Tomsk Polytechnic University

V.N.Kozlov

TECHNOLOGY of MECHANICAL ENGINEERING

Part 2 (part 2a)

Textbook

Tomsk 2002

UDK 621.91.002(075)

V.N.Kozlov. Technology of Mechanical Engineering. Part 2. Textbook. Tomsk: TPU Press, 2002, 148 pp.

This textbook is devoted to technological opportunities of machine tools and designing of technological processes

The textbook is prepared at the Department of Mechanical Engineering of Tomsk Polytechnic University. It is recommended for foreign students following the Bachelor Degree Program in Mechanical Engineering at Tomsk Polytechnic University.

Reviewed by: V.F. Skvorsov, Head of the Department “Technology of Mechanical Engineering, Cutting and Tools”, TPU, K. Sc.

PREFACE

The discipline "Technology of Mechanical Engineering" is a finishing rate in preparation of the experts under the program "Mechanical Engineering". For its study the knowledge of disciplines "Processing of materials", "Resistance of materials" is required.

The discipline "Technology of mechanical engineering" is studied on the senior rate for the bachelor level in 7 and 8 semesters and is divided into two parts. The part "Fixing of accuracy in mechanical engineering and bases of the theory of cutting tools” are studied in the seventh semester (3 credit.) The part second "Technological opportunities of machine tools and designing of technological processes" is studied in the eighth semester (3 credit.) At the end of each semester the examination in the appropriate parts is stipulated.

This textbook provides the most comprehensive introduction to technology of mechanical engineering. Measurements throughout the textbook are given according to the SI Metric system of measurement. The content is generously illustrated, and the language used is simple and direct.

This text book is written on the basis of the book of Victor E. Repp and Willard J. McCarthy “Machine Tool Technology”. The author of this textbook expresses gratitude to the authors of the book for the given opportunity to use its material.

Suggestions on improvements of future editions of the textbook are welcome.

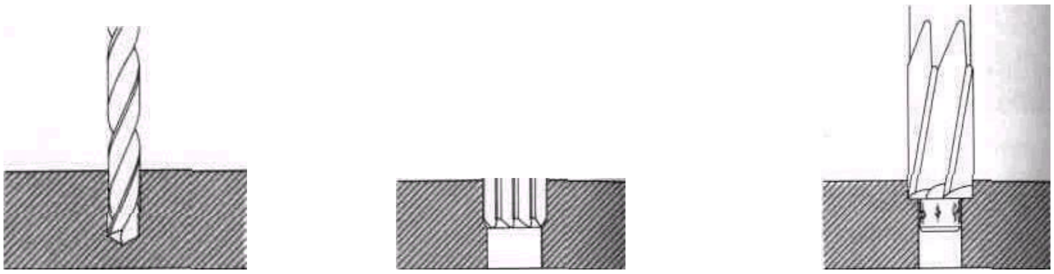
Welcome to the textbook.

Part 2. Technological Opportunities of Machine Tools and Designing of Technological Processes

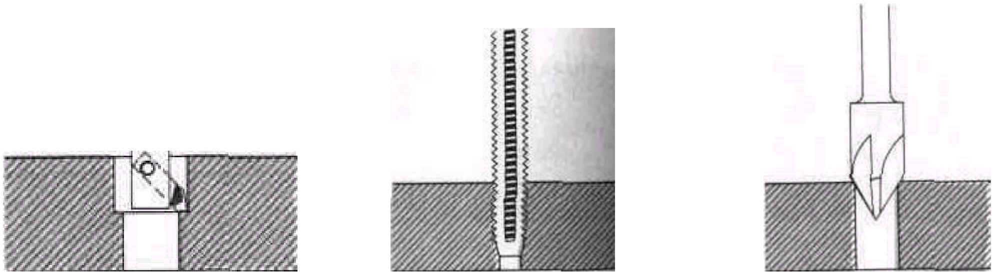
Chapter 18. Drills and Drilling Machine Operations

18.1. The Drill Press

The drill press is one of the most important metalworking machine tools. In addition to its principal task of drilling holes, the drill press is widely used for hole-machining operations such as reaming, boring, counterboring, countersinking, and tapping, Fig. 18-1. The drill press also may be used for honing, lapping, sport-finishing.



Drilling Reaming Counterboring



Boring Tapping Countersinkin

Fig. 18-1. Six common operations that can be performed on a drill press.

The floor and bench model drill presses are sensitive drilling machines. They are equipped with a hand feed which enables the operator to feel the progress of the cutting tool. This type machine also may be equipped with power feed. Othen they are arranged in a line to make up a gang drilling machine for mass-production purposes where a number of drilling operations must be performed in a certain sequence.

The principal parts of a drilling machine, as shown in Fig. 18-2, include the following: base and column, spindle, motor and head, table, feed mechanism, and quill. Other important parts include a drill chuck, on-off switch, table-raising crank, table lock or clamp, and a depth stop.

Variation of spindle **speed** is accomplished through several types of drive systems. One common method involves the **step-pulley drive** with a V-belt. A second drive system is the **variable-speed drive**. With this system, the speed is infinitely variable throughout each speed range. The drill press in Fig. 18-1 has a variable-speed countershaft drive. A third method for variation of spindle speed involves a **gear drive system**. Here speed-selector handles are used to shift the gears for the desired spindle rpm. Usually the machine must be stopped while speed changes are made. This type of drive system normally is used on heavy-duty drilling machines.

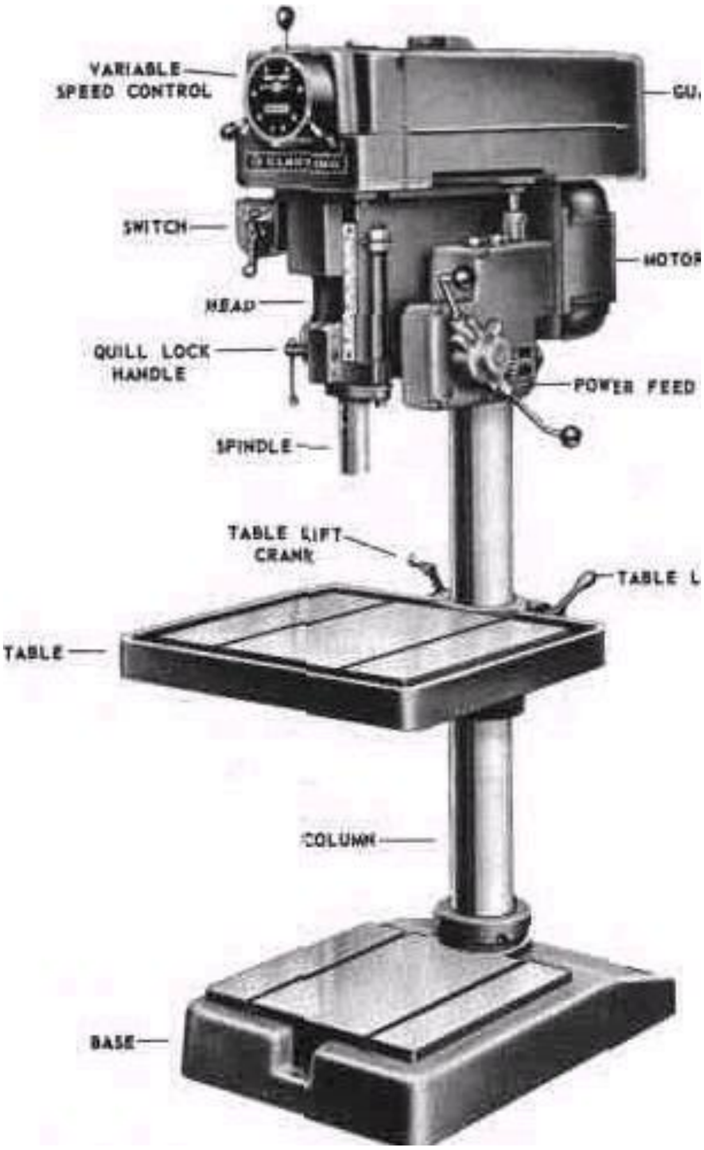


Fig. 18-2. Floor model of a sensitive drill press.

A two-speed motor may be used in conjunction with any of the above drive systems. This conveniently provides both a low- and a high-speed range for the spindle. The speed ranges on drill presses may vary from about 30 to 5000 rpm. Correct spindle speed is determined mainly by the kind and size of tool being used and the kind of material being cut.

All drill presses are equipped with a hand feed. In addition, many are equipped with an automatic feed. Mechanical automatic feeds usually are set at a certain feed per revolution of the drill press spindle. The range of feeds varies from 0.05 to 0.64 mm per revolution.

An air-hydraulic feed mechanism may be set at infinitely variable feeds from 12.7 to 1524 mm per minute. It operates from air pressure and may be either automatic or semiautomatic in operation. Liquid hydraulic feed mechanisms also are widely used on large production drill presses.

Several attachments are available for use on drill presses. The universal compound vise is useful in clamping work securely for many drilling operations. The attachment should be bolted to the drill press table. With slides providing movement in both directions, workpieces can be easily and accurately positioned.

Figure 18-3 shows a small multiple spindle drilling head arranged for drilling two vertical holes at the same time. A drilling jig is normally used with this type drilling head to assure accurate hole location. A portable drilling head positioned for drilling a horizontal hole at the same time the vertical holes are being drilled also are available. Several styles of multiple spindle drilling heads are made, the largest being capable of drilling 30 or more holes simultaneously.

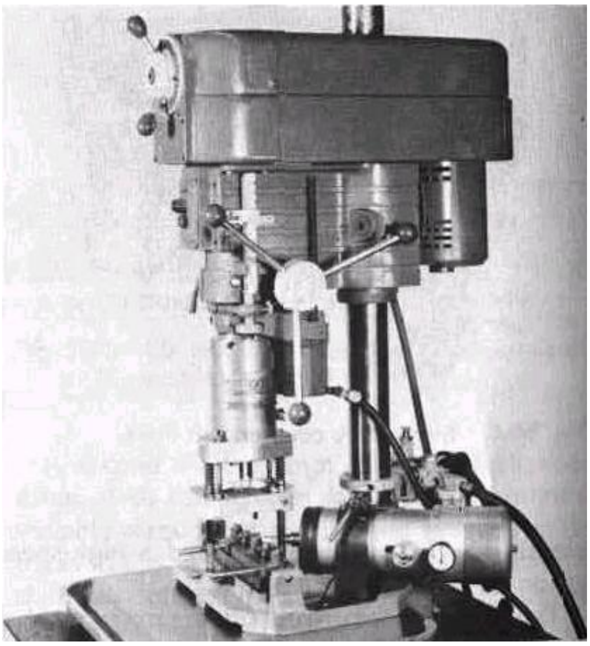


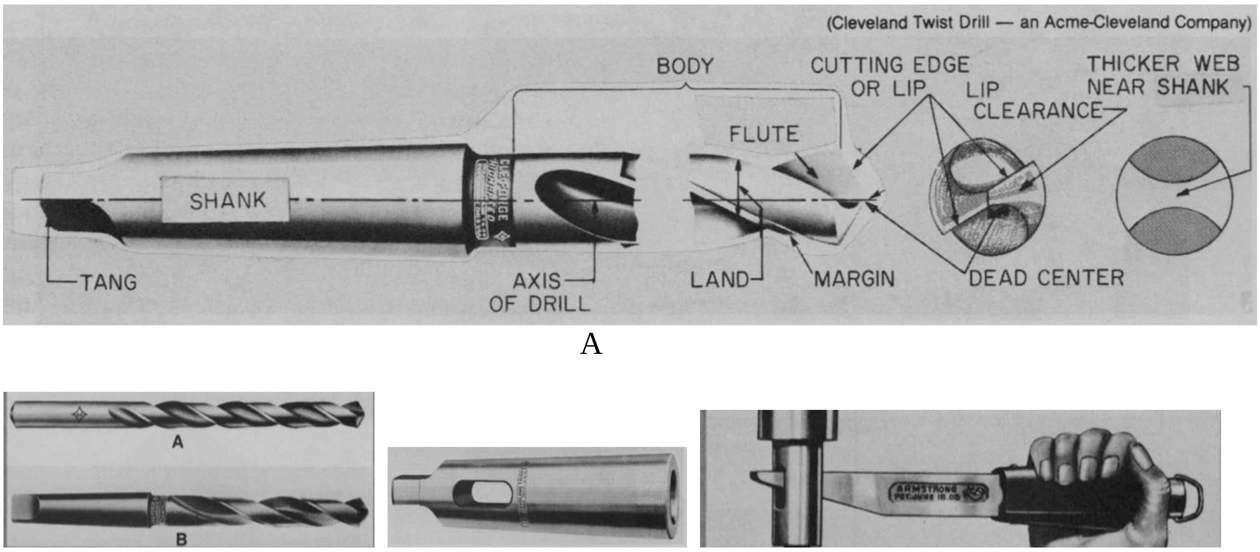
Fig. 18-3. Multiple-spindle drill head mounted on single-spindle drill press.

Usually, the size of a drill press is given in terms of the distance from the column to the center of the spindle. A 400 mm drill press will drill a hole in the center of a circle 400 mm in diameter. The vertical capacity of the machine is determined by the distance from the table (in its lowest position) to the bottom of the jaws of the chuck when fully elevated, less the amount the drill projects. Other factors determining capacity include the distance of quill travel and the size of drill which the spindle or chuck will accommodate. Drill presses are made in many sizes and for many special purposes. In Russia usually the size of a drill press is given in terms of the maximum diameter of hole which can be drilled in the steel 45 (1112 steel.) A drill press 2135 can drill the hole 35 mm diameter.

18.2. Drills

Drills are used for cutting holes into or through material. There are many kinds of drills. However, twist drills are by far the most commonly used.

Generally speaking, a drill has three principal parts: the point or dead center, the body, and the shank. Fig. 18-4. The spiral grooves that wind around the body of the drill are called flutes. They provide a means whereby (1) a suitable lip or cutting edge may be formed on the point of the bit; (2) the chip removed by the cutting lip may be carried by a channel to the surface; and (3) a lubricant can be carried easily to the cutting edge. The body surface between the flutes is known as the land. The narrow strip of metal, labeled margin, is formed by grinding away some of the land to give the drill body clearance.


(Cleveland Twist Drill — an Acme-Cleveland Company)

A

B C D

Fig.18-4. Parts of a drill (A), types of drill shank (B), sleeve or shell socket (C), drill drift (D)

The thin wall between the flutes is called the web. It is this part that gives rigidity and strength to the drill. As the web approaches the shank, it thickens. Cutting the flutes somewhat shallower but slightly wider permits free passage of the chips.

The shank (that part of the drill which fits the spindle or chuck of the drill press) varies in shape according to its size or the purpose for which it was designed. An ordinary **straight shank** (which is clamped with a chuck) and a **taper-shank** drills are available . Taper-shank drills have standard Morse tapers and will fit the spindles of standard drill presses or auxiliary sleeves. The tang on shank fits a slot in the spindle to prevent the drill from slipping or turning in the spindle.

Morse-taper shanks are standard on taper-shank drills. They also are used on a variety of other tools such as reamers, milling cutters, counterbores, and spot-facing tools. Morse tapers are made in various sizes ranging from Nos. 0 through 7. The No. 2, 3, and 4 tapers are used most commonly on drills from 9.5 mm to 37.5 mm diameter.

Since a 12.7 mm drill with a No. 2 Morse-taper shank will not fit a drill press spindle with a larger No. 3 or 4 taper hole, a reducing fitting called a **sleeve** or **shell socket** must be used. These sleeves are available in several standard sizes. The sleeve is placed over the drill shank and tapped lightly with a hammer. It fits the taper shank securely because of a close friction fit. The drill together with its sleeve then is mounted in the drill press spindle by tapping the end of the drill, lightly with a lead hammer for a snug fit.

To remove the drill from the spindle or from the sleeve, a drill drift is used. The drift is inserted in the hole in the sleeve and tapped lightly with a hammer, thus forcing the taper shank to separate from the taper hole.

Fitted sockets are used to adapt tool shanks to fit into machine sockets which they otherwise would not fit. Fitted sockets are available with a variety of standard external and internal tapers so that a large shank may be adapted to a small hole or vice versa. (They also can be used simply to extend the reach of a drill.)

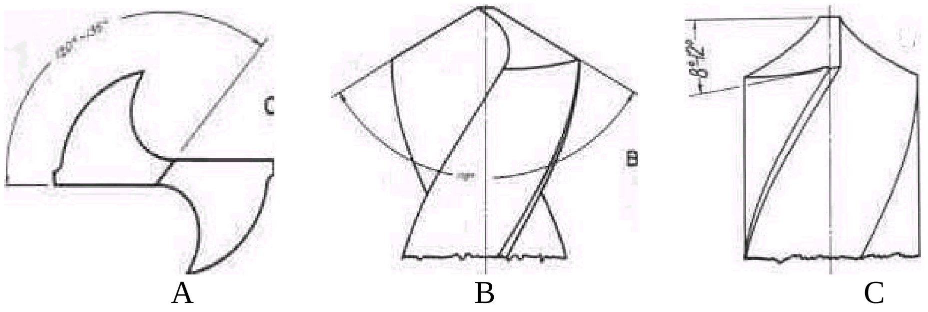


Fig.18-5. Correct angles for standard general-purpose drills.
A. Chisel edge angle; B. Point angle ; C. Lip clearance.

Drill sizes are indicated in four ways: (1) ***in millimeters***, (2) ***by number***, (3) ***by letter***, and (4) ***by fractional parts of an inch***. Metric drill sizes range from 0.1499 mm (0.0059") to 100 mm (4"). Drill sizes by number are given in terms of wire gage, and they range from No. 80, which has a diameter of 0.0135", up to No. 1 with a diameter of 0.2280". The series continues with lettered sizes from A up to Z. An **A** drill has a diameter of 0.2340" and **Z**, a diameter of 0.4130". Drill sizes given in fractional parts of an inch are available as small as 1/64" in diameter to 3-1/2" or larger on request. Drill sizes designated in fractions of an inch increase in size by 1/64" up to 3", after which stock drills increase by 1/32" or 1/16".

Most of the difficulties encountered in drilling may be attributed to improper grinding. When grinding, three factors are important: (1) correct chisel edge angle as at A, Fig. 18-5; (2) correct point angle as at B; (3) correct lip clearance as at C. Correct **lip clearance** for standard general-purpose drills used for drilling most steels is 8° to 12°. Within this range, provide smaller drills with greater clearance than larger drills. Clearance may be increased 50% when drilling soft or free-machining material. Lip clearance permits the cutting edge of the lip to penetrate the workpiece. A drill with no lip clearance will not cut (the cutting lip and the heel are at the same level horizontally) because the cutting lips cannot penetrate the workpiece.

The correct **point angle** for general-purpose drills used for drilling most machine steels and many other metals is 59° measured to the center line of the drill, B, Fig. 18-5. Thus, the total included angle of the drill point is 118°. To check the lip angle, use a drill point gage. An angle of 59° is recommended because drills ground at that angle cut more rapidly and with less exertion of power than when ground at any other angle. The **chisel edge** of a drill should have an angle of 120° to 135° and be centered exactly in line with the center of the drill, A, Fig. 18-5.

When grinding drills for drilling hard materials such as manganese steel, the lip angle should be 75°; total point angle of 150°. This material is very hard and tough.

Consequently, the shorter lip secured when ground at the flatter angle takes less power to operate and does not cause as great a strain on the drill.

When drills are ground for use in drilling softer materials such as bakelite, hard rubber, molded plastics, fiber, and wood, the lip angle should be ground to an angle of 45°, a total angle of 90°.

A drill with a standard 118° point is used for drilling brass and bronze. Drills with standard rakes ***tend to screw themselves*** into free machining brass and some plastics, sometimes breaking the drill, the material, or both. This can be avoided if the drills are reground to provide a neutral or slightly negative rake. This also is recommended for drilling very hard steel, because it reduces the angle of the cutting edge of the drill. It increases the strength of the cutting edge, thus preventing chipping.

With the conventional drill, a center-drilled hole is required to start the drill. Without the center-drilled hole, the drill has a tendency to **walk** to one side or another, thus drilling a slanted hole or a hole in the wrong location. With the conventional chisel-point drill, jigs and drill bushings normally are required to start and guide the drill most accurately for production setups.

Most two-lipped drills produce holes several hundredths of a millimeter **oversize**, depending on the material being drilled, the accuracy of the ground point, the rigidness of the setup, and whether or not fixtures with bushings are used to guide the drill. With the use of spiral-point drills, the average range of hole oversize is reduced by more than 50 percent.

Spiral-point drills also produce holes which are more accurate in roundness. Where accurate size and shape are required for holes drilled with chisel-point drills, it is common practice to drill the hole slightly undersize, followed by reaming or boring. In many instances, spiral-point drills hold size and shape sufficiently to eliminate the need for reaming and boring.

The **speed** of a drill refers to the rate at which it travels at the circumference. This is called **peripheral** or **outside speed** and is given in terms of meters traveled per minute. The **feed** is the rate at which the drill advances into the work per revolution, measured in hundredths of a millimeter.

Numerous **styles and types of drills** are made (Fig. 18-6). Each type of drill is designed for certain applications. The types of drills used most are the straight-shank and the taper-shank twist drill. Less common types of drills include:

1. Drills **over** 12.7 mm but with a 12.7 mm **shank** are designed for use either in portable electric drills or in drill chucks on smaller drill presses (Fig. 18-6, A). A ***pilot hole should be drilled first***.
2. Straight-shank **drills with carbide tips** are recommended for production drilling of cast iron, cast steel, and nonferrous materials (Fig. 18-6, B). They are not recommended for drilling steel.
3. **Carbide-tipped die drills** (**gun** drills) with the straight-shank type are recommended for use in drilling **long holes** and even for **hardened steel** in the range from 48 to 65 Rockwell-C hardness (HRC 48...65) (Fig. 18-6, C). Holes may

be drilled without annealing the metal. A steady hand feed with a good flow of cutting fluid should be used.

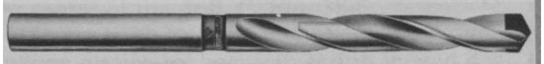
4. **Three-fluted core drills** are used for enlarging cored holes or previously punched or drilled holes (Fig. 18-6, D). Because of their wide use in drilling cored holes in castings, they are known as **core drills**. This type of core drill can enlarge holes as small as 60 percent of the drill diameter. The advantages of a multiflute drill include increased rate of metal removal, increased accuracy in hole size and location, and improved finish. Four-fluted core drills are also made.

5. **Subland drills** are special multicut drills which can drill several diameters in one operation (Fig. 18-6, E). Drilling operations of this type are called **step-drilling operations**. Many variations of step-drilling operations are possible with multicut drills.

6. **Oil hole drills** are used on high-production screw machines (Fig. 18-6, F). Oil is forced through the machine spindle and through the oil holes to the cutting edges. The oil not only serves as a lubricant and coolant, but it also helps force the chips out of the drilled hole.

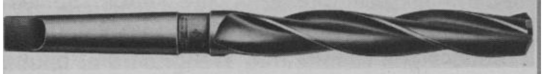
(Cleveland Twist Drill; DoAll Company)

A. Drill with 12.7 mm (1/2″) shank

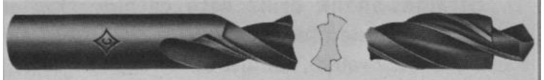


B. Straight-shank drill with carbide tip

C. Carbide-tipped die drill

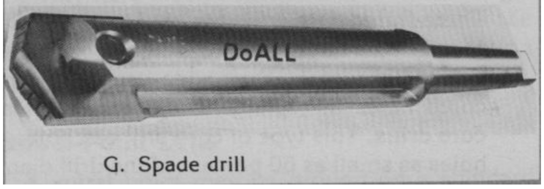


D. Three-fluted core drill



E. Subland drill

F. Oil hole drill



G. Spade drill

Fig. 18-6. Types of drills

7. **Spade drills** are made up of a tool shank and a replaceable spade bit (Fig. 18-6, G). They are most widely used for drilling holes 25 mm to 125 mm in diameter. Straight and taper shank holders are available in several lengths. For deep hole drilling, an oil hole is provided through the shank to allow cutting fluid to be delivered to the drill point under pressure. Spade drills are much less expensive than twist drills of like size.

When a drill of a particular dimension is not available, or when a very straight, accurate hole is desired, a **boring tool** may be used. In such cases, a hole large enough to permit entry of the boring tool is drilled with a standard drill.

The **boring head** is adjustable to within 0.01 mm or less. This type is available in a wide range of sizes. Boring capacity may range from 19 mm to 50 mm offset from center, or a hole capacity of over 100 mm. Adjustable boring heads eliminate the need for a complete inventory of expensive, large-size drills. When used with care on sturdy machines with power feed and rigid setups, extremely accurate holes may be machined with this type of tool.

The **counterboring tool**, Fig. 18-1, is used to spot-face or counterbore for bolts and screws, to enlarge holes to receive the head of fillister head screws and for similar purposes. It has a **pilot** or **guide** on the point which frequently is interchangeable. Ordinarily, the pilot has a diameter about 0.05 mm smaller than that of the drilled hole. Counterboring tools are made in many sizes. They are available in high-speed steel and with carbide insert teeth. Counterboring tools should be run at lower cutting speeds than a drill of corresponding diameter. A cutting fluid should be used freely.

Countersinks, Fig. 18-1, are used to machine a cone-shaped enlargement at the end of a hole. They are made in many styles and sizes, and with point angles of 60° for ***lathe centers***, 82° for flat-headed machine screws, 90° for ***deburring***, and 100, 110, and 120 degrees for some types of ***rivets***. Other angles are made for special applications. There are six types of countersinks: one-, three-, and four-flute; add-on; chatter-less; and piloted. Other types are made for special applications. The single-flute is essentially chatter-free but is limited to relatively shallow countersinking. The chatterless style provides additional cutting edges offset from the main cutting edges, which effectively defeats the tendency to chatter. Add-on countersinks are often used for countersinking on the back side of a surface which would be inaccessible otherwise. Piloted countersinks help attain close concentricity between the hole and the countersink. For best results, countersinks should be run at 1/2 to 2/3 the rpm of drills of like size.

Combination drill and countersinks, also called **center drills**, Fig. 18-7 are used for two main purposes: (1) for providing a starting hole for drills, and (2) for drilling center holes in stock to be held in a lathe or cylindrical grinder. They are made in many sizes., and of either high carbon or high-speed steel. The plain style has a single 60° countersink angle. The bell style has a secondary angle of 120°. The hole made by the bell style is desirable because it protects the edges of the 60° center hole from being nicked during handling, thus preserving the accuracy of the workpiece.

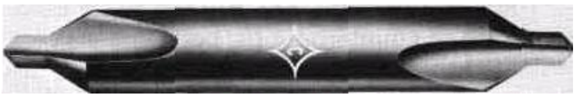


Fig. 18-7. Combined drill and countersinks.

18.3. Reamers

Reamers are multiple cutting edge tools used to enlarge and finish a drilled hole to exact dimension and smoothness, Figs. 18-1, 18-8 and 17-10. Drilled holes usually have a rough finish and are inaccurate in size and roundness. Drilled holes are usually satisfactory for bolts and screws, however, where round, straight, accurately sized holes with a smooth finish are required for use with bushings or bearings, the holes often are reamed, either by hand or by machine.

