METHODS OF MOUNTING WORK

Mounting Workpieces in Chucks

When installing the chuck or any attachment that screws onto the lathe headstock spindle, the threads and bearing surfaces of both spindle and chuck must be cleaned and oiled. In cleaning the internal threads of the chuck, a spring thread cleaner is very useful (Figure 7-34).

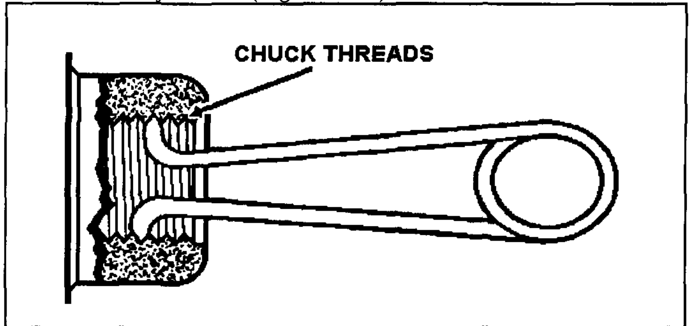


Figure 7-34. Spring thread cleaner.

Turn the spindle so that the key is facing up and lock the spindle in position. Make sure that the spindle and chuck taper are free of grit and chips. Place the chuck in position on the spindle. Engage the draw nut thread and tighten by applying four or five hammer blows on the spanner wrench engaged with the draw nut. Rotate the spindle 180°, engage the spanner wrench, and give four or five solid hammer blows to the spanner wrench handle. The workpiece is now ready for mounting.

Work automatically centers itself in the universal (3 jaw) scroll chuck, drill chuck, collet chucks, and step chuck, but must be manually centered in the independent (4 jaw) chuck. To center work in the independent chuck, line the four jaws up to the concentric rings on the face of the chuck, as close to the required diameter as possible.

Mount the workpiece and tighten the jaws loosely onto the workpiece (Figure 7-35). Spin the workpiece by hand and make approximate centering adjustments as needed, then firmly tighten the jaws.

For rough centering irregularly shaped work, first measure the outside diameter of the workpiece, then open the four jaws of the chuck until the workpiece slides in. Next tighten each opposing jaw a little at a time until the workpiece is held firmly, but not too tightly. Hold a piece of chalk near the workpiece and revolve the chuck slowly with your left hand. Where the chalk touches is considered the high side. Loosen the jaw opposite and tighten the jaw where the chalk marks are found. Repeat the process until the workpiece is satisfactorily aligned.

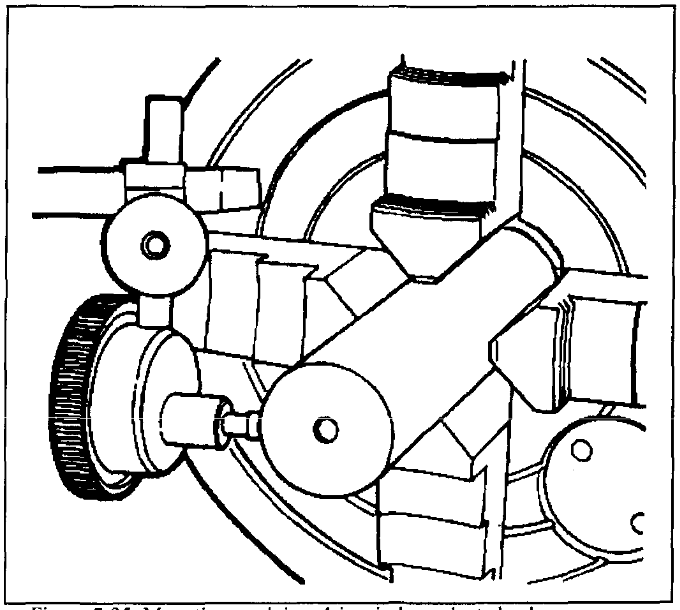


Figure 7-35. Mounting work in a 4-jaw independent chuck.

To center a workpiece having a smooth surface such as round stock, the best method is to use a dial test indicator. Place the point of the indicator against the outside or inside diameter of the workpiece. Revolve the workpiece slowly by hand and notice any deviations on the dial. This method will indicate any inaccuracy of the centering in thousandths of an inch.

If an irregularly shaped workpiece is to be mounted in the independent chuck, then a straight, hardened steel bar can be used with a dial indicator to align the workpiece. Experienced machinists fabricate several sizes of hardened steel bars, ground with a 60° point, that can be mounted into the drill chuck of the tailstock spindle and guided into the center-punched mark on the workpiece. A dial indicator can then be used to finish aligning the workpiece to within 0.001 inch. If a hardened steel bar is not readily available, a hardened center mounted in the tailstock spindle may be used to align the work while using a dial indicator on the chuck jaws. This method is one of several ways to align a workpiece in an independent chuck. Ingenuity and experience will increase the awareness of the machine operator to find the best method to set up the work for machining.

When removing chucks from the lathe, always use a wooden chuck block under the chuck to support the chuck on the lathe ways. Use care to avoid dropping the chuck on the ways, since this can greatly damage the lathe ways or crush the operator's hands.

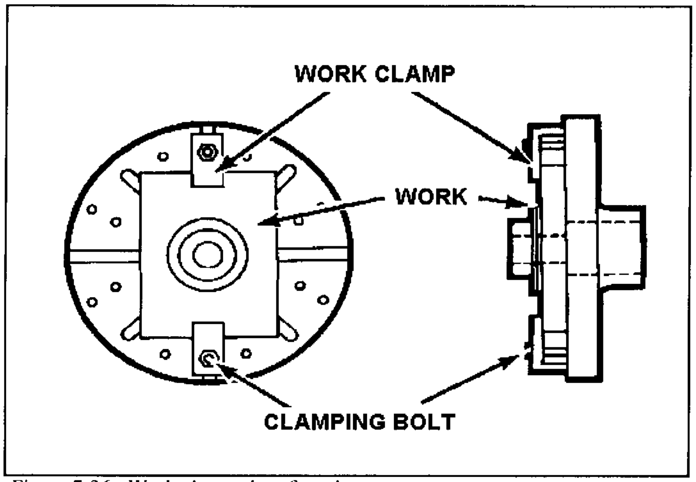


Figure 7-36. Work clamped on faceplate.

Mounting Work to Faceplates

Mount faceplates in the same manner as chucks. Check the accuracy of the faceplate surface using a dial indicator, and true the-faceplate surface by taking a light cut if necessary. Do not use faceplates on different lathes, since this will cause excessive wear of the faceplate due to repeated truing cuts having to be taken. Mount the workpiece using T-bolts and clamps of the correct sizes (Figure 7-36). Ensure all surfaces are wiped clean of burrs, chips, and dirt. When a heavy piece of work is mounted off center, such as when using an angle plate, use a counterweight to offset the throw of the work and to minimize vibration and chatter. Use paper or brass shims between the work and the faceplate to protect the delicate surface of the faceplate. After mounting the work to an approximate center location, use a dial indicator to finish accurate alignment.

Mounting Work Between Centers

Before mounting a work- piece between centers, the workpiece ends must be center- drilled and countersunk. This can be done using a small twist drill followed by a 60° center countersink or, more commonly, using a countersink and drill (also commonly called a center drill). It is very important that the center holes are drilled and countersunk so that they will fit the lathe centers exactly. Incorrectly drilled holes will subject the lathe centers to unnecessary wear and the workpiece will not run true because of poor bearing surfaces. A correctly drilled and countersunk hole has a uniform 60° taper and has clearance at the bottom for the point of the lathe center. Figure 7-37 illustrates correctly and incorrectly drilled center holes. The holes should have a polished appearance so as not to score the lathe centers. The actual drilling and countersinking of center holes can be done on a drilling machine or on the lathe itself. Before attempting to center drill using the lathe, the end of the workpiece must be machined flat to keep the center drill from running off center.

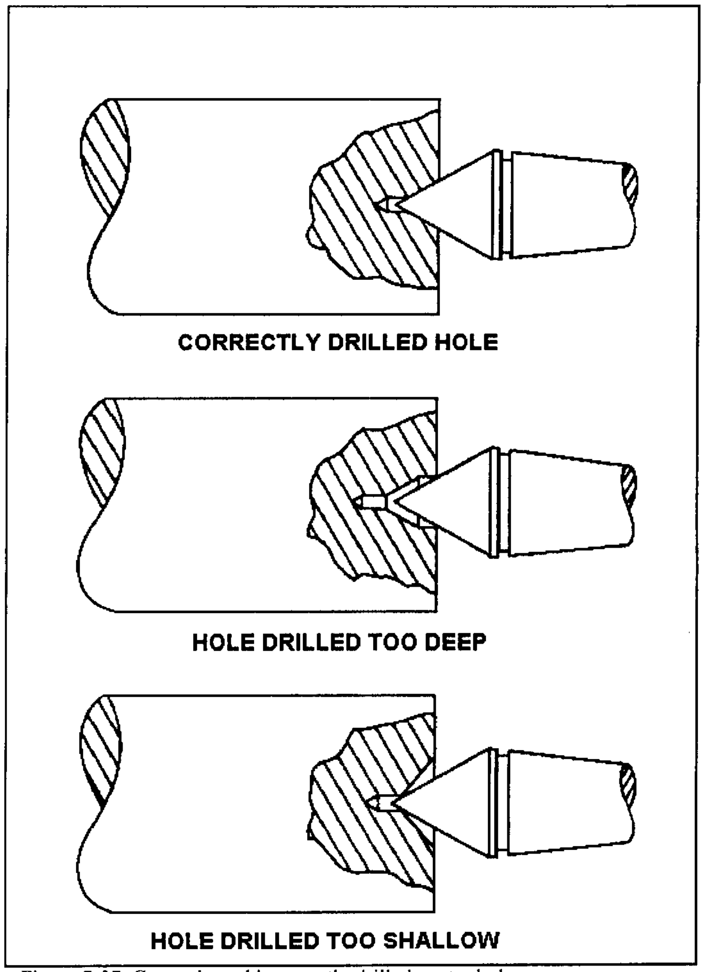


Figure 7-37. Correctly and incorrectly drilled center hole.

Mount the work in a universal or independent chuck and mount the center drill in the lathe tailstock (Figure 7-38). Refer to the section of this chapter on facing and drilling on the lathe, prior to doing this operation. Center drills come in various sizes for different diameters of work (Figure 7-39). Calculate the correct speed and hand feed into the workpiece. Only drill into the workpiece about 2/3 of the body diameter.

high speeds and feed them into the work slowly to avoid breaking off the drill point inside the work. If this happens, the work must be removed from the chuck and the point extracted. This is a time-consuming job and could ruin the workpiece.

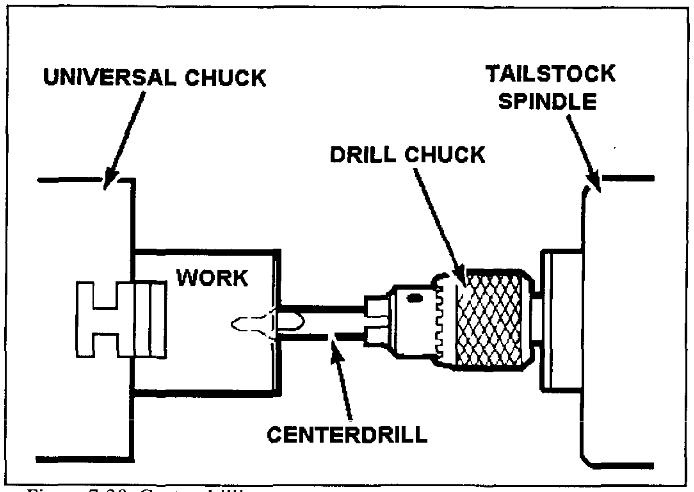


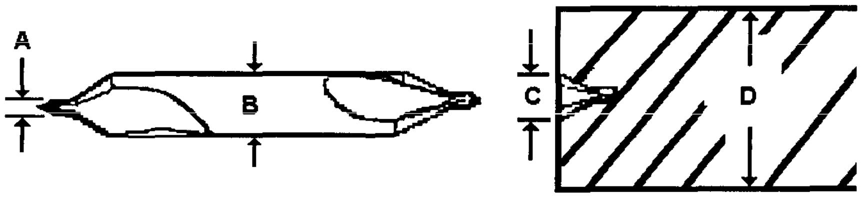
Figure 7-38. Center drilling.

To mount work between centers, the operator must know how to insert and remove lathe centers. The quality of workmanship depends as much on the condition of the lathe centers as on the proper drilling of the center holes. Before mounting lathe centers in the headstock or tailstock, thoroughly clean the centers, the center sleeve, and the tapered sockets in the headstock and tailstock spindles. Any dirt or chips on the centers or in their sockets will prevent the centers from seating properly and will cause the centers to run out of true.

Install the lathe center in the tailstock spindle with a light twisting motion to ensure a clean fit. Install the center sleeve into the headstock spindle and install the lathe center into the center sleeve with a light twisting motion.

To remove the center from the headstock spindle, hold the pointed end with a cloth or rag in one hand and give the center a sharp tap with a rod or knockout bar inserted through the hollow headstock spindle.

To remove the center from the tailstock, turn the tailstock handwheel to draw the tailstock spindle into the tailstock. The center will contact the tailstock screw and will be bumped loose from its socket.



SIZE	A	B	C	D
1	3/64”	.125	3\32	3/16”TO 5/16”
2	5/64”	.1875	9/64	3/8” TO 1”
3	7/64”	.250	3/16	1 1/4” TO 2”
4	1/8”	.3125	15/64	2 1/4” TO 4”
5	3/16”	.4375	21/64	4” TO 6”

Figure 7-39. Common sizes for combination countersink and centerdrill.

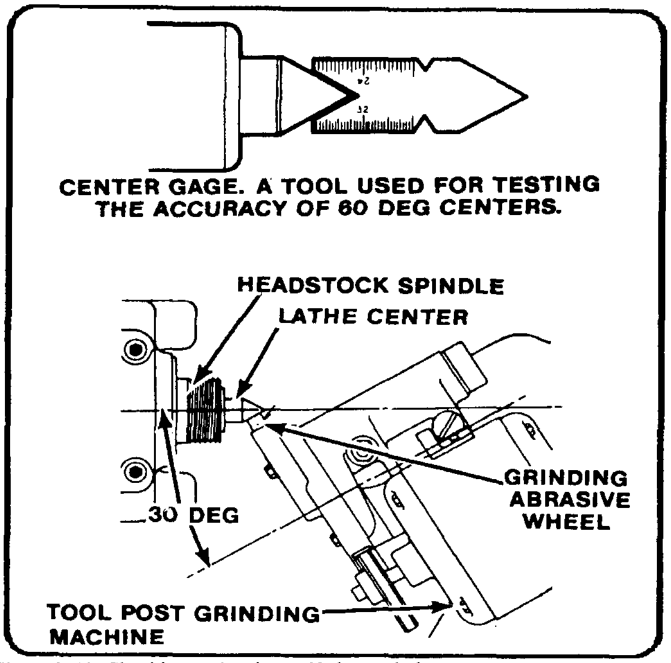


Figure 7-40. Checking and truing a 60 degree lathe center.

After mounting the headstock and tailstock centers, the accuracy of the 60° point should be checked using a center gage or a dial indicator. If the center in the headstock is not at 60°, or is scarred and burred, it must be trued while inserted in the lathe headstock spindle. If the headstock center is a soft center (a center that is not heat-treated and hardened), it can be turned true with the lathe tool bit. If the center in the headstock is hardened, it must be ground with a tool post grinding machine to get a true surface (Figure 7-40).

To turn a soft center true with the lathe, first set up the tool bit for right hand turning, center the tool bit; then, rotate the compound rest to an angle of 30° to the axis of the lathe (Figure 7-41). The lathe speed should be set for a finish cut, and the feed is supplied by cranking the handwheel of the compound rest, thus producing a clean and short steep taper with an included angle of 60°. Once trued, the center should stay in place until the operation is completed. If the center must be removed, mark the position on the center and headstock for easy realignment later.

Lathe centers must be parallel with the ways of the lathe in order to turn workplaces straight and true. Before beginning each turning operation, the center alignment should be checked.

The tailstock may be moved laterally to accomplish this alignment by means of adjusting screws after it has been released from the ways. Two zero lines are located at the rear of the tailstock and the centers are approximately aligned when these lines coincide (Figure 7-42). This alignment may be checked by moving the tailstock up close to the headstock so that the centers almost touch, and observing their relative positions (Figure 7-42).

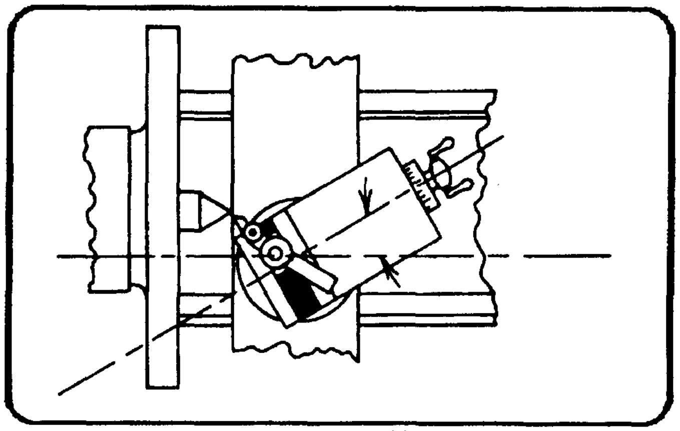


Figure 7-41. Turning of soft center true with the compound rest.

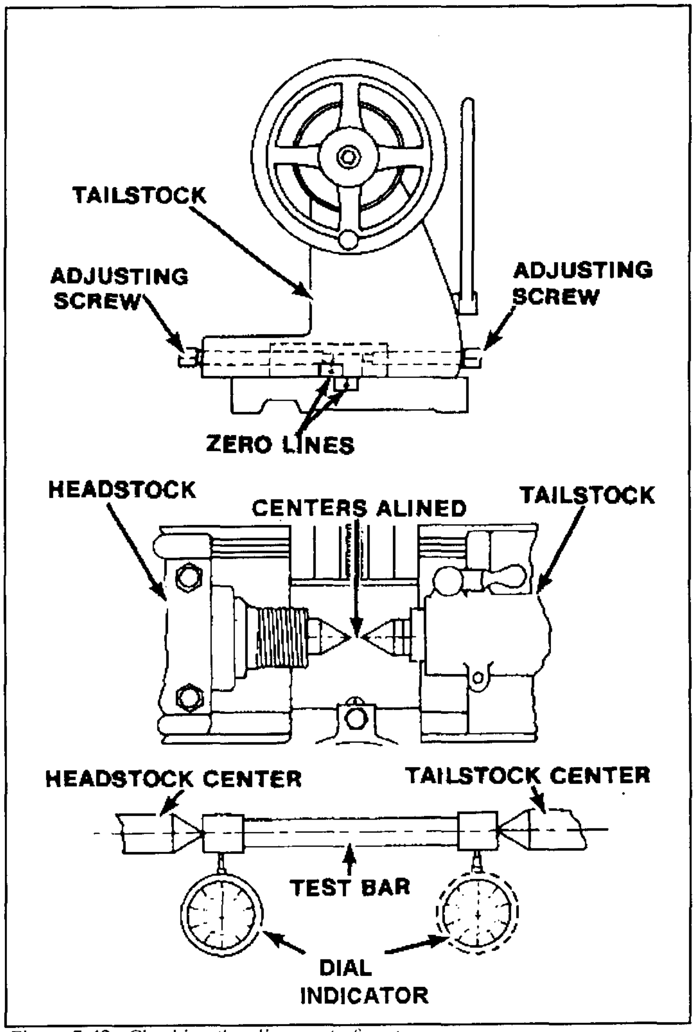


Figure 7-42. Checking the alignment of centers.

The most accurate method of checking alignment of centers is by mounting the workpiece between centers and taking light cuts at both ends without changing the carriage adjustments. Measure each end of this cut with calipers or a micrometer. If the tailstock end is greater in diameter than the headstock end, the tailstock is moved toward the operator. If the tailstock end is smaller in diameter than the headstock end, the tailstock is moved away from the operator. Take additional cuts in the same manner after each adjustment until both cuts measure the same.

To setup the workpiece between centers on the lathe, a driving faceplate (drive plate) and lathe dog must be used. (Figure 7-43). Make sure that the external threads of the headstock spindle are clean before screwing on the driving faceplate. Screw the faceplate securely onto the spindle. Clamp the lathe dog on the workpiece so that its tail hangs over the end of the workpiece. If the workpiece is finished, place a shim of soft material such as brass between the setscrew of the dog and workpiece. Mount the workpiece between the centers. Make sure that the lathe dog tail tits freely in the slot of the faceplate and does not bind. Sometimes, the tailstock center is a dead center and does not revolve with the workpiece, so it may require lubrication. A few drops of oil mixed with white lead should be applied to the center before the workpiece is set up. The tailstock should

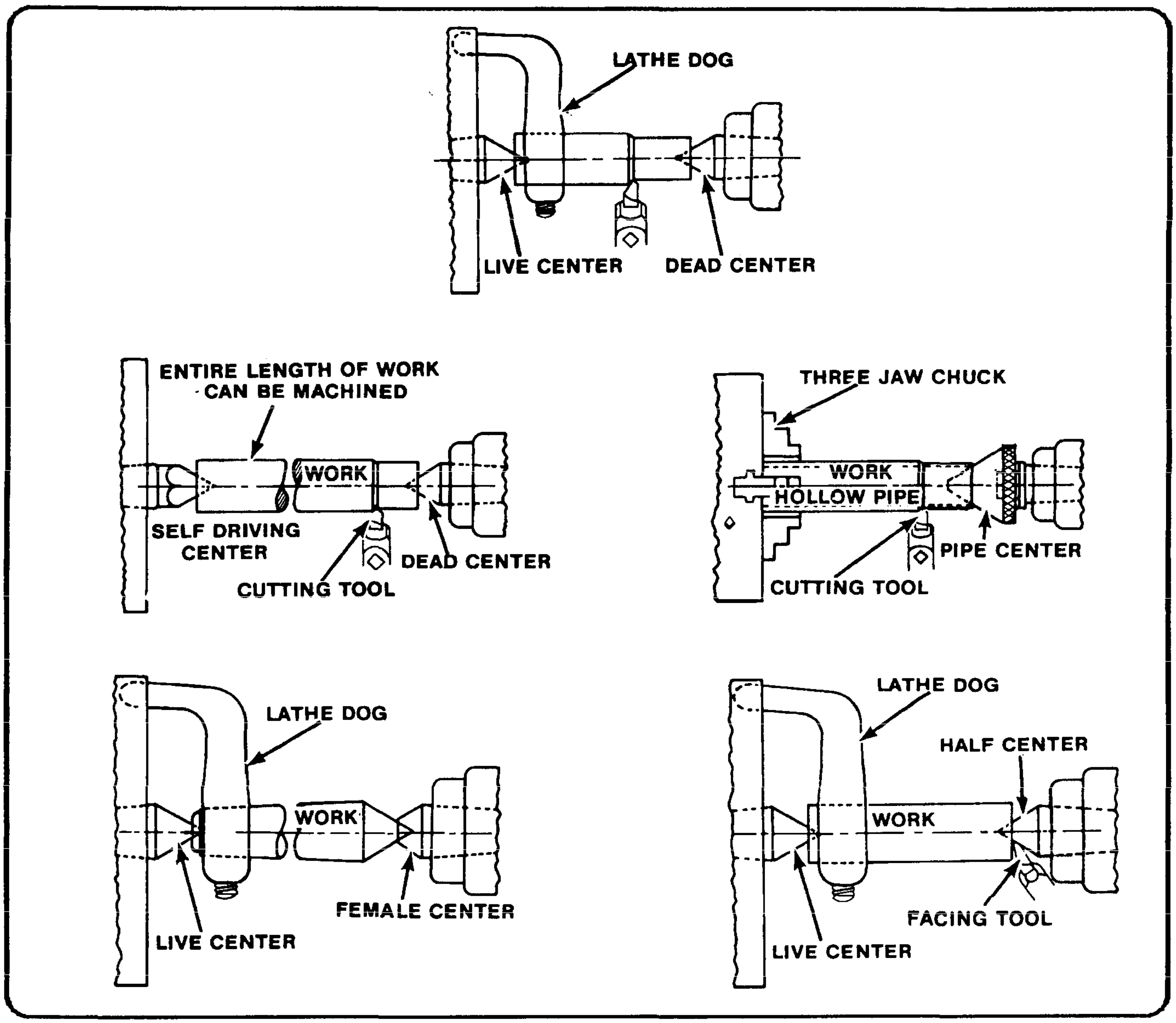


Figure 7-43. Holding work between centers.

be adjusted so that the tailstock center fits firmly into the center hole of the workpiece but does not bind. The lathe should be stopped at intervals and additional oil and white lead mixture applied to the dead center to prevent overheating harm to the center and the workpiece.

Mounting Work on Mandrels

To machine a workpiece of an odd shape, such as a wheel pulley, a tapered mandrel is used to hold and turn the work. The mandrel must be mounted between centers and a drive plate and lathe dog must be used. The centers must be aligned and the mandrel must be free of burrs. Mount the workpiece onto a lubricated mandrel of the proper size by using an arbor press. Ensure that the lathe dog is secured to the machined flat on the end of the mandrel and not on the smooth surface of the mandrel taper (Figure 7-44). If expansion bushings are to be used with a mandrel, clean and care for the expansion bushings in the same manner as a normal mandrel.

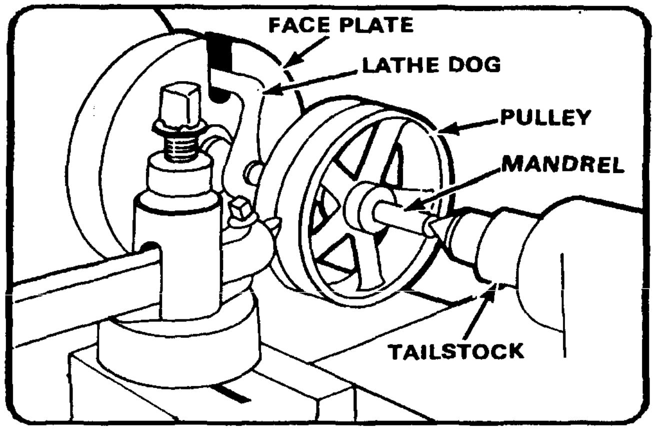


Figure 7-44. Pulley mounted on a mandrel.

Always feed the tool bit in the direction of the large end of the mandrel, which is usually toward the headstock end, to avoid pulling the work out of the mandrel. If facing on a mandrel, avoid cutting into the mandrel with the tool bit..

GENERAL LATHE OPERATIONS

LATHE SPEEDS, FEEDS, AND DEPTH OF CUTS

General operations on the lathe include straight and shoulder turning, facing, grooving, parting, turning tapers, and cutting various screw threads. Before these operations can be done, a thorough knowledge of the variable factors of lathe speeds, feeds, and depth of cut must be understood. These factors differ for each lathe operation, and failure to use these factors properly will result in machine failure or work damage. The kind of material being worked, the type of tool bit, the diameter and length of the workpiece, the type of cut desired (roughing or finishing), and the working condition of the lathe will determine which speed, feed, or depth of cut is best for any particular operation. The guidelines which follow for selecting speed, feed, and depth of cut are general in nature and may need to be changed as conditions dictate.

Cutting Speeds.

The cutting speed of a tool bit is defined as the number of feet of workpiece surface, measured at the circumference, that passes the tool bit in one minute. The cutting speed, expressed in FPM, must not be confused with the spindle speed of the lathe which is expressed in RPM. To obtain uniform cutting speed, the lathe spindle must be revolved faster for workplaces of small diameter and slower for workplaces of large diameter. The proper cutting speed for a given job depends upon the hardness of the material being machined, the material of the tool bit, and how much feed and depth of cut is required. Cutting speeds for metal are usually expressed in surface feet per minute, measured on the circumference of the work. Spindle revolutions per minute (RPM) are determined by using the formula:

$$\frac{12 \times SFM}{3.1416 \times D} = RPM$$

Which is simplified to:

$$\frac{4 \times SFM}{D} = RPM$$

Where **SFM** is the rated surface feet per minute, also expressed as cutting speed.

RPM is the spindle speed in revolutions per minute

D is the diameter of the work in inches.

in order to use the formula simply insert the cutting speed of the metal and the diameter of the workpiece into the formula and you will have the RPM.

Turning a one-half inch piece of aluminum. cutting speed of 200 SFM. would result in the following:

$$\frac{4 \times 200}{1/2} = 1600 \text{ RPM}$$

Table 7-2 in Appendix A lists specific ranges of cutting speeds for turning and threading various materials under normal lathe conditions, using normal feeds and depth of cuts. Note that in Table 7-2 the measurement calculations are in inch and metric measures. The diameter measurements used in these calculations are the actual working diameters that are being machined. and not necessarily the largest diameter of the material. The cutting speeds have a wide range so that the lower end of the cutting speed range can be used for rough cutting and the higher end for finish cutting. If no cutting speed tables are available, remember that, generally. hard materials require a slower cutting speed than soft or ductile materials. Materials that are machined dry. without coolant. require a slower cutting speed than operations using coolant. Lathes that are worn and in poor condition will require slower speeds than machines that are in good shape. If carbide-tipped tool bits are being used, speeds can be increased two to three times the speed used for high-speed tool bits.

Feed

Feed is the term applied to the distance the tool bit advances along the work for each revolution of the lathe spindle. Feed is measured in inches or millimeters per revolution, depending on the lathe used and the operator's system of measurement. Table 7-3 in Appendix A is a guide that can be used to select feed for general roughing and finishing operations. A light feed must be used on slender and small workplaces to avoid damage. If an irregular finish or chatter marks develop while turning. reduce the feed and check the tool bit for alignment and sharpness. Regardless of how the work is held in the lathe, the tool should feed toward the headstock. This results in most of the pressure of the cut being put on the work holding device. If the cut must be fed toward the tailstock. use light feeds and light cuts to avoid pulling the workpiece loose.

Depth of Cut

Depth of cut is the distance that the tool bit moves into the work. usually measured in thousandths of an inch or in millimeters. General machine practice is to use a depth of cut up to five times the rate of feed, such as rough cutting stainless steel using a feed of 0.020 inch per revolution and a depth of cut of 0.100 inch. which would reduce the diameter by 0.200 inch. If chatter marks or machine noise develops. reduce the depth of cut.

MICROMETER COLLAR

Graduated micrometer collars can be used to accurately measure this tool bit movement to and away from the lathe center axis. Thus. the depth of cut can be accurately measured when moving the tool bit on the cross slide by using the cross slide micrometer collar. The compound rest is also equipped with a micrometer collar. These collars can measure in inches or in millimeters, or they can be equipped with a dual readout collar that has both. Some collars measure the exact tool bit movement. while others are designed to measure the amount of material removed from the workpiece (twice the tool bit movement). Consult the operator's instruction manual for specific information on graduated collar use.

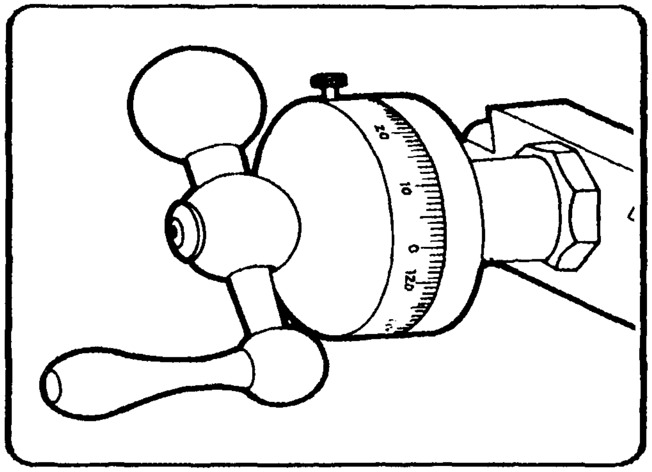


Figure 7-45. Graduated micrometer collar.

FACING

Facing is machining the ends and shoulders of a piece of stock smooth. flat, and perpendicular to the lathe axis. Facing is used to cut work to the desired length and to produce a surface from which accurate measurements may be taken.

Facing Work in a Chuck

Facing is usually performed with the work held in a chuck or collet. Allow the workpicce to extend a distance no more than 1 1/2 times the work diameter from the chuck jaws. and use finishing speeds and feeds calculated using the largest diameter of the workpiece. The tool bit may be fed from the outer edge to the center or from the center to the outer edge. Normal facing is done from the outer edge to the center since this method permits the operator to observe the tool bit and layout line while starting the cut. This method also eliminates the problem of feeding the tool bit into the solid center portion of the workpiece to get a cut started.. Use a left-hand finishing tool bit and a right-hand tool holder when facing from the outer edge toward the center. Work that has a drilled or bored hole in the center may be faced from the center out

to the outer edge if a right-hand finishing tool bit is used. Avoid excessive tool holder and tool bit overhang when setting up the facing operation. Set the tool bit exactly on center to avoid leaving a center nub on the workpiece (Figure 7-46). Use the tailstock center point as a reference point when setting the tool bit exactly on center. If no tailstock center is available, take a trial cut and readjust as needed. If using the cross slide power feed to move the tool bit (into the center), disengage power when the tool bit is within l/16 inch of the center and finish the facing cut using hand feed.

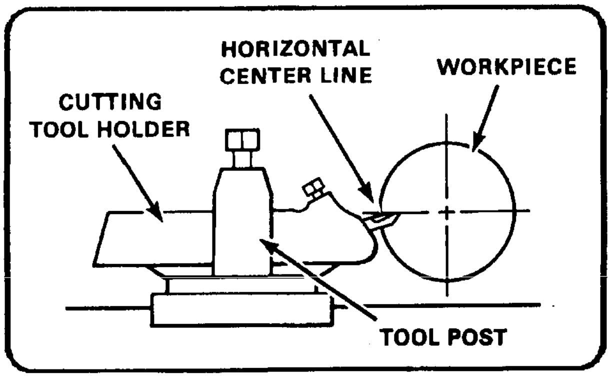


Figure 7-46. Positioning tool bit for facing.

Facing Work Between Centers

Sometimes the workpiece will not fit into a chuck or collet, so facing must be done between centers. To properly accomplish facing between centers, the workpiece must be center-drilled before mounting into the lathe. A half male center (with the tip well lubricated with a white lead and oil mixture) must be used in the lathe tailstock to provide adequate clearance for the tool bit. The tool bit must be ground with a sharp angle to permit facing to the very edge of the center drilled hole (Figure 7-47). Start the facing cut at the edge of the center-drilled hole after checking for tool bit clearance, and feed the cutting tool out to the edge. Use light cuts and finishing feeds, which will reduce the tension put on the half male center. Replace the half male center with a standard center after the facing operation, since the half male center will not provide adequate support for general turning operations. Only a small amount of material can be removed while facing between centers. If too much material is removed, the center-drilled hole will become too small to support the workpiece.

Precision Facing

Special methods must be used to face materials to a precise length. One method is to mount the work in a chuck and lightly face one end with a cleanup cut. Then, reverse the stock and face it to the scribed layout line. This method may not be as accurate as other methods, but it will work for most jobs. A more precise method to face a piece of stock to a specified length is to turn the compound rest to an angle of 30 degrees to the cross slide and then use the graduated micrometer collar to measure tool bit movement, Figure 7-48. At this angle of the compound rest, the movement of the cutting tool will always be half of the reading of the graduated collar. Thus, if the compound rest feed is turned 0.010 inch, the tool bit will face off 0.005 inch of material. With the compound rest angled at 30°, a light cut may be made on the first end, then the piece reversed and faced to accurate length. Always lock the carriage down to the bed. This provides the most secure and accurate base for the cutting tool and helps eliminate unwanted vibration during facing operations. Another way to face to a precise length is to use the lathe carriage micrometer stop to measure the carriage and tool bit movement. Using the micrometer stop can sometimes be faster and easier than using the compound rest graduated collar for measuring tool bit movement.

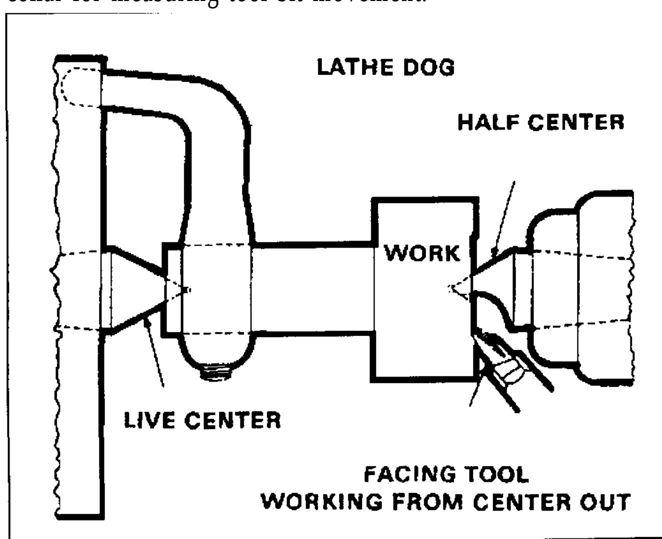


Figure 7-47. Facing using a side finishing tool and a half-male center.

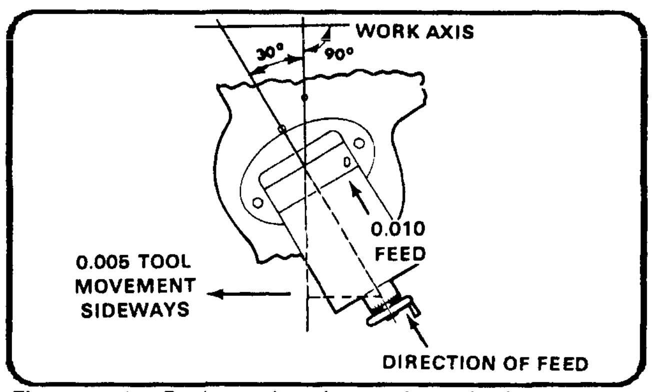


Figure 7-48. Facing using the graduated micrometer collar to measure tool bit movement.

STRAIGHT TURNING

Straight turning, sometimes called cylindrical turning, is the process of reducing the work diameter to a specific dimension as the carriage moves the tool along the work. The work is machined on a plane parallel to its axis so that there is no variation in the work diameter throughout the length of the cut. Straight turning usually consists of a roughing cut followed by a finishing cut. When a large amount of material is to be removed, several roughing cuts may need to be taken. The roughing cut should be as heavy as the machine and tool bit can withstand. The finishing cut should be light and made to cut to the specified dimension in just one pass of the tool bit. When using power feed to machine to a specific length, always disengage the feed approximately 1/16-inch away from the desired length dimension, and then finish the cut using hand feed.

Setting Depth of Cut

In straight turning, the cross feed or compound rest graduated collars are used to determine the depth of cut, which will remove a desired amount from the workpiece diameter. When using the graduated collars for measurement, make all readings when rotating the handles in the forward direction. The lost motion in the gears, called backlash, prevents taking accurate readings when the feed is reversed. If the feed screw must be reversed, such as to restart a cut, then the backlash must be taken up by turning the feed screw handle in the opposite direction until the movement of the screw actuates the movement of the cross slide or compound rest. Then turn the feed screw handle in the original or desired direction back to the required setting.

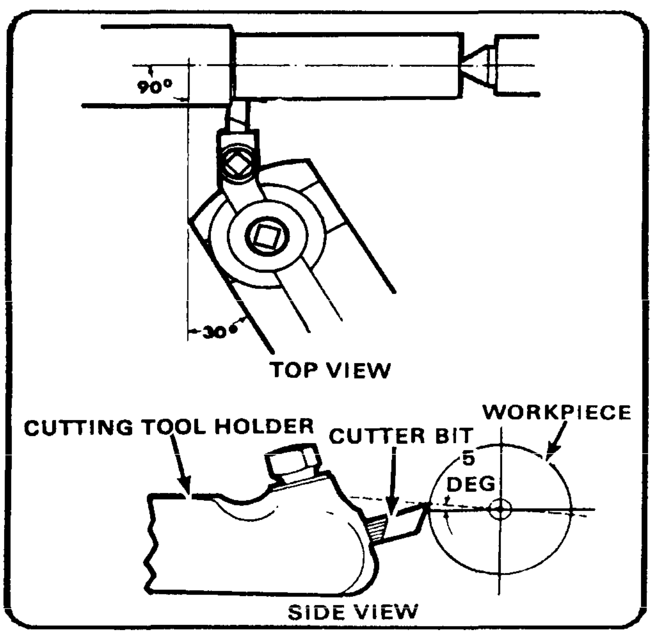


Figure 7-49. Set up for straight turning.

Setting Tool Bit for Straight Turning

See Figure 7-49. For most straight turning operations, the compound rest should be aligned at an angle perpendicular to the cross slide, and then swung 30° to the right and clamped in position. The tool post should be set on the left-hand side of the compound rest T-slot, with a minimum of tool bit and tool holder overhang.

When the compound rest and tool post are in these positions, the danger of running the cutting tool into the chuck or damaging the cross slide are minimized. Position the roughing tool bit about 5° above center height for the best cutting action. This is approximately 3/64-inch above center for each inch of the workpiece diameter. The finishing tool bit should be positioned at center height since there is less torque during finishing. The position of the tool bit to the work should be set so that if anything occurs during the cutting process to change the tool bit alignment, the tool bit will not dig into the work, but instead will move away from the work. Also, by setting the tool bit in this position, chatter will be reduced. Use a right-hand turning tool bit with a slight round radius on the nose for straight turning. Always feed the tool bit toward the headstock unless turning up to an inside shoulder. Different workplaces can be mounted in a chuck, in a collet, or between centers. Which work holding device to use will depend on the size of the work and the particular operation that needs to be performed.

Turning Work Between Centers

Turning work that is held between centers is one accurate method that is available. The chief advantage of using this method is that the work can be removed from the lathe and later replaced for subsequent machining operations without disturbing the trueness of the turned surface in relation to the center holes of the workpiece. The lathe centers must be in good condition and carefully aligned if the turning operation is to be accurate. If necessary, true the centers and realign as needed. After the workpiece is center-drilled, place a lathe dog (that is slightly larger in diameter than the workpiece) on the end of the work that will be toward the headstock, and tighten the lathe dog bolt securely to the workpiece). If using a dead center in the tailstock, lubricate the center with a mixture of white lead and motor oil. A ball bearing live center is best for the tailstock center since this center would not need lubrication and can properly support the work. Extend the tailstock spindle out about 3 inches and loosen the tailstock clamp-down nut. Place the work with the lathe dog end on the headstock live center and slide the tailstock forward until the tailstock center will support the work; then, secure the tailstock with the clamp-down nut. Adjust the tail

of the lathe dog in the drive plate slot, making sure that the tail does not bind into the slot and force the work out of the center. A good fit for the lathe dog is when there is clearance at the top and bottom of the drive plate slot on both sides of the lathe dog tail. Tension should be applied to hold the work in place, but not so much tension that the tail of the lathe dog will not move freely in the drive -plate slot.

Check tool bit clearance by moving the tool bit to the furthest position that can be cut without running into the lathe dog or the drive plate. Set the lathe carriage stop or micrometer carriage stop at this point to reference for the end of the cut and to protect the lathe components from damage. Set the speed, feed, and depth of cut for a roughing cut and then rough cut to within 0.020 inch of the final dimension. Perform a finish cut, flip the piece over, and change the lathe dog to the opposite end. Then rough and finish cut the second side to final dimensions.

Turning Work in Chucks

Some work can be machined more efficiently by using chucks, collets, mandrels, or faceplates to hold the work. Rough and finish turning using these devices is basically the same as for turning between centers. The workpiece should not extend too far from the work holding device without adequate support. If the work extends more than three times the diameter of the workpiece from the chuck or collet, additional support must be used such as a steady rest or a tailstock center support. When turning using a mandrel or faceplate to hold an odd-shaped workpiece, use light cuts and always feed the cutting tool toward the headstock. Every job may require a different setup and a different level of skill. Through experience, each machine operator will learn the best methods for holding work to be turned.

MACHINING SHOULDERS, CORNERS, UNDERCUTS, GROOVES, AND PARTING

Shoulders

Frequently, it will be necessary to machine work that has two or more diameters in its length. The abrupt step, or meeting place, of the two diameters is called a shoulder. The workpiece may be mounted in a chuck, collet, or mandrel, or between centers as in straight turning. Shoulders are turned, or formed, to various shapes to suit the requirements of a particular part. Shoulders are machined to add strength for parts that are to be fitted together, make a corner, or improve the appearance of a part. The three common shoulders are the square, the filleted, and the angular shoulder (Figure 7-50).

Square shoulders are used on work that is not subject to excessive strain at the corners. This shape provides a flat clamping surface and permits parts to be fitted squarely together. There are many different ways to accurately machine a square shoulder. One method is to use a parting tool bit to locate and cut to depth the position of the shoulder. Straight-tuming the diameter down to the desired size is then the same as normal straight turning. Another method to machine a square shoulder is to rough out the shoulder slightly oversize with a round-nosed tool bit, and then finish square the shoulders to size with a side-finishing tool bit. Both of these methods are fine for most work, but may be too time-consuming for precise jobs. Shoulders can be machined quickly and accurately by using one type of tool bit that is ground and angled to straight turn and face in one operation (Figure 7-51).

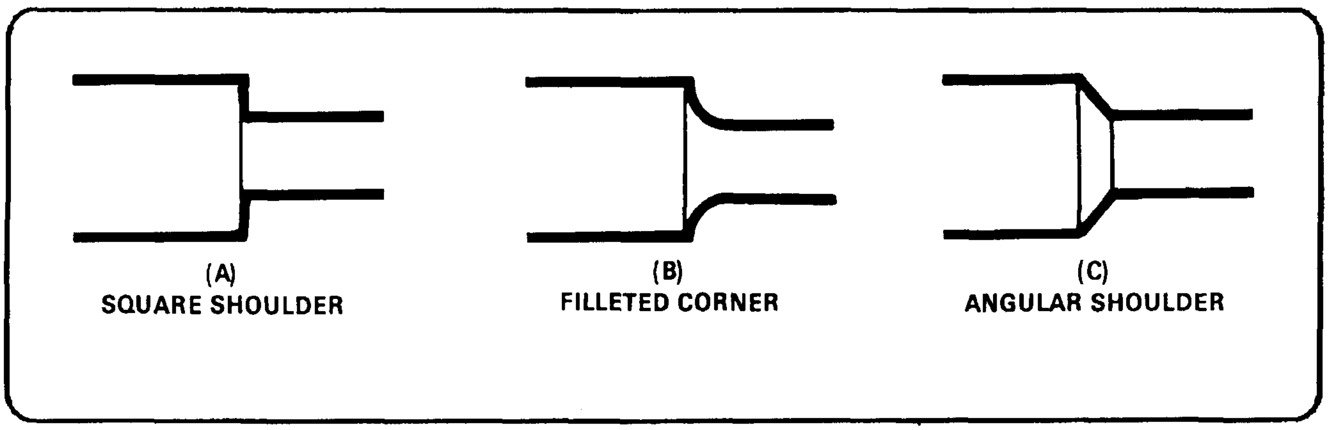


Figure 7-50. Common shoulders.

Set up the micrometer carriage stop to align the shoulder dimension; then, in one pass of the tool bit, feed the tool bit left to turn the smaller diameter until contact is made with the carriage stop. Change the direction to feed out from center and face the shoulder out to the edge of the workpiece. The lathe micrometer stop measures the length of the shoulder and provides for a stop or reference for the tool bit. Shoulder turning in this manner can be accomplished with a few roughing cuts and a finishing cut.

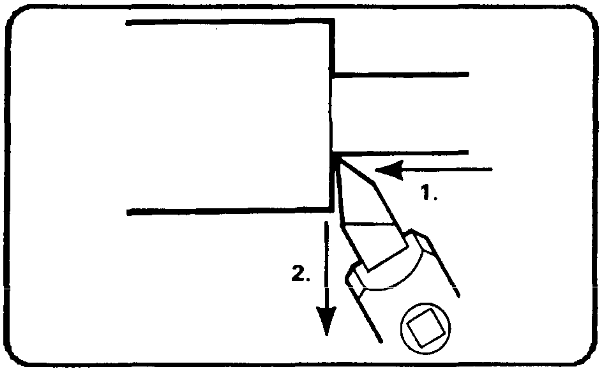
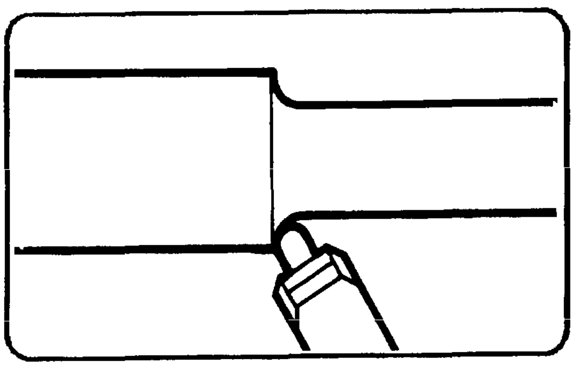


Figure 7-51. Straight and shoulder turning in one pass.



Figurfe 7-52. Cutting a filleted corner.

Filleted Shoulders

Filleted shoulders or comers, are rounded to be used on parts which require additional strength at the shoulder. These shoulders are machined with a round-nose tool bit or a specially formed tool bit (Figure 7-52). This type of shoulder can be turned and formed in the same manner as square shoulders. Filleted corners are commonly cut to double-sided shoulders (see Undercuts).

Angular Shoulders

Angular shoulders although not as common as filleted shoulders, are sometimes used to give additional strength to corners, to eliminate sharp corners, and to add to the appearance of the work. Angular shoulders do not have all the strength of filleted corners but are more economical to produce due to the simpler cutting tools. These shoulders are turned in the same manner as square shoulders by using a side turning tool set at the desired angle of the shoulder, or with a square-nosed tool set straight into the work (Figure 7-53).

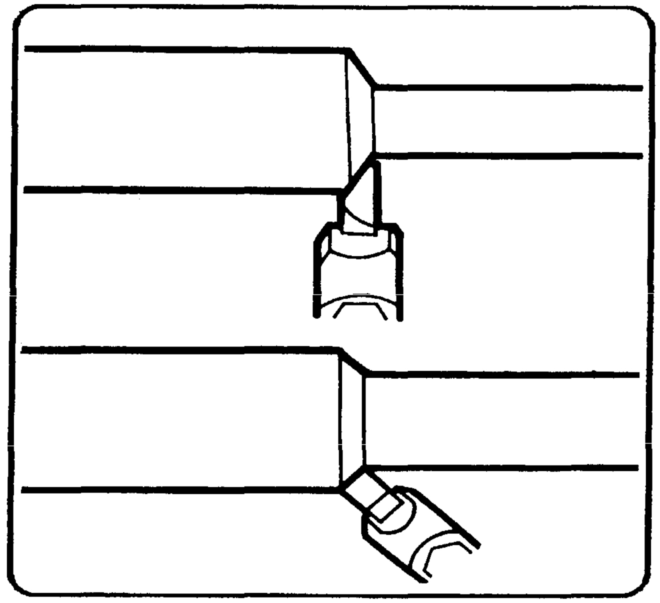


Figure 7-53. Cutting angular shoulders using two tool

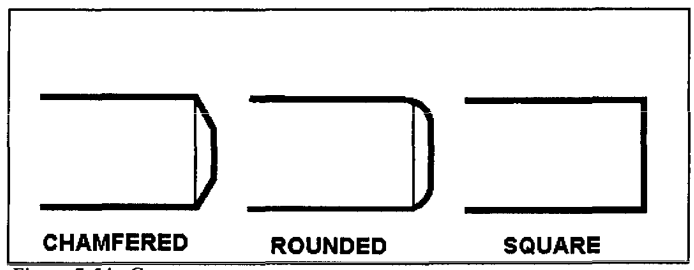


Figure 7-54. Corners.

Corners

Corners are turned on the edges of work to break down sharp edges and to add to the general appearance of the work. Common types of corners are chamfered, rounded, and square (Figure 7-54). Chamfered (or angular) corners may be turned with the side of a turning tool or the end of a square tool bit, as in angular shoulder turning. Round corners are produced by turning a small radius on the ends of the work. The radius may be formed by hand manipulation of the cross slide and carriage using a turning tool. An easier method is to use a tool bit specifically ground for the shape of the desired corner. Still another method is to file the radius with a standard file. A square corner is simply what is left when making a shoulder, and no machining is needed.

Undercuts

Undercuts are the reductions in diameter machined onto the center portion of workplaces (Figure 7-55) to lighten the piece or to reduce an area of the part for special reasons, such as holding an oil seal ring. Some tools, such as drills and reamers, require a reduction in diameter at the ends of the flutes to provide clearance or runout for a milling cutter or grinding wheel. Reducing the diameter of a shaft or workpiece at the center with filleted shoulders at each end may be accomplished by the use of a round-nosed turning tool bit. This tool bit may or may not have a side rake angle, depending on how much machining needs to be done. A tool bit without any side rake is best when machining in either direction. Undercutting is done by feeding the tool bit into the workpiece while moving the carriage back and forth slightly. This prevents gouging and chatter occurring on the work surface.

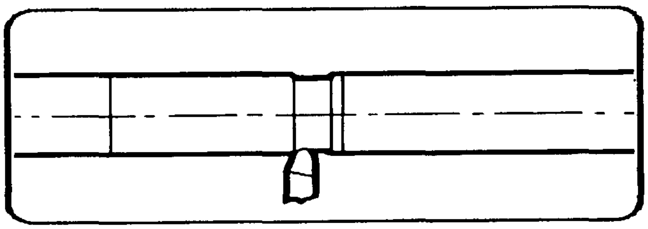


Figure 7-55. Machining an undercut.

Grooves

Grooving (or necking) is the process of turning a groove or furrow on a cylinder, shaft, or workpiece. The shape of the tool and the depth to which it is fed into the work govern the shape and size of the groove. The types of grooves most commonly used are square, round, and V-shaped (Figure 7-56). Square and round grooves are frequently cut on work to provide a space for tool runout during subsequent machining operations, such as threading or knurling. These grooves also provide a clearance for assembly of different parts. The V-shaped groove is used extensively on step pulleys made to fit a V-type belt. The grooving tool is a type of forming tool. It is ground without side or back rake angles and set to the work at center height with a minimum of overhang. The side and end relief angles are generally somewhat less than for turning tools.

In order to cut a round groove of a definite radius on a cylindrical surface, the tool bit must be ground to fit the proper radius gage (Figure 7-57). Small V-grooves may be machined by using a form tool ground to size or just slightly undersize. Large V-grooves may be machined with the compound rest by finishing each side separately at the desired angle. This method reduces tool bit and work contact area, thus reducing chatter, gouging, and tearing. Since the cutting surface of the tool bit is generally broad, the cutting speed must be slower than that used for general turning. A good guide is to use half of the speed recommended for normal turning. The depth of the groove, or the diameter of the undercut, may be checked by using outside calipers or by using two wires and an outside micrometer (Figure 7-58).

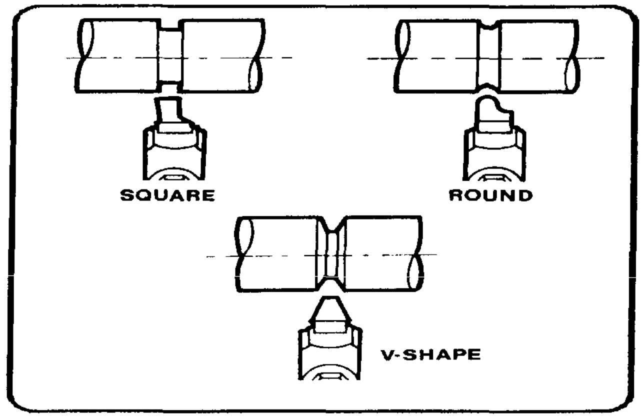


Figure 7-56. Common grooves.

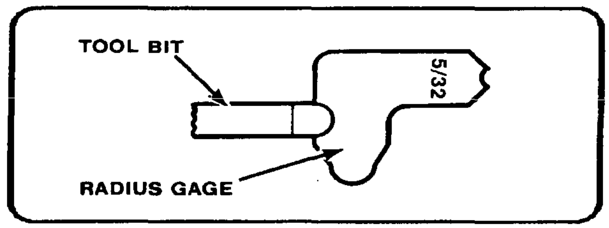


Figure 7-57. Checking tool bit with a radius gage.

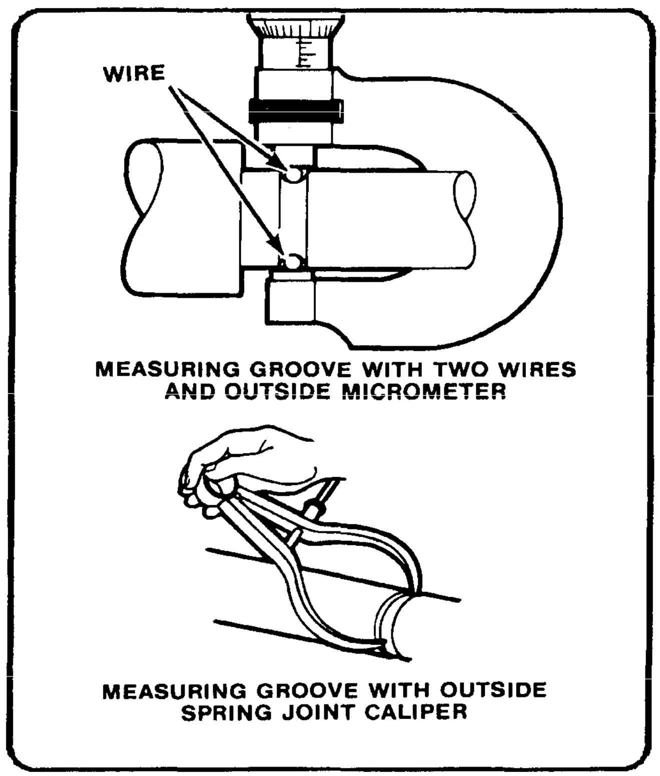


Figure 7-58. Checking the depth of a groove.

When a micrometer and two wires are used. the micrometer reading is equal to the measured diameter of the groove plus two wire diameters.

To calculate measurement over the wires, use the following formula:

Measurement = Outside Diameter+ (2 x wires) – 2 x radius).

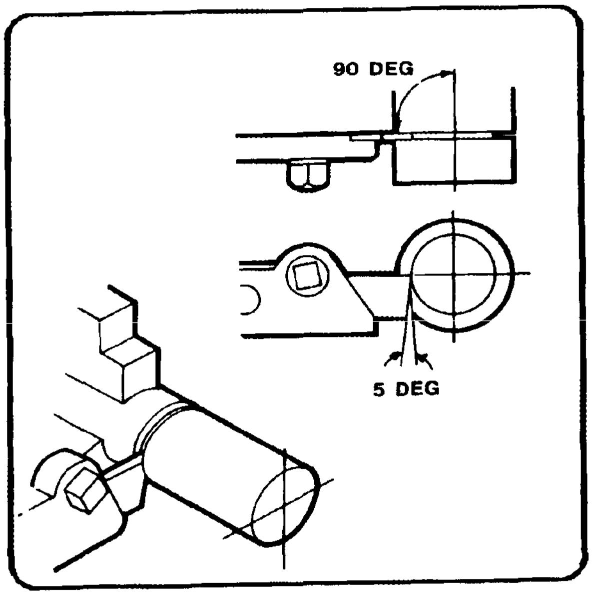


Figure 7-59. Parting.

Parting

Parting is the process of cutting off a piece of stock while it is being held in the lathe. This process uses a specially shaped tool bit with a cutting edge similar to that of a square-nosed tool bit. When parting. be sure to use plenty of coolant, such as a sulfurized cutting oil (machine cast iron dry). Parting tools normally have a 5° side rake and no back rake angles. The blades are sharpened by grinding the ends only. Parting is used to cut off stock. such as tubing. that is impractical to saw off with a power hacksaw.

Parting is also used to cut off work after other machining operations have been completed (Figure 7-59). Parting tools can be of the forged type. inserted blade type. or ground from a standard tool blank. In order for the tool to have maximum strength, the length of the cutting portion of the blade should extend only enough to be slightly longer than half of the workpiece diameter (able to reach the center of the work). Never attempt to part while the work is mounted between centers,

Work that is to be parted should be held rigidly in a chuck or collet, with the area to be parted as close to the holding device as possible. Always make the parting cut at a right angle to the centerline of the work. Feed the tool bit into the revolving work with the cross slide until the tool completely severs the work. Speeds for parting should be about half that used for straight turning. Feeds should be light but continuous. If chatter occurs. decrease the feed and speed. and check for loose lathe parts or a loose setup. The parting tool should be positioned at center height unless cutting a piece that is over 1-inch thick. Thick pieces should have the cutting tool just slightly above center to account for the stronger torque involved in parting. The length of the portion to be cut off can be measured by using the micrometer carriage stop or by using layout lines scribed on the workpiece. Always have the carriage locked down to the bed to reduce vibration and chatter. Never try to catch the cutoff part in the hand; it will be hot and could burn.

RADII AND FORM TURNING

Occasionally, a radius or irregular shape must be machined on the lathe. Form turning is the process of machining radii and these irregular shapes. The method used to form-turn will depend on the size and shape of the object. the accuracy desired. the time allowed. and the number of pieces that need to be formed. Of the several ways to form-turn. using a form turning tool that is ground to the shape of the desired radius is the most common. Other common methods are using hand

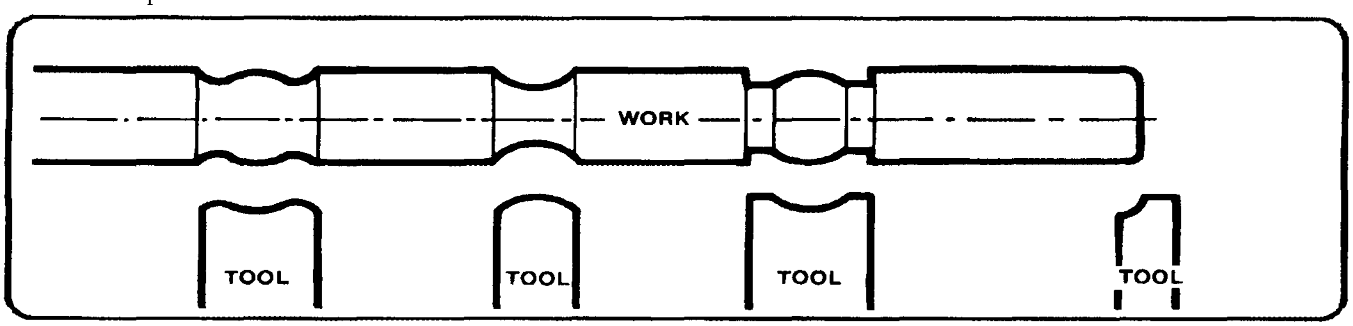


Figure 7-60. Forming tools.

manipulation and filing, using a template and following rod, or using the compound rest and tool to pivot and cut. Two radii are cut in form turning, concave and convex. A concave radius curves inward and a convex radius curves outward.

Forming a Radius Using a Form Turning Tool

Using a form turning tool to cut a radius is a way to form small radii and contours that will fit the shape of the tool. Forming tools can be ground to any desired shape or contour (Figure 7-60), with the only requirements being that the proper relief and rake angles must be ground into the tool's shape. The most practical use of the ground forming tool is in machining several duplicate pieces, since the machining of one or two pieces will not warrant the time spent on grinding the form tool. Use the proper radius gage to check for correct fit. A forming tool has a lot of contact with the work surface, which can result in vibration and chatter. Slow the speed, increase the feed, and tighten the work setup if these problems occur.

Forming a Radius Using Hand Manipulation

Hand manipulation, or free hand, is the most difficult method of form turning to master. The cutting tool moves on an irregular path as the carriage and cross slide are simultaneously manipulated by hand. The desired form is achieved by watching the tool as it cuts and making small adjustments in the movement of the carriage and cross slide. Normally, the right hand works the cross feed movement while the left hand works the carriage movement. The accuracy of the radius depends on the skill of the operator. After the approximate radius is formed, the workpiece is filed and polished to a finished dimension.

Forming a Radius Using a Template

To use a template with a follower rod to form a radius, a full scale form of the work is laid out and cut from thin sheet metal. This form is then attached to the cross slide in such a way that the cutting tool will follow the template. The accuracy of the template will determine the accuracy of the workpiece. Each lathe model has a cross slide and carriage that are slightly different from one another, but they all operate in basically the same way. A mounting bracket must be fabricated to hold the template to allow the cutting tool to follow its shape. This mounting bracket can be utilized for several different operations, but should be sturdy enough for holding clamps and templates. The mounting bracket must be positioned on the carriage to allow for a follower (that is attached to the cross slide) to contact the template and guide the cutting tool. For this operation, the cross slide must be disconnected from the cross feed screw and hand pressure applied to hold the cross slide against the follower and template. Rough-cut the form to the approximate shape before disconnecting the cross feed screw. This way, a finish cut is all that is required while applying hand pressure to the cross slide. Some filing may be needed to completely finish the work to dimension.

Forming a Radius Using the Compound Rest

To use the compound rest and tool to pivot and cut (Figure 7-6 1), the compound rest bolts must be loosened to allow the compound rest to swivel. When using this method, the compound rest and tool are swung from side to side in an arc. The desired radius is formed by feeding the tool in or out with the compound slide. The pivot point is the center swivel point of the compound rest. A concave radius can be turned by positioning the tool in front of the pivot point, while a convex radius can be turned by placing the tool behind the pivot point. Use the micrometer carriage stop to measure precision depths of different radii.

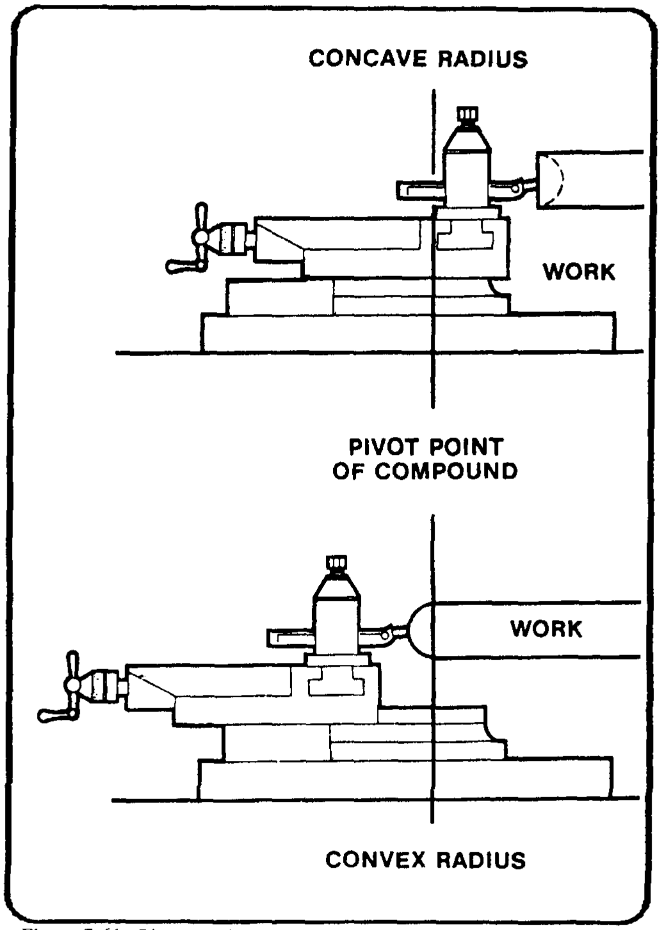


Figure 7-61. Pivots of the compound radius.

TAPER TURNING

When the diameter of a piece changes uniformly from one end to the other, the piece is said to be tapered. Taper turning as a machining operation is the gradual reduction in diameter from one part of a cylindrical workpiece to another part, Tapers can be either external or internal. If a workpiece is tapered on the outside, it has an external taper; if it is tapered on the inside, it has an internal taper. There are three basic methods of turning tapers with a lathe. Depending on the degree, length, location of the taper (internal or external), and the number of pieces to be done, the operator will either use the compound rest, offset the tailstock, or use the taper attachment. With any of these methods the cutting edge of the tool bit must be set exactly on center with the axis of the workpiece or the work will not be truly conical and the rate of taper will vary with each cut.

Compound Rests

The compound rest is favorable for turning or boring short, steep tapers, but it can also be used for longer, gradual tapers providing the length of taper does not exceed the distance the compound rest will move upon its slide. This method can be used with a high degree of accuracy, but is somewhat limited due to lack of automatic feed and the length of taper being restricted to the movement of the slide.

The compound rest base is graduated in degrees and can be set at the required angle for taper turning or boring. With this method, it is necessary to know the included angle of the taper to be machined. The angle of the taper with the centerline is one-half the included angle and will be the angle the compound rest is set for. For example, to true up a lathe center which has an included angle of 60°, the compound rest would be set at 30° from parallel to the ways (Figure 7-41).

If there is no degree of angle given for a particular job, then calculate the compound rest setting by finding the taper per inch, and then calculating the tangent of the angle (which is the: compound rest setting) .

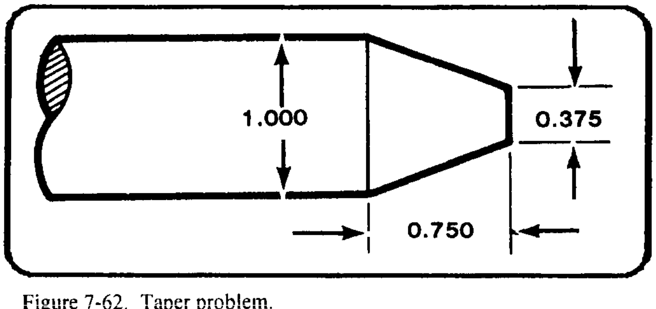


Figure 7-62. Taper problem.

For example, the compound rest setting for the workpiece shown in Figure 7-62 would be calculated in the following manner

$$\textbf{TPI} = \frac{D - d}{L} \qquad \textbf{angle} = \text{TAN}\left(\frac{TPI}{2}\right)$$

Where TPI = taper per inch

D = large diameter,

d = small diameter,

L = length of taper

angle = compound rest setting

The problem is actually worked out by substituting numerical values for the letter variables:

$$TPI = \frac{1.000 - 0.375}{0.750}$$

$$TPI = \frac{0.625}{0.750}$$

$$TPI = 0.833$$

Apply the formula to find the angle by substituting the numerical values for the letter variables:

$$\text{angle} = \text{TAN}\left(\frac{0.833}{2}\right)$$

$$\text{angle} = \text{TAN}\ 0.41650$$

Using the trig charts in TC 9-515 or any other source of trig charts, the TAN of 0.41650 is found to be 22º37'. This angle is referred to as 22 degrees and 37 minutes.

To machine the taper shown in Figure 7-62, the compound rest will be set at 22°37 '. Since the base of the compound rest is not calibrated in minutes, the operator will set the base to an approximate degree reading, make trial cuts, take measurements, and readjust as necessary to obtain the desired angle of taper. The included angle of the workpiece is double that of the tangent of angle (compound rest setting). In this case, the double of 22°37' would equal the included angle of 45°14'.

To machine a taper by this method, the tool bit is set on center with the workpiece axis. Turn the compound rest feed handle in a counterclockwise direction to move the compound rest near its rear limit of travel to assure sufficient traverse to complete the taper. Bring the tool bit into position with the workpiece by traversing and cross-feeding the carriage. Lock the carriage to the lathe bed when the tool bit is in position. Cut from right to left, adjusting the depth of cut by moving the cross feed handle and reading the calibrated collar located on the cross feed handle. feed the tool bit by hand-turning the compound rest feed handle in a clockwise direction.

Offsetting the Tailstock

The oldest and probably most used method of taper turning is the offset tailstock method. The tailstock is made in two pieces: the lower piece is fitted to the bed, while the upper part can be adjusted laterally to a given offset by use of adjusting screws and lineup marks (Figure 7-63).

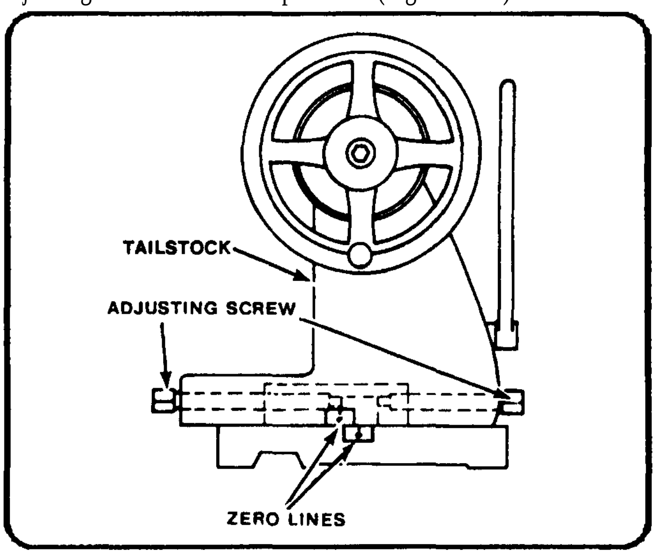


Figure 7-63. Tailstock offset for taper turning.

Since the workpiece is mounted between centers, this method of taper turning can only be used for external tapers. The length of the taper is from headstock center to tailstock center, which allows for longer tapers than can be machined using the compound rest or taper attachment methods.

The tool bit travels along a line which is parallel with the ways of the lathe. When the lathe centers are aligned and the workpiece is machined between these centers, the diameter will remain constant from one end of the piece to the other. If the tailstock is offset, as shown in Figure 7-64, the centerline of the workpiece is no longer parallel with the ways; however, the tool bit continues its parallel movement with the ways, resulting in a tapered workpiece. The tail stock may be offset either toward or away from the operator. When the offset is toward the operator, the small end of the workpiece will be at the tailstock with the diameter increasing toward the headstock end.

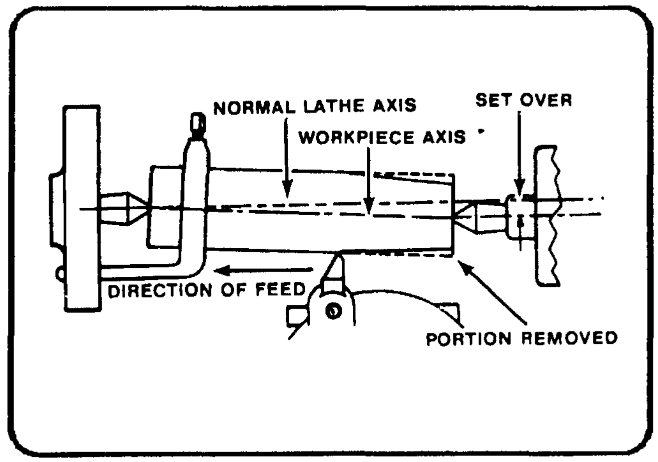


Figure 7-64. Taper turning with tailstock set over.

The offset tailstock method is applicable only to comparatively gradual tapers because the lathe centers, being out of alignment, do not have full bearing on the workpiece. Center holes are likely to wear out of their true positions if the lathe centers are offset too far, causing poor results and possible damage to centers.

The most difficult operation in taper turning by the offset tailstock method is determining the proper distance the tailstock should be moved over to obtain a given taper. Two factors affect the amount the tailstock is offset: the taper desired and the length of the workpiece. If the offset remains constant, workplaces of different lengths, or with different depth center holes, will be machined with different tapers (Figure 7-65).

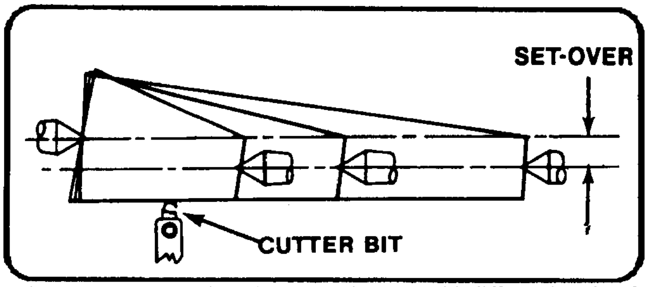


Figure 7-65. Effect of fixed amount of setover with different lengths of workpieces.

The formula for calculating the tailstock offset when the taper is given in taper inches per foot (tpf) is as follows

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

Where: Offset = tailstock offset (in inches)
TPF = taper (in inches per foot)
L = length of taper (in feet) measured along the axis of the workpiece

For example, the amount of offset required to machine a bar 42 inches (3.5 feet) long with a taper of 1/2 inch per foot is calculated as follows:

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

$$\text{OFFSET} = \frac{1/2 \times 42}{24}$$

$$\text{OFFSET} = \frac{0.5 \times 42}{24}$$

$$\text{OFFSET} = \frac{21}{24}$$

OFFSET = 0.875 inch.

Therefore, the tailstock should be offset 0.875 inch to machine the required taper. The formula for calculating the tailstock offset when the taper is given in TPF is as follows:

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

Where **OFFSET** = tailstock offset
TPI = taper per inch
L = length of taper in inches

For example, the amount of offset required to machine a bar 42 inches long with a taper of 0.0416 TPI is calculated as follows:

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

$$\text{OFFSET} = \frac{0.0416 \times 42}{2}$$

$$\text{OFFSET} = \frac{1.7472}{2} \text{ or rounded up } \frac{1.75}{2}$$

OFFSET = .875 inch

Therefore, the tailstock should be offset 0.875 inch to machine the required taper.

If the workpiece has a short taper in any par of it's length and the **TPI** or **TPF** is not given. use the following formula:

$$\text{OFFSET} = \frac{L \times (D-d)}{2 \times L1}$$

Where :

D = Diameter of large end
d = Diameter of small end
L = Total length of workpiece in inches diameter (in inches)
L1 = Length of taper

For example, the amount of tailstock offset required to machine a bar 36 inches (3 feet) in length for a distance of 18 inches (1.5 feet) when the large diameter is 1 3/4 (1 .750) inches and the small diameter is 1 1/2 (1.5) inches is calculated as follows

$$\text{OFFSET} = \frac{L \times (D-d)-}{2XL1}$$

$$\text{OFFSET} = \frac{36 \times (1.750 - 1.5)}{36}$$

$$\text{OFFSET} = \frac{36 \times 0.25}{36}$$

OFFSET = 9/36

OFFSET = 0.25 inch

Therefore, the tailstock would be offset (toward the operator) 0.25 inch to machine the required taper.

Metric tapers can also be calculated for taper turning by using the offset tailstock method. Metric tapers are expressed as a ratio of 1 mm per unit of length. Figure 7-66 shows how the work would taper 1 mm in a distance of 20 mm. This taper would then be given as a ratio of 1:20 and would be annotated on small diameter (d) will be 1 mm greater (d +). Refer to the following formula for calculating the dimensions of a metric taper. If the small diameter (d), the unit length of taper (k), and the total length of taper (1) are known, then the large diameter (D) may be calculated. The large diameter (D) will be equal to the small diameter plus the amount of taper. The amount of taper for the unit length (k) is (d + 1) - (d). Therefore, the amount of taper per millimeter of unit length = (l/k). The total amount of taper will be the taper per millimeter (l/k) multiplied by the total length of taper (l).

$$\text{Total Taper} = \frac{1 \times 1}{k} \text{ or } \frac{1}{k}$$

$$D = d + \text{total amount of taper}$$

For example, to calculate for the large diameter D for a 1:30

$$D = \frac{d}{k} + \frac{1}{k}$$

taper having a small diameter of 10 mm and a length of 60 mm, do the following:

Since the taper is the ratio 1:30, then (k)= 30, since 30 is the unit of length.

$$D = \frac{d + 1}{k}$$

$$D = \frac{10 + 60}{30}$$

$$D = 10 + 2$$

$$D = 12 \text{ mm}$$

Tailstock offset is calculated as follows:

$$\text{Tailstock offset} = \frac{D - d}{2 \times 1} \times L$$

D = large diameter

d = small diameter

I = length of taper

L = length of the workpiece

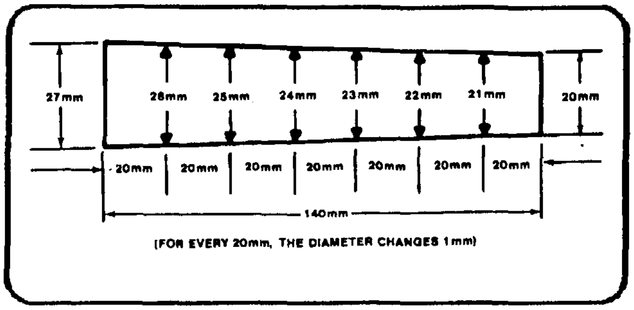


Figure 7-66. Metric taper, 1:200mm.

Thus, to determine the tailstock offset in millimeters for the taper in Figure 7-67, substitute the numbers and solve for the offset. Calculate the tailstock offset required to turn a 1:50 taper 200 mm long on a workpiece 800 mm long. The small diameter of the tapered section is 49 mm.

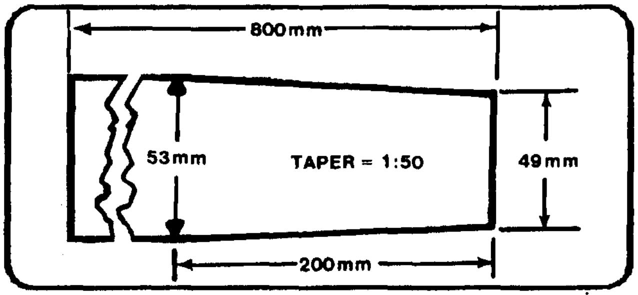


Figure 7-67. Metric taper problem.

$$D = d + \frac{1}{k}$$

$$D = 49 + \frac{200}{50}$$

$$D = 49 + 4 \text{ or } D = 53$$

The tailstock would be moved toward the operator 8 mm.

$$(to) = \frac{53 - 49}{2 \times 200} \times 800$$

$$(to) = \frac{4}{400} \times 800$$

$$(to) = 0.01 \times 800 \text{ or } 8 \text{ mm}$$

Another important consideration in calculating offset is the distance the lathe centers enter the workpiece. The length of the workpiece (L) should be considered as the distance between the points of the centers for all offset computations.

Therefore, if the centers enter the workpiece 1/8 inch on each end and the length of the workpiece is 18 inches, subtract 1/4 inch from 18 inches and compute the tailstock offset using 17 3/4 inches as the workpiece length (L).

The amount of taper to be cut will govern the distance the top of the tailstock is offset from the centerline of the lathe. The tailstock is adjusted by loosening the clamp nuts, shifting the upper half of the tailstock with the adjusting screws, and then tightening them in place.

There are several methods the operator may use to measure the distance the tailstock has been offset depending upon the accuracy desired (Figure 7-68).

One method is to gage the distance the lineup marks on the rear of the tailstock have moved out of alignment. This can be done by using a 6-inch rule placed near the lineup marks or by transferring the distance between the marks to the rule's surface using a pair of dividers.

Another common method uses a rule to check the amount of offset when the tailstock is brought close to the headstock.

Where accuracy is required, the amount of offset may be measured by means of the graduated collar on the cross feed screw. First compute the amount of offset; next, set the tool holder in the tool post so the butt end of the holder faces the tailstock spindle. Using the cross feed, run the tool holder in by hand until the butt end touches the tailstock spindle. The pressure should be just enough to hold a slip of paper placed between the tool holder and the spindle. Next, move the cross slide to bring the tool holder toward you to remove the backlash. The reading on the cross feed micrometer collar may be recorded, or the graduated collar on the cross feed screw may be set at zero. Using either the recorded reading or the zero setting for a starting point, bring the cross slide toward you the distance computed by the offset. Loosen and offset the tailstock until the slip of paper drags when pulled between the tool holder and the spindle. Clamp the tailstock to the lathe bed.

