PDHonline Course M 515 (7PDH)

General Machining Operations Best Practices

Instructor: Jurandir Primo, PE

2014

PDH Online | PDH Center

5272 Meadow Estates Drive
Fairfax, VA 22030-6658
Phone: 703-988-0088

www.PDHonline.com
www.PDHcenter.org

GENERAL MACHINING OPERATIONS – BEST PRACTICES

CONTENTS:

DRILLING MACHINES:

A drilling machine, also called as a drill press, is used to cut holes into or through metal, wood, or other materials (Figure 4-1). 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 and may be used to perform other operations as countersinking, boring, counter-boring, spot facing, reaming, and tapping.

Drill press operators must know how to set up the work, set speed and feed, and provide for coolant to get an acceptable finished product. The size or capacity of the drilling machine is usually determined by the largest piece of stock that can be center-drilled (Figure 4-2). For instance, a 15-inch drilling machines can center-drill a 30-inch-diameter piece of stock. Other ways to determine the size of the drill press are by the largest hole that can be drilled, the distance between the spindle and column, and the vertical distance between the worktable and spindle.

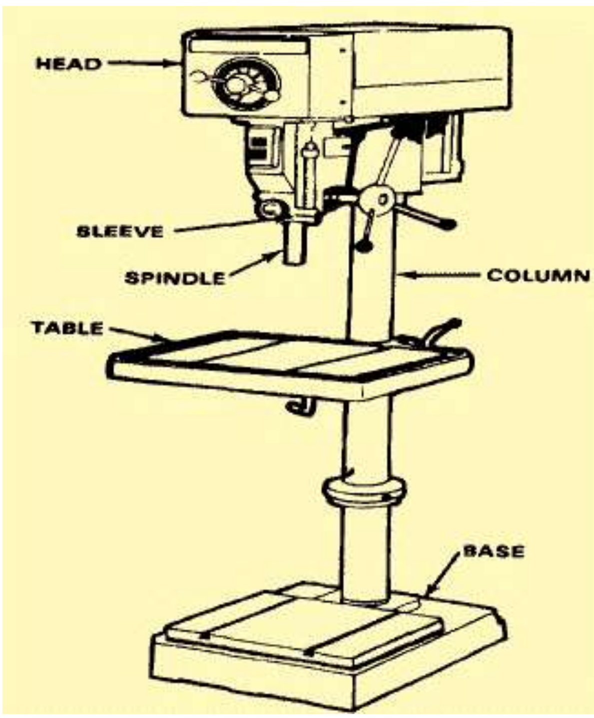


Figure 4-1. Upright drilling machine.

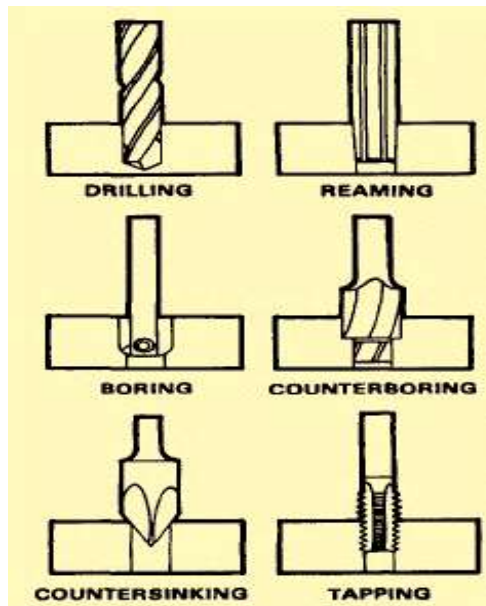


Figure 4-2. Operations with an upright drilling machine.

1) Characteristics:

The spindle holds the drill or cutting tools and revolves in a fixed position in a sleeve. In most drilling machines, the spindle is vertical and the work is supported on a horizontal table. The sleeve or quill assembly does not revolve but may slide in its bearing in a direction parallel to its axis. When the sleeve carrying the spindle with a cutting tool is lowered, the cutting tool is fed into the work: and when it is moved upward, the cutting tool is withdrawn from the work.

Feed pressure applied to the sleeve by hand or power causes the revolving drill to cut its way into the work a few thousandths of an inch per revolution. The column of most drill presses is circular and built rugged and solid. The column supports the head and the sleeve or quill assembly. The head of the drill press is composed of the sleeve, spindle, electric motor, and feed mechanism. The head is bolted to the column.

The worktable is supported on an arm mounted to the column. The worktable can be adjusted vertically to accommodate different heights of work or it may be swung completely out of the way. It may be tilted up to 90° in either direction, to allow for long pieces to be end or angled drilled. The base of the drilling machine supports the entire machine and when bolted to the floor, provides for vibration-free operation and best machining accuracy. The top of the base is similar to a worktable and maybe equipped with T-slots for mounting work too large for the table.

Lubrication: Lubrication is important because of the heat and friction generated by the moving parts. Follow the manufacturer's manual for proper lubrication methods. Clean each machine after use. Clean T-slots grooves and dirt from belts and pulleys. Remove chips to avoid damage to moving parts. Wipe all spindles and sleeves free of grit to avoid damaging the precision fit. Put a light coat of oil on all unpainted surfaces to prevent rust. Operate all machines with care to avoid overworking the electric motor.

Operation: The operator of the drill machine must use a sense of feel while feeding the cutting tool into the work. The operator must pay attention and be alert, to when the drill breaks through the work, because of the tendency of the drill to grab or snag the workpiece, wrenching it free of its holding device. Due to the high speed of these machines, operations that require higher drilling speeds cannot be performed less than 450 RPM. Reaming, counterboring, and counter-sinking may require slower speeds than drilling and may not be able to be performed for all materials on these machines.

2) Types of Drilling Machines:

The most common of drilling machines used by maintenance for repairing and fabricating needed parts, are: hand-feed and power-feed. Other types of drilling machines, such as the radial drill press, CNC drilling machine, multiple spindle drilling machine, gang drilling machine, and turret drill press, are all variations of the basic hand and power-feed drilling machines. They are designed for high-speed production and industrial shops.

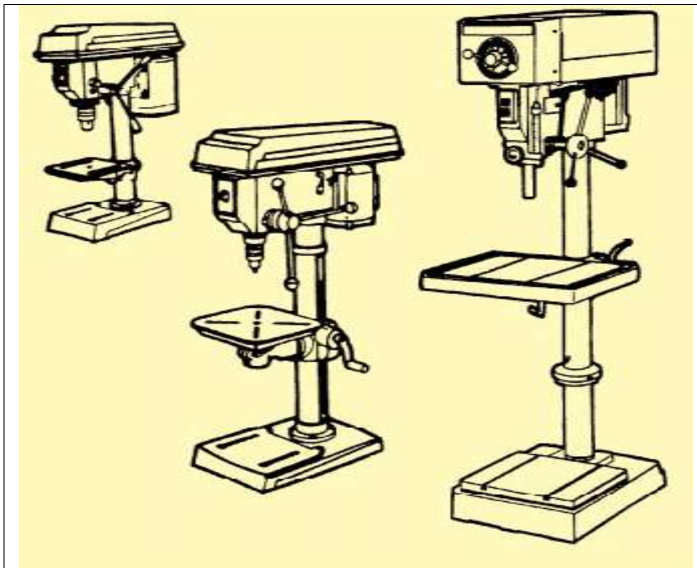


Figure 4-5. Hand-feed drilling machines.

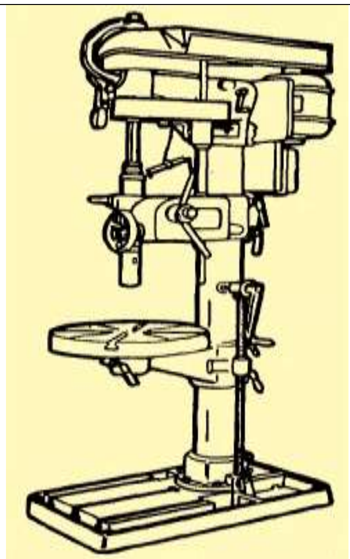


Figure 4-6. Power-feed drilling machine.

Hand-Feed: The hand-feed drilling machines (Figure 4-5) are the simplest and most common type of drilling machines in use today. These drilling machines can be bench or floor mounted. They are driven by an electric motor that turns a drive belt on a motor pulley that connects to the spindle pulley. Hand-feed machines are essentially high-speed machines and are used on small workplaces that require holes 1/2 inch or smaller. Normally, the head can be moved up and down on the column by loosening the locking bolts, which allows the drilling machine to drill different heights of work.

Power-Feed: The power-feed drilling machines (Figure 4-6) are usually **larger and heavier** than the hand-feed. They are equipped with the ability to feed the cutting tool into the work automatically, at a preset depth of cut per revolution of the spindle, usually in thousandths of an inch per revolution.

These machines are used in maintenance shops for medium-duty work, or work that uses large drills that require power feeds. The power-feed capability is needed for drills or cutting took that are over 1/2 inch in diameter, because they require more force to cut than that which can be provided by using hand pressure. The speeds available on power-feed machines can vary from about 50 RPM to about 1,800 RPM. The slower speeds allow for special operations, such as counterboring, countersinking, and reaming.

The sizes of these machines generally range from 17-inch to a 22-inch center-drilling capacity, and are usually floor mounted. They can handle drills up to 2 inches in diameter, which mount into tapered Morse sockets. Larger workplaces are usually clamped directly to the table or base using T-bolts and clamps, while small workplaces are held in a vise. A depth-stop mechanism is located on the head, near the spindle, to aid in drilling to a precise depth.

3) Tools and Characteristics:

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 (Figure 4-7). 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 tipped 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** (Figure 4-7). Straight shank twist drills are usually l/2-inch or smaller and usually mounted into geared drill chucks. Tapered shank drills are usually larger, because need more mechanical strength, commonly mounted in tapered socket chucks. Common twist drill sizes range from 0.0135 (wire gage size No. 80) to 3.500 inches in diameter.

Larger holes are cut by special drills that are not considered as twist drills. The standard sizes used in the United States are the wire gage numbered drills, letter drills, fractional drills, and metric drills (See Fig. 4-8). Twist drills can also be classified by the diameter and length of the shank and by the length of the fluted portion of the twist drill.

Wire gage twist drills and letter twist drills are generally used where other than standard fractional sizes are required, such as drilling holes for tapping. In this case, the drilled hole forms the minor diameter of the thread to be cut, and the major diameter which is cut by tapping corresponds to the common fractional size of the screw.

Wire gage twist drills range from the smallest to the largest size; from No 80 (0.01 35 inch) to No 1 (0.2280 inch), the larger the number, the smaller the diameter of the drill. Letter size twist drills range from A (0.234 inch) to Z (0.413 inch). As the letters progress, the diameters become larger. Fractional drills range from 1/64 to 1 3/4 inches in l/64-inch units; from 1/32 to 2 1/4 inches in 1/32-inch units, and from 1/1 6 to 3 1/2 inches in 1/16-inch units.

Metric twist drills are ranged in three ways: miniature set, straight shank, and taper shank. Miniature metric drill sets range from 0.04 mm to 0.99 mm in units of 0.01 mm. Straight shanks metric drills range from 0.05 mm to 20.0 mm in units from 0.02 mm to 0.05 mm depending on the size of the drill. Taper shank: drills range in size from 8 mm to 80 mm in units from 0.01 mm to 0.05 mm depending on the size of the drill.

The drill gage (Figure 4-8) is used to check the diameter size of a twist drill. The gage consists of a plate having a series of holes. These holes can be numbered, lettered, fractional, or metric-sized twist drills. The cutting end of the drill is placed into the hole to check the size. A micrometer can also be used to check the size of a twist drill by measuring over the margins of the drill (Figure 4-9). The smaller sizes of drills are not usually marked with the drill size or worn drills may have the drill size rubbed off, thus a drill gage or micrometer must be used to check the size.

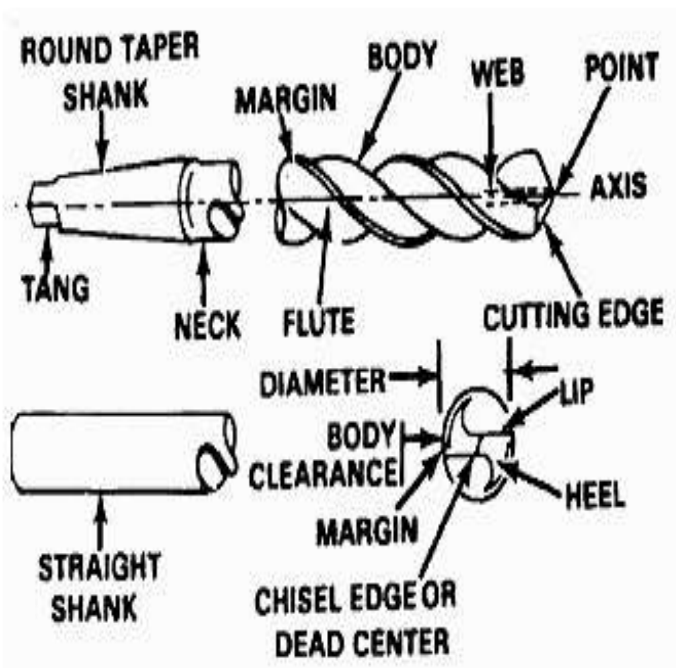


Figure 4-7. Twist drill nomenclature

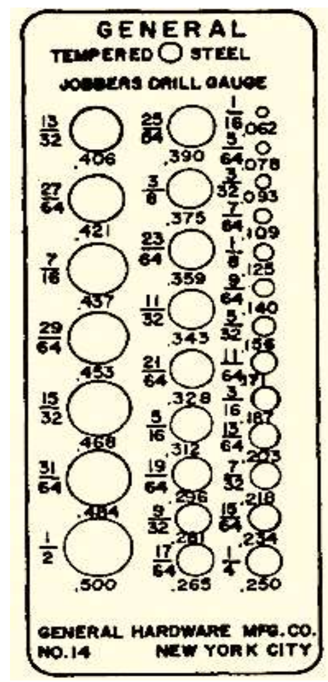


Figure 4-8. Drill gage.

It is important to know the parts of the twist drill for proper identification and sharpening (Figure 4-7). The point is the entire conical shaped end of the drill containing the cutting edges and chisel edge. The body is the part of the drill that is fluted and relieved. The shank is the part that fits into the holding device, whether it is a straight shank or a tapered shank. The chisel edge is the point at which the two lips meet.

The chisel edge acts as a chisel when the drill is turning and cuts into the workpiece. The chisel edge must always be centered exactly on the drill's axis for accurate cutting action. The cutting edge lips cut like knives when fed and rotated into the workpiece. The lips are sharp edges formed by grinding the flutes to a conical point. The heel is the conical shaped portion of the point in back of the cutting edge lips. The amount of slope given to the heel in back of the drill lips is called lip clearance.

This clearance is necessary to keep the heel from rubbing the bottom of the hole being drilled. Rubbing would prevent the drill from cutting. The flute is the helical groove on the drill. It carries out the chips and admits coolant to the cutting edges.

The margin is the narrow surface along the flutes that determines the size of the drill and keeps the drill aligned. The portion of the drill body that is relieved behind the margin is known as the body clearance. The diameter of this part is less than that of the margin and provides clearance so that all of the body does not rub against the side of the hole and cause friction. The body clearance also permits passage of lubricants around the drill.

The narrowed end of the tapered shank drill is called the tang. The tang fits the slot in the innermost end of the drill spindle, drill chuck, or other drill holding device and aids in driving the tool. It also prevents the drill from slipping. The web of the drill is the metal section separating the flutes. It runs the length of the body between the flutes. The web gradually increases in thickness toward the shank, increasing the rigidity of the drill. An imaginary line through the center of the drill from end to end is the axis. The drill must rotate evenly about the axis at all times.

4) Special Drills:

Special drills are needed for some applications that a normal general purpose drill cannot accomplish quickly or accurately. Special drills can be twist drill type, straight fluted type, or special fluted. Special drills can be known by the job that they are designed for, such as aircraft length drills, which have an extended shank. Special drills are usually used in high-speed industrial operations. Other types of special drills are: left hand drill, Silver and Deming, spotting, slow spiral, fast spiral, half round, die, flat, and core drills. The general purpose high-speed drill, which is the common twist drill used for most field and maintenance shops, can be reground and adapted for most special drilling needs.

5) Sharpening Twist Drills:

Twist drills become dull and must be re-sharpened. The preferred method of re-sharpening a twist drill is with the drill grinding machine, but this machine is not always available in field and maintenance units, so the offhand method of drill sharpening must be used (Figure 4-10). The offhand method requires that the operator have a knowledge of the drilling geometry (Figure 4-11) and how to change drill angles as needed for any drilling job (see Table 4-2).

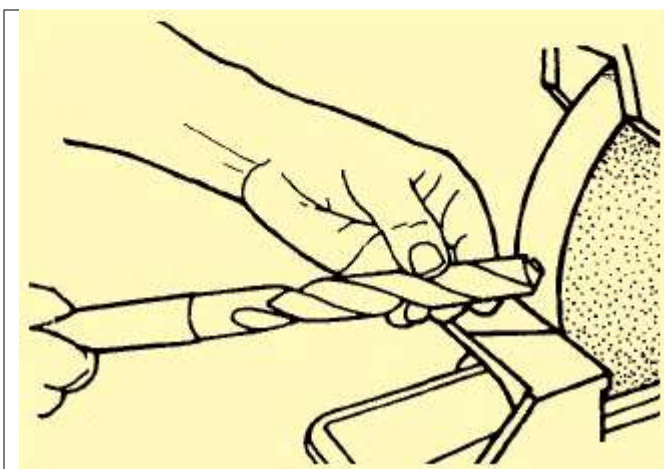


Figure 4-10. Offhand method of drill sharpening

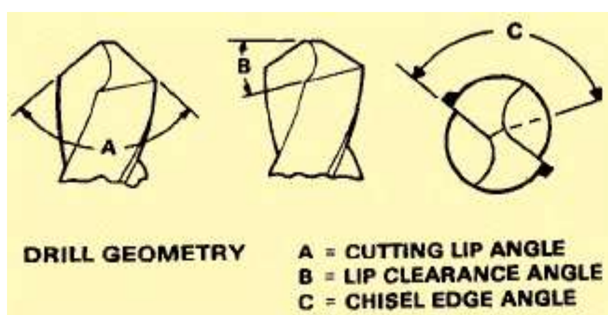


Figure 4-11. Drill geometry

6) Drill Point Gage:

Tools needed are a utility or bench grinder with a dressed wheel and a drill point gage (Figure 4-12) or protractor head on the combination square. The drill point gage is set at 59° and adjusted along the steel rule to fit the drill to be sharpened. The cutting lips must be of the same angle, the lip clearance angle must be within a specific degree range, and the cutting lips must be of an equal length. There are several basic characteristics that all twist drills must have to cut properly. The following will cover those characteristics.

Before sharpening a twist drill, the operator must check the condition of the drill for chipped and cracked lips or edges that must be ground off during the sharpening process. The operator must also check the references for the proper lip angle and lip clearance angle for the material to be drilled. After setting up the bench grinder for offhand drill sharpening, the operator assumes a comfortable stance in front of the grinding wheel to sharpen the twist drill.

The suggested method is to grind the lip angle first, then concentrate on grinding the lip clearance angle, which will then determine the lip length. The usual lip angle is an included angle of 118° (59° x 2) (Figure 4-13), which is the lip angle of general purpose drills. Use the drill point gage frequently to check lip angle and lip length. When grinding, do not allow the drill to become overheated. Overheating will cause the drill edges to become blue which is an indication that the drill's temper has been lost. The blue area must be ground completely away to reestablish the drill's temper. If a drill becomes too hot during sharpening, the lips can crack when dipped into cold water or coolant.

Figure 4-12. Checking the tip angle

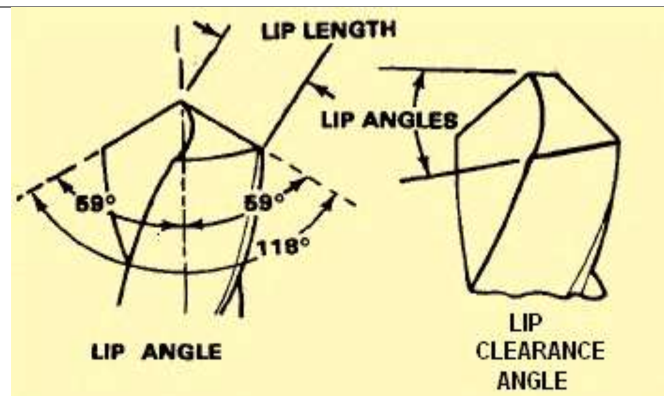


Figure 4-13. Twist drill angles

When grinding the lip angle, use the drill point gage and grind one lip perfectly straight and at the required angle (usually 59°). Then flip the drill over and grind the other lip. Once the angle is established the lip clearance angle and lip length can be ground, if both lips are not straight and of the same angle, then the chisel edge (Figure 4-14) will not be established.

It is it important to have a sharp and centered chisel edge or the drill will not rotate exactly on its center and the hole will be oversized. If the drill point is too flat, it will not center properly on the workpiece. If the drill point is too steep, the drill will require more power and cut slowly. When the angles of the cutting lips are different, then the drill will only have one lip cutting as it revolves. The hole will be oversized and the drill will wear very rapidly. When both the angles and the length of the angles are incorrect, then excessive wear is put on both the drill and machine, which will result in poor workmanship (Figure 4-15).

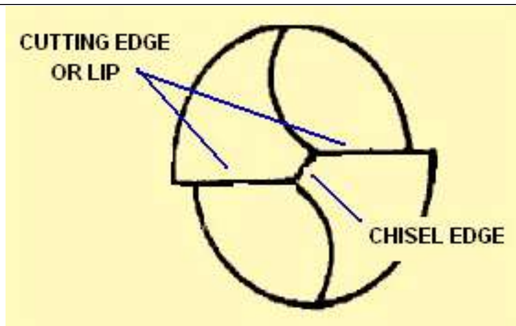


Figure 4-14. The drill point

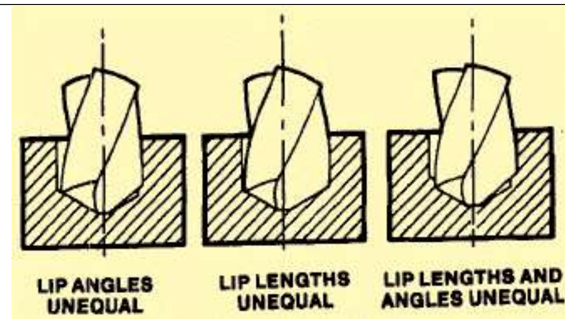


Figure 4-15. Results of improperly ground drills

7) Clearance Angle:

When grinding the lip clearance angle, (Figure 4-13), relief must be given to both cutting edges allowing them to enter into the workpiece to do the cutting. General purpose drills have a clearance of 8° to 12°. The chisel edge of a correctly ground drill should be at an angle of about 45° with the line of the cutting edges. The angle of the chisel edge to the lips is a guide to the clearance (Figure 4-16). Too much clearance will cause the drill to break down because of insufficient support of the lip, and there will not be enough lip thickness to carry away the generated heat.

Too little clearance will result in the drill having little or no cutting edges, and the increased pressure required to feed it into the hole will cause the drill to break. By looking straight onto the cutting tip of the drill, the operator can see if the chisel edge is correct. If the chisel edge is correct at 45° to the lips, then it is an indication that the lip clearance angle is correct. An incorrect chisel edge is usually produced by holding the drill at an incorrect angle to the wheel (Figure 4- 17) when grinding. A good guide is to hold the drill parallel to the ground, and make slight adjustments.

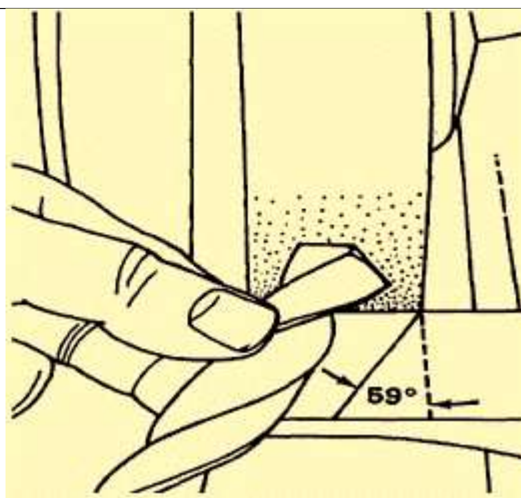


Figure 4-17. Adjusting the drill for grinding the top angle.

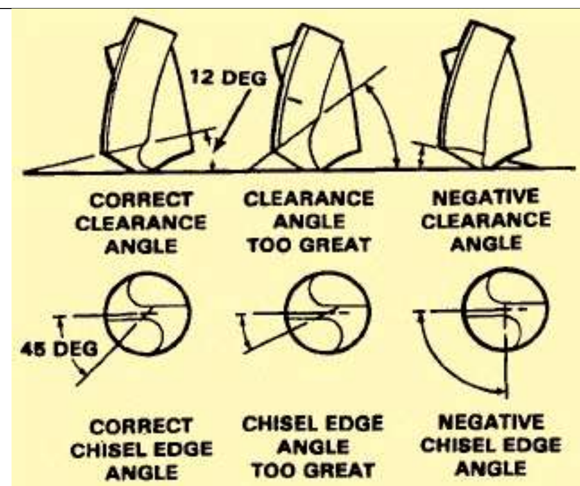


Figure 4-16. Lip clearance angle is directly proportional to the chisel point.

8) Rake Angle:

The angle between the flute and the axis forms the cutting edge is known as the rake angle (Figure 4-18). Generally, the rake angle is between 180 and 450, with 30° being the most common. Drills used on armor plate or other very hard materials need a reduced rake angle to increase the support behind the cutting edge. Soft materials, like brass and bronze, also use a reduced rake angle to prevent the drill from grabbing. The rake angle partially governs the tightness with which the chips curl and the amount of space they occupy. If the rake angle is too small, the lips may be too thin and break under the strain of drilling since too large of a rake angle makes the drill chatter and vibrate excessively.

The web of a drill is made thicker toward the shank to strengthen the tool. In smaller size drills, the difference is not noticeable, but in larger drills, when the point is ground back by repeated sharpening, the thickness of the web becomes greater and the chisel edge of the drill becomes wider. This causes the chisel edge to scrape on the bottom of the hole and requires excessive pressure to be applied to the drill.

This can be corrected by thinning the web (Figure 4-19). The point is ground thinner on a thin grinding wheel with a rounded face to fit into the flute. An equal amount of metal should be ground from each flute. The web should not be ground too thin as this may weaken the web and cause the drill to split in the middle.

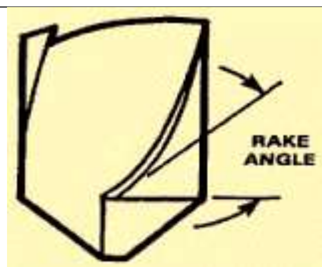


Figure 4-18. Rake angle

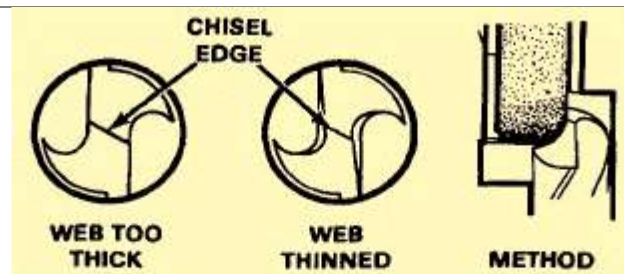


Figure 4-19. Thinning the web

9) Drill Grinding Machines:

Drill grinding machines (Figure 4-20) make the accurate grinding of all types and sizes of drills an easy job. Comparatively little skill is required to sharpen drills with these machines while following the operating instructions. They are particularly valuable when a large number of the same general types of drills are to be sharpened.

Two basic designs for the bench-type drill grinding machines are available. Both perform the same operations but use different drill holding devices. The capacity of these machines is stated in the horsepower of the electric motor and the sizes of drills which can be accommodated by the drill holding devices.

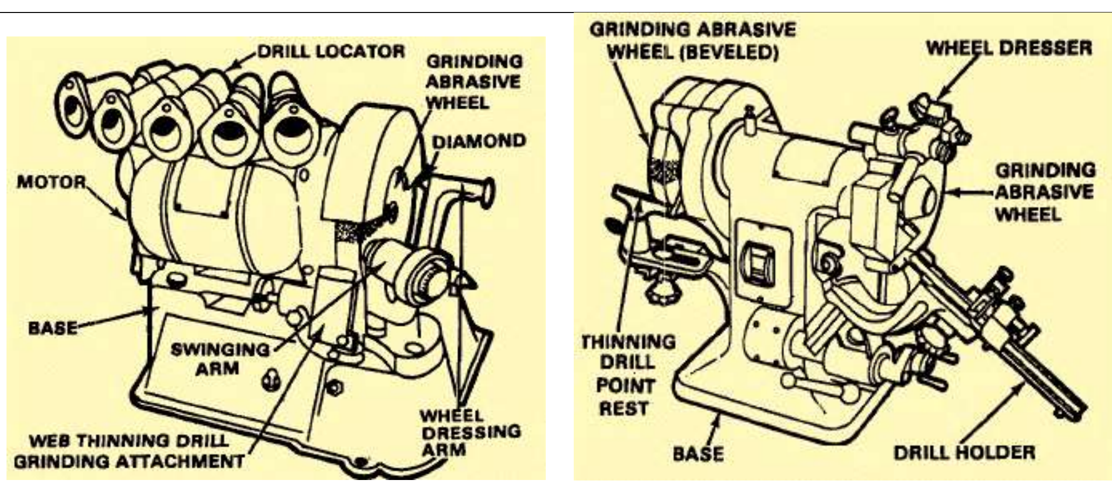


Figure 4-20. Drill grinding machines.

Single Wheel Fixture: Consists of an electric motor, a grinding abrasive wheel attached to the motor shaft, and fixtures to hold and position all types of twist drills for drill grinding. A drill holder assembly and swinging arm hold the drill in a fixed position for each grinding operation and permit the cutting edge lips to be ground symmetrically at the correct angle and with the correct clearance to ensure long life and efficient cutting. Collets and bushings are supplied with the drill grinding machine to hold a wide range of different sized drills. The grinding machine has a diamond set in the wheel-dressing arm to dress the grinding wheel as necessary.

Double Wheel Swing Arm: Another kind of bench type drill grinding machine is equipped with two grinding abrasive wheels, one at each end of the motor shaft. One wheel is beveled for thinning the web of the drill at the point. The other wheel is used for lip grinding. The grinder includes a wheel holder assembly for mounting the drill and providing a means for bringing the drill into contact with the grinding wheel at the correct angle and feed to obtain proper clearance angles.

10) Drilling Machine Cutters:

Drilling machines use cutters that are not drills, to produce special holes. Below are listed the most common types. A thinning drill point rest is mounted forward of the beveled grinding abrasive wheel to rest and guide the drill during web thinning operations. A wheel dresser is provided to dress the grinding wheel as necessary.

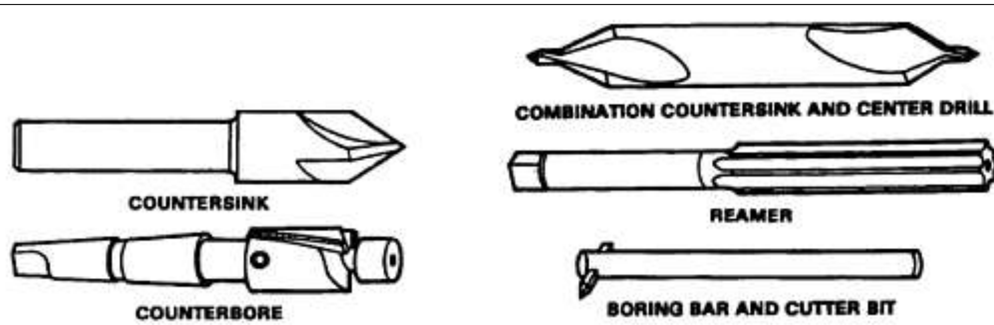
Countersinks: Countersinks are special angled cutters used to countersink holes for flathead screws so they are flush with the surface when mounted and the most common countersinks are cone shaped with angles of 82°. Cone angles of 60°, 90°, 100°, 110°, and 120° are for special needs. (Figure 4-21).

Counterbores: Counterbores are special cutters that use a pilot to guide the cutting action to enlarge a portion of a hole, commonly for enlarging a hole to make a bolt head fit flush with the surface. (Figure 4-21).

Combined Countersink and Center Drill: This special drilling tool is used to start holes accurately. These tools are mainly used to center drill and countersink the end of round stock in a lathe machine. (Figure 4-21).

Reamers: Reamers are cutting tools that are used to enlarge a drilled hole by a few thousandths of an inch for a precise fit. (Figure 4-21).

Boring Tools: Boring tools are not usually considered with drilling, but they can be used to bore a hole using the power-feed drilling machines. These tools consist of an arbor with a tool bit attached that cuts a preset sized hole according to the distance that the tool bit protrudes from the arbor. (Figure 4-21).



Boring Tools. Figure 4-21.

Tap and Die Work: Hand tapping and hand die work can be done on a drilling machine. The drill chuck is used to align the tap or die.

11) Drill Holding Devices:

The revolving vertical spindle of the drilling machine holds and drives the cutting tool. In order to use various sizes and shapes of drills in various machines three types of drill holding devices, which fit the spindle of the drilling machines, are used: the geared drill chuck, the drill sleeve, and the drill socket. The larger drilling machines have a spindle that has a standard Morse taper at the bottom end. There are **three types** of drill holding devices: the geared drill chuck, the drill sleeve, and the drill socket. (Figure 4-22).

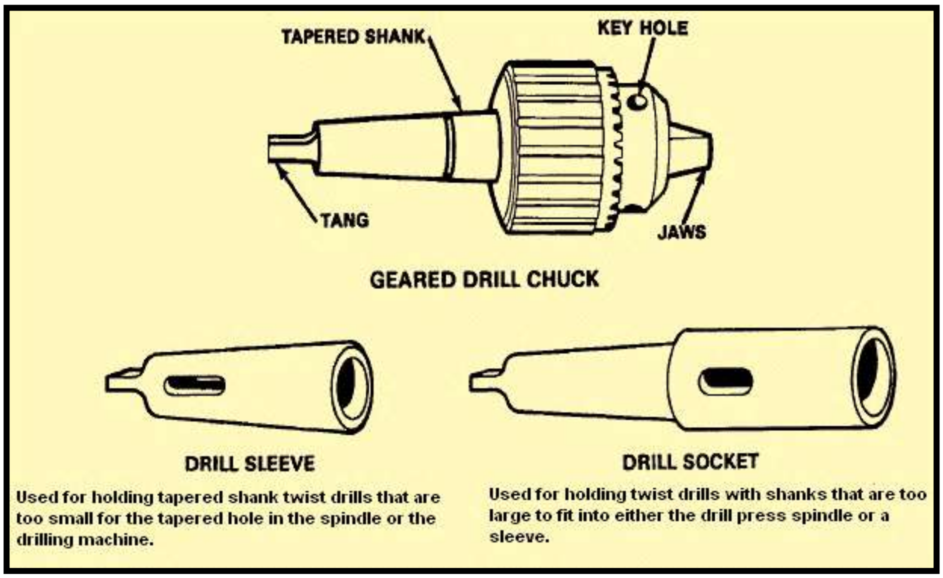


Figure 4-22. Drill holding devices.

12) 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, when 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.

13) Drill Sockets and 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 of size #2, #3, or #4. It is also common for a drilling machine spindle to have a Morse taper of size #3 or #4, and it can be adapted for many other Morse taper sizes, depending on the size of the drill.

A drill too small for the machine spindle may be fitted into a socket or sleeve which has a taper hole of the proper size to hold the drill and a taper shank of the proper size to fit the drill spindle. Sometimes, more than one socket or sleeve is needed to build up the shank to tit into the drilling machine spindle. Sockets and sleeves may be obtained in a number of different sizes and hole shank taper combinations. Sockets, sleeves, and taper shank drills are mounted into the aligning slots of the spindle and lightly tapped with a soft hammer to seat in place.

14) Drill Drifts:

Drill drifts are flat tapered keys with one rounded edge designed to fit into a spindle chuck's slot to force a tapered shank drill loose. The rounded top of the small end of the drill drift is designed to face upward while inserting the drift into the slot. There are **two types** of drill drifts, the **standard type and the safety type** (Figure 4-23). The **standard** drift must be inserted into the chuck's slot and then struck with a soft hammer to the taper shank drill. The drill will fall quickly if not held by the hand and could break or cause injury. The **safety** drill drift has a sliding hammer weight on the drift itself, to allow for a free hand to stay constantly on the drill.

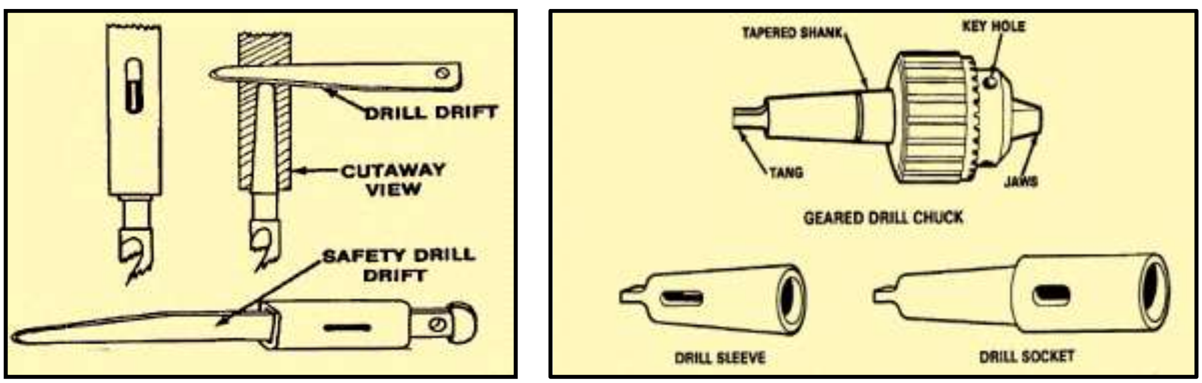


Figure 4-23.

15) Drill Table Vises:

Work holding devices are used to hold the work steady for an accurate hole to be drilled, and so a safe drilling operation can be accomplished. Drilling support devices are used to keep the workpiece above the worktable or vise surface and to keep the workpiece aligned for drilling. Some devices are fairly simple and are used for drilling operations that do not require a perfect hole. Other devices are designed for more accurate drilling. Many work holding devices are used with one another to produce the most stable work setup for drilling.

The machine table vises is the most useful device, equipped with jaws which clamp against the workpiece, holding it secure. The vise can be bolted to the drilling table or the tail can be swung around to lie against the column to hold itself steady. Below are listed many types of special purpose machine table vises available to machine operators. 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 (Figure 4-24). 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. Many other vises are available. They include the compound vise, universal vise, magnetic vise, and contour vise.

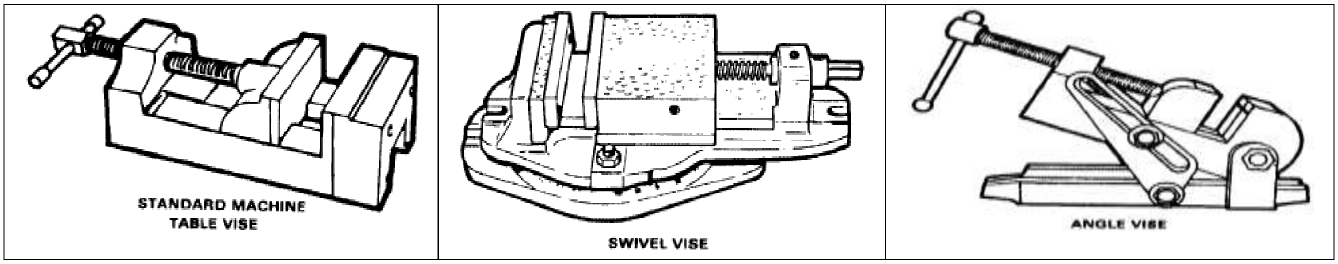


Figure 4-24. Types of vises

16) Step Blocks and Clamps:

Step blocks are holding devices are built like stairs to allow for height adjustments in mounting drilling jobs and are used with strap clamps and long T-slot bolts (Figure 4-25). Clamps are small, portable vises or plates which bear against the workpiece and holding devices to steady the job. Clamps are made in numerous shapes to meet various work-holding needs. Common types of clamps are the C-clamp, the parallel clamp, the machine strap clamp, the bent-tail machine clamp, the U-clamp, and the finger machine clamp (Figure 4-25).