Reamers may be made of carbon tool steel, high-speed steel, or may have carbide-tipped cutting edges. They are available in a wide variety of types and sizes.

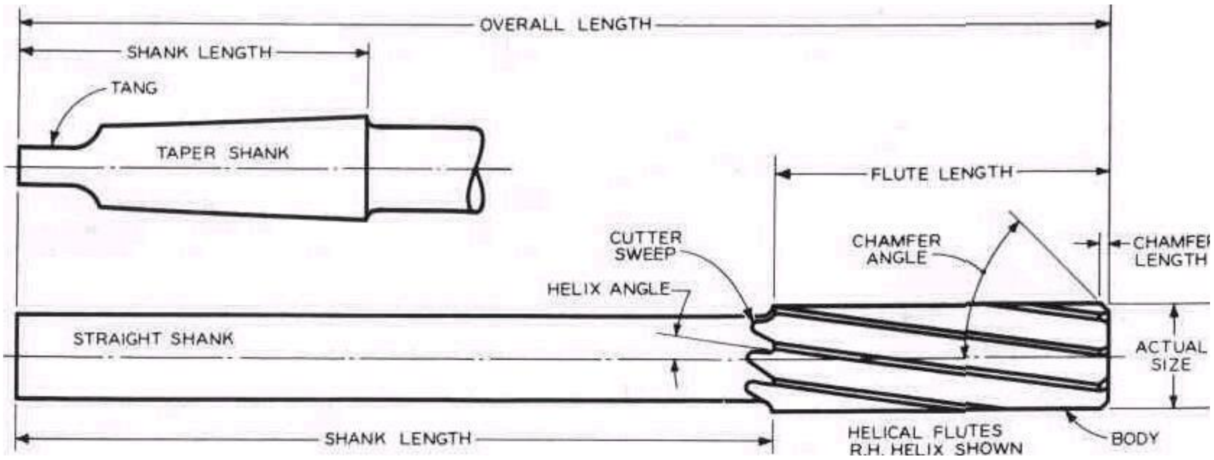


Fig. 18-8. Terms applying to reamers.

Reamers may be classified into two general groups: **machine reamers**, Fig. 18-9; and **hand reamers**, Fig. 18-10. Hand reamers are designed for hand operation and light cuts. They are equipped with a straight shank having a square tang which is turned with a tap wrench. Machine reamers are designed for use on drill presses, engine lathes, turret lathes, vertical milling machines, and other special production machines.

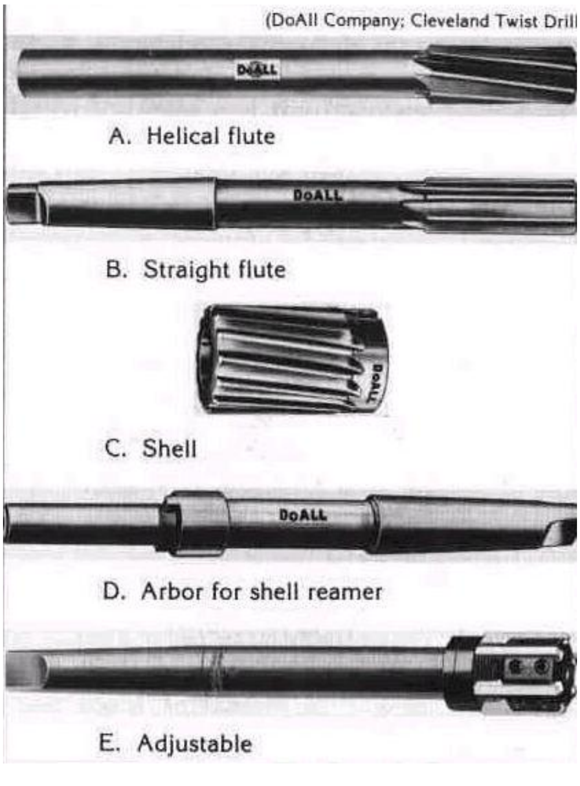

(DoAll Company; Cleveland Twist Drill

Fig. 18-9. Types of machine reamers

Machine reamers are made with either Morse taper shanks or straight shanks, Fig. 18-9. Machine reamers have a bevel of from 40° to 50° (usually 45°) on the cutting end. They cut differently than hand reamers. Machine reamers are end-cutting reamers which cut on the beveled end in the same manner as a drill cuts as it enters a hole. When the reamer becomes dull, it is resharpened on the 45° beveled end only. It must be sharpened on a tool-and-cutter grinder so that all cutting edges are exactly even.

Hand reamers, on the other hand, cut on the periphery of the reamer. The flutes are ground straight for the whole length, except near the cutting end which is ground with a starting taper, Fig. 18-9. This taper permits the reamer to enter the hole easily. Thus, hand reamers are essentially scraping tools, and nearly all cutting takes place along the starting taper at the cutting end of the reamer. Hence, the hand reamer is designed for removing only very small amounts of material. When the hand reamer becomes dull, it must be resharpened on the starting taper portion only. Like a machine reamer, it must be sharpened on a tool-and-cutter grinder.

Types of Machine Reamers. Rose reamers are intentionally designed to rough-ream cored or drilled holes 0.07 to 0.25 mm undersize. They are designed **for two-step reaming operations**. After rough reaming, the hole is finish-reamed to final size with a fluted reamer.

Rose reamers have a 45° end-cutting angle which does all of the cutting and which is capable of cutting rapidly. The reamer also has a back taper; that is, it is tapered a maximum of 0.025 mm per 25 mm of length for the length of the flutes. The back taper prevents binding in deep holes with heavy cuts. The out-side diameter of the reamer is ground cylindrically, thus leaving a wide circular margin. There are no cutting edges along the flutes, which merely provide space for cutting fluid and chip ejection.

Fluted reamers generally have more flutes than rose reamers and are designed for reaming holes to finished size. They may be used for one-step reaming operations or after using a rose reamer in a two-step operation.

The cutting action takes place at the 45° beveled cutting end. A narrow circular margin, from 0.12 to 0.50 mm in width, runs along the entire length of the flute. The lands are slightly beveled along the length of the flute to provide a body clearance angle. The reamer is not provided with significant back taper.

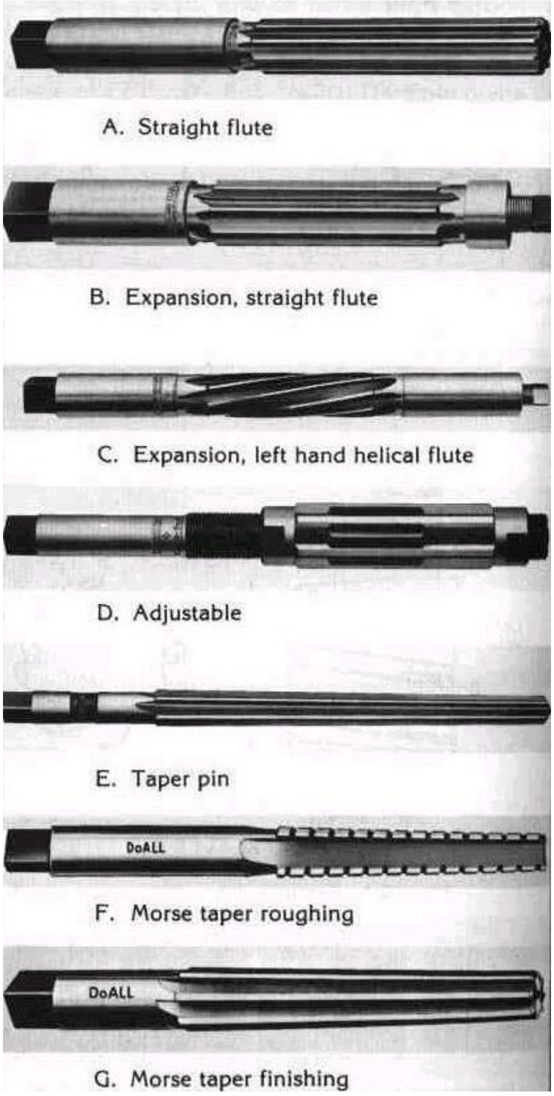


Fig. 18-10. Types of hand reamers.

Several common types of machine reamers are shown in Fig. 18-9. The **helical fluted reamer** has a straight shank and is designed for reaming materials which ordinarily are considered difficult to ream. The helical flutes provide a free-cutting action which aids in producing smooth, accurate holes.

Straight fluted machine reamers are designed for use with materials which possess average reaming properties.

Shell reamers are made with helical flutes or with straight flutes. They are also made in the rose type or the fluted type. Shell reamers are designed with the hole

for economy reasons. The hole is tapered to fit snugly and accurately on a special arbor, which is available with either a straight or taper shank. Several sizes of shell reamers may fit the same arbor. When the reamer is worn out, it may be discarded, and a new reamer may be used on the old arbor.

Adjustable machine reamers, as shown at E in Fig. 18-9, are made with either high-speed steel blades or carbide-tipped high-speed steel blades, and with either straight or taper shanks. These reamers are easily adjusted for size within a range of about 1 mm diameter. As the blades become dull, they can be expanded and reground several times. When worn out, the blades may be replaced.

Carbide tipped reamers are more abrasion-resistant and can withstand higher temperatures and higher cutting speeds than high-speed steel reamers. They are particularly useful for reaming castings, both ferrous and nonferrous, which have sand or scale inclusions.

Special machine reamers of many types and sizes are available for special purposes. One important type is the combination drill and reamer, which has a drill at the end and a reamer farther back. This design makes it possible to drill and ream a hole in one operation.

Straight-fluted hand reamers of the solid type are made of either carbon tool steel or high-speed steel, Fig. 18-10 A. The cutting end is ground with a starting taper for easy entry into a hole. This type reamer is recommended for general-purpose reaming of holes to finished size. **Helical-fluted hand reamers** are recommended for reaming holes with interruptions or keyways. The helical flute produces a smooth cutting action with minimum chatter.

Expansion hand reamers may be of the straight-flute type, Fig. 18-10 B, or the helical-fluted type, Fig. 18-10 C. The amount of expansion possible depends on the diameter of the reamer. It may vary from about 0.15 mm to about 0.3 mm. These reamers are provided with an adjusting screw for expansion. An undersize pilot is provided on the end to aid in alignment.

Adjustable hand reamers, Fig. 18-10 D, may be adjusted for any size, above or below basic size, within the range of the reamer. These reamers are available in standard sizes from 6.35 mm to about 85 mm diameter. Each reamer may be expanded to the smallest size of the next-size reamer. Blades are available in carbon steel or highspeed steel. They slide in accurately tapered slots and may be adjusted by loosening one nut and tightening the other, thus moving the blades in the slots.

Taper pin reamers, Fig. 18-10 E, are used for reaming holes for standard taper pins. Best results are achieved when the drilled hole is slightly larger than the small end of the taper pin. These reamers are made with straight- or left-hand helical flutes, and in carbon steel or highspeed steel.

Taper socket reamers are made for reaming standard Morse or Brown and Sharpe taper holes. Straight-shank hand reamers are intended for maintenance of taper holes in machine tools and accessories, Fig. 18-10 F and G. Taper-shank roughing reamers and finishing reamers are made for production reaming of standard taper holes.

Burring reamers are used for removing burrs from cut pipe and conduit. They also may be used for enlarging holes in thin materials.

Reaming speeds may vary considerably, depending on the type of material to be reamed, the type of machine used, the type of finish required, and the accuracy required. As a general rule, machine reaming is done between half and two-thirds the speed used for drilling the same material.

Reaming feeds are two to three times higher than for drilling. If the feed is too slow, excessive reamer wear will result. If the feed is too fast, the hole will be inaccurate. The feed should be sufficient to cause each flute to cut a chip rather than burnish or rub the material. A good starting point is to use a feed from 0.04 to 0.10 mm per flute per revolution. Then the feed can be adjusted as required for desired results.

Reamer alignment is one of the most important factors in reaming accuracy. When reaming, the spindle, reamer, reamer bushing, and the hole to be reamed should be in perfect alignment. Any variation in these factors detracts from reaming accuracy and results in excessive reamer wear. The effects of reamer misalignment may be reduced through the use of a **floating reamer holder** which allows the reamer to be self-centering. When possible, holes should be drilled and reamed in the same setup without moving the workpiece.

Chatter can affect the accuracy and finish of reamed holes. The following are possible causes for chatter: excessive speed; too light a feed; setup not rigid; spindle too loose; excessive clearance on reamer; excessive looseness in floating holder.

Cutting fluids aid in producing good finishes when reaming. Mineral-lard oils and sulfurized oils are desirable for most reaming applications. Gray cast iron generally should be reamed dry, or the reamer may be cooled with a jet of compressed air.

The usual procedure for producing a reamed hole is to produce an undersize hole first. The hole may be produced in one of three ways: drilling, drilling with three-fluted core drill or boring. Holes for machine reaming should be produced undersize an amount which will provide the following **material allowances** ($2z$): for 6 - 12 mm holes: $2z = 0.2 - 0.4$ mm; for 12 - 25 mm holes: $2z = 0.4 - 0.5$ mm; for 25 - 50 mm holes: $2z = 0.5 - 0.8$ mm; for 50 - 80 mm holes: $2z = 0.8 - 1.2$ mm.

Where extreme accuracy is required, hand reaming often is required. A cut of 0.05 mm usually is recommended. Never leave over 0.3 mm of material for hand reaming. Never reverse a reamer or the workpiece when reaming. To do so will cause chips to damage the margins of the reamer, thus causing an inaccurate hole with a poor-quality finish.

18.4. Cutting Speeds and Feeds for Drilling

Cutting speed for drills and other rotating tools is expressed in terms of meters per minute (mpm). It is the distance that a point on the circumference of the drill will travel in one minute. If the cutting speed for drilling is too high, the drill will become overheated and will dull easily. If the cutting speed is too low, the production rate will be low and the drill may break easily.

There is no one correct speed for drilling all materials. First of all, it is necessary to select cutting feed in accordance with the drill diameter, material drilled and its hardness looking at the handbook table for machinists. The feed for drilling refers to the rate at which the drill advances into the work in one revolution. Hence, with a feed setting of 0.10 mmpr, the drill advances into the work 0.10 mm deeper each revolution.

Feeds are governed by the size of the drill and the material to be drilled. The general rule when drilling mild steel is to use a feed of 0.02 to 0.06 mm per revolution for drills smaller than 3 mm; 0.06 to 0.10 mm for drills 3 mm to 6 mm ; 0.10 to 0.2 mm for drills 6 to 12 mm; 0.2 to 0.4 mm for drills 12 to 25 mm; 0.4 to 0.6 mm for drills larger than 25 mm. Alloy and hard steels should be drilled with a lighter feed than given, while cast iron, brass, and aluminum usually may be drilled with a heavier feed.

It is common practice to select an average cutting speed for each type of material. The cutting speed selected may then be increased or decreased according to conditions which affect the particular job setup and the material drilled. The following factors affect the cutting speed selected for drilling operations:

1. The kind of material being drilled. Softer materials generally are drilled at higher cutting speeds than harder materials. See Table 18-1.
2. The kind of cutting tool material. Carbon-steel drills are used with cutting speeds which are about one-half those used with high-speed steel drills. Tungsten-carbide drills generally are run at higher speeds and lighter feeds than high-speed steel drills.
3. Whether or not a cutting fluid is used.
4. The size and the type of drilling machine used, and the rigidness of the work setup.
5. The quality of finish desired in the hole.

Extreme speed or feed will cause drills to chip or break at the cutting edges or to split the web. Similar damage also may result from improper grinding. **Rapid wearing at the outer corners of the cutting edges** usually is an indication of too much speed, the colour of steel chip becomes blue.

Table 18-1. Cutting Speeds for High-Speed Steel Drills

Material	Meters Per Minute	Feet Per Minute
Low-carbon steel (0.05-0.30% carbon)	24.4-33.5	80-110
Medium-carbon steel (0.30-0.60% carbon)	21.4-24.4	70-80
High-carbon steel (0.60-1.70% carbon)	15.2-18.3	50-60
Steel forgings	15.2-18.3	50-60
Alloy steel	15.2-21.4	50-70
Stainless steel	9.1-12.2	30-40
Cast iron, soft gray	30.5-45.7	100-150
Cast iron, hard-chilled	21.4-30.5	70-100
Cast iron, malleable	24.4-27.4	80-90
Ordinary brass and bronze	61.0-91.4	200-300
High-tensile bronze	21.4-45.7	70-150
Monel metal	12.2-15.2	40-50
Aluminum and its alloys	61.0-91.4	200-300
Magnesium and its alloys	76.2-122.0	250-400
Slate, marble,and stone	4.6-7.6	15-25
Bakelite and similar plastics	30.5-45.7	100-150
Wood	91.4-122.0	300-400

Carbon-steel drills should be run at speeds of from 40% to 50% of those given above. Carbide-tipped or solid-carbide drills may be run two to three times faster than high-speed steel drills.

Obtaining the recommended cutting speed from a table or chart completes the data needed to calculate the correct rpm:

$$n = 1000V/\pi D, \qquad (18.1)$$

where: n - rotation frequency of spindle, revolution per minute (rpm); V - cutting speed, mpm; D - diameter of a drill or a reamer, mm; $\pi \approx 3.14$.

The cutting speed can be calculated with the help of empirical formulas for drilling (t = 0.5D):

$$V = \frac{C_V \cdot D^q}{T^m \cdot s^y} \cdot K_V\ , \qquad (18.2)$$

and for enlarging cored holes [t = 0.5(D-d)]:

$$V = \frac{C_V \cdot D^q}{T^m \cdot t^x \cdot s^y} \cdot K_V\,, \qquad (18.3)$$

where: V - cutting speed, mpm; C_V - factor taking into account the kind of material being cut, the kind of processing, the material of the cutting tool, the range of cutting feed, cutting fluid; T - value of tool life, minutes; t - depth of cut, mm; s - cutting feed, mmpr; m, x, y - exponents of degrees which are taking into account the kind of material being cut, the kind of processing, the material of the cutting tool, the range of cutting feed; K_V - product of factors which are taking into account influence of a material being cut, depth of cut, material of the cutting tool.

Further information concerning factors and exponents of degrees is included in standard handbooks for machinists.

The **torsional moment of cutting M** and **axial force P_a** can be calculated with the help of empirical formulas for drilling:

$$M = 10 \cdot C_M \cdot D^q \cdot s^y \cdot K_P \ , \qquad (18.4)$$

$$P_a = 10 \cdot C_P \cdot D^q \cdot s^y \cdot K_P \ , \qquad (18.5)$$

and for enlarging cored holes [t = 0.5(D-d)]:

$$M = 10 \cdot C_M \cdot D^q \cdot s^y \cdot t^x \cdot K_P \ , \qquad (18.6)$$

$$P_a = 10 \cdot C_P \cdot t^x \cdot s^y \cdot K_P \ , \qquad (18.7)$$

where: M - torsional moment of cutting, Nm; P_a - axial cutting force, N; C_P - factor taking into account the kind of material being cut, the material of the cutting tool; t - depth of cut, mm; s - cutting feed, mmpr; q, y, x - exponents of degrees which are taking into account the kind of material being cut, the material of the cutting tool; K_P - factor which is taking into account influence of the material being cut.

Further information concerning factors and exponents of degrees is included in standard handbooks for machinists.

Cutting power is also calculated:

$$W = \frac{M \cdot n}{9.75} , \qquad (18.8)$$

where: W - cutting power, Watt; M - torsional moment of cutting, Nm; n - rotation frequency of spindle, rpm.

The above formulas also can be used to calculate the cutting speed and rpm for all types of cylindrical cutting tools, including taps, reamers, counterboring tools, and milling cutters. In addition, they may be used to calculate cutting speeds for turning or boring on a lathe. However, in this case, the diameter (D) refers to the diameter of the hole being bored.

To maintain the feeds and speeds recommended, it is necessary to use good **cutting fluids**. The following are recommended for drilling operations:

1. Hard refractory steel – turpentine, kerosene, emulsifiable oils (soluble oils), mineral-lard oils.
2. Soft steel and wrought iron – emulsifiable oils (soluble oils), sulfurized oils, mineral-lard oils.
3. Malleable iron – emulsifiable oils, mineral-lard oils.
4. Brass and bronze – emulsifiable oils, or dry.
5. Aluminum and aluminum alloys – mineral oils, emulsifiable oils, mineral-lard oils.
6. Gray cast iron – dry or with a jet of compressed air.

The selection of cutting fluid also varies with the machinability of the material and the severity of the operation being performed.

18.5. Safety with Drilling Operations

1. Always wear appropriate safety goggles.
2. Never leave the chuck key in the drill chuck or the drill drift in the spindle.
3. Mount the work securely before drilling. Do not hold thin or small pieces in the hands.
4. Remove chips with a brush or piece of wood. Never use the hands.
5. Ease up on the feed pressure as the drill begins to break through a hole. This will prevent the drill from catching or breaking, or pulling the work loose.
6. Do not attempt to stop the spindle with the hands after turning the machine off.
7. Never drill copper alloys, including brass and bronze, with a drill which is ground for steel. Request the instructor to show you how to grind the drill for this purpose. A drill ground for steel may dig in, break, ruin your work, or cause injury.
8. Keep long sleeves, other loose clothing, and especially long hair away from the revolving spindle or belts.

8. Do not operate the drill with the covers or guards removed.

Chapter 19. The Metalworking Lathe

The metalworking lathe is the most basic of all the metalworking machine tools. Lathes are built primarily for making cylindrical and conical parts such as pins, bolts, shafts, discs, pulleys gear blanks, and for boring holes more precisely or larger than can be made by drilling and reaming. Some lathes can be equipped with attachments which enable them to perform milling, grinding, and broaching operations. A well-constructed lathe, when properly operated, can produce work accurate to 0.01 mm or less (grade of tolerance 6 -7.)

Lathes are made in a wide variety of types and sizes, from the small precision lathe found in watch repair shops to the immense machines used in manufacturing big guns or mill rolls. A heavy-duty lathe with a geared-head drive is used for the duty operations.

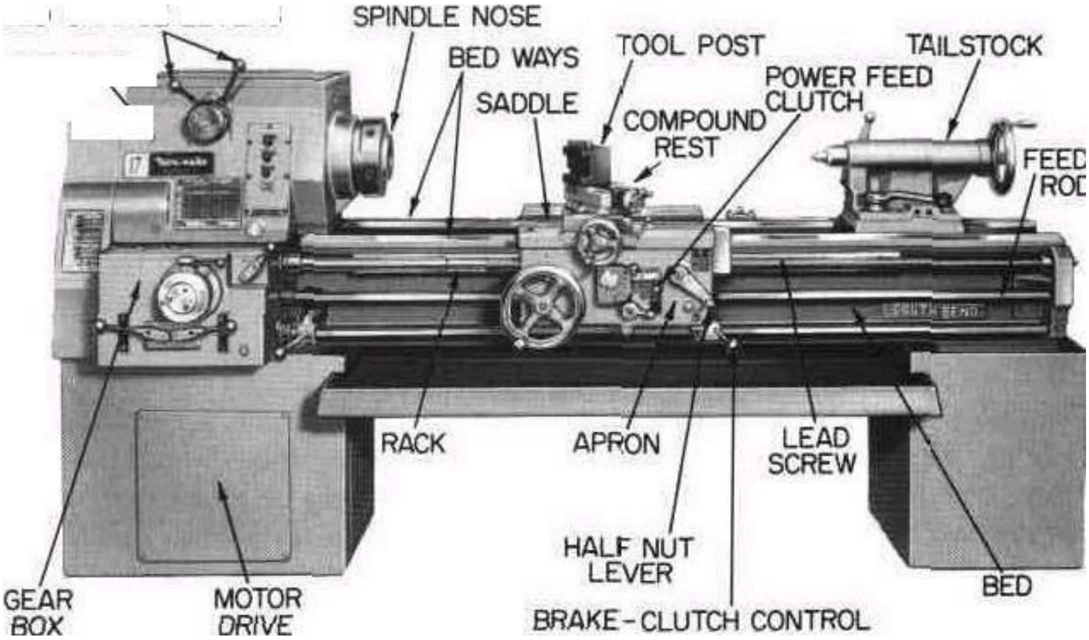


Fig. 19-1. Parts of a lathe.

Modern lathes perform basically the same operations which were performed on lathes at the turn of this century. However, because of improved design features and improved cutting tools, they accomplish these operations more accurately and efficiently today. Without the metal-working lathe, modern industrial machines and equipment could not be produced.

19.1. Lathe Accessories

Numerous accessories are required for various machining operations on a lathe. Several standard accessories needed for beginning lathe operations are described in this unit. Other accessories are described in those units dealing with lathe operations for which they are commonly used.

Two **lathe centers** are required for turning work between centers on a lathe. The centers have a Morse-taper shank which fits into the tapered hole in the tailstock and the tapered hole in the spindle. Centers which are too small to fit the tapered hole in the headstock spindle can be adapted to fit by the use of a **spindle sleeve**.

The center in the headstock rotates with the spindle and the work, and, therefore, is called the **live center**; that in the tailstock is stationary and is called the **dead center**. The center in the tailstock always must be made of hardened steel or carbide.

That in the headstock may be either hardened or unhardened steel. Hard lathe centers are made of either carbon tool steel or high-speed steel, or they may be equipped with carbide tips which are highly wear- and heat-resistant.

Center holes are drilled into the workpiece, thus permitting the workpiece to be supported between the centers. The center hole in which the dead center is inserted must be lubricated to prevent it from becoming overheated and scored or burned. A mixture of white lead or red lead and oil is a satisfactory lubricant.

Some manufacturers make **rotating live centers** for the tailstock. Such centers are equipped with ball bearings or tapered roller bearings which permit the center to rotate with the work. It is not necessary to lubricate the center holes when mounting work on centers of this type.

When hard lathe centers become scored, nicked, or damaged, they must be resharpened by grinding. They are ground with a tool post grinder mounted on the compound rest on a lathe. The point angle of the lathe center is 60° and is checked with a center gage.

When a workpiece is mounted between centers on a lathe, it is driven with a **lathe dog**, Fig. 19-9.

Other lathe accessories include toolholders, cutting tools, steady rest, follower rest, knurling tools, chucks, faceplates, types of spindle noses, boring tools, mandrels, taper attachments, and milling attachments.

Lathe chucks and **faceplates** are devices for mounting and holding work while it is being machined on a lathe. When work is mounted on a faceplate, it usually is clamped with bolts and metal straps. The plate is equipped with slotted holes to allow for work of varying size and shape. Figures 19-2 and 19-3 show work mounted ready for machining.

Chucks hold work by means of **jaws** which may be adjusted to accommodate work of varying size or shape. The jaws of the **four-jaw independent chuck**, Fig. 19-4B, move independently. This enables the chuck to hold work which is round, square, rectangular, or irregular in shape. As pictured, the jaws are installed for holding work of fairly large diameter. They should be removed and reversed for holding work of small diameter.

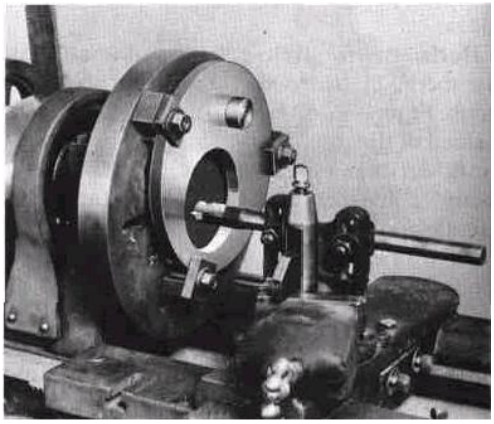


Fig. 19-2. Work mounted on faceplate with metal straps.

