EXPERIMENTS IN MANUFACTURING PROCESSES-I

SMT. S. R. PATEL ENGINEERING COLLEGE DABHI, UNJHA

PIN- 384 170

DEPARTMENT OF MECHANICAL ENGINEERING

SUBJECT : MANUFACTURING PROCESSES-I

SUBJECT CODE: 131903

INDEX

LATHES AND LATHE MACHINING OPERATIONS

LATHE

- Lathe is a machine, which removes the metal from a piece of work to the required shape & size to be produced.
- For this purpose, work is held between to centres (Live & Dead Centres) or in a chuck or face plate.
- The cutting tool is held and supported rigidly in a tool post and is made to move either parallel to or inclined to the axis to cut the material.

There are a number of different types of lathes installed in the machine shops in various Navy ships. These include the engine lathe, the horizontal turret lathe, and several variations of the basic engine lathe, such as bench, toolroom, and gap lathes. All lathes, except the vertical turret type, have one thing in common. For all usual machining operations, the workpiece is held and rotated about a horizontal axis, while being formed to size and shape by a cutting tool. In the vertical turret lathe, the workpiece is rotated about a vertical axis. Of the various types of lathes, the type you are most likely to use is the engine lathe. Therefore, this chapter deals only with engine lathes and the machining operations you may have to perform.

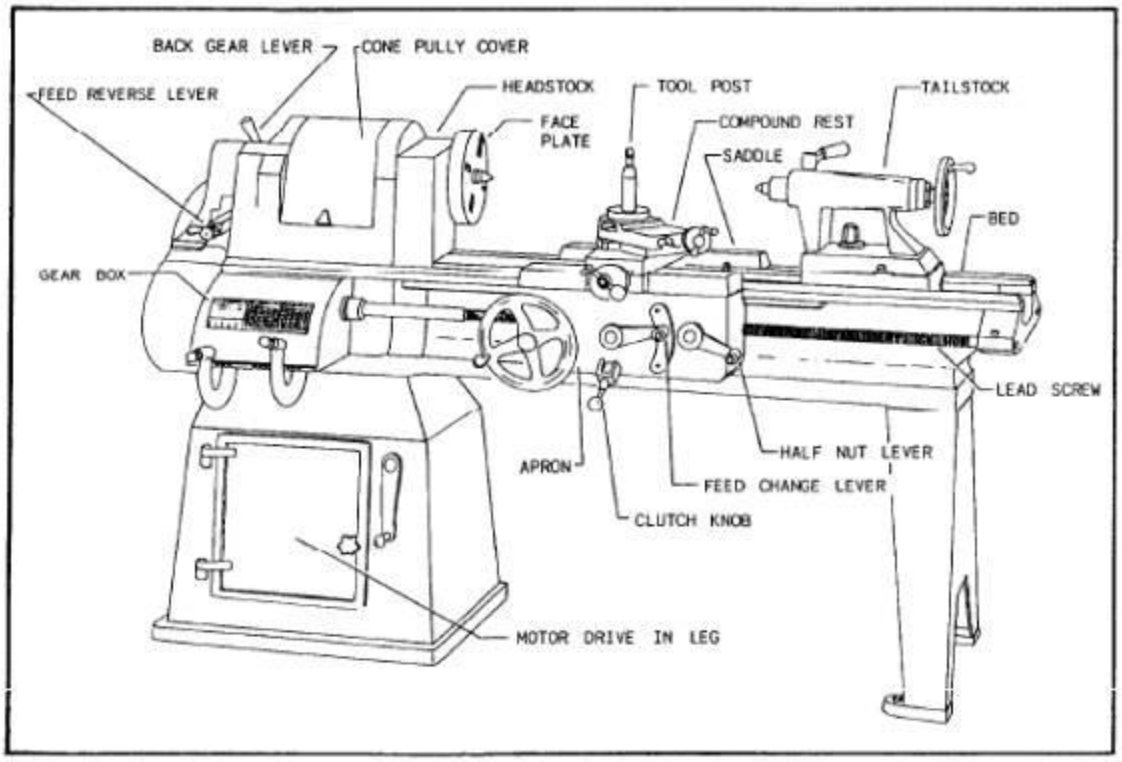
ENGINE LATHE

An engine lathe similar to the one shown in figure 9-1 is found in every machine shop. It is used mostly for turning, boring, facing, and thread cutting. But it may also be used for drilling, reaming, knurling, grinding, spinning, and spring winding. Since you will primarily be concerned with turning, boring, facing, and thread cutting, we will deal primarily with those operations in this chapter.

The work held in the engine lathe can be revolved at any one of a number of different speeds, and the cutting tool can be accurately controlled by hand or power for longitudinal feed and crossfeed. (Longitudinal feed is the movement of the cutting tool parallel to the axis of the lathe; crossfeed is the movement of the cutting tool perpendicular to the axis of the lathe.)

Lathe size is determined by two measurements: (1) the diameter of work it will swing (turn) over the ways and (2) the length of the bed. For example, a 14-inch by 6-foot lathe will swing work up to 14 inches in diameter and has a bed that is 6 feet long.

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



Typical engine lathe.

PRINCIPAL PARTS

Bed and Ways

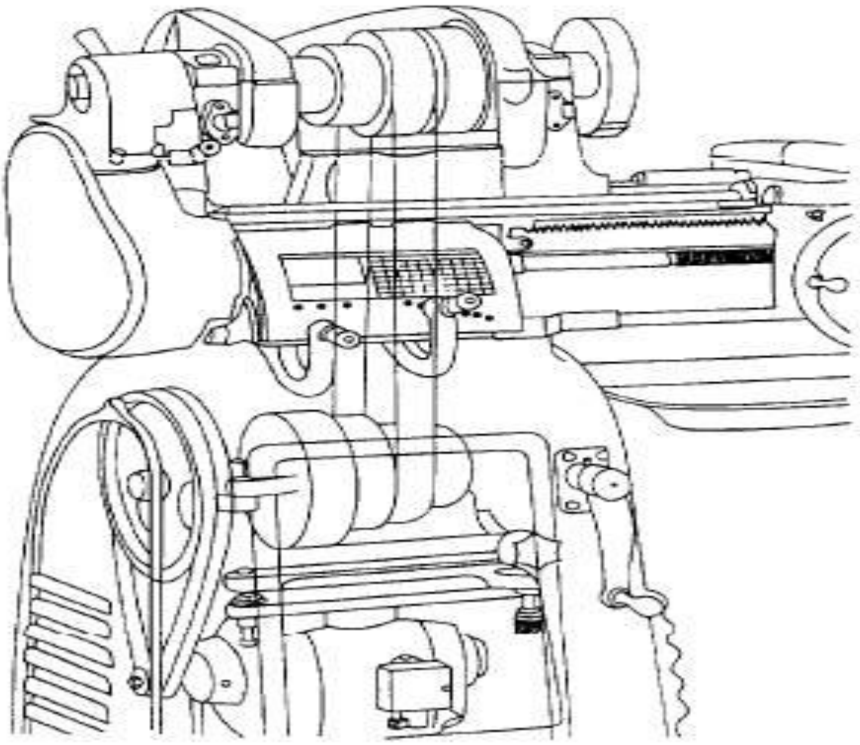
The bed is the base or foundation of the parts of the lathe. The main feature of the bed is the ways, which are formed on the bed's upper surface and run the full length of the bed. The ways keep the tailstock and the carriage, which slide on them, in alignment with the headstock.

Headstock

The headstock contains the headstock spindle and the mechanism for driving it. In the belt-driven type, shown in figure 9-2, the driving mechanism consists of a motor-driven cone pulley that drives the spindle cone pulley through a drive belt. The spindle can be rotated either directly or through back gears.

When the headstock is set up for direct drive, a bull-gear pin, located under a cover to the right of the spindle pulley, connects the pulley to the spindle. This connection causes the spindle to turn at the same speed as the spindle pulley.

When the headstock is set up for gear drive, the bull-gear pin is pulled out, disconnecting the spindle pulley from the spindle. This allows the spindle to turn freely inside the spindle pulley. The back-gear lever, on the left end of the headstock, is moved to engage the back-gear set with a gear on the end of the spindle and a gear on the end of the spindle pulley. In this drive mode, the drive belt turns the spindle pulley, which turns the back-gear set, which turns the spindle. Each drive mode provides four spindle speeds, for a total of eight. The back-gear drive speeds are less slower than the direct-drive speeds.

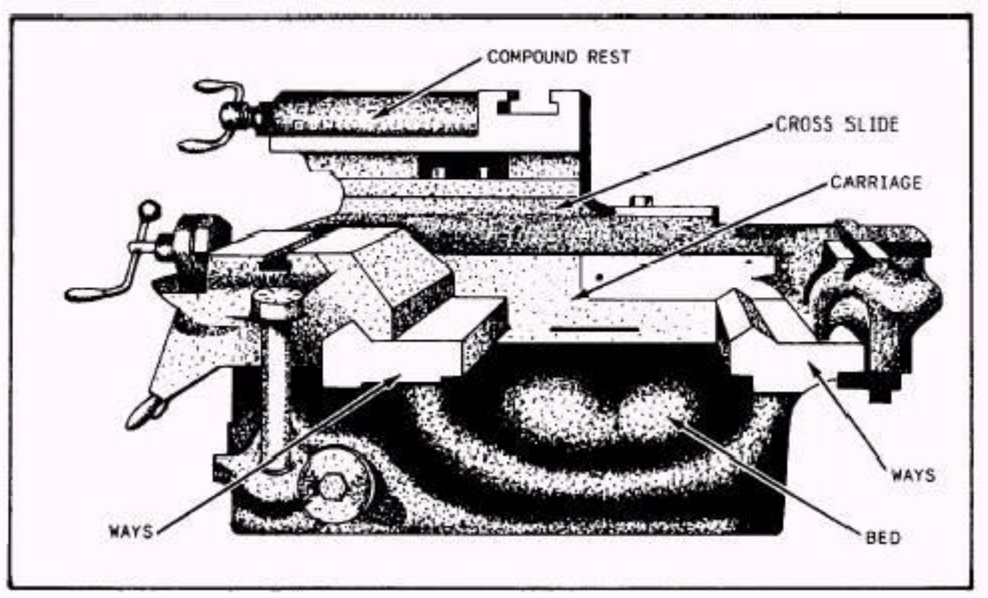


Belt-driven type of headstock.

Tailstock

The primary purpose of the tailstock is to hold the dead center to support one end of the work being machined. However, the tailstock can also be used to hold tapered shank drills, reamers, and drill chucks. It can be moved on the ways along the length of the bed and can be clamped in the desired position by tightening the tailstock clamping nut. This movement allows for the turning of different lengths of work. The tailstock can be adjusted laterally (front to back) to cut a taper by loosening the clamping screws at the bottom of the tailstock.

Before you insert a dead center, drill, or reamer, carefully clean the tapered shank and wipe out the tapered hole of the tailstock spindle. When you hold drills or reamers in the tapered hole of the spindle, be sure they are tight enough so they will not revolve. If you allow them to revolve, they will score the tapered hole and destroy its accuracy.



Side view of a carriage mounted on a bed.

Carriage

The carriage is the movable support for the crossfeed slide and the compound rest. The compound rest carries the cutting tool in the tool post. Figure 9-3 shows how the carriage travels along the bed over which it slides on the outboard ways.

The carriage has T-slots or tapped holes to use for clamping work for boring or milling. When the carriage is used for boring and milling operations, carriage movement feeds the work to the cutting tool, which is rotated by the headstock spindle.