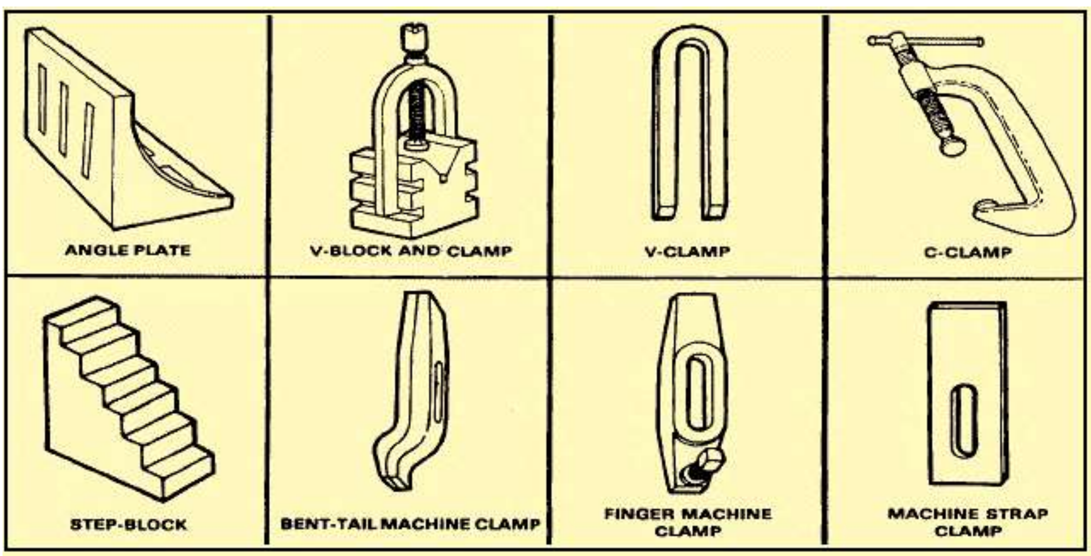


Figure 4-25. Work holding devices.

17) V-Blocks, Angle Plates and Accessories:

V-blocks are precision made blocks with special slots made to anchor clamps that hold workplaces. The V-slot of the block is designed to hold round workplaces. The V-block and clamp set is usually used to hold and drill round stock. Angle plates are made in a 900 angle with slots and bolt holes for securing work to the table or to other work holding devices (Figure 4-25).

Blocks are used with clamps to aid in securing and supporting the work. These blocks are usually precision ground of hard steel for long life. Parallels are precision ground rectangular bars are used to keep the workpiece parallel with the worktable when the workpiece must be raised above the worktable surface, such as when drilling completely through a workpiece (Figure 4-26.

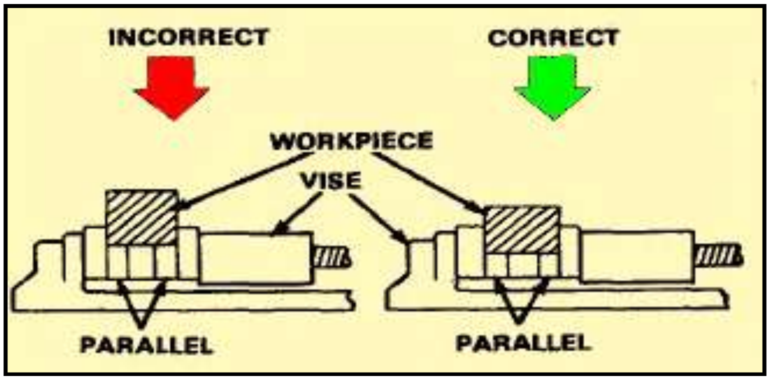


Figure 4-26. Parallels being used to support a workpiece

T-bolts are specially made bolts that have a T-shaped head designed to slide into T-slots of the drilling machine's worktable. A heavy duty washer and nut are used with the T-bolt to secure the work. Drill jigs are devices designed for production drilling jobs. The workplaces are clamped into the jig so that the holes will be drilled in the same location on each piece. The jig may guide the drill through a steel bushing to locate the holes accurately.

18) Drilling Process:

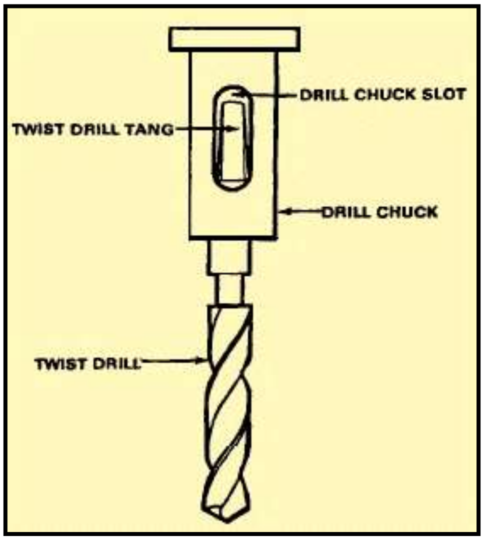
After a workpiece is laid out and properly mounted, the drilling process can begin. The drilling process, or complete operation, involves selecting the proper twist drill or cutter for the job, properly installing the drill into the machine spindle, setting the speed and feed, starting the hole on center, and drilling the hole to specifications within the prescribed tolerance. Tolerance is the allowable deviation from standard size.

The drilling process must have some provisions for tolerance because of the oversizing that naturally occurs in drilling. Drilled holes are always slightly oversized, or slightly larger than the diameter of the drill's original designation. For instance, an l/4-inch twist drill will produce a hole that may be several thousandths of an inch larger than l/4-inch.

Oversizing is due to several factors that affect the drilling process: the actual size of the twist drill, the accuracy of the drill point, the accuracy of the machine chuck and sleeve, the accuracy and rigidity of the drilling machine spindle, the rigidity of the entire drilling machine, and the rigidity of the workpiece and setup. Field and maintenance shop drilling operations allow for some tolerance, but oversizing must be kept to the minimum by the machine operator.

Drill Tool: Selecting the proper twist drill means getting the right tool for the job. The material to be drilled, the size of that material, and the size of the drilled hole must all be considered when selecting the drill. Also, the drill must have the proper lip angles and lip clearances for the job. The shank of the drill must also be clean and free of burrs to fit into the chuck. Most drills wear on the outer edges and on the chisel point, so these areas must be checked, and resharpened if needed, before drilling can begin. If the twist drill appears to be excessively worn, replace it.

Before installing the drill into the drilling machine spindle, clean the spindle socket and drill shank of all dirt, chips, and burrs. Use a small tile inside the socket to remove any tough burrs. Slip the tang of the drill or geared drill chuck into the sleeve and align the tang into the keyway slot (Figure 4-30). Tap the end of the drill lightly with a soft hammer to seat firmly. Another method used to seat the drill into the sleeve is to place a block of wood on the machine table and force the drill down onto the block.



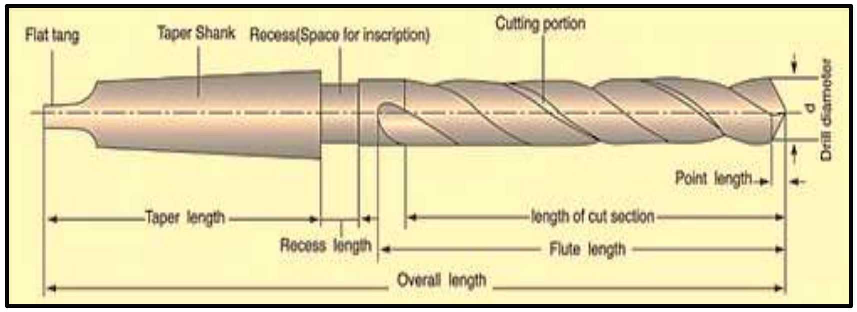


Figure 4-30. Installing a taper shank drill

Depth of Drilled Holes: To accurately check the depth of a drilled hole, the length of the sides of the hole must be measured. Do not measure from the bottom point of the hole (Figure 4-36). A thin depth gage is inserted into the hole, along the side, and the measurement taken. If the hole is too small for the gage to fit down into it then a twist drill of the same size as the hole can be inserted into the hole upside down, then removed and measured with a rule...

To start a twist drill into the workpiece, the point of the drill must be aligned with the **center-punched mark** on the workpiece. Some drilling operations may not require a precise alignment of the drill to the work, so alignment can be done by lining up the drill by hand and eye alone. If a greater precision in center alignment is required, than, more preparation is needed before starting to drill.

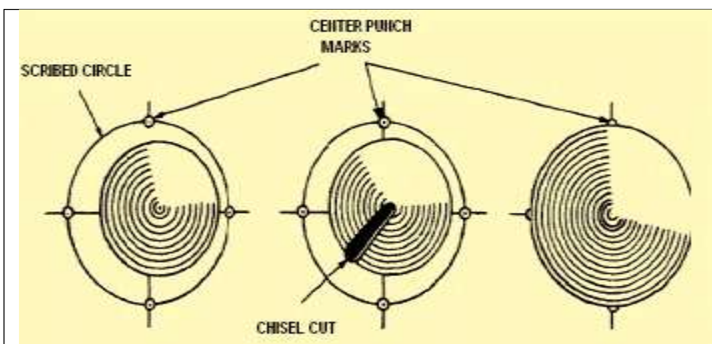


Figure 4-31. Center punching mark

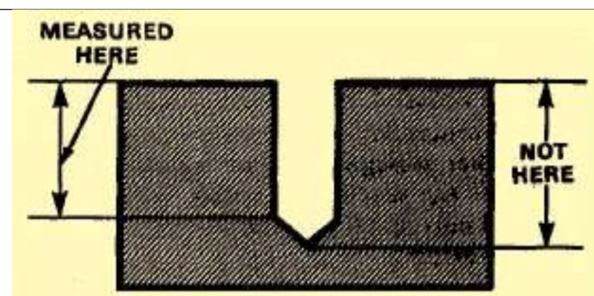


Figure 4-36. Checking the depth of the drill holes

The best method to align and start a hole is to use the combination countersink and drill, known as the center drill (Figure 4-31). Set the drilling machine speed for the diameter of the tip of the center drill, start the machine, and gently lower the center drill into contact with the work, using hand and eye coordination. The depth of the center-drilled hole should be **no deeper than two third the length** of the tapered portion of the center drill.

19) Drilling Calculations:

Selecting Drill Speed: Speed refers to the revolutions per minute (RPM) of the drilling machine spindle. For drilling, the spindle should rotate at a set speed that is selected for the material being drilled. Correct speeds are Essen - ntial for satisfactory drilling. The speed at which a drill turns and cuts is called the peripheral speed.

Peripheral speed is the speed of a drill at its circumference expressed in surface feet per minute (SFPM). This speed is related to the distance a drill would travel if rolled on its side. For example, a peripheral speed of 30 feet per minute means the drill would roll 30 feet in 1 minute if rolled on its side.

Speed Calculation: If the cutting speed of a material is known, then a simple formula can be used to find the recommended RPM of the twist drill. The slower of **two recommended speeds** may be used for the following formulas due to the varying conditions that may exist, such as the rigidity of the setup, the size of the drilling machine, and the quality of finish.

$$\text{RPM} = \frac{\text{CS x 4}}{\text{D}}$$

Where:
RPM = drill speed in revolutions per minute.
CS = Recommended cutting speed in surface feet per minute.
4 = A constant in all calculations for RPM (except metric).
D = Diameter of the drill itself.

Example:

If a 1/2-inch (0.500-inch) twist drill is to cut aluminum, the formula would be setup as follows:

$$\text{RPM} = \frac{200 \text{ x } 4}{0.500} = \frac{800}{0.500} = \mathbf{1600\ RPM}$$

Thus, the drilling machine would be set up to drill as close to 1,600 RPM as possible. It is best to use the machine speed that is closest to the recommended RPM. When using the metric system of measurement, a different formula must be used to find RPM:

$$\text{RPM} = \frac{\text{CS (m) x 320}}{\text{D (mm)}}$$

Where:
RPM = Drill speed in revolutions per minute.
CS = Recommended cutting speed in surface meters per minute.
320 = A constant for all metric RPM calculations.
D = Diameter of the twist drill in millimeters.

Example:

If a 15-mm twist drill is to cut medium-carbon steel, with a recommended cutting speed of 21.4 meters per minute, the formula would be set up as follows:

$$\text{RPM} = \frac{21.4 \text{ x } 320}{15} = \frac{6848}{15} = \mathbf{456.533\ or\ 457\ RPM}$$

The RPM is rounded to the nearest machine speed. The speeds on these tables are just recommendations and can be adjusted lower if needed, or to higher speeds if conditions permit.

20) Selecting the Drill Feed:

Feed is the distance a drill travels into the workpiece during each revolution of the spindle. It is expressed in thousandths of an inch or in millimeters. Hand-feed drilling machines have the feed regulated by the hand pressure of the operator; thus, the skill of the operator will determine the best feeds for drilling. Power feed drilling machines have the ability to feed the drill into the work at a preset depth of cut per spindle revolution, so the best feeding rate can be determined (see Table 4-2 and 4-4).

Table 4-2: Drill information for different materials (high-speed drills)

MATERIAL	CUTTING SPEEDS 1. (METERS/MINUTE) MPM	(FEET/MINUTE) FPM	POINT ANGLE	LIP CLEARANCE	COOLANTS
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 - 30.50	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

NOTE: Too much feed will cause the drill to split; too little feed will cause chatter, dull the drill, and possibly harden the workpiece so it becomes more difficult to drill. Drills 1/2 inch or smaller can generally be hand-fed, while the larger drills require more downward torque and should be power-fed.

The selection of the best feed depends upon the size of the drill, the material to be drilled, and the condition of the drilling machine. Feed should increase as the size of the drill increases. After starting the drill into the workpiece by hand, a lever on the power-feed drilling machine can be activated, which will then feed the drill into the work until stopped or disengaged.

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	4,889	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	3,668	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]	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	286	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	306	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	916	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.

21) Drilling Best Practices:

After the drill has been aligned and the hole started, then insert the proper size drill (Figure 4-32) and continue drilling into the workpiece (Figure 4-33), while applying cutting fluid. The cutting fluid to use will depend on what material is being machined (see Table 4-3). Use the cutting fluids freely. Often, the drill will not be on center, sometimes due to a poorly made center-punched mark or a hard spot on the metal.

To draw the twist drill back to the position desired (Figure 4-31), a sharp chisel is used to make one or more nicks or grooves on the side toward which the drill is to be drawn. The chisel marks will draw the drill over because of the tendency of the drill to follow the line of least resistance. After the chisel mark is made, the drill is again hand-fed into the work and checked for being on center.

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 holes 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. When 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.

Pilot Hole: As the drill size increases, both the size of the web and the width of the chisel edge increase (Figure 4-34). 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 (Figure 4-34) 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.

Thin Material: When drilling thin workpieces, such as sheet metal, place another piece of metal or wood under the workpiece to provide support and prevent bending the workpiece or ruining the hole due to the up thrust created when the drill breaks through. If thin metal must be drilled and a support cannot be rigged under the thin metal, then a drill designed for thin metal, such as a low helix drill with zero rake angle, commonly called a sheet metal drill, must be used.

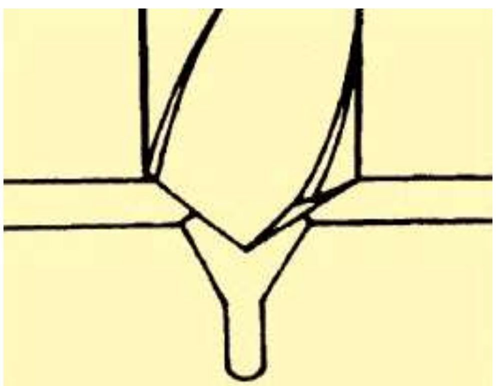


Figure 4-32. Drifting the center drilled hole

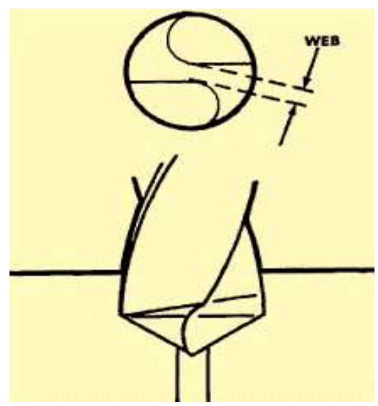


Figure 4-34. Using a pilot hole

Stop Device: The depth stop mechanism on the drilling machine (Figure 4-35) should be used whenever drilling to a desired depth, and to prevent the twist drill from traveling too far after cutting through the workpiece. The depth stop is designed to be used whenever a number of holes of the same depth are to be drilled, or when drilling holes deep into the workpiece (blind holes).

Depth Setting: Make sure that drills are chucked tightly to avoid slipping and changing the depth setting. Most depth stops have a way to measure the distance that the drill travels. Some may have a fractional gage on the depth stop rod, and some may have a micrometer dial located on the depth stop for very precise measurements.

Round Stock: When drilling shafts, rods, pipes, dowels, or other round stock, it is important to have the center punch mark aligned with the drill point (Figure 4-37). Use V-blocks to hold the round stock for center punching and drilling. Align the center of the round stock with a square or by lining the workpiece up with the twist drill point. Another method to drill round stock is to use a V-block drill jig that automatically centers the work for drilling.

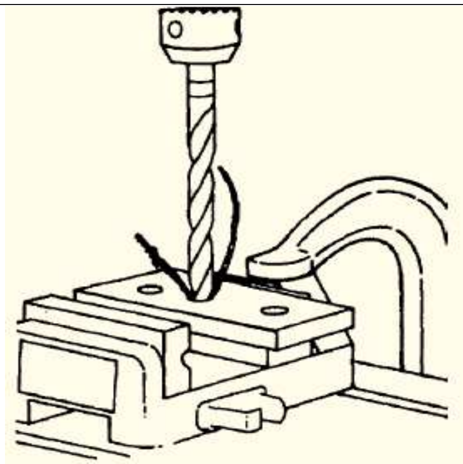


Figure 4-33. Drilling a workpiece.

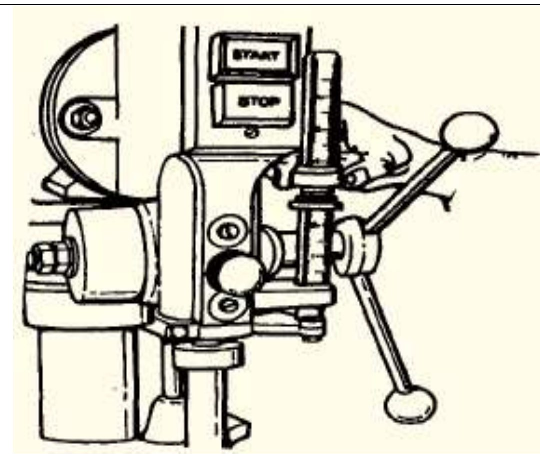


Figure 4-35. Depth stop mechanism.

Countersink: 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 (Figure 4-38) and is similar to counterboring.

Types of Countersinks: Machine countersinks for machining recessed screw heads commonly have an included angle of 82°. Another common countersink has an included angle of 60° machining lathe centers. Some countersinks have a pilot on the tip to guide the countersink into the recess. Since these pilots are not interchangeable, these types of countersinks can be used for only one size of hole and are not practical for field or maintenance shops.

Countersink Alignment: Proper alignment of the countersink and the hole to be recessed are important. Failure to align the tool and spindle with the axis of the hole, or failure to center the hole, will result in an eccentric or out-of-round recess.

Countersinking Procedures: Good procedures require that the countersink be run at a speed approximately one-half of the speed for the same size drill. Feed should be light, but not too light to cause chatter. A proper cutting fluid should be used to produce a smooth finish. Rough countersinking is caused by too much speed, dull tools, failure to securely hold the work, or inaccurate feed. The depth stop mechanism should be used when countersinking to ensure the recess will allow the flathead screw to be flush with the surface (Figure 4-39).

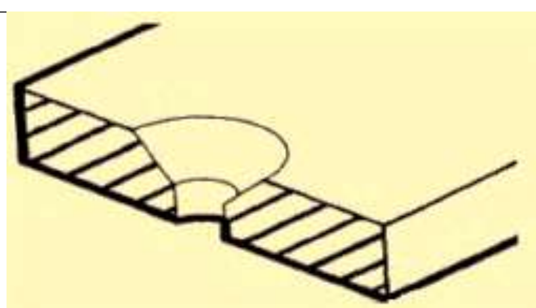


Figure 4-38. Countersunk hole.

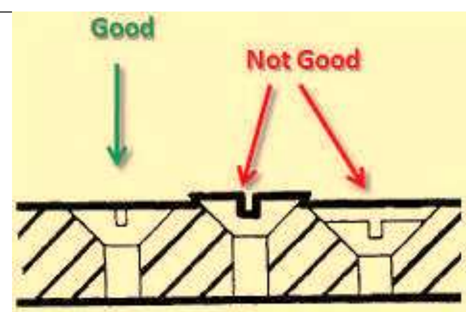


Figure 4-39. Proper and improper countersinking.

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 (Figure 4-40). When counterboring, mount the tool into the drill chuck and set the depth stop ‘mechanism for the required depth of shoulder cut. Set the speed to approximately one-half that for the same size of twist drill.

Counterboring Procedures: Compute for the actual cutter size and not the shank size when figuring speed. Mount the workpiece firmly to the table or vise. Align the workpiece on the center axis of the counterbore by fitting the pilot into the drilled hole. The pilot should fit with a sliding motion inside the hole. If the pilot fits too tightly, then the pilot could be broken off when attempting to counterbore. If the pilot fits too loosely, the tool could wander inside the hole, causing chatter marks and making the hole out of round.

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 spot facing can be accomplished with standard counterbore cutters.

Spot Facing Procedures: Is basically the same as counter boring, using the same tool, speed, feed, and lubricant but the operation of spot facing is slightly different, usually done above a surface or on a curved surface. Care must be taken when starting the spot facing cut to avoid too much feed. If the tool grabs the workpiece because of too much feed, the cutter may break or the workpiece may be damaged.

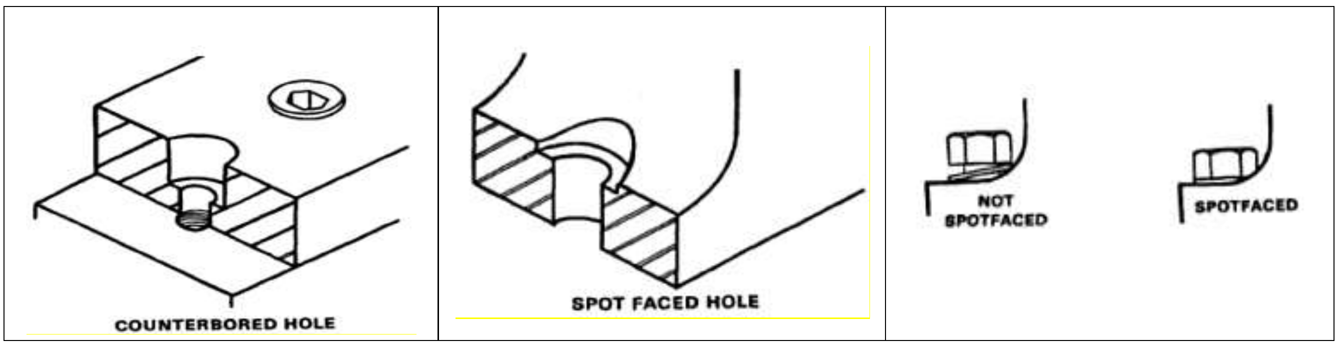


Figure 4-40. Counterboring and spot facing

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 holes combinations can be accomplished.

Feeds: Generally **0.002 to 0.005** inch per revolution, but the condition of the tool and the type of metal will affect the cutting operation. Slow the speed and feed if needed. The pilot must be lubricated with lubricating oil during counterboring to prevent the pilot seizing into the work. Use an appropriate cutting fluid if the material being cut requires it. Use hand feed to start and accomplish counterboring operations. Power feed counterboring is used mainly for production shops.

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.

Large Holes Tapping: One method of hand tapping is to mount an adjustable tap and reamer wrench on the square shank of the tap and install a pointed tool with a center in the drilling machine spindle (Figure 4-41). The tap is placed in the drilled hole and the tool's center point is placed in the center hole. The tap is held steady, without forcing, by keeping light pressure on it with the hand feed lever of the drilling machine, while turning the wrench and causing the tap to cut into the hole.

Small Holes Tapping: Another method of hand tapping, without power, is to connect the tap directly into the geared drill chuck of the drilling machine and then turn the drill chuck by hand, while applying light pressure on the tap with the hand feed lever. This method works well on small hand-feed drilling machines when using taps smaller than 1/2-inch diameter.

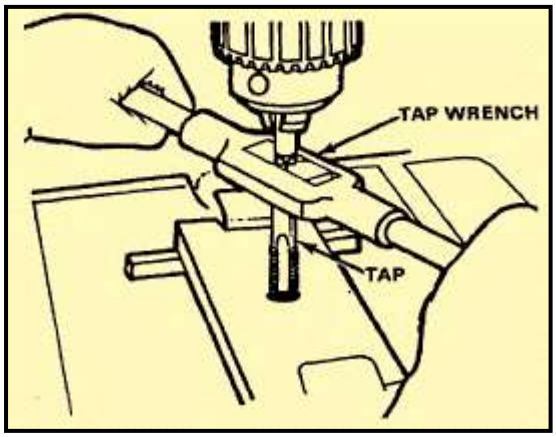


Figure 4-41. Tapping with an upright drilling machine

Reaming: 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 (Figure 4-42). Reaming can be done on a drilling machine by using a hand reamer or using a machine reamer.

Procedures for Reaming: To drill and ream a hole, the setup cannot be 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 Reamers: Solid hand reamers should be used when a greater accuracy in size is required. The cutting action of a hand reamer is performed on the taper (approximately 0.015 per inch) which extends 3/8- to 1/2- inch above the chamfer. This slight taper limits the stock allowance, or metal to be removed by the reamer, from 0.001- to 0.003-inch depending on the size of the reamer. As the reamer is turned by hand into the hole, only a slight pressure is applied to the hand feed lever to keep the center in contact with the reamer and maintain accuracy in alignment.

Machine Reamers: The allowance for machine reamers is generally 1/64 inch for reamers l/2-inch to 1 inch in diameter, a lesser amount for smaller holes, and greater than 1/64-inch for holes over 1 inch. Machine reamers for use on drilling machines or lathes have taper shanks to fit the machine spindle or straight shanks for inserting into a drill chuck. A reamer must run straight and true to produce a smooth finish. The proper cutting fluid for the metal being cut should be used. Generally, the speed used for machine reaming would be approximately one-half that used for the same size drill.

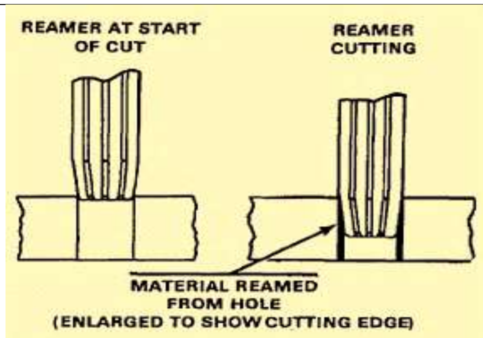


Figure 4-42. Reaming operations.

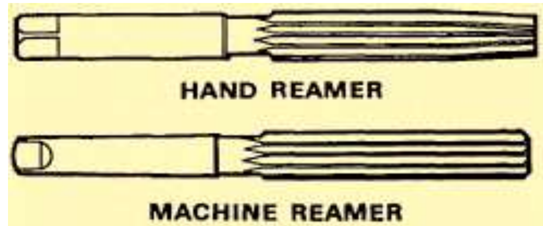


Figure 4-43. Hand and machine reamers.

Boring: Occasionally a straight and smooth hole is needed when the hole is too large or odd sized for drills or reamers. A boring tool can be inserted into the drilling machine and bore any size hole into which the tool holder will fit. A boring bar with a tool bit installed is used for boring on the larger drilling machines. To bore accurately, the setup must be rigid, machine must be sturdy, and power feed must be used. Boring is not recommended for hand-feed drilling machines. Hand feed is not smooth enough for boring and can be dangerous. The tool bit could catch the workpiece and throw it back at the operator.

Boring Procedures: First, secure the work and drill a hole for the boring bar. Then, insert the boring bar without changing the setup. Use a dial indicator to set the size of bored hole desired by adjusting the tool bit in the boring tool holder; then, set the machine speed and feed. The speed is set at the speed recommended for drilling a hole of the same size. Feed should be light, such as 0.005 to 0.010 inch per revolution. Start the machine and take a light cut. Check the size of the hole and make necessary adjustments. Continue boring with a more rough cut, followed by a smoother finishing cut. When finished, check the hole with an internal measuring device before changing the setup in case any additional cuts are required.

Operational Checks: After the hole is drilled to specifications, always back the drill out of the hole and shut off the machine. Allowing a drill to run on in the hole will cause the hole to be oversized. At any time during the drilling process, a problem could occur. If so, it should be fixed as soon as possible to avoid any damage or injury. Operators must observe the drilling machine for any excessive vibration or wobble, overheating of the electric motor, and unusual noises coming from the machine.

A high pitched squeal coming from the drill itself may indicate a dull drill. A groaning or rumbling sound may indicate that the drill is overloaded and the feed needs to be reduced. A chattering sound may indicate an off-center drill or a poorly sharpened drill. These or other noises could also be caused by internal parts of the machine. Consult the operator's manual and correct all problems before attempting to continue drilling.

22) Drilling Cutting Fluids:

Drilling lubricants, cutting fluids and coolants are used in drilling work to lubricate the chip being formed for easier removal, to help dissipate the high heat caused by friction, to wash away the chips, improve the finish and to permit greater cutting speeds for best efficiency. In drilling work, the cutting fluid can be sprayed, dripped, or machine pumped onto the work and cutting tool, cooling the action and provide for maximum tool life.

Drilling, reaming, and tapping of various materials can be improved by using the proper cutting fluids. Cutting- fluids can be produced from animal, vegetable, or mineral oils. Some cutting fluids are very versatile and can be used for any operation, while other cutting fluids are specially designed for only one particular metal.

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

GRINDING MACHINES:

From the simplest grinding machine to the most complex, grinding machines can be classified as utility grinding machines, cylindrical grinding machines and surface grinding machines. The average machinist will be concerned mostly with floor-mounted and bench-mounted utility grinding machines, electric motor-driven machines, buffing machines, and tool-post grinding machines.

Utility Grinding Machine: Is intended for offhand grinding where the workpiece is supported in the hand and brought to bear against the rotating grinding abrasive wheel. The accuracy of this type of grinding machine depends on the operator's skill and knowledge of the machine's capabilities and the nature of the work. The utility grinding machine consists of a horizontally mounted motor with a grinding abrasive wheel attached to each end of the motor shaft.

Electric Motor-Driven Machine: Is simple and common. It may be bench-mounted or floor-mounted. Generally, the condition and design of the shaft bearings, as well as, the motor rating determine the wheel size capacity of the machine. Suitable wheel guards and tool rests are provided for safety and ease of operation. Grinding machines come in various sizes and shapes as listed below:

1) Grinding Machines Types:

Floor Mounted Utility Grinding Machine: The typical floor-mounted utility grinding machine stands waist-high and is secured to the floor by bolts. The floor-mounted utility grinding machine shown in Figure 5-1 mounts two 12-inch diameter by 2-inch-wide grinding abrasive wheels. The two wheel arrangement permits installing a coarse grain wheel for roughing purposes on one end of the shaft and a fine grain wheel for finishing purposes on the other end this saves the time that would be otherwise consumed in changing wheels.

Each grinding abrasive wheel is covered by a wheel guard to increase the safety of the machine. Transparent eye shields spark arresters and adjustable tool rests are provided for each grinding wheel. A tool tray and a water pan are mounted on the side of the base or pedestal. The water pan is used for quenching carbon steel cutting took as they are being ground. Using the 12-inch wheel, the machine provides a maximum cutting speed of approximately 5.500 SFPM. The 2-HP electric motor driving this machine has a maximum speed of 1.750 RPM.

Bench Type Utility Grinding Machine: Like the floor mounted utility grinding machine, one coarse grinding wheel and one fine grinding wheel are usually mounted on the machine for convenience of operation (Figure 5-2). Each wheel is provided with an adjustable table tool rest and an eye shield for protection.

On this machine, the motor is equipped with a thermal over-load switch to stop the motor if excessive wheel pressure is applied thus preventing the burning out of the motor. The motor revolve at 3.450 RPM maximum to provide a maximum cutting speed for the 7 inch grinding wheels of about 6,300 surface feet per minute (SFPM).

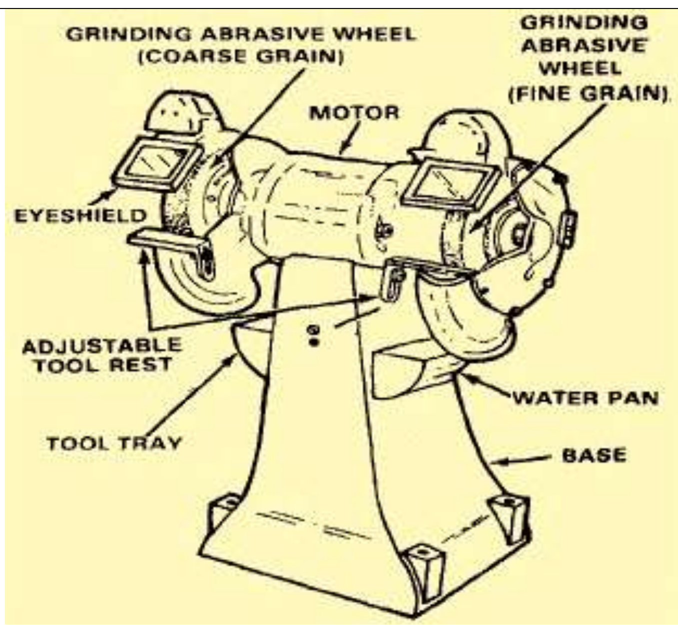


Figure 5-1. Floor mounted utility grinding machine

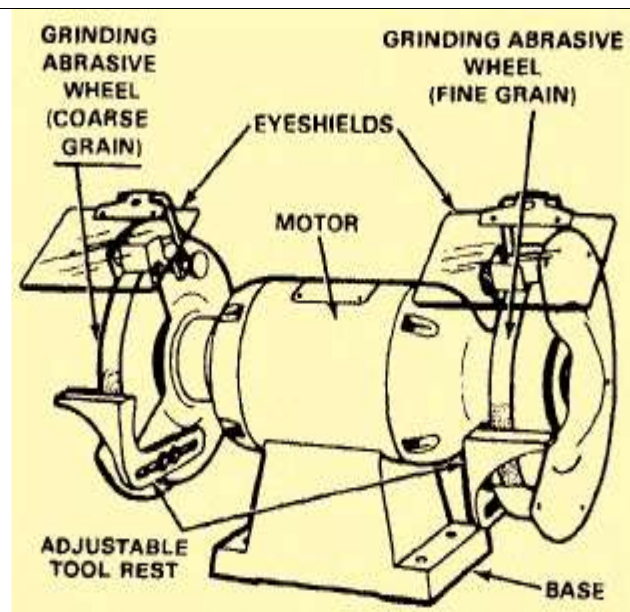


Figure 5-2. Bench-type utility grinding machine.

Bench-Type Utility Drill Grinding Machine: The bench-type drill grinding machine is intended for sharpening many types of drills. The accuracy of this type of grinder is not dependent on the capacity and skill of the operator since the drill is placed in a holding device.

Bench-Type Utility Grinding and Buffing Machine: The bench-type utility grinding and buffing machine is more suitable for miscellaneous grinding, cleaning, and buffing, but not recommended for tool grinding since it contains no tool rests, eye shields, or wheel guards. This machine normally mounts a 4 inch-diameter wire wheel on one end. The wire wheel is used for cleaning and the abrasive wheel is used for general grinding. One of the two wheels can be removed and a buffing wheel mounted in its place for buffing and polishing. The 1/4-HP electric motor revolves at a maximum of 3,450 RPM. The maximum cutting speed of the 4-inch-diameter wheel is approximately 3,600 SFPM. (Figure 5-3).

Tool Post Grinding Machine: The tool post grinding machine, see Figure 5-4, is a machine tool attachment designed to mount to the tool post of engine lathes. As described in a later lesson, it is used for internal and external grinding of cylindrical workplaces.

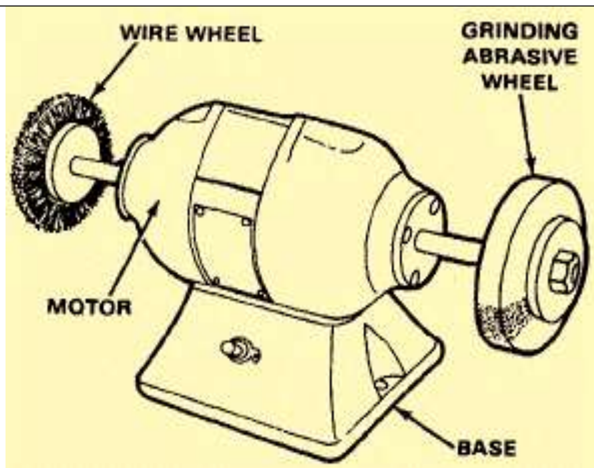


Figure 5-3. Bench-type utility grinding and buffing machine.

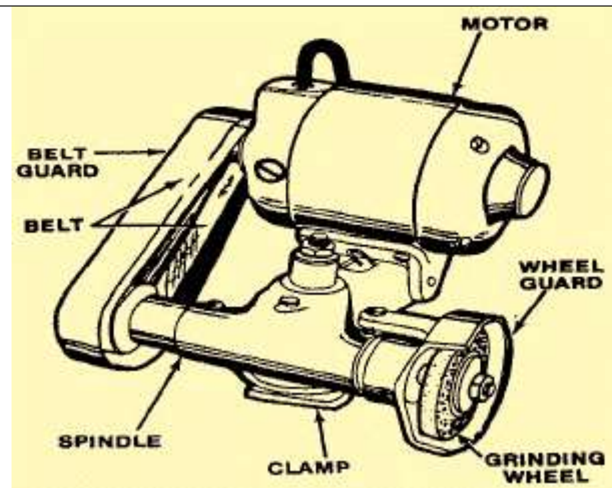


Figure 5-4. Tool post grinding machine.

Bench-Type Tool and Cutter Grinder: Is used to grind a large variety of small wood and steel cutters, as well as, slitting saw cutters up to 12 inches in diameter using the saw grinding attachment. The Bench-Type Tool and Cutter grinder was designed primarily to grind end mills. (Figure 5-5).

Milling and Grinding Lathe (Versa-Mill) Attachment: Also called as Versa-Mill, this attachment is a versatile machine tool attachment that is mounted on the carriage of a lathe. As described in a later lesson, it performs internal and external cylindrical grinding among other functions.

Surface Grinding Machine: Is used for grinding flat surfaces. The workpiece is supported on a rectangular table which moves back and forth and reciprocates beneath the grinding wheel. Reciprocating surface grinding machines generally have horizontal wheel spindles and mount straight or cylinder-type grinding abrasive wheels.

Reciprocating Surface Grinding Machine: Is a horizontal-type surface grinding machine. Workpieces are fastened to the table and can be moved beneath the grinding abrasive wheel by hand or power feed. A magnetic chuck maybe used for fastening the workpiece to the table. This grinding machine has an internal pump and piping network for automatic application and recirculation of a coolant to the workpiece and wheel. The grinding abrasive wheel, mounted on the horizontal spindle is straight and cuts on its circumferential surface only. Grinding wheel speeds are adjustable. (Figure 5-6).

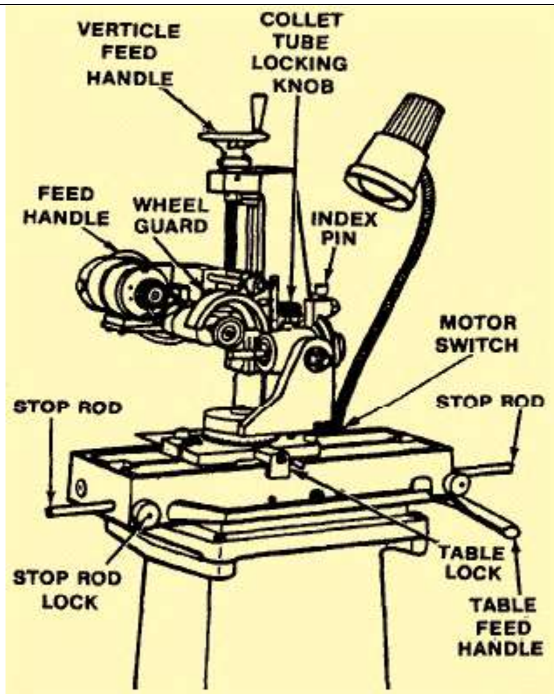


Figure 5-5. Bench-type tool and cutter grinder.