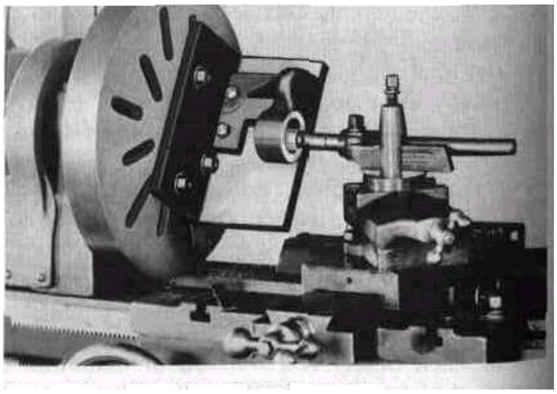
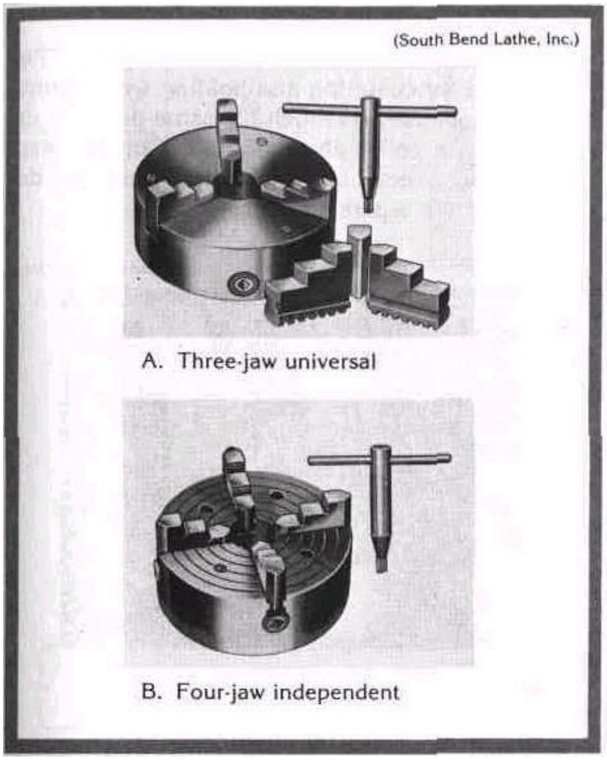


Fig. 19-3. Work bolted on faceplate.

All jaws of the **three-jaw universal chuck** in Fig. 19-4A operate as a unit, opening or closing together, **automatically centering** the workpiece. This makes it possible to center workpieces easily and rapidly. **Two sets** of jaws normally are included with the universal chuck because the jaws are not reversible. Universal chucks also are available with six jaws.


(South Bend Lathe, Inc.)

A. Three-jaw universal

B. Four-jaw independent

Fig. 19-4. Chucks.

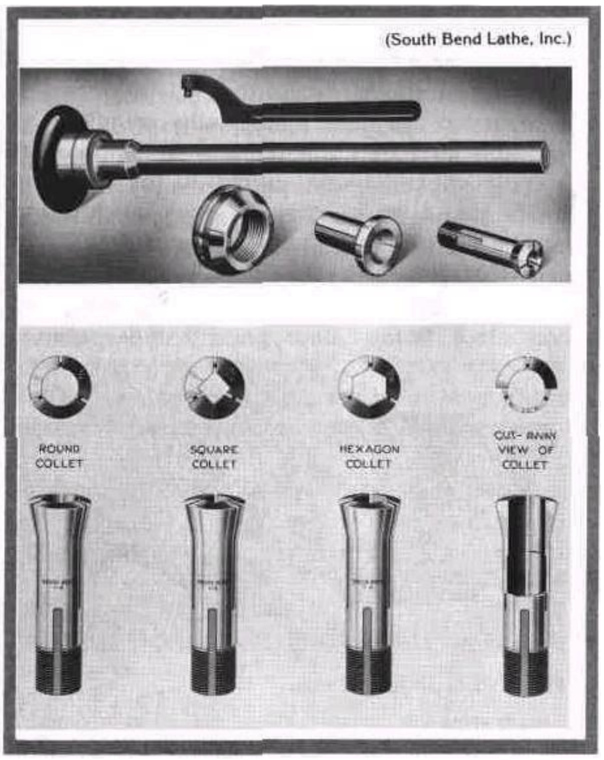

(South Bend Lathe, Inc.)

Fig. 19-5. Collet chuck and collets.

Universal three-jaw and six-jaw chucks are intended primarily for holding round workpieces. They hold work accurately to within 0.05 or 0.08 mm and retain this accuracy until either the jaws or the internal threads become worn or damaged.

The four-jaw independent chuck may be more accurate than the universal chuck, since the work-piece may be centered exactly with the use of a dial indicator.

The **collet chuck** is used widely in production work because it centers accurately, holds very tightly, and does not mark the workpiece. Collets are made in every standard bar stock size and are designed for holding round, square, or hexagon stock, Fig. 19-5.

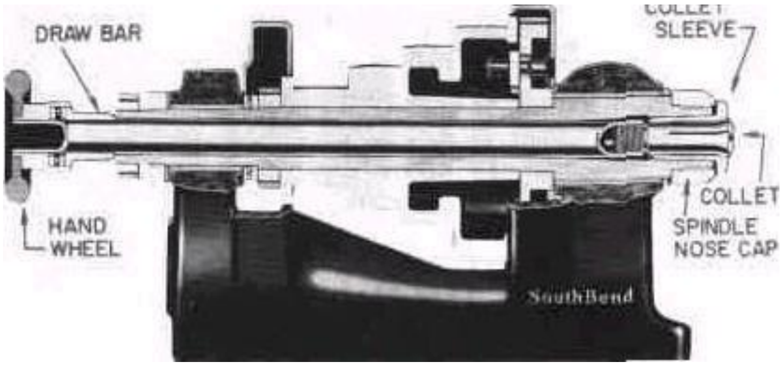


Fig.19-6.Assembly of draw-in collet chuck

A spring collet should be used only for the size of stock it is designed to hold. To avoid danger of breakage, it should not be used for holding stock that is more than 0.08 mm oversize or undersize.

Figure 19-6 shows an assembly of one form of draw-in collet chuck. The chuck is drawn tightly into the collet sleeve by turning the handwheel at the rear end of the drawbar toward the right, and loosened by turning the wheel toward the left.

Before installing a collet chuck, always clean the hole in the spindle, the collet sleeve, and the collet. Grit or chips left between mating surfaces can spoil the accuracy of the chuck and the parts held in it.

The **spindle chuck**, Fig. 19-7, resembles an ordinary **drill chuck**, except that it is designed to screw into the nose of the lathe spindle. The movable jaws are tightened with a pinion key. A spindle chuck is used primarily for chucking small round work, which it does quickly and accurately within 0.05 to 0.08 mm.

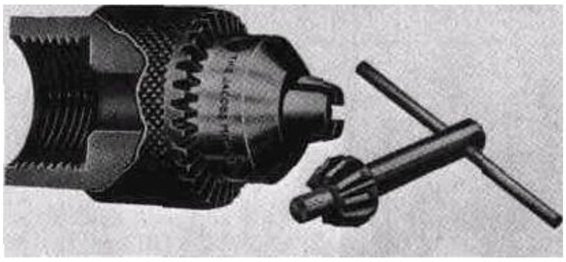


Fig. 19-7. Spindle chuck.

The **step chuck** and **closer unit** is designed for centering and holding small round discs. It operates on much the same principle as the draw-in collet chuck. Small diameter step chucks fit directly into the collet sleeve and do not require a separate closer.

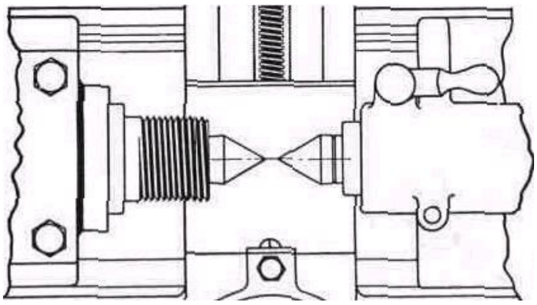


Fig. 19-8. Testing lathe centers for alignmen.

For efficient machining on a metalworking lathe, the correct type of cutting tool, called a tool bit, must be used. A tool bit is also referred to as a single-point cutting tool. For machining, the tool bit must be mounted in a toolholder.

Mounting work between centers is a common method of holding a workpiece while it is being machined. To rotate the workpiece, a faceplate having an open slot on one side is mounted on the spindle. A lathe dog is mounted on the stock and engaged in the slot in the faceplate.

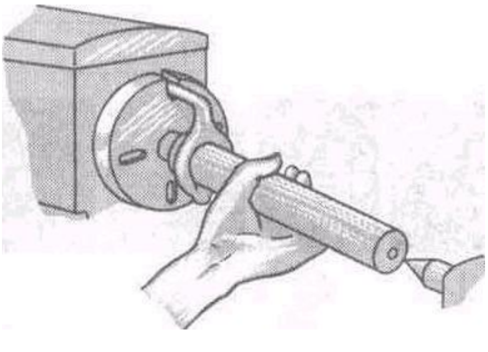


Fig. 19-9. Engaging the live center. The lathe dog is mounted on the workpies.

There are three types of **spindle noses** in use for attaching chucks and faceplates. They are (1) the threaded spindle nose, Fig. 19-10; (2) the spindle nose with the long taper key drive, Fig. 19-11; and (3) the spindle with the cam-lock drive, Fig. 19-13.

With the threaded spindle, the chuck (or the faceplate) is threaded directly to the spindle.

With the key-drive spindle, the chuck is mounted with the internal key slot properly aligned with the key on the spindle nose. The threaded collar is then threaded to the shoulder of the chuck and tightened with a spanner wrench.

With the cam-lock spindle, the notched holding pins on the chuck are inserted into the holes in the spindle flange. The chuck then is locked in position by turning the cam-locking screws in the flange with a T-handle chuck key.

(South Bend Lathe, Inc.)

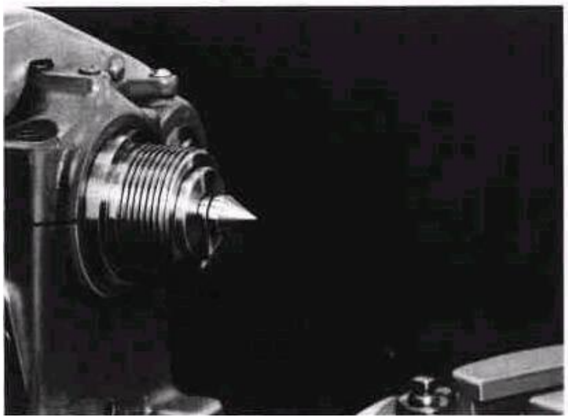


Fig. 19-10. Threaded spindle nose.

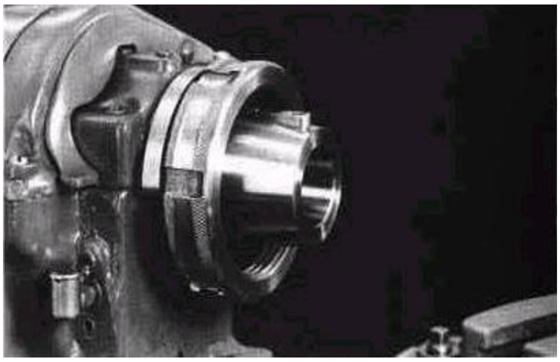


Fig. 19-11. Spindle nose with long taper key drive.

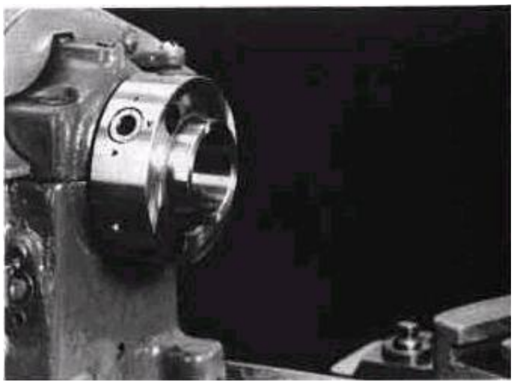


Fig. 19-12. Cam-lock spindle.

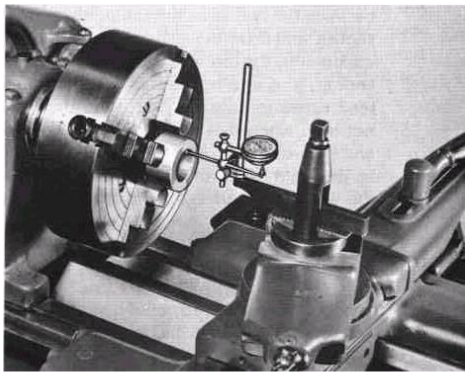


Fig. 19-13. Centering work with dial indicator.

The **steady rest**, sometimes called a **center rest**, Fig. 19-14, is a device used to support long shafts or spindles of small diameter while they are being turned, bored, or threaded. When in use, the rest is mounted on the lathe bed and held in position with a clamp. A **follower rest**, Fig. 19-14, is a supporting device which, when correctly attached to the saddle of the lathe with the supporting jaws adjusted to the work, follows along the finished face of the work and holds it steady against the cutting tool.

When turning very long rods, shafts, or spindles, it sometimes is necessary to use both a steady and a follower rest in combination, Fig. 19-14.

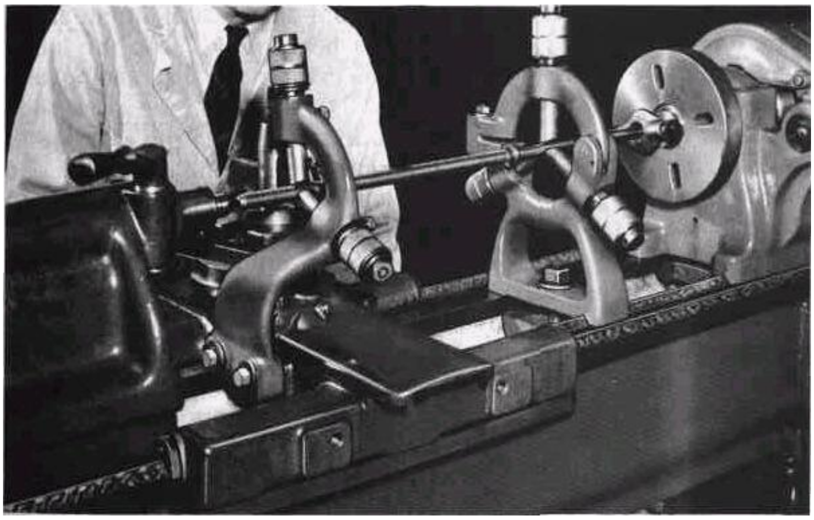


Fig. 19-14. Using both steady rest and follower rest for cutting threads.

19.2. Milling and Grinding Attachments

Some manufacturers offer milling and grinding attachments which extend the usefulness of the lathe. These are particularly useful to small shops which have only occasional need to do milling and grinding.

The attachment shown in Fig. 19-15 is a vise for holding the work in the position desired. It may be swiveled horizontally or pivoted vertically for angular cuts. The cutter may be mounted on a taper shank which fits the tapered hole in the lathe spindle or, if a cutter with a straight shank is used, it may be mounted in a collet or spindle chuck. Cutters also may be mounted on an arbor held between centers.

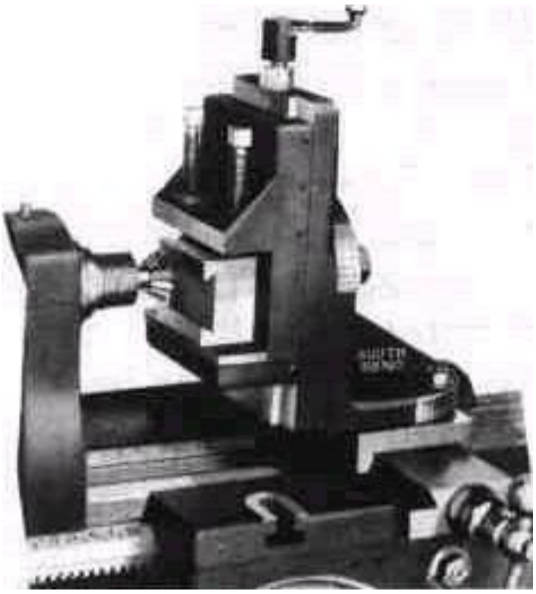


Fig. 19-15. Milling a dovetail on a lathe.

The workpiece is brought into contact with the cutter by means of the handwheel on the carriage, the cross-slide crank, and the vertical adjusting screw at the top of the milling vise. The cut is controlled through the power cross-feed system.

Lathe-grinding attachments are mounted on the compound rest in place of the tool post, and thus are also known as **tool post grinders**. An external grinding attachment enables the lathe to do precision cylindrical grinding of hardened steel bushings and shafts. With appropriate accessories, it is also capable of sharpening milling cutters, Fig. 19-17, reamers, Fig. 19-18, and other precision cutting tool.

An **internal grinding attachment** is used to finish round holes in parts like steel bushings and cutting dies after they have been hardened.

A hydraulic tracer attachment, Fig. 19-19, converts an ordinary lathe into a copying lathe. The tracing attachment consists mainly of (1) a hydraulic cylinder to which a toolholder is attached, (2) a sensitive valve which detects the movement of the tracing stylus, (3) a rigid arm connecting the valve body to the toolholder, (4) a template holder, and (5) a hydraulic pump. Tracer attachments are capable of turning parts of almost any shape, but are valued most for their ability to efficiently turn parts with curved contours.

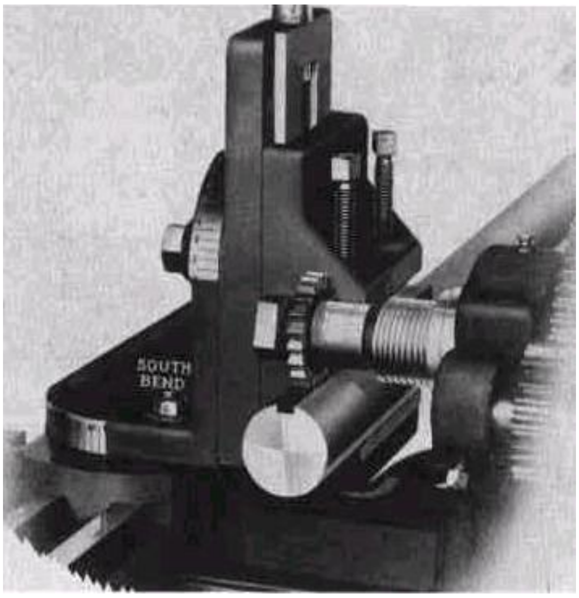


Fig. 19-16. Milling a keyway on a lathe.

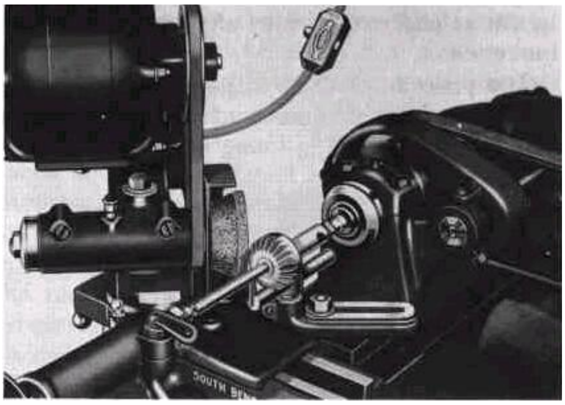


Fig. 19-17. Grinding a milling cutter on a lathe.

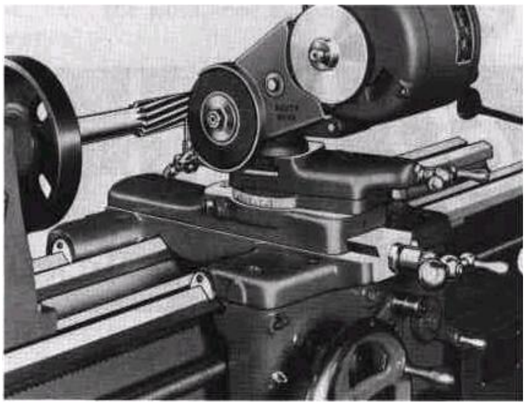


Fig. 19-18. Grinding a reamer on a lathe.

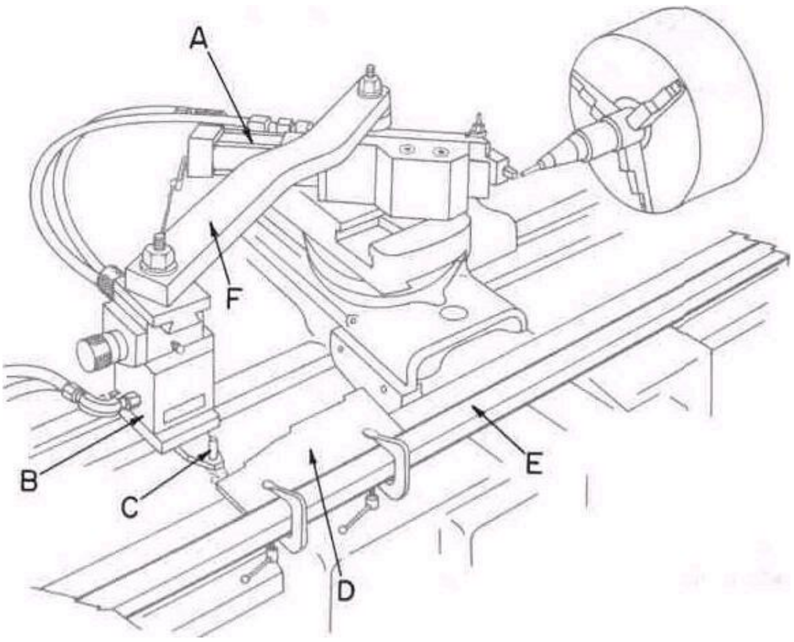


Fig. 19-19. A lathe hydraulic tracer attachment. A- hydraulic cylinder; B- valve body; C- tracing stylus; D- template; E- template holder; F- arm connecting valve body with tool slide.

The shape of the object to be machined is either turned on a round model or cut into a flat template. As the tracing stylus is drawn across the round model or template, it signals its movements through the valve to the hydraulic cylinder, causing a like amount of tool movement. The profile of the template or round model must be made to the same size and shape as that desired on the workpiece. The surface which is traced should be finished smoothly, since any roughness will be transmitted to the workpiece. It is important to provide about 25 mm of land area on each end of the template. The surplus land area at the starting end of the template is necessary to provide a location for the stylus to contact the template without the tool touching the workpiece. At the trailing end, the surplus land area ensures that the tool traverses over the entire length of the workpiece. If only a portion of the workpiece is involved, the template may be shaped to automatically withdraw the tool from the workpiece surface at the desired location.

Manually operated **turret lathes** are designed for low and moderate volume production of duplicate parts. Turret lathes appear much more complex than a standard lathe. However, all lathes are operated by the same basic principles.

Turret lathes are fitted with all of the tools needed to machine a particular part. Some parts can be completely machined in a single setup. Often, however, a second setup is required in order to machine surfaces on the back of the piece which were inaccessible during the first setup. Lengthwise movement of tools is controlled by setting adjustable stops built into the machine. Diameters are usually determined by the machine operator who sets the tool location according to a micrometer dial reading. In the hands of skilled machine operators, these machines are capable of making duplicate parts of almost any shape with remarkably consistent accuracy.

A great variety of toolholders and cutting tools have been designed especially for use on the bed turret.

Tooling may be either **shank-mounted** or **flange-mounted**. Shank-mounted tooling fits into a hole in the turret face where it is held in place by a built-in clamp, Fig. 19-2I B. Flange-mounted tooling is bolted directly to the face of the turret, Fig. 19-21 A. Shank-mounted tooling can also be held in flange-mounted toolholders. **Multiple toolholders** are also available, permitting several operations to be carried out from the same position, Fig. 19-20.

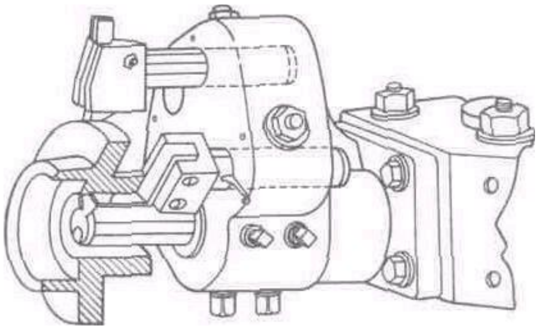
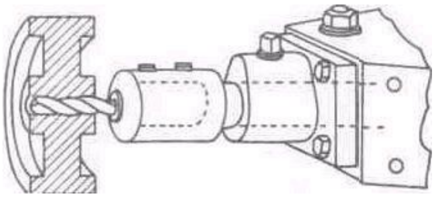
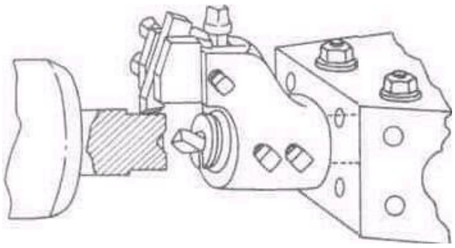


Fig. 19-20. Multiple toolholder for turret lathe.

The square turret mounted on the cross slide uses conventional turning tools. Large highspeed steel or cast alloy tool bits are mounted directly in the turret. Throwaway carbide inserts are mounted in their special toolholders which in turn are mounted in the turret. The square turret is tooled for any of the usual facing and turning operations. The rear tool post normally mounts an inverted cutoff blade; but chamfering, grooving, and other cuts may be made from this position as well. When fully tooled, the turret lathe may carry from 11 to 16 or more cutting and forming tools.



A. Adjustable toolholder suitable for holding drills, reamers, counterbores, etc.



B. Adjustable knee tool for turning and drilling short lengths at the same time if necessary.

Fig. 19-21. Turret lathe tools**.**

Turrets are often provided with **permanent setups** of universal toolholders capable of accepting a wide variety of tools.

Chapter 20. Milling Machine Operations

20.1. Types of Milling Machines

A milling machine uses one or more revolving cutters to shape the workpiece, Figs. 20-1 and 20-2. The workpiece is usually held in a vise or fixture attached to a movable table. Cutting takes place by feeding the workpiece against the revolving cutter. On some very large machines, however, the revolving cutter is fed past the stationary workpiece.

