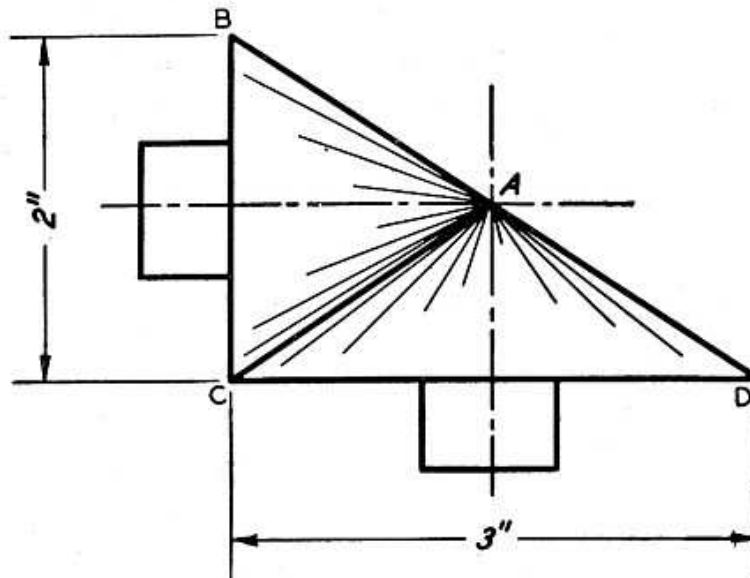
FIGURE 77.—Pitch cones of bevel gears.

(1) The tangent of the pitch cone angles of any set of bevel gears whose center angles equal 90° may be obtained by dividing the number of teeth in the gear by the number of teeth in the pinion. This angle, subtracted from 90°, equals the pitch angle of the pinion. If the sum of the pitch cone angles of the gears is more or less than 90°, this rule does not hold true.

(2) Although the pitch diameters are calculated for bevel gears in the same manner as for spur gears, this does not apply to their outside diameters as the depth dimensions of the teeth are measured in a plane at right angles to the pitch line. As illustrated in figure 78, outside bevel gear diameters may be calculated by means of the following formula:

Outside diameter = pitch diameter + (cosine of pitch cone angle × 2 addenda).

(3) If the faces of the bevel gears were made the full length of their pitch cones, the teeth cut thereon would be of zero dimensions at the apex; therefore, the gears are made as frustums of these cones. This length of face does not generally exceed one-third the outside cone radius, which may be found by means of the following formula:

Outside cone radius = secant of pinion pitch angle ×
$$\frac{\text{pitch diameter of gear}}{2}$$

(4) By reference to figure 78, it may be noted that the face, pitch, and bottom angles of the gear tooth converge at the pitch cone apex,
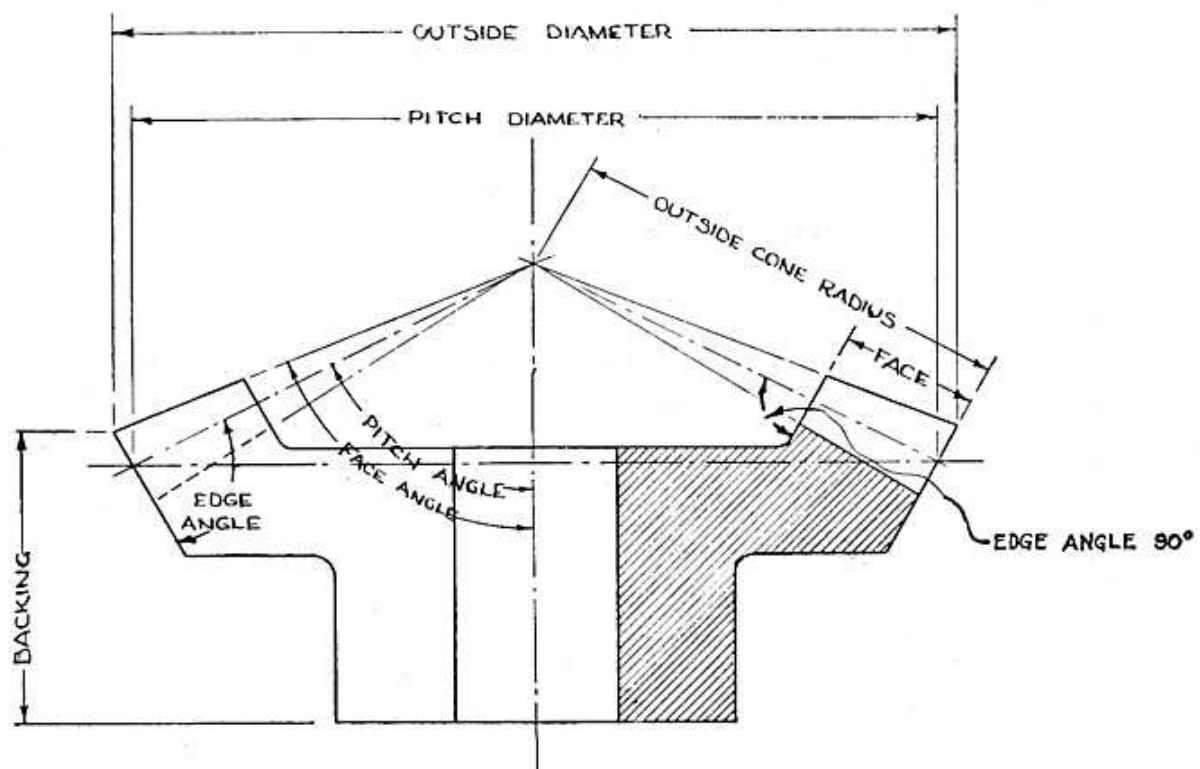
FIGURE 78.—Bevel gear nomenclature.

causing the face angle to be greater than that of the pitch angle and the cutting angle of the tooth to be less. To obtain the face angle, it is therefore necessary to add the pitch angle to the angular distance necessary to form the face angle. This angular distance may be found by means of the following formula:

$$\text{Tangent of angle} = \frac{\text{addendum}}{\text{outside cone radius}}$$

(5) The compound rest of the lathe is used to machine the angles of the bevel gears, and the back and front edges of the blank are machined at 90° to the pitch cone or center angle.

56. Worm gears.—*a.* Worm gears are used to connect shafts which are perpendicular to each other and where a great reduction of speed is required with a corresponding increase in power. A pair of worm gears consists of a worm, made similarly to a screw, whose threads mesh with the teeth of a gear called a worm wheel (fig. 79).
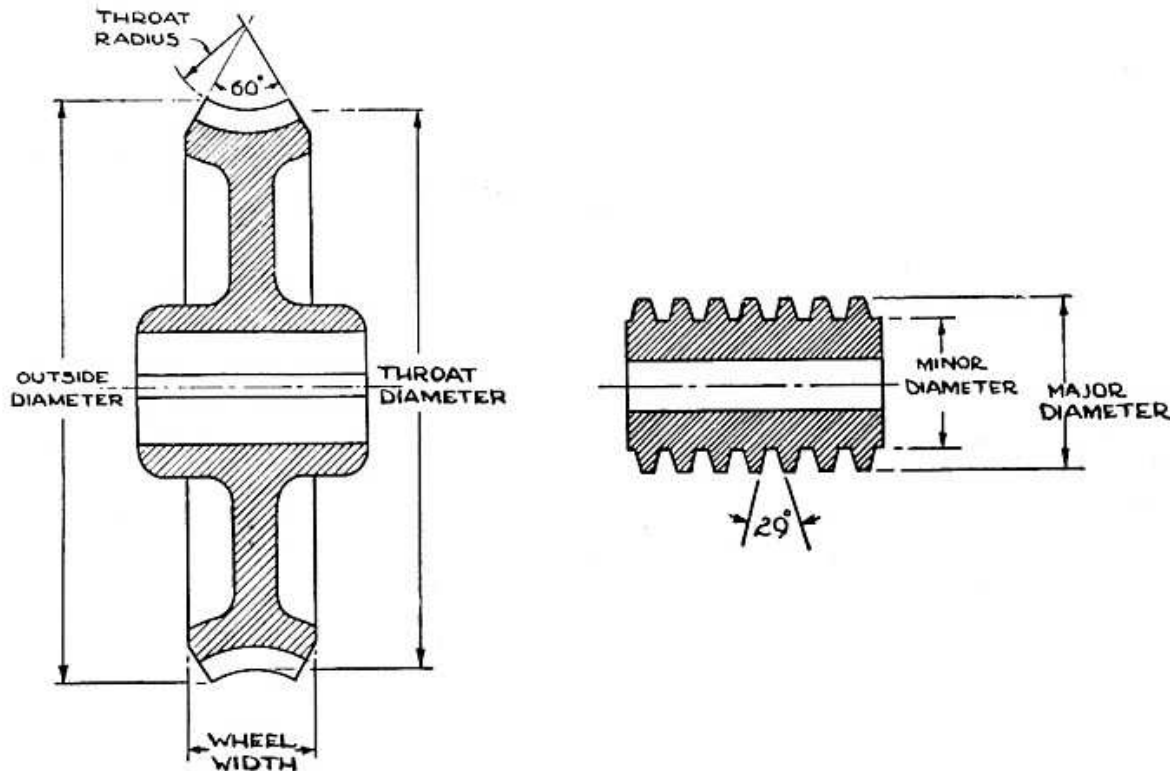
FIGURE 79.—Worm and worm wheel nomenclature.

b. The complete worm is machined in the lathe, including the cutting of the Brown and Sharpe 29° worm thread. The commonly used proportions are—

Pitch diameter=2.4×linear pitch+1.1.
Outside diameter=3.036×linear pitch+1.1.
Minor diameter=1.664×linear pitch+1.1.

c. Worms may be double or triple threaded to increase the speed without decreasing the center distance.

d. The worm wheel blank is machined in the lathe, and the teeth are gashed and hobbed in the milling machine.

(1) As shown in figure 79, the throat diameter of a worm wheel is measured at the center of the concave face, and the outside diameter is measured over the corners.

(2) The throat curvature may be machined by the use of a radius forming tool or by turning free hand and fitting to a template.

(3) The commonly used proportions for turning the worm wheel blank are—

Pitch diameter = number of teeth × 0.3183 × linear pitch.
Throat diameter = pitch diameter + (0.636 × linear pitch).
Outside diameter = throat diameter + (0.4775 × linear pitch).
Radius of wheel face = 0.882 × linear pitch + 0.55.
Face width = 2.38 × linear pitch + 0.25.

INDEX

[A. G. 062.11 (9–24–40).]

By order of the Secretary of War:

G. C. MARSHALL,
Chief of Staff

Official:

E. S. ADAMS,
Major General,
The Adjutant General.