Another and possibly the most precise method of measuring the offset is to use a dial indicator. The indicator is set on the center of the tailstock spindle while the centers are still aligned. A slight loading of the indicator is advised since the first 0.010 or 0.020 inches of movement of the indicator may be inaccurate due to mechanism wear causing fluctuating readings. Load the dial indicators follows: Set the bezel to zero and move tailstock towards the operator the calculated Famount. Then clamp the tailstock to the way.

Whichever method is used to offset the tailstock, the offset must still be checked before starting to cut. Set the dial indicator in the tool post with its spindle just barely touching far right side of the workpiece. Then, rotate the carriage toward the headstock exactly I inch and take the reading from the dial indicator. One inch is easily accomplished using the thread chasing dial. It is 1 inch from one number to another.

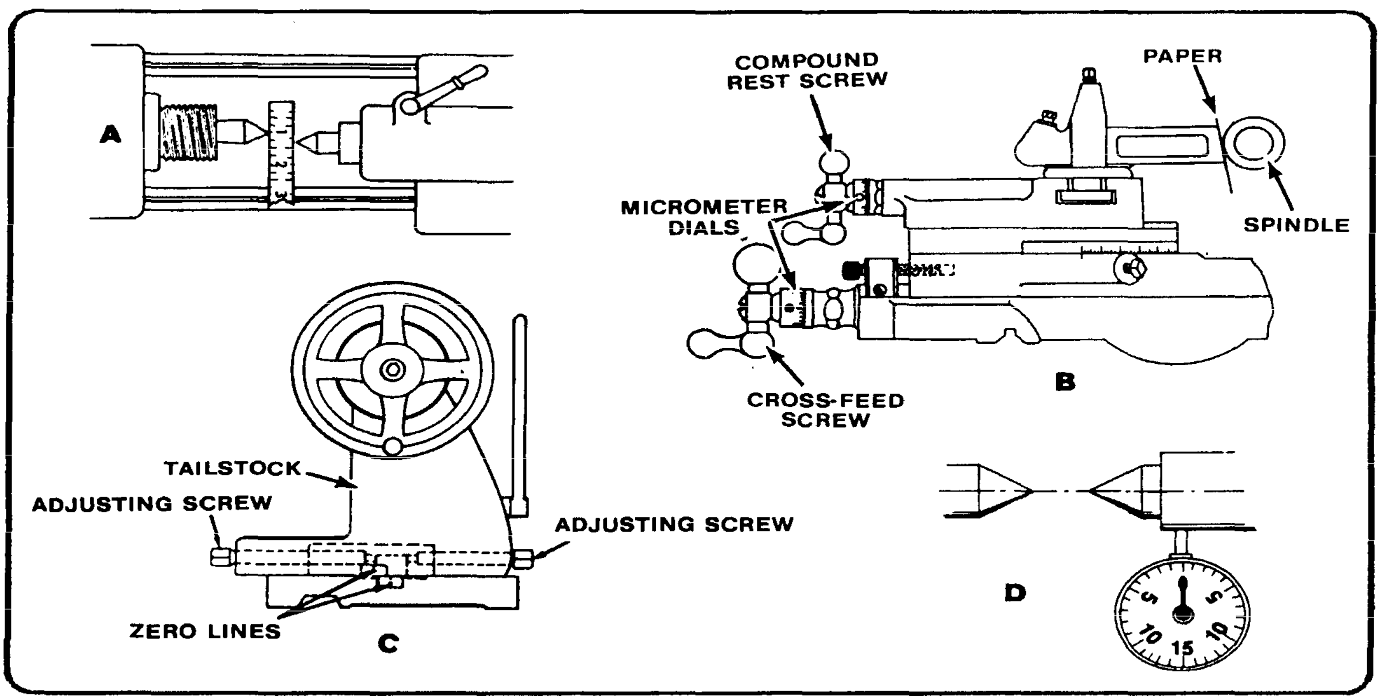


Figure 7-68. Measuring tailstock offset.

Alternatively, 1 inch can be drawn out on the workpiece. The dial indicator will indicate the taper for that 1 inch and, if needed, the tailstock can be adjusted as needed to the precise taper desired. If this method of checking the taper is not used, then an extensive trial and error method is necessary.

To cut the taper, start the rough turning at the end which will be the small diameter and feed longitudinally toward the large end (Figure 7-64). The tailstock is offset toward the operator and the feed will be from right to left. The tool bit, a right-hand turning tool bit or a round-nose turning tool bit, will have its cutting edge set exactly on the horizontal centerline of the workpiece, not above center as with straight turning

Taper Attachment

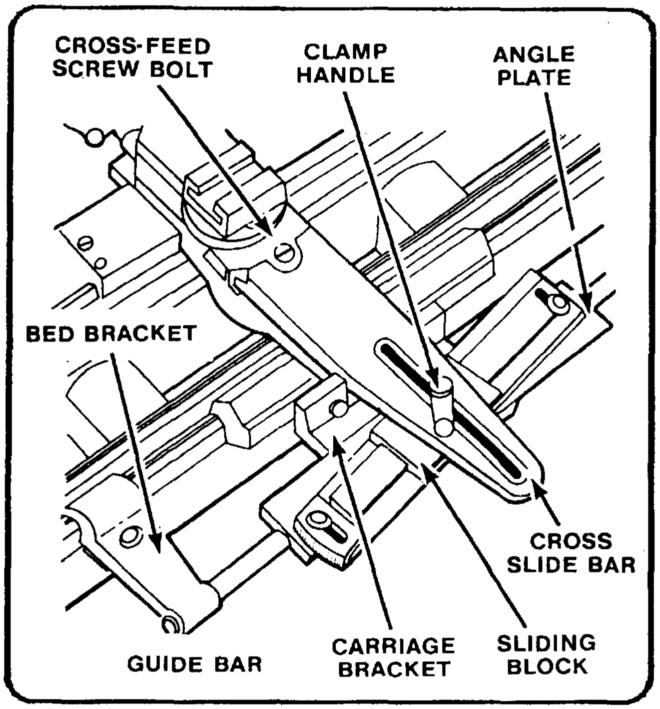


Figure 7-69. Taper attachment.

The taper attachment (Figure 7-69) has many features of special value, among which are the following:

- The lathe centers remain in alignment and the center holes in the work are not distorted.
- The alignment of the lathe need not be disturbed, thus saving considerable time and effort.
- Taper boring can be accomplished as easily as taper turning.
- A much wider range is possible than by the offset method. For example, to machine a 3/4-inch-per-foot taper on the end of a bar 4 feet long would require an offset of 1 1/2 inches, which is beyond the capabilities of a regular lathe but can be accomplished by use of the taper attachment.

Some engine lathes are equipped with a taper attachment as standard equipment and most lathe manufacturers have a taper attachment available. Taper turning with a taper attachment, although generally limited to a taper of 3 inches per foot and to a set length of 12 to 24 inches, affords the most accurate means for turning or boring tapers. The taper can be set directly on the taper attachment in inches per foot; on some attachments, the taper can be set in degrees as well.

Ordinarily, when the lathe centers are in line, the work is turned straight, because as the carriage feeds along, the tool is always the same distance from the centerline. The purpose of the taper attachment is to make it possible to keep the lathe centers in line, but by freeing the cross slide and then guiding it (and the tool bit) gradually away from the centerline, a taper can be cut or, by guiding it gradually nearer the centerline (Figure 7-70), a taper hole can be bored.

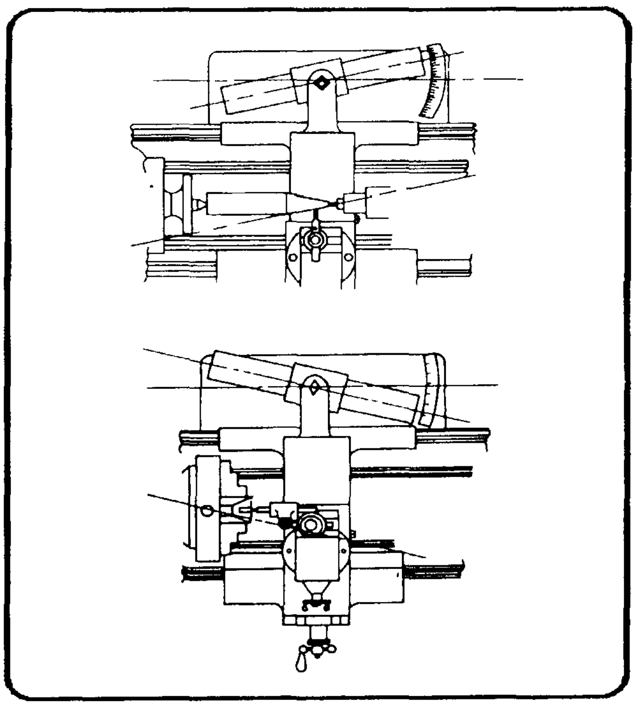


Figure 7-70. Taper turning and boring.

A plain taper attachment for the lathe is illustrated in Figure 7-69. A bed bracket attaches to the lathe bed and keeps the angle plate from moving to the left or the right. The carriage bracket moves along the underside of the angle plate in a dovetail and keeps the angle plate from moving in or out on the bed bracket. The taper to be cut is set by placing the guide bar, which clamps to the angle plate, at an angle to the ways of the lathe bed. Graduations on one or both ends of the guide bar are used to make this adjustment. A sliding block which rides on a dovetail on the upper surface of the guide bar is secured during the machining operation to the cross slide bar of the carriage, with the cross feed screw of the carriage being disconnected. Therefore, as the carriage is traversed during the feeding operation, the cross slide bar follows the guide bar, moving at the predetermined angle from the ways of the bed to cut the taper. It is not necessary to remove the taper attachment when straight turning is desired. The guide bar can be set parallel to the ways, or the clamp handle can be released permitting the sliding block to move without affecting the cross slide bar, and the cross feed screw can be reengaged to permit power cross feed and control of the cross slide from the apron of the carriage.

Modern lathes use a telescopic taper attachment. This attachment allows for using the cross feed, and set up is a bit faster than using a standard taper attachment. To use the telescopic attachment, first set the tool bit for the required diameter of the work and engage the attachment by tightening the binding screws, the location and number of which depend upon the design of the attachment. The purpose of the binding screws is to bind the cross slide so it may be moved only by turning the cross feed handle, or, when loosened, to free the cross slide for use with the taper attachment. To change back to straight turning with the telescopic attachment, it is necessary only to loosen the binding screws.

When cutting a taper using the taper attachment, the direction of feed should be from the intended small diameter toward the intended large diameter. Cutting in this manner, the depth of cut will decrease as the tool bit passes along the workpiece surface and will assist the operator in preventing possible damage to the tool bit, workpiece, and lathe by forcing too deep a cut.

The length of the taper the guide bar will allow is usually not over 12 to 24 inches, depending on the size of the lathe. It is possible to machine a taper longer than the guide bar allows by moving the attachment after a portion of the desired taper length has been machined; then the remainder of the taper can be cut. However, this operation requires experience.

If a plain standard taper attachment is being used, remove the binding screw in the cross slide and set the compound rest perpendicular to the ways. Use the compound rest graduated collar for depth adjustments.

When using the taper attachment, there may be a certain amount of "lost motion" (backlash) which must be eliminated or serious problems will result. In every slide and every freely revolving screw there is a certain amount of lost motion which is very noticeable if the parts are worn. Care must be taken to remove lost motion before proceeding to cut or the workpiece will be turned or bored straight for a short distance before the taper attachment begins to work. To take up lost motion when turning tapers, run the carriage back toward the dead center as far as possible, then feed forward by hand to the end of the workpiece where the power feed is engaged to finish the cut. This procedure must be repeated for every cut.

The best way to bore a taper with a lathe is to use the taper attachment. Backlash must be removed when tapers are being bored with the taper attachment, otherwise the hole will be bored straight for a distance before the taper starts. Two important factors to consider: the boring tool must be set exactly on center with the workpiece axis, and it must be small enough in size to pass through the hole without rubbing at the small diameter. A violation of either of these factors will result in a poorly formed, inaccurate taper or damage to the tool and workpiece. The clearance of the cutter bit shank and boring tool bar must be determined for the smaller diameter of the taper. Taper boring is accomplished in the same manner as taper turning.

To set up the lathe attachment for turning a taper, the proper TPF must be calculated and the taper attachment set-over must be checked with a dial indicator prior to cutting. Calculate the taper per foot by using the formula:

$$TPF = \frac{D - d}{L} \times 12$$

TPF = taper per foot,

D = large diameter (in inches),

d = small diameter (in inches),

L = length of taper

After the TPF is determined, the approximate angle can be set on the graduated TPF scale of the taper attachment. Use a dial indicator and a test bar to set up for the exact taper. Check the taper in the same manner as cutting the taper by allowing for backlash and moving the dial indicator along the test bar from the tailstock end of the head stock end. Check the TPI by using the thread-chasing dial, or using layout lines of 1-inch size, and multiply by 12 to check the TPF. Make any adjustments needed, set up the work to be tapered, and take a trial cut. After checking the trial cut and making final adjustments, continue to cut the taper to required dimensions as in straight turning. Some lathes are set up in metric measurement instead of inch measurement. The taper attachment has a scale graduated in degrees, and the guide bar can be set over for the angle of the desired taper. If the angle of the taper is not given, use the following formula to determine the amount of the guide bar set over:

$$\text{Guide Bar Set Over (in millimeters)} = \frac{D + d}{2} \times \frac{L}{l}$$

D = large diameter of taper (mm)

d = small diameter of taper (mm)

I = length of taper (mm)

L = length of guide bar (mm)

Reference lines must be marked on the guide bar an equal distance from the center for best results.

A metric dial indicator can be used to measure the guide bar set over, or the values can be changed to inch values and an inch dial indicator used.

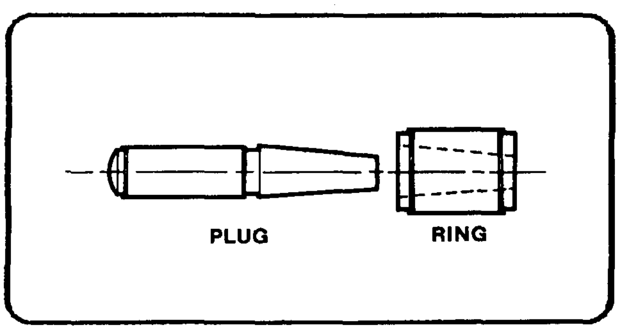


Figure 7-71. Taper gages.

Checking Tapers for Accuracy

Tapers must be checked for uniformity after cutting a trial cut. Lay a good straight edge along the length of the taper and look for any deviation of the angle or surface. Deviation is caused by backlash or a lathe with loose or worn parts. A bored taper may be checked with a plug gage (Figure 7-71) by marking the gage with chalk or Prussian blue pigment. Insert the gage into the taper and turn it one revolution. If the marking on the gage has been rubbed evenly, the angle of taper is correct. The angle of taper must be increased when there is not enough contact at the small end of the plug gage, and it must be decreased when there is not enough contact at the large end of the gage. After the correct taper has been obtained but the gage does not enter the workpiece far enough, additional cuts must be taken to increase the diameter of the bore.

An external taper may be checked with a ring gage (Figure 7-71). This is achieved by the same method as for checking internal tapers, except that the workpiece will be marked with the chalk or Prussian blue pigment rather than the gage. Also, the angle of taper must be decreased when there is not enough contact at the small end of the ring gage and it must be increased when there is not enough contact at the large end of the gage. If no gage is available, the workpiece should be tested in the hole it is to fit. When even contact has been obtained, but the tapered portion does not enter the gage or hole far enough, the diameter of the piece is too large and must be decreased by additional depth of cut

Another good method of checking external tapers is to scribe lines on the workpiece 1 inch apart (Figure 7-72); then, take measurements with an outside micrometer. Subtracting the small reading from the large reading will give the taper per inch.

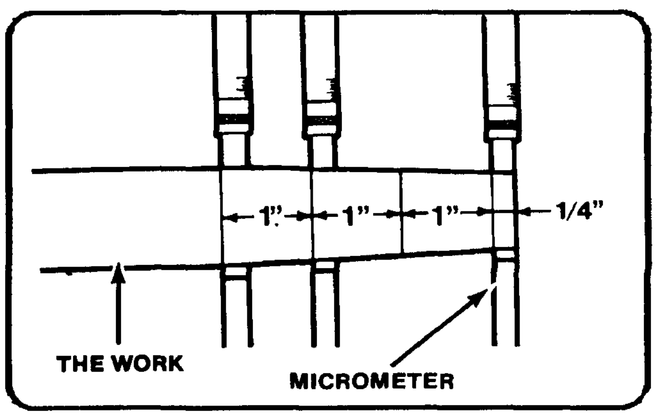


Figure 7-72. Measuring a taper with a micrometer.

Duplicating a Tapered Piece

When the taper on a piece of work is to be duplicated and the original piece is available, it may be placed between centers on the lathe and checked with a dial indicator mounted in the tool post.. When the setting is correct, the dial indicator reading will remain constant when moved along the length of taper.

This same method can be used on workplaces without centers provided one end of the workpiece can be mounted and held securely on center in the headstock of the lathe. For example, a lathe center could be mounted in the lathe spindle by use of the spindle sleeve, or a partially tapered workpiece could be held by the nontapered portion mounted in a collet or a chuck. Using either of these two methods of holding the work, the operator could use only the compound rest or the taper attachment for determining and machining the tapers.

Standard Tapers

There are various standard tapers in commercial use, the most common ones being the Morse tapers, the Brown and Sharpe tapers, the American Standard Machine tapers, the Jarno tapers, and the Standard taper pins.

Morse tapers are used on a variety of tool shanks, and exclusively on the shanks of twist drills. The taper for different numbers of Morse tapers is slightly different, but is approximately 5/8 inch per foot in most cases. Dimensions for Morse tapers are given in Table 7-4 in Appendix A.

Brown and Sharpe tapers are used for taper shanks on tools such as end mills and reamers. The taper is approximately ½ inch per foot for all sizes except for taper No 10, where the taper is 0.5161 inch per foot.

The American Standard machine tapers are composed of a self-holding series and a steep taper series. The self-holding taper series consists of 22 sizes which are given in Table 7-5 in Appendix A. The name “self-holding” has been applied where the angle of the taper is only 2° or 3° and the shank of the tool is so firmly seated in its socket that there is considerable frictional resistance to any force tending to. turn or rotate the tool in the holder. The self-holding tapers are composed of selected tapers from the Morse, the Brown and Sharpe, and the ¾-inch-per foot machine taper series. The smaller sizes of self-holding tapered shanks are provided with a tang to drive the cutting tool. Larger sizes employ a tang drive with the shank held by a key, or a key drive with the shank held with a draw bolt. The steep machine tapers consist of a preferred series and an intermediate series as given in Table 7-6 in Appendix A. A steep taper is defined as a taper having an angle large enough to ensure the easy or self-releasing feature. Steep tapers have a 3 ½-inch taper per foot and are used mainly for aligning milling machine arbors and spindles, and on some lathe spindles and their accessories.

The Jarno taper is based on such simple formulas that practically no calculations are required when the number of taper is known. The taper per foot of all Jarno tapers is 0.600 inch per foot. The diameter at the large end is as many eighths, the diameter at the small end is as many tenths, and the length as many half-inches as indicated by the number of the taper. For example: A No 7 Jarno taper is 7/8 inch in diameter at the large end; 7/10 or 0.7 inch in diameter at the small end; and 7/2, or 3 1/2 inches long. Therefore, formulas for these dimensions would read:

$$\text{Diameter at small end} = \frac{\text{No. of timer}}{8}$$

$$\text{Diameter at small end} = \frac{\text{No. of taper}}{10}$$

$$\text{Length of taper} = \frac{\text{No. of taper}}{2}$$

The Jarno taper is used on various machine tools, especially profiling machines and die-sinking machines. It has also been used for the headstock and tailstock spindles on some lathes.

The Standard taper pins are used for positioning and holding parts together and have a ¼-inch taper per foot. Standard sizes in these pins range from No 7/0 to No 10 and are given in Table 7-7 in Appendix A. The tapered holes used in conjunction with the tapered pins utilize the processes of step-drilling and taper reaming.

To preserve the accuracy and efficiency of tapers (shanks and holes), they must be kept free from dirt, chips, nicks, or burrs. The most important thing in regard to tapers is to keep them clean. The next most important thing is to remove all oil by wiping the tapered surfaces with a soft, dry cloth before use, because an oily taper will not hold.

SCREW THREAD CUTTING

Screw threads are cut with the lathe for accuracy and for versatility. Both inch and metric screw threads can be cut using the lathe. A thread is a uniform helical groove cut inside of a cylindrical workpiece, or on the outside of a tube or shaft. Cutting threads by using the lathe requires a

thorough knowledge of the different principles of threads and procedures of cutting. Hand coordination, lathe mechanisms, and cutting tool angles are all interrelated during the thread cutting process. Before attempting to cut threads on the lathe a machine operator must have a thorough knowledge of the principles, terminology and uses of threads.

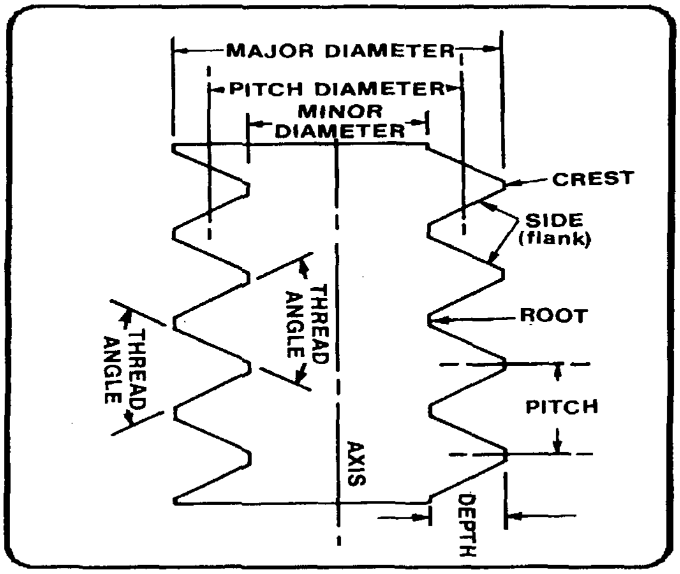


Figure 7-73. Screw thread terminology.

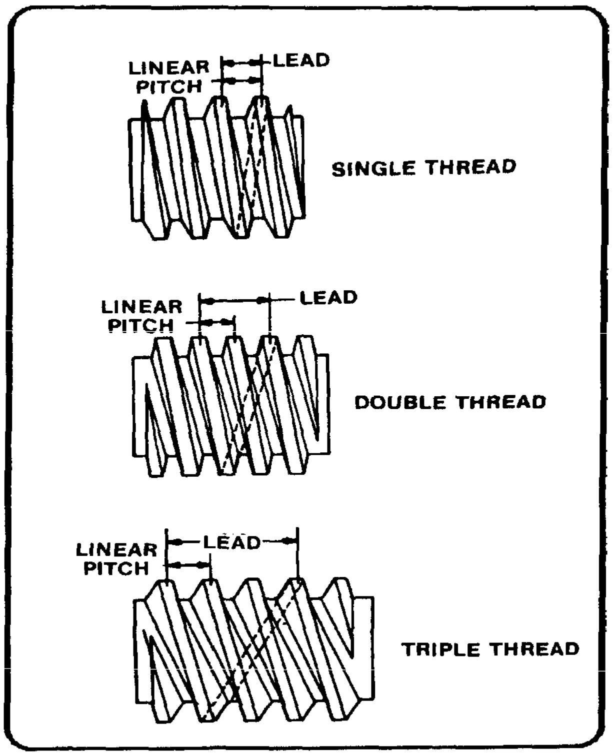


Figure 7-74. Screw thread types.

Screw Thread Terminology

The common terms and definitions below are used in screw thread work and will be used in discussing threads and thread cutting.

- External or male thread is a thread on the outside of a cylinder or cone.
- Internal or female thread is a thread on the inside of a hollow cylinder or bore.
- Pitch is the distance from a given point on one thread to a similar point on a thread next to it, measured parallel to the axis of the cylinder. The pitch in inches is equal to one divided by the number of threads per inch.
- Lead is the distance a screw thread advances axially in one complete revolution. On a single-thread screw, the lead is equal to the pitch. On a double-thread screw, the lead is equal to twice the pitch, and on a triple-thread screw, the lead is equal to three times the pitch (Figure 7-74).
- Crest (also called “flat”) is the top or outer surface of the thread joining the two sides.
- Root is the bottom or inner surface joining the sides of two adjacent threads.
- Side is the surface which connects the crest and the root (also called the flank).
- Angle of the thread is the angle formed by the intersection of the two sides of the threaded groove.
- Depth is the distance between the crest and root of a thread, measured perpendicular to the axis.
- Major diameter is the largest diameter of a screw thread.
- Minor diameter is the smallest diameter of a screw thread.
- Pitch diameter is the diameter of an imaginary cylinder formed where the width of the groove is equal to one-half of the pitch. This is the critical dimension of threading as the fit of the thread is determined by the pitch diameter (Not used for metric threads).

- Threads per inch is the number of threads per inch may be counted by placing a rule against the threaded parts and counting the number of pitches in 1 inch. A second method is to use the screw pitch gage. This method is especially suitable for checking the finer pitches of screw threads.

- A single thread is a thread made by cutting one single groove around a rod or inside a hole. Most hardware made, such as nuts and bolts, has single threads. Double threads have two grooves cut around the cylinder. There can be two, three, or four threads cut around the outside or inside of a cylinder. These types of special threads are sometimes called multiple threads.

- A right-hand thread is a thread in which the bolt or nut must be turned to the right (clockwise) to tighten.

- A left hand thread is a thread in which the bolt or nut must turn to the left (counterclockwise) to tighten.

- Thread fit is the way a bolt and nut fit together as to being too loose or too tight.

- Metric threads are threads that are measured in metric measurement instead of inch measurement.

Screw Thread Forms

The most commonly used screw thread forms are detailed in the following paragraphs. One of the major problems in industry is the lack of a standard form for fastening devices. The screw thread forms that follow attempt to solve this problem; however, there is still more than one standard form being used in each industrial nation. The International Organization for Standardization (IS0) met in 1975 and drew up a standard metric measurement for screw threads, the new IS0 Metric thread Standard (previously known as the Optimum Metric Fastener System). Other thread forms are still in general use today, including the American (National) screw thread form, the square thread, the Acme thread, the Brown and Sharpe 29° worm screw thread, the British Standard Whitworth thread, the Unified thread, and different pipe threads. All of these threads can be cut by using the lathe.

- The IS0 Metric thread standard is a simple thread system that has threaded sizes ranging in diameter from 1.6 mm to 100 mm (see Table 7-8 in Appendix A). These metric threads are identified by the capital M, the nominal diameter, and the pitch. For example, a metric thread with an outside diameter of 5 mm and a pitch of 0.8 mm would be given as M 5 x 0.8. The IS0 metric thread standard simplifies thread design, provides for good strong threads, and requires a smaller inventory of screw fasteners than used by other thread forms. This IS0 Metric thread has a 60° included angle and a crest that is 1.25 times the pitch (which is similar to the National thread form). The depth of thread is 0.6134 times the pitch, and the flat on the root of the thread is wider than the crest. The root of the ISO Metric thread is 0.250 times the pitch (Table 7-9).

- The American (National) screw thread form is divided into four series, the National Coarse (NC), National Fine (NF), National Special (NS), and National Pipe threads (NPT), 11 series of this thread form have the same shape and proportions. This thread has a 60° included angle. The root and crest are 0.125 times the pitch. This thread form is widely used in industrial applications for fabrication and easy assembly and construction of machine parts. Table 7-9 in Appendix A gives the different values for this thread form.

- The British Standard Whitworth thread form thread has a 55° thread form in the V-shape. It has rounded crests and roots.

- The Unified thread form is now used instead of the American (National) thread form. It was designed for interchangeability between manufacturing units in the United States, Canada, and Great Britain. This thread is a combination of the American (National) screw thread form and the British Whitworth screw thread forms. The thread has a 60° angle with a rounded root, while the crest can be rounded or flat. (In the United States, a flat crest is preferred.) The internal thread of the unified form is like the American (National) thread form but is not cut as deep, leaving a crest of one-fourth the pitch instead of one-eighth the pitch. The coarse thread series of the unified system is designated UNC, while the fine thread series is designated UNF. (See Tble 7-9 in Appendix A for thread form and values.

- The American National 29° Acme was designed to replace the standard square thread, which is difficult to machine using normal taps and machine dies. This thread is a power transmitting type of thread for use in jacks, vises, and feed screws. Table 7-9 lists the values for Acme threads.

The Brown and Sharpe 29° worm screw thread uses a 29° angle, similar to the Acme thread. The depth is greater and the widths of the crest and root are different (Table 7-9 in Appendix A). This is a special thread used to mesh with worm gears and to transmit motion between two shafts at right angles to each other that are on separate planes. This thread has a self-locking feature making it useful for winches and steering mechanisms.

- The square screw thread is a power transmitting thread that is being replaced by the Acme thread. Some vises and lead screws may still be equipped with square threads. Contact areas between the threads are small, causing screws to resist wedging, and friction between the parts is minimal (Table 7-9 in Appendix A).

- The spark plug thread (international metric thread type) is a special thread used extensively in Europe, but seen only on some spark plugs in the United States. It has an included angle of 60° with a crest and root that are 0.125 times the depth.

- Different types of pipe thread forms are in use that have generally the same characteristics but different fits. Consult the Machinery's Handbook or a similar reference for this type of thread.

THREAD FIT AND CLASSIFICATIONS

The Unified and American (National) thread forms designate classifications for fit to ensure that mated threaded parts fit to the tolerances specified. The unified screw thread form specifies several classes of threads which are Classes 1A, 2A, and 3A for screws or external threaded parts, and 1B, 2B, and 3B for nuts or internal threaded parts. Classes 1 A and 1 B are for a loose fit where quick assembly and rapid production are important and shake or play is not objectionable. Classes 2A and 2B provide a small amount of play to prevent galling and seizure in assembly and use. and sufficient clearance for some plating. Classes 2A and 2B are recommended for standard practice in making commercial screws. bolts. and nuts. Classes 3A and 3B have no allowance and 75 percent of the tolerance of Classes 2A and 2B A screw and nut in this class may vary from a fit having no play to one with a small amount of play. Only high grade products are held to Class 3 specifications.

Four distinct classes of screw thread fits between mating threads (as between bolt and nut) have been designated for the American (National) screw thread form. Fit is defined as "the relation between two mating parts with reference to ease of assembly. " These four fits are produced by the application of tolerances which are listed in the standards.

The four fits are described as follows:

- Class 1 fit is recommended only for screw thread work where clearance between mating parts is essential for rapid assembly and where shake or play is not objectionable.

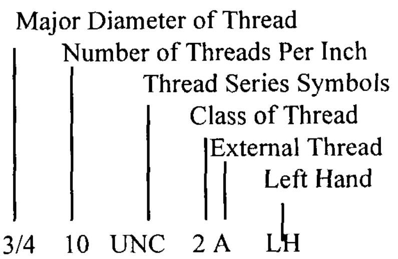
- Class 2 fit represents a high quality of thread product and is recommended for the great bulk of interchangeable screw thread work.

- Class 3 fit represents an exceptionally high quality of commercially threaded product and is recommended only in cases where the high cost of precision tools and continual checking are warranted.

- Class 4 fit is intended to meet very unusual requirements more exacting than those for which Class 3 is intended. It is a selective fit if initial assembly by hand is required. It is not. as yet. adaptable to quantity production.

Thread Designations

In general. screw thread designations give the screw number (or diameter) first. then the thread per inch. Next is the thread series containing the initial letter of the series. NC (National Coarse). UNF (Unified Fine). NS (National Special). and so forth. followed by the class of fit. If a thread is left-hand. the letters LH follow the fit. An example of designations is as follows:



Two samples and explanations of thread designations are as follows:

- No 12 (0.216) -24 NC-3. This is a number 12 (0.216-inch diameter) thread. 24 National Coarse threads per inch. and Class 3 ways of designating the fit between parts. including tolerance grades. tolerance positions. and tolerance classes. A simpler fit.

- 1/4-28 UNF-2A LH. This is a l/4-inch diameter thread. 28 Unified Fine threads per inch, Class 2A fit, and left-hand thread.

Metric Thread Fit and Tolerance

The older metric screw thread system has over one hundred different thread sizes and several ways of designating the fit between parts. including tolerance grades. tolerance positions. and tolerance classes. A simple system was devised with the latest ISO Metric thread standard that uses one internal fit and two external fit designations to designate the tolerance (class) of fit. The symbol 6H is used to designate the fit for an internal thread (only the one symbol is used). The two symbols 6g and 5g6g are used to designate the fit for an external thread. 6g being used for general purpose threads and Sg6g used to designate a close fit. A fit between a pair of threaded parts is indicated by the internal thread (nut) tolerance fit designation followed by the external thread (bolt) tolerance fit designation with the two separated by a stroke. An example is M 5 x 0.8-Sg6g/6H. where the nominal or major diameter is 5 mm. the pitch is 0.8 mm. and a close tit is intended for the bolt and nut. Additional information on ISO metric threads and specific fits can be found in any updated engineer's handbook or machinist's handbook.

THREAD CUTTING TOOL BITS

Cutting V-threads with a 60 degrees thread angle is the most common thread cutting operation done on a lathe. V-threads. with the 60 degree angle. are used for metric thread cutting and for American (National) threads and Unified threads. To properly cut V-shaped threads. the single point tool bit must be ground for the exact shape of the thread form. to include the root of the thread (Figure 7-75).

For metric and American (National) thread forms. a flat should be ground at the point of the tool bit (Figure 7-76). perpendicular to the center line of the 600 thread angle. See the thread form table for the appropriate thread to determine the width of the Sat. For unified thread forms. the tip of the tool bit should be ground with a radius formed to fit the size of the root of the thread. Internal unified threads have a flat on the tip of the tool bit. In all threads listed above. the tool bit should be ground with enough side relief angle and enough front clearance angle (Figure 7-76). Figure 7-77 illustrates the correct steps involved in grinding a thread-cutting tool bit.

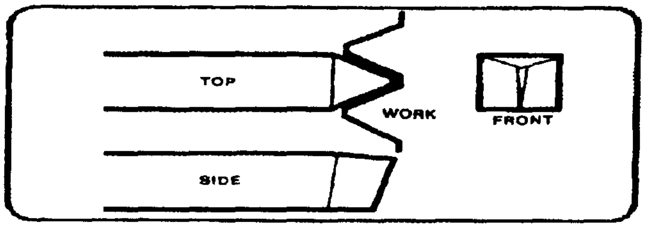


Figure 7-75. V-shaped thread cutter.

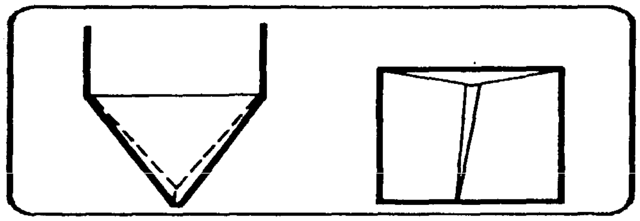


Figure 7-76. Relief angles on a thread cutting tool bit.

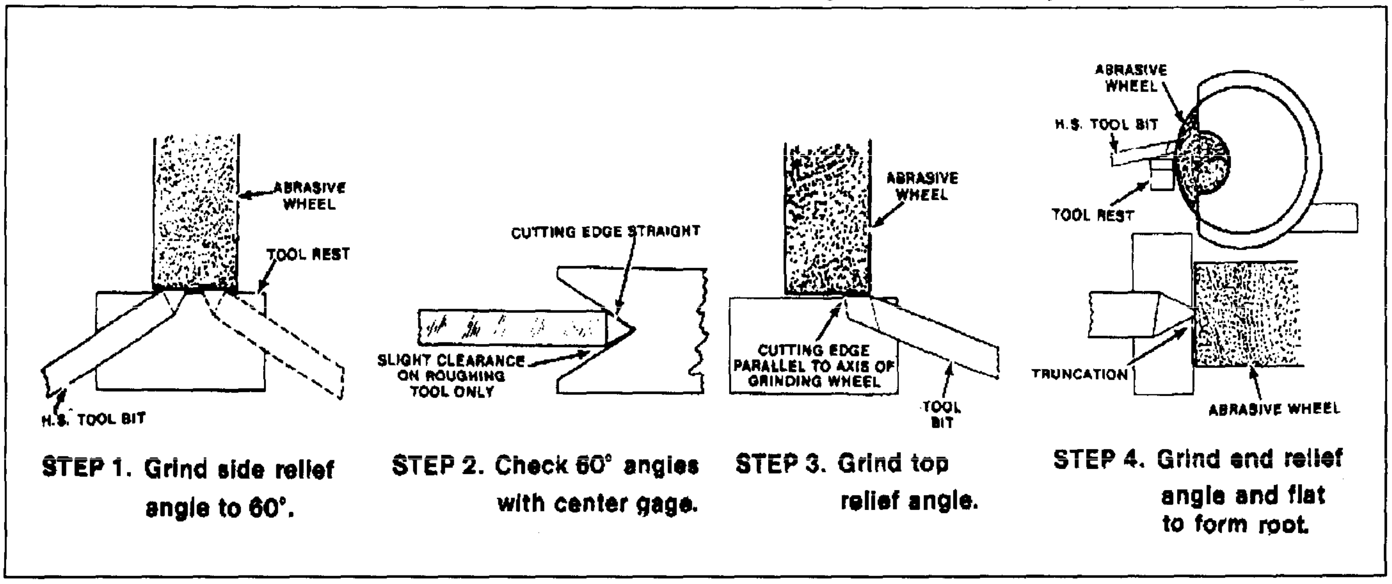


Figure 7-77. Grinding a thread cutting tool bit.

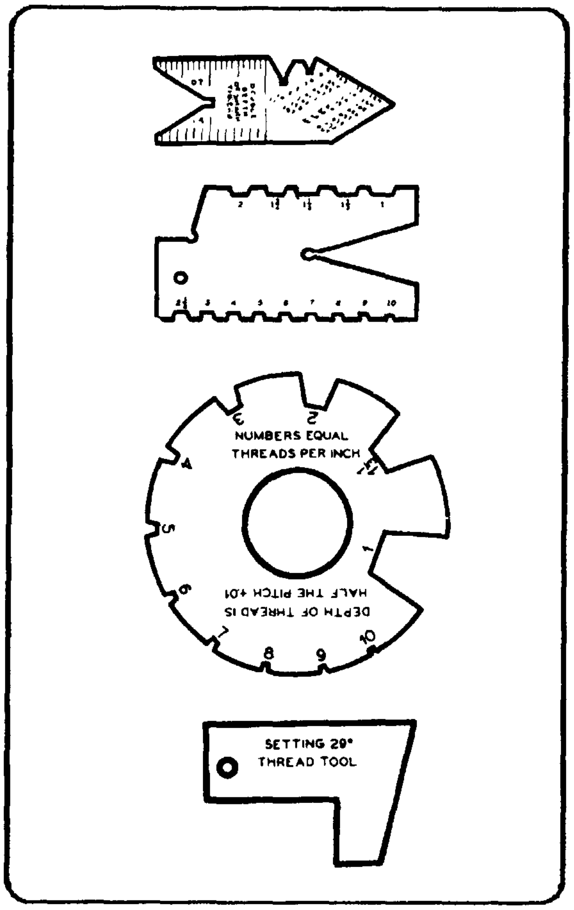


Figure 7-78. Common gages for checking threading tool bits.

For Acme and 29° worm screw threads, the cutter bit must be ground to form a point angle of 29°. Side clearances must be sufficient to prevent rubbing on threads of steep pitch. The end of the bit is then ground to a flat which agrees with the width of the root for the specific pitch being cut. Thread-cutting tool gages (Figure 7-78) are available to simplify the procedure and make computations unnecessary.

To cut square threads, a special thread-cutter bit is required. Before the square thread-cutter bit can be ground, it is necessary to compute the helix angle of the thread to be cut (Figure 7-79). Compute the helix angle by drawing a line equal in length to the thread circumference at its minor diameter (this is accomplished by multiplying the minor diameter by 3.1416 [pi]). Next, draw a line perpendicular to and at one end of the first line, equal in length to the lead of the thread. If the screw is to have a single thread, the lead will be equal to the pitch. Connect the ends of the angle so formed to obtain the helix angle.

The tool bit should be ground to the helix angle. The clearance angles for the sides should be within the helix angle. Note that the sides are also ground in toward the shank to provide additional clearance.

The end of the tool should be ground flat, the flat being equal to one-half the pitch of the thread to produce equal flats and spaces on the threaded part.

When positioning the thread-cutter bit for use, place it exactly on line horizontally with the axis of the workpiece. This is especially important for thread-cutter bits since a slight variation in the vertical position of the bit will change the thread angle being cut.

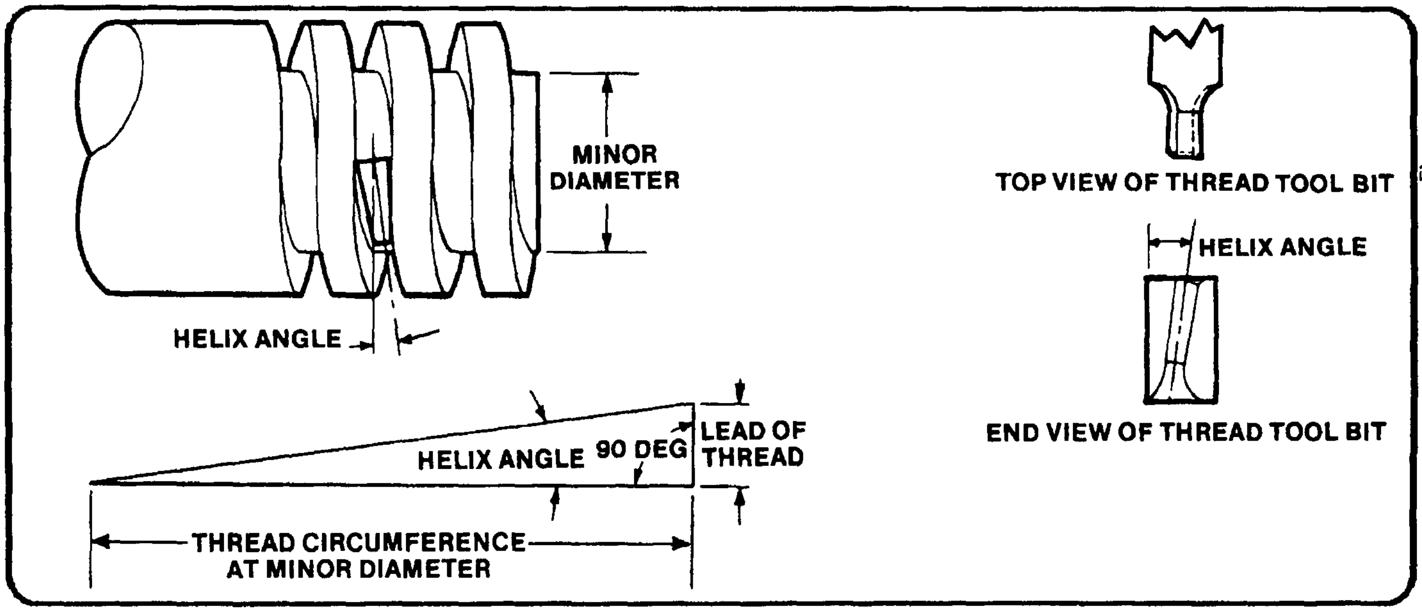


Figure 7-79. Thread tool bit for square threads.

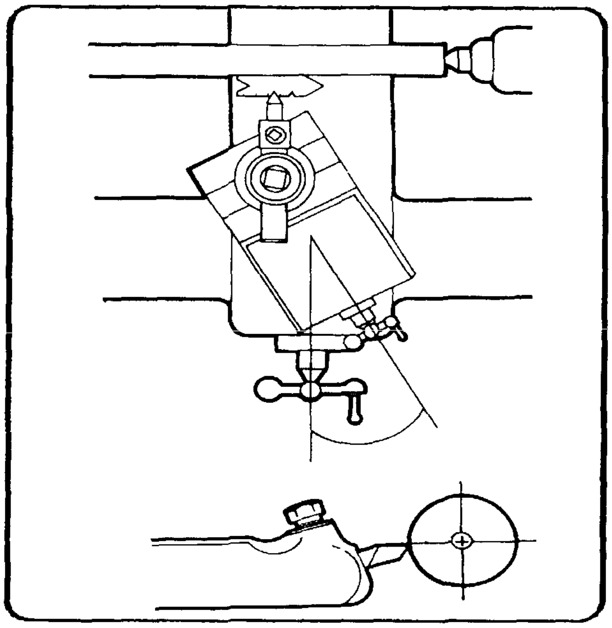


Figure 7-80. Positioning thread cutter bit.

The thread-cutter bit must be positioned so that the centerline of the thread angle ground on the bit is exactly perpendicular to the axis of the workpiece. The easiest way to make this alignment is by use of a center gage. The center gage will permit checking the point angle at the same time as the alignment is being effected. The center gage is placed against the workpiece and the cutter bit is adjusted on the tool post so that its point fits snugly in the 60° angle notch of the center gage (Figure 7-80).

In cutting threads on a lathe, the pitch of the thread or number of threads per inch obtained is determined by the speed ratio of the headstock spindle and the lead screw which drives the carriage. Lathes equipped for thread cutting have gear arrangements for varying the speed of the lead screw. Modern lathes have a quick-change gearbox for varying the lead screw to spindle ratio so that the operator need only follow the instructions on the direction plates of the lathe to set the proper feed to produce the desired number of threads per inch. Once set to a specific number of threads per inch, the spindle speed can be varied depending upon the material being cut and the size of the workpiece without affecting the threads per inch.

The carriage is connected to the lead screw of the lathe for threading operations by engaging the half nut on the carriage apron with the lead screw. A control is available to reverse the direction of the lead screw for left or right-hand threading as desired. Be sure the lead screw turns in the proper direction. Feed the cutter bit from right to left to produce a right-hand thread. Feed the cutter bit from left to right to produce a left-hand thread.

Direction of feed. For cutting standard 60° right-hand threads of the sharp V-type, such as the metric form, the American (National) form, and the Unified form, the tool bit should be moved in at an angle of 29° to the right (Figure 7-8 1), (Set the angle at 29° to the left for left-hand threads). Cutting threads with the compound rest at this angle allows for the left side of the tool bit to do most of the cutting, thus relieving some strain and producing a free curling chip. The direction is controlled by setting the compound rest at the 29° angle before adjusting the cutter bit perpendicular to the workpiece axis. The depth of cut is then controlled by the compound rest feed handle.

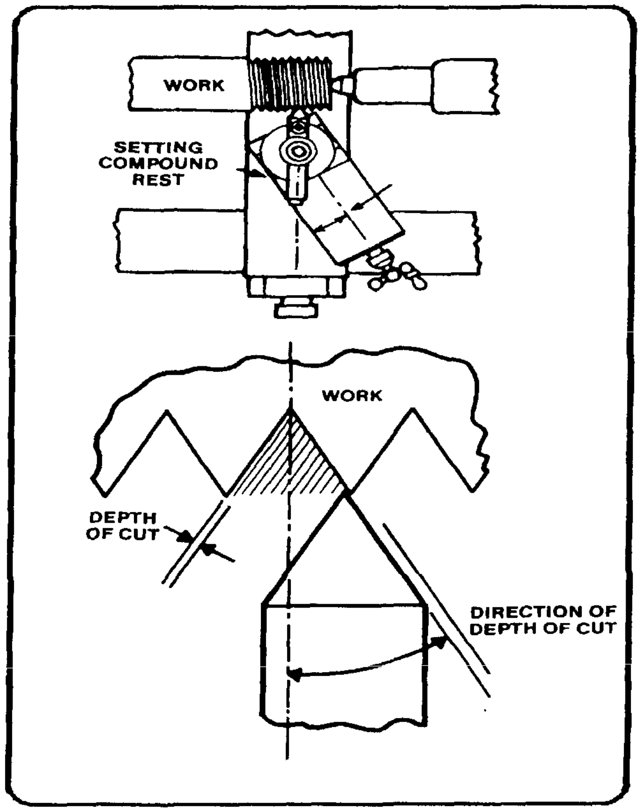


Figure 7-81. External threading setup.

For Acme and 29° worm threads, the compound rest is set at one-half of the included angle (14 1/2°) and is fed in with the compound rest. For square threads, the cutter bit is fed into the workpiece at an angle perpendicular to the workpiece axis.

THREAD CUTTING OPERATIONS

Before cutting threads, turn down the workpiece to the major diameter of the thread to be cut and chamfer the end. Engineering and machinist's handbooks have special tables listing the recommended major and minor diameters for all

thread forms. These tables list a minimum and a maximum major diameter for the external threads, and a minimum and maximum minor diameter for internal threads. Table 7-10 in Appendix A lists the most common screw thread sizes. The difference between the maximum and minimum major diameters varies with different sizes of threads. Coarse threads have a larger difference between the two than fine threads. It is common practice, when machining threads on the lathe, to turn the outside diameter down to the maximum major diameter instead of the minimum major diameter, thus allowing for any error.

The workpiece may be set up in a chuck, in a collet, or between centers. If a long thread is to be cut, a steady rest or other support must be used to help decrease the chance of bending the workpiece. Lathe speed is set for the recommended threading speed (Table 7-2 in Appendix A).

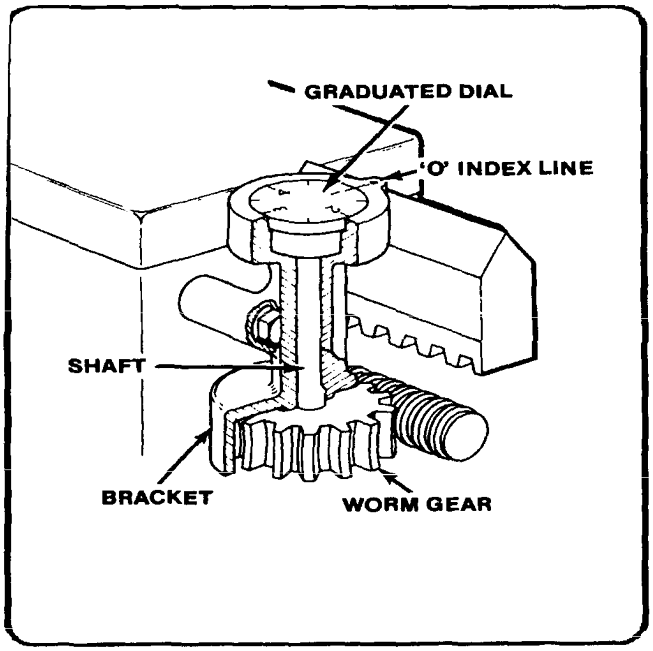


Figure 7-82. Thread chasing dial.

To cut threads, move the threading tool bit into contact with the work and zero the compound rest dial. The threading tool bit must be set at the right end of the work; then, move the tool bit in the first depth of cut by using the graduated collar of the compound rest. Position the carriage half nut lever to engage the half nut to the lead screw in order to start the threading operation. The first cut should be a scratch cut of no more than 0.003 inch so the pitch can be checked. Engaging the half nut with the lead screw causes the carriage to move as the lead screw revolves. Cut the thread by making a series of cuts in which the threading tool follows the original groove for each cut. Use the thread chasing dial, Figure 7-82, to determine when to engage the half nut so that the threading tool will track properly. The dial is attached to the carriage and is driven by means of the lead screw. Follow the directions of the thread chasing dial, Figure 7-83, to determine when to engage the half nut lever.

After making the first pass check for proper pitch of threads by using one of the three methods in Figure 7-84. After each pass of the threading tool bit, the operator must move the threading tool bit out of the threaded groove by backing out the compound rest handle, taking note of the setting. Traverse the carriage back to the start of the thread and move the compound rest dial back to the original setting plus the new depth of cut. At the end of each cut, the half nut lever is usually disengaged and the carriage returned by hand. (The cross slide dial can also be used to move the tool bit in and out, depending on the preference of the operator.)

After cutting the first depth of thread, check for the proper pitch of threads by using one of the three methods in Figure 7-84. If the thread pitch is correct as set in the quick-change gearbox, continue to cut the thread to the required depth. This is determined by measuring the pitch diameter and checking the reference table for-the proper pitch diameter limits for the desired tit.

Some lathes are equipped with a thread chasing stop bolted to the carriage which can be set to regulate the depth of cut for each traverse of the cutter bit or can be set to regulate the total depth of cut of the thread.

When the thread is cut the end must be finished in some way. The most common means of finishing the end is with a specially ground or 45 degree angle chanifer cutting bit. To produce a rounded end, a cutter bit with the desired shape should be specially ground for that purpose.

Metric Thread Cutting Operations

Metric threads, are cut one of two ways by using the lathe, designed and equipped for metric measurement or by using a standard inch lathe and converting its operation to cut metric threads. A metric measurement lathe has a quick-change gear box used to set the proper screw pitch in millimeters. An inch- designed lathe must be converted to cut metric threads by switching gears in the lathe headstock according to the directions supplied with each lathe.

Most lathes come equipped with a set of changeable gears for cutting different, or nonstandard screw threads. Follow the directions in the lathe operator manual for setting the proper metric pitch. (A metric data plate may be attached to the lathe headstock.) Most lathes have the capability of quickly attaching these change gears over the existing gears then realigning the gearing. One change gear in needed for the lead screw gear and one for the spindle, or drive gear.

THREADS PER INCH TO BE CUT	WHEN TO ENGAGE SPLIT NUT
Even Number Of Threads	Engage At Any Graduation On The Dial
Odd Number Of Threads	Engage At Any Main Division
Fractional Number Of Threads	1/2 Threads, E.G. 11 1/2 Engage At Any Other Main Division 1&3, Or 2& 4 Other Fractional Threads Engage At Same Division Every Time
Threads That Are A Multiple of The Number of The Threads per Inch In The Lead Screw	Engage At Any Time That Split Nut Meshes

Figure 7-83. Thread chasing dial instructions.

The metric thread diameter and pitch can be easily measured with a metric measuring tool. If there are no metric measuring tools available, the pitch and diameter must be converted from millimeters to inch measurement, and then a inch micrometer and measuring tools can be used to determine the proper pitch and diameter. Millimeters may be converted to inch measurement either by dividing millimeters by 25.4 inches or multiplying by 0.03937 inches.

For example, a thread with a designation M20 x 2.5 6g/6h is read as follows: the M designates the thread is metric. The 20 designates the major diameter in millimeters. The 2.5 designates the linear pitch in millimeters. The 6g/6h designates that a general purpose fit between nut and bolt is intended. Therefore, to machine this metric thread on a inch designed lathe, convert the outside diameter in millimeters to a decimal fraction of an inch and machine the major diameter to the desired diameter measurement. Convert the linear pitch in millimeters, to threads per inch by dividing the linear pitch of 2.5 by 25.4 to get the threads per inch (10.16 TPI).

Now. a 8-13 TPI thread micrometer can be used to measure the pitch diameter for this metric thread.

To sum up how to convert metric threads to inch measurement:

- Convert major diameter from millimeters to inch measure.
- Convert pitch and pitch diameter to inch measure,
- Set quick change gears according to instructions.

Set up the lathe for thread cutting as in the preceding paragraphs on screw thread cutting, Take a light trial cut and check that the threads are of the correct pitch using a metric screw pitch gage. At the end of this trial cut, and any cut when metric threading, turn off the lathe and back out the tool bit from the workpiece without disengaging the half-nut- lever. Never disengage the lever until the metric thread is cut to the proper pitch diameter, or the tool bit will have to be realigned and set for chasing into the thread.

After backing the tool bit out from the workpiece, traverse the tool bit back to the starting point by reversing the lathe spindle direction while leaving the half-nut lever engaged. If the correct pitch is being cut, continue to machine the thread to the desired depth.

NOTE: If the tool bit needs to be realigned and chased into the thread due to disengagement, of the half-nut lever or having to remove the piece and start again, then the lathe must be reset for threading. Start the lathe, with the tool bit clear of the workpiece engage the lever. Allow the carriage to travel until the tool bit is opposite any portion of the unfinished thread; and then turn off the lathe, leaving the engaged. Now the tool bit can be set back into a thread groove by advancing the cross slide and reference. Restart the lathe, and the tool bit should follow the groove that was previously cut, as long as the half-nut lever stays engaged.

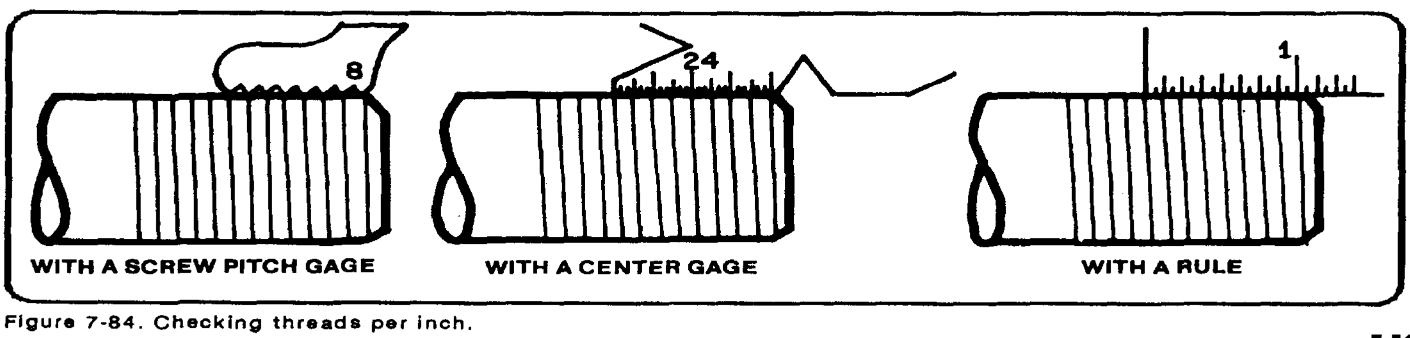


Figure 7-84. Checking threads per inch.

TAPERED SCREW THREADS

Tapered screw threads or pipe threads can be cut on the lathe by setting the tailstock over or by using a taper attachment. Refer to the references for taper per inch and nominal measurements of tapered thread forms. When cutting a tapered thread, the tool bit should be set at right angles to the axis of the work. Do not set the tool bit at a right angle to the taper of the thread. Check the thread tool bit carefully for clearances before cutting since the bit will not be entering the work at right angles to the tapered workpiece surface.

MEASURING EXTERNAL V-SHAPED SCREW THREADS

The fit of the thread is determined by its pitch diameter. The pitch diameter is the diameter of the thread at an imaginary point on the thread where the width of the space and the width of the thread are equal. The fact that the mating parts bear on this point or angle of the thread, and not on the top of it, makes the pitch diameter an important dimension to use in measuring screw threads.

The thread micrometer (Figure 7-85) is an instrument used to gage the thread on the pitch diameter. The anvil is V-Shaped to fit over the V-thread. The spindle, or movable point, is cone-shaped (pointed to a V) to fit between the threads. Since the anvil and spindle both contact the sides of the threads, the pitch diameter is gaged and the reading is given on the sleeve and spindle where it can be read by the operator.

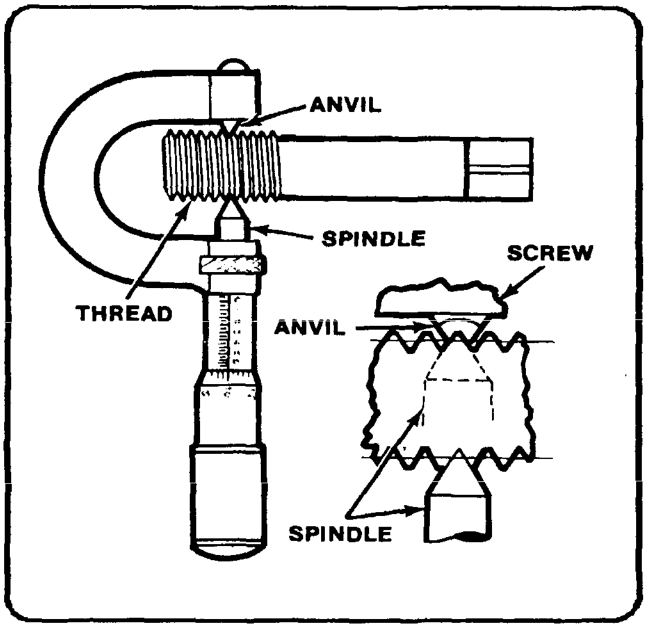


Figure 7-85. Thread micromenter.

Thread micrometers are marked on the frame to specify the pitch diameters which the micrometer is used to measure. One will be marked, for instance, to measure from 8 to 13 threads per inch, while others are marked 14 to 20, 22 to 30, or 32 to 40; metric thread micrometers are also available in different sizes.

The procedure in checking the thread is first to select the proper micrometer, then calculate or select from a table of threads the correct pitch diameter of the screw. Lastly, fit the thread into the micrometer and take the reading.

The 3-wire method is another method of measuring the pitch diameter for American National (60 degree) and Unified threads. It is considered the "best" method for extremely accurate measurement. Page A-28 in Appendix A shows three wires of correct diameter placed in threads with the micrometer measuring over them. The pitch diameter can be found by subtracting the wire constant from the measured distance over the wires. It can be readily seen that this method is dependent on the use of the "'best'" wire for the pitch of the thread. The "best" wire is the size of wire which touches the thread at the middle of the sloping sides. in other words, at the pitch diameter. A formula by which the proper size wire may be found is as follows: Divide the constant 0.57735 by the number of threads per inch to cut. If. for example, 8 threads per inch have been cut, we would calculate 0.577358 = 0.072. The diameter of wire to use for measuring an 8-pitch thread is 0.072.

The wires used in the three-wire method should be hardened and lapped steel wires. they, should be three times as accurate as the accuracy desired in measurement of the threads. The Bureau of Standards has specified an accuracy of 0.0002 inch. The suggested procedure for measuring threads is as follows:

After the three wires of equal diameter have been selected by using the above formula, they are positioned in the thread grooves as shown on page A-28 in Appendix A. The anvil and spindle of an ordinary micrometer are then placed against the three wires and the reading is taken. To determine what the reading of the micrometer should be if a thread is the correct finish size. use the following formula (for measuring Unified National Coarse threads): add three times the diameter of the wire to the diameter of the screw; from the sum, subtract the quotient obtained by dividing the constant 1.5155 by the number of threads per inch. Written concisely, the formula is:

$$m = (D + 3W) - \frac{1.5155}{n}$$

Where m = micrometer measurement over wires,
D = diameter of the thread,
n = number of threads per inch,
W = diameter of wire used

Example: Determine m (measurement over wires) for 1/2 inch, 12-pitch UNC thread. We would proceed to solve as follows:

where W = 0.04811 inch
D = 0.500 inch
n=12

$$\text{Then } m = (0.500 + 0.14433) - \frac{15155}{12}$$

$$m = (0.500 + 0.14433) - 0.1263$$

m = 0.51803 inch (micrometer measurement)

When measuring a Unified National Fine thread, the same method and formula are used. Too much pressure should not be applied when measuring over wires.

Metric threads can also be checked by using the three-wire method by using different numerical values in the formula. Three-wire threads of metric dimensions must have a 60° angle for this method.

M= PD+CPD=M-C

M = measurement over the wires
PD = pitch diameter
C = N constant (This is found in Table 7-11 in Appendix A)

The "best" wire size can be found by converting from inch to metric, or by using Table 7-11 in Appendix A.

An optical comparator must be used to check the threads if the tolerance desired is less than 0.001 inch (0.02 mm). This type of thread measurement is normally used in industrial shops doing production work.

CUTTING INTERNAL THREADS

Internal threads are cut into nuts and castings in the same general manner as external threads. If a hand tap is not available to cut the internal threads, they must be machined on the lathe.

An internal threading operation will usually follow a boring and drilling operation, thus the machine operator must know drilling and boring procedures before attempting to cut internal threads. The same holder used for boring can be used to hold the tool bit for cutting internal threads. Lathe speed is the same as the speed for external thread cutting.

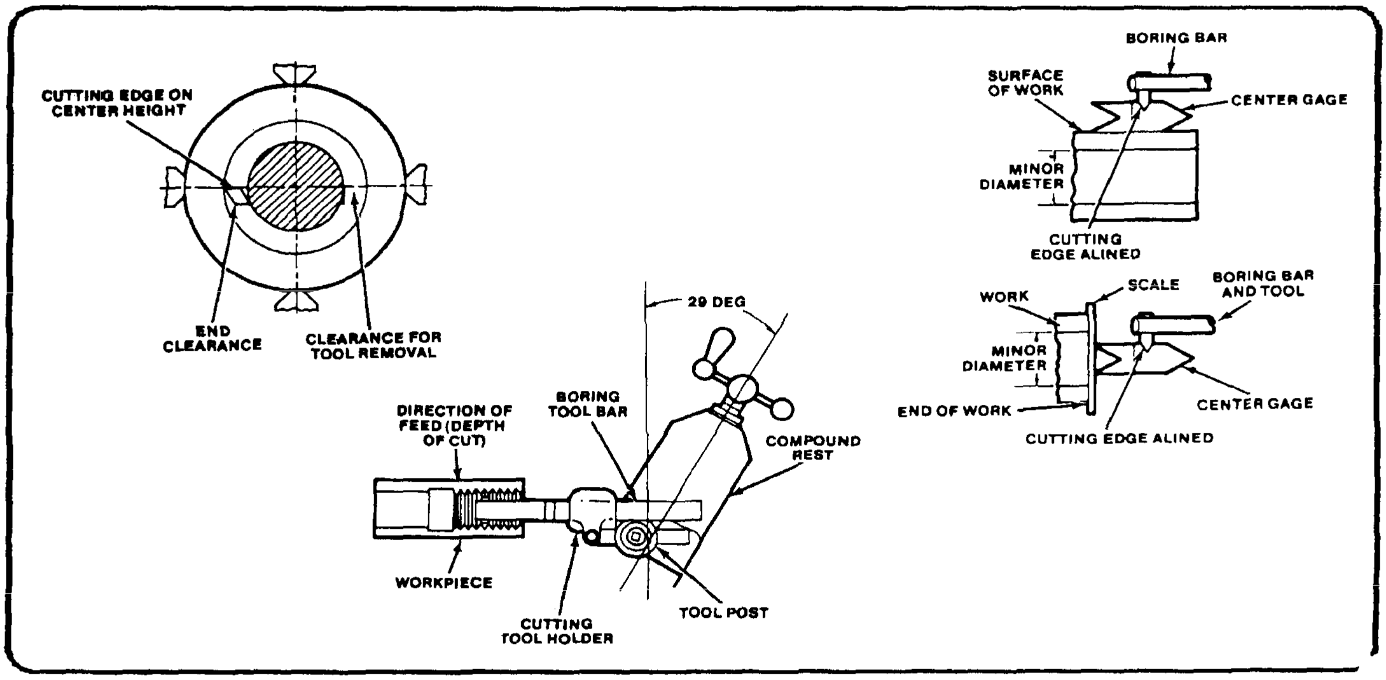


Figure 7-86. Internal thread cutting.

To prevent rubbing, the clearance of the cutter bit shank and boring tool bar must be greater for threading than for straight boring because of the necessity of moving the bit clear of the threads when returning the bit to the right after each cut.

The compound rest should be set at a 29° angle to the saddle so that the cutter bit will feed after each cut toward the operator and to his left.

Although the setup shown in Figure 7-86 would be impractical on extremely large lathes, it allows a degree of safety on common sized machines by having the compound ball crank positioned away from any work holding device that would be in use on the lathe, eliminating the possibility of the operator's hands or the compound rest contacting the revolving spindle and work holding devices.

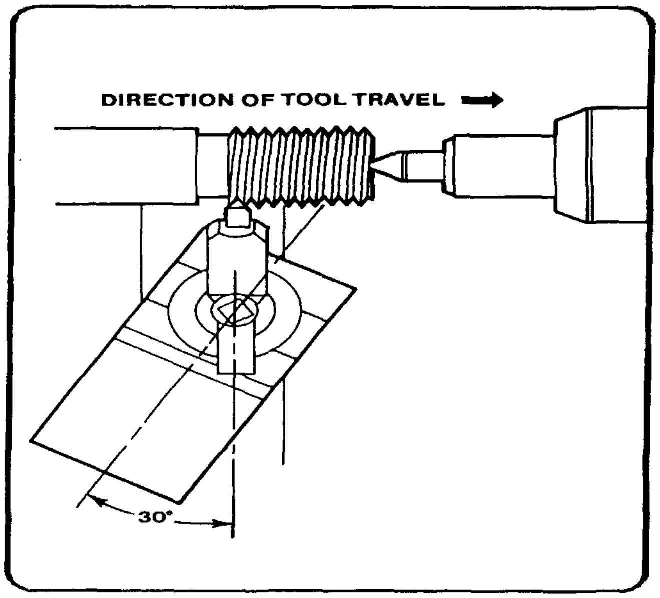


Figure 7-87. Left-hand threading.

Cutting 60° left-hand threads. A left-hand thread is used for certain applications where a right-hand thread would not be practicable, such as on the left side of a grinder where the nut may loosen due to the rotation of the spindle. Left-hand threads are cut in the same manner as right hand threads, with a few changes. Set the feed direction lever so that the carriage feeds to the right, which will mean that the lead screw revolves opposite the direction used for right-hand threading. Set the compound rest 29° to the left of perpendicular. Cut a groove at the left end of the threaded section, thus providing clearance for starting the cutting tool (see Figure 7-87). Cut from left to right until the proper pitch dimension is achieved.

CUTTING EXTERNAL ACME THREADS

The first step is to grind a threading tool to conform to the 29° included angle of the thread. The tool is first ground to a point, with the sides of the tool forming the 29° included angle (Figure 7-88). This angle can be checked by placing the tool in the slot at the right end of the Acme thread gage.

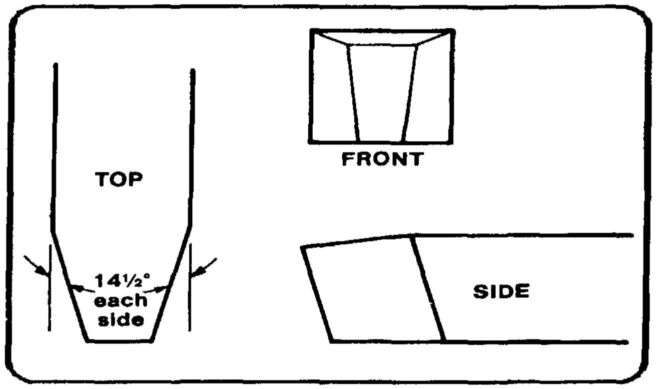


Figure 7-88. Acme thread cutting tool bit.

If a gage is not available, the width of the tool bit point may be calculated by the formula:

Width of point= 0.3707P -0.0052 inch

Where P = Number of threads per inch

Be sure to grind this tool with sufficient side clearance so that it will cut. Depending upon the number of threads per inch to be cut, the point of the tool is ground flat to fit into the slot on the Acme thread gage that is marked with the number of threads per inch the tool is to cut. The size of the flat on the tool point will vary depending upon the thread per inch to be machined.

After grinding the tool, set the compound rest to one-half the included angle of the thread (14 1/2°) to the right of the vertical centerline of the machine (Figure 7-89). Mount the tool in the holder or tool post so that the top of the tool is on the axis or center line of the workpiece. The tool is set square to the work, using the Acme thread gage. This thread is cut using the compound feed. The depth to which you feed the compound rest to obtain total thread depth is determined by the formula given and illustrated in Table 7-9 in Appendix A. The remainder of the Acme thread-cutting operation is the same as the V-threading operation previously described. The compound rest should be fed into the work only 0.002 inch to 0.003 inch per cut until the desired depth of thread is obtained.

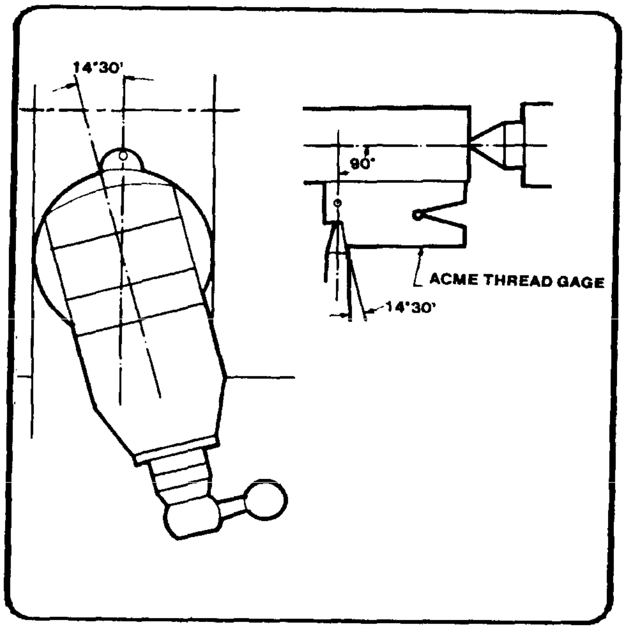


Figure 7-89. Acme and 29° worm thread setup.

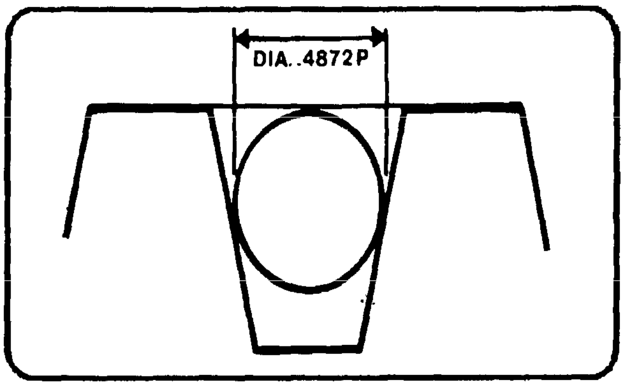


Figure 7-90. Using one wire to measure an Acme

The formulas used to calculate Acme thread depth are in Table 7-9 in Appendix A. The single wire method can be used to measure the accuracy of the thread (Figure 7-90). A single wire or pin of the correct diameter is placed in the threaded groove and measured with a micrometer. The thread is the correct size when the micrometer reading over the wire is the same as the major diameter of the thread and the wire is placed tightly into the thread groove. The diameter of the wire to be used can be calculated by using this formula:

Wire diameter = 0.4872 x pitch

Thus, if 6 threads per inch are being cut, the wire size would be:

0.4872 x 1/6= 0.081 inch

Cutting the 29° worm screw thread (Brown and Sharpe). The tool bit used to cut 29° worm screw threads will be similar to the Acme threading tool, but slightly longer with a different tip. Use Table 7-9 in Appendix A to calculate the length of the tool bit and tip width. The cutting is done just like cutting an Acme thread.

CUTTING SQUARE THREADS

Because of their design and strength, square threads are used for vise screws, jackscrews, and other devices where maximum transmission of power is needed. All surfaces of the square thread form are square with each other, and the sides are perpendicular to the center axis of the threaded part. The depth, the width of the crest, and root are of equal dimensions. Because the contact areas are relatively small and do not wedge together, friction between matching threads is reduced to a minimum. This fact explains why square threads are used for power transmission.

Before the square thread cutting tool can be ground, it is necessary first to determine the helix angle of the thread. The sides of the tool for cutting the square thread should conform with the helix angle of the thread (Figure 7-79).

For cutting the thread, the cutting edge of the tool should be ground to a width exactly one-half that of the pitch. For cutting the nut, it should be from 0.001 to 0.003 of an inch larger to permit a free fit of the nut on the screw.

The cutting of the square thread form presents some difficulty. Although it is square, this thread, like any other, progresses in the form of a helix, and thus assumes a slight twist. Some operators prefer to produce this thread in two cuts, the first with a narrow tool to the full depth and the second with a tool ground to size. This procedure relieves cutting pressure on the tool nose and may prevent springing the work. The cutting operation for square threads differs from cutting threads previously explained in that the compound rest is set parallel to the axis of the workpiece and feeding is done only with the cross feed. The cross feed is fed only 0.002 inch or 0.003 inch per cut. The finish depth of the thread is determined by the formula.

Depth = 1/2P

The width of the tool point is determined by this formula also and will depend upon the number of threads per inch to be machined. It is measured with a micrometer, as square thread gages are not available.

SPECIAL OPERATIONS ON THE LATHE

KNURLING ON THE LATHE

Knurling is a process of impressing a diamond shaped or straight line pattern into the surface of a workpiece by using specially shaped hardened metal wheels to improve its appearance and to provide a better gripping surface. Straight knurling is often used to increase the workpiece diameter when a press fit is required between two parts.

Holding Devices for Knurling

The setup for knurling can be made between centers or mounted in a solid chuck. Never attempt to knurl by holding the work in a rubber or metal collet chuck, since the great pressures of knurling could damage these devices. It is important to support the work while knurling. If mounting the work between centers, make the center holes as large as possible to allow for the strongest hold. If using a chuck to hold the work, use the tailstock center to support the end of the work. If doing a long knurl, use a steady rest to support the work and keep the piece from springing away from the tool.

Knurling Tools

The knurling tool (Figure 7-10) can be designed differently, but all accomplish the same operation. Two common types of knurling tools are the knuckle joint and revolving head type of knurling tools. The knuckle joint type is equipped with a single pair of rollers that revolve with the work as it is being knurled. The revolving head type of tool is fitted with three pairs of rollers so that the pitch can be changed to a different knurl without having to change the setup. There are two knurl patterns, diamond and straight.

There are three pitches of rollers, coarse, medium, and tine (Figure 7-9l).

The diamond is the most common pattern and the medium pitch is used most often. The coarse pitch is used for large-diameter work; the fine pitch is used for small-diameter work.

Knurling

The knurling operation is started by determining the location and length of the knurl, and then setting the machine for knurling. A slow speed is needed with a medium feed. Commonly, the speed is set to 60 to 80 RPM, while the feed is best from 0.015 to 0.030 inch per revolution of the spindle. The knurling tool must be set in the tool post with the axis of the knurling head at center height and the face of the knurls parallel with the work surface. Check that the rollers move freely and are in good cutting condition; then oil the knurling tool cutting wheels where they contact the workpiece. Bring the cutting wheels (rollers) up to the surface of the work with approximately 1/2 of the face of the roller in contact with the work.

If the face of the roller is placed in this manner, the initial pressure that is required to start the knurl will be lessened and the knurl may cut smoother. Apply oil generously over the area to be knurled. Start the lathe while forcing the knurls into the work about 0.010 inch. As the impression starts to form, engage the carriage feed lever (Figure 7-92). Observe the knurl for a few revolutions and shut off the machine. Check to see that the knurl is tracking properly, and that it is not on a "double track" (Figure 7-93).

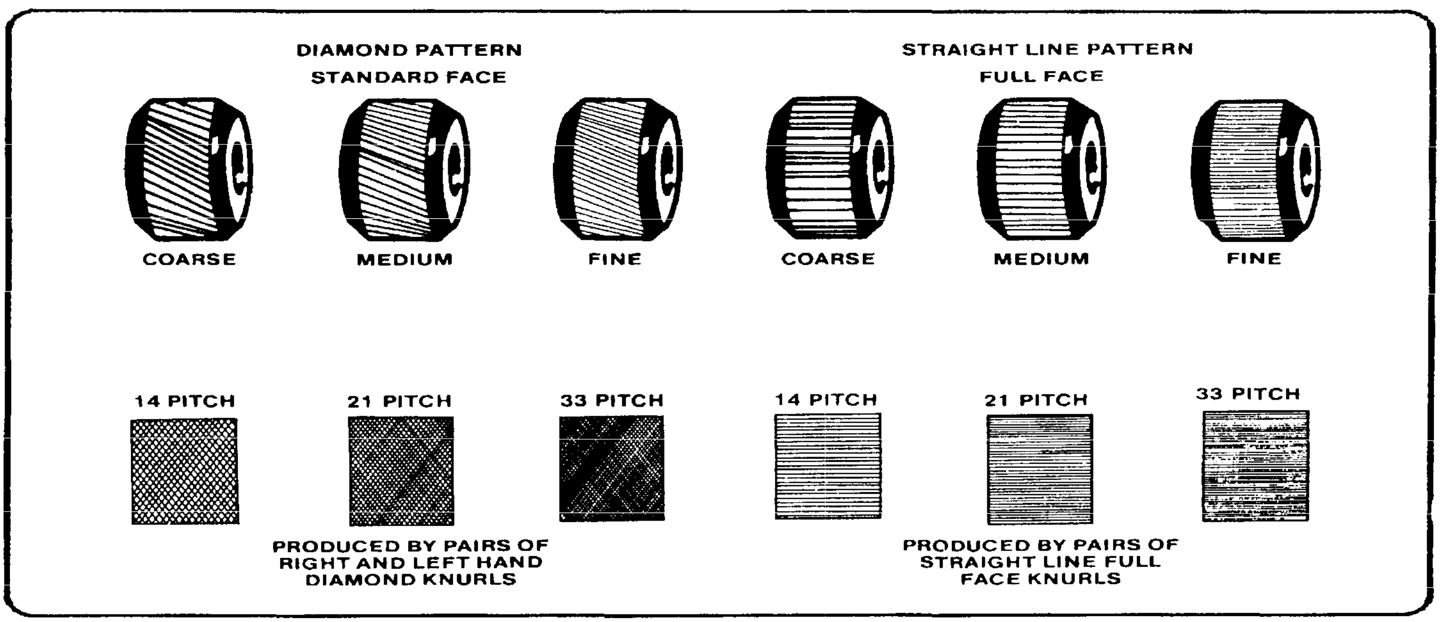


Figure 7-91. Knurling patterns and pitches.

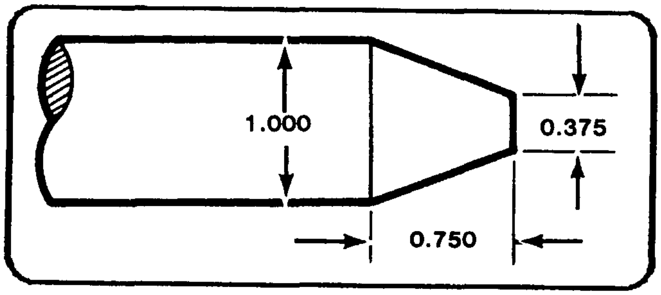


Figure 7-92. Starting the knurl.

rings on the work surface (Figure 7-94). Check the operation to ensure that the knurling tool is not forcing the work from the center hole. Keep the work and knurling tool well oiled during the operation. Never allow a brush or rag to come between the rollers and the work or the knurl will be ruined.