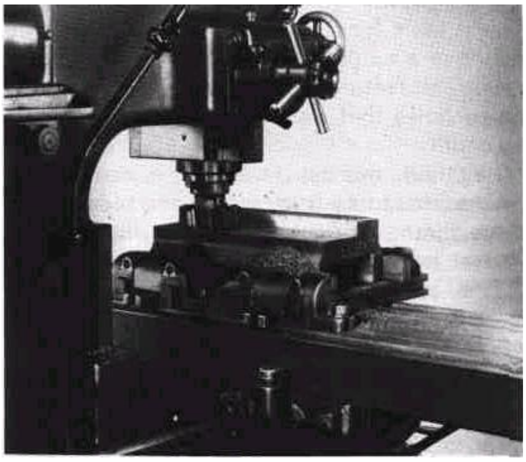
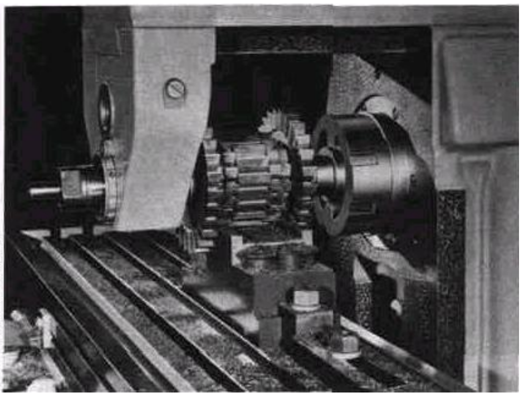


Fig. 20-1. A single heavy-duty plain milling cutter mounted in a vertical milling machine spindle.

The milling machine is one of the most important machine tools. It is capable of a wide variety of machining operations, and its high metal removal rates make it very efficient. For these reasons, it ranks equal in importance with the metalworking lathe.

Milling machines are widely used for machining flat surfaces. Shoulders, grooves, keyways, T-slots, and dovetails are also common milling operations. Curved and irregular surfaces can also be made with special milling techniques and milling cutters.

Milling machines may also be used for all of the common hole-machining operations normally done on a drill press. With the milling machine, holes may be precisely located without the use of drill jigs. The micrometer collars that are attached to each table feed screw are graduated in hundredths of a millimeter or less. This feature greatly simplifies the production of holes that must have accurate location and depth.



(Cincinnati Milacron)

Fig. 20-2. Milling a casting with several cutters mounted on the arbor of a horizontal milling machine. This is known as gang milling.

Milling machines may be classified in a general way according to two types: the column and knee type and the bed type. Machines of the **column** and **knee** type, Figs. 20-1 and 20-2, are the most versatile. Milling machines of the column and knee type have a table that travels longitudinally on the saddle. The saddle, which travels transversely (crosswise), is mounted on the knee. The knee is mounted on the column in a way that permits it with the table, to be lowered or raised to the desired height. Thus, the three directions in which the table may be moved on a column and knee machine are longitudinal, transverse, and vertical.

The **bed** type generally is a larger machine used for more specialized production milling operations. The table on a bed machine rests on a stationary bed. Therefore, it is very sturdy and rigid. On many bed-type machines, the table travels longitudinally only. Bed-type milling machines are often equipped with more than one spindle. These are located in spindle heads that may be advanced or withdrawn to adjust the depth of cut. The heads also may be adjusted so that the cutter can be properly located over the workpiece.

Milling machines are also classified by whether the spindle is normally vertical, Fig. 20-1, or horizontal, Fig. 20-2.

The **vertical milling machine** shown in Fig. 20-1 is known as a **turret milling machine**. The turret feature allows the head to be rotated in a horizontal plane wherever needed to be in the best operating position to machine a workpiece. With this type of mill, attachments mounted on the rear of the ram may be brought into operating position over the table. The spindle is carried in a quill that can move in and out of the head like a drill press quill. The universal head may be both swiveled and tilted for angular milling operations or for drilling holes at any angle. This type of machine is popular in schools and in tool and die, maintenance, and job shops. Some vertical milling machines have heads that cannot be swiveled or tilted. They are built to withstand the stress of heavy milling operations.

Vertical milling machines use end-milling cutters of various types and sizes. Vertical mills can machine horizontal surfaces, vertical surfaces, angular surfaces, shoulders, grooves, fillets, keyways, T-slots, and dovetails.

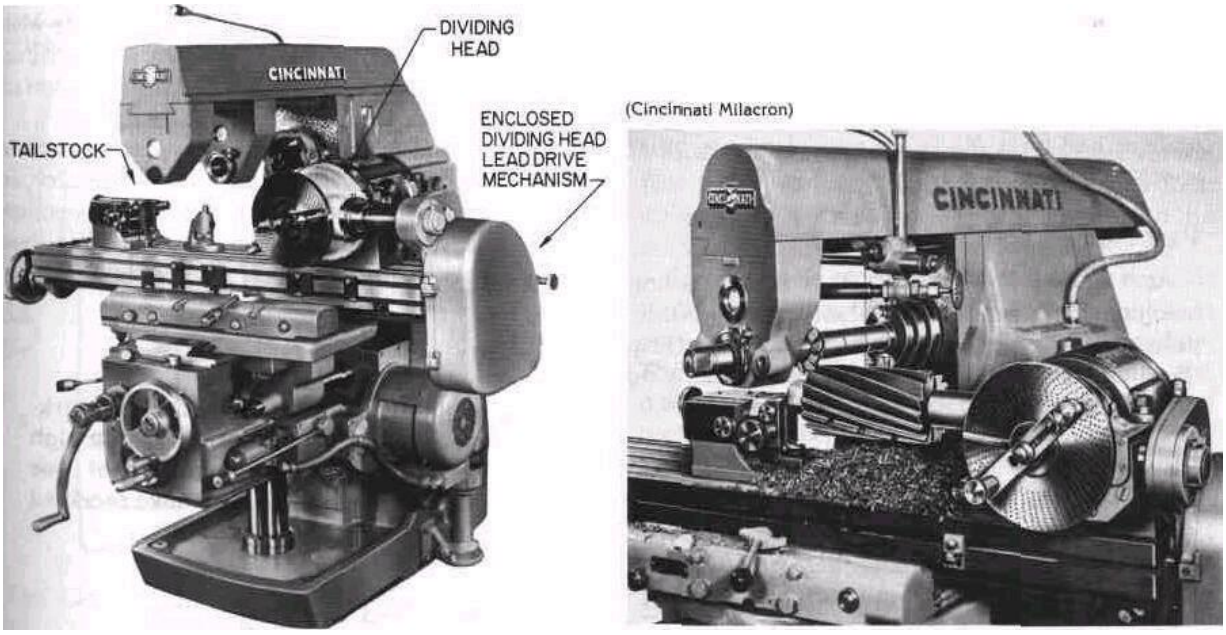


Fig. 20-3. A universal horizontal milling machine equipped for helical milling. The close-up shows the workpiece mounted between the dividing head centers. A fluting cutter is mounted on the arbor.

Vertical milling machines often are used for hole-machining operations that require extreme accuracy in hole location, such as jig boring operations.

Horizontal milling machines are made in two types, **plain** and **universal**. Plain milling machines have the table fixed at a right angle to the knee. Universal milling machines have a table that can be pivoted in a horizontal plane, Fig. 20-3. This feature allows the universal milling machine table to be swiveled to whatever angle is needed for milling a helical groove.

Four basic size factors are used in identifying milling machines: **power, capacity, model, and type.**

The power rating of a machine is based on the power of the spindle drive motor. On machines of standard size, this rating may vary from 2.238 to 37.3 kW. On smaller machines, it may be as low as 0.373 kW.

The capacity or size of the machine is based on the amount of longitudinal table travel. The overall capacity of the machine is also related to the amount of cross and vertical table travel. However, only the longitudinal travel is used in identifying the machine size. The six standard sizes that apply to column and knee milling machines follow: No.1 - 559 mm; No.2 - 711 mm; No.3 - 864 mm; No.4 - 1067 mm; No.5 - 1270 mm; No.6 - 1524 mm.

The **model** designation is determined by manufacture. Features vary with different brands. The **type** of milling machine is either plain or universal, horizontal or vertical, and column and knee or bed type.

In **up milling**, the cutter tooth starts with a chip of zero thickness and ends with a thick chip. The cut starts in clean metal and ends by lifting off the rough surface scale. This procedure increases cutting tool life where a hard or dirty surface exists on the workpiece. The forces caused by the cutter on the workpiece act in a direction that tends to pull the workpiece out of the vise or fixture. Hence, the workpiece must be fastened very securely for up milling. Up milling also tends to push the workpiece away from the cutter, thus eliminating backlash.

Up milling should, therefore, be used on all milling machines not equipped with anti-backlash devices or ball bearing table screws. Up milling is also recommended for cutting the softer steels and other ductile metals. Because of its hard surface, cast iron should be machined this way.

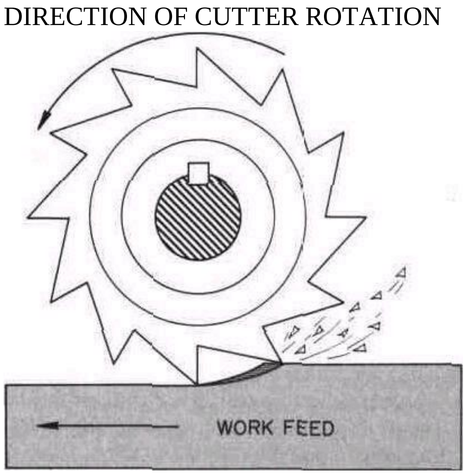


Fig. 20-4. Up or conventional milling.

With **down milling**, the cutter tooth starts with a chip of maximum thickness and ends with a thin chip of zero thickness. A scraping action results as the thinned edge of the chip is removed. Hence, down milling generally produces a good-quality surface finish.

The direction of cutter rotation in down milling tends to push the work down against the table, thus resulting in more rigid setups. This factor is an advantage when cutting thin workpieces held in a vise or workpieces held on a magnetic chuck.

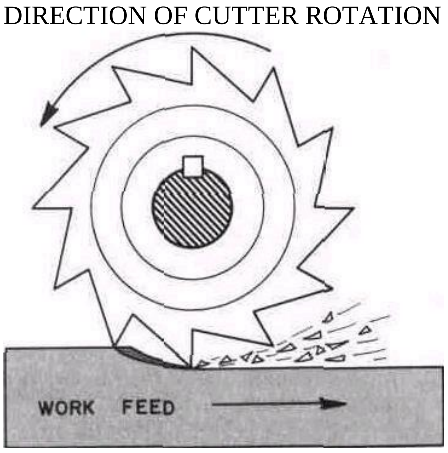


Fig. 20-5. Down or climb milling.

Metal sawing with thin cutters tends to cause the cutter to **walk** while up milling. This tendency is reduced and straighter cuts are made by using the down-milling technique. Down milling often is recommended where carbide cutters are used. Since the cut starts with a chip of maximum thickness, there is less rubbing action and, therefore, less wear on the cutting tool.

There are many ways of holding a workpiece while it is being machined on a milling machine. The most common are: (1) using a special jig to accurately position the work for machining and at the same time hold the work securely with clamping bolts, studs, and screws; (2) fastening the work securely to an angle plate with clamps or bolts; (3) using a vise that is bolted to the machine table; (4) supporting the work between the centers of a dividing head; and (5) using a chuck that screws onto the spindle of a dividing head.

20.2. Milling Cutters

Milling cutters are multiple-toothed cutting tools made especially for use on milling machines. They are made in many standard shapes and sizes, Fig. 20-6. Milling cutters are designed three ways: (1) as solid cutters made of one piece of high-speed steel, cast alloy, or cemented carbide; (2) as solid cutters made of cemented carbide teeth brazed to an alloy steel body; and (3) as inserted-tooth cutters, which have an alloy steel body with replaceable ctters made of high-speed steel, cast alloy, or cemented carbide.

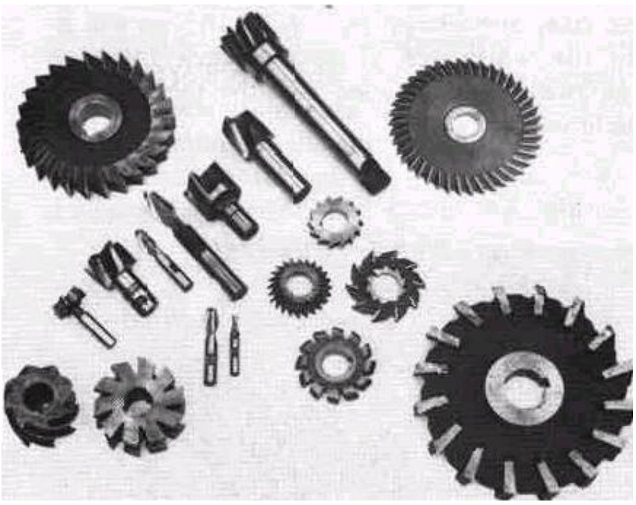


Fig. 20-6. An assortment of milling cutters.

Increasingly, milling cutters with replaceable carbide inserts are being used for production milling. Their ability to withstand high cutting speeds and feed rates makes them the most economical choice for many applications.

Most cutters, however, are

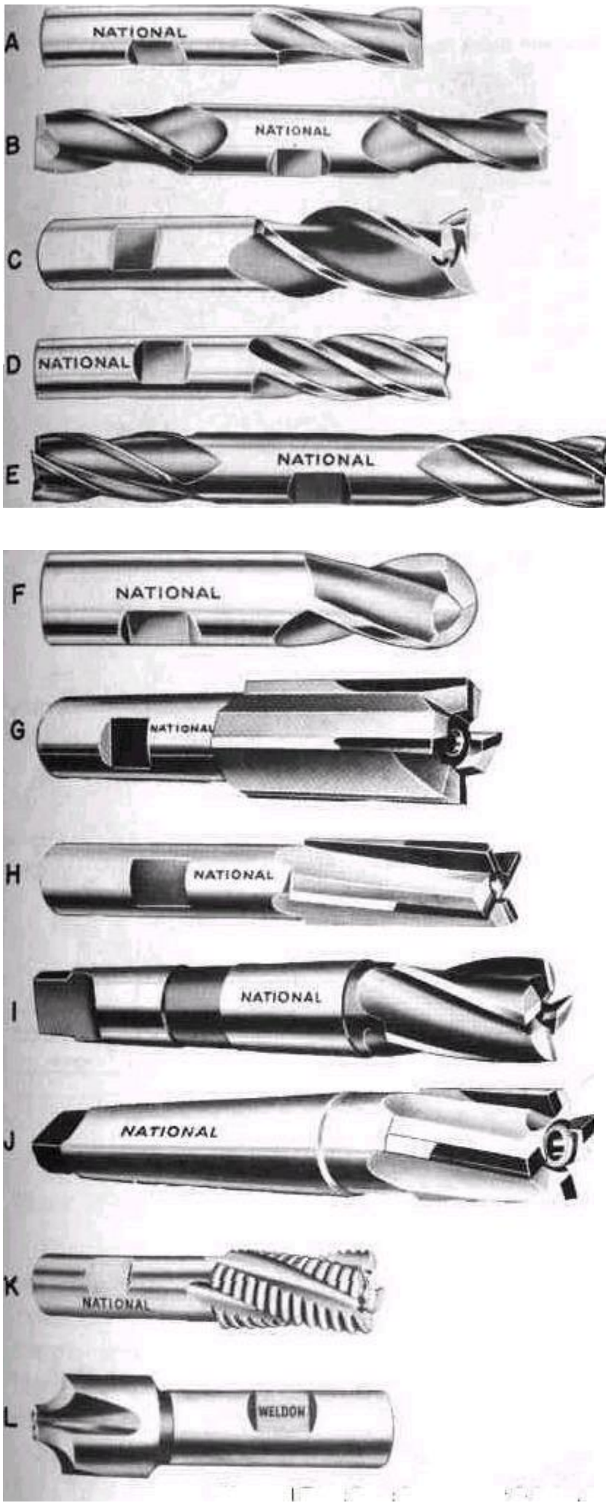


Fig. 20-7. End mills: (A) two-flute single-end, (B) two-flute double-end, (C) three-flute single-end, (D) four-flute single-end, (E) four-flute double-end, (F) two-flute ball-end, (0) carbide-tipped straight-flute, (H) carbide-tipped right-hand helical-flute, (I) four-flute with taper shank, (J) carbide-tipped with taper shank, (K) roughing, (L) corner-rounding.

made of solid high-speed steel or cast alloy. They have good impact and wear resistance. They also are able to withstand the vibration that often occurs on lightweight milling machines.

Rigidity of the machine and the workpiece is important when machining with carbide cutters. Vibration or chatter will often cause the cutters to chip or shatter because of their extreme hardness.

Milling cutters fall into two major classifications — standard and special. Standard types are made according to dimensional standards. The dimensions apply to diameter, width, hole size, size of keyway, etc. Special types may or may not be made to standard dimensions. Sometimes they are designed to combine several different milling operations. Some of the most common types of milling cutters and their uses are explained below.

Solid end-milling cutters are made of one piece of cutting tool material, Fig. 20-7. Some end mills are of the **shell type**, Fig. 20-11, in which the cutter body and its shank are separate. End-milling cutters have teeth on the periphery (circumference) and on the end. The teeth on the periphery may be either straight or helical. Solid end mills may have either a straight shank or a tapered shank. Shell end mill adapters normally have tapered shanks.

Fig. 20-8. An end mill designed to use three replaceable insert cutters.

End mills may be used for machining horizontal, vertical, angular, or irregular surfaces. Common operations include the milling of slots, keyways, pockets, shoulders, and flat surfaces, Fig. 20-9.

Two-flute end mills have only two teeth. The end teeth are designed so that they can cut to the center of the mill. Therefore, two-flute end mills may be fed into the work like a drill; they then may be fed lengthwise to form a slot. These mills may be either the single-end type with teeth on one end only, or they may be the double-end type, Fig. 20-7.

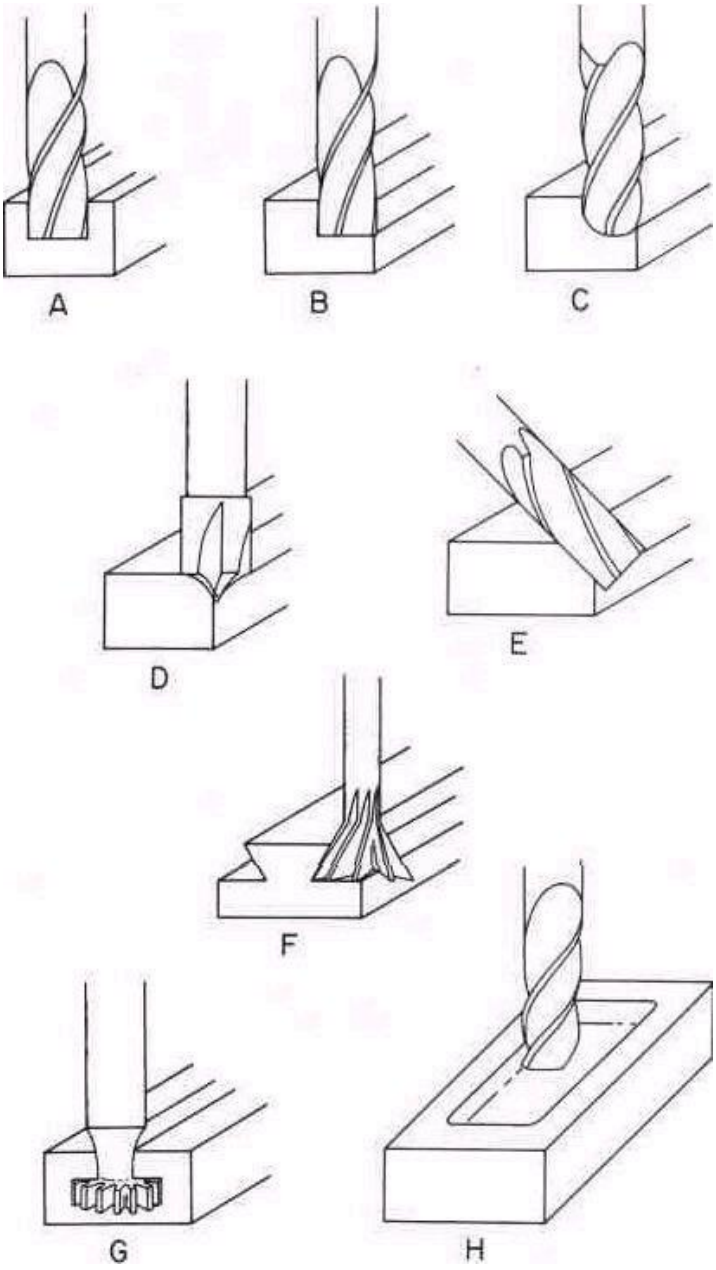


Fig. 20-9. Commonly performed end-milling operations: (A) slot or groove, (B) step, (C) concave radius or fillet, (D) convex radius, (E) angled surfaces, (F) dovetail, (G) T-slot, (H) pocket.

Multiple-flute end mills have three, four, six, or eight flutes, and normally are available in diameters up to 50 mm. They also may be either the single- or double-end type.

Ball end mills are used for milling fillets or slots with a radius bottom, for rounding pockets and the bottoms of holes, and for all-round die-sinking and die-making work. Four-fluted ball end mills with center cutting lips also are available.

Roughing end mills are made to rapidly remove excess metal with minimum

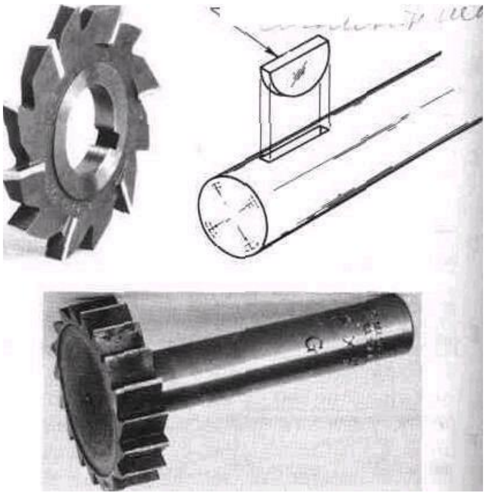


Fig. 20-10. Woodruff key seat cutters and assembly drawing of a shaft, key seat, and Woodruff key.

horsepower. They have from three to eight flutes depending on the tool diameter.

Insert-type end mills, Fig. 20-8, use replaceable inserts of high-speed steel or carbide. Sizes as small as 25 mm diameter use two inserts. Larger sizes use three or four inserts.

Key seat cutters are of special design for cutting key seats for Woodruff keys (which have the shape of a half circle). An end mill Woodruff key seat cutter is shown at the bottom of Fig. 20-10. An arbor type also is shown in the same illustration.

Shell end mills, Fig. 20-11, have a hole for mounting the cutter on a short (stub) arbor. The center of the shell is recessed to provide space for the screw or nut that fastens the cutter to the arbor. The teeth usually are helical. These mills are made in larger sizes than solid end mills; normally they are available in diameters from 32 mm to 152 mm. Cutters of this type are intended for milling wide, flat surfaces.

A **face milling cutter** is a special form of a large end mill. They are made in sizes 152 mm in diameter or over. Similar cutters under 152 mm in diameter are called **shell end mills**. Facing cutters usually have inserted teeth that cut on the periphery and the face. Most of the cutting takes place on the periphery, but some finishing also is done by the face teeth. Face milling cutters are used for milling of large flat surfaces.

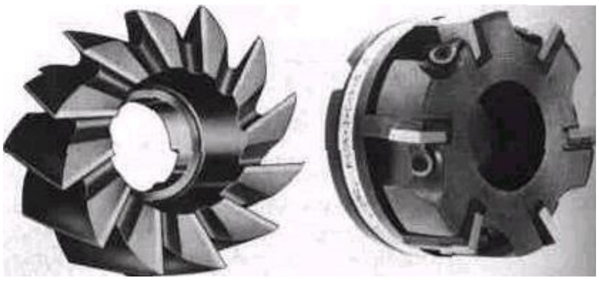


Fig. 20-11. Shell end mills: one-piece type and replaceable-insert typs

Plain milling cutters are cylindrical, with teeth on the periphery only, Fig. 20-12. They have an accurately ground hole for mounting on the milling machine arbor. They are used primarily for milling plain flat surfaces. However, they may be combined with cutters of other types to produce surfaces with various shapes. Plain milling cutters are available in many widths and diameters.

The names of the different parts of milling cutters are shown in Fig. 20-12. Refer to this figure as you read the following discussion of the three groups of plain milling cutters:

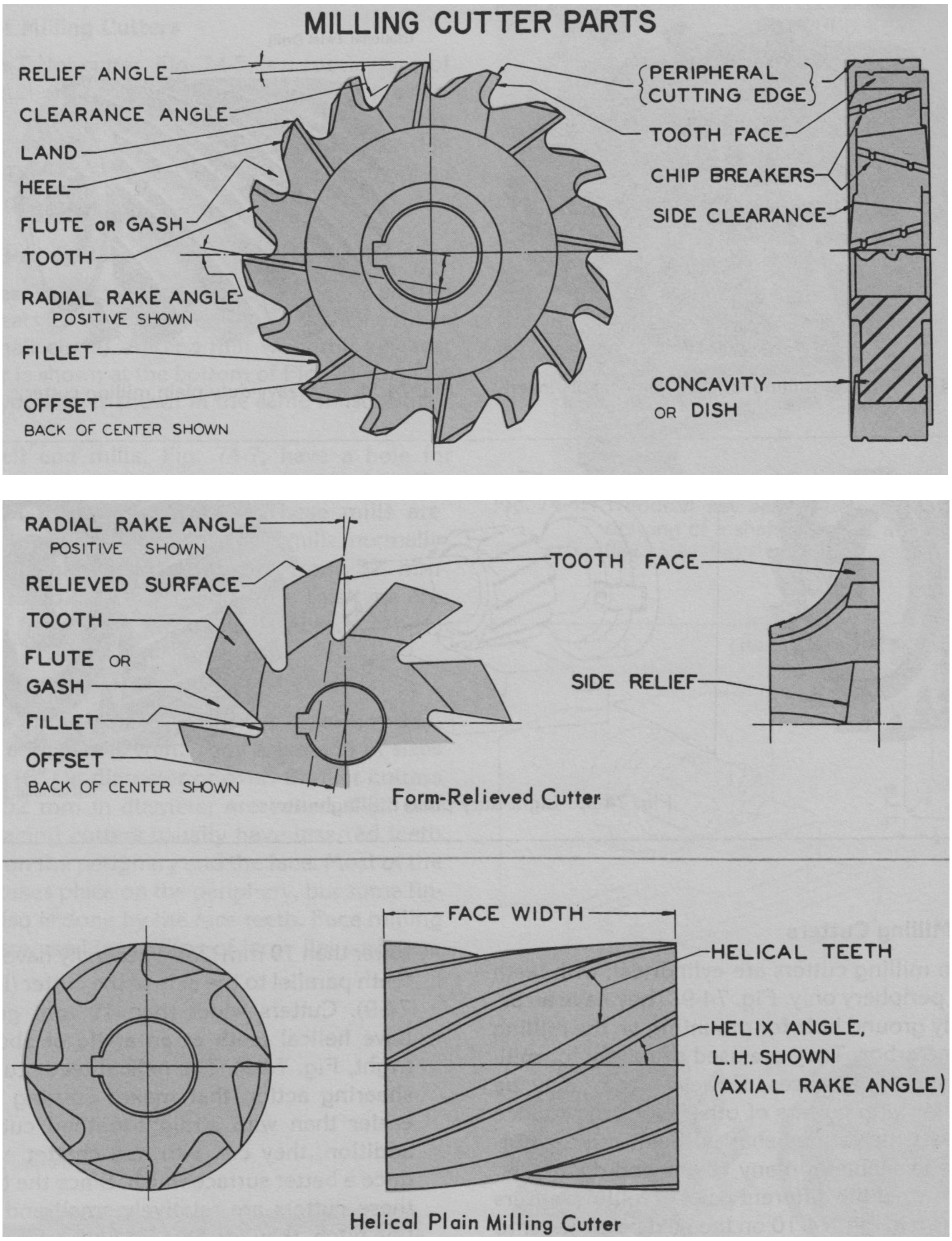


Fig. 20-12. Parts of milling cutters

1. **Light-duty plain milling cutters** are available in two different forms. Those that are narrower than 19 mm generally have straight teeth parallel to the axis of the cutter. Cutters wider than 19 mm generally have helical teeth at an angle of about 25°, Fig. 20-13. The helical teeth cut with a shearing action that makes starting the cut easier than with straight-toothed cutters. In addition, they cut with less chatter and produce a better surface finish. Since the teeth on these cutters are relatively small and have a fine pitch, they are best for light cuts with fine feeds.