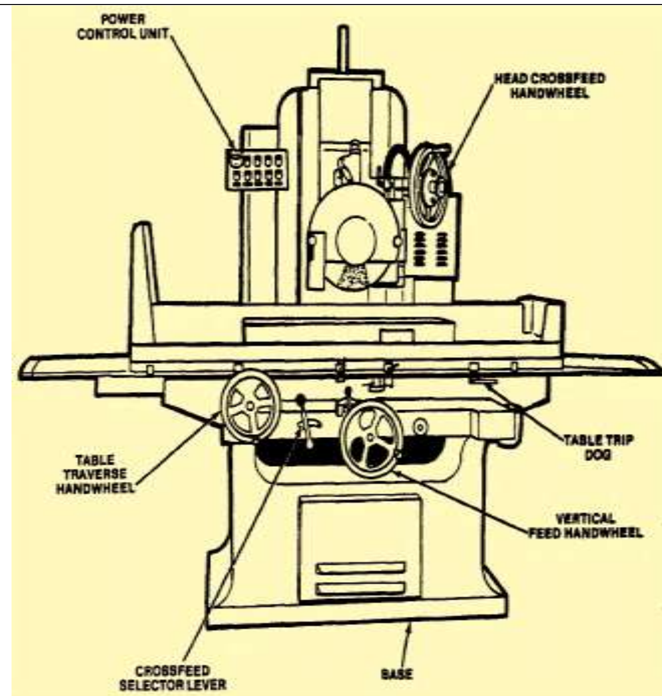


Figure 5-6. Reciprocating surface grinding machine.

The capacity of the **typical bench-type tool** and **cutter grinder** are:

Grinding wheel travel - 7 l/2-inch vertical.
Grinding wheel travel - 5 1/2-inch horizontal.
Table travel - 6 inches.
Slitting saws with attachment - 12-inch diameter.
Distance between centers - 14 inches.
Swing on centers (diameter) - 4 l/2-inch diameter.
Swing in work head (diameter) - 4 l/2-inch diameter.
Non-specialized cylindrical grinding machines include the tool post grinding machine and the Versa-Mil attachment.

2) Abrasives:

Most grinding wheels are made of silicon carbide or aluminum oxide, both of which are artificial (manufactured) abrasives. Silicon carbide is extremely hard but brittle. Aluminum oxide is slightly softer but is tougher than silicon carbide. It dulls more quickly, but it does not fracture easily therefore it is better suited for grinding materials of relatively high tensile strength.

Abrasive grain size: The abrasive grains are selected according to the mesh of a sieve through which they are sorted. For example, grain number 40 indicates that the abrasive grain passes through a sieve having approximately 40 meshes to the linear inch. A grinding wheel is designated coarse, medium, or fine according to the size of the individual abrasive grains making up the wheel.

3) Bonding Materials:

Bond. The abrasive particles in a grinding wheel are held in place by the bonding agent. The percentage of bond in the wheel determines, to a great extent, the "hardness" or "grade" of the wheel. The greater the percentage and strength of the bond, the harder the grinding wheel will be. "Hard" wheels retain the cutting grains longer, while "soft" wheels release the grains quickly.

If a grinding wheel is "too hard" for the job, it will glaze because the bond prevents dulled abrasive particles from being released so new grains can be exposed for cutting. Besides controlling hardness and holding the abrasive, the bond also provides the proper safety factor at running speed. It holds the wheel together while centrifugal force is trying to tear it apart. The most common bonds used in grinding wheels are vitrified, silicate, shellac, resinoid, and rubber.

Vitrified. A vast majority of grinding wheels have a vitrified bond. Vitrified bonded wheels are unaffected by heat or cold and are made in a greater range of hardness than any other bond. They adapt to practically all types of grinding with one notable exception: if the wheel is not thick enough, it does not withstand side pressure as in the case of thin cutoff wheels.

Silicate. Silicate bond **releases** the abrasive grains **more readily** than vitrified bond. Silicate bonded wheels are well suited for grinding, where heat must be kept to a minimum, such as grinding edged cutting tools. It is **not suited** for heavy-duty grinding. Thin cutoff wheels are sometimes made with a **shellac bond** because it provides a fast cool cutting.

Resinoid. Resinoid bond is strong and flexible. It is widely used in snagging wheels (for grinding irregularities from rough castings), which operate at 9,500 SFPM. It is also used in cutoff wheels.

Rubber. In rubber-bonded wheels, pure rubber is mixed with sulfur. It is extremely flexible at operating speeds and permits the manufacture of grinding wheels as thin as 0.006 inch for slitting nibs. Most abrasive cutoff machine wheels have a rubber bond.

4) Abrasive Grades of Hardness:

The grade of a grinding wheel designates the hardness of the bonded material. Listed below are examples of those grades. A soft wheel is one on which the cutting particles break away rapidly while a hard wheel is one on which the bond successfully opposes this breaking away of the abrasive grain. Most wheels are graded according to hardness by a letter system. Most manufacturers of grinding abrasive wheels use a letter code ranging from A (very soft) to Z (very hard). Vitrified and silicate bonds usually range from very soft to very hard, shellac and resinoid bonds usually range from very soft to hard, and rubber bonds are limited to the medium to hard range.

The grade of hardness should be selected as carefully as the grain size. A grinding abrasive wheel that is too soft will wear away too rapidly; the abrasive grain will be discarded from the wheel before its useful life is realized. On the other hand, if the wheel is too hard for the job, the abrasive particles will become dull because the bond will not re-lease the abrasive grain, and the wheel's efficiency will be impaired.

Figure 5-8 illustrates sections of three grinding abrasive wheels with different spacing of grains. If the grain and bond materials in each of these are alike in size and hardness, the wheel with the wider spacing will be softer than the wheel with the closer grain spacing. Thus, the actual hardness of the grinding wheel is equally dependent on grade of hardness and spacing of the grains or structure.

5) Abrasive Wheel & Structure Markings:

Bond strength of a grinding wheel is not wholly dependent upon the grade of hardness but depends equally on the structure of the wheel, that is, the spacing of the grain or its density. The structure or spacing is measured in number of grains per cubic inch of wheel volume.

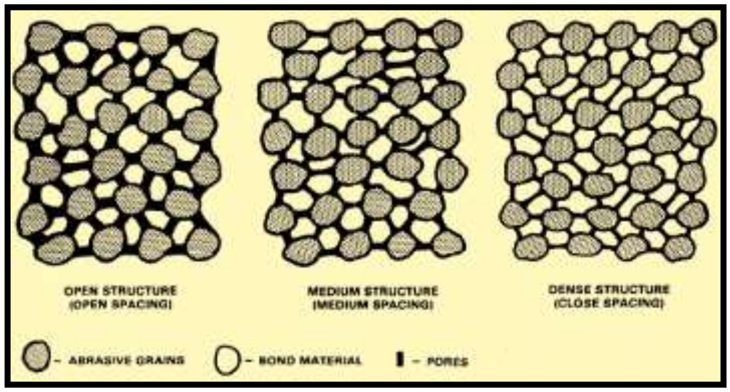


Figure 5-8. Grinding wheel abrasive.

Every grinding wheel is marked by the manufacturer with a stencil or a small tag. The manufacturers have worked out a standard system of markings, shown in Figure 5-9.

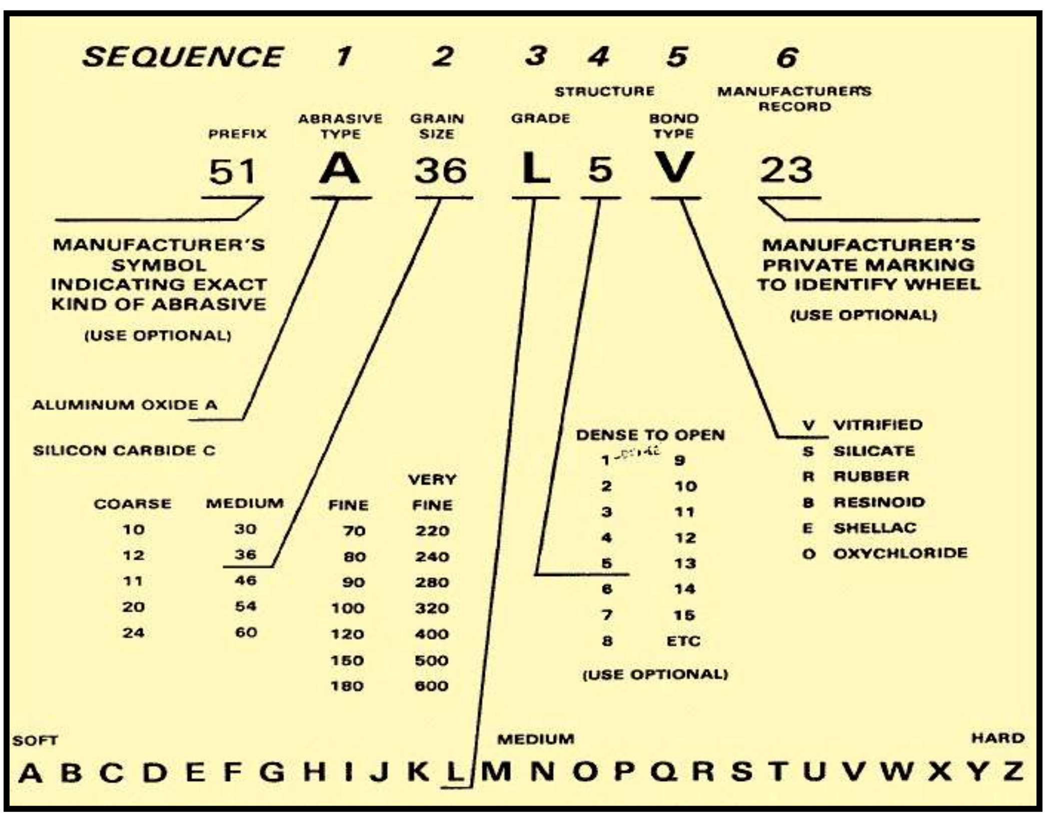


Figure 5-9. Standard system of markings.

For an example use a wheel marked A36-L5-V23. The A refers to the abrasive which is aluminum oxide. The 36 represents the grain size. The L shows the grade or degree of hardness, which is medium. The 5 refers to the structure of the wheel and the V refers to the bond type.

6) Standard Shapes of Grinding Wheel Faces:

Figure 5-10 illustrates standard shapes of grinding wheel faces. The nature of the work dictates the shape of the face to be used. For instance, shape A is commonly used for straight cylindrical grinding and shape E for grinding threads.

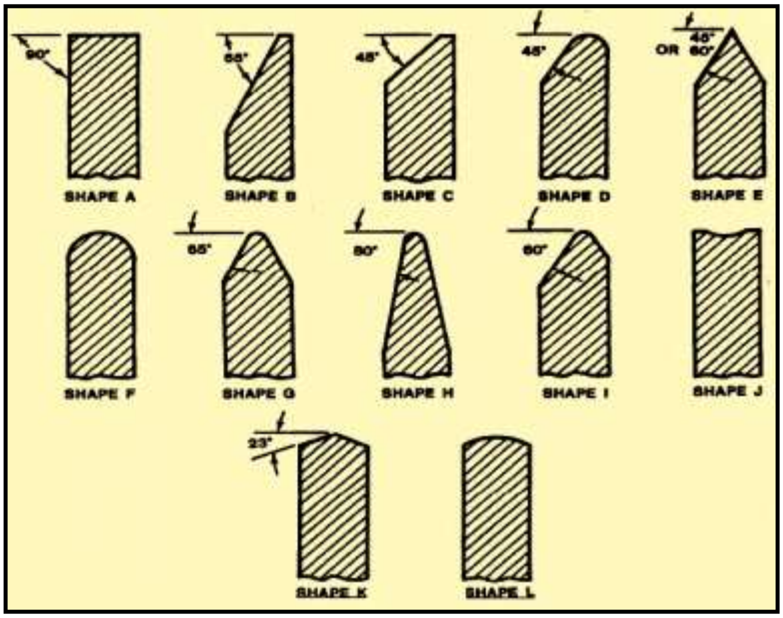


Figure 5-10. Standard shapes of grinding wheel faces.

7) Selection of Grinding Wheels:

Conditions under which grinding wheels are used vary considerably, and a wheel that is satisfactory on one machine may be too hard or soft for the same operation on another machine. The following basic factors are considered when selecting grinding wheels, though it should be understood that the rules and conditions listed are flexible and subject to occasional exceptions.

Tensile Strength of Material: The tensile of material to be ground is the main factor in the selection of the abrasive to be used. Two types of abrasives are suited to different materials as shown below.

<u>Silicon Carbide</u>	<u>Aluminum Oxide</u>
Gray and chilled iron	Carbon steels
Brass and soft bronze	Alloy steels
Aluminum and copper	High speed steels
Marble and other stone	Annealed malleable iron
Rubber and leather	Wrought iron
Very hard alloys	Hard bronzes
Cemented carbides	Unannealed malleable iron

Factors Affecting the Grain Size: Grain size to be chosen when selecting a grinding wheel depends upon the factors described below:

- ✓ The softer and more ductile the material, the coarser the grain size.
- ✓ The larger the amount of stock to be removed, the coarser the grain size.
- ✓ The finer the finish desired, the finer the grain size.
- ✓ Factors Affecting the Grade of Hardness
- ✓ The factors described below determine the proper grade of hardness of the grinding wheel.
- ✓ The harder the material, the softer the wheel.
- ✓ The smaller the arc of contact, the harder the grade should be. The arc of contact is the arc, measured along the periphery of the wheel that is in contact with the work at any instance. It follows that the larger the grinding wheel, the greater is the arc of contact and, therefore, a softer wheel can be used.
- ✓ The higher the wheel speed, the milder the grinding action and harder the grade should be.
- ✓ The better the condition of the grinding machine and spindle bearings, the softer the wheel can be.

Factors Affecting the Structure: The structure or spacing of the abrasive grains of wheel depends upon the four factors described below:

- ✓ The softer, tougher, and more ductile the material, the wider the grain spacing.
- ✓ The finer the finish desired, the closer or denser the grain spacing should be.
- ✓ Surfacing operations require open structure (wide grain spacing).
- ✓ Cylindrical grinding tool and cutter grinding are best performed with wheels of medium structure (medium grain spacing).

Factors Affecting Bonding Material: The factors described below affect the selection of bonding material for the wheel desired, are:

- ✓ Thin cutoff wheels and other wheels subject to bending strains require resinoid, shellac, or rubber bonds.
- ✓ Solid wheels of very large diameters require a silicate bond.
- ✓ Vitrified wheels are usually best for speeds up to 6,500 SFPM and resinoid, shellac, or rubber wheels are best for speeds above 6,500 SFPM.
- ✓ Resinoid, shellac, or rubber bonds are generally best where a high finish is required.

Inspection of Grinding Wheels: When a grinding wheel is received in the shop or removed from storage, it should be inspected closely for damage or cracks. Check a small wheel by suspending it on one finger or with a piece of string. Tap it gently with a light nonmetallic instrument, such as the handle of a screwdriver (Figure 5-11).

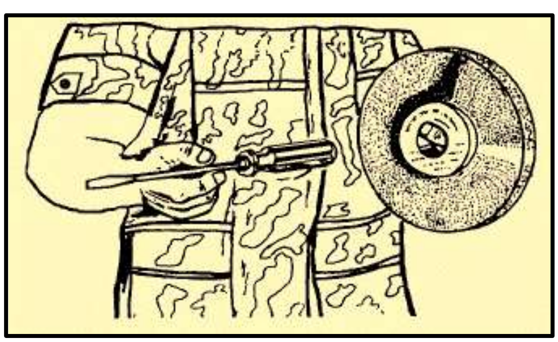


Figure 5-11. Checking for cracks.

Check a larger wheel by striking it with a wooden mallet. If the wheel does not give a clear ring, discard it. All wheels do not emit the same tone; a low tone does not necessarily mean a cracked wheel. wheels are often filled with various resins or greases to modify their cutting action, and resin or grease deadens the tone. Vitrified and silicate wheels emit a clear metallic ring. Resin, rubber, and shellac bonded wheels emit a tone that is less clear. Regardless of the bond, the sound of a cracked wheel is easy to identify.

Selection of Grinding Wheels – 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	60 & 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		

8) Mounting Grinding Wheels:

The proper mounting of a grinding wheel is very important. An improperly mounted wheel may become potentially dangerous at high speeds. The specified wheel size for the particular grinding machine to be used should not be exceeded either in wheel diameter or in wheel width. Figure 5-12 illustrates a correctly mounted grinding wheel.

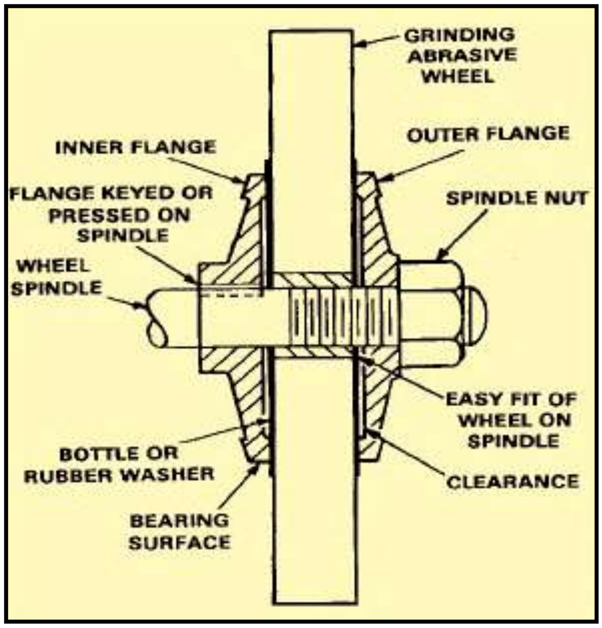


Figure 5-12. Correctly mounted wheel.

The following four items are methods and procedures for mounting grinding wheels:

- ✓ Note that the wheel is mounted between two flanges which are relieved on their inner surfaces so that they support the wheel only at their outer edges. This holds the wheel more securely with less pressure and with less danger of breaking. For good support, the range diameter should be about one-third of the wheel diameter.
- ✓ The spindle hole in the wheel should be no more than 0.002 inch larger than the diameter of the spindle, since a loose fit will result in difficulty in centering the wheel. If the spindle hole is oversize, select another wheel of the proper size. If no others are available, fit a suitable bushing over the spindle to adapt the spindle to the hole.
- ✓ Paper blotters of the proper size usually come with The grinding wheel. If the proper blotters are missing, cut them from heavy blotter paper (no more than 0.025-inch thick :) and place them between the grinding wheel and each flange. The blotters must be large enough to cover the whole area of contact between the flanges and the wheel. These blotters serve as cushions to minimize wheel breakage.
- ✓ When installing the grinding wheel on the wheel spindle, tighten the spindle nut firmly, but not so. tight that undue strain will be put on the wheel.

9) Dressing and Truing:

Dressing is cutting the face of a grinding wheel to restore its original cutting qualities. **Truing** is restoring the wheel's concentricity or reforming its cutting face to a desired shape. Both operations are performed with a tool called an abrasive wheel dresser (Figure 5- 13).

Grinding wheels wear unevenly under most general grinding operations due to uneven pressure applied to the face of the wheel when it cuts. Also, when the proper wheel has not been used for certain operations, the wheel may become charged with metal particles, or the abrasive grain may become dull before it is broken loose from the wheel bond. [n these cases, it is necessary that the wheel be dressed or trued to restore its efficiency and accuracy.

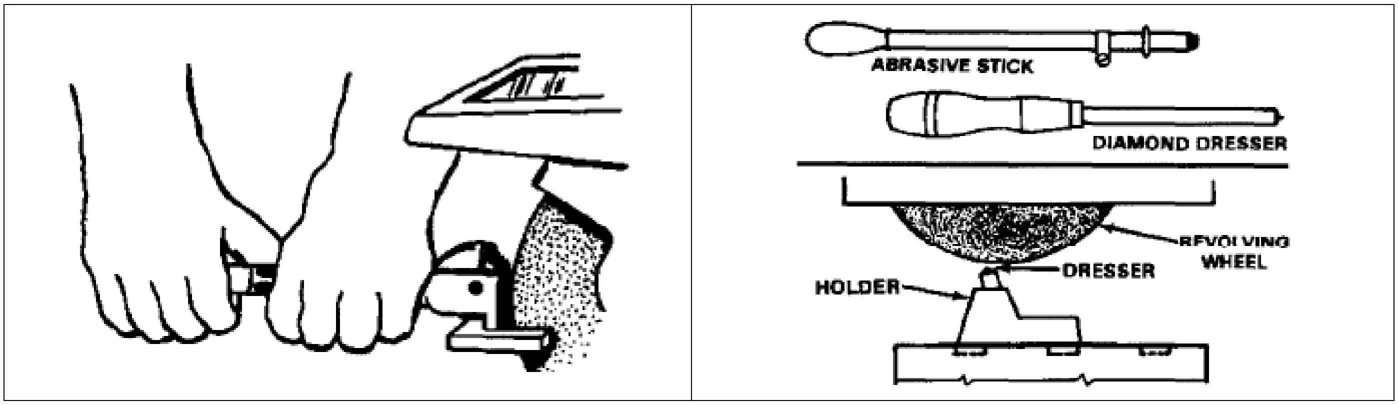


Figure 5-13. Dressing tools.

Mechanical Dresser: The hand-held mechanical dresser has alternate pointed and solid discs which are loosely mounted on a pin. This dresser is used to dress coarse-grit wheels and wheels used in hand grinding operations.

Abrasive Stick Dresser. The abrasive stick dresser comes in **two shapes**: square for hand use, and round for mechanical use. It is often used instead of the more expensive diamond dresser for dressing shaped and form wheels. It is also used for general grinding wheel dressing.

Abrasive Wheel Dresser: The abrasive wheel dresser is a bonded silicon carbide wheel that is fastened to the machine table at a slight angle to the grinding wheel and driven by contact with the wheel. This dresser produces a smooth, clean-cutting face that leaves no dressing marks on the work.

Diamond Dresser: The diamond dresser is the most efficient for truing wheels for precision grinding, where accuracy and high finish are required. A dresser may have a single diamond or multiple diamonds mounted in the end of a round steel shank. Inspect the diamond point frequently for wear. It is the only usable part of the diamond, and is worn away it cannot dress the wheel properly.

The correct use is to slant the diamond **3° to 15°** in the direction of rotation and 30° to the plane to prevent chatter and gouging. Rotate the diamond slightly in its holder between dressing operations to keep it sharp. A dull diamond will force the abrasive grains into the bond pores and load the face of the wheel, reducing the cutting ability.

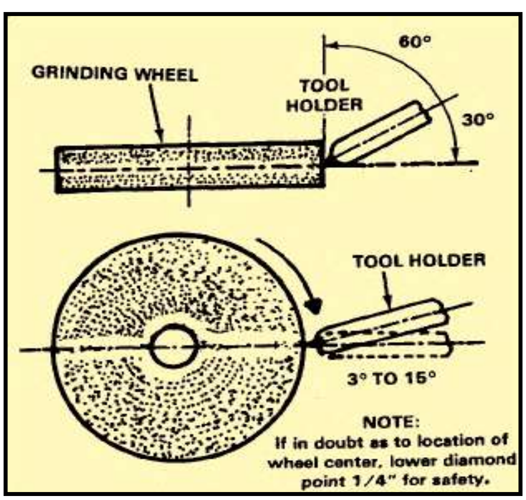


Figure 5-14. Position of the diamond dresser

When using a **diamond dresser to dress or true** a grinding wheel, the wheel should be turning at, or slightly less than, normal operating speed never at the higher speed. For wet grinding, flood the wheel with coolant when you dress or true it. For dry grinding, the wheel should be dressed dry. The whole dressing operation should simulate the grinding operation as much as possible. Whenever possible, hold the dresser by some mechanical device. It is a good idea to round off wheel edges with a handstone after dressing to prevent chipping.

This is especially true of a fine finishing wheel. Do not round off the edges if the work requires sharp corners. The grinding wheel usually wears more on the edges, leaving a high spot towards the center. When starting the dressing or truing operation, be certain that the point of the dressing tool touches the highest spot of the wheel first, to prevent the point from digging in. Feed the dresser tool point progressively, **0.001 inch at a time**, into the wheel until the sound indicates that the wheel is perfectly true.

The rate at which you move the point across the face of the wheel depends upon the grain and the grade of the wheel and the desired finish. A slow feed gives the wheel a fine finish, but if the feed is too slow, the wheel may glaze. A fast feed makes the wheel free cutting, but if the feed is too fast, the dresser will leave tool marks on the wheel. The correct feed can only be found by trial, but a uniform rate of feed should be maintained during any one pass.

10) Polishing, Buffing and Wire Wheels:

Polishing Wheels: Are formed of layers of cloth felt or leather glued or sewed together to form a flexible soft wheel, commonly made of canvas, felt, or leather sewed or glued together to provide various wheel grades from soft to hard. The harder or firmer wheels are generally used for heavier work while the softer and more flexible wheels are used for delicate contour polishing and finishing of parts on which corners and edges must be kept within rather strict specifications.

Buffing Wheels: Are usually made in the form of cakes, paste, or sticks which are applied to the wheel in this form. Polishing abrasives are fixed to polishing wheels with a glue.

OBS.: The **canvas** wheels are generally suitable for use with **medium grain abrasives**, while felt, leather, and muslin wheels are suitable for **fine grain abrasives**.

Wire Wheels: Consist of many strands of wire bound to a hub and radiating outward from the hub in the shape of a wheel. The wire wheel is used in place of a grinding wheel for cleaning operations such as removal of rust or corrosion from metal objects and for rough-polishing castings, hot-rolled steel, and so forth. The wire wheel fastens to the wheel spindle of the grinding machine in the same manner as a grinding wheel.

HACKSAW MACHINES:

All Hacksaw Machines are basically similar in design Figure 6-1 shows a typical power hacksaw and identifies its main parts, discussed below. Some machines are fed by gravity, the saw frame having weights that can be shifted to give greater or less pressure on the blade. Other machines are power fed with the feed being adjustable.

On these machines, the feed is usually stopped or reduced automatically when a hard spot is encountered in the material, thus allowing the blade to cut through the hard spot without breaking. The shift lever mechanism allows a number of strokes per minute to be changed so that a variety of metals may be sawed at the proper speeds. Some saws have a diagram showing the number of strokes per minute when the shift lever is indifferent positions; others are merely marked **"F," M," and "S"** (fast, medium, and slow).

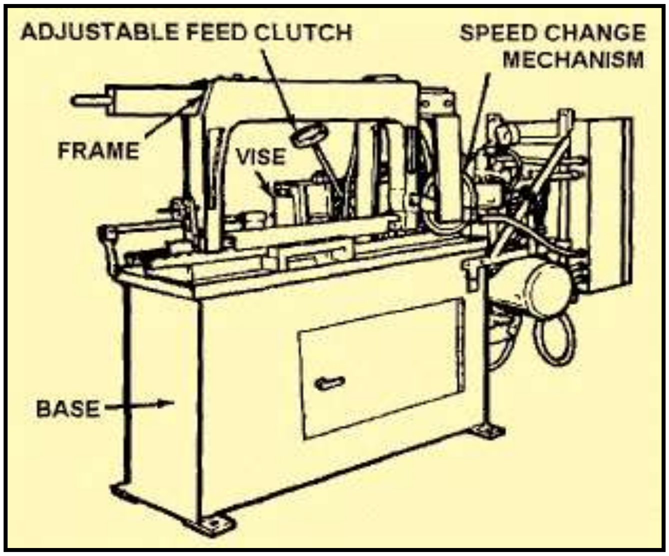


Figure 6-1: Power Hacksaw Machines

Base. The base of the saw usually contains a coolant reservoir and a pump for conveying the coolant to the work. The reservoir contains baffles which cause the chips to settle to the bottom of the tank. A table which supports the vise and the metal being sawed is located on top of the base and is usually referred to as part of the base.

Vise. The vise is adjustable so that various sizes and shapes of metal may be held. On some machines the vise may be swiveled so that stock may be sawed at an angle. The size of a power hacksaw is determined by the largest piece of metal that can be held in the vise and sawed.

Frame. The frame of the saw supports and carries the hacksaw blade. The machine is designed so that the saw blade contacts the work only on the cutting stroke. This. action prevents unnecessary wear on the saw blade. The cutting stroke is on the draw or back stroke.

Adjustable Feed Clutch: Is a ratchet-and-pawl mechanism that is coupled to the feed screw. The feed clutch may be set to a desired amount of feed in thousandths of an inch. Because of the ratchet-and-pawl action, the feed takes place at the beginning of the cutting stroke. The clutch acts as a safety device and permits slippage if too much feed pressure is put on the saw blade. It may also slip because of a dull blade or if too large a cut is attempted. This slippage helps prevent excessive blade breakage.

1) Bandsaw Machines:

Metal-Cutting Bandsaw machines fall into two basic categories:

- Vertical machines (Figure 6-2)
- Horizontal machines (Figure 6-3).

Bandsaws use a continuous saw blade. Chip removal is rapid, because each tooth is a precision cutting tool and accuracy can be held to close tolerances eliminating or minimizing many secondary machining operations.

a) Vertical Bandsaw Machine:

The metal-cutting vertical bandsawing machine, also called a contour machine, is made in a variety of sizes and models by several manufacturers. The size of a contour machine is determined by the throat depth, which is the distance from the saw band to the column. Figure 6-2 shows a typical contour machine and identifies its main parts, which are discussed below.

The head is the large unit at the top of the contour machine that contains the saw band idler wheel, the drive motor switch, the tension adjustment handwheel and mechanism, a flexible airline (directs a jet of air at the work to keep layout lines free from chips), and the adjustable post which supports the upper saw guide. The job selector dial is also located on the head.

The column contains the speed indicator dial, which is driven by a cable from the transmission and indicates the speed in feet per minute (FPM). The butt welder is also mounted on the column. The base contains the saw band drive wheel, the motor, and the transmission. The transmission has two speed ranges. The low range gives speeds from 50 FPM to 375 FPM. The high range gives speeds from 260 FPM to 1,500 FPM.

A shift lever on the back of the base can be placed in the high, low, or neutral position. Low is recommended for all speeds under 275 FPM. The base also supports the table and contains the lower saw band guide, which is mounted immediately under the table slot. The power feed mechanism is located within the base, and the feed adjustment handle and foot pedal are located on the front of the base.

b) Horizontal Bandsaw Machine:

The horizontal bandsawing machine does the same job as the power hacksaw but does it more efficiently. The blade of the bandsaw is actually a continuous band which revolves around a drive wheel and idler wheel in the band support frame. Two band guides use rollers to twist the band so that the teeth are in the proper cutting position. The guides should be adjusted so that they are just slightly further apart than the width of the material to be cut. This will give maximum support to the saw band and help assure a straight cut.

The vise on the horizontal bandsaw is much like the one on the power hacksaw. However, the horizontal bandsaw has a much greater capacity for large stock than does the power hacksaw. The stationary jaw can be set at several angles, adjusted automatically to whatever position the stationary jaw is in when the vise handwheel is tightened.

The horizontal bandsaw is operated hydraulically by controls on a control box, which is located on the front side of the machine. A motor and pump assembly supplies hydraulic fluid from a reservoir in the base to a cylinder, which raises and lowers the support arm and also controls the feed pressure and band tension.

A speed and feed chart is sometimes provided on the machine, but when it is not, consult the operator's manual for the proper settings for sawing. A coolant pump is located in one of the legs of the base, which serves as a coolant reservoir. The coolant cools the saw band and also washes away chips from the cut before they can clog the band.

Variable speed unit: Is located within the base of the machine. This unit consists of two V-type pulleys which are mounted on a common bearing tube. A belt on one pulley is driven by the transmission, while the belt on the other pulley drives the saw band drive wheel. The two outside cones of the pulleys are fixed, but the middle cone is shifted when the speed change wheel is turned. A shift in the middle cone causes the diameter of one pulley to increase and the diameter of the other pulley to decrease. This slowly changes the ratio between the two pulleys and permits a gradual increase or decrease in the speed of the machine.

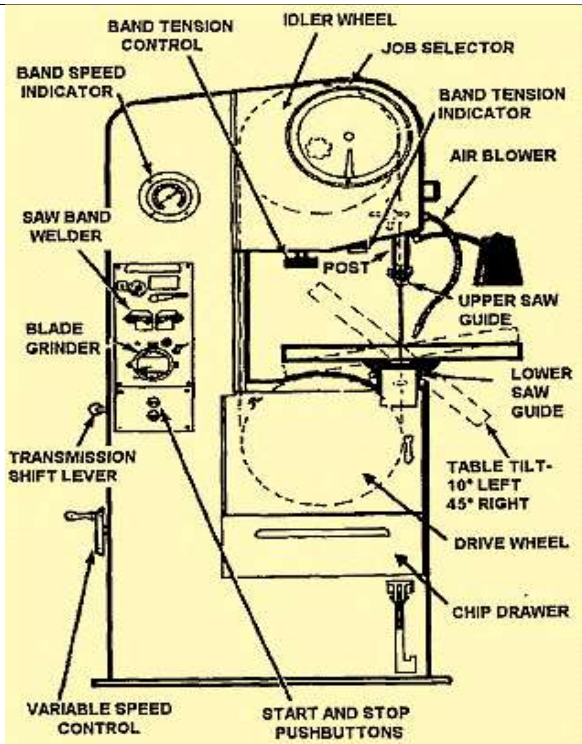


Figure 6-2. Vertical bandsawing

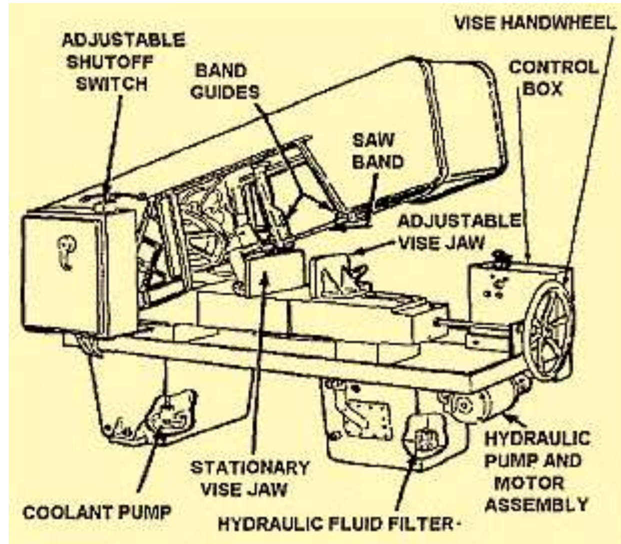


Figure 6-3. Horizontal bandsaw.

2) Power Hacksaw Blades:

Power hacksaw blades differ from hand hacksaw blades in that they are generally heavier, made in longer sizes, and have fewer teeth per inch. Hacksaw blades are discarded when they become dull; sharpening is not practical. Materials commonly used in manufacturing power hacksaw blades are high-speed tungsten steel and high-speed molybdenum steel.

On some blades only the teeth are hardened, leaving the body of the blade flexible. Other blades are hardened throughout. The set is the amount of bend given the teeth. The set makes it possible for a saw to cut a kerf or slot wider than the thickness of the band back (gage), thus providing side clearance. This is the pattern in which the teeth are set. There are three set patterns: raker, wave, and straight, as shown in Figure 6-4.

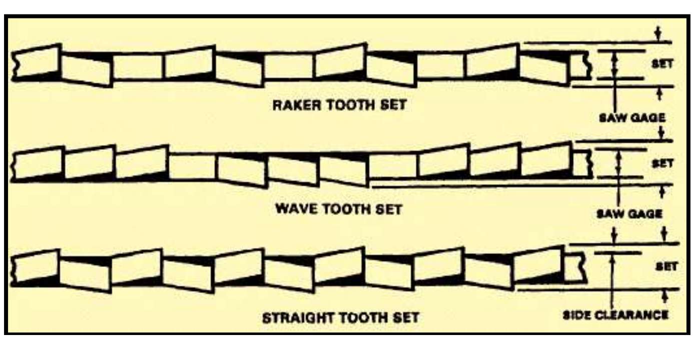


Figure 6-4. Set patterns

The pitch of hacksaw blade teeth (Figure 6-5) is expressed as the number of teeth per linear inch of blade. For example, a blade having 10 teeth per inch is said to be 10 pitch.

MATERIAL	HACKSAW BLADE TEETH PER INCH (PITCH)
SHEET METAL	14
SOLID STOCK: 1	
ALUMINUM	4
BRASS	10
BRONZE	4
CAST IRON	4
COPPER	4
STEEL, ALLOY	6
STEEL, HIGH-SPEED	6
STEEL, MACHINE	4
STEEL, STAINLESS	6
STEEL, TOOL (ANNEALED)	6
STEEL, TOOL (UNANNEALED)	4
TUBING, THIN	14
TUBING, HEAVY	10

1. Three or more teeth must contact the workpiece at all times to prevent blade damage. If the recommended pitch for a material fails to meet this requirement, a blade with more teeth to the inch should be used.

Figure 6-5. Selection of power hacksaw blades.

OBS.: Note that sheet metal and tubing are listed separately from solid stock. It is assumed that solid stock will be sufficiently thick that three or more teeth will be in contact with the stock at all times.

Power hacksaw blades are coarser in pitch (fewer teeth per inch) than hand hacksaw blades. Common pitches for power hacksaw blades range from 4 to 14 teeth per inch. The following guidelines for the selection of power hacksaw blades are:

- ✓ Select power hacksaw blades for material to be cut.
- ✓ Soft materials require a coarser blade to provide adequate spaces between the teeth for removal of chips. Hard material requires a finer blade to distribute the cutting pressure to a greater number of teeth, thereby reducing wear to the blade.
- ✓ At least three teeth must be in contact with the workpiece at all times or the blade will snag on the workpiece and break teeth from the blade. Therefore, a blade must be selected with sufficient pitch so that three or more teeth will be in contact with the workpiece, no matter what type of material is being cut.

3) Bandsaw Blades:

Bandsaw blades are manufactured in two forms. They are supplied in rolls of 50 to 500 feet for use on machines that have butt welders for forming their own blade bands. Bandsaw blades are also supplied in continuous welded bands for machines having no provisions for welding thickest saw band possible. However, consider the curvature of the cut, since wide saw blades cannot cut sharp curves.

Figure 6-6 shows saw band selection for various radii. They are supplied in rolls of 50 to 500 feet for use on machines that have butt welders for forming their own blade bands. Bandsaw blades are also supplied in continuous welded bands for machines having no provisions for welding.

Materials: Bandsaw blades are made from special alloy steels. The blades are made flexible by annealing the body of the blade and hardening only the teeth. Metal cutting bandsaw blades have their teeth bent (Figure 6-4). This bend produces a kerf slightly wider than the thickness of the blade, which prevents the blade from being pinched by the stock. There are three set patterns: raker, wave, and straight, as shown in Figure 6-4.

Pitch: The pitch of bandsaw blades is expressed as the number of teeth per linear inch of the blade. Metal cutting blades range from 6 to 32 teeth per inch, the coarser tooth blades being used for sawing large stock and soft metals.

Selection of Bandsaw Blades: Select bandsaw blades according to the type of material to be cut, the thickness of the material to be cut, and the sawing operation to be performed. Always use the widest and thickest saw band possible. However, consider the curvature of the cut, since wide saw blades cannot cut sharp curves. Figure 6-6 shows saw band selection for various radii. For general sawing, use the raker set pattern. The wave set pattern is used where thin work sections are encountered during the cut, such as tubing, angles, and channels.

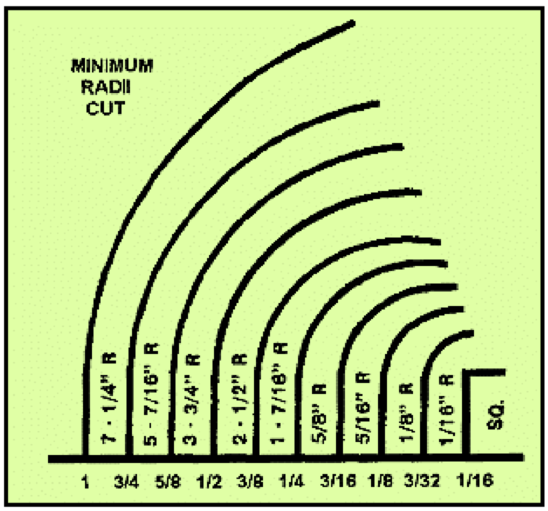


Figure 6-6. Bandsaw selection for various radii.

Three teeth of the bandsaw blade must be in contact with the workpiece at all times to prevent chatter and shearing off teeth. Therefore, use fine tooth blades to cut sheet metal and tubing. If the sheet metal is too thin to meet this requirement with the finest tooth blade available, place the metal between plywood fiberboard, or thicker metal. Figure 6-7 is a guide for selecting the proper pitch band saw blade for different metals and metal thickness.