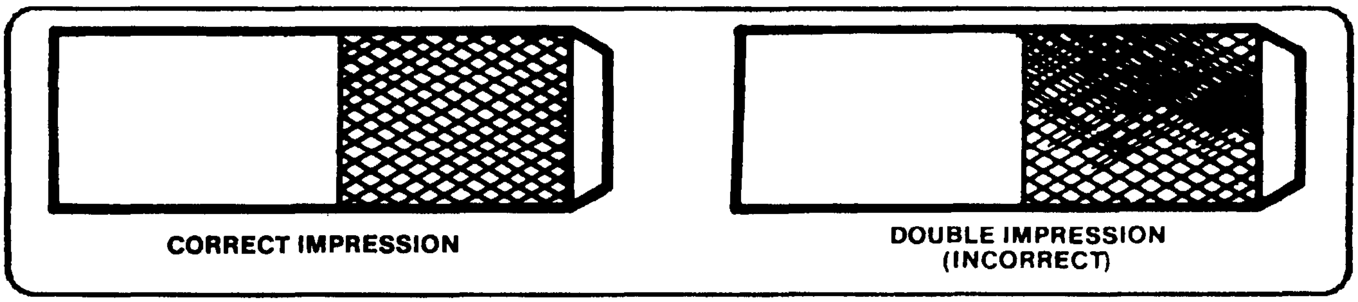


Figure 7-93. Correct and incorrect knurls

Reset the tool if needed; otherwise, move the carriage and tool back to the starting point and lightly bring the tool back into the previously knurled portion. The rollers will align themselves with the knurled impressions. Force the knurling tool into the work to a depth of about 1/64 inch and simultaneously engage the carriage to feed toward the headstock. Observe the knurling action and allow the tool to knurl to within 1/32 inch of the desired end of cut, and disengage the feed. Hand feed to the point where only one-half of the knurling wheel is off the work, change the feed direction toward the tailstock and force the tool deeper into the work.

Engage the carriage feed and cut back to the starting point. Stop the lathe and check the knurl for completeness. Never allow the knurling tool to feed entirely off the end of the work, or it could cause damage to the work or lathe centers. The knurl is complete when the diamond shape (or straight knurl) is fully developed. Excessive knurling after the knurl has formed will wear off the full knurl and ruin the work diameter. Move the tool away from the work as the centers. The knurl is complete when the diamond shape (or work revolves and shut off the lathe. Clean the knurl with a brush and then remove any burrs with a file.

Special Knurling Precautions

Never stop the carriage while the tool is in contact with the work and the work is still revolving as this will cause wear

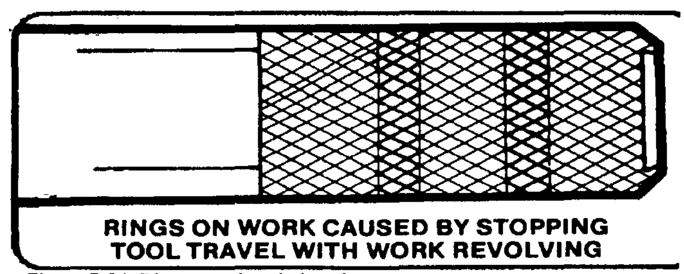


Figure 7-94. Rings on a knurled surface

DRILLING WITH THE LATHE

Frequently, holes will need to be drilled using the lathe before other internal operations can be completed, such as boring, reaming, and tapping. Although the lathe is not a drilling machine, time and effort are saved by using the lathe for drilling operations instead of changing the work to another machine. Before drilling the end of a workpiece on the lathe, the end to be drilled must be spotted (center- punched) and then center- drilled so that the drill will start properly and be correctly aligned. The headstock and tailstock spindles should be aligned for all drilling, reaming, and spindles should be aligned for drilling, reaming, and tapping operations in order to produce a true hole and avoid damage to the work and the lathe. The purpose for which the hole is to be drilled will determine the proper size drill to use. That is, the drill size must allow sufficient material for tapping, reaming, and boring if such operations are to follow.

The correct drilling 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 recommended drilling speeds for various materials, such as Table 4-2 in Appendix A.

Supporting drills in the tailstock

Methods of supporting the twist drill in the tailstock can vary (Figure 7-95). Straight shank drills are usually held in a drill chuck, which is placed in the taper socket of the tailstock spindle. Combination drill and countersinks (center drills), counterbores, reamers, taps, and other small shank cutters can also be supported in this way.

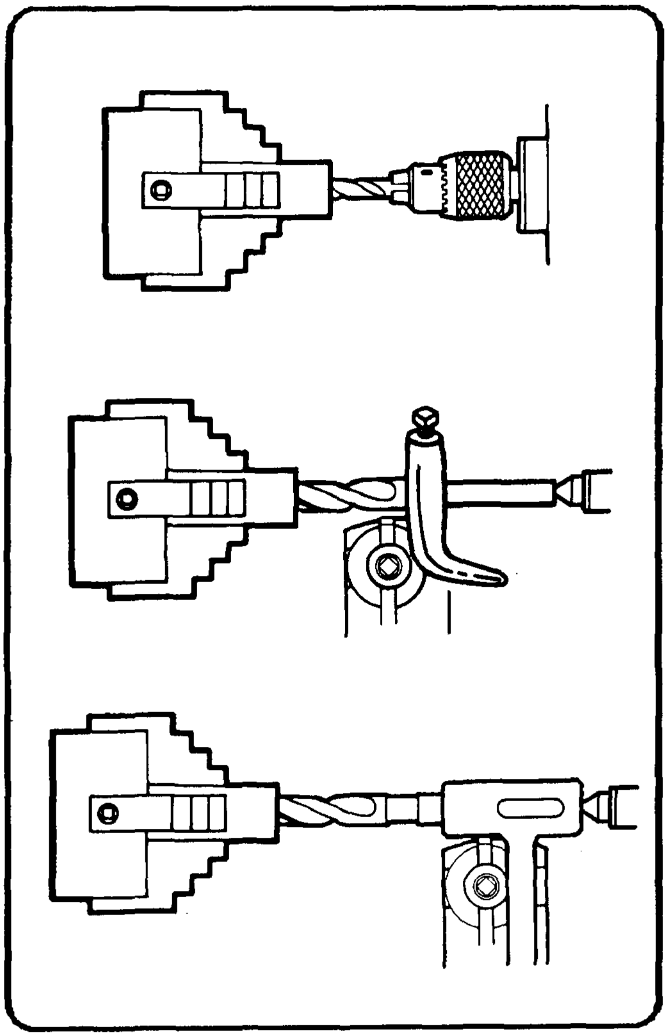


Figure 7-95- Set ups for drilling with the lathe.

Tapered-shank twist drills may be held directly in the tailstock tapered spindle as long as a good fit exists. If the drill shank is not the correct size, then a drill socket or sleeve may be used in the tailstock spindle.

A twist drill holder is used to support large twist drills with the tailstock center. The drill is inserted into the holder and the tailstock center is placed in the center hole which is located at the rear of the drill holder. The holder will rest on the cross slide or compound rest and must be supported by hand until it is held secure by pressure between the tailstock and headstock. When using this method, never withdraw or loosen the tailstock spindle while the lathe is rotating or the workpiece can be thrown out at the operator. Always stop the machine before attempting to withdraw the twist drill.

Another method of supporting a large twist drill in the tailstock is to fasten a lathe dog to the drill shank and support the rear of the drill with the tailstock center in the center hole in the tang of the drill.

Supporting Drills in the Headstock

The drill can also be held and rotated in the headstock with the work held stationary against the tailstock. Straight shank twist drills are supported in the headstock by a drill chuck or collet which is mounted in the headstock spindle. A universal or independent jaw chuck can also be used to hold and turn twist drills if a headstock drill chuck is not available. Tapered shank twist drills can be mounted in the headstock by using a special adapter, such as a sleeve with an internal taper to hold the tapered drill, while the outside of the sleeve is made to fit into the headstock spindle.

Mounting Work for Drilling

If the work is to be rotated and the twist drill will be fed into the end of the work, the work should be mounted in a chuck, on a faceplate, or in a collet. The center of the hole to be drilled should be accurately marked and punched as described for drilling setups.

Always start holes by using a center drill, since this method will be the most accurate and the most efficient. Center-drill by rotating the spindle at computed drill speed and gently bringing the point of the center drill into the end of the work until the proper depth is reached.

If the twist drill is to be rotated by the headstock spindle and the workpiece is to be supported by a V-center mounted in the tailstock, the work should be carefully positioned by hand and

the drill moved lightly into contact with the workpiece before starting the lathe. The workpiece must be well supported during drilling operations to prevent the work from being thrown from the lathe or rotating with the drill.

Drilling Operations

To start the drilling operation, compute the correct RPM for the drill and set the spindle speed accordingly. Ensure the tailstock is clamped down on the lathe ways. The feed is controlled by turning the tailstock handwheel. The graduations on the tailstock spindle are used to determine the depth of cut.

If a large twist drill is used, it should be proceeded by a pilot drill, the diameter of which should be wider than the larger drills web.

Use a suitable cutting fluid while drilling (Table 4-3 in Appendix A). Always withdraw the drill and brush out the chips before attempting to check the depth of the hole. If the drill is wobbling and wiggling in the hole, use a tool holder turned backwards (Figure 7-96) to steady the drill. Always use a drill that is properly ground for the material to be drilled. Use care when feeding the drill into the work to avoid breaking the drill off in the work. The drill should never be removed from the work while the spindle is turning because the drill could be pulled off the tailstock spindle and cause injury or damage.

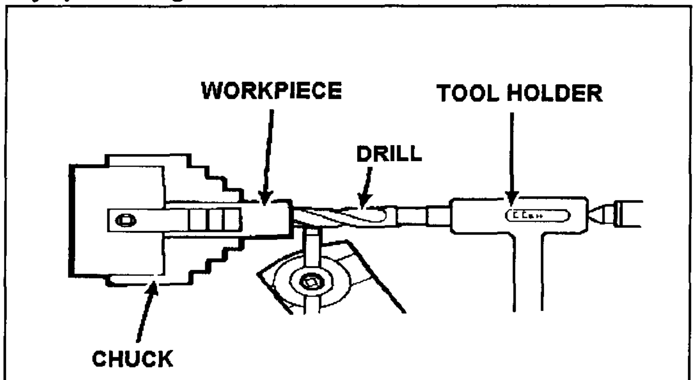


Figure 7-96. Steadying the drill.

BORING WITH THE LATHE

Boring is the enlarging and truing of a hole by removing material from internal surfaces with a single-point cutter bit. On the lathe, boring is accomplished in either of these two methods:

- Mounting the holder and boring tool bar with cutter bit on the tool post and revolving the workpiece.
- Mounting the workpiece in a fixed position to the carriage and revolving the boring tool bar and cutter bit in a chuck attached to the headstock spindle. (This is a special process and not used in most machine shops).

Mounting Workpiece for Boring

The workpiece may be supported in a chuck or fastened to a faceplate for boring operations depending upon of the material to be machined. When boring is to be performed on the ends of long stock, the workpiece is mounted in a chuck and a steady rest is used to support the right end near the cutter bit. Some boring operations require the use of special chuck-mounted mandrels to hold workplaces that cannot be successfully mounted otherwise.

Purpose for Boring

Boring is necessary in many cases to produce accurate holes. Drilled holes are seldom straight due to imperfections in the material which cause drills to move out of alignment. Therefore, where accuracy is important, drilled holes are usually made undersize and then bored or reamed to the proper dimensions. Boring is also useful in truing large holes in flat material. In this case, the hole is cut undersize using a bandsaw or trepanning tool and is trued to proper dimensions by boring.

Boring Cutter Bit Setup

The cutter bit used for boring is similar to that used for external turning on the lathe. The bit is usually held in a **soft** or semisoft bar called a boring tool bar. The boring tool bar (Figure 7-11) is supported by a cutting tool holder which fits into the lathe tool post.

Boring tool bars are supplied in several types and sizes for holding different cutter bits. The bit is supported in the boring tool bar at a 90°, 30°, or 45° angle, depending upon the nature of the workpiece being bored. Most general boring is accomplished with a 90° cutter bit. The bit is mounted at a 30° or 45° angle to the axis of the boring tool bar when it is necessary to cut up to the bottom of a hole or finish the side of an internal shoulder. It is desirable that the boring tool bar be as large as possible without interfering with the walls of the hole. The cutter bit should not extend far beyond the boring tool bar and the bit securely in the bar, yet not have the shank-end protrude far from the bar.

The cutter bits used for boring are shaped like left-hand turning and facing cutter bits. Greater attention must be given to the end clearance angle and the back rake angle because of the curvature of the hole (Figure 7-97).

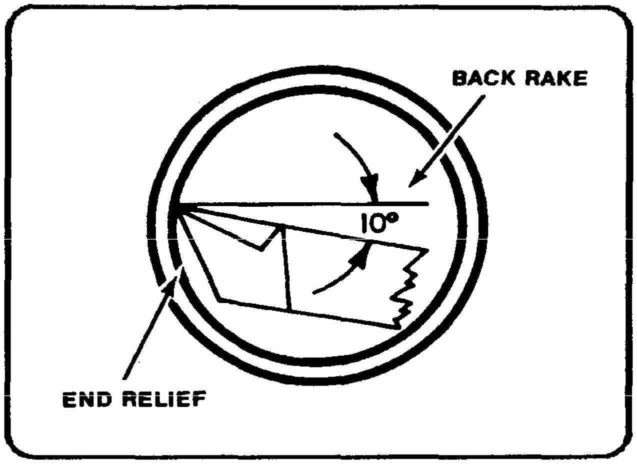


Figure 7-97. Proper position of boring cutter bit.

The boring tool bar should be clamped as close to the holder and tool post as possible considering the depth of boring to be done. The bar will have a tendency to spring away from the workpiece if the bar overhangs the tool post too far. If deep boring is to be performed, it will be necessary that the bar be as thick as possible to counteract this springing tendency.

Straight Boring Operation

The cutter bit is positioned for straight boring operations with its cutting edge set slightly above center. Depending on the rigidity of the setup, the boring tool will have a tendency to spring downward as pressure is applied to the cutting edge. By setting the cutter slightly above center, compensation has been made for the downward spring and the cutter will actually be positioned on the exact center of the workpiece during machining operations (Figure 7-98). The cutting edge faces forward for most operations so the lathe can turn in its normal counterclockwise direction. If it becomes necessary to position the cutter bit against the rear wall of the hole for a special operation, a right-hand turning cutter bit is used and the spindle rotation is reversed.

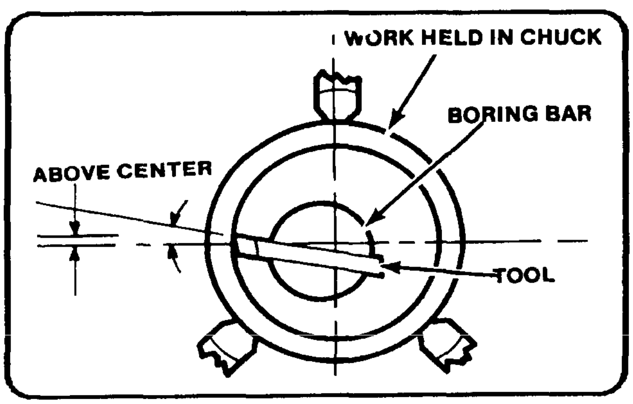


Figure 7-98. Boring cutter bit above center.

Position the cutter bit so that the cutting edge is immediately to the right of the workpiece and clears the wall of the hole by about 1/1 6 inch. Traverse the carriage by hand, without starting the lathe, to move the cutter bit and boring tool bar into the hole to the depth of the intended boring and out again to determine whether there is sufficient clearance to prevent the back of the cutter bit and the boring tool bar from rubbing the inside of the hole. When the clearance is satisfactory, position the cutter bit to the right of the workpiece ready for the first cut. Use the micrometer carriage stop to control the depth of tool travel.

The same speeds recommended for straight turning should be used for straight boring. Feeds for boring should be considerably smaller than feeds used for straight turning because there is less rigidity in the setup. Decrease the depth of cut for each pass of the tool bit for the same reason. It is often advisable to feed the cutter bit into the hole to the desired depth and then reverse the feed and let the cutter bit move out of the hole without changing the depth of feed. It is also good practice to take a free cut every several passes to help eliminate bell mouthing of the workpiece. This practice will correct any irregularities caused by the bit or boring tool bar springing because of the pressure applied to the bit.

TAPPING AND HAND DIE THREADING

The lathe can be used as a device to hold and align a tap or hand die to cut internal or external threads quickly for threads that do not require a high degree of accuracy or a fine finish. More information on taps and dies can be found in TM 9-243.

Hand Tapping on the Lathe

Tapping can be done on the lathe by power or by hand. Regardless of the method, the hole must be drilled with the proper sized tap drill and chamfered at the end. The shank end of the tap is supported by the tailstock center. A slight pressure is maintained against the tap to keep its center hole on the center and to help the cutting teeth of the tap engage the work (Figure 7-99).

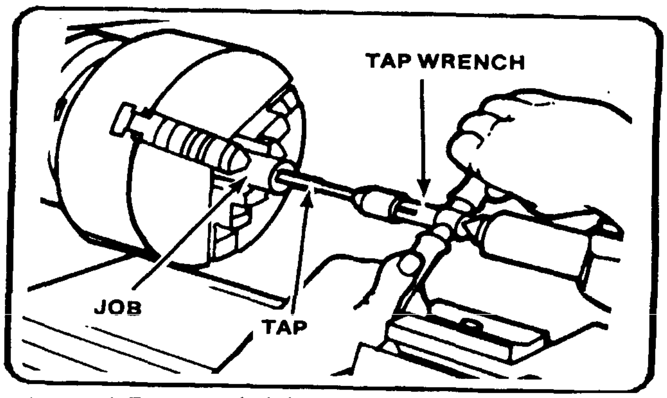


Figure 7-99. Tapping on the lathe.

The work will rotate when tapping using lathe power. Use a very slow spindle speed (10 to 30 RPM) and plenty of cutting fluid or coolant. Install a tap and reamer wrench on the end of the tap to keep it from turning. Support the wrench on the compound rest. Power is not recommended for taps under 1/2 inch in diameter or when tapping steel. Ensure that the tap wrench handle contacts the compound rest before engaging power or the end of the handle will whip around and could crush a finger or cause other injury or damage. Do not attempt to start the tap into the hole with the work revolving. Always keep the tap snug in the center hole to prevent the tap from coming out of alignment and ruining the threads.

The setup for hand tapping in a lathe is similar to that used in power tapping. The headstock chuck is held steady and not rotated. The tap is turned by using an adjustable wrench. Lock the lathe gears so that the headstock will not move when using a large tap. Back off the tap frequently when tapping to break the chips and allow for a clean thread.

Hand Die Threading on the Lathe

Die threading on a lathe is very similar to tapping on a lathe, except that the die is aligned perpendicular to the work axis by pressure exerted against the back surface of the die. This pressure can be exerted by means of a drill pad, by using the tailstock spindle, or by using the head of the drill chuck for small dies. Die threading can be done using power or by hand, using the same procedures as tapping. Power can be used to remove the die from the work if the die stock handle is swung to the opposite side and low reverse power is used. It is difficult to cut very coarse threads with a die because of the great amount of force needed to turn the die. It is advisable to open up the die to its full width, rough-cut the threads, and then close up the die and go over the threads for a finished size. Always use a lubricant or coolant for this operation.

REAMING ON THE LATHE

Reamers are used to finish drilled holes or bores quickly and accurately to a specified diameter. When a hole is to be reamed, it must first be drilled or bored to within 0.004 to 0.012 inch of the finished size since the reamer is not designed to remove much material.

Reaming with a Machine Reamer

The hole to be reamed with a machine reamer must be drilled or bored to within 0.012 inch of the finished size so that the machine reamer will only have to remove the cutter bit marks.

The workpiece is mounted in a chuck at the headstock spindle and the reamer is supported by the tailstock in one of the methods described for holding a twist drill in the tailstock.

The lathe speed for machine reaming should be approximately one-half that used for drilling.

Reaming with a Hand Reamer

The hole to be reamed by hand must be within 0.005 inch of the required finished size.

The workpiece is mounted to the headstock spindle in a chuck and the headstock spindle is locked after the piece is accurately setup The hand reamer is mounted in an adjustable tap and reamer wrench and supported with the tailstock center. As the wrench is revolved by hand, the hand reamer is fed into the hole simultaneously by turning the tailstock handwheel.

The reamer should be withdrawn from the hole carefully, turning it in the same direction as when reaming. Never turn a reamer backward. See Table 4-3 in Appendix A for the proper cutting fluid for reaming. Never use power with a hand reamer or the work could be ruined.

FILING AND POLISHING ON THE LATHE

Filing and polishing are performed on the lathe to remove tool marks, reduce the dimension slightly, or improve the finish.

Filing on the Lathe

Mill files are generally considered best for lathe filing. The bastard cut mill type hand file is used for roughing and the second cut mill-type hand file for the finer class of work. Other types such as the round, half-round, and flat hand files may also be used for finishing irregular shaped workplaces. Never use a file without a handle.

For filing ferrous metals, the lathe spindle speed should be four or five times greater than the rough turning speed. For filing nonferrous metals, the lathe spindle speed should be only two or three times greater than the roughing speed. Too slow a speed may cause the workpiece to be filed out of round, while too high a speed will cause the file to slide over the workpiece, dulling the file and glazing the piece.

NOTE: When filing, file left-handed if at all possible to avoid placing your arm over the revolving chuck or lathe dog.

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 a shearing action (Figure 7-100). The direction of stroke and angle should never be the opposite, as this will cause chatter marks on the piece. The file should be passed slowly over the workpiece so that the piece will have made several revolutions before the stroke is completed. 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 workpiece, the file must be cleaned frequently to avoid scratching.

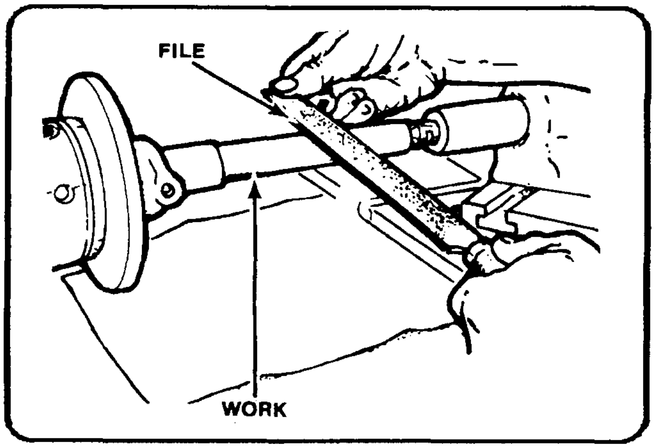


Figure 7-100. Filing on the lathe.

Since filing should be used for little more than to remove tool marks from the workpiece, only 0.002 to 0.005 inch should be left for the tiling operation.

Polishing on the Lathe

Polishing with either abrasive cloth or abrasive paper is desirable to improve the surface finish after filing. Emery abrasive cloth is best for ferrous metals while abrasive paper often gives better results on nonferrous materials. The most effective speed for polishing with ordinary abrasives is approximately 5,000 feet per minute. Since most lathes are not capable of a speed this great for an average size workpiece, it is necessary to select as high a speed as conditions will permit.

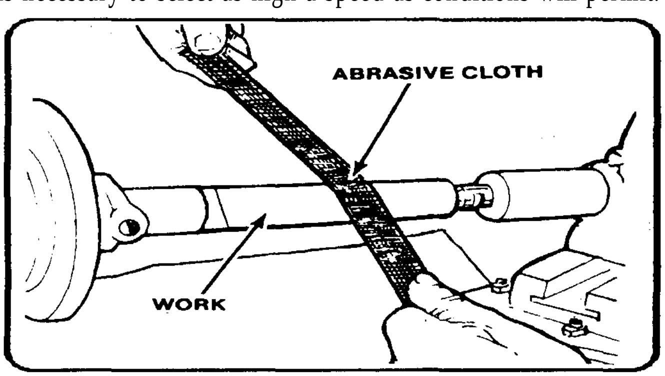


Figure 7-101. Polishing on the lathe.

In most cases the abrasive cloth or paper is held directly in the hand and applied to the workpiece, although it may be tacked over a piece of wood and used in the same manner as a file. Improvised clamps may also be used to polish plain round work.

Since polishing will slightly reduce the dimensions of the workpiece, 0.00025 to 0.0005 inch should be allowed for this operation. Figure 7-101 shows how to hold the abrasive strip when polishing. Note that the ends of the strip are separated. This prevents the strip from grabbing and winding around the work, which could pull the operator's hand into the work. Move the polishing strip slowly back and forth to prevent material building up on the strip which causes polishing rings to form on the work. To produce a bright surface, polish the work dry. To produce a dull satin finish, apply oil as the polishing operation is in progress.

ECCENTRIC WORK ON THE LATHE

Eccentric work is work that is turned off center, or not on the normal center axis. An engine crankshaft is a good example of an eccentric workpiece. Crankshafts normally have a main center axis, called a main journal, and offset axes, which produce the throw and the eccentric diameters of the mechanism. An eccentric shaft may have two or more diameters and several different center axes. The amount of eccentricity, or half of the throw, is the linear distance that a set of center holes has been offset from the normal center axis of the workpiece. Eccentric turning on the lathe is used for the following eccentric turning situations:

When the throw is large enough to allow all centers to be located on the workpiece at the same time.

When the throw is too small to allow all centers to fit into the end of a workpiece at the same time. (The center drilled holes are too large.)

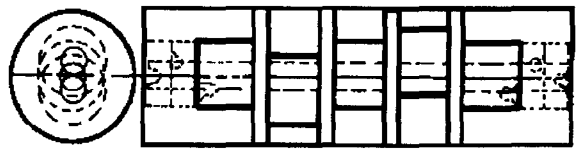
When the throw is so great that all centers cannot be located on the work, or in other words, a throw larger than the largest diameter of the workpiece. (This type of crank is usually made in separate pieces and connected together, since the cost of wasted material would be too great if constructed from one piece on the lathe).

Turning an Eccentric with Center Holes

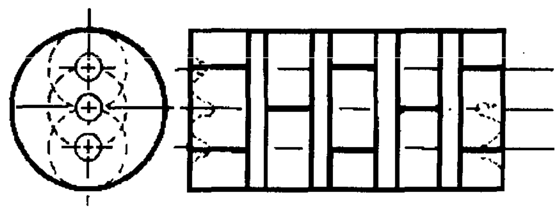
Before an eccentric workpiece can be machined, it is necessary to center-drill both ends of the workpiece, including the offset centers. If the workpiece is large enough to position all center axes on the work at the same time, the machining operation will be simple and easy.

- First determine the stock required by adding the throws plus 1/8 inch for machining (Figure 7-102).
- Face the work to length in a chuck.
- Remove the piece and apply layout dye to both ends.
- Mount the work in a V- block and, using a surface plate and venire height scriber, lay out the normal center axis and the offset center axes on both ends.
- Accurately prick punch the intended centers, check for accuracy, and then enlarge the punch marks with a center punch.
- Center- drill both sets of center punch marks by using a milling machine, a drilling machine, or the four-jaw independent chuck of the lathe with a dial indicator to line up the centers.
- Mount the work in the lathe between centers and turn the largest diameter first. If all diameters are the same, turn the middle diameter journal first.
- After turning the center journal down to the required diameter, remount the work in an offset center hole and machine the throw diameter to the finished size.
- Accurately prick punch the intended centers, check for accuracy, and then enlarge the punch marks with a center punch.
- Center- drill both sets of center punch marks by using a milling machine, a drilling machine, or the four-jaw independent chuck of the lathe with a dial indicator to line up the centers.
- Mount the work in the lathe between centers and turn the largest diameter first. If all diameters are the same, turn the middle diameter journal first.
- After turning the center journal down to the required diameter, remount the work in an offset center hole and machine the throw diameter to the finished size.

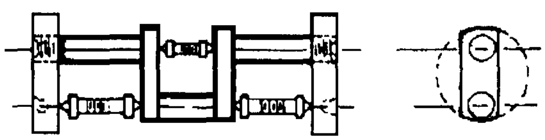
Additional throws are machined in the same manner. Throw positions may be started by cutting with a parting tool to establish the shoulders, which may aid the turning operation. The tool bit selected will depend on the material to be machined and on the depth of cut desired.



Eccentric turning when the throw is too small to allow properly drilled center holes (add 3/4" length for each center hole).



Eccentric turning when the throw is large enough to allow all centers to be located on the workpiece at the same time. In this case, the eccentric throws must be turned before the main journal.



Eccentric turning a large throw is not normally done on a lathe, however, it may be accomplished with special end and center supports.

Figure 7-102. Eccentric turning.

Turning an Eccentric with Close Center Holes

If turning an eccentric that has the different centers placed too close together, a different procedure should be used. Cut

the stock 3/4 inch oversized and just face both ends to clean up the saw cuts Lay out and center-drill the normal. center axis and turn down those diameters on the center axis with the work mounted between centers. Remove the work and remount into a chuck. Face both ends to the required length and center-drill the offset centers. Remount the work between these centers and machine the eccentric diameters to size. For eccentric work that has a limited distance between each center, this method is safer than trying to use a very shallow center-drilled hole to hold the work between centers (Figure 7-102).

Turning an Eccentric Using Throw Plates

If the lathe is to be used to turn a crank with a great throw, or a throw that is greater than normally machined on a lathe (Figure 7- 102), special throw plates must be fabricated to hold the ends of the work while turning. The special throw plates will be used as support blocks to enable the offset center holes to be machined into the throw plates and allow for eccentric turning. eccentric turning, it is not recommended for normal lathe operations. Special crankshaft turning and grinding equipment is available for this type of machining.

RECESSING DRILLED AND BORED HOLES

General

Recessing, sometimes called channeling or cambering, is the process of cutting a groove inside of a drilled, bored, or reamed hole. Recesses (Figure 7-103) are usually machined to provide room for the tool runout needed for subsequent operations such as internal threading.

A boring bar and holder may be used as a recessing tool, since recessing tools have the same tool angles and are similar in shape to boring tools. A high-speed steel cutting tool bit, ground with a square nose, makes a satisfactory tool for cutting small chambers (Figure 7-103). The sides of the tool bit taper in from the cutting edge so that the nose of the tool is the widest part. The tool bit must extend from the holder a distance slightly greater than the depth of the chamber to prevent the holder from rubbing the bore of the work.

Machining a Recess

To cut a recess, set up the lathe as in a boring operation. Reference the face of the tool bit to the face of the work; then move the tool bit forward the required distance to the recess by using the micrometer stop or by using the compound rest graduated collar. The compound rest must be set parallel with the ways of the bed for this method. Add the width of the tool bit into the measurement or the recess will not be cut correctly. Position A (Figure 7-103) is the tool aligning to the work, position B is set over to the front shoulder of the recess, and position C is the set over to the back of the recess. Use the cross slide graduated collar to measure the distance to move the tool bit toward the operator. inside of the hole. Spindle speed may have to be reduced due to the shape of the tool bit causing chatter on the work. After cutting the recess, use inside calipers to check the diameter.

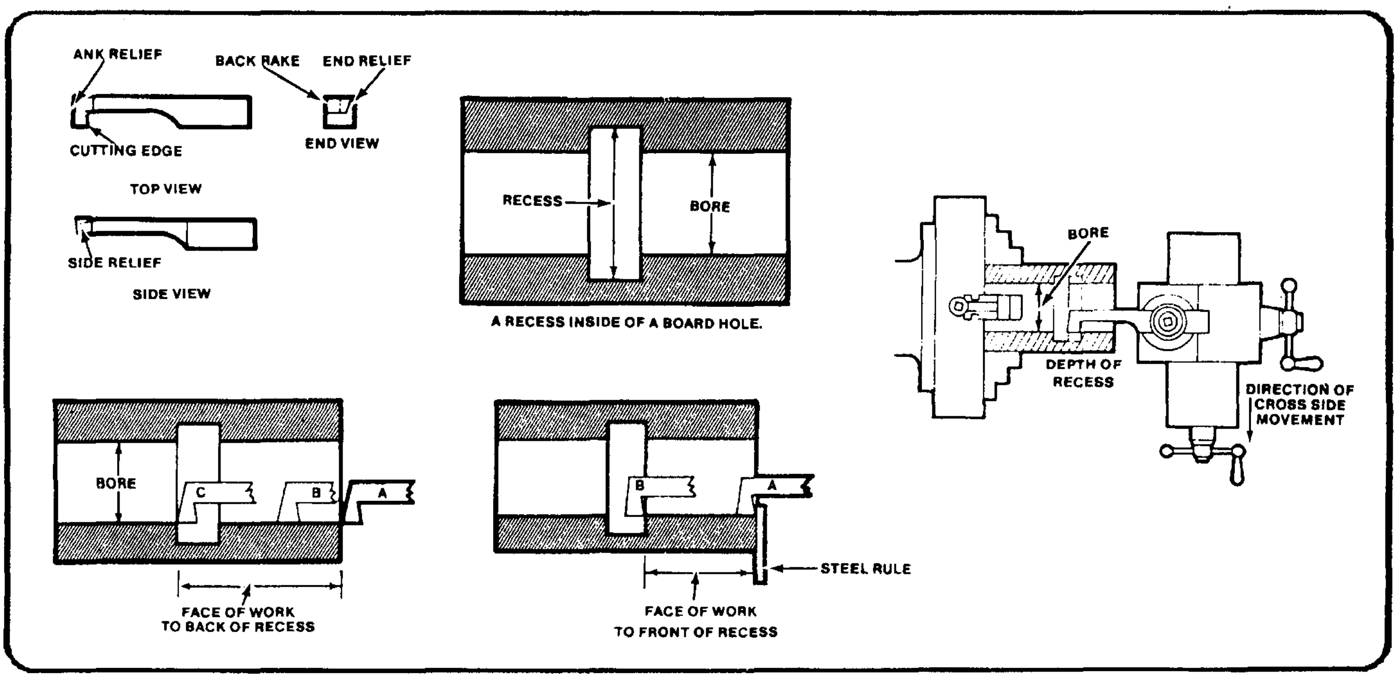


Figure 7-103. Recessing.

LATHE TOOL POST GRINDER

General

The tool post grinder is a portable grinding machine that can be mounted on the compound rest of a lathe in place of the tool post. It can be used to machine work that is too hard to cut by ordinary means or to machine work that requires a very fine finish. Figure 7-29 shows a typical tool post grinder. The grinder must be set on center, as shown in Figure 7-104. The centering holes located on the spindle shaft are used for this purpose. The grinding wheel takes the place of a lathe cutting tool. It can perform most of the operations that a cutting tool is capable of performing. cylindrical, tapered, and internal surfaces can be ground with the tool post grinder. Very small grinding wheels are mounted on tapered shafts known as quills to grind internal surfaces.

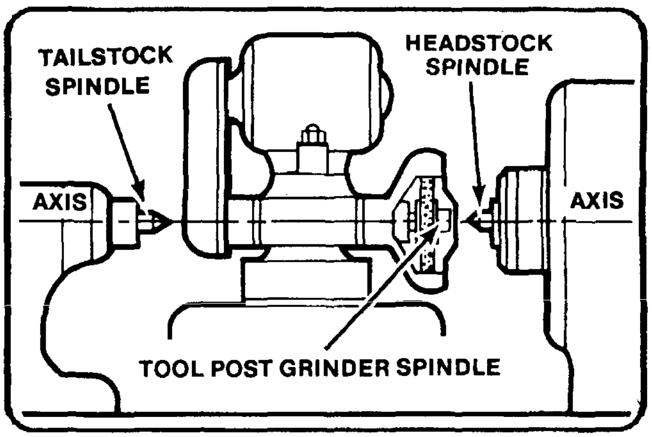


Figure 7-104. Aligning tool post grinder.

Selection of Grinding Wheels and Speeds

The grinding wheel speed is changed by using various sizes of pulleys on the motor and spindle shafts. An instruction plate on the grinder gives both the diameter of the pulleys required to obtain a given speed and the maximum safe speed for grinding wheels of various diameters. Grinding wheels are safe for operation at a speed just below the highest recommended speed. A higher than recommended speed may cause the wheel to disintegrate. For this reason, wheel guards are furnished with the tool post grinder to protect against injury. Always check the pulley combinations given on the instruction plate of the grinder when you mount a wheel. Be sure that the combination is not reversed, because this may cause the wheel to run at a speed far in excess of that recommended. During all grinding operations, wear goggles to protect your eyes from flying abrasive material.

Dressing the Grinding Wheel

The grinding wheel must be dressed and trued. Use a diamond wheel dresser to dress and true the wheel. The dresser is held in a holder that is clamped to the drive plate. Set the point of the diamond at center height and at a 10° to 15° angle in the direction of the grinding wheel rotation. The 10° to 15° angle prevents the diamond from gouging the wheel. Lock the lathe spindle by placing the spindle speed control lever in the low RPM position.

NOTE: The lathe spindle does not revolve when you are dressing the grinding wheel.

Remove the diamond dresser holder as soon as the dressing operation is completed. Bring the grinding wheel in contact with the diamond by carefully feeding the cross slide by hand. Move the wheel clear of the diamond and make a cut by means of the cross slide. The maximum depth of cut is 0.002 inch. Move the wheel slowly by hand back and forth over the point of the diamond. Move the carriage if the face of the wheel is parallel to the way of the lathe. Move the compound rest if the face of the wheel is at an angle. Make the final depth of cut of 0.0005 inch with a slow, even feed to obtain a good wheel finish.

Before you begin the grinding operation, cover the ways with a heavy piece of paper or use a shallow pan of water placed on the ways to collect the grinding dust that will accumulate from the grinding. This is to ensure none of the grinding burns to the ways or gets under the carriage which will cause the lathe premature wear. If you use a piece of paper, pay close attention that the sparks from the grinding operation do not cause the paper to ignite. If you use a shallow pan of water, make sure water is not spilled on the ways of the lathe. After all grinding operations, thoroughly clean and oil the lathe to remove any grinding dust that the paper pan of water missed.

Grinding Feeds, Speeds, and Depth of Cuts

Rotate the work at a fairly low speed during the grinding operations. The recommended surface foot speed is 60 to 100 FPM. The depth of cut depends upon the hardness of the work, the type of grinding wheel, and the desired finish.

Never take grinding cuts deeper than 0.002 inch Use a fairly low rate of feed, You will soon be able to judge whether the feed should be increased or decreased. Never stop the rotation of the work or the grinding wheel while they are in contact with each other.

Marking Position of Lathe Centers

Tool post grinders are often used to refinish damaged lathe centers. If the lathe is to be used for turning between centers in the near future, grind the tailstock center first, then the headstock center. Leave the headstock center in position for the turning operation. This method provides the greatest degree of accuracy. If you must remove the headstock center in order to perform other operations, marks placed on the headstock center, the sleeve, and the center will enable you to install them in the same position they were in when the center was ground. This will ensure the greatest degree of accuracy for future operations involving turning work between centers.

Setup for Grinding Lathe Centers

To refinish a damaged lathe center, you should first install headstock and tailstock centers after ensuring that the spindle holes, drill sleeves, and centers are clean and free of burrs. Next, position the compound rest parallel to the ways; then, mount the tool post grinder on the compound rest. Make sure that the grinding wheel spindle is at center height and aligned with the lathe centers. Move the compound rest 30° to the right of the lathe spindle axis, as shown in Figure 7-40. Mount the wheel dresser, covering the ways and carriage with rags to protect them from abrasive particles. Wear goggles to protect your eyes.

Grinding Lathe Centers

Start the grinding motor. Turn it on and off alternately, but let it run a bit longer each time, until the abrasive wheel is brought up to top speed. Dress the wheel, feeding the grinder with the compound rest. Then move the grinder clear of the headstock center and remove the wheel dresser. Set the lathe for the desired spindle speed and engage the spindle. Pick up the surface of the center. Take a light depth of cut and feed the grinder back and forth with the compound rest. Do not allow the abrasive wheel to feed entirely off the center. Continue taking additional cuts until the center cleans up. To produce a good finish, reduce the feed rate and the depth of cut to 0.0005. Grind off the center's sharp point, leaving a flat with a diameter about 1/32 inch. Move the grinder clear of the headstock and turn it off.

MILLING ON THE LATHE

Milling operations may be performed on the lathe by using the Versa-Mil, which is discussed in Chapter 9, and by using the lathe milling fixture. The lathe milling fixture complements the Versa-Mil and adds to the basic capabilities of the machine shop. If the Versa-Mil is out of action or being used for another job, many milling operations can still be accomplished by using the milling fixture (Figure 7-105). Capabilities, functions, and uses are outlined in the appropriate operator's manual, either TM 9-3465-200-10 or TM 9-3465-201-10.

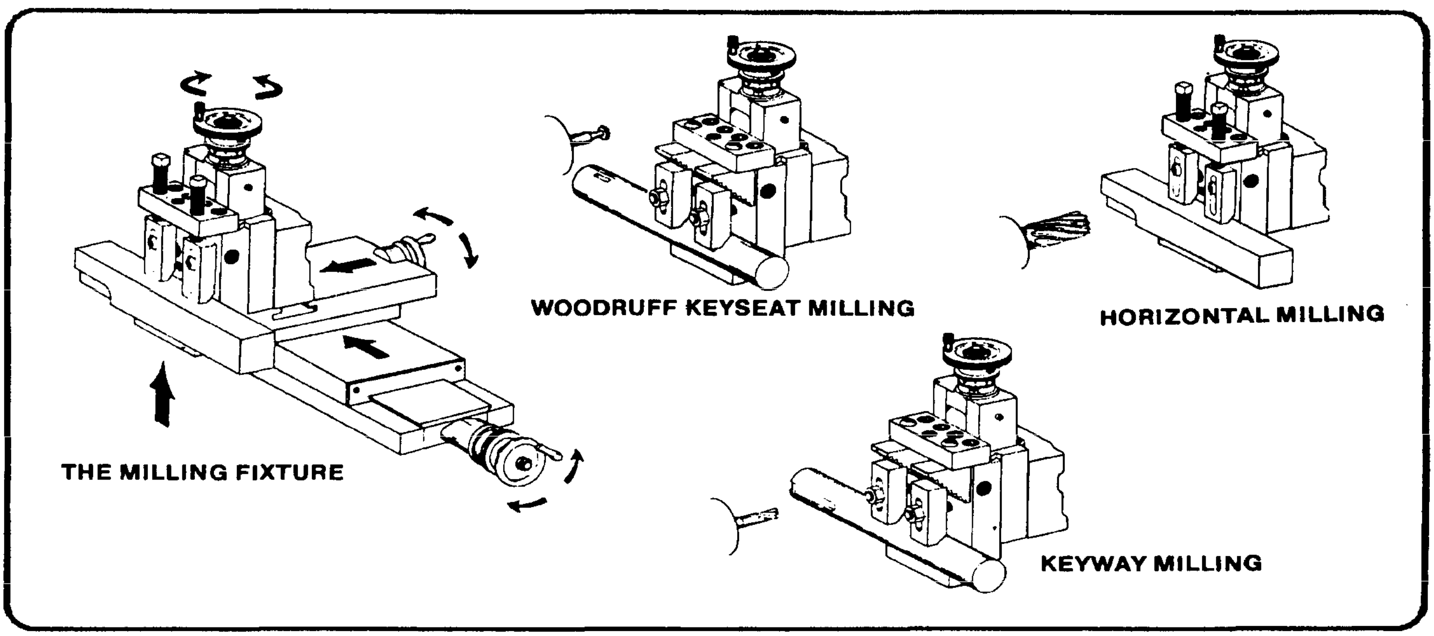


Figure 7-105. Lathe milling fixture operations.

USING MICROMETER CARRIAGE STOP

The micrometer carriage stop, shown in Figure 7-28, is used to accurately position the lathe carriage. Move the carriage so that the cutting tool is approximately positioned. Clamp the micrometer carriage stop to the ways of the lathe, with the spindle in contact with the carriage. The spindle of the micrometer carriage stop can be extended or retracted by means of the knurled adjusting collar. The graduations on the collar, which indicate movement in thousandths of an inch, make it possible to set the spindle accurately. Next, bring the carriage in contact with the micrometer spindle again. The carriage can be accurately positioned within 0.001 inch. This is very useful when you are facing work to length, machining shoulders to an exact length, or accurately spacing internal and external grooves. After making a cut, bring the tool back to the start of the cut by means of the carriage stop. This feature is very useful when you must remove a tool, such as the internal recessing tool, from the hole to take measurements and then reposition it to take additional cuts. Always bring the carriage into contact with the stop by hand. Use power feed to bring the carriage within 1/32 inch of the stop. Move the carriage by hand the remaining distance.

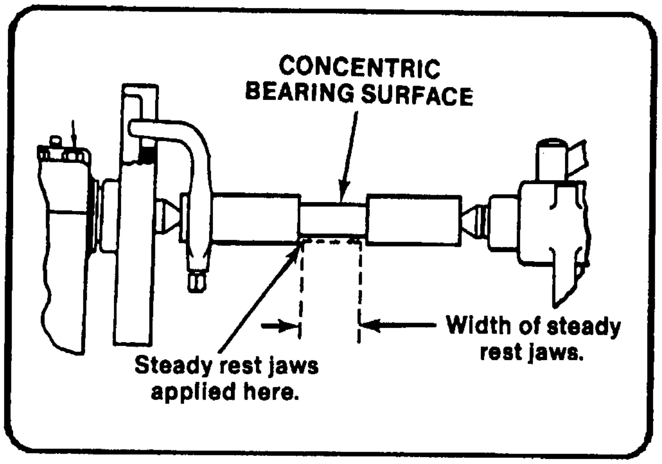


Figure 7-106. Using the steady rest.

USING STEADY AND FOLLOWER RESTS

General

The steady rest consists of a frame and three adjustable jaws which support the work, as shown in Figure 7-27. One purpose of the steady rest is to prevent springing or deflection of slender, flexible work; another is to furnish auxiliary support for the work to permit heavy cuts to be made; a third is to support work for drilling, boring, or internal threading. The over arm containing the top jaw can be unfastened and swung out of the way so that identical pieces can be removed and replaced without adjusting the jaws.

Bearing Surface

A bearing surface must be provided for the steady rest jaws. The bearing surface is usually machined directly on the work, as shown in Figure 7-106. When the work is too small in diameter to machine the bearing surface or shaped so that it would be impractical to machine one, you can use a cathead to provide the bearing surface. The cathead shown in Figure 7-27, has a bearing you surface, a hole through, which the work extends, and adjusting screws. The adjusting screws fasten the cathead to the work. They are also used to align the bearing surface so can use a cathead to provide the bearing surface so that t is concentric to the work axis. Use a dial indicator to ensure concentricity.

Setting up the Steady Rest

To setup the rest, first machine and polish the portion of the work that is to be used as the bearing surface. Clean the portion of the ways where the steady rest is to be mounted, place the steady rest on the ways and clamp loosely. Open the top of the steady rest and place the workpiece in the chuck with the bearing surface over the adjustable jaws. Clamp the steady rest securely to the ways. Close the top of the steady rest and adjust the jaws to the workpiece. There should be 0.001 inch clearance between the jaws and the workpiece. Tighten the locking screws on the adjustable jaws. Lubricate the bearing surface generously with a heavy oil before turning the lathe on. Proceed with the machining operation Continuously watch the bearing surface and the adjustable jaws to ensure a film of heavy oil is between them. As the machining operation continues, also check the bearing surface and adjustable jaws as when the workpiece heats up it will expand, closing the distance between the jaws and the workpiece.

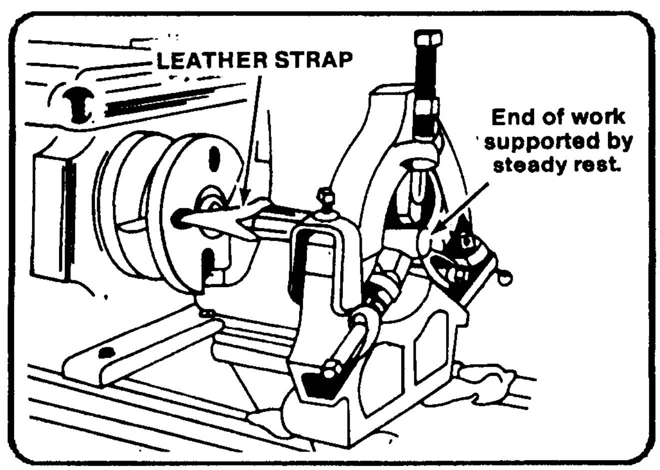


Figure 7-107. Tying the lathe dog.

Using Steady Rest with Headstock Center

When it is not possible to hold the work in the chuck, you can machine with one end supported by the headstock center and the other end supported by the steady rest. Use a leather strap or rawhide thong to tie the work to the driveplate and to prevent it from moving off the headstock center, as shown in Figure 7-107. Mount the work between centers and machine the bearing surface. Set up the steady rest. With the work mounted between the centers, tie the lathe dog, then remove the tailstock center and perform the necessary machining,

Using the Follower Rest

Long slender shafts that tend to whip and spring while they are being machined require the use of a follower rest (Figure 7-27). The follower rest is fastened to the carriage and moves with the cutting tool. The upper jaw prevents the work from climbing the cutting tool, The lower jaw prevents the work from springing away from the cutting tool The follower rest jaws are adjusted in the same manner as steady rest jaws. The follower rest is often used when long, flexible shafts are threaded, as shown in Figure 7-108, At the completion of each threading cut, remove any burrs that may have formed to prevent them from causing the work to move out of alignment.

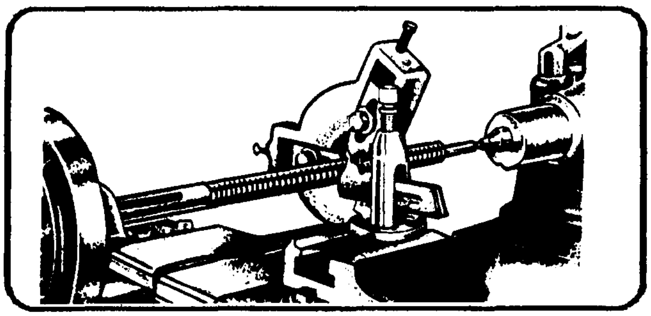


Figure 7-108. Using the follower rest in threading.

Chapter 8

MILLING OPERATIONS

Milling is the process of machining flat, curved, or irregular surfaces by feeding the workpiece against a rotating cutter containing a number of cutting edges. The milling machine consists basically of a motor driven spindle, which mounts and revolves the milling cutter, and a reciprocating adjustable worktable, which mounts and feeds the workpiece.

Milling machines are basically classified as vertical or horizontal. These machines are also classified as knee-type, ram-type, manufacturing or bed type, and planer-type. Most milling machines have self-contained electric drive motors, coolant systems, variable spindle speeds, and power-operated table feeds

TYPES OF MILLING MACHINES

KNEE-TYPE MILLING MACHINE

Knee-type milling machines are characterized by a vertically adjustable worktable resting on a saddle which is supported by a knee. The knee is a massive casting that rides vertically on the milling machine column and can be clamped rigidly to the column in a position where the milling head and milling machine spindle are properly adjusted vertically for operation.

The plain vertical machines are characterized by a spindle located vertically, parallel to the column face, and mounted in a sliding head that can be fed up and down by hand or power. Modern vertical milling machines are designed so the entire head can also swivel to permit working on angular surfaces,

The turret and swivel head assembly is designed for making precision cuts and can be swung 360° on its base. Angular cuts to the horizontal plane may be made with precision by setting the head at any required angle within a 180” arc.

The plain horizontal milling machine's column contains the drive motor and gearing and a fixed position horizontal milling machine spindle. An adjustable overhead arm containing one or more arbor supports projects forward from the top of the column. The arm and arbor supports are used to stabilize long arbors. Supports can be moved along the overhead arm to support the arbor where support is desired depending on the position of the milling cutter or cutters.

The milling machine's knee rides up or down the column on a rigid track. A heavy, vertical positioning screw beneath past the milling cutter. The milling machine is excellent for forming flat surfaces, cutting dovetails and keyways, forming and fluting milling cutters and reamers, cutting gears, and so forth. Many special operations can be performed with the attachments available for milling machine use.the knee is used for raising and lowering. The saddle rests upon the knee and supports the worktable. The saddle moves in and out on a dovetail to control cross feed of the worktable. The worktable traverses to the right or left upon the saddle for feeding the workpiece past the milling cutter. The table may be manually controlled or power fed.

UNIVERSAL HORIZONTAL MILLING MACHINE

The basic difference between a universal horizontal milling machine and a plain horizontal milling machine is the addition of a table swivel housing between the table and the saddle of the universal machine. This permits the table to swing up to 45° in either direction for angular and helical milling operations. The universal machine can be fitted with various attachments such as the indexing fixture, rotary table, slotting and rack cutting attachments, and various special fixtures.

RAM-TYPE MILLING MACHINE

The ram-type milling machine is characterized by a spindle mounted to a movable housing on the column to permit positioning the milling cutter forward or rearward in a horizontal plane. Two popular ram-type milling machines are the universal milling machine and the swivel cutter head ram-type milling machine.

UNIVERSAL RAM-TYPE MILLING MACHINE

The universal ram-type milling machine is similar to the universal horizontal milling machine, the difference being, as its name implies, the spindle is mounted on a ram or movable housing.

SWIVEL CUTTER HEAD RAM-TYPE MILLING MACHINE

The cutter head containing the milling machine spindle is attached to the ram. The cutter head can be swiveled from a vertical spindle position to a horizontal spindle position or can be fixed at any desired angular position between vertical and horizontal. The saddle and knee are hand driven for vertical and cross feed adjustment while the worktable can be either hand or power driven at the operator's choice.

Basic milling machine configurations are shown in Figure 8-1.

SAFETY RULES FOR MILLING MACHINES

Milling machines require special safety precautions while being used. These are in addition to those safety precautions described in Chapter 1.

- Do not make contact with the revolving cutter.
- Place a wooden pad or suitable cover over the table surface to protect it from possible damage.
- Use the buddy system when moving heavy attachments.

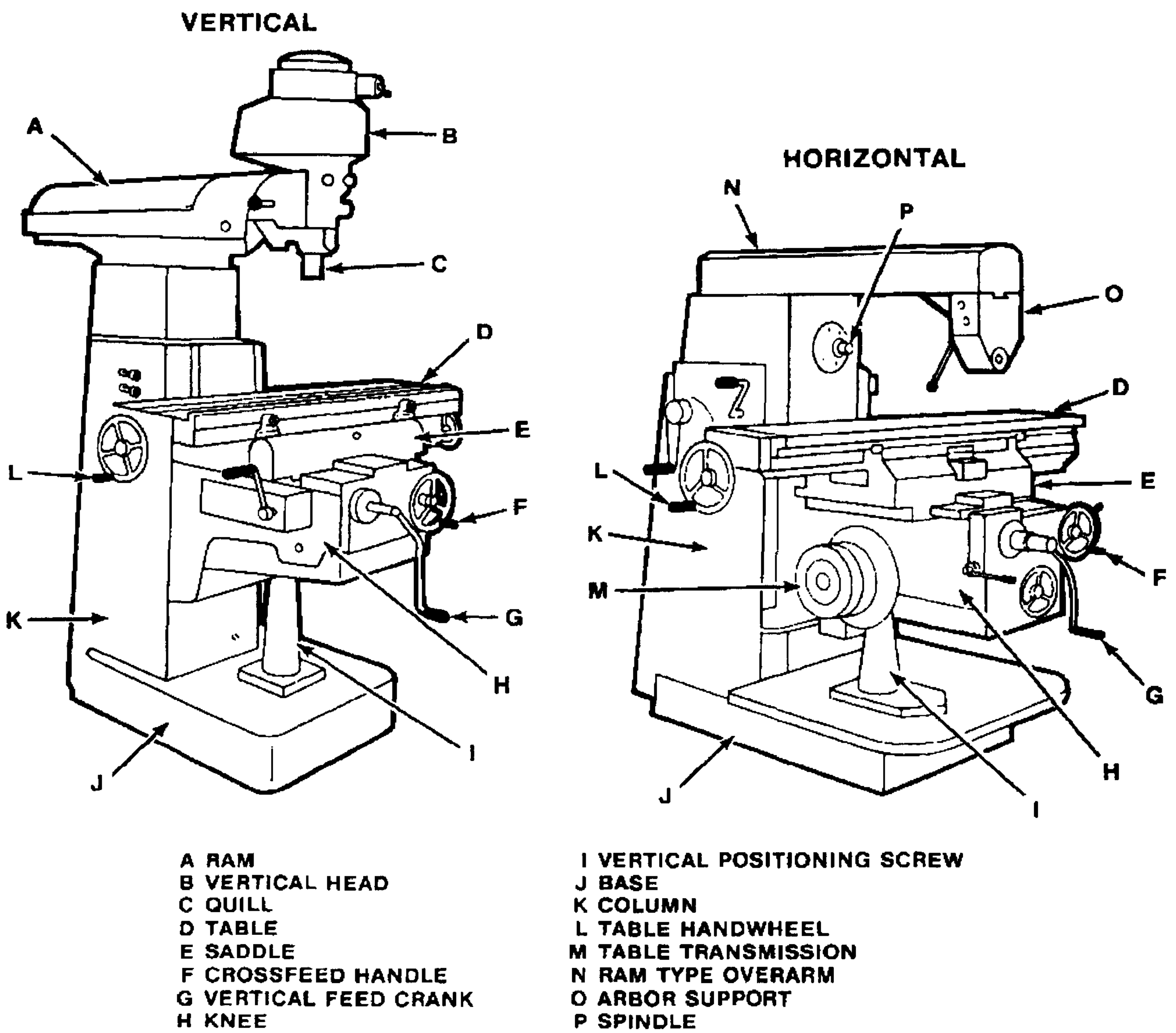


Figure 8-1. Milling machines.

- Do not attempt to tighten arbor nuts using machine power.
- When installing or removing milling cutters, always hold them with a rag to prevent cutting your hands.
- While setting up work, install the cutter last to avoid being cut.
- Never adjust the workpiece or work mounting devices when the machine is operating.
- Chips should be removed from the workpiece with an appropriate rake and a brush.

NOTE Chip rake should be fabricated to the size of the T-slots (Figure 8-2).

- Shut the machine off before making any adjustments or measurements.
- When using cutting oil, prevent splashing by using appropriate splash guards. Cutting oil on the floor can cause a slippery condition that could result in operator injury

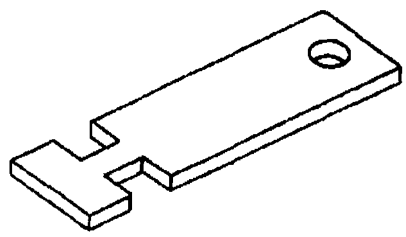


Figure 8-2. Chip rake.

TOOLS AND EQUIPMENT

MILLING CUTTERS

Classification of Milling Cutters

Milling cutters are usually made of high-speed steel and are available in a great variety of shapes and sizes for various purposes. You should know the names of the most common classifications of cutters, their uses, and, in a general way, the sizes best suited to the work at hand.

Milling Cutter Nomenclature

Figure 8-3 shows two views of a common milling cutter with its parts and angles identified. These parts and angles in some form are common to all cutter types.

- The pitch refers to the angular distance between like or adjacent teeth.

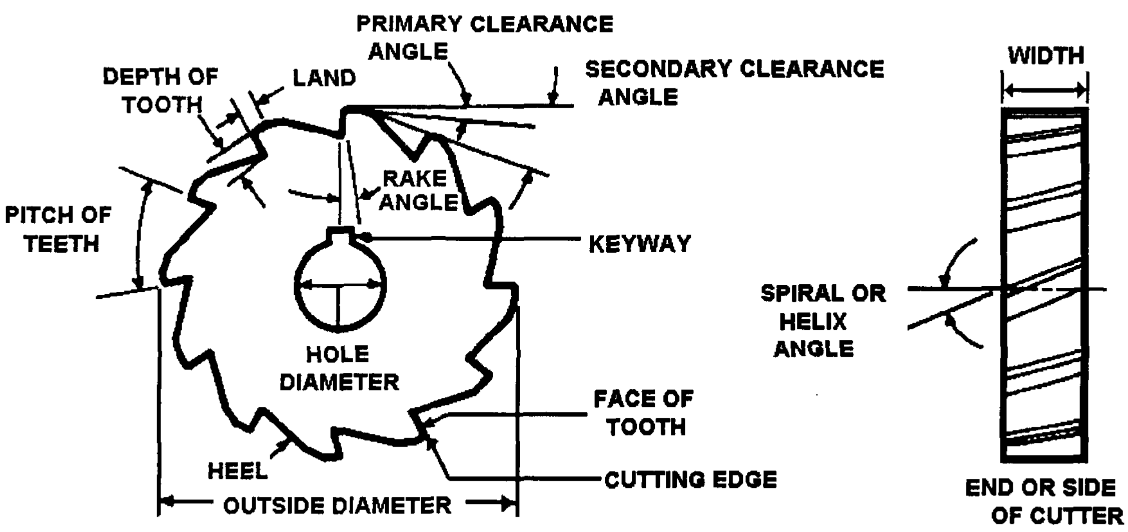


Figure 8-3. Milling cutter nomenclature.

- The pitch is determined by the number of teeth. The tooth face is the forward facing surface of the tooth that forms the cutting edge.
- The cutting edge is the angle on each tooth that performs the cutting.
- The land is the narrow surface behind the cutting edge on each tooth.
- The rake angle is the angle formed between the face of the tooth and the centerline of the cutter. The rake angle defines the cutting edge and provides a path for chips that are cut from the workpiece.
- The primary clearance angle is the angle of the land of each tooth measured from a line tangent to the centerline of the cutter at the cutting edge. This angle prevents each tooth from rubbing against the workpiece after it makes its cut.
- This angle defines the land of each tooth and provides additional clearance for passage of cutting oil and chips.
- The hole diameter determines the size of the arbor necessary to mount the milling cutter.
- Plain milling cutters that are more than 3/4 inch in width are usually made with spiral or helical teeth. A plain spiral-tooth milling cutter produces a better and smoother finish and requires less power to operate. A plain helical-tooth milling cutter is especially desirable when milling an uneven surface or one with holes in it.

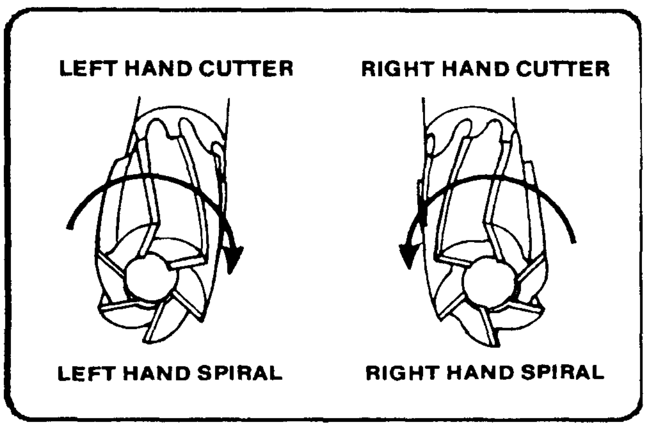


Figure 8-4. Left and right cutters

Types of Teeth

The teeth of milling cutters may be made for right-hand or left-hand rotation, and with either right-hand or left-hand helix. Determine the hand of the cutter by looking at the face of the cutter when mounted on the spindle. A right-hand cutter must rotate counterclockwise; a left-hand cutter must rotate clockwise. The right-hand helix is shown by the flutes leading to the right; a left-hand helix is shown by the flutes leading to the left. The direction of the helix does not affect the cutting ability of the cutter, but take care to see that the direction of rotation is correct for the hand of the cutter (Figure 8-4).

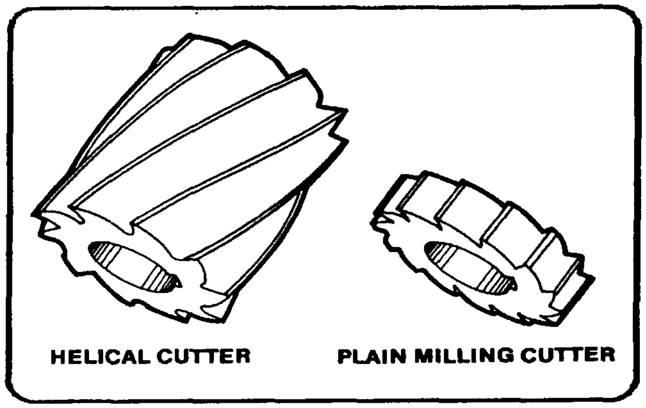


Figure 8-5. Plain and helical milling cutters.

Saw Teeth

Saw teeth similar to those shown in Figure 8-3 are either straight or helical in the smaller sizes of plain milling cutters, metal slitting saw milling cutters, and end milling cutters. The cutting edge is usually given about 5 degrees primary clearance. Sometimes the teeth are provided with off-set nicks which break up chips and make coarser feeds possible.

Helical Milling Cutters

The helical milling cutter is similar, to the plain milling cutter, but the teeth have a helix angle of 45° to 60°. The steep helix produces a shearing action that results in smooth, vibration-free cuts. They are available for arbor mounting, or with an integral shank with or without a pilot. This type of helical cutter is particularly useful for milling elongated slots and for light cuts on soft metal. See Figure 8-5.

Metal Slitting Saw Milling Cutter

The metal slitting saw milling cutter is essentially a very thin plain milling cutter. It is ground slightly thinner toward the center to provide side clearance. These cutters are used for cutoff operations and for milling deep, narrow slots, and are made in widths from 1/32 to 3/16 inch.

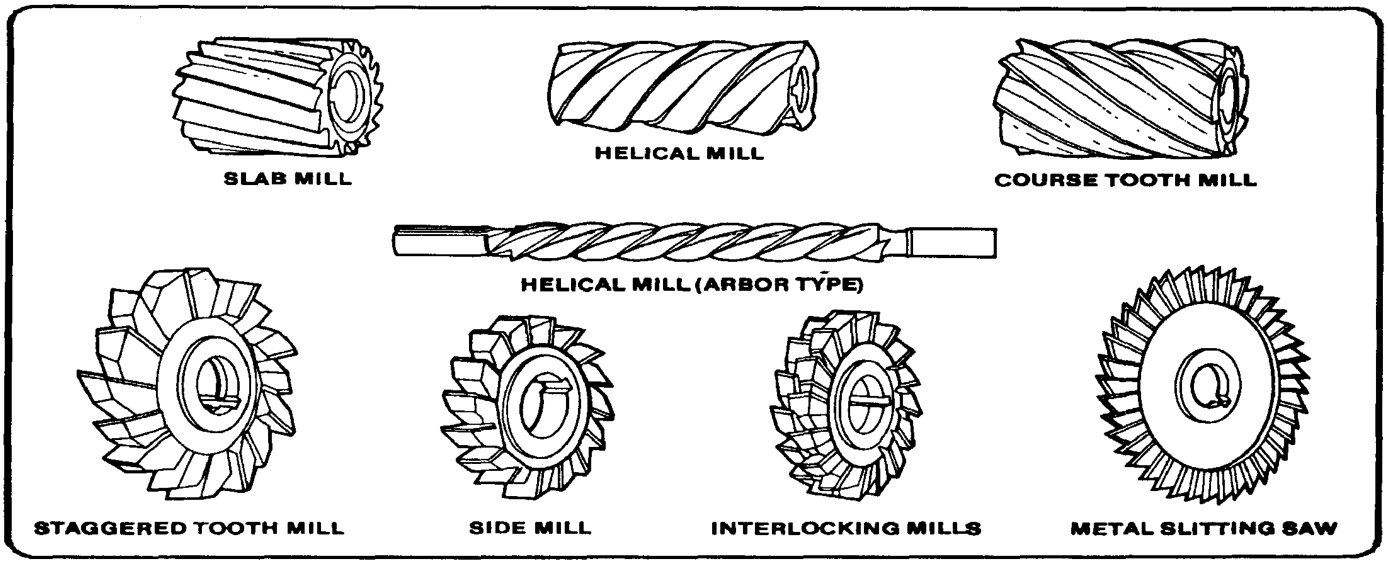


Figure 8-6. Various milling cutters.

Side Milling Cutters

Side milling cutters are essentially plain milling cutters with the addition of teeth on one or both sides. A plain side milling cutter has teeth on both sides and on the periphery. When teeth are added to one side only, the cutter is called a half-side milling cutter and is identified as being either a right-hand or left-hand cutter. Side milling cutters are generally used for slotting and straddle milling.

Interlocking tooth side milling cutters and staggered tooth side milling cutters are used for cutting relatively wide slots with accuracy (Figure 8-6). Interlocking tooth side milling cutters can be repeatedly sharpened without changing the width of the slot they will machine.

After sharpening, a washer is placed between the two cutters to compensate for the ground off metal. The staggered tooth cutter is the most washer is placed between the two cutters to compensate for efficient type for milling slots where the depth exceeds the width.

End Milling Cutters

The end milling cutter, also called an end mill, has teeth on the end as well as the periphery. The smaller end milling cutters have shanks for chuck mounting or direct spindle mounting. End milling cutters may have straight or spiral flutes. Spiral flute end milling cutters are classified as left-hand or right-hand cutters depending on the direction of rotation of the flutes. If they are small cutters, they may have either a straight or tapered shank.

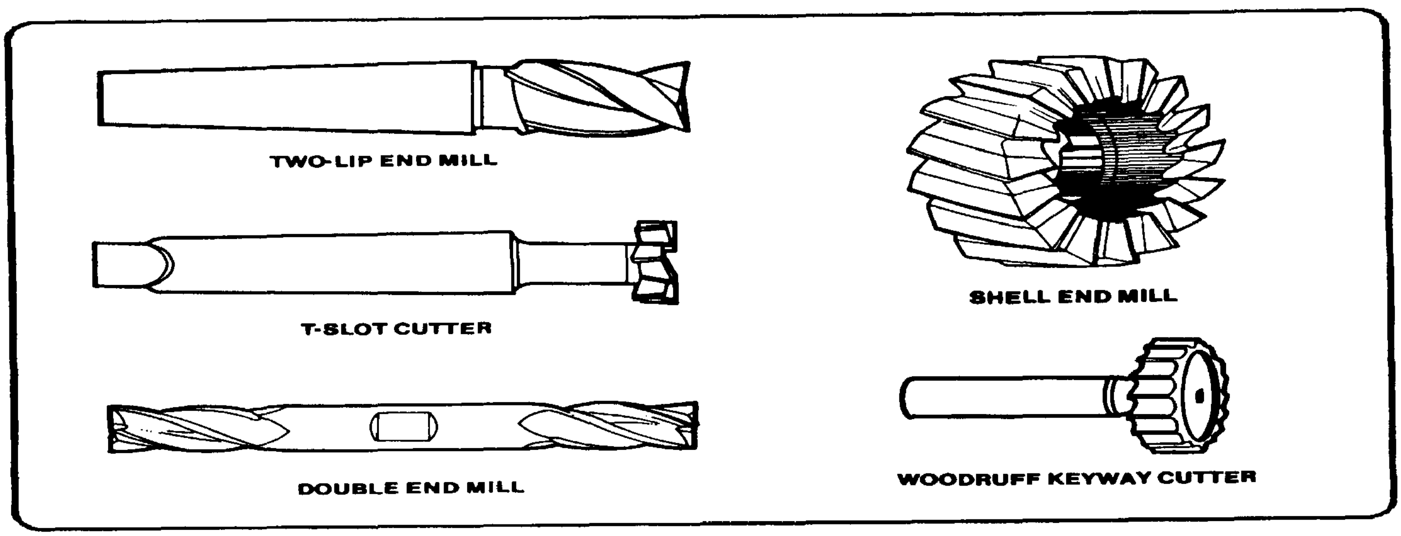


Figure 8-7. End mill, T-slot, and Woodruff keyway cutters.

The most common end milling cutter is the spiral flute cutter containing four flutes. Two-flute end milling cutters, sometimes referred to as two-lip end mill cutters, are used for milling slots and keyways where no drilled hole is provided for starting the cut. These cutters drill their own starting holes. Straight flute end milling cutters are generally used for milling both soft or tough materials, while spiral flute cutters are used mostly for cutting steel.

Large end milling cutters (normally over 2 inches in diameter) (Figure 8-10) are called shell end mills and are recessed on the face to receive a screw or nut for mounting on a separate shank or mounting on an arbor, like plain milling cutters. The teeth are usually helical and the cutter is used particularly for face milling operations requiring the facing of two surfaces at right angles to each other.

T-Slot Milling Cutter

The T-slot milling cutter is used to machine T-slot grooves in worktables, fixtures, and other holding devices. The cutter has a plain or side milling cutter mounted to the end of a narrow shank. The throat of the T-slot is first milled with a side or end milling cutter and the headspace is then milled with the T-slot milling cutter.

Woodruff Keyslot Milling Cutters

The Woodruff keyslot milling cutter is made in straight, tapered-shank, and arbor-mounted types. See Figure 8-7. The most common cutters of this type, under 1 1/2 inches in diameter, are provided with a shank. They have teeth on the periphery and slightly concave sides to provide clearance. These cutters are used for milling semicylindrical keyways in shafts.

Angle Milling Cutters

The angle milling cutter has peripheral teeth which are neither parallel nor perpendicular to the cutter axis. See Figure 8-8. Common operations performed with angle cutters are cutting V-notches and serration's. Angle cutters may be single-angle milling cutters or double-angle milling cutters. The single-angle cutter contains side-cutting teeth on the flat side of the cutter. The angle of the cutter edge is usually 30°, 45°, or 60°, both right and left. Double-angle cutters have included angles of 45, 60, and 90 degrees.

Gear Hob

The gear hob is a formed tooth milling cutter with helical teeth arranged like the thread on a screw. These teeth- are fluted to produce the required cutting edges. Hobs are generally used for such work as finishing spur gears, spiral gears, and worm gears. They may also be used to cut ratchets and spline shafts.

Concave and Convex Milling Cutters

Concave and convex milling cutters are formed tooth cutters shaped to produce concave and convex contours of 1/2 circle or less. The size of the cutter is specified by the diameter of the circular form the cutter produces.

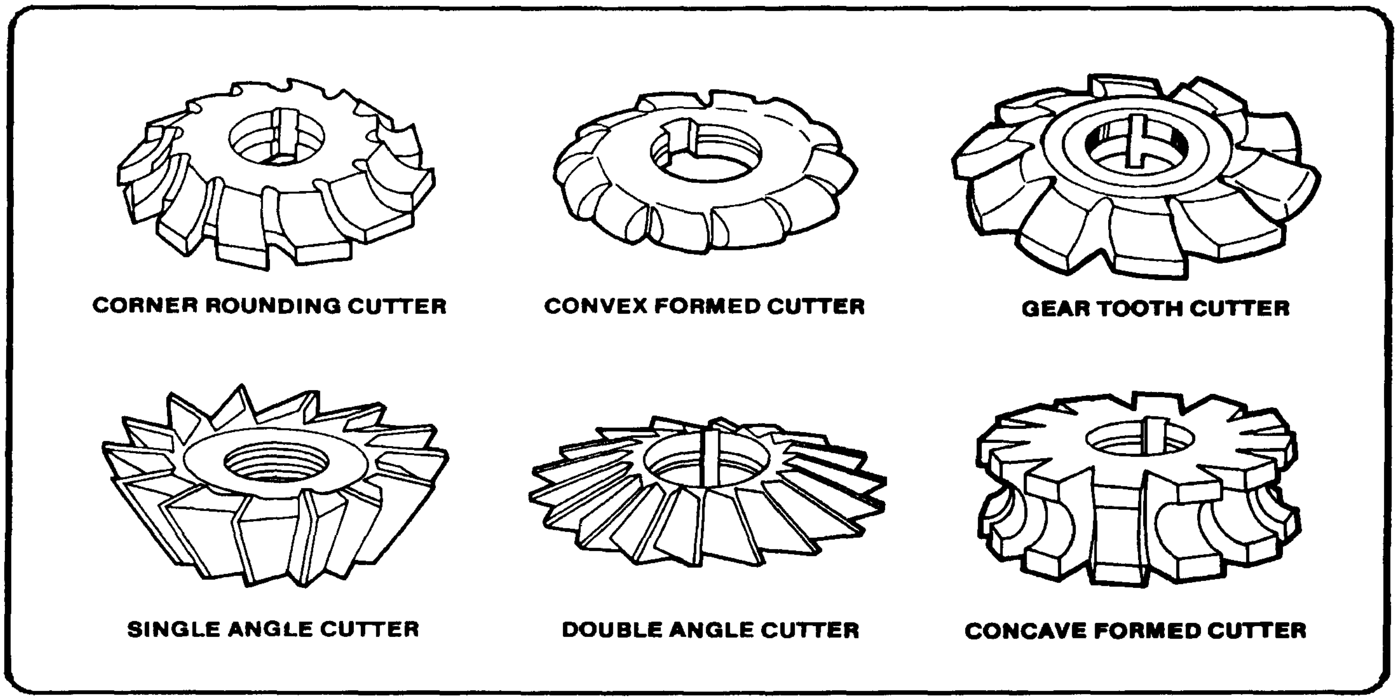


Figure 8-8. Angle, concave, convex, corner,and gear cutters.

Corner Rounding Milling Cutter

The corner-rounding milling cutter is a formed tooth cutter used for milling rounded corners on workplaces up to and including one-quarter of a circle. The size of the cutter is specified by the radius of the circular form the cutter produces, such as concave and convex cutters generally used for such work as finishing spur gears, spiral gears, and worm wheels. They may also be used to cut ratchets and spline shafts.

Special Shaped-Formed Milling Cutter

Formed milling cutters have the advantage of being adaptable to any specific shape for special operations. The cutter is made specially for each specific job. In the field, a fly cutter is formed by grinding a single point lathe cutter bit for mounting in a bar, holder, or fly cutter arbor. The cutter can be sharpened many times without destroying its shape.

Selection of Milling Cutters

Consider the following when choosing milling cutters:

- High-speed steel, stellite, and cemented carbide cutters have a distinct advantage of being capable of rapid production when used on a machine that can reach the proper speed.
- 45° angular cuts may either be made with a 45° single-angle milling cutter while the workpiece is held in a swivel vise, or with an end milling cutter while the workpiece is set at the required angle in a universal vise.
- The harder the material, the greater will be the heat that is generated in cutting. Cutters should be selected for their heat-resisting properties,
- Use a coarse-tooth milling cutter for roughing cuts and a finer-toothed milling cutter for light cuts and finishing operations.
- When milling stock to length, the choice of using a pair of side milling cutters to straddle the workpiece, a single-side milling cutter, or an end milling cutter will depend upon the number of pieces to be cut.
- Some operations can be done with more than one type of cutter such as in milling the square end on a shaft or reamer shank. In this case, one or two side milling cutters, a fly cutter, or an end milling cutter may be used. However, for the majority of operations, cutters are specially designed and named for the operation they are to accomplish.

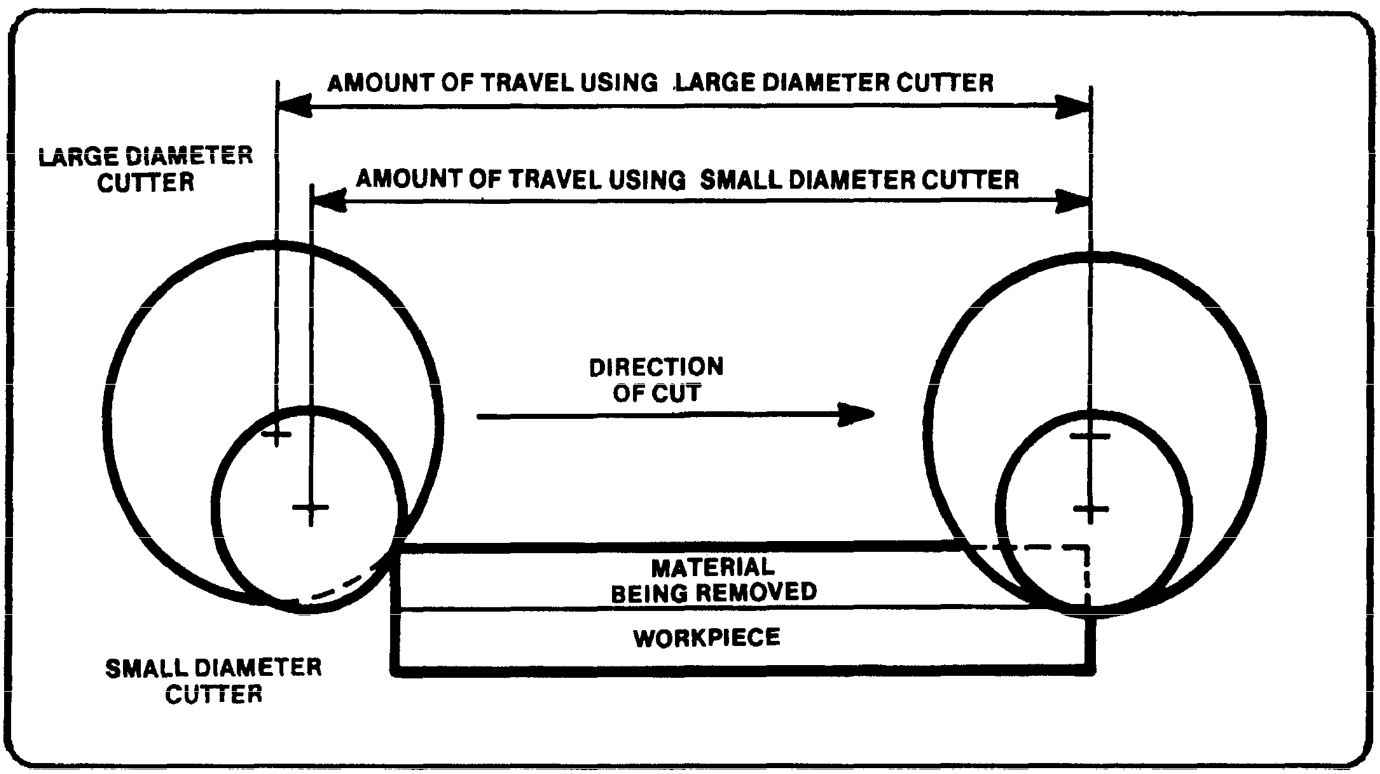


Figure 8-9. Effect of milling cutter diameter on workpiece travel.

- The milling cutter should be small enough in diameter so that the pressure of the cut will not cause the workpiece to be sprung or displaced while being milled.

Size of Milling Cutter

- In selecting a milling cutter for a particular job, choose one large enough to span the entire work surface so the job can be done with a single pass. If this cannot be done, remember that a small diameter cutter will pass over a surface in a shorter time than a large diameter cutter which is fed at the same speed. This fact is illustrated in Figure 8-9.

Care and Maintenance of Milling Cutters

- The life of a milling cutter can be greatly prolonged by intelligent use and proper storage. General rules for the care and maintenance of milling cutters are given below.
- New cutters received from stock are usually wrapped in oil paper which should not be removed until the cutter is used.
- Take care to operate the machine at the proper speed for the cutter being used, as excessive speed will cause the cutter to wear rapidly from overheating.
- Take care to prevent the cutter from striking the hard jaws of the vise, chuck, clamping bolts, or nuts.
- Whenever practical, use the proper cutting oil on the cutter and workpiece during operations, since lubrication helps prevent overheating and cutter wear.
- Keep cutters sharp. Dull cutters require more power to drive and this power, being transformed into heat, softens the cutting edges. Dull cutters should be marked as such and set aside for grinding. For further information on cutter grinding, refer to Chapter 5, Grinding Machines.
- Thoroughly clean and lightly coat milling cutters with oil before storing.
- Place cutters in drawers or bins so that their cutting edges will not strike each other. Hang small cutters on hooks or pegs, and set large cutters on end. Place taper and straight shank cutters in separate drawers, bins, or racks provided with suitable sized holes to receive the shanks.
- Never operate a cutter backwards. Due to the clearance angle, the cutter will rub, producing a great deal of friction. Operating the cutter backward may result in cutter breakage.

ARBORS

Milling machine arbors are made in various lengths and in standard diameters of 7/8,1,1 1/4, and 1 1/2 inch. The shank is made to fit the taper hole in the spindle while the other end is threaded.

NOTE: The threaded end may have left or right-handed threads.