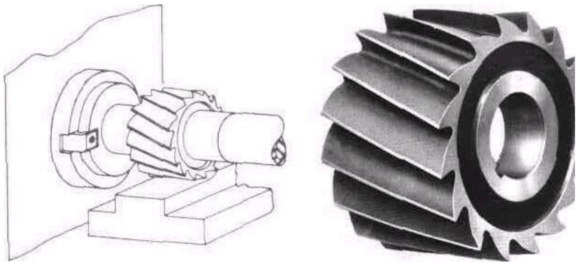


Fig. 20-13. Light-duty plain milling cutters.

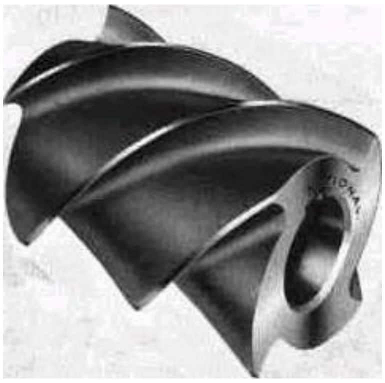


Fig. 20-14. A helical plain milling cutter.

2. **Heavy-duty plain milling cutters** or **coarse-tooth milling cutters** are made in the larger widths only an have larger and fewer teeth than light-duty cutters. The teeth have a helix angle between 25° and 45°. These cutters are designed for heavy plain milling cuts. The strongly supported cutting edges and the wide flutes provide strength and adequate space for heavy chip removal. Wide cutters often are called **slab mills**.

3. **Helical plain milling cutters**, Fig. 20-14, have fewer and coarser teeth than the heavy-duty type. The helix angle of the teeth is between 45° and 60°, and it may be even greater. The high helix angle tends to absorb the load in end thrust. This type of cutter is efficient for taking wide, shallow profiling cuts on brass or soft steel. However, it is not as efficient as the heavy-duty type for heavy feeds and deep cuts on wide, flat surfaces.

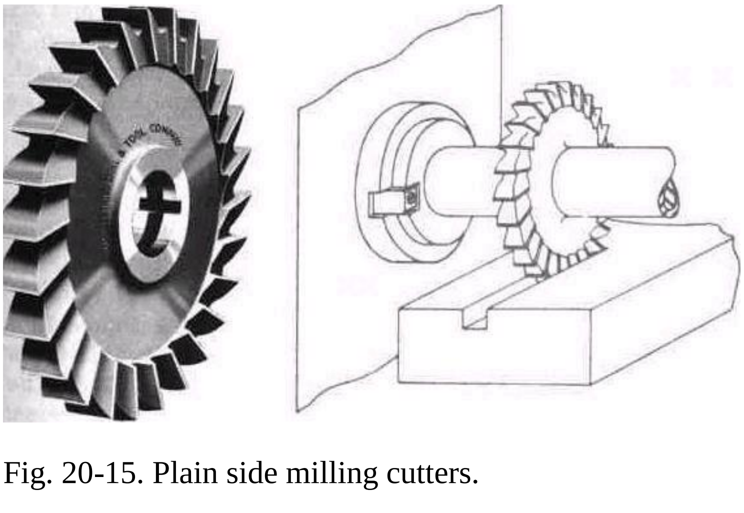


Fig. 20-15. Plain side milling cutters.

4. **Side milling cutters**, Fig. 20-15, are similar to plain milling cutters, but they also have teeth on one or both sides. The teeth on the periphery do most of the actual cutting, while those on the sides finish the side of the cut to size. The teeth may be either straight or helical. These cutters are used for side milling, slotting, and straddle milling. In **straddle milling** two side milling cutters are mounted on a milling arbor, with the desired distance between the

cutters established with spacers. Thus, both sides of the part are machined parallel to each other simultaneously. Several types of side milling cutters are in common use:

1. **Plain side milling cutters**, Fig. 20-15, have straight teeth on the periphery and on both sides. The teeth on the side are provided with concavity or taper toward the center of the cutter, thus giving side relief or clearance, Fig. 20-12.

2. **Half side milling cutters** have helical teeth on the periphery and on one side only. Cutters of this type are recommended for heavy-duty face milling and straddle-milling operations where teeth are needed only on one side of the cutter. The teeth are deeper and longer on the side, and thus provide more chip clearance.

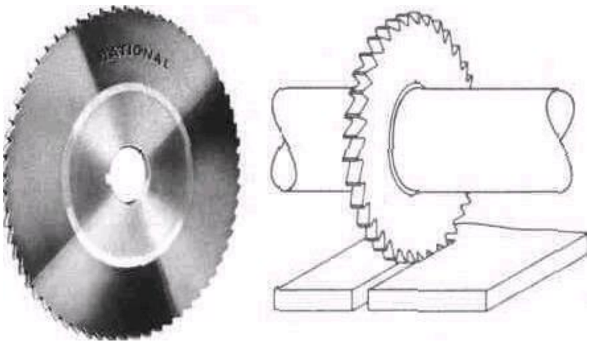


Fig. 20-16. A plain slitting saw.

3. **Staggered-tooth side milling cutters,** Fig. 20-17, are narrow cutters with teeth alternating on opposite sides. This tooth arrangement reduces dragging and scoring, thus providing a free cutting action. It also provides more space for chip removal. This cutter is recommended for heavy-duty keyway and slotting operations.

Slitting saws are cutters designed for cutoff operations and for cutting narrow slots. For deep cuts, saws with side chip clearance should be used. **Plain slitting saws,** Fig. 20-16, are essentially thin plain milling cutters. However, the sides are slightly tapered toward the hole, thus providing side relief, which prevents binding on the sides of the cutter. The teeth are much finer than on plain milling cutters. Hence, the feed rate must be much less, usually from one-eighth to one-quarter of that used with plain milling cutters. Plain slitting saws are available in widths from 0.8 mm to 4.8 mm and in diameters from 60 mm to 200 mm.

Slitting saws with side teeth, Fig. 20-19, are similar to side milling cutters. The side teeth provide clearance for chips and prevent the cutter from binding on the sides. These cutters are available in widths from 1.5 mm to 4.8 mm. Cutters of this type are designed for deeper slotting and cutoff operations than those normally done with plain slitting saws.

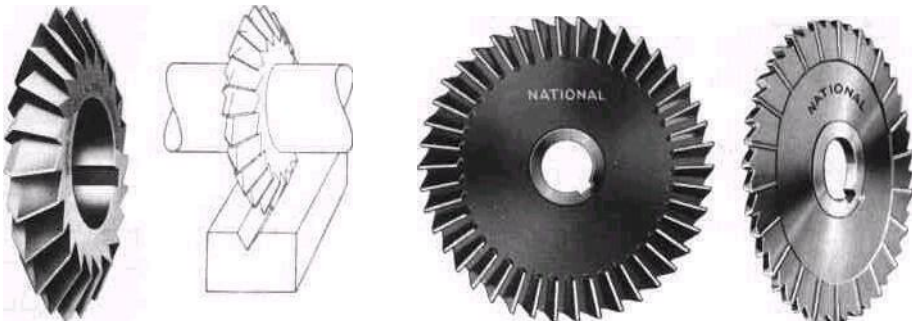
Fig. 20-17. A staggered-tooth side milling cutter.

Staggered-tooth slitting saws, Fig. 20-19, are similar to staggered-tooth side milling cutters. They are recommended for cuts of 4.8 mm and wider. They may be used for deeper cuts and with standard feeds.

Fig. 20-18. A single-angle milling cutter.

Screw-slotting cutters are plain slitting saws that are designed for cutting slots in screw heads. These cutters have fine-pitch teeth, and therefore are designed for fine feeds. The sides of the cutter are ground straight and parallel. Hence, no side relief is provided. They are available in widths from 0.5 mm to 4.6 mm, and with a maximum diameter of 70 mm.

Angle milling cutters, Fig. 20-18, are used for angle milling operations such as cutting V-grooves, dovetails, serrations, and reamer teeth. Two basic types of angle cutters are available. **Single-angle cutters,** Fig. 20-18, have one angular surface, and they have teeth on the angular surface as well as the straight side. These cutters are normally made with 45° or 60° angles. **Double-angle cutters** are used for machining V-grooves.



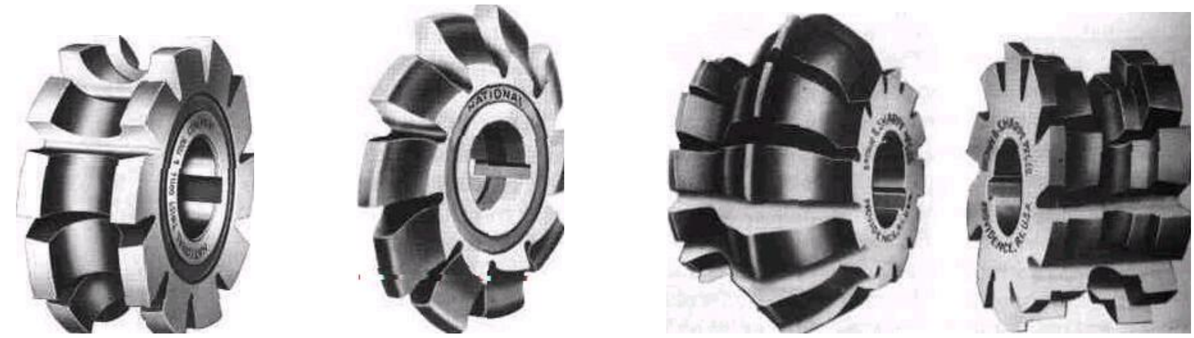
A. With side teeth.

B. With staggered teeth.

Fig. 20-19. Slitting saws.

Double-angle cutters with equal angles on both faces normally are available with an included angle of 45°, 60°, or 90°. The angles, however, need not be the same on both sides of the cutter.

Formed-tooth milling cutters are used for machining surfaces with a curved outline. **Concave milling cutters** are used to mill convex half-circles, Fig. 20-20.



A. A concave milling cutter

B. A gear milling cutter

C. Special form-relieved cutters

Fig. 20-20. Form-relieved milling cutters.

Convex milling cutters are used to cut concave surfaces. Fig. 20-21. **Corner-**

rounding cutters are used for rounding outside corners. They are available in either a right-hand or left-hand style. **Gear cutters**, Fig. 20-20, are used for cutting gear teeth. **Fluting cutters** are used for cutting flutes in reamers and milling cutters. Formed-tooth cutters also are available in special shapes, Fig. 20-20.

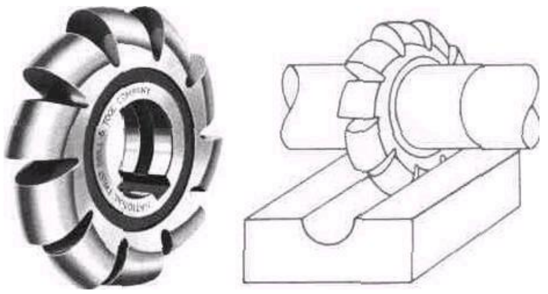


Fig. 20-21. A convex milling cutter.

A **fly cutter** consists of one or more single-point tool bits or cutters mounted in a bar or cylinder. There are three types of fly cutters.

One kind of fly cutter is used **for boring holes**. This type consists of an arbor into which a single-point cutting tool is fastened, Fig. 20-22. Special multiple cutting-tool fly cutters are custom-made for boring several holes of different diameters simultaneously, Fig. 20-23. Each cutting tool is individually adjustable.

A second kind of fly cutter is used primarily **for planing flat surfaces**. A third kind of fly cutter is used **for cutting grooves**. It is also used to cut holes by plunging a single-point cutting tool into the workpiece. This method of cutting is known as **trepanning**.

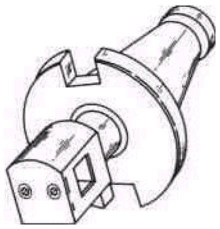


Fig. 20-22. A fly-cutter adapter used with a single-point cutting tool.

Fig. 20-23. A special fly cutter for boring several diameters simultaneously.

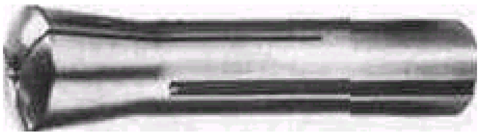
Helical fluted cutters are either left-handed or right-handed. Looking at the cutting end of the cutter, if the flutes twist to the right, the cutter is **right-handed**. The hand of the cutter determines the direction it must rotate in order to cut. A right-handed cutter requires counterclockwise rotation when viewed from the cutting end. A **left-handed** cutter requires clockwise rotation.

Milling cutters should not be nicked by bumping against tools, machines, or accessories. When not in use, they should be stored in a way that prevents damage. When cutters show evidence of becoming dull, they should be sharpened. If they are allowed to become very dull, extreme forces build up at the cutting edge of the teeth, thus causing possible chipping or fracture. Dull cutters also cause extreme forces on the milling arbor and other parts of the machine.

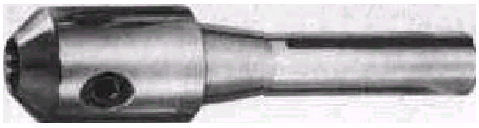
Milling cutters never should be forced into the arbor. Doing so may damage the cutter or the arbor.

20.3. Methods of Holding Cutters

Several different methods are used to hold milling cutters on milling machines. These methods include (1) collets and special holders, (2) quick-change toolholding systems, (3) arbors, and (4) adapters.



A. Spring collet



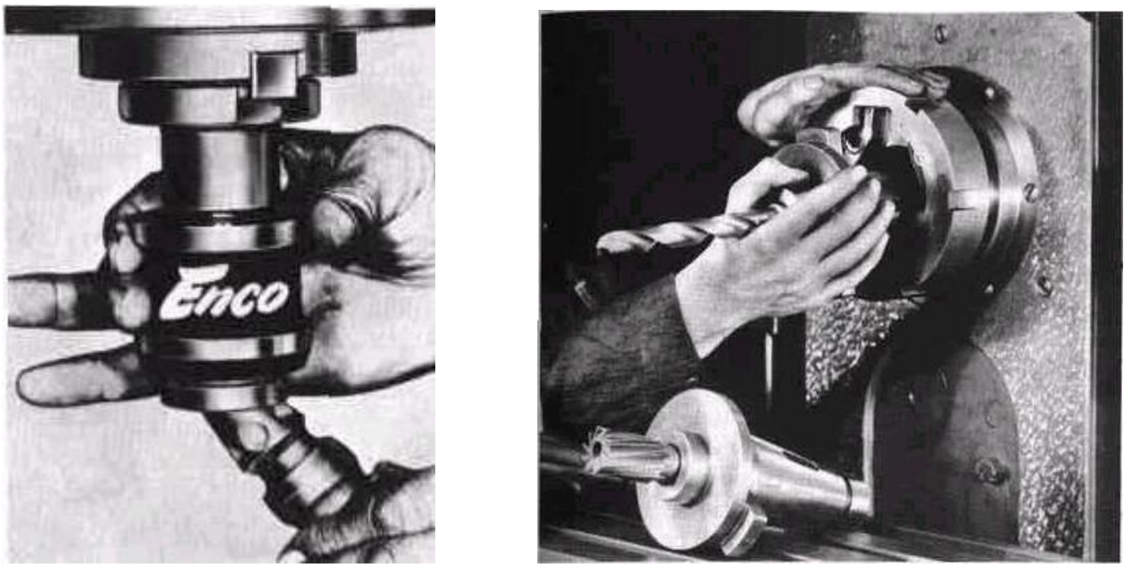
B. Holder for straight-shank end mill

Fig. 20-24. Devices for holding end mills.

Straight-shank end mills are held either in spring collets or in end mill holders, Fig. 20-24. When a spring collet is tightened, its hole is reduced in size and the collet grips the end mill shank evenly around its circumference. If the end mill is not tightened securely in a spring collet, it will slip under the pressure of cutting. It may then pull out of the collet and damage the workpiece, the cutter, or both.

End mill holders for straight-shank end mills use a setscrew to lock the end mill in place, Fig. 20-24B. Take care to ensure that the setscrew is tightened against the flat surface provided on the end mill, see Fig. 20-7.

Taper-shank end mills are held in adapters that have holes with matching tapers. If the taper shank on the tool is smaller than the tapered hole in the adapter, a reducing sleeve is used in the adapter. Shell end mill adapters are made in several standard sizes to accept the different size shell end mills. Many **quick-change toolholding systems** are now available, Fig. 20-25. They speed tool-changing and thereby increase valuable machining time. A full range of adapters is available for holding end mills as well as drill chucks, drills, reamers, and other common drilling tools.



A B

Fig. 20-25. Quick-change tool systems for milling machines using a ball bearing (A) and a flange-and-ring (B) locking mechanisms.

Plain, side, angle, and form-relieved cutters are used mostly on horizontal milling machines. They are usually held on either a style A or style B **arbor**, Fig. 20-26.

Style A arbors have a pilot at the outer end. The pilot fits in a bearing in the style A arbor support, which is suspended from the overarm. This type of arbor is used chiefly in smaller milling machines and primarily for light-duty milling applications. It provides the distinct advantage of allowing the work to be brought up close to the arbor. Hence, in many setups, small-diameter milling cutters may be used more readily than with the style B arbor. It also is possible to use an inner arbor support with the style A arbor. With this setup, a bearing sleeve is keyed to the arbor and runs in the bearing of the inner arbor support.

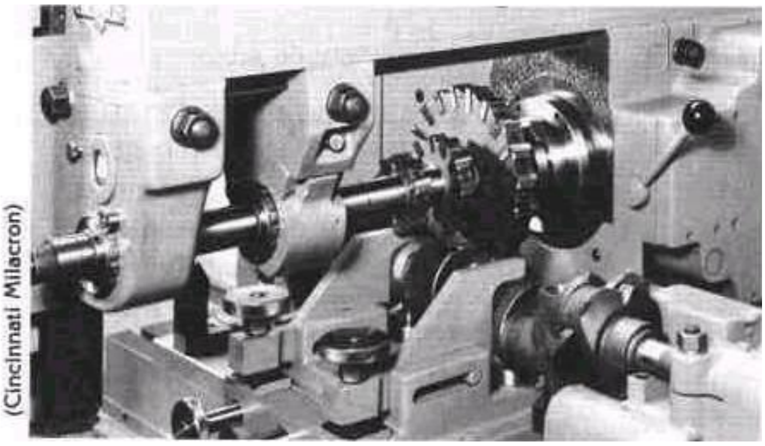

(Cincinnati Milacron)

Fig. 20-26. Milling a crankshaft in a special fixture. A style B arbor is used with two arbor supports.

Style B arbors do not have a pilot. Rather, they are provided with one or more bearing sleeves which are keyed to the arbor. The bearing sleeves run in the bearings of style B arbor supports, as illustrated in Fig. 20-26. Style B arbors are used on both large and small machines for heavy-duty milling operations where maximum clearance is not required under the arbor supports. Care must be taken to see that the bushings in the support fit the bearing sleeves properly. If they are too loose, chatter will develop; if they are too tight, heat will develop.

Spacing collars are provided on both style A and style B arbors for spacing the cutters and the bearing sleeves. They also keep the arbor straight and rigid. The collars are precision-ground and are lapped on the faces to hold the arbor straight.

Style C arbors are also called shell end mill arbors. They are used for holding shell end mills and face mills that are too small to be bolted directly to the spindle nose of the machine.


(Cincinnati Milacron)

Fig. 20-27. Milling machine spindle adapters: face mill and collet adapters.

Adapters are devices that are used to mount cutters of various types and sizes on a milling machine spindle. The arbor adapter (Fig. 20-27) is used to mount face mills on the spindle. The collet adapter is used for mounting end mills on the spindle. The tapered hole in this adapter is a self-holding type, the most common being the Morse taper and the Brown and Sharpe taper.

Most manufacturers have adopted the **national milling machine taper**, which is available in four sizes for use on milling machine arbors. These sizes are designated by the numbers 30, 40, 50, and 60. The No. 50 taper is the most common on production machines. No. 30 and No. 40 tapers are used on smaller machines.

Standard milling machine tapers are steep tapers of the self-releasing type. The amount of taper is 291.67 mm/m (16° 36' included angle). Arbors or adapters with this type of taper must be retained in the spindle socket with a locking device such as a **collar** or a **draw-in bolt**. The draw-in bolt is also called a **drawbar**. Positive drive is provided by two keys bolted to the face of the spindle. The keys engage the slots in the backs of arbors, adapters, and face mills.

20.4. Cutting Speeds and Feeds for Milling

Cutting speed for milling and other rotating tools is circumferential speed of the milling cutter, expressed in meters per minute (mpm). It is the distance that a point on the circumference of a milling cutter tooth travels in one minute. If the cutting speed is too high, the cutter becomes overheated and dulls rapidly. If the cutting speed is too low, the production rate is low and inefficient.

There is no exact cutting speed for milling any single type of material. It is common practice to start with an average cutting speed, which then increased or decreased according to the results produced.

One of the most important factors determining cutting speeds is the **machinability** which refers to the ease with which the metal may be machined. Machinability ratings are normally compared with that of 1112 steel, which is rated at 100 percent. 1015 steel (a low-carbon steel) is rated at 50 percent. As the machinability rating increases, the cutting speed also may be increased. Since the rating for 1112 steel is double that for 1015 steel, the cutting speed may be doubled also.

It is common practice to select an average cutting speed and feed for each type of material. The cutting speed and feed selected may then be increased or decreased according to conditions which affect the particular job setup and the material milled. The following factors affect the cutting speed and feed selected for milling operations:

1. Machinability rating of material being machined. Softer materials generally are drilled at higher cutting speeds than harder materials. See Table 20-1.

2. Kind of cutting tool material to be used. Carbon-steel tools are used with cutting speeds which are about one-half those used with high-speed steel tools. Cemented carbide tools generally are run at higher speeds and lighter feeds than high-speed steel mills.

3. Whether or not a cutting fluid is used, and, if so, the kind of fluid. The use of a cutting fluid is recommended for all milling operations on steel, aluminum, and

copper alloys. Gray cast iron may be machined dry, or it may be cooled with compressed air.

4. The size and the type of milling machine used, and the rigidness of the work setup.
5. Type of cutter, its size, and the coarseness of the teeth.
6. Amount of metal being removed (rough cut or finish cut).
7. The quality of finish desired on the surface.

Extreme speed or feed will cause mills to chip or break at the cutting edges. Similar damage also may result from improper grinding. **Rapid wearing at the outer corners of the cutting edges** usually is an indication of too much speed, the colour of steel chip becomes blue - it is unpermissiblly for high-speed steel.

First of all, it is necessary to select cutting feed in accordance with the mill diameter, material milled and its hardness looking at the handbook table for machinists. The feed is the rate at which the work is moved past the cutter. The feed rate, in conjunction with the width and depth of cut, determines the number of cubic millimeters of metal removed per minute.

Table 20-1. Feeds per tooth S_z (mmpt) for milling with high-speed steel cutters

Material	End mills[1]	Shell end and face mills	Plain mills		Slitting saws	Form-relieved mills
			Heavy-duty	Light-duty		
Low-carbon steel, free-machining	0.08	0.20	0.25	0.13	0.08	0.10
Low-carbon steel	0.08	0.15	0.20	0.13	0.08	0.08
Medium-carbon steel	0.08	0.15	0.20	0.13	0.05	0.08
High-carbon steel	0.05	0.10	0.10	0.05	0.05	0.05
Stainless steel, free-machining	0.08	0.20	0.20	0.15	0.05	0.05
Stainless steel	0.05	0.10	0.15	0.10	0.05	0.05
Cast iron, soft	0.10	0.20	0.31	0.20	0.10	0.10
Cast iron, medium	0.08	0.15	0.25	0.15	0.08	0.08
Malleable iron	0.10	0.20	0.25	0.15	0.08	0.08
Brass and bronze, medium	0.08	0.20	0.25	0.15	0.08	0.08
Aluminum, wrought, cold-drawn	0.10	0.25	0.36	0.20	0.10	0.13

[1] Feeds cited are for 12 mm diameter end mills taking a 6 mm depth of cut. For finishing cuts, reduce the above feed rates by about 50%.

The tendency with inexperienced milling machine operators is to use a cutting speed that is too high and a feed rate that is too low. The feed rate should be as great as the machine, the cutter, the workholding method, and the workpiece will safely stand, and, at the same time, produce a satisfactory finish. Of course, if the feed rate is too great, the cutter may fracture or the machine or the workpiece may be damaged. Suggested feed rates per tooth per revolution S_z are included in Tables 20-1 and 20-2.

Table 20-2. Feeds per tooth S_z (mmpt) for milling with cemented carbide cutters

Material	Brinell hardness	Shell end and face mills	Slotting and side mills
Low-carbon steel, free-machining	140 - 180	0.20 - 0.50	0.15 - 0.31
Low-carbon steel	150 - 200	0.20 - 0.50	0.15 - 0.31
Medium- and high-carbon steel	120 - 180 180 - 220	0.20 - 0.50 0.15 - 0.50	0.15 - 0.31 0.13 - 0.25
Alloy steels with less than 0.3% carbon	220 - 300 125 - 220 220 - 280 280 - 320	0.13 - 0.25 0.15 - 0.50 0.10 - 0.31 0.08 - 0.20	0.08 - 0.20 0.13 - 0.31 0.08 - 0.25 0.05 - 0.15
Alloy steels with more than 0.3% carbon	170 - 220 220 - 280 280 - 320	0.13 - 0.50 0.10 - 0.31 0.08 - 0.20	0.13 - 0.31 0.08 - 0.20 0.05 - 0.15
Tool steel, annealed	. . . .	0.15 - 0.50	0.13 - 0.31
Stainless steel	135 - 185	0.20 - 0.38	0.15 - 0.31
Gray cast iron	150 - 220 220 - 300	0.20 - 0.50 0.15 - 0.31	0.13 - 0.31 0.10 - 0.20
Malleable iron	110 - 160	0.20 - 0.50	0.10 - 0.31
Zinc die-casting alloys	. . . .	0.13 - 0.50	0.10 - 0.38
Brass and bronze	100 - 150 150 - 250	0.20 - 0.50 0.15 - 0.36	0.15 - 0.31 0.10 - 0.25
Cast aluminum, as cast	. . . .	0.25 - 0.50	0.20 - 0.41
Wrought aluminum, cold-drawn	. . . .	0.25 - 0.50	0.15 - 0.41

1. For finishing cuts, these feed rates should be reduced by about 50%.

Milling-machine feed is usually expressed in millimeters of table movement per minute (mmpm) - S_m. It may also be expressed as hundredths or tenths of a millimeters per tooth for one revolution of tool (mmpt) - S_z.

$$S_m = S_z \times z \times n, \qquad (20.1)$$

where: z is amount of cutting teeth; n - rotation frequency of spindle, revolution per minute (rpm).