MATERIAL	BANDSAW BLADE (TPI)	MATERIAL	BANDSAW BLADE (TPI)
SHEET METAL UNDER 1/8 INCH THICK	24-32	SOLID STOCK CONTINUED	
SHEET METAL OVER 1/8 INCH THICK	18	STEEL, ALLOY	12-14
SOLID STOCK: 1		STEEL,HIGH-SPEED	12-14
ALUMINUM	6-10	STEEL, MACHINE	10-14
BRASS	10-12	STEEL, STAINLESS	12-14
BRONZE	12-14	STEEL, TOOL	12-14
CAST IRON	10-12	TUBING UNDER 1/8-INCH WALL THICKNESS	24-32
COPPER	10-12	TUBING OVER 1/8-INCH WALL THICKNESS	18

1. Three or more teeth must contact the workpiece at all times to prevent shearing of the blade teeth. If the recommended pitch for solid stock fails to meet this requirement, a blade with finer pitch must be selected.

Figure 6-7. Selection of bandsaw blades.

OBS.: The finish depends largely upon the saw pitch. The faster the saw speed and finer the saw pitch, the finer the finish. Lubricating improves the finish. A fine saw pitch, high velocity, and light feed produce the finest finish.

Bandsaw Blade Wear: Bandsaw blades naturally become dull from prolonged use, but some conditions promote greater than normal wear on the blades. Blades dull quickly if used at too high a speed for the material being cut. Also, if the material to be cut is too hard for the pitch of the blade, abnormal wear will result. The most common cause of premature blade dulling occurs from using too fine a pitch blade and from feeding too heavily.

The following symptoms indicate a dull bandsaw blade. When these symptoms are noticed, the blade should be replaced.

- ✓ It becomes difficult to follow a line, the blade being forced to one side or the other.
- ✓ The chips are granular (except for cast iron, which produces granular chips with both sharp and dull blades).
- ✓ The bandsaw blade cuts workpiece is fed by hand. slowly or not at all when the
- ✓ With the machine stopped or the bandsaw blade removed, run a finger slowly over the teeth in the cutting direction. If sharp edges are not felt the blade is dull.

File Bands: The band sawing machine is adapted for filing by use of the band file attachment. A band file is fitted over the drive and idler wheels and in place of the bandsaw blade. The band is made up of several parts or segments which are riveted at one end (the leading end) to a spring steel band. The trailing end of each segment is free to lift during the time when the band bends over the drive and idler wheels of the band saw. When the band straightens out, the segments lock together. Figure 6-8 shows the construction of and terminology for file band parts.

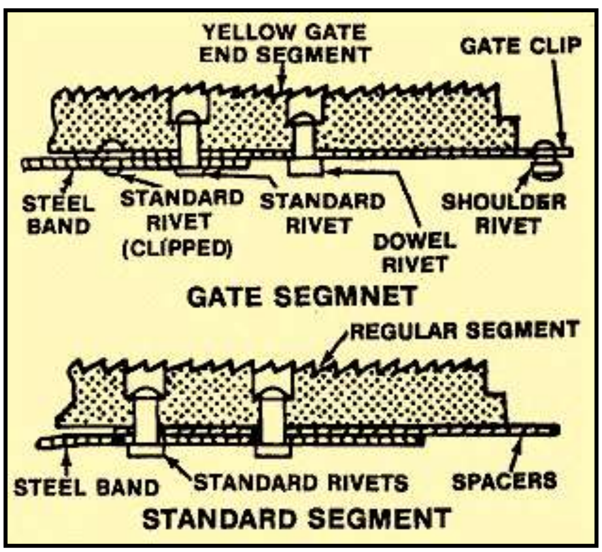


Figure 6-8. Construction and parts of a file band.

Note that the gate segment (a segment at one end of the band that is specially designed to allow the two band ends to be locked together) has a shoulder rivet and a dowel rivet protruding from beneath it. The shoulder rivet locks into the other file band end, and the dowel rivet aligns the two end segments and prevents the shoulder rivet from sliding out of the locked position during tiling. The gate segment of a file band is identified by yellow paint.

Cut of File Teeth: File bands are either coarse or bastard cut or normally range in pitch from 10 to 20 teeth per inch. The coarse **10-pitch bands** are used for filing **softer metals** such as aluminum, brass, copper, and cast iron. A bastard-cut 14-pitch band is a good choice for general steel tiling, while 16 to 20 pitch bastards are recommended for filing tool steel.

Choose band files on the basis of workpiece thickness and type of material to be filed. In general, the thicker the workpiece, the coarser the file should be. This is due to a large; chip accumulation from the larger area of the workpiece, thus requiring additional space for the chips between the teeth. On thin sheet metal, a fine pitch file is required to prevent chatter. Use fine pitch files for filing tough carbon and alloy steels; use coarser pitch files for filing softer metals. Figure 6-9 is provided to aid in selecting the proper file for filing specific materials.

BAND FILE		
MATERIAL	CUT OF TEETH	TEETH PER INCH
ALUMINUM	SHORT ANGLE- OR BASTARD-CUT	10-12
BRASS	SHORT ANGLE- OR BASTARD-CUT	10-12
BRONZE	SHORT ANGLE- OR BASTARD-CUT	10-12
CAST IRON	SHORT ANGLE- OR BASTARD-CUT	10-12
COPPER	SHORT ANGLE- OR BASTARD-CUT	10-12
FIBER	SHORT ANGLE- OR BASTARD-CUT	10-12
MAGNESIUM	SHORT ANGLE- OR BASTARD-CUT	10-12
STEEL, ALLOY	BASTARD-CUT	14-24
STEEL, MACHINE	BASTARD-CUT	14-16
STEEL, TOOL	BASTARD-CUT	14-24

Figure 6-9. Selection of band files.

Care and Cleaning of Band Files: Clean the file often, using a stiff brush or a file card. Move the brush in the direction of each cut of the file to dislodge all particles hidden between the teeth. The file band should not be coiled into more than three loops. The best means of storing file bands is in a cabinet looped over a 16-inch radius support with the ends hanging free.

Band File Attachment: A band file attachment (Figure 6-10) is provided with most bandsaw machines to permit the use of band files. A typical band file attachment consists of a band file guide and upper and lower guide supports that attach to the frame and part of the band saw. A special filing filler plate is provided to adapt the table slot to the extra width and depth required for the band file and file band guide.

4) Hacksawing Best Practices:

Power hacksawing machines cut by drawing the hacksaw blade toward the motor end of the machine. At the completion of this movement called the draw stroke, the hacksaw blade is lifted slightly to clear the material being cut and moved an equal distance in the opposite direction.

Power Hacksaw Speeds: Since the cutting speed of hacksawing machines is measured in strokes per minute, the length of the stroke is an important consideration. A longer stroke at a given speed will cut faster than a shorter stroke at the same speed. Thus, to obtain a proper cutting speed the length of the stroke must be specified.

Length of the Stroke: For most power hacksaws is between 4 and 10 inches depending upon the size of the machine. On machines with an adjustable stroke, the wider the stock being cut, the shorter the stroke to prevent the blade holders from hitting the stock. With most power hacksaws, the stroke length is adjustable within 2 or 3 inches.

and on some machines more than one speed can be selected. On single-speed hacksawing machines, the speed must be regulated by changing the stroke.

If the stroke is doubled the machine will cut twice as fast, and if the stroke is decreased by one-half, the machine will cut half as fast. This proportion can be applied to any fraction to increase or decrease the cutting speed of the machine. The speeds given in the chart, Figure 6-16, bellow are for example only.

The correct speeds for cutting various metals will depend on the type of machine you are using. In general the faster speeds are used for cutting soft materials and the slower speeds are used for cutting harder materials. If a recommended speed cannot be approximated either by changing the stroke or changing the speed, the feed can be decreased to prevent undue wear to the hacksaw blade.

MATERIAL	SPEED IN STROKES PER MINUTE	
	4 TO 6 INCH	8 TO 10 INCH
ALUMINUM	135	65
BRASS	135	65
BRONZE	90	45
CAST IRON	90	45
COPPER	135	65
STEEL, ALLOY	90	45
STEEL, HIGH-SPEED	60	30
STEEL, MACHINE	135	65
STEEL, STAINLESS	60	30
STEEL, TOOL (ANNEALED)	90	45
STEEL, TOOL (UNANNEALED)	60	30

Figure 6-16. Power hacksawing machine speeds.

Power hacksaw machines having a **mechanical feed** can usually be regulated to feed the saw downward **from 0.001 to 0.025 inch per stroke**, depending upon the type and size of the material to be cut. On these machines, a device to stop the feed when hard spots are encountered is usually incorporated into the design.

The feed of machines having gravity feed is regulated by the weight of the saw frame and any additional weights or springs that might be connected or attached to the frame to increase or decrease the downward force of the hacksaw blade. Maximum and minimum blade pressures obtainable are determined by the manufacturer of the hacksawing machine, and are specified as relatively light or heavy. The following **general rules** apply for selecting proper feeds for hacksawing machines:

- ✓ The feed should be very light when starting a cut and can be increased after the cut is well started.
- ✓ Hard materials require a lighter feed than soft materials; reduce the feed when welds or hard spots in materials are encountered.
- ✓ Wide material requires a heavier feed than narrow material because the pressure is distributed over a larger surface.
- ✓ Sharp hacksaw blades will cut well with lighter feeds. Heavier feeds are necessary for cutting with dull blades.

Bandsawing Speeds: Proper bandsaw speeds are important in conserving bands blades. Too great a speed for the material being cut will cause abnormally rapid blade wear, while too slow a speed will result in inefficient production. The chart of recommended speeds (Figure 6-17) are guidelines only. It shows the speeds for a given type of machine. The cutting speed always depends on the type of machine you are using and the manufactures' recommendations. All bandsawing machines have several cutting speeds. Since the diameter of the drive wheel of the bandsaw machine establishes a fixed ratio between the motor or transmission speed in RPM to the blade speed in FPM, it is not necessary to convert RPM into FPM as with most other machine tools. The speeds are identified in FPM on the sawing machine speed selector controls. Some machines have a speed indicator so a careful check of sawing speeds may be made when the machine is operating with or without a load. In general the **following principles** apply to speeds of bandsaw blades:

- ✓ The harder the material, the slower the speed; conversely, the softer the material, the faster the speed.
- ✓ The faster the speed, the finer the finish produced on the cut surface. This principle applies to light feeds in conjunction with fast feeds.

MATERIAL	BANDSAWING SPEED (fpm)	MATERIAL	BANDSAWING SPEED (fpm)
ALUMINUM	200 TO 2,000	RUBBER, HARD	150 TO 250
BAKELITE	200 TO 900	STEEL, ALLOY	50 TO 100
BRASS, SOFT	175 TO 300	STEEL, HIGH CARBON	50 TO 100
BRASS, HARD	75 TO 150	STEEL, HIGH-SPEED	50 TO 90
BRASS, SHEETS	200 TO 900	STEEL, MACHINE	75 TO 175
BRONZE	75 TO 150	STEEL, SHEEET	150 TO 200
CAST IRON	50 TO 100	STEEL, STAINLESS	50 TO 75
COPPER	115 TO 175	STEEL, TOOL	50 TO 150
MONEL METAL	50 TO 100		

Figure 6-17. Bandsawing speeds.

Horizontal Bandsawing Machine Feeds: Feed of horizontal bandsaw machines is controlled by adjusting the pressure applied by the saw blade against the material being cut, as with hacksawing machines. The horizontal saw has a spring counterbalance and a sliding weight to adjust the pressure of the blade. When the sliding weight is moved toward the pivot point of the saw frame the band saw blade pressure is reduced. When the weight is moved away from the pivot point, the pressure is increased. The following **general principles** apply when regulating the feed of horizontal band saw machines:

- ✓ The feed should be very light when starting a cut. After the cut is started, increase the feed.
- ✓ Wider material requires a heavier feed than narrow material.
- ✓ Wide blades will stand greater pressure than narrow blades and can therefore be used with heavier feeds.
- ✓ A lighter feed is required for hard materials; a heavier feed can be used for soft materials.
- ✓ Reduce the feed when hard spots in the material are encountered such as chilled spots in cast iron and welds in joined sections.

Vertical Bandsawing Machine Feeds: With vertical machines, the feed is the pressure applied to the saw blade by the material being cut. The workpiece may be hand fed or power fed depending upon the operation to be performed. Cutting curves or special contours requires that the workpiece be guided and fed into the saw blade by hand. The power feed on bandsaw machines is operated by adjustable weights in the machine pedestal.

The weights are connected by cables to one of the work-holding attachments of the sawing machine to pull the workpiece against the bandsaw blade. To operate the power feed, the weights are raised by depressing a pedal and the cables are then fixed to the work-holding attachment. When the pedal is released the weights pull the piece into the blade. The following general rules apply to feeding workpieces on bandsawing machines:

- ✓ The feed should be light when starting a cut. The pressure can be increased after the cut is established.
- ✓ Hard materials require lighter feeds than softer materials.
- ✓ Wider band saw blades will stand greater pressure than narrow blades and can therefore be used with heavier feeds.
- ✓ When hard spots in the material being cut are encountered, reduce the feed until the spots are cut through.
- ✓ Use a light feed when cutting curves; straight-line cutting.

Power Hacksawing: The power hacksaw machine is designed primarily for straight-line sawing. A typical sawing operation is outlined below:

- ✓ Select a hacksaw blade of the proper length for the machine and proper pitch for the material to be cut.
- ✓ Install the hacksaw blade with the teeth pointing downward and toward the motor end of the hacksaw.
- ✓ Check the alignment of the vise and hacksaw blade and mount the workpiece in the vise. Make sure the vise holds the workpiece securely.
- ✓ Check the stroke and adjust if necessary. Readjust the position of the vise if necessary.
- ✓ Position the hacksaw blade about 1/4 inch above the workpiece and set the feed control to its lightest feed setting.
- ✓ Set the desired speed of the hacksawing machine. Start the machine and let the blade feed lightly into the workpiece for about 1/4 inch. Readjust the feed to whatever the material will stand for normal cutting.
- ✓ Permit the hacksaw blade to cut completely through workpiece. The blade frame will trip a switch on sawing machine bed to stop the sawing machine.

Horizontal Bandsawing: Like hacksawing machines, the horizontal bandsaw machine is used primarily for straight-line sawing. The typical sequence of operation for this machine is:

- ✓ Select and install a bandsaw blade of the proper pitch for the type and size of material to be cut.
- ✓ Set the vise to the desired angle and check the angle by measuring it from the line of the band saw blade.
- ✓ Mount the workpiece in the vise. Make sure the work piece is secured and will not loosen during cutting.
- ✓ Check the alignment of the blade guides for vertical positioning and adjust if necessary.
- ✓ Position the saw frame so that the bandsaw blade is 1/4 inch above the workpiece. The power feed weight should be placed at its lightest feed setting.
- ✓ Set the desired speed on the horizontal band sawing machine. Start the machine and let the bandsaw blade cut into the workpiece about 1/4 inch. After the cut has been established, readjust the feed weight to exert the desired amount of pressure on the workpiece.

Vertical Bandsawing: Straight-line sawing is performed on the vertical band saw machine by using one or a combination of several mechanisms or attachments:

- ✓ The miter guide attachment, with or without power feed,
- ✓ With or without the work-holding jaw device with power feed and angular blade guide attachment.

The miter guide attachment on some machines can be connected to the power feed mechanism and on others must be fed by hand. The workpiece is clamped or handheld against the miter guide attachment and the workpiece and attachment are moved on a track parallel to the blade, thereby assuring a straight-line cut. The work-holding jaw device on some machines can be connected to the power feed to produce straight-line cuts (Figure 6-18).

The angular blade guide attachment is used for straight-line sawing when the workpiece cannot be cut in the usual manner because it is too large or too long to clear the column of the bar, sawing machine frame. A typical example of straight-line sawing is outlined below:

- ✓ Select a band saw blade of the desired pitch for the nature of material to be cut. The blade should be as wide as possible for straight-line sawing.
- ✓ Set the desired speed on the bandsawing machine. Position the workpiece at the desired angle in one of the machine attachments and connect the cable to the power feed mechanism if power feed is to be used.
- ✓ Start the bandsawing machine and feed the workpiece lightly into the blade to start the cut. Once the cut is started, the feed can be increased. If feeding is by hand, the pressure applied to the workpiece by the operator can be varied to find the best cutting conditions.

Radius Sawing: Is performed on the bandsaw by either guiding the workpiece by hand or by using the disk-cutting attachment. Care must be taken to select a bandsaw blade of the proper width for the radius or circle to be cut. If the blade is too wide for the radius, the heel of the blade will press against the outer edge of the kerf (Figure 6-19). When the heel contacts this edge, any further twisting of the workpiece in an attempt to cut a sharper radius will twist the bandsaw blade and may result in the blade breaking.

Cutting Pressure: When cutting a radius, apply a slight side pressure at the inner cutting edge of the bandsaw blade (Figure 6-19). This pressure will give the blade a tendency to provide additional clearance.

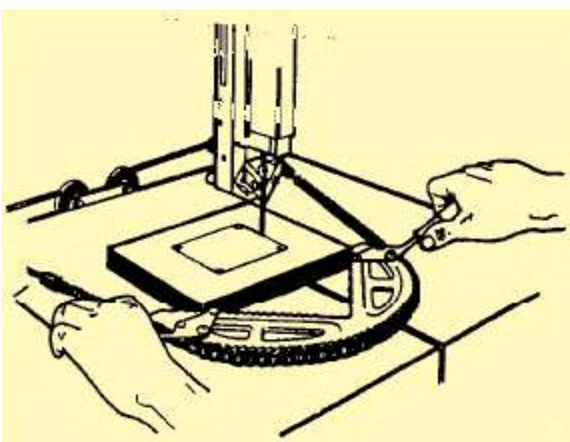


Figure 6-18. Work holding jaw device used for straight-line sawing with power feed

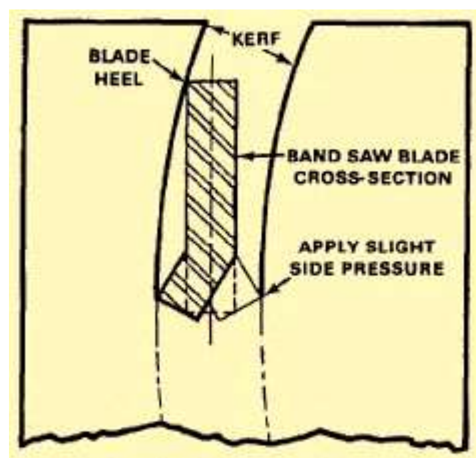


Figure 6-19. Radius limitation for bandsaw blade

Contour Sawing: Is the process of cutting shapes in which the direction of the cut must be changed at intervals. Holes larger in diameter than the width of the saw blade must be drilled at each corner where a change of direction of the bandsaw blade will occur. Figure 6-20 illustrates the methods of changing direction of a cut at a hole.

Sawing Away From the Hole: To saw away from the hole on a line tangent to the hole, the saw blade must cut away from the center of the hole, or the blade will bow and cause a belly in the cut. The cut should be started as in A, Figure 6-20, in which a curve is cut outward from the hole to meet the layout line, leaving apiece of excess metal which can be removed later by filing. An alternate method is shown at B, Figure 6-20, in which a section of metal is notched out with a saw blade by several short cuts to give the blade clearance for starting the cut along the layout line.

Sawing Toward the Hole: The diagrams at C and D, Figure 6-20, show the proper method of sawing up to a hole in two cuts. The excess metal can be removed later by tiling. After the shape is cut and the slug or waste material is removed, the comers should be finished by filing or notching. The bandsaw blade should not be used for these operations because the blade will bow and cut unevenly.

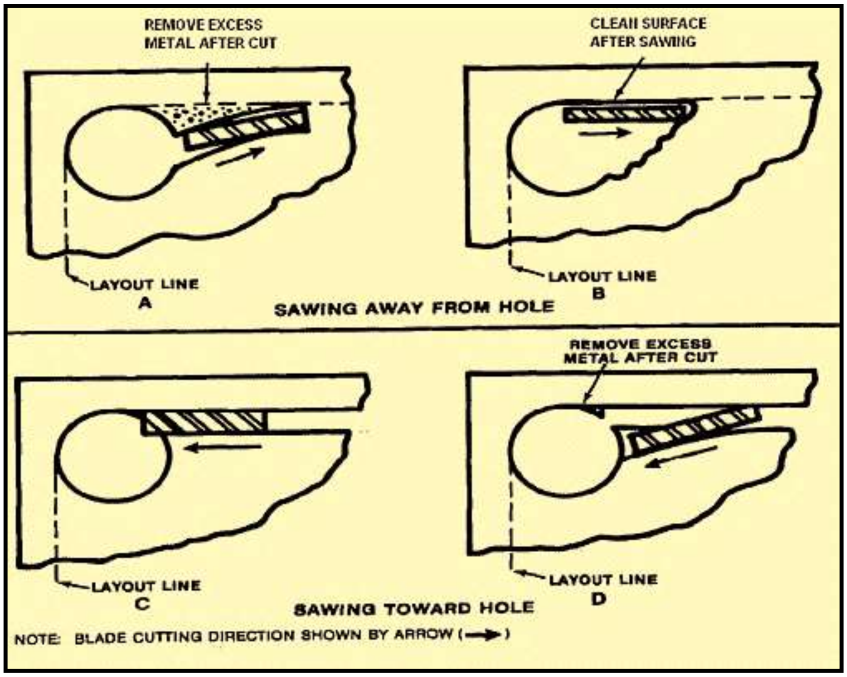


Figure 6-20. Methods of sawing toward and away from holes.

5) Hacksaw Polishing Bands:

Polishing can be performed on the bandsaw using a polishing attachment and polishing band. The polishing band is usually 1.0 inch wide and has a heavy fabric backing.

Types of Polishing Bands: Polishing bands for bandsawing machines are usually supplied in various grain sizes of aluminum-oxide or silicon carbide abrasive: No 50 grain (coarse) for heavy stock removal and soft material, No 80 (medium) for general surface finishing, and No 120 or No 150 grain (fine) for high polishing and light stock removal.

Selection of Polishing Bands: Polishing bands should be selected according to the particular job to be performed, For removing tool marks and deburring edges, use the No 50 grain polishing band. Finer grain polishing bands should not be used on soft metals like aluminum or cast iron because the band will quickly fill with metal particles, reducing the cutting action.

Polishing Attachment: The polishing attachment (Figure 6-11), similar to the band file attachment, provides support for the polishing band. The polishing band plate acts as a solid backing for the polishing band to prevent stretching and distorting the band when the workpiece is held against it. Use a polishing band filler plate to fill the table slot so the workpiece can be supported close to the polishing band.

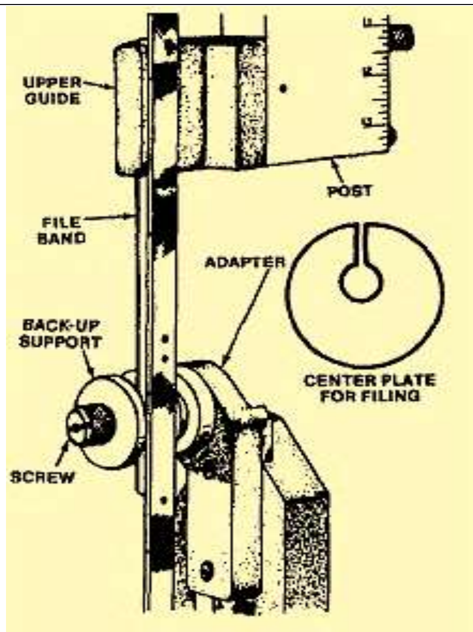


Figure 6-10. Band file attachment installed on a band sawing machine.

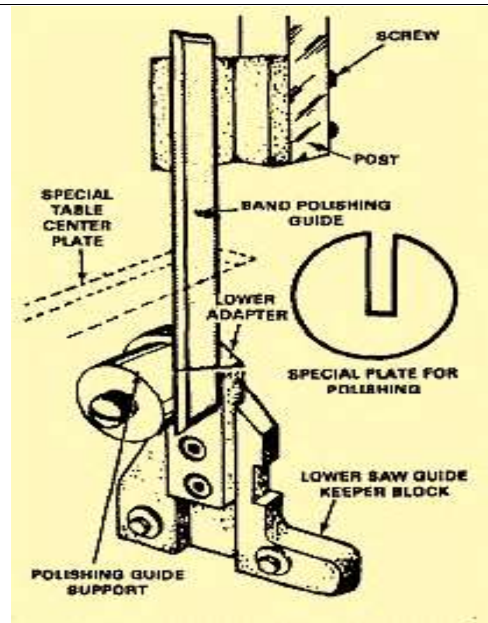


Figure 6-11. Polishing attachment installed on a band sawing machine.

6) Disc-Cutting Attachment:

Use the disc-cutting attachment (Figure 6-12) to saw internal or external circles and discs. The diameter of the circle that can be cut is limited to the length of the cylindrical bar on the attachment or to the throat depth of the machine. The disc-cutting attachment consists of three main parts a clamp and cylindrical bar, which is fastened to the saw guidepost; an adjustable arm, which slides on the cylindrical bar; and a pivot or centering pin. The disc must be laid out and center-drilled to a depth of 1/8 inch to 3/16 inch to provide a pivot point for the centering pin. The centerline of the centering pin must be in line with the front edge of the saw teeth and at the desired distance from the saw band.

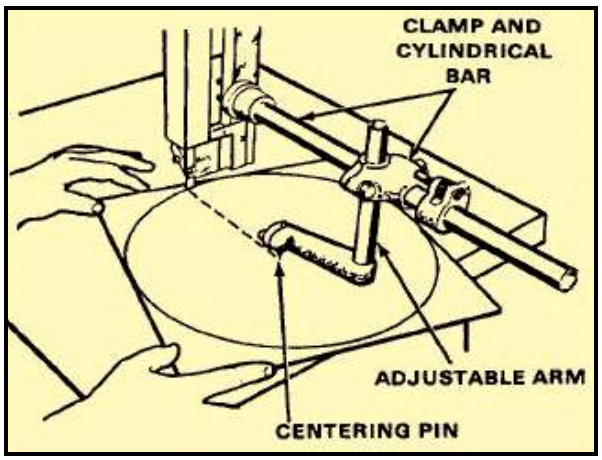


Figure 6-12. Disc-cutting attachment.

Angular Blade Guide Attachment: This attachment (Figure 6-13) twists the blade so that long workpieces that would not normally clear the machine column can be cut. The blade is twisted to a 30 degree angle on most machines.

Miter Guide Attachment: Is illustrated in Figure 6-14. The workpiece is supported against the miter head which attaches to the slide arm. The attachment can be set at an angle with a protractor, using the table slot as a reference line. A gage rod can be extended from the attachment and used as a stop when identical lengths are sawed. When not in use, swing the attachment on the slide rod so that it hangs below the table.

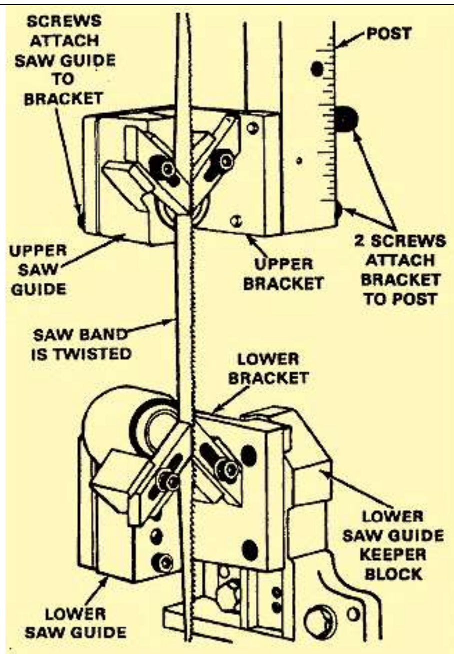


Figure 6-13. Angular saw guides.

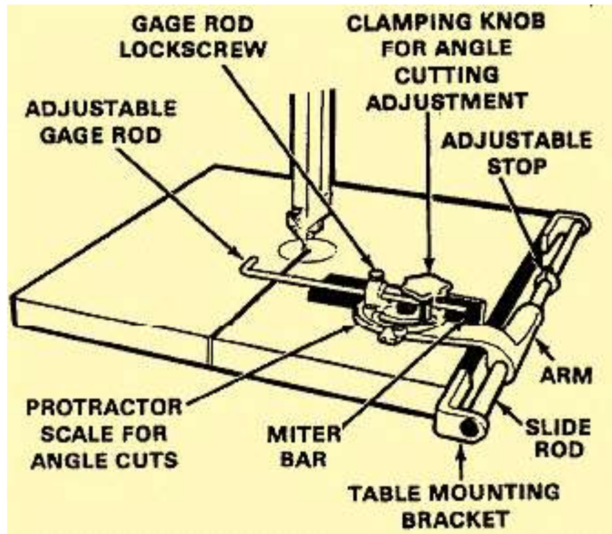


Figure 6-14. Miter guide attachment.

7) Hacksaw Machine Coolants:

Most sawing machines are **dry cutting** machines; that is, they are not intended for use with liquid coolants. However, some power hacksaws and horizontal bandsaws are equipped with a coolant attachment. Soluble oil products, when mixed with water to form emulsions, are used for these machines. This type of coolant has proven very satisfactory for sawing where cooling is an important factor. Most manufacturers of water oil emulsion coolants add a rust inhibitor to the solution to prevent rusting caused by the water in the coolant.

LATHES:

Lathes can be divided into three types for easy identification: engine lathes, turret lathes, and special purpose lathes. Small lathes can be bench mounted, are lightweight, and can be transported in wheeled vehicles easily. The larger lathes are floor mounted and may require special transportation if they must be moved. Field and maintenance shops generally use a lathe that can be adapted to many operations and that is not too large to be moved from one work site to another.

The engine lathe (Figure 7-1) is ideally suited for this purpose. A trained operator can accomplish more machining jobs with the engine lathe than with any other machine tool. Turret lathes and special purpose lathes are usually used in production or job shops for mass production or specialized parts. while basic engine lathes are usually used for any type of lathe work. Further reference to lathes in this chapter will be about the various engine lathes.

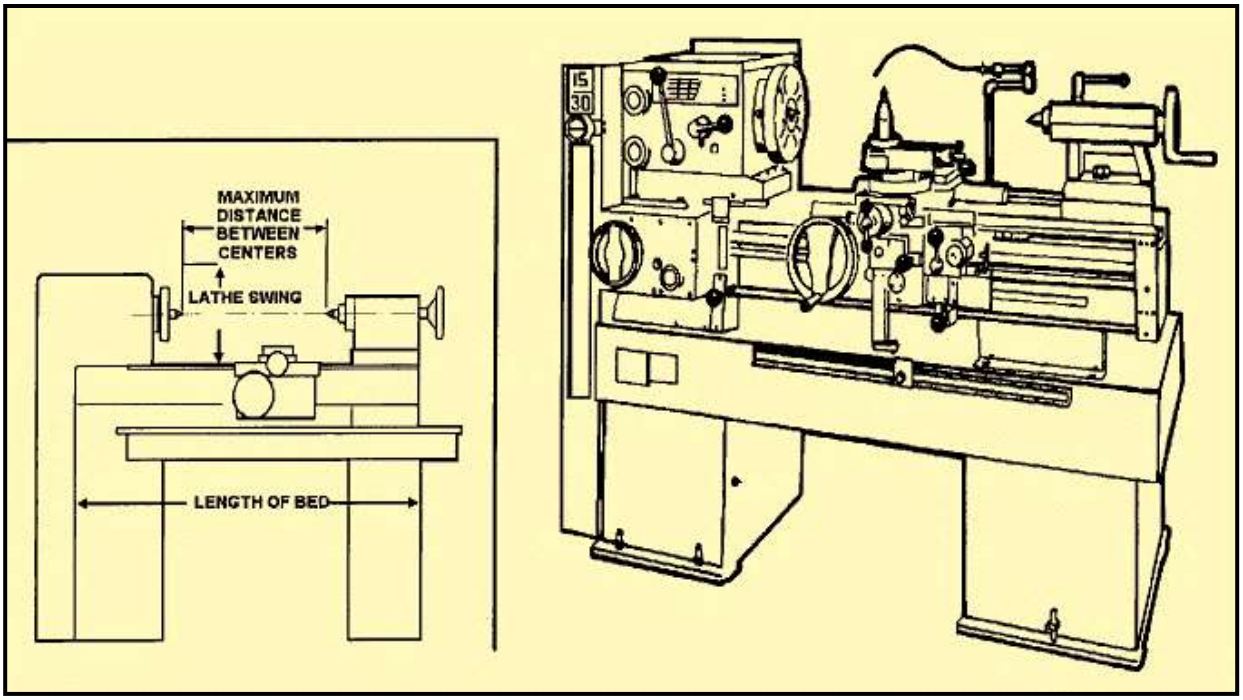


Figure 7-1. Lathe Categories.

1) Engine Lathes:

Application: Lathes are used in woodturning, metalworking, metal spinning, thermal spraying, maintenance parts, glass-working, shape pottery and many other operations. Most suitably equipped metalworking lathes can also be used to produce most solids of revolution, plane surfaces and screw threads or helices. Ornamental lathes can produce three-dimensional solids of incredible complexity.

Sizes: The size of an engine lathe is determined by the largest piece of stock that can be machined. Before machining a workpiece, the following measurements must be considered: the diameter of the work that will swing over the bed and the length between lathe centers (Figure 7-1).

Categories: Slight differences in the various engine lathes make it easy to group them into three categories: light-weight bench engine lathes, precision tool room lathes, and gap lathes, which are also known as extension- type lathes. These lathe categories are shown in Figure 7-2 Different manufacturers may use different lathe categories.

Lightweight: Lightweight bench engine lathes are generally small lathes with a swing of 10 inches or less, mounted to a bench or table top. These lathes can accomplish most machining jobs, but may be limited due to the size of the material that can be turned.

Precision: Precision tool room lathes are also known as standard manufacturing lathes and are used for all lathe operations, such as turning, boring, drilling, reaming, producing screw threads, taper turning, knurling, and radius forming, and can be adapted for special milling operations with the appropriate fixture.

Tool room lathe: This type of lathe can handle workplaces up to 25 inches in diameter and up to 200 inches long. However, the general size is about a 15-inch swing with 36 to 48 inches between centers. Many tool room lathes are used for special tool and die production due to the high accuracy of the machine.

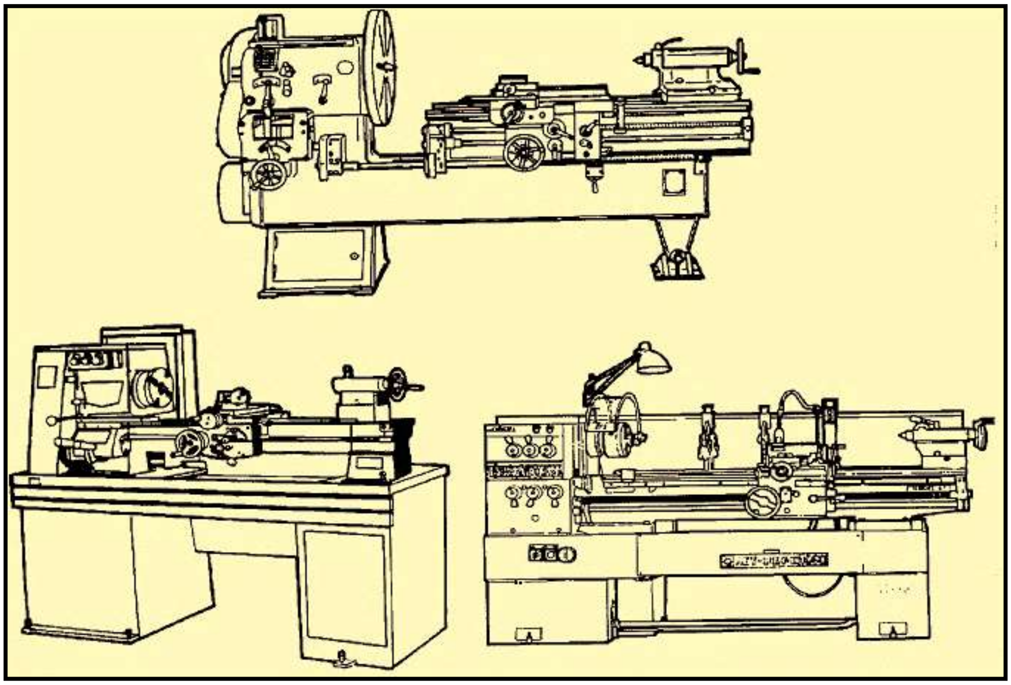


Figure 7-2. Lathe categories.

Gap or extension-type lathes: Are similar to toolroom lathes except that gap lathes can be adjusted to machine larger diameter and longer workplaces The operator can increase the swing by moving the bed a distance from the headstock, which is usually one or two feet. By sliding the bed away from the headstock, the gap lathe can be used to turn very long workplaces between centers.

2) Lathe Components:

Engine Lathes: All have the same general functional parts even though the specific location or shape of a certain part may differ from one manufacturer. The bed is the foundation of the working parts of the lathe to another (Figure 7-3).

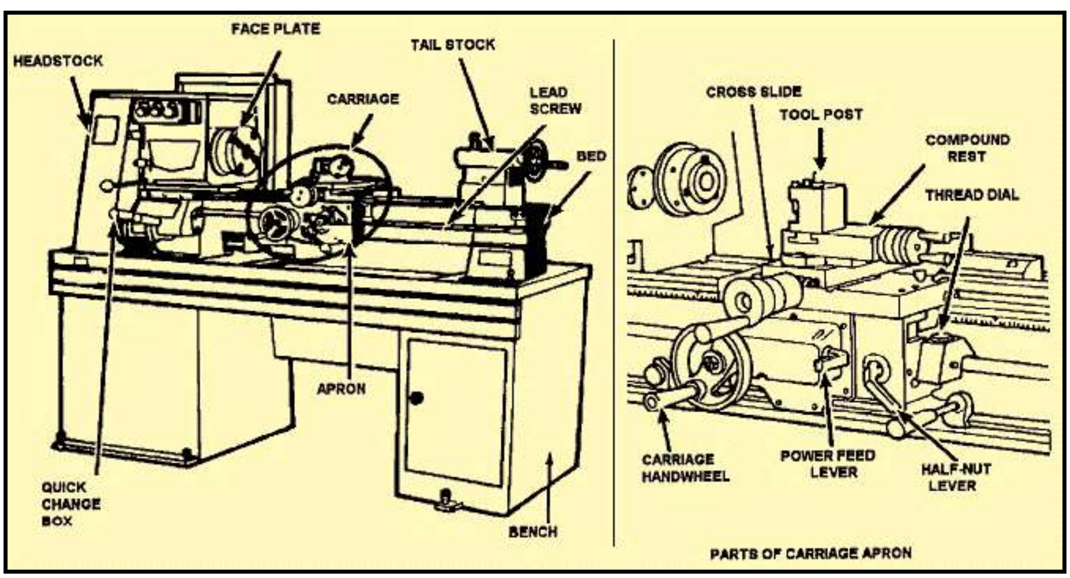


Figure 7-3. Lathe components.

The Headstock: Is located on the operator's left end of the lathe bed. It contains the main spindle and oil reservoir and the gearing mechanism for obtaining various spindle speeds and for transmitting power to the feeding and threading mechanism. The headstock mechanism is driven by an electric motor connected either to a belt or pulley system or to a geared system.

The Tailstock: May be adjusted laterally (toward or away from the operator) by adjusting screws. It can be moved by hand or by power and can be clamped into position with a locking nut. The saddle carries the cross slide and the compound rest. The cross slide is mounted on the dovetail ways on the top of the saddle and is moved back and forth at 90° to the axis of the lathe by the cross slide lead screw.

The Lead Screw: Can be hand or power activated. A feed reversing lever, located on the carriage or headstock, can be used to cause the carriage and the cross slide to reverse the direction of travel. The cutting tool and tool holder are secured in the tool post which is mounted directly to the compound rest. The apron contains the gears and feed clutches which transmit motion from the feed rod or lead screw to the carriage and cross slide.

The Main Spindle: Is mounted on bearings in the headstock and is hardened and specially ground to fit different lathe holding devices. The spindle has a hole through its entire length to accommodate long workplaces. The hole in the nose of the spindle usually has a standard Morse taper which varies with the size of the lathe.

Centers, Collets, Drill Chucks, Tapered Shank Drills and Reamers: Are located on the opposite end of the lathe from the headstock. It supports one end of the work when machining between centers, supports long pieces held in the chuck, and holds various forms of cutting tools, such as drills, reamers, and taps. The tailstock is mounted on the ways and is designed to be clamped at any point along the ways. It has a sliding spindle that is operated by a hand wheel and clamped in position by means of a spindle clamp.

3) Lathe Cutting Tools:

The lathe cutting tool or tool bit must be made of the correct material and ground to the correct angles to machine a workpiece efficiently. The most common tool bit is the general all-purpose bit made of high-speed steel. These tool bits are generally inexpensive, easy to grind on a bench or pedestal grinder, take lots of abuse and wear, and are strong enough for all-around repair and fabrication.

High-Speed Steel Tool Bits: Can handle the high heat that is generated during cutting and are not changed after cooling. These tool bits are used for turning, facing, boring and other lathe operations. Tool bits made from special materials such as carbides, ceramics, diamonds, cast alloys are able to machine workplaces at very high speeds but are brittle and expensive for normal lathe work. High-speed steel tool bits are available in many shapes and sizes to accommodate any lathe operation.

Single Point Tool Bits: Can be one end of a high-speed steel tool bit or one edge of a carbide or ceramic cutting tool or insert. Basically, a single point cutter bit is a tool that has only one cutting action proceeding at a time. A machinist or machine operator should know the various terms applied to the single point tool bit to properly identify and grind different tool bits (Figure 7-4).

The Shank: Is the main body of the tool bit. The nose is the part of the tool bit which is shaped to a point and forms the corner between the side cutting edge and the end cutting edge. The nose radius is the rounded end of the tool bit. The face is the top surface of the tool bit upon which the chips slide as they separate from the work piece.

The Tool Flank: Is the surface just below and adjacent to the cuffing edge of the tool bit, which cuts the workpiece, located behind the nose and adjacent to the side and face. The base is the bottom surface of the tool bit, which usually is ground flat during tool bit manufacturing. The end of the tool bit is the near-vertical surface which, with the side of the bit, forms the profile of the bit. The end is the trailing surface of the tool bit when cuffing. The heel is the portion of the tool bit base immediately below and supporting the face.

Angles of Tool Bits: The successful operation of the lathe and the quality of work that may be achieved depend largely on the angles that form the cutting edge of the tool bit. The angles are the side and back rake angles, the side and end cutting edge angles, and the side and end relief angles. Other angles to be considered are the radius on the end of the tool bit and the angle of the tool holder. After knowing how the angles affect the cutting action, some recommended cutting tool shapes can be considered. (Figure 7-4).

Rake Angle: Is the top surface of the tool bit, and can be **positive, negative, or have no rake angle** at all. There are **two types** of rake angles: the side and back rake angles. The tool holder can have an angle, known as the tool holder angle, which averages about 150°, depending on the model of toot holder selected. The tool holder angle combines with the back rake angle to provide clearance for the heel of the tool bit from the workpiece and to facilitate chip removal. (Figure 7-4).

Rake angles cannot be too great or the cutting edge will lose strength to support the cuffing action. The side rake angle determines the type and size of chip produced during the cutting action and the direction that the chip travels when leaving the cutting tool. Chip breakers can be included in the side rake angle to ensure that the chips break up and do not become a safety hazard.

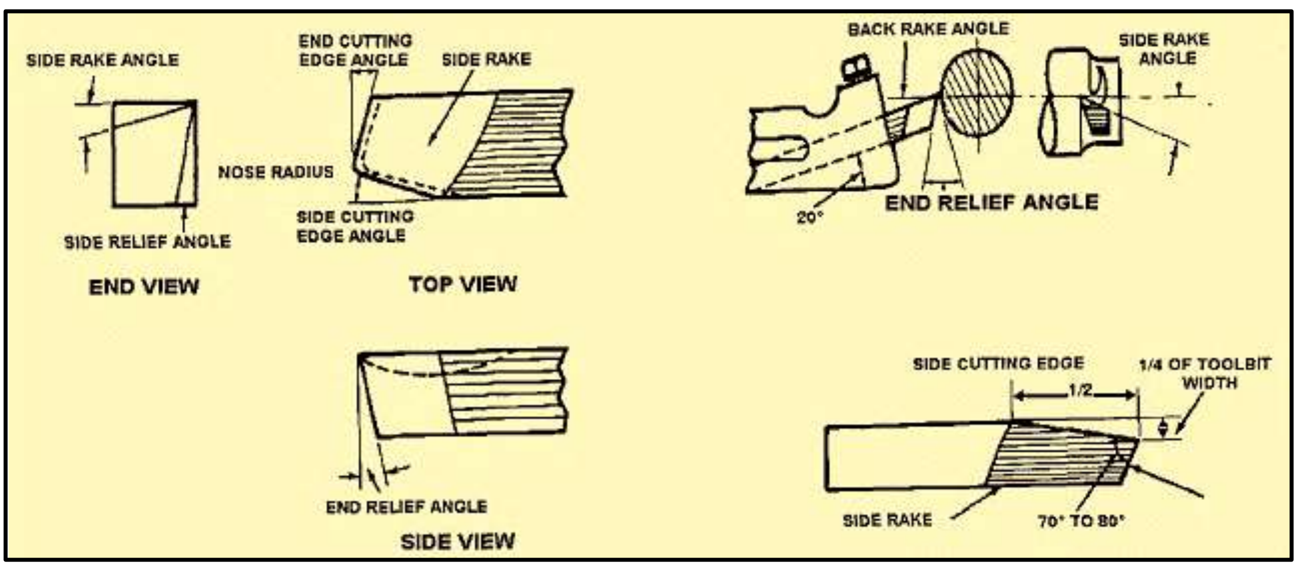


Figure 7-4. Tool bit angles.

Side and Relief Angles or Clearance Angles: Are the angles formed behind and beneath the cutting edge that provide clearance or relief to the cutting action of the tool. There are **two types** of relief angles: side relief and end relief. Side relief is the angle ground into the tool bit, under the side of the cutting edge, to provide clearance in the direction of tool bit travel. End relief is the angle ground into the tool bit to provide front clearance to keep the tool bit heel from rubbing. The end relief angle is supplemented by the tool holder angle and makes up the effective relief angle for the end of the tool bit.

Side and Cutting Edge Angles: Are the angles formed by the cutting edge with the end of the tool bit (the end cutting edge angle, or the side of the tool bit (the side cutting edge angle). The end cutting edge angle permits the nose of the tool bit to make contact with the work and aids in feeding the tool bit into the work. The side cutting edge angle reduces the pressure on the tool bit as it begins to cut. The side rake angle and the side relief angle combine to form the wedge angle providing the cutting action. A radius ground onto the nose of the tool bit can help strengthen the tool bit and provide for a smooth cutting action. (Figure 7- 4).

Shapes of Tool Bits: The overall shape of the lathe tool bits can be rounded, squared, or another shape as long as the proper angles are included. Tool bits are identified by the function they perform such as turning or facing. They can also be identified as roughing tools or finishing tools. Generally, a roughing tool has a radius ground onto the nose of the tool bit that is smaller than the radius for a finishing or general-purpose tool bit as shown in Figure 7-5.

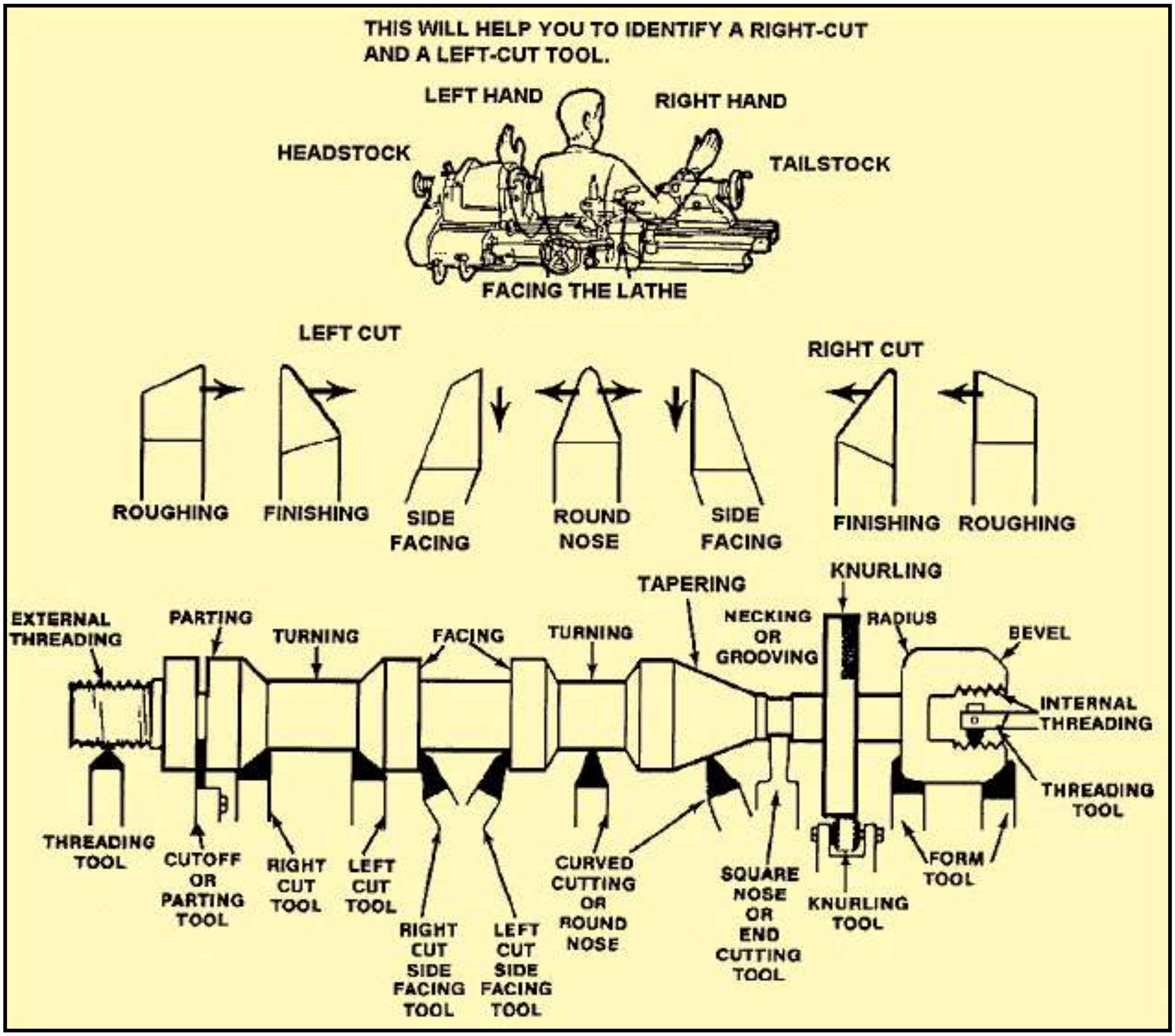


Figure 7-5. Shapes of tool bits.

The Right-Hand Turning Tool Bit: Is shaped to be fed from right to left. The cutting edge is on the left side of the tool bit and the face slopes down away from the cutting edge. The left side and end of the tool bit are ground with sufficient clearance to permit the cutting edge to bear upon the workpiece without the heel rubbing on the work. The right-hand turning tool bit is ideal for taking light roughing cuts as well as general all- around machining.

The Right-Hand Facing Tool Bit: Is intended for facing on right-hand side shoulders and the right end of a workpiece. The cutting edge is on the left-hand side of the bit. and the nose is ground very sharp for machining into a square corner. The direction of feed for this tool bit should be away from the center axis of the work, not going into the center axis. A left-hand facing tool bit is the opposite of the right-hand facing tool bit and is intend to machine and face the left sides of shoulders.

The Round-Nose Turning Tool Bit: Is very versatile and can be used to turn in either direction for roughing and finishing cuts. No side rake angle is ground into the top face when used to cut in either direction, but a small back rake angle may be needed for chip removal. The nose radius is usually ground in the shape of a half-circle with a diameter of about 1/32 inch.

The Parting Tool Bit: Is also known as the cutoff tool bit. This tool bit has the principal cutting edge at the squared end of the bit that is advanced at a right angle into the workpiece. Both sides should have sufficient clearance to prevent binding and should be ground slightly narrower at the back than at the cutting edge. Besides being used for parting operations, this tool bit can be used to machine square corners and grooves. (Figure 7-6).

Thread-Cutting Tool Bits: Are ground to cut the type and style of threads desired. Side and front clearances must be ground, plus the special point shape for the type of thread desired. Thread-cutting tool bits can be ground for standard 60° thread forms or for square, Acme, or special threads. Thread-cutting forms are discussed in greater detail later in this chapter. (Figure 7-7).

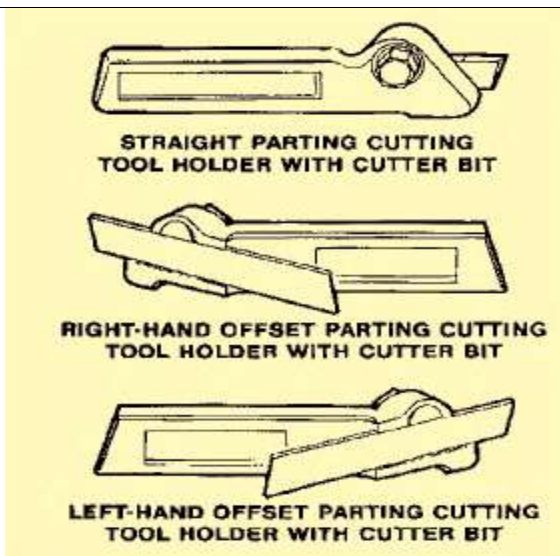


Figure 7-6. Parting tool bits.

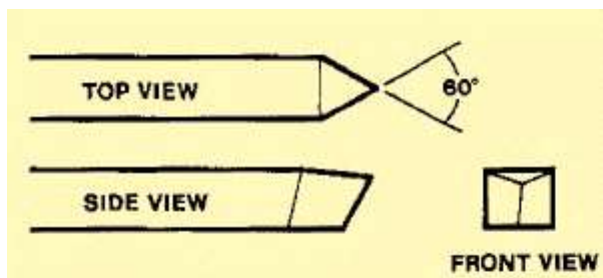


Figure 7-7. Thread cutting tool bit.

4) Special Lathe Cutting Tools:

Besides the common shaped tool bits, special lathe operations and heavy production work require special types of cutting tools. **Tungsten carbide, tantalum carbide, titanium carbide, ceramic, oxide, diamond-tipped tool bits, and cutting tool inserts** are commonly used in high-speed production work when heavy cuts are necessary and where exceptionally hard and tough materials are encountered, usually designed to be indexed or rotated. When the cutting edges get amorphous the inserts are discarded. Carbide and ceramic inserts can be square, triangular, round, or other shapes. Cutting tool inserts are not intended for reuse after sharpening. Standard shapes for tipped tool bits are similar to high-speed steel-cutting tool shapes. (Figure 7-8).

The Thread Cutter Tool: Is specially mounted in a thread "cutter holder" specially designed for production of high-speed thread cutting operations. The special design of the cutter allows for sharp and strong cutting edges which need only to be re-sharpened occasionally by grinding the face. The cutter mounts into a special tool holder that mounts to the lathe tool post. (Figure 7-9).

The Knurling Tool: Consists of two cylindrical cutters, called knurls, which rotate in a specially designed tool holder. The knurls contain teeth which are rolled against the surface of the workpiece to form depressed patterns on the workpiece. The common knurling tool accepts different pairs of knurls, each having a different pattern or pitch. The diamond pattern is most widely used and comes in three pitches: 14, 21, or 33. These pitches produce coarse, medium, and fine knurled patterns. (Figure 7-10).

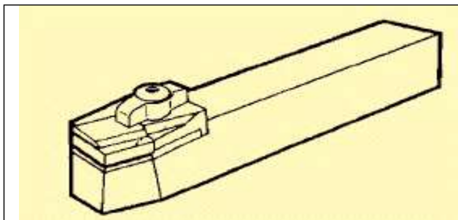


Figure 7-8. Tipped tool bit.

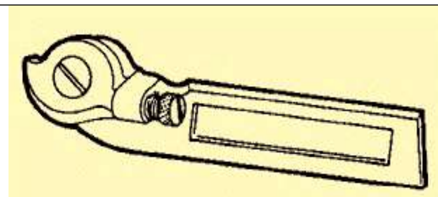


Figure 7-9. Thread cutting tool holder and cutter.

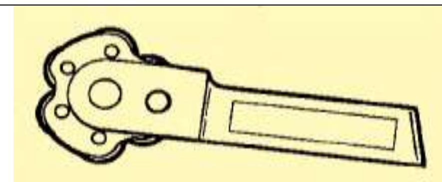


Figure 7-10. The common knurling tool.

Boring Tool Bits: Boring is the enlarging of a hole by removing material from internal surfaces with a single-point cutter bit, ground similar to left-hand turning tool bits and thread-cutting tool bits, but with more end clearance angle to prevent the heel of the tool bit from rubbing against the surface of the bored hole. The boring tool bit is usually clamped to a boring tool holder, but it can be a one-piece unit. The boring tool bit and tool holder clamp into the lathe tool post. (Figure 7-11).

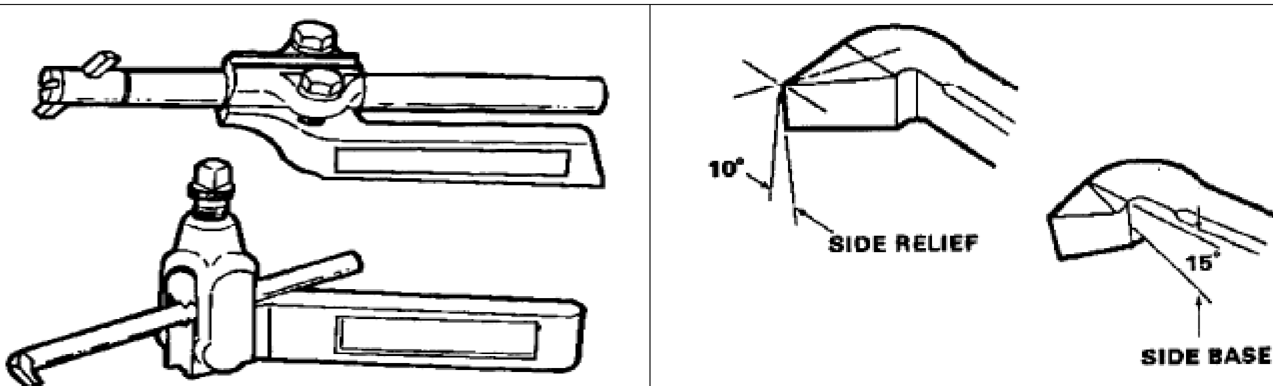


Figure 7-11. Boring tool bits and holders.

Grinding the Lathe Tool Bit Shapes: Carbide tool bits must be ground on a silicon carbide grinding wheel to remove the very hard metal, however, there are no set procedures, but only general guidelines to be followed. In order to effectively grind a tool bit, the grinding wheel must have a true and clean face and be of the appropriate material for the cutting tool to be ground. Do not attempt to use the bench or pedestal grinder without becoming fully educated as to its safety, operation, and capabilities.

Grinding High-Speed Steel Tool Bits: Are the **only tool bits** that can effectively be ground on the bench or pedestal grinder when equipped with the aluminum oxide grinding wheel which is standard for most field and maintenance shops. Before grinding, shaping, or sharpening a high-speed steel tool bit, inspect the entire grinder for a safe setup and adjust the tool rests and guards as needed for tool bit grinding. (Figure 7- 12).

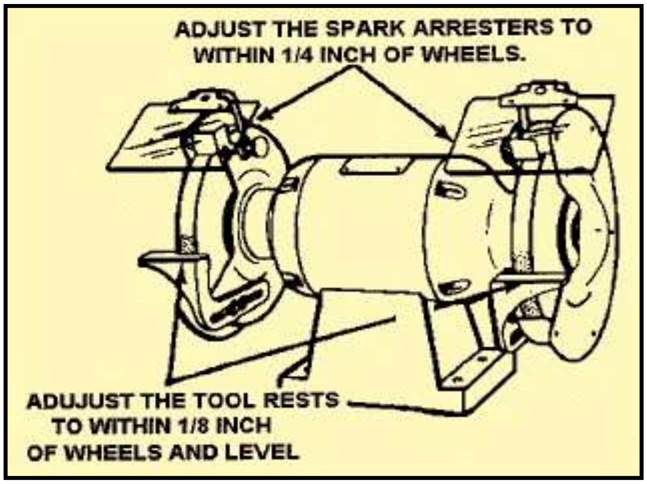


Figure 7-12. Grinder setup for lathe and tool bit grinding.

Grinding Tool Bits: Each grinder is usually equipped with a coarse-grained wheel for rough grinding and a fine-grained wheel for fine and finish grinding. Set in 1/8 inch or less from the wheel, and adjust the spark arrestor 1/4 inch or less. Dress the face of the grinding wheels to keep a smooth, flat grinding surface for the tool bit. When grinding the side and back rake angles, ensure the grinding wheel has a sharp corner for shaping the angle.

Dip the tool bit in water occasionally while grinding to keep the tool bit cool enough and frequently inspect the tool bit angles with a protractor or special grinding gage. Grind the tool bit to the recommended angles in the reference for tool bit geometry. The smoother the finish on the cutting tool, the smoother the finish on the work. As a safety note, never use the side of the grinding wheel to grind a tool bit, as this could weaken the bonding of the wheel and cause it to crack and explode as shown in Figure 7-13, below.

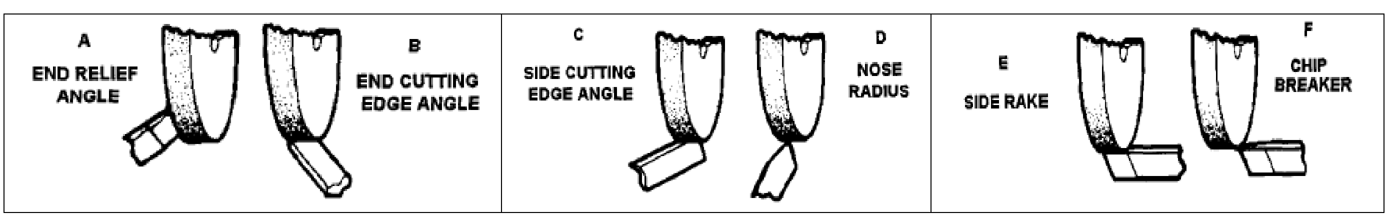


Figure 7-13. Grinding tool bits.

5) Tool Holders and Tool Posts:

The Tool Post: Consists of the post, screw, washer, collar, and rocker, and fits into the T-slot of a compound rest. The lathe tool holders are designed to securely and rigidly hold the tool bit at a fixed angle for properly machining a workpiece and come in various types for different uses. (Figure 7-14).

Standard Tool Holders: Are designed for high-speed steel cutting tools and commonly have a square slot made to fit a standard size tool bit shank. Tool bit shanks can be 1/4-inch, 5/16-inch, 3/8-inch, and greater, with all the various sizes being manufactured for all the different lathe manufacturer's tool holder models. These tool holders are designed to be used with the standard round tool post that usually is supplied with each engine lathe. (Figure 7-15).

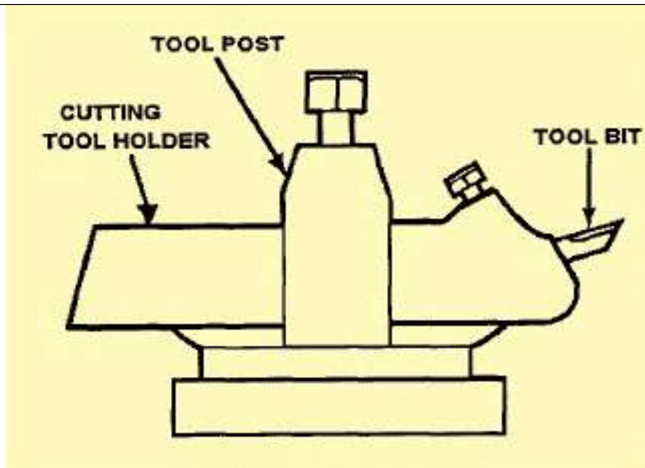


Figure 7-14. Tool holder with a tool bit mounted in a tool pose.

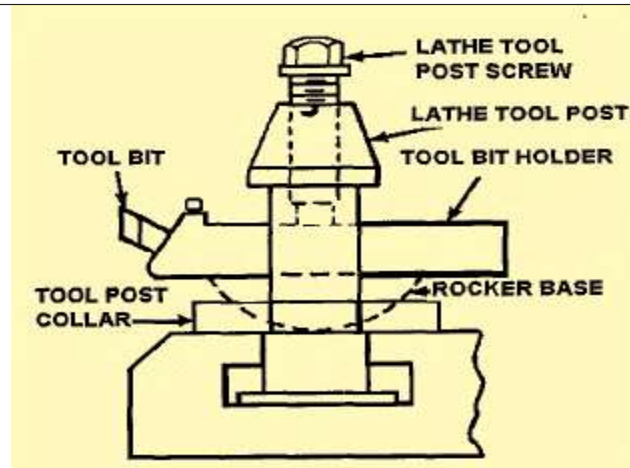


Figure 7-15. Standard round tool post.

OBS.: Some standard tool holders for steel tool bits are the straight tool holder, right and left offset tool holder, and the zero rake tool holder designed for special carbide tool bits. Other tool holders to fit the standard round tool post include straight, left, and right parting tool holders, knurling tool holders, boring bar tool holders, and specially formed thread cutting tool holders.

The Turret Tool Post: Is a swiveling block that can hold many different tool bits or tool holders. Each cutting tool can quickly be swiveled into cutting position and clamped into place using a quick clamping handle. The turret tool post is used mainly for high-speed production operations. (Figure 7-16).

The Heavy-Duty or Open-Sided Tool Post: Is used for holding a single carbide-tipped tool bit or tool holder. It is used mainly for very heavy cuts that require a rigid tool holder. (Figure 7-17).

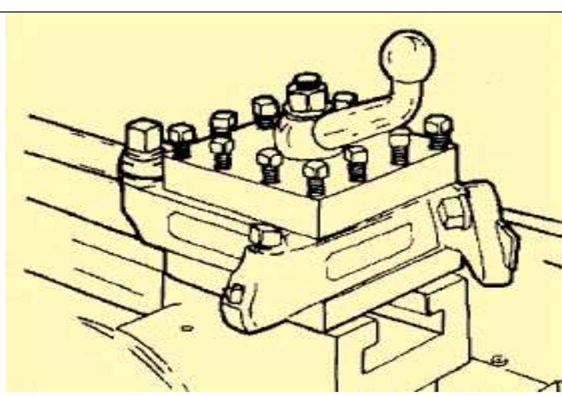


Figure 7-16. Turret tool post.

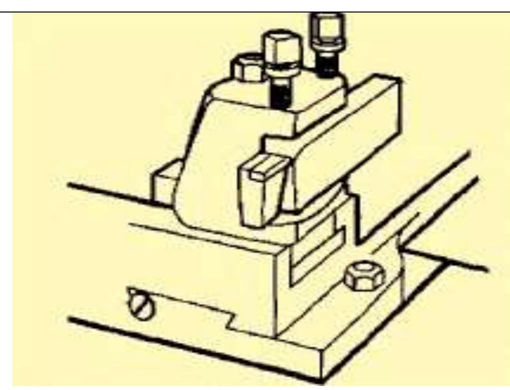


Figure 7-17. Heavy-duty or open-sided tool post.

The Quick-Change Tool System: Consists of a quick-change dovetail tool post with a complete set of matching dovetailed tool holders that can be quickly changed as different lathe operations become necessary. This system has a quick-release knob on the top of the tool post that allows tool changes in less than 5 seconds, which makes this system valuable for production machine shops. (Figure 7-18).

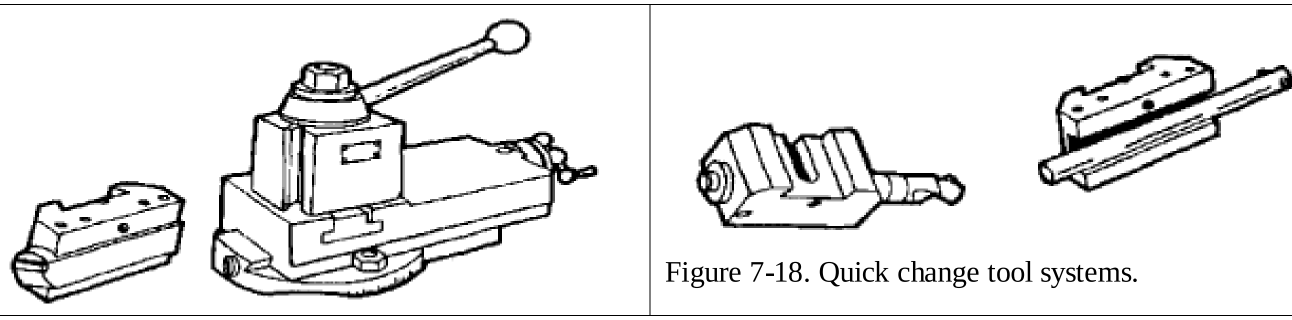


Figure 7-18. Quick change tool systems.

6) Lathe Holding Devices:

Chucks, Collets, Faceplates, Drive Plates, Mandrels and Lathe Centers: Are different devices used to hold and control the size and type of work to be machined. The particular operation that needs to be done will determine which work holding device is best for any particular job, considering how much accuracy is needed for a job, since some work holding devices are more accurate than others. The operational details for some of the more common work holding devices are:

The Independent Chuck: Generally have four jaws which are adjusted by means of adjusting screws, used to hold **square, round, octagonal, or irregularly shaped** workplaces in either a **concentric or eccentric** position due to the independent operation of each jaw. The chuck face is scribed with concentric circles which are used for rough alignment of the jaws when chucking round workplaces. The final adjustment is made by turning the workpiece slowly by hand and using a dial indicator to determine its concentricity. The jaws are then readjusted as necessary to align the workpiece within the desired tolerances.

The Universal Scroll Chuck: Generally have three jaws which move in unison as an adjusting pinion is rotated, used **to hold** and automatically center **round or hexagonal** workplaces. Having only three jaws, the chuck **cannot** be used effectively to hold **square, octagonal, or irregular shapes**. The advantage of the universal scroll chuck is its ease of operation in centering work for concentric turning. This chuck is not as accurate as the independent chuck, but when in good condition it will center work within **0.002 to 0.003 inches** of runout. (Figure 7-19).

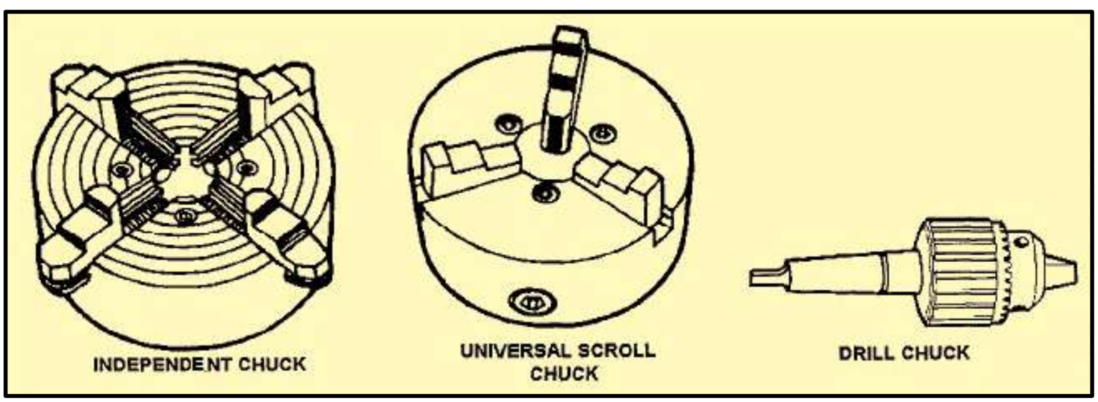
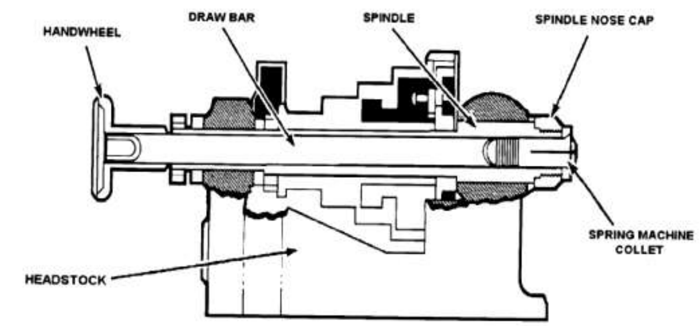


Figure 7-19. Lathe chucks.

The Drill Chuck: Is a small universal chuck which can be used in either the headstock spindle or the tailstock for holding straight-shank drills, reamers, taps, or small diameter workpieces. The drill chuck has three or four hardened steel jaws which are moved together or apart by adjusting a tapered sleeve within which they are contained. The drill chuck is capable of centering tools and small-diameter workplaces to within **0.002 or 0.003 inch** when firmly tightened. (Figure 7-19).

The Spring Collet Chuck: Is a thin metal bushing with an accurately machined bore and a tapered exterior, the most accurate means of holding small workplaces in the lathe. To grip the workpiece accurately, the collet must be no more than **0.005 inch larger or smaller** than the **diameter of the piece** to be chucked. For this reason, spring machine collets are available in increments of **1/64 inch**. For general purposes, the spring machine collets are limited in capacity to **1 1/8 inch** in diameter. (Figure 7-20).



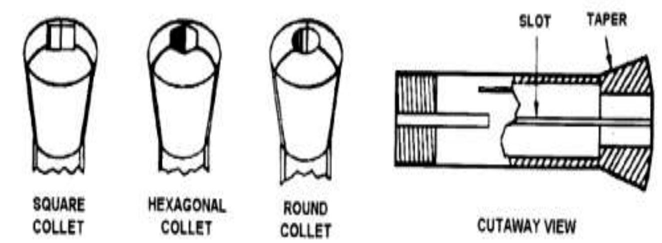


Figure 7-20. Spring machine collet chucks and installation method.

Collets: Are holding devices that form a collar around the machining tool or a workpiece to be held, exerting a strong clamping force when it is tightened, usually by means of a tapered outer collar. The collet sleeve is fitted to the right end of the headstock spindle. The drawbar passes through the headstock spindle and is threaded to the spring machine collet. Spring machine collets are available in different shapes to chuck square and hexagonal workplaces of small dimensions as well as round workplaces.

Jacob's Spindle-Nose Collet Chuck: Is a special chuck used for the Jacob's rubber flex collets. This chuck combines the functions of the standard collet chuck and drawbar into one single compact unit. The chuck housing has a handwheel on the outer diameter that turns to tighten or loosen the tapered spindle which holds the rubber flex collets. (Figure 7-21).

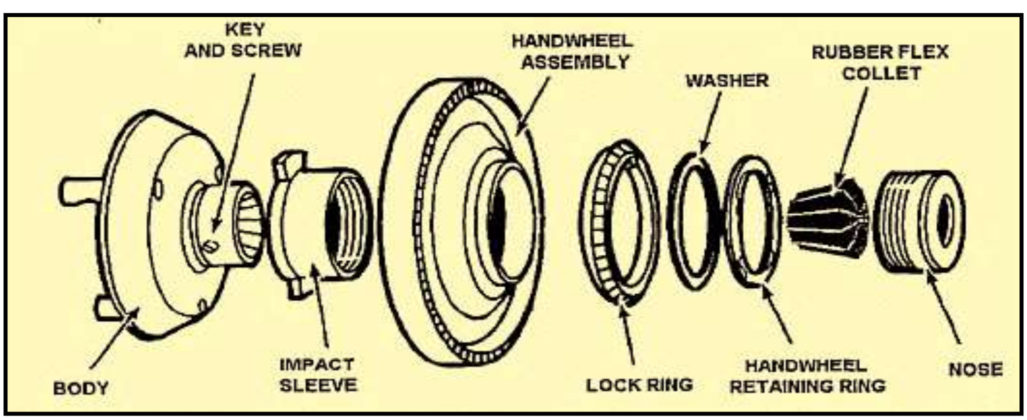


Figure 7-21. Jacob's spindle nose collet chuck and rubber flex collet.

Jacob´s Rubber Flex Collets: Are made of hardened steel jaws in solid rubber housing and have a range of **1/8 inch** per collet, designed for heavy duty turning and possess two to four times the grip of the conventional steel collet. The different sets of these collets are stored in steel boxes designed for holding the collets. Collets are normally stored in steel boxes designed for holding the collets. The gripping power and accuracy remain constant throughout the entire collet capacity.

The Step Chuck: Is a variation of the collet chuck, and it is intended for holding small round workplaces or discs for special machining jobs. Step chucks are blank when new, and then are machined in the lathe for an exact fit for the discs to be turned. The step chuck machine collet, which is split into three sections like the spring machine collet, is threaded to the drawbar of the collet attachment. (Figure 7-22).

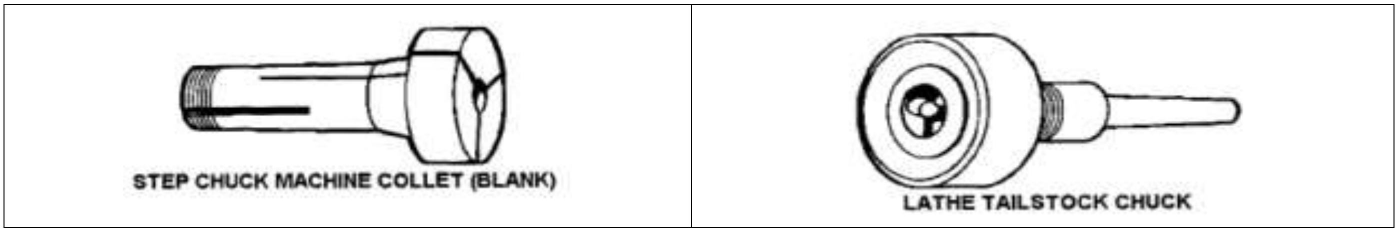


Figure 7-22. Step chuck machine collet and tailstock chuck.

The Lathe Tailstock Chuck: is a device designed to support the ends of workplaces in the tailstock when a lathe center cannot be used conveniently. The chuck has a taper arbor that fits into the lathe tailstock spindle. The three bronze self-centering jaws of the chuck will accurately close upon workplaces between 1/4 and 1 inch in diameter. The bronze jaws provide a good bearing surface for the workpiece. The jaws are adjusted to the diameter of the workpiece and then locked in place. (Figure 7-22).

The Lathe Faceplate: Is a flat, round plate that threads to the headstock spindle of the lathe. The faceplate is used for irregularly shaped workplaces that cannot be successfully held by chucks or mounted between centers, valuable for mounting workplaces, in which an eccentric hole or projection is to be machined. The workpiece is either attached to the faceplate using angle plates or brackets or bolted directly to the plate. Radial T-slots in the faceplate surface facilitate mounting workplaces. (Figure 7-23).

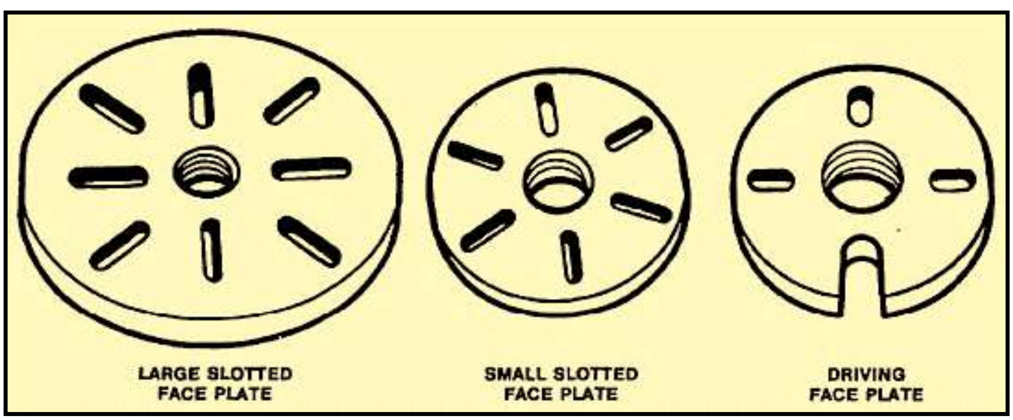


Figure 7-23. Faceplates.

Lathe Centers: Are referred to as **live centers or dead centers**. A **live center revolves** with the work and **does not need** to be **lubricated** and hardened. A **dead center does not revolve** with the work and **must be** heavily **lubricated** and hardened, when holding work. Live and dead centers commonly come in matched sets, with the hardened dead center marked with a groove near the conical end point.

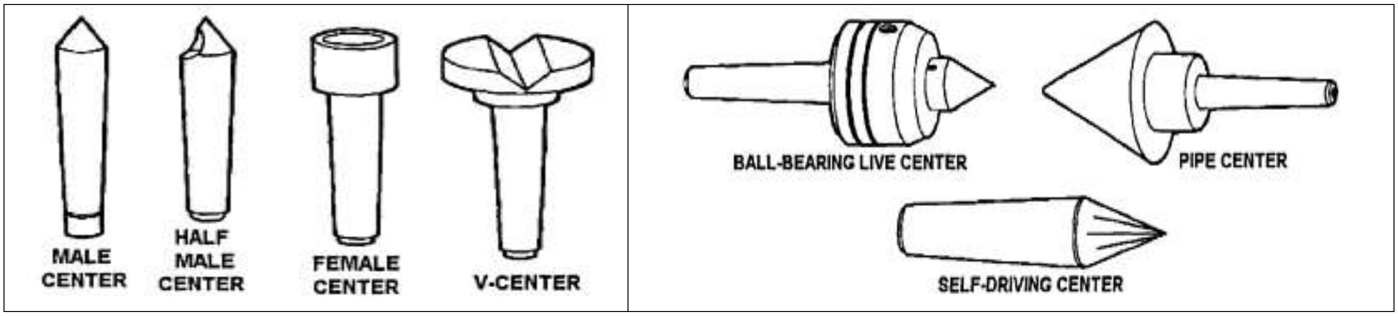


Figure 7-24. Lathe centers.

OBS.: Most lathe centers have a tapered point with a 60° included angle to fit workplace holes with the same angle. The workpiece is supported between two centers, one in the headstock spindle and one in the tailstock spindle. Centers for lathe work have standard tapered shanks that fit directly into the tailstock and into the headstock spindle using a center sleeve to convert the larger bore of the spindle to the smaller tapered size of the lathe center. (Figure 7-24).

The Ball Bearing Live Center: Is a special center mounted in a ball bearing housing that lets the center turn with the work and **eliminates the need** for a heavily lubricated dead center. Ball bearing types of centers can have interchangeable points which make this center a versatile tool in all lathe operations. Modern centers of this type can be very accurate. Descriptions for some common lathe centers follow.

The Male Center or Plain Center: Is used in pairs for most general lathe turning operations. When used in the **headstock spindle** where it revolves with the workpiece, it is commonly called a **live center**. When used in the **tailstock spindle**, where it remains stationary when the workpiece is turned, it is called a **dead center**, and must be lubricated very often to prevent overheating. The point is ground to a 60° cone angle.

The Half-Male Center: Is used as a **dead center in the tailstock** where facing is to be performed. The cutaway portion of the center faces provides the necessary clearance for the tool when facing the surface immediately around the drilled center in the workpiece. Is a male center that has a portion of the 60° cone cut away.

The V-Center: Is used to support round workpieces at right angles to the lathe axis for special operations such as drilling or reaming.

The Pipe Center: Is similar to the male center, but its cone is ground to a greater angle and is larger in size, commonly used for holding pipe and tubing in the lathe. The female center is conically bored at the tip and is used to support workplaces that are pointed on the end.

The Self-Driving Lathe Center: Is a center with serrated ground sides that can grip the work while turning between centers without having to use lathe dogs. A self-driving center is a center that has grips installed on the outer edge of the center diameter that can be forced into the work to hold and drive the work when turning between centers without using lathe dogs.

Lathe Dogs: Are cast metal devices used to provide a firm connection between the headstock spindle and the workpiece mounted between centers. This firm connection permits the workpiece to be driven at the same speed as the spindle under the strain of cutting. Three common lathe dogs are illustrated in Figure 7-25. Lathe dogs may have bent tails or straight tails.

- **Bent-Tail Dogs**: The tail fits into a slot of the driving faceplate.
- **Straight-Tail Dogs**: The tail bears against a stud projecting from the faceplate.

OBS.: The bent-tail lathe dog with headless setscrew is considered safer than the dog with the square head screw because the headless setscrew reduces the danger of the dog catching in the operator's clothing and causing an accident. The bent-tail clamp lathe dog is used primarily for rectangular workplaces.

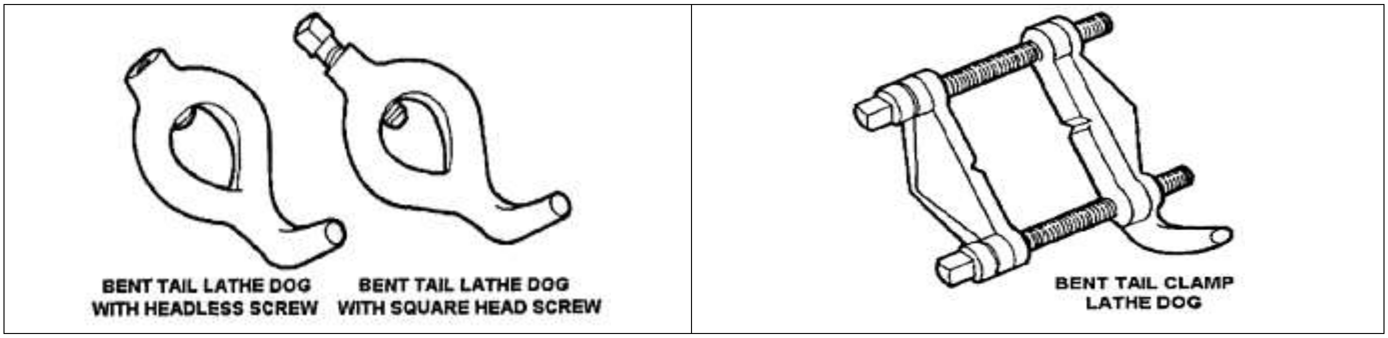


Figure 7-25. Lathe dogs.

7) Mandrels:

A mandrel is a tapered axle pressed into the bore of the workpiece to support it between centers. When turning work on a mandrel, feed toward the large end which should be nearest the headstock of the lathe. A workpiece which cannot be held between centers, because its axis has been drilled or bored and not suitable for holding it in a chuck or against a faceplate, is usually machined using a mandrel. To prevent damage to the work, the mandrel should always be oiled before being forced into the hole. (Figure 7-26).

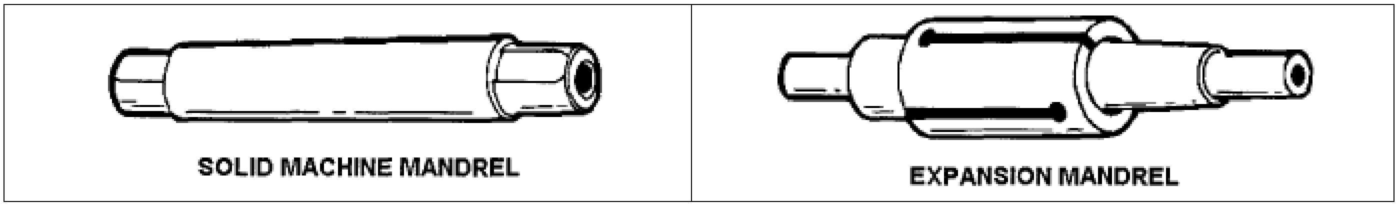


Figure 7-26. Mandrels.

Solid machine mandrels have a very slight taper, and the size is always stamped on the large end of the taper, but limited to workplaces with specific inside diameters. The ends of the mandrel are smaller than the body and have machined flats for the lathe dog to grip, generally made from hardened steel and ground to a slight taper from **0.0005 to 0.0006** inch per inch. **Expansion mandrels** accept workplaces having a greater range of sizes. The expansion mandrel is a chuck arranged so that the grips can be forced outward against the interior of the hole in the workpiece.

8) Lathe Attachments:

The variety of work that can be performed is greatly increased by the use of various lathe attachments. Some lathes come equipped with special attachments which must be ordered separately. Some common lathe attachments are the steady rest with cathead, the follower rest, the tool post grinding machine, the lathe micrometer stop, the lathe milling fixture, the lathe coolant attachment, the lathe indexing fixture, and the milling-grinding-drilling-slotting attachment (or Versa-Mil). The descriptions for lathe attachments are:

9) Rests:

Workpieces often need extra support, especially long, thin workplaces that tend to spring away from the tool bit. Rests are also used for internal threading operations where the workpiece projects a considerable distance from the chuck or faceplate. The common supports are the **steady rest, the cathead, and the follower rest** (Figure 7-27).

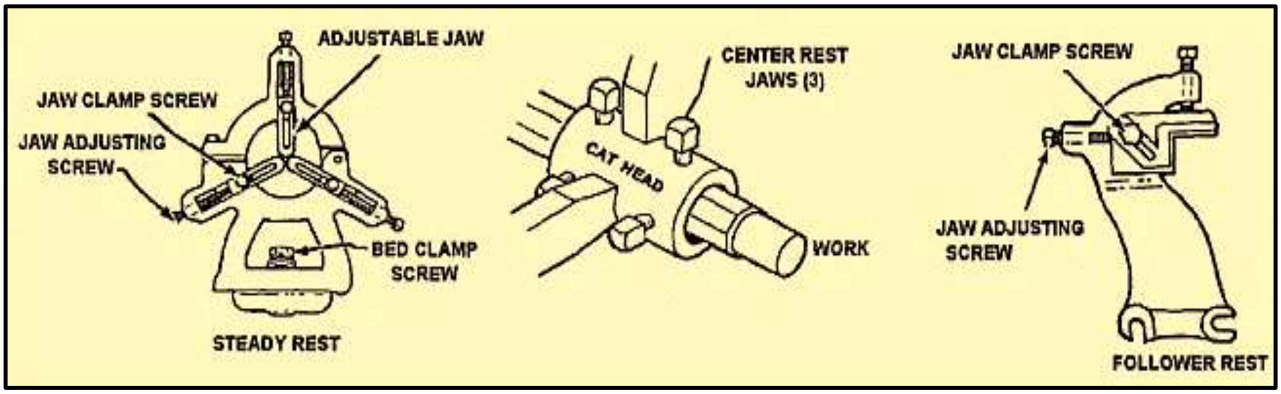


Figure 7-27. Lathe rests.

Steady Rest: The steady rest, also called a center rest, is used to support long workplaces for turning and boring operations. The steady rest is clamped to the lathe bed at the desired location and supports the workpiece within three adjustable jaws. The workpiece must be machined with a concentric bearing surface at the point where the steady rest is to be applied. The area of contact must be lubricated frequently. The top section of the steady rest swings away from the bottom section to permit removal of the workpiece without disturbing the jaw setting.

Cathead: When the work is too small to machine a bearing surface for the adjustable jaws to hold, then a cathead should be used. The cathead has a bearing 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 that it is concentric to the work axis. A dial indicator must be used to set up the cathead to be concentric and accurate.

Follower Rest: The follower rest has one or two jaws that bear against the workpiece. The rest is fastened to the lathe carriage so that it will follow the tool bit and bear upon the portion of the workpiece that has just been turned. The follower rest is generally used only for straight turning and for threading long, thin workplaces. These types of rests can be used without excessive lubricant or having to machine a polished bearing surface.

Micrometer Carriage Stop: The micrometer carriage stop (Figure 7-28) is used to accurately position the lathe carriage. The micrometer stop is designed- so the carriage can be moved into position against the retractable spindle of the stop and locked into place. A micrometer gage on the stop enables carriage movement of as little as **0.001 inches**. This tool is very useful when facing work to length, turning a shoulder, or cutting an accurate groove.

Tool Post Grinder: The tool post grinder (Figure 7-29) is a machine tool attachment specially designed for cylindrical grinding operations on the lathe. It consists primarily of a 1/4- or 1/ 3- horsepower electric motor and a wheel spindle connected by pulleys and a belt. The machine fastens to the compound rest of the lathe with a T-slot bolt which fits in the slot of the compound rest in the same manner as the lathe tool post. The tool post grinding machine mounts grinding abrasive wheels ranging from 1/ 4 inch to 3 or 4 inches in diameter for internal and external grinding operations.

Pulleys on the Wheel Spindle and Motor Shaft: Are interchangeable to provide proper cutting speeds with various wheel sizes. The larger grinding abrasive wheels used for external grinding are attached to the wheel spindle with an arbor. Small, mounted grinding abrasive wheels for internal grinding are fixed in a chuck which screws to the wheel spindle. The electric motor is connected to an electrical power source by a cable and plug.

Lathe Milling Fixture: This is a fixture designed to provide the ability for limited milling operations. Many repair and fabrication jobs cannot be satisfactorily completed on the standard engine lathe, but with the **milling attachments**, the **lathe can mill** key slots, keyways, flats, angles, hex heads, squares, splines, and holes.

10) Mounting the Lathe Work:

There is relatively little layout work to be done for most lathe work because of the lathe's ability to guide the cutting tool accurately to the workpiece. Some suggested methods are to use a bell-type center punch between centers and this cannot be accomplished on the lathe, (Figure 7-32), use hermaphrodite calipers to scribe intersecting arcs, use the centering head of the combination square, or use dividers (Figure 7-33).

Figure 7-32. Bell-type center punch.

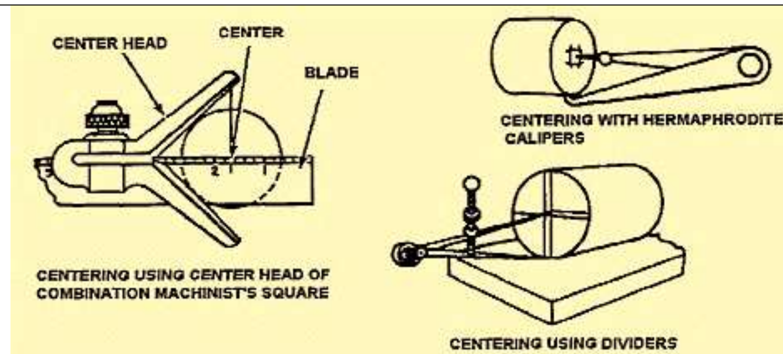


Figure 7-33. Laying out center holes.

Rough Centering: For irregularly shaped work, first measure the outside diameter of the workpiece, then opens the four jaws of the chuck until the workpiece slides in. Loosen the jaw opposite and tighten the jaw where the chalk marks are found. To center a workpiece having a smooth surface such as round stock, the best method is to use a dial test indicator. 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.

Mounting Workpieces in Chucks: Turn the spindle so that the key is facing up and lock the spindle in position. 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. (Figure 7-34).

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 and lathe dog must be used. 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. If expansion bushings are to be used with a mandrel, clean and care for the expansion bushings, and always feed the tool bit in the direction of the large end of the mandrel, to avoid pulling the work out of the mandrel. If facing on a mandrel, avoid cutting into the mandrel with the tool bit. (Figure 7-35).

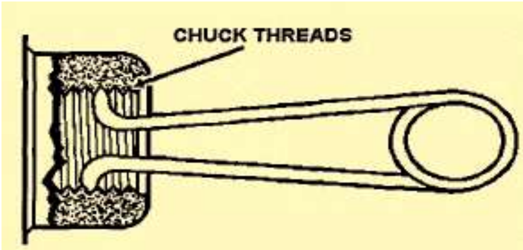


Figure 7-34. Spring thread cleaner.

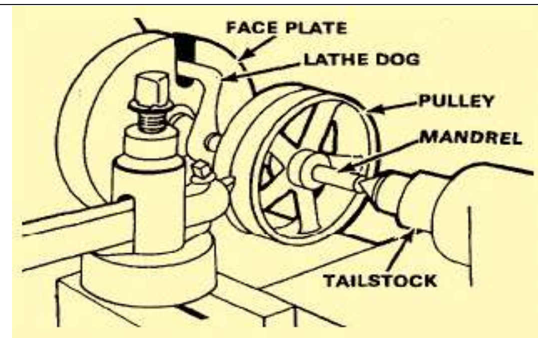


Figure 7-35. Pulley mounted on a mandrel.

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 7-36).

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 7-37).

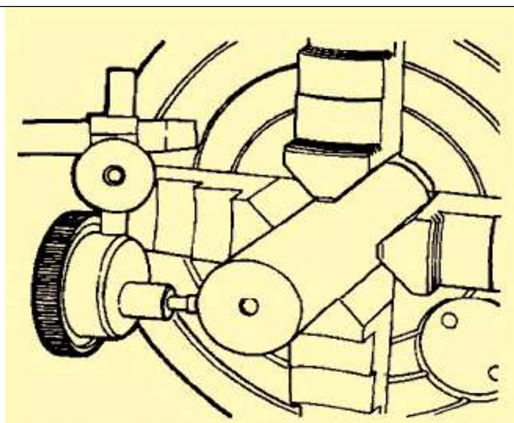


Figure 7-36. Mounting work on a 4-jaw independent chuck.

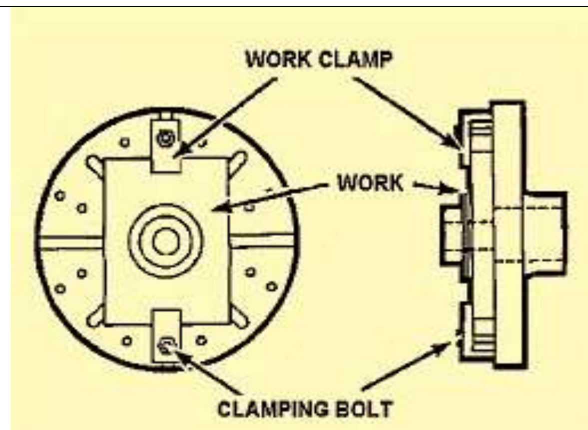


Figure 7-37. Work clamped on faceplate.

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.

Drilling Holes: Correct drillings and countersunk holes have a uniform 60° taper and has clearance at the bottom for the point of the lathe center. 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 7-38).

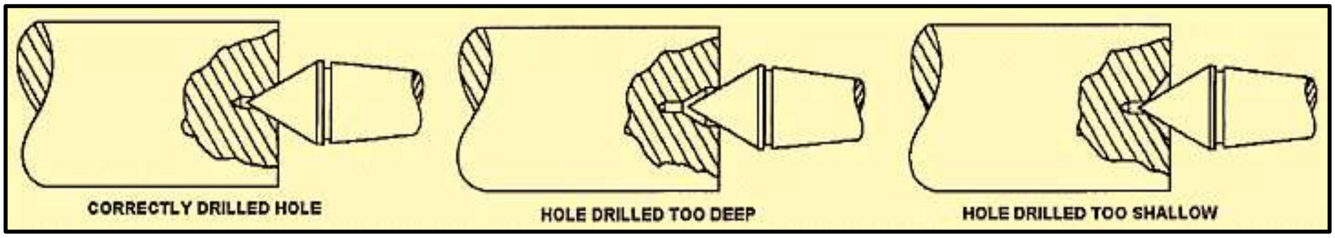


Figure 7-38. Correctly and incorrectly drilled center holes.

Work Between Centers: 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. 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 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. (Figure 7-39).

Headstock and Tailstock Centers: After mounting, 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).

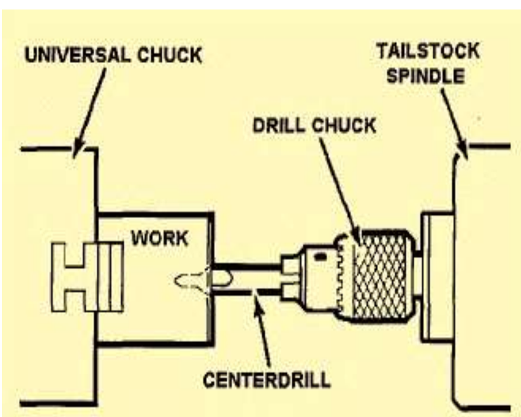


Figure 7-39. Center drilling.

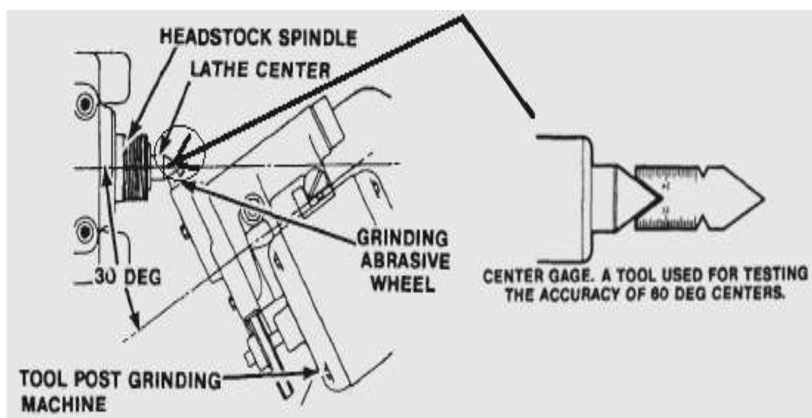
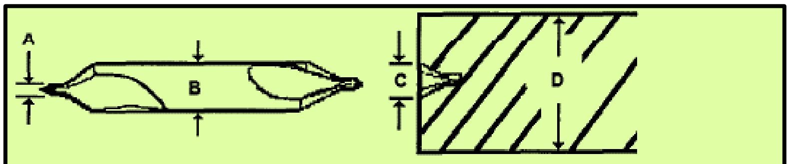


Figure 7-40. Checking a truing a 60-degree lathe center.

Turning a Soft Center True: 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. 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. (Figure 7-42).



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 7-41. Common sizes for countersink and center drill.

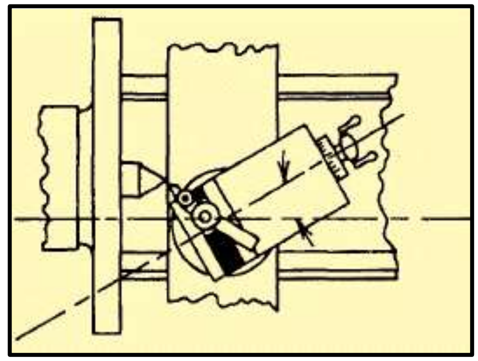


Figure 7-42. Turning of soft center true.

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 and this alignment may be checked by moving the tailstock up close to the headstock (Figure 7-43).

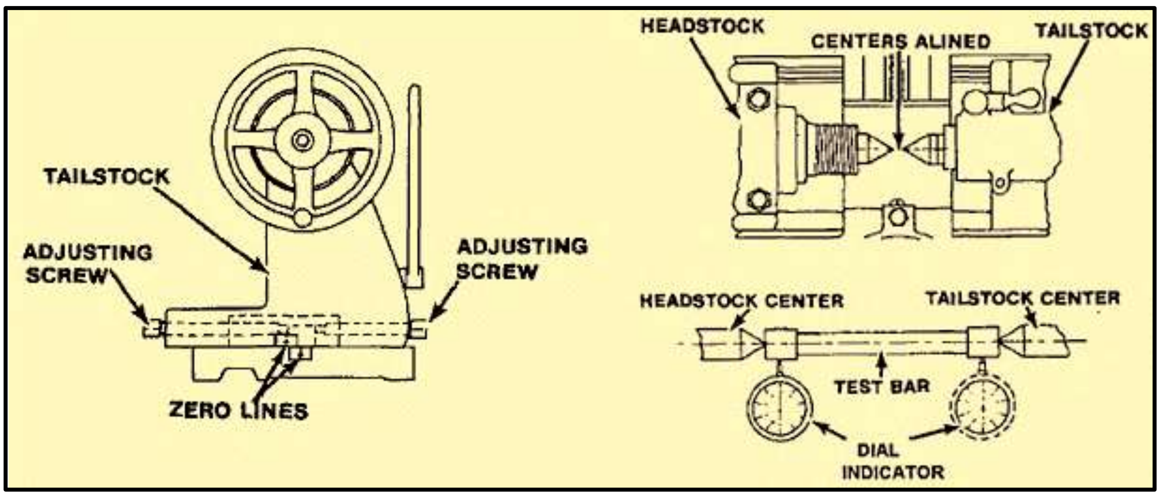


Figure 7-43. Checking the alignment of centers.

Headstock Spindle and Faceplate: Screw the sure that the external threads of the clean before screwing on the driving faceplate securely onto the spindle. 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.

Method of Checking the Alignment of Centers: The most accurate 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, drive plate and lathe dog must be used (Figure 7-44).

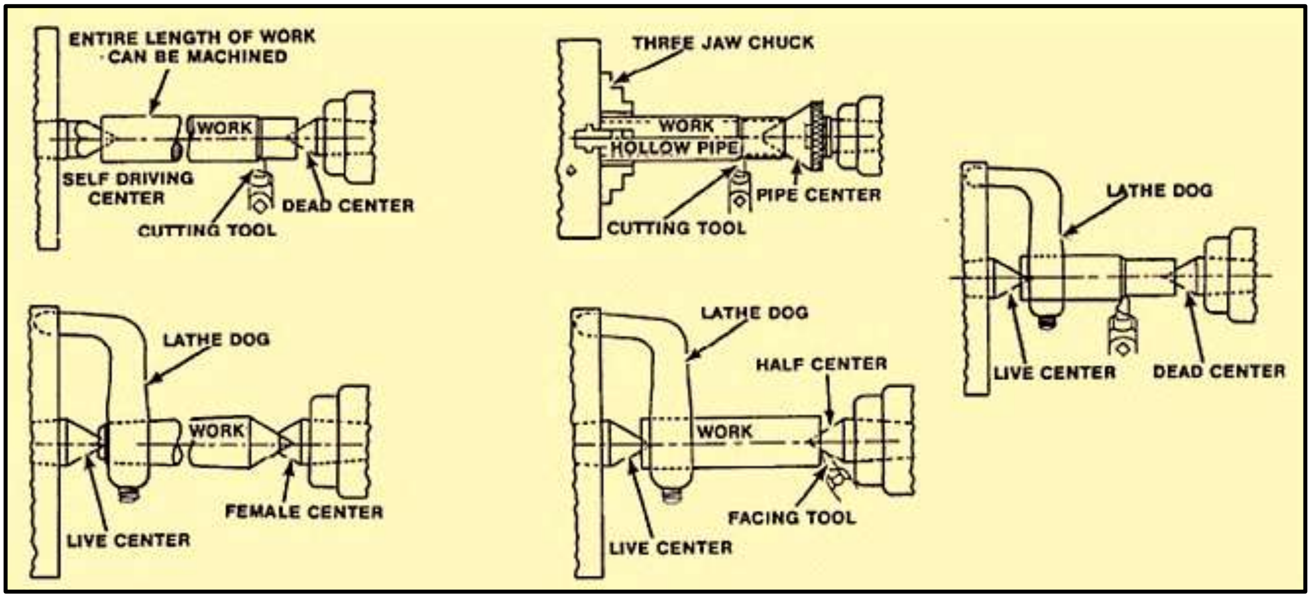


Figure 7-44. Holding work between centers.

Micrometer Collars: The compound rest may be equipped with a micrometer collar to measure in inches or in millimeters, or they can be equipped with a dual readout collar that has both measurements. 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). Graduated micrometer collars can be used to accurately measure this tool bit movement to and away from the lathe center axis. (Figure 7-45).

Facing Work in a Chuck: Facing is machining the ends and shoulders of a piece of stock smooth, usually performed with the work held in a chuck or collet. Allow the workpiece 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. This method also eliminates the problem of feeding the tool bit into the solid center portion of the workpiece to get a cut started. 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).

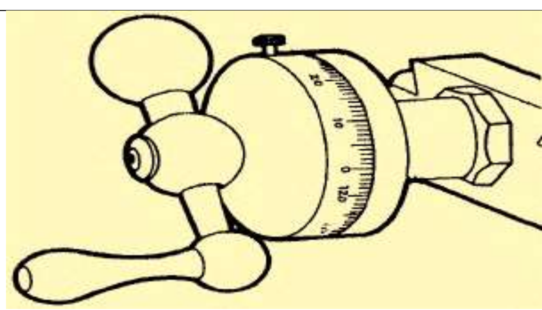


Figure 7-45. Graduated micrometer collar.

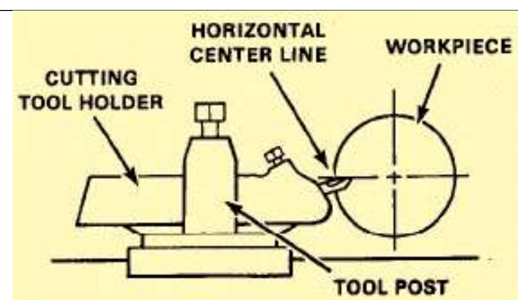


Figure 7-46. Positioning tool bit for facing.

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).

Precision Facing: Special methods must be used to face materials to a precise length. 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. 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. (Figure 7-48).

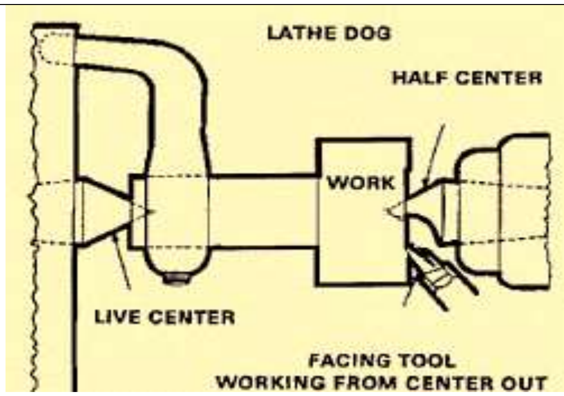


Figure 7-47. Facing using a side finishing tool and a half-male center.

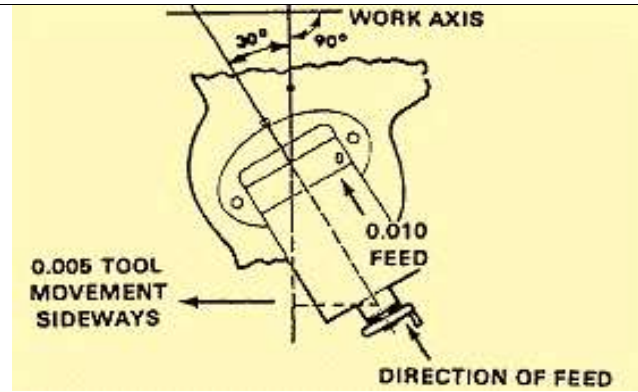
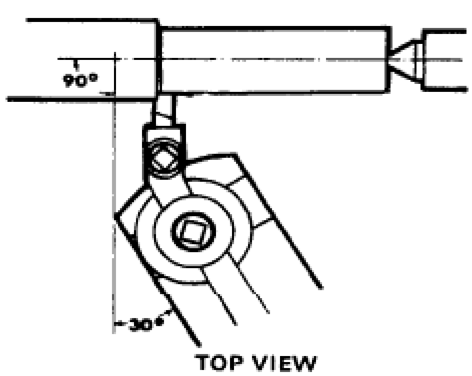


Figure 7-48. Facing using a graduated micrometer collar to measure bit movement.

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.

Setting Tool Bit for Straight Turning: 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. (Figure 7-49).



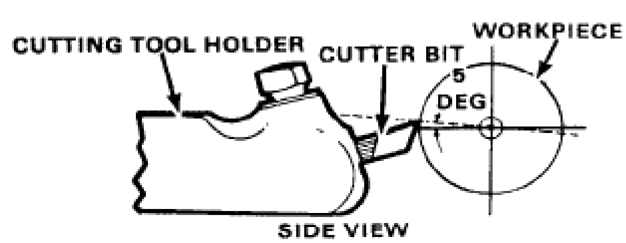


Figure 7-49. Setup for straight turning.

Turning Work Between Centers: Is one of the most accurate method available. 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.

Tool Bit Clearance: Check it 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.

11) Shoulders, Corners, 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).

Straight and Shoulder Turning: Straightening the diameter down to the desired size is 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).

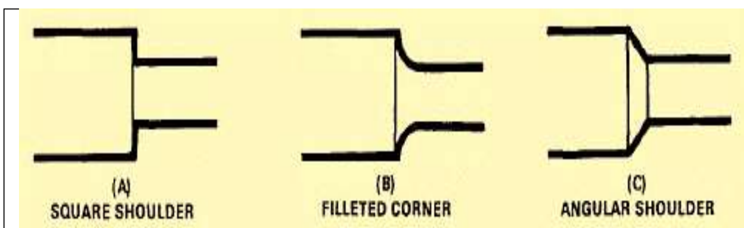


Figure 7-50. Common shoulders.

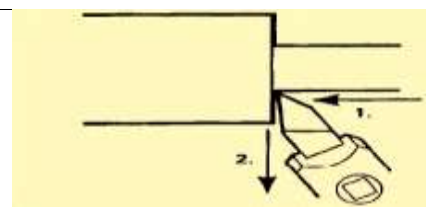
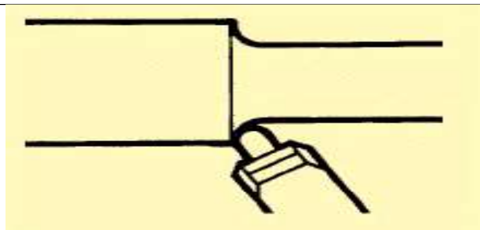
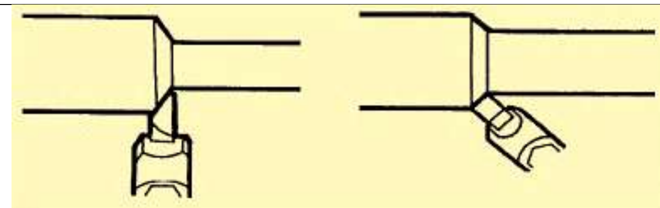


Figure 7-51. Straight and shoulder turning in one pass.

Filleted Shoulders: Filleted shoulders or corners 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. 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. (Figure 7-52).

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 7-52. Cutting a filleted corner.	 Figure 7-53. Two different tools for cutting angular shoulders.

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. 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. (Figure 7-54).

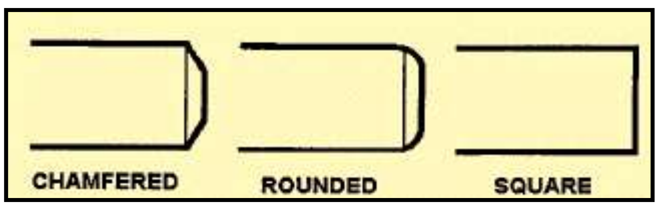


Figure 7-54. Corners.

Undercuts: Undercuts are the reductions in diameter machined onto the center portion of workplaces 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. (Figure 7-55).

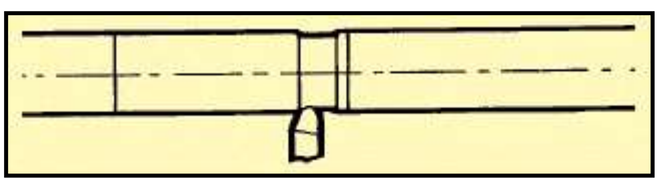


Figure 7-55. Machining an undercut.

Grooves: Grooving (or necking) is the process of turning a groove or furrow on a cylinder, shaft, or workpiece. The V-shaped groove is used extensively on step pulleys made to fit a V-type belt. The types of grooves most commonly used are square, round, and V-shaped. 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, without side or back rake angles. The grooving tool is a type of forming tool. (Figure 7- 56).

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. 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. (Figure 7-57).

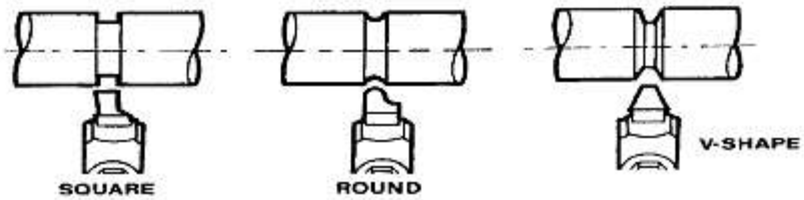


Figure 7-56. Common grooves.

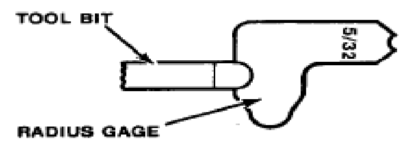


Figure 7-57. Checking tool bit with a radius gage.

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. When a micrometer and two wires are used, the micrometer reading is equal to the measured diameter of the groove plus two wire diameters. (Figure 7-58). To calculate measurement over the wires, use the following formula:

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

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. Parting tools normally have a 5° side rake and no back rake angles. 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. Parting tools can be of the forged type, inserted blade type, or ground from a standard tool blank. (Figure 7-59).

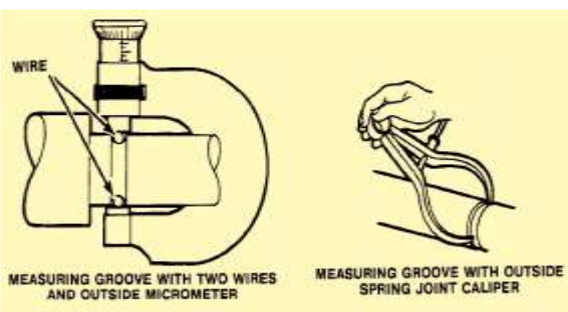


Figure 7-58. Checking the depth of a groove.

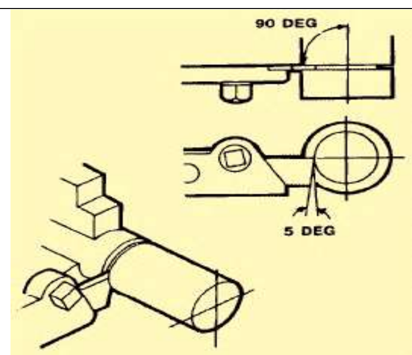


Figure 7-59. Parting.

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. 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 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. (Figure 7-60).

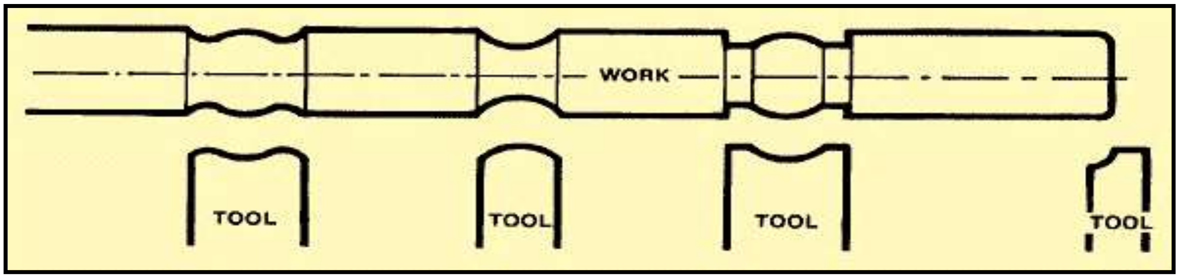


Figure 7-60. Forming tools.

Forming a Radius Using the Compound Rest: To use the compound rest and tool to pivot and cut 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).

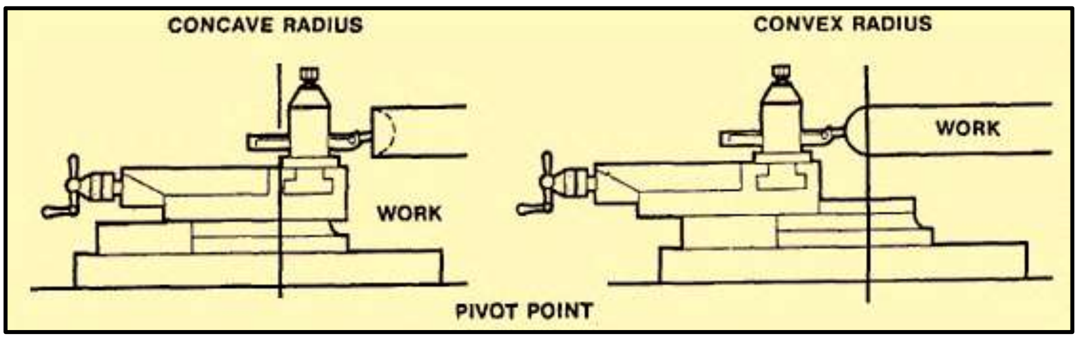


Figure 7-61. Pivots of the compound radius.

12) Tapered Pieces:

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.

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. 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. When the lathe centers are aligned and the workpiece is machined between these centers, the diameter will remain constant. (Figure 7-63).

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. The compound rest base is graduated in degrees and can be set at the required angle for taper turning or boring. 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 7-62).

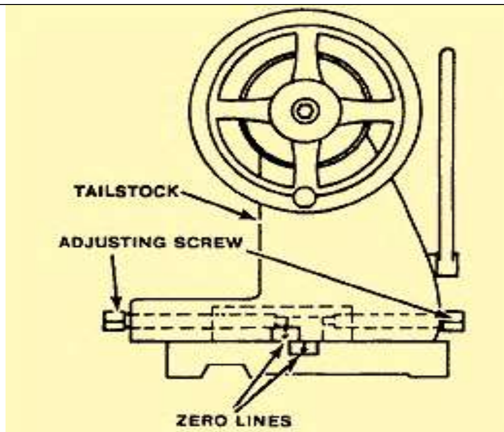


Figure 7-63. Tailstock offset for taper turning.

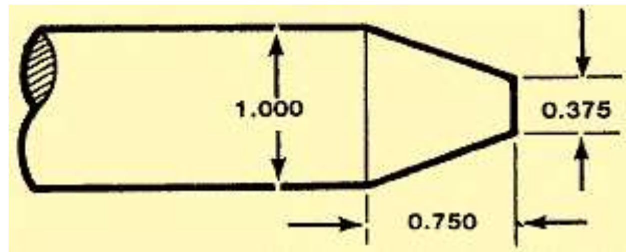


Figure 7-62. Taper problem.

Example: The compound rest setting for the workpiece shown in Figure 7-62, would be calculated with these two formulas:

$$TPI = \frac{D - d}{L}$$

Where:
TPI = taper per inch
D = large diameter
d = small diameter
L = length of taper
Angle = tan (TPI/2)

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

$$TPI = \frac{D - d}{L} = \frac{1.000 - 0.375}{0.750} = \frac{0.625}{0.750} = \mathbf{0.833}$$

angle = tan (TPI/2) = tan (0.833/2) = **tan (0.41650)**

Using a handheld calculator, you will see that, **tan (0.41650) = 22º 37'**. Then, to machine the taper shown in Figure 7-62, the compound rest will be set at 22° 37'.

Tailstock Offset: 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).

Tailstock Offset Factors: 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. Center holes are likely to wear out of their true positions if the lathe centers are offset too far, causing poor results, and the taper turning is determined by the proper distance the tailstock should be moved over to obtain a given taper (Figure 7-65).

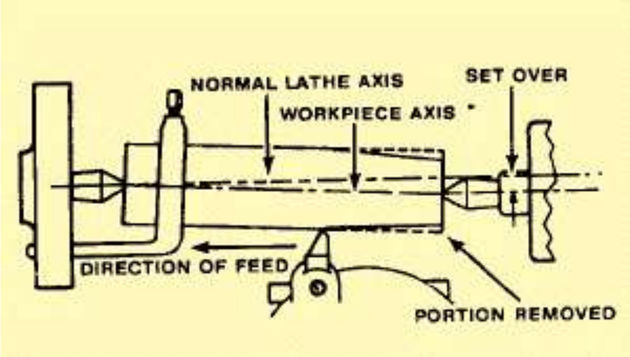


Figure 7-64. Taper turning with tailstock set over.

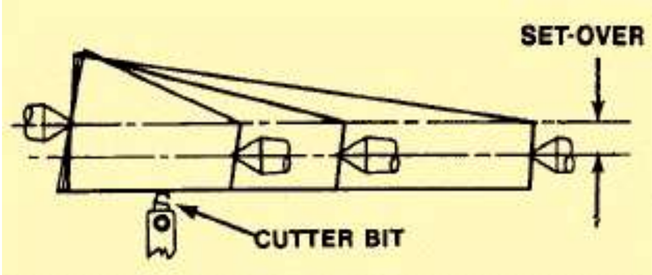


Figure 7-65. Effect of fixed amount of set over with different lengths of workpieces.

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

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

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.

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:

$$\text{OFFSET} = \frac{1/2 \text{ X } 42}{24} = \frac{21}{24} = \mathbf{0.875\ inch}$$

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:

OFFSET = TPI X L

Where:

OFFSET = TPI (taper per inch);
L = length of taper in inches;

Example: The amount of offset required to machine a bar 42 inches long with a taper of 0.0416 TPI is calculated as follows:
OFFSET = TPI X L
OFFSET = 0.0416 x 42

OFFSET = 1.7472 or rounded up 1.75
OFFSET = **0.875 inch**

Therefore, the tailstock should be offset 0.875 inch to machine the required taper. If the workpiece has a short taper in any part of its length and the TPI or TPF is not given. use the following formula:

OFFSET = L x (D - d) / 2 x 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

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. Calculate the offset.

OFFSET = L x (D - d) / 2 x L1
OFFSET = 36 x (1.750 - 1.5) / 2 x 18
OFFSET = 36 x 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 are expressed as a ratio of 1.0 mm per unit of length, which is, given as a ratio of 1:20.

Taper Attachment: Some 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. The **taper attachment** has many features of special value, among which are the taper attachment, as shown below. (Figure 7-69).

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.

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, thus, a taper hole can be bored. (Figure 7.70).

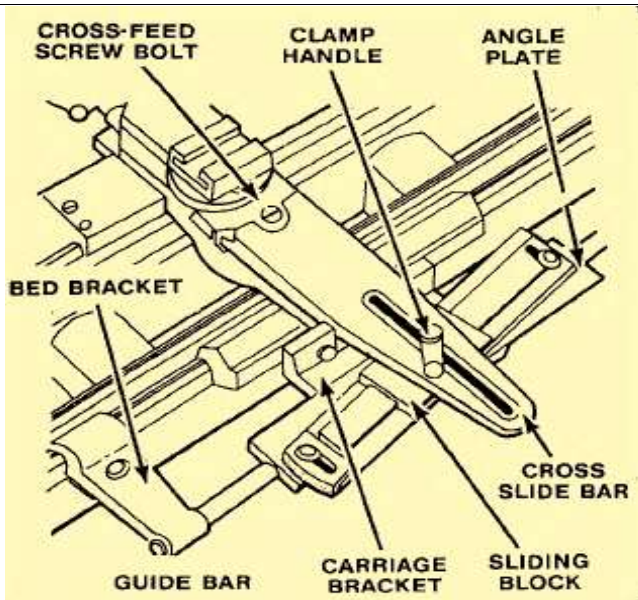


Figure 7-69. Taper attachment.

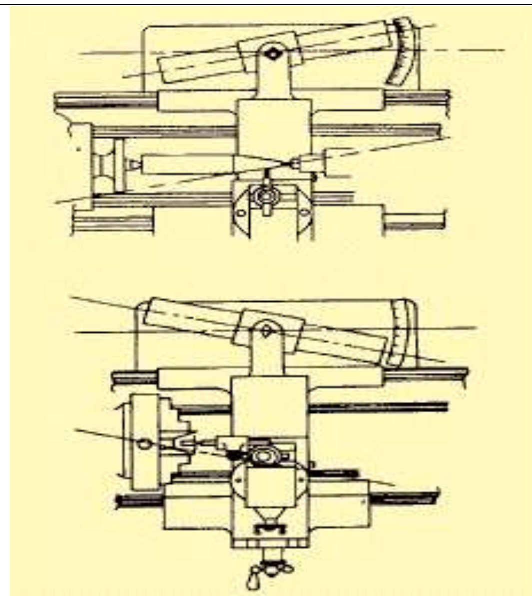


Figure 7-70. Taper turning and boring.

Calculate the taper per foot by using the formula:

TPF = D - d x 12

Where:

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:

Guide Bar Set Over (in millimeters) =

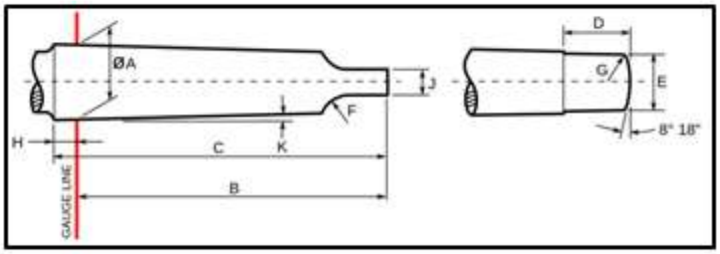
D = large diameter of taper (mm)
d = small diameter of taper (mm)
I = length of taper (mm)
L = length of guide bar (mm)

13) Standard Tapers:

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

Morse Tapers Dimensions: Are given in Table 7-4. 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.

Table 7.4 – Morse Tapers.



Morse Taper Number	Taper	A	B (max)	C (max)	D (max)	E (max)	F	G	H	J	K
0	1:19.212	9.045	56.5	59.5	10.5	6	4	1	3	3.9	1° 29' 26"
1	1:20.047	12.065	62	65.5	13.5	8.7	5	1.2	3.5	5.2	1° 25' 43"
2	1:20.020	17.780	75	80	16	13.5	6	1.6	5	6.3	1° 25' 50"
3	1:19.922	23.825	94	99	20	18.5	7	2	5	7.9	1° 26' 16"
4	1:19.254	31.267	117.5	124	24	24.5	8	2.5	6.5	11.9	1° 29' 15"
5	1:19.002	44.399	149.5	156	29	35.7	10	3	6.5	15.9	1° 30' 26"
6	1:19.180	63.348	210	218	40	51	13	4	8	19	1° 29' 36"
7	1:19.231	83.058	285.75	294.1	34.9	-	-	19.05	-	19	1° 29' 22"

The American Standard Machine Tapers: Are composed of a self-holding series and a steep taper series. 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 3/4-inch-per foot machine taper series. Brown and Sharpe tapers are used for taper shanks on tools such as end mills and reamers. The taper is approximately 1/2 inch per foot for all sizes except for taper No 10, where the taper is 0.5161 inch per foot.

Table 7-5: Brown and Sharpe Tapers:

Size	Lg. Dia.	Sm. Dia.	Length	Taper (in/ft)	Size	Lg. Dia.	Sm. Dia.	Length	Taper (in/ft)
1	0.2392	0.2000	0.94	0.5020	10	1.2597	1.0447	5.00	0.5161
2	0.2997	0.2500	1.19	0.5020	11	1.4978	1.2500	5.94	0.5010
3	0.3753	0.3125	1.50	0.5020	12	1.7968	1.5001	7.13	0.4997
4	0.4207	0.3500	1.69	0.5024	13	2.0731	1.7501	7.75	0.5002
5	0.5388	0.4500	2.13	0.5016	14	2.3438	2.0000	8.25	0.5000
6	0.5996	0.5000	2.38	0.5033	15	2.6146	2.2500	8.75	0.5000
7	0.7201	0.6000	2.88	0.5010	16	2.8854	2.5000	9.25	0.5000
8	0.8987	0.7500	3.56	0.5010	17	3.1563	2.7500	9.75	0.5000
9	1.0775	0.9001	4.25	0.5009	18	3.4271	3.0000	10.25	0.5000

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. See Table 7.5, below.

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:

Diameter at small end= No. of timer, 8
Diameter at small end= No. of taper, 10
Length of taper= No. of taper, 2

Table 7-6: Jarno Tapers:

Taper	Large end	Small end	Length	Taper/ ft	Taper/ in	Angle from center/°	Taper	Large end	Small end	Length	Taper/ ft	Taper/ in	Angle from center/°
#2	0.2500	0.2000	1.00	.6000	.0500	1.4321	#11	1.3750	1.1000	5.50	.6000	.0500	1.4321
#3	0.3750	0.3000	1.50	.6000	.0500	1.4321	#12	1.5000	1.2000	6.00	.6000	.0500	1.4321
#4	0.5000	0.4000	2.00	.6000	.0500	1.4321	#13	1.6250	1.3000	6.50	.6000	.0500	1.4321
#5	0.6250	0.5000	2.50	.6000	.0500	1.4321	#14	1.7500	1.4000	7.00	.6000	.0500	1.4321
#6	0.7500	0.6000	3.00	.6000	.0500	1.4321	#15	1.8750	1.5000	7.50	.6000	.0500	1.4321
#7	0.8750	0.7000	3.50	.6000	.0500	1.4321	#16	2.0000	1.6000	8.00	.6000	.0500	1.4321
#8	1.0000	0.8000	4.00	.6000	.0500	1.4321	#17	2.1250	1.7000	8.50	.6000	.0500	1.4321
#9	1.1250	0.9000	4.50	.6000	.0500	1.4321	#18	2.2500	1.8000	9.00	.6000	.0500	1.4321
#10	1.2500	1.0000	5.00	.6000	.0500	1.4321	#19	2.3750	1.9000	9.50	.6000	.0500	1.4321
							#20	2.5000	2.0000	10.00	.6000	.0500	1.4321

14) Lathe Calculations:

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:

$$\frac{12 \text{ X SFM}}{3.1416 \text{ X D}} = \text{RPM}$$

This can be closely approximated as:

$$\frac{4 \text{ X SFM}}{\text{D}} = \text{RPM}$$

Where:

SFM: The rated surface feet per minute, also expressed as peripheral or cutting speed.
RPM: The spindle speed in revolutions per minute.
D: The diameter of the work in inches.

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.

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

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

Table 7-7 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 7-7 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.

Table 7-7: 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

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. A light feed must be used on slender and small workplaces to avoid damage.

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.

15) Screw Threads:

Screw Thread Terminology: Before attempting to cut threads on the lathe a machine operator must have a **thorough knowledge** of the principles, terminology and uses of threads.

- ✓ 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.
- ✓ Pitch Diameter is the diameter of an imaginary cylinder formed where the width of the groove is equal to one-half of the pitch.
- ✓ 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.
- ✓ 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.
- ✓ 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.

Threads per Inch: Is the number of threads per inch that may be counted by placing a rule against the threaded parts and counting the number of pitches in **1.0 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.

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 (ISO) 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 ISO Metric Thread Standard: Is a simple thread system that has threaded sizes ranging in diameter from 1.6 mm to 100 mm. 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 5.0 mm and a pitch of 0.8 mm would be given as **M 5 x 0.8**. This ISO Metric thread has a 60° included angle and a crest that is 1.25 times the pitch. 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.

The 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. The **British Standard Whitworth** 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.

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.

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. 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.

16) Threads Fit and Classifications:

The Unified and American (National) Thread: Forms designate classifications, 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, being

1A and 1B for a loose fit and quick assembly. Classes 2A and 2B prevent galling and seizure in assembly, using sufficient clearances for plating, recommended for standard practice in making commercial screws, bolts and nuts. Classes 3A and 3B have no allowance. 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 unusual requirements more exacting than those for which Class 3 is intended. It is a selective fit if initial assembly by hand is required.

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. 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. Cutting V-threads with a 60 degrees thread angle is the most common thread cutting operation done on a lathe.

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. The two symbols **6g and 5g6g** are used to designate the fit for an external thread, **6g** being used for general purpose threads and **5g6g** used to designate a close fit. 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 symbol 6H is used to designate the fit for an internal thread.

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. 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.

Thread Cutting Tool Bits: The tool bit should be ground to the helix angle. The clearance angles for the sides should be within the helix angle. 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.

17) Thread Cutting Operations:

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. Engineering and **Machinist's Handbooks** have **special tables** listing the recommended major and minor diameters for all thread forms.

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. To machine this metric thread on an inch designed lathe, convert the outside diameter in millimeters to a decimal fraction of an inch, and machine the major diameter pitch in millimeters, to threads per inch by dividing the linear pitch of **2.5 by 25.4** to get the threads per inch.

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.

Thread Micrometers: Specify the pitch diameters to measure threads. 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 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. 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. (See Table 7-8, below).

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. For example, **8 threads per inch** to cut; we would calculate **0.57735 / 8 = 0.072.** The **diameter of wire** to use for measuring an 8-pitch thread is **0.072**.

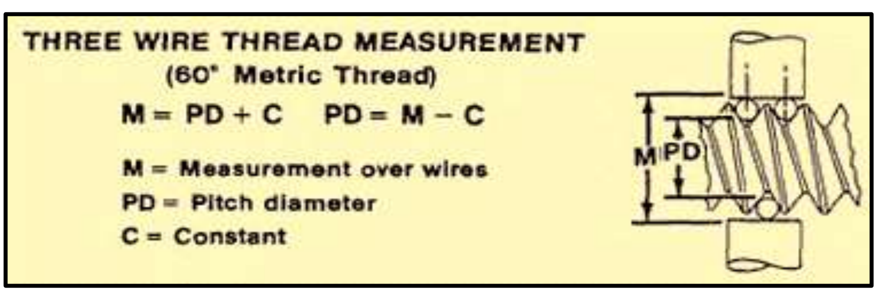


Table 7-8: Three-wire measurement for metric threads.

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

Note: The Bureau of Standards has specified an accuracy of **0.0002 inch**.

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, where: W = 0.04811 inch; D = 0.500 inch; n= 12. Proceed to solve as follows:

$$m = (0.500 + 0.14433) - \frac{1.5155}{12}$$

m = (0.500 + 0.14433) – 0.1263
m = **0.51803 inch** (micrometer measurement)

Cutting Internal Threads: Are cut into nuts and castings in the same general manner as external threads. 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.

Cutting External ACME Threads: The first step is to grind a threading tool to conform to the **29°** included angle of the thread. This angle can be checked by placing the tool in the slot at the right end of the Acme thread gage. 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. 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. 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

The single wire method can be used to measure the accuracy of the ACME thread. 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:
Wire size = 0.4872 x 1/6= **0.081 inch**

Cutting Square Threads: 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. 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. 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 = 1/2 P

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.

18) Tapping and Hand-Die Threading:

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. Always use a lubricant or coolant for this operation. (Figure 7-76).

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.

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. (Figure 7-71).

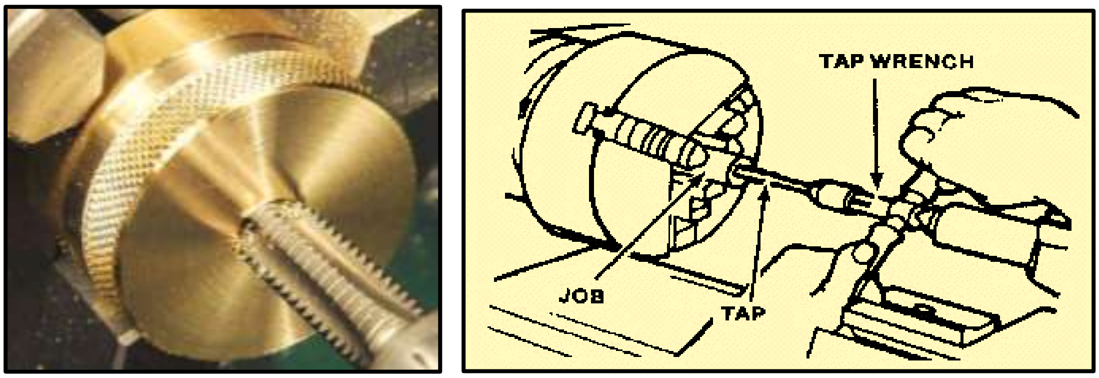


Figure 7-71: Tapping on a lathe by power and tapping on a lathe by hand.

19) Knurling:

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.

Knurling Tools: The knurling tool 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. (Figure 7-72).

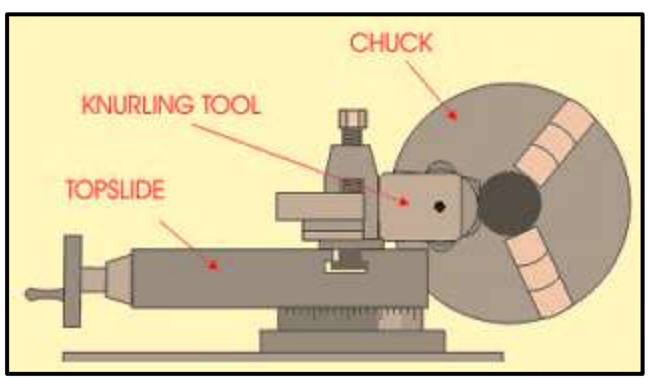


Figure 7-72: The knurling tool.

Holding Devices for Knurling: The setup for knurling can be made between centers or mounted in a solid chuck. 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.

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. 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. (Figure 7-73).

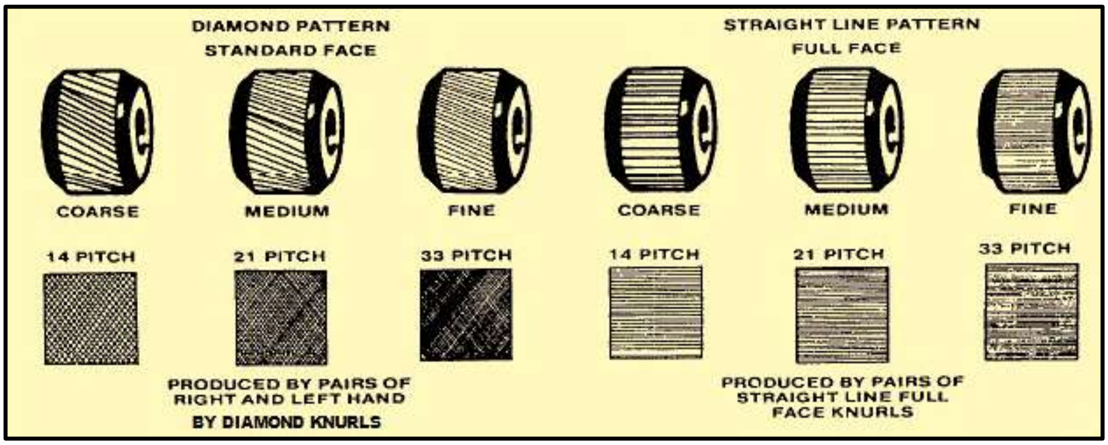


Figure 7-73. Knurling patterns and pitches.

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. 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.

Apply oil generously over the area to be knurled. Start the lathe forcing the knurls into the work about **0.010 inch**. As the impression starts to form, engage the carriage feed lever. 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 7-74).

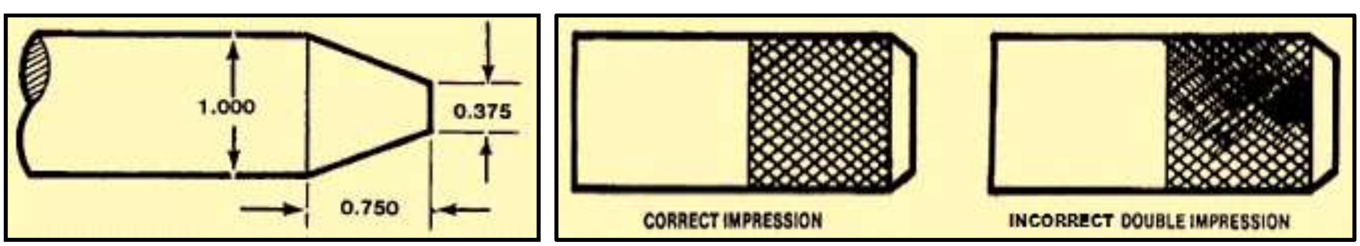


Figure 7-74. Starting the knurl, Correct and incorrect knurls.

The knurling operation is started by determining the location and length of the knurl, and then setting the machine for knurling. 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. Never stop the carriage while the tool is in contact with the work and the work is still revolving as this will cause wear rings on the work surface. 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.

20) 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, 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. If the workpiece is to be rotated, should be mounted in a chuck. The twist drill will be fed into the end of the workpiece, on a faceplate, or in a collet. That is, the drill size must allow sufficient material for tapping, reaming, and boring if such operations, as shown below (Figure 7-75).

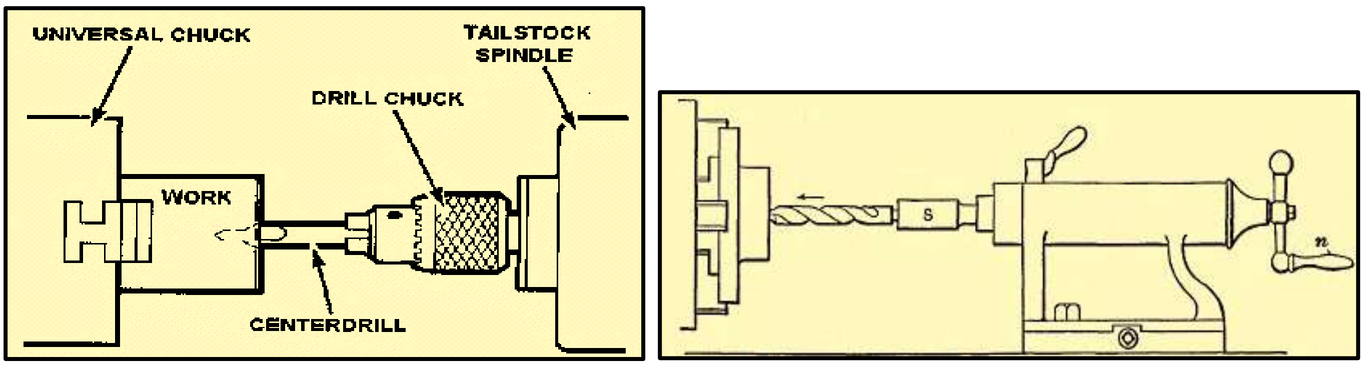


Figure 7-75: Lathe drilling operations

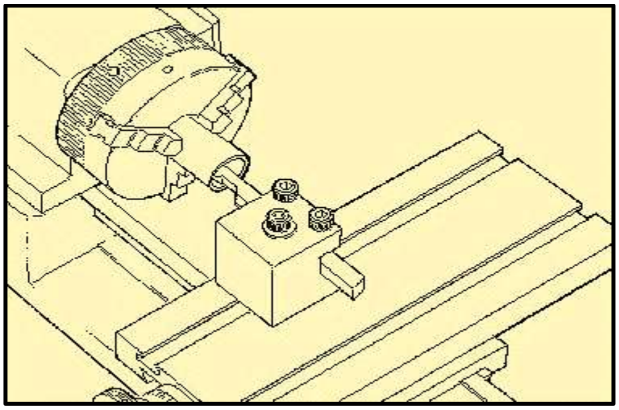
Supporting Drills in the Headstock: A universal or independent jaw chuck can also be used to hold and turn twist drills if a headstock drill chuck is not available. Tapered-shank twist drills can be mounted in the headstock by using a special adapter, such as a sleeve with an internal taper to hold the tapered drill, while the outside of the sleeve is made to fit into the headstock spindle. 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.

Drilling Operations: To start the drilling operation, compute the correct RPM and set the spindle speed accordingly. Ensure the tailstock is clamped down on the lathe ways, and use a suitable cutting fluid while drilling. The graduations on the tailstock spindle are used to determine the depth of cut. If a large twist drill is used, it should be preceded by a pilot drill, the diameter of which should be wider than the larger drills. 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 to steady the drill.

21) 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. 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. 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.

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. 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, and most general boring is accomplished with a 90° cutter bit. (Figure 7-76).



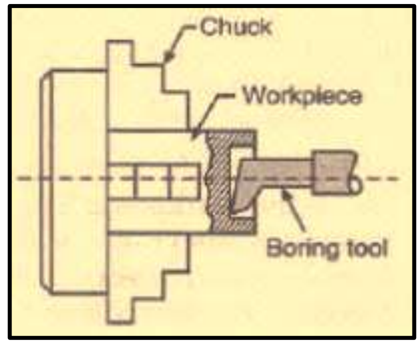


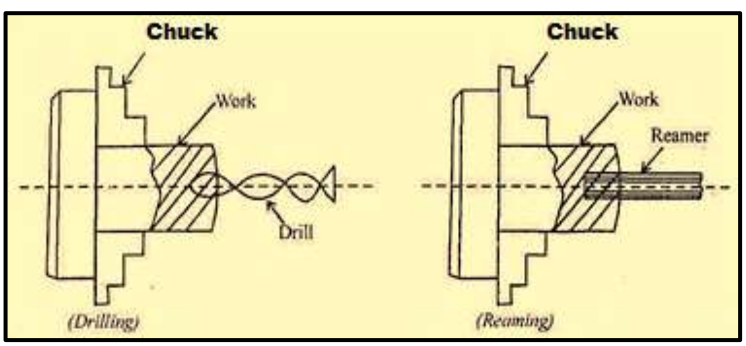
Figure 7-76: Boring with the lathe

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. 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.

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.

22) Reaming on the Lathe:

Reamers are used to **finish drilled holes or bores** quickly and accurately to a **specified diameter**. 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. 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.



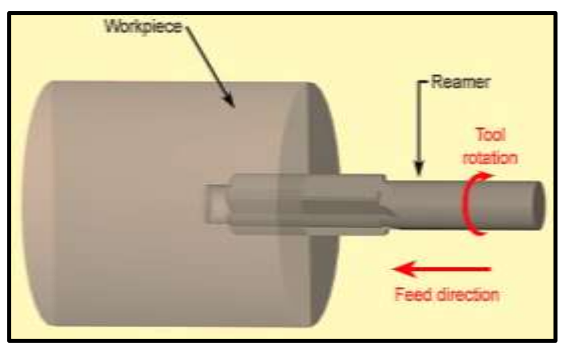


Figure 7-77: Reaming on the lathe.

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.

23) 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. The **bastard mill-type** hand file is used for roughing, and the second **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. For filing **ferrous metals**, the lathe spindle speed should be **four or five times** greater than the rough turning speed. For **nonferrous metals**, the lathe spindle speed should be only **two or three times** greater than the roughing speed.

Filing Operations: Mill files are generally considered best for lathe filing. The file is held at an angle of **about 10°** to the right and moved with a slow sliding motion from left to right. 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 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. (Figure 7-78).

Polishing on the Lathe: Polishing with either **abrasive cloth or abrasive paper** is desirable to improve the surface finish after filing. 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 polishing will slightly reduce the dimensions of the workpiece, **0.00025 to 0.0005 inch** should be always allowed for this operation.

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. (Figure 7- 78).

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

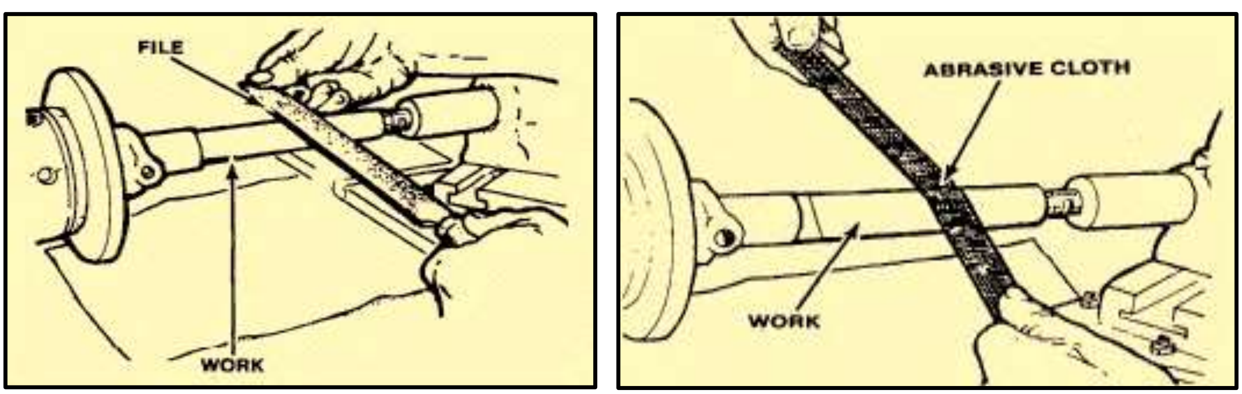


Figure 7-78: Filing and polishing on a lathe.

24) 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.

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. First determine the stock required by adding the throws plus 1/8 inch for machining. When turning an eccentric that has the different centers placed too close together, 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. 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 7-79).

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.

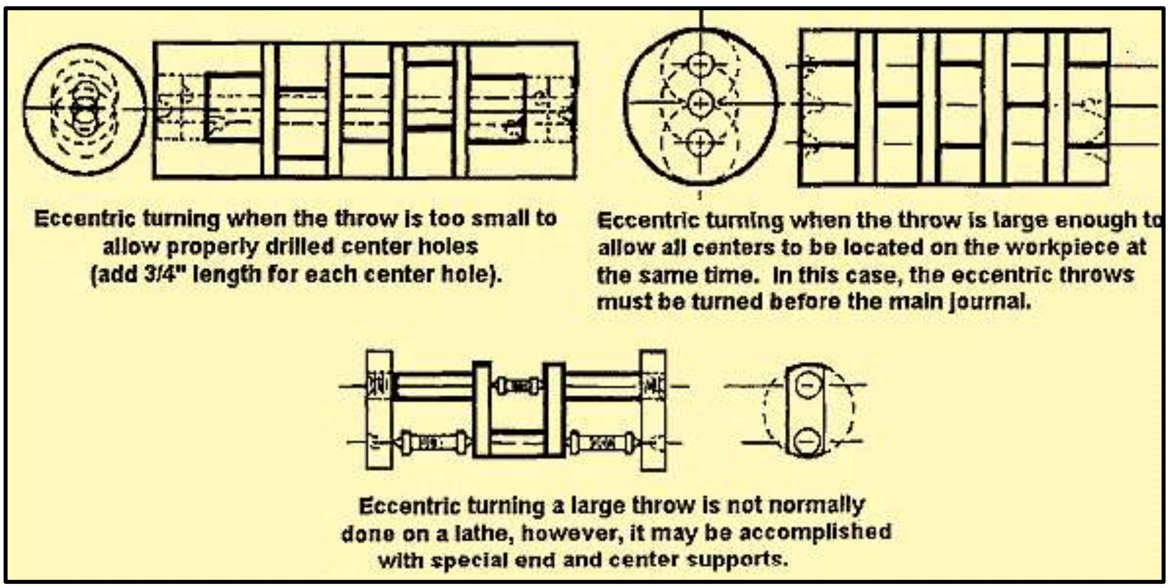


Figure 7-79. Eccentric turning.

25) Recessing Drilled and Bored Holes:

Recessing, sometimes called channeling or cambering, is the process of cutting a groove inside of a drilled, bored, or reamed hole. Recesses 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 7-80).

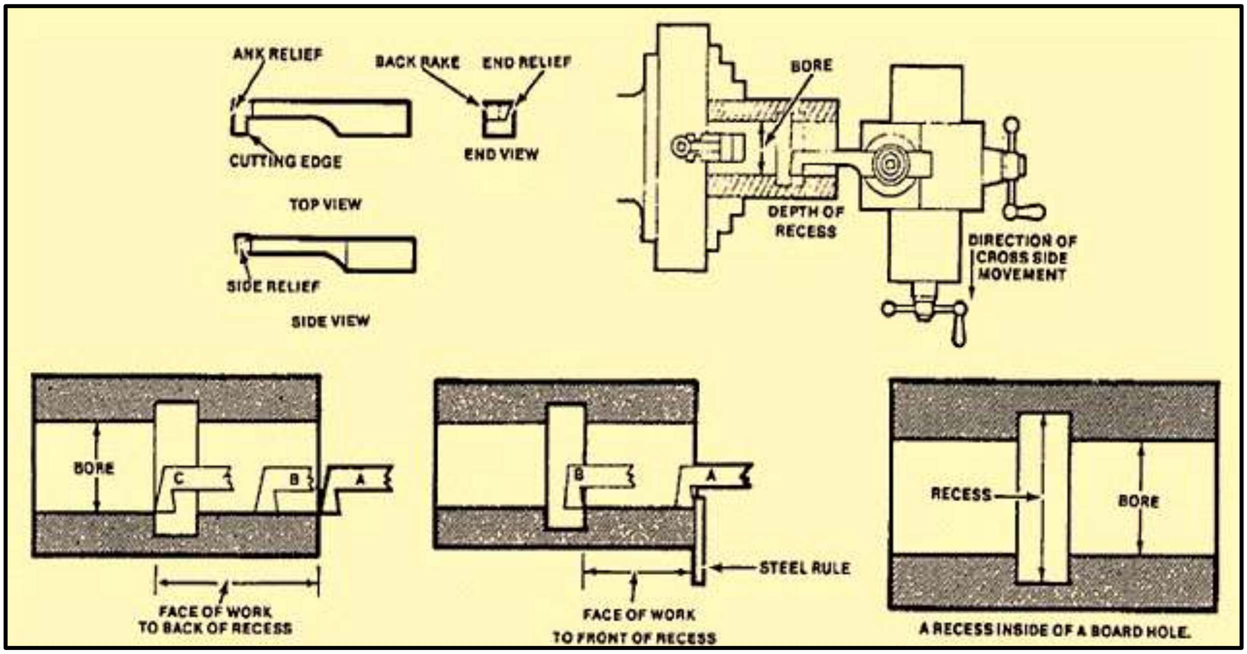


Figure 7-80: Recessing.

26) Grinding with Lathe Tool Post Grinder:

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. The grinding wheel takes the place of a lathe cutting tool, to 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.

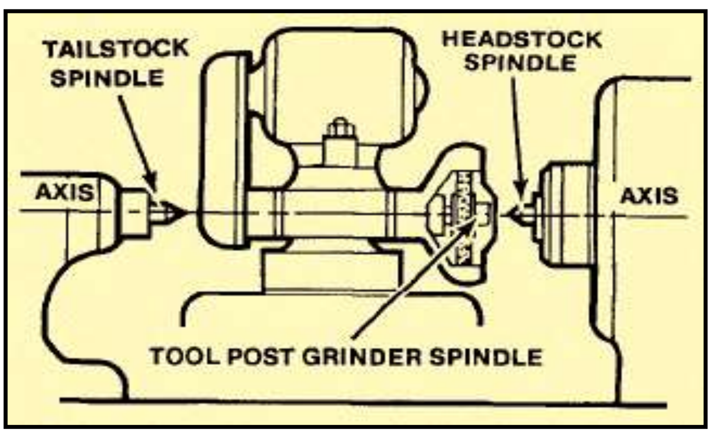


Figure 7-81. Aligning the tool post grinder.

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. 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. Be sure that the combination is not reversed. 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. 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. The maximum depth of cut is **0.002 inch**.

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

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.** Never stop the rotation of the work or the grinding wheel while they are in contact with each other.

27) Milling on the Lathe:

Milling operations may be performed on the lathe using the Versa-Mil and the lathe milling fixture. The lathe milling fixture complements the Versa-Mil and adds basic capabilities for milling. If the Versa-Mil is out of action or used for another job, many milling operations can still be accomplished by using only the milling fixture (Figure 7-82).

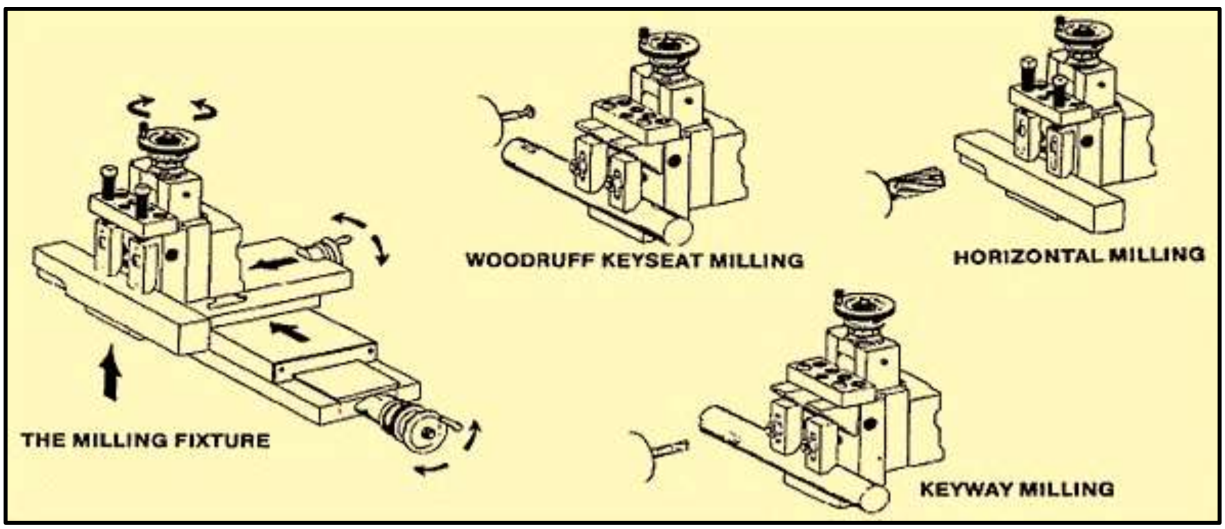


Figure 7-82. Lathe milling fixture operations.

28) Using the Micrometer Carriage Stop:

The micrometer carriage stop is used to accurately position the lathe carriage. The carriage can be accurately positioned within **0.001 inch**. 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.

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. Use power feed to bring the carriage within **1/ 32 inch of the stop**. Move the carriage by hand the remaining distance. (Figure 7-83).

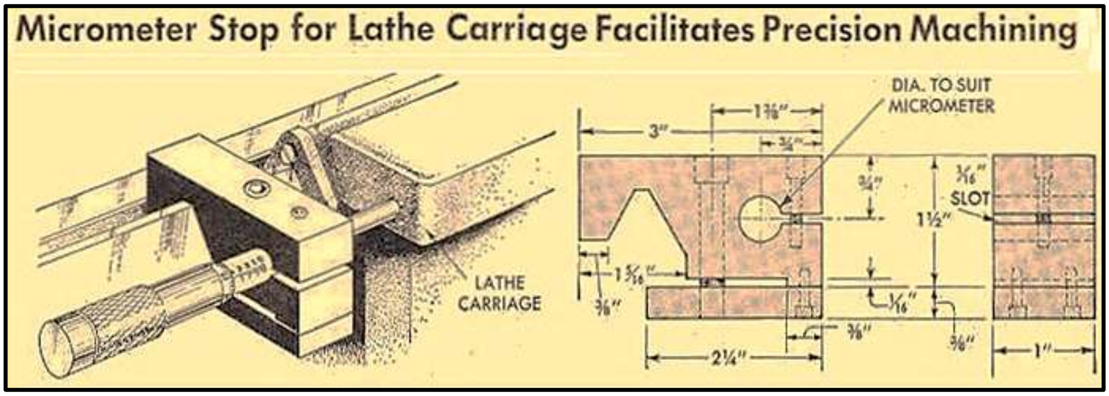


Figure 7-83: Using the micrometer stop lathe carriage.

29) Steady Rest, Lathe Dog and Follower Rest:

The steady rest consists of a frame and three adjustable jaws to set the workpiece. The main purpose of the steady rest is to prevent springing or deflection of long, slender, flexible workpieces, and to support heavy cuts to be made, or to support work for drilling, boring, or internal threading. 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 below, has a bearing you surface, a hole through, which the work extends, and adjusting screws. Use a dial indicator to ensure concentricity. (Figure 7-84).

Using Steady Rest with the Lathe Dog: 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. With the workpiece mounted between the centers, **tie the lathe dog**, then remove the tailstock center and perform the necessary machining. Use a leather strap or rawhide thong to tie the work to the drive plate and to prevent it from moving off the headstock center. Mount the work between centers and machine the bearing surface. (Figure 7-85).

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. 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. 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 7-86).

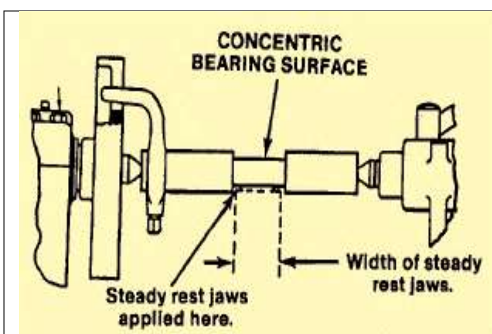


Figure 7-84. Using the steady rest.

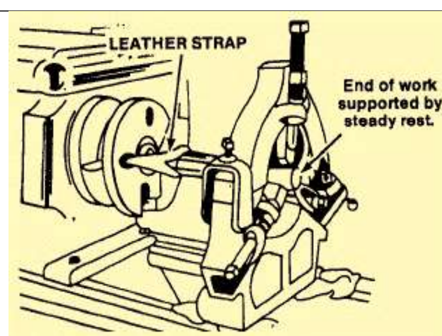


Figure 7-85. Tying the lathe dog.

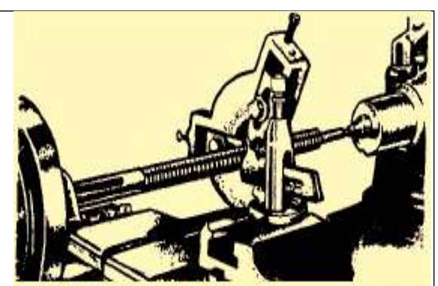


Figure 7-86. Using the follower rest in threading

Figure 7-83, 7-84, and 7-85: Using the steady rest, lathe dog and the follower rest.

30) Lathe Cutting Fluids:

The purposes of using cutting fluids on the lathe are to cool the tool bit and workpiece that are being machined, increase the life of the cutting tool, make a smoother surface finish, deter rust, and wash away chips. Cutting fluids can be sprayed, dripped, wiped, or flooded onto the point where the cutting action is taking place. Generally, cutting fluids should only be used if the speed or cutting action requires the use of cutting fluids, as follows:

Lard Oil: Is one of the **oldest and best** cutting oils, especially good for thread cutting, tapping, deep hole drilling and reaming. Lard oil has a high degree of adhesion or oiliness, a relatively high specific heat, and its fluidity changes only slightly with temperature. It is an excellent rust preventive and produces a smooth finish on the workpiece. Because lard oil is **expensive**, is seldom used in a **pure** state, but combined with other ingredients.

Mineral Oil: Mineral oils are **petroleum-based oils** that range in viscosity from kerosene to light paraffin oils. Mineral oil is very stable and does not develop disagreeable odors like lard oil, however, it lacks some of the good qualities of lard oil such as adhesion, oiliness, and high specific heat. Because it is relatively **inexpensive**, is commonly mixed with lard oil or other chemicals to provide cutting oils with desirable characteristics. Two mineral oils, kerosene and turpentine, are often used alone for machining **aluminum and magnesium**. Paraffin oil is used alone or with lard oil for machining **copper and brass**.

Mineral-Lard Cutting Oil Mixture: Various mixtures of mineral oils and lard oils are used as cutting oils which combine the good points of both ingredients but prove more economical and often as effective as pure lard oil.

Sulfurized Fatty-Mineral Oil. Most good cutting oils contain mineral oil and lard oil with various amounts of **sulfur and chlorine** which give the oils good antiweld properties and promote free machining. These oils play an important part in present-day machining because they provide **good finishes** on most materials and aid the cutting of tough material.

Soluble Cutting Oils: Water is an excellent cooling medium but has little lubricating value and hastens rust and corrosion. Therefore, mineral oils or lard oils which can be mixed with water are often used to form a cutting oil. A soluble oil and water mix has lubricating qualities dependent upon the strength of the solution. Generally, soluble oil and water is used for rough cutting where quick dissipation of heat is most important. Borax and trisodium phosphate (TSP) are sometimes added to the solution to improve its corrosion resistance.

Soda-Water Mixtures: Salts, such as soda ash, are sometimes added to water to help **control rust**. This mixture is the cheapest of all coolants and has practically no lubricating value. Lard oil and soap in small quantities are sometimes added to the mixture to improve its lubricating qualities. Soda water is used only where **cooling** is the prime consideration and **lubrication** a secondary consideration. It is especially suitable in reaming and threading operations on cast iron where a better finish is desired.

White Lead and Lard Oil Mixture: White lead can be mixed with either lard oil or mineral oil to form cutting oils especially suitable for **difficult machining** of very hard metals.

31) Lathe Machining Safety:

All lathe operators must be constantly aware of the safety hazards that are associated with using the lathe and must know all safety precautions to avoid accidents and injuries. Carelessness and ignorance are two great menaces to personal safety. Other hazards can be mechanically related to working with the lathe, such as proper machine maintenance and setup. Some important safety precautions to follow when using lathes are:

- ✓ Correct dress is important, remove rings and watches, and roll sleeves above elbows.
- ✓ Don´t change spindle speeds until the lathe comes to a complete stop.
- ✓ Handle heavy chucks with care and protect the lathe ways with a block of wood when installing a chuck.
- ✓ Know where the emergency stop is before operating the lathe.
- ✓ Keep tools overhang as short as possible.
- ✓ Never attempt to measure work while it is turning.
- ✓ Protect the lathe ways when grinding or filing.
- ✓ Do not wrap sand paper or emery cloth around the workpiece.

MILLING MACHINES:

The main functions of milling machines is to produce flat surfaces in any orientation, as well as, surfaces of revolution, helical surfaces and contoured surfaces of various configurations. Milling machines are designed the way that the cutter can make many individual cuts on the workpiece in a single run or, accomplished by cutters with many teeth at high speed, advance the workpiece through the cutter slowly. The speed at which the piece advances through the cutter is called feed rate, or only feed, most often measured in length of material per complete revolution of the cutter. Basic milling machine configurations are shown in Figure 8-1.

1) Milling Machine Types:

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.

Plain Vertical Milling 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. 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.

Plain Horizontal Milling Machines: Contains the drive motor and gearing in a fixed horizontal position. 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.

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. The most 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 is, the **spindle** is mounted on a **ram** or movable housing, as its name implies.

- **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.

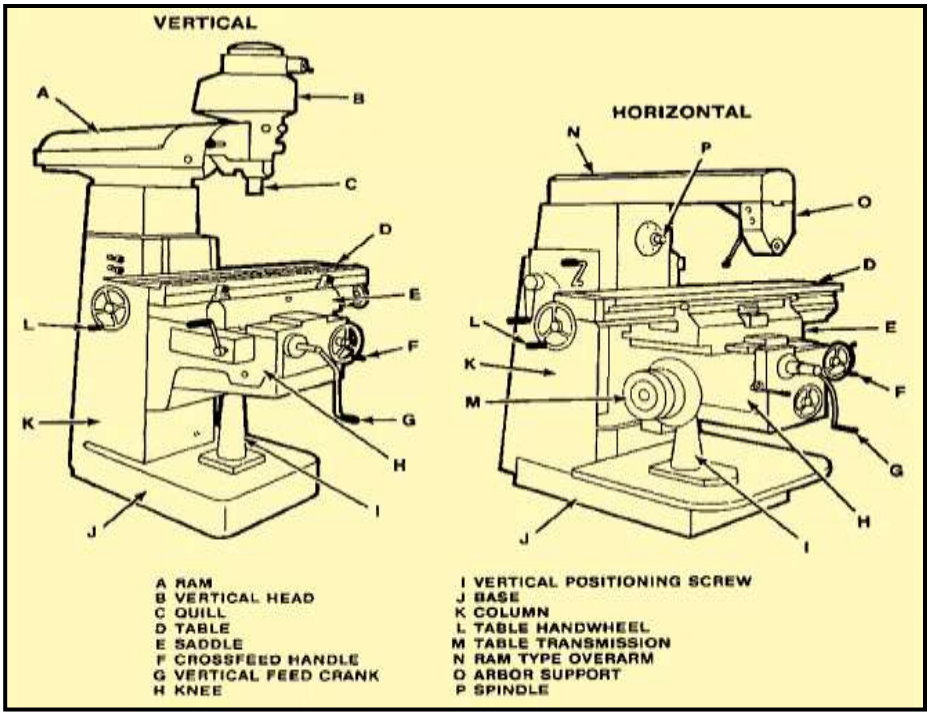


Figure 8-1: Milling machine parts.

2) Milling Tools and Equipment:

Milling cutters are usually made of high-speed steel and are available in a great variety of shapes and sizes for various purposes. The Figure 8-2 shows two views of a common milling cutter with its parts and angles identified. These parts and angles are common to all cutter types.

- The pitch is determined by the number of teeth. The facing tooth face is the forward face or 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 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, 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.

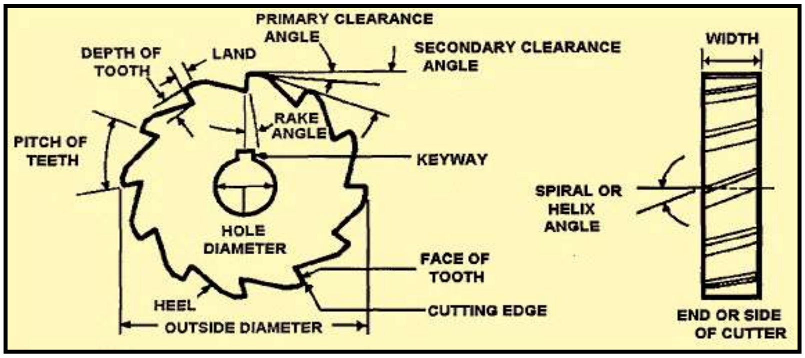


Figure 8-2: Milling cutter nomenclature.

Types of Teeth: The teeth of milling cutters may be **right-hand** or **left-hand** rotation, and with either right-hand or left-hand helix. The hand of the cutter can be determined 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 8-3).

Saw Teeth: 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° primary clearance**. Sometimes the teeth are provided with off-set nicks which break up chips and make coarser feeds possible. (Figure 8-4).

Helical Milling Cutters: Are 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.

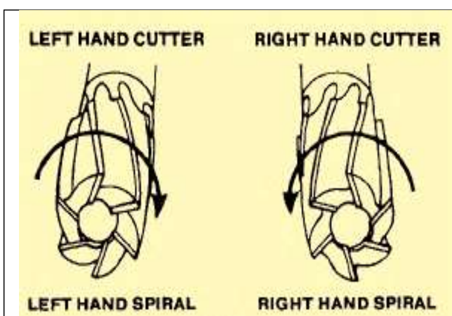


Figure 8-3. Left and right cutters.

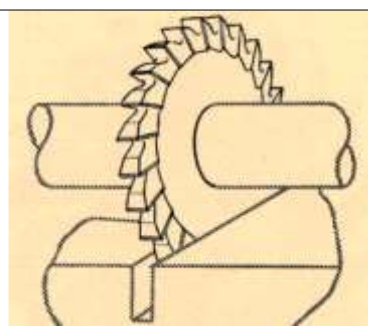


Figure 8-4. Saw teeth

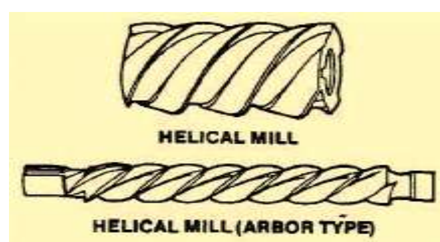


Figure 8-5: Helical Milling Cutters

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).

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. (Figure 8-6).

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

Staggered Tooth Cutters: Have 10° positive radial rake and 10° positive axial rake (helix), right and left-hand on alternate teeth. These cutters are ground with clearance and slight concavity on the side to avoid having the side tooth drag in the cut. The positive axial and radial rake design, with alternate side teeth cutting, makes the staggered tooth cutter a free cutting tool with excellent chip disposal characteristics. (Figure 8-6).

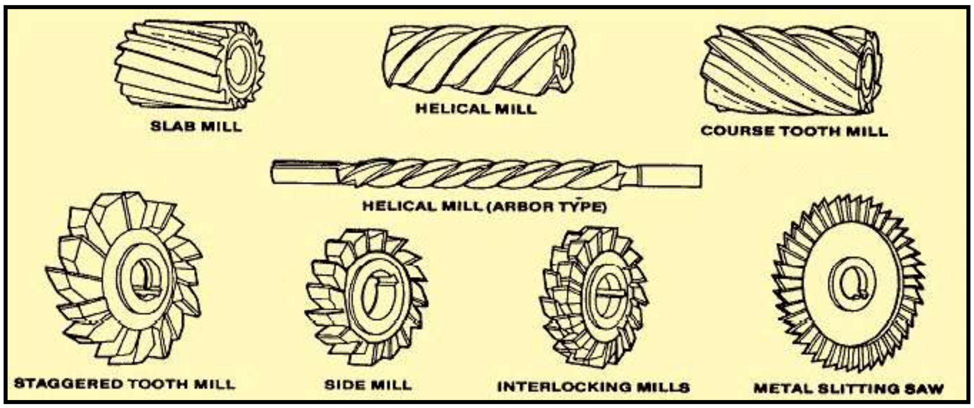


Figure 8-6. Various milling cutters.

End Milling Cutters: The end milling cutter, have straight or tapered shank, also called an end mill, has teeth on the end as well as the periphery, and may have straight or spiral flutes, classified as left-hand or right-hand cutters depending on the direction of rotation of the flutes. (Figure 8-7).

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.

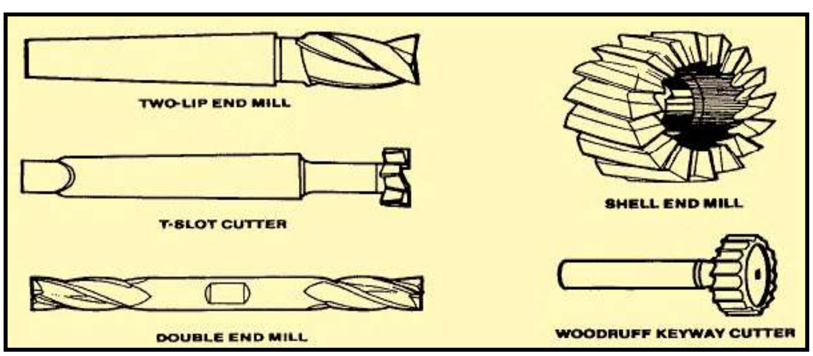


Figure 8-7. End milling cutters, T-slot, and Woodruff cutters.

Large end milling cutters (normally over 2 inches in diameter) 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: Is made in straight, tapered-shank, and arbor-mounted types. The most common cutters of this type, under 1 1/2 inches, are provided with a shank. They have teeth on the periphery and slightly concave sides to provide clearance. These cutters are used for milling semi cylindrical keyways in shafts. (Figure 8-7).

Angle Milling Cutters: Has peripheral teeth which are neither parallel nor perpendicular to the cutter axis. Common operations performed with angle cutters are cutting V-notches and serrations. 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. (Figure 8-8).

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. (Figure 8-8).

Concave and Convex Milling Cutters: Are formed tooth cutters 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).

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. (Figure 8-8).

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 especially 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. (Figure 8-8).

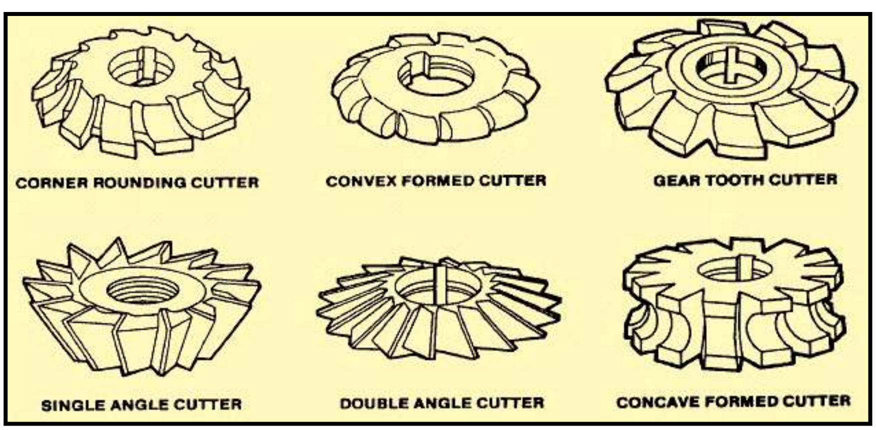


Figure 8-8. Angle, concave, convex, corner, and gear cutters.

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 generated in cutting.

Cutters are selected for 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. 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 Cutters: 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. (Figure 8-9).

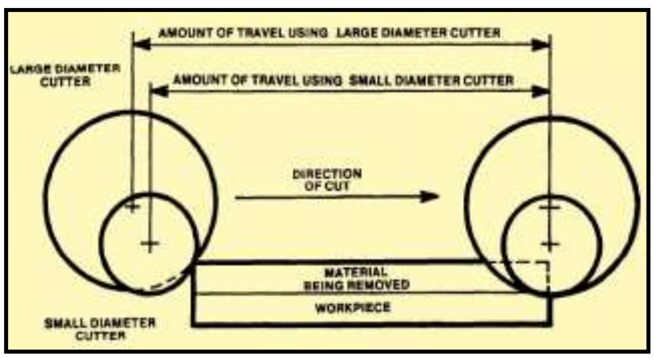


Figure 8-9. Effect of milling cutting diameter on workpiece travel.

3) Arbors:

Milling machine arbors are made in various lengths and in standard diameters of 7/8,1,1¼, and 1½ inch. The shank is made to fit the taper hole in the spindle (self-holding or self-releasing), while the other end is threaded. The spindle taper in most milling machines is self-releasing. The Standard Milling Machine tapers are: the Brown and Sharpe taper, and the Brown and Sharpe taper with tang (Figure 8-10).

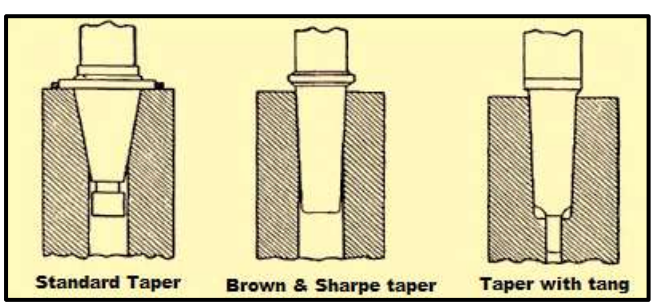


Figure 8-10. Tapers used for milling machine arbors.

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 tapers are identified by the number 30, 40, 50, or 60. Number 50 is the most commonly used size on all modern machines. (Figure 8-11).

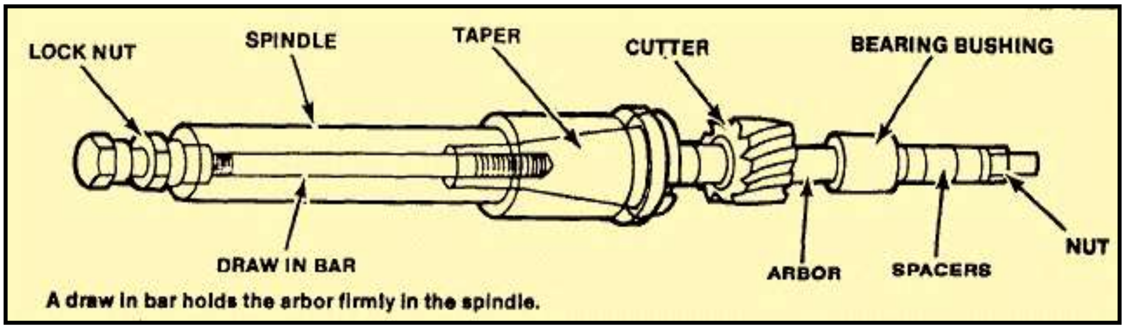


Figure 8-11. Standard milling machine arbor.

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

Arbor Installation: The end of the arbor, opposite the taper, has one or more supports 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 of the arbor fitting inside a bushing of the arbor support. The arbor may also be firmly supported as it turns in the arbor support bearing suspended from the over-arm. (Figure 8-12).

Arbor Styles: Arbor "**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. Arbor "**Style B**" has one or more bearing collars positioned to any part of the arbor, close to the cutter to-obtain rigid setups in heavy duty milling operations. Arbor "**Style C**" is 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. (Figure 8-13).

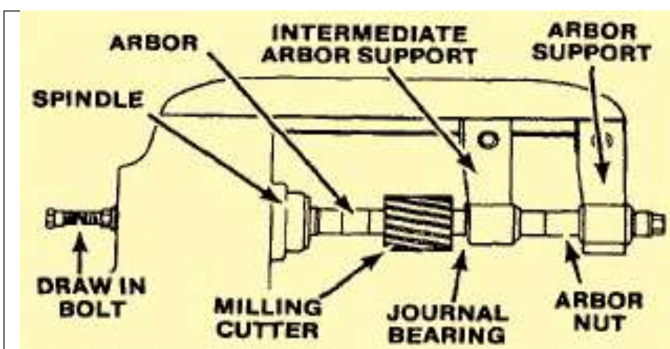


Figure 8-12. Arbor installation.

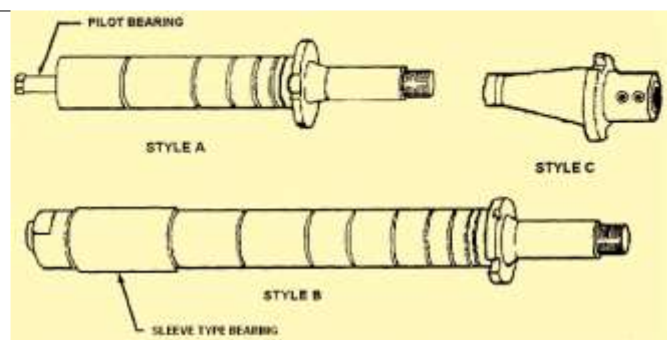


Figure 8-13. Typical milling arbors.

Screw Arbors: Are used to hold small cutters with 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. The slitting saw milling cutter arbor 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. (Figure 8-14).

Chuck Adaptor: 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 8-15).

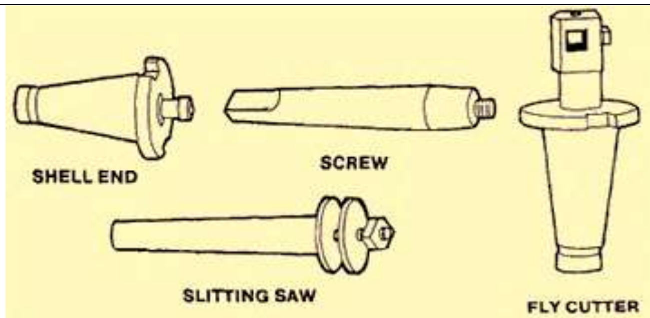


Figure 8-14. Arbor variations.

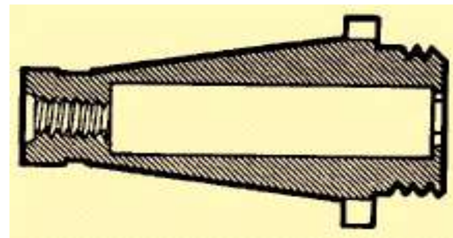


Figure 8-17. Chuck adaptor.

4) Milling Tools and Accessories:

Collets: 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. They are made in several forms, similar to drilling machine sockets and sleeves, except that their tapers are not alike. (Figure 8-16).

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 8-17).

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).

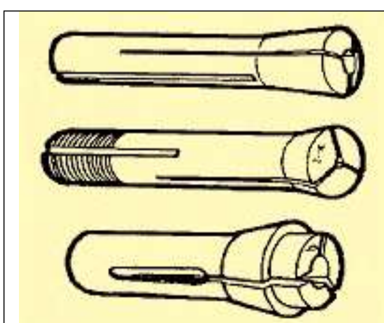


Figure 8-16. Solid and spring collets.

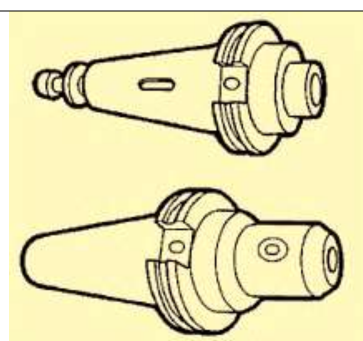


Figure 8-17. Milling machine adaptors.

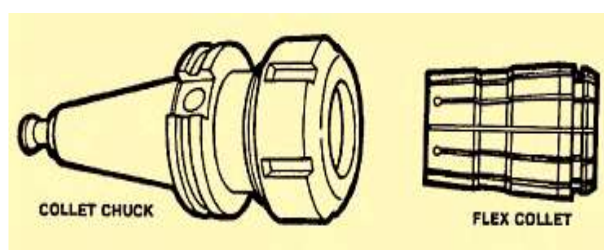


Figure 8-18. Quick-change adaptor and tool holder.

Vises: Either a plain or swivel-type vise is furnished with each milling machine. The **plain vise** is used for milling straight workplaces, 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** can be set with both horizontal and vertical angles, used for flat and angular milling. 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.

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.

5) Holding Workpieces:

Clamping Workpieces to the Table: 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. 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.

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.

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. 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.

Holding Workpieces in a Vise: The 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. 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. (Figure 8-21).

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. (Figure 8-22).

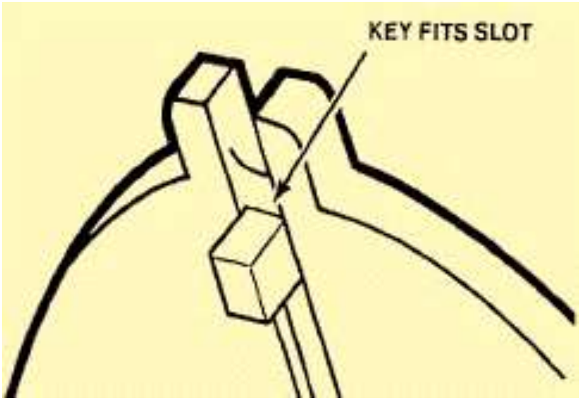


Figure 8-21. Locating key on a vise.

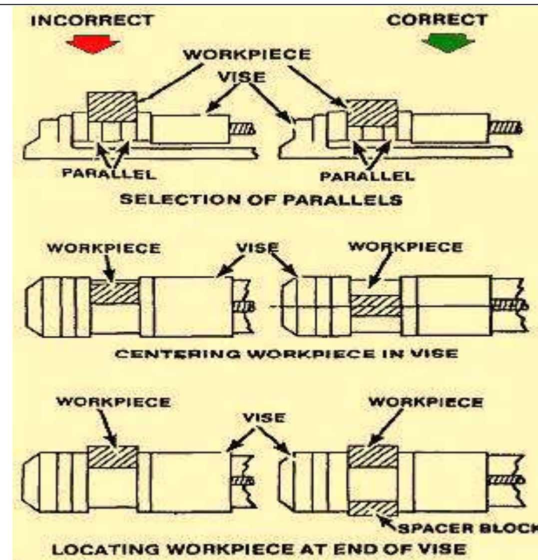


Figure 8-22. Mounting workpiece in the vise

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. 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. 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, but it is not adaptable for heavy milling.

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. 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. (Figure 8-21).

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. If the workpiece is so thin that it is impossible to let it extend over the top of the vise, hold down straps is generally used. (Figure 8-24).

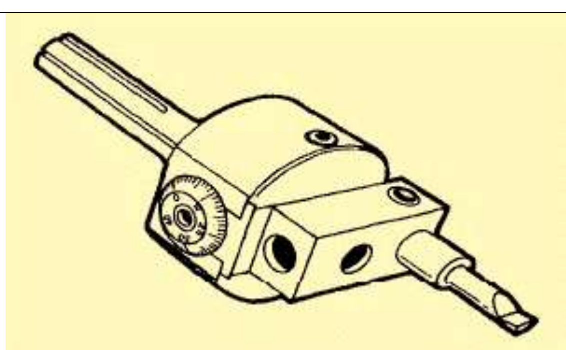


Figure 8-21. Offset boring head.

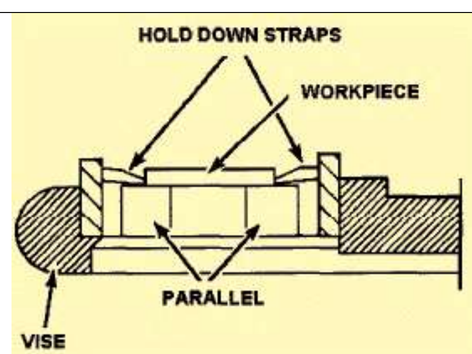


Figure 8-24. Application of hold-down straps.

6) 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. The 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. (Figure 8-19).

Index Fixture: Is also called a dividing head. Consists of an index head, and footstock similar to the tailstock of a lathe, attached to the worktable of the milling machine by T-slot bolts. The index plate containing graduations is used to control the rotation of the index head spindle, 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, and may also be held in a chuck mounted to the index head spindle. (Figure 8-19).

Index Head: The index head of the indexing fixture contains an indexing mechanism which is used to **control the rotation** of the index head spindle to space or divide a workpiece accurately. Is a simple indexing mechanism which **consists of a 40-tooth** worm wheel fastened to the index head spindle, a single-cut worm, a crank for turning the worm shaft, and an index plate with sectors. 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. (Figure 8-19).

Rotary Table: 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).

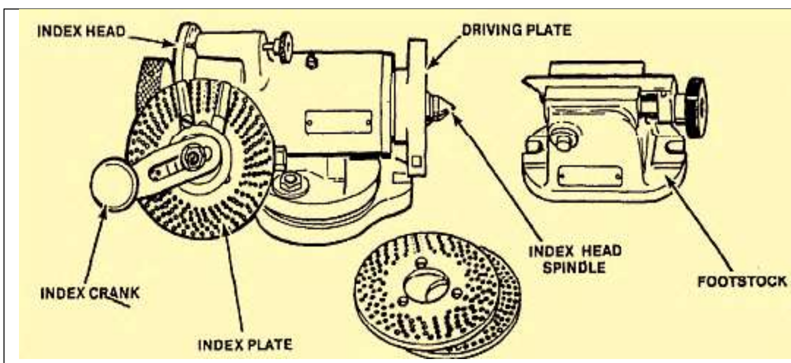


Figure 8-19. Indexing fixture.

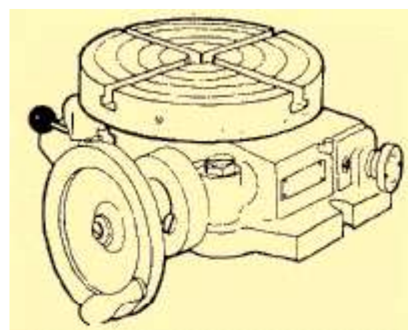


Figure 8-20. Rotary table.

Index Plate: The indexing plate 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 of any circle. With the interchangeable plates regularly furnished with most index heads, the spacing necessary for most gears, bolt heads, milling cutters, splines, and so forth can be obtained. The following sets of plates are standard equipment. (Figure 8-21).

Index Plate Sector: 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. (Figure 8-21).

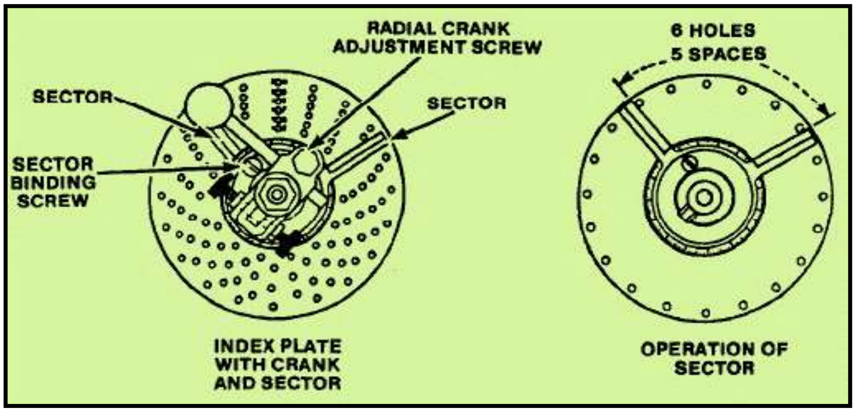


Figure 8-21. Index plate and sector.

Brown and Sharpe Index 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 Index 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.

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 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 worm shaft 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. 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.

7) General Milling Operations:

Setup: Setting up the job, the table, the taper in the spindle, selecting the proper milling cutter and holding the cutter by the best means under the circumstances, means the success of any milling operation. 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 the main practices are mentioned below:

Plain Milling: Is 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: 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, serrations, and grooves. Milling dovetails is a typical example of angular milling.

Dovetails Milling: 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.

Straddle Milling: When two or more parallel vertical surfaces are machined at a single cut, the operation is called straddle milling. Parallel slots of equal depth can be milled by using straddle mills of equal diameters. The workpiece is usually mounted between centers in the indexing fixture or mounted vertically in a swivel vise. 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.

Face Milling: Is the milling of surfaces that are perpendicular to the cutter axis. 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.

Gang Milling: Is the term applied to an operation, when two or more milling cutters are mounted on the same arbor, used to cut horizontal surfaces; for example, several workplaces need a slot, a flat surface and an angular groove. Then, all cutters may perform the same type of operation or each cutter may perform a different type of operation. The best method to cut these would be the gang milling operation.

Form Milling: Is the machining of special contours composed of curves, straight lines, beads, half-round recesses, or entirely of curves, at a single cut. These operations are 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 several complex surfaces in a single cut such as are produced by gang milling.

Fly cutting: 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, held and rotated by a flying cutter arbor. You can grind this cutter when you need a special form cutter for a very limited number of parts, or 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.

Flat Surfaces: Another type of fly cutter, which differs mainly in the design of the arbor, easily manufactured in the shop using common lathe tool bits, can be used to mill flat surfaces as in plain or face milling. 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, as the arbor is too short, that only very shallow holes can be bored.

Keyway Milling: Keyways are grooves of different shapes cut along the axis of the cylindrical surface of shafts to provide a positive method of locating and driving members on the shafts. 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-end Machine Key and the Round-end Machine Key.

Woodruff Keys: 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.

Square-end Machine Keys: May be square or rectangular in section and several times as long as they are wide. Commonly the key width is 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 and depth of the keyway (or rectangular section keys) is 1/2 the thickness of the key.

Round-end Machine Keys: 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.

T-Slot Milling: 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 necessary T-slot. The size depends upon the size of the T-bolts which the T-slot will be used. Dimensions of T-slot bolts and T-slots are standardized for specific bolt diameters.

Sawing and Parting: The metal slitting saw milling cutters are used to part stock on a milling machine. While parting thin material such as sheet metal, the workpiece 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.

Helical Milling: When milling a helix, a universal index head should be 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 for a given longitudinal movement of the table. Helical parts require the use of special formed milling cutters and double-angle milling cutters and calculations and formulas are necessary to compute the proper machining. Include helical gears, spiral flute milling cutters, twist drills, and helical cam grooves.

Gear Cutting: Gear teeth may be cut on milling machines, using formed milling cutters, called involute gear cutters, manufactured in many pitch sizes and shapes for different numbers of teeth per gear. 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 this operation.

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, 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 straddle milling, 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 spline fittings have 4, 6, 10, or 16 splines, and their dimensions depend upon the class of tit for the desired application.

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.

8) Milling Calculations:

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 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.

Selecting Proper Cutting Speeds: The approximate values given in Table 8-1 may be used as a guide for selecting the proper cutting speed. 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 %**. For roughing cuts, a moderate speed and coarse feed often give best results; for finishing cuts, use higher speed and lighter feed. The formula for calculating spindle speed in revolutions per minute is as follows:

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

Where:

RPM = Spindle speed (in revolutions per minute)
CS = cutting speed of milling cutter (in SFM)
D = diameter of milling cutter (in inches)

Milling-Machine Operations
Table 8-1. Recommended Cutting Speed for Milling in Feet per Minute (fpm)

Work Material	Hardness, Bhn	Cutting Speed, fpm	
		High-Speed Steel	Carbide
Plain Carbon Steel, AISI 1010 to AISI 1030	Up to 150 150 to 200	110 100 to 140 100 80 to 120	600 400 to 900 450 300 to 700
AISI B1111, AISI B1112, AISI B1113, Steel	140 to 180	140 110 to 200	650 400 to 1200
Plain Carbon Steel, AISI 1040 to 1095	120 to 180 180 to 220 220 to 300	195 80 to 120 85 70 to 110 60 30 to 80	600 400 to 800 350 300 to 500 200 100 to 300
All Alloy Steels Having 0.3% or Less Carbon Content: AISI 1320, AISI 3120, AISI 4130, AISI 4020, AISI 5020, AISI 4118, AISI 9310, etc.	180 to 220 220 to 300 300 to 400	80 65 to 100 60 30 to 80 40 30 to 50	350 300 to 600 300 200 to 350 125 100 to 150
All Alloy Steels Having More Than 0.3% Carbon Content: AISI 1340, AISI 2340, AISI 4140, AISI 4150, AISI 4340, AISI 5140, AISI 5150, AISI 52100, AISI 8660, AISI 9260, etc.	180 to 220 220 to 300 300 to 400	80 60 to 100 55 30 to 80 30 20 to 50	325 275 to 450 250 180 to 300 100 80 to 130

Example: A milling cut is to be taken with a 0.50 inch high speed steel (HSS) end mill on a piece of 1018 steel with a Brinnel Hardness of 200. Calculate the RPM setting to perform this cut.

Cutting Speed = 90 (fpm)
Diameter of Cutter = 0.50

$$RPM = \frac{CS \times 4}{D} = \frac{90 \times 4}{0.50} = \frac{360}{0.50} = \mathbf{720\ RPM}$$

Example: The spindle speed for machining a piece of steel at a speed of 35 SFM with a cutter 2 inches in diameter is calculated as follows:

$$RPM = \frac{CS \times 4}{D} = \frac{35 \times 4}{2} = \frac{140}{2} = \mathbf{70\ RPM}$$

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

Feeds for Milling: 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.

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), the number of teeth on the cutter, and the revolutions per minute of the cutter. The formula used to find the feed in **inches per minute** is:

Fr = CPT x N x RPM

Where:

Fr = Feed rate, in inches per minute
CPT = Chip per tooth
N = Number of teeth per minute of the milling cutter

Example: The spindle speed for machining a piece of steel at a speed of 300 SFM with a cutter 1/2 inches in diameter. Calculate the feed rate, considering CPT = 0.005 and N = 2 teeth.

$$RPM = \frac{CS \times 4}{D} = \frac{300 \times 4}{1/2} = \frac{1200}{0.5} = \mathbf{2400\ RPM}$$

Then, calculating the feed rate, considering CPT = 0.005 and N = 2 teeth of the milling cutter:

Fr = CPT x N x RPM = 0.005 x 2 x 2,400 = **24 in./min** (**inches per minute)**.

Direction of Feed: It is usually regarded as standard practice to feed the workpiece 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.

9) Milling Cutting Fluids:

Cutting oils are basically **water-based soluble oils, petroleum oils, and synthetic 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. Water-based coolants have excellent heat transfer qualities; other oils result in good surface finishes. 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.

10) Milling Machines Safety Rules:

- ✓ Milling machines require special safety precautions while being used.
- ✓ Do not make contact with the revolving cutter.
- ✓ Use the buddy system when moving heavy attachments.
- ✓ 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.
- ✓ 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

REFERENCES & LINKS:

SENAI/Brazil: Processos Mecânicos de Usinagem, 2006;
Sandvik: Metal Cutting Technical Guide, Catalogues;
Machine Shop Work, by Frederick W. Turner, Oscar E. Perrigo, Howard P. Fairfield, 1918;
Metals Handbook, 9th Edition, 1989;
Indian Institute of Technology, Kharagpur, Study Materials.

Links:

http://www.cnccookbook.com/CCCNCGCodeCourse.htm
http://www.hsmworks.com/docs/cncbook/en/#start
http://www.helmancnc.com/how-to-make-cnc-program/
http://www.cncci.com/resources/articles/CNC%20basics%201.htm
http://users.bergen.org/jdefalco/CNC/
http://en.wikipedia.org/wiki/Numerical_control
http://en.wikipedia.org/wiki/Machining
http://www.ntma.org/
https://www.nims-skills.org/web/nims/home
http://www.iso.org/iso/products/
http://www.efunda.com/processes/machining/mill_design.cfm
http://www.astm.org/Standards/