$K_p = .78$ (From Table 5-11); $C = .83$ (From Table 5-13); $E = .80$ (From Table 5-15)

$Q = 12\ V\ f\ d = 12 \times 60 \times .030 \times .375 = 8.1$ in.^{3}/min

$P_c = K_p\ C\ Q\ W = .78 \times .83 \times 8.1 \times 1.3 = 6.8$ hp

$$P_m = \frac{P_c}{E} = \frac{6.8}{.8} = 8.5 \text{ hp}$$

CHAPTER 4

Milling Machine Construction

The milling process is used to produce a variety of surfaces by using a circular-type cutter with multiple teeth or cutting edges which successively produce chips as the cutter rotates. These cutting edges are located on the periphery and also often on the face of the cutter. The shape of the milling cutter and the path that it takes determine the shape of the surface produced. The function of the milling machine is to provide the means of holding and rotating the milling cutter, of holding and feeding the workpiece into the cutter, and of transmitting the necessary power to cut the metal at the desired rate. The great variety in the size and shape of parts, as well as advances in technology, has led to the development of many different types and styles of milling machines. They are extensively used at the present time for machining both large and small workpieces.

Classification of Milling Machines

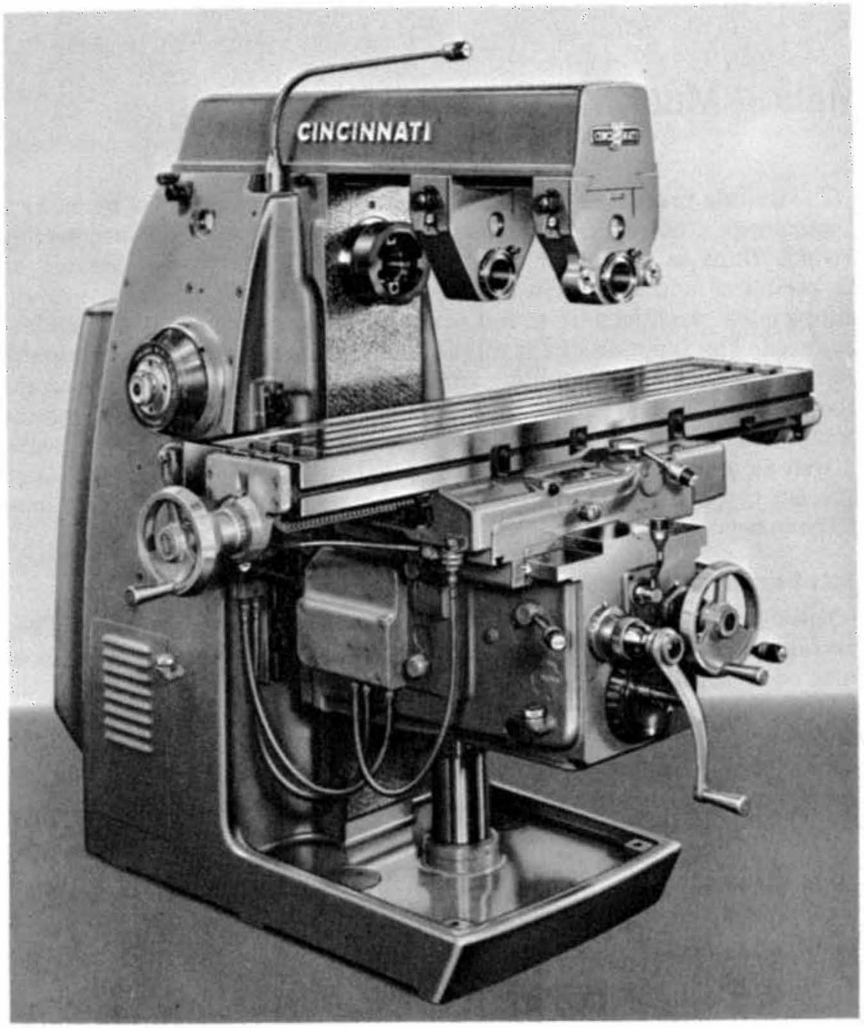
Milling machines, as a class of machine tools, are very versatile. They are capable of machining economically one or two piece lots as well as parts on a large-volume production basis. The inherent advantage of the milling process is the circular cutter, which is economical in first cost and which has a high metal removal rate since it can bring a large number of cutting edges into the cut in a relatively short space of time. This advantage has led to the design of a large variety of machine tools primarily for the purpose of milling. These can be classified as follows:

1. Knee-and-column- (or column-and-knee-)type milling machines
2. Fixed-bed-type milling machines
3. Planer-type milling machines
4. Special milling machines.

Further classifications of the milling machines are made on the basis of the type of control used, such as numerically controlled milling machines, and on the basis of the position of the spindle—i.e., horizontal or vertical.