Suggested cutting speeds are included in Table 20-3.

The cutting speed can be calculated with the help of empirical formula for milling:

$$V = \frac{C_V \cdot D^q}{T^m \cdot t^x \cdot s_z{}^y \cdot B^u \cdot z^p} \cdot K_V\,, \qquad (20.2)$$

where: V - a cutting speed, mpm; C_V - a factor taking into account the kind of material being cut, the kind of processing, the material of the cutting tool, the range of cutting depth, width and feed, cutting fluid; T - a value of tool life, minutes; t - a depth of cut, mm; s_z - a cutting feed, millimeters per tooth per revolution, mmpt; m, x, y, u, p - exponents of degrees which are taking into account the kind of material being cut, the kind of processing, the material of the cutting tool, the range of cutting depth, width and feed; B - an width of cut, mm; z - a number of cutting teeth; K_V - a product of factors which are taking into account the condition of milled surface, influence of material being cut, material of the cutting tool.

Further information concerning factors and exponents of degrees is included in standard handbooks for machinists.

Table 20-3. Cutting speeds V (mpm) is recommended for milling

Material	Brinell hardness	High-seed steel cutters	Carbide cutters
Free-machining low-carbon steel, resulphurized, 1111, 1112	100 - 150 150 - 200	35 - 50 35 - 55	120 - 180 120 - 270
Free-machining low-carbon steel, leaded, 10L18, 12L14	100 - 150 150 - 220	30 - 70 34 - 75	75 - 150 75 - 180
Plain low-carbon steels, 1006 - 1026	100 - 125 125 - 175	25 - 45 25 - 42	90 - 180 75 - 150
Plain medium-carbon steels, 1030 - 1052	125 - 175 175 - 225	25 - 42 20 - 40	75 - 150 70 - 120
Plain high-carbon steels, 1060 - 1095	125 - 175 175 - 225	20 - 35 18 - 33	75 - 140 70 - 120
Tool steels, W1 -W7 Tool steels, H20 - H43 Tool steels, D1 - D7	150 - 200 200 - 250 200 - 250	25 - 37 12 - 25 9 - 18	90 - 105 53 - 90 30 - 60
Stainless steel, 302 Stainless steel, 430F	135 - 185 135 - 185	21 - 30 30 - 42	70 - 105 105 - 137
Gray cast iron, ASTM Class 20 Through scale: Under scale:	 100 - 140	 24 - 37 33 - 37	 100 - 150 120 - 180
Malleable iron, ferritic, 32510 Through scale Under scale	 110 - 160	 42 - 60 40 - 70	 105 - 215 120 - 245
Aluminum, cold-drawn wrought alloys	. . . .	150 - 245	300 - 550
Aluminum, casting alloys (as cast)	. . . .	180 - 300	365 - 610
Brass, 360 free-cutting, cold-drawn	. . . .	90 - 150	180 - 550
Bronze, 220 commercial, annealed	. . . .	24 - 42	55 - 85

Obtaining the recommended cutting speed from a formula, a table or chart completes the data needed to calculate the correct rotation frequency of spindle:

$n = 1000V/\pi D,$ (20.3)

where: n - rotation frequency of spindle, revolution per minute (rpm); V - cutting speed, mpm; D - diameter of a drill or a reamer, mm; $\pi \approx 3.14$.

The main milling force is **circumferential force P_z** which can be calculated with the help of empirical formula for milling:

$$P_z = 10 \cdot C_P \cdot t^x \cdot s_z{}^y \cdot B^m \cdot z \cdot \frac{K_{mp}}{D^q \cdot n^w}, \qquad (20.4)$$

where: P_z - a circumferential cutting force, N; C_P - a factor taking into account the kind of mill, the kind of material being cut, the material of the cutting tool; t - a depth of cut, mm; s_z - a cutting feed, mmpt; B - an width of cut, mm; z- a number of cutting teeth; D - a diameter of the mill, mm; n - a rotation frequency of spindle, rpm; x, y, m, q, w - exponents of degrees which are taking into account the kind of mill, the kind of material being cut, the material of the cutting tool; K_{mp} - a product of factors which are taking into account the condition of milled surface, influence of material being cut, material of the cutting tool.

The value of rest components of cutting force is calculated according to ratio, see Table 20-4.

Table 20-4. Ratios of components of cutting force

Kind of milling	$P_h : P_z$	$P_v : P_z$	$P_y : P_z$	$P_x : P_z$
Plain, end (working as plain mills), angle, form-relieved mills, and plain slitting saws				
Up milling	1.1 - 1.2	0 - 0.25	0.4 - 0.6	(0.2 - 0.4)tgω
Down milling	0.8 - 0.9	0.7 - 0.9	0.4 - 0.6	(0.2 - 0.4)tgω
Face and end (working as face mills) mills				
Symmetrical milling	0.3 - 0.4	0.85 - 0.95	0.3 - 0.4	0.5 - 0.55
Non symmetrical up milling	0.6 - 0.8	0.6 - 0.7	0.3 - 0.4	0.5 - 0.55
Non symmetrical down milling	0.2 - 0.3	0.9 - 1.0	0.3 - 0.4	0.5 - 0.55

1. P_h - force of cutting feed (horizontal force); P_v - vertical force; P_y - radial force; P_x - axial force.

2. The force, which is taking for calculation of sleeve bending, is $P_{yz} = \sqrt{P_y{}^2 + P_z{}^2}$.

The **torsion moment of cutting M:**

$$M = \frac{P_z \cdot D}{2 \cdot 100}, \qquad (20.5)$$

where: M - torsional moment of cutting, Nm; P_z - a circumferential cutting force, N; D - diameter of mill, mm.

Further information concerning factors and exponents of degrees is included in standard handbooks for machinists.

Cutting power is also calculated:

$$W = \frac{M \cdot n}{9.75}, \qquad (20.6)$$

where: W - cutting power, Watt; M - torsional moment of cutting, Nm; n - rotation frequency of spindle, rpm.

Chapter 21. Shaping and Broaching Operations

21.1. Shaping Operations

Shaping and planing produce flat surfaces with a single-point cutting tool. In shaping, the cutting tool on a shaper reciprocates or moves back and forth while the work is fed automatically towards the tool. A small amount of material is removed with each stroke. Neither the tool nor the workpiece rotates. The shaper is commonly used for machining short, flat surfaces and for making grooves and keyways. The surfaces may be horizontal, vertical, angular, or irregular.

In planing, the workpiece is attached to a worktable that reciprocates past the cutting tool. The cutting tool is automatically fed into the workpiece a small amount on each stroke. The planer is used for machining large flat surfaces and long grooves. It can machine horizontal, vertical, angular, or irregular surfaces also.

Horizontal shapers are used mostly for machining flat surfaces, which may be horizontal, vertical, or angular. **Vertical shapers** are also known as **slotters.** They are used more for machining slots, keyways, splines, and other shapes in large gears, pulleys, and flywheels that are difficult to machine. Vertical shaping attachments allow vertical shaping to be done on milling machines. A highly specialized type of shaper is the **gear shaper**.

Planers cut by moving a single-point cutting tool across the path of a reciprocating workpiece. **Double-housing planers** have two columns to support a **cross rail** on which **overhead tool heads** ride. A **side tool head** is also provided on the column on the operator's side of the machine. Cutting tools in all three tool heads often cut simultaneously to speed the machining of parts. **Open-side planers** support the cross rail from a single column. This allows wide workpieces to overhang the table on the open side if necessary.

Planers are used mostly for machining flat surfaces on workpieces too large for shapers. However, both planers and shapers may be fitted with hydraulic tracing attachments that enable them to cut curved surfaces.

Shapers and planers require many strokes of the tool or workpiece to complete a cutting operation. Because milling machines can remove metal at a more rapid rate, horizontal and vertical mills have almost completely replaced horizontal shapers and planers for production work.

Horizontal shapers are inexpensive to maintain, and they use inexpensive single-point cutting tools. For these reasons and because of their ability to machine internal shapes, horizontal shapers continue in limited use.

The size of a shaper is designated by the maximum length of stroke (cut) it can take. Shapers are also designed for light-duty, medium-duty, or heavy-duty work.

Shaper driving mechanisms use either belts, gears, or hydraulic systems. The **ram** is driven by a slotted **rocker arm.** The rocker arm is connected to the **driving wheel by a crankpin** and a **sliding block**. As the driving wheel revolves, the crankpin and the sliding block are carried through a circular path. This causes the rocker arm to move back and forth, driving the ram in a straight line.

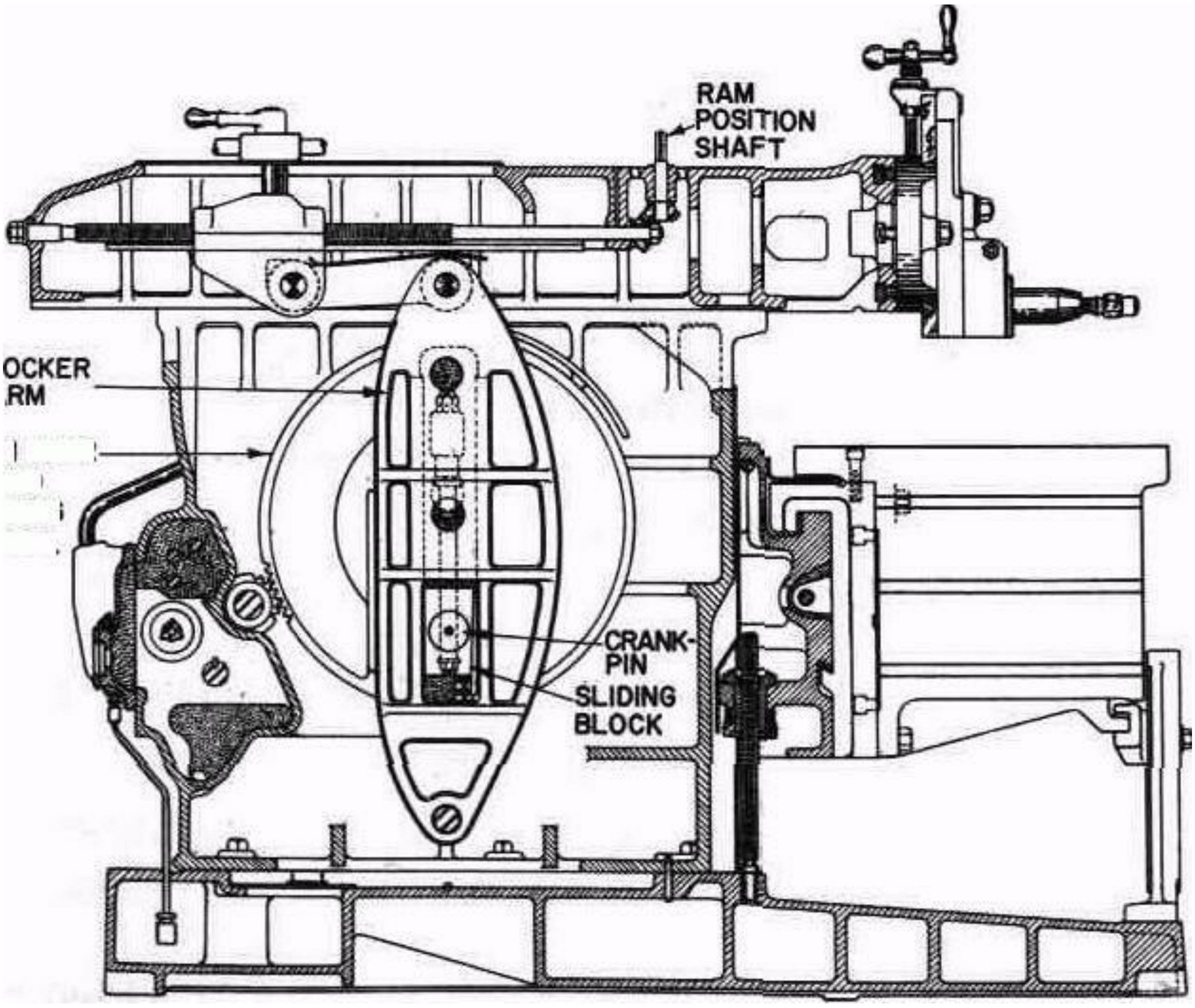


Fig. 21-1. A sectional side view of a shaper.

When set for any particular **operating speed,** a shaper will make a constant number of strokes, **regardless of the length of the stroke.** However to maintain a constant **cutting speed,** the shaper must make twice as many strokes when making a cut 76 mm long as when making one 152 mm long. On larger machines, speed change is accomplished through a system of gears. An index plate shows where to set the control levers for each speed. Some smaller shapers are equipped with a variable speed drive that is continuously variable in the range of 12 to 180 strokes per minute.

The table of a shaper is bolted to a saddle that slides in a horizontal plane along the cross rail. The saddle is moved by a longitudinal feed screw. The cross rail may be raised or lowered with a vertical adjusting screw. Before vertical adjustment of the cross rail is made, the clamping bolts holding it to the column should be loosened. If the table is to be lowered, the table support bolts must also be loosened.

After the cross rail has been adjusted, retighten the table support bolts and cross rail clamp bolts. The top and sides of the table are provided with T-slots for clamping the work to the table, or for attaching vises or fixtures.

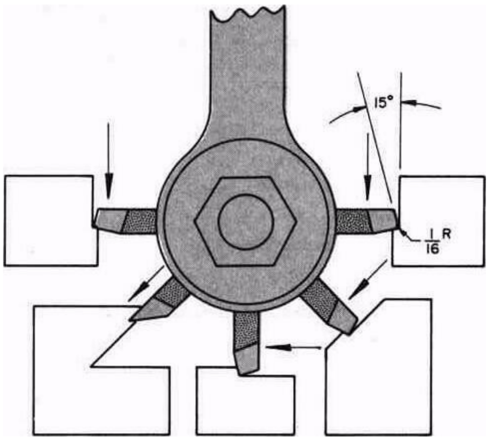
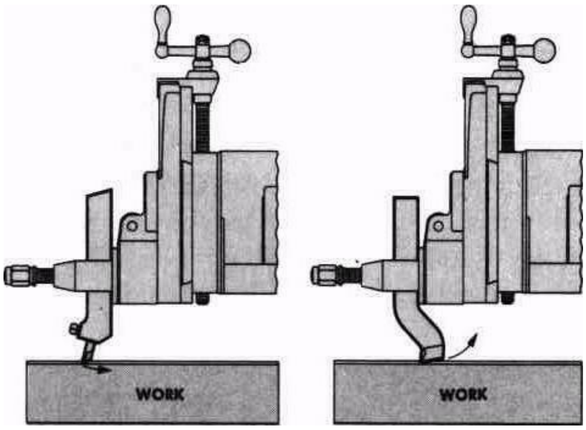


Fig. 21-2. Tool positions and types of cuts possible with a shaper or planer toolholder.

The tool head consists of a tool slide, a swivel base, an apron, a clapper box, a clapper block, and a tool post. A feed screw on the tool head has a collar graduated in hundredths of a millimeter or thousandths of an inch. The swivel base allows vertical or angular cuts. The tool is held in a tool post that is attached to the hinged clapper block. During the cut, the clapper block is forced against the clapper box and is thus solidly supported. On the return stroke, the clapper block swings free. This allows the tool to be drawn back across the work with only a slight rubbing effect, minimizing wear on the- cutting edge.

The apron may be swiveled to the left or right to cause the cutting tool to swing away from the direction of cut. However, for cutting grooves, the apron should be kept vertical.

Shapers are provided with an automatic longitudinal feed. This is controlled with either a ratchet or cam mechanism. With a ratchet mechanism, the feed rate is determined by the number of notches the ratchet pawl moves on the feed wheel for each stroke of the ram. The number of notches, in turn, is determined by the position of the knob in the feed crank wheel. On a cam-actuated feed, a direct-reading dial is set to the desired feed rate, for example 0.15 mm per stroke.



WRONG RIGHT

Fig. 21-3. The tool's position and the angle at which the tool is held affect its cutting action.

Shaper, planer, and lathe cutting tools are very similar. Two basic kinds are used: forged tools and tool bits. Tool bits are more widely used than forged tools because of their lower cost.

The terminology used for shaper, planer, and lathe tools is the same. A side-relief angle of 3° to 5° is enough for most shaper tools. An end relief of 3° to 5° is also sufficient. The side rake for shaper tools is usually 8° or 10°. Little or no back rake is needed for most applications.

Forged tools and other larger shaper tools are mounted directly in the tool post.

The type of toolholder shown in Fig. 21-4 is used for holding shaper or planer tools bits. With this toolholder, the tool bit may positioned at different angles for various types of cuts, Fig.21 -2.

The position of the tool in a planer and shaper toolholder in relation to the direction of tool travel is important. For light cuts, the tool generally is clamped at the front of the tool-holder. For moderate to heavy cuts, the toolholder and the tool are reversed. With this procedure, the cutting edge is brought back of the shank of the toolholder, causing an effect similar to a gooseneck tool. This reduces the tendency of the tool to chatter or dig in, Fig. 21-3.

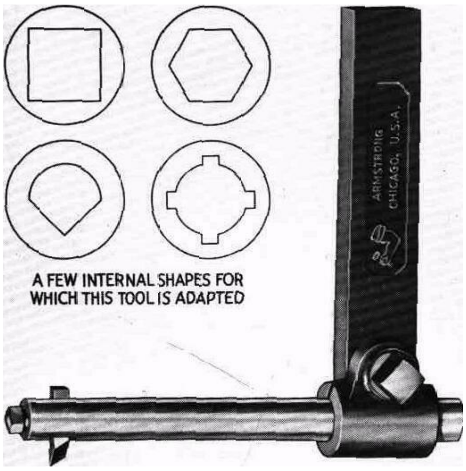


Fig. 21-4. An extension shaper tool.

An extension shaper tool consists of a boring bar mounted in a toolholder, Fig. 21-4. This tool is used for cutting internal keyways and holes of various shapes.

Calculating Shaper Cutting Speeds

On most shapers, the cutting stroke takes approximately one and one-half times as long as the return stroke. The ratio of 1:1-1/2, when each is multiplied by two, may be represented by the products 2 and 3 respectively. Thus three-fifths of each full minute of running time is spent in making cutting strokes and two-fifths in return strokes.

Cutting speed is always given in meters per minute and is determined as follows:

When: V - cutting speed in meters per minute; N - number of strokes per minute; L - length of stroke in millimeters; 3/5 - portion of time spent actually cutting; 1000 mm = 1 meter.

Then: $V = (N \times L) / (3/5 \times 1000) = (N \times L) / 600$

When the shaper is used, the cutting speed desired and the length of the stroke will be known. Since it will be necessary to determine the number of strokes per minute required to operate at the given cutting speed, the above formulas may be converted into the necessary form: $N = (V \times 600) / L$.

The length of the stroke of the ram is determined by the distance the crankpin is from the axis of the driving wheel.

In many cases, horizontal cuts may be used to machine rectangular stock on all its surfaces. In other cases, both horizontal and vertical cuts are required.

A vertical cut usually is made when cutting a groove, a shoulder or a keyway, and when planing the end of wide stock. An angular cut is made when cutting dovetails or bevels on the edges or ends of work. When such cuts are being made, the vertical feed is used.

21.2. Broaching Operations

Most broaching machines, like shapers and planers, move the cutting tool or the workpiece through a cutting stroke and a return stroke. Instead of using a single-point cutting tool, however, broaching machines use a multiple cutting edge tool called a broach, Fig. 21-5. With this tool the broaching operation is completed in one or two strokes. As a result, broaching is a very rapid, efficient process.

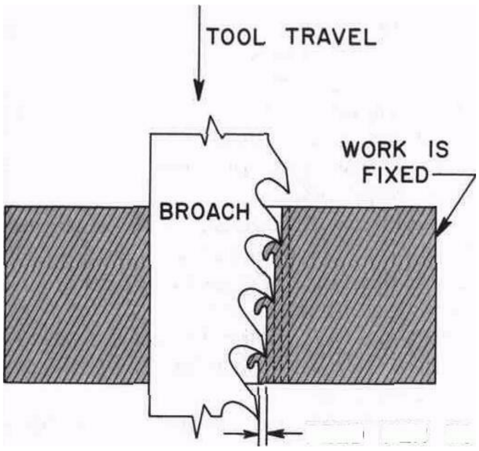


Fig. 21-5. Each tooth on a broaching tool removes a small amount of material.

Broaching machines are of two main types according to the direction of tool orientation: **horizontal** or **vertical**. On some machines the broach is fixed and the workpiece moves. Vertical machines may either push, pull down, or pull up the broach. Most horizontal machines pull the broach. To shorten the cycle time, the return (noncutting) stroke is often two or more times faster than the cutting stroke.

Pot-broaching machines are a type of vertical broaching machine made for use with pot broaches. The pot broach is a tubular broach with internal cutting teeth. Pot broaches simultaneously cut grooves, gear teeth, and other shapes around the outside of the workpiece.

Continuous broaching machines use short broaches attached to an endless chain. These machines rapidly and repeatedly cut slots, gear teeth, or similar surface shapes without moving the broach back and forth.

Rotary broaching machines are production machines that have broaches mounted around a column. The workpieces are mounted in fixtures on a rotary table that encircles the column. After each cutting stroke, the rotary table is automatically indexed to carry the workpieces to the next broach. The parts are thus gradually shaped by each broach as they move from station to station. When there is a workpiece at each station, each cutting stroke results in one finished workpiece.

Broaching operations are classified into two categories. Internal broaching is done inside a hole in the workpiece. It is used especially to form holes of complex shapes, but simple shapes are also done this way. Keyways, splines, gear teeth, and other holes with complex or irregular shapes are easily done by internal broaching. **Surface or external broaching** is done on the outside of the workpiece. Any shape from a flat surface to helical teeth can be made by surface broaching.

Broach Design

Broaches are designed so that each tooth cuts a small amount of the total metal to be removed, as shown in Fig. 21-5. If a fine finish is required, the broach will be designed with roughing, semi-finishing, and finishing teeth. Note that the broach is provided with a front **pilot** and a **rear pilot.** The front pilot insures that the broach will be accurately aligned in the starting hole provided in the workpiece. The rear pilot supports the broach in the finished hole until the broach can be removed from the workpiece.

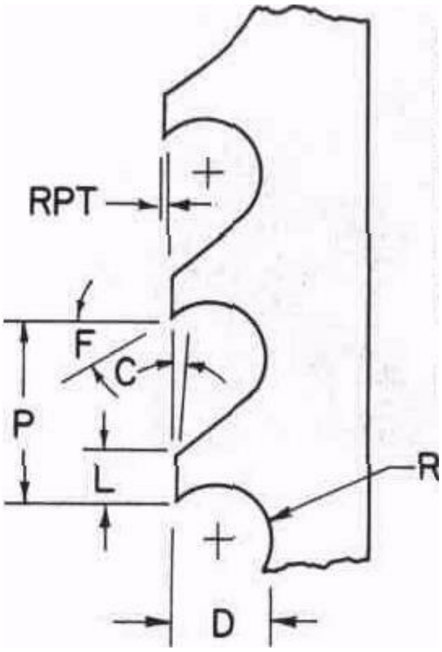


Fig. 21-6. Broach tooth design.

Each tooth on a broach is shaped much like any single-point cutting tool, Fig. 21-6, where: C - clearance angl; D - depth of tooth; F - face or rake angle; L - land width; P - pitch of teeth; R - radius of tooth gullet; RPT - rise per tooth. A shallow clearance angle of 1/2° to 4° provides maximum support for the cutting edge. A rake angle between 12 and 15 is used for cutting workpieces made of most steels.

Chapter 22. Abrasives and Grinding Wheels

An abrasive is a hard and tough substance. It has many sharp edges. An abrasive cuts or wears away materials that are softer than itself. Abrasives are used as cutting tools or cutting materials. They are used in several forms which include **grinding wheels, sharpening stones** or **sticks,** and **coated abrasives**. Coated abrasive is cloth or paper with a coating of abrasive grains cemented to its surface. Coated abrasives used for machine grinding usually are in belt or disc form. They are used in sheet or strip form for hand polishing. **Loose abrasive grains** also are used for certain **polishing** and **lapping operations**.

22.1. Properties of Abrasives

Penetration hardness, fracture resistance, and wear resistance are the necessary properties of abrasives. **Penetration hardness** refers to the ability of the abrasive to scratch or cut a softer material.

Fracture resistance refers to the ability of an abrasive material to resist breaking or cracking under load. When an abrasive grain is fractured, sharp edges should appear without loss of the entire grain. Fracture of the grain should occur after the original point has started to dull, but before it becomes too dull. Excessive resistance to fracture causes excessive pressure and heat while grinding.

Wear resistance refers to the ability of the abrasive grain to maintain sharpness. Wear resistance is largely related to penetration hardness and tensile strength of the abrasive.

Table 22-1. Relative Hardness Values of Abrasives and Other Materials

Materials	Moh scale	Knoop scale
Common glass (depending on composition)		300 - 500
Hard steel, Rockwell C. 60.5		740
Quartz	7	820
Synthetic blue spinel		1270
Topaz	8	1350
Garnet		1350
Cemented carbides		1400 - 1800
Tungsten carbide (not cemented)		1880
Aluminum oxide (Alundum) and corundum	9	2000
Silicon carbide (Crystolon)		2500
Boron carbide (Norbide)		2800
Diamond (mined or manufactured)	10	greater than 7000

The hardness of abrasives often is rated according to the **Mohs hardness scale**. The scale ranges from a rating of No. 1 for talc to No. 10 for diamond, the hardest substance known. Except for diamond, the natural abrasives rank below 9, and most artificial abrasives above 9. The following are the approximate Mohs hardness ratings for natural abrasives: Crocus, 6.0; flint, 6.9; garnet, 7.5 to 8.5: emery, 8.5 to 9.0. In the measurement of abrasives, the range from 9 to 10 actually is as broad and significant as the whole range from 1 to 9.

A more recent method, developed by the National Bureau of Standards, is the **Knoop hardness value**. The material is tested with a micro-hardness tester, which in principle is similar to the Rockwell hardness tester for measuring the hardness of metals. Under a certain load, the tester presses a diamond point into the material being tested. The depth of the impression is indicated by a number value. As with the Mohs system, the higher the number, the harder the material. However, the Knoop number more clearly indicates the relative difference in the hardness of various abrasive materials. The hardness values (100-gram load) of several abrasives and other materials ground by abrasives are indicated in Table 22-1.

Abrasives are grouped in two broad classifications - **natural abrasives** and **artificial** or **manufactured abrasives**. Natural abrasives are obtained from nature. They are being replaced rapidly by artificial abrasives. Those **natural abrasives** still used in industry include flint, garnet, emery, crocus, and diamond. Except for diamond, the natural abrasives are relatively soft in comparison with artificial abrasives. Flint and garnet are used in the form of coated abrasives in the woodworking industry. Emery, crocus, and diamond are used to work metals. For this reason, their properties and uses should be understood.

Crocus is a reddish-brown oxide of iron and may be natural or synthetic. It is used in very fine powder form as a rouge, or as a coating on cloth known as crocus cloth. Crocus cloth or rouge is used for polishing corroded metals or rare metals where a minimum of base metal is to be removed.

Emery is one of the oldest natural abrasives used in the metalworking industry. It is black and is composed largely of a combination of corundum and iron oxide. Corundum is aluminum oxide, Al_2O_3. Emery used for abrasives usually is composed of about 60% corundum and 40% iron oxide and other impurities. Emery is used in making the coated abrasive, emery cloth. Although emery cloth still is manufactured in four grades of fineness, artificial abrasives rapidly are replacing it for use in grinding and polishing metals. Emery grains are not as sharp or as hard as artificial abrasive grains; hence they are slower cutting.

Diamond, the hardest material known, is used in the form of grains bonded together to form an abrasive stick or grinding wheel. A diamond cluster abrasive stick or nib is used to cut or true other softer grinding wheels. Diamond grinding wheels are used to grind very hard materials such as cemented carbide cutting tools, ceramic cutting tools, glass, stone, and other types of ceramic materials. Industrial diamonds are relatively inexpensive when compared with the clear diamonds used for jewelry.

Artificial abrasives are also known as manufactured or synthetic abrasives. The commonly used artificial abrasives include silicon carbide, aluminum oxide, boron carbide, and synthetic diamond. The manufactured abrasives are harder and have greater impact toughness than any of the natural abrasives except diamond.

The first artificial abrasive was crystalline **silicon carbide**. It was discovered by Dr. Edward G. Acheson about 1891. The principal ingredients in silicon carbide are silica sand, which contains the silicon, and coke, which provides the carbon. A small amount of sawdust is added to make the mixture porous.

The properties of silicon carbide depend upon its purity in manufacture. Its hardness and sharpness are ideal. Its Knoop hardness value is approximately 2500, in comparison with diamond rated at approximately 7000. However, silicon carbide is brittle, as compared with aluminum oxide, and its grain fracturing properties limit its use to grinding specific materials. It is hard enough to cut aluminum-oxide abrasive materials.

Silicon carbide abrasives are used for grinding wheels, abrasive stones or sticks, and coated abrasives. The grinding wheels are used for materials of low-tensile strength, including: cast iron, bronze, aluminum, copper, tungsten carbide, rubber, glass, marble, ceramics, pottery, plastics, magnesium, and fiber.

About 1897, several years after silicon carbide was developed, **aluminum oxide** was discovered by Charles P. Jacobs, an engineer in the laboratories of the Ampere Electro-Chemical Company at Ampere, New Jersey. The principal ingredient used in manufacturing aluminum oxide is bauxite ore. This is the same material from which metallic aluminum is derived. The bauxite ore is purified to crystalline form by heating to extremely high temperatures in large electric furnaces. Greater toughness is imparted to the aluminum oxide by adding titanium.

The properties of aluminum oxide are dependent on its purity in manufacture. Aluminum oxide of 99% purity is available with modern manufacturing methods. Since the addition of titanium imparts varying degrees of toughness, several types of aluminum oxide are available with slightly varying characteristics.

Aluminum oxide is not as hard as silicon carbide. The Knoop hardness value of a typical aluminum oxide material is 2000, in comparison with silicon carbide at 2500 or diamond at approximately 7000. However, aluminum oxide is tougher and more shock resistant than silicon carbide.

Aluminum oxide abrasives are used for grinding wheels, abrasive sticks and sharpening stones, and coated abrasives. The grinding wheels are tough and shock resistant. They are used for grinding materials of high-tensile strength, including: carbon steels, alloy steels, soft or hard steels, wrought iron, malleable iron, and tough bronze. Approximately 75 percent of all grinding wheels used today are manufactured from aluminum oxide.

A third kind of artificial abrasive material is **boron carbide (norbide)**. A typical Knoop hardness value of boron carbide is 2800, which is harder than silicon carbide, but not as hard as diamond. It is produced from coke and boric acid at tremendously high temperatures in an electric furnace.

Boron carbide is used in stick form to dress grinding wheels 254 mm in diameter or smaller. It is also used in powder form in place of diamond dust for lapping operations on very hard materials, such as hardened steels and cemented-tungsten-carbide materials. Norbide is very resistant to hard wear. It is also used in solid form for such items as linings for nozzles used in high-pressure sandblasting.

Synthetic diamonds were first produced in 1955 by the General Electric Company. They were made by compressing graphite to pressures of nearly 140,909 kg per square centimeter while heated to temperatures in excess of 2760^{o} C.

Synthetic diamonds are not of gem quality but are suitable for industrial applications. Most are so small that they are used as manufactured. They are used mostly for diamond grinding wheels with grit sizes from 80 to 500.

22.2. Grinding Wheels

Grinding wheels are made of thousands of crushed abrasive grains held together by a substance called a **bond**. Between the abrasive and the bonding material are pores or air spaces. These spaces provide clearance for chips removed in the grinding process, and they minimize wheel loading.

Each abrasive grain in a grinding wheel is a cutting tool. Each has sharp cutting edges which cut off tiny particles from the metal being ground. Under a magnifying glass, the small particles appear as metal chips similar to chips removed by a shaper or lathe tool. Because of the heat created by the speed of metal removal, the chips appear as sparks which are readily visible.

On **finish-grinding** operations, grinding wheels remove metal relatively slowly in comparison with other cutting tools. Finish grinding usually follows other rough-machining operations, and it generally involves machining to very close tolerances.

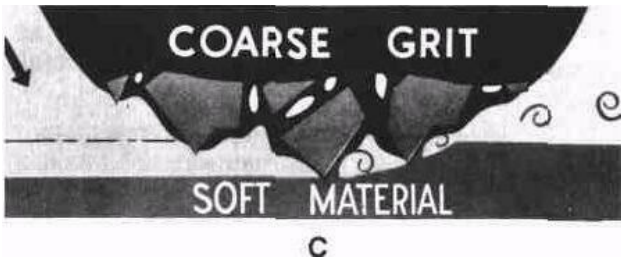


Fig. 22-1. Cutting action of a grinding wheel.

On **abrasive-machining** operations, metal is removed more rapidly than on finish-grinding operations. The term **abrasive machining** is used for grinding applications which involve the removal of a relatively large amount of metal, usually 1.5 mm or more in depth. Both grinding wheels and coated abrasives are used. Abrasive machining involves machining castings, forgings, weldments, and bar stock to commercial tolerance and finish without previous machining operations. It is used in applications which are machined more profitably by grinding than by other machining methods.

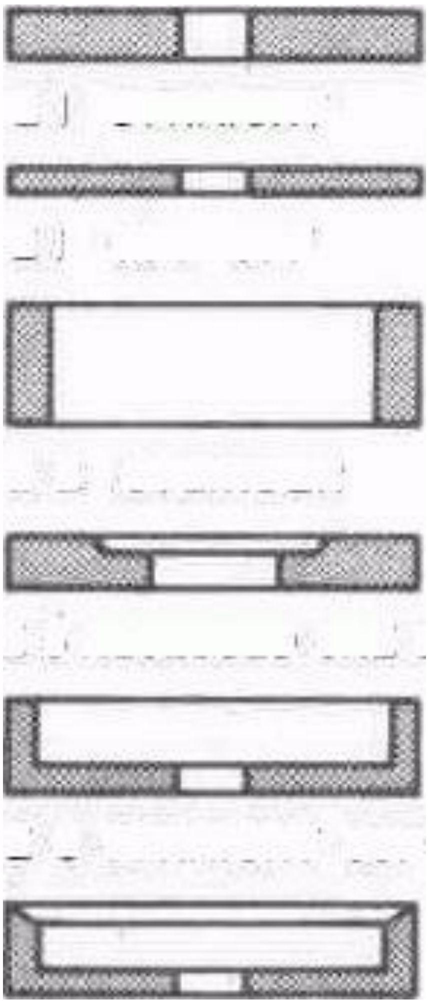


Fig. 22-2. Some standard grinding wheel shapes with type-number designations.

The quality of the work achieved by grinding is controlled to a large extent by selecting a proper grinding wheel. As a grinding wheel is used, the cutting edges of the abrasive grains become dulled. When this happens, the grinding pressure should cause these dull edges to break off, exposing new sharp edges without breaking off the entire grain. Once the grain has been broken down sufficiently, the grinding pressure should cause the bonding material to release the remaining portion of the grain, thereby exposing a new, sharp grain. This process should continue repeatedly when the right grinding wheel is selected for the job.

Grinding wheels are classified according to their size and shape, type of abrasive used, grain size, type of bond, grade or hardness, and structure.

The **size of a grinding wheel** is given in terms of its outside diameter, the diameter of the spindle hole, and the width of the face. Grinding wheels are made in many standard shapes and sizes.

Manufacturers have adopted standard **type-number** designations for most of the basic **shapes** for grinding wheels. Cross-sectional views of shapes most commonly used in toolroom and cylindrical grinding are shown in Fig. 22-2.

Grinding wheel faces may be shaped for grinding contoured surfaces on cutting tools such as milling cutters, taps, and special tools. Grinding wheels which are manufactured with standard wheel faces are designated by letters.

The most common **types of abrasive** used in grinding wheels are aluminum oxide, silicon carbide, and diamond. Diamond grinding wheels are treated separately later in this unit.

Aluminum oxide grains or crystals, although not the hardest artificial abrasive, are tough and are best for grinding materials of high-tensile strength. They are used to grind carbon steels, alloy steels, soft or hard steels, cast-alloy cutting tools, wrought iron, and tough bronze.

Silicon carbide abrasive grains are harder and more brittle than aluminum oxide. Its grain fractures more readily than aluminum oxide. Therefore, it is used to grind materials that are easily penetrated, such as copper, aluminum, rubber, plastics, magnesium, and fiber. It is also used to grind hard materials of low-tensile strength, such as cast iron, cast bronze, glass, marble, ceramics, and pottery. Cemented-tungsten-carbide cutting tools must be ground on either silicon carbide or diamond grinding wheels.

Grain refers to the size of the abrasive particles used in the manufacture of the grinding wheel. The **grain size** is determined by the **mesh number** of the finest screen through which the grain will pass. For example, a 36-grain wheel is one made of particles of abrasive which just pass through a 36-mesh screen, but which will be retained on a 46-mesh screen, the next finer screen. (A 36-mesh screen has 36 openings each lineal 25.4 mm, or 200 openings per square centimeter. Grain numbers are sometimes called **grit numbers**.

Grain sizes vary from from 10 to 600 and are classified as follows:

Coarse: 10, 12, 14, 16, 20, 24

Medium: 30, 36, 46, 54, 60

Fine: 70, 80, 90,100,120,150,180

Very Fine: 220, 240, 280,320.400,500,600

Fine-grain wheels are used on small-diameter work to produce small fillets or for fine finishes. Fine wheels also are preferred for grinding hard materials, since they have more cutting edges and, therefore cut faster than coarse-grain wheels. Because coarse-grain wheels have fewer grains, and the grains cannot penetrate hard material deeply without burning, they cannot cut as rapidly on hard materials. **Coarse-grain** wheels are used for rapid metal removal on softer materials. Coarse wheels also are used for grinding large workpieces. The grain size selected should be determined by the type of material to be ground, the finish desired, and the amount of metal to be removed.

The **bond** is the material which holds the abrasive grains together to form the grinding wheel. As the grains get dull, pressure on the wheel causes the bond to break down and release the dull grains, thus exposing new sharp grains. The bond holds the individual grain in much the same manner as a toolholder holds a tool bit. There are five basic types of bonds used in grinding wheels: **vitrified, silicate, rubber, shellac, and resinoid**. Additional modifications of these five materials are also produced by some manufacturers.

Approximately 75% of all wheels are made with **vitrified** or a modified vitrified bond. Vitrified-bond wheels are strong, porous, and are not affected by rapid changes in temperature, oils, acid, or water. These wheels are uniform in structure, free from hard spots, and hold their form well. The bond is formed when special clays are mixed with abrasive grains and heated to high temperatures. The mixture forms a molten glass which cements the grains together.

Wheels bonded with **silicate** (silicate of soda) are known as silicate- or semi-vitrified-bond wheels. Silicate-bonded wheels release the grains more readily than vitrified bond. Hence, the wheel is softer and it breaks down more readily, thereby exposing new sharp grains. Silicate-bonded wheels are used for grinding edge tools, drills, reamers, milling cutters, and similar tools.

Wheels which are **rubber** bonded are elastic, very strong, and shock resistant. This bond is used for very thin wheels, such as cutoff wheels for abrasive cutoff machines. Cutoff wheels are used for cutting pipe, angle iron, or bar stock. For safety, rubber bond is used for high-speed grinding. It also produces a very good finish.

Wheels with a **shellac** bond are also elastic in nature, resilient, and cool cutting. They produce a very fine finish. Hence, they are used to grind such items as mill rolls, camshafts, and fine cutlery.

Wheels with a **resinoid** bond have high strength and mechanical shock resistance. Resinoid bond is used for large, heavy-duty, high-speed wheels. They are used for rough grinding operations involving rapid stock removal, or for cutoff wheels. Resinoid wheels frequently are used in foundries and steel mills for snagging castings and for cleaning steel billets. (Snagging means to grind off the rough spots or surplus metal.)

Wheels from which the grit or abrasive is readily torn are termed **soft grade**. Conversely, wheels that do not readily release the grain are called **hard grade**. **Hard-grade** wheels generally are used **for grinding soft** metals such as mild steel. **Soft-grade** wheels generally are used **for grinding hard** metals such as high-carbon steel.

It should be remembered that the term hard as used with respect to grinding wheels has no relationship to the hardness of the abrasive, but rather to the ease or difficulty with which the worn particles of the abrasive are torn from the face of the wheel. With a given bond material, it is the amount of bond which determines the hardness or softness of the wheel - the more bond material, the harder the wheel.

The grade of grinding wheels is designated by letters of the alphabet, A being the softest and Z the hardest, Table 22-2.

The **structure** of a grinding wheel refers to the spacing between the grains, or the density of the wheel. Grains which are very closely spaced are more dense or close, while grains which are wider apart are less dense or open, Fig. 22-1.

The structure of a wheel is rated with numbers from 1 (dense) to 15 (open). The rate of metal removal usually is greater for wheels with an open structure. However, those with dense structure usually produce a finer finish.

The system of **grinding wheel markings** adopted by the American Standards Association is shown in Table 22-2. A separate system is used for diamond wheels. Most manufacturers use this system for identification markings on grinding wheels. The standard system for marking wheels includes six parts in sequence, as listed across the top of Table 22-2. Note that the prefix to item one in the sequence is optional for each manufacturer. For example, where several types of a given abrasive are available, such as several variations of aluminum oxide, the prefix number indicates the exact type of aluminum oxide. Also note that items four and six in the sequence are optional with the manufacturer.

Table 22-2. Standard Marking System for Grinding Wheels
(Example Is a Typical Marking: 32A46 — H8VBE)

(Prefix)	(1)	(2)	(3)	(4)	(5)	(6)
	Kind of Abrasive	Grain Size	Grade or Hardness	Structure	Bond Type	Manufacturer's Record
32	A	46	H	8	V	BE
Manufacturer's symbol indicating exact kind of abrasive	A Aluminum oxide C Silicon carbide	Coarse- 10; 12; 14; 15; 20; 24; Medium- 30; 36; 46; 54; 60; Fine- 70; 80; 90; 100; 120; 150; 180; Very Fine- 220; 240; 280; 320; 400; 500; 600	Soft to Hard: A; B; C; D; E; F; G; H; I; J; K; L; M; N; O; P; Q; R; S; T; U; V; X; Y; Z	Dense to Open: 1; 2; 3; 4; 5; 6; 7; 8; 9; 10; 11; 12; 13; 14; 15	V -Vitrified S - Silicate R - Rubber B- Resinoid O-Oxychloride	Manufacturer's private marking to identify wheel. May be a letter or number or both to designate modifcation of bond or wheel characteristics

The marking on the grinding wheel indicated in Table 22-2 is 32A46-H8VBE. This marking indicates that the abrasive is type 32 Alundum; with a 46 medium grain size; with H grade (which is rated between soft and medium); structure 8 (middle density); bond type V (which is vitrified); and BE represents the manufacturer's mark for the specific type of vitrified bond. A grinding wheel of this type will do a good job in surface-grinding hardened carbon tool steel.

Several manufacturers may use the same number to identify a given type of grinding wheel, but this does not mean that all of the wheels so identified will produce equal results or equivalent grinding action. The physical properties of the materials used may vary with different manufacturers.

Grinding wheels should be operated at **cutting speeds** as near as possible to those recommended by the wheel manufacturer. Cutting speed refers to the speed at which the circumference (the cutting face) of the wheel is traveling in mpm (meters per minute) or fpm (feet per minute). The cutting speed of a wheel is increased or decreased by changing the rpm of the grinding wheel spindle.

The rpm of the wheel spindle may or may not be adjustable. Bench grinders and floor-model grinders used for offhand grinding generally cannot be adjusted. On these grinders, the wheel usually is mounted directly on the motor spindle. On some cylindrical grinders, tool grinders, and surface grinders, the rpm of the wheel can be varied through the use of step pulleys or a variable-speed mechanism.

The maximum speed at which a wheel should be operated is indicated on the wheel. This speed should never be exceeded. For example, a wheel designed for a maximum of 1800 rpm should not be used on a grinding machine which has a spindle speed of 3600 rpm. When used at speeds above those recommended, a grinding wheel may fly apart, and may cause serious injury to the operator.

The following are general recommended cutting speeds:

General offhand grinding with vitrified-bonded wheels, 1524 to 1829 mpm. Surface grinding, 1219 to 1981 mpm. Tool and cutter grinding, 1372 to 1829 mpm. Cylindrical grinding, 1676 to 1981 mpm.

Hence, an average speed of about 1524 mpm is recommended.

Specific cutting speeds to be used with different types of grinding wheels on various grinding applications are available in standard handbooks for machinists.

Diamond grinding wheels are in a class by themselves. They are used to grind cemented-tungsten-carbide cutting tools, ceramic-oxide cutting tools, wear-resistant die steel, ceramics, glass, granite, marble, and jewels.

Diamond grinding wheels are made of fine particles or grains of natural or manufactured diamond, which are held together with a bonding material. The diamond particles are graded in grain sizes ranging from 36 to 500. Wheels with 80-120 grain size often are used for rough grinding, 180-320 for finish grinding, and 120-150 for combination rough and finish grinding. For fine lapping operations, diamond abrasive in bulk form as fine as 2000-grain size is available.

Diamond wheels usually are made of a special composition material to which a layer of abrasive mixture is applied on the cutting surface. The mixture is made of diamond grains and bonding materials (either metal, resinoid, or vitrified). The abrasive layer is available in thicknesses from 0.8 to 6.35 mm. It also is available in several different concentrations or proportions of diamond to bonding material - low, medium, and high. In addition, numerous grades of hardness are manufactured and are indicated by letters of the alphabet.

At the present time, there is no standard marking system used by all manufacturers of diamond grinding wheels. However, the system developed by the Morton Company is being adopted by several other manufacturers of diamond wheels, Table 22-3.

Table 22-3. Marking chart for diamond wheels and hones.
(Example Is a Typical Marking: SD100-N100B56 1/8)

(1)	(2)	(3)	(4)	(5)	(6)	(7)
Abrasive	Grit size	Grade	Concentration	Bond type	Bond modification	Depth of diamond section
Diamond: D- naturral; SD- manufactured; SND- selected natural	24; 36; 46; 60; 80; 100; 100S; 120; 150; 180; 220; 240; 320; 400; 400S; 500; 500S; 600S; 800S; 1200S; 1500S; 2000S	Resinoid: H; R; J; L; N. Metal: L; R; N; P; Q. Vitrified: J; R; L; T; N; P	25% 50% 75% 100%	B- resinoid M- metal V- vitrified	Numeral to designate special bond modification. Example: Resinoid- 56 and 11. This symbol may be omitted.	1/16; 1/8; 1/4

Since diamond grinding wheels have a high initial cost, care must be taken in selecting and using them. Cutting speeds from 1372 to 1829 mpm usually should be used. A liberal supply of cutting fluid also may be used. The manufacturer's recommendations should be carefully followed. Several manufacturers of diamond wheels supply literature without cost concerning the selection, care, and use of their wheels. Recommendations concerning the selection of diamond grinding wheels for particular job applications are also available in standard handbooks for machinists.

22.3. Selecting, Using and Ordering a Grinding Wheel

The following factors must be taken into consideration in recommending and selecting a wheel for a particular job:

1. Type of grinding operation: offhand grinding, surface grinding, tool grinding, cylindrical grinding, internal grinding, etc.
2. Material to be ground.
3. Type of abrasive and bond to be used.
4. Amount of stock to be removed.
5. Finish required.
6. Area of wheel in contact with work: a wide wheel face may require a soft-grade wheel.
7. Wheel speed.
8. Work speed: for surface grinding and cylindrical grinding.

9. Whether grinding is wet or dry.

10. Machine condition: capacity and rigidity.

11. Abrasive grain size, grade, and structure.

Recommended grinding wheels for use on a number of basic grinding applications are indicated in Table 22-2. Recommendations for other grinding applications are available in standard handbooks for machinists and from manufacturers.

In ordering a grinding wheel, one may use the standard identification number to specify the type of abrasive, grain size, grade, structure, and specific bond type. In addition, the following specifications must be indicated: wheel type, wheel size (including diameter and width of face), type of face, diameter of hole or opening at the center, and the maximum rpm of the wheel.

Precautions in Using Wheels

1. The grinding machine should be rigid in order to prevent vibration or chatter.

2. Spindle bearings on the grinding machine should be adjusted properly in order to prevent vibration and chatter.

3. Wheels always should be mounted with proper cardboard or blotting paper discs between the wheel and properly relieved steel flanges or washers, Fig. 22-3.

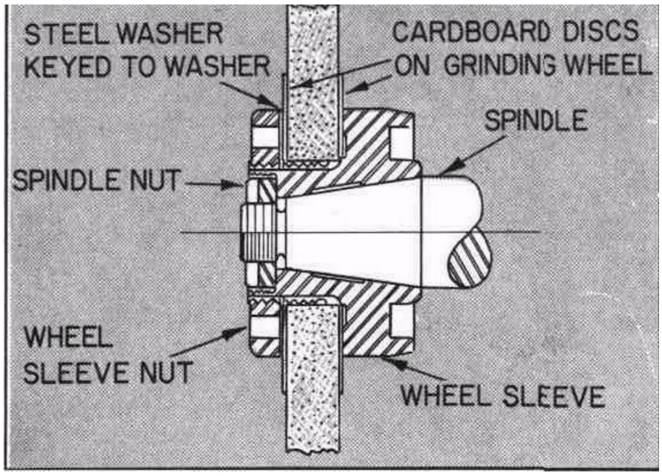


Fig. 22-3. Cross section of wheel mounted on arbor.

4. Wheels should be mounted with the spindle nut fastened snuggly. If the nut is too tight, the wheel may crack.

5. Sound the wheel before installing it in order to test for cracks. The wheel will ring when struck very lightly with a nonmetallic object. A dull thud will be heard if the wheel is cracked.

6. The wheel should not be forced on the machine spindle. If the wheel is too tight, scrape the inside of the wheel bushing lightly and evenly all around until it will just slide on.

7. A new grinding wheel should be allowed to run at full speed, with the operator standing to one side, before it is used.

8. Wheel glazing is indicated by a smooth, glass-like appearance. It is caused when the abrasive grains wear too much before being released. This condition may be corrected by using a wheel of softer grade.

9. Wheel loading is caused by grinding a soft metal with the wheel or by using too heavy a grinding action. The wheel must be cleaned with a wheel dressing tool.

10.The wheel must be kept true and in balance.

22.4. Coated Abrasives

A coated abrasive is composed of a flexible backing material to which abrasive grains are cemented. The coated abrasives used in the metalworking industry include emery, aluminum oxide, silicon carbide, and crocus.

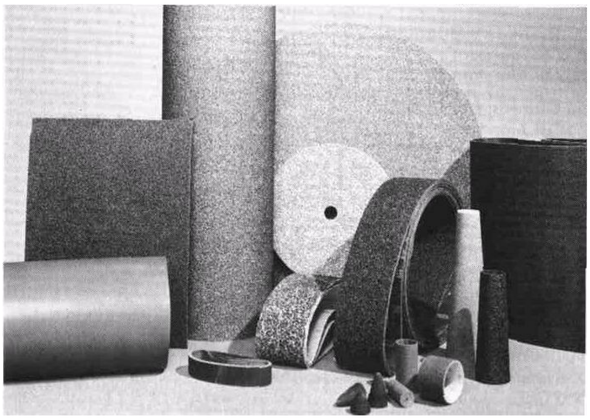


Fig. 22-4. Commonly used coated abrasives.

Coated abrasives used for metalworking and other industrial jobs are available in the form of belts, rolls, sheets, discs, spiral points, and cones. Sheets usually are used for hand polishing, while the other forms are used for machine grinding and polishing.

Coated abrasives differ in the following characteristics: the abrasive material, the backing material, the bonding material, the method of coating, and the size of the grains. Each of these factors must be considered in selecting and purchasing the type to be used.

The type of abrasive selected is determined by the type of material to be cut or polished, the amount of material to be removed, and the quality of finish desired. **Silicon carbide** abrasive is used on metals of low-tensile strength, such as cast iron, aluminum, brass, and copper. It also is used on plastics, glass, and marble.

Aluminum oxide abrasive is used on metals of high-tensile strength, such as carbon steel, alloy steel, stainless steel, and tough bronze. When considerable material is to be removed and a fine finish is desired, several grades of abrasive must be used - from coarse to medium to fine. **Crocus** cloth is made with a very fine crocus powder. It is used for polishing corroded metals or rare metals where a minimum of base metal is to be removed.

The backing materials used for coated abrasives include paper, cloth, fiber, or various combinations of these materials. Paper backing is used largely for hand applications on woodwork. Cloth or fiber is used as backing on coated abrasives used in metalworking and for industrial machine grinding and polishing applications on other materials.

The **cloth backing** used for coated abrasives is of two types: a flexible lightweight cloth, called Jean, and a heavier weight cloth, called **drill**. Drill is more stretch resistant and is used for machine grinding or polishing with belt or disc machines. **Fiber backing** is strong and durable and is used for tough disc applications.

Several types of adhesive materials are used to hold the abrasive grain to the backing material. Some are only suitable for use dry, while others may be used either wet or dry. **Hide glue** is used as a bond for dry, light, hand or mechanical polishing. **Synthetic resins** are used for waterproof cloth and for wet belt grinding and polishing. In addition, many modifications or combinations of glue, resin, and varnish are used for **bonding materials**. These materials add toughness, heat resistance, and moisture resistance to the abrasive cloth or belt.

Coatings are of two basic types - open coating and closed coating. With an open coating, part of the backing surface is not covered with abrasive grains, thus leaving open space which resists filling or clogging when used with certain materials. With a closed coating, all of the backing surface is covered. Closed coating is used for cutting operations which involve a high rate of stock removal. Special heavy-duty coatings of adhesive and abrasive grain also are available for severe grinding operations.

The system used for designating abrasive grain sizes for coated abrasives is essentially the same as that used for grinding wheels. However, since coated abrasives have more of the grain exposed, the relative rating for coarseness is different. The following abrasive grain sizes, listed according to relative coarseness are used for coated abrasives. Extra coarse: 12, 16, 20, 24, 30, 36. Coarse: 40, 50. Medium: 60, 80, 100. Fine: 120, 150, 180. Extra fine: 220, 240, 280, 320. 360, 400, 500, 600.

Polishing with Abrasive Cloth

1. Tear a strip of abrasive from a sheet or roll.

2. For hand use, place the abrasive strip under a piece of wood. Apply a few drops of oil to the abrasive. Using straight strokes, polish the workpiece in the direction of the longest dimension of the surface.

3. When considerable stock is to be removed and a fine finish is to be produced, use several grades of abrasives. First, use medium, follow with fine, and finish with an extra-fine grade.

4. When polishing work on a lathe, use a strip of abrasive cloth under a stick of wood or a flat file. Apply several drops of oil to the abrasive, and polish with overlapping strokes. Another method of polishing is to loop the abrasive around the workpiece and, holding one end of the strip in each hand, pulling the strip back and forth with a slow sawing motion against the revolving workpiece. Be careful to avoid striking the lathe dog or the chuck with the fingers or hands. A high speed should be used.

22.5. Abrasive Machining

Grinding machines are precision machine tools. They machine metal parts to very close tolerances. They produce high-quality surface finishes. Grinding machines are available for grinding flat surfaces, external and internal cylindrical surfaces, tapered surfaces, and irregular surfaces.

It is common industrial practice to grind many mass-produced parts to tolerances of plus or minus 0.02 mm. Special parts for precision instruments are ground to tolerances of plus or minus 0.0005mm. Another distinct advantage of grinding is that it often is the only method by which parts hardened by heat treatment may be machined. Precision-ground parts range in size from the small parts in a wristwatch to the large rolls used for rolling sheet steel in steel mills.

SURFACE GRINDING

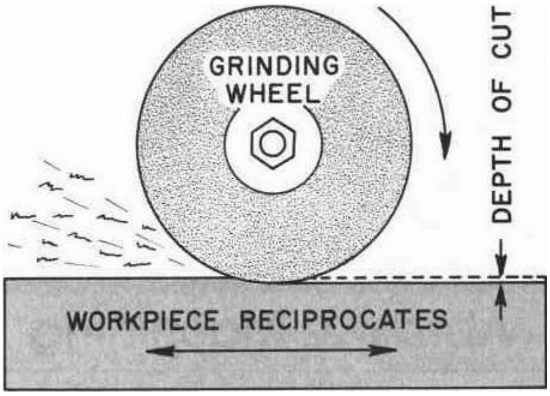


Fig. 22-5. Relationship of grinding wheel and work-piece with horizontal-spindle surface grinder.

There are several basic kinds of precision grinding operations.

Surface grinding produces an accurate flat surface on a part. Several types of surface grinders are made, Fig. 22-5. **Cylindrical grinding** produces a cylindrical or conical shape on a workpiece. The workpiece is mounted between centers or in a chuck, and the face of the grinding wheel passes over the external surface of the revolving piece, Fig. 22-6. Cylindrical grinding is done on either plain or universal cylindrical grinding machines.

CYLINDRICAL GRINDING

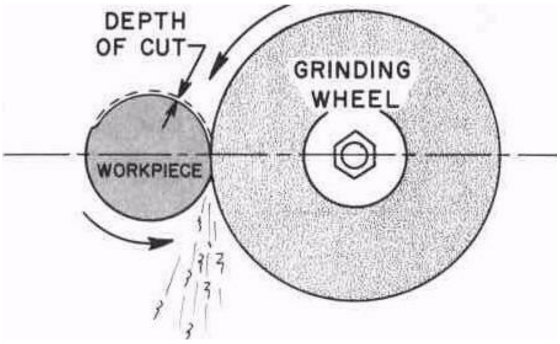


Fig. 22-6. Relationship of grinding wheel

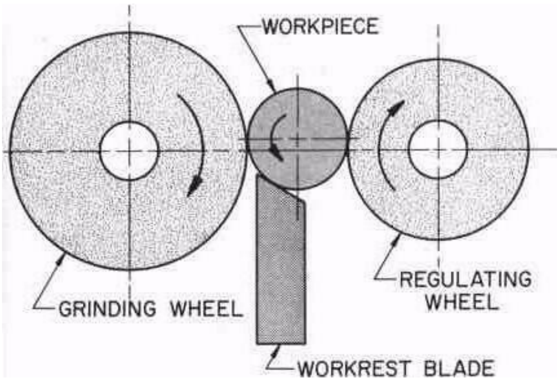


Fig. 22-7. Relationship of grinding wheel and work-piece in cylindrical centerless grinding.

Internal grinding produces a smooth and accurate surface in a cylindrical hole. The surface may be straight, tapered, or irregular. Internal grinding is a form of cylindrical grinding. This type of grinding may be done on universal grinding machines, internal grinding machines, and with tool post grinders mounted on a lathe.

Form grinding produces a smooth and accurate surface of a special shape. It is done with a grinding wheel which usually is shaped to conform to the contour of the surface it is designed to produce. An example of form grinding is the grinding of a thread from solid stock. The grinding of fillets, rounds, or irregular shapes is another example. Form grinding may be performed with various types of grinding machines, including surface grinders, cylindrical grinders, internal grinders, and special grinding machines.

Plunge grinding is another form of cylindrical grinding which may produce a straight, tapered, or formed surface on a workpiece. Instead of traversing the workpiece past the wheel as is usually the case, the wheel is fed directly into the revolving workpiece with little or no side movement. Automotive crankshafts and similar objects with deep shoulders often are ground by this method. A cylindrical grinding machine equipped with a plunge-type grinding wheel head generally is used for operations of this type.

Centerless grinding is a form of cylindrical grinding. It produces accurately ground parts without requiring them to be mounted between centers. Parts are held in position on a workrest blade which is located between a grinding wheel and a regulating wheel, Fig. 22-7. The regulating wheel rotates the work which rests on the blade. At the same time, the grinding wheel grinds the surface. Straight or tapered objects may be ground in this manner.

Lathe centers, piston pins, roller bearings, and similar objects without center holes are examples of objects ground by the centerless grinding method.

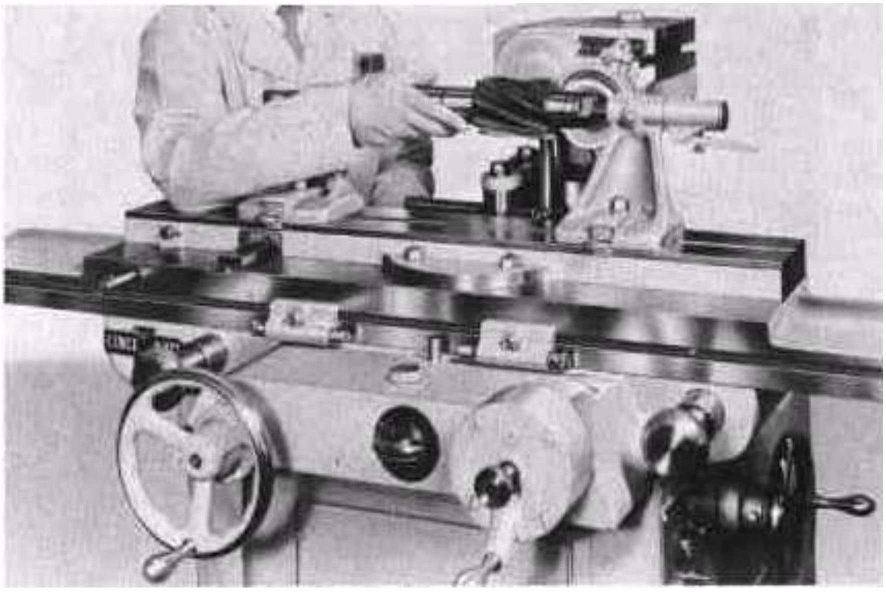


Fig. 22-8. Grinding a plain milling cutter on a tool and cutter grinder.

Tool and cutter grinding involves the grinding of milling cutters, counterbores, reamers, and many other kinds of cutting tools. This type of grinding normally is done on a tool and cutter grinder, Fig. 22-8.

Offhand grinding is the nonprecision type of grinding done on bench- or floor-model tool grinders. Cold chisels, center punches, lathe tool bits, and shaper tool bits often are ground by offhand grinding. The tool or object being ground is held by hand, with or without a guiding device.

22.6. Kinds of Grinding Machines

Like other metalworking machines, grinding machines designed for many purposes are available. In this unit, the most common types of grinding machines, their applications, and their principal parts are described briefly. More detailed information concerning machine accessories, controls, and other operational factors is explained in succeeding units which include the procedures for specific types of grinding operations.

Grinding machines are often classified as to size by an arbitrary number assigned by the manufacturer. All, however, can be classified in terms of their maximum capacity to accommodate a workpiece.

Surface grinding machines are designed primarily for grinding flat surfaces. However, with special setups, angular and formed surfaces also may be ground. Surface grinding machines are of two general types: the horizontal-spindle type (Fig. 22-8, 22-9) and the vertical-spindle type.

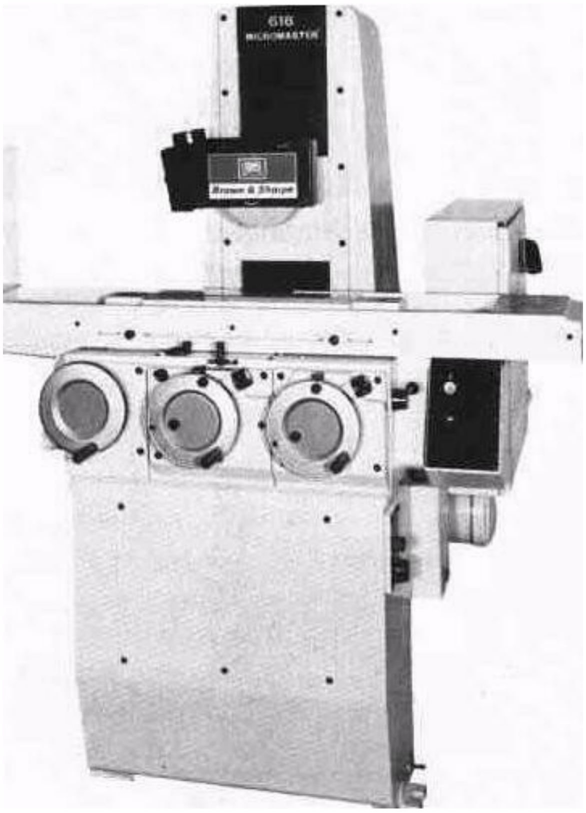


Fig. 22-9. Surface grinder with horizontal spindle.

Machines with horizontal spindles are most common in tool- and die-shops, toolrooms, maintenance shops, and schools. With this type of machine, the work may be mounted on the table in a number of ways, often with a magnetic chuck. The work table reciprocates back and forth under the grinding wheel. With each succeeding table stroke, the work is fed crosswise under the wheel. Machines of this type are available with either manual or power longitudinal and transverse (crosswise) feeds. They may be operated dry or wet.

For dry grinding, an exhaust attachment generally is used to catch the dust. For wet grinding, a special attachment pumps fluid or a mist to the grinding area.

A second type of surface grinder with a horizontal spindle is the **rotary type**. This machine has a rotary table mounted on a supporting table which travels longitudinally. The workpiece revolves under the grinding wheel, and, at the same time, the table is fed longitudinally under the wheel. Workpieces of larger diameter may be ground on machines of this type.

Surface grinders with a **vertical spindle** generally cuts much faster than those with horizontal spindles. As the grinding wheel revolves, the work is fed back and forth under the wheel. Because of the amount of heat developed with this type of grinder, a cutting fluid always should be used.

Plain grinding machines are designed primarily for production grinding of external cylindrical surfaces. They are used for cylindrical grinding of straight surfaces, tapered surfaces, and shoulders. They also may be used for plunge grinding of formed surfaces which conform to the shape of the grinding wheel.

The principal parts on a plain grinding machine are: a heavy bed which gives the machine stability; a wheel head mounted on a slide base; a headstock mounted on a table which can be swiveled through 8°; a footstock or tailstock; a sliding table on which the swivel table is mounted; a longitudinal table feed mechanism; and a manual or automatic cross-feed mechanism.

The wheel head is set permanently at right angles with the table travel and cannot be swiveled. The headstock on many plain grinding machines cannot be swiveled. On some machines the head-stock may be swiveled up to 45° for grinding steep tapers. The headstock spindle has four step pulleys which provide work speeds ranging from about 200 to 800 rpm.

Longitudinal table travel may be operated manually or automatically. Six rates of power travel are provided, ranging from about 152 mm to 3378 mm per minute. The feeds are selected through the use of table speed-selector levers. Table reverse dogs may be set for automatic table reversal. Plain grinding machines often are equipped with features such as wheel slide rapid travel, independent automatic cross feed, and a wheel spindle reciprocating mechanism.

A **universal grinding machine**, Fig. 22-10, is far more versatile than the plain grinder. It is designed to perform both external and internal cylindrical grinding operations. It can grind straight surfaces, tapered surfaces, shoulders, steep tapers, and face grinding. Also, straight fluted reamers and milling cutters can be ground on a universal grinder.

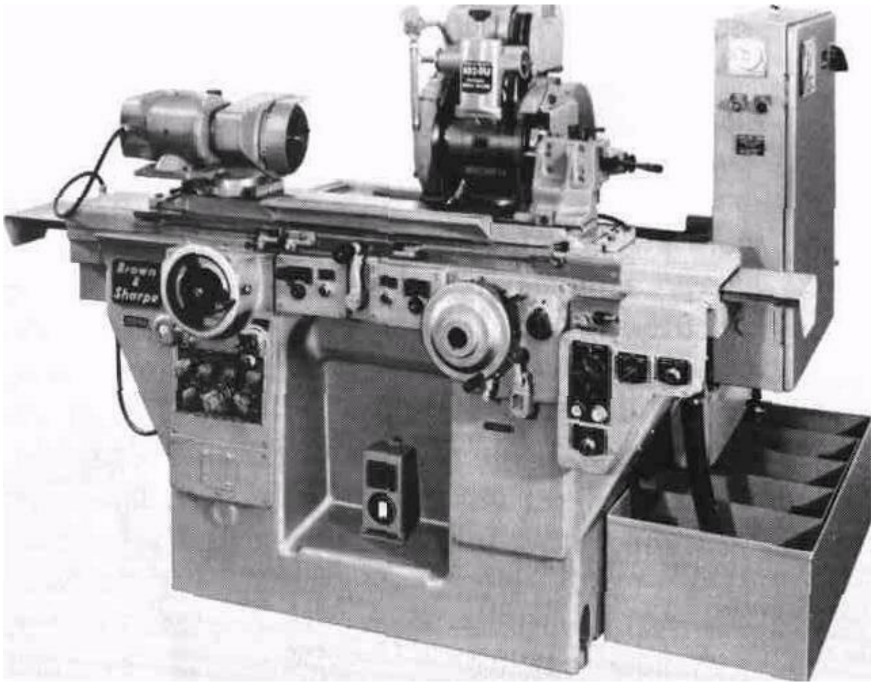


Fig.22-10. Universal grinding machine.

Universal grinding machines are made in sizes ranging in swing capacity from 254 mm to 355 mm or more. Their capacity between centers ranges from 508 mm to 1524 mm or more.

The principal parts of a typical universal grinding machine include: a heavy base which gives the machine stability, a table which can be swiveled 8° for grinding tapers, a wheel spindle head which may be swiveled for grinding angles, a headstock which may be swiveled for grinding steep angles or faces, a footstock for holding work between centers, and an internal grinding unit which is mounted directly above the external grinding spindle.

A variety of work-holding devices and accessories also is available to extend further the versatility of this machine. Work may be mounted in a chuck on the headstock spindle, for either face or angular grinding.

The **internal grinding machine** is used for finishing cylindrical or tapered holes. It is a highly specialized machine which rarely is found in schools or small commercial shops. In such shops, internal grinding usually is done with a universal grinding machine equipped with an internal grinding fixture. Internal grinding also may be done with a tool post grinder mounted on a lathe.

The **tool and cutter grinding machine** is designed for grinding milling cutters, reamers, taps, and other precision cutting tools used on milling and drilling machines, Fig. 22-8. When equipped with the appropriate accessories, tool and cutter grinding machines also may be used for accurately grinding single-point cutting tools.

The universal and tool grinding machine is a general-purpose machine which may be used for an unusually large number of grinding applications. It is an extremely versatile machine which is particularly useful in toolrooms, small commercial shops, and schools. It will perform small and medium cylindrical grinding operations which normally are performed on a universal grinder, including both external and internal grinding operations.

The machine can also be used for grinding all of the common cutting tools (such as milling cutters, reamers, and taps) and many special cutting tools. Numerous accessories and holding devices are available for the machine, including a universal chuck, internal grinding attachment, wet grinding attachment, surface grinding attachment, collect chuck, magnetic chuck, index centers, end mill sharpening attachment, and many convenient accessories commonly used in conjunction with cutter sharpening.

When many parts are to be ground to the same size on a cylindrical grinding machine, the size can be quickly checked with an indicating grinding gage. **This gage indicates the size of the work while the machine is running**.

Several types of dial indicating snap gages may also be used to gauge the work while mounted in the machine. These gages indicate size directly and are convenient for measuring work in the grinder. Go and not-go snap gages also may be used. However, gages of this type show only whether the workpiece is within specified tolerances.

General Procedure

1. Select a grinding wheel suited to the work to be performed, and mount it on the wheel spindle. Be sure the wheel is sound. If the wheel is changed, it will be necessary to true and balance the new wheel before attempting to grind a workpiece. If the workpiece is to be form-ground, it will be necessary to shape the wheel accordingly.
2. Mount the work on the table of the machine: a) by clamping; b) by using a permanent magnetic chuck. For safety, use back or end stops if the workpiece has only small contact with the chuck. Flat pieces of metal that are thinner than the workpiece make good stops; c) by using a vise. Plain, swivel, and

tilting vises may be attached to the worktable or held on a magnetic chuck. Workpieces may be held at any angle required; d) by using index centers; e) by using a sine plate or a perma sine that may be used when workpieces are to be ground to very precise angles. The work is attached to the sine plate by bolting or clamping; the perma sine holds the work-piece magnetically; f) by using V-blocks for round workpieces.

3. Properly lubricate the machine before starting it. If the machine has been idle for some time, alternately press the start and stop buttons in rapid succession three or four times before running the machine at operating speed.

4. Protect eyes by wearing properly fitted goggles.

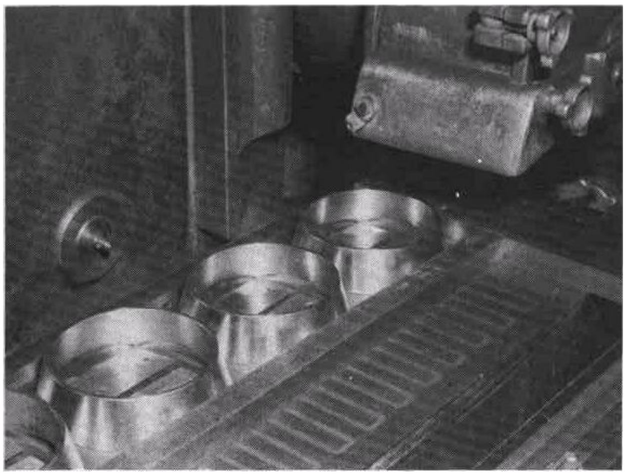


Fig. 22-11. Work mounted in V-blocks on magnetic chuck.

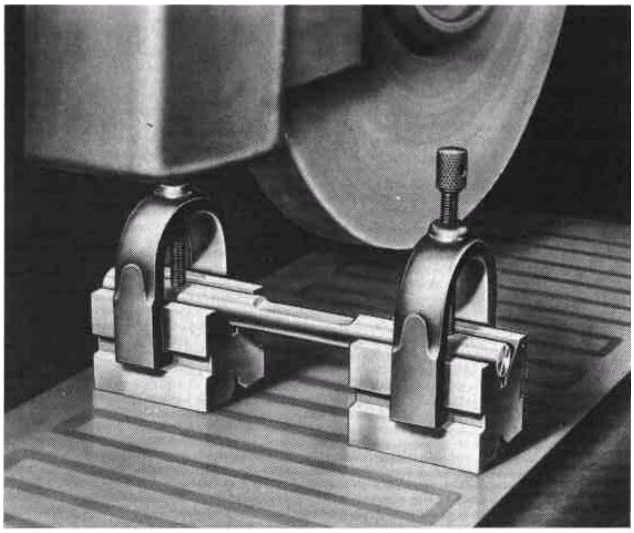


Fig. 22-12. Method of blocking workpieces with small contact area on magnetic chuck to prevent their moving under pressure of grinding.

If the wheel is dull it is nececcary to dress it. Position the fixture on the table of the machine so that the diamond tool may be applied to the face of the wheel. The contact point of the diamond tool should be slightly left of the vertical center of the wheel (for clockwise rotation). The toolholder should also be inclined slightly in the direction of wheel travel. This is necessary to prevent gouging and a tendency to chatter.

To avoid excessive vibration, it is essential that a **grinding wheel be balanced**. Most manufacturers balance their wheels before issuing them. Consequently, wheels 254 mm or less in diameter rarely need further balancing. Larger wheels may need to be balanced because of wear and changes which may have developed within the wheel.

Balance may be achieved by: (1) Adding weight in the form of lead to the light side. This may be accomplished by removing small amounts of the wheel beneath the flanges and then filling the holes thus made with lead. (2) On some wheel units, balance is achieved by adjustment of segments attached to the inner sleeve flange.

22.7. Cutting Speed and Possible Grinding Troubles

Speeds of between 1 372 mpm and 1 981 mpm are recommended for grinding cutters of high-speed or cast nonferrous alloy. Sintered carbide cutters should be ground at 1 524 to 1 676 mpm. Dry grinding is recommended except when using diamond wheels.

A maximum depth of cut of 0.01 mm should be used when grinding carbide cutters. The cut should not exceed 0.076 mm per pass when grinding high-speed steel or cast alloy cutters.

Clearance angles should be as small as possible so as to provide as much metal as possible to support the cutting edge, carry away heat, and minimize the possibility of chatter. Recommended clearance angles vary from 12° or more to as little as 3°, depending on the cutter diameter and type of material to be cut. For general purpose use, cutters up to 76 mm should have 6° to 7° of clearance. Cutters over 76 mm should have 4° to 5° clearance. For best results, the primary clearance angle should be adjusted for the material being cut as follows:

Low carbon steels .... .5°-7°
High carbon and alloy steels........ 3°-5°
Cast iron, medium and hard bronze . . 4°-7°
Brass, soft bronze, aluminum, magnesium, plastics .... 10°-12^0

When resharpening milling cutters, a secondary clearance angle of 3°-5° is normally provided so as to maintain the original land width. This prevents the heel of the land from interfering with the surface being milled. Original land widths are usually 0.4 mm, 0.8 mm, and 1.6 mm for small, medium and large cutters respectively. End and side teeth are ground with less clearance, usually 2°.

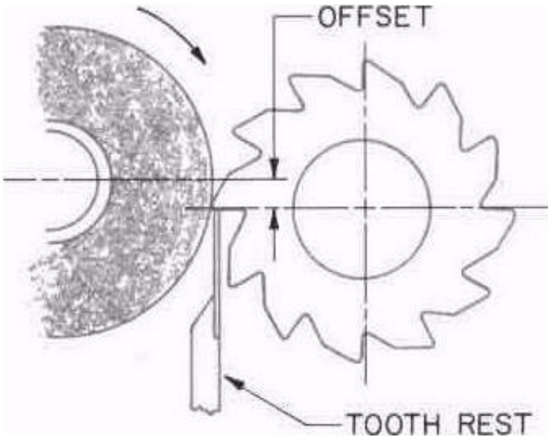


Fig. 22-13. Arrangement of tooth rest, cutter, and grinding wheel for disc wheel cutter sharpening.

Figure 22-13 shows the arrangement of cutter, tooth rest, and grinding wheel for sharpening cutters with a disc wheel. Note that the tooth rest is set to the same height as the cutter centerline. The amount of offset between the cutter and grinding wheel centerlines determines the clearance angle ground on the cutter.

Table 22-4. Possible Grinding Troubles

Symptom	Probable Cause
Work shows chatter finish.	Grinding wheel out of balance or not clamped properly on the wheel sleeve. Grinding wheel dull, glazed, or loaded. Poor choice of wheel for material being ground. Work not well-supported; centers worn or need lubricant. Not a sufficient number of back rests used, or back rests not properly adjusted. Too high a work speed or rate of table travel. Cut too heavy, caused by excessive cross feed. Unbalanced workpiece (for example, a crankshaft) running at a speed which is too high or running away from the driving dog (too much momentum). A worn or defective driving belt; check headstock, spindle, and table belts. Machine located on an insufficiently rigid floor or a floor which transmits vibration to the machine.
Scratches on the work.	Using a dirty coolant. Grinding wheel not trued properly. Truing diamond dull, cracked, or broken; or not held rigidly in the holder; not clamped securely in the truing fixture; the fixture not rigidly clamped in position; or footstock spindle not clamped. Too rapid table feed, or a too deep cut when truing or dressing the wheel. Wheel too coarse for the work.
Spiral marks on the work.	Point of truing diamond too high. The wheel should be trued with the diamond point as near the heights of the work centers as possible.
Wheel burning the work.	Insufficient coolant used, or coolant not properly directed at the point of contact of the wheel and the work. Grinding wheel dull, glazed, or loaded; needs dressing. Wheel too hard, wheel speed too high, or work speed too low. Excessive cross feed.
Work not ground parallel.	Swivel table not set accurately at zero. Swivel table pivot shoe may need adjusting. Headstock or footstock not seated properly on the table. Centers not seated properly in the spindle, or center points worn out of round. Center holes in workpiece dirty, out of round, or do not fit the centers properly. Radial play in the footstock spindle. Spindle clamp not properly adjusted. Back rests needed, or if used, not properly adjusted.
Work not sizing uniformly.	Wheel slide rapid travel arrangement motor brake needs adjusting. Cross-feed screw thrust bearing needs adjusting.
Wheel spindle runs too hot or stalls.	Insufficient oil in spindle reservoir, or wrong kind of oil. Cross feed too heavy, beyond capacity of the machine. Spindle driving belts too tight.

Bibliography

1. Repp V.E., McCarthy W.J. “Machine Tool Technology”. McKnight Publishing Company, Glencol, 1984.
2. Orlov P. “Fundamentals of Machine Design”, v.3. Mir Publishers, Moscow, 1980, -272 pp.
3. Orlov V.N. “Technology of Transport Machines Details Manufacturing”, Kurgan, 2000.- 262 pp.
4. Matalin A.A. “Technology of Machinebuilding”, Machinebuilding Publishers, Moscow, 1985, -512 pp.
5. Орлов В.Н. “Технология изготовления деталей транспортных машин”, Курган, 2000. -262 pp.

CONTENTS

Victor N. Kozlov
Technology of Mechanical Engineering
Part 2 a
Textbook
Editor ____________