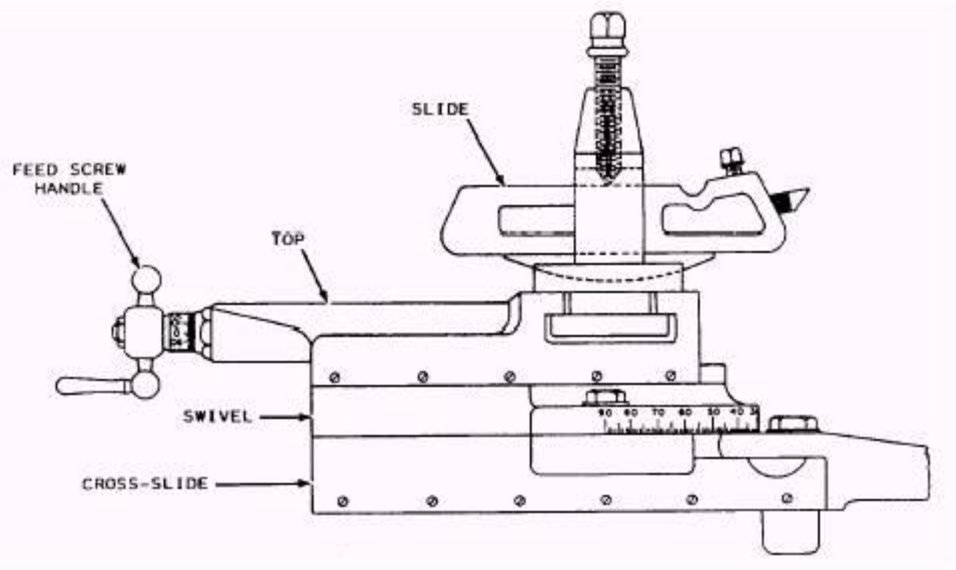
You can lock the carriage in any position on the bed by tightening the carriage clamp screw. But you do this only when you do such work as facing or parting-off, for which longitudinal feed is not required. Normally the carriage clamp is kept in the released position. Always move the carriage by hand to be sure it is free before you engage its automatic feed.

Apron

The apron is attached to the front of the carriage and contains the mechanism that controls the movement of the carriage and the crossslide.

Feed Rod

The feed rod transmits power to the apron to drive the longitudinal feed and crossfeed mechanisms. The feed rod is driven by the spindle through a train of gears. The ratio of feed rod speed to spindle speed can be varied by using change gears to produce various rates of feed.



Compound rest.

The rotating feed rod drives gears in the apron; these gears in turn drive the longitudinal feed and crossfeed mechanisms through friction clutches.

Some lathes do not have a separate feed rod, but use a spline in the lead screw for the same purpose.

Lead Screw

The lead screw is used for thread cutting. It has accurately cut Acme threads along its length that engage the threads of half-nuts in the apron when the half-nuts are clamped over it. The lead screw is driven by the spindle through a gear train. Therefore, the rotation of the lead screw bears a direct relation to the rotation of the spindle. When the half-nuts are engaged, the longitudinal movement of the carriage is controlled directly by the spindle rotation. Consequently, the cutting tool is moved a definite distance along the work for each revolution that the spindle makes.

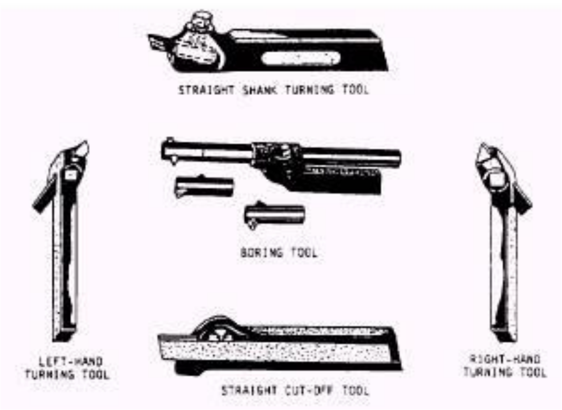
Crossfeed Slide

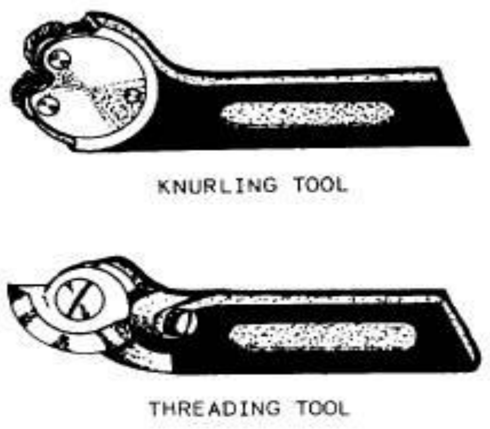
The crossfeed slide is mounted to the top of the carriage in a dovetail and moves on the carriage at a right angle to the axis of the lathe. A crossfeed screw allows the slide to be moved toward or away from the work in accurate increments.

Compound Rest

The compound rest, mounted on the compound slide, provides a rigid adjustable mounting for the cutting tool. The compound rest assembly has the following principal parts:

1. The compound rest **SWIVEL,** which can be swung around to any desired angle and clamped in position. It is graduated over an arc of 90° on each side of its center position for easier setting to the angle selected. This feature is used for machining short, steep tapers, such as the angle on bevel gears, valve disks, and lathe centers.
2. The compound rest, or **TOP SLIDE,** which is mounted on the swivel section on a dovetailed slide. It is moved by the compound rest feed screw.





Common types of toolholders **Knurling and threading toolholders.**

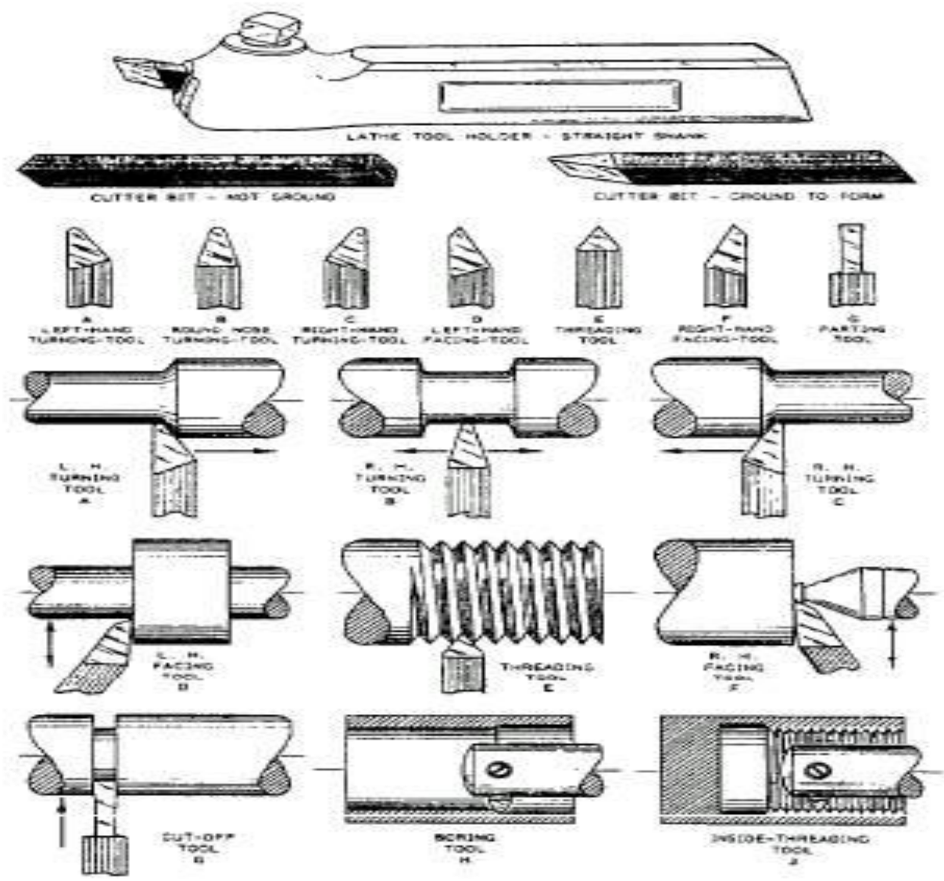
This arrangement permits feeding the tool to the work at any angle (determined by the angular setting of the swivel section). The graduated collars on the crossfeed and compound rest feed screws read in thousandths of an inch for fine adjustment in regulating the depth of cut.

Accessories and Attachments

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

TOOL POST.—The sole purpose of the tool post is to provide a rigid support for the tool. It is mounted in the T-slot of the compound rest. A forged tool or a toolholder is inserted in the slot in the tool post. By tightening a setscrew, you will firmly clamp the whole unit in place with the tool in the desired position.

TOOLHOLDERS—Some of the common toolholders used in lathe work are illustrated in figure 9-5. Notice the angles at which the tool bits are set in the various holders. These angles must be considered with respect to the angles ground on the tools and the angle that the toolholder is set with respect to the axis of the work.



Lathe tools and their applications

A knurling toolholder carries two knurled rollers which impress their patterns on the work as it revolves. The purpose of the knurling tool is to provide a roughened surface on round metal parts, such as knobs, to give a better grip in handling. The knurled rollers come in a variety of patterns.

ENGINE LATHE TOOLS.—Figure shows the most popular shapes of ground lathe cutter bits and their applications. In the following paragraphs we will discuss each of the types shown.

Left-Hand Turning Tool.—This tool is ground for machining work when it is fed from left to right, as indicated in figure 9-7, view A. The cutting edge is on the right side of the tool, and the top of the tool slopes down away from the cutting edge.

Round-Nosed Turning Tool.–This tool is for general-purpose machine work and is used for taking light roughing cuts and finishing cuts. Usually, the top of the cutter bit is ground with side rake so the tool may be fed from right to left. Sometimes this cutter bit is ground flat on top so the tool may be fed in either direction.

Right-Hand Turning Tool.–This is just the opposite of the left-hand turning tool and is designed to cut when it is fed from right to left. The cutting edge is on the left side. This is an ideal tool for taking roughing cuts and for all-around machine work.

Left-Hand Facing Tool.–This tool is intended for facing on the left-hand side of the work. The direction of feed is away from the lathe center. The cutting edge is on the right-hand side of the tool, and the point of the tool is sharp to permit machining a square corner.

Threading Tool.–The point of the threading tool is ground to a 60-degree included angle for machining V-form screw threads. Usually, the top of the tool is ground flat, and there is clearance on both sides of the tool so it will cut on both sides.

Right-Hand Facing Tool.–This tool is just the opposite of the left-hand facing tool and is intended for facing the right end of the work and for machining the right side of a shoulder.

Square-Nosed Parting (Cutoff) Tool.–The principal cutting edge of this tool is on the front. Both sides of the tool must have sufficient clearance to prevent binding and should be ground slightly narrower at the back than at the cutting edge. This tool is convenient for machining necks and grooves and for squaring comers and cutting off.

Boring Tool.–The boring tool is usually ground the same shape as the left-hand turning tool so that the cutting edge is on the right side of the cutter bit and may be fed in toward the headstock.

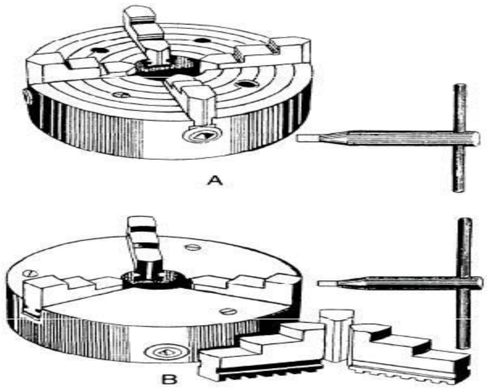
Inside-Threading Tool. —The inside-threading tool has the same shape as the threading tool, but it is usually much smaller. Boring and inside-threading tools may require larger relief angles when used in small diameter holes.

