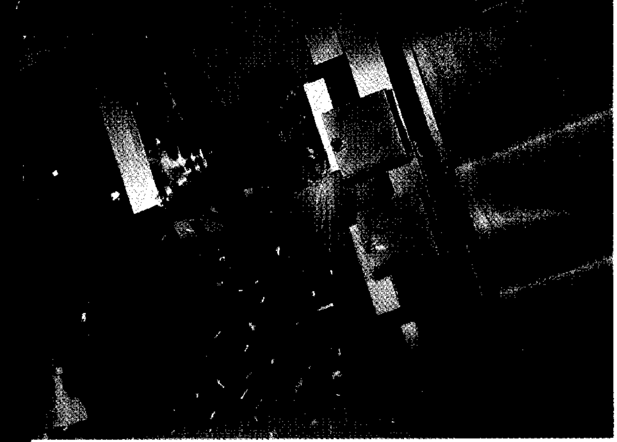


Chapter 14

Cutting Tapers and Screw Threads on the Lathe

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- Describe how a taper is turned on a lathe.
- Calculate tailstock setover for turning a taper.
- Safely set up and operate a lathe for taper turning.
- Describe the various forms of screw threads.
- Cut screw threads on a lathe.

IMPORTANT TERMS

external threads
internal threads
major diameter
minor diameter
offset tailstock method
pitch diameter
setover
taper attachment
thread cutting stop
three-wire method of measuring threads

14.1 TAPER TURNING

A section of material is considered to be ***tapered*** when it increases or decreases in diameter at a uniform rate, **Figure 14-1**. A cone is an example of a taper. The "wedging" action of a taper makes it ideal as a means for driving drills, milling arbors, end mills, and centers. In addition, it can be assembled and disassembled easily, and will automatically align itself in a similarly tapered hole each time. Taper can be stated in taper per inch, taper per foot, degrees, millimeters per 25 mm of length, or as a ratio, **Figure 14-2**.

There are five principal methods of machining tapers on a lathe. Each has its advantages and disadvantages. The five methods are listed in **Figure 14-3.**

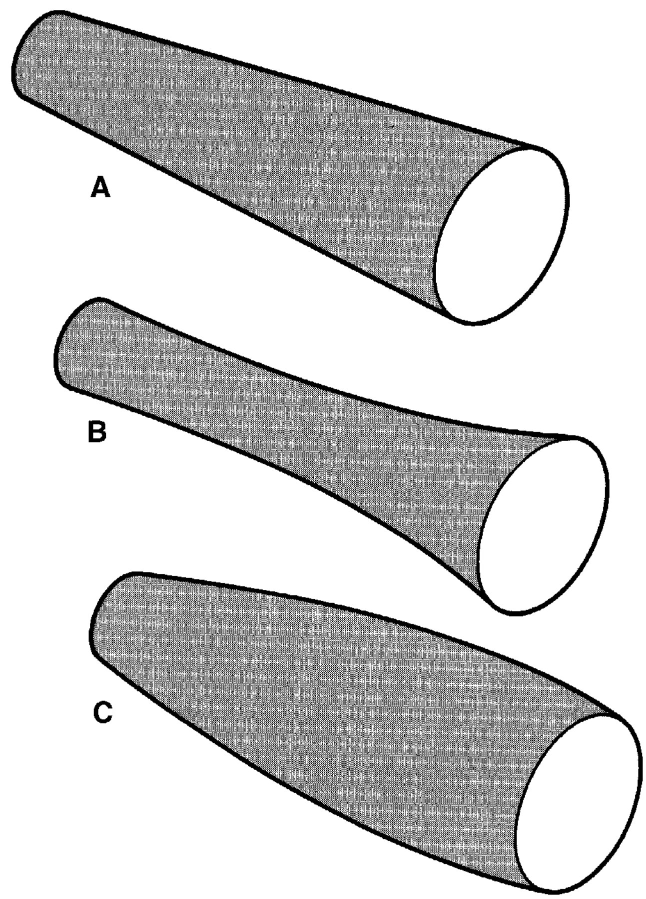


Figure 14-1*. Taper. A—The diameter of a taper increases or decreases at a uniform rate. B and C—These pieces are "bell shaped," rather than tapered.*

14.1.1 Taper turning with compound rest

The compound rest method of turning a taper is the easiest. Either internal or external tapers can be machined, as shown in **Figure 14-4**.

Taper length is limited, however, by the movement of the compound rest. Because the compound rest base is graduated in degrees, **Figure 14-5**, the taper must be converted to degrees. A conversion table may be used. See **Figure 14-6.**

A careful study of the print will show whether the angle given is from center, or is the included angle. **Figure 14-7** shows the difference in methods of measuring angles. If an ***included angle*** is given, it must be divided by two to obtain the ***angle from the centerline.***

With the lathe's centerline representing 0°, pivot the compound rest to the desired angle and lock it in position. It is the usual practice to turn a taper from the smaller diameter to the larger diameter. Refer to **Figure 14-4A**.

As will be the case when turning all tapers, the cutting tool must be set on *exact center*. A toolholder that will provide ample clearance should be selected.

To machine a taper, bring the cutting tool into position with the work and lock the carriage to prevent it from shifting during the turning operation.

Since there is no power feed for the compound rest, the cutting tool must be fed evenly with both

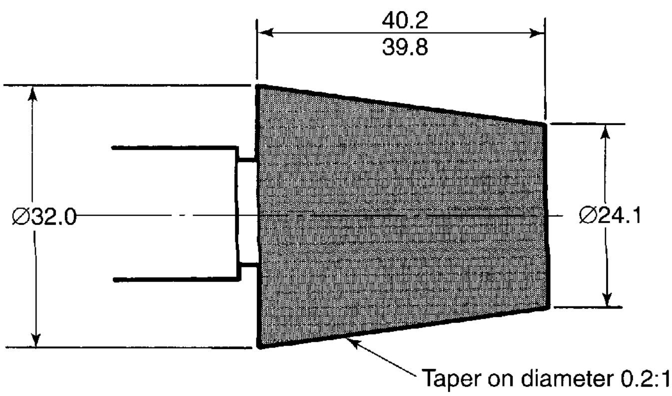


Figure 14-2*. Taper may be stated as a ratio (0.2:1, in the example above), taper per inch, taper per foot, degrees, or in millimeters per 25 mm.*

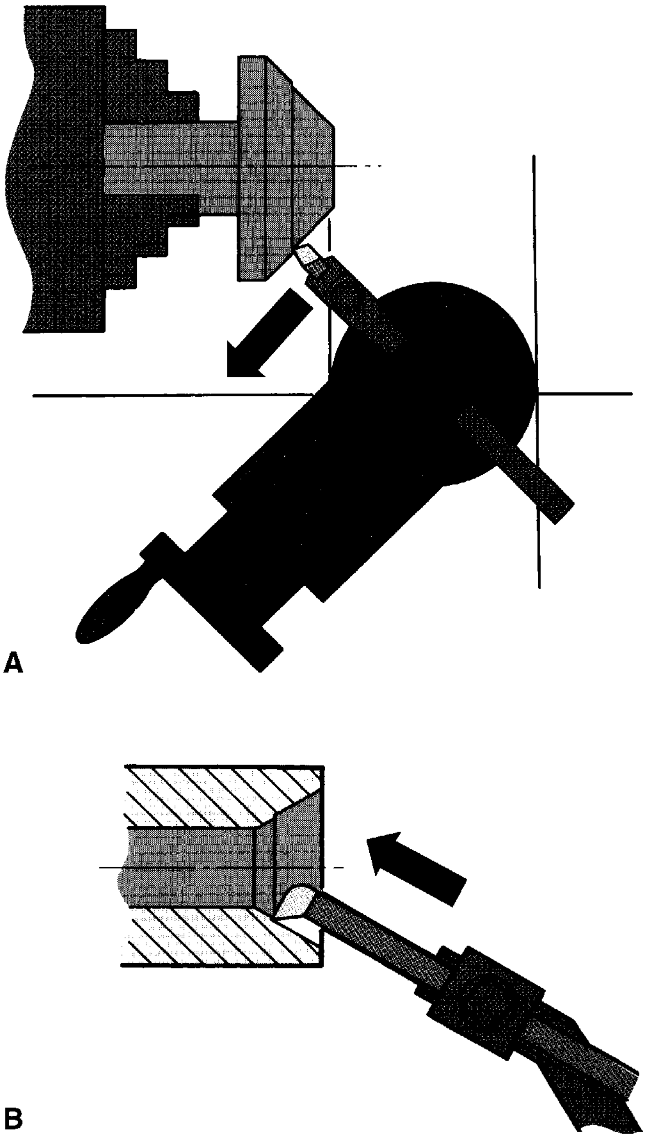


Figure 14-4*. Cutting tapers using the compound rest. A—External taper. Note that the cut is being made from small diameter to large diameter. B—Internal taper being turned with the compound rest.*

Ways of Machining Tapers		
Method	**Advantages and disadvantages**	**Information needed**
1. Compound	Length of taper limited. Will cut external and internal taper.	Must know the taper angle.
2. Offset tailstock	External taper only. Must work between centers.	Taper per inch or taper per foot.
3. Taper attachment	Best method to use.	Angle or taper per inch or foot.
4. Tool bit	Very short taper.	Taper angle.
5. Reamer	Internal only.	Taper number.

Figure 14-3*. Methods by which tapers can be turned on a lathe.*

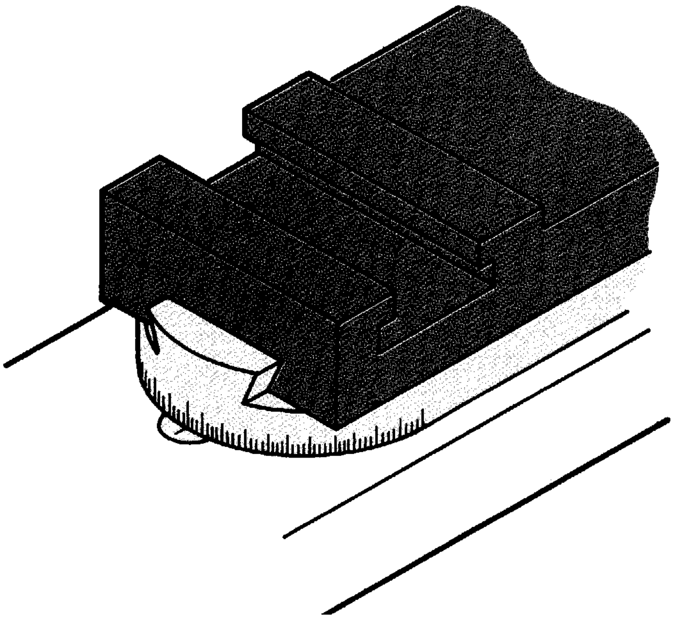


Figure 14-5. Base of the compound rest is marked in degrees to aid in precise positioning.

Taper per Foot with Corresponding Angles		
Taper per foot	Included angle	Angle with centerline
1/16	0° 17′ 53″	0° 8′ 57″
1/8	0° 35′ 47″	0° 17′ 54″
3/16	0° 53′ 44″	0° 26′ 52″
1/4	1° 11′ 38″	0° 35′ 49″
5/16	1° 29′ 31″	0° 44′ 46″
3/8	1° 47′ 25″	0° 53′ 42″
7/16	2° 5′ 18″	1° 2′ 39″
1/2	2° 23′ 12″	1° 11′ 36″
9/16	2° 41′ 7″	1° 20′ 34″
5/8	2° 58′ 3″	1° 29′ 31″
11/16	3° 16′ 56″	1° 38′ 28″
3/4	3° 34′ 48″	1° 47′ 24″
13/16	3° 52′ 42″	1° 56′ 21″
7/8	4° 10′ 32″	2° 5′ 16″
15/16	4° 28′ 26″	2° 14′ 13″
1	4° 46′ 19″	2° 23′ 10″

Figure 14-6. You can use this table to convert taper per foot into corresponding angles for adjustment of the compound rest.

hands to achieve a smooth finish. The entire cut must be made without stopping the cutting tool. The compound rest is moved back to the starting point and positioned with the cross-slide for the next cut.

When tapers are cut with a compound rest, the work can be mounted between centers or held in a chuck. A suitable boring bar is needed when machining internal tapers. Some internal tapers are finished to size with a taper reamer.

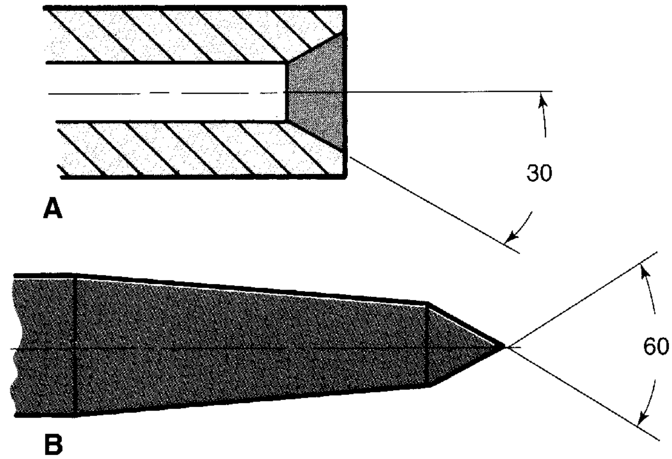


Figure 14-7. The two methods used to measure angles. A—Angle measured from the centerline of the workpiece. B—Measurement of the included angle.

14.1.2 Taper turning by offset tailstock method

The *offset tailstock method,* also known as the *tailstock setover method,* is also employed for taper turning, **Figure 14-8**. Jobs that can be turned between centers may be taper turned by this technique. Only *external* tapers can be machined in this way, however.

Most lathe tailstocks consist of two parts, which permits the upper portion to be shifted off center, **Figure 14-9**. This movement, referred to as *setover,* is accomplished by loosening the anchor bolt that locks the tailstock to the ways, then making the proper adjustments with screws on the tailstock. After the setover has been made, the screws are drawn up *snug,* but not tight.

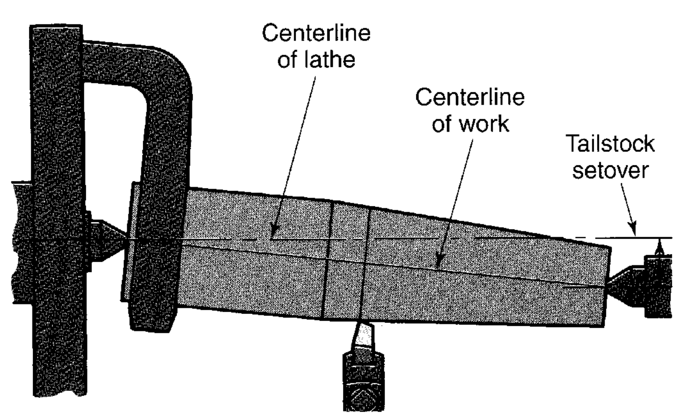


Figure 14-8. Machining a taper using the offset tailstock method.

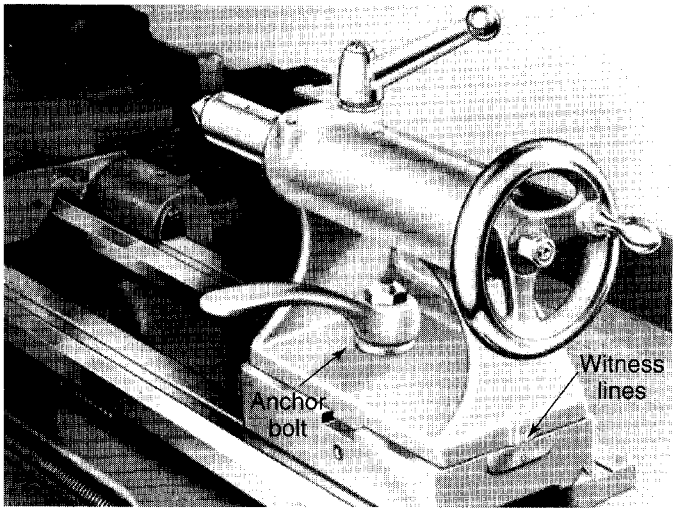


Figure 14-9. The tailstock is usually constructed in two parts. This allows the section mounting the center to be shifted relative to the lathe's centerline. The distance off center, or setover, can be checked by observing the witness lines. (Rockwell International)

14.2 CALCULATING TAILSTOCK SETOVER

Taper turning by this technique is *not* a precise method and requires some "trial and error" adjustments to produce an accurate tapered section. The approximate setover can be calculated when certain basic information is known.

Offset must be calculated for each job, because the length of the piece plays an important part in the calculations. When lengths of the pieces vary, different tapers will be produced with the same tailstock offset, **Figure 14-10**.

The following terms are used with calculating tailstock setover. See **Figure 14-11**.

D = Diameter at large end
d = Diameter at small end
ℓ = Length of taper
L = Total length of piece
TPI = Taper per inch
TPF = Taper per foot

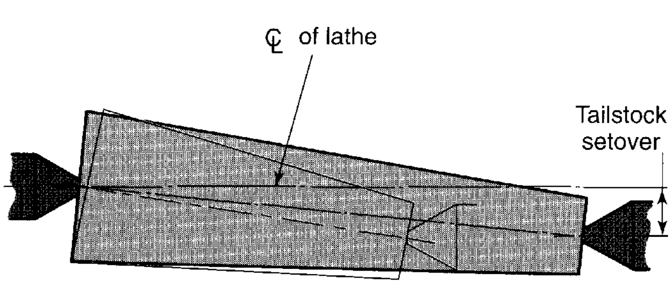


Figure 14-10. Length of work causes taper to vary even though tailstock offset remains the same.

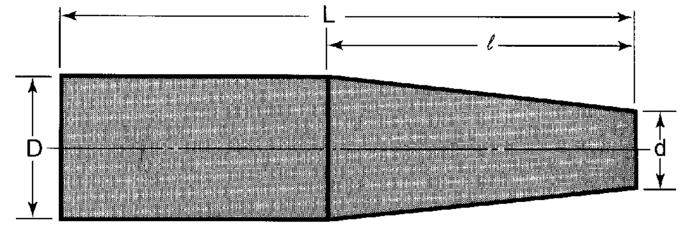


Figure 14-11. Basic taper information. D = diameter at large end of taper; d = diameter at small end of taper; ℓ = length of taper; L = total length of piece.

14.2.1 Calculating setover when taper per inch is known

Information needed:

TPI = Taper per inch
L = Total length of piece

Formula used: $\text{Offset} = \frac{L \times TPI}{2}$

Example: What will be the tailstock setover for the following job?

$$\begin{aligned} \text{Taper per inch} &= 0.0125 \\ \text{Total length of piece} &= 8.000 \\ \text{Offset} &= \frac{L \times TPI}{2} \\ &= \frac{8.000 \times 0.125}{2} \\ &= 0.500'' \end{aligned}$$

Note: The same procedure would be followed when using metric units. However, all dimensions would be in millimeters.

14.2.2 Calculating setover when taper per foot is known

When taper per foot (TPF) is known, it must be converted to taper per inch (TPI). The following formula takes this into account:

$$\text{Offset} = \frac{TPF \times L}{24}$$

14.2.3 Calculating setover when dimensions of tapered sections are known but TPI or TPF is not given

Plans often do not specify TPI or TPF, but do give other pertinent information. Calculations will be easier if all fractions are converted to decimals. All dimensions must either be in inches or in millimeters.

Information needed:

D = Diameter at large end
d = Diameter at small end
ℓ = Length of taper
L = Total length of piece

Formula used: $\text{Offset} = \dfrac{L \times (D - d)}{2\ell}$

Example: Calculate the tailstock setover for the following job.

$$D = 1.250''$$
$$d = 0.875''$$
$$\ell = 3.000''$$
$$L = 9.000''$$
$$\text{Offset} = \frac{L \times (D-d)}{2\ell} = \frac{9.000 \times (1.250 - 0.875)}{2 \times 3.000} = \frac{9.000 \times 0.375}{6} = 0.562''$$

14.2.4 Calculating setover when taper is given in degrees

The space available in this text does not permit the introduction of basic trigonometry, which is necessary to make these calculations. However, any good machinist's handbook will provide this information. At least one such book should be part of every machinist's toolbox.

Figure 14-12. Measuring setover. A—Approximate tailstock setover can be determined by measuring distance between center points. B—Approximate setover can also be determined by measuring distance between witness lines on the tailstock.

14.3 MEASURING TAILSTOCK SETOVER

When an ample tolerance is allowed, (±0.015″ or 0.05 mm), the setover can be measured with a steel rule. There are two ways to measure:

- Place a rule that has graduations on both edges between the center points, **Figure 14-12A**. Measure the distance between the center points.
- Measure the distance between the two witness lines on the tailstock base, **Figure 14-12B**.

Accurate work requires care in making the tailstock setover. An additional factor enters into the calculations—the distance that the center point enters the piece. Typically, 1/4″ (6.5 mm) is an ample allowance; it must be subtracted from the total length of the piece.

Use the appropriate method to calculate the offset. A precise setover may be made using the *micrometer collar* on the lathe cross-slide. See **Figure 14-13**.

1. Clamp the toolholder in a reverse position in the tool post.
2. Turn the cross-slide screw back to remove all play.
3. Turn in the compound rest until the toolholder can be felt with a piece of paper between the toolholder and tailstock spindle.
4. Use the micrometer collar and turn out the cross-slide screw the distance the tailstock is to be set over.
5. Move the tailstock over until the spindle touches the paper in same manner described in Step 3.
6. Check the setting again after "snugging up" the adjusting screws.

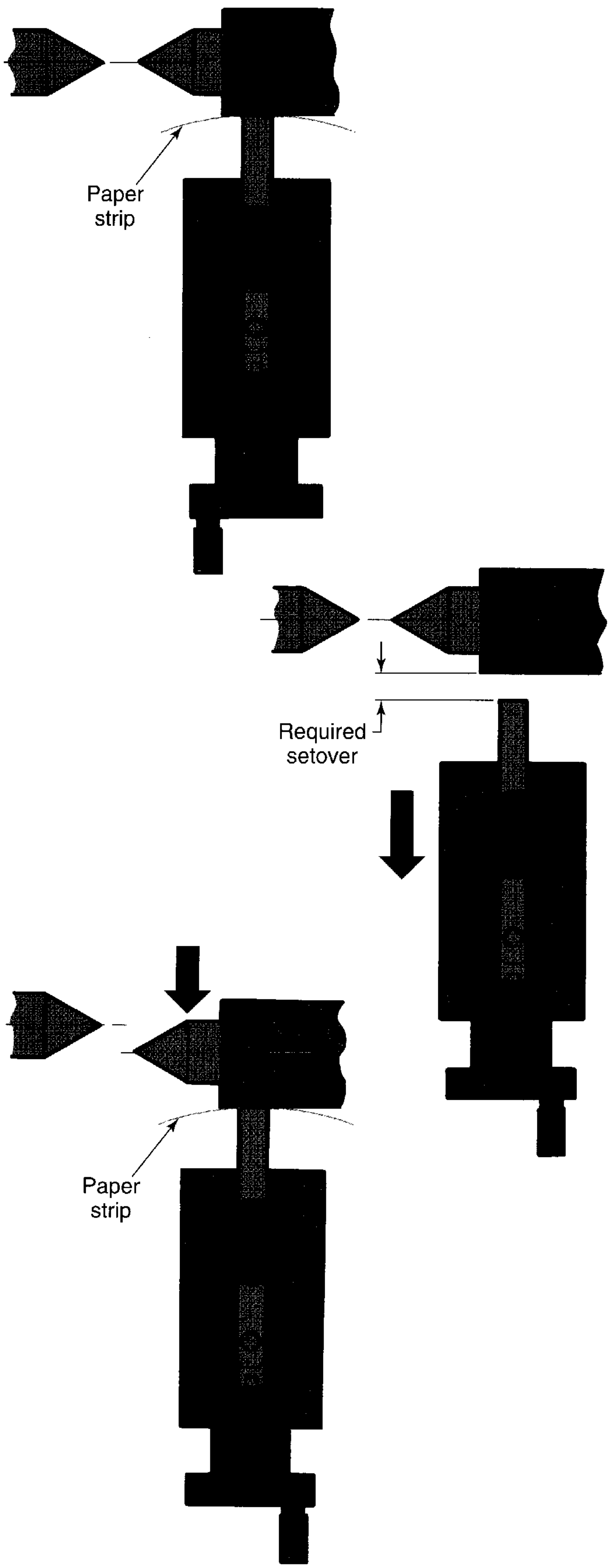


Figure 14-13. *Using the micrometer collar of the compound rest to make the setover measurement.*

In place of the toolholder and paper strip, a *dial indicator* can be employed to establish the offset. See **Figure 14-14.**

1. Mount the dial indicator in the tool post.
2. Position it with the cross-slide until the indicator reads zero when in contact with the tailstock spindle. There should be *no* "play" in the cross-slide.
3. Set the tailstock over the required distance using the dial indicator to make the measurement.
4. Recheck the reading after "snugging up" the adjusting screws. Make additional adjustments if any deviation in the indicator reading occurs.

14.4 CUTTING A TAPER

When cutting a taper, additional strain is imposed on the centers because they are out-of-line and do not bear true in the center holes. Because the pressures imposed are uneven, the work is more apt to heat up than when doing conventional turning between centers. It must be checked frequently for binding. A bell-type center drill offers some advantage in reducing strain. Some machinists prefer a center with a ball tip to produce an improved bearing surface. See **Figure 14-15.**

Make the cuts as in conventional turning. However, cutting should start at the small end of the taper.

14.4.1 Turning a taper with a taper attachment

A ***taper attachment*** is a guide that can be attached to most lathes. It is an accurate way to cut tapers and offers advantages over other methods of machining tapers.

Both internal and external tapers can be cut. This helps assure an accurate fit for mating parts. Once the attachment has been set, the taper can be machined on material of various lengths. Work can be held by any conventional means. One end of the taper attachment ***swivel bar*** is graduated in total taper in inches per foot. The other end is graduated to indicate the included angle of the taper in degrees.

The lathe does not have to be altered. The machine can be used for straight turning by locking out the taper attachment. No realignment of the lathe is necessary.

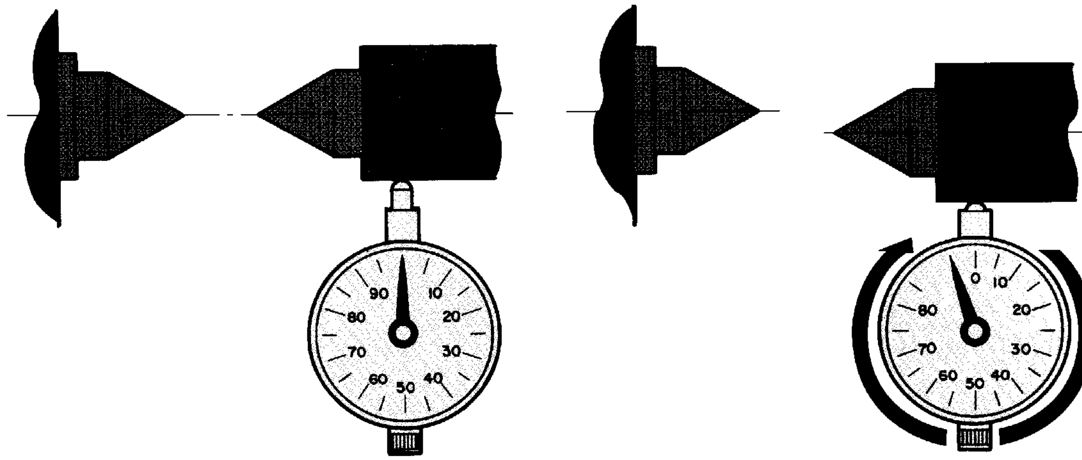


Figure 14-14. *A dial indicator can also be used to measure amount of setover.*

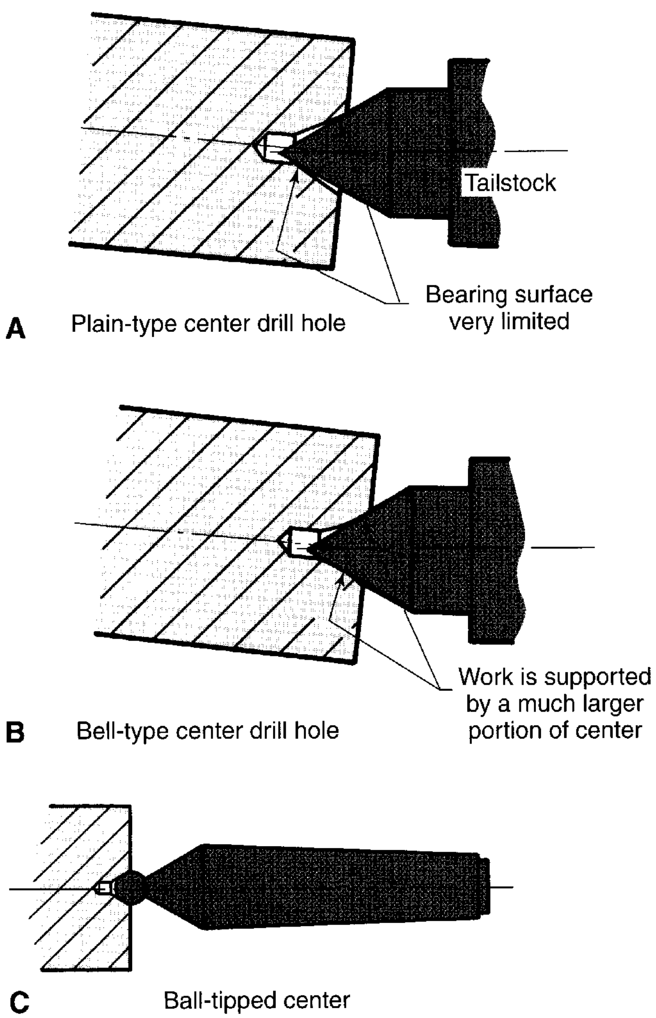


Figure 14-15. *Taper turning done by the offset tailstock method is hard on the tailstock center. A—Center point does not bear evenly in conventional center hole. B—A center hole drilled with a bell-type center drill reduces the problem by providing more bearing surface. C—A ball-tipped center lessens pressure on tail center when turning tapers.*

14.4.2 Types of taper attachments

There are two types of taper attachments, plain and telescopic. See **Figure 14-16**. The ***plain taper attachment*** requires disengaging the cross-slide screw from the cross-slide feed nut. The cutting tool is advanced by using the compound rest feed screw.

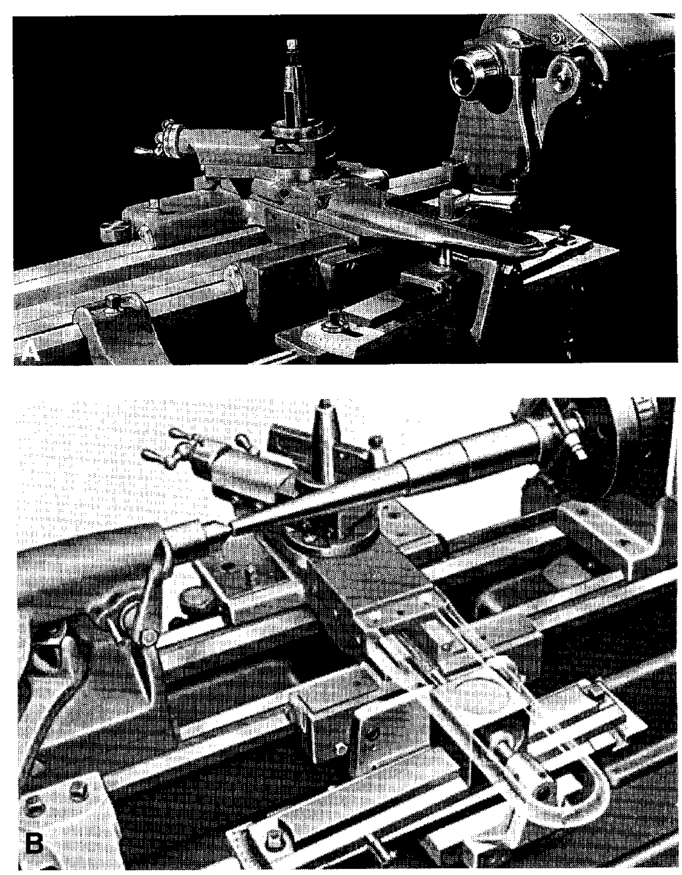


Figure 14-16. *Taper attachments. A—Plain taper attachment. (South Bend Lathe Corp.) B—Telescopic taper attachment. (Clausing Industrial, Inc.)*

The *telescopic taper attachment* is made in such a way that it is not necessary to disconnect the cross-slide feed nut. The tool can be advanced into the work with the cross-slide screw in the usual manner.

14.4.3 Setting a taper attachment

1. Study the plans and, if necessary, calculate the taper. Set the swivel bar as specified from the calculations.
2. Mount the work in the machine.
3. Slide the taper attachment unit to a position that will permit the cutting tool to travel the full length of the taper. Lock it to the ways.
4. Move the carriage to the right until the cutting tool is about 1″ (25 mm) away from the end of the work. This will permit any play to be taken up before the tool starts to cut.
5. If the machine is fitted with a plain taper attachment, tighten the binding screw that engages the cross-slide feed to the attachment.
6. Oil the bearing surfaces of the taper attachment and make a trial cut. If necessary, readjust until the taper is being cut to specifications. Complete the cutting operation.

14.4.4 Turning a taper with a square-nose tool

Using a square-nose tool is a taper technique limited to the production of *short tapers*, **Figure 14-17**. The cutter bit is ground with a square nose and set to the correct angle with the protractor head and blade of a combination set.

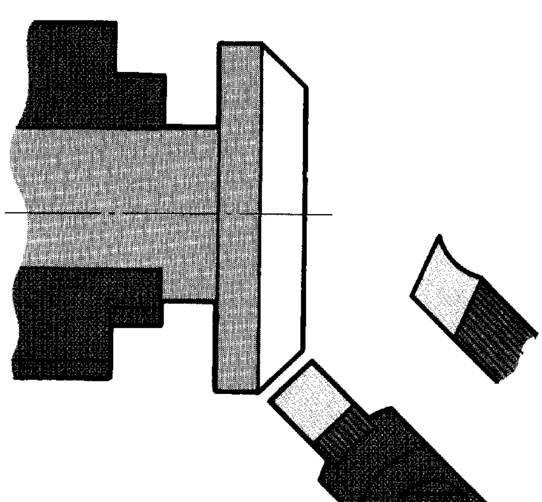


Figure 14-17*. A short taper can be turned with a square-nose tool.*

The tool is positioned on center and fed into the revolving work. "Chatter" can be minimized by running the work at a slow spindle speed. The carriage must be locked to the ways.

Before using any of the taper-turning techniques on work mounted between centers, it is very important that centers be ***"zeroed in"*** (put in perfect alignment). Then the necessary adjustments (tailstock setover, taper attachment adjustment) can be made.

14.5 MEASURING TAPERS

There are two basic methods of testing the accuracy of machined tapers. One is a ***comparison method;*** the other involves ***direct taper measurement.***

14.5.1 Measuring tapers by comparison

Taper plug gages and ***taper ring gages*** serve two purposes, **Figure 14-18**. They measure the basic diameter of the taper as well as the angle of slope. The angle is checked by applying ***bluing*** (usually a liquid known as "Prussian blue") to the machined surface or plug gage. The blued section is inserted into the mating part and slowly rotated. If the bluing rubs off *evenly,* it indicates that the taper is correct. If the bluing rubs off *unevenly,* **Figure 14-19**, the remaining material will show where the taper is incorrect and indicate what machine adjustments are needed.

Gages are also provided with notches to indicate the specified tolerance in taper diameter. The indentations show the *go* and *no-go* limits, **Figure 14-20.**

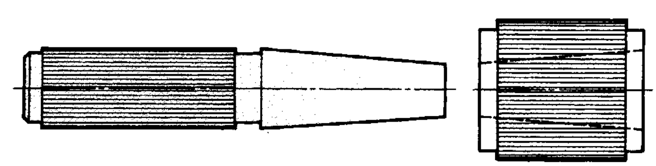


Figure 14-18*. Left—Plug gage. Right—Ring gage.*

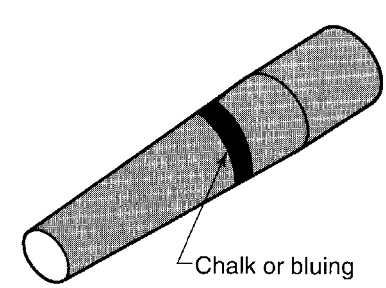


Figure 14-19*. When chalk or bluing does not rub off evenly, it indicates that taper does not fit properly and additional machine adjustments will have to be made.*

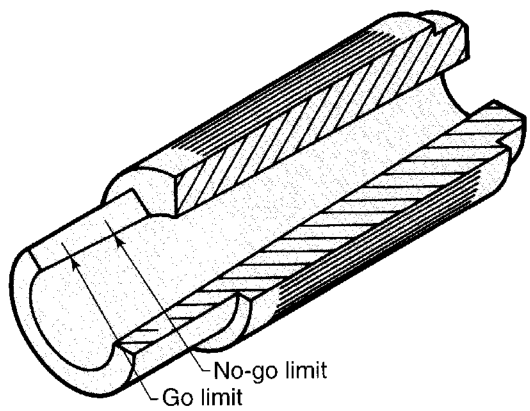


***Figure 14-20**. Typical go and no-go ring gage for measuring tapers.*

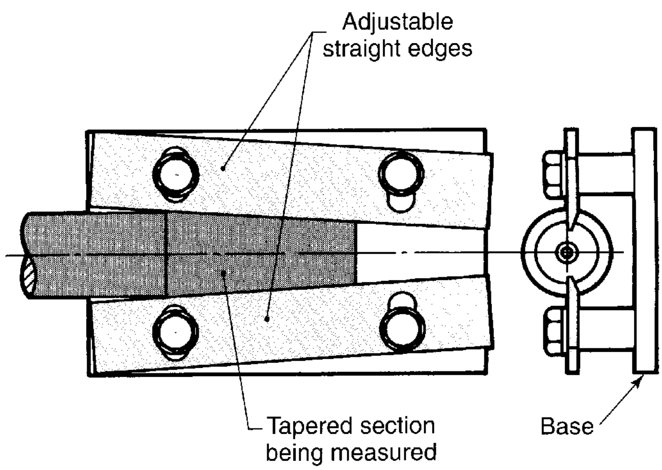


***Figure 14-21**. A taper test gage can be set for different tapers.*

14.5.2 Direct measurement of tapers

A ***taper test gage*** is sometimes employed to check taper accuracy, **Figure 14-21**. It consists of a base with two adjustable straight edges. Slots in the straight edges permit adapting the gage to check different tapers. The taper test gage is set by using two discs of known size which are located the correct distance apart.

Another technique for checking and/or measuring tapers is to set the tapered section on a surface plate. Two gage blocks or ground parallels of the same height are placed on opposite sides of the taper. Two cylindrical rods (sections of drill rod are satisfactory) of the same diameter are placed on the blocks. See **Figure 14-22**. The distance across the rods is then measured with a micrometer.

Blocks 1″, 3″, or 6″ (25, 75, or 150 mm) taller than those used for the first reading are substituted. The rods are the same diameter as those used to make the first reading. A second reading is made, **Figure 14-22B.** The taper per foot then can be determined. First, subtract to find the difference between the two measurements. Then multiply it by twelve (if the readings were made 1″ apart), by four (if they were made 3″ apart), or by two (if they were made 6″ apart).

A ***sine bar*** is a very accurately machined bar with edges that are parallel, **Figure 14-23**. The bar is used in conjunction with gage blocks and sine tables to precisely measure angles.

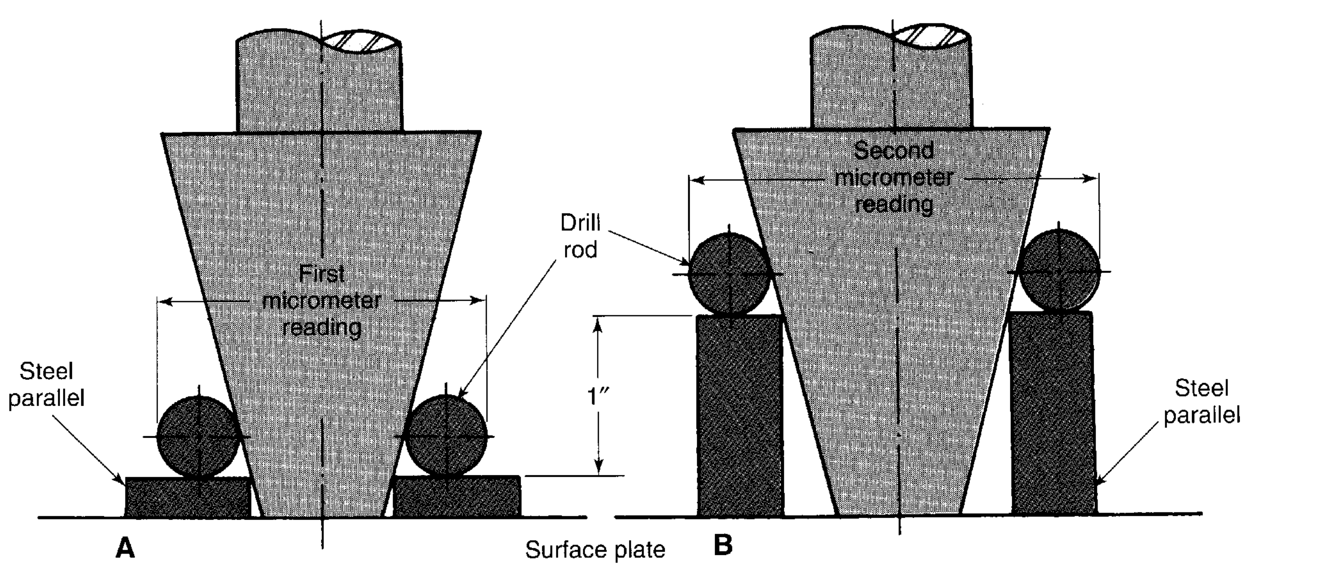


***Figure 14-22**. Measuring a taper using parallels, drill rod, micrometer, and a surface plate. A—Setup for first measurement. B—Setup for second measurement.*

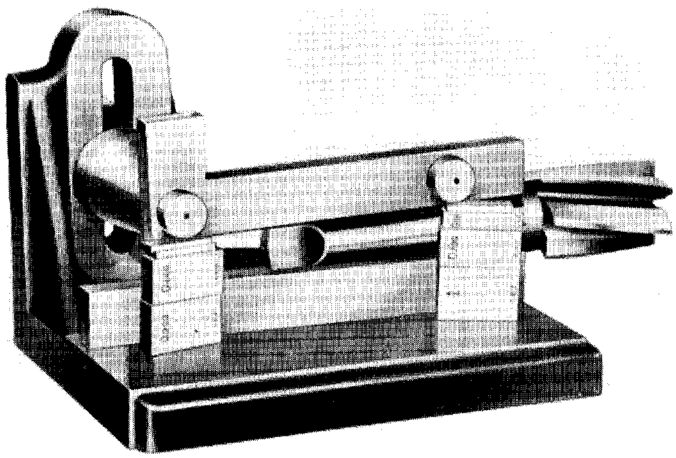


Figure 14-23. A sine bar and precision gage blocks can also be used to measure a taper. (C.E. Johannson Co.)

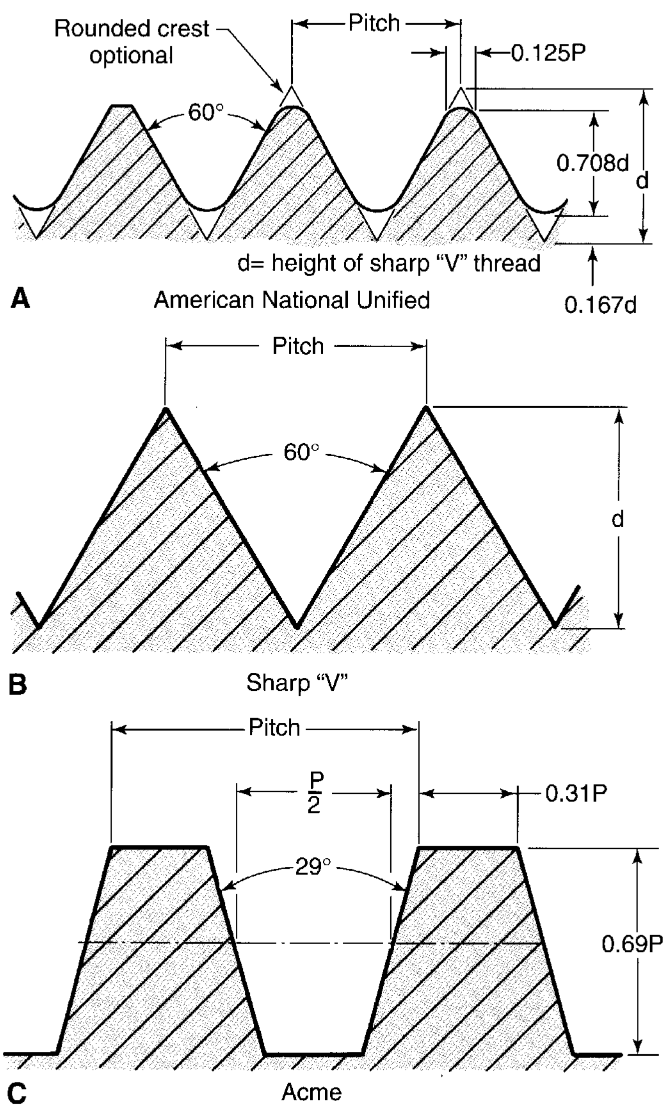
14.6 CUTTING SCREW THREADS ON THE LATHE

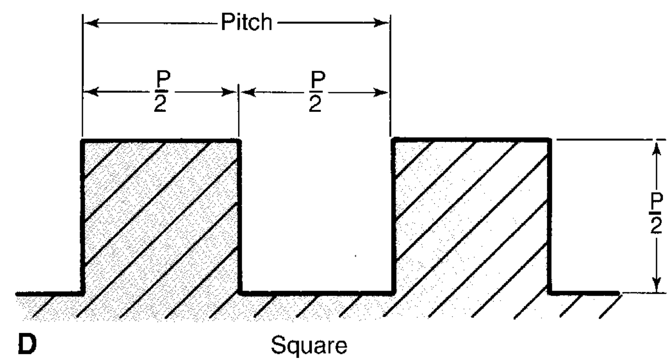
Screw threads are utilized for many applications. The more important are:

- Making adjustments (cross-feed on a lathe).
- Assembling parts (nuts, bolts, and screws).
- Transmitting motion (lead screw on a lathe).
- Applying pressure (clamps).
- Making measurements (micrometer).

14.6.1 Screw thread forms

The first screw threads cut by machine were square in cross-section. Since that time, many different thread forms have been developed, including American National, Unified, Sharp V, Acme, Worm threads, and others. Each thread form has a specific use and a formula for calculating its shape and size. See **Figure 14-24**. More than 75% of all threads cut in the United States are of the Unified (UN) 60° type.





Unified Thread Pitch = $\frac{1}{N}$ $d = \frac{0.866}{N}$

Sharp V Thread Pitch = $\frac{1}{N}$ $d = \frac{0.866}{N}$

Acme Thread Pitch = $\frac{1}{N}$ $d = \frac{P}{2 + 0.010}$

Flat = 0.371P

Root = 0.71P–0.0052

Square Thread Pitch = $\frac{1}{N}$ $d = \frac{P}{2}$

Flat or space = $\frac{P}{2}$

***Figure 14-24**. Common thread forms. A—Unified thread form, interchangeable with American National Thread. B—Sharp "V" thread form. C—Acme thread form. D—Square thread form. Note: In formulas above, N = Number of threads per inch; P = Pitch; d = depth of thread.*

The following terms relate to *screw threads,* as shown in **Figure 14-25**:

- *External threads* are cut on the outside surface of piece.
- *Internal threads* are cut on the inside surface of piece.
- *Major diameter* is the largest diameter of the thread.
- *Minor diameter* is the smallest diameter of the thread.
- *Pitch diameter* is the diameter of an imaginary cylinder that would pass through threads at such points to make width of thread and width of the spaces at these points equal.
- *Pitch* is the distance from one thread point to the next thread point, measured parallel to the thread axis. Pitch of inch-based threads is equal to 1 divided by the number of threads per inch.
- *Lead* is the distance that a nut will travel in one complete revolution of the screw. On a single thread, the lead and pitch are the same. Multiple thread screws have been developed to secure an increase in lead without weakening the thread. See **Figure 14-26.**

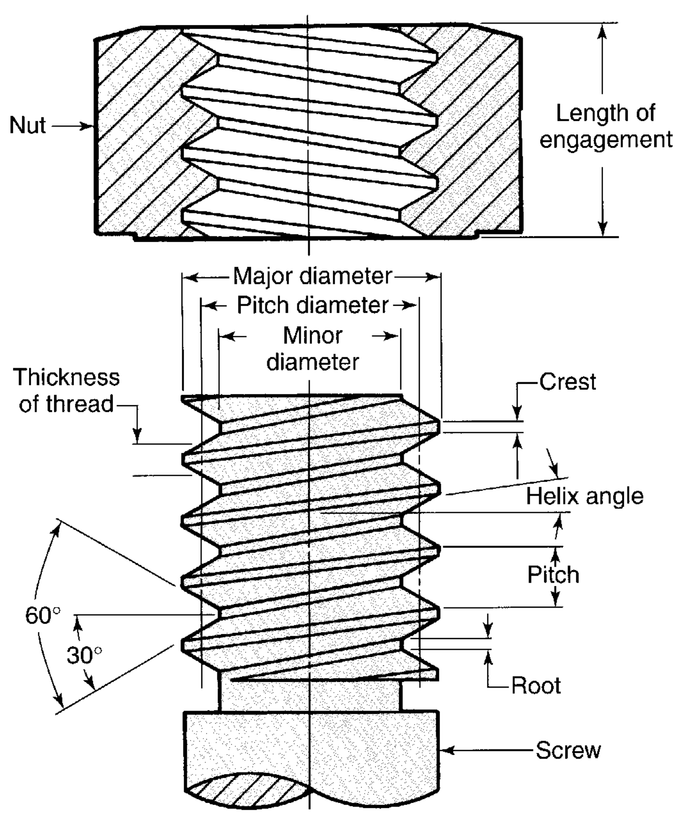


Figure 14-25. *Nomenclature of a thread.*

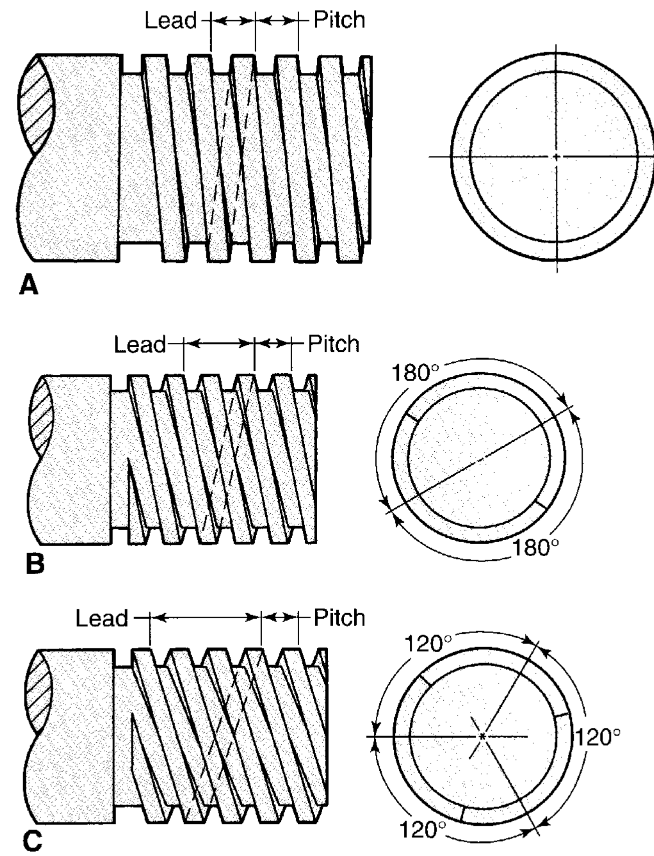


Figure 14-26. *The difference between lead and pitch. A—Single thread screw, the pitch and lead are equal. B—Double thread screw, the lead is twice the pitch. C—Triple thread screw, the lead is three times the pitch.*

14.6.2 Preparing to cut 60° threads on a lathe

Sharpen the cutting tool to the correct shape, including the proper clearance. The top is ground flat with no side or back rake, **Figure 14-27**. An oilstone is used to touch up the cutting edges and form the radius on the tip.

A *center gage* is used for grinding and setting the tool bit in position, **Figure 14-28**. The gage is often referred to as a *fishtail.*

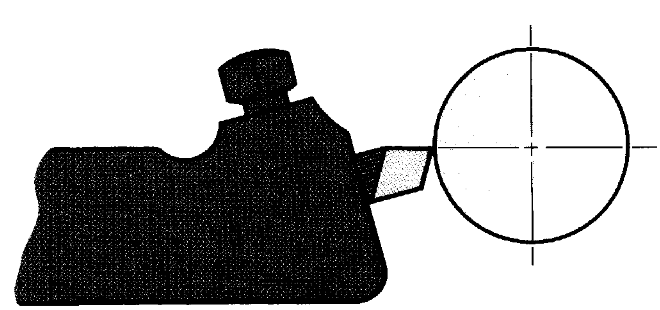


Figure 14-27. *Cutting tool positioned for cutting 60° threads. The tool is set on center as shown.*

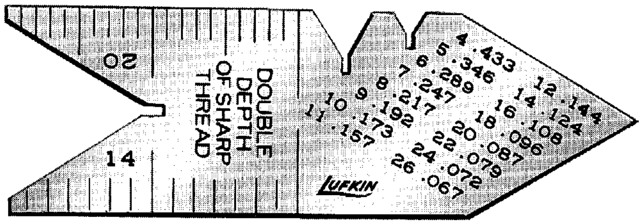


Figure 14-28. *A center gage or "fishtail." (Lufkin Rule Co.)*

The work is set up in the same manner as for straight turning. If mounted between centers, the centers must be precisely aligned; otherwise, a tapered thread will be produced. If this occurs, the thread will not be usable unless it is cut excessively deep at one end. The work must also run true with no "wobble." The tail of the lathe dog must have no play in the face plate slot.

A groove is frequently cut at the point where the thread is to terminate, **Figure 14-29**. The ***thread end groove*** is cut equal to the minor diameter of the thread and serves two purposes:

- It provides a place to stop the threading tool at the end of its cut.
- It permits a nut to be run up to the end of the thread.

Several methods may be employed to terminate a thread, as shown in **Figure 14-29**. Ordinarily, the beginner should use a groove until sufficient experience has been gained. However, the design of some parts does not permit a groove to be used. In such a case, the threads must be terminated by another method. They require perfect coordination and very rapid operation of the cross-slide to get the tool out of position at the end of the cut.

The gearbox is adjusted to cut the correct number of threads. Make apron adjustments to permit the half-nuts to be engaged. After the proper apron and gear adjustments have been made, pivot the compound rest to 29° to the right, **Figure 14-30**. Then set the threading tool in place.

It is essential that the tool be set on center with the tool axis at 90° to the centerline of the work.

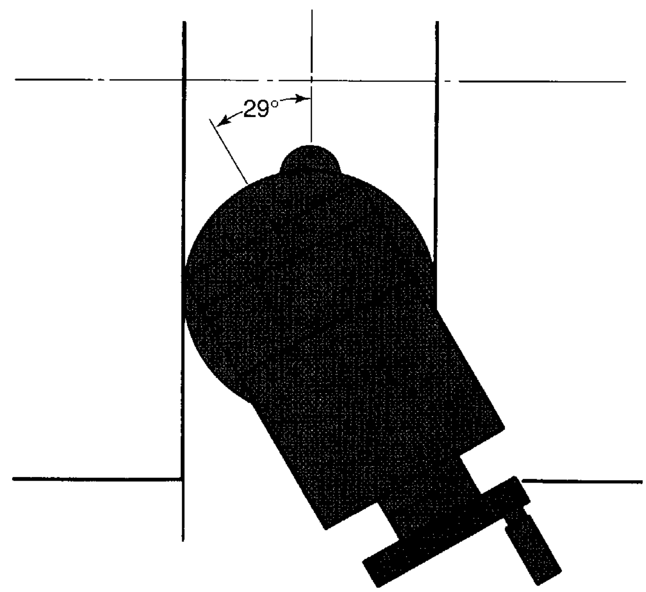


Figure 14-30. *The compound rest is set up for machining right-hand external threads.*

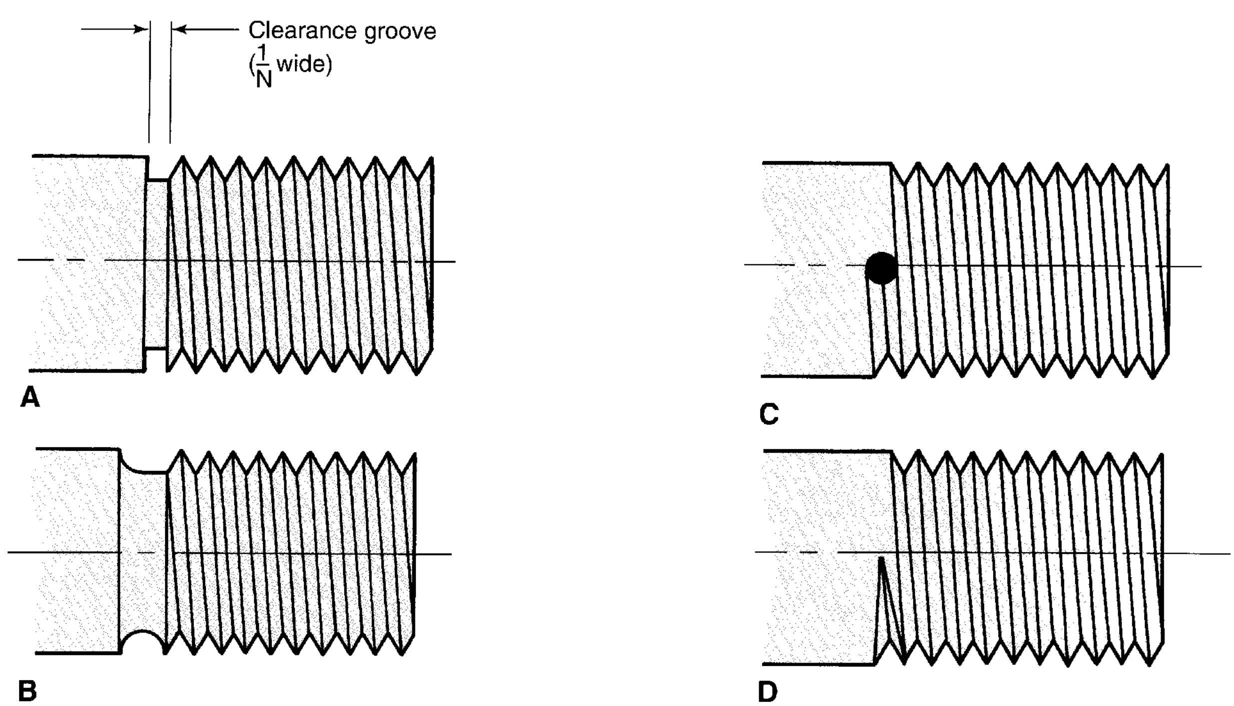


Figure 14-29. *Techniques for terminating a screw thread. A—Square groove. B— Round groove. C—Small shallow hole. D—Tool withdrawn from thread at end of cut.*

This is done with the aid of a center gage. Place the gage against the work while the tool is set into a V, **Figure 14-31**. Tool height can be set by using the centerline scribed on the tailstock spindle or with the center point.

The compound rest is set at 29° to permit the tool to shear the chip better than if it were fed straight into the work, **Figure 14-32**. Since the angle of the tool is 30° and it is fed in at an angle of 29°, the slight shaving action that results will produce a smooth finish on the right side of the thread. At the same time, not enough metal is removed to interfere with the main chip that is removed by the left edge of the tool.

Since the tool must be removed from the work after each cut and repositioned before the next cut can be started, a ***thread cutting stop*** may be used. After the point of the tool is set to just touch the work, lock the stop to the saddle dovetail with the adjusting screw just bearing on the stop, **Figure 14-33**.

After a cutting pass has been made, move the tool back from the work with the cross-slide screw. Move the carriage back to start another cut. Feed the tool into the work until the adjusting screw again bears against the thread cutting stop. By turning the compound rest in a distance of 0.002″ to 0.005″ (0.05 mm to 0.12 mm), the tool will be positioned for the next cut.

A thread dial that meshes with the lead screw is fitted to the carriage of most lathes, **Figure 14-34**. The ***thread dial*** is used to indicate when to engage the half-nuts, which permit the tool to follow exactly in the original cut. The thread dial eliminates the need to reverse spindle rotation after each cut to bring the tool back to the starting point.

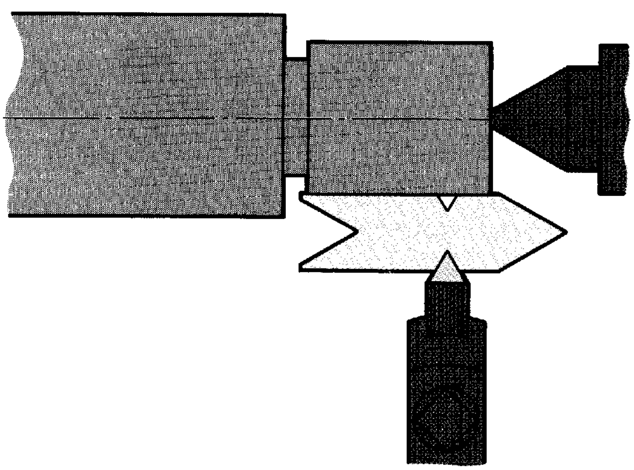


Figure 14-31*. Positioning a cutting tool for machining threads, using a center gage.*

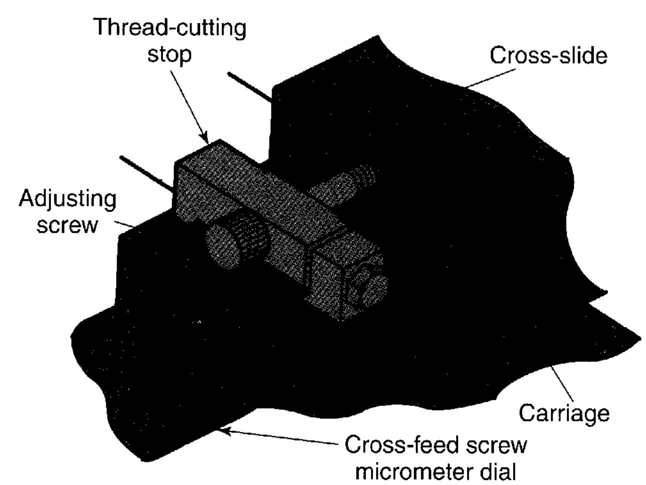


Figure 14-33*. After being properly adjusted, the thread cutting stop will let you start next cut in same location.*

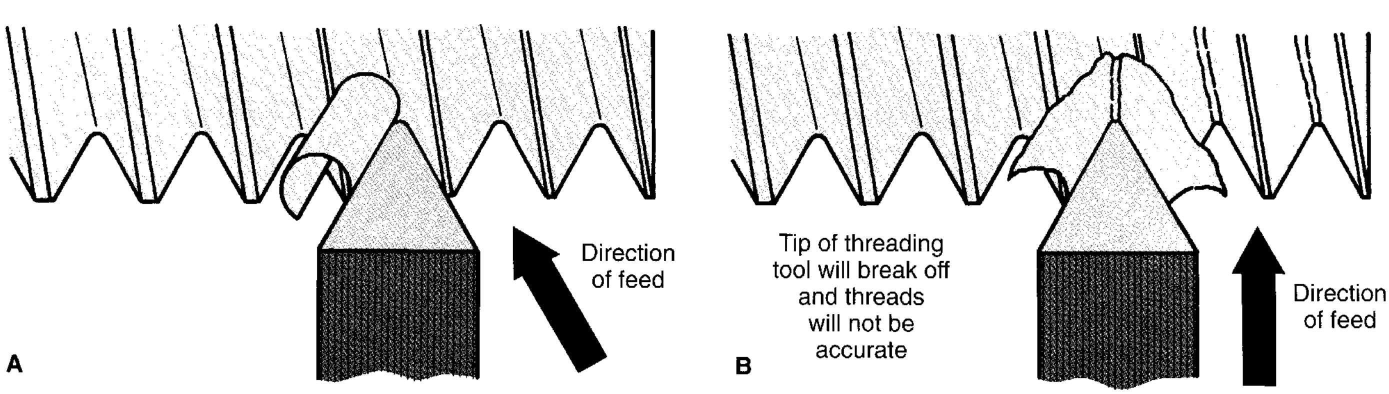


Figure 14-32*. Cutting action of tool. A—When the tool is fed in at 29° angle, note that only one edge is cutting, and that the cutting load is distributed evenly across the edge. B—When fed straight in, note that both edges are cutting and weakest part of tool, the point, is doing hardest work.*

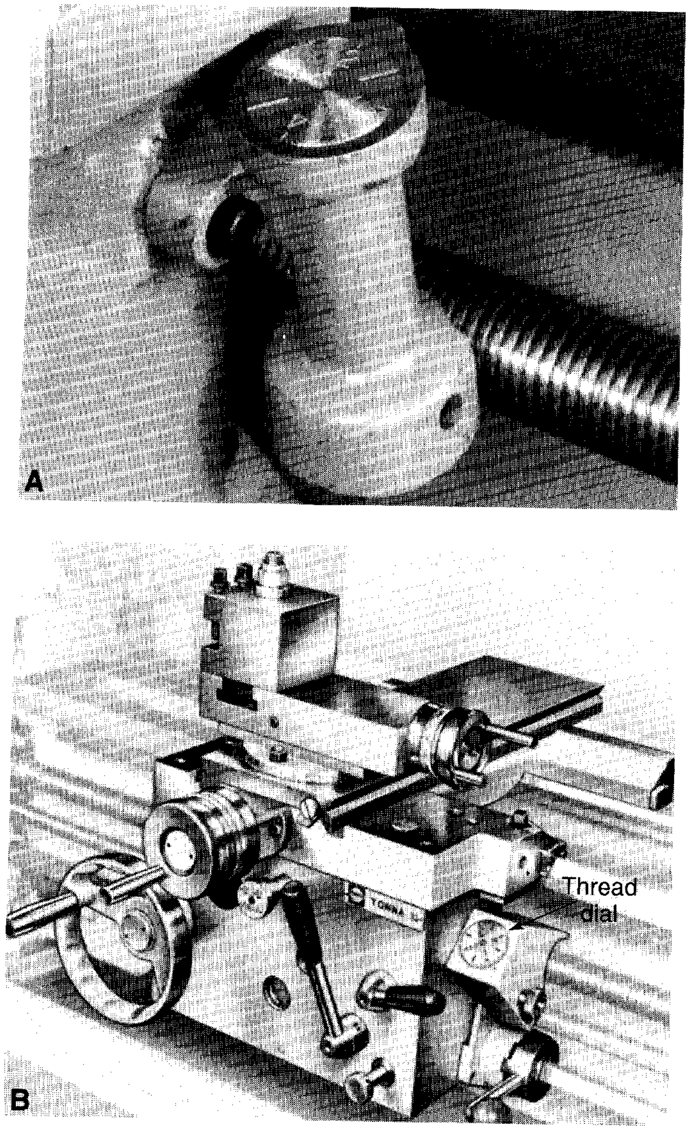


Figure 14-34. Thread dials. A—Thread dial for cutting inch-based threads. B—Dial used for cutting either inch-based or metric-based threads. The housing contains a series of gears, with gear selection depending upon threads being cut. (Clausing Industrial, Inc.)

The face of the thread dial, **Figure 14-35,** rotates when the half-nuts are *not* engaged. When the desired graduation moves into alignment with the index line, the half-nuts can be engaged.

The thread dial is used as follows for all inch-based threads:

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

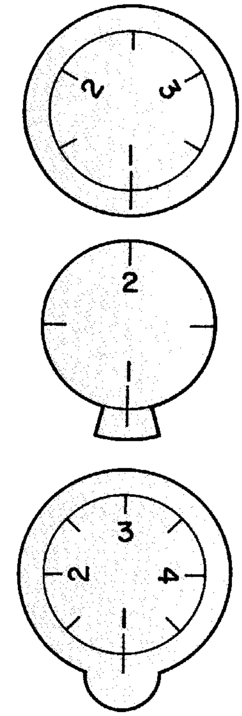


Figure 14-35. Typical thread dial faces.

- For all threads involving *one-fourth of a thr in each inch* (such as 4 3/4), return to the or inal starting line before closing the half-nu

On lathes that have been *converted* to me threading capability, the thread dial cannot be us When thread cutting with such a lathe, the half-n (once closed) must not be opened until the threac completely cut. The spindle rotation must reversed after each cut to return the tool to its sta ing position.

The thread dial *can* be used, however, on lath with full metric capabilities. The thread dial w vary with the lathe manufacturer and must be co sidered individually. To be sure of correct threa dial procedure, consult the manufacturer's han book for the machine.

14.6.3 Making the cut

Set the spindle speed to about one-fourth th speed that is used for conventional turning. Feed i the tool until it just touches the work. Then, mov the tool beyond the right end of the work and adjus it to take a 0.002" (0.05 mm) cut.

Turn on the power and engage the half-nut when indicated by the thread dial. This cut is made to check whether the lathe is producing the correc threads. Thread pitch can be checked with a rule or with a screw pitch gage, **Figure 14-36.** When

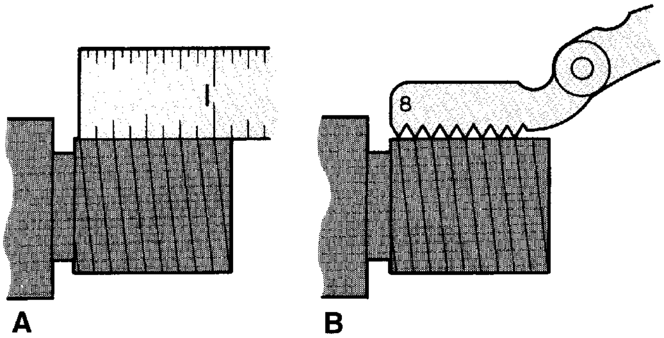


Figure 14-36. Always check thread pitch after first light cut has been made. A—Checking with a rule. B—Checking with a screw pitch gage.

everything checks, make additional cuts, working in 0.005″ (0.12 mm) increments, until the thread is almost to size. The last few cuts should be no more than 0.002″ (0.05 mm) deep. Note that all advances of the cutting tool are made with the compound rest feed screw.

A liberal application of cutting oil, before each cut, will help to obtain a smooth finish.

14.6.4 Resetting tool in thread

It is sometimes necessary to replace a broken cutting tool, or to resharpen it for the finish cuts. After replacing the tool, you must realign it with the portion of the thread already cut. This can be done as follows:

1. Set the tool on center and position it with a center gage.
2. Engage the half-nuts at the proper thread dial graduation.
3. Move the tool back from the work and rotate the spindle until the tool reaches a position about halfway down the threaded section.
4. Using the compound rest screw and the cross-slide screw, align the tool in the existing thread. Reset the thread cutting stop after the tool has been aligned.

14.6.5 Cutting threads with insert-type cutting tools

There are two basic types of 60° threading serts, the partial profile insert and the full-profile ert.

ᴾ*artial-profile inserts,* **Figure 14-37,** are most nly used because they can cut a range of itches. However, the major diameter (OD) ead must be cut to size prior to threading.

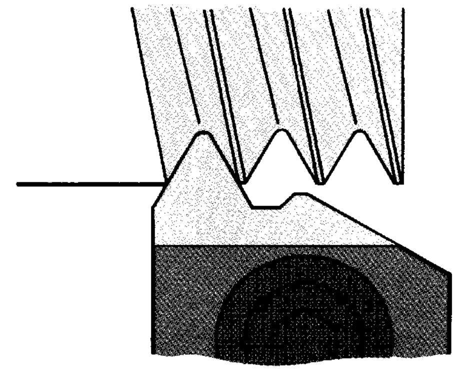


Figure 14-37. Cutting threads with a partial-profile insert. The major (outside) diameter of the thread must be cut to size before using this type insert.

Deburring may be required when cutting threads on most metals.

Full-profile inserts, **Figure 14-38,** produce the best thread form and finish. The tool cuts the leading flank, the root, and the trailing flank simultaneously. The machinist needs only to check the pitch diameter to determine if the major and minor diameters of the thread are to size. No deburring is necessary since the insert trims the thread crest. The disadvantage of the full-profile insert is that a separate insert is required for each thread pitch.

14.6.6 Measuring threads

Measure threads at frequent intervals during the machining operation to assure accuracy. The easiest way to check thread size is to try fitting the threaded piece into a threaded hole or nut of the proper size. If the piece does not fit, it is too large and further machining is necessary. This technique is not very accurate, but is usually satisfactory when close tolerances are not specified.

A ***thread micrometer*** can be used to make quick, accurate thread measurements. It has a pointed

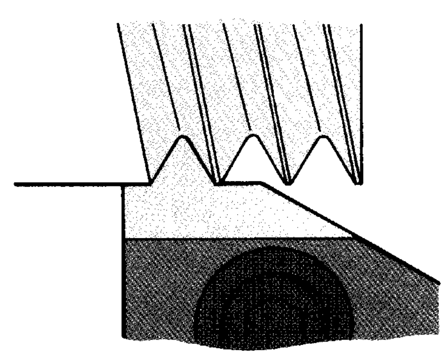


Figure 14-38. Using a full-profile insert to cut a thread. A separate insert is required for each thread pitch.

spindle and a double-V anvil to engage the thread. See **Figure 14-39**.

The micrometer reading given is the true pitch diameter. It equals the outside diameter of the screw minus the depth of one thread. Each micrometer is designed to read a limited number of thread pitches and is available in both inch and millimeter graduations.

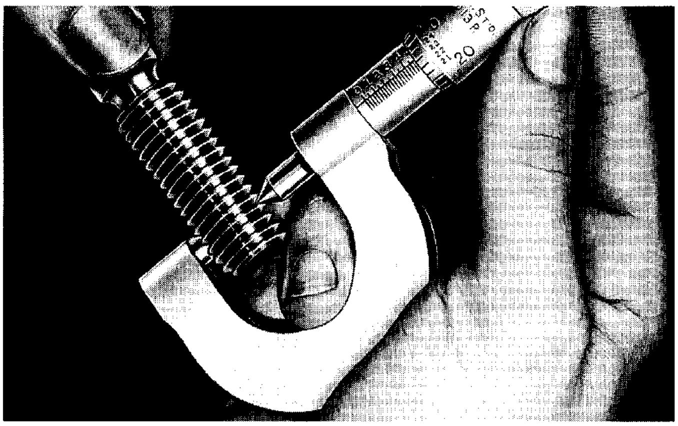


Figure 14-39. A thread micrometer can be used to check cut threads precisely. (L.S. Starrett Co.)

The *three-wire method of measuring threads* has proven to be quite satisfactory. As shown in **Figure 14-40A,** three wires of a specific diameter are fitted into the threads and a micrometer measurement is made over the wires. The formula in **Figure 14-40B** will provide the information necessary to calculate the correct measurement over the wires.

A three-wire thread measuring system has been developed to simplify and speed up the measuring process. It consists of a digital micrometer mounted in a special fixture that holds the threaded workpiece and the three wires. See **Figure 14-41.**

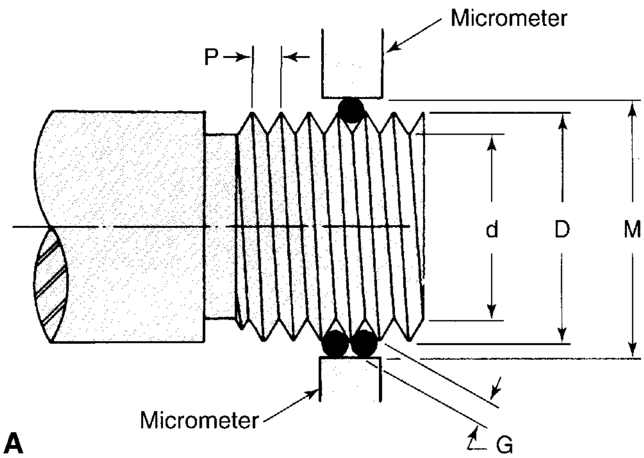
14.6.7 Cutting left-hand threads

Left-hand threads are cut in basically the same manner as right-hand threads. The major differences involve pivoting the compound to the *left* and changing the lead screw rotation so the carriage travels toward the tailstock (left to right), **Figure 14-42**.

14.6.8 Cutting square threads

Square threads are employed to transmit motion. They are more difficult to cut than 60° threads.

To cut a square thread, first calculate the width of the required tool bit (0.5 × thread pitch). If the



A

$$M = D + 3G - \frac{1.5155}{N}$$

Where: M = Measurement over the wires
D = Major diameter of thread
d = Minor diameter of thread
G = Diameter of wires

$P = \text{Pitch} = \frac{1}{N}$

N = Number of threads per inch

The smallest wire size that may be used for a given thread:

$$G = \frac{0.560}{N}$$

The largest wire size that can be used for a given thread:

$$G = \frac{0.900}{N}$$

The three-wire formula will work only if "G" is no larger or smaller than the sizes determined above. Any wire diameter between the two extremes may be used. All wires must be the same diameter.

B

Figure 14-40. Three-wire method of measuring 60° screw threads. A—Arrangement of the workpiece, wires, and micrometer. B—The three-wire thread measuring formula.

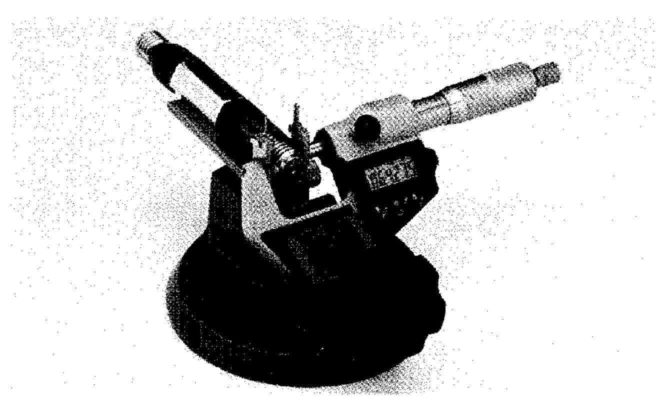


Figure 14-41. Thread measurements can be made in a of the time normally needed with this new three-wire system. Wires are mounted in individual holders tha clamping fixture. (Mitutoyo/MTI Corp.)

square thread is fairly coarse, a roughing tool is ground 0.010" to 0.015" (0.2 mm to 0.4 mm) smaller than the thread groove width. The cutting point of the finishing tool is ground 0.002" to 0.003" (0.05 mm to 0.08 mm) wider than the calculated groove width. Be sure adequate clearance is ground on the cutting tool, **Figure 14-43.**

14.6.9 Cutting Acme threads

On the *Acme thread,* the top and bottom are flat, but the sides have a 29° included angle. It was originally developed to replace the square thread. Its advantages are the strength and ease with which it can be cut, compared to the square thread. The thread form is employed in machine tools for precise control of component movement.

The *Acme screw thread gage* is the standard for grinding and setting Acme thread cutting tools. The tool angle is ground to fit a V in the thread gage. The width of the flat section varies with the pitch of the thread. This width is obtained by grinding back the tool point until it fits into the notch appropriate for the thread being cut. See **Figure 14-44.**

In cutting the threads, the groove is usually roughed out with a square nosed tool to approximate depth, then finished with an Acme-shaped tool. The compound rest is set to 14° and the tool is positioned using the thread gage, **Figure 14-45.** Other than this, Acme threads are cut in the same manner as the Sharp V thread.

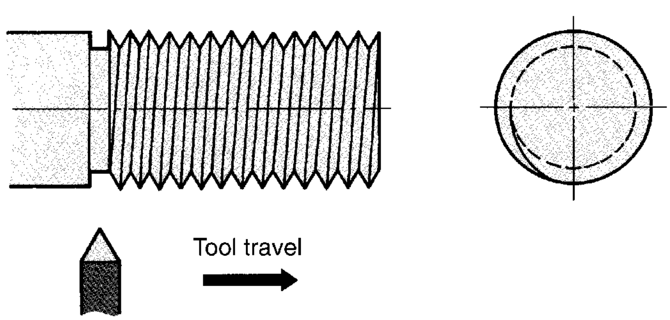


Figure 14-42. *Direction of tool travel for cutting left-hand threads.*

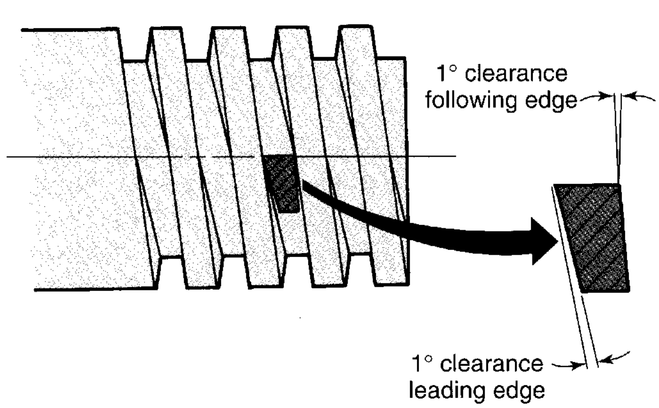


Figure 14-43. *Allow adequate side clearance when sharpening a tool to cut square threads.*

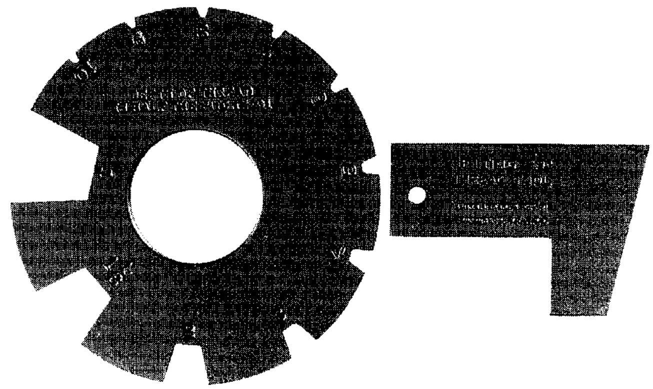


Figure 14-44. *The Acme screw thread gage and tool setup gage will allow you to check lathe settings.*

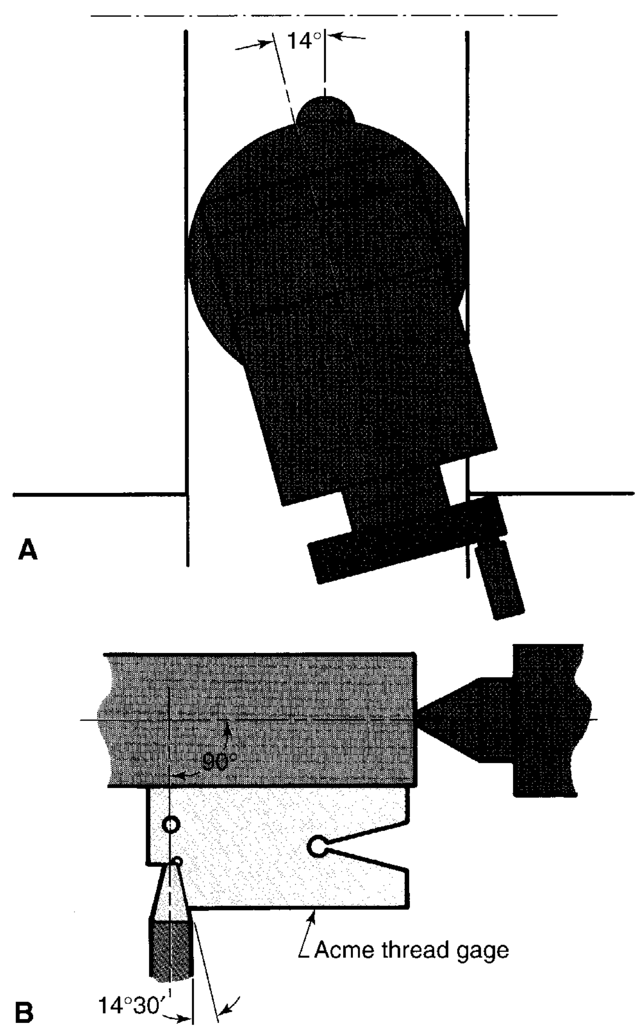


Figure 14-45. *Cutting Acme threads. A—Compound setting for cutting Acme threads. B—Cutting tool is positioned with an Acme thread gage.*

14.6.10 Cutting internal threads

Internal threads, **Figure 14-46,** are made on the lathe with a conventional boring bar and a cutting tool sharpened to the proper shape.

Before internal threads can be machined, the work must be prepared. A hole is drilled and bored to correct size for the thread's minor diameter. A recess is then machined with a square-nosed tool at the point where the thread terminates, **Figure 14-47**. The diameter of the recess is equal to the major diameter of the thread.

To cut right-hand internal threads, pivot the compound rest 29° to the *left,* as shown in **Figure 14-48**. Mount the tool on center and align it, using a center gage, **Figure 14-49.**

Bring the tool up until it just touches the work surface. Adjust the micrometer collar on the cross-slide to zero with the tool in position. Usi the compound rest screw, adjust the cutter to ma a cut of 0.002" (0.05 mm).

Remember that, when cutting *internal* threac tool infeed and removal from the cut are the rever of those used when cutting external threads.

A problem may arise in trying to determi when the tool has traveled far enough into the ho so the half-nuts can be disengaged. One metho makes use of a line that has been lightly scribed ir blued area on the flat way of the lathe bed. The to will have advanced far enough when the carria reaches this point.

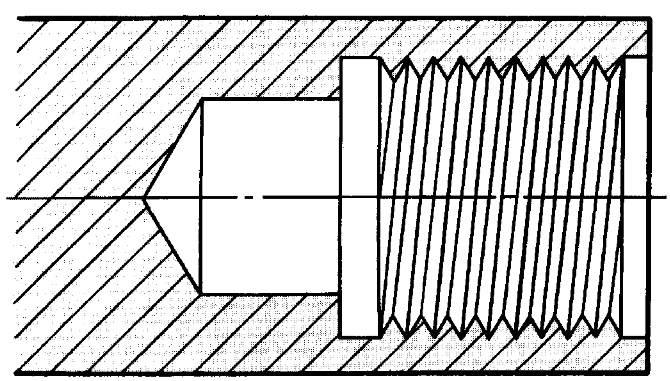


Figure 14-46. Internal threads.

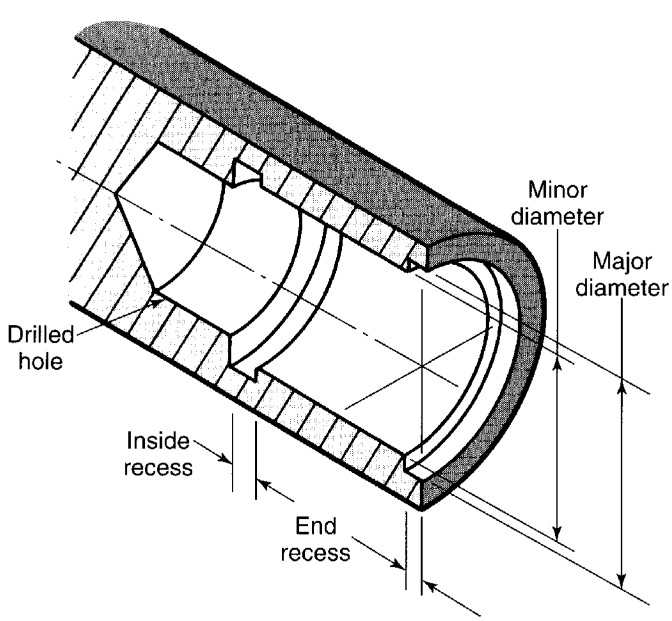


Figure 14-47. Opening for internal screw threads has been drilled and grooves machined to the major diameter.

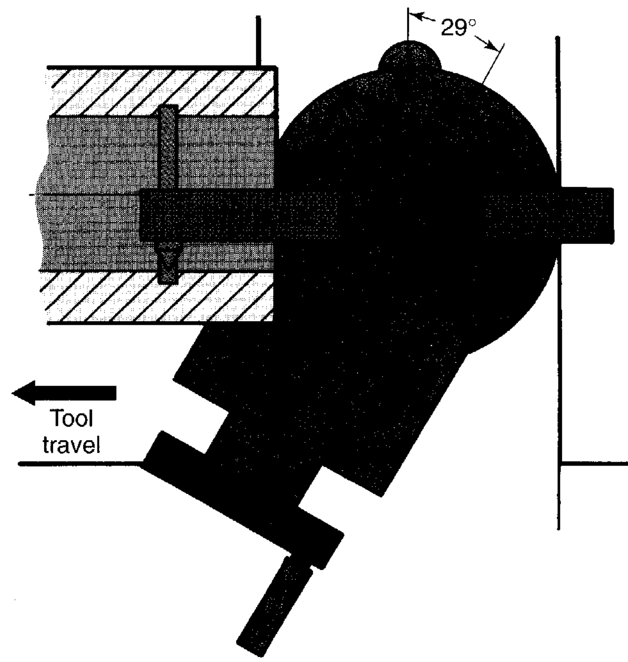


Figure 14-48. Compound setting for cutting internal right-ha screw threads.

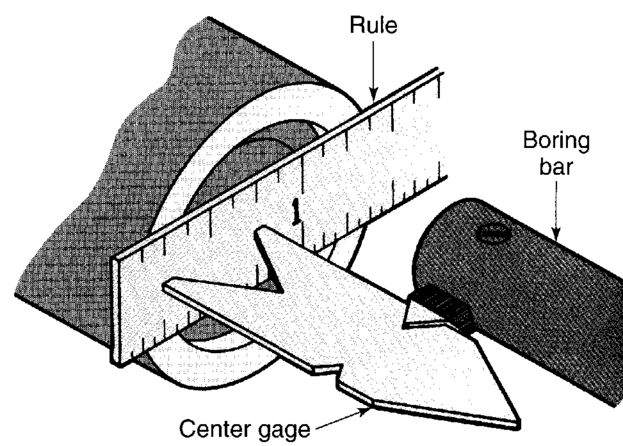


Figure 14-49. How to position cutting tool for machining inter screw threads.

Another technique allows you to start at the back of the hole when cutting internal threads. Pivot the compound rest 29° to the *right*. Place the threading tool to the rear of the boring bar with the cutting edge up. See **Figure 14-50**.

The lathe spindle is run in reverse. To prevent the tool from being placed too far into the hole to start the cut, mount a micrometer carriage stop on the ways. See **Figure 14-51**. The carriage is returned until it touches the stop. For cutting the threads, follow the same general procedure previously described.

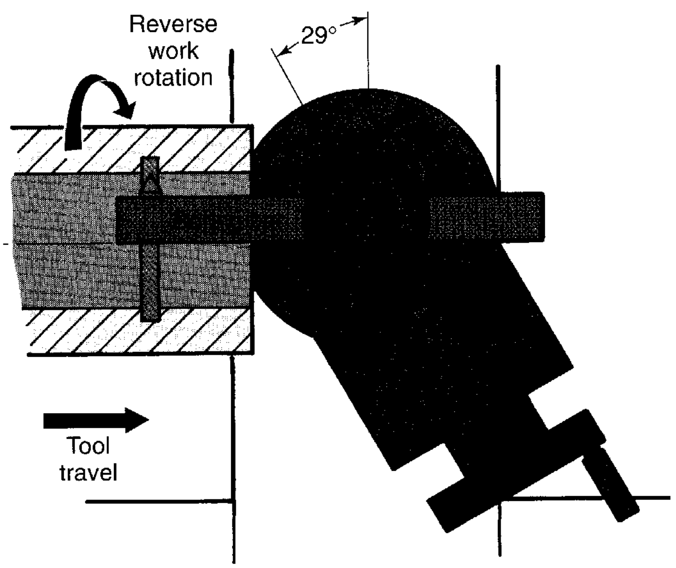


Figure 14-50*. An alternative setup for cutting internal right-hand threads. The work rotates in a direction opposite that of normal turning operations.*

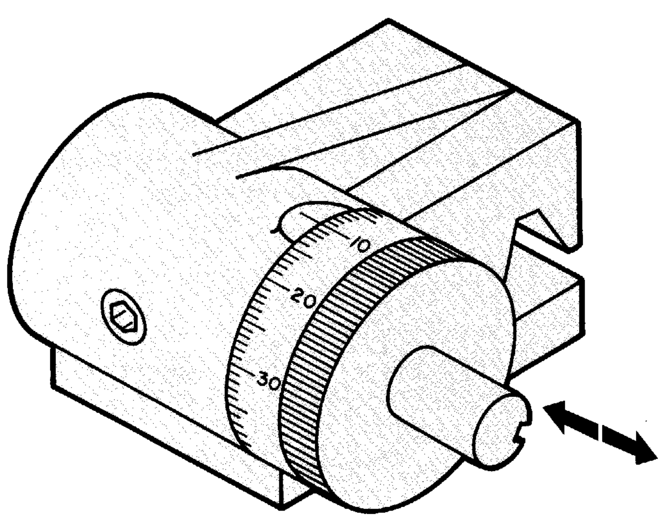


Figure 14-51*. When using alternate technique for cutting internal right-hand threads, mount a micrometer carriage stop on the ways. Adjust it to prevent the tool from being placed too far into hole when starting each cut.*

Continue making additional cuts until the threads are finished. Because the toolholder is not as rigid, lighter cuts must be taken when cutting internal threads than when machining external threads,. Keep the work flooded with cutting fluid.

14.6.11 Cutting threads on a taper surface

Tapered threads must be cut, at times, to obtain a fluid- or gas-tight joint. When this situation arises, the threading tool must be positioned in relation to the centerline of the taper rather than to the taper itself, **Figure 14-52**.

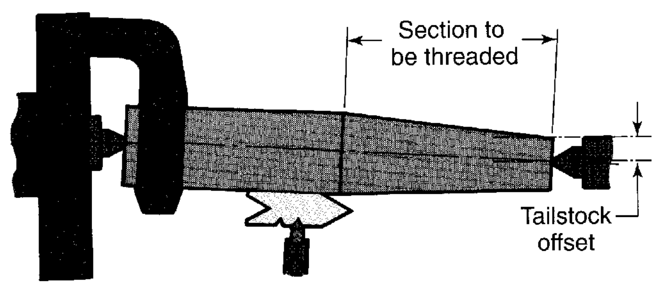


Figure 14-52*. Tool setup method for machining screw threads on a taper. Note that the tool is* not *positioned on the taper.*

TEST YOUR KNOWLEDGE

Please do not write in the text. Write your answers on a separate sheet of paper.

1. There are five ways of machining tapers on a lathe. List them, with their advantages and disadvantages.
2. When is a section of material considered tapered?
3. Machine adjustments must be calculated for each tapering job. The information given below will enable you to calculate the necessary tailstock setover for the problems given.

   Formula: When taper per inch is known,

   $$\text{Offset} = \frac{L \times TPI}{2}$$

   When taper per foot is known,

   $$\text{Offset} = \frac{L \times TPF}{24}$$

   When dimensions of tapered section are known but TPI or TPF is not given,

   $$\text{Offset} = \frac{L \times (D-d)}{2 \times \ell}$$

Where:

TPI = Taper per inch
TPF = Taper per foot
D = Diameter at large end of taper
d = Diameter at small end of taper
ℓ = Length of taper
L = Total length of piece

Note: These formulas, except for the TPF formula, can be used when dimensions are in mm.

Problem A: What will the tailstock setover be for the following job?

Taper per inch = 0.125″
Total length of piece = 4.000″

Problem B: What will the tailstock setover be for the following job?

D = 2.50″
d = 1.75″
ℓ = 6.00″
L = 9.00″

Problem C: What will the tailstock setover be for the following job?

D = 45.0 mm
d = 25.0 mm
ℓ = 175.0 mm
L = 275.0 mm

4. Screw threads are used for many reasons. List five or more important uses.

The following questions are of the matching type. Place the letter of the correct explanation on your paper.

5. _____ External thread.
6. _____ Internal thread.
7. _____ Major diameter.
8. _____ Minor diameter.
9. _____ Pitch diameter.
10. _____ Pitch.
11. _____ Lead.
    a. Smallest diameter of thread.
    b. Largest diameter of thread.
    c. Distance from one point on a thread to a corresponding point on next thread.
    d. Cut on outside surface of piece.
    e. Diameter of imaginary cylinder that would pass through threads at such points as to make width of thread and width of space at these points equal.
    f. Cut on inside surface of piece.
    g. Distance a nut will travel in one complete revolution of screw.

12. A groove is cut at the point where a thread is to terminate. It is cut to the depth of the thread and serves to:
    a. Provide a place to stop the threading tool after it makes a cut.
    b. Permits a nut to be run up to the end of the thread.
    c. Terminate the thread.
    d. All of the above.
    e. None of the above.
13. The tip of a cutting tool to cut a Sharp V thread is sharpened using a _____ to check that it is the correct shape. This tool is frequently called a _____.
14. The _____ is fitted to many lathe carriages. It meshes with the lead screw and is used to indicate when to engage the half-nuts to permit the thread cutting tool to follow exactly in the original cut.
15. The compound rest is set at _____ when cutting threads to permit the cutting tool to shear the material better than if it were fed straight into the work.
16. The three-wire thread measuring formula for inch-based threads is:

$$M = D + 3G - \frac{1.5155}{N}$$

Where: G = Wire diameter
D = Major diameter of thread (Convert to decimal size)
M = Measurement over the wires
N = Number of threads per inch

Problem: Calculate the correct measurement over the wires for the following threads. Use the wire size given in the problem.

a. 1/2-20 UNF (wire size 0.032″)
b. 1/4-20 UNC (wire size 0.032″)
c. 3/8-16 UNC (wire size 0.045″)
d. 7/16-14 UNC (wire size 0.060″)