The milling machine spindle may be self-holding or self-releasing. The self-holding taper is held in the spindle by the high wedging force. The spindle taper in most milling machines is self-releasing; tooling must be held in place by a draw bolt extending through the center of the spindle.

Arbors are supplied with one of three tapers to fit the milling machine spindle: the Standard Milling Machine taper, the Brown and Sharpe taper, and the Brown and Sharpe taper with tang (Figure 8-10).

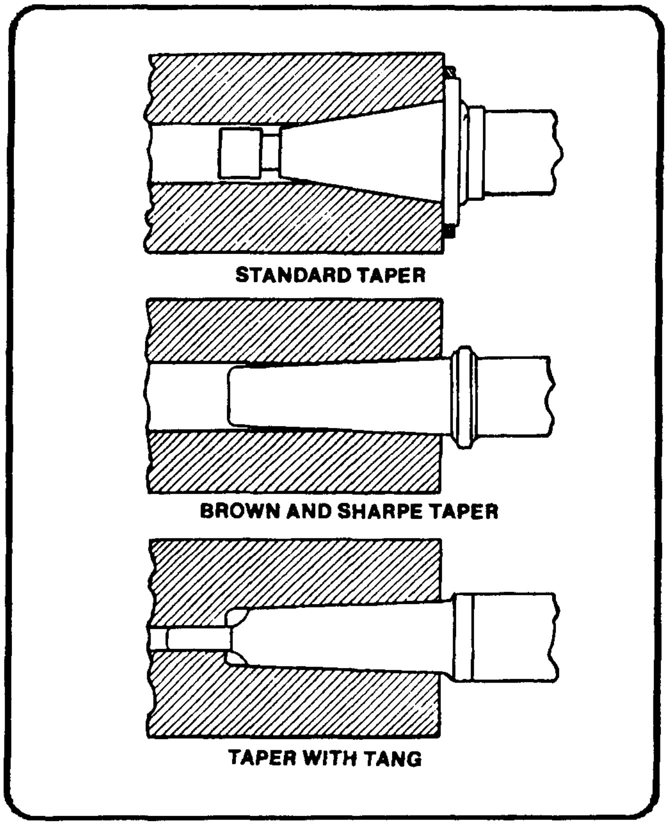


Figure 8-10. Tapers used for milling machine arbors.

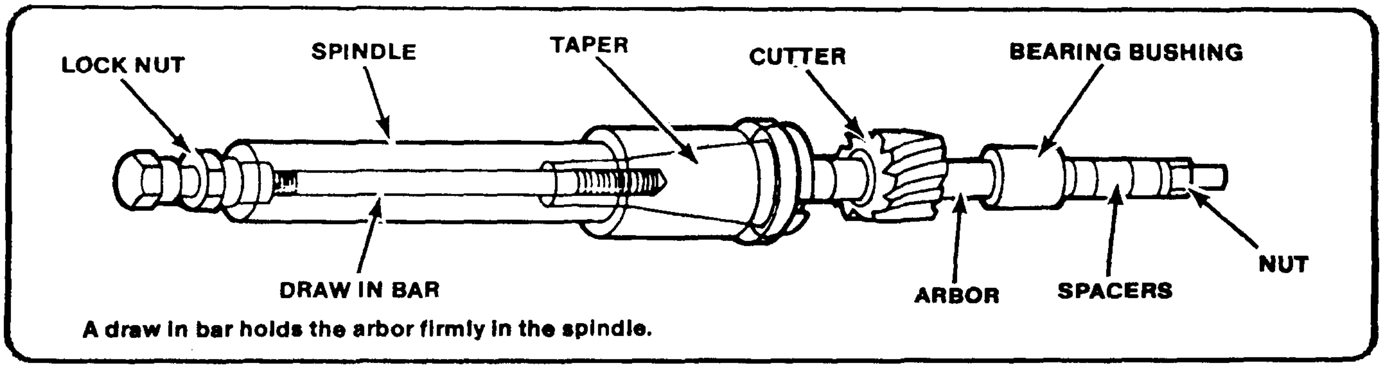


Figure 8-11. Standard milling machine arbor.

The Standard Milling Machine Taper is used on most machines of recent manufacture. See Figure 8-11. These tapers are identified by the number 30, 40, 50, or 60. Number 50 is the most commonly used size on all modern machines.

The Brown and Sharpe taper is found mostly on older machines. Adapters or collets are used to adapt these tapers to fit machines whose spindles have Standard Milling Machine tapers.

The Brown and Sharpe taper with tang is used on some older machines. The tang engages a slot in the spindle to assist in driving the arbor,

Standard Milling Machine Arbor

The standard milling machine arbor has a tapered, cylindrical shaft with a standard milling taper on the driving end and a threaded portion on the opposite end to receive the arbor nut. One or more milling cutters may be placed on the straight cylindrical portion of the arbor and held in position by sleeves and the arbor nut. The standard milling machine arbor is usually splined and keys are used to lock each cutter to the arbor shaft. These arbors are supplied in three styles, various lengths and, standard diameters.

The most common way to fasten the arbor in the milling machine spindle is to use a draw bar. The bar threads into the taper shank of the arbor to draw the taper into the spindle and hold it in place. Arbors secured in this manner are removed by backing out the draw bar and tapping the end of the bar to loosen the taper.

The end of the arbor opposite the taper is supported by the arbor supports of the milling machine. One or more supports reused depending on the length of the arbor and the degree of rigidity required. The end may be supported by a lathe center bearing against the arbor nut or by a bearing surface 0f the arbor fitting inside a bushing of the arbor support.

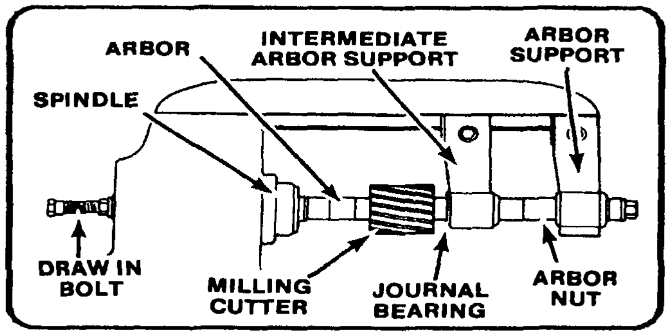


Figure 8-12. Arbor installation.

The arbor may also be firmly supported as it turns in the arbor support bearing suspended from the over-arm (Figure 8-12).

Typical milling arbors are illustrated in Figure 8-13. Listed on the next page are several types of Style C arbors.

Style A has a cylindrical pilot on the end that runs in a bronze bearing in the arbor support. This style is mostly used on small milling machines or when maximum arbor support clearance is required.

Style B is characterized by one or more bearing collars that can be positioned to any part of the arbor. This allows the bearing support to be positioned close to the cutter, to-obtain rigid setups in heavy duty milling operations).

Style C arbors are used to mount the smaller size milling cutters, such as end mills that cannot be bolted directly on the spindle nose. Use the shortest arbor possible for the work.

Screw Arbor

Screw arbors are used to hold small cutters that have threaded holes. See Figure 8-14. These arbors have a taper next to the threaded portion to provide alignment and support for tools that require a nut to hold them against a taper surface. A right-hand threaded arbor must be used for right-hand cutters while a left-hand threaded arbor is used to mount left-hand cutters.

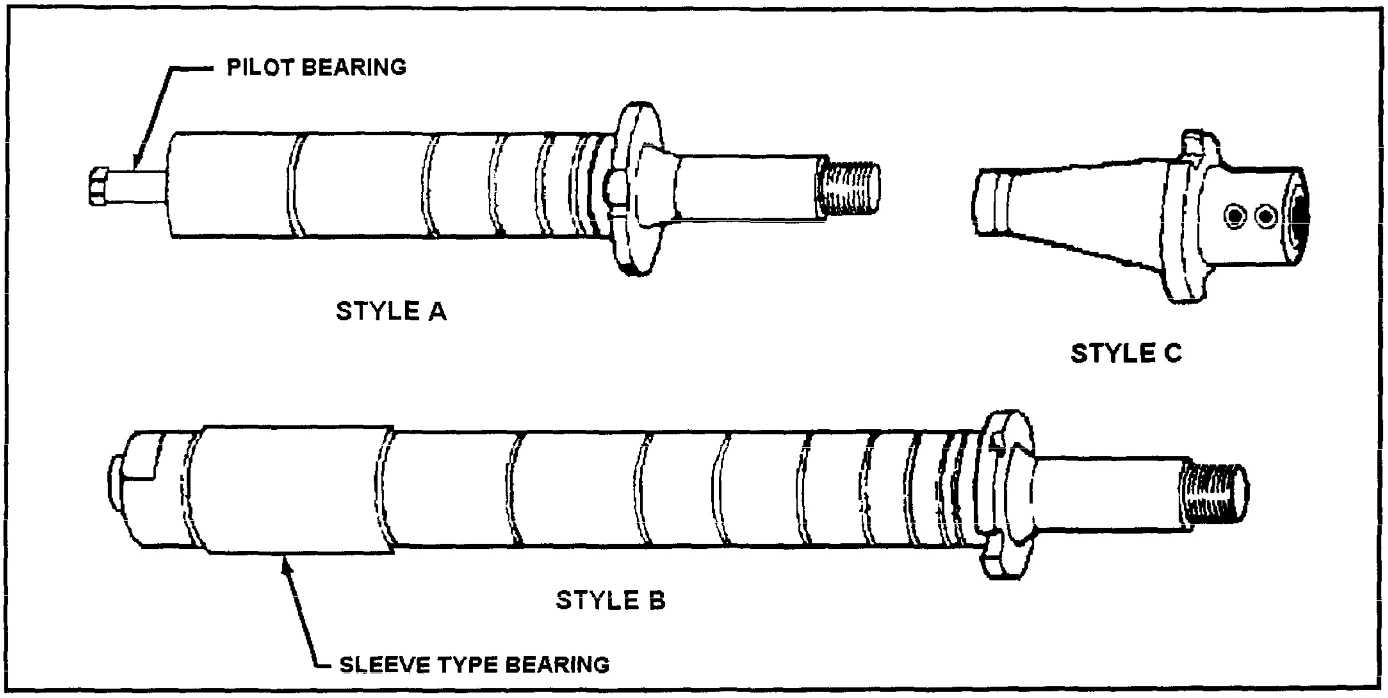


Figure 8-13. Typical milling arbors.

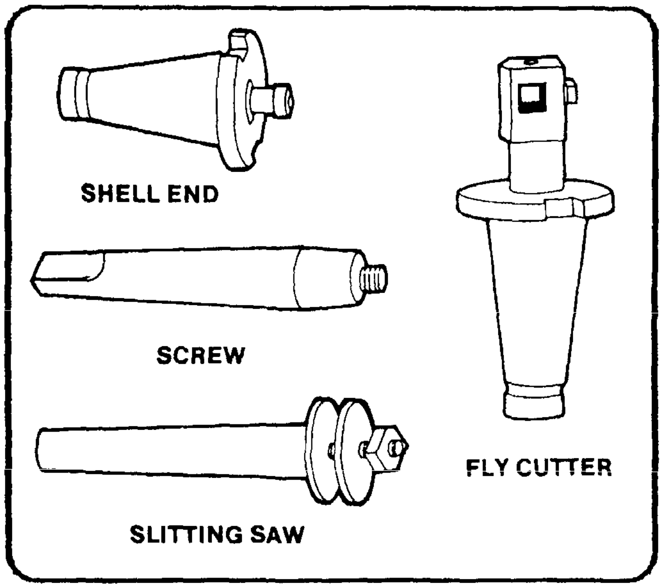


Figure 8-14. Arbor variations.

Screw arbors are used to hold small cutters that have threaded holes. These arbors have a taper next to the threaded portion to provide alignment and support for tools that require a nut to hold them against a taper surface. A right-hand threaded arbor must be used for right-hand cutters while a left-hand threaded arbor is used to mount left-hand cutters.

The slitting saw milling cutter arbor (Figure 8-14) is a short arbor having two flanges between which the milling cutter is secured by tightening a clamping nut. This arbor is used to hold metal slitting saw milling cutters used for slotting, slitting, and sawing operations.

The shell end milling cutter arbor has a bore in the end in which shell end milling cutters fit and are locked in place by means of a cap screw.

The fly cutter arbor is used to support a single-edge lathe, shaper, or planer cutter bit for boring and gear cutting operations on the milling machine.

COLLETS, SPINDLE ADAPTERS, AND QUICK-CHANGE TOOLING

Description

Milling cutters that contain their own straight or tapered shanks are mounted to the milling machine spindle with collets, spindle adapters, and quick-change tooling which adapts the cutter shank to the spindle.

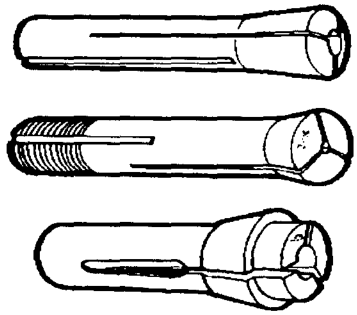


Figure 8-15. Solid and spring collets.

Collets

A collet is a form of a sleeve bushing for reducing the size of the hole in the milling machine spindle so that small shank tools can be fitted into large spindle recesses (Figure 8-15). They are made in several forms, similar to drilling machine sockets and sleeves, except that their tapers are not alike.

Spindle Adapters

A spindle adapter is a form of a collet having a standardized spindle end. They are available in a wide variety of sizes to accept cutters that cannot be mounted on arbors. They are made with either the Morse taper shank or the Brown and Sharpe taper with tang having a standard spindle end (Figure 8-16).

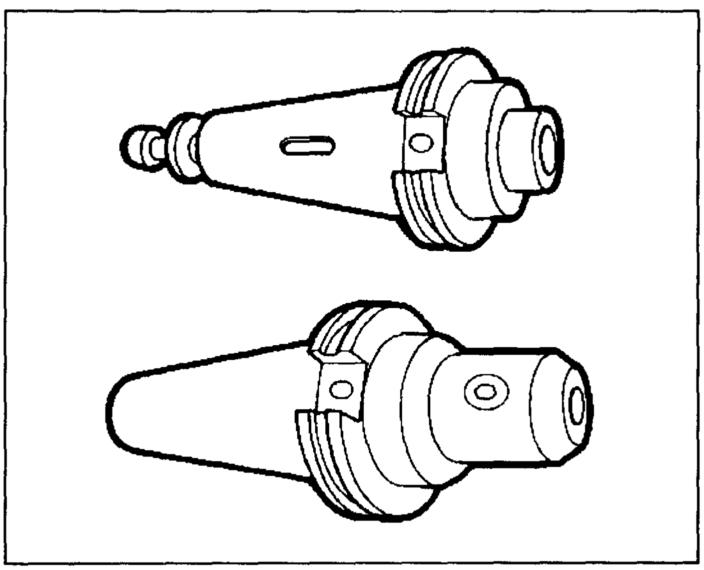


Figure 8-16. Milling machine adaptors.

Chuck Adapter

A chuck adapter (Figure 8-17) is used to attach chucks to milling machines having a standard spindle end. The collet holder is sometimes referred to as a collet chuck. Various forms of chucks can be fitted to milling machines spindles for holding drills, reamers, and small cutters for special operations.

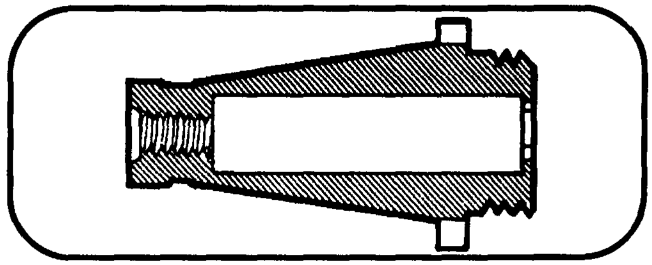


Figure 8-17. Chuck adaptor.

Quick-Change Tooling

The quick-change adapter mounted on the spindle nose is used to speed up tool changing. Tool changing with this system allows you to set up a number of milling operations such as drilling, end milling, and boring without changing the setup of the part being machined. The tool holders are mounted and removed from a master holder mounted to the machine spindle by means of a clamping ring (Figure 8-18).

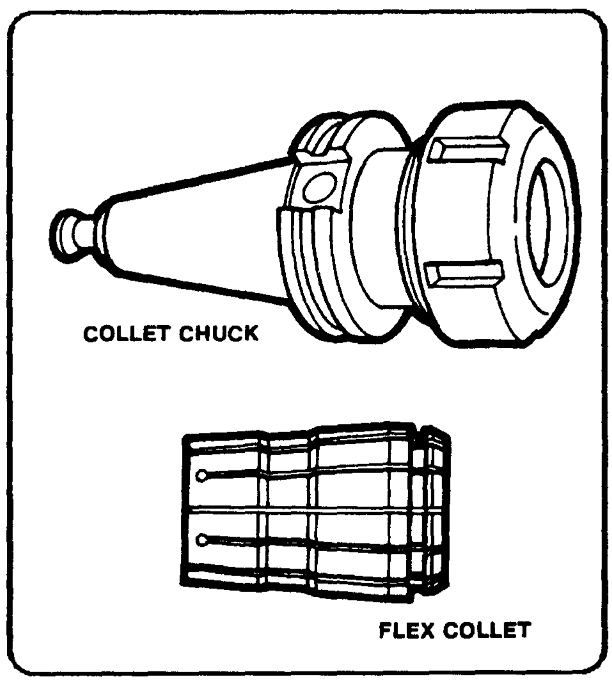


Figure 8-18. Quick-change adaptor and tool holder.

VISES

Either a plain or swivel-type vise is furnished with each milling machine. The plain vise, similar to the machine table vise, is used for milling straight workplaces and is bolted to the milling machine table either at right angles or parallel to the machine arbor. The swivel vise can be rotated and contains a scale graduated in degrees at its base to facilitate milling workplaces at any angle on a horizontal plane. The universal vise, which may be obtained as extra equipment, is designed so that it can be set at both horizontal and vertical angles. This type of vise maybe used for flat and angular milling. The all-steel vise is the strongest setup because the workpiece is clamped closer to the table. The vise can securely fasten castings, forgings, and rough-surfaced workplaces. The jaw can be positioned in any notch on the two bars to accommodate different shapes and sizes. The air or hydraulically operated vise is used more often in production work. This type of vise eliminates tightening by striking the crank with a lead hammer or other soft face hammer. See page 4-13 for examples of various vises.

ADJUSTABLE ANGLE PLATE

The adjustable angle plate is a workpiece holding device, similar to the universal vise in operation. Workpieces are mounted to the angle plate with T-bolts and clamps in the same manner used to fasten workplaces to the worktable of the milling machine. The angle plate can be adjusted to any angle so that bevels and tapers can be cut without using a special milling cutter or an adjustable cutter head.

INDEXING FIXTURE

The index fixture (Figure 8-19) consists of an index head, also called a dividing head, and footstock which is similar to the tailstock of a lathe. The index head and footstock attach to the worktable of the milling machine by T-slot bolts. An index plate containing graduations is used to control the rotation of the index head spindle. The plate is fixed to the index head, and an index crank, connected to the index head spindle by a worm gear and shaft. Workpieces are held between centers by the index head spindle and footstock. Workpieces may also be held in a chuck mounted to the index head spindle or may be fitted directly into the taper spindle recess of some indexing fixtures. There are many variations of the indexing fixture. Universal index head is the name applied to an index head designed to permit power drive of the spindle so that helixes may be cut on the milling machine. Gear cutting attachment is another name applied to an indexing fixture; in this case, one that is primarily intended for cutting gears on the milling machine.

HIGH-SPEED MILLING ATTACHMENT

The rate of spindle speed of the milling machine may be increased from 1 1/2 to 6 times by using the high-speed milling attachment. This attachment is essential when using cutters and twist drills which must be driven at a high rate of speed in order to obtain an efficient surface speed. The attachment is clamped to the column of the machine and is driven by a set of gears from the milling machine spindle.

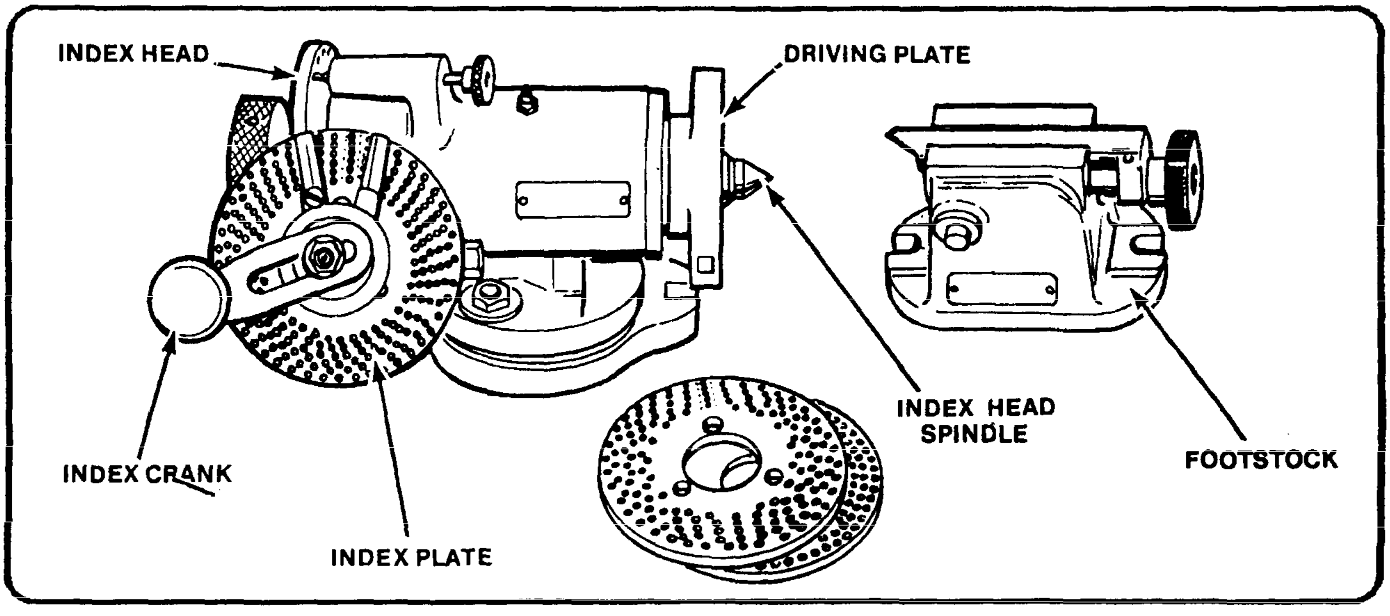


Figure 8-19. Indexing fixture.

VERTICAL SPINDLE ATTACHMENT

This attachment converts the horizontal spindle of a horizontal milling machine to a vertical spindle. It is clamped to the column and driven from the horizontal spindle. It incorporates provisions for setting the head at any angle, from the vertical to the horizontal, in a plane at right angles to the machine spindle. End milling and face milling are more easily accomplished with this attachment, because the cutter and the surface being cut are in plain view.

UNIVERSAL MILLING ATTACHMENT

This device is similar to the vertical spindle attachment but is more versatile. The butterhead can be swiveled to any angle in any plane, whereas the vertical spindle attachment only rotates in one place from horizontal to vertical.

ROTARY TABLE OR CIRCULAR MILLING ATTACHMENT

This attachment consists of a circular worktable containing T-slots for mounting workplaces. The circular table revolves on a base attached to the milling machine worktable. The attachment can be either hand or power driven, being connected to the table drive shaft if power driven. It may be used for milling circles, angular indexing, arcs, segments, circular slots, grooves, and radii, as well as for slotting internal and external gears. The table of the attachment is divided in degrees (Figure 8-20).

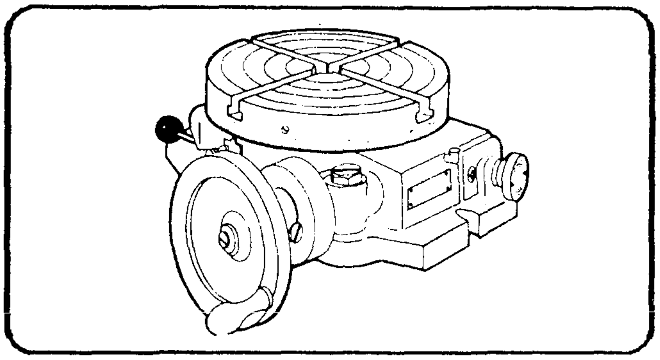


Figure 8-20. Rotary table (circular milling attachment)

OFFSET BORING HEAD

Boring, an operation that is too often restricted to a lathe, can be done easily on a milling machine. The offset boring head is an attachment that fits to the milling machine spindle and permits most drilled holes to have a better surface finish and greater diameter accuracy.

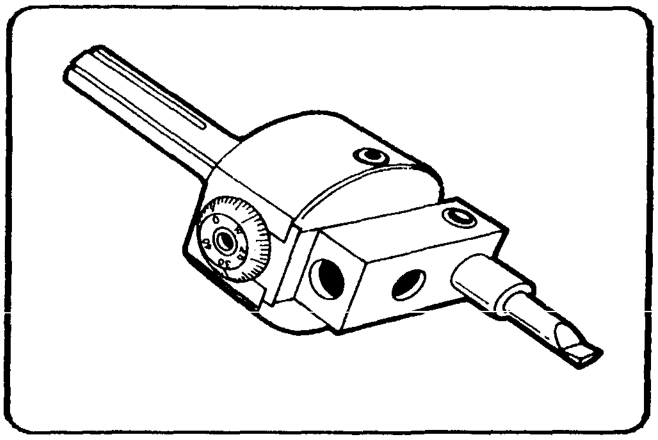


Figure 8-21. Offset boring head.

OFFSET BORING HEAD AND TOOLS

Figure 8-21 shows an offset boring head. Note that the boring bar can be adjusted at a right angle to the spindle axis. This feature makes it possible to position the boring cutter accurately to bore holes of varying diameters.

This adjustment is more convenient than adjusting the cutter in the boring bar holder or changing the boring bar. Another advantage of the offset boring head is the fact that a graduated micrometer collar allows the tool to be moved accurately a specified amount (usually in increments of 0.001) without the use of a dial indicator or other measuring device.

NOTE: On some boring heads, the reading on the tool slide is a direct reading. On other boring heads, the tool slide advances twice the amount shown on the micrometer dial.

MOUNTING AND INDEXING WORK

An efficient and positive method of holding workplaces to the milling machine table is important if the machine tool is to be used to its fullest advantage. The most common methods of holding are clamping a workpiece to the table, clamping a workpiece to the angle plate, clamping the workpiece in fixtures, holding a workpiece between centers, holding the workpiece in a chuck, and holding the workpiece in a vise. Page 4-13 of this manual shows a variety of mounting and holding devices. Regardless of the method used in holding, there are certain factors that should be observed in every case. The workpiece must not be sprung in clamping, it must be secured to prevent it from springing or moving away from the cutter, and it must be so aligned that it may be correctly machined T-slots, Milling machine worktables are provided with several T-slots which are used either for clamping and locating the workpiece itself or for

mounting the various holding devices and attachments. These T-slots extend the length of the table and are parallel to its line of travel. Most milling machine attachments, such as vises and index fixtures, have keys or tongues on the underside of their bases so that they may be located correctly in relation to the T-slots.

METHODS OF MOUNTING WORKPIECES

Clamping Workpieces to the Table

When clamping a workpiece to the worktable of the milling machine, the table and the workpiece should be free from dirt and burrs. Workpieces having smooth machined surfaces may be camped directly to the table, provided the cutter does not come in contact with the table surface during milling. When clamping workplaces with unfinished surfaces in this way, the table face should be protected from damage by using a shim under the workpiece. Paper, plywood, and sheet metal are shim materials. Clamps should be located on both sides of the workpiece if possible to give a full bearing surface. These clamps are held by T-slot bolts inserted in the T-slots of the table. Clamp supports must be the same height as the workpiece. Never use clamp supports that are lower than the workpiece. Adjustable step blocks are extremely useful to raise the clamps, as the height of the clamp bar may be adjusted to ensure maximum clamping pressure. Clamping bolts should be placed as near to the workpiece as possible so that the full advantage of the fulcrum principle may be obtained. When it is necessary to place a clamp on an overhanging part, a support should be provided between the overhang and the table to prevent springing or possible breakage. A stop should be placed at the end of the workpiece where it will receive the thrust of the cutter when heavy cuts are being taken.

Clamping a Workpiece to the Angle Plate

Workpieces clamped to the angle plate may be machined with surfaces parallel, perpendicular, or at an angle to a given surface. When using this method of holding a workpiece, precautions should be taken similar to those mentioned for clamping work directly to the table. Angle plates are either adjustable or nonadjustable and are generally held in alignment by keys or tongues that fit into the table T-slots.

Clamping Workpieces in Fixtures

Fixtures are generally used in production work where a number of identical pieces are to be machined. The design of the fixture depends upon the shape of the piece and the operations to be performed. Fixtures are always constructed to secure maximum clamping surfaces and are built to use a minimum number of clamps or bolts in order to reduce the setup time required. Fixtures should always be provided with keys to assure positive alignment with the table T-slots.

Holding Workpieces Between Centers

The indexing fixture is used to support workplaces which are centered on both ends. When the piece has been previously reamed or bored, it may be pressed upon a mandrel and then mounted between the centers.

Two types of mandrels may be used for mounting workplaces between centers. The solid mandrel is satisfactory for many operations, while one having a shank tapered to fit into the index head spindle is preferred in certain cases.

A jackscrew is used to prevent springing of long slender workplaces held between centers or workplaces that extend some distance from the chuck.

Workpieces mounted between centers are fixed to the index head spindle by means of a lathe dog. The bent tail of the dog should be fastened between the setscrews provided in the driving center clamp in such a manner as to avoid backlash and prevent springing the mandrel. When milling certain types of workpieces, a milling machine dog is held in a flexible ball joint which eliminates shake or spring of the dog or the workpiece. The flexible ball joint allows the tail of the dog to move in a radius along the axis of the workpiece, making it particularly useful in the rapid milling of tapers.

Holding Workpieces in a Chuck

Before screwing the chuck to the index head spindle, it should be cleaned and any burrs on the spindle or chuck removed. Burrs may be removed with a smooth-cut, three cornered file or scraper, while cleaning should be accomplished with a piece of spring steel wire bent and formed to fit the angle of the threads. The chuck should not be tightened on the spindle so tightly that a wrench or bar is required to remove it. Cylindrical workplaces held in the universal chuck may be checked for trueness by using a test indicator mounted upon a base resting upon the milling machine table. The indicator point should contact the circumference of small diameter workpieces, or the circumference and exposed face of large diameter pieces. While checking, the workpiece should be revolved by rotating the index head spindle.

Holding Workpieces in the Vise

As previously mentioned, five types of vises are manufactured in various sizes for holding milling machine workplaces. These vises have locating keys or tongues on the underside of their bases so they may be located correctly in relation to the T-slots on the milling machine table (Figure 8-22).

The plain vise similar to the machine table vise is fastened to the milling machine table. Alignment with the milling machine table is provided by two slots at right angles to each other on the underside of the vise. These slots are fitted with removable keys that align the vise with the table T-slots either parallel to the machine arbor or perpendicular to the arbor.

The swivel vise can be rotated and contains a scale graduated in degrees at its base which is fastened to the milling machine table and located by means of keys placed in the T-slots. By loosening the bolts which clamp the vise to its graduated base, the vise may be moved to hold the workpiece at any angle in a horizontal plane. To set a swivel vise accurately with the machine spindle, a test indicator should be clamped to the machine arbor and a check made to determine the setting by moving either the transverse or the longitudinal feeds, depending upon the position of the vise jaws. Any deviation as shown by the test indicator should be corrected by swiveling the vise on its base.

The universal vise is used for work involving compound angles, either horizontally or vertically. The base of the vise contains a scale graduated in degrees and can rotate 360° in the horizontal plane and 90° in the vertical plane. Due to the flexibility of this vise, it is not adaptable for heavy milling.

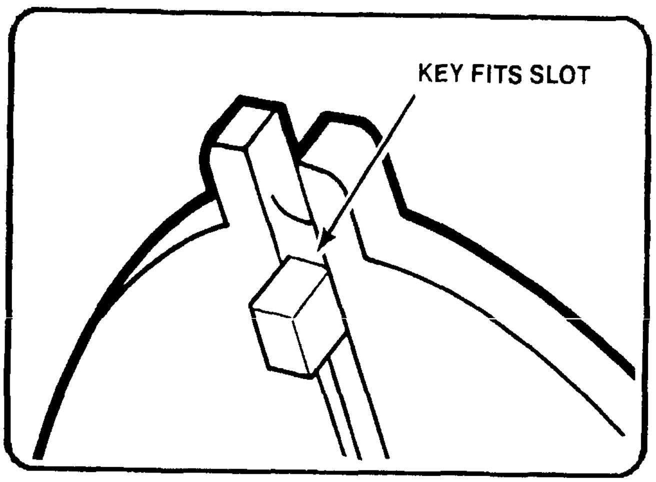


Figure 8-22. Locating key on vises.

The all-steel vise is the strongest setup where the workpiece is clamped close to the table. This vise can securely fasten castings, forgings, and rough-surface workplaces. The jaws can be positioned in any notch on the two bars to accommodate different shapes and sizes.

The air or hydraulically operated vise is used more often in production work. This type of vise eliminates the tightening by striking the crank with a lead hammer or other soft face hammer.

When rough or unfinished workplaces are to be vise mounted, a piece of protecting material should be placed between the vise and the workpiece to eliminate marring by the vise jaws.

When it is necessary to position a workpiece above the vise jaws, parallels of the same size and of the proper height should be used. These parallels should only be high enough to allow the required cut, as excessive raising reduces the holding ability of the jaws. When holding a workpiece on parallels, a soft hammer should be used to tap the top surface of the piece after the vise jaws have been tightened. This tapping should be continued until the parallels cannot be moved by hand. After the workpiece is set, additional tightening of the vise should not be attempted, as such tightening has a tendency to raise the work off the parallels. Correct selection of parallels is illustrated in Figure 8-23.

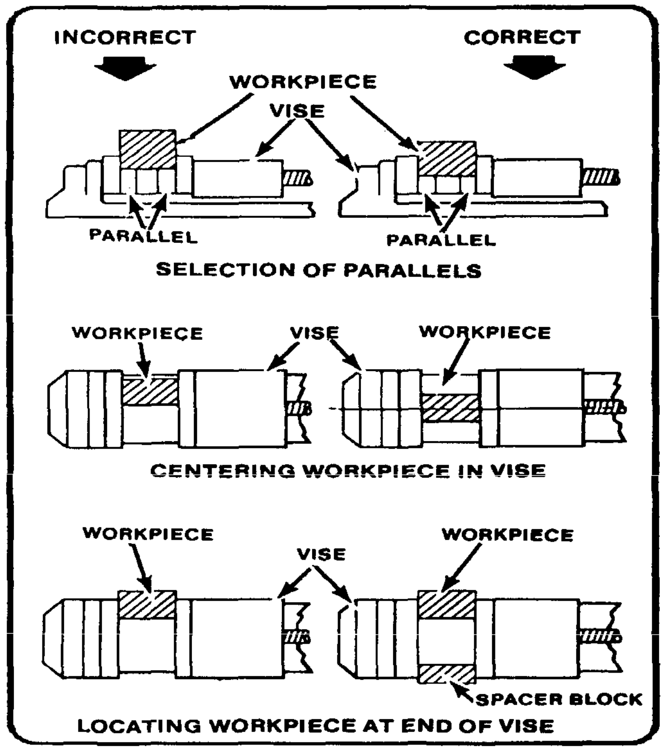


Figure 8-23. Mounting workpiece in the vise.

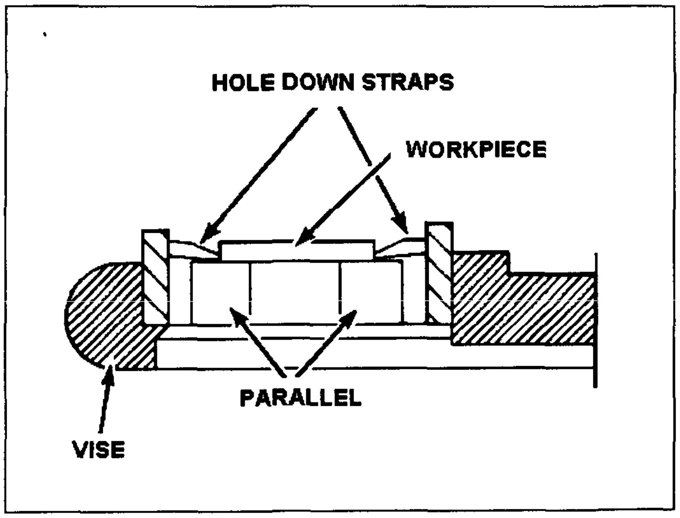


Figure 8-24. Application of holddown straps.

Whenever possible, the workpiece should be clamped in the center of the vise jaws. However, when necessary to mill a short workpiece which must be held at the end of the vise, a spacing block of the same thickness as the piece should be placed at the opposite end of the jaws. This will avoid strain on the movable jaw and prevent the piece from slipping. If the workpiece is so thin that it is impossible to let it extend over the top of the vise, hold down straps are generally used. See Figure 8-24. These straps are hardened pieces of steel, having one vertical side tapered to form an angle of about 92° with the bottom side and the other vertical side tapered to a narrow edge. By means of these tapered surfaces, the workpiece is forced downward into the parallels, holding them firmly and leaving the top of the workpiece fully exposed to the milling cutter.

Indexing

Indexing is the process of evenly dividing the circumference of a circular workpiece into equally spaced divisions, such as in cutting gear teeth, cutting splines, milling grooves in reamers and taps, and spacing holes on a circle. The index head of the indexing fixture is used for this purpose.

Index Head

The index head of the indexing fixture (Figure 8-19) contains an indexing mechanism which is used to control the rotation of the index head spindle to space or divide a workpiece accurately. A simple indexing mechanism consists of a 40-tooth worm wheel fastened to the index head spindle, a single-cut worm, a crank for turning the wormshaft, and an index plate and sector. Since there are 40 teeth in the worm wheel, one turn of the index crank causes the worm, and consequently, the index head spindle to make 1/40 of a turn; so 40 turns of the index crank revolve the spindle one full turn.

Index Plate

The indexing plate (Figure 8-25) is a round plate with a series of six or more circles of equally spaced holes; the index pin on the crank can be inserted in any hole in any circle. With the interchangeable plates regularly furnished with most index heads, the spacing necessary for most gears, boltheads, milling cutters, splines, and so forth can be obtained. The following sets of plates are standard equipment:

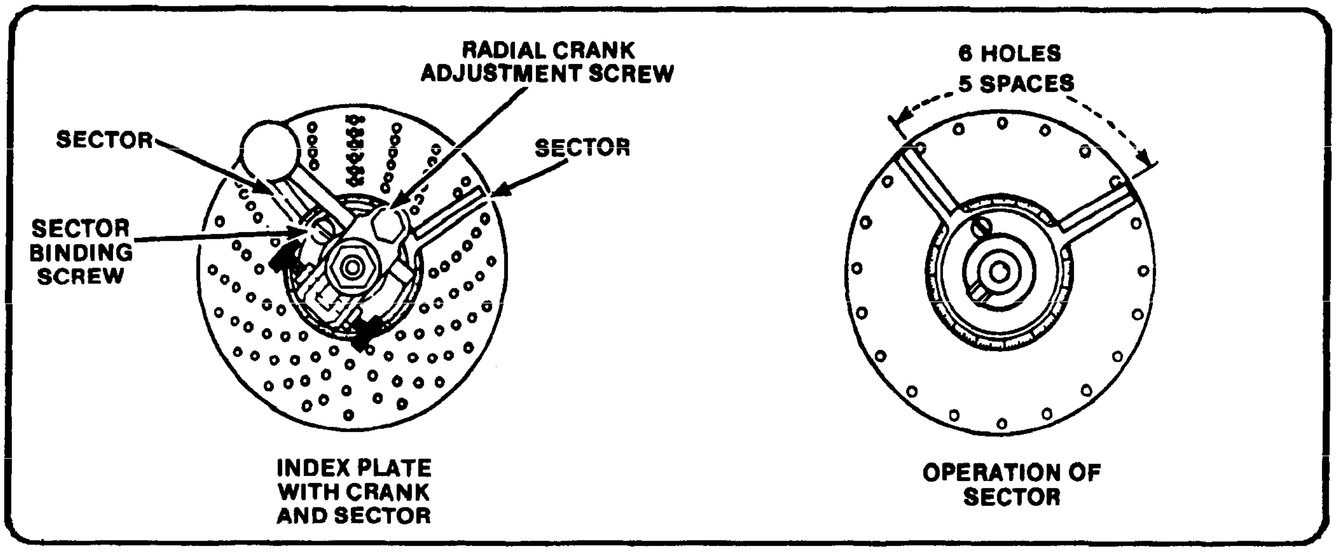


Figure 8-25. Index plate and sector.

Brown and Sharpe type consists of 3 plates of 6 circles each drilled as follows:

Plate I -15, 16, 17, 18, 19, 20 holes

Plate 2-21, 23, 27, 29, 31, 33 holes

Plate 3-37, 39, 41, 43,47,49 holes

Cincinnati type consists of one plate drilled on both sides with circles divided as follows:

First side -24, 25, 28, 30, 34, 37,38, 39,41,42,43 holes

Second side -46, 47, 49, 51, 53, 54, 57, 58, 59, 62, 66 holes

Sector

The sector (Figure 8-25) indicates the next hole in which the pin is to be inserted and makes it unnecessary to count holes when moving the index crank after each cut. It consists of two radial, beveled arms which can be set at any angle to each other and then moved together around the center of the index plate. Suppose that, as shown in Figure 8-25, it is desired to make a series of cuts, moving the index crank 1 1/4 turns after each cut. Since the circle illustrated has 20 holes, turn the crank one full turn plus five spaces after each cut, Set the sector arms to include the desired fractional part of a turn or five spaces between the beveled edges of its arms, as shown. If the first cut is taken with the index pin against the left-hand arm, to take the next cut, move the pin once against the right-hand arm of the sector. Before taking the second cut, move the arms so that the left-hand arm is again against the pin; this moves the right-hand arm another five spaces ahead of the pin. Then take the second cut, and repeat the operation until all the cuts have been completed.

NOTE: It is good practice always to index clockwise on the plate to eliminate backlash.

Plain Indexing

The following principles apply to basic indexing of workpieces:

Suppose it is desired to mill a project with eight equally spaced teeth. Since 40 turns of the index crank will turn the spindle one full turn, l/8th of 40 or 5 turns of the crank after each cut will space the gear for 8 teeth, If it is desired to space equally for 10 teeth, 1/10 of 40 or 4 turns would produce the correct spacing.

The same principle applies whether or not the divisions required divide equally into 40, For example, if it is desired to index for 6 divisions, 6 divided into 40 equals 6 2/3 turns; similarly, to index for 14 spaces, 14 divided into 40 equals 2 6/7 turns. These examples may be multiplied indefinitely and from them the following rule is derived: to determine the number of turns of the index crank needed to obtain one division of any number of equal divisions on the workpiece, divide 40 by the number of equal divisions desired (provided the worm wheel has 40 teeth, which is standard practice).

Direct Indexing

The construction of some index heads permits the worm to be disengaged from the worm wheel, making possible a quicker method of indexing called direct indexing. The index head is provided with a knob which, when turned through part of a revolution, operates an eccentric and disengages the worm.

Direct indexing is accomplished by an additional index plate fastened to the index head spindle. A stationary plunger in the index head fits the holes in this index plate. By moving this plate by hand to index directly, the spindle and the workpiece rotate an equal distance. Direct index plates usually have 24 holes and offer a quick means of milling squares, hexagons, taps, and so forth. Any number of divisions which is a factor of 24 can be indexed quickly and conveniently by the direct indexing method.

Differential Indexing

Sometimes, a number of divisions is required which cannot be obtained by simple indexing with the index plates regularly supplied. To obtain these divisions, a differential index head is used. The index crank is connected to the wormshaft by a train of gears instead of a direct coupling as with simple indexing. The selection of these gears involves calculations similar to those used in calculating change gear ratio for lathe thread cutting.

Indexing in Degrees

Workpieces can be indexed in degrees as well as fractions of a turn with the usual index head. There are 360 degrees in a complete circle and one turn of the index crank revolves the spindle 1/40 or 9 degrees. Therefore, 1/9 turn of the crank rotates the spindle 1 degree. Workpieces can therefore be indexed in degrees by using a circle of holes divisible by 9. For example, moving the crank 2 spaces on an 18-hole circle, 3 spaces on a 27-hole circle, or 4 spaces on a 36-hole circle

will rotate the spindle 1 degree, Smaller crank movements further subdivide the circle: moving 1 space on an 18-hole circle turns the spindle 1/2 degree (30 minutes), 1 space on a 27-hole circle turns the spindle 1/3 degree (20 minutes), and so forth.

Indexing Operations

The following examples show how the index plate is used to obtain any desired part of a whole spindle turn by plain indexing,

- Milling a hexagon. Using the rule previously given, divide 40 by 6 which equals 6 2/3 turns, or six full turns plus 2/3 of a turn or any circle whose number is divisible by 3. Take the denominator which is 3 into which of the available hole circles it can be evenly divided. In this case, 3 can be divided into the available 18-hole circle exactly 6 times. Use this result 6 as a multiplier to generate the proportional fraction required.

Example: 2 x 6 = 12
3x6 = 18-

Therefore, 6 full turns of the crank plus 12 spaces on an 18-hole circle is the correct indexing for 6 divisions.

- Cutting a gear. To cut a gear of 52 teeth, using the rule again, divide 40 by 52. This means that less than one full turn is required for each division, 40/52 of a turn to be exact. Since a 52-hole circle is not available, 40/52 must be reduced to its lowest term which is 10/13. Take the denominator of the lowest term 13, and determine into which of the available hole circles it can be evenly divided. In this case, 13 can be divided into a 39-hole circle exactly 3 times. Use this result 3 as a multiplier to generate the proportional fraction required.

Example: 10 x 3 = 30
13 x 3 = 39

Therefore, 30 holes on a 39-hole circle is the correct indexing for 52 divisions. When counting holes, start with the first hole ahead of the index pin.

GENERAL MILLING OPERATIONS

GENERAL

Setup

The success of any milling operation depends, Before setting up a job, be sure that the to a great extent, upon judgment in setting up the job, workpiece, the table, the taper in the spindle, selecting the proper milling cutter, and holding the cutter by the best means under the circumstances Some fundamental practices have been proved by experience to be necessary for and the arbor or cutter shank are all clean and good results on all jobs. Some of these practices are mentioned be low...

- Before setting up a job, be sure that the workpiece, table, the taper in the spindle, and the arbor or cutter shank are free from chips, nicks, or burrs.
- Do not select a milling cutter of larger diameter than is necessary.
- Check the machine to see if it is in good running order and properly lubricated, and that it moves freely, but not too freely in all directions.
- Consider direction of rotation. Many cutters can be reversed on the arbor, so be sure you know whether the spindle is to rotate clockwise or counterclockwise.
- Feed the workpiece in a direction opposite the rotation of the milling cutter (conventional milling).
- Do not change feeds or speeds while the milling machine is in operation.
- When using clamps to secure a workpiece, be sure that they are tight and that the piece is held so it will not spring or vibrate under cut.
- Use a recommended cutting oil liberally.
- Use good judgment and common sense in planning every job, and profit from previous mistakes.
- Set up every job as close to the milling machine spindle as circumstances will permit.

Milling Operations

Milling operations may be classified under four general headings as follows:

- Face milling. Machining flat surfaces which are at right angles to the axis of the cutter,
- Plain or slab milling. Machining flat surfaces which are parallel to the axis of the cutter.
- Angular milling. Machining flat surfaces which are at an inclination to the axis of the cutter.
- Form milling. Machining surfaces having an irregular outline.

Special Operations

Explanatory names, such as sawing, slotting, gear cutting, and so forth have been given to special operations. Routing is a term applied to milling an irregular outline while controlling the workpiece movement by hand feed. Grooving reamers and taps is called fluting. Gang milling is the term applied to an operation in which two or more milling cutters are used together on one arbor. Straddle milling is the term given to an operation in which two milling cutters are used to straddle the workpiece and mill both sides at the same time.

SPEEDS FOR MILLING CUTTERS

The speed of milling is the distance in FPM at which the circumference of the cutter passes over the work. The spindle RPM necessary to give a desired peripheral speed depends on the size of the milling cutter. The best speed is determined by the kind of material being cut and the size and type of cutter used, width and depth of cut, finish required, type of cutting fluid and method of application, and power and speed available are factors relating to cutter speed.

Factors Governing Speed

There are no hard and fast rules governing the speed of milling cutters; experience has shown that the following factors must be considered in regulating speed:

- A metal slitting saw milling cutter can be rotated faster than a plain milling cutter having a broad face.
- Cutters having undercut teeth (positive rake) cut more freely than those having radial teeth (without rake); hence, they may run at higher speeds.
- Angle cutters must be run at slower speeds than plain or side cutters.
- Cutters with inserted teeth generally will stand as much speed as a solid cutter.
- A sharp cutter may be operated at greater speeds than a dull one.
- A plentiful supply of cutting oil will permit the cutter to run at higher speeds than without cutting oil

Selecting Proper Cutting Speeds

The approximate values given in Table 8-1 in Appendix A may be used as a guide for selecting the proper cutting speed. If the operator finds that the machine, the milling cutter, or the workpiece cannot be handled suitably at these speeds, immediate readjustments should be made.

Table 8-1 lists speeds for high-speed steel milling cutters. If carbon steel cutters are used, the speed should be about one-half the recommended speed in the table. If carbide-tipped cutters are used, the speed can be doubled.

If a plentiful supply of cutting oil is applied to the milling cutter and the workpiece, speeds can be increased 50 to 100 percent. For roughing cuts, a moderate speed and coarse feed often give best results; for finishing cuts, the best practice is to reverse these conditions, using a higher speed and lighter feed.

Speed Computation

The formula for calculating spindle speed in revolutions per minute is as follows:

$$\mathbf{RPM} = \frac{\mathbf{CS} \times 4}{\mathbf{D}}$$

Where **RPM** = Spindle speed (in revolutions per minute).

CS = cutting speed of milling cutter (in SFPM)

D = diameter of milling cutter (in inches)

For example, the spindle speed for machining a piece of steel at a speed of 35 SFPM with a cutter 2 inches in diameter is calculated as follows:

$$RPM = \frac{CSx4}{D} = \frac{35x4}{2} = \frac{140}{2} = 70\ RPM$$

Therefore, the milling machine spindle would be set for as near 70 RPM as possible.

Table 8-2 in Appendix A is provided to facilitate spindle speed computations for standard cutting speeds and standard milling cutters.

FEEDS FOR MILLING

The rate of feed, or the speed at which the workpiece passes the cutter, determines the time required for cutting a job. In selecting the feed. there are several factors which should be considered.

Forces are exerted against the workpiece, the cutter, and their holding devices during the cutting process. The force exerted varies directly with the amount of feed and depth of cut. and in turn are dependent upon the rigidity and power of the machine. Milling machines are limited by the power they can develop to turn the cutter and the amount of vibration they can resist when using coarse feeds and deep cuts. The feed and depth of the cut also depend upon the type of milling cutter being used. For example. deep cuts or coarse feeds should not be attempted when using a small diameter end milling cutter. Coarse cutters with strong cutting teeth can be fed at a faster rate because the chips maybe washed out more easily by the cutting oil.

Coarse feeds and deep cuts should not be used on a frail workpiece if the piece is mounted in such a way that its holding device is not able to prevent springing or bending.

Experience and judgment are extremely valuable in selecting the correct milling feeds. Even though suggested rate tables are given. remember that these are suggestions only. Feeds are governed by many variable factors, such as the degree of finish required. Using a coarse feed, the metal is removed more rapidly but the appearance and accuracy of the surface produced may not reach the standard desired for the finished product. Because of this fact. finer feeds and increased speeds are used for finer. more accurate finishes. while for roughing. to use a comparatively low speed and heavy feed. More mistakes are made on overspeeding and underfeeding than on underspeeding and overfeeding.

Overspeeding may be detected by the occurrence of a squeaking. scraping sound. If vibration (referred to as chattering) occurs in the milling machine during the cutting process. the speed should be reduced and the feed increased. Too much cutter clearance. a poorly supported workpiece, or a badly worn machine gear are common causes of chattering.

Designation of Feed

The feed of the milling machine may be designated in inches per minute or millimeters per minute The milling feed is determined by multiplying the chip size (chip per tooth) desired (see Table 8-3 in Appendix A), the number of teeth on the cutter, and the revolutions per minute of the cutter.

Example: the formula used to find the workfeed in inches per minute.

IPM = CPTxNxRPM
IPM = Feed rate in inches per minute.
CPT = Chip pert
N = Number of teeth per minute of the milling cutter.

The first step is to calculate the spindle speed before the feed rate can be calculated,

$$RPM = \frac{CSD\ 4}{D} = \frac{300 \times 4}{1/2} = \frac{1,200}{0.5} = 2,400$$

The second step is to calculate the feed rate.

$$IPM = CPT \times N \times RPM$$
$$= 0.005 \times 2 \times 2,400$$
$$= 24$$

Therefore, the RPM for a l/2-inch-diameter end mill machining aluminum revolves at 2.400 RPM and the feed rate should be 24 inches per minute.

The formula used to find workfeed in millimeters per minute is the same as the formula used to find the feed in IPM, except that mm/min is substituted for IPM.

Direction of Feed

It is usually regarded as standard practice to feed the workpicce against the milling cutter. When the workpiece is fed against the milling cutter. the teeth cut under any scale on the workpiece surface and any backlash in the feed screw is taken up by the force of the cut. See Figure 8-26.

As an exception to this recommendation. it is advisable to feed with the milling cutter when cutting off stock or when milling comparatively deep or long slots.

The direction of cutter rotation is related to the manner in which the workplace is held. The cutter should rotate so that the piece springs away from the cutter; then there will be no tendency for the force of the cut to loosen the piece. No milling cutter should ever be rotated backward; this will break the teeth. If it is necessary to stop the machine during a finishing cut, the power feed should never be thrown out, nor should the workpiece be fed back under the cutter unless the cutter is stopped or the workpiece lowered. Never change feeds while the cutter is rotating.

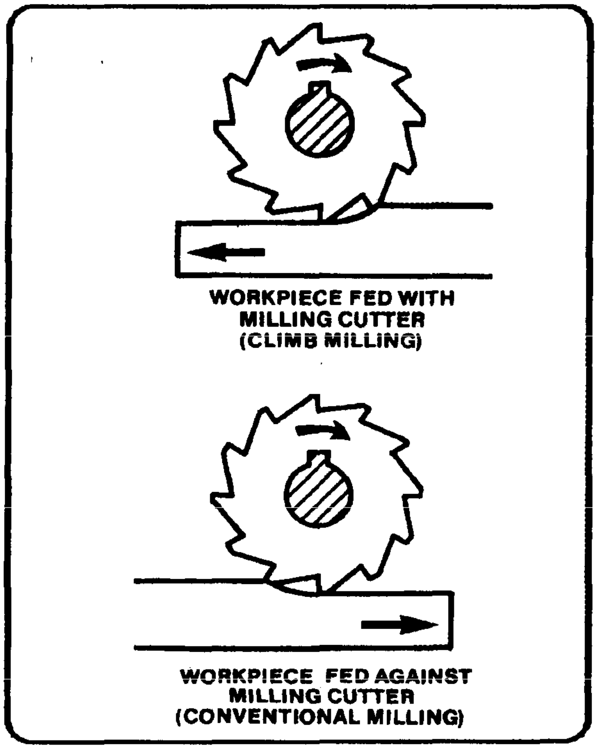


Figure 8-26. Direction of feed.

CUTTING OILS

The major advantage of using a coolant or cutting oil is that it dissipates heat, giving longer life to the cutting edges of the teeth. The oil also lubricates the cutter face and flushes away the chips, consequently reducing the possibility of marring the finish.

Types

Cutting oils are basically water-based soluble oils, petroleum oils, and synthetic oils. Water-based coolants have excellent heat transfer qualities; other oils result in good surface finishes. The cutting oil compounds for various metals are given in Table 4-3 in Appendix A. In general, a simple coolant is all that is required for roughing. Finishing requires a cutting oil with good lubricating properties to help produce a good finish on the workpiece. Plastics and cast iron are almost always machined dry.

Method of Use

The cutting oil or coolant should be directed by means of coolant drip can, pump system, or coolant mist mix to the point where the cutter contacts the workpiece. Regardless of method used, the cutting oil should be allowed to flow freely over the workpiece and cutter.

PLAIN MILLING

General

Plain milling, also called surface milling or slab milling, is milling flat surfaces with the milling cutter axis parallel to the surface being milled. Generally, plain milling is done with the workpiece surface mounted parallel to the surface of the milling machine table and the milling cutter mounted on a standard milling machine arbor. The arbor is well supported in a horizontal plane between the milling machine spindle and one or more arbor supports.

Mounting the Workpiece

The workpiece is generally clamped directly to the table or supported in a vise for plain milling. The milling machine table should be checked for alignment before starting to cut. If the workpiece surface to be milled is at an angle to the base plane of the piece, the workpiece should be mounted in a universal vise or on an adjustable angle plate. The holding device should be adjusted so that the workpiece surface is parallel to the table of the milling machine.

Selecting the Cutter

A careful study of the drawing must be made to determine what cutter is best suited for the job. Flat surfaces may be milled with a plain milling cutter mounted on an arbor. Deeper cuts may generally be taken when using narrow cutters than with wide cutters. The choice of milling cutters should be based on the size and shape of the workpiece. If a wide area is to be milled, fewer traverses will be required using a wide cutter. If large quantities of metal are to be removed, a coarse tooth cutter should be used for roughing and a finer tooth cutter should be used for finishing. A relatively slow cutting speed and fast table feed should be used for roughing, and a relatively fast cutting speed and slow table feed used for finishing. The surface should be checked for accuracy after each completed cut.

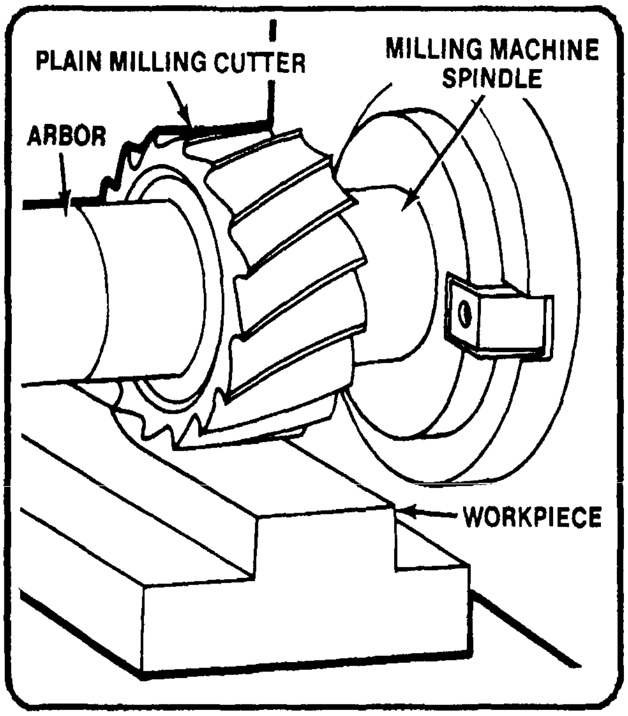


Figure 8-27. Plain milling.

Setup

A typical setup for plain milling is illustrated in Figure 8-27. Note that the milling cutter is positioned on the arbor with sleeves so that it is as close as practical to the milling machine spindle while maintaining sufficient clearance between the vise and the milling machine column. This practice reduces torque in the arbor and permits more rigid support for the cutter.

ANGULAR MILLING

General

Angular milling, or angle milling, is milling flat surfaces which are neither parallel nor perpendicular to the axis of the milling cutter. A single angle milling cutter is used for angular surfaces, such as chamfers, serration's, and grooves.
Milling dovetails (Figure 8-28) is a typical example of angular milling.

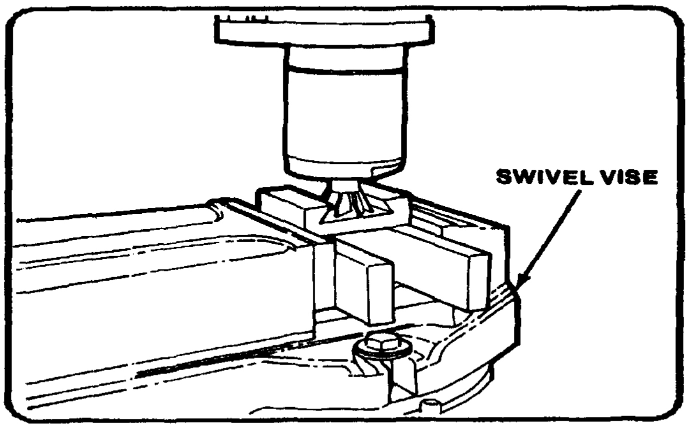


Figure 8-28. Angular milling.

Milling Dovetails

When milling dovetails, the usual angle of the cutter is 45°, 50°, 55°, or 60° based on common dovetail designs.

When cutting dovetails on the milling machine, the workpiece may be held in a vise, clamped to the table, or clamped to an angle plate. The tongue or *groove is* first roughed out using a side milling cutter, after which the angular sides and base are finished with an angle milling cutter.

In general practice, the dovetail is laid out on the workpiece surface before the milling operation is started. To do this, the required outline should be inscribed and the line prick-punched. These lines and punch marks may then be used as a guide during the cutting operation.

STRADDLE MILLING

When two or more parallel vertical surfaces are machined at a single cut, the operation is called straddle milling. Straddle milling is accomplished by mounting two side milling cutters on the same arbor, set apart at an exact spacing. Two sides of the workpiece are machined simultaneously and final width dimensions are exactly controlled.

MILLING A HEXAGON

Straddle milling has many useful applications introduction machining. Parallel slots of equal depth can be milled by using straddle mills of equal diameters. Figure 8-29 illustrates a typical example of straddle milling. In this case a hexagon is being cut, but the same operation may be applied to cutting squares or splines on the end of a cylindrical workpiece. The workpiece is usually mounted between centers in the indexing fixture or mounted vertically in a swivel vise. The two side milling cutters are separated by spacers, washers, and shims so that the distance between the cutting teeth of each cutter is exactly equal to the width of the workpiece area required. When cutting a square by this method, two opposite sides of the square are cut, and then the spindle of the indexing fixture or the swivel vise is rotated 90°, and the other two sides of the workpiece are straddle milled.

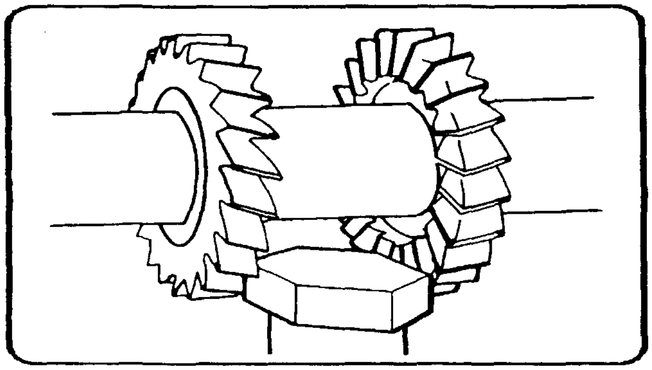


Figure 8-29. Straddle milling.

FACE MILLING

General

Face milling is the milling of surfaces that are perpendicular to the cutter axis, as shown in Figure 8-30. Face milling produces flat surfaces and machines work to the required length. In face milling, the feed can be either horizontal or vertical.

In face milling, the teeth on the periphery of the cutter do practically all of the cutting. However, when the cutter is properly ground, the face teeth actually remove a small amount of stock which is left as a result of the springing of the workpiece or cutter, thereby producing a finer finish.

It is important in face milling to have the cutter securely mounted and to see that all end play or sloppiness in the machine spindle is eliminated.

Mounting the Workpiece

When face milling, the workpiece may be clamped to the table or angle plate or supported in a vise, fixture, or jig.

Large surfaces are generally face milled on a vertical milling machine with the workpiece clamped directly to the milling machine table to simplify handling and clamping operations.

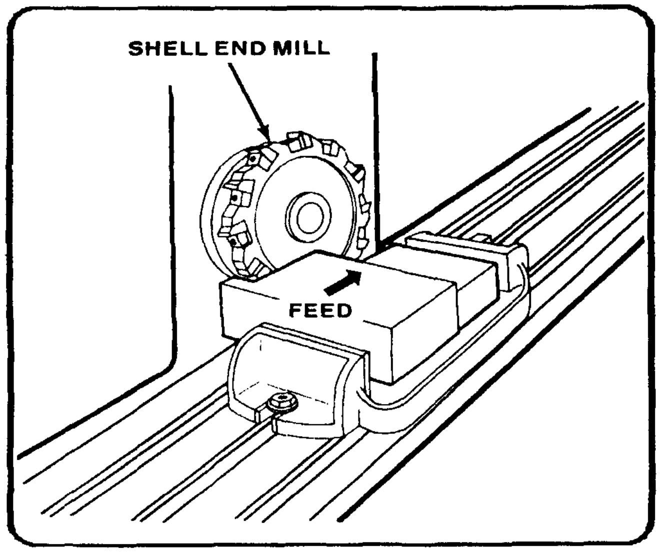


Figure 8-30. Face milling.

Angular surfaces can also be face milled on a swivel cutter head milling machine (Figure 8-31). In this case, the workpiece is mounted parallel to the table and the cutter head is swiveled to bring the end milling cutter perpendicular to the surface to be produced.

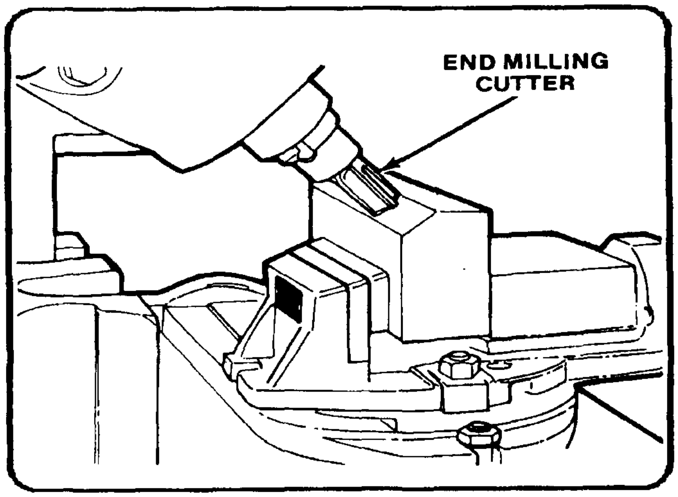


Figure 8-31. Angular face milling.

During face milling operations, the workpiece should be fed against the milling cutter so that the pressure of the cut is downward, thereby holding the piece against the table.Whenever possible, the edge of the workpiece should be in line with the center of the cutter. This position of the workpiece in relation to the cutter will help eliminate slippage.

Depth of Cut

When setting the depth of cut, the workpiece should be brought up to just touch the revolving cutter. After a cut has been made from this setting, measurement of the workpiece is taken. At this point, the graduated dial on the traverse feed is locked and used as a guide in determining the depth of cut.

When starting the cut, the workpiece should be moved so that the cutter is nearly in contact with its edge, after which the automatic feed may be engaged.

When a cut is started by hand, care must be taken to avoid pushing the corner of the workpiece between the teeth of the cutter too quickly, as this may result in cutter tooth breakage. In order to avoid wasting time during the operation, the feed trips should be adjusted to stop the table travel just as the cutter clears the workpiece.

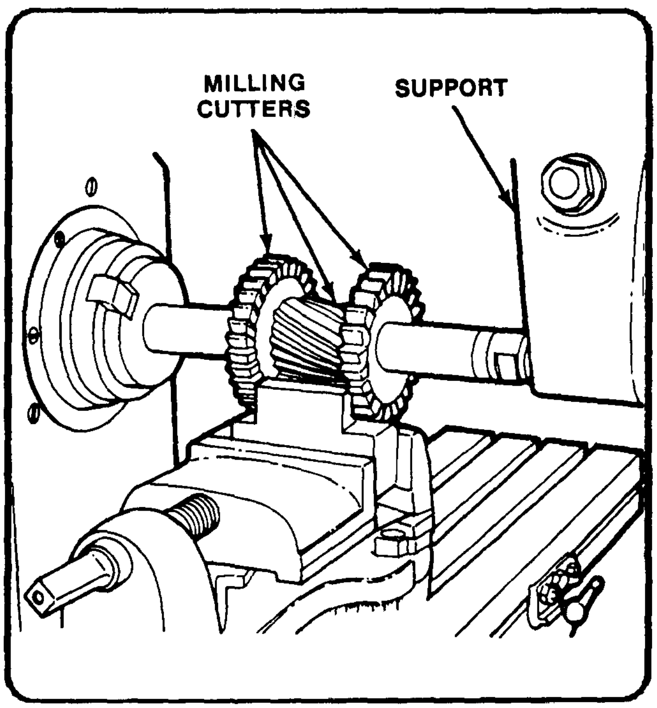


Figure 8-32. Gang milling.

GANG MILLING

Gang milling is the term applied to an operation in which two or more milling cutters are mounted on the same arbor and used when cutting horizontal surfaces. All cutters may perform the same type of operation or each cutter may perform a different type of operation. For example, several workplaces need a slot, a flat surface, and an angular groove. The best method to cut these would be gang milling as shown in Figure 8-32. All the completed workplaces would be the same. Remember to check the cutters carefully for proper size.

FORM MILLING

Form milling is the process of machining special contours composed of curves and straight lines, or entirely of curves, at a single cut. This is done with formed milling cutters, shaped to the contour to be cut. The more common form milling operations involve milling half-round recesses and beads and quarter-round radii on workplaces (Figure 8-33), This operation is accomplished by using convex, concave, and corner rounding milling cutters ground to the desired circle diameter. Other jobs for formed milling cutters include milling intricate patterns on workplaces and milling several complex surfaces in a single cut such as are produced by gang milling.

FLY CUTTING

General

Fly cutting, which is also called single point milling, is one of the most versatile milling operations. It is done with a single-point cutting tool shaped like a lathe tool bit. It is held and rotated by a fly cutter arbor. You can grind this cutter to almost any form that you need, as shown in Figure 8-34. Formed cutters are expensive. There are times when you need a special form cutter for a very limited number of parts. It is more economical to grind the desired form on a lathe-type tool bit than to buy a preground form cutter, which is very expensive and usually suitable only for one particular job.

Gear Cutting

The single-point or fly cutter can be used to great advantage in gear cutting. A II that is needed is enough of the broken gear to grind the cutting tool to the proper shape. It can also be used in the cutting of splines and standard and special forms.

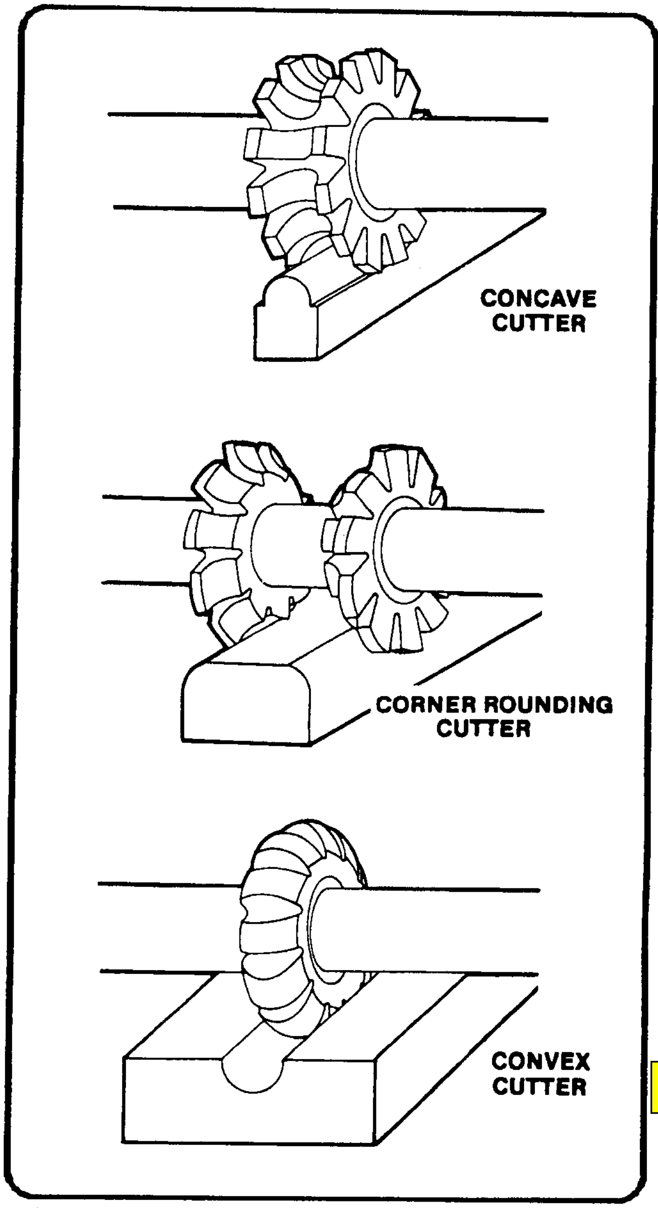


Figure 8-33. Form milling.

Flat Surfaces

Another type of fly cutter, which differs mainly in the design of the arbor, can be used to mill flat surfaces as in plain or face milling (Figure 8-34). The arbor can easily be manufactured in the shop using common lathe tool bits. This type of fly cutter is especially useful for milling flat surfaces on aluminum and other soft nonferrous metals, since a high quality finish can be easily obtained. Boring holes with this type of fly cutter is not recommended. The arbor is so short that only very shallow holes can be bored.

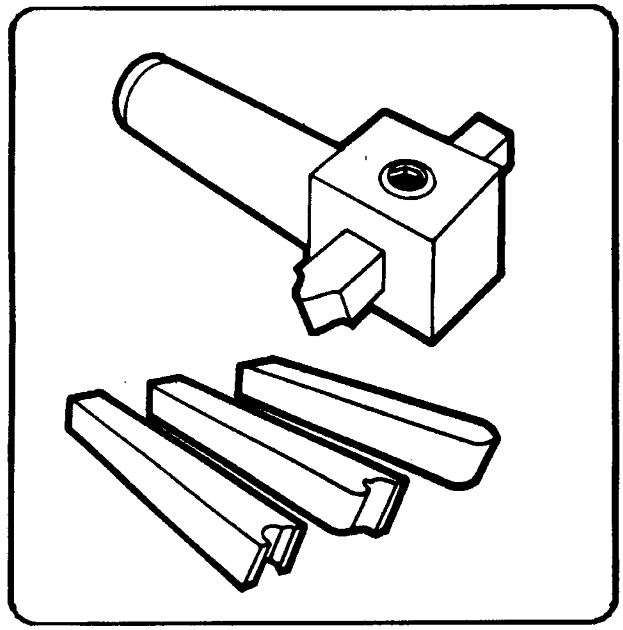


Figure 8-34. Fly cutter arbor and special-formed cutters.

KEYWAY MILLING

Keyways are grooves of different shapes cut along the axis of the cylindrical surface of shafts, into which keys are fitted to provide a positive method of locating and driving members on the shafts. A keyway is also machined in the mounted member to receive the key.

The type of key and corresponding keyway to be used depends upon the class of work for which it is intended. The most commonly used types of keys are the Woodruff key, the square-ends machine key, and the round-end machine key (Figure 8-35).

Woodruff Key

The Woodruff keys are semicylindrical in shape and are manufactured in various diameters and widths. The circular side of the key is seated into a keyway which is milled in the shaft. The upper portion fits into a slot in a mating part, such as a pulley or gear. The Woodruff key slot milling cutter (Figure 8-36) must have the same diameter as that of the key.

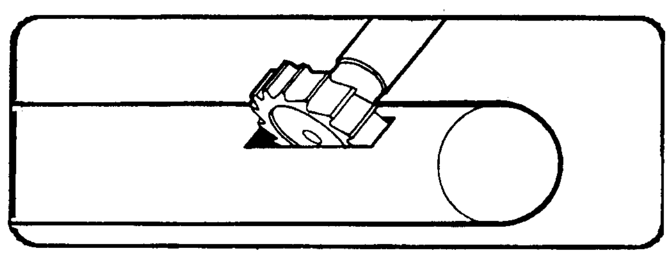


Figure 8-36. Woodruff keyslot.

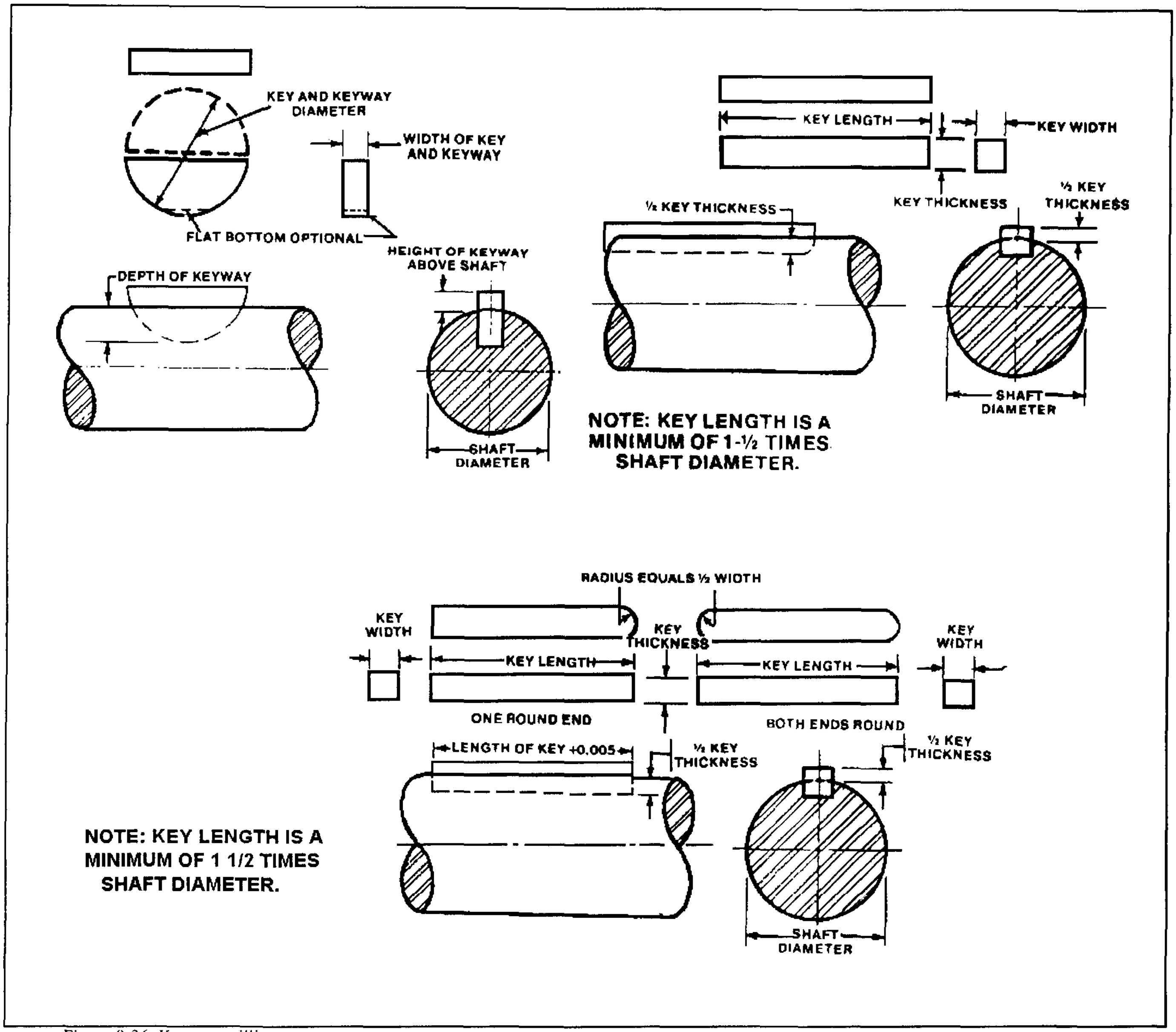


Figure 8-36. Keyway milling.

Woodruff key sizes are designated by a code number in which the last two digits indicate the diameter of the key in eighths of an inch, and the digits preceding the last two digits give the width of the key in thirty-seconds of an inch. Thus, a number 204 Woodruff key would be 4/8 or 1/2 inch in diameter and 2/32 or 1/16 inch wide, while a number 1012 Woodruff key would be 12/8 or 1 1/2 inches in diameter and 10/32 or 5/16 inch wide. Table 8-4 in Appendix A lists Woodruff keys commonly used and pertinent information applicable to their machining.

For proper assembly of the keyed members to be made, a clearance is required between the top surface of the key and the keyway of the bore. This clearance may be from a minimum of 0.002 inch to a maximum of 0.005 inch. Positive fitting of the key in the shaft keyway is provided by making the key 0.0005 to 0.001 inch wider than the keyway.

Square-End Machine Key

Square-ends machine keys are square or rectangular in section and several times as long as they are wide. For the purpose of interchangeability and standardization, these keys are usually proportioned with relation to the shaft diameter in the following method:

- Key width equals approximately one-quarter of the shaft diameter.
- Key thickness for rectangular section keys (flat keys) equals approximately 1/6 of the shaft diameter.
- Minimum length of the key equals 1 1/2 times the shaft diameter.
- Depth of the keyway for square section keys is 1/2 the width of the key.
- Depth of the keyway for rectangular section keys (flat keys) is 1/2 the thickness of the key,

Table 8-5 in Appendix A lists common sizes for square-end machine keys. The length of each key is not included because the key may be of any length as long as it equals at least 1 1/2 times the shaft diameter.

Round-end machine keys (Figure 8-35). The round-ends machine keys are square in section with either one or both ends rounded off. These keys are the same as square-ends machine keys in measurements (see Table 8-5 in Appendix A).

Milling Cutters Used for Milling Keyways

Shaft keyways for Woodruff keys are milled with Woodruff keyslot milling cutters (Figure 8-35). The Woodruff keyslot milling cutters are numbered by the same system employed for identifying Woodruff keys, Thus, a number 204 Woodruff keyslot cutter has the proper diameter and width for milling a keyway to fit a number 204 Woodruff key.

Square-end keyways can be cut with a plain milling cutter or side milling cutter of the proper width for the key

Round-end keyways must be milled with end milling cutters (Figure 8-37) so that the rounded end or ends of the key may fit the ends of the keyway. The cutter should be equal in diameter to the width of the key.

Alignment of Milling Cutters

When milling keyways. the shaft may be supported in the vise or chuck, mounted between centers. or clamped to the milling machine table. The cutter must be set centrally with the axis of the workpiece. This alignment is accomplished by using one of the following methods:

When using a Woodruff keyslot milling cutter, the shaft should be positioned so that the side of the cutter is tangential to the circumference of the shaft. This is done by moving the shaft transversely to a point that permits the workpiece to touch the cutter side teeth. At this point the graduated dial on the cross feed is locked and the milling machine table is lowered. Then, using the cross feed graduated dial as a guide, the shaft is moved transversely a distance equal to the radius of the shaft plus 1/2 the width of the cutter.

End mills may be aligned centrally by first causing the workpiece to contact the periphery of the cutter, then proceeding as in the paragraph above.