Knee-and-Column-Type Milling Machines

The knee-and-column-type milling machine is also frequently called a column-and-knee-type milling machine. As a very versatile machine capable of performing a wide variety of operations, it is extensively used in machine shops and tool and die shops. The distinguishing characteristic of knee-and-column-type milling machines is that the table can be moved three directions in space, as can be seen in Fig. 4-2. Thus, a workpiece



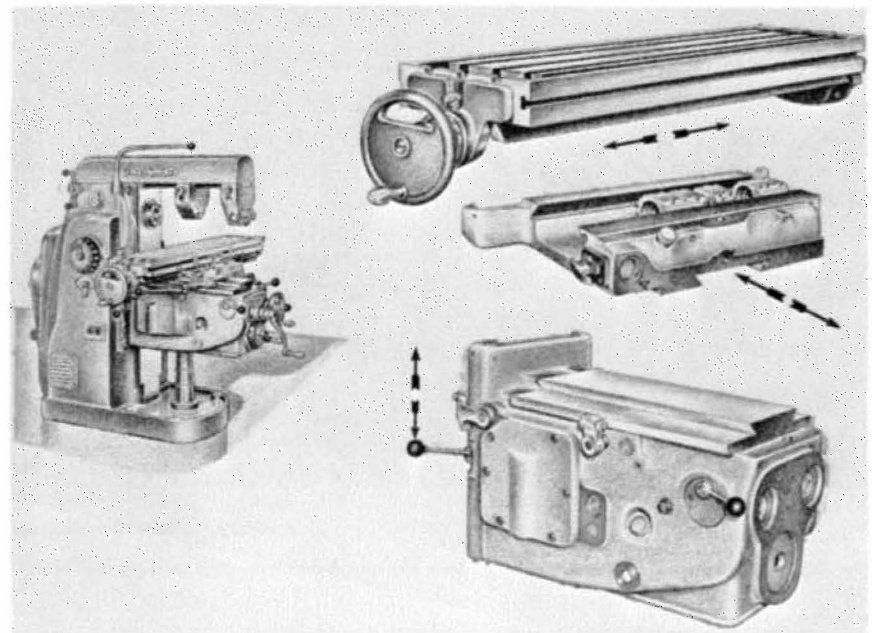
Courtesy of Cincinnati Milacron

Fig. 4-1. Plain knee-and-column-type milling machine.

mounted on the table of the milling machine can be easily and accurately positioned relative to the spindle which contains the cutting tools, and many different kinds of operations can be performed. There are several different types of knee and column milling machines which are described here.

Plain Knee and Column Milling Machine. A plain knee and column milling machine is illustrated in Figs. 4-1 and 4-2. The main supporting frame of knee and column milling machines is the column, including the base. The front face of the column is a machined and precision hand-scraped surface which supports and guides the knee. Being a precision bearing, the column face should be kept free of scratches, nicks, and other damage which would impair its accuracy.

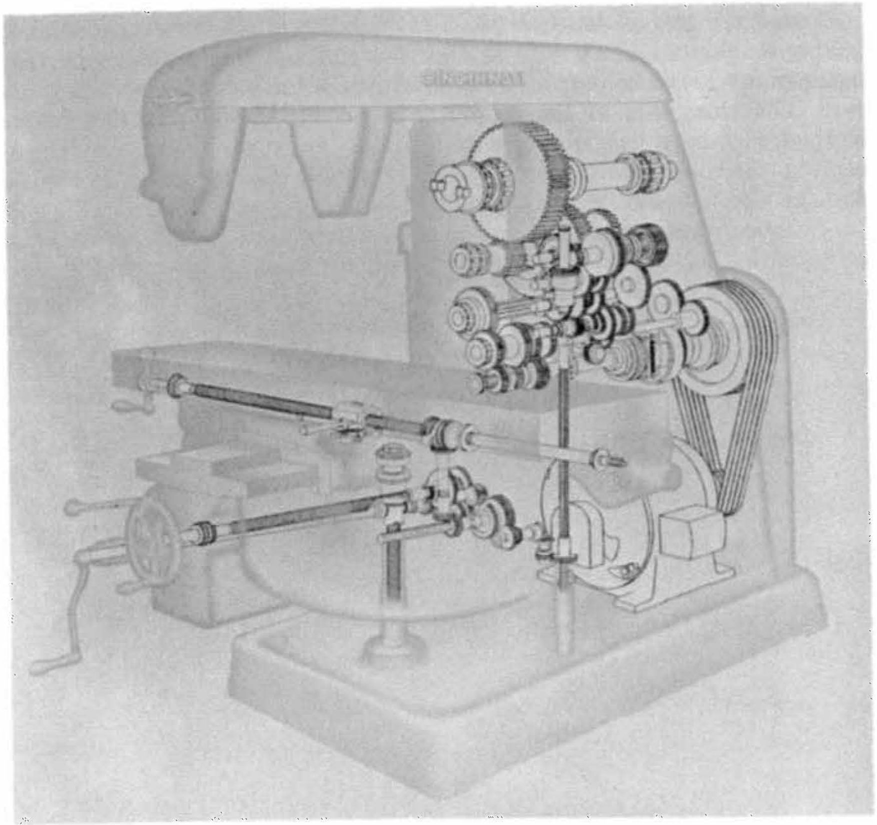
The knee slides up and down on the column face. This movement is actuated by the elevating screw seated on the base of the machine. The