LATHE CHUCKS.—The lathe chuck is a device for holding lathe work. It is mounted on the nose of the spindle. The work is held by jaws which can be moved in radial slots toward the center of the chuck to clamp down on the sides of the work. These jaws are moved in and out by screws turned by a special chuck wrench.

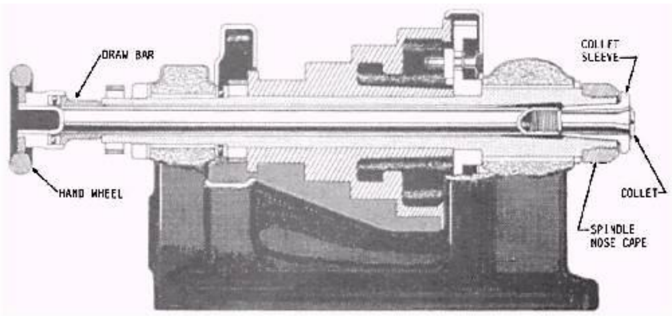
The four-jaw independent lathe chuck, view A in figure 9-8, is the most practical chuck for general work The four jaws are adjusted one at a time, making it possible to hold work of various shapes and to adjust the center of the work to coincide with the axis of the spindle. The jaws are reversible.

The three-jaw universal or scroll chuck, view B in figure 9-8, can be used only for holding round or hexagonal work All three jaws move in and out together in one operation and bring the work on center automatically. This chuck is easier to operate than the four-jaw type, but, when its parts become worn, its accuracy in centering cannot be relied upon. Proper lubrication and constant care are necessary to ensure reliability.

The draw-in collet chuck is used to hold small work for machining in the lathe. It is the most accurate type of chuck made and is intended for precision work. Figure 9-9 shows the parts assembled in place in the lathe spindle. The collet, which holds the work, is a split-cylinder with an outside taper that fits into the tapered closing sleeve and screws into the threaded end of the hollow drawbar. As the handwheel is turned clockwise, the drawbar is moved toward the handwheel. This tightening up on the drawbar pulls the collet back into the tapered sleeve, thereby closing it firmly over the work and centering the work accurately and quickly. The size of the hole in the collet determines the diameter of the work the chuck can handle.



A. Four-Jaw chuck. B. Three-Jaw chuck.

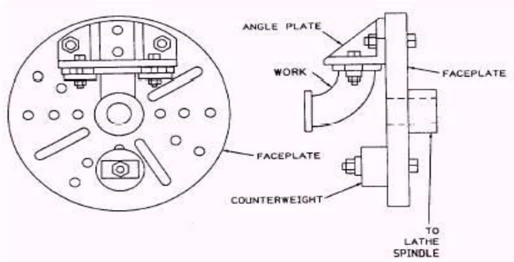


Draw-in collet chuck.

Faceplates

The faceplate is used for holding work that, because of its shape and dimensions, cannot be swung between centers or in a chuck. The T-slots and other openings on its surface provide convenient anchors for bolts and clamps used in securing the work to it. The faceplate is mounted on the nose of the spindle.

The driving plate is similar to a small faceplate and is used mainly for driving work that is held between centers. The primary difference between a faceplate and a driving plate is that a faceplate has a machined face for precision mounting, while the face of a driving plate is left rough. When a driving plate is used, the bent tail of a dog clamped to the work is inserted into a slot in the faceplate. This transmits rotary motion to the work.

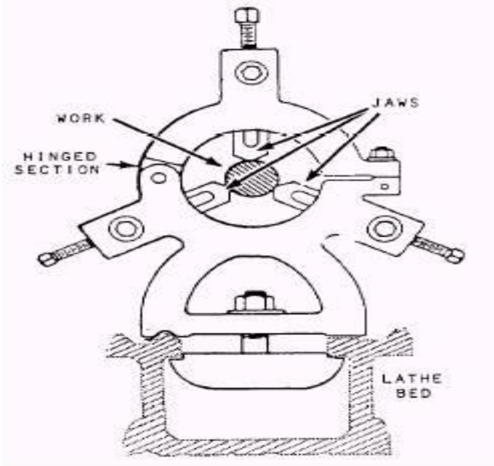


Lathe Centers

The 60-degree lathe centers provide a way to hold the work so it can be turned accurately on its axis. The headstock spindle center is called the **LIVE CENTER** because it revolves with the work. The tailstock center is called the **DEAD CENTER** because it does not turn. Both live and dead centers have shanks turned to a Morse taper to fit the tapered holes in the spindles; both have points finished to an angle of 60°. They differ only in that the dead center is hardened and tempered to resist the wearing effect of the work revolving on it. The live center revolves with the work and is usually left soft. The dead center and live center must **NEVER** be interchanged. (There is a groove around the hardened dead center to distinguish it from the live center.)

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

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



Center rest.

Lathe Dogs

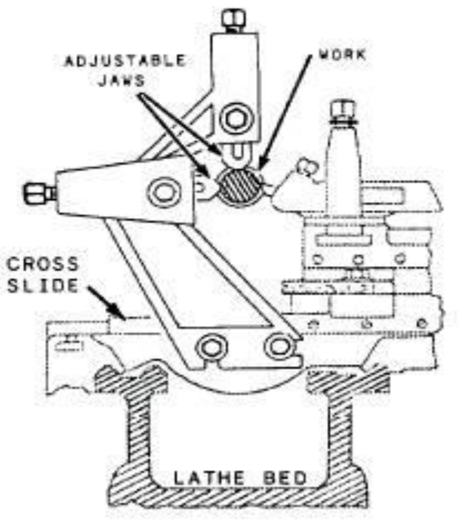
Lathe dogs are used with a driving plate or faceplate to drive work being machined on centers; the frictional contact alone between the live center and the work is not sufficient to drive the work.

Center Rest

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

1. To provide an intermediate support for long slender bars or shafts being machined between centers. The center rest prevents them from springing, or sagging, as a result of their otherwise unsupported weight.
2. To support and provide a center bearing for one end of the work, such as a shaft, being bored or drilled from the end when it is too long to be supported by a chuck alone. The center rest is clamped in the desired position on the bed and is kept aligned by the ways, as illustrated in figure 9-13. The jaws (A) must be carefully adjusted to allow the work (B) to turn freely and at the same time remain accurately centered on the axis of the lathe. The top half of the frame is a hinged section (C) for easier positioning without having to remove the work from the centers or to change the position of the jaws.

Follower Rest



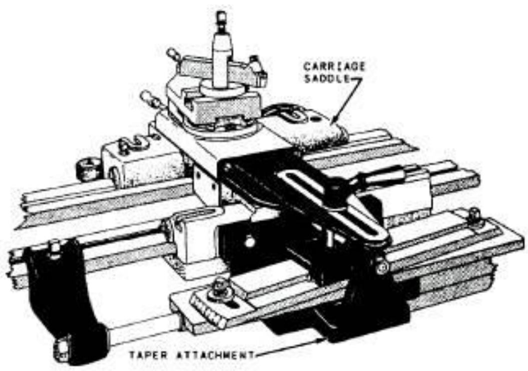
Follower rest.

The follower rest is used to back up small diameter work to keep it from springing under the cutting pressure. It can be set to either precede or follow the cutting action. As shown in figure, it is attached directly to the saddle by bolts. The adjustable jaws bear directly on the part of the work opposite the cutting tool.

Taper Attachment

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

The desired angle of taper is set on the guide bar of the attachment. The guide bar support is clamped to the lathe bed Since the cross slide is connected to a shoe that slides on this guide bar, the tool follows along a line parallel to the guide bar and at an angle to the work axis corresponding to the desired taper.



Thread Dial Indicator

The thread dial indicator, shown in figure 9-16, eliminates the need to reverse the lathe to return the carriage to the starting point each time a successive threading cut is taken. The dial, which is geared to the lead screw, indicates when to clamp the half-nuts on the lead screw for the next cut.

Thread dial Indicator.

The threading dial consists of a worm wheel which is attached to the lower end of a shaft and meshed with the lead screw. On the upper end of the shaft is the dial. As the lead screw revolves, the dial is turned and the graduations on the dial indicate points at which the half-nuts may be engaged.

Cutting Speeds and Feeds

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

Feed is the amount the tool advances for each revolution of the work. It is usually expressed in thousandths of an inch per revolution of the spindle. Cutting speeds and tool feeds are determined by various considerations: the hardness and toughness of the metal being cut; the quality, shape, and sharpness of the cutting tool; the depth of the cut; the tendency of the work to spring away from the tool; and the strength and power of the lathe. Since conditions vary, it is good practice to find out what the tool and work will stand and then select the most practical and efficient speed and feed for the finish desired.

When **ROUGHING** parts down to size, use the greatest depth of cut and feed per revolution that the work, the machine, and the tool will stand at the highest practical speed. On many pieces where tool failure is the limiting factor in the size of the roughing cut, you may be able to reduce the speed slightly and increase the feed to remove more metal. This will prolong tool life. Consider an example where the depth of cut is 1/4 inch, the feed 0.020 inch per revolution, and the speed 80 fpm. If the tool will not permit additional feed at this speed, you can drop the speed to 60 fpm and increase the feed to about 0.040 inch per revolution without having tool trouble. The speed is therefore reduced 25 percent, but the feed is increased 100 percent. Thus the actual time required to complete the work is less with the second setup.

For the **FINISH TURNING OPERATION,** take a very light cut, since you removed most of the stock during the roughing cut. Use a fine feed to run at a high surface speed. Try a 50 percent increase in speed over the roughing speed. In some cases, the finishing speed may be twice the roughing speed. In any event, run the work as fast as the tool will withstand to obtain the maximum speed during this operation. Be sure to use a sharp tool when you are finish turning.

COOLANTS

A cutting lubricant serves two main purposes: (1) It cools the tool by absorbing a portion of the heat and reducing the friction between the tool and the metal being cut. (2) It also keeps the cutting edge of the tool flushed clean.

The best lubricants to use for cutting metal must often be determined by experiment. Water-soluble oil is acceptable for most common metals. Special cutting compounds containing such ingredients as tallow, graphite, and lard, marketed under various names, are also used. But these are expensive and used mainly in manufacturing where high cutting speeds are the rule.

Some common materials and their cutting lubricants are as follows:

Metal	Lubricant
Cast iron	Usually worked dry
Mild steel	Oil or soluble oil
Hard steel	Mineral lard oil
Monel metal	Dry (or soluble oil)
Bronze	Dry (or soluble oil)
Brass	Dry (or soluble oil)
Copper	Dry (or soluble oil)
Babbitt	Dry (or soluble oil)
Aluminum	Dry (or soluble oil)

A lubricant is more important for threading than for straight turning. Mineral lard oil is recommended for threading the majority of metals that are used by the Navy.

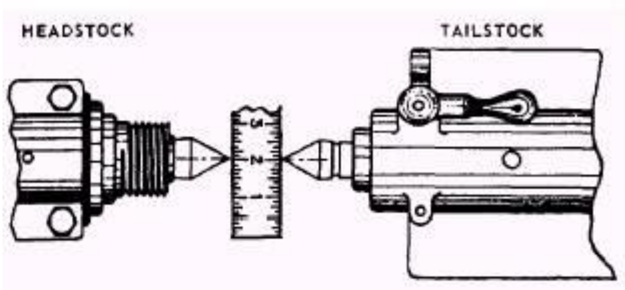
PRELIMINARY PROCEDURES

Before starting a lathe machining operation, always ensure that the machine is set up properly. If the work is mounted between centers, check the alignment of the dead center and the live center and make any necessary changes. Ensure that the toolholder and cutting tool are set at the proper height and angle. Check the work-holding accessory to ensure that the workpiece is held securely. Use the center rest or follower rest to support long workpieces.

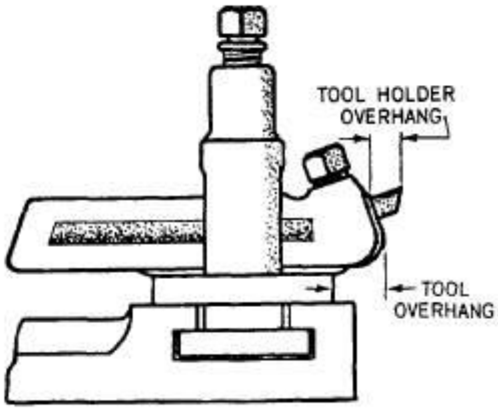
PREPARING THE CENTERS

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

A center's point must be finished accurately to an angle of 60°. Figure 9-18 shows the method of checking this angle with a center gauge. The large notch of the center gauge is intended for this purpose. If this test shows that the point is not perfect, you must true it in the lathe by taking a cut over the point with the compound rest set at 30°. You must anneal the hardened tail center before it can be machined in this manner, or you can grind it if a grinding attachment is available.



Aligning lathe centers.



Tool overhang.

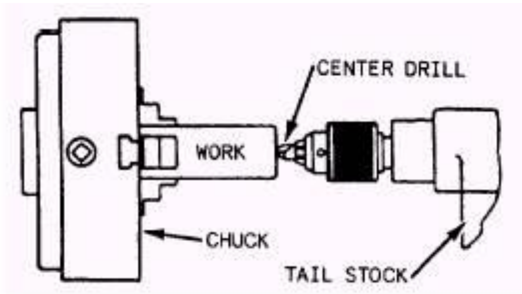
CHECKING ALIGNMENT

To turn a shaft straight and true between centers, be sure the centers are aligned in a plane parallel to the ways of the lathe. You can check the approximate alignment of the centers by moving the tailstockup until the centers almost touch and observing their relative positions as shown in figure 9-19.

To test center alignment for very accurate work, take a light cut over at each end with a micrometer and, if readings are found to differ, adjust the tailstock accordingly. Repeat the procedure until alignment is obtained.

SETTING THE TOOLHOLDER AND THE CUTTING TOOL

The first requirement for setting the tool is to have it rigidly mounted on the tool post holder. Be sure the tool sets squarely in the tool post and that the setscrew is tight. Reduce overhang as much as possible to prevent the tool bit from springing during cutting. If the tool has too much spring, the point of the tool will catch in the work, causing chatter and damaging both the tool and the work The distances represented by A and B in figure show the correct overhang for the tool bit and the holder.



Drilling a center hole.

The point of the tool must be correctly positioned on the work Place the cutting edge. slightly above the center for straight turning of steel and cast iron and exactly on the center for all other work To set the tool at the height desired, raise or lower the point of the tool by moving the wedge in or out of the tool post ring. By placing the point opposite the tailstock center point, you can adjust the setting accurately.

HOLDING THE WORK

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

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

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

Other methods of holding work to suit special conditions are (1) one end on the live center or in a chuck and the other end supported in a center rest, and (2) one end in a chuck and the other end on the dead center.

Holding Work Between Centers

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

CENTERING THE WORK.—To center round stock where the ends are to be turned and must be concentric with the unturned body, mount the work on the head spindle in a universal chuck or a draw-in collet chuck If the work is long and too large to pass through the spindle, use a center rest to support one end. Mount a center drill in a drill chuck in the tailstock spindle and feed it to the work by turning the tailstock handwheel (fig. 9-21).

For center drilling a workpiece, the combined drill and countersink is the most practical tool. These combined drills and countersinks vary in size and the drill points also vary. Sometimes a drill point on one end will be 1/8 inch in diameter, and the drill point on the opposite end will be 3/16 inch in diameter. The angle of the center drill must always be 60° so that the countersunk hole will fit the angle of the lathe center point. If a center drill is not available, center the work with a small twist drill. Let the drill enter the work a sufficient distance on each end; then follow with a 60° countersink.

In center drilling, use a drop or two of oil on the drill. Feed the drill slowly and carefully to prevent breaking the tip. Take extreme care when the work is heavy, because you will be less able to “feel” the proper feed of the work on the center drill.

If the center drill breaks during countersinking and part of the broken drill remains in the work, you must remove this part. Sometimes you can drive the broken piece out by a chisel or by jarring it loose, but it may stick so hard that you cannot remove it this way. Then you must anneal the broken part of the drill and drill it out.

We cannot overemphasize the importance of proper center holes in the work and a correct angle on the point of the lathe centers. To do an accurate job between centers on the lathe, you must ensure that the center-drilled holes are the proper size and depth and that the points of the lathe centers are true and accurate.

Holding Work on a Mandrel

Many parts, such as bushings, gears, collars, and pulleys, require all the finished external surfaces to run true with their center hole, or bore.

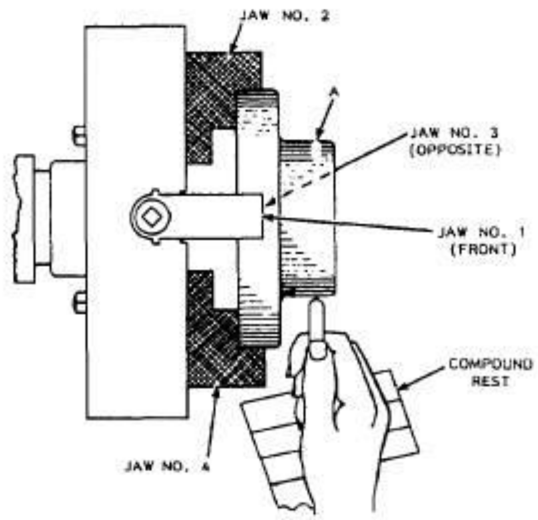
General practice is to finish the bore to a standard size within the limit of the accuracy desired. Thus a 3/4-inch standard bore would have a finished diameter of from 0.7495 to 0.7505 inch This variation is due to a tolerance of 0.0005 inch below and above the true standard of exactly 0.750 inch. First drill the hole to within a few thousandths of an inch of the finished size; then remove the remainder of the material with a machine reamer, following with a hand reamer if the limits are extremely close.

Then press the piece on a mandrel tightly enough so the work will not slip while being machined Clamp a dog on the mandrel, which is mounted between centers. Since the mandrel surface runs true with respect to the lathe axis, the turned surfaces of the work on the mandrel will be true with respect to the bore of the piece. A mandrel is simply a round piece of steel of convenient length which has been center drilled and ground true with the center holes. Commercial mandrels are made of tool steel, hardened and ground with a slight taper (usually 0.0005 inch per inch). This taper allows the standard hole in the work to vary according to the usual shop practice and still provides a drive to the work when the mandrel is pressed into the hole. The taper is not great enough to distort the hole in the work The center-drilled centers of the mandrel are lapped for accuracy. The ends are turned smaller than the body of the mandrel and provided with flats, which give a driving surface for the lathe dog.

Holding Work in Chucks

The independent chuck and universal chuck are used more often than other work-holding devices in lathe operations. The universal chuck is used for holding relatively true cylindrical work when the time required to do the job is more important than the concentricity of the machined surface and the holding power of the chuck When the work is irregular in shape, must be accurately centered, or must be held securely for heavy feeds and depth of cuts, an independent chuck is used. FOUR- JAW INDEPENDENT CHUCK.-Figure 9-23 shows a rough cylindrical casting mounted in a four-jaw independent lathe chuck on the spindle of the lathe. Before truing the work, determine which part you wish to have turned true. To mount this casting in the chuck, proceed as follows:

1. Adjust the chuck jaws to receive the casting. The same point on each jaw should touch the same ring on the face of the chuck If there are no rings, put each jaw the same distance from the outside edge of the body of the chuck.



Work mounted in a four-jaw chuck.

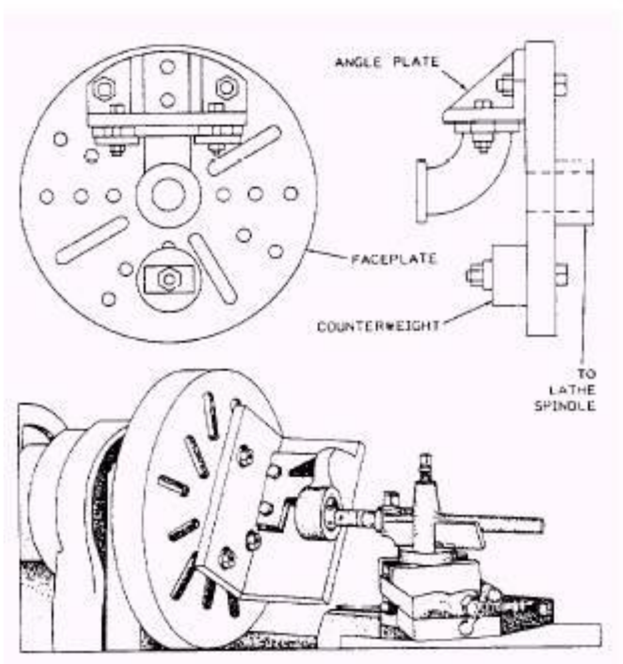
2. Fasten the work in the chuck by turning the adjusting screw on jaw 1 and then on jaw 3, a pair of jaws which are opposite each other. Next, tighten jaws 2 and 4.
3. At this stage the work should be held in the jaws just tightly enough so it will not fall out of the chuck while you turn it.
4. Revolve the spindle slowly by hand and, with a piece of chalk, mark the high spot on the work while it is revolving. Steady your hand on the tool post while holding the chalk.
5. Stop the spindle. Locate the high spot on the work and move the high spot toward the center of the chuck by releasing the jaw opposite the chalk mark and tightening the one nearest the mark

Sometimes the high spot on the work will be located between adjacent jaws. In that case, loosen the two opposite jaws and tighten the jaws adjacent to the

Holding Work on a Faceplate

A faceplate is used for mounting work that cannot be chucked or turned between centers because of its size or shape.

Work is secured to the faceplate by bolts, clamps, or any suitable clamping means. The holes and slots in the faceplate are used for anchoring the holding bolts. Angle plates may be used to position the work at the desired angle, as shown in figure 9-24. Note the counterweight added for balance.



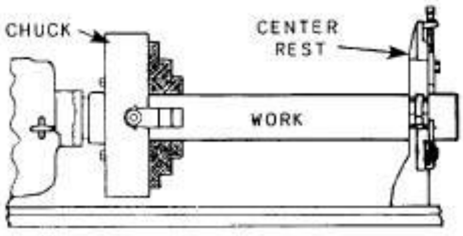
Work clamped to an angle plate.

For work to be mounted accurately on a faceplate, the surface of the work in contact with the faceplate must be accurately faced. It is good practice to place a piece of paper between the work and the faceplate to prevent slipping.

Before you clamp the work securely, move it about on the surface of the faceplate until the point to be machined is centered accurately with the axis of the lathe. Suppose you wish to bore a hole, the center of which has been laid out and marked with a prick punch. First, clamp the work to the approximate position on the faceplate. Slide the tailstock up until the dead center just touches the work.

Using the Center Rest and Follower Rest

Place the center rest on the ways where it will give the greatest support to the workpiece. This is usually at about the middle of its length.



Work mounted in a chuck and center rest.

Ensure that the jaws of the center rest are adjusted to support the work while allowing it to turn freely. Figure shows how a chuck and center rest are used for machining the end of a workpiece.

The follower rest differs from the center rest in that it moves with the carriage and provides support against the forces of the cut only. Set the tool to the diameter selected, and turn a "spot" about 5/8 to 3/4 inch wide. Then adjust the follower rest jaws to the finished diameter to follow the tool along the entire length to be turned.

Use a thick oil on the center rest and follower rest to prevent "seizing" and scoring of the workpiece. Check the jaws frequently to see that they do not become hot. The jaws may expand slightly if they get hot, pushing the work out of alignment (when using the follower rest) or binding (when using the center rest).

Holding Work in a Draw-In Collet Chuck

The draw-in collet chuck is used for very fine, accurate work of small diameter. Long work can be passed through the hollow drawbar. Short work can be placed directly into the collet from the front. The collet is tightened on the work by rotating the drawbar to the right; this draws the collet into the tapered closing sleeve. The opposite operation releases the collet. Accurate results are obtained when the diameter of the work is exactly the same size as the dimension stamped on the collet. In some cases, the diameter may vary as much as 0.002 inch; that is, the work may be 0.001 inch smaller or larger than the collet size. If the work diameter varies more than this, it will impair the accuracy and efficiency of the collet. That is why a separate collet should be used for each small variation or work diameter, especially if precision is desired.

MACHINING OPERATIONS

FACING

Facing is the machining of the end surfaces and shoulders of a workpiece. In addition to squaring the ends of the work, facing provides a way to cut work to length accurately. Generally, only light cuts are required since the work will have been cut to approximate length or rough machined to the shoulder.

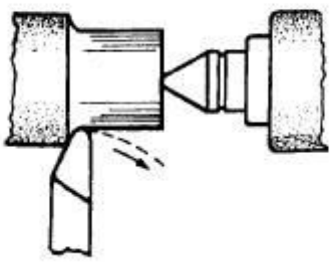
TURNING

Turning is the machining of excess stock from the periphery of the workpiece to reduce the diameter. In most lathe machining operations requiring removal of large amounts of stock, a series of roughing cuts is taken to remove most of the excess stock Then a finishing cut is taken to accurately "size" the workpiece.

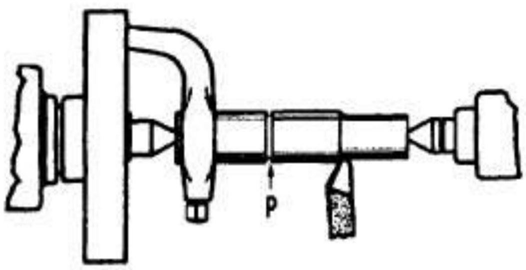
Rough Turning

When a great deal of stock is to be removed, you should take heavy cuts to complete the job in the least possible time. This is called rough turning. Select the proper tool for taking a heavy chip. The speed of the work and the amount of feed of the tool should be as great as the tool will stand.

When you take a roughing cut on steel, cast iron, or any other metal that has a scale on its surface, be sure to set the tool deep enough to get under the scale in the first cut. Unless you do, the scale on the metal will dull or break the point of the tool.



Position of the tool for a heavy cut.



Machining to a shoulder.

Finish Turning

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

If you must finish the work to close tolerances, be sure the work is not hot when you take the finish cut. If you turn the workpiece to exact size when it is hot, it will be undersize when it has cooled.

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

Turning to a Shoulder

Machining to a shoulder is often done by locating the shoulder with a parting tool. Insert the parting tool about 1/32 inch from the shoulder line toward the small diameter end of the work Cut to a depth 1/32 inch larger than the small diameter of the work. Then machine the stock by taking heavy chips up to the shoulder. This procedure eliminates detailed measuring and speeds up production.

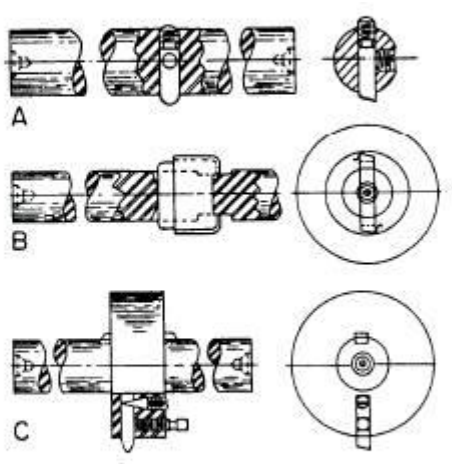
Figure 9-29 illustrates this method of shouldering. A parting tool has been used at P and the turning tool is taking a chip. It will be unnecessary to waste any time in taking measurements. You can devote your time to rough machining until the necessary stock is removed. Then you can take a finishing cut to accurate measurement.

Boring

Boring is the machining of holes or any interior cylindrical surface. The piece to be bored must have a drilled or cored hole, and the hole must be large enough to insert the tool. The boring process merely enlarges the hole to the desired size or shape. The advantage of boring is that a true round hole is obtained, and two or more holes of the same or different diameters may be bored at one setting, thus ensuring absolute alignment of the axis of the holes.

Work to be bored may be held in a chuck, bolted to the faceplate, or bolted to the carriage. Long pieces must be supported at the free end in a center rest. When the boring tool is fed into the hole of work being rotated on a chuck or faceplate, the process is called single point boring. It is the same as turning except that the cutting chip is taken from the inside. The cutting edge of the boring tool resembles that of a turning tool. Boring tools may be the solid forged type or the inserted cutter bit type.

When the work to be bored is clamped to the top of the carriage, a boring bar is held between centers and driven by a dog. The work is fed to the tool by the automatic longitudinal feed of the carriage. Three types of boring bars are shown in figure 9-30. Note the center holes at the ends to fit the lathe centers.



Boring bars

TAPERS

A taper is the gradual decrease in the diameter of a piece of work toward one end. The amount of taper in any given length of work is found by subtracting the size of the small end from the size of the large end. Taper is usually expressed as the amount of taper per foot of length or taper per inch of length.

Exercise: -

PLANER AND PLANER OPERATIONS

PLANER

A **planer** is a type of metalworking machine tool that is some what similar to a shaper, but larger, and with the entire workpiece moving beneath the cutter, instead of the cutter moving above a stationary workpiece. The work table is moved back and forth on the bed beneath the cutting head either by mechanical means, such as a rack and pinion gear, or by a hydraulic cylinder.

Planers and shapers are used generally for two types of work: generating accurate flat surfaces and cutting slots.

Classification of Planer

1. Double Housing Planer
2. Open Side Planer
3. Edge Type Planer
4. Divide Table Planer
5. Pit Type Planer

- **Double Housing Planer**

It is the most common type of planer. It consists of mainly a massive bed on which the worktable reciprocates, and two vertical columns or housing, one on each side of the bed. Each column carries a tool head that can be slide up and down on the column. A cross rail fitted between the two columns may carry one or two tool heads that can slide horizontally on the cross rail. All the tool heads can be clamped in position, and can be used collectively or individually depending on the requirements.

- **Open Side Planer**

Open side planer consists of only one vertical column or housing on which the cross rail is mounted. The column and the cross rail carry single and double tool heads respectively. This type of machine permits machining of wide work pieces.

- **Edge Type Planer**

This type of machine is used for machining the edges of heavy workpieces. The workpiece is clamped on the bed and the side mounted carriage supporting the cutting tool is reciprocated along the edge of the workpiece. Cutting can take place during both directions of carriage travel.

- **Divide Type Planer**

It is also called tandem planer and consists of two worktables, which may be reciprocated together or separately. It is quite well known that, mounting and setting of workpieces on the worktable consumes more time thereby restricting the machine for continuous mass production applications. In such cases, divided table planer can be used, where in, one worktable can be used for setting up a new workpiece, while the second worktable carrying workpiece is being machined. The two tables can be joined together to hold large workpieces.

- **Pit Type Planer**

A pit type planer differs from other planer in the sense that, the table and the work piece resting on it remain stationary and the tool reciprocates across the work surface. This type of machine is preferred for very large work, where the weight of the work piece and the tool required table would make reciprocating movement difficult. The job is either mounted on a stationary table, or on the floor inside a pit, and hence the name pit type planer. One or two tool heads can be mounted on the cross rail and two side tool posts the housings, if required. The entire unit travels along the horizontal ways and fro and, thus the tool moves past the work surface during operation.

PLANING MACHINE PARTS

The principle parts of the planer are:

- Bed
- Table
- Column
- Cross rail
- Tool head

- **BED**

1. The bed of a planer is large in size and heavy in weight.
2. It supports the column and all other moving parts of the machine.
3. It is made slightly longer than twice the length of the table so that the full length of the table may be moved on it.

TABLE

1. The table supports the work and reciprocates along the ways of the bed.
2. Table is made from good quality cast iron.
3. The top face of the table is accurately finished in order to locate the work correctly.
4. T-slots are provided on the entire length of the table so that the work and work holding devices may be bolted upon it.

COLUMN

1. These are rigid box-like vertical structures placed on each side of the bed and are fastened to the sides of the bed.
2. They are heavily ribbed to trace up severe forces due to cutting.
3. The cross rail may be made to slide up and down for accommodating different heights of work

CROSSRAIL

1. It is a rigid box-like casting connecting the two columns.
2. It may be raised or lowered on the face of the housing and can be clamped at a desired position by manual or electrical clamping devices.
3. It should remain absolutely parallel to the top surface of the table.
4. It is necessary to generate a flat horizontal surface on a work piece because the tool follows the part on the cross rail during cross feed.

TOOLHEAD

1. Tool heads are mounted on the cross rail by a saddle.
2. The saddle may be made to move transversely on the cross rail to give cross feed.
3. The swivel base is pivoted on the saddle and is graduated on each side to 60 degrees.
4. The clapper block is hinged at hinge pins to the clapper block and it holds the tool post in which the tool is clamped by straps.

PLANER OPERATIONS

- Planning flat horizontal surfaces.
- Planning vertical surfaces.
- Planning at an angle and machining dovetails.
- Planning curved surfaces.
- Planning slots and grooves.

Planning horizontal surfaces:-

While machining horizontal surface, the work is given a reciprocating movement along with the table and the tool is fed crosswise to complete the cut. Both the railheads may be used for simultaneous removal of the metal from two cutting edges.

Planning vertical surfaces:-

The vertical surface of a work is planed by adjusting the saddle horizontally along the cross rail until the tool is in a position to give the required depth of cut. The vertical slide is adjusted perpendicular to the planer table and the apron is swiveled in a direction so that the tool will swing clear out of the machined surface during the return stroke.

Planning angular surfaces:-

For dovetail work, cutting v –grooves etc. the tool head is swiveled to the required angle and the apron is then further swiveled away from the work to give relief to the tool cutting edges during the return stroke.

Planning curved surfaces:-

This illustrates simple method of planning a concave surface with the aid of a special fixture consisting of a radius arm and a bracket. the bracket is connected to the cross member attached to the two housings. One end of the radius arm is pivoted on the bracket and the other end to the vertical slide of the tool head. The down feed screw of the tool head is removed.

Planning slots or grooves:-

Slots or grooves are cut by using slotting tools. The operations is similar to that of a shaper.

CUTTING SPEED ,FEED AND DEPTH OF CUT

CUTTING SPEED:

The cutting speed of a planner is the rate at the metal is removed during the forward cutting stroke.

FEED:-

The feed in planning machine is the distance the tool head travels at the beginning of each cutting stroke expressed in mm per double stroke.

DEPTH OF CUT:-

It is the thickness of metal removed in one cut and is measured by the perpendicular distance between the machined and no machined surface expressed in mm.

MACHINING TIME:-

The cutting speed, feed, length of cutting stroke, breadth of the job and number of double strokes per minute for a planer operation are known, the machining time required for one complete cut may be calculated.

MILLING MACHINE AND MILLING OPERATIONS

Milling Machine

A milling machine is a power driven machine that cuts by means of a multitooth rotating cutter. The mill is constructed in such a manner that the fixed workpiece is fed into the rotating cutter. Varieties of cutters and holding devices allow a wide rage of cutting possibilities.

Mill Construction:-

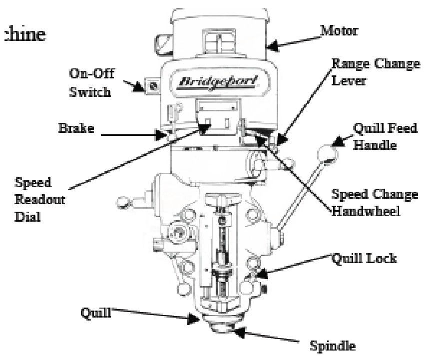
The vertical milling machine is made up of five major groups: base and column, knee, saddle, table, and head. The base and column are one piece that forms the major structural component of the milling machine. They are cast integrally, ad provide the mill with its stability and rigidity. The front of the column has a machined face which provides the ways for the vertical movement of the knee.

The knee supports the saddle and table. It contains the controls for raising and lowering the saddle. Sitting atop the knee is the saddle which supports the table. The saddle slides in dovetailed grooves into and away from the machine, providing the mill with its Y-axis movement. On top of the saddle sits the table.

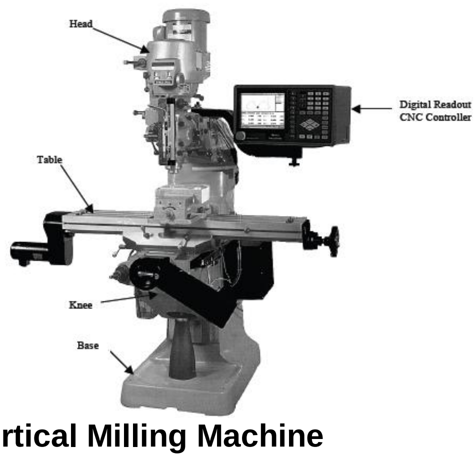
Being moved side-to-side, left-right, over the saddle furnishes the mill with its X-axis movement. The workpiece is secured to the table through the use of various types of holding devices.

The head is the most complex assembly in the major parts groups. This contains the following components:

1. The drive motor and on/off switch.
2. Drive belt, gear train, and range lever selector.
3. Quill, spindle, and draw bar.
4. Quill feed, lock, and digital depth read out (Z-axis).



Vertical Milling Machine-Head



Vertical Milling Machine

Tooling-

End mills are the most common cutter used on the vertical milling machine. They are extremely versatile in that they can be used for surface cuts, slotting, and side (or profiling) cuts. End mills come in many types, each being suited for a particular application. End mills are fluted, much like drills, and the number of flutes determines what the end mill can do. Two fluted end mills are used for machining aluminum, and are favored for plunge cuts.

Four fluted end mills are used in machining the harder metals like steel. Generally, it is not a good idea to use a four fluted end mill when machining aluminum or brass however, since the flutes can fill with material, and no longer cut. The other main characteristic of the end mill is the cutting end of the tool its self. Some end mills are bottom cutting, meaning they can be plunged into material much like a drill, while some are not, and are only useful for cutting on the side.

Be careful not to plunge an end mill that is not bottom cutting. Other types of end mills includes the ball end mill, which has a radiused end used to produce a fillet, and corner radiusing end mills, used to round the edges of a workpiece.

Other types of cutters include slitting saws for cutting grooves, shell milling cutters for faster milling of surfaces than is possible with an end mill, flycutters (which are single point cutters for facing large workpieces), formed cutters for cutting special shapes like gears; and groove cutters like T-slot and dovetail cutters.

Milling cutters are expensive and easily ruined if not taken care of when using or storing. Failure to obtain satisfactory results on a job can many times be attributed to inappropriate selection of the proper milling cutter.

Machining Operations:-

Once the preliminary operations and selections have been accomplished, a quick check should be made to be sure that work and fixtures will clear any parts of the machine, and that the cutter will not strike the table or fixtures. All table movements that will not be used on a cut should be locked, and those that will be used should be unlocked. The head controls should be checked for proper range and speed.

When starting the motor, make certain the cutter is rotating in the proper direction. Do not stop the cutter in mid cut and make no adjustments with the cutter in contact with the workpiece. There are two types of milling to be discussed. Conventional milling is where the workpiece is fed opposite the direction of the rotation of the cutter, and climb milling is when the workpiece is fed in the direction of rotation of the cutter. Each has its own advantages and disadvantages.

Climb milling draws the part into the cutter, and can violently take up any backlash in the table. However, it does produce a smoother finish.

Conventional milling is the more preferred method, and will be used for every cut except the finishing cut.

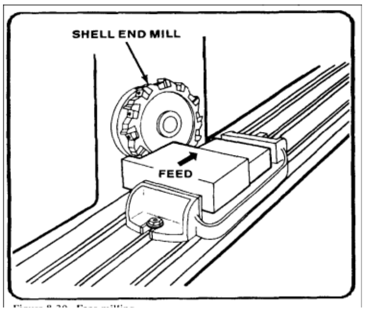
When using an end mill, there are certain general rules that should be followed when making cuts.

1. The greatest depth of cut should never be more than 1/2 the diameter of the end mill.
2. Do not plunge an end mill more then 1-1/2 times its diameter. This is also true for slotting.
3. Do not, in a single pass, cut a slot deeper than 1-1/2 its width.
4. Do not edge mill to a depth of more than 1-1/2 times the diameter of the cutter.

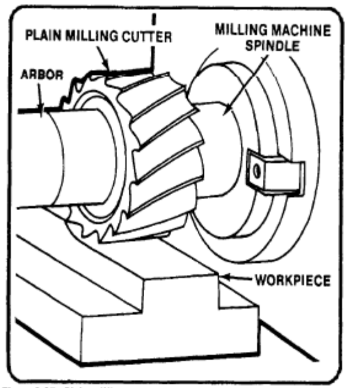
Milling Operations

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.

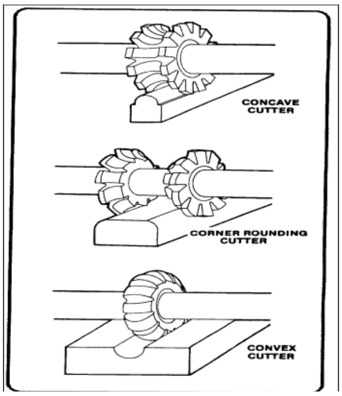


Plain milling, also called surface milling or slab milling, is milling flat surfaces with the milling cutter axis parallel to the surface being milled. Generally, plain milling is done with the workpiece surface mounted parallel to the surface of the milling machine table and the milling cutter mounted 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.

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

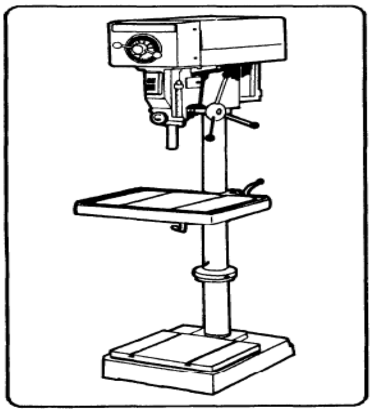
Explanatory names, such as sawing, slotting, gear cutting, and so forth have been given to special operations. Routing is a term applied to milling an irregular outline while controlling the workpiece movement by hand feed. Grooving reamers and taps is called fluting. Gang milling is the term applied 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.

DRILLING MACHINE AND DRILLING OPERATIONS

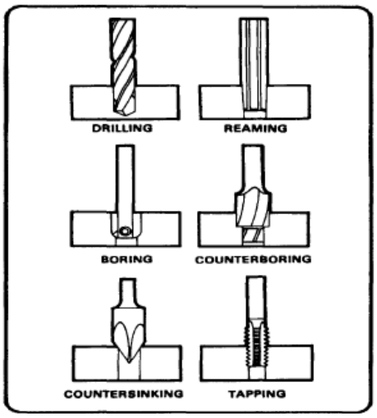
Drilling Machine: -

A drilling machine, called a drill press, is used to cut holes into or through metal, wood, or other materials.

Drilling machines use a drilling tool that has cutting edges at its point. This cutting tool is held in the drill press by a chuck or Morse taper and is rotated and fed into the work at variable speeds. Drilling machines may be used to perform other operations. They can perform countersinking, boring, counterboring, spot facing, reaming, and tapping



Upright Drilling Machine

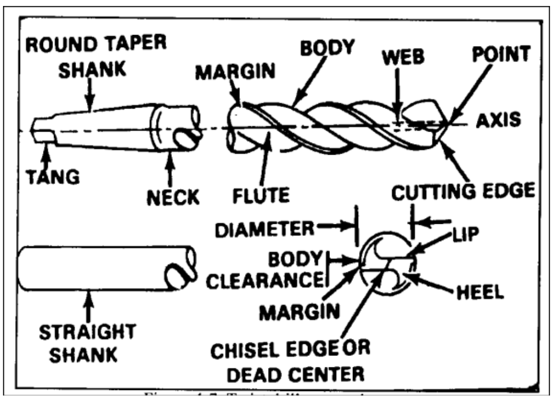


Operations of Upright Drilling Machine

TOOLS AND EQUIPMENTS

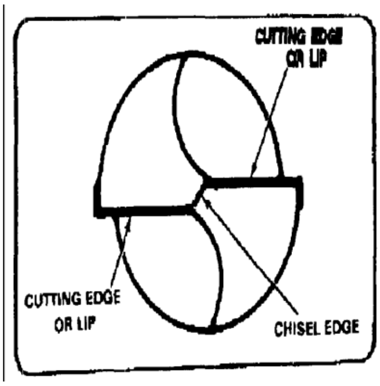
TWIST DRILLS

Twist drills are the most common cutting tools used with drilling machines. Twist drills are designed to make round holes quickly and accurately in all materials. They are called twist drills mainly because of the helical flutes or grooves that wind around the body from the point to the neck of the drill and appear to be twisted. Twist drills are simply constructed but designed very tough to withstand the high torque of turning, the downward pressure on the drill, and the high heat generated by friction.

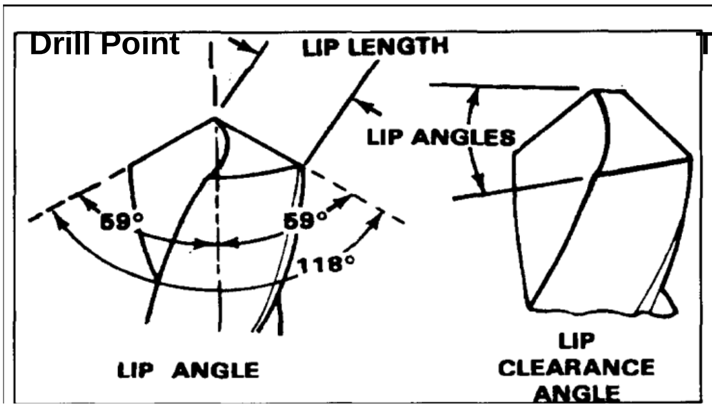


There are two common types of twist drills, high-speed steel drills, and carbide-tipped drills. The most common type used for field and maintenance shop work is the high-speed steel twist drill because of its low cost. Carbide-tipped metal drills are used in production work where the drill must remain sharp for extended periods, such as in a numerically controlled drilling machine.

Other types of drills available are: carbide tipped masonry drills, solid carbide drills, TiN coated drills, parabolic drills and split point drills. Twist drills are classified as straight shank or tapered shank. Straight shank twist drills are usually l/2-inch or smaller and tit into geared drill chucks, while tapered shank drills are usually for the larger drills that need more strength which is provided by the taper socket chucks.

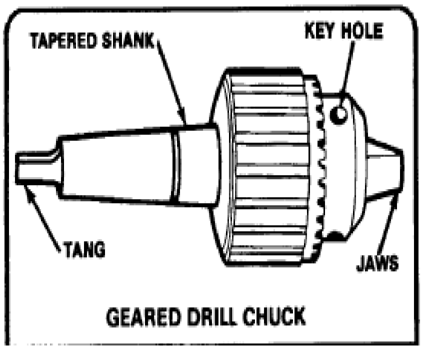


DRILL



HOLDING DEVICES

GEARED DRILL CHUCKS



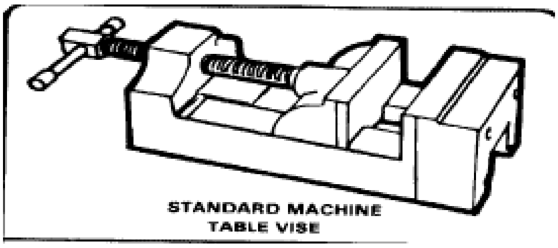
Drills with straight shanks are held in geared drill chucks which have three adjustable jaws to clamp onto the drill. Smaller size drills are made with straight shanks because of the extra cost of providing these sizes if tapered. Geared drill chucks come in various sizes, with the 3/8 or 1/2-inch capacity chuck being the most common. The shank of the chuck is set into the spindle of the drilling machine by inserting the chuck's shank into the spindle's internal taper and seating the shank into the taper with a light blow with a soft hammer. Both the internal and external taper surfaces must be clean and free of chips for the shank to seat and lock properly. The drill is locked into the chuck by using the chuck key to simultaneously tighten the three chuck jaws. Geared drill chucks can also come with a morse tapered shank and may have a different method of attaching They may screw on, have a Jarno taper, or a Jacob's back taper.

DRILL SOCKETS AND DRILL SLEEVES

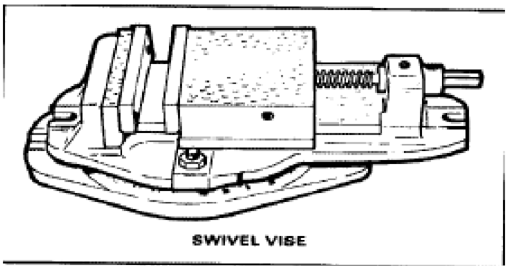
Morse taper shank drills come in several sizes, thus, adapters must be used for mounting them into various drilling machine spindles. Drill sleeves and drill sockets are designed to add to or subtract from the Morse taper for fitting a drill into the chuck spindle. For example, it is common for a 3/4 inch twist drill to have a Morse taper size. It is also common for a drilling machine spindle to have a Morse taper size, and it can be adapted for many other Morse taper sizes, depending on the size of the drill.

MACHINE TABLE VISES

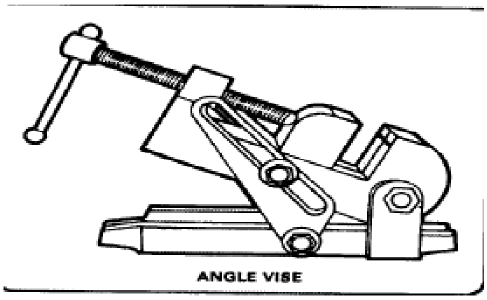
The standard machine table vise is the simplest of all vises. It is equipped with two precision ground jaws for holding onto the work and a lead screw to tighten the one movable jaw to the work.



The swivel vise is a machine vise that has an adjustable base that can swivel through 360° on a horizontal plane.



The angle vise is very similar to the table vise, except this vise can be tilted to 90° to be perpendicular to the work table.



GENERAL DRILLING OPERATIONS

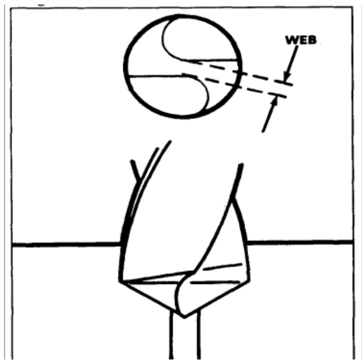
Drilling Deep Holes

If the depth of the hole being drilled is greater than four times the diameter of the drill, remove the drill from the workpiece at frequent intervals to clean the chips from the flutes of the drill and the hole being drilled. A slight increasing speed and decrease in feed is often used to give the chips a greater freedom of movement. In deep hole drilling, the flutes of the smaller drills will clog up very quickly and cause the drill to drag in the hole, causing the diameter of the hole to become larger than the drill diameter. The larger drills have larger flutes which carry away chips easier.

The depth of the hole being drilled is four times the diameter of the drill itself, remove the drill at frequent intervals and clean the chips from the flutes of the drill and from the hole being drilled.

Drilling a Pilot Hole

As the drill size increases, both the size of the web and the width of the chisel edge increase. The chisel edge of drill does not have a sharp cutting action, scraping rather than cutting occurs. In larger drills, this creates a considerable strain on the machine. To eliminate this strain when drilling a large hole, a pilot hole is drilled first and then followed with the larger drill. A drill whose diameter is wider than the web thickness of the large drill is used for the pilot hole. This hole should be drilled accurately as the larger drill will follow the small hole.



Pilot Drill

SPECIAL OPERATIONS ON DRILLING MACHINES

COUNTERSINKING

Countersinking is the tapering or beveling of the end of a hole with a conical cutter called a machine countersink. Often a hole is slightly countersunk to guide pins which are to be driven into the workpiece; but more commonly, countersinking is used to form recesses for flathead screws and is similar to counterboring.

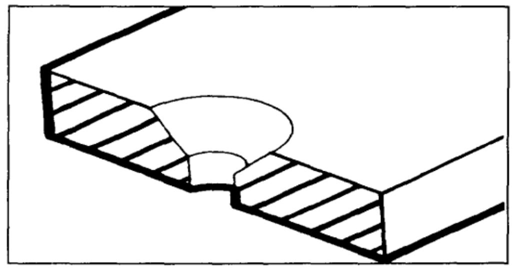


Figure 4-38. Countersunk hole.

COUNTERBORING AND SPOT FACING

Counterboring is the process of using a counterbore to enlarge the upper end of a hole to a predetermined depth and machine a square shoulder at that depth. Spot facing is the smoothing off and squaring of a rough or curved surface around a hole to permit level seating of washers, nuts, or bolt heads. Counterbored holes are primarily used to recess socket head cap screws and similar bolt heads slightly below the surface. Both counterboring and spotfacing can be accomplished with standard counterbore cutters.

Counterbore cutters have a pilot to guide the counterbore accurately into the hole to be enlarged. If a counterbore is used without a pilot, then the counterbore flutes will not stay in one spot, but will wander away from the desired hole. The shank of counterbores can be straight or tapered. The pilots of counterbores can be interchangeable with one another so that many hole combinations can be accomplished.

TAPPING

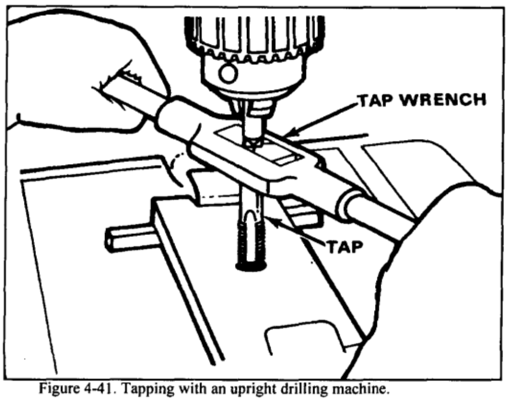


Figure 4-41. Tapping with an upright drilling machine.

Tapping is cutting a thread in a drilled hole. Tapping is accomplished on the drilling machine by selecting and drilling the tap drill size, then using the drilling machine chuck to hold and align the tap while it is turned by hand. The drilling machine is not a tapping machine, so it should not be used to power tap. To avoid breaking taps, ensure the tap aligns with the center axis of the hole, keep tap flutes clean to avoid jamming, and clean chips out of the bottom of the hole before attempting to tap.

REAMING

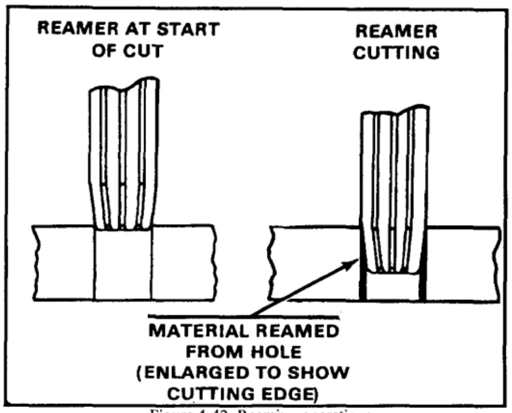


Figure 4-42. Reaming operations.

Reaming a drilled hole is another operation that can be performed on a drilling machine. It is difficult, if not impossible, to drill a hole to an exact standard diameter. When great accuracy is required, the holes are first drilled slightly undersized and then reamed to size. Reaming can be done on a drilling machine by using a hand reamer or using a machine reamer. When you must drill and ream a hole, it is best if the setup is not changed. For example, drill the hole (slightly undersized) and then ream the hole before moving to another hole. This method will ensure that the reamer is accurately aligned over the hole. If a previously drilled hole must be reamed, it must be accurately realigned under the machine spindle. Most hand and machine reamers have a slight chamfer at the tip to aid in alignment and starting.

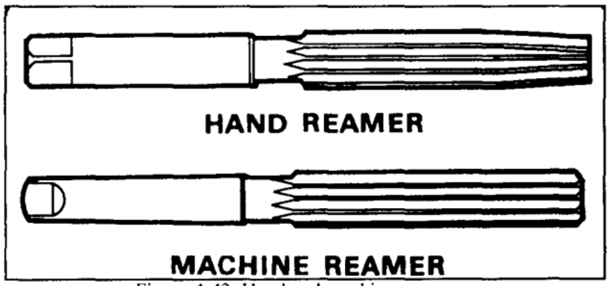


Figure 4-43. Hand and machine reamers.

BRAOCHING MACHINE AND BROACHING OPERATIONS

Broaching: -

- Broaching is a method of removing metal by pushing or pulling a cutting tool called a broach, which cuts in fixed path.
- Surface finished by this method may be flat or contoured and maybe either internal or external.
- The term broaching is derived from an ancient roman word BRACES, which meant an object having projecting teeth.

A broach is a multiple edges cutting tool that has successively higher cutting edges along the length of the tool.

Types of Broaches: -

- According to type of operation: - Internal & External
- According to Method of Operation: - Push & Pull
- According to Type of Construction: - solid, build-up, inserted, over lapping, double jump
- According to Function: - surface ,keyway, round roll, spiral, splint, burnishing etc.

Tools used in Broaching Machine: -

Broach Elements: -

- Pull end
- Front pilot
- Roughing and semi finished
- Finishing teeth
- Reer pilot and follower rest
- Land
- Back off or clearance angle
- Rake or hook angel(or face angel)
- Pitch

Broaching Machines: -

- Broaching machines are probably the simplest of all machine tools.
- They consist of a work holding fixture, a broaching tool, a drive mechanism and a suitable supporting frame.
- Several various of design are possible.
- Broaching machines usually pull or push the broach through or pass a work piece that is held in a fixture.

Types:-

1. **Horizontal broaching machines:**

Nearly horizontal machines are of pull type. They maybe used for either internal or external broaching. These are used for broaching keyways, splines, slots, round holes and other internal shapes or contours.

They have the disadvantage of taking more floor space than do the vertical machines.

2. **Vertical Broaching Machine:**

The vertical types maybe obtained either push or pull types. The push type is most popular. Vertical machines are employed in multiple operations since they are convenient to pass work from one machine to another and they are more likely to be found doing surface operations.

Vertical machines require an operator platform or a pit and are economical of floor space.

3. <u>Surface Broaching Machine</u>:

Surface broaching machines have their broaching tools attached to a ramp forced in a straight path along guide ways past the work piece. On some machines the ram moves horizontally on others vertically. When two ramps are used the machine is called a duplex broach.

4. <u>Continous Broaching Machine:</u>

For mass production of small parts the highly productive continuous broaching method is used on rotary or horizontally continuous broaching machine.

In Rotary broaching machines the work piece is loaded on the table which rotates continuously.

In Horizontal broaching machines the work pieces travel as they are carried by an endless chain.

Broaching Methods: -

Broaching, according to the method of operation maybe classified as follows:-

- Pull broaching
- Push broaching
- Surface broaching
- Continuous broaching

Pull broaching:

The work is held stationary and the broach is pulled through the work. Broaches are usually long and are held in a special head. This is used mostly for internal broaching but it can do some surface broaching.

Push broaching:

The work is held stationary and the broach is pushed through the work. Hand and hydraulic arbor presses are popular for push broaching. This method is used usually for sizing holes or cutting keyways.

Surface broaching:

This method has rapidly become an important means of surface finishing. Many irregular or intricate shapes can be broached by surface broaching. But the tools must be specially for each job.

Continuous broaching:

The work is moved continuously and the broach is held stationary. The path of movement may be straight, horizontal or circular. This method is very suitable for broaching a number of similar works at a time.

Broaching Operations: -

- Broaching is applied for machining various internal and external surfaces. For round or irregular shaped holes from 6 to 100 mm in diameter for external, flat and contour surfaces.
- Most broaching operations are completed in one pass but some are arranged for repeated cuts to simplify the design of the broach.
- Broaching a key in a hole with a keyway broach.
- Broaching hole with a round back.
- Surface broaching with a contour broach.
- Broaching a spline hole with a spline broach.

Advantages: -

- Rate of production is very high.
- Little skill is required to perform a broaching operation.
- High accuracy and a high class of surface finish is possible.
- Both roughing and finishing cuts are completed in one pass of the tool.
- The process can be used for either internal or external surface finishing
- Any form that can be reproduced on a broaching can be machined.
- Cutting fluid maybe readily applied where it is most effective.

Limitations: -

- High tool cost.
- Very large work pieces cannot be broached.
- The surfaces to be broached cannot have an obstruction.
- Broaching cannot be used for the removal of a large amount of stock.
- Parts to be broached must be capable of being rigid and must withstand the forces that set up during cutting.

SLOTTING MACHINE AND SLOTTING OPERATIONS

- The slotting machine was developed by Brunel in 1800 much earlier than a shaper was invented.
- The slotting machine falls under the category of reciprocating type of machine tool similar to a shaper or a planer.
- The major difference between a slotter and a shaper is that the ram holding the tool moves in a vertical axis in a slotter and in a horizontal axis in a shaper.

Types of Slotting Machine: -

PUNCHER SLOTTER

- The puncher slotter is a heavy rigid machine designed for removal of large amount of metal from large forgings or castings.
- The puncher slotter ram is usually driven by a spiral pinion meshing with the rack teeth cut on the underside of the ram.
- The pinion is driven by a variable speed reversible electric motor similar to that of a planer.
- The feed is also controlled by electrical gears.

PRECISION SLOTTER

- The precision slotter is a lighter machine and is operated at high speeds.
- The machine is designed to take light cuts giving accurate finish.

- Using special jigs, the machine can handle a number of identical works on a production basis.
- The machine is used for general purpose work. They are usually fitted with Whitworth quick return mechanism.

Slotter Size: -

- The size of a slotter like that of a shaper is specified by the maximum length of stroke of the ram, expressed in mm.
- The size of a general purpose or precision slotter usually ranges from 80 to 900 mm.
- To specify a slotter correctly the diameter of the table in mm, amount of cross and longitudinal travel of the table expressed in mm, number of speeds and feeds available, h.p of the motor, floor space required etc should also be stated.

Cutting Speed, Feed And Depth Of Cut:-

CUTTING SPEED :

The cutting speed of a slotter is defined by the rate with which the metal is removed during downward cutting stroke and is expressed in meters per minute.

FEED :

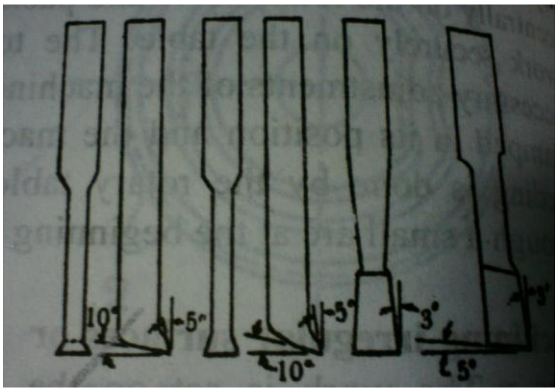
It is the movement of the work per double stroke expressed in mm.

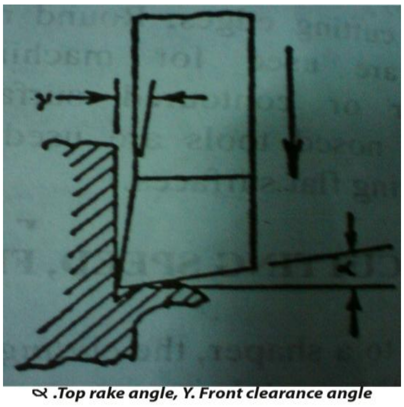
DEPTH OF CUT:

It is the perpendicular distance measured between the machined surface and unmachined expressed in mm.

Slotter Tools: -

- A slotting machine tool differs widely from a shaper or a planner tool. A slotter removes metal during its vertical cutting stroke .
- This changed cutting condition presents a lot of difference in the tool shape .In a slotter the pressure acts along the length of the tool.
- The rake and the clearance angle of a slotter tool apparently look different from a lathe or a shaper tool as these angles are determined with respect to a vertical plane rather than the horizontal.
- Slotter tools are provided with top rake, front clearance and side clearance but no side rake is given.
- The nose of the tool projects slightly beyond the shank to provide clearance. The amount of rake angle given is similar to that of a shaper tool.
- The slotter tools are robust in cross-section and are usually of forged type; of course, bit type tools fitted in heavy duty tool holders are also used.





∝ .Top rake angle, Y. Front clearance angle

Slotting Machine Parts: -

- Base
- Column or Pillar
- Saddle
- Table and Cross Slide
- Rotating Table
- Ram and Tool head Assembly
- Gear Box

Base:

The base is rigidly built to take up all the cutting forces and entire load of the machine.

The top of the bed is accurately finished to provide guide ways on which the saddle is mounted. The guide ways are perpendicular to the column face.

Column or Pillar:

The column is the vertical member which is cast integral with the base and houses driving mechanism. It is also called the pillar.

The front vertical face of the column is accurately finished for providing ways on which the ram reciprocates.

Saddle:

The saddle is the entire unit which is mounted upon the guide ways and may be moved toward or away from the column either by power or manual control to supply longitudinal feed to the work.
The top face of the saddle is accurately finished to provide guide ways for the cross-slide. These guide ways are perpendicular to the guide ways on the base.

Rotary Table:

The rotary table is a circular table which is mounted on the top of the cross-slide. The table may be rotated by rotating a worm which meshes with a worm gear connected to the underside of the table.

The rotation of the table may be effected either by hand or power. In some machines the table is graduated in degrees that enable the table to be rotated for indexing or dividing the periphery of a job in equal number of parts.

T-slots are cut on the table for holding the work by different clamping devices. The rotary table enables a circular or contoured surface to be generated on the work piece. The table has grades on it.

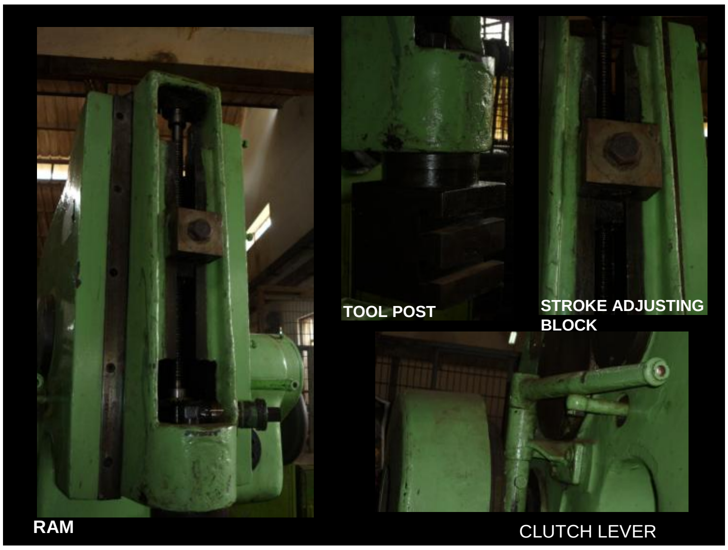
Cross slide:

The cross slide is mounted upon the guide ways of the saddle and maybe moved parallel to the face of the column. The movement of the slide maybe controlled by hand or power to supply cross-feed.

The rotating wheel provided on the cross slide can be controlled manually to guide the cross slides. The top one is called a table. And the bottom is called a cross slide.

Ram and Tool Head Assembly:

- The ram is the reciprocating member of the machine mounted on the guide ways of the column. It supports the tool at its bottom end on a tool head. A slot is cut on the ram for changing the position of stroke. In some machines, special types of tool-holders are provided to relieve the tool during its return stroke.
- Stroke adjusting block is adjusted by changing the bevel gauges using a square shaft. This helps in adjusting the position of the bevel gauges, adjusting the stroke.
- Tool Head or Tool Post helps in holding the cutting tool.
- Clutch Lever is used for disengaging the machine for a while.



Gear Box:

- The gear box has a gear lever which is used for changing the speed of the movement of the ram.
- Minimum strokes per minute are 30 while maximum is 120.
- To remove less material the movement is made faster.
- For softer materials like brass and copper the stroke movement is preferably faster than one for the hard materials.