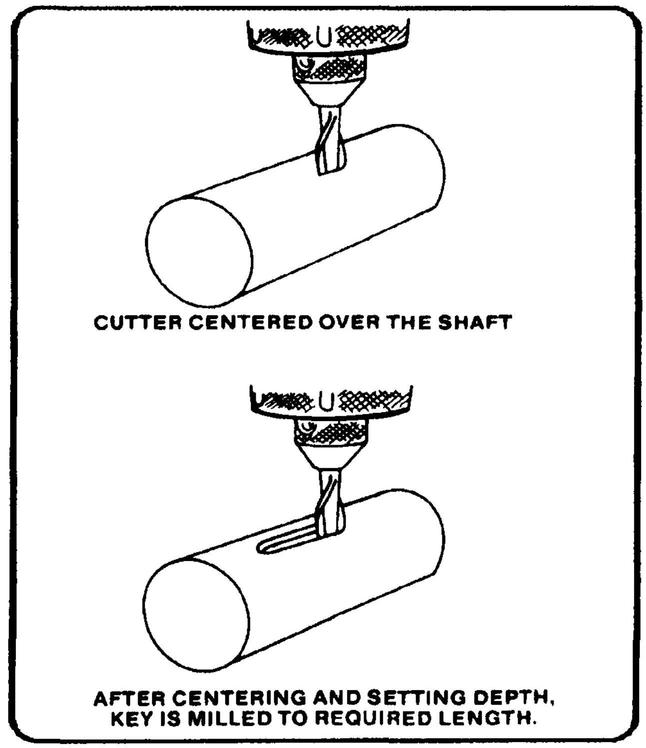


Figure 8-37. Round-end keyway.

Milling Woodruff Key Slot

The milling of a Woodruff keyslot is relatively simple since the proper sized cutter has the same diameter and thickness as the key. With the milling cutter located over the position in which the keyway is to be cut, the workpiece should be moved up into the cutter until you obtain the desired keyseat depth. Refer to Table 8-4 in Appendix A for correct depth of keyslot cut for standard Woodruff key sizes. The work may be held in a vise. chuck. between centers. or clamped to the milling machine table. Depending on its size, the cutter is held in an arbor or in a spring collet or drill chuck that has been mounted in the spindle of the milling machine.

Milling Keyslot for Square-End Machine Key

The workpiece should be properly mounted, the cutter centrally located, and the workpiece raised until the milling cutter teeth come in contact with the workpiece. At this point, the graduated dial on the vertical feed is locked and the workpiece moved longitudinally to allow the cutter to clear the workpiece. The vertical hand feed screw is then used to raise the workpiece until the cutter obtains the total depth of cut. After this adjustment. the vertical adjustment control should be locked and the cut made by feeding the table longitudinally.

Milling Keyway for Round-End Machine Key

Rounded keyways are milled with an end milling cutter Of the proper diameter. As in the case of square-ends machine key keyways, the workpiece should be properly mounted and the cutter centrally located with respect to the shaft. The shaft or cutter is then positioned to permit the end of the cutter to tear a piece of thin paper held between the cutter and the workpiece. At this point the graduated feed dial should be locked and used as a guide for setting the cutter depth. The ends of the keyway should be well marked and the workpiece moved back and forth making several passes to eliminate error due to spring of the cutter.

T-SLOT MILLING

Cutting T-slots in a workpiece holding device is a typical milling operation. The size of the T-slots depends upon the size of the T-slot bolts which will be used. Dimensions of T-slots and T-slot bolts are standardized for specific bolt diameters. The dimensions for bolt diameters commonly used are given in Table 8-6 (Appendix A).

Selection of Milling Cutters

Two milling cutters are required for milling T-slots, a T-slot milling cutter and either a side milling cutter or an end milling cutter. The side milling cutter (preferably of the staggered tooth type) or the end milling cutter is used to cut a slot in the workpiece equal in width to the throat width of the T-slot and equal in depth to slightly less than the head space depth plus the throat depth). The T-slot milling cutter is then used to cut the head space to the prescribed dimensions.

Milling the T-Slot

The position of the T-slot is laid out on the workpiece. The throat depth is determined by considering the thickness of the workpiece and the maximum and minimum dimensions allowable (Table 8-6. Appendix A).

A side milling cutter or an end milling cutter is then selected. The cutter should be of proper size to mill a slot equal in width to the throat width prescribed for the T-slot size desired. Cut a plain groove equal to about 1/16 inch less than the combined throat depth and head space depth.

Select a T-slot milling cutter for the size T-slot to be cut. T-slot milling cutters are identified by the T-Slot bolt diameter and remanufactured with the proper diameter and width to cut the head space to the dimensions given in Table 8-6 in Appendix A. Position the T-slot milling cutter over the edge of the workpiece and align it with the previously cut groove. Feed the table longitudinally to make the cut. Flood the cutter and workpiece with cutting oil during this operation. Figure 8-38 shows a T-slot milling cutter and dimension locations for T-slots.

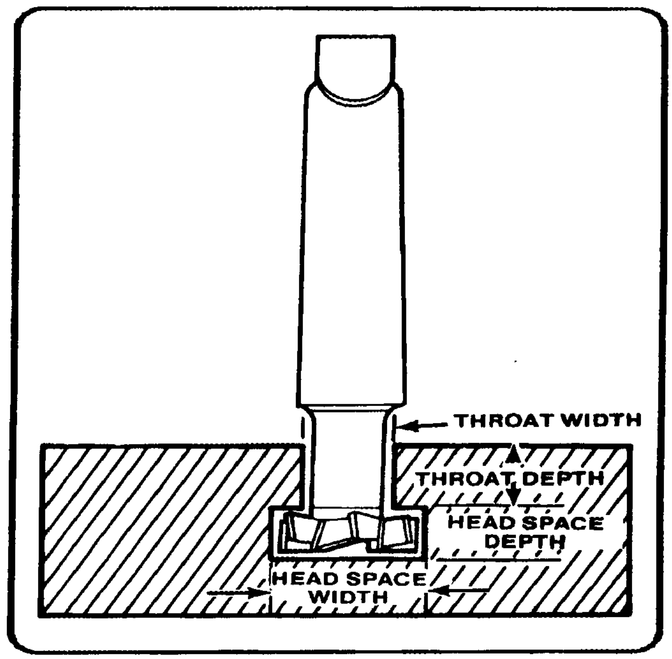


Figure 8-38. T-slot. milling.

SAWING AND PARTING

Metal slitting saw milling cutters are used to part stock on a milling machine. Figure 8-39 illustrates parting solid stock. The workpiece is being fed against the rotation of the cutter. For greater rigidity while parting thin material such as sheet metal, the vvorkpiece may be clamped directly to the table with the line of cut over one of the table T-slots. In this case, the workpiece should be fed with the rotation of the milling cutter (climb milling) to prevent it from being raised off the table. Every precaution should be taken to eliminate backlash and spring in order to prevent climbing or gouging the workpiece.

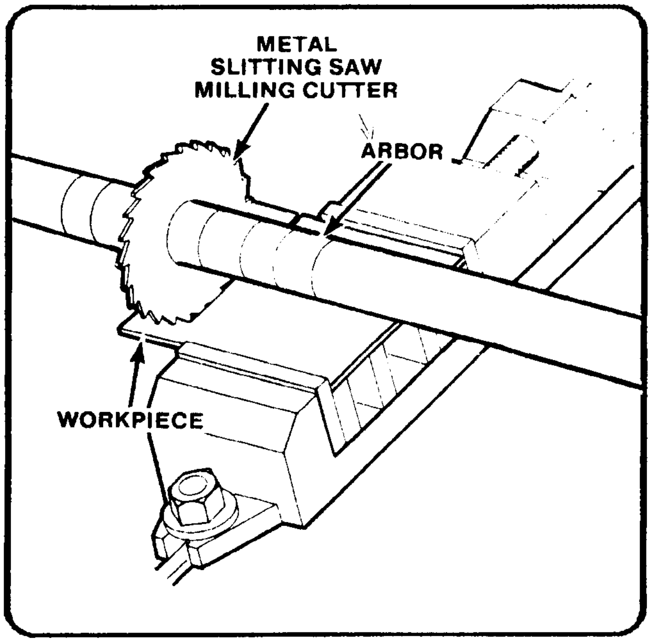


Figure 8-39. Parting solid stock.

HELICAL MILLING

A helix may be defined as a regular curved path. such as is formed by winding a cord around the surface of a cylinder. Helical parts most commonly cut on the milling machine include helical gears. spiral flute milling cutters, twist drills. and helical cam grooves. When milling a helix. a universal index head is used to rotate the workpiece at the proper rate of speed while the piece is fed against the cutter. A train of gears between the table feed screw and the index head serves to rotate the workpiece the required amount for a given longitudinal movement of the table. Milling helical parts requires the use of special formed milling cutters and double-angle milling cutters, The calculations and formulas necessary to compute proper worktable angles, gear adjustments. and cutter angles and positions for helical milling are beyond the scope of this manual,

GEAR CUTTING

Gear teeth are cut on the milling machine using formed milling cutters called involute gear cutters. These cutters are manufactured in many pitch sizes and shapes for different numbers of teeth per gear (Table 8-7, Appendix A).

If involute gear cutters are not available and teeth must be restored on gears that cannot be replaced. a lathe cutter bit ground to the shape of the gear tooth spaces may be mounted in a fly cutter for the operation. The gear is milled in the following manner:

NOTE: This method of gear cutting is not as accurate as using an involute gear cutter and should be used only for emergency cutting of teeth which have been built up by welding,

Fasten the indexing fixture to the milling machine table. Use a mandrel to mount the gear between the index head and footstock centers. Adjust the indexing fixture on the milling machine table or adjust the position of the cutter to make the gear axis perpendicular to the milling machine spindle axis. Fasten the cutter bit that has been ground to the shape of the gear tooth spaces in the fly cutter arbor. Adjust the cutter centrally with the axis of the gear. Rotate the milling machine spindle to position the cutter bit in the fly cutter so that its cutting edge is downward.

Align the tooth space to be cut with the fly cutter arbor and cutter bit by turning the index crank on the index head.

Proceed to mill the tooth in the same manner as milling a keyway.

SPLINE MILLING

Splines are often used instead of keys to transmit power from a shaft to a hub or from a hub to a shaft. Splines are. in effect. a series of parallel keys formed integrally with the shaft. mating with corresponding grooves in the hub or fitting (Figure 8-40). They are particularly useful where the hub must slide axially on the shaft, either under load or freely. Typical applications for splines are found in geared transmissions, machine tool drives. and in automatic mechanisms.

Splined Shafts and Fittings

Splined shafts and fittings are generally cut by bobbing and broaching on special machines. However. when spline shafts must be cut for a repair job. the operation may be accomplished on the milling machine in a manner similar to that described for cutting keyways. Standard spline shafts and splint fittings have 4, 6, 10, or 16 splines, and theirdimensions depend upon the class of tit for the desired application: a permanent fit, a sliding fit when not under load, and a sliding fit under load. Table 8-8 in Appendix A lists the standard dimensions for 4, 6, 10, and 16-spline shafts.

Milling Splines

Spline shafts can be milled on the milling machine in a manner similar to the cutting of keyways.

The shaft to be splined is set up between centers in the indexing fixture.

Two side milling cutters are mounted to an arbor with a spacer and shims inserted between them. The spacer and shims are chosen to make space between the inner teeth of the cutters equal to the width of the spline to be cut (Table 8-8, Appendix A).

The arbor and cutters are mounted to the milling machine spindle. and the milling machine is adjusted so that the cutters are centered over the shaft.

The splines are cut by straddle milling each spline to the required depth (Table 8-8. Appendix A) and using the index head of the indexing fixture to rotate the workpiece the correct distance between each spline position.

After the splines are milled to the correct depth, mount a narrow plain milling cutter in the arbor and mill the spaces between the splines to the proper depth. It will be necessary to make several passes to cut the groove uniformly so that the spline fitting will not interfere with the grooves. A formed spline milling cutter, if available, can be used for this operation.

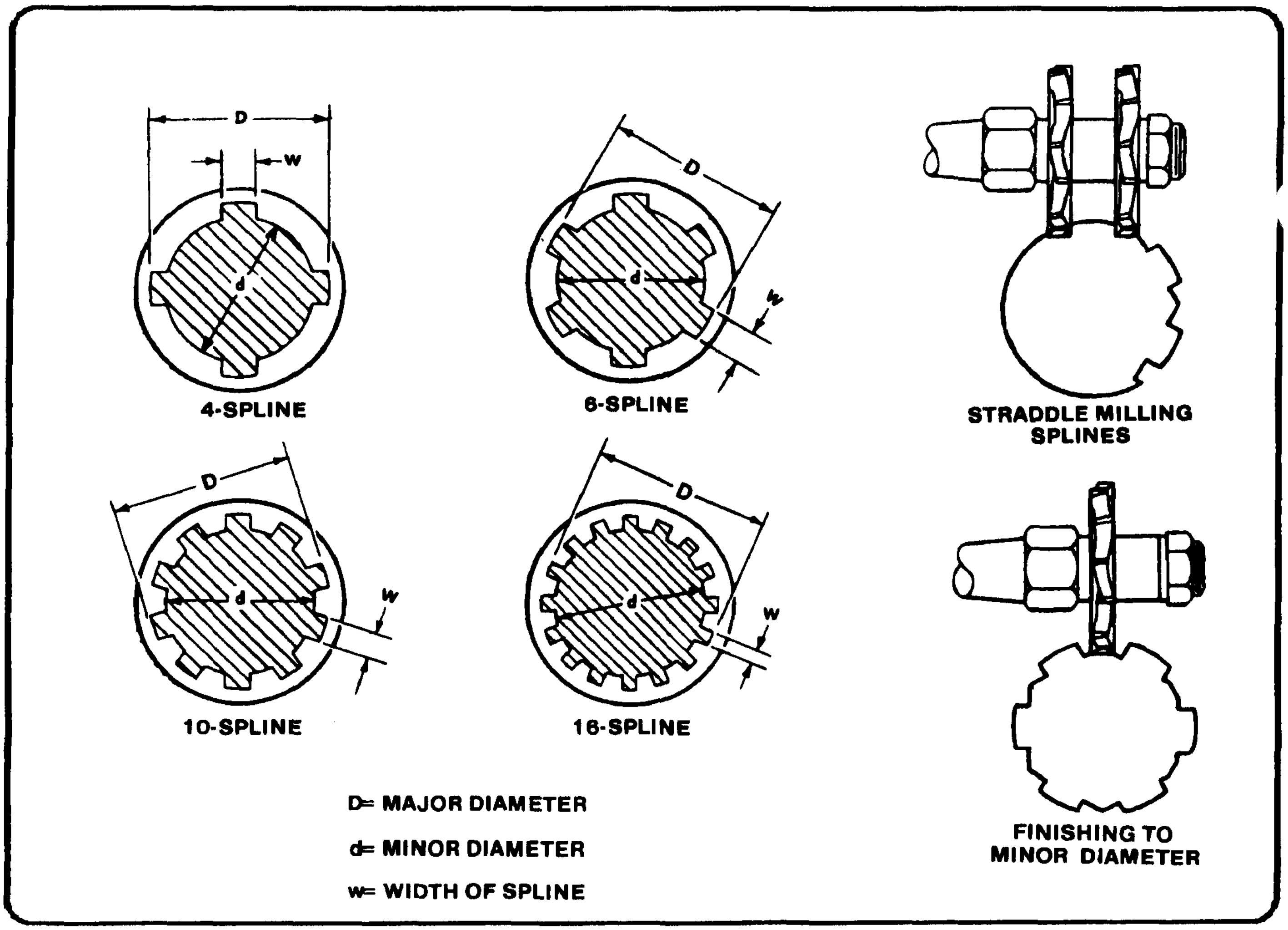


Figure 8-40. Milling spline shafts.

DRILLING

The milling machine may be used effectively for drilling, since accurate location of the hole may be secured by means of the feed screw graduations. Spacing holes in a circular path, such as the holes in an index plate, may be accomplished by indexing with the index head positioned vertically.

Twist drills may be supported in drill chucks fastened in the milling machine spindle or mounted directly in milling machine collets or adapters. The workpiece to be drilled is fastened to the milling machine table by clamps, vises, or angle plates.

BORING

Various types of boring tool holders may be used for boring on the milling machine. the boring tools being provided with either straight shanks to be held in chucks and holders or taper shanks to fit collets and adapters. The two attachments most commonly used for boring are the fly cutter arbor and the offset boring head.

The single-edge cutting tool used for boring on the milling machine is the same as a lathe cutter bit. Cutting speeds, feeds, and depth of cut should be the same as that prescribed for lathe operations.

Chapter 9

MILLING-GRINDING-DRILLING AND SLOTTING ATTACHMENT

(VERSA-M IL)

GENERAL

DESCRIPTION

The milling-grinding-drilling and slotting attachment is commonly referred to as a Versa-Mil. It is a compact, portable unit capable of doing many machining operations that normally require expensive single-purpose machines. With the different attachments that are available with the unit, drilling, shaping, milling, and grinding can be performed quickly and inexpensively. This self-powered, vertical-feed, variable-speed precision tool may be mounted in any position on the carriage, table, ram, turret, or tool arm of other machine tools. With a two-directional feed table, the Versa-Mil unit becomes a complete machining tool for bench or inplace machining of parts too large to be moved or held in conventional machine tools.

USES

An important factor in the efficiency of the Versa-Mil is that machine tools already in the shop area provide the power for feeds, a means of holding and moving the work, and the rigidity needed for machining. Faced with unusual machining problems, the Versa-Mil offers many solutions either as a separate tool or combined with other machine tools and machinery already in the shop to create special machines. The Versa-Mil increases the capabilities of standard machines by doing secondary operations without changing setups. The Versa-Mil provides power to interchangeable attachments allowing the unit to be used on site to perform different machining operations on equipment being repaired or rebuilt. Where space is limited, as in a shop area, floor space is needed only for the lathe. Different sizes of the Versa-Mil unit are available for light, medium, and heavy machining. This chapter will be limited to the Series 31 (light machining unit).

SAFETY PRECAUTIONS

Safety in the shop area or around power equipment cannot be overemphasized. Each piece of equipment has safety procedures unique to that particular piece of equipment. Listed below are safety procedures that pertain to the Versa-Mil.

- Avoid dangerous environments. Do not use the Versa-Mil in damp or wet locations. Do not expose the Versa-Mil to rain.
- Keep visitors away from running equipment. Keep visitors a safe distance from the Versa-Mil while it is in operation.
- Store tools when not in use. Store or lock tools and equipment in the Versa-Mil cabinet.
- Do not force the equipment. The Versa-Mil will do the job better and safer at the rate for which it was designed.
- Wear proper apparel. Keep shirt sleeves above the elbow. Remove ID tags, watches, rings, and any other jewelry when working around the Versa-Mil.
- Use safety glasses. Wear safety glasses when operating any type of machine shop equipment.
- Do not abuse the electrical cord. Never carry the Versa-Mil by the electrical cord or pull on the cord to disconnect it from the receptacle. Keep the cord away from excessive heat, oil, and sharp edges. Replace end connectors or cords when excessive wear or damage is apparent.
- Maintain tools with care. Keep tools and cutters sharp and clean for the best performance. Follow instructions in the Versa-Mil Operation and Service Manual for lubricating the basic unit and changing accessories.
- Disconnect equipment not in use. Ensure the Versa-Mil is disconnected when not in use, before servicing, and when changing attachments, speeds, cutters, or arbors.
- Remove chuck keys and wrenches. Form a habit of checking to see that chuck keys and wrenches are removed from the unit prior to operating the equipment. Remove all tools from the area that may vibrate off the equipment and into moving parts.

- Avoid accidental starting. Place protective cover around the switch to help prevent accidental starting of the Versa-Mil. Ensure switch is off before connecting the unit to a power supply.

- Outdoor use of extension cords. When using the Versa-Mil outdoors, use only extension cords designed and marked for outdoor use.

- Reversing switch. Ensure that the reversing switch is in the correct position for proper cutter rotation. Failure to do this could result in damage or injury by having a cutter or arbor dislodged from the basic unit.

- Pulley guard. The pulley guard must be in place before operating the Versa-Mil. This prevents fingers or clothing from getting caught between the belt and pulleys.

- Handle cutters with care. Handle all cutters with a cloth to prevent accidental cutting of fingers or hands.

- Grinding wheels. Use grinding wheels with the safe speed at least as high as the no-load RPM rating of the Versa-Mil grinding attachment.

TOOLS AND EQUIPMENT

VERSA-MIL BASIC UNIT

The Versa-Mil basic unit (Figure 9-1) has a powered machining head which moves vertically on four hardened ground guide posts by means of a precision-ground lead screw calibrated to 0.001 inch. Thirteen different speeds are available to the head through the use of different size pulleys to accommodate all types of machining and cutter sizes within the range of the unit. The circular T-slot on the face of the basic unit accommodates a variety of attachments. The graduation marks on the basic unit indicate the degree angle an attachment is to be positioned for various machining operations.

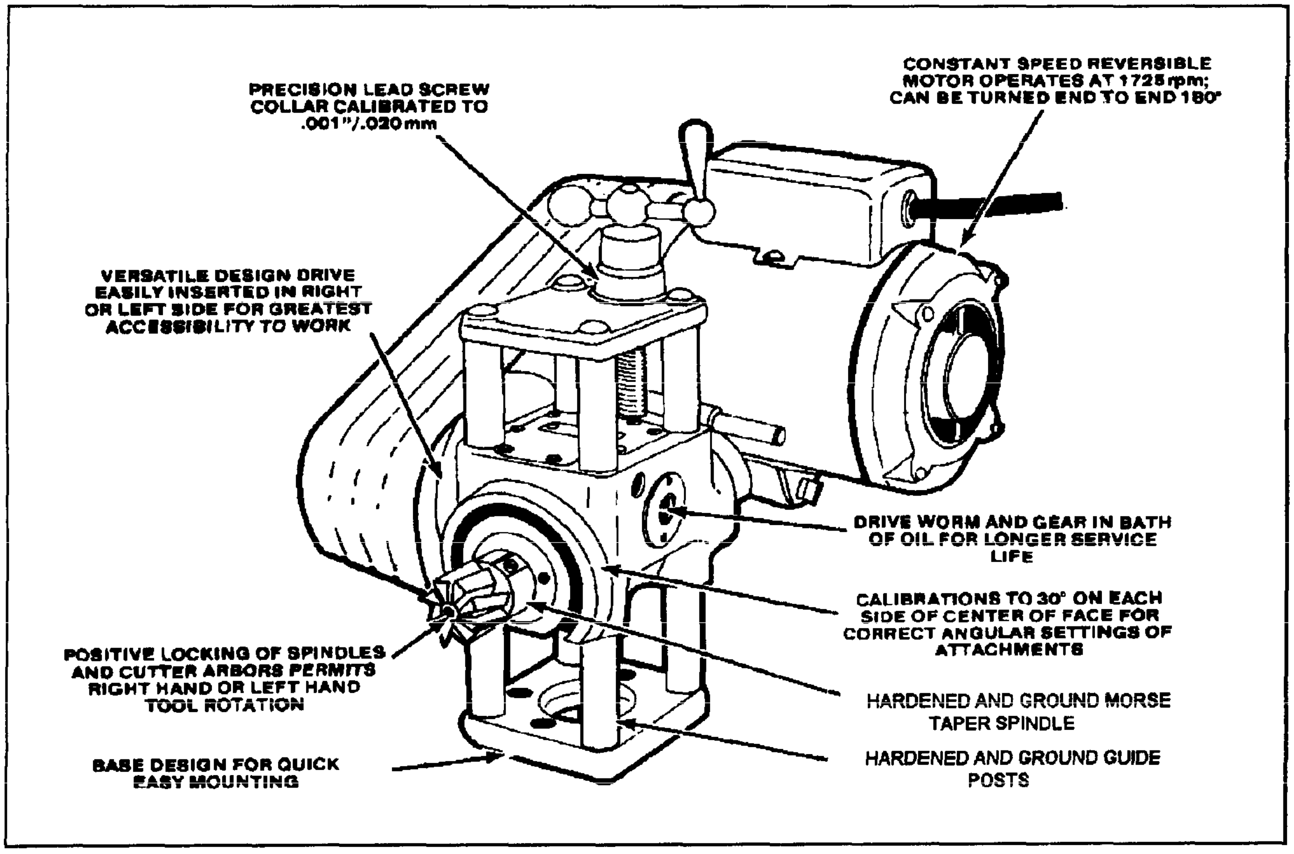


Figure 9-1. Versa-Mil basic unit.

ATTACHMENTS

External Grinding Attachment

The external grinding head (Figure 9-2) bolts to the face of the Versa-Mil making the unit a precision external grinder. The head adjusts to 30 degrees range of angle to either side. A flat belt from the motor provides power to the head for smooth operation. Different pulley diameters allow matching spindle speeds to the grinding wheel size and rating. A wheel guard on the head offers protection to the operator from debris coming off the wheel during grinding.

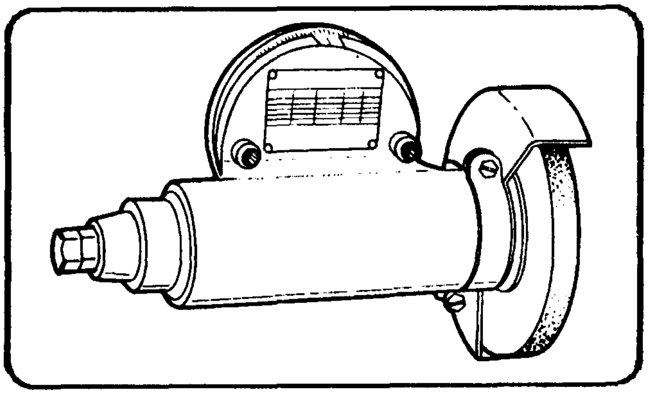


Figure 9-2. External grinding attachment.

Internal Grinding Attachment

A wide variety of internal grinding jobs can be handled on a lathe with the Versa-Mil basic unit and the internal grinding unit (Figure 9-3). The internal grinding attachment bolts to the face of the basic unit and is driven by a flat belt from the motor. The internal grinder handles grinding wheels from 5/8 inch to 2 1/2 inches in diameter and grinds to a depth of 4 inches. Five different speeds are available to match the spindle speed to the grinding wheel diameter and rating.

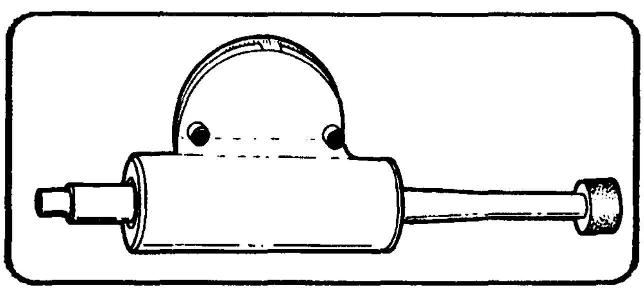


Figure 9-3. Internal grinding attachment.

Heavy-Duty Deep-Hole Grinder

The heavy-duty deep-hole grinder (Figure 9-4) may be attached to the face of the Versa-Mil for deep internal grinding. The deep-hole grinder accommodates grinding wheels 3 to 5 inches in diameter and grinds to a depth of 10 inches. A flat belt from the motor drives the deep-hole grinder for smooth operation. Six spindle speeds are available to match the spindle speed to the grinding wheel diameter and rating.

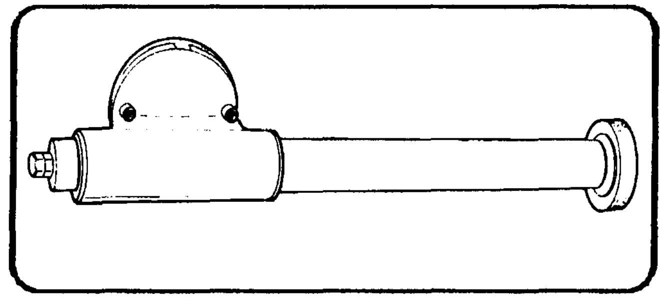


Figure 9-4. Heavy-duty, deep-hole grinder.

Tooth Stop Rest

Cutters held in the lathe chuck, collet, or between lathe centers can be ground quickly and accurately with the Versa-Mil unit equiped with an external or internal grinding head. The tooth stop rest (Figure 9-5) assures uniform grinding of cutter teeth because the finger on the gage ratchets over the teeth stopping each tooth in the exact same position. The tooth stop rest is completely adjustable for height and position.

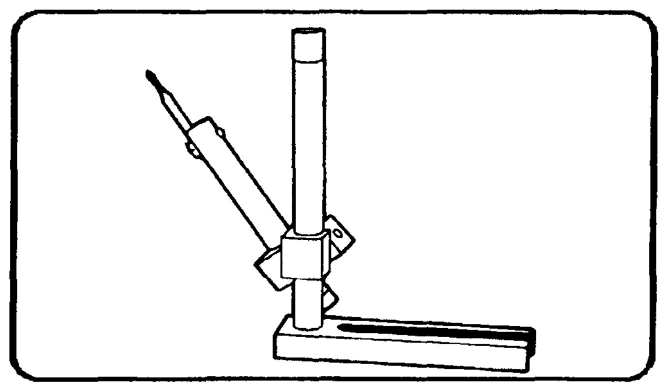


Figure 9-5. Tooth stop rest.

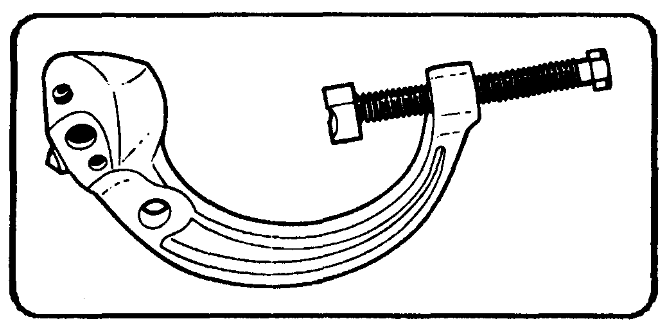


Figure 9-6. Diamond dresser.

Diamond Dresser

The diamond dresser (Figure 9-6) is used with all Versa-Mil grinding attachments and clamps to the workpiece, tailstock, or lathe face plate to true the grinding wheel. A 0.35-karat industrial diamond mounts in either of two positions to dress the face or side of the grinding wheel. The cast-iron frame with V-notch clamps securely to round shapes up to 3 1/2 inches in diameter.

Universal Milling Head

The universal milling head (Figure 9-7) mounts to the face of the Versa-Mil and is driven by the spindle of the basic unit. This feature eliminates the need for special belts and permits the head to operate at any angle. The milling head and the basic unit have the same spindle taper and use the same arbors. With the universal head, machining can be performed along the side of the work, allowing the machining of much larger parts. Angular operations such as thread milling can easily be performed on large diameter material using the universal head.

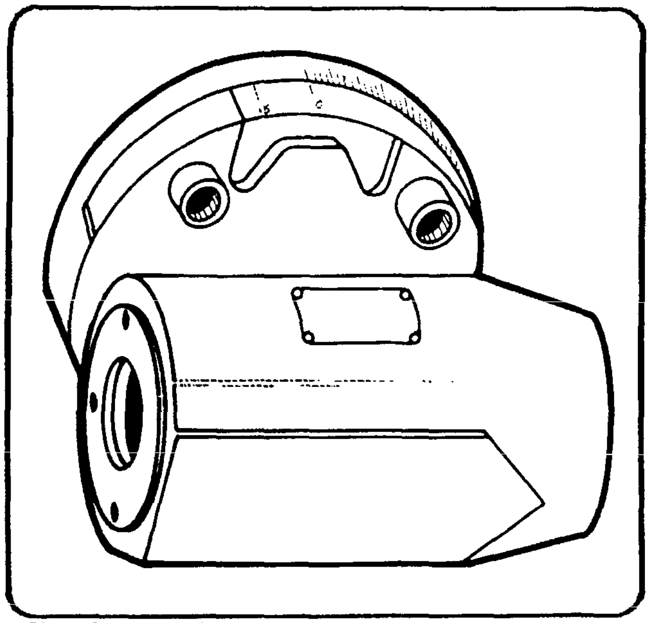


Figure 9-7. Heavy-duty, deep-hole grinder.

Internal Keyseater and Slotter

This unit bolts to the face of the Versa-Mil and is driven by the basic unit spindle. The An internal keyseater and slotter (Figure 9-8) commonly called a “Versa-Shaper,” bolts to the face of the Versa-Mil Versa-Shaper operates in any angular position and in either direction of stroke for cutting internal keyways, slotting, or shaping. Thestroke length adjusts from 0 to 4 inches with a speed of 44 to 450 strokes per minute. Tool holders for 1/8”, 3/16”, 1/4”, 5/16”, and 1/2” cutters are available for use in the Versa-Shaper.

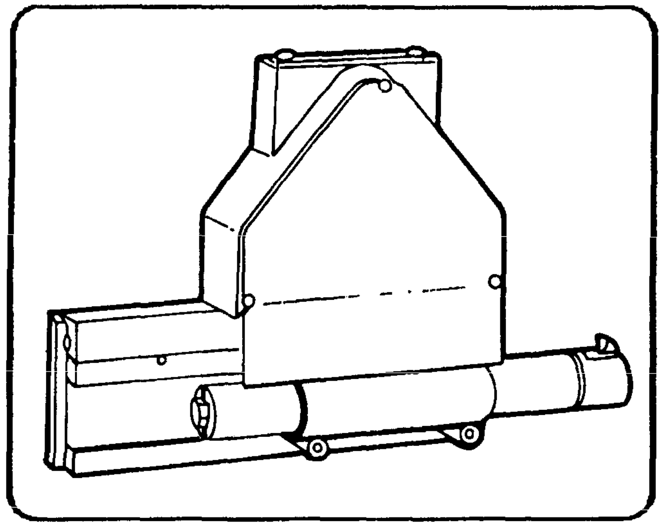


Figure 9-8. Internal keyseater and slotter (Versa-Shaper).

HIGH-SPEED END MILLING

For speeds higher than the basic unit can provide, a high-speed end milling and drilling head (Figure 9-9) bolts to the face of the Versa-M il. The head rotates 30° in either direction from center. Graduation marks on the face of the basic unit indicate the angle setting. Thirteen spindle speeds are available to the head directly from the motor through the use of a V-belt and pulleys. Arbors may be mounted in either end of the high-speed head. The spindle taper is the same as the basic unit. The high-speed head is used mostly for small diameter work such as end milling, drilling, or other related operations.

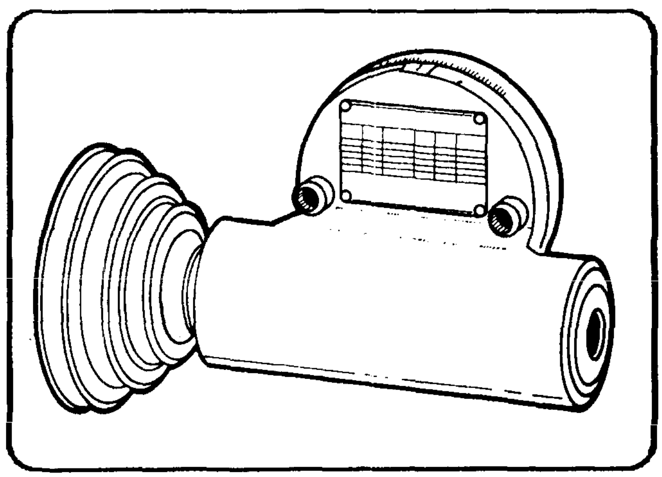


Figure 9-9. High-speed end milling and drilling head.

Indexing Head

The indexing head (Figure 9-10) mounts in the lathe head stock spindle to index work held in the lathe chuck, collet, or between lathe centers. The indexing head mandrel locks into a 1 1/8-inch or larger spindle bore; however, adapters for other bores are available. Forty turns of the dividing head crank rotates the lathe spindle one revolution. The indexing plate has 18 circles of holes allowing for divisions to be made in degrees, number of sides, or the number of teeth on gears or splines.

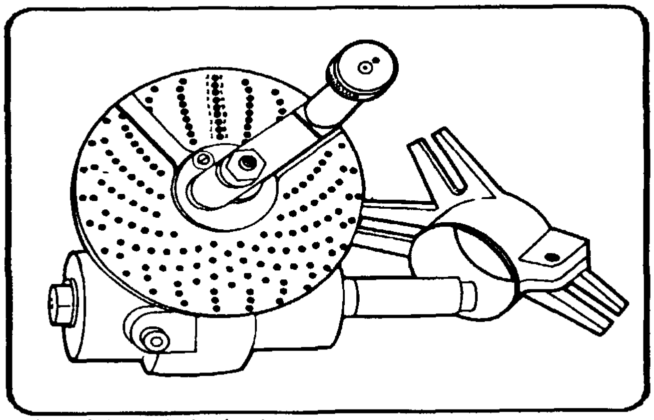


Figure 9-10. Indexing head.

T-Slot Mounting Adapter

Versa-Mil units are furnished with an adapter (Figure 9-11) that fits the T-slot of the compound rest on most conventional lathes to lock the Versa-Mil unit to the compound rest with two hex-head bolts. Four holes in the base of the Versa-Mil unit allow mounting the basic unit in any of four positions 90° apart. Mounting the basic unit by this method permits the use of the compound rest for angular movement where low mounting of the Versa-Mil is not required. Any operation normally done above the centerline of the workpiece is usually accomplished by using the T-slot adapter and the compound rest. Such operations include milling keyways, slots, and splines, angle milling, and gear cutting. Other operations such as drilling or boring may also be accomplished if they are performed above the center line of the work.

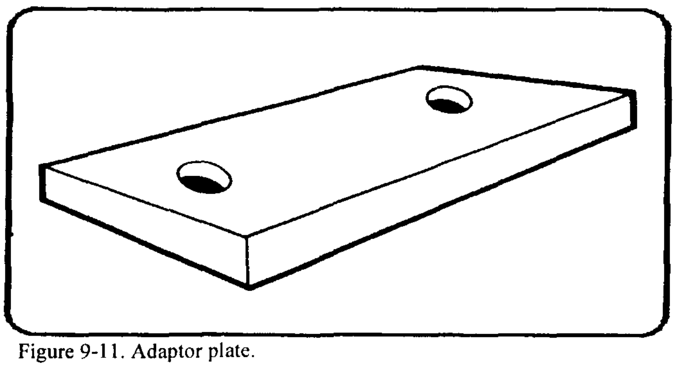


Figure 9-11. Adaptor plate.

Adapter Base Mounting

When a lower mounting of the Versa-Mil unit is required, the compound rest can be removed and replaced with a special adapter base (Figure 9- 12) that mounts directly on the cross-slide. The base plates are semifinished and may require drilling two mounting bolt holes and a pivot pin hole. The location of these holes depends upon the lathe model and size. The base plate adapter should be used for operations on or below the centerline of the workpiece. Such operations include milling keyways along the side of a shaft, surface milling with a shell end mill, and drilling or boring on the centerline of the workpiece. The compound rest must be removed prior to mounting the base plate adapter.

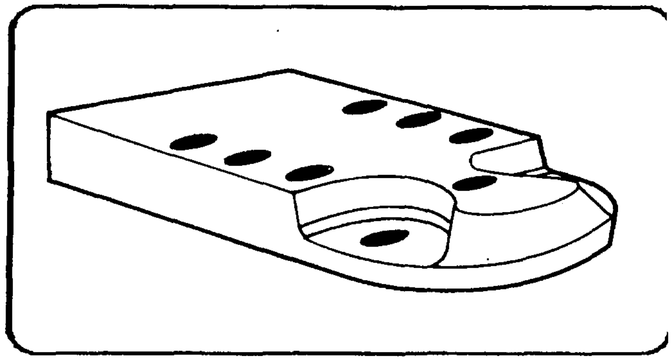


Figure 9-12. Adapter base mounting.

FEED TABLES

Although not part of the basic unit accessories, the feed table may be found in some shop sets. Rigid accurate feed tables (Figure 9-13) make the Versa-Mil unit a portable machine tool by providing two additional directions of. travel. Precision finished ways, adjustable gibs. and accurate lead screws calibrated to 0.001 inch assure accurate positioning and feed for the most precise machining. Feed tables for Versa-Mil units are available in four different models and all feed tables can be quickly converted to reduce table height when only one direction of travel is required.

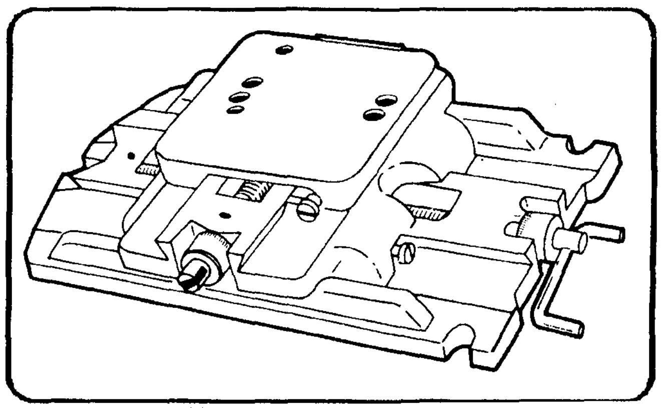


Figure 9-13. Feed tables.

Machining done on the lathe with a Versa-Mil allows the cutter to move along three different axes: vertical, lateral, and longitudinal (x, y, z). However, not all machining can be done using the lathe. Because the lathe allows longitudinal and lateral movement, mounting the Versa-Mil directly to a bench or piece of equipment would severely restrict its machining capabilities. Feed tables eliminate that restriction by providing those two additional directions of travel. Feed tables mounted directly to a bench or piece of equipment allow the Versa-Mil to perform machining in all three directions.

SELECTION OF ARBORS

When the basic unit is to be used independently or with an attachment other than the grinding attachments, an arbor and cutter must be selected and mounted. The cutter should be mounted onto the arbor first. The arbor should be secured in a vise to properly mount the cutter.

This ensures a properly torqued cutter and prevents the arbor from bending or causing damage to the Versa-Mil basic unit. When tightening the arbor nut, the pressure applied to the wrench should always be in the direction of the operator in case of slippage. Listed in the following paragraphs are various arbor styles and some of their uses. Note that they are similar to, but smaller than those used on a milling machine. Refer to chapter 8 for illustrations not listed.

Taper Arbors

Taper arbors (Figure 9-14) are designed primarily for use with Brown and Sharpe, or Morse standard taper shank tools.

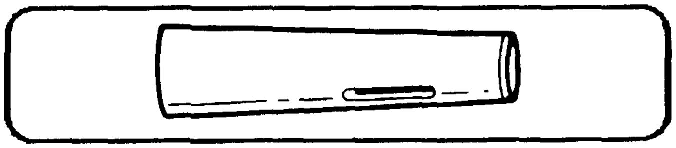


Figure 9-14. Taper arbor.

Fly-Cutting Arbor

The fly-cutting arbor may be used for boring, facing, gear repair, keyway milling, and form milling. This type of arbor allows the tool bit to be positioned at either 45° or 90° to the arbor axis.

Side-Milling Arbor

The side-milling arbor (Figure 9-15) is used with arbor-type cutters and slitting saws. This arbor is supplied with 1/8” and 3/8” spacing collars.

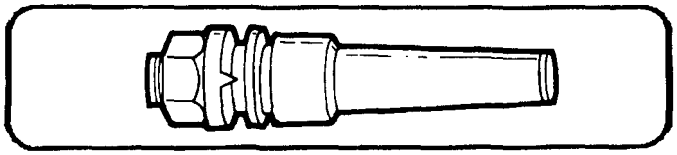


Figure 9-15. Side milling arbor.

Shell End Mill Arbor

The shell end milling arbor is used primarily for facing; however, milling a wide slot with a shell end mill can be accomplished.

Geared Chuck Arbor

This type of arbor is used for mounting chucks with a #3 Jacobs taper. The chuck itself is used primarily for drilling.

Straight Shank Arbor

The straight shank arbor with setscrews is used with straight shank drills of the correct size, end mills, and Woodruff key seat cutters.

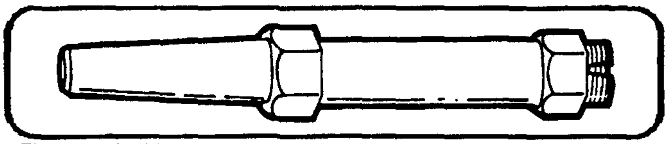


Figure 9-16. Side milling arbor.

Straddle Mill Arbor

The straddle mill arbor is used for milling splines on a shaft, milling hexagon or square shapes, and large keyways. Six spacers come with the Versa-Mil accessories, allowing milling of areas from 1/8 to 3 inches wide in 1/16 inch increments.

Threaded Angle Mill Arbor

The threaded angle mill arbor is used for milling angular grooves and dovetails.

SELECTION OF CUTTERS

After selecting the arbor, select the desired cutter for the machining process, mount the cutter on or in the arbor, and mount the arbor in the Versa-Mil unit or attachment. Ensure the arbor and spindle are free of dirt and burrs.

Woodruff Key Slot Cutters

This cutter has a 1/2-inch straight shank and is used for cutting Woodruff keyslots in a shaft. This cutter may also be used for cutting straight keyways in a shaft or similar operations.

Side Cutters

Side cutters are available in two basic styles. The stagger tooth side milling cutter should be selected for milling keyways and deeply milled slots, while the straight side milling cutters are usually used in matched sets for straddle milling or individually for side milling.

Shell End Mill Cutter

This cutter is used for slabbing or surfacing cuts and end or face milling.

Form Cutters

Form cutters are manufactured in a variety of shapes. Selection of the cutter depends upon the desired shape or form to be machined.

Fly Cutters

Fly cutters are usually square tool bits ground with the proper clearances for boring, facing, or counterboring. Fly cutters can also be ground to particular shapes for special jobs such as gear repair or spline milling.

END MILLS

End mills are manufactured in a variety of shapes and styles and should be selected in accordance with the job to be performed. The two fluted end mills are recommended for cutting keyways and for deep milling while the multiple flute end mills are designed for end milling and routing work.

SLITTING SAWS

Slitting saws are manufactured in a variety of styles and sizes and should be selected in accordance with the job to be performed. Use slitting saws to cut deep slots in the work and for cutting slots.

SELECTION OF GRINDING WHEELS

When the external grinding head, internal grinding head, or deep-hole grinding head is selected and mounted on the Versa-Mil, a wide range of grinding operations is made available. The data books published by the leading abrasive manufactures should be referred to for proper selection of grinding wheels as the variety of grinding done by Versa-Mil is to great for complete coverage of wheels in this manual.

WARNING

Use only abrasive wheels designed for the external or internal grinding heads that have been tested and found to be safe when operating at the speeds attained by these heads. Using incorrect untested wheels may result in breaking the abrasive wheel causing wheel fragments to be projected into the work area endangering personnel and equipment.

Straight Abrasive Wheels

Straight abrasive wheels are furnished in 46 and 60 grit sizes. The 46 grit wheel is a general-purpose wheel and should be selected for rough-grinding cylindrical parts, face plate grinding, and so forth. Select the 60 grit wheel for finishing and for tool and cutter grinding where finer finishes are required.

Straight Cup Wheels

Select a straight cup wheel should be selected for tool and cutter grinding, face plate grinding, and internal grinding of large holes.

Flare Cup Wheels

Select a flare cup wheel for general tool and cutter grinding.

Dish Wheels

Select a dish wheel for tool and cutter grinding such as grinding flutes and individual teeth of milling cutters.

VERSA-MIL OPERATIONS

SETUP

The Versa-Mil adds important machining functions to a lathe. With built-in power and vertical feed, it adds a third machining dimension, allowing the operator to mill, drill, bore, slot, shape, grind, and perform other special operations. The success of any Versa-Mil operation depends largely upon the judgment of the operator in setting up the Versa-Mil, selecting the proper cutter, and holding the cutter by the best means possible under the circumstances.

Preoperational Checks

Gibs should be as snug as possible and still allow the movement needed. Tighten all gibs not required for the operation being done to prevent movement and chatter. The adjusting bar on the back of the lathe carriage that holds the carriage onto the lathe bed should be snug enough to still allow a slight drag when feeding the lathe carriage. If the work is held between centers, they should be tight against the work and long pieces should be supported at the point where machining is being done. Unless both the Versa-Mil and the work are rigidly supported, it is difficult to obtain accurate results.

Mounting on a Lathe

The Versa-Mil may be mounted on the front or the rear of the lathe carriage. On the front, it may be set on the compound rest or directly on the cross slide. A more permanent and generally more useful mounting is at the rear of the lathe carriage, where it may be left until it is needed.

Squaring the Versa-Mil to the Lathe

For accurate milling cuts, it is necessary to square the Versa-Mil to the lathe (Figure 9-17). The front compound face of the Versa-Mil is a reference surface machined in relation to the spindle. A square can be set across this face and squared to the chuck or face plate of the lathe. For work between centers, the Versa-Mil can be squared to the workpiece. After the machine has been squared on the compound rest of the lathe, the compound rest can be loosened for adjusting the spindle to various angles using the graduated scale on the compound rest. For extremely precise adjustments and settings, use the dial indicator or vernier protractor.

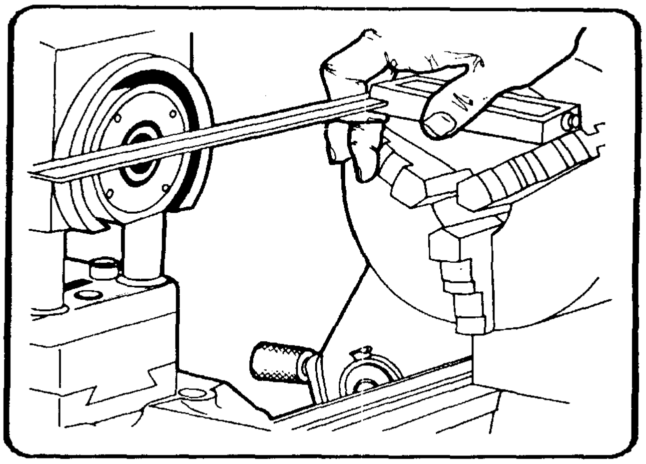


Figure 9-17. Squaring the Versa-Mil to the lathe.

MILLING SQUARE END KEYWAYS

Conventional milling is recolmmended when using the Versa-Mil on a lathe as the lathe's feeds and bearings are not designed for upward pressure on the carriage. Cutting square end keyways (Figure 9-18) can be accomplished with the Versa-Mil using a variety of different cutters and speeds. The Versa-Mil is usually set on top of the compound rest with the spindle of the Versa-Mil parallel with the travel of the compound rest. Select and mount the cutter to the appropriate arbor. A stagger tooth side milling cutter the width of the keyway is the most satisfactory cutter to use for square end keyway milling operations; however, plain milling cutters may be used. Mount the arbor into the Versa-Mil spindle and tighten.

CAUTION Do not over tighten as the pin in the back of the Versa Mil may shear

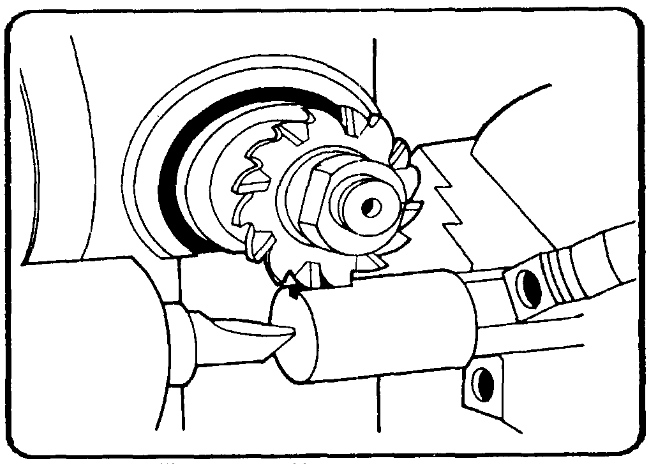


Figure 9-18. Milling square end keyways.

Speed Selection

If a good flow of coolant is available to the cutter, choose or select speeds near the top of the recommended cutting speeds for the operation being performed, type of cutter used, and material being milled. If milling is to be done dry, then use a speed at the lower end of the recommended cutting speeds.

Centering the Cutter

To center the cutter over the work, first ensure the backlash is removed from the cross slide. Next, start the Versa-Mil and reference the cutter to the side of the work using a paper shim. Zero the cross feed dial; then, raise the Versa--Mil above the top of the work. To determine the distance the cutter must move, add one-half of the diameter of the cutter plus one-half the diameter of the workpiece plus the thickness of the paper shim. Keep in mind some latches only move half the distance shown on the crossfeed dial. After the cutter has been moved over the center of the work lock the cross slide to prevent movement during milling. See Figure 9-19.

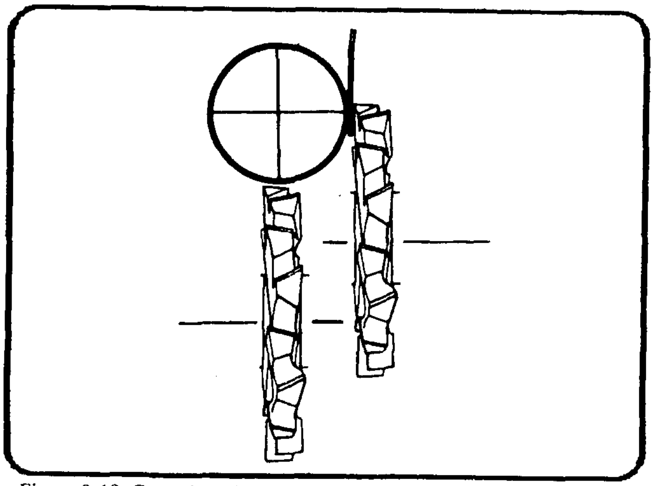


Figure 9-19. Centering the cutter.

Depth of Cut

Start the Versa-Mil and reference the cutter to the top of the workpiece using a paper shim. The depth of cut equals one-half the key thickness plus the chordal height plus the thickness of the paper shim. Tables for chordal height may be found in the new American Machinist's Handbook or Machinery's Handbook. A simple approximate formula for chordal height is key thickness squared, divided by four times the shaft diameter. After the depth of cut is determined and set, tighten the post binding setscrew to prevent the basic unit from moving during machining.

Feed Rate

The rate of feed will vary from 0.001-inch chip thickness per tooth to as much as 0.008 inch per tooth. Determine the feed rate by multiplying the number of teeth on the cutter times the desired chip thickness times the RPM of the cutter. A chip thickness of 0.001 to 0.004 is considered a finishing cut while a chip thickness heavier than 0.004 is considered a roughing cut. Most milling operations involving the Versa-Mil are fed by hand. The operator should attempt to feed the cutter at a consistent rate with each tooth taking the same chip thickness. Power feeding is recommended when long cuts along a shaft or workpiece are necessary. To do this, mount the steady rest on the lathe close to the headstock and clamp the steady rest tightly against the workpiece. Lubricate the headstock center or use a ball bearing type center to allow the headstock spindle to rotate freely while the workpiece remains stationary. If a ball bearing center is not used, maintain low spindle speeds to prevent overheating the work. Feed rates during power feeding are adjusted using of the quick change gearbox on the lead screw.

INTERNAL KEYWAY AND SPLINE CUTTING

After the internal diameter of gears or sleeves have been machined to size, keyways or splines may be cut into the work with the Versa-Shaper without removing the work from the lathe chuck (Figure 9-20). This has a major advantage of saving time by not having to change setups.

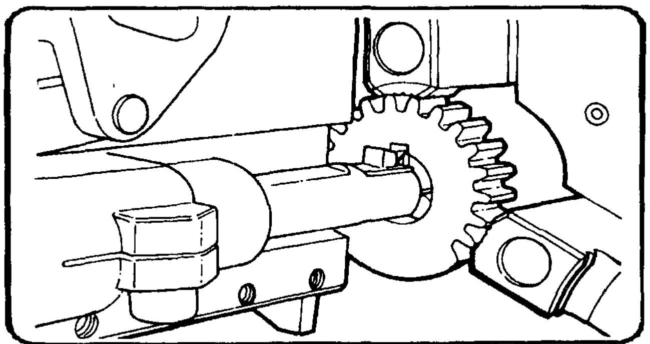


Figure 9-20. Internal keyway and spline cutting.

Sizes of Keyways

Each of the standard widths of keyways from 1/8 to 1/2 inch may be cut with one of the standard keyway cutters available with the Versa-Mil. Wider keyways may be cut with one of the standard cutters by cutting the slot to the proper depth and enlarging it by feeding the cutter first to one side and then the other through the use of the cross slide lead screw.

Depth of Cut

Determine the depth of cut by the amount of feed applied to the basic unit lead screw. However, it is necessary to allow the Versa-Shaper to take additional cuts (free cuts) until no further material is removed before taking a measurement. This will assure accurate keyways or splines being machined in the gear or sleeve.

Direction of Feed

Whenever practical, mill keyways and splines by feeding upward with the Versa-Shaper. This will cause the lathe carriage to be held more firmly in contact with the lathe ways and the lathe bed, permitting heavier cuts to be taken.

Clearance

After the Versa-Shaper is set up, run through the entire stroke cycle turning the worm sheave by hand. This will ensure that the cutter clears the work at both ends and does not strike the lathe chuck or encounter any other obstructions.

PLAIN MILLING

Plain milling or slabbing (Figure 9-21) is a term applied to many operations such as face milling, milling a hex or square shape, or milling flat surfaces along the side of a workpiece. The process of plain milling normally involves removing large amounts of material with either a shell end mill or side milling cutters to form a flat surface. Work may be held either in the lathe chuck or between centers for plain milling.

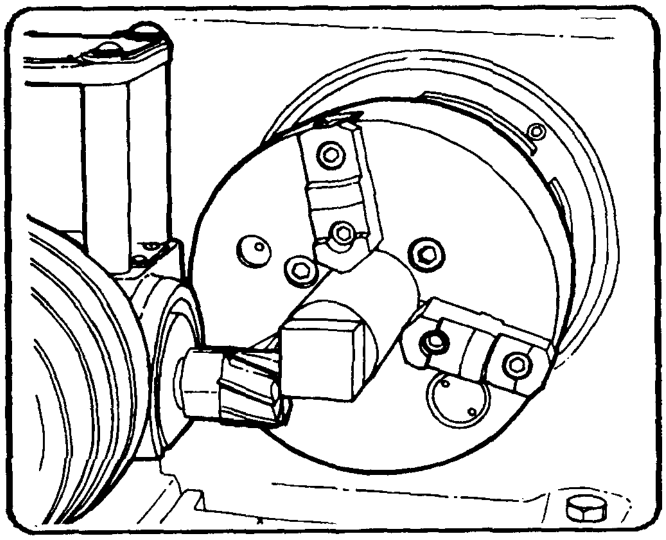


Figure 9-21. Plain milling.

Depth of Cuts

In the case of shell end mills, the depth of cut should not exceed the depth of the teeth or flutes. With side milling cutters, the depth of cut is controlled by the diameter of the cutter. For deep cuts, a staggered tooth, side milling cutter is recommended. Extremely light cuts should be avoided if possible as the cutter tends to slide over the work, heating and dulling the cutter which may result in putting undo pressure on the arbor and carriage causing excessive chatter.

Milling Feeds

The best milling performance is obtained when each tooth of the cutter takes a full chip. When milling steel, for example, the ideal feed is 0.005 inch. Depending on the width of the cutter and machinability of the material, it may be desirable to reduce the depth of cut and increase the rate of feed to maintain chip thickness. Chatter is likely to result when chips are too thin, causing cutter life between grindings to be reduced.

DRILLING

Many drilling and boring operations not ordinarily possible on the lathe are easily performed with the Versa-Mil mounted on the lathe. The Versa-Mil is usually fed by hand using either the either carriage, cross slide, or compound rest. Check the operators manual supplied with the Versa-Mil for information concerning power feeding when drilling.

Off-Center Drilling

Off-center drilling and boring may be performed by positioning the Versa-Mil spindle parallel with the lathe axis and maneuvering the drill by means of the cross slide and the Versa-Mil lead screw. This allows the complete machining of irregularly-shaped items without removing them from the lathe chuck. See Figure 9-22.

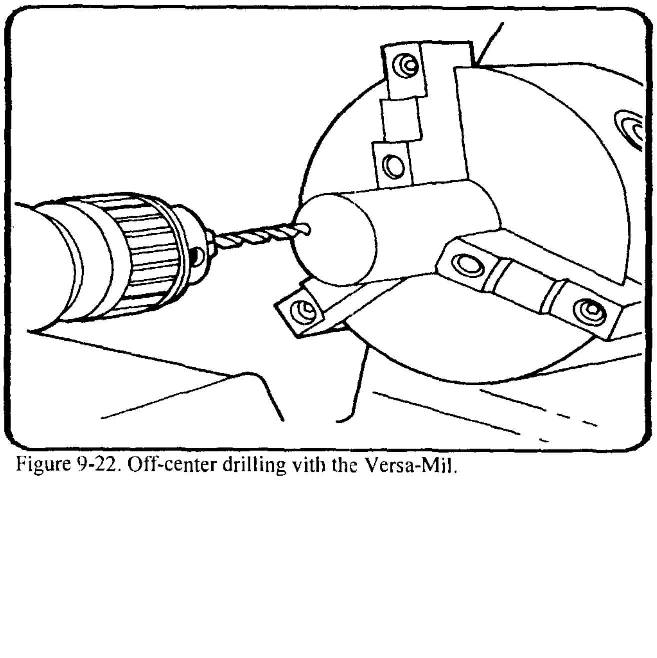


Figure 9-22. Off-center drilling vith the Versa-Mil.

Angular Drilling

With the Versa-Mil mounted on the compound rest, holes may be drilled at any angle in relation to the lathe axis by setting the compound rest at the desired angle and feeding the drill into the work with the compound rest lead screw. To use power feeding with the taper attachment, set the taper attachment and Versa-Mil spindle parallel with the hole to be drilled. The work must be held in position to prevent turning when the lathe carriage feed and head stock spindle are engaged. See Figure 9-23.

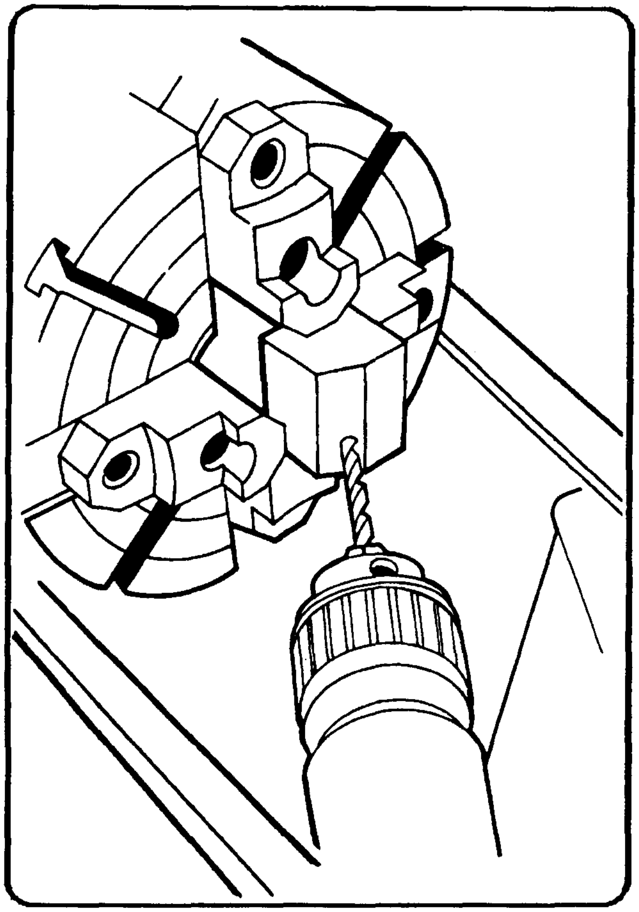


Figure 9-23. Angular drilling with the Versa-Mil.

Index Drilling

Stock held in the lathe chuck or between centers can be drilled at regular intervals around the center or perimeter of a workpiece by using the indexing head to position the work. A considerable amount of setup time and effort is saved after positioning the drill for the first hole to be drilled.

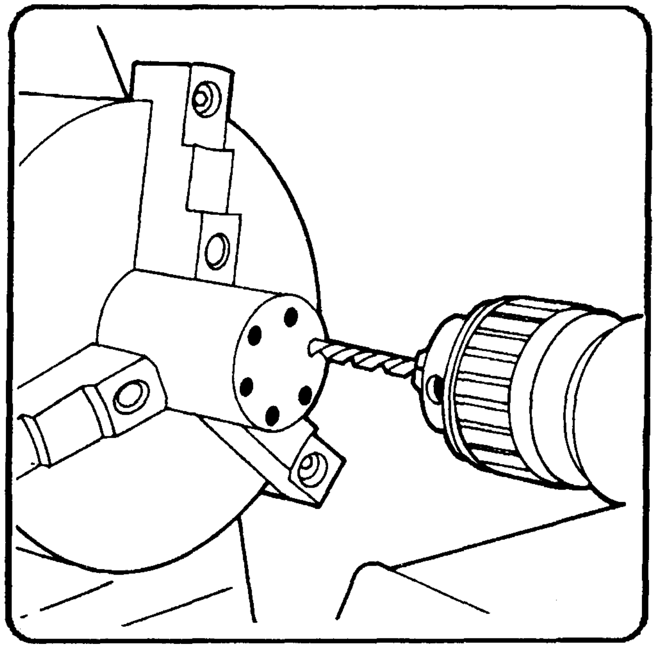


Figure 9-24. Index drilling.

Additional Drilling Applications

Drilling with the Versa-Mil attached to a feed table, turret lathe, or vertical boring mill is unique. Special drilling operations with these pieces of equipment are covered in the operator's manual on the Versa-Mil. See Figure 9-24.

WOODRUFF KEYSLOT MILLING

Milling Woodruff keyslots (Figure 9-25) in shafts is very similar to milling straight keyways in the basic setup, centering the cutter, and feed rate. The only difference in milling a Woodruff keyslot is that the carriage must be locked down in addition to the cross slide, if cutting from the top of the workpiece, to prevent the basic unit from moving during milling. Cutting a Woodruff keyslot is relatively simple since the proper size cutter has the same diameter and width of the key to be inserted. The work may be held in the lathe chuck or between centers and the cutter may be on an arbor or in a drill chuck. After the cutter has been centered on the work, the cutter is fed directly into the work until the proper depth of cut has been achieved.

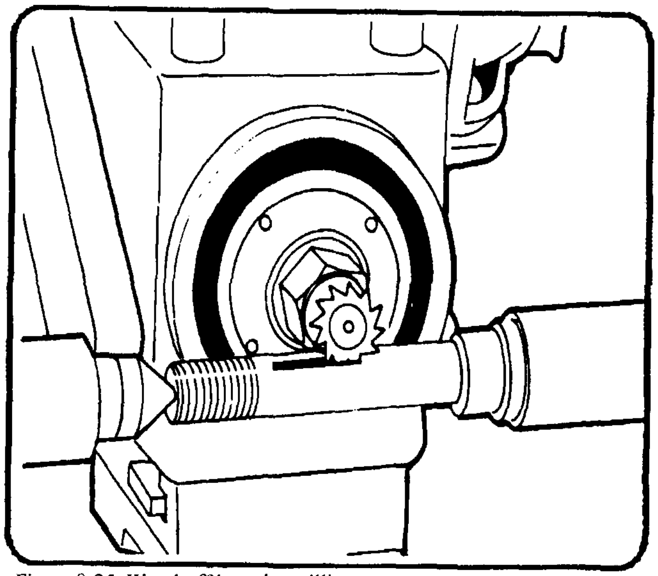


Figure 9-25. Woodruff key-slot milling.

INDEXING

An indexing head comes with the Versa-Mil and is installed on the headstock of the lathe to permit indexing a workpiece. Even though the workpiece is mounted in a conventional manner in the lathe, the headstock spindle should never be allowed to rotate under power with the indexing head attached as this would cause severe damage to the equipment. It is always a good practice to unplug or turn off the main power switch on the lathe in this situation.

Mounting the Workpiece

A workpiece may be supported in the lathe between centers, against the faceplate, or in the lathe chuck. If the work is mounted between centers, a lathe dog is mounted on the work and used to transfer movement from the faceplate to the work.

Indexing the Work

Indexing is the process of controlling the rotational position of a workpiece during machining. The indexing head attaches to the left end of the lathe headstock and locks into the headstock spindle using an expansion adapter. With the indexing head mounted to the lathe, the work will not rotate unless the crank arm of the indexing head is moved. Forty complete turns of the crank arm move the lathe spindle one revolution. The indexing plate contains a series of concentric rings with each ring containing a different number of holes. The workpiece is indexed by moving the crank arm from one hole to another through a calculated pattern of turns and holes. To determine the correct pattern of turns and holes and which ring to use, refer to Chapter 8, Indexing a Workpiece.

FORM MILLING

Form milling is the process of machining special contours, composed of curves and straight lines or entirely of curves, in a single cut. Gear cutting may be considered form milling by definition; however, the definition is usually restricted to the use of convex, concave, or corner rounding cutters. These form cutters are manufactured in a variety of radii and sizes and may be grouped or ganged together on an arbor to mill intricate shapes. Convex (curved or rounded outward) cutters mill concave (curved or rounded inward) shapes while concave cutters are used to mill convex shapes.

ANGLE MILLING

Angle milling is milling flat surfaces which are neither parallel nor perpendicular to the work. Angular milling can be divided into several different types of setups.

Single Angle Milling Cutters

Single angle milling cutters are mounted on an arbor and the arbor is then mounted to the basic unit or universal head. The unit is then squared to the workpiece and the work is milled in a conventional manner. This type of cutter is manufactured in a variety of angles with the most common angles being 45°, 50°, 55°, or 60°.

Dovetail Milling

When cutting dovetails with the Versa-Mil, the workpiece is usually held in the lathe chuck or mounted on a face plate. The tongue or groove of the dovetail is first roughed out using a side milling cutter, after which the angular sides and base are finished with the dovetail cutter. See Figure 9-26.

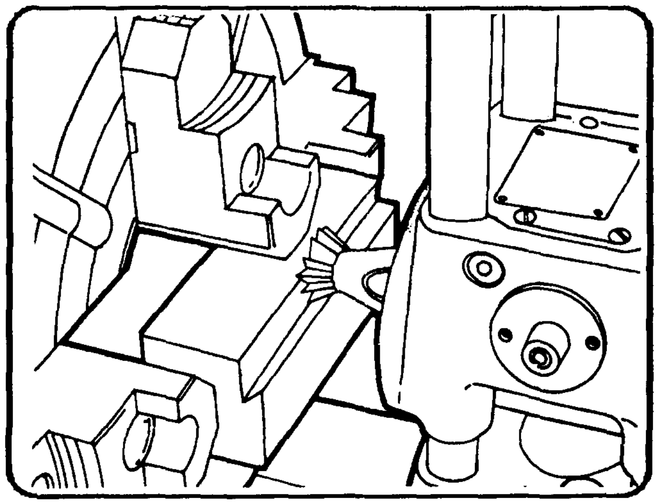


Figure 9-26. Dovetail milling.

Compound Rest

Angular milling may also be accomplished on the Versa-Mil by squaring the Versa-Mil on the compound rest and setting the compound rest to the desired angle. With this method of angular milling, the cutter is usually a shell end mill and the work is either held in the lathe chuck or mounted on the faceplate.

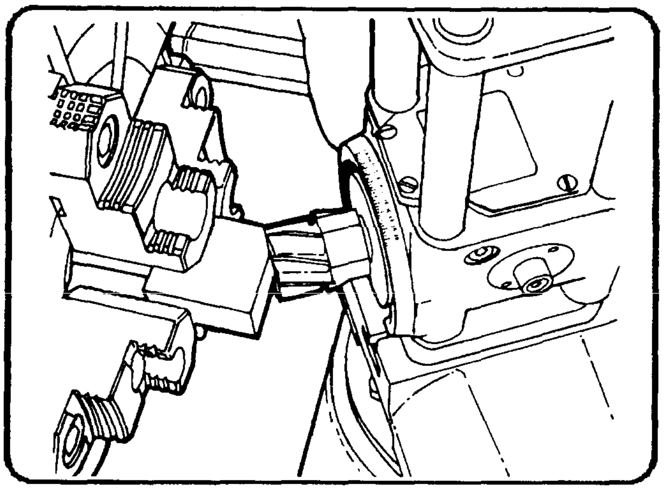


Figure 9-27. Compound rest.

Universal Head

Angles may also be milled on a workpiece using the universal head. This head may be tilted to 180° in either direction of center. Complex angles may be machined with the universal head used in conjunction with the compound rest or the tailstock offset method. See Figure 9-28.

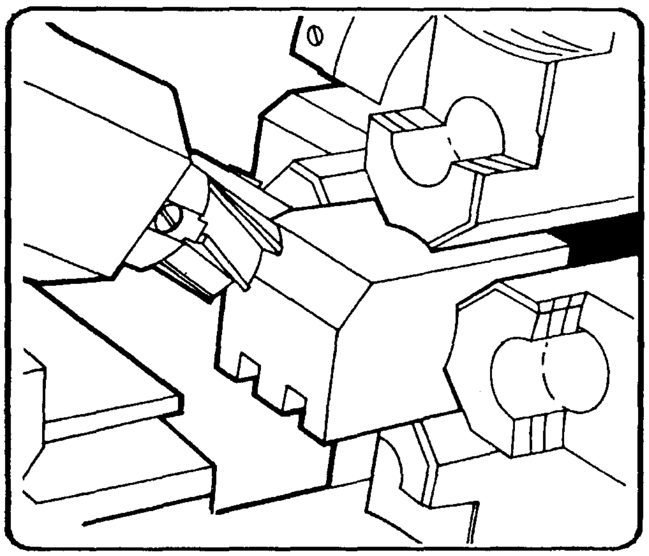


Figure 9-28. Universal head angle milling.

Tailstock Offset

This type of angular milling is accomplished by squaring the unit to the tailstock spindle or faceplate. Normally, a shell end mill is used in this type of milling. Work is mounted between centers and the tailstock is offset to the desired angle for milling. The work may be rotated with the indexing head to mill additional surfaces on the workpiece. See Figure 9-29.

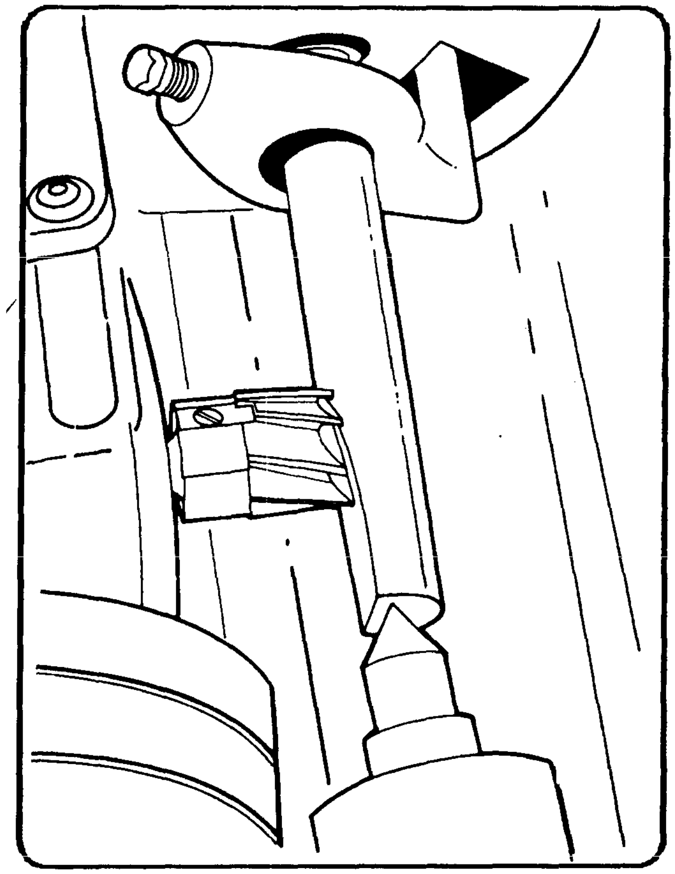


Figure 9-29. Tailstock offset milling.

STRADDLE MILLING

Straddle milling (Figure 9-30) is the machining of two parallel surfaces in a single cut by using two cutters separated by spacers, washers, or shims. Use straddle milling in spline milling or the cutting of squares or hexagons on the end of a cylindrical workpiece. The workpiece is mounted between centers to mill splines on a shaft and mounted in the lathe chuck to mill squares or hexagons. In both cases, the indexing head is used to rotate the work after each cut.

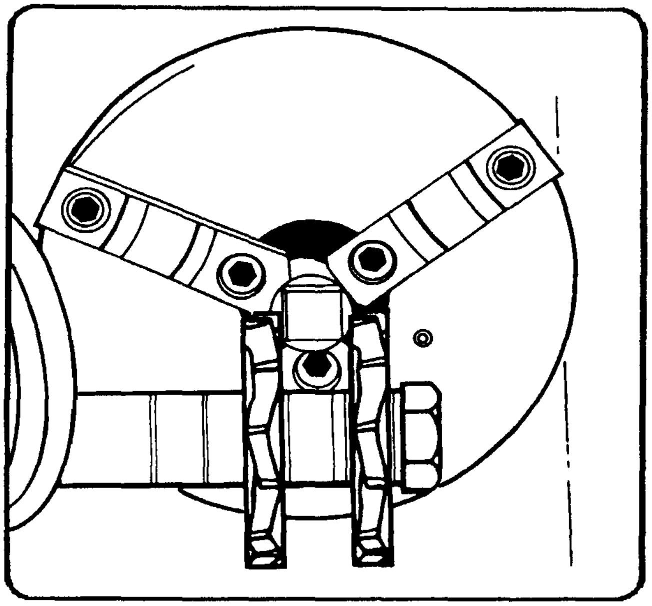


Figure 9-30. Straddle milling.

GANG MILLING

Gang milling differs from straddle milling in that two or more cutters of different diameters or shapes are mounted on the same arbor to mill horizontal surfaces. Cutter combinations in gang milling are virtually unlimited and are determined by the desired shape of the finished product.

SPLINE MILLING (EXTERNAL)

Splines are often used instead of keys and keyways to transmit power from the shaft to a hub or gear. Splines are a series of parallel keys and keyways evenly spaced around a shaft or interior of a hub. Splines allow the hub to slide on the shaft either under load or freely. This feature is found in transmissions, automotive mechanisms, and machine tool drives. Manufactured splines are generally cut by bobbing and broaching; however, this discussion will be limited to field expedient methods. Standard splines on shafts and spline fittings are cut with 4, 6, 10, or 16 splines.

The dimensions depend upon the class of fit and the shaft diameter. The class of fit may be permanent, sliding fit not under load, and sliding fit under load. Table 8-8 in Appendix A lists the standard dimensions for the different classes of fits. Shafts may be milled several different ways.

The most common way is to use two side milling cutters separated by spacers, with the width of the spacers equal to the width of the spline. The splines are cut by straddle milling each spline to the proper depth and indexing around the shaft for each spline. A narrow plain milling cutter is used to mill the spaces between the splines to the proper depth. It may be necessary to make several passes to mill the groove uniformly around the shaft. A formed cutting tool or cutter may also be used for this operation.

SPLINE MILLING (INTERNAL)

After a hub or gear has been drilled and bored to the finished internal minor diameter, internal splines may be cut into the hub or gear by using the Versa-Shaper (Figure 9-31). The indexing head provides the means to locate each spline to be cut. For this operation, the milling is continued until the desired class of fit is obtained. For field expedience, it is best to machine the mating parts to match if possible.

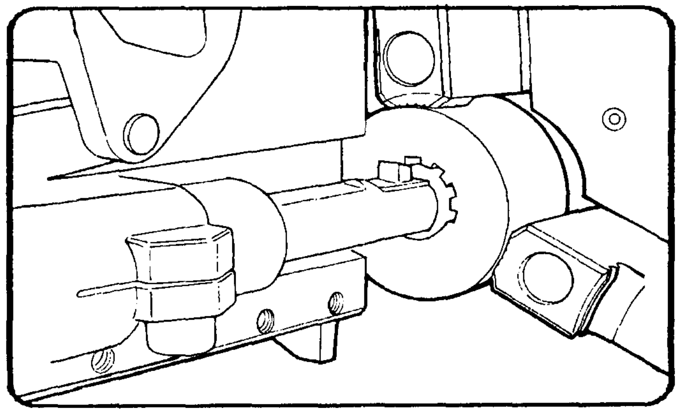


Figure 9-31. Spline milling internal splines.

SLOTTING

Slotting with the Versa-Mil (Figure 9-32) covers a wide variety of operations from milling long wide slots in material to cutting curved or thin slots. Workpieces may be mounted in the lathe chuck or between centers for slotting operation.

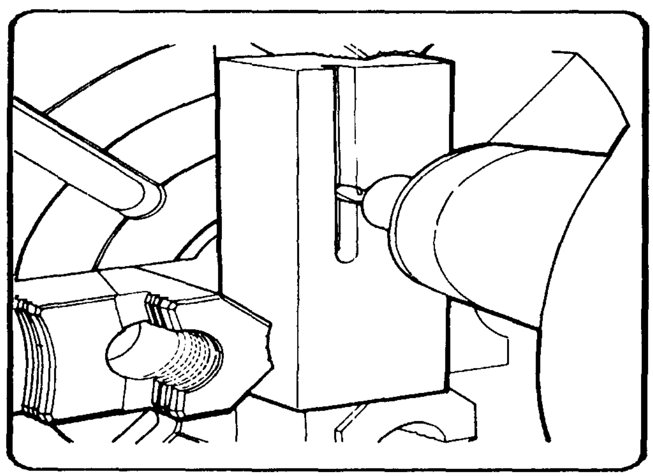


Figure 9-32. Slotting with the Versa-Mil.

Longitudinal Slots

Longitudinal slots along a shaft or other large piece may be cut in the material in the same manner as milling keyways with end mills. It is often desirable to use a cutter smaller than the width of the slot. The reason for this is, when the cutter is as wide as the slot, one side of the cutter is climb milling while the opposite side of the milling cutter is performing conventional milling. This causes a difference in the finish between the two sides of the slot. A roughing out of the slot should be made first, followed by a finishing cut down one side of the slot and returning on the other side.

Narrow Slots

For narrow slots, use slitting saws rather than end milling cutters. When using slitting saws, reduce speeds and feeds to extend the life of the cutter.

FLY CUTTING

Fly cutting (Figure 9-33), also called single-point milling, is one of the most versatile milling operations available to the machinist. Fly cutting is done with a single-point cutting tool, like the lathe or shaper cutting tool, held in a fly cutting arbor. Formed cutters are not always available and there are times when special form cutters are needed only for a very limited number of parts or operations; therefore, it is moreeconomical to grind the desired form on a lathe cutter bit rather than order a special form cutter. The fly cutter is used to great extent in the reshaping of repaired gears because the tool bit can be ground to the shape of gear teeth available. Fly cutting can also be used in cutting standard and special forms of splines.

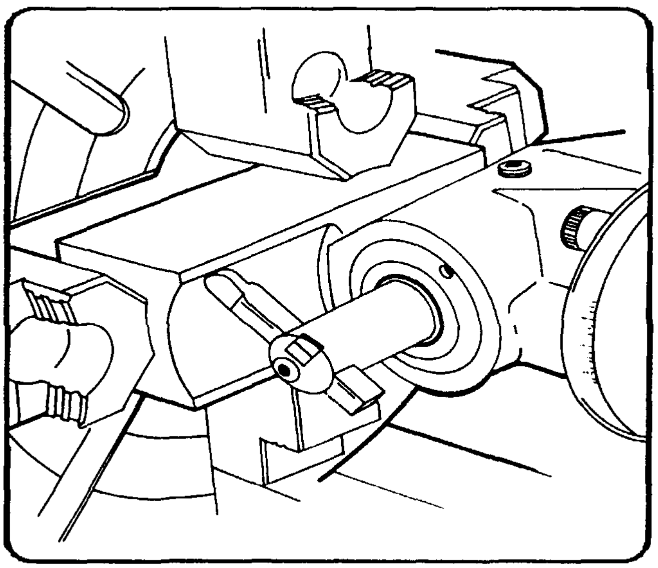


Figure 9-33. Fly cutting.

Plain or Face Milling of Soft Nonferrous Metals

Plain or face milling of soft nonferrous" metals such as aluminum, with a fly cutter produces a high quality finish. Boring holes with a fly cutter is generally not desirable because of the difficulty in positioning the cutter and controlling the diameter. The short arbor allows boring of only very shallow holes.

Gear Cutting

A variety of gears, pinions, and sprockets can be fabricated on the lathe using the Versa-M il. By referring to various texts and references for detailed data and instructions on gears and gear cutting, the operator can develop different methods of mounting the Versa-Mil to the lathe to perform gear cutting. The basic unit and the indexing head are the two basic elements needed to cut gears. When large diameter gears need to be cut, the universal head is used to mill the side of the gear.

Spur gears are the most common type of gear used in the field and the correct cutter to use for this type of gear is determined by the pitch of the teeth and the number of teeth required. Standard cutter catalogs supply the data necessary to select the correct cutter.

Gear Cutting with the Basic Unit and an Involute Gear Cutter

In this setup, Figure 9-34, the gear blank is first turned to the correct diameter using a mandrel mounted between centers. The blank should remain on the mandrel after turning. The lathe dog should be wedged against the faceplate to eliminate backlash and the indexing head mounted to the lathe spindle to position the individual teeth. The basic unit is mounted on the compound rest with the faceplate parallel to the lathe center and an arbor with an involute gear cutter, stamped with the correct pitch and number of teeth, is installed in the basic unit. After the cutter is positioned, lock down the cross feed by tightening the gibs. When the correct depth is reached, tighten the post locking screw on the basic unit. The cutter is then fed into the blank by hand using the lathe carriage wheel.

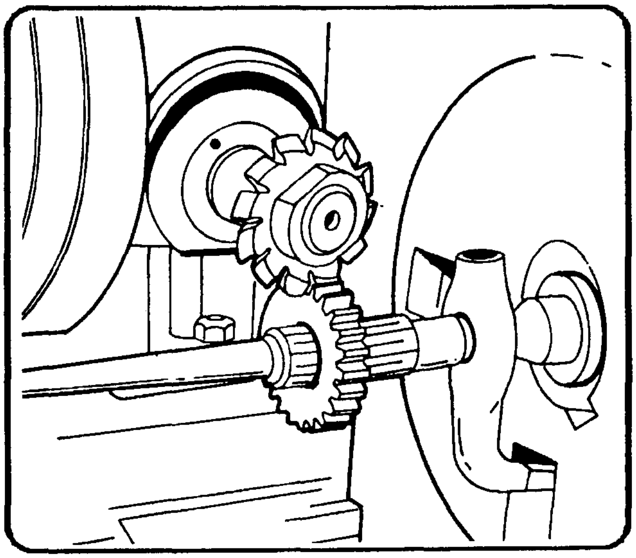


Figure 9-34. Gear cutting with an involute gear cutter.

Gear Cutting with the Basic Unit and a Fly Cutter

When an involute gear cutter is not available or delay in obtaining one is too great, a fly cutter is used. The only difference is that a fly cutter with a 5/16-inch square tool bit, ground to the correct shape, is used instead of an involute gear cutter. See Figure 9-35.

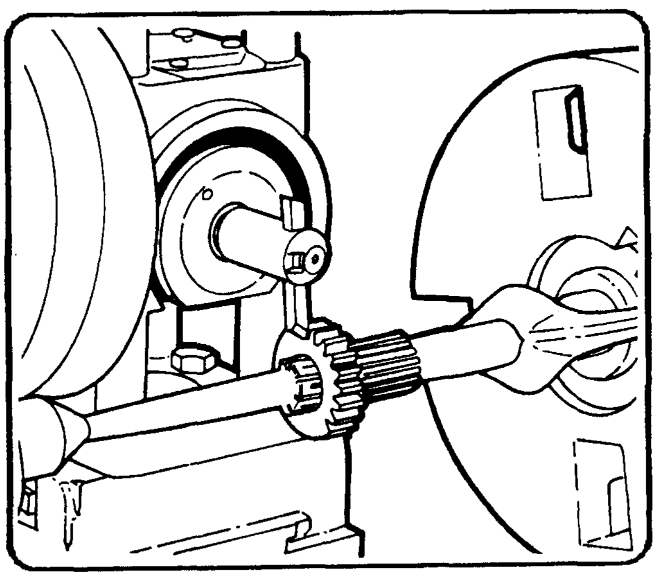


Figure 9-35. Gear cutting with a fly cutter.

Gear Cutting with a Universal Head

Used this setup with either a fly cutter or an involute gear cutter on gear blanks larger than 8 inches in diameter. See Figure 9-36.

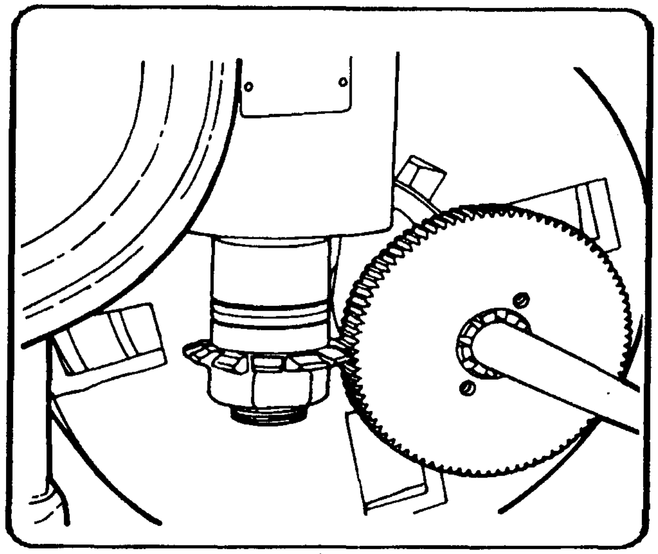


Figure 9-36. Gear cutting with the universal head.

WHEEL DRESSING

Wheel dressing (Figure 9-37) with the diamond dresser is a must for accurate precision grinding. Dress wheels before starting any grinding job and again prior to the finishing cut. The diamond dresser is the most efficient type of wheel dresser for truing wheels used in precision grinding. The diamond point is the only usable part of the diamond and must be inspected frequently for wear. Rotate the diamond slightly in the holder between dressings to keep the point sharp. A dull diamond will press the wheel cuttings into the bonded ores of the wheel, increasing the wheel's hardness. When truing the wheel, the diamond should be centered on the wheel and slanted between 5° and 15° in the direction of wheel rotation to prevent chatter and gouging.

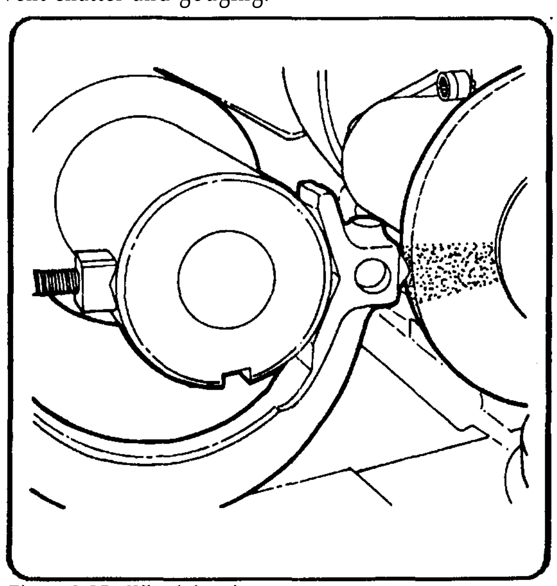


Figure 9-37. Wheel dressing.

The grinding wheel should rotate at or slightly less than operating speed when truing or dressing, never at a higher speed After truing, slightly round the edges of the wheel with an oilstone to prevent the wheel from chipping, unless the work requires sharp comers. Start the dressing process at the highest spot on the wheel, normally the center, and feed at a uniform rate with a 0.002 inch depth of cut per pass. Too slow a feed will glaze the wheel while too fast a feed rate will leave dresser marks on the wheel.

GRINDING

A wide range of grinding is made available to the machinist by using the Versa-Mil and the different grinding heads supplied with the unit. Refer to references published by the leading abrasive manufacturers when selecting the proper wheel for the job being performed. For maximum metal removal and minimum wheel wear, surface speeds of the grinding wheel should be near the highest allowable speed for the wheel size. Light cuts at full speed will remove metal faster than deep cuts at slow speeds. In general, rough cuts average 0.002 inch per pass, while finishing cuts average 0.0005 inch. The spindle rotation should be selected to throw wheel and metal debris away from the operator. When movement of the work is required during grinding, the work and the wheel should rotate in the same direction. This allows the wheel and work to move in opposite directions at the point of contact. The precision grinding may be done either wet or dry.

GRINDING LATHE CENTERS

Before grinding work between centers takes place the centers should be ground true (Figure 9-38). With the center mounted in the lathe headstock, mount the Versa-Mil on the compound rest and set the compound rest at one-half the included angle of the center. Grind the center by feeding the compound lead screw by hand at a uniform rate of feed.

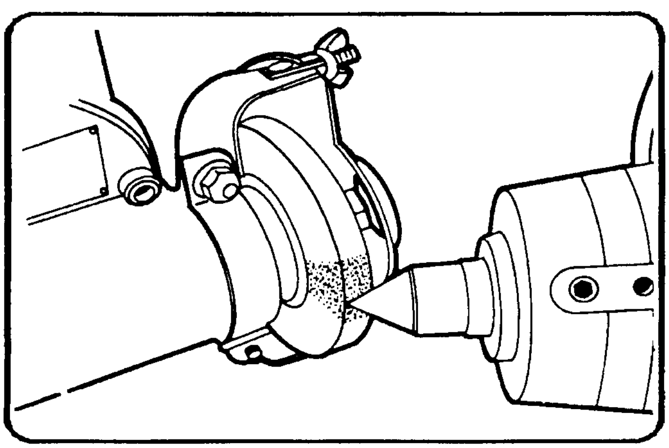


Figure 9-38. Grinding lathe centers.

CYLINDRICAL GRINDING

The lengths and diameters of shafts ground on a lathe are determined by the lathe swing and the distance between the lathe centers. Mount the Versa-Mil on the compound rest with the face of the basic unit parallel to the work surface. In cylindrical grinding (Figure 9-39), the work rotates slowly while the wheel rotates close to the highest allowable speed. The wheel should never leave the work at either end of the cut in order to produce a smooth surface free of wheel marks. Direct the spark pattern downward onto a dampened cloth to prevent very small particles of material from getting into and destroying machined surfaces. A spark pattern directed downward and away from the operator indicates the wheel is too low on the work, while a spark pattern that is directed downward and toward the operator indicates the wheel is too high on the work. Conical grinding can be accomplished with either the taper attachment or by the tailstock offset method.

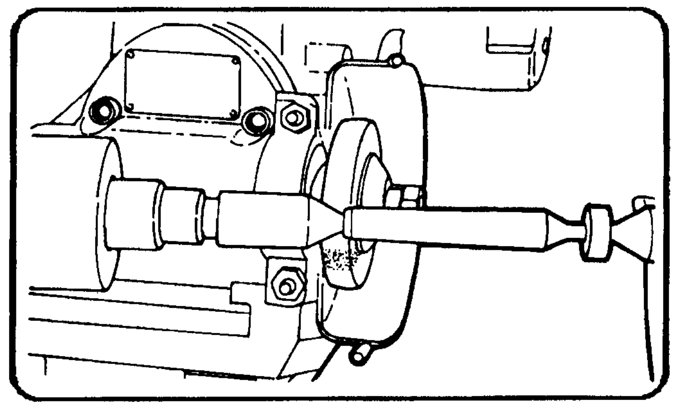


Figure 9-39. Cylindrical grinding.

INTERNAL GRINDING

Holes and bores as deep as 18 inches may be internally ground using the Versa-Mil. The diameter of the hole may be any size larger than 3/4 inch. Either the internal grinder with the taper spindle or the deep-hole grinder may be used, depending on the hole dimensions. Internal grinding differs from external grinding basically in one area. The surface contact between the work and the wheel is much greater in internal grinding, causing the wheel to load and glaze much more quickly. This loading or glazing will cause unnecessary vibration and produce a poor surface finish. A coarser wheel grain structure, which provides better chip clearance, or a softer wheel that will break down more easily, should be used for internal grinding. While grinding, the wheel should clear the end of the work at least one half the wheel thickness but not more than two thirds. If the wheel is allowed to clear the end of the work entirely, a bell-shaped effect will be produced.

Tapered Spindle Grinder

For shallow and small diameter holes up to 6 inches in depth, use the tapered spindle internal grinder. Tapers may also be ground on the work by using either the taper attachment or the compound rest. See Figure 9-40.

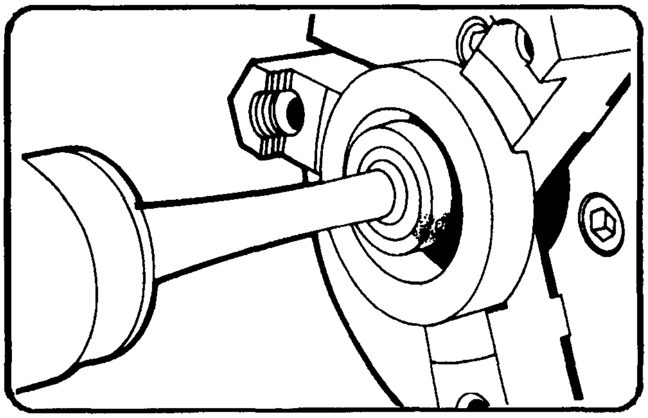


Figure 9-40. Tapered spindle grinder.

Deep Hole Grinder

The deep-hole grinder with the extended housing offers a rigid precision grinder for holes as deep as 18 inches. Tapers may also be ground with the deep-hole grinder. See Figure 9-41.

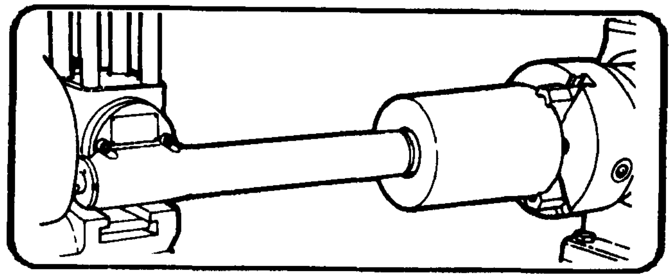


Figure 9-41. Deep-hole grinder.

Versa Grinder Head

The Versa-Mil external grinder with the wheel guard removed may be used for internal grinding of large bored pieces if a considerable amount of stock must be removed and the hole depth does not exceed the unit clearance. This setup permits the operator to grind internally, externally, and face in one setup, assuring a true relation between the three different surfaces. See Figure 9-42.

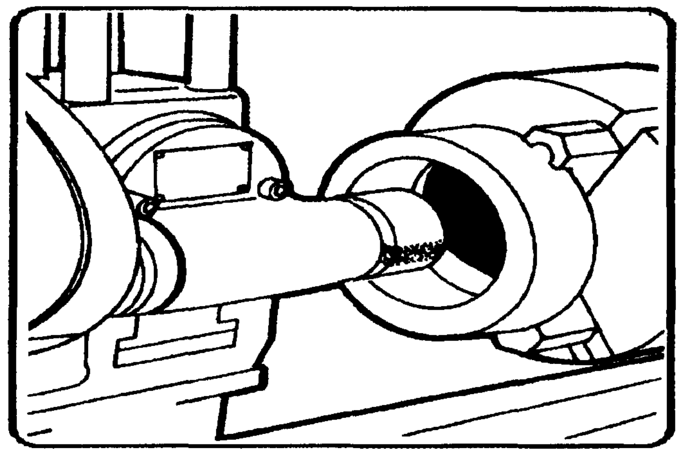


Figure 9-42. Versa grinder head.

SPECIAL OPERATIONS

TOOL GRINDING

The Versa-Mil mounted on the compound rest of a lathe will duplicate the full range of tool and cutter grinding offered by conventional tool grinders. For successful results, the lathe should be in excellent operating condition and preferably small in size to permit the close setting of feeds and angles. Versa-Mil spindles use precision, spring-loaded duplex bearings to eliminate play in the grinding wheel forsuccessful tool grinding. The Versa-Mil tool rest is solidly constructed to provide rigid support with a tip that is designed for smooth, solid contact under the teeth or flutes of the tool being ground. The operator familiar with tool grinding and the use of the Versa-Mil soon develops methods for grinding the various types and forms of cutters. Tool grinding cannot be completely covered in this manual, and it is suggested that reference material covering tool grinding be consulted for complete detailed instructions.

Selection of Grinding Wheels

Grinding wheels should be in the medium grit range for tool and cutter grinding. The shape of the cutting tool will determine which wheel design to use. Abrasive manufacturers' catalogs should be referred to for proper wheel selection.

Depth of Cut

Light traversed cuts should be used to avoid overheating and burning the cutting edge of the tool. Dry grinding is recommended for sharpening high speed steel because coolant removes heat from the cutting edge too quickly causing cracking.

Direction of Wheel Rotation

It is generally safer to have the wheel rotate off and away from the tool cutting edge. This allows the tooth rest to position the tooth and prevent the cutter from turning. This method, however, has some drawbacks, in that the heat from grinding is directed toward the tool cutting edge and leaves a burr which must be removed with an oilstone.

TOOL SHARPENING

The efficiency of a cutter is determined by the sharpness of its cutting edge. Therefore, it is important to sharpen a cutter at the first sign of dullness. A dull cutter not only produces a poorly finished surface, but if used continuously, the cutter will need excessive sharpening to restore it to its original efficiency.

Grinding Cutters Cylindrically

Certain types of cutting tools, such as reamers and plain milling cutters, are ground cylindrically to remove warpage from heat treating, to remove nicks, to obtain a specific diameter, or to produce a cutting edge with a slight clearance. When grinding tools or cutters, the work rotates in the opposite direction from that used in conventional grinding. This allows movement in the same direction at the point of contact. Mount the cutter so that the heel of the tooth makes contact with the grinding wheel first, allowing the heel of the tooth to be ground slightly lower than the cutting edge. This clearance will vary slightly depending on the rigidity of the tool being ground and the job setup. The tool to be ground can be held in one of three ways: between centers, on a mandrel, or on a short arbor mounted in the lathe headstock spindle. There are actually two methods of sharpening the cutting edges of individual teeth or flutes found on cutters.

Down Method

In this method, the rotation of the wheel is from the body of the tooth off and away from the cutting edge. The direction of wheel rotation holds the cutter on the tooth but will raise a burr on the cutting edge, which must be removed by stoning. This method has a tendency to draw temper from the metal. See Figure 9-43.

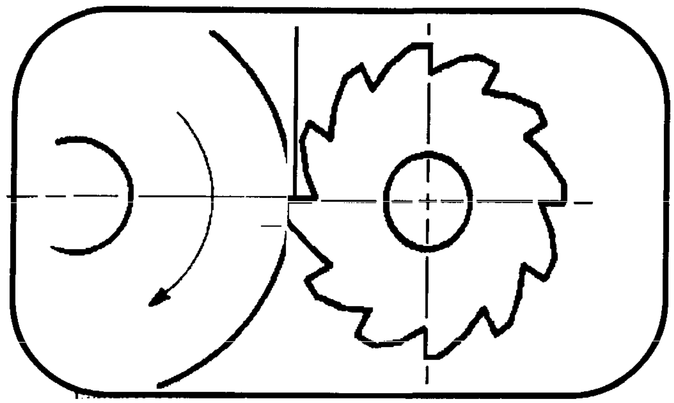


Figure 9-43. Down method.

Up Method

In this method, the wheel rotation is from the cutting edge towards the body of the tooth. With this method, there ‘is less danger of burning the tooth. However, the operator must ensure that the cutter is held firmly against the tool rest. If the cutter turns during grinding, the cutter will be ruined.

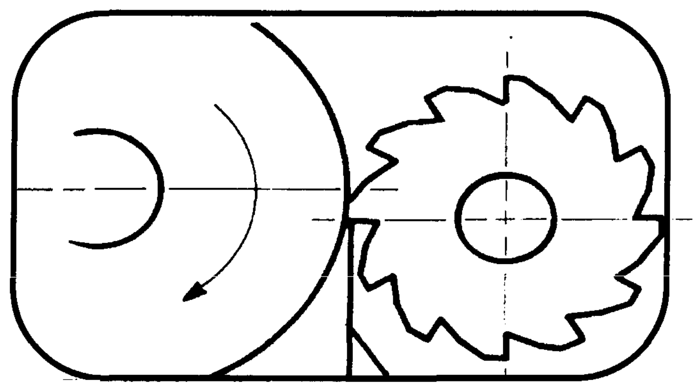


Figure 9-44. Up method.

Cutting Tool Clearance

Correct clearance on the cutting edge of any tool is essential for heat distribution, resistance to wear, and cutting efficiency. Not enough clearance will cause the teeth on the cutter to drag, producing heat caused by friction, and slow cutting. Too much clearance produces chatter and dulls the teeth rapidly. Primary clearance angles are determined by the type of material the cutter wil1 be used on. Secondary clearance angles are usually 3° to 5° more than primary clearance angles. This produces a strong tooth that provides easy control over the width of the cutting land. The width of the land depends on the diameter of the cutter and varies between 1/64 inch to 1/16 inch. When the width of the land becomes too wide after several sharpening, the secondary clearance angle must be ground to restore the land to its original width.

Clearance angles are produced by positioning the wheel, cutter, and tooth rest in different locations. When using the Versa-Mil, it is easier to reposition the wheel by raising or lowering the basic unit. To determine the distance in thousands of an inch, multiply the desired clearance angle by the diameter of the cutter times the constant 0.0088. The constant 0.0088 is the decimal equivalent of the distance moved 10 on the circumference of a l-inch-diameter circle.

EXAMPLE: Using the following formula clearance angle x cutter diameter x 0.0088, a clearance angle of 7° on a 1 1/2-inch-diameter cutter would be 7 x 1.5 x 0.0088, or a movement of 0.0924 of an inch.

Grinding Form Cutters

Formed or eccentricity relieved cutters (such as gear cutters) and concave and convex cutters cannot be sharpened in the same manner as profile cutters. Form cutters have a definite shape that must be retained, even after several sharpening. To retain this shape, only the face of the cutter is ground. Increasing or decreasing the rake on these cutters alters the final shape of the cutter, so care must be taken to ensure that the rake remains at the original angle. The indexing head may be used to assure even spacing of the teeth faces.

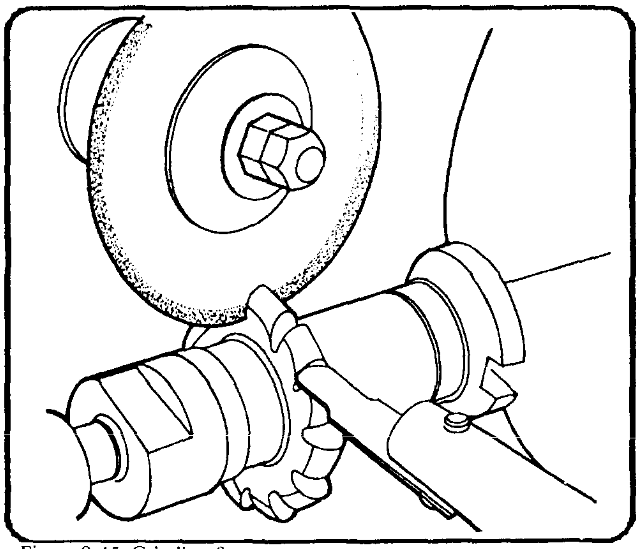


Figure 9-45. Grinding form cutters.

THREAD MILLING

The Versa-Mil with the universal head will enable a lathe to mill threads to full depth and complete profile in a single pass (Figure 9-46). Milling threads saves time and reduces the chance for error over single pointing. USS threads may be cut with standard 60° included angle cutters.

Acme and special form threads are cut with cutters designed for the pitch diameter required. The Versa-Mil will cut internal, external, right-handed, or left-handed threads. Square threads can be cut with an end mill mounted in either the basic or the milling and drilling head.

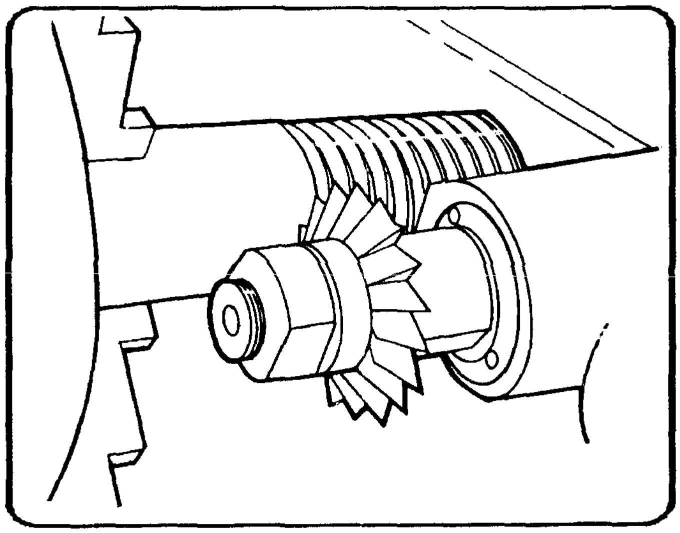


Figure 9-46. Thread milling.

Lathe Preparation

Thread milling speeds and feeds are approximately the same as those used for keyway millillg and slotting. The lathe spindle speeds needed for thread milling are generally lower than those available on standard lathes. It is usually necessary to use a reduction unit mounted to the lathe to obtain the required lathe spindle speed. Large diameter workplaces may require speeds as low as 1/2 or 1/3 RPM. Other than lathe spindle reduction, no other modification of the lathe is needed for thread milling. The quick change gearbox and lead screw are set the same as for single point threading. The indexing head may be mounted to the lathe and used to rotate the lathe spindle when a reduction unit is not available.

Mounting the Versa-Mil

Even though the cutter is at or below the centerline of the work when the basic unit is mounted on the compound rest, it is advisable to mount the unit directly to the cross slide for rigidity.

Supporting the Work

Work of sufficient diameter and rigidity may be supported easily between centers. For long or small diameter work, a steady rest or follower should be used to prevent the work from bending away from the cutter thereby reducing the depth of cut.

Depth of Cut

For external threads, the cutter is fed into the work with the cross feed lead screw. For internal threads, the cutter is fed into the work with the basic unit lead screw. Because thread milling with the Versa-Mil is a one-pass operation, total depth of cut is calculated and set before cutting the thread.

Cutter Rotation

Consideration should be taken when mounting the cutter and selecting the spindle rotation. Conventional milling should be used to put pressure downward onto the carriage. A key may have to be inserted in the arbor to prevent the cutter from loosening the spindle nut.

Accessibility to Work

Because the universal head spindle may be operated in either direction and mounted on either side of the basic unit, threads may be milled at either end of the work and very close to shoulders and flanges.

Helix Angles

The graduations on the basic unit faceplate and the mounting plate of the universal head are used to set the approximate helix angle. Refer to the Versa-Mil operator's manual for helix angles of different threads.

Thread Milling Cutters

Cutters as small as 2 3/4-inches in diameter may be used with the universal head for external thread milling. The cutter diameter for internal threads is governed by the internal diameter of the work. Standard 60° included angle cutters may be modified for use for American Standard Threads by grinding a flat on the point. The width of the flat equals 1/8 the thread pitch and must have relief clearance the same as other cutting tools.

GLOSSARY

ACRONYMS AND ABBREVIATIONS

TC - Training Circular

TM - Technical Manual

AR - Army Regulation

DA - Department of the Army

RPM - revolutions per minute

SAE - Society of Automotive Engineers

SFPM - surface feet per minute

tpf -taper per foot

tpi taper per inch

UNC - Unified National Coarse

UNF - Unified National Fine

SF -standard form

Med - medical

WRPM - revolutions per minute of workpiece

FF - fraction of finish

WW - width of wheel

TT - table travel in feet per minute

sd - small diameter

Id - large diameter

ID - inside diameter

TOS- Intentional Organization for Standardization

LH - left hand

NC - National Coarse

NF - National Fine

OD - outside diameter

RH - right hand

CS - cutting speed

AA - aluminum alloys

IPM - feed rate in inches per minute

FPM - feet per minute of workpiece

pd - pitch diameter

tan L - tangent angle formula

It - length of taper

DEFINITIONS

abrasive - natural - (sandstone, emery, corundum. diamonds) or artificial (silicon carbide, aluminum oxide) material used for making grinding wheels, sandpaper, abrasive cloth, and lapping compounds.

abrasive wheels - Wheels of a hard abrasive, such as Carborundum used for grinding.

accurate - Conforms to a standard or tolerance.

Acme thread - A screw thread having a 29 degree included angle. Used largely for feed and adjusting screws on machine tools.

acute angle - An angle that is less than 90 degrees.

adapter - A tool holding device for fitting together various types or sizes of cutting tools to make them interchangeable on different machines.

addendum - That portion of a gear tooth that extends from the pitch circle to the outside diameter.

align - To adjust or set to a line or center.

allowance - The prescribed difference in dimensions of mating parts to provide a certain class of fit.

alloy - A metal formed by a mixture of two or more different metals.

angle iron - An iron or steel structural member that has been cast, rolled, or bent (folded) so that its cross section is L-shaped.

angle plate - A precision holding fixture made of cast iron. steel, or granite. The two principal faces are at right angles and may be slotted for holding the work **or** clamping to a table.

annealing - The controlled heating and cooling of a metal to remove stresses and to make it softer and easier to work with.

anvil - A heavy iron or steel block upon which metal is forged or hammered-. also the fixed jaw on a micrometer against which parts are measured.

apron - That portion of a lathe carriage that contains the clutches, gears, and levers for moving the carriage. It also protects the mechanism.

arbor - A shaft or spindle for holding cutting tools; most usually on a milling machine.

arbor press - A hand-operated machine tool designed for applying high pressure for the purpose of pressing together or removing parts.

assembly - A unit of fitted parts that make up a mechanism or machine, such as the headstock assemble of a lathe.

automatic stop - A device which may be attached to any of several parts of a machine tool to stop the operation of the machine at any predetermined point.

axis - The line, real or imaginary, passing through the center of an object about which it could rotate; a point of reference.

babbitt - An antifriction metal alloy used for bearing inserts; made of tin, antimony, lead, and copper.

back gears - Gears fitted to a machine to increase the number of spindle speeds obtainable with a cone or step pulley belt drive.

back rake - The angular surface ground back from the cutting edge of cutting tools. On lathe cutting tools. the rake is positive if the face slopes down from the cutting edge toward the shank, and negative if the face slopes upward toward the shank.

backlash - The lost motion or looseness (play) between the faces of meshing gears or threads.

bandsaw - A power saw. the blade of which. is a continuous, narrow, steel band having teeth on one edge and passing over two large pulley wheels.

bar stock - Metal bars of various lengths, made in flat, hexagon, octagon, round, and square shapes from which parts are machined.

bastard - Threads, parts, tools, and sizes that are not standard, such as ‘bastard nuts,’ “bastard plus,” “bastard fittings,’ and so forth. The term also refers to a standard coarse cut file.

bearing - Rollers, and balls placed between moving parts to reduce friction and wear.

bed - One of the principal parts of a machine tool, having accurately machined ways or bearing surfaces for supporting and aligning other parts of the machine.

bell mouth - The flaring or tapering of a machined hole, usually made at the entrance end because of misalignment or spring of the cutting tool.

bench grinder - A small grinding machine for shaping and sharpening the cutting edges of tools.

bench lathe - A small lathe mounted on a bench or table.

bench work - Work done primarily at a bench with hand tools. occasionally supplemented by small power-driven tools.

bevel - Any surface that is not at right angles to another surface. Also, the name given a tool used for measuring, laying out, or checking the accuracy of work machined at an angle or bevel.

bit, tool (cutter) - A hardened steel bar or plate that is shaped according to the operation to be performed and the material to be machined.

blind bole - A hole made in a workpiece that does not pass through it.

block, Jo - Shop name for a Johannson gage block, a very accurate measuring device.

blowhole - A defect in a casting caused by trapped steam or gas.

blueprint - A pen or ink line drawing reproduced (printed) on sensitized paper by direct exposure.

blue vitriol copper sulfate - A layout solution which turns a copper color when applied to a clean. polished metal surface.

bond - The material that holds the abrasive grains together to form a grinding wheel.

bore - To enlarge and finish the surface of a cylindrical hole by the action of a rotating boring bar (cutting tool) or by the action of a stationary tool pressed (fed) against the surface as the part is rotated.

boring bar (cuffer bar) - A combination tool holder and shank.

boring tool - A cutting tool in which the tool bit'. the boring bar and, in some cases, the tool holder are incorporated in one solid piece.

boss - A projection or an enlarged section of a casting through which a hole may be machined.

brass - A nonferrous alloy consisting essentially of copper and zinc.

brazing - Joining metals by the fusion of nonferrous alloys having a melting temperature above 800 degrees F, but below that of the metals being joined.

brine - A saltwater solution for quenching or cooling when heat treating steel.

Brinell hardness - A method of testing the hardness of a metal by controlled pressure of a hardened steel ball of a given size.

broach - A long, tapered cutting tool with serration's which, when forced through a hole or across a surface, cuts a desired shape or size.

bronze - A nonferrous alloy consisting essentially of copper and tin.

buff - To polish to a smooth finish of high luster with a cloth or fabric wheel to which a compound has been added.

bull gear - The large crank gear of a shaper.

burnishing - The process of finishing a metal surface by contact with another harder metal to improve it. To make smooth or glossy by or as if by rubbing; polish.

burr - The sharp edge left on metal after cutting or punching-, also, a rotary cutting tool designed to be attached to a drill.

bushing - A sleeve or a lining for a bearing or a drill jig to guard against wear.

caliper - A device used to measure inside or outside dimensions.

caliper, gear tooth - A special caliper used to measure both the "choral thickness" and the depth of a gear tooth.

cam - A device for converting regular rotary motion to irregular rotary or reciprocating motion. Sometimes the effect of off-center lathe operations.

carbide tool bits - Lathe cutting tools to which carbide tip inserts have been brazed, to provide cutting action on harder materials than the high speed cutters are capable of.

carbon steel - A broad term applied to tool steel other than high-speed or alloy steel.

Carborundum - A trade name for an abrasive compounded of silicon and carbon (silicon carbide).

carbonizing - The process of adding carbon to the outer surface of steel to improve its quality by heat treating it in contact with a carbonaceous material.

carriage - A principal part of a lathe that carries the cutting tool and consists of the saddle, compound rest. and apron.

case hardening - A heat treating process, basically carbonizing. that makes the surface layer or case of steel substantially harder than the interior or core.

castigated nut (castle nut) - A nut with grooves cut entirely across the top face.

casting - A part made by pouring molten metal into a mold.

cathead - A collar or sleeve which fits loosely over a shaft to which it is clamped by setscrews.

center - A point or axis around which anything revolves or rotates. 1 n the lathe. one of the parts upon which the work to be turned is placed. The center in the headstock is referred to as the "live' center and the one mounted in the tailstock as the 'dead' center.

center, dead - A center that does not rotate; commonly found on the tailstock of a lathe. A1so, an expression for the exact center of an object.

center drill - A combined countersink and drill used to prepare work for mounting centers.

center gage - A small ', flat gage having 60 degree angles that is used for grinding and setting the thread cutting tools in a lathe. It may also be used to check the pitch of threads and the points of center.

center, half male - A dead center that has a portion of the 60 degree cone cut away.

center head - A part of a combination square set that is used to find the center of or to bisect a round or square workpiece.

center, live - A center that revolves with the work. Generally. this is the headstock center; however, the ball bearing type tailstock center is also called a live center.

center punch - A pointed hand tool made of hardened steel and shaped somewhat like a pencil.

ceramic - A new type of cutting tool material made of aluminum oxide . or silicon carbide that is finding increased use where high speed and resistance to high temperatures and wear are factors.

chain gearing (chain drive) - Power transmission by means of an endless chain running around chain wheels (chain pulley) and/or sprocket wheels.

chamfer - The bevel or angular surface cut on the edge or a corner of a machined part.

chasing threads - Cutting threads in a lathe or screw machine.

chatter - The vibrations caused between the work and the cutting tool which leave distinctive tool marks on the finished surface that are objectionable.

chip breaker - A small groove ground back of the cutting edge on the top of a cutting tool to keep the chips short.

chipping - The process of cutting metal with a cold chisel and hammer.

chisel - Any one of a variety of small hand cutting tools, generally wedge-shaped.

chuck - A device on a machine tool to hold the workpiece or a cutting tool.

chuck, independent jaw - A chuck, each of whose jaws (usually four) is adjusted with a screw action independently of the other jaws.

chuck, universal (self-centering chuck, concentric chuck) - A chuck whose jaws are so arranged that they are all moved together at the same rate by a special wrench.

circular pitch - The distance measured on the pitch circle from a point on a gear tooth to the same point on the next gear tooth.

clearance - The distance or angle by which one objector surface clears another.

clearance angle - The angle between the rear surface of a cutting tool and the surface of the work at the point of contact.

climb milling - A method of milling in which the work table moves in the same direction as the direction of rotation of the milling center. Sometimes called down cutting or down milling.

clutch, friction (friction coupling) - A shaft coupling used where it is necessary to provide a connection that can be readily engaged or disengaged while one of the shafts is in motion.

cog - A tooth in the rim of a wheel - a gear tooth in a gear wheel.

cold-rolled steel - Steel that has been rolled to accurate size and smooth finish when made. In contrast, hot-rolled steel may have a rough, pitted surface and slag inclusion.

collet - A precision work holding chuck which centers finished round stock automatically when tightened. Specialized collets are also a-, available in shapes for other than round stock.

color method - A technique of heat treating metal by observing the color changes that occur to determine the proper operation to perform to achieve the desired results.

combination square - A drafting and layout tool combining a square, a level. A protractor, and a center head.

compound (rest) - The part of a lathe set on the carriage that carries the tool post and holder. It is designed to swing in any direction and to provide feed for turning short angles or tapers.

concave - A curved depression in the surface of an object.

concentric - Accurately centered or having a common center.

cone pulley - A one-piece stepped pulley having two or more diameters.

contour - The outline of an object.

convex - The curved surface of a cylinder, as a sphere when viewed from without.

coolant - A common term given to the numerous cutting fluids or compounds used with cutting tools to increase the tool life and to improve surface finish on the material.

corrosion - Oxidation (rusting) or similar chemical change in metals.

counterbore - To enlarge the top part of a hole to a specific size, as for the head of a socket-head or cap screw. Also, the tool that is used.

countersink - To enlarge the top part of a hole at an angle for a flat-head screw. Also, the tool that is used.

cross feed - The feed that operates across the axis of the workpiece or at right angles to the main or principal feed on a machine.

cross section - A view showing an internal structure as it would be revealed by cutting through the piece in any plane.

crucible steel - A high-grade tool steel made by melting selected materials in a crucible.

cutting fluid - A liquid used to cool and lubricate the cutting to improve the work surface finish.

cutting speed - The surface speed of the workpiece in a lathe or a rotating cutter, commonly expressed in feet per minute (FPM) and converted to revolutions per minute (RPM) for proper setting on the machine.

cutting tool - A hardened piece of metal (tool steel) that is machined and ground so that it has the shape and cutting edges appropriate for the operation for which it is to be used.

cyaniding - A process of case hardening steel by heating in molten cyanide.

dead center - See center, dead.

dead smooth - The term applied to the finest cut of a file.

deburr - To remove sharp edges.

decalescence - A decrease in temperature that occurs while heating metal through a range in which change in structure occurs.

dedendum - The depth, or that portion of a gear tooth from the pitch circle to root circle of gear.

diametral pitch - Ratio of the number of teeth on a gear to the number of inches of pitch diameter or the number of teeth to each inch of pitch diameter.

die - A tool used to form or stamp out metal parts', also, a tool used to cut external threads.

die stock - The frame and two handles (bars) which hold the dies (chasers) used for cutting (chasing) external screw threads.

dividers, spring - Dividers whose legs are held together at the hinged end by the pressure of a C-shaped spring.

dividing head (index bead) - A machine tool holding fixture which positions the ",-work for accurately spacing holes. slots. flutes. and gear teeth and for making geometric shapes. When geared to the table lead screw, it can be used for helical milling operations.

Do-Al l saw - A trade name given to a type of band saw used for sawing metal.

dog - A clamping device (lathe dog) used to drive work being machined between centers. Also, a part projecting on the side of a machine worktable to trip the automatic feed mechanism off or to reverse the travel.

dovetail - A two-part slide bearing assembly used in machine tool construction for the precise alignment and smooth operation of the movable components of the machine.

dowel - A pin fitted or keyed in two adjacent parts to accurately align the parts when assembling them.

down feed (climb cutting, climb milling)- A seldom used method of feeding work into milling cutters. The work is fed in the same direction as the portion of the cutter which comes in contact with it.

draw - See tempering.

dressing - The act of removing the glaze and dulled abrasives from the face of a grinding wheel to make it clean and sharp. See truing.

drift - A tapered. flat steel used to remove drills and other tapered shank tools from spindles, sockets, or sleeves. Also a round, tapered punch used to align or enlarge holes.

drill - A pointed tool that is rotated to cut holes in material.

drill bushing - A hardened steel guide inserted in jigs, fixtures. or templates for the purpose of providing a guide for the drill in drilling holes in their proper or exact location.

drill, center - A combination drill and countersink-

drill chuck - A device used to grip drills and attach them to a rotating spindle.

drill, twist - A commonly used metal-cutting drill, usually made with two flutes running around the body.

drill jig - A jig which holds parts or units of a structure and. by means of bushings, guides the drill so that the holes are properly located.

drill press - An upright power-driven machine for drilling holes in metal, wood, or other material.

drill press, radial (radial drill) - A machine tool for drilling holes. The drill head is so supported that it may be moved over a large area to drill holes in objects of large size or to drill several holes in an object without shifting the object.

drill press - A drilling machine with a counterbalanced spindle which makes it possible for the operator to control accurately the rate at which the drill is fed into the work. The sensitive drill press usually contains drills that are less than 1/2 inch in diameter and which rotate at high speeds.

drill rod - A high-carbon steel rod accurately ground to size with a smooth finish. It is available in many sizes and is used extensively in tool making.

drill sleeve - An adapter with an internal and external taper which fits tapered shank tools such as drills or reamers to adapt them to a larger size machine spindle.

drill socket - An adapter similar to a sleeve except that it is made to adapt a larger tapered-shank tool to a smaller size spindle.

drill, twist - A commonly used metal-cutting drill, usually made with two flutes running around the body.

drive fit - One of several classes of fits in which parts are assembled by pressing or forcing one part into another.

ductility - The property of a metal that permits it to be drawn. rolled, or hammered without fracturing or breaking.

eccentric - A circle not having a geometric center. Also, a device such as a crankshaft or a cam for converting rotary motion to reciprocating motion.

element - Matter which cannot be broken up into simpler substances by chemical action, that is, whose molecules are all composed of only one kind of atom.

elongation - Lengthening or stretching out.

emery - A natural abrasive used for grinding or polishing. It is being largely replaced by artificial abrasives.

emulsion - A coolant formed by mixing soluble oils or compounds with water.

extruded - Metal which had been shaped by forcing through a die.

extrusion - A shaped part resulting from forcing a plastic material such as lead, tin. aluminum. zinc., copper, rubber, and so forth. through a die opening

EZY OUT (trademark) - A tool for removing broken bolts or studs from a hole.

face - To machine a flat surface, as in the end of a shaft in the lathe. The operation is known as facing.

face milling - Milling a large flat surface with a milling cutter that operates in a plane that is at right angles to its axis.

faceplate - A large circular plate with slots and holes for mounting the workpiece to be machined. It is attached to the headstock of a lathe.

facing - The process of making a flat or smooth surface (usually the end) on a piece of stock or material.

fatigue - The effect on certain materials, especially metals, undergoing repeated stresses.

feed - The rate of travel of a cutting tool across or into the work-, expressed in inches per minute or in inches per revolution.

feed mechanism - The mechanism, often automatic. which controls the advancing movement (feed) of the cutting tools used in machines.

female part - A concave piece of equipment which receives a mating male (convex) part.

ferrous - A metal alloy in which iron is the major ingredient.

rile test - A test for hardness in which a corner of a file is run across the piece of metal being tested. The hardness is shown by the dent the file makes.

fillet - A curved surface connecting two surfaces that form an angle.

fishtail - A common name for the center gage. It is used to set thread cutting tools and has scales on it for determining the number of threads per inch.

fit - The relation between mating or matching parts, that is, the amount of, or lack of. play between them.

fitting - Any small part used in aircraft construction.

fixture - A production work-holding device used for machining duplicate workplaces. Although the term is used interchangeably with a jig, a fixture is not designed to guide the cutting tools as the jig does.

flange - A relatively thin rim around a part.

flash - A thin edge of metal formed at the parting line of a casting or forging where it is forced out between the edges of the form or die.

flute - The groove in a cutting tool which provides a cutting edge and a space for the chips to escape and permits the cutting fluids to reach the cutting edges.

fly cutter - A single-point cutter mounted on a bar in a fly cutter holder or a fly cutter arbor- used for special applications for which a milling cutter is not available.

follower rest - A support for long, slender work turned in the lathe. It is mounted on the carriage, travels close to and with the cutting tool, and keeps the work from springing away.

footstock - Part of an indexing, attachment which has a center and serves the same purpose as the tail stock of a lathe.

force fit - A fitting which one part is forced of pressed into another to form a single unit. There are different classes of force fits depending on standard limits between mating parts.

forge - To form or shape heated metal by hammering. Also. the name of the unit used for heating metal, as the blacksmith’s forge.

formed cutters - Milling cutters which will produce shaped surfaces with a single cut', and so designed that they may be sharpened without. changing their outline or shape.

forming tool - Tool ground to a desired shape to reproduce this shape on the workpiece.

free cut - An additional cut with no advancement of depth.

free cutting steel - Bar stock containing a high percentage of sulfur. making it very easy to machine. Also known as Bessemer screw stock.

free fit - A class of fit intended for use where accuracy is not essential. or where large temperature variations are likely to be encountered, or both conditions.

fulcrum - The point or support on which a lever turns.

gage - Any one of a large variety of devices for measuring or checking the dimensions of objects.

gage blocks - Steel blocks machined to extremely accurate dimensions.

gage, center - See center gage.

gage, depth - A tool used in measuring the depth of holes or recesses.

gage, drill - A flat steel plate drilled with holes of various sizes, each marked with the correct size or number. into which small twist drills may be fitted to determine the size of their diameters.

gage, drill point - A gage use to check the 59" angle on drills.

gage, feeler (thickness gage) - A gage consisting of a group of very thin blades, each of which is accurately ground to a specific thickness.

gage, indicating (dial indicator) - A gage consisting of a dial, commonly graduated (marked) in thousandths of an inch, to which is fastened an adjustable arm.

gage, radius (fillet gage) - Any one of a number of small. flat, standard-shaped metal leafs or blades used for checking the accuracy of regular concave and convex surfaces.

gage, screw pitch - A gage consisting of a group of thin blades. used for checking the number of screw threads per unit of distance, usually per inch, on a screw, bolt, nut, pipe, or fitting.

gage, surface (scribing block) - A gage used to check the accuracy, of plane surfaces, to scribe lines at desired distances from a given surface and to check the height of a point or points on a piece of work from a given surface.

gage, telescoping - A T-shaped gage used to measure the diameter **or** width of holes.

gang milling - A milling setup where a number of cutters are arranged on an arbor so that several surfaces can be machined at one time. It is commonly used for production purposes.

gear blank - A stamping, casting. or any, piece of material from which a gear is to be machined. It is usually a disk.

gib - A tapered strip of metal placed between the bearing surface of two machine parts to ensure a precision fit and provide an adjustment for wear.

hacksaw - A metal blade of hardened steel having small, close teeth on one edge. It is held under tension in a U-shaped frame.

half nut - A lever-operated mechanism that resembles a split nut that can be closed on the lead screw of a lathe when threads are being cut.

handwheel - Any adjusting or feeding mechanism shaped like a wheel and operated by hand.

hardening - A heat-treating process for steel which increases its hardness and tensile strength and reduces its ductility.

hardness tests - Tests to measure the hardness of metals.

headstock - The fixed or stationary end of a lathe or' similar machine tool.

heat treatment - The process of heating and cooling a solid metal or alloy to obtain certain desired properties or characteristics.

helical gear - A gear with teeth cut at some angle other than at a right angle across the face of the gear, thus permitting more than one tooth to be engaged at all times and providing a smoother and quieter operation than the spur gear.

helix - A path formed as a point advances uniformly around a cylinder, as the thread on a screw or the flutes on a drill.

helix angle - The angle between the direction of the threads around a screw and a line running at a right angle to the shank.

hex - A term used for anything shaped like a hexagon.

high-speed steel - An alloy steel commonly used for cutting tools because of its ability to remove metal at a much faster rate than carbon steel tools.

hob - A cylindrical cutting tool shaped like a worm thread and used in industry to cut gears.

hobbing - The operation of cutting gears with a bob.

hog - To remove in excess of what is considered normal, sometimes causing accidents or tool breakage; also, to rough out haphazardly.

hole saw - A cutting tool used to cut a circular groove into solid material.

honing - The process of finishing ground surfaces to a high degree of accuracy and smoothness with abrasive blocks applied to the surface under a light controlled pressure. and with a combination of rotary and reciprocating motions.

hot-rolled steel - Steel which is rolled to finished size. while hot. Identified by a dark oxide scale left on the surface.

idler - A gear or gears placed between two other gears to transfer motion from one gear to the other gear without changing their speed or ratio.

independent chuck - A chuck in which each jaw may be moved independently of the others.

indexing - The process of positioning a workpiece for machining it into equal spaces. dimensions. or angles using an. index or dividing head.

indexing fixture - A complete indexing unit composed of a dividing head and rootstock. (See dividing head.)

index plate - A metal disk or plate punched with many holes arranged in a series of rings. one outside the other each ring containing a different number of holes.

indicator - A precision instrument which shows variations of thousandths of an inch or less when testing the trueness or alignment of a workpiece, fixture, or machine.

inserted-tooth cutter - A milling cutter designed with replaceable cutting tooth inserts to save the expense of a new cutter whenever the teeth become damaged or worn. Generally, they are made 6 inches or more in diameter.

intermediate gear - See idler.

jack, leveling - Small jacks (usually screw jacks) for leveling and holding work on planner beds and similar places.

Jacobs chuck - Common term for the drill chuck used in either the headstock spindle or in the tailstock for holding straight-shank drills, taps, reamers, or small diameter workplaces.

Jarno - A standard taper having 0.600-inch taper per foot used on some machine tools.

jig - A production work holding device that locates the workpiece and guides the cutting tool (see fixture).

Johannson blocks (Jo blocks) - Common term for the precision gage blocks used and accepted as dimensional standards by machinists, toolmakers. and inspectors.

kerf - The width of cut made by a Saw.

key - One of the several types of small metal objects designed to fit mating slots in a shaft and the hub of a gear or pulley to provide a positive drive between them: also. the name of the T-handle wrench used on chucks.

key seat - A recessed groove (slot) machined into a shaft or a part going on the shaft (usually a wheel or gear).

knee - That part of a column of a knee-type milling machine which carries the saddle and the table and provides the machine with vertical feed adjustments. Also, the name of a precision angle plate called a "toolmaker's knee".

knurl - A decorative gripping surface of straight-line or diagonal design made by uniformly serrated rolls called knurls.

knurling - The process of finishing a part by scoring (pressing) patterns on the surface of the work.

land - That surface on the periphery of a rotary cutting tool, such as a milling cutter. drill tap, or reamer, which joins the face of the flute or tooth to make up the basic cutting edge.

lap - A tool made of soft metal and charged With fine abrasives for precision finishing of metal surfaces. Also, to perform the operation using a lap-

lard oil - A cutting oil made from animal fats usually mixed with mineral oils to reduce its cost and improve its qualities.

layout - To locate and scribe on blank stock the shape and size dimensions required to machine or form the part.

lead - The distance a thread will advance along its axis in one complete revolution. Also, a heavy, soft', malleable metal having a low melting point. It has a bright. silvery color when freshly cut or poured and turns to a dull gray with aging.

lead hole - See pilot hole.

lead screw - The long, precision screw located in front of the lathe bed geared to the spindle, and used for cutting threads. Also, the table screw on the universal milling machine when geared to the indexing head for helical milling.

limits - The smallest and largest dimension which are tolerable (allowed).

lip of a drill - The sharp cutting edge on the end of a twist drill.

live center - See center, live.

loading - A condition caused by grinding the wrong material with a grinding wheel or using too heavy a grinding action.

machinability - The degree of difficulty with which a metal may be machined; may be found in appropriate handbooks.

machine tool - A power-driven machine designed to bore, cut, drill, or grind metal or other materials.

machining, Finish - Machining a surface to give it the desired finish.

machinist - A person who is skilled in the operation of machine tools. He must be able to plan his own procedures and have a knowledge of heat-treating principles.

machining, rough (rough finishing) - Removing excess stock (material) with a machine tool thus shaping it in preparation for finish machining.

magnesium - A lightweight, ductile metal similar to but 1ighter than aluminum.

magnetic chuck - A flat’. smooth-surfaced work holding device which operates by magnetism to hold ferrous metal workplaces for grinding.

malleable - Capable of being extended or shaped by hammering or rolling.

mandrel - A precision-made tapered shaft to support work for machining between centers.

mesh - To engage. as the teeth between two gears.

mic; mike - A term used for micrometer, or to measure with a micrometer.

micrometer, depth - A micrometer in which the spindle projects through a flat, accurately machined bar.. used to measure the depth of holes or recesses.

micrometer, thread - A micrometer in which the spindle is ground to a point having a conical angle of 60 degrees. The anvil, instead of being flat. has a 60 degree V-Shaped groove which fits the thread.

mild steel - A term used for low-carbon machine steel.

mill - A milling machine; also, the act of performing an operation on the milling machine.

milling, climb - See climb milling. milling, face-See face milling.

milling cuffer - A cutting tool, generally cylindrica] in shape. used on a milling machine and operated essentially like a circular saw.

minor diameter - The smallest diameter of a screw thread. Also known as the “root diameter."

Morse taper - A self-holding standard taper largely used on small cutting tools such as drills, end mills, and reamers, and, on some machines, spindles in which these tools are used.

multiple-thread screw - A screw made of two or more threads to provide an increased lead with a specified pitch.

music wire - A high-quality steel wire used for making springs. Also called piano wire.

necking - Machining a groove or undercut in a shaft to permit mating parts to be screwed tightly against a shoulder or to provide clearance for the edge of a grinding wheel.

nickel - An alloying element which increases the strength, toughness, and wear and corrosion resistance of steels.

nitrating - A case hardening process in which ammonia or some other form of nitrogen is introduced to the surface of certain alloys.

nonferrous - Metal containing no iron, such as brass and aluminum.

normalizing - Process of heating a ferrous metal or alloy to above its critical temperature and cooling in still air to **room** temperature to relieve Internal stresses.

off center - Not centered; offset, eccentric, or inaccurate.

oil hardening - The process of quenching in oil when heat treating alloy steel to bring out certain qualities.

oilstones - Molded abrasives in various shapes used to hand-sharpen cutting tools.

overarm - The support for the end of a milling cutter which is on the opposite side of the cutter from the spindle and column.

pack hardening - A heat-treating process in which the workpiece is packed into a metal box together with charcoal, charred leather. or other carbonaceous material to case-harden the part.

parallels - Hardened steel bars accurately ground to size and ordinarily made in pairs in many different sizes to support work in precision setups.

parting - The operation of cutting off a piece from a part held in the chuck of a lathe.

pawl - A pivoted lever or sliding bolt that secures as an automatic directional table control on a grinder.

peen - To draw, bend. or flatten, also, the formed side of a hammer opposite the face.

pilot - A guide at the end of a counterbore which keeps it aligned with the hole,

pilot hole - A starting hole for large drills to serve as a guide, reduce the resistance, and aid in maintaining the accuracy of the larger hole. Also called a lead hole.

pinning - A term used to describe the condition of a file clogged with metal filings causing it to scratch the work.

pitch - The distance from any point on a thread to the corresponding point on the adjacent thread. measured parallel to the axis. Also applied to spur gears-. see diametral pitch.

pitch circle - The line (circle) of contact between two meshing gears.

pitch diameter - The diameter of a thread at an imaginary point where the width of the groove and the with of the thread are equal.

pitch line - An imaginary line which passes through threads at such points that the length of the part of the line between adjacent threads is equal to the length of the line within a thread.

plain cutter - A milling cutter with cutting teeth on the periphery (circumference) only.

play - The looseness of fit (slack) between two pieces press tit-See force fit.

punch, prick - A solid punch with a sharp point, used to mark centers or other locations on metal.

pyrometer - A device for measuring the high temperatures in a heat-treating furnace.

quench - To rapidly cool heated metal in water, oil brine, or air in the process of heat treating.

quick return - A mechanism on some machine tools that provides rapid movement of the ram or table on the return or anointing stroke of the machine.

rack - An array of gears spaced on a straight bar.

radial - In a direction directly outward from the center of a circle or sphere or from the axis of a cylinder. The spokes of a wheel, for example, are radial.

radius - The distance from the center of a circle to its circumference (outside).

rake - That surface of a cutting tool against which the chips bear while being severed. If this surface is less than 90" from the surface being cut, the rake is positive-, if more, the rake is negative.

ram - That part of a shaper which moves back and forth and carries the tool head assembly.

rapid traverse - A lever-controlled, power-operated feature of some machines that permits the rapid movement of the worktable from one position to another.

reaming, line - The process of reaming two or more holes to bring them into very accurate alignment.

recalescence - An increase of temperature that occurs while cooling metal through a range of temperatures in which changes in metal occur.

recess - An internal groove. See undercut.

relief - A term for clearance or clearance angle.

root diameter - See minor diameter.

roughing - The fast removal of stock to reduce a workpiece to approximate dimensions'. leaving only enough material to finish the part to specifications.

rule, hook - A rule with a hook on the end for measuring through pulley holes and in similar places.

running fit - A class of fit intended for use on machinery with moderate speeds, where accurate location and minimum play are desired.

SAE steel - Steel manufactured under the specifications by the Society of Automotive Engineers.

sandblasting - A process of blowing sand by compressed air with considerable force through a hose against an object.

scale - The rough surface on hot. finished steel and castings. Also, a shop term for steel rules.

scraper - A hardened steel hand tool used to scrape surfaces very smooth by removing minute amounts of metal.

scribe (scribe; scratch awl) - A steel rod 8 to 12 inches long and about 3/16 inches in diameter. It has a long, slender, hardened steel point on one or both ends.

sector - A device that has two radial, beveled arms which can be set to include any number of holes on the indexing plate of a dividing head to eliminate recounting the holes for each setting.

set - The bend or offset of a saw tooth to provide a clearance for the blade while cutting. Also, the permanent change in the form of metal as the result of repeated or excessive strain.

set screw - A plain screw used principally for locking adjustable parts in position.

setup - The preparation of a machine tool to complete a specific operation. It includes mounting the workpiece and necessary tools and fixtures, and selecting the proper speeds. feeds, depth of cut and coolants.

shank - That part of a tool or similar object which connects the principal operating part to the handle, socket', or chuck by which it is held or moved.

shims - Very thin sheets of metal made in precise thickness and used between parts to obtain desired fits. Sometimes they are laminated, to be pulled off to the desired depth.

shoulder - A term for the step made between two machined surfaces.

shrink fit - A class of fit made when the outer member is expanded by heating to fit over a shaft, and then contracts or shrinks tightly to the shaft when cooled.

side cutter - A milling cutter that has cutting teeth on the side as well as on the periphery or circumference.

side rake - That surface which slopes to the side of the cutting edge. It may be positive or negative and is combined with the back rake. See rake.

sine bar - A precision instrument for laying out, setting, testing, and otherwise dealing with angular work.

slabbing cutter - A wide, plain milling cutter having helical teeth. Used for producing large, flat surfaces.

sleeve - See drill sleeve.

slitting saw - A narrow milling cutter designed for cutoff operations or for cutting narrow slots.

slotter - An attachment which operates with a reciprocating motion. Used for machining internal slots and surfaces.

soft hammer - A hammer made of brass, copper, lead'. or plastic to a, non-marring finished surfaces on machines or workplaces.

spherodizing - A process of heat treating steel to produce a grain structure that is relatively soft and machinable.

spindle - A rotating device widely used in machine tools. such as lathes., milling machines, drill presses, and so forth, to hold the cutting tools or the work, and to give them their rotation.

spindle speed - The RPM at which a machine is set. See cutting speed.

spot facing - Finishing a bearing surface around the top of a hole.

spring collet - See collet.

spur gear - A gear having teeth parallel to the axis of the shaft on which it is mounted.

square, solid (toolmaker's tri square) - A very accurate try square in which a . steel blade is set firmly into a solid, rectangular-shaped handle so that each edge of the blade makes an angle of exactly 90" with the inner face (side) of the handle.

square surface - A surface at a right angle with another surface.

square threads - A thread hating a depth. width. and space between threads that are equal. It is used on heavy jack screws. vise screws. and other similar items.

steady rest - A support that is clamped to the bed of a lathe used when machining a long workpiece. Sometimes called a center rest.

stellite - A cast al alloy of chromium, cobalt. and sometimes tungsten, used to make lathe cutter bits that will stand exceptionally fast speeds and heavy cuts.

step block - A fixture designed like a series step to provide support at various heights required for setups.

stock - A term for the materials used to make parts in a machine tool. Also, the die stock used for threading dies.

stop - A device attached to a machine tool to limit the travel of the worktable and sometimes the work head.

straddle milling - A milling setup where two side milling cutters are spaced on an arbor to machine two parallel surfaces with a single cut.

stress - The internal force or resistance developed in steel which was hardened, extensively machined. or cold worked.

surface grinding - The process of grinding flat surfaces on a surface grinding machine. With special setups, angular and form surfaces may also be ground.

surface plate - An accurately machined and scraped flat metal piece (usually of cast iron) used to check the flatness of surfaces.

swing - The dimension of a lathe determined by the maximum diameter of the work that can be rotated over the ways of the bed.

tailstock - That part of a machine tool. such as a lathe or cylindrical grinder which supports the end of a workpiece with a center. It may be positioned at any point along the way of the bed, and may be offset from center to machine tapers.

tang - The flat on the shank of a cutting tool, such as a drill, reamer or end mill, that fits a slot in -the spindle of a machine to keep the tool from slipping. Also, the part of a file that fits into a handle.

tap - A tool used to cut threads on the inside of a round hole.

taper - A uniform increase or decrease in the size or diameter of a workpiece.

tapping - The process of cutting screw threads in a round hole with a tap (an internal thread cutting tool).

T-bolt - Term for the bolts inserted in the T-slots of a worktable to fasten the workpiece or work-holding device to the table.

tempering - A heat-treating process to relieve the stresses produced when hardening and to impart certain qualities‘, such as toughness-, sometimes called “drawing.”

template - A pattern or a guide for laying out or machining to a specific shape or form.

tensile strength - The property of a metal which resists force applied to pull it apart.

thread - A helical projection of uniform section on the internal or external surface of cylinder or cone. Also, the operation of cutting a screw thread.

thread angle - The angle formed by the two sides of the thread (or their projections) with each other.

thread axis - A line running lengthwise through the center of the screw.

thread crest - The top surface joining the two sides of a thread.

thread depth - The distance between the crest and the root of a thread.

thread pitch - The distance from a point on one screw thread to a corresponding point on the next thread.

thread pitch diameter - The diameter of a screw thread measured from the thread pitch line on one side to the thread pitch line on the opposite side.

thread root - The bottom surface joining the sides of two adjacent threads.

throw - The crankpin on a crankshaft. Also, the length of the radius of a crank, an eccentric, or a cam.

tolerance - The allowable deviation from a standard size.

tool steel - A general classification for high-carbon steel that can be heat treated to a hardness required for metal cutting tools such as punches, dies. drills. taps’, reamers, and so forth.

traverse - One movement across the surface of the work being machined.

truing - The act of centering or aligning a workpiece or cutting tool so that an operation may be performed accurately. Also, correcting the eccentricity or out of-round condition when dressing a grinding wheel.

T-slot - The slots made in the tables of machine tools for the square-head bolts used to clamp the workpiece,, attachments, or work-holding fixtures in position for performing the machining operations.

tumbler gears - A pair of small lever-mounted gears on a lathe used to engage or to change the direction of the lead screw.

two-lip end mill - An end milling cutter designed with teeth that cut to the center so that it may be used to feed into the work like a drill.

universal grinder - A versatile grinding machine designed to perform both internal and external grinding operations. including straight and tapered surfaces on tools and cutters.

universal milling machine - A milling machine with a worktable that can be swiveled for milling helical work. It is always supplied with attachments, including an indexing fixture.

universal vise - A vise designed for holding work at a double or compound angle. Also, a toolmaker’s vise.

Ways - The flat or V-shaped bearing surfaces on a machining tool that guide and align the parts which they support.

wheel dresser - A tool or device for dressing or truing a grinding wheel.

work - A common term for a workpiece or part being machined.

working drawing - A drawing. blueprint, or sketch of a part, structure, or machine.

worm - The threaded cylinder or shaft designed to mesh with a worm gear.

worm gear - A gear with helical teeth made to conform with the thread of the mating worm.

wrought iron - A commercially pure form of iron with minute slag inclusions which make it soft, tough, and malleable.

REFERENCES

ARMY REGULATIONS (AR)

34-4	Army Standardization Policy
310-50	Catalog of Abbreviations and Brevity Codes 340-21 + The Army Privacy Program
385-10	The Army Safety Program
385-40	Accident Reporting and Records

DA PAMPHLETS (DA Pam)

25-30	Consolidated Index of Army Publications and Blank Forms
738-750	The Army Maintenance Management System

COMPONENT LIST

3405/70-IL	Technical Bulletins (TB)
MED 501	Occupational and Environmental Health: Hearing Conservation MED
MED-502	Occupational and Environmental Health Respiratory Protection program
MED 5060	Occupational and Environmental Health Occupational Vision

TECHNICAL MANUALS (TM)

9-3415-226-10 Operator's Manual, Grinding Machine, Drill

9-3415-230-10 Operator's Manual, Grinding Machine, Tool and Cutter, Universal Type, Bench Mounting

9-3416-221-10 Operator's Manual for Lathe, Engine Floor Mtd, Sliding Bed Gap

9-3417-209-10 Operator's Manual, Milling Machine, Horizontal, Universal, Floor Mounting

9-3417-210-10 Operator's Manual, Milling Machine, Ram Type, Swivel Head Plain Table, Power Feed

REFERENCES (cont.)

TECHNICAL MANUALS (TM) (cont.)

9-3419-224-10 Operator's Manual, Saw, Band, Metal Cutting

9-243 Use and Care of Hand Tools and Measuring Tools

9-3460-207-12 Organizational Maintenance Manual Milling, Grinding & Drilling, and Slotting Attachment

9-3465-200-10 Operator's Manual, Fixture, Lathe, Metal Cutting for 2-Inch Diameter Round or 2-.Inch Square Stock

9-3419-228-10 Operator's Manual, Saw, Band, Metal Cutting, Floor Mounting

9-3444-203-10 Operator's Manual for Press, Arbor, Hand Operated

FORMS

SF 368 Product Quality Deficiency Report *(Category II)*

OTHER

Machinery Handbook, 21st ed, Industrial Press Inc, New York, 1981

TRADOC Pamphlets (TRADOC Pam)

310-6 Armywide Doctrinal and Training Literature (ADTL), Development and Preparation

APPENDIX A

TABLES

TABLE 3-1. Proper wire gages for extension cords.

AMPERAGE RATING ON NAMEPLATE	3.5-5.0	5.1-7.0	7.1-12.0	12.1-16.0	16.1-20.0
EXTENSION CORD LENGTH		WIRE GAGES			
25 ft.	18	18	16	14	12
50 ft.	18	16	14	12	10
75 ft.	16	14	12	10	
100 ft.	14	12	10		
150 ft.	12	12			
200 ft.	12	10			
300 ft.	10				

Sizes smaller than wire gage 10 are not normally available as flexiable extension cord.

TABLE 3-2. *Selection of abrasive disks.*

OPERATION	ABRASIVE DISK	
	GRAIN NO.	TYPE OF COAT
REMOVING RUST	16 to 30	OPEN
REMOVING PAINT	16 to 36	OPEN
SANDING METAL (ROUGH CUTS)	24 to 36	CLOSED
SANDING METAL (MEDIUM CUTS)	36 to 60	CLOSED
SANDING METAL (FINISHING CUTS)	36 to 80	CLOSED
SANDING WOOD (ROUGH CUTS)	16 to 24	OPEN
SANDING WOOD (MEDIUM CUTS)	24 to 50	OPEN
SANDING WOOD (FINISHING CUTS)	60 to 120	OPEN

TABLE 3-3. Recommended use of bandsaw blades.

RECOMENDED USE	THICKNESS OF MATERIAL TO BE CUT	TEETH PER INCH	RECOMENDED USE	THICKNESS OF MATERIAL TO BE CUT	TEETH PER INCH
CARBON STEEL			ALLOY STEEL		
High speed cutting of aluminum, brass, copper, soft bronze, magnesium, wood, mild steel and tougher steels at slow speed.	1/2"-3 3/8"	6	High speed cutting of aluminum, brass iron, cast iron, bronze, brass, copper, galvanized pipe, mild steel and tougher steel including chrome and tungsten steels at slower speed.	3/16"-1"	10
High speed cutting of aluminum, brass copper, magnesium, mild steels and tougher steels at slow speed.	3/8"-1"	8	High speed cutting of aluminum, angle iron, cast iron, bronze, brass, copper galvanized pipe, mild steel and tougher steels including chrome, tungsten steels and electric cable at slow speed.	5/32"-3/4"	14
High speed cutting of aluminum, angle iron, cast iron, bronze, brass, copper galvanized pipe, mild steel and tougher steels at slow speed.	3/16"-3/4"	10	High speed cutting of angle iron, bronze brass, copper, galvanized pipe, thin wall tubing, mild steel and tougher steels including chrome and tungsten steels at slow speed.	1/8"-1/2"	18
High speed cutting of aluminum, angle iron, cast iron, bronze, brass, copper, galvanized pipe, mild steel and tougher steels and electric cable at slow speed.	5/32"-1/2"	14	High speed cutting of angle iron, bronze brass, copper, galvanized pipe, thin wall tubing, mild steel and tougher steels at slow speed.	3/32"-1/8"	24
High speed cuting of angle iron, bronze brass, copper, galvanized pipe, thin wall tubing, mild steel and tougher steels at slow speed.	1/8"-1/4" 3/32"-1/8"	18 24	HIGH SPEED STEEL		
			High speed cutting of aluminum, angle iron, bronze, brass, copper, galvanized pipe, mild steel and tougher steels including stainless, chrome, tungsten steels plus other problem material at slow speed.	7/32"-7/8" 3/16"-5/8" 5/32"-3/8"	10 14 18

TABLE 4-1 Common Twist Drill Sizes

Drill Designation	Decimal Equivalent	Drill Designation	Decimal Equivalents	Drill Designation	Decimal Equivalents	Drill Designation	Decimal Equivalents	Drill Designation	Decimal Equivalents
No. 80	.0135	No. 49	.0730	4.10mm	.1614	6.80.mm	.2877	27/64	.4219
35mm	.0138	1.90mm	.0748	4.20mm	.1654	6.90mm	.2717	11.00mm	.4331
No. 79	.0145	No. 48	.0760	No. 19	.1660		.2720	7/16	.4375
1/64	.0156	1.95mm	.0768	4.25mm	.1673	7.00mm	.2756	11.50mm	.4528
.40mm	.0158	5/64	.0781	4.30mm	.1693	I	.2770	29/64	.4531
								11.80mm	.4646
No. 78	.0160	No. 47	.0785	No. 18	.1695	7.10mm	.2795	15/32	.4888
.45mm	.0177	2.00mm	.0787	11/64	.1719	K	.2810	12.00mm	.4724
No. 77	.0180	2.05mm	.0807	No. 17	.1730	9/32	.2812	31/64	.4844
.50mm	.0197	No. 46	.0810	4.40mm	.1732	7.20rom	.2835	12.50mm	.4921
No. 76	.0200	No. 45	.0820	No. 16	.1770	7.25nlm	.2854	1/2	.5000
								12.80mm	.5039
No. 75	.0210	2.10mm	.0827	4.50mm	.1772	7.30fhm	.2874	13.0mm	.5118
.55mm	.0217	2.15mm	.0846	No. 15	.1800	L	.2900	33/64	.5156
No. 74	.0225	No. 44	.0860	4.60mm	.1811	7.40mm	.2913	17/32	.5312
.60mm	.0236	2.20mm	.0866	No. 14	.1820	m	.2950	13.5mm	.5315
.61 mm	.0240	2.25mm	.0886	No. 13	.1850	7.50mm	.2953	35/64	.5469
No. 73	.0240	No. 43	.0890	4.70mm	.1850	19/64	.2969	14.0mm	.5512
No. 72	.0250	2.30mm	.0906	4.75mm	.1870	7.60mm	.2992	9/16	.5625
.65mm	.0256	2.35mm	.0925	3/16	.1875	N	.3020	14.5mm	.5709
No. 71	.0260	No. 42	.0935	4.80mm	.1890	7.70mm	.3031	37/64	.5781
.70mm	.0276	3/32	.0938	No. 12	.1890	7.75mm	.3051	15.0mm	5906
No. 70	.0280	2.40mm	.0945	No. 1 1	.1910	7.80mm	.3071	19/32	.5938
No.69	.0292	No. 41	.0960	4.90mm	.1929	7.90mm	.3110	39/64	.6094
.75mm	.0295	2.45mm	.0965	No. 10	.1935	5/18	.3125	15.5mm	.6102
No. 68	.0310	No. 40	.0980	No. 9	.1960	8.00mm	.3150	5/8	.6250
1/32	.0312	2.50mm	.0984	5.00mm	.1968	0	.3160	16.0mm	.6299
								1625mm	.6388
.80mm	.0315	No. 39	.0995	No. 8	.1990	8.10mm	.3189	41/64	.6406
No. 67	.0320	No. 38	.1015	5.10mm	.2008	8.20mm	.3228	16.5mm	.6496
No. 68	.0330	2.60mm	.1024	No. 7	.2010	p	.3230	21/32	.6562
.85mm	.0335	No. 37	.1040	13/64	.2031	8.25mm	.3248	17.0mm	.6693
No. 65	.0350	2.70mm	.1063	No. 6	.2040	8.30mm	.3268	43/64	.6719
								17.25mm	.6791
.90mm	.0354	No. 36	.1065	5.20mm	.2047	21/64	.3281	11/16	.6875
No. 64	.0360	2.75mm	.1083	No. 5	.2055	8.40mm	.3307	17.5mm	.6890
No. 63	.0370	7/64	.1094	5.25mm	.2067	n	.3320	45/64	.7031
.95mm	.0374	No. 35	.1100	5.30mm	.2087	8.50mm	.3346	Is.Omm	.7087
No. 62	.0380	2.80mm	.1102	No. 4	.2090	8.60mm	.3386	23/32	.7188
No.6t	.0390	No. 34	.1110	5.40mm	2126	R	.3390	18.5mm	.7283
1.00mm	.0394	No. 33	.1130	No. 3	.2130	8.70mm	.3425	47/64	.7344
No. W	.0400	2.90mm	.1142	5.50mm	.2165	11/32	.3437	19.0mm	.7480
No. 59	.0410	No. 32	.1160	7/32	.2187	8.75mm	.3445	3/4	.7500
1.05mm	.0413	3.00mm	.1181	5.60mm	.2205	S.Somm	.3465	49/64	.7656
No. 58	0420	No: 31	1200	No. 2	.2210	S	.3480	19.5mm	.7677
No. 57	.0430	3.10mm	.1220	5.70mm	.2244	8.90mm	.3504	25/32	.7812
1.10mm	.0433	1/8	.1250	5.75mm	.2264	9.00mm	.3543	20.0mm	.7874
1.15mm	.0453	3.20mm	.1260	No. 1	.2280	T	.3580	51/64	.7969
No. 56	.0465	3.25mm	.1280	5.80mm	2283	9.10mm	.3583	20.5mm	.8071
3/64	.0469	No. 30	.1285	5.90mm	.2323	23/64	.3594	13/16	.8125
1.20mm	.0472	3.30mm	.1299	A	.2340	9.20mm	.3622	21.0mm	.8268
1.25mm	.0492	3.40mm	.1339	15/64	.2344	9.25mm	.3642	53/64	.8281
1.30mm	.0512	No. 29	.1360	6.00mm	.2362	9.30mm	.3661	27/32	.8438
No. 55	.0520	3.50mm	.1378	8	.2380	u	.3680	21.5mm	.8465
1.35mm	0531	No.28	.1405	6.10mm	.2402	9.40mm	.3701	55/64	.8594
No. 54	.0550	9/54	.1406	c	.2420	9.50mm	.3740	22.0mm	.8661
1.40mm	.0551	3.Wmm	.1417	6.20mm	.2"1	3/8	.3750	7/8	.8750
1.45mm	.0571	No. 27	.1440	D	.2460	v	.3770	22.5mm	.8858
1.50mm	0591	3.70mm	1457	6.25mm	.2461	9.60mm	.3780	57/64	.8906
No. 53	.0595	No. 26	.1470	6.30mm	.2480	9.70mm	.3819	23.0mm	.9055
1.55mm	.0610	3.75mm	.1476	1/4	.2500	9.75mm	.3839	29/32	.9063
1/16	.0625	No. 25	1495	E	.2500	9.Wmm	.3858	59/64	.9219
1.60mm	.0630	3.80mm	1496	6.40mm	.2520	w	.3860	23.5mm	.9252
No. 52	.0635	No. 24	.1520	6.50mm	.2559	9.90mm	.3898	15/16	.9375
1.65mm	.0650	3.90mm	.1535	F	.2570	25/64	.3906	24.0mm	.9"9
1.70mm	.0669	No. 23	.1540	6.60mm	.2598	10.0mm	.3937	61/64	.9531
No. 51	.0670	5/32	.1562	G	.2610	x	.3970	24.5mm	.9646
1.75mm	.0689	No. 22	.1570	6.70mm	.2638	y	.4040	31/32	.9688
No. 50	.070	4.00mm	.1575	17/64	2686	13/32	.4062	25.0mm	.9843
1.5omm	.0709	No. 21	.1590	6.75mm	.2657	z	.4130	63/64	.9844
1.85mm	.0728	No. 20	.1610	H	.2860	10.5mm	.4134	1	1.0000

TABLE 4-2 Drill Information

Drill information for different materials (High Speed Drills)

MATERIAL	CUTTING SPEEDS 1. (METERS/MINUTE) (FEET/MINUTE)		POINT ANGLE	LIP CLEARANCE	COOLANTS
	MPM	FPM			
Aluminum And Alloys	61.00 - 91.50	200 - 300	90 - 130 deg	12 - 15 deg	Kerosene/Kerosene & Lard Oil/ Soluble Oil
Armor Plate	12.20 - 18.25	40 - 50	135 - 140 deg	6 - 9 deg	Light Machine Oil
Brass	61.00 - 91.50	200 - 300	118 - 118 deg	12 - 15 deg	Dry/ Soluble Oil/Kerosene/Lard Oil
Bronze	61.00 - 91.50	200 - 300	110 - 118 deg	12 - 15 deg	Dry/ Soluble Oil/Mineral Oil/Lard Oil
Bronze, High Tensile	21.35 - 45.75	70 - 150	100 - 110 deg	12 - 15 deg	Dry/ Soluble Oil/Mineral Oil/Lard Oil
Cast Iron, Soft	30.50 - 45.75	100 - 150	90 - 100 deg	12 - 15 deg	Air Jet Dry/ Soluble Oil
Cast Iron, Medium	21.35 - 30.50	70 - 100	100 - 110 deg	12 - 15 deg	Air Jet Dry/ Soluble Oil
Cast Iron, Hard	21.35 - 3O.50	70 - 100	100 - 118 deg	8 - 12 deg	Air Jet Dry/ Soluble Oil
Cast Iron, Chilled	9.15 - 12.20	30 - 40	118 - 135 deg	5 - 9 deg	Air Jet Dry/ Soluble Oil
Copper	61.00 - 91.50	200 - 300	100 - 118 deg	12 - 15 deg	Air Jet Dry/ Soluble Oil
Copper Graphite Alloy (Carbon Drills)	18.30 - 21.35	60 - 70	**--**	**----**	Soluble Oil/Dry/Mineral Oil/Kerosene
Glass (Carbon Drills)	6.10 - 9.15	20 - 30	**--**	**--**	Soluble Oil/Dry/Mineral Oil/Kerosene
Iron, Malleable	15.25 - 27.45	50 - 90	90 - 100 deg	12 - 15 deg	Light Machine Oil
Magnesium And Alloys	76.25 - 122.0	250 - 400	70 - 118 deg	12 - 15 deg	Soluble Oil
Monel Nickel	4.15 - 15.28	30 - 50	118 - 125 deg	10 - 12 deg	Compressed Air/Mineral Oil
Nickel Alloys	12.20 - 18.30	40 - 60	135 - 140 deg	5 - 7 deg	Lard Oil/Soluble Oil
Plastic, Hot Set	30.50 - 91.50	100 - 300	60 - 90 deg	10 - 12 deg	Lard Oil/Soluble Oil
Plastic, Cold Set	30.50 - 91.50	100 - 300	118 - 135 deg	12 - 20 deg	Soap Solution
Steel, Low Carbon, 0.2-0.3ct	24.40 - 33.55	80 - 110	110 - 118 deg	7 - 9 deg	Soap Solution
Steel, Medium Carbon 0.4-0.5c	21.35 - 24.40	70 - 80	118 - 125 deg	7 - 9 deg	Soluble Oil/Mineral Oil/Sulfur Oil/Lard Oil
Steel (High Carbon 1.2c)	15.25 - 18.30	50 - 60	118 - 145 deg	7 - 9 deg	Soluble Oil/Mineral Oil/Sulfur Oil/Lard Oil
Steel, Forged	15.25 - 18.30	50 - 60	118 - 145 deg	7 - 12 deg	Soluble Oil/Mineral Oil/Sulfur Oil/Lard Oil
Steel, Alloy	15.25 - 21.35	50 - 70	118 - 125 deg	10 - 12 deg	Mineral Lard Oil
Steel, Alloy 300 To 400 Brinnel	6.10 - 9.15	20 - 30	130 - 140 deg	7 - 10 deg	Soluble Oil
Steel, Stainless, Free Machining	9.15 - 24.40	30 - 80	110 - 118 deg	8 - 12 deg	Soluble Oil
Steel, Stainless, Hard	4.57 - 15.25	15 - 50	118 - 135 deg	6 - 8 deg	Soluble Oil
Steel, Manganese	3.66 - 4.57	12 - 15	140 - 150 deg	7 - 10 deg	Soluble Oil
Stone (Carbide Drills)	7.63 - 9.15	25 - 30	**--**	**--**	Water Solution
Wood	91.50 - 122.2	300 - 400	60 - 70 deg	10 - 15 deg	Dry

1. Cutting speeds are for high speed steel drills except as indicated. Carbon drills are approximately 200 to 300% than high speed steel drills.

** Cabide drill point angles and lip clearance angles very with different manufacturers. Consult the manufacturers data on the type of material being drilled for correct point and clearance angles

Table 4-3 Recommended Cutting Fluids For Various Materials

MATERIAL	DRILLING	REAMING	TAPPING	TURNING	THREADING	MILLING
Aluminum	Soluble Oil Kerosene Kerosene & Lard Oil	Soluble Oil Kerosene Mineral Oil	Soluble Oil Mineral Oil	Soluble Oil	Soluble Oil Kerosene & Lard Oil	Soluble Oil Lard Oil Lard Or Mineral Oil
Brass	Dry Soluble Oil Kerosene & Lard Oil	Soluble Oil Dry	Soluble Oil Lard Oil Dry	Soluble Oil	Soluble Oil Lard Oil	Soluble Oil Dry
Bronze	Dry Soluble Oil Lard Oil Mineral Oil	Soluble Oil Lard Oil Dry	Soluble Oil Lard Oil Dry	Soluble Oil	Soluble Oil Lard Oil	Soluble Oil Lard Oil Dry
Cast Iron	Dry Soluble Oil Air Jet	Soluble Oil Mineral Lard Oil	Mineral Lard Oil	Soluble Oil Mineral Lard Oil Dry	Dry Sulfurized Oil	Dry Soluble Oil
Copper	Dry Soluble Or Lard Oil Kerosene Mineral Lard Oil	Soluble Oil Lard Oil Dry	Soluble Oil Mineral Lard Oil	Soluble Oil	Soluble Oil Lard Oil	Soluble Oil Dry
Malleable Iron	Dry Soda Water	Dry Soda Water	Soluble Oil	Soluble Oil	Lard Oil Soda Water	Dry Soda Water
Monel Metal	Soluble Oil Lard Oil	Soluble Oil Lard Oil	Mineral Lard Oil Sulfurized Oil	Soluble Oil	Lard Oil	Soluble Oil
Steel Alloys	Soluble Oil Sulfurized Oil Mineral Lard Oil	Soluble Oil Mineral Lard Oil	Sulfurized Oil Mineral Oil	Soluble Oil	Lard Oil Sulfurized Oil	Soluble Oil Mineral Lard Oil
Steel Forgings Low Carbon	Soluble Oil Sulfurized Lard Oil Lard Oil Mineral Lard Oil	Soluble Oil Mineral Lard Oil	Soluble Oil Lard Oil	Soluble Oil	Soluble Oil Mineral Lard Oil	Soluble Oil Mineral Lard Oil
Tool Steel	Soluble Oil Sulfurized Oil Mineral Lard Oil	Soluble Oil Sulfurized Oil Lard Oil	Mineral Lard Oil Sulfurized Oil	Soluble Oil	Lard Oil Sulfurized Oil	Soluble Oil Lard Oil

TABLE 4-4. Rotational speeds and feeds for high-speed twist drills

Diameter of drill (in.)	Aluminum	Brass & Bronze	Cast iron	Mild steel 0.2-0.3 carbon (LOW)	Steel 0.4-0.5 carbon (MED)	Tool steel 1.2 carbon and drop forgings	Conn. rod molybdenum steel	3.5 nickel steel	Stainless steel and monel metal	Malleable iron	Feed per revolution (in.)
MATERIAL AND CUTTING SPEED (FT PER MINUTE)	300	200	100	110	80	60	55	60	50	85	
	Revolutions per minute										
1/16	18,336	12,224	6,112	6,724	4,883	3,668	3,404	3,976	3,056	5.192	0.0015
1/8	9,168	6,112	3,056	3,362	2,444	1,834	1,702	1,988	1,528	2,596	0.002-0.003
3/16	6,108	4,072	2,036	2,242	1,630	1,222	1,120	1,324	1,018	1,734	0.004
1/4	4,584	3,056	1,528	1,681	1,222	917	851	994	764	1,298	0.005
5/16	3,666	2,444	1,222	1,344	978	733	672	794	611	1,039	0.005
3/8	3,054	2,036	1,018	1,121	815	611	560	662	509	867	0.006
7/16	2,622	1,748	874	921	699	524	481	568	437	742	0.007
1/2	2,292	1,528	764	840	611	459	420	497	382	649	0.008
9/16	2,037	1,358	679	747	543	407	373	441	340	577	0.008
5/8	1,836	1,224	612	673	489	367	337	398	306	520	0.009
11/16	1,665	1,110	555	611	444	333	300	360	273	472	0.009
3/4	1,524	1,016	508	559	408	306	279	330	254	433	0.010
13/16	1,422	948	474	521	379	285	261	308	237	403	0.010
7/8	1,314	876	438	482	349	262	241	285	219	371	0.011
15/16	1,221	814	407	448	326	244	224	265	204	346	0.012
1	1,146	764	382	420	308	229	210	258	191	325	0.013
1 1/16	1,077	718	359	395	287	215	197	233	180	305	0.013
1 1/8	1,020	680	340	374	272	204	187	221	170	288	0.014
1 3/16	966	644	322	354	258	193	177	209	161	274	0.014
1 1/4	918	612	306	337	245	183	168	199	153	260	0.015
1 5/16	873	582	291	320	233	175	160	189	146	248	0.015
1 3/8	834	556	278	306	222	167	153	180	139	236	0.015
1 7/16	795	530	265	292	212	159	146	172	133	225	0.015
1 1/2	762	508	254	279	204	153	140	165	127	216	0.015
1 9/16	732	488	244	268	195	146	134	159	122	207	0.016
1 5/8	702	468	234	257	188	141	129	152	117	201	0.016
1 11/16	678	452	226	249	181	136	124	147	113	192	0.016
1 3/4	654	436	218	240	175	131	120	142	109	186	0.016
1 13/16	630	420	210	231	168	126	116	137	105	179	0.016
1 7/8	612	408	204	224	163	122	112	133	102	173	0.016
1 15/16	591	394	197	216	158	118	108	128	99	168	0.016
2	573	382	191	210	153	115	105	124	96	162	0.016

1. Rotational speed value for carbide twist drills are 200 to 300 percent higher than H.S.S.

TABLE 4-5. Screw thread pitches and tap drill sizes.

Screw Thread Size and Pitch	Outside Diameter of Screw (in.)	Tap Drill Size	Decimal Equivalent of Drill Size
National Coarse (NC) Series			
No. 1-64	0.073	53	0.0595
No. 2-56	0.086	50	0.0700
No. 3-48	0.099	47	0.0785
No. 4-40	0.112	43	0.0890
No. 5-40	0.125	38	0.1015
No. 6-32	0.138	36	0.1065
No. 8-32	0.164	29	0.1360
No. 10-24	0.190	25	0.1495
No. 12-24	0.216	16	0.1770
No. 1/4-20	0.250	07	0.2010
No. 5/16-18	0.3125	F	0.2570
No. 3/8-16	0.375	5/16	0.3125
No. 7/16-14	0.4375	U	0.3680
No. 1/2-13	0.500	27/64	0.4219
No. 9/16-12	0.5625	31/64	0.4843
No. 5/8-11	0.625	17/32	0.5312
No. 3/4-10	0.750	21/32	0.6562
No. 7/8-9	0.875	49/64	0.7656
No. 1-8	1.000	7/8	0.875
National Finese (NF) Series			
No. 0-80	0.060	3/64	0.0469
No. 1-72	0.073	53	0.0595
No. 2-64	0.086	50	0.0700
No. 3-56	0.099	45	0.0820
No. 4-48	0.112	42	0.0935
No. 5-44	0.125	37	0.1040
No. 6-40	0.138	33	0.1130
No. 8-36	0.164	29	0.1360
No. 10-32	0.190	21	0.1590
No. 12-18	0.216	14	0.1820
No. 1/4-28	0.250	3	0.2130
No. 5/16-24	0.3125	I	0.2720
No. 3/8-24	0.375	Q	0.3320
No. 7/16-20	0.4375	25/64	0.3906
No. 1/2-20	0.500	29/64	0.4531
No. 9/16-18	0.5625	33/64	0.5156
No. 5/8-18	0.625	37/64	0.5781
No. 3/4-16	0.750	11/16	0.6875
No. 7/8-16	0.875	13/16	0.8125
No. 1-14	1.000	15/16	0.9375
METRIC SERIES			
1.6mm x .35	.0630	1.20mm	.0472
2.0mm x .40	.0787	1.60mm	.0630
2.5mm x .45	.0984	2.05mm	.0807
3.0mm x .50	.1181	2.50mm	.0984
3.5mm x .60	.1378	2.90mm	.1142

TABLE 4-5. Screw thread pitches and tap drill sizes (cont.).

METRIC SERIES			
Screw thread size and pitch	Outside diameter of screw (in.)	Tap drill size	Decimal equivalent of drill size
4.0mm x .70	.1575	3.30mm	.1299
5.0mm x .80	.1968	4.20mm	.1654
6.3mm x 1.00	.2480	5.30mm	.2087
8.0mm x 1.25	.3150	6.80mm	.2677
10.0mm x 1.50	.3937	8.50mm	.3346
12.0mm x 1.75	.4724	10.20mm	.4016
14.0mm x 2.00	.5512	12.00mm	.4724
16.0mm x 2.00	.6299	14.00mm	.5512
20.0mm x 2.50	.7874	17.50mm	.6890
24.0mm x 3.00	.9449	21.00mm	.8268
30.0mm x 3.50	1.1811	26.50mm	1.0433
36.0mm x 4.00	1.4173	32.00mm	1.2598
42.0mm x 4.50	1.6535	37.50mm	1.4764
48.0mm x 5.00	1.8898	43.00mm	1.6929
56.0mm x 5.50	2.2047	50.50mm	1.9882
64.0mm x 6.00	2.5197	58.00mm	2.2837
72.0mm x 6.00	2.8346	66.00mm	2.5984
80.0mm x 6.00	3.1456	74.00mm	2.9134
90.0mm x 6.00	3.5433	84.00mm	3.3071
100.0mm x 6.00	3.9370	94.00mm	3.7008

NATIONAL TAPER PIPE THREAD PITCHES AND TAP DRILL SIZES				
Nominal thread size (in.)	Threads per inch	Major pipe diameter (in.)	Tape drill size (in.)	Dercimal equivalent of drill size (in.)
1/8	27	0.405	21/64	0.32813
1/4	18	0.540	29/64	0.45313
3/8	18	0.675	19/32	0.59375
1/2	14	0.840	23/32	0.71875
3/4	14	1.050	15/16	0.9375
1	11 1/2	1.315	1 3/16	1.1875
1 1/4	11 1/2	1.660	1 15/32	1.46875
1 1/2	11 1/2	1.900	1 23/32	1.71875
2	11 1/2	2.375	2 3/16	2.1875
2 1/2	8	2.875	2 11/16	2.6875
3	8	3.500	3 5/16	3.3125
3 1/2	8	4.00	3 13/16	3.8125
4	8	4.500	4 3/16	4.1875

Table____ Formulas for calculating the tap drill size for inch and metric threads.

$$TDS = OD - \frac{1}{N}$$

TDS = Tap Drill size (in Inches)
OD = Outside Diameter
1 = Constant
N = Number of th reads per inch

NOTE: This formula will determine a recommended decimal size, then use the numbered, lettered, or fractional size drill th at is closest to thje computed size.

FOR METRIC SIZES:

The recommended tap drill size is equal to the outside diameter minus the pitch. Metric tap sizes are designated by a capital M, the outside diameter in millimeters, and by the pitch in millimeters; such as M22 x 1.5. To find the recommended tap drill size, subtract 1.5 from 22, to get 20.5, which is the recommended tap drill size. If a metric or inch is not available for the recommended tap drill size, the round up to the nearest available drill.

TABLE 5-1. Grinding wheel selection and application.

GRINDING WHEEL SELECTION AND APPLICATION			
SUITABLE FOR	**WHEEL MATERIAL**	**GRAIN**	**GRADE**
External Cylindrical Grinding			
	Aluminox	2946	L
Good all-around wheels;	Alundum	3836	L
best adapted to soft steel	Aloxite	401	N
Hardened steel	Aluminox or Alundum	46	K
Soft steel of small diam.	Aluminox or Alundum	36	M½
Reamers, drills and general tool work	Aluminox or Alundum	80	K
Hard steel, dry grinding	Aluminox or Alundum	100	I
Cast iron and bronze	Crystolon	45	L
Facing Shoulders			
Ordinary work	Aluminox or Alundum	60	H or I
Fine finish	Aluminox or Alundum	80	I½
Surface Grinding			
	Alundum or Aluminox	46	H
Hardened steel	Alundum or Aluminox	46	G[2]
	Alundum or Aluminox	60	F[2]
Hardened high-speed steel or very thin	Aloxite	367	U
pieces of hardened carbon steel	Alundum or Aluminox	46	G
Cast iron	Carborundum or	36	M
	Crystolon	36	J
Disk Grinding			
Thick pieces, wet grinding	Aluminox or Alundum	30	K
Thin pieces, wet grinding	Aluminox or Alundum	30	J
High-speed steel, dry grinding	Aluminox or Alundum	60 or 80	H or I
Washers and similar pieces	Aluminox or Alundum	60	I
Internal Cylindrical Grinding			
Good all around wheel	Aluminox or Alundum	46	2½ I½
Roughing hardened steel	Aluminox or Alundum	46	J or K
Finishing hardened steel	Aluminox or Alundum	120	J or K
Ordinary finishing without roughing	Aluminox or Alundum	80 & 90	J or K
Roughing brass	Cryatolon	36	H or I
Finishing brass	Crystolon	80	H
Automobile cylinders	Crystolon	46	K
Automobile cylinders	Carborundum	36	M or P
Automobile cylinders, roughing or fair finish	Carbolite	36	H or I
Automobile cylinders, fine finish	Carbolite	60	H
Sharpening Carbon-Steel Cutters, Dry Grinding			
Milling cutters	Aluminox or Alundum	46 or 60	I
Formed and gear cutters	Aluminox or Alundum		

TABLE 7-1. Rake and Relief Angles in Degrees for High-Speed Steel Lathe Tools.

MATERIAL	SIDE RELIEF	FRONT RELIEF	SIDE RAKE	BACK RAKE
ALUMINUM	12	8	15	35
BRASS	10	8	5 to -4	0
BRONZE	10	8	5 to -4	0
CAST IRON	10	8	12	5
COPPER	12	10	20	16
MACHINE STEEL	10 to 12	8	12 to 18	8 to 15
TOOL STEEL	10	8	12	8
STAINLESS STEEL	10	8	15 to 20	8

TABLE 7-2. Cutting speeds for Straight Turning and Threading With HSS Tool Bits.

MATERIAL	STRAIGHT TURNING SPEED		THREADING SPEED	
	FEET PER MINUTE	METERS PER MINUTE	FEET PER MINUTE	METERS PER MINUTE
LOW-CARBON STEEL	80-100	24.4-30.5	35-40	10.7-12.2
MEDIUM-CARBON STEEL	60-80	18.3-24.4	25-30	7.6-9.1
HIGH-CARBON STEEL	35-40	10.7-12.2	15-20	4.6-6.1
STAINLESS STEEL	40-50	12.2-15.2	15-20	4.6-6.1
ALUMINUM AND ITS ALLOYS	200-300	61.0-91.4	50-60	15.2-18.3
ORDINARY BRASS AND BRONZE	100-200	30.5-61.0	40-50	12.2-15.2
HIGH-TENSILE BRONZE	40-60	12.2-18.3	20-25	6.1-7.6
CAST IRON	50-80	15.2-24.4	20-25	6.1-7.6
COPPER	60-80	18.3-24.4	20-25	6.1-7.6

NOTE: Speeds for carbide-tipped bits can be 2 to 3 times the speed recommended for high-speed steel tool bits.

Simple formulas to use for English and Metric calculations:

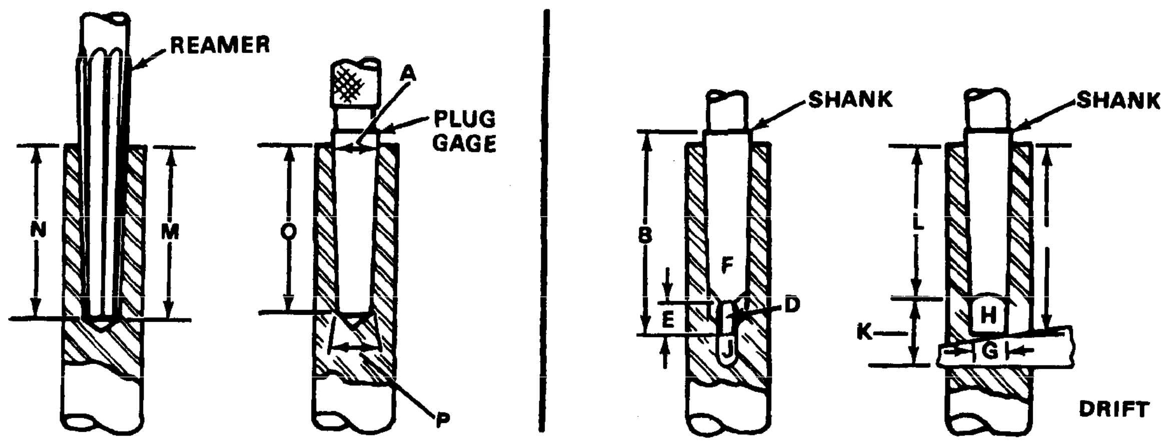
ENGLISH	METRIC
$\frac{\text{cs (in feet)} \times 4}{\text{D (in inches)}}$	$\frac{\text{cs (in meters)} \times 320}{\text{D (in millimeters)}}$
RPM = revolutions per minute CS = recommended cutting speed 4 = a constant for this calculation in feet per minute D = diameter of workpiece in inches	RPM = revolutions per minute CS = recmmended cutting speed 320 = a constant for this calculation in meters per mintue D = diameter of workpiece in milimeters

TABLE 7-3. Feeds for various materials (using high-speed steel or carbide-tipped tool bits).

MATERIAL	FINISHING CUTS		ROUGHING CUTS	
	Inches	Millimeters	Inches	Millimeters
LOW-CARBON STEEL	.012	0.3	.025	0.6
MEDIUM-CARBON STEEL	.012	0.3	.015	0.4
HIGH-CARBON STEEL	.005	0.1	.012	0.3
STAINLESS STEEL	.020	0.5	.010	0.2
ALUMINUM (AND ALLOYS)	.003	0.08	.020	0.5
BRASS AND BRONZE	.003	0.08	.020	0.5
HIGH-TENSILE BRONZE	.003	0.08	.020	0.5
CAST IRON	.003	0.08	.020	0.5
COPPER	.003	0.08	.020	0.5

NOTE: Use less feed on thin, long work to avoid bending the work.

TABLE 7-4. Morse Tapers.



Dia of plug at small end P	Dia of gage line A	Whole lengh B	Depth C	Depth of drilled hole M	Depth of reamed hole N	Stand-ard plug depth O	Thickness D	TANG Length E	Radius F	Diameter G	TAND SLOT Radius H	Width J	Length K	End of socket to tang slot L	Taper per inch	Taper per foot	No of drift	No of taper
0**	0.252	0.356	2-11/32	2-7/32	2-1/16	2-1/32	2	0.156	1/4	5/32	15/64	3/64	0.166	9/16	1-15/16	0.0520	0.6240	0**
1	0.369	0.475	2-9/16	2-7/16	2-3/16	2-5/32	2-1/8	0.203	3/8	3/16	11/32	3/64	0.213	3/4	2-1/16	0.0498	0.5986	1
2	0.572	0.700	3-1/8	2-15/16	2-21/32	2-39/64	2-9/16	0.250	7/16	1/4	17/32	1/16	0.260	7/8	2-1/2	0.0499	0.5994	2
3	0.778	0.938	3-7/8	3-11/16	3-5/16	3-1/4	3-3/16	0.312	9/16	9/32	23/32	5/64	0.322	1-3/16	3-1/16	0.0501	0.6023	3
4	1.020	1.231	4-7/8	4-5/8	4-3/16	4-1/8	4-1/16	0.469	5/8	5/16	31/32	3/32	0.479	1-1/4	3-7/8	0.0519	0.6232	4
5	1.475	1.748	6-1/8	5-7/8	5/5/16	5-1/4	5-3/16	0.625	3/4	3/8	1-13/32	1/8	0.635	1-1/2	4-15/16	0.0526	0.6315	5
6	2.116	2.494	8-9/16	8-1/4	7-13/32	7-21/64	7-1/4	0.750	1-1/8	1/2	2	5/32	0.760	1-3/4	7	0.0521	0.6256	5**
7	2.750	3.270	11-5/8	11-1/4	10-5/32	10-5/64	10	1.125	1-3/8	3/4	2-5/8	3/16	1.135	2-5/8	9-1/2	0.0520	0.6240	

* THE DIMENSIONS AGREE ESSENTIALLY WITH DIMENSIONS OF THE AMERICAN STANDARD ON MACHINE TAPERS.
** THE SIZE 0 TAPER IS NOT LISTED IN THE AMERICAN STANDARD ON MACHINE TAPERS.
*** THE NO 6 DRIFT WILL ALSO EJECT NO 6 TAPER SHANK TOOLS.

TABLE 7-5. Self-holding tapers basic dimensions.

NO. OF TAPER	TAPER PER FT.	DIA. AT GAGE LINE A	ORIGIN OF SERIES
0.239	0.50200	0.23922	Brown & Sharpe taper series
0.299	0.50200	0.29968	
0.375	0.50200	0.37525	
1	0.59858	0.47500	Morse taper series
2	0.59941	0.70000	
3	0.60235	0.93800	
4	0.62326	1.23100	
4-1/2	0.62400	1.50000	
5	0.63151	1.74800	
6*	0.62565	2.49400	
7*	0.62400	3.27000	
200	0.750	2.000	3/4 in. per ft. taper series
250	0.750	2.500	
300	0.750	3.000	
350	0.750	3.500	
400	0.750	4.000	
450	0.750	4.500	
500	0.750	5.000	
600	0.750	6.000	
800	0.750	8.000	
1000	0.750	10.000	
1200	0.750	12.000	

* These sizes are continued in the tang drive series for the present to meet special needs.

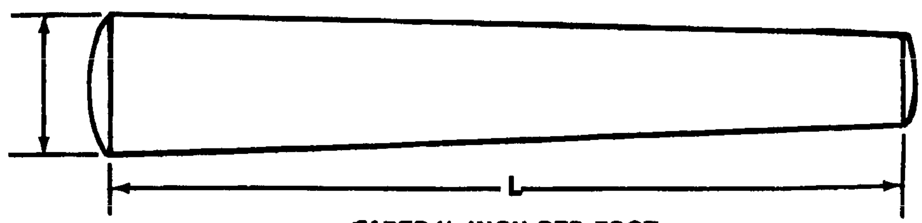
TABLE 7-6. Dimensions for steep machine tapers.

NO. OF TAPER	TAPER PER FT.*	DIA. AT GAGE LINE	LENGTH ALONG AXIS
5	3.500	1/2 0.500	11/16 0.6875
10	3.500	5/8 0.625	7/8 0.8750
15	3.500	3/4 0.750	1- 1/16 1.0625
20	3.500	7/8 0.875	1- 5/16 1.3125
25	3.500	1 1.000	1- 9/16 1.5625
30	3.500	1-1/4 1.250	1- 7/8 1.8750
35	3.500	1-1/2 1.500	2- 1/4 2.2500
40	3.500	1-3/4 1.750	2-11/16 2.6875
45	3.500	2-1/4 2.250	3- 5/16 3.3125
50	3.500	2-3/4 2.750	4 4.0000
55	3.500	3-1/2 3.500	5- 3/16 5.1875
60	3.500	4-1/4 4.250	6- 3/8 6.3750

Note: The tapers numbered 10,20,30,40,50, and 60 are designated as the "Preferred Series."
The tapers numbered 5,15,25,35, and 45 are designated as the "Intermediate Series."

*This taper corresponds to an included angle of 16° 35' 33.4".

TABLE 7-7. American Standard Taper Pins.



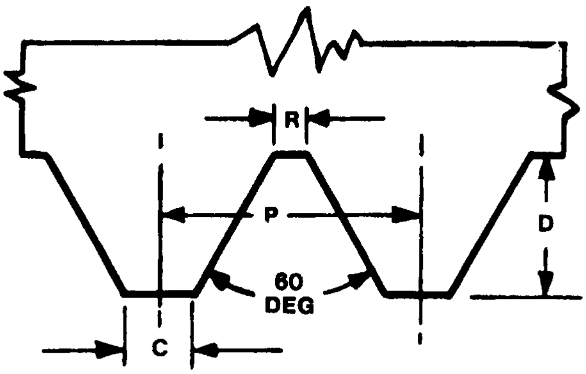
TAPER ¼ INCH PER FOOT.

No. of Taper Pin	Diameter Large End D	Approx. Size D	Range of Lengths L	No. of Taper Pin	Diameter Large End D	Approx. Size D	Range of Lengths L
7/0	0.0625	1/16	3/8 to 5/8	3	0.219	7/32	3/4 to 1-3/4
6/0	0.078	5/64	3/8 to 3/4	4	0.250	1/4	3/4 to 2
5/0	0.094	3/32	1/2 to 1	5	0.289	19/64	1 to 2-1/4
4/0	0.109	7/64	1/2 to 1	6	0.341	11/32	1-1/4 to 3
3/0	0.125	1/8	1/2 to 1	7	0.409	13/32	2 to 3-3/4
2/0	0.141	9/64	1/2 to 1-1/4	8	0.492	1/2	2 to 4-1/2
0	0.156	5/32	1/2 to 1-1/4	9	0.591	19/32	2-3/4 to 5-1/4
1	0.172	11/64	5/8 to 1-1/4	10	0.706	45/64	3-1/2 to 6
2	0.193	3/16	3/4 to 1-1/2				

TABLE 7-8. ISO Metric Pitch & Diameter Combinations.

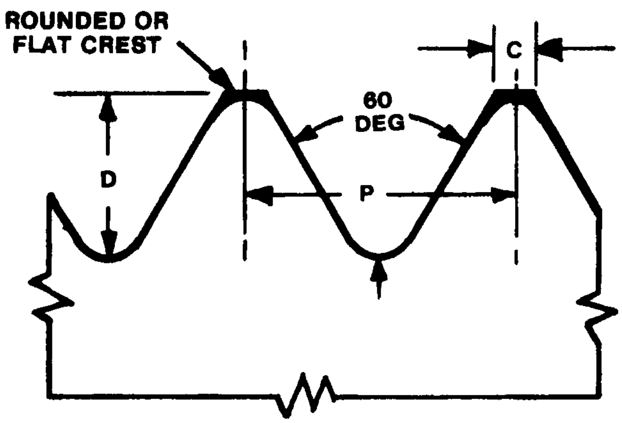
NOMINAL DIA.		THREAD PITCH	NOMINAL DIA.		THREAD PITCH
MM	INCHES	MM	MM	INCHES	MM
1.6	.0630	0.35	20	.7874	2.5
2	.0787	0.40	24	.9449	3.0
2.5	.0984	0.45	30	1.1811	3.5
3	.1181	0.50	36	1.4173	4.0
3.5	.1378	0.60	42	1.6535	4.5
4	.1575	0.70	48	1.8898	5.0
5	.1969	0.80	56	2.2047	5.5
6.3	.2480	1.00	64	2.5197	6.0
8	.3150	1.25	72	2.8346	6.0
10	.3937	1.50	80	3.1496	6.0
12	.4724	1.75	90	3.5433	6.0
14	.5512	2.00	100	3.9370	6.0
16	.6299	2.00			

TABLE 7-9. General Form Dimensions for Standard Screw Threads.



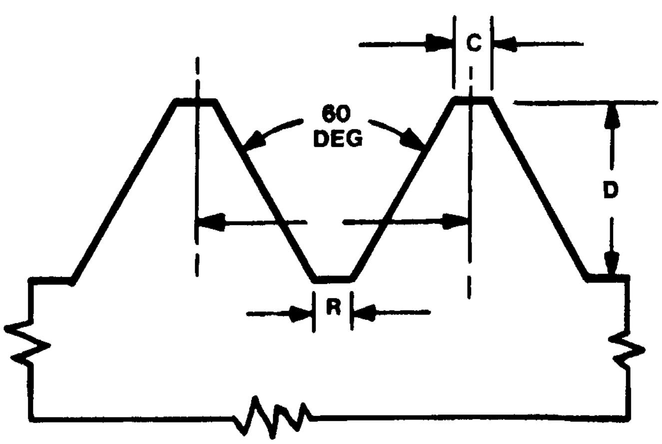
D =DEPTH=0.54127 X PITCH
C =CREST=PITCH÷4

UNIFIED SCREW THREAD
(INTERNAL THREAD)



D =DEPTH=0.61344 X PITCH
C =CREST=PITCH÷8

UNIFIED SCREW THREAD
(EXTERNAL THREAD)



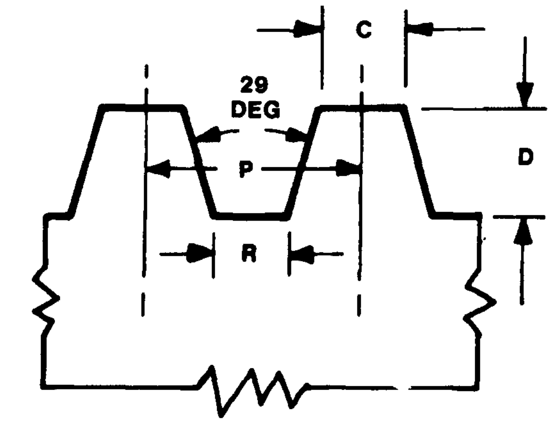
D =DEPTH=0.64952 × PITCH
C =CREST=PITCH ÷8

D = DEPTH = 0.64952 × PITCH
C = CREST = PITCH ÷ 8

AMERICAN NATIONAL STANDARD THREAD

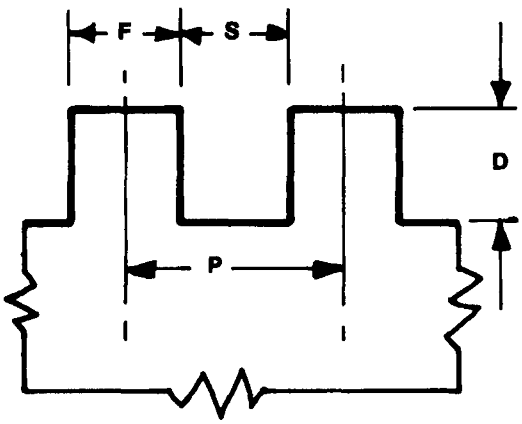
FOR ABOVE THREAD FORMS, P=PITCH=1÷THREADS PER INCH, AND R=ROOT=PITCH÷8

TABLE 7-9. General Form Dimensions for Standard Screw Threads(cont).



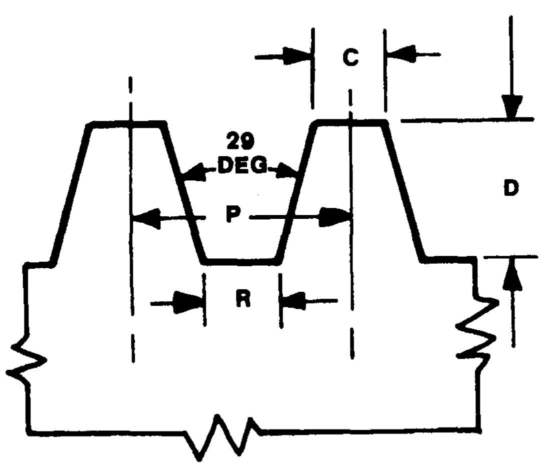
D = DEPTH = 1/2 PITCH + 0.01 INCH
C = CREST = 0.03707 × PITCH
R = ROOT = CREST − 0.0052 INCH

ACME SCREW THREAD



D = DEPTH = 1/2 PITCH
F = FLAT = 1/2 PITCH
S = SPACE =
FOR SCREW : 1/2 PITCH
FOR NUT : 1/2 PITCH + 0.001 TO 0.002 INCH CLEARANCE

SQUARE SCREW THREAD

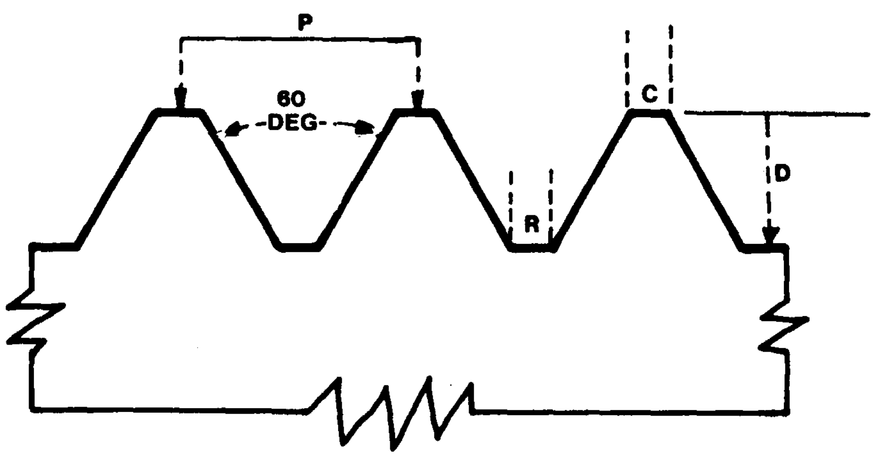


D = DEPTH = 0.6866 X PITCH
C = CREST = 0.335 × PITCH
R = ROOT = 0.310 × PITCH

29-DEG WORM SCREW THREAD (BROWN AND SHARPE)

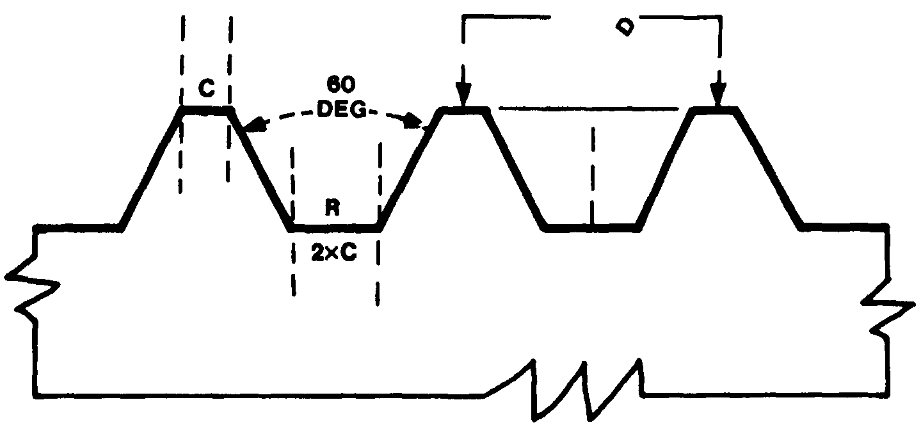
FOR ABOVE THREAD FORMS, P=PITCH=1÷THREADS PER INCH

TABLE 7-9. General Form Dimensions for Standard Screw Threads(cont).



D = DEPTH = 0.7035 × P (max)
= 0.6855 × P (min)
C = CREST = ROOT = P ÷ 8

INTERNATIONAL METRIC THREAD
(SPARK PLUG THREAD)



D = DEPTH = 0.54127 × P
C = CREST = P ÷ 8
R = ROOT = P ÷ 4

ISO METRIC THREAD STANDARD

TABLE 7-10. Standard Series Limits of Size-Unified and American Screw Threads.

Nominal size and threads per inch	Series designation	External									Internal						
		Class	Allowance	Major diameter limits			Pitch diameter limits			Minor diameter	Class	Minor diameter limits		Pitch diameter limits			Major diameter
				Max	Min	Min	Max	Min	Tolerance			Min	Max	Min	Max	Tolerance	Min
			in.	in.	in.	in.	in.	in.	in.	in.		in.	in.	in.	in.	in.	in.
0—80	NF	2A	0.0005	0.0595	0.0563	....	0.0514	0.0496	0.0018	0.0442	2B	0.0465	0.0514	0.0519	0.0542	0.0023	0.0600
		3A	.0000	.0600	.0568	....	.0519	.0506	.0013	.0447	3B	.0465	.0514	.0519	.0536	.0017	.0600
1—64	NC	2A	.0006	.0724	.0686	....	.0623	.0603	.0020	.0532	2B	.0561	.0623	.0629	.0655	.0026	.0730
		3A	.0000	.0730	.0692	....	.0629	.0614	.0015	.0538	3B	.0561	.0623	.0629	.0648	.0019	.0730
1—72	NF	2A	.0006	.0724	.0689	....	.0634	.0615	.0019	.0554	2B	.0580	.0635	.0640	.0665	.0025	.0730
		3A	.0000	.0730	.0695	....	.0640	.0626	.0014	.0560	3B	.0580	.0635	.0640	.0659	.0019	.0730
2—56	NC	2A	.0006	.0854	.0813	....	.0738	.0717	.0021	.0635	2B	.0667	.0737	.0744	.0772	.0028	.0860
		3A	.0000	.0860	.0819	....	.0744	.0728	.0016	.0641	3B	.0667	.0737	.0744	.0765	.0021	.0860
2—64	NF	2A	.0006	.0854	.0816	....	.0753	.0733	.0020	.0662	2B	.0691	.0753	.0759	.0786	.0027	.0860
		3A	.0000	.0860	.0822	....	.0759	.0744	.0015	.0668	3B	.0691	.0753	.0759	.0779	.0020	.0860
3—48	NC	2A	.0007	.0983	.0938	....	.0848	.0825	.0023	.0727	2B	.0764	.0845	.0855	.0885	.0030	.0990
		3A	.0000	.0990	.0945	....	.0855	.0838	.0017	.0734	3B	.0764	.0845	.0855	.0877	.0022	.0990
3—56	NF	2A	.0007	.0983	.0942	....	.0867	.0845	.0022	.0764	2B	.0797	.0865	.0874	.0902	.0028	.0990
		3A	.0000	.0990	.0949	....	.0874	.0858	.0016	.0771	3B	.0797	.0865	.0874	.0895	.0021	.0990
4—40	NC	2A	.0008	.1112	.1061	....	.0950	.0925	.0025	.0805	2B	.0849	.0939	.0958	.0991	.0033	.1120
		3A	.0000	.1120	.1069	....	.0958	.0939	.0019	.0813	3B	.0849	.0939	.0958	.0982	.0024	.1120
4—48	NF	2A	.0007	.1113	.1068	....	.0978	.0954	.0024	.0857	2B	.0894	.0968	.0985	.1016	.0031	.1120
		3A	.0000	.1120	.1075	....	.0985	.0967	.0018	.0864	3B	.0894	.0968	.0985	.1008	.0023	.1120
5—40	NC	2A	.0008	.1242	.1191	....	.1080	.1054	.0026	.0935	2B	.0979	.1062	.1088	.1121	.0033	.1250
		3A	.0000	.1250	.1199	....	.1088	.1069	.0019	.0943	3B	.0979	.1062	.1088	.1113	.0025	.1250
5—44	NF	2A	.0007	.1243	.1195	....	.1095	.1070	.0025	.0964	2B	.1004	.1079	.1102	.1134	.0032	.1250
		3A	.0000	.1250	.1202	....	.1102	.1083	.0019	.0971	3B	.1004	.1079	.1102	.1126	.0024	.1250
6—32	NC	2A	.0008	.1372	.1312	....	.1169	.1141	.0028	.0989	2B	.104	.114	.1177	.1214	.0037	.1380
		3A	.0000	.1380	.1320	....	.1177	.1156	.0021	.0997	3B	.1040	.1140	.1177	.1204	.0027	.1380
6—40	NF	2A	.0008	.1372	.1321	....	.1210	.1184	.0026	.1065	2B	.111	.119	.1218	.1252	.0034	.1380
		3A	.0000	.1380	.1329	....	.1218	.1198	.0020	.1073	3B	.1110	.1186	.1218	.1243	.0025	.1380
8—32	NC	2A	.0009	.1631	.1571	....	.1428	.1399	.0029	.1248	2B	.130	139	.1437	.1475	.0038	.1640
		3A	.0000	.1640	.1580	....	.1437	.1415	.0022	.1257	3B	.1300	.1389	.1437	.1465	.0028	.1640
8—36	NF	2A	.0008	.1632	.1577	....	.1452	.1424	.0028	.1291	2B	.134	.142	.1460	.1496	.0036	.1640
		3A	.0000	.1640	.1585	....	.1460	.1439	.0021	.1299	3B	.1340	.1416	.1460	.1487	.0027	.1640
10—24	NC	2A	.0010	.1890	.1818	....	.1619	.1586	.0033	.1379	2B	.145	.156	.1629	.1672	.0043	.1900
		3A	.0000	.1900	.1828	....	.1629	.1604	.0025	.1389	3B	.1450	.1555	.1629	.1661	.0032	.1900
10—32	NF	2A	.0009	.1891	.1831	....	.1688	.1658	.0030	.1508	2B	.156	.164	.1697	.1736	.0039	.1900
		3A	.0000	.1900	.1840	....	.1697	.1674	.0023	.1517	3B	.1560	.1641	.1697	.1726	.0029	.1900
12—24	NC	2A	.0010	.2150	.2078	....	.1879	.1845	.0034	.1639	2B	.171	.181	.1889	.1933	.0044	.2160
		3A	.0000	.2160	.2088	....	.1889	.1863	.0026	.1649	3B	.1710	.1807	.1889	.1922	.0033	.2160
12—28	NF	2A	.0010	.2150	.2085	....	.1918	.1886	.0032	.1712	2B	.177	.186	.1928	.1970	.0042	.2160
		3A	.0000	.2160	.2095	....	.1928	.1904	.0024	.1722	3B	.1770	.1857	.1928	.1959	.0031	.2160
12—32	NEF	2A	.0009	.2151	.2091	....	.1948	.1917	.0031	.1768	2B	.182	.190	.1957	.1998	.0041	.2160
		3A	.0000	.2160	.2100	....	.1957	.1933	.0024	.1777	3B	.1820	.1895	.1957	.1988	.0031	.2160

NOTE.—The following seven sizes have been standardized as between American, Canadian, and British military services or industry for purposes of attachment, e.g., an instrument or accessory to a panel: 0—80 NF, 2—56 NC, 4—40 NC, 8—32 NC, 10—24 NC, and 10—32 NF, with 10—32 preferred over 10—24.

TABLE 7-10. Standard Series Limits of Size-Unified and American Screw Threads(cont.).

Nominal size and threads per inch	Series designation	External									Internal						
		Class	Allowance	Major diameter limits			Pitch diameter limits			Minor diameter	Class	Minor diameter limits		Pitch diameter limits			Major diameter
				Max	Min	Min	Max	Min	Tolerance			Min	Max	Min	Max	Tolerance	Min
			in.	in.	in.	in.	in.	in.	in.	in.		in.	in.	in.	in.	in.	in.
1/4—20	UNC	1A	.0011	.2489	.2367	....	.2164	.2108	.0056	.1876	1B	.196	.207	.2175	.2248	.0073	.2500
		2A	.0011	.2489	.2408	0.2367	.2164	.2127	.0037	.1876	2B	.196	.207	.2175	.2223	.0048	.2500
		3A	.0000	.2500	.2419	....	.2175	.2147	.0028	.1887	3B	.1960	.2067	.2175	.2211	.0036	.2500
1/4—28	UNF	1A	.0010	.2490	.2392	....	.2258	.2208	.0050	.2052	1B	.211	.220	.2268	.2333	.0065	.2500
		2A	.0010	.2490	.2425	....	.2258	.2225	.0033	.2052	2B	.211	.220	.2268	.2311	.0043	.2500
		3A	.0000	.2500	.2435	....	.2268	.2243	.0025	.2062	3B	.2110	.2190	.2268	.2300	.0032	.2500
1/4—32	NEF	2A	.0010	.2490	.2430	....	.2287	.2255	.0032	.2107	2B	.216	.224	.2297	.2339	.0042	.2500
		3A	.0000	.2500	.2440	....	.2297	.2273	.0024	.2117	3B	.2160	.2229	.2297	.2328	.0031	.2500
5/16—18	UNC	1A	.0012	.3113	.2982	....	.2752	.2691	.0061	.2431	1B	.252	.265	.2764	.2843	.0079	.3125
		2A	.0012	.3113	.3026	....	.2752	.2712	.0040	.2431	2B	.252	.265	.2764	.2817	.0053	.3125
		3A	.0000	.3125	.3038	....	.2764	.2734	.0030	.2443	3B	.2520	.2630	.2764	.2803	.0039	.3125
5/16—24	UNF	1A	.0011	.3114	.3006	....	.2843	.2788	.0055	.2603	1B	.267	.277	.2854	.2925	.0071	.3125
		2A	.0011	.3114	.3042	....	.2843	.2806	.0037	.2603	2B	267	.277	.2854	.2902	.0048	.3125
		3A	.0000	.3125	.3053	....	.2854	.2827	.0027	.2614	3B	.2670	.2754	.2854	.2890	.0036	.3125
5/16—32	NEF	2A	.0010	.3115	.3055	....	.2912	.2880	.0032	.2732	2B	.279	.286	.2922	.2964	.0042	.3125
		3A	.0000	.3125	.3065	....	.2922	.2898	.0024	.2742	3B	.2790	.2847	.2922	.2953	.0031	.3125
3/8—16	UNC	1A	.0013	.3737	.3595	....	.3331	.3266	.0065	.2970	1B	.307	.321	.3344	.3429	.0085	.3750
		2A	.0013	.3737	.3643	.3595	.3331	.3287	.0044	.2970	2B	.307	.321	.3344	.3401	.0057	.3750
		3A	.0000	.3750	.3656	....	.3344	.3311	.0033	.2983	3B	.3070	.3182	.3344	.3387	.0043	.3750
3/8—24	UNF	1A	.0011	.3739	.3631	....	.3468	.3411	.0057	.3228	1B	.330	.340	.3479	.3553	.0074	.3750
		2A	.0011	.3739	.3667	....	.3468	.3430	.0038	.3228	2B	.330	.340	.3479	.3528	.0049	.3750
		3A	.0000	.3750	.3678	....	.3479	.3450	.0029	.3239	3B	.3300	.3372	.3479	.3516	.0037	.3750
3/8—32	NEF	2A	.0010	.3740	.3680	....	.3537	.3503	.0034	.3357	2B	.341	.349	.3547	.3591	.0044	.3750
		3A	.0000	.3750	.3690	....	.3547	.3522	.0025	.3367	3B	.3410	.3469	.3547	.3580	.0033	.3750
7/16—14	UNC	1A	.0014	.4361	.4206	....	.3897	.3826	.0071	.3485	1B	.360	.376	.3911	.4003	.0092	.4375
		2A	.0014	.4361	.4258	.4206	.3897	.3850	.0047	.3485	2B	.360	.376	.3911	.3972	.0061	.4375
		3A	.0000	.4375	.4272	....	.3911	.3876	.0035	.3499	3B	.3600	.3717	.3911	.3957	.0046	.4375
7/16—20	UNF	1A	.0013	.4362	.4240	....	.4037	.3975	.0062	.3749	1B	.383	.395	.4050	.4131	.0081	.4375
		2A	.0013	.4362	.4281	....	.4037	.3995	.0042	.3749	2B	.383	.395	.4050	.4104	.0054	.4375
		3A	.0000	.4375	.4294	....	.4050	.4019	.0031	.3762	3B	.3830	.3916	.4050	.4091	.0041	.4375
7/16—28	UNEF	2A	.0011	.4364	.4299	....	.4132	.4096	.0036	.3926	2B	.399	.407	.4143	.4189	.0046	.4375
		3A	.0000	.4375	.4310	....	.4143	.4116	.0027	.3937	3B	.3990	.4051	.4143	.4178	.0035	.4375
1/2—12	N	2A	.0016	.4984	.4870	....	.4443	.4389	.0054	.3962	2B	.410	.428	.4459	.4529	.0070	.5000
		3A	.0000	.5000	.4886	....	.4459	.4419	.0040	.3978	3B	.4100	.4223	.4459	.4511	.0052	.5000
1/2—13	UNC	1A	.0015	.4985	.4822	....	.4485	.4411	.0074	.4041	1B	.417	.434	.4500	.4597	.0097	.5000
		2A	.0015	.4985	.4876	.4822	.4485	.4435	.0050	.4041	2B	.417	.434	.4500	.4565	.0065	.5000
		3A	.0000	.5000	.4891	....	.4500	.4463	.0037	.4056	3B	.4170	.4284	.4500	.4548	.0048	.5000

TABLE 7-10. Standard Series Limits of Size-Unified and American Screw Threads(cont.).

Nominal size and threads per inch	Series designation	External									Internal							
		Class	Allowance	Major diameter limits			Pitch diameter limits			. eter	Class	Minor diameter limits		Pitch diameter limits			Major diameter
				Max	Min	Min	Max	Min	Tolerance			Min	Max	Min	Max	Tolerance	Min
			in.	in.	in.	in.	in.	in.	in.	in.		in.	in.	in.	in.	in.	in.
1/2—20	UNF	1A	0.0013	0.4987	0.4865	....	0.4662	0.4598	0.0064	0.4374	1B	0.416	0.457	0.4675	0.4759	0.0084	0.5000
		2A	.0013	.4987	.4906	....	.4662	.4619	.0043	.4374	2B	.446	.457	.4675	.4731	.0056	.5000
		3A	.0000	.5000	.4919	....	.4675	.4643	.0032	.4387	3B	.4460	.4537	.4675	.4717	.0042	.5000
1/2—28	UNEF	2A	.0011	.4989	.4924	....	.4757	.4720	.0037	.4551	2B	.461	.470	.4768	.4816	.0048	.5000
		3A	.0000	.5000	.4935	....	.4768	.4740	.0028	.4562	3B	.4610	.4676	.4768	.4804	.0036	.5000
9/16—12	UNC	1A	.0016	.5609	.5437	....	.5068	.4990	.0078	.4587	1B	.472	.490	.5084	.5186	.0102	.5625
		2A	.0016	.5609	.5495	0.5437	.5068	.5016	.0052	.4587	2B	.472	.490	.5084	.5152	.0068	.5625
		3A	.0000	.5625	.5511	....	.5084	.5045	.0039	.4603	3B	.4720	.4843	.5084	.5135	.0051	.5625
9/16—18	UNF	1A	.0014	.5611	.5480	....	.5250	.5182	.0068	.4929	1B	.502	.515	.5264	.5353	.0089	.5625
		2A	.0014	.5611	.5524	....	.5250	.5205	.0045	.4929	2B	.502	.515	.5264	.5323	.0059	.5625
		3A	.0000	.5625	.5538	....	.5264	.5230	.0034	.4943	3B	.5020	.5106	.5264	.5308	.0044	.5625
9/16—24	NEF	2A	.0012	.5613	.5541	....	.5342	.5303	.0039	.5102	2B	.517	.527	.5354	.5405	.0051	.5625
		3A	.0000	.5625	.5563	....	.5354	.5325	.0029	.5114	3B	.5170	.5244	.5354	.5392	.0038	.5625
5/8—11	UNC	1A	.0016	.6234	.6062	....	.5644	.5561	.0083	.5119	1B	.527	.546	.5660	.5767	.0107	.6250
		2A	.0016	.6234	.6113	.6052	.5644	.5589	.0055	.5119	2B	.527	.546	.5660	.5732	.0072	.6250
		3A	.0000	.6250	.6129	....	.5660	.5619	.0041	.5135	3B	.5270	.5391	.5660	.5714	.0054	.6250
5/8—12	N	2A	.0016	.6234	.6120	....	.5693	.5639	.0054	.5212	2B	.535	.553	.5709	.5780	.0071	.6250
		3A	.0000	.6250	.6136	....	.5709	.5668	.0041	.5228	3B	.5350	.5463	.5709	.5762	.0053	.6250
5/8—18	UNF	1A	.0014	.6236	.6105	....	.5875	.5805	.0070	.5554	1B	.565	.578	.5889	.5890	.0091	.6250
		2A	.0014	.6236	.6149	....	.5875	.5828	.0047	.5554	2B	.565	.578	.5889	.5949	.0060	.6250
		3A	.0000	.6250	.6163	....	.5889	.5854	.0035	.5568	3B	.5650	.5730	.5889	.5934	.0045	.6250
5/8—24	NEF	2A	.0012	.6238	.6166	....	.5967	.5927	.0040	.5727	2B	.580	.590	.5979	.6031	.0052	.6250
		3A	.0000	.6250	.6178	....	.5979	.5949	.0030	.5739	3B	.5800	.5869	.5979	.6018	.0039	.6250
11/16—12	N	2A	.0016	.6859	.6745	....	.6318	.6264	.0054	.5837	2B	.597	.615	.6334	.6405	.0071	.6875
		3A	.0000	.6875	.6761	....	.6334	.6293	.0041	.5853	3B	.5970	.6085	.6334	.6387	.0053	.6875
11/16—24	NEF	2A	.0012	.6863	.6791	....	.6592	.6552	.0040	.6352	2B	.642	.652	.6604	.6656	.0052	.6875
		3A	.0000	.6875	.6803	....	.6604	.6574	.0030	.6364	3B	.6420	.6494	.6604	.6643	.0039	.6875
3/4—10	UNC	1A	.0018	.7482	.7288	....	.6832	.6744	.0088	.6255	1B	.642	.663	.6850	.6965	.0115	.7500
		2A	.0018	.7482	.7353	.7288	.6832	.6773	.0059	.6255	2B	.642	.663	.6850	.6927	.0077	.7500
		3A	.0000	.7500	.7371	....	.6850	.6806	.0044	.6273	3B	.6420	.6545	.6850	.6907	.0057	.7500
3/4—12	N	2A	.0017	.7483	.7369	....	.6942	.6887	.0055	.6461	2B	.660	.678	.6959	.7031	.0072	.7500
		3A	.0000	.7500	.7386	....	.6959	.6918	.0041	.6478	3B	.6600	.6707	.6959	.7013	.0054	.7500
3/4—16	UNF	1A	.0015	.7485	.7473	....	.7079	.7004	.0075	.6718	1B	.682	.696	.7094	.7192	.0098	.7500
		2A	.0015	.7485	.7391	....	.7079	.7029	.0050	.6718	2B	.682	.696	.7094	.7159	.0065	.7500
		3A	.0000	.7500	.7406	....	.7094	.7056	.0038	.6733	3B	.6820	.6908	.7094	.7143	.0049	.7500
3/4—20	UNEF	2A	.0013	.7487	.7406	....	.7162	.7118	.0044	.6874	2B	.696	.707	.7175	.7232	.0057	.7500
		3A	.0000	.7500	.7419	....	.7175	.7142	.0033	.6887	3B	.6960	.7037	.7175	.7218	.0043	.7500
13/16—12	N	2A	.0017	.8108	.7994	....	.7567	.7512	.0055	.7086	2B	.722	.740	.7584	.7656	.0072	.8125
		3A	.0000	.8125	.8011	....	.7584	.7543	.0041	.7103	3B	.7220	.7329	.7584	.7638	.0054	.8125
13/16—16	UN	2A	.0015	.8110	.8016	....	.7704	.7655	.0049	.7343	2B	.745	.759	.7719	.7782	.0063	.8125
		3A	.0000	.8125	.8031	....	.7719	.7683	.0036	.7358	3B	.7450	.7533	.7719	.7766	.0047	.8125

TABLE 7-10. Standard Series Limits of Size-Unified and American Screw Threads (cont).

Nominal size and threads per inch	Series designation	External									Internal						
		Class	Allowance	Major diameter limits			Pitch diameter limits			Minor diameter	Class	Minor diameter limits		Pitch diameter limits			Major diameter
				Max	Min	Min	Max	Min	Tolerance			Min	Max	Min	Max	Tolerance	Min
			in.	in.	in.	in.	in.	in.	in.	in.		in.	in.	in.	in.	in.	in.
13/16—20	UNEF	2A	.0013	.8112	.8031	....	.7787	.7743	.0044	.7498	2B	.758	.770	.7800	.7857	.0057	.8125
		3A	.0000	.8125	.8044	....	.7800	.7767	.0033	.7512	3B	.7580	.7662	.7800	.7843	.0043	.8125
7/8—9	UNC	1A	.0019	.8731	.8523	....	.8009	.7914	.0095	.7368	1B	.755	.778	.8028	.8151	.0123	.8750
		2A	.0019	.8731	.8592	.8523	.8009	.7946	.0063	.7368	2B	.755	.778	.8028	.8110	.0082	.8750
		3A	.0000	.8750	.8611	....	.8028	.7981	.0047	.7387	3B	.7550	.7681	.8028	.8089	.0061	.8750
7/8—12	N	2A	.0017	.8733	.8619	....	.8192	.8137	.0055	.7711	2B	.785	.803	.8209	.8281	.0072	.8750
		3A	.0000	.8750	.8636	....	.8209	.8168	.0041	.7728	3B	.7850	.7952	.8209	.8263	.0054	.8750
7/8—14	UNF	1A	.0016	.8734	.8579	....	.8270	.8189	.0081	.7858	1B	.798	.814	.8286	.8392	.0106	.8750
		2A	.0016	.8734	.8631	....	.8270	.8216	.0054	.7858	2B	.798	.814	.8286	.8356	.0070	.8750
		3A	.0000	.8750	.8647	....	.8286	.8645	.0041	.7874	3B	.7980	.8068	.8286	.8339	.0053	.8750
7/8—16	UN	2A	.0015	.8735	.8641	....	.8329	.8280	.0049	.7968	2B	.807	.821	.8344	.8407	.0063	.8750
		3A	.0000	.8750	.8656	....	.8344	.8308	.0036	.7983	3B	.8070	.8158	.8344	.8391	.0047	.8750
7/8—20	UNEF	2A	.0013	.8737	.8656	....	.8412	.8368	.0044	.8124	2B	.821	.832	.8425	.8482	.0057	.8750
		3A	.0000	.8750	.8669	....	.8425	.8392	.0033	.8137	3B	.8210	.8287	.8425	.8468	.0043	.8750
15/16—12	UN	2A	.0017	.9358	.9244	....	.8817	.8760	.0057	.8336	2B	.847	.865	.8834	.8908	.0074	.9375
		3A	.0000	.9375	.9261	....	.8834	.8793	.0041	.8353	3B	.8470	.8575	.8834	.8889	.0055	.9375
15/16—16	UN	2A	.0015	.9360	.9266	....	.8954	.8904	.0050	.8593	2B	.870	.884	.8969	.9034	.0065	.9375
		3A	.0000	.9375	.9281	....	.8969	.8932	.0037	.8608	3B	.8700	.8783	.8969	.9018	.0049	.9375
15/16—20	UNEF	2A	.0014	.9361	.9280	....	.9036	.8991	.0045	.8748	2B	.883	.895	.9050	.9109	.0059	.9375
		3A	.0000	.9375	.9294	....	.9050	.9016	.0034	.8762	3B	.8830	.8912	.9050	.9094	.0044	.9375
1—8	UNC	1A	.0020	.9980	.9755	....	.9168	.9067	.0101	.8446	1B	.865	.890	.9188	.9320	.0132	1.0000
		2A	.0020	.9980	.9830	.9755	.9168	.9100	.0068	.8446	2B	.865	.890	.9188	.9276	.0088	1.0000
		3A	.0000	1.0000	.9850	....	.9188	.9137	.0051	.8466	3B	.8650	.8797	.9188	.9254	.0066	1.0000
1—12	UNF	1A	.0018	.9982	.9810	....	.9441	.9353	.0088	.8960	1B	.910	.928	.9459	.9573	.0114	1.0000
		2A	.0018	.9982	.9868	....	.9441	.9382	.0059	.8960	2B	.910	.928	.9495	.9535	.0076	1.0000
		3A	.0000	1.0000	.9886	....	.9459	.9415	.0044	.8978	3B	.9100	.9198	.9495	.9516	.0057	1.0000
1—16	UN	2A	.0015	.9985	.9891	....	.9579	.9529	.0050	.9218	2B	.932	.946	.9594	.9659	.0065	1.0000
		3A	.0000	1.0000	.9906	....	.9594	.9557	.0037	.9233	3B	.9320	.9408	.9594	.9643	.0049	1.0000
1—20	UNEF	2A	.0014	.9986	.9905	....	.9661	.9616	.0045	.9373	2B	.946	.957	.9675	.9734	.0059	1.0000
		3A	.0000	1.0000	.9919	....	.9675	.9641	.0034	.9387	3B	.9460	.9537	.9675	.9719	.0044	1.0000
1-1/16—12	UN	2A	.0017	1.0608	1.0494	....	1.0067	1.0010	.0057	.9586	2B	.972	.990	1.0084	1.0158	.0074	1.0625
		3A	.0000	1.0625	1.0511	....	1.0084	1.0042	.0042	.9603	3B	.9720	.9823	1.0084	1.0139	.0055	1.0625
1-1/16—16	UN	2A	.0015	1.0610	1.0516	....	1.0204	1.0154	.0050	.9843	2B	.995	1.009	1.0219	1.0284	.0065	1.0625
		3A	.0000	1.0625	1.0531	....	1.0219	1.0182	.0037	.9858	3B	.9950	1.0033	1.0219	1.0268	.0049	1.0625
1-1/16—18	NEF	2A	.0014	1.0611	1.0524	....	1.0250	1.0203	.0047	.9929	2B	1.002	1.015	1.0264	1.0326	.0062	1.0625
		3A	.0000	1.0625	1.0538	....	1.0264	1.0228	.0036	.9943	3B	1.0020	1.0105	1.0264	1.0310	.0046	1.0625
1-1/8—7	UNC	1A	.0022	1.1228	1.0982	....	1.0300	1.0191	.0109	.9475	1B	.970	.998	1.0322	1.0463	.0141	1.1250
		2A	.0022	1.1228	1.1064	1.0982	1.0300	1.0228	.0072	.9475	2B	.970	.998	1.0322	1.0416	.0094	1.1250
		3A	.0000	1.1250	1.1086	....	1.0322	1.0268	.0054	.9497	3B	.9700	.9875	1.0322	1.0393	.0071	1.1250
1-1/8—8	N	2A	.0021	1.1229	1.1079	1.1004	1.0417	1.0348	.0069	.9695	2B	.990	1.015	1.0438	1.0528	.0090	1.1250
		3A	.0000	1.1250	1.1100	....	1.0438	1.0386	.0052	.9716	3B	.9900	1.0047	1.0438	1.0505	.0067	1.1250
1-1/8—12	UNF	1A	.0018	1.1232	1.1060	....	1.0691	1.0601	.0090	1.0210	1B	1.035	1.053	1.0709	1.0826	.0117	1.1250
		2A	.0018	1.1232	1.1118	....	1.0691	1.0631	.0060	1.0210	2B	1.035	1.053	1.0709	1.0787	.0078	1.1250
		3A	.0000	1.1250	1.1136	....	1.0709	1.0664	.0045	1.0228	3B	1.0350	1.0448	1.0709	1.0768	.0059	1.1250

TABLE 7-10. Standard Series Limits of Size-Unified and American Screw Threads (cont.).

Nominal size and threads per inch	Series designation	External									Internal						
		Class	Allowance	Major diameter limits			Pitch diameter limits			Minor diameter	Class	Minor diameter limits		Pitch diameter limits			Major diameter
				Max	Min	Min	Max	Min	Tolerance			Min	Max	Min	Max	Tolerance	Min
			in.	in.	in.	in.	in.	in.	in.	in.		in.	in.	in.	in.	in.	in.
1-1/8—16	UN	2A	0.0015	1.1235	1.1141	....	1.0829	1.0779	0.0050	1.0468	2B	1.057	1.071	1.0844	1.0909	1.0065	1.1250
		3A	.0000	1.1250	1.1156	....	1.0844	1.0807	.0037	1.0483	3B	1.0570	1.0658	1.0844	1.0893	.0049	1.1250
1-1/8—18	NEF	2A	.0014	1.1236	1.1149	....	1.0875	1.0828	.0047	1.0554	2B	1.065	1.078	1.0889	1.0951	.0062	1.1250
		3A	.0000	1.1250	1.1163	....	1.0889	1.0853	.0036	1.0568	3B	1.0650	1.0730	1.0889	1.0935	.0046	1.1250
1-3/16—12	UN	2A	.0017	1.1858	1.1744	....	1.1317	1.1259	.0058	1.0836	2B	1.097	1.115	1.1334	1.1409	.0075	1.1875
		3A	.0000	1.1875	1.1761	....	1.1334	1.1291	.0043	1.0853	3B	1.0970	1.1073	1.1334	1.1390	.0056	1.1875
1-3/16—16	UN	2A	.0015	1.1860	1.1766	....	1.1454	1.1403	.0051	1.1093	2B	1.120	1.134	1.1469	1.1535	.0066	1.1875
		3A	.0000	1.1875	1.1781	....	1.1469	1.1431	.0038	1.1108	3B	1.1200	1.1283	1.1469	1.1519	.0050	1.1875
1-3/16—18	NEF	2A	.0015	1.1860	1.1773	....	1.1499	1.1450	.0049	1.1178	2B	1.127	1.140	1.1514	1.1577	.0063	1.1875
		3A	.0000	1.1875	1.1788	....	1.1514	1.1478	.0036	1.1193	3B	1.1270	1.1355	1.1514	1.1561	.0047	1.1875
1-1/4—7	UNC	1A	.0022	1.2478	1.2232	....	1.1550	1.1439	.0111	1.0725	1B	1.095	1.123	1.1572	1.1716	.0144	1.2500
		2A	.0022	1.2478	1.2314	1.2232	1.1550	1.1476	.0074	1.0725	2B	1.095	1.123	1.1572	1.1668	.0096	1.2500
		3A	.0000	1.2500	1.2336	....	1.1572	1.1517	.0055	1.0747	3B	1.0950	1.1125	1.1572	1.1644	.0072	1.2500
1-1/4—8	N	2A	.0021	1.2479	1.2329	1.2254	1.1667	1.1597	.0070	1.0945	2B	1.115	1.140	1.1688	1.1780	.0092	1.2500
		3A	.0000	1.2500	1.2350	....	1.1688	1.1635	.0053	1.0966	3B	1.1150	1.1297	1.1688	1.1757	.0069	1.2500
1-1/4—12	UNF	1A	.0018	1.2482	1.2310	....	1.1941	1.1849	.0092	1.1460	1B	1.160	1.178	1.1959	1.2079	.0120	1.2500
		2A	.0018	1.2482	1.2368	....	1.1941	1.1879	.0062	1.1460	2B	1.160	1.178	1.1959	1.2039	.0080	1.2500
		3A	.0000	1.2500	1.2386	....	1.1959	1.1913	.0046	1.1478	3B	1.1600	1.1698	1.1959	1.2019	.0060	1.2500
1-1/4—16	UN	2A	.0015	1.2485	1.2391	....	1.2079	1.2028	.0051	1.1718	2B	1.182	1.196	1.2094	1.2160	.0066	1.2500
		3A	.0000	1.2500	1.2406	....	1.2094	1.2056	.0038	1.1733	3B	1.1820	1.1908	1.2094	1.2144	.0050	1.2500
1-1/4—18	NEF	2A	.0015	1.2485	1.2398	....	1.2124	1.2075	.0049	1.1803	2B	1.190	1.203	1.2139	1.2202	.0063	1.2500
		3A	.0000	1.2500	1.2413	....	1.2139	1.2103	.0036	1.1818	3B	1.1900	1.1980	1.2139	1.2186	.0047	1.2500
1-5/16—12	UN	2A	.0017	1.3108	1.2994	....	1.2567	1.2509	.0058	1.2086	2B	1.222	1.240	1.2584	1.2659	.0075	1.3125
		3A	.0000	1.3125	1.3011	....	1.2584	1.2541	.0043	1.2103	3B	1.2220	1.2323	1.2584	1.2640	.0056	1.3125
1-5/16—16	UN	2A	.0015	1.3110	1.3016	....	1.2704	1.2653	.0051	1.2343	2B	1.245	1.259	1.2719	1.2785	.0066	1.3125
		3A	.0000	1.3125	1.3031	....	1.2719	1.2681	.0038	1.2358	3B	1.2450	1.2533	1.2719	1.2769	.0050	1.3125
1-5/16—18	NEF	2A	.0015	1.3110	1.3023	....	1.2749	1.2700	.0049	1.2428	2B	1.252	1.265	1.2764	1.2827	.0063	1.3125
		3A	.0000	1.3125	1.3038	....	1.2764	1.2728	.0036	1.2443	3B	1.2520	1.2605	1.2764	1.2811	.0047	1.3125
1-3/8—6	UNC	1A	.0024	1.3726	1.3453	....	1.2643	1.2523	.0120	1.1681	1B	1.195	1.225	1.2667	1.2823	.0156	1.3750
		2A	.0024	1.3726	1.3544	1.3453	1.2643	1.2563	.0080	1.1681	2B	1.195	1.225	1.2667	1.2771	.0104	1.3750
		3A	.0000	1.3750	1.3568	....	1.2667	1.2607	.0060	1.1705	3B	1.1950	1.2146	1.2667	1.2745	.0078	1.3750
1-3/8—8	N	2A	.0022	1.3728	1.3578	1.3503	1.2916	1.2844	.0072	1.2194	2B	1.240	1.265	1.2938	1.3031	.0093	1.3750
		3A	.0000	1.3750	1.3600	....	1.2938	1.2884	.0054	1.2216	3B	1.2400	1.2547	1.2938	1.3008	.0070	1.3750
1-3/8—12	UNF	1A	.0019	1.3731	1.3559	....	1.3190	1.3096	.0094	1.2709	1B	1.285	1.303	1.3209	1.3332	.0123	1.3750
		2A	.0019	1.3731	1.3617	....	1.3190	1.3127	.0063	1.2709	2B	1.285	1.303	1.3209	1.3291	.0082	1.3750
		3A	.0000	1.3750	1.3636	....	1.3209	1.3162	.0047	1.2728	3B	1.2850	1.2948	1.3209	1.3270	.0061	1.3750
1-3/8—16	UN	2A	.0015	1.3735	1.3641	....	1.3329	1.3278	.0051	1.2968	2B	1.307	1.321	1.3344	1.3410	.0066	1.3750
		3A	.0000	1.3750	1.3656	....	1.3344	1.3306	.0038	1.2983	3B	1.3070	1.3158	1.3344	1.3394	.0050	1.3750
1-3/8—18	NEF	2A	.0015	1.3735	1.3648	....	1.3374	1.3325	.0049	1.3053	2B	1.315	1.328	1.3389	1.3452	.0063	1.3750
		3A	.0000	1.3750	1.3663	....	1.3389	1.3353	.0036	1.3068	3B	1.3150	1.3230	1.3389	1.3436	.0047	1.3750
1-7/16—12	UN	2A	.0018	1.4357	1.4243	....	1.3816	1.3757	.0059	1.3335	2B	1.347	1.365	1.3834	1.3910	.0076	1.4375
		3A	.0000	1.4375	1.4261	....	1.3834	1.3790	.0044	1.3353	3B	1.3470	1.3573	1.3834	1.3891	.0057	1.4375
1-7/16—16	UN	2A	.0016	1.4359	1.4265	....	1.3953	1.3901	.0052	1.3592	2B	1.370	1.384	1.3969	1.4037	.0068	1.4375
		3A	.0000	1.4375	1.4281	....	1.3969	1.3930	.0039	1.3608	3B	1.3700	1.3783	1.3969	1.4020	.0051	1.4375

TABLE 7-10. Standard Series Limits of Self-Unified and American Screw Threads (cont.).

Nominal size and threads per inch	Series designation	External										Internal						
		Class	Allowance	Major diameter limits			Pitch diameter limits			Minor diameter	Class	Minor diameter limits		Pitch diameter limits			Major diameter
				Max	Min	Min	Max	Min	Tolerance			Min	Max	Min	Max	Tolerance	Min
			in.	*in.*	*in.*	*in.*	*in.*	*in.*	*in.*	*in.*		*in.*	*in.*	*in.*	*in.*	*in.*	*in.*
1-7/16—18	NEF	2A	.0015	1.4360	1.4273	....	1.3999	1.3949	.0050	1.3678	2B	1.377	1.390	1.4014	1.4079	.0065	1.4375
		3A	.0000	1.4375	1.4288	....	1.4014	1.3977	.0037	1.3693	3B	1.3770	1.3855	1.4014	1.4062	.0048	1.4375
1-1/2—6	UNC	1A	.0024	1.4976	1.4703	....	1.3893	1.3772	.0121	1.2931	1B	1.320	1.350	1.3917	1.4075	.0158	1.5000
		2A	.0024	1.4976	1.4794	1.4703	1.3893	1.3812	.0081	1.2931	2B	1.320	1.350	1.3917	1.4022	.0105	1.5000
		3A	.0000	1.5000	1.4818	....	1.3917	1.3856	.0061	1.2955	3B	1.3200	1.3396	1.3917	1.3996	.0079	1.5000
1-1/2—8	N	2A	.0022	1.4978	1.4828	1.4753	1.4166	1.4093	.0073	1.3444	2B	1.365	1.390	1.4188	1.4283	.0095	1.5000
		3A	.0000	1.5000	1.4850	....	1.4188	1.4133	.0055	1.3466	3B	1.3650	1.3797	1.4188	1.4259	.0071	1.5000
1-1/2—12	UNF	1A	.0019	1.4981	1.4809	....	1.4440	1.4344	.0096	1.3959	1B	1.410	1.428	1.4459	1.4584	.0125	1.5000
		2A	.0019	1.4981	1.4867	....	1.4440	1.4376	.0064	1.3959	2B	1.410	1.428	1.4459	1.4542	.0083	1.5000
		3A	.0000	1.5000	1.4886	....	1.4459	1.4411	.0048	1.3978	3B	1.4100	1.4198	1.4459	1.4522	.0063	1.5000
1-1/2—16	UN	2A	.0016	1.4984	1.4890	....	1.4578	1.4526	.0052	1.4217	2B	1.432	1.446	1.4594	1.4662	.0068	1.5000
		3A	.0000	1.5000	1.4906	....	1.4594	1.4555	.0039	1.4233	3B	1.4320	1.4408	1.4594	1.4645	.0051	1.5000
1-1/2—18	NEF	2A	.0015	1.4985	1.4898	....	1.4624	1.4574	.0050	1.4303	2B	1.440	1.452	1.4639	1.4704	.0065	1.5000
		3A	.0000	1.5000	1.4913	....	1.4639	1.4602	.0037	1.4318	3B	1.4400	1.4480	1.4639	1.4687	.0048	1.5000
1-9/16—16	N	2A	.0016	1.5609	1.5515	....	1.5203	1.5151	.0052	1.4842	2B	1.495	1.509	1.5219	1.5287	.0068	1.5625
		3A	.0000	1.5625	1.5531	....	1.5219	1.5180	.0039	1.4858	3B	1.4950	1.5033	1.5219	1.5270	.0051	1.5625
1-9/16—18	NEF	2A	.0015	1.5610	1.5523	....	1.5249	1.5199	.0050	1.4928	2B	1.502	1.515	1.5264	1.5329	.0065	1.5625
		3A	.0000	1.5625	1.5538	....	1.5264	1.5227	.0037	1.4943	3B	1.5020	1.5105	1.5264	1.5312	.0048	1.5625
1-5/8—8	N	2A	.0022	1.6228	1.6078	1.6003	1.5416	1.5342	.0074	1.4694	2B	1.490	1.515	1.5438	1.5535	.0097	1.6250
		3A	.0000	1.6250	1.6100	....	1.5438	1.5382	.0056	1.4716	3B	1.4900	1.5047	1.5438	1.5510	.0072	1.6250
1-5/8—12	UN	2A	.0018	1.6232	1.6118	....	1.5691	1.5632	.0059	1.5210	2B	1.535	1.553	1.5709	1.5785	.0076	1.6250
		3A	.0000	1.6250	1.6136	....	1.5709	1.5665	.0044	1.5228	3B	1.5350	1.5448	1.5709	1.5766	.0057	1.6250
1-5/8—16	UN	2A	.0016	1.6234	1.6140	....	1.5828	1.5776	.0052	1.5167	2B	1.557	1.571	1.5844	1.5912	.0068	1.6250
		3A	.0000	1.6250	1.6156	....	1.5844	1.5805	.0039	1.5483	3B	1.5570	1.5658	1.5844	1.5895	.0051	1.6250
1-5/8—18	NEF	2A	.0015	1.6235	1.6148	....	1.5874	1.5824	.0050	1.5553	2B	1.565	1.578	1.5889	1.5954	.0065	1.6250
		3A	.0000	1.6250	1.6163	....	1.5889	1.5852	.0037	1.5568	3B	1.5650	1.5730	1.5889	1.5937	.0048	1.6250
1-11/16—16	N	2A	.0016	1.6859	1.6765	....	1.6453	1.6400	.0053	1.6092	2B	1.620	1.634	1.6469	1.6538	.0069	1.6875
		3A	.0000	1.6875	1.6781	....	1.6469	1.6429	.0040	1.6108	3B	1.6200	1.6283	1.6469	1.6521	.0052	1.6875
1-11/16—18	NEF	2A	.0015	1.6860	1.6773	....	1.6499	1.6448	.0051	1.6178	2B	1.627	1.640	1.6514	1.6580	.0066	1.6875
		3A	.0000	1.6875	1.6788	....	1.6514	1.6476	.0038	1.6193	3B	1.6270	1.6355	1.6514	1.6563	.0049	1.6875
1-3/4—5	UNC	1A	.0027	1.7473	1.7165	....	1.6174	1.6040	.0134	1.5019	1B	1.534	1.568	1.6201	1.6375	.0174	1.7500
		2A	.0027	1.7473	1.7268	1.7165	1.6174	1.6085	.0089	1.5019	2B	1.534	1.568	1.6201	1.6317	.0116	1.7500
		3A	.0000	1.7500	1.7295	....	1.6201	1.6134	.0067	1.5046	3B	1.5340	1.5575	1.6201	1.6288	.0087	1.7500
1-3/4—8	N	2A	.0023	1.7477	1.7327	1.7252	1.6665	1.6590	.0075	1.5943	2B	1.615	1.640	1.6688	1.6786	.0098	1.7500
		3A	.0000	1.7500	1.7350	....	1.6688	1.6632	.0056	1.5966	3B	1.6150	1.6297	1.6688	1.6762	.0074	1.7500
1-3/4—12	UN	2A	.0018	1.7482	1.7368	....	1.6941	1.6881	.0060	1.6460	2B	1.660	1.678	1.6959	1.7037	.0078	1.7500
		3A	.0000	1.7500	1.7386	....	1.6959	1.6914	.0045	1.6478	3B	1.6600	1.6698	1.6959	1.7017	.0058	1.7500

TABLE 7-10. Standard Series Limits of Self-Unified and American Screw Threads (cont).

Nominal size and threads per inch	Series designation	External: Class	External: Allowance	External: Major diameter limits Max	External: Major diameter limits Min	External: Major diameter limits Min	External: Pitch diameter limits Max	External: Pitch diameter limits Min	External: Pitch diameter limits Tolerance	External: Minor diameter	Internal: Class	Internal: Minor diameter limits Min	Internal: Minor diameter limits Max	Internal: Pitch diameter limits Min	Internal: Pitch diameter limits Max	Internal: Pitch diameter limits Tolerance	Internal: Major diameter Min
			in.	in.	in.	in.	in.	in.	in.	in.		in.	in.	in.	in.	in.	in.
1-3/4—16	UNEF	2A	0.0016	1.7484	1.7390	....	1.7078	1.7025	0.0053	1.6717	2B	1.682	1.696	1.7094	1.7163	0.0069	1.7500
		3A	.0000	1.7500	1.7406	....	1.7094	1.7054	.0040	1.6733	3B	1.6820	1.6908	1.7094	1.7146	.0052	1.7500
1-13/16—16	N	2A	.0016	1.8109	1.8015	....	1.7703	1.7650	.0053	1.7342	2B	1.745	1.759	1.7719	1.7788	.0069	1.8125
		3A	.0000	1.8125	1.8031	....	1.7719	1.7679	.0040	1.7358	3B	1.7450	1.7533	1.7719	1.7771	.0052	1.8125
1-7/8—8	N	2A	.0023	1.8727	1.8577	1.8502	1.7915	1.7838	.0077	1.7193	2B	1.740	1.765	1.7938	1.8038	.0100	1.8750
		3A	.0000	1.8750	1.8600	....	1.7938	1.7881	.0057	1.7216	3B	1.7400	1.7547	1.7938	1.8013	.0075	1.8750
1-7/8—12	UN	2A	.0018	1.8732	1.8618	....	1.8191	1.8131	.0060	1.7710	2B	1.785	1.803	1.8209	1.8287	.0078	1.8750
		3A	.0000	1.8750	1.8636	....	1.8209	1.8164	.0045	1.7728	3B	1.7850	1.7948	1.8209	1.8267	.0058	1.8750
1-7/8—16	UN	2A	.0016	1.8734	1.8640	....	1.8328	1.8275	.0053	1.7967	2B	1.807	1.821	1.8344	1.8413	.0069	1.8750
		3A	.0000	1.8750	1.8656	....	1.8344	1.8304	.0040	1.7983	3B	1.8070	1.8158	1.8344	1.8396	.0052	1.8750
1-15/16—16	N	2A	.0016	1.9359	1.9265	....	1.8953	1.8899	.0054	1.8592	2B	1.870	1.884	1.8969	1.9039	.0070	1.9375
		3A	.0000	1.9375	1.9281	....	1.8969	1.8929	.0040	1.8608	3B	1.8700	1.8783	1.8969	1.9021	.0052	1.9375
2—4-1/2	UNC	1A	.0029	1.9971	1.9641	....	1.8528	1.8385	.0143	1.7245	1B	1.759	1.795	1.8557	1.8743	.0186	2.0000
		2A	.0029	1.9971	1.9751	1.9641	1.8528	1.8433	.0095	1.7245	2B	1.759	1.795	1.8557	1.8681	.0124	2.0000
		3A	.0000	2.0000	1.9780	....	1.8557	1.8486	.0071	1.7274	3B	1.7590	1.7861	1.8557	1.8650	.0093	2.0000
2—8	N	2A	.0023	1.9977	1.9827	1.9752	1.9165	1.9087	.0078	1.8443	2B	1.865	1.890	1.9188	1.9289	.0101	2.0000
		3A	.0000	2.0000	1.9850	....	1.9188	1.9130	.0058	1.8466	3B	1.8650	1.8797	1.9188	1.9264	.0076	2.0000
2—12	UN	2A	.0018	1.9982	1.9868	....	1.9441	1.9380	.0061	1.8960	2B	1.910	1.928	1.9459	1.9538	.0079	2.0000
		3A	.0000	2.0000	1.9886	....	1.9459	1.9414	.0045	1.8978	3B	1.9100	1.9198	1.9459	1.9518	.0059	2.0000
2—16	UNEF	2A	.0016	1.9984	1.9890	....	1.9578	1.9524	.0054	1.9217	2B	1.932	1.946	1.9594	1.9664	.0070	2.0000
		3A	.0000	2.0000	1.9906	....	1.9594	1.9554	.0040	1.9233	3B	1.9320	1.9408	1.9594	1.9646	.0052	2.0000
2-1/16—16	N	2A	.0016	2.0609	2.0515	....	2.0203	2.0149	.0054	1.9842	2B	1.995	2.009	2.0219	2.0289	.0070	2.0625
		3A	.0000	2.0625	2.0531	....	2.0219	2.0179	.0040	1.9858	3B	1.9950	2.0033	2.0219	2.0271	.0052	2.0625
2-1/8—8	N	2A	.0024	2.1226	2.1076	2.1001	2.0414	2.0335	.0079	1.9692	2B	1.990	2.015	2.0438	2.0540	.0102	2.1250
		3A	.0000	2.1250	2.1100	....	2.0438	2.0379	.0059	1.9716	3B	1.9900	2.0047	2.0438	2.0515	.0077	2.1250
2-1/8—12	UN	2A	.0018	2.1232	2.1118	....	2.0691	2.0630	.0061	2.0210	2B	2.035	2.053	2.0709	2.0788	.0079	2.1250
		3A	.0000	2.1250	2.1136	....	2.0709	2.0664	.0045	2.0228	3B	2.0350	2.0448	2.0709	2.0768	.0059	2.1250
2-1/8—16	UN	2A	.0016	2.1234	2.1140	....	2.0828	2.0774	.0054	2.0467	2B	2.057	2.071	2.0844	2.0914	.0070	2.1250
		3A	.0000	2.1250	2.1156	....	2.0844	2.0803	.0041	2.0483	3B	2.0570	2.0658	2.0844	2.0896	.0052	2.1250
2-3/16—16	N	2A	.0016	2.1859	2.1765	....	2.1453	2.1399	.0054	2.1092	2B	2.120	2.134	2.1469	2.1539	.0070	2.1875
		3A	.0000	2.1875	2.1781	....	2.1469	2.1428	.0041	2.1108	3B	2.1200	2.1283	2.1469	2.1521	.0052	2.1875
2-1/4—4-1/2	UNC	1A	.0029	2.2471	2.2141	....	2.1078	2.0882	.0146	1.9745	1B	2.009	2.045	2.1057	2.1247	.0190	2.2500
		2A	.0029	2.2471	2.2251	2.2141	2.1028	2.0931	.0097	1.9745	2B	2.009	2.045	2.1057	2.1183	.0126	2.2500
		3A	.0000	2.2500	2.2280	....	2.1057	2.0984	.0073	1.9774	3B	2.0090	2.0361	2.1057	2.1152	.0095	2.2500
2-1/4—8	N	2A	.0024	2.2476	2.2326	2.2251	2.1664	2.1584	.0080	2.0942	2B	2.115	2.140	2.1688	2.1792	.0104	2.2500
		3A	.0000	2.2500	2.2350	....	2.1688	2.1628	.0060	2.0966	3B	2.1150	2.1297	2.1688	2.1766	.0078	2.2500
2-1/4—12	UN	2A	.0018	2.2482	2.2368	....	2.1941	2.1880	.0061	2.1460	2B	2.160	2.178	2.1959	2.2038	.0079	2.2500
		3A	.0000	2.2500	2.2386	....	2.1959	2.1914	.0045	2.1478	3B	2.1600	2.1698	2.1959	2.2018	.0059	2.2500
2-1/4—16	UN	2A	.0016	2.2484	2.2390	....	2.2078	2.2024	.0054	2.1717	2B	2.182	2.196	2.2094	2.2164	.0070	2.2500
		3A	.0000	2.2500	2.2406	....	2.2094	2.2053	.0041	2.1733	3B	2.1820	2.1908	2.2094	2.2146	.0052	2.2500
2-5/16—16	N	2A	.0017	2.3108	2.3014	....	2.2702	2.2647	.0055	2.2341	2B	2.245	2.259	2.2719	2.2791	.0072	2.3125
		3A	.0000	2.3125	2.3031	....	2.2719	2.2678	.0041	2.2358	3B	2.2450	2.2533	2.2719	2.2773	.0054	2.3125

TABLE 7-10. Standard Series Limits of Self-Unified and American Screw Threads (cont.).

Nominal size and threads per inch	Series designation	External: Class	External: Allowance	External: Major diameter limits Max	External: Major diameter limits Min	External: Major diameter limits Min	External: Pitch diameter limits Max	External: Pitch diameter limits Min	External: Pitch diameter limits Tolerance	External: Minor diameter	Internal: Class	Internal: Minor diameter limits Min	Internal: Minor diameter limits Max	Internal: Pitch diameter limits Min	Internal: Pitch diameter limits Max	Internal: Pitch diameter limits Tolerance	Internal: Major diameter Min
			in.	in.	in.	in.	in.	in.	in.	in.		in.	in.	in.	in.	in.	in.
2-3/8—12	UN	2A	.0019	2.3731	2.3617	....	2.3190	2.3128	.0062	2.2709	2B	2.285	2.303	2.3209	2.3290	.0081	2.3750
		3A	.0000	2.3750	2.3636	....	2.3209	2.3163	.0046	2.2728	3B	2.2850	2.2948	2.2309	2.3269	.0060	2.3750
2-3/8—16	UN	2A	.0017	2.3733	2.3639	....	2.3327	2.3272	.0055	2.2966	2B	2.307	2.321	2.3344	2.3416	.0072	2.3750
		3A	.0000	2.3750	2.3656	....	2.3344	2.3303	.0041	2.2983	3B	2.3070	2.3158	2.3344	2.3398	.0054	2.3750
2-7/16—16	N	2A	.0017	2.4358	2.4264	....	2.3952	2.3897	.0055	2.3591	2B	2.370	2.384	2.3969	2.4041	.0072	2.4375
		3A	.0000	2.4375	2.4281	....	2.3969	2.3928	.0041	2.3608	3B	2.3700	2.3783	2.3969	2.4023	.0054	2.4375
2-1/2—4	UNC	1A	.0031	2.4969	2.4612	....	2.3345	2.3190	.0155	2.1902	1B	2.229	2.267	2.3376	2.3578	.0202	2.5000
		2A	.0031	2.4969	2.4731	2.4612	2.3345	2.3241	.0104	2.1902	2B	2.229	2.267	2.3376	2.3511	.0135	2.5000
		3A	.0000	2.5000	2.4762	....	2.3376	2.3298	.0078	2.1933	3B	2.2290	2.2594	2.3376	2.3477	.0101	2.5000
2-1/2—8	N	2A	.0024	2.4976	2.4826	2.4751	2.4164	2.4082	.0082	2.3442	2B	2.365	2.390	2.4188	2.4294	.0106	2.5000
		3A	.0000	2.5000	2.4850	....	2.4188	2.4127	.0061	2.3466	3B	2.3650	2.3797	2.4188	2.4268	.0080	2.5000
2-1/2—12	UN	2A	.0019	2.4981	2.4867	....	2.4440	2.4378	.0062	2.3959	2B	2.410	2.428	2.4459	2.4540	.0081	2.5000
		3A	.0000	2.5000	2.4886	....	2.4459	2.4413	.0046	2.3978	3B	2.4100	2.4198	2.4459	2.4519	.0060	2.5000
2-1/2—16	UN	2A	.0017	2.4983	2.4889	....	2.4577	2.4522	.0055	2.4216	2B	2.432	2.446	2.4594	2.4666	.0072	2.5000
		3A	.0000	2.5000	2.4906	....	2.4594	2.4553	.0041	2.4233	3B	2.4320	2.4408	2.4594	2.4648	.0054	2.5000
2-5/8—12	UN	2A	.0019	2.6231	2.6117	....	2.5690	2.5628	.0062	2.5209	2B	2.535	2.553	2.5709	2.5790	.0081	2.6250
		3A	.0000	2.6250	2.6136	....	2.5709	2.5663	.0046	2.5228	3B	2.5350	2.5448	2.5709	2.5769	.0060	2.6250
2-5/8—16	UN	2A	.0017	2.6233	2.6139	....	2.5827	2.5772	.0055	2.5466	2B	2.557	2.571	2.5844	2.5916	.0072	2.6250
		3A	.0000	2.6250	2.6156	....	2.5844	2.5803	.0041	2.5483	3B	2.5570	2.5658	2.5844	2.5898	.0054	2.6250
2-3/4—4	UNC	1A	.0032	2.7468	2.7111	....	2.5844	2.5686	.0158	2.4401	1B	2.479	2.517	2.5876	2.6082	.0206	2.7500
		2A	.0032	2.7468	2.7230	2.7111	2.5844	2.5739	.0105	2.4401	2B	2.479	2.517	2.5876	2.6013	.0137	2.7500
		3A	.0000	2.7500	2.7262	....	2.5876	2.5797	.0079	2.4433	3B	2.4790	2.5094	2.5876	2.5979	.0103	2.7500
2-3/4—8	N	2A	.0025	2.7475	2.7325	2.7250	2.6663	2.6580	.0083	2.5941	2B	2.615	2.540	2.6688	2.6796	.0108	2.7500
		3A	.0000	2.7500	2.7350	....	2.6688	2.6626	.0062	2.5966	3B	2.6150	2.6297	2.6688	2.6769	.0081	2.7500
2-3/4—12	UN	2A	.0019	2.7481	2.7367	....	2.6940	2.6878	.0062	2.6459	2B	2.660	2.678	2.6959	2.7040	.0081	2.7500
		3A	.0000	2.7500	2.7386	....	2.6959	2.6913	.0046	2.6478	3B	2.6600	2.6698	2.6959	2.7019	.0060	2.7500
2-3/4—16	UN	2A	.0017	2.7483	2.7389	....	2.7077	2.7022	.0055	2.6716	2B	2.682	2.696	2.7094	2.7166	.0072	2.7500
		3A	.0000	2.7500	2.7406	....	2.7094	2.7053	.0041	2.6733	3B	2.6820	2.6908	2.7094	2.7148	.0054	2.7500
2-7/8—12	UN	2A	.0019	2.8731	2.8617	....	2.8190	2.8127	.0063	2.7709	2B	2.785	2.803	2.8209	2.8291	.0082	2.8750
		3A	.0000	2.8750	2.8636	....	2.8209	2.8162	.0047	2.7728	3B	2.7850	2.7948	2.8209	2.8271	.0062	2.8750
2-7/8—16	UN	2A	.0017	2.8733	2.8639	....	2.8327	2.8271	.0056	2.7966	2B	2.807	2.821	2.8344	2.8417	.0073	2.8750
		3A	.0000	2.8750	2.8656		2.8344	2.8302	.0042	2.7983	3B	2.8070	2.8158	2.8344	2.8399	.0055	2.8750
3—4	UNC	1A	.0032	2.9968	2.9611	....	2.8344	2.8183	.0161	2.6901	1B	2.729	2.767	2.8376	2.8585	.0209	3.0000
		2A	.0032	2.9968	2.9730	2.9611	2.8344	2.8237	.0107	2.6901	2B	2.729	2.767	2.8376	2.8515	.0139	3.0000
		3A	.0000	3.0000	2.9762	....	2.8376	2.8296	.0080	2.6933	3B	2.7290	2.7594	2.8376	2.8480	.0104	3.0000
3—8	N	2A	.0026	2.9974	2.9824	2.9749	2.9162	2.9077	.0085	2.8440	2B	2.865	2.890	2.9188	2.9299	.0111	3.0000
		3A	.0000	3.0000	2.9850	....	2.9188	2.9124	.0064	2.8466	3B	2.8650	2.8797	2.9188	2.9271	.0083	3.0000
3—12	UN	2A	.0019	2.9981	2.9867	....	2.9440	2.9377	.0063	2.8959	2B	2.910	2.928	2.9459	2.9541	.0082	3.0000
		3A	.0000	3.0000	2.9886	....	2.9459	2.9412	.0047	2.8978	3B	2.9100	2.9198	2.9459	2.9521	.0062	3.0000
3—16	UN	2A	.0017	2.9983	2.9889	....	2.9577	2.9521	.0056	2.9216	2B	2.932	2.946	2.9594	2.9667	.0073	3.0000
		3A	.0000	3.0000	2.9906	....	2.9594	2.9552	.0042	2.9233	3B	2.9320	2.9408	2.9594	2.9649	.0055	3.0000

TABLE 7-10. Standard Series Limits of Unified and American Screw Threads (cont).

Nominal size and threads per inch	Series designation	External									Internal						
		Class	Allowance	Major diameter limits			Pitch diameter limits			Minor diameter	Class	Minor diameter limits		Pitch diameter limits			Major diameter
				Max	Min	Min	Max	Min	Tolerance			Min	Max	Min	Max	Tolerance	Min
			in.	in.	in.	in.	in.	in.	in.	in.		in.	in.	in.	in.	in.	in.
3-1/8—12	UN	2A	0.0019	3.1231	3.1117	....	3.0690	3.0627	0.0063	3.0209	2B	3.035	3.053	3.0709	3.0791	0.0082	3.1250
		3A	.0000	3.1250	3.1136	....	3.0709	3.0662	.0047	3.0228	3B	3.0350	3.0448	3.0709	3.0771	.0062	3.1250
3-1/8—16	UN	2A	.0017	3.1233	3.1139	....	3.0827	3.0771	.0056	3.0466	2B	3.057	3.071	3.0844	3.0917	.0073	3.1250
		3A	.0000	3.1250	3.1156	....	3.0844	3.0802	.0042	3.0483	3B	3.0570	3.0658	3.0844	3.0899	.0055	3.1250
3-1/4—4	UNC	1A	.0033	3.2467	3.2110	....	3.0843	3.0680	.0163	2.9400	1B	2.979	3.017	3.0876	3.1088	.0212	3.2500
		2A	.0033	3.2467	3.2229	3.2110	3.0843	3.0734	.0109	2.9400	2B	2.979	3.017	3.0876	3.1017	.0141	3.2500
		3A	.0000	3.2500	3.2262	....	3.0876	3.0794	.0082	2.9433	3B	2.9790	3.0094	3.0876	3.0982	.0106	3.2500
3-1/4—8	N	2A	.0026	3.2474	3.2324	3.2249	3.1662	3.1575	.0087	3.0940	2B	3.115	3.140	3.1688	3.1801	.0113	3.2500
		3A	.0000	3.2500	3.2350	....	3.1688	3.1623	.0065	3.0966	3B	3.1150	3.1297	3.1688	3.1772	.0084	3.2500
3-1/4—12	UN	2A	.0019	3.2481	3.2367	....	3.1940	3.1877	.0063	3.1459	2B	3.160	3.178	3.1959	3.2041	.0082	3.2500
		3A	.0000	3.2500	3.2386	....	3.1959	3.1912	.0047	3.1478	3B	3.1600	3.1698	3.1959	3.2021	.0062	3.2500
3-1/4—16	UN	2A	.0017	3.2483	3.2389	....	3.2077	3.2021	.0056	3.1716	2B	3.182	3.196	3.2094	3.2167	.0073	3.2500
		3A	.0000	3.2500	3.2406	....	3.2094	3.2052	.0042	3.1733	3B	3.1820	3.1908	3.2094	3.2149	.0055	3.2500
3-3/8—12	UN	2A	.0019	3.3731	3.3617	....	3.3190	3.3126	.0064	3.2709	2B	3.285	3.303	3.3209	3.3293	.0084	3.3750
		3A	.0000	3.3750	3.3636	....	3.3209	3.3161	.0048	3.2728	3B	3.2850	3.2948	3.3209	3.3272	.0063	3.3750
3-3/8—16	UN	2A	.0017	3.3733	3.3639	....	3.3327	3.3269	.0058	3.2966	2B	3.307	3.321	3.3344	3.3419	.0075	3.3750
		3A	.0000	3.3750	3.3656	....	3.3344	3.3301	.0043	3.2983	3B	3.3070	3.3158	3.3344	3.3400	.0056	3.3750
3-1/2—4	UNC	1A	.0033	3.4967	3.4610	....	3.3343	3.3177	.0166	3.1900	1B	3.229	3.267	3.3376	3.3591	.0215	3.5000
		2A	.0033	3.4967	3.4729	3.4610	3.3343	3.3233	.0110	3.1900	2B	3.229	3.267	3.3376	3.3519	.0143	3.5000
		3A	.0000	3.5000	3.4762	....	3.3376	3.3293	.0083	3.1933	3B	3.2290	3.2594	3.3376	3.3484	.0108	3.5000
3-1/2—8	N	2A	.0026	3.4974	3.4824	3.4749	3.4162	3.4074	.0088	3.3440	2B	3.365	3.390	3.4188	3.4303	.0115	3.5000
		3A	.0000	3.5000	3.4850	....	3.4188	3.4122	.0066	3.3466	3B	3.3650	3.3797	3.4188	3.4274	.0086	3.5000
3-1/2—12	UN	2A	.0019	3.4981	3.4867	....	3.4440	3.4376	.0064	3.3959	2B	3.410	3.428	3.4459	3.4543	.0084	3.5000
		3A	.0000	3.5000	3.4886	....	3.4459	3.4411	.0048	3.3978	3B	3.4100	3.4198	3.4459	3.4522	.0063	3.5000
3-1/2—16	UN	2A	.0017	3.4983	3.4889	....	3.4577	3.4519	.0058	3.4216	2B	3.432	3.446	3.4594	3.4669	.0075	3.5000
		3A	.0000	3.5000	3.4906	....	3.4594	3.4551	.0043	3.4233	3B	3.4320	3.4408	3.4594	3.4650	.0056	3.5000
3-5/8—12	UN	2A	.0019	3.6231	3.6117	....	3.5690	3.5626	.0064	3.5209	2B	3.535	3.553	3.5709	3.5793	.0084	3.6250
		3A	.0000	3.6250	3.6136	....	3.5709	3.5661	.0048	3.5228	3B	3.5350	3.5448	3.5709	3.5772	.0063	3.6250
3-5/8—16	UN	2A	.0017	3.6233	3.6139	....	3.5827	3.5769	.0058	3.5466	2B	3.557	3.571	3.5844	3.5919	.0075	3.6250
		3A	.0000	3.6250	3.6156	....	3.5844	3.5801	.0043	3.5483	3B	3.5570	3.5658	3.5844	3.5900	.0056	3.6250
3-3/4—4	UNC	1A	.0034	3.7466	3.7109	....	3.5842	3.5674	.0168	3.4399	1B	3.479	3.517	3.5876	3.6094	.0218	3.7500
		2A	.0034	3.7466	3.7228	3.7109	3.5842	3.5730	.0112	3.4399	2B	3.479	3.517	3.5876	3.6021	.0145	3.7500
		3A	.0000	3.7500	3.7262	....	3.5876	3.5792	.0084	3.4433	3B	3.4790	3.5094	3.5876	3.5985	.0109	3.7500
3-3/4—8	N	2A	.0027	3.7473	3.7323	3.7248	3.6661	3.6571	.0090	3.5939	2B	3.615	3.640	3.6688	3.6805	.0117	3.7500
		3A	.0000	3.7500	3.7350	....	3.6688	3.6621	.0067	3.5966	3B	3.6150	3.6297	3.6688	3.6776	.0088	3.7500
3-3/4—12	UN	2A	.0019	3.7481	3.7367	....	3.6940	3.6876	.0064	3.6459	2B	3.660	3.678	3.6959	3.7043	.0084	3.7500
		3A	.0000	3.7500	3.7386	....	3.6959	3.6911	.0048	3.6478	3B	3.6600	3.6698	3.6959	3.7022	.0063	3.7500
3-3/4—16	UN	2A	.0017	3.7483	3.7389	....	3.7077	3.7019	.0058	3.6716	2B	3.682	3.696	3.7094	3.7169	.0075	3.7500
		3A	.0000	3.7500	3.7406	....	3.7094	3.7051	.0043	3.6733	3B	3.6820	3.6908	3.7094	3.7150	.0056	3.7500
3-7/8—12	UN	2A	.0020	3.8730	3.8616	....	3.8189	3.8124	.0065	3.7708	2B	3.785	3.803	3.8209	3.8294	.0085	3.8750
		3A	.0000	3.8750	3.8636	....	3.8209	3.8160	.0049	3.7728	3B	3.7850	3.7948	3.8209	3.8273	.0064	3.8750
3-7/8—16	UN	2A	.0018	3.8732	3.8638	....	3.8326	3.8267	.0059	3.7965	2B	3.807	3.821	3.8344	3.8420	.0076	3.8750
		3A	.0000	3.8750	3.8656	....	3.8344	3.8300	.0044	3.7983	3B	3.8070	3.8158	3.8344	3.8401	.0057	3.8750
4—4	UNC	1A	.0034	3.9966	3.9609	....	3.8342	3.8172	.0170	3.6899	1B	3.729	3.767	3.8376	3.8597	.0221	4.0000
		2A	.0034	3.9966	3.9728	3.9609	3.8342	3.8229	.0113	3.6899	2B	3.729	3.767	3.8376	3.8523	.0147	4.0000
		3A	.0000	4.0000	3.9762	....	3.8376	3.8291	.0085	3.6933	3B	3.7290	3.7594	3.8376	3.8487	.0111	4.0000

TABLE 7-10. Standard Series Limits of Self-Unified and American Screw Threads (cont.).

Nominal size and threads per inch	Series designation	External									Internal						
		Class	Allowance	Major diameter limits			Pitch diameter limits			Minor diameter	Class	Minor diameter limits		Pitch diameter limits			Major diameter
				Max	Min	Min	Max	Min	Tolerance			Min	Max	Min	Max	Tolerance	Min
			in.	in.	in.	in.	in.	in.	in.	in.		in.	in.	in.	in.	in.	in.
4—8	N	2A	.0027	3.9973	3.9823	3.9748	3.9161	3.9070	.0091	3.8439	2B	3.865	3.890	3.9188	3.9307	.0119	4.0000
		3A	.0000	4.0000	3.9850	....	3.9188	3.9120	.0068	3.8466	3B	3.8650	3.8797	3.9188	3.9277	.0089	4.0000
4—12	UN	2A	.0020	3.9980	3.9866	....	3.9439	3.9374	.0065	3.8958	2B	3.910	3.928	3.9459	3.9544	.0085	4.0000
		3A	.0000	4.0000	3.9886	....	3.9459	3.9410	.0049	3.8978	3B	3.9100	3.9198	3.9459	3.9523	.0064	4.0000
4—16	UN	2A	.0018	3.9982	3.9888	....	3.9576	3.9517	.0059	3.9215	2B	3.932	3.946	3.9594	3.9670	.0076	4.0000
		3A	.0000	4.0000	3.9906	....	3.9594	3.9550	.0044	3.9233	3B	3.9320	3.9408	3.9594	3.9651	.0057	4.0000
4-1/4—8	N	2A	.0028	4.2472	4.2322	4.2247	4.1660	4.1567	.0093	4.0938	2B	4.115	4.140	4.1688	4.1809	.0121	4.2500
		3A	.0000	4.2500	4.2350	....	4.1688	4.1618	.0070	4.0966	3B	4.1150	4.1297	4.1688	4.1778	.0090	4.2500
4-1/4—12	UN	2A	.0020	4.2480	4.2366	....	4.1939	4.1874	.0065	4.1458	2B	4.160	4.178	4.1959	4.2044	.0085	4.2500
		3A	.0000	4.2500	4.2386	....	4.1959	4.1910	.0049	4.1478	3B	4.1600	4.1698	4.1959	4.2023	.0064	4.2500
4-1/4—16	UN	2A	.0018	4.2482	4.2388	....	4.2076	4.2017	.0059	4.1715	2B	4.182	4.196	4.2094	4.2170	.0076	4.2500
		3A	.0000	4.2500	4.2406	....	4.2094	4.2050	.0044	4.1733	3B	4.1820	4.1908	4.2094	4.2151	.0057	4.2500
4-1/2—8	N	2A	.0028	4.4972	4.4822	4.4747	4.4160	4.4066	.0094	4.3438	2B	4.365	4.390	4.4188	4.4310	.0122	4.5000
		3A	.0000	4.5000	4.4850	....	4.4188	4.4117	.0071	4.3466	3B	4.3650	4.3797	4.4188	4.4280	.0092	4.5000
4-1/2—12	UN	2A	.0020	4.4980	4.4866	....	4.4439	4.4374	.0065	4.3958	2B	4.410	4.428	4.4459	4.4544	.0085	4.5000
		3A	.0000	4.5000	4.4886	....	4.4459	4.4410	.0049	4.3978	3B	4.4100	4.4198	4.4459	4.4523	.0064	4.5000
4-1/2—16	UN	2A	.0018	4.4982	4.4888	....	4.4576	4.4517	.0059	4.4215	2B	4.432	4.446	4.4594	4.4670	.0076	4.5000
		3A	.0000	4.5000	4.4906	....	4.4594	4.550	.0044	4.4233	3B	4.4320	4.4408	4.4594	4.4651	.0057	4.5000
4-3/4—8	N	2A	.0029	4.7471	4.7321	4.7246	4.6659	4.6564	.0095	4.5937	2B	4.615	4.640	4.6688	4.6812	.0124	4.7500
		3A	.0000	4.7500	4.7350	....	4.6688	4.6616	.0072	4.5966	3B	4.6150	4.6297	4.6688	4.6781	.0093	4.7500
4-3/4—12	UN	2A	.0020	4.7480	4.7366	....	4.6939	4.6872	.0067	4.6458	2B	4.660	4.678	4.6959	4.7046	.0087	4.7500
		3A	.0000	4.7500	4.7386	....	4.6959	4.6909	.0050	4.6478	3B	4.6600	4.6698	4.6959	4.7025	.0066	4.7500
4-3/4—16	UN	2A	.0018	4.7482	4.7388	....	4.7076	4.7015	.0061	4.6715	2B	4.682	4.696	4.7094	4.7133	.0079	4.7500
		3A	.0000	4.7500	4.7406	....	4.7094	4.7049	.0045	4.6733	3B	4.6820	4.6908	4.7094	4.7153	.0059	4.7500
5—8	N	2A	.0029	4.9971	4.9821	4.9746	4.9159	4.9062	.0097	4.8137	2B	4.865	4.890	4.9188	4.9314	.0126	5.0000
		3A	.0000	5.0000	4.9850	....	4.9188	4.9116	.0072	4.8466	3B	4.8650	4.8797	4.9188	4.9282	.0094	5.0000
5—12	UN	2A	.0020	4.9980	4.9866	....	4.9439	4.9372	.0067	4.8958	2B	4.910	4.928	4.9459	4.9546	.0087	5.0000
		3A	.0000	5.0000	4.9886	....	4.9459	4.9409	.0050	4.8978	3B	4.9100	4.9198	4.9459	4.9525	.0066	5.0000
5—16	UN	2A	.0018	4.9982	4.9888	....	4.9576	4.9515	.0061	4.9215	2B	4.932	4.946	4.9594	4.9673	.0079	5.0000
		3A	.0000	5.0000	4.9906	....	4.9594	4.9549	.0045	4.9233	3B	4.9320	4.9408	4.9594	4.9653	.0059	5.0000
5-1/4—8	N	2A	.0029	5.2471	5.2321	5.2246	5.1659	5.1561	.0098	5.0937	2B	5.115	5.140	5.1688	5.1815	.0127	5.2500
		3A	.0000	5.2500	5.2350	....	5.1688	5.1615	.0073	5.0966	3B	5.1150	5.1297	5.1688	5.1783	.0095	5.2500
5-1/4—12	UN	2A	.0020	5.2480	5.2366	....	5.1939	5.1872	.0067	5.1458	2B	5.160	5.178	5.1959	5.2046	.0087	5.2500
		3A	.0000	5.2500	5.2386	....	5.1959	5.1909	.0050	5.1478	3B	5.1600	5.1698	5.1959	5.2025	.0066	5.2500

TABLE 7-10. Standard Series Limits of Self-Unified and American Screw Threads (cont.).

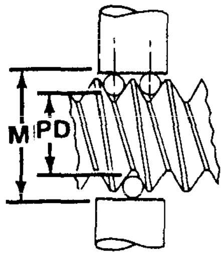
Nominal size and threads per inch	Series designation	Class	External: Allowance	External: Major diameter limits Max	External: Major diameter limits Min	External: Major diameter limits Min	External: Pitch diameter limits Max	External: Pitch diameter limits Min	External: Pitch diameter limits Tolerance	External: Minor diameter	Class	Internal: Minor diameter limits Min	Internal: Minor diameter limits Max	Internal: Pitch diameter limits Min	Internal: Pitch diameter limits Max	Internal: Pitch diameter limits Tolerance	Internal: Major diameter Min
			in.	in.	in.	in.	in.	in.	in.	in.		in.	in.	in.	in.	in.	in.
5-1/4—16	UN	2A	0.0018	5.2482	5.2388	....	5.2076	5.2015	0.0061	5.1715	2B	5.182	5.196	5.2094	5.2173	0.0079	5.2500
		3A	.0000	5.2500	5.2406	....	5.2094	5.2049	.0045	5.1733	3B	5.1820	5.1908	5.2094	5.2153	.0059	5.2500
5-1/2—8	N	2A	.0030	5.4970	5.4820	5.4745	5.4158	5.4059	.0099	5.3436	2B	5.365	5.390	5.4188	5.4317	.0129	5.5000
		3A	.0000	5.5000	5.4850	....	5.4188	5.4114	.0074	5.3466	3B	5.3650	5.3797	5.4188	5.4285	.0097	5.5000
5-1/2—12	UN	2A	.0020	5.4980	5.4866	....	5.4439	5.4372	.0067	5.3958	2B	5.410	5.428	5.4459	5.4546	.0087	5.5000
		3A	.0000	5.5000	5.4886	....	5.4459	5.4409	.0050	5.3978	3B	5.4100	5.4198	5.4459	5.4525	.0066	5.5000
5-1/2—16	UN	2A	.0018	5.4982	5.4888	....	5.4576	5.4515	.0061	5.4215	2B	5.432	5.446	5.4594	5.4673	.0079	5.5000
		3A	.0000	5.5000	5.4906	....	5.4594	5.4549	.0045	5.4233	3B	5.4320	5.4408	5.4594	5.4653	.0059	5.5000
5-3/4—8	N	2A	.0030	5.7470	5.7320	5.7245	5.6658	5.6558	.0100	5.5936	2B	5.615	5.640	5.6688	5.6818	.0130	5.7500
		3A	.0000	5.7500	5.7350	....	5.6688	5.6613	.0075	5.5966	3B	5.6150	5.6297	5.6688	5.6786	.0098	5.7500
5-3/4—12	UN	2A	.0021	5.7479	5.7365	....	5.6938	5.6869	.0069	5.6457	2B	5.660	5.678	5.6959	5.7049	.0090	5.7500
		3A	.0000	5.7500	5.7386	....	5.6959	5.6907	.0052	5.6478	3B	5.6600	5.6698	5.6959	5.7026	.0067	5.7500
5-3/4—16	UN	2A	.0019	5.7481	5.7387	....	5.7075	5.7013	.0062	5.6714	2B	5.682	5.696	5.7094	5.7175	.0081	5.7500
		3A	.0000	5.7500	5.7406	....	5.7094	5.7047	.0047	5.6733	3B	5.6820	5.6908	5.7094	5.7155	.0061	5.7500
6—8	N	2A	.0030	5.9970	5.9820	5.9745	5.9158	5.9056	.0102	5.8436	2B	5.865	5.890	5.9188	5.9320	.0132	6.0000
		3A	.0000	6.0000	5.9850	....	5.9188	5.9112	.0076	5.8466	3B	5.8650	5.8797	5.9188	5.9287	.0099	6.0000
6—12	UN	2A	.0021	5.9979	5.9865	....	5.9438	5.9369	.0069	5.8957	2B	5.910	5.928	5.9459	5.9549	.0090	6.0000
		3A	.0000	6.0000	5.9886	....	5.9459	5.9407	.0052	5.8978	3B	5.9100	5.9198	5.9459	5.9526	.0067	6.0000
6—16	UN	2A	.0019	5.9981	5.9887	....	5.9575	5.9513	.0062	5.9214	2B	5.932	5.946	5.9594	5.9675	.0081	6.0000
		3A	.0000	6.0000	5.9906	....	5.9594	5.9547	.0047	5.9233	3B	5.9320	5.9408	5.9594	5.9655	.0061	6.0000

TABLE 7-11. Three Wire Measurement for Metric Threads.

THREE WIRE THREAD MEASUREMENT
(60° Metric Thread)

M = PD + C   PD = M − C

M = Measurement over wires
PD = Pitch diameter
C = Constant



PITCH		BEST WIRE SIZE		CONSTANT	
MM	INCHES	MM	INCHES	MM	INCHES
0.35	.01378	0.2021	.00796	0.3031	.01193
0.4	.01575	0.2309	.00909	0.3464	.01364
0.45	.01772	0.2598	.01023	0.3897	.01534
0.5	.01969	0.2887	.01137	0.4330	.01705
0.6	.02362	0.3464	.01364	0.5196	.02046
0.7	.02756	0.4041	.01591	0.6062	.02387
0.8	.03150	0.4619	.01818	0.6928	.02728
1.0	.03937	0.5774	.02273	0.8660	.03410
1.25	.04921	0.7217	.02841	1.0815	.04262
1.5	.05906	0.8660	.03410	1.2990	.05114
1.75	.06890	1.0104	.03978	1.5155	.05967
2.0	.07874	1.1547	.04546	1.7321	.06819
2.5	.09843	1.4434	.05683	2.1651	.08524
3.0	.11811	1.7321	.06819	2.5981	.10229
3.5	.13780	2.0207	.07956	3.0311	.11933
4.0	.15748	2.3094	.09092	3.4641	.13638
4.5	.17717	2.5981	.10229	3.8971	.15343
5.0	.19685	2.8868	.11365	4.3301	.17048
5.5	.21654	3.1754	.12502	4.7631	.18753
6.0	.23622	3.4641	.13638	5.1962	.20457

TABLE 8-1. Milling Machine Cutting Speeds for High-Speed Steel Milling Cutters.

MATERIAL	CUTTING SPEED (sfpm)[1, 2]			
	PLAIN MILLING CUTTERS		END MILLING CUTTERS	
	Roughing	Finishing	Roughing	Finishing
Aluminum	400 to 1,000	400 to 1,000	400 to 1,000	400 to 1,000
Brass, composition	125 to 200	90 to 200	90 to 150	90 to 150
Brass, yellow	150 to 200	100 to 250	100 to 200	100 to 200
Bronze, phosphor and manganese	30 to 80	25 to 100	30 to 80	30 to 80
Cast iron (hard)	25 to 40	10 to 30	25 to 40	20 to 45
Cast iron (soft and medium)	40 to 75	25 to 80	35 to 65	30 to 80
Monel metal	50 to 75	50 to 75	40 to 60	40 to 60
Steel, hard	25 to 50	25 to 70	25 to 50	25 to 70
Steel, soft	60 to 120	45 to 110	50 to 85	45 to 100

1 For carbon steel cutters, decrease values by 50 percent.   2 For carbide-tipped cutters, increase values by 100 percent.

TABLE 8-2. Milling Cutter Rotational Speeds.

dia. of cutter (in.)	CUTTING SPEED (sfpm) 25	30	35	40	50	60	70	80	90	100	120	140	160	200
	CUTTER REVOLUTIONS PER MINUTE													
1/4	382	458	535	611	764	917	1,070	1,222	1,376	1,528	1,834	2,139	2,445	3,056
5/16	306	367	428	489	611	733	856	978	1,100	1,222	1,466	1,711	1,955	2,444
3/8	255	306	357	408	509	611	713	815	916	1,018	1,222	1,425	1,629	2,036
7/16	218	262	306	349	437	524	611	699	786	874	1,049	1,224	1,398	1,748
1/2	191	229	268	306	382	459	535	611	688	764	917	1,070	1,222	1,528
5/8	153	184	214	245	306	367	428	489	552	612	736	857	979	1,224
3/4	127	153	178	203	254	306	357	408	458	508	610	711	813	1,016
7/8	109	131	153	175	219	262	306	349	392	438	526	613	701	876
1	95.5	115	134	153	191	229	267	306	344	382	458	535	611	764
1 1/4	76.3	91.8	107	123	153	183	214	245	274	306	367	428	490	612
1 1/2	63.7	76.3	89.2	102	127	153	178	204	230	254	305	356	406	508
1 3/4	54.5	65.5	76.4	87.3	109	131	153	175	196	218	262	305	349	438
2	47.8	57.3	66.9	76.4	95.5	115	134	153	172	191	229	267	306	382
2 1/2	38.2	45.8	53.5	61.2	76.3	91.7	107	122	138	153	184	213	245	306
3	31.8	38.2	44.6	51	63.7	76.4	89.1	102	114	127	152	178	203	254
3 1/2	27.3	32.7	38.2	43.6	54.5	65.5	76.4	87.4	98.1	109	131	153	174	218
4	23.9	28.7	33.4	38.2	47.8	57.3	66.9	76.4	86	95.6	115	134	153	191
5	19.1	22.9	26.7	30.6	38.2	45.9	53.5	61.1	68.8	76.4	91.7	107	122	153
6	15.9	19.1	22.3	25.5	31.8	38.2	44.6	51.0	57.2	63.6	76.3	89	102	127
7	13.6	16.4	19.1	21.8	27.3	32.7	38.2	43.7	49.1	54.6	65.5	76.4	87.4	109
8	11.9	14.3	16.7	19.1	23.9	28.7	33.4	38.2	43	47.8	57.4	66.9	76.5	95.6

TABLE 8-3. Chip Sizes Per Tooth for Various Milling Cutters.

TYPE OF CUTTER	ALUMINUM HSS	ALUMINUM CARBIDE	BRONZE HSS	BRONZE CARBIDE	CAST IRON HSS	CAST IRON CARBIDE	FREE MACHINING STEEL HSS	FREE MACHINING STEEL CARBIDE	ALLOY STEEL HSS	ALLOY STEEL CARBIDE
FACE MILLS	.007 to .022	.007 to .020	.005 to .014	.004 to .012	.004 to .016	.006 to .020	.003 to .012	.004 to .016	.002 to .008	.003 to .014
HELICAL MILLS	.006 to .018	.006 to .016	.003 to .011	.004 to .010	.004 to .018	.002 to .018	.002 to .010	.003 to .013	.002 to .007	.003 to .012
SIDE CUTTING MILLS	.004 to .013	.004 to .012	.003 to .008	.003 to .007	.002 to .009	.003 to .012	.002 to .007	.003 to .009	.001 to .005	.002 to .008
END MILLS	.003 to .011	.003 to .010	.003 to .007	.002 to .006	.002 to .008	.003 to .010	.001 to .006	.002 to .008	.001 to .004	.002 to .007
FORM RELIEVED CUTTERS	.002 to .007	.002 to .006	.001 to .004	.001 to .004	.002 to .005	.002 to .006	.001 to .004	.002 to .005	.001 to .003	.001 to .004
CIRCULAR SAWS	.002 to .005	.002 to .005	.001 to .003	.001 to .003	.001 to .004	.002 to .006	.001 to .003	.001 to .004	.005 to .002	.001 to .004

TABLE 8-4. Sizes of Woodruff Keys.

KEY NUMBER	KEY DIMENSIONS		SHAFT DIAMETER		HEIGHT OF KEY ABOVE SHAFT	DEPTH OF KEYWAY
	DIAMETER	WIDTH	MINIMUM	MAXIMUM		
204	1/2	1/16	5/16	3/8	0.0312	0.1718
304	1/2	3/32	7/16	1/2	0.0469	0.1561
404	1/2	1/8	9/16	3/4	0.0625	0.1405
405	5/8	1/8	9/16	3/4	0.0625	0.1875
505	5/8	5/32	13/16	15/16	0.0781	0.1719
406	3/4	1/8	11/16	3/4	0.0625	0.2505
606	3/4	3/16	1	1 1/8	0.0937	0.2193
507	7/8	5/32	7/8	15/16	0.0781	0.2969
807	7/8	1/4	15/16	1 1/8	0.1250	0.2500
608	1	3/16	1	1 7/16	0.0937	0.3443
1008	1	5/16	1 1/16	1 5/8	0.1562	0.2818
609	1 1/8	3/16	1 1/16	1 7/16	0.0937	0.3903
810	1 1/4	1/4	1 1/4	1 3/4	0.1250	0.4220
1210	1 1/4	3/8	1 1/2	1 7/8	0.1875	0.3595
1011	1 3/8	5/10	1 13/16	2	0.1562	0.4378

TABLE 8-5. Dimensions of Square-Ends Machine Keys.

SHAFT DIAMETER (in.)	SQUARE SECTION KEYS		FLAT SECTION KEYS		
	Width and thickness (in.)	Bottom of keyway to opposite side of shaft (in.)	Width (in.)	Thickness (in.)	Bottom of keyway to opposite side of shaft (in.)
1/2	1/8	0.430	1/8	3/32	0.455
9/16	1/8	0.493	1/8	3/32	0.509
5/8	3/16	0.517	3/16	1/8	0.548
11/16	3/16	0.581	3/16	1/8	0.612
3/4	3/16	0.644	3/16	1/8	0.676
13/16	3/16	C.708	3/16	1/8	0.739
7/8	3/16	0.771	3/16	1/8	0.802
15/16	1/4	0.796	1/4	3/16	0.827
1	'/4	0.859	1/4	3/16	0.890
1 1/16	1/4	0.923	1/4	3/16	0.954
1 1/8	1/4	0.986	1/4	3/16	1.017
1 3/16	1/4	1.049	1/4	3/16	1.081
1 1/4	1/4	1.112	1/4	3/16	1.144
1 5/16	5/16	1.137	5/16	1/4	1.169
1 3/8	5/16	1.201	5/16	1/4	1.232
1 7/16	3/8	1.225	3/8	1/4	1.288
1 1/2	3/8	1.289	3/8	1/4	1.351
1 9/16	3/8	1.352	3/8	1/4	1.415
1 5/8	3/8	1.416	3/8	1/4	1.478
1 11/16	3/8	1.479	3/8	1/4	1.542
1 3/4	3/8	1.542	3/8	1/4	1.605
1 13/16	1/2	1.527	1/2	3/8	1.590
1 7/8	1/2	1.591	1/2	3/8	1.654
1 15/16	1/2	1.655	1/2	3/8	1.717
2	1/2	1.718	1/2	3/8	1.781

TABLE 8-6. Dimensions of T-Slots.

T-BOLT SIZE (DIAMETER IN in.)	THROAT WIDTH (in.)	THROAT DEPTH (in.)		HEAD SPACE (in.)	
		MAX.	MIN.	WIDTH	DEPTH
1/4	9/32	3/8	1/8	9/16	15/64
5/16	11/32	7/16	5/32	21/32	17/64
3/8	7/16	9/16	7/32	25/32	31/64
1/2	9/16	11/16	5/16	31/32	35/64
3/8	11/16	7/8	7/16	1 1/4	31/64
3/4	1/16	1 1/16	9/16	1 15/32	1 3/32

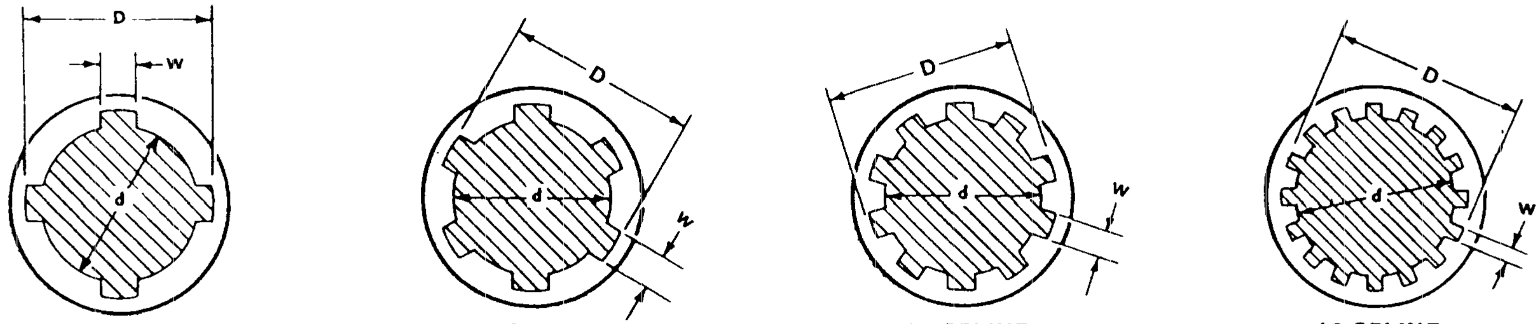
TABLE 8-7. Series of Involute Milling Cutters for each Pitch.

NUMBER OF CUTTER	WILL CUT GEAR FROM:	NUMBER OF CUTTER	WILL CUT GEAR FROM:
1	135 teeth to a rack	5	21 to 25 teeth
2	55 to 134 teeth	6	17 to 20 teeth
3	35 to 54 teeth	7	14 to 16 teeth
4	26 to 34 teeth	8	12 to 13 teeth

NOTE: THE REGULAR CUTTERS LISTED ABOVE ARE USED ORDINARILY. THE CUTTERS LISTED BELOW (AN INTERMEDIATE SERIES HAVING HALF-NUMBERS) MAY BE USED WHEN GREATER ACCURACY OF TOOTH SPACE IS ESSENTIAL IN CASES WHERE THE NUMBER OF TEETH ARE BETWEEN THE NUMBER FOR WHICH THE REGULAR CUTTERS ARE INTENDED.

NUMBER OF CUTTER	WILL CUT GEAR FROM:	NUMBER OF CUTTER	WILL CUT GEAR FROM:
1-1/2	80 to 134 teeth	5-1/2	19 to 20 teeth
2-1/2	42 to 54 teeth	6-1/2	15 to 16 teeth
3-1/2	30 to 34 teeth	7-1/2	13 teeth
4-1/2	23 to 25 teeth		

TABLE 8-8. Standard Spline Dimensions.



4-SPLINE 6-SPLINE 10-SPLINE 16-SPLINE

spline shaft	Width of spline (all fits)*				Permanent fit ** Minor diameter				Sliding fit when not under load** Minor diameter				Sliding fit when under load** Minor diameter		
	4-Spline (0.241 D)	6-Spline (0.250 D)	10-Spline (0.156 D)	16-Spline (0.098 D)	4-Spline (0.850 D)	6-Spline (0.900 D)	10-Spline (0.910 D)	16-Spline (0.910 D)	4-Spline (0.750 D)	6-Spline (0.850 D)	10-Spline (0.860 D)	16-Spline (0.860 D)	6-Spline (0.800 D)	10-Spline (0.810 D)	16-Spline (0.810 D)
¾ (.750)	0.181	0.188	0.117	....	0.637	0.675	0.683	....	0.562	0.638	0.645	....	0.600	0.608	....
⅞ (.875)	0.211	0.219	0.137	....	0.744	0.788	0.796	....	0.656	0.744	0.753	....	0.700	0.709	....
1 (1.000)	0.241	0.250	0.156	....	0.850	0.900	0.910	....	0.750	0.850	0.860	....	0.800	0.810	....
1⅛ (1.125)	0.271	0.281	0.176	....	0.956	1.013	1.024	....	0.844	0.956	0.968	....	0.900	0.911	....
1¼ (1.250)	0.301	0.313	0.195	....	1.062	1.125	1.138	....	0.937	1.063	1.075	....	1.000	1.013	....
1⅜ (1.375)	0.331	0.344	0.215	....	1.169	1.238	1.251	....	1.031	1.169	1.183	....	1.100	1.114	....
1½ (1.500)	0.361	0.375	0.234	....	1.275	1.350	1.365	....	1.125	1.275	1.290	....	1.200	1.215	....
1⅝ (1.625)	0.391	0.406	0.254	....	1.381	1.463	1.479	....	1.219	1.381	1.398	....	1.300	1.316	....
1¾ (1.750)	0.422	0.438	0.273	....	1.487	1.575	1.593	....	1.312	1.488	1.505	....	1.400	1.418	....
2 (2.000)	0.482	0.500	0.312	0.196	1.700	1.800	1.820	1.820	1.500	1.700	1.720	1.720	1.600	1.620	1.020
2¼ (2.250)	0.542	0.563	0.351	....	1.912	2.025	2.043	....	1.687	1.913	1.935	....	1.800	1.823	....
2½ (2.500)	0.602	0.625	0.390	0.245	2.125	2.250	2.275	2.275	1.875	2.125	2.150	2.150	2.000	2.025	2.025
3 (3.000)	0.723	0.750	0.468	0.294	2.550	2.700	2.730	2.730	2.250	2.550	2.580	2.580	2.400	2.430	2.430
3½ (3.500)	....	....	0.546	0.343	....	....	3.185	3.185	....	....	3.010	3.010	....	2.835	2.835
4 (4.000)	....	....	0.624	0.392	....	....	3.640	3.640	....	....	3.440	3.440	....	3.240	3.240
4½ (4.500)	....	....	0.702	0.441	....	....	4.095	4.095	....	....	3.870	3.870	....	3.645	3.645
5 (5.000)	....	....	0.780	0.490	....	....	4.550	4.550	....	....	4.300	4.300	....	4.050	4.050
5½ (5.500)	....	....	0.858	0.539	....	....	5.005	5.005	....	....	4.730	4.730	....	4.455	4.455
6 (6.000)	....	....	0.936	0.588	....	....	5.460	5.460	....	....	5.160	5.160	....	4.860	4.860

*Tolerance allowed is - 0.002 inch for shafts ¾ to 1¾ inches in diameter inclusive, and - 0.003 inch for larger sizes.

**Tolerance allowed is - 0.001 inch for shafts ¾ to 1¾ inches in diameter inclusive, - 0.002 inch for shafts 2 to 3 inches in diameter inclusive, and - 0.003 inch for larger sizes.

TABLE 9-1. Versa-Mil Cutting Speeds.

1. To determine the cutting speed of the material:
   a. Locate the column for the operation to be performed.
   b. Determine the type of cutter being used.
   c. Follow down the column to the type of material being used.
   d. Select the desired cutting speed from the chart.
   (lower speeds are for roughing while higher speeds are for finishing)

2. After the cutting speed has been selected from the chart, select the pulley ratio from the ratio chart.
   a. Locate the column the speed selected (determined from cutting speed chart) is located in.
   b. Follow the column down to the diameter closest to the cutter being used.
   c. Select the pulley ratio to ber used.
   d. Ratio selected will determine the head to use.

MATERIAL	END MILLING		MILLING SHELL/SIDE/FORM			DRILLING	FLY CUTTING	
	HSS	CARBIDE	HSS	CARBIDE	HSS	CARBIDE	HSS	CARBIDE
ALUMINUM	400-1000	600 & UP	400-1000	600 & UP	200-300	300-400	200-300	400-600
BRASS	100-200	150-300	150-200	200-500	200-300	300-450	150-200	300-400
BRONZE	30-80	45-120	25-100	50-200	200-300	300-400	30-100	60-200
CAST IRON	30-80	45-120	25-80	500-160	100-150	150-225	50-80	100-160
COPPER	60-80	90-120	125-175	250-350	60-70	90-105	60-80	120-160
MACHINERY STEEL	60-80	90-120	25-150	50-100	80-100	120-150	60-80	120-160
STEEL (HARD)	25-70	40-105	25-70	50-140	20-30	30-45	35-40	70-80
STEEL (SOFT)	45-100	70-150	45-110	90-220	50-60	75-90	80-100	160-200
STEEL STAINLESS	20-40	30-60	35-105	70-210	30-40	45-60	40-50	80-100
TOOL STEEL	40-60	60-90	70-105	140-210	50-60	75-90	35-40	70-80

TABLE 9-2. Versa-Mil Pulley Combinations.

DIAM. OF CUTTER	20-40 PULLEY RATIO	40-60 PULLEY RATIO	60-100 PULLEY RATIO	100-140 PULLEY RATIO	140-180 PULLEY RATIO	180-230 PULLEY RATIO	230-280 PULLEY RATIO	280-330 PULLEY RATIO	330-400 PULLEY RATIO	400-600 PULLEY RATIO	600-1000 PULLEY RATIO	HEAD
6"	*	*	(2:6) (2:5)	(3:6) (3:5)	(3:4) (4:5)	(4:4)	(5:4)	(4:3)	(5:3)	(6:3) (5:2)	(6:2)	BASIC UNIT/UNIVERSAL HEAD
	*	*	*	*	*	*	*	*	*	*	(2:6)	HIGH SPEED HEAD
5"	*	(2:6)	(2:5) (3:6)	(3:5) (3:4)	(4:5)	(4:4)	(5:4) (4:3)	(5:3)	(6:3)	(5:2) (6:2)	*	BASIC UNIT/UNIVERSAL HEAD
	*	*	*	*	*	*	*	*	*	*	(2:6) (2:5)	HIGH SPEED HEAD
4"	*	(2:6) (2:5)	(3:6) (3:5)	(3:4) (4:5)	(4.4)	(5:4) (4:3)	(5:3)	(6:3)	(5:2)	(6:2)	*	BASIC UNIT/UNIVERSAL HEAD
	*	*	*	*	*	*	*	*	*	(2:6)	(2:5) (3:6)	HIGH SPEED HEAD
3"	(2:6)	(2:5) (3:6)	(3:5) (3:4) (4:5)	(4:4) (5:4)	(4:3)	(5:3) (6:3)	*	(5:2)	(6:2)	*	*	BASIC UNIT/UNIVERSAL HEAD
	*	*	*	*	*	*	*	*	*	(2:6) (2:5)	(3:6) (3:5)	HIGH SPEED HEAD
2½"	(2:6) (2:5)	(3:6) (3:5)	(3:4) (4:5) (4:4)	(5:4) (4:3)	(5:3)	(6:3)	(5:2)	(6:2)	*	*	*	BASIC UNIT/UNIVERSAL HEAD
	*	*	*	*	*	*	*	*	(2:6)	(2:5) (3:6)	(3:5) (3:4) (4:5)	HIGH SPEED HEAD
2"	(2:6) (2:5) (3:6)	(3:5) (3:4) (4:5)	(4:4) (5:4) (4:3)	(5:3)	(6:3)	(5:2)	(6:2)	*	*	*	*	BASIC UNIT/UNIVERSAL HEAD
	*	*	*	*	*	*	*	(2:6)	(2:5)	(3:6) (3:5)	(3:4) (4:5) (4:4)	HIGH SPEED HEAD
1½"	(2:5) (3:6) (3:5)	(3:4) (4:5) (4:4)	(5:4) (4:3) (5:3)	(6:3)	(5:2) (6:2)	*	*	*	*	*	*	BASIC UNIT/UNIVERSAL HEAD
	*	*	*	*	*	(2:6)	(2:5)	*	(3:6) (3:5)	(3:4) (4:5)	(4:4) (5:4) (4:3)	HIGH SPEED HEAD

TABLE 9-2. Versa-Mil Pulley Combinations (cont.).

DIAM. OF CUTTER	20-40 PULLEY RATIO	40-60 PULLEY RATIO	60-100 PULLEY RATIO	100-140 PULLEY RATIO	140-180 PULLEY RATIO	180-230 PULLEY RATIO	230-280 PULLEY RATIO	280-330 PULLEY RATIO	330-400 PULLEY RATIO	400-600 PULLEY RATIO	600-1000 PULLEY RATIO	HEAD
1¼"	(3:6) (3:5) (3:4) (4:5)	(4:4) (5:4)	(4:3) (5:3) (6:3)	(5:2)	(6:2)	*	*	*	*	*	*	BASIC UNIT/UNIVERSAL HEAD
	*	*	*	*	(2:6)	(2:5)	*	(3:6)	(3:5)	(3:4) (4:5) (4:4)	(5:4) (4:3) (5:3)	HIGH SPEED HEAD
1"	(3:5) (3:4) (4:5) (4:4)	(5:4) (4:3)	(5:3) (6:3) (5:2)	(6:2)	*	*	*	*	*	*	*	BASIC UNIT/UNIVERSAL HEAD
	*	*	*	*	(2:6) (2:5)	(3:6)	(3:5)	*	(3:5) (4:5)	(4:4) (5:4)	(4:3) (5:3) (6:3)	HIGH SPEED HEAD
¾"	(4:5) (4:4) (5:4) (4:3)	(5:3) (6:3)	(5:2) (6:2)	*	*	*	*	*	*	*	*	BASIC UNIT/UNIVERSAL HEAD
	*	*	*	(2:6) (2:5)	(3:6)	(3:5)	(3:4) (4:5)	*	(4:4)	(5:4) (4:3) (5:3)	(6:3) (5:2)	HIGH SPEED HEAD
⅝"	(4:4) (5:4) (4:3) (5:3)	(6:3)	(5:2) (6:2)	*	*	*	*	*	*	*	*	BASIC UNIT/UNIVERSAL HEAD
	*	*	(2:6)	(2:5)	(3:5)	(3:4) (4:5)	*	(4:4)	(5:4) (4:3)	(5:3) (6:3)	(5:2) (6:2)	HIGH SPEED HEAD
½"	(5:4) (4:3) (5:3) (6:3)	(5:2) (6:2)	*	*	*	*	*	*	*	*	*	BASIC UNIT/UNIVERSAL HEAD
	*		(2:6) (2:5)	(3:5)	(3:4)	(4:5)	(4:4)	(5:4) (4:3)	(5:3)	(6:3)	(5:2) (6:2)	HIGH SPEED HEAD
⅜"	(5:3) (6:3) (5:2)	(6:2)	*	*	*	*	*	*	*	*	*	BASIC UNIT/UNIVERSAL HEAD
	*	(2:6)	(2:5)	(3:5) (3:4) (4:5)	(4:4)	(5:4) (4:3)		(5:3)	(6:3)	(5:2) (6:2)	*	HIGH SPEED HEAD

TABLE 9-2. Versa-Mil Pulley Combinations (cont.).

DIAM. OF CUTTER	20-40 PULLEY RATIO	40-60 PULLEY RATIO	60-100 PULLEY RATIO	100-140 PULLEY RATIO	140-180 PULLEY RATIO	180-230 PULLEY RATIO	230-280 PULLEY RATIO	280-330 PULLEY RATIO	330-400 PULLEY RATIO	400-600 PULLEY RATIO	600-1000 PULLEY RATIO	HEAD
¼"	(5:2) (6:2)	*	*	*	*	*	*	*	*	*	*	BASIC UNIT/UNIVERSAL HEAD
	(2:6)	(2:5)	(3:5) (3:4) (4:5)	(4:4)	(5:4) (4:3)	(5:3) (6:3)	*	(5:2)	(6:2)	*	*	HIGH SPEED HEAD
1/8"	(2:5) (3:5)	(3:4) (4:4) (4:4)	(5:4) (4:3) (5:3)	(6:3)	(5:2)	(6:2)	*	*	*	*	*	HIGH SPEED HEAD
1/16"	(3:4) (4:5) (4:4) (5:4) (4:3)	(5:3) (6:3)	(5:2) (6:2)	*	*	*	*	*	*	*	*	HIGH SPEED HEAD

NOTE: An "*" indicates a pulley ratio is not available for that size cutter and cutter speed.

APPENDIX B

WEIGHTS AND MEASURES

Length Measure

	Miles	Furlongs	Rods	Yards	Feet	inches
Mile	1	8	320	1,760	5,280	63,360
Furlong		1	40	220	660	7,920
Rod			1	5.5	16.5	198
Yard				1	3	36
Foot					1	12
Inches						I

Square Measure

	Sq. Miles	Acres	Sq. Rods	Sq. Yards	Sq. Feet	Sq. Inches
Sq. Mile	1	640	120,400	3,097,600	27,878,400	4,014,489,600
Acre		1	160	4,840	43,560	62,729,640
Sq. Rod			1	30.25	272.25	39,204
Sq. Yard				1	9	1.296
Sq. Foot					1	12
Sq. Inch						1

Dry Measure

	Bushels	Pecks	Quarts	Pints
Bushel	1	4	32	64
Peck		1	8	16
Quart			1	2
Pint				1

1 Bushel (US) = 2125.42 cubic inches
I Bushel (British) = 2218.19 cubic inches

Liquid Measure

	Hogshead	Barrels	Gallons	Quarts	Pints	Gills
Hogshead	1	2	63	252	504	2,016
Barrel		1	31.5	126	252	504
Gallon			1	8	16	32
Quart				1	2	4
Pint					1	2
Gill						1

The US gallon contains 231 cu in = 0.134 cu ft
One cubic foot = 7.481 gallons
One cubic foot weighs 62.425 lb. at 39.2'F
One gallon weighs 8.345 lb.
British Imperial gallon weighs 10 lb.
For rough calculations, I cu ft is called 71/4 gallons and 1 gallon is 8 1/3 lb.

Weight Measure

	Long tons	Tons	Pounds	Ounces	Grains
Long ton	1	1.12	2,240	35,846	250,880,000
Ton		1	2,000	32,000	224,000,000
Pound			1	16	7000
Grain					1

Angles of Arcs

	Circles	Degrees	Minutes	Seconds
Circle	1	360	21,600	1,269,000
Degree		1	60	3,600
Minute			1	60
Second				1

Water Conversion Factors

US gallons	X	8.33	=	pounds
US gallons	X	0.13368	=	cubic feet
US gallons	X	231	=	cubic inches
US gallons	X	0.83	=	British gallons
US gallons	X	3.78	=	liters

British gallons (Imperial)	X	10	=	pounds
British gallons (Imperial)	X	0.16	=	cubic feet
British gallons (Imperial)	X	277.274	=	cubic inches
British gallons (Imperial)	X	1.2	=	US gallons
British gallons (Imperial)	X	4..537		liters

Cubic inches of water (39.2'F)	X	0.036125	=	pounds
Cubic inches of water (39.2'F)	X	0.004329	=	US gallons
Cubic inches of water (39.2'F)	X	0.003607	=	British gallons
Cubic inches of water (39.2'F)	X	0.576384	=	ounces
Cubic inches of water (39.2'F)	X	62.425	=	pounds
Cubic feet (of water) (39.2'F)	X	7.48	=	US gallons
Cubic feet (of water) (39.2'F)	X	6.232	=	British gallons
Cubic feet (of water) (39.2F)	X	.028	=	tons

Pounds of water	X	7 72	=	cubic inches
Pounds of water	X	.01602	=	cubic feet
Pounds of water	X	0.12	=	US gallons
Pounds of water	X	0.10	=	British gallons

METRIC SYSTEM

Length Measures

	Kilometers	Hectometers	Dekameters	Meters	Decimeters	Centimeters	Millimeters
km	1		100	1,000	10,000	100,000	1,000,000
hm	0.1	1	10	100	1,000	10,000	100,000
dkm	0.01	0.1	1	10	100	1,000	10,000
m	0.0001	0.01	0.1	1	10	100	1,000
dm	0.0001	0.001	0.01	0.1	1	10	100
cm	0.00001	0.0001	0.001	0.01	0.1	1	10
mm	0.000001	0.00001	0.0001	0.001	0.01	0.1	1

Square Measure

1	sq.	kilometer	=	100	sq.	hectometers
I	sq.	hectometer	=	100	sq.	decameters
I	sq.	decameter	=	100	sq.	meters
I	sq.	meter	=	100	sq.	decimeters
I	sq.	decimeter	=	100	sq.	centimeters
I	sq.	centimeter	=	100	sq.	millimeters

Capacity Measure

	Kiloliters	Hectoliters	Dekaliters	Liters	Deciliters	Centiliters	Milliliters
kl	1		100	1,000	10,000	100,000	1,000,000
hl	0.1	1	10	100	1,000	10,000	100,000
dlm	0.01	0.1	1	10	100	1,000	10,000
ml	0.0001	0.01	0.1	1	10	100	1,000
dl	0.0001	0.001	0.01	0.1	1	10	100
ci	0.00001	0.0001	0.001	0.01	0.1	1	10
ml	0.000001	0.00001	0.0001	0.001	0.01	0.1	1

Weight Measure

	Kilograms	Hectograms	Dekagrams	Grams	Decigrams	Centigrams	Milligrams
kg	1		100	1,000	10,000	100,000	1,000,000
hg	0.1	1	10	100	1,000	10,000	100,000
dkg	0.01	0.1	1	10	100	19000	10,000
mg	0.0001	0.01	0.1	1	10	100	1,000
dg	0.0001	0.001	0.01	0.1	1	10	100
C9	0.00001	0.0001	0.001	0.01	0.1	1	10
mg	0.000001	0.00001	0.0001	0.001	0.01	0.1	I

BRITISH AND METRIC CONVERSION TABLES

Measure of Length

I inch	=	2.54 centimeters or 25.4 millimeters
I foot	=	0.3048 meter, 30.48 centimeters, 304.8 millimeters
I yard	=	0.9144 meters, 91.44 centimeters, 914.4 millimeters
I rod	=	5.0292 meters
I mile	=	1.609 kilometers, 1,609.34 meters
I millimeter	=	0.03937 inch
I centimeter	=	0.3937 inch
1 meter	=	39.37 inches, 3.28083 feet 1.0936 yards
1 kilometer	=	0.62137 mile

Surface Measure

1 sq. inch	=	6.452 sq. centimeters, 645.2 sq. millimeters
I sq. foot	=	0.0929 sq. meter, 929.03 centimeters
1 sq. yard	=	0.836 sq. meter
I sq. millimeter	=	0.00155 sq. inch
I sq. centimeter	=	0.155 sq. inch
I sq. meter	=	1.196 sq. yards, 10.764 sq. feet 1,550.003 sq. inches

Volume and Capacity Measure

I cubic inch	=	16.387 cubic centimeters, 16,387.06 millimeters
I cubic foot	=	0.02832 cubic meter, 28.317 cubic decimeters, 28.317 liters
I cubic yard	=	0.7645 cubic meter
I cubic centimeter	=	0.061 cubic inch
I cubic decimeter	=	61.023 cubic inches, 0.0353 cubic foot
I cubic meter	=	231 cubic inches, 1.308 cubic yards, 35.314 cubic feet 264.2 gallons

Weight Measure

I gram		
I kilogram	=	0.03527 ounce, 15.432 grains
I metric ton	=	2.2046 pounds, 35.274 ounces avoirdupois
1 grain	=	0.9842 long ton (2,240 lb.), 1.1023 ton (2,000 lb.), 2,204.6 pounds
I ounce avoirdupois	=	0.0648 grams
1 pound	=	28.35 grams
1 ton (2,000 lb.)		
I long ton (2,240 lb.)	=	0.4536 kilogram, 453.6 grams
	=	907.2 kilograms
	=	1.016 metric tons, 1,016 kilograms

Temperature Conversion

Fahrenheit		
Celsius		
	=	(Celsius x 1.8) + 32
	=	(Fahrenheit 32) /.8

POWER UNITS

1 HORSEPOWER = 33,000 FOOT-POUNDS PER MINUTE, 746 WATTS.
I WATT = 0.00134 HORSEPOWER, 44.24 FOOT-POUNDS PER MINUTE
I KILOWATT = 1,000 WATTS, 1.34 HORSEPOWER 44,240 FOOT-POUNDS PER MINUTE

WEIGHTS OF MATERIALS

Material	Weight in pounds per cubic foot	Weight in pounds per cubic inch	Weight in kilograms per cubic meter	Weight in grams per cubic centimeter
Aluminum	168.5	0.0975	2,699.11	2.6988
Brass, 80% C, 20% Z	536.6	0.3105	8,595.51	8.5946
Brass, 70% C, 30% Z	526.7	0.3048	8,436.92	8.4368
Brass, 60% C, 40% Z	521.7	0.3019	8,356.83	8.3538
Brass, *50% C,* 50% Z	511.7	0.2961	8,196.65	8.1960
Brick, common	112	0.0648	1,794.07	1.7937
Brick, fire	143	0.0827	2,290.64	2.2891
Brick, pressed	137	0.0793	2,194.53	2.1950
Brick, hard,	125	0.0723	29002.31	2.0013
Bronze, 90% *C, 10%* T	547.9	0.3171	8,776.51	8.7773
Cement portland, loose	90	0.0521	1,441.66	1.4421
Cement portland, set	183	0.1059	2,931.38	2.9313
Chromium	432.4	0.2502	6,926.38	6.9255
Clay, loose	63	0.0365	1,009.16	1.0103
Coat broke, loose, anthracite	54	0.0313	864.99	0.8664
Coal broken loose, bituminous	49	0.0294	784.90	0.7861
Concrete	137	0.0793	2,194.53	2.1950
Copper	554.7	0.3210	8,885.44	8.8852
Earth, common, loam	75	0.0434	1,201.38	1.2013
Earth, packed	100	0.0579	1,601.85	1.6027
Glass	162	0.0938	2,594.99	2.5964
Gravel dry, loose	90 to 106		1,441.66 to 1,697.96	
gravel well shaken	99 to 117		1,585.83 to 1,874.16	
Gold	19204.3	0.6969	19,291.03	19.2901
Ice	56	0.0324	897.03	0.8968
Iron, cast	450	0.2604	7,08.31	7.2078
Iron, wrought	486.7	0.2817	7,796.18	7.7974
Lead	707.7	0.4095	11,336.26	11.3349
Lime	53	0.0307	848.98	0.8498
Magnesium	108.6	0.0628	1,739.60	1.7383
Masonry	150	0.0868	2,402.77	2.4026
Masonry, dry rubble	138	0.0799	2,210.55	2.2116
Molybdenum	636.5	0.3683	10,195.75	10.1945
Mortar, set	103	*0.05%*	1,649.90	1.6497
Nickel	549	0.3177	8,794.13	8.7939
Petroleum, benzene	46	0.0266	736.85	0.7363
Petroleum gasoline	42	0.0243	672.78	0.6726
Plaster of Paris	112	0.0648	1,794.07	1.7937
Platinum	1,333.5	0.7717	21,360.62	21.3606
Quartz	162	0.0938	2,594.99	2.5964

WEIGHTS OF MATERIALS (continued)

.... Material	Weight in pounds per cubic foot	Weight in pounds per cubic inch	Weight in kilograms per cubic meter	Weight in grams per cubic centimeter
Salt, common	48	0.0278	768.88	0.7695
Sand dry, loose	90 to 106		1,441.66 to 1,697.96	
Sand, well shaken	99 to 106 .		1,585.83 to 1,874.16	
Silver	657	0.3802	10,524.13	10.5239
Snow, freshly fallen	5 to 12		80.09 to 192.22	
Snow, wet and compacted	15 to 50 ...		240.28 to 800.92	
Steel	490	0.2836	7,849.05	7.8500
Stone, gneiss	168	0.0972	2,691.10	2.6905
Stone, granite	168	0.0972	2,691.10	2.6905
Stone, limestone	162	0.0938	2,594.99	2.5964
Stone, marble	168	0.0972	2,691.10	2.6905
Stone, sandstone	143	0.0828	2,290.64	2.2919
Stone, shale	162	0.0938	2,594.99	2.5964
Stone, slate	175	0.1013	21803.23	2.8040
Tar	75	0.0434	1,201.38	1.2013
Tin	455	0.2632	7,288.40	7.2881
Titanium	280.1	0.1621	4,486.77	4.4869
Tungsten	1,192	0.6898	19,094.00	19.0936
Water, fresh	62.5	0.0362	1,001.15	1,0020
Water, sea water	64	0.0370	1,025.18	1.0242
Wood, dry				
As, black	28	0.0162	448.52	0.4484
Ask white	41	0.0237	656.76	0.6560
Beech	45	0.0260	720.83	0.7197
Birch	44	0.0255	704.81	0.7058
Birch paper	38	0.0220	608.70	0.6090
Cedar, Alaska	31	0.0179	496.57	0.4955
Cedar, eastern red	33	0.0191	528.61	0.5287
Cedar, southern white	23	0.0133	368.42	0.3681
Cedar, western re	23	0.0133	368.42	0.3681
Cherry	42	0.0203	672.78	0.6726
Cherry, black	35	0.0203	560.65	0.5619
Chestnut	41	0.0237	656.76	0.6560
Cypress	30	0.0174	480.55	0.4816
Elm	45	0.0260	720.83	0.7197
Hemlock	29	0.0168	464.54	0.4650
Hickory	49	0.0284	784.90	0.7861
Locust	46	0.0266	736.85	0.7363
Mahogany	53	0.0307	848.98	0.8498
Maple, hard	43	0.0249	688.79	0.6892
Maple, white	33	0.0191	528.61	0.5287
Oak, chestnut	54	0.0313	864.99	0,8664
Oak, live	59	0.0341	954.09	0.9439
Oak, red black	41	0.0237	656.76	0.6560
Oak, white	46	0.0266	738.85	0.7363

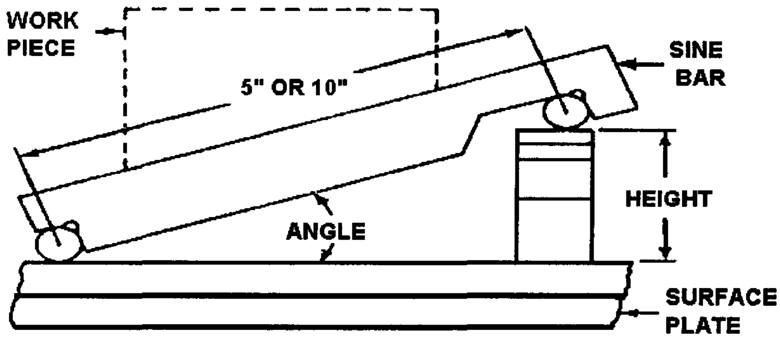
WEIGHTS OF MATERIALS (Continued)

<u>Material</u>	Weight in pounds per cubic foot	Weight in pounds per cubic inch	Weight in kilograms per cubic meter	Weight in grams per cubic centimeter
Pine, white	26	0.0150	416.48	0.4152
Pine, yellow, longleaf	44	0.0255	656.76	0. 7058
Pine, yellow, short leaf	36	0.0208	576.66	0.5757
Poplar	28	0.0162	448.52	0.4484
Red wood California	26	0.0150	416.48	0.4152
Spruce, white, black	27	0.0156	432.50	0.4318
Sycamore	37	0.0214	592.68	0.5923
Walnut black	38	0.0220	608.70	0.6089
Walnut, white	26	0.0150	416.48	0.4152
Zinc	439.3.....	0.2542	7,036.91	7.0362

APPENDIX C

FORMULAS

SINE BAR OR SINE PLATE SETTING



Sine bars or sine plates usually have a length of 5 inches or 10 inches. These standard lengths are commonly used by the tool maker or inspector. The sine bar or sine plate is used for accurately setting up work for machining or for inspection. Gage blocks are usually used for establishing the height.

Rule for determining the height of the sine bar setting for a given angle: multiply the sine of the angle by the length of the sine bar. The sine of the angle is taken from the tables of trigonometric functions.

Problem: What would be the height to set a sine bar for establishing an angle of 230 41'? **Solution:** The sine of 23'41' is 0.40168. Multiply this by 5 because a 5-inch sine bar is used; 5 x 0.40168 = 2.0084, which is the height to set the sine bar.

RULES FOR FIGURING TAPERS

TO FIND	GIVEN	RULE
Taper per inch	Taper per foot	Divide the taper per foot by 12.
Taper per foot	Taper per inch	Multiply the taper per inch by 12.
Taper per foot	End diameters and length of taper in inches	Subtract small diameter from large, divided by length of taper, and multiply quotient by 12.
Diameter at small end in inches	Large diametre, length of taper in inches, and taper foot	Divide taper per foot by 12, multiply by length of taper, and subtract from large diameter.
Diameter at large end in inches	Small diameter, length of taper in inches, and taper per foot	Divide taper per foot by 12, multiply by length of taper, and add results to small diameter.
Distance between two given diameters in inches	Taper per foot and two diameters in inches	Subtract small diameter from large, divide remainder by taper per foot and multiply quotient by 12.
Amount of taper in a certain length given in inches	Taper per foot	Divide taper per foot by 12 and multiply by given length of tappered part

To find the circumference of a circle Ξ xD or D / 0.3183.

To find the diameter of a circle 0.31831 x C or C / Ξ.

To find the area of a circle Ξr2.

To find size of round stock needed to machine a hexagon, D = 1.1547 x distance across the flats

To find size of round stock needed to machine a square, D = 1.4142 x distance across the flats

To find the area of a square, square one side

To find the area of a rectangle, multiply length times width

To find the volume of a cube, multiply length times width times depth

To find the volume of a square prism, multiply length times width times depth

To find the volume of a cylinder, multiply Ξ times radius squared times height

To find the area of a triangle, multiply base times height divided by 2

To find the area of a ring, subtract the area of inside diameter from the area of the outside diameter.

TRIGONOMETRY FORMULAS

Formulas for Finding Functions of Angles

$$\frac{\text{Side opposite}}{\text{Hypotenuse}} = \text{sine}$$

$$\frac{\text{Side adjacent}}{\text{Hypotenuse}} = \text{cosine}$$

$$\frac{\text{Side opposite}}{\text{Side adjacent}} = \text{tangent}$$

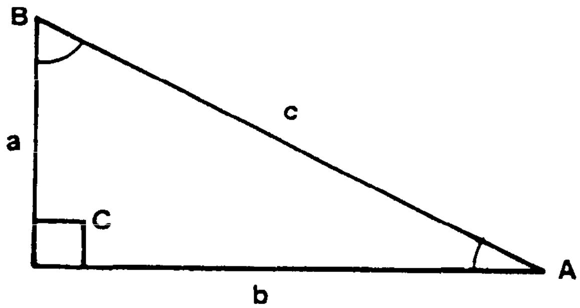
$$\frac{\text{Side adjacent}}{\text{Side opposite}} = \text{cotangent}$$

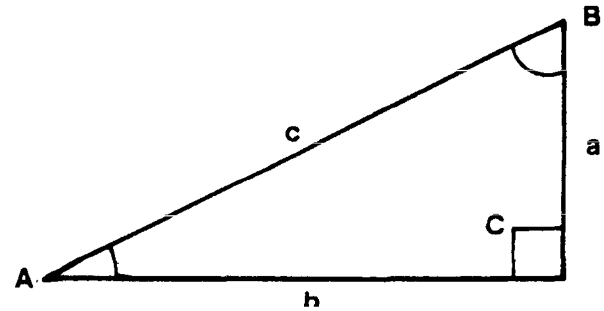
$$\frac{\text{Hypotenuse}}{\text{Side adjacent}} = \text{secant}$$

$$\frac{\text{Hypotenuse}}{\text{Side opposite}} = \text{cosecant}$$

Formulas for Finding the Length of Slides for Right-Angle Triangle When an Angle and Side are Known

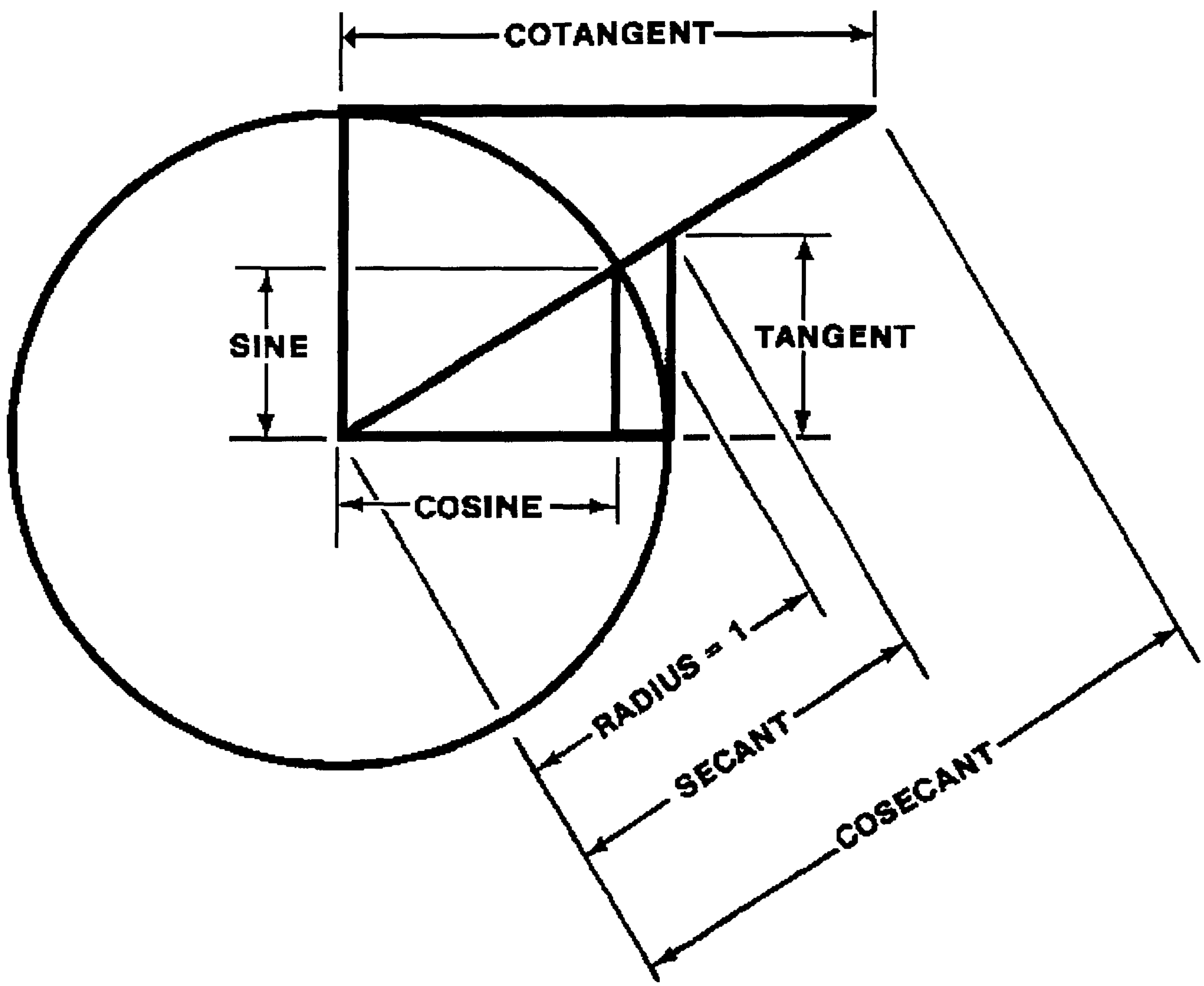
Length of side adjacent	Hypotenuse * sine Hypotenuse/cosecant Side adjacent * tangent Side adjacent/cotangent
Length of side adjacent	Hypotenuse * sine Hypotenuse/secant Side opposite * cotangent Side opposite/tangent
Length of hypotenuse	Side opposite * cosecant Side opposite/sine Side adjacent * secant Side adjacent/cosine





RIGHT TRIANGLES

KNOWN	SIDE a	TO FIND SIDE b	SIDE c
Side c, Angle B	Cosine B x c or c / Secant B	Sine B x c or c / Cosecant B	Angle A = 90° - B
Side c, Angle A	Sine A x c or c / Cosecant A	Cosine A x c or c / Secant A	Angle B = 90° - A
Side b, Angle B	Cotangent B x b or b / Tangent B	Angle A = 90° - B	Cosecant B x b or b / Sine B
Side b, Angle A	Tangent A x b or b / Cotangent A	Angle B = 90° - A	Secant A x b or b / Cosine A
Side a, Angle B	Angle A = 90° - B	Tangent B x b or a / Cotangent B	Secant B x b or a / Cosine B
Side a, Angle A	Angle B = 90° - A	Cotangent A x a or a / Tangent A	Cosecant A x a or a / Sine A
	ANGLE A	**ANGLE B**	**SIDE x**
Side c and b	Cosine A = b+c or Secant A = c+b	Sine B = b+c or Cosecant B = c+b	Side A = $c^2 - b^2$
Side c and a	Sine A = a+c or Cosecant A = c+a	Cosine B = a+c or Secant B = c+a	Side b = $c^2 - a^2$
Side c and a	Tangent A = a+b or Cotangent A = b+a	Cotangent B = a+b or Tangent B = b+a	Side c = $a^2 - b^2$


COTANGENT
SINE
TANGENT
COSINE
RADIUS = 1
SECANT
COSECANT

INDEX