Courtesy of Cincinnati Milacron

Fig. 4-2. Principal components of a plain knee-and-column-type milling machine.

elevating screw is a telescoping screw—i.e., one screw working inside another screw—which can be seen, in part, toward the front of the machine in Fig. 4-3. The elevating screw can be turned manually to raise or to lower the knee by turning the large hand crank on the front of the knee (Fig. 4-1). A micrometer dial behind this crank permits accurate vertical adjustments to be made. The vertical feed can be power-actuated on all but light-duty knee and column milling machines. The top surfaces of the knee are precision-machined and hand-scraped to form a bearing surface on which the saddle can slide. This surface should be protected from damage; i.e., wrenches and other tools should not be placed on it.



Courtesy of Cincinnati Milacron

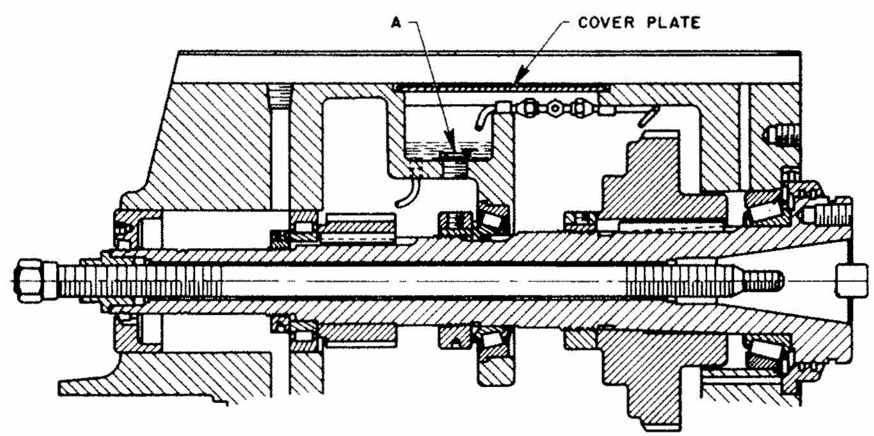
Fig. 4-3. Illustration of power train of a plain knee-and-column-type milling machine.

The saddle slides on the knee in a horizontal direction that is parallel to the spindle. This feed direction is called the *transverse feed*. The handwheel in front of the knee (Fig. 4-1) is used to actuate the transverse feed by turning the transverse feed screw, which can be seen in Fig. 4-3. A micrometer dial behind the handwheel permits accurate adjustments to be made in this direction. On all except light-duty machines, the transverse feed can be actuated by power. The surfaces on the top of the saddle are precisely machined and hand-scraped to form a slide that is perpendicular to the lower slide, which works on the knee.

The milling-machine table moves over the upper slide of the saddle in a direction perpendicular to the axis of the spindle. This feeding movement, called the *longitudinal feed*, is actuated by a handwheel or crank located at the end of the table. Thus the longitudinal feed screw or lead screw (Fig. 4-3) is caused to rotate in a stationary nut attached to the saddle. A micrometer dial behind the longitudinal feed handwheel or crank

permits accurate adjustments of the table to be made in this direction. A power longitudinal feed is also available on most milling machines. The tabletop is a precision reference and locating surface used to locate workpieces, vises, angle plates, and other work-holding fixtures in a plane that is parallel to the axis of the spindle. Since the accuracy of the operation being performed often depends upon the condition of the tabletop, it should not be abused by carelessly clamping rough work surfaces to it or by placing heavy tools on it. A nick could result in a scraped workpiece. T-slots milled into the top of the table are used to retain or to anchor bolts for clamping workpieces and attachments to the table. The T-slots, which are machined parallel to the lengthwise direction of the table, can be used as reference surfaces when workpieces and accessories are set up on the table.

The power train of a knee and column milling machine is shown in Fig. 4-3. The electric drive motor is located in the column of the machine, in a position where the heat that it generates has the least effect on the precision-sliding movements. The power is transmitted from the motor through a large multiple V-belt to a shaft containing a clutch that starts and stops the spindle. Then the power goes directly to the spindle through a selective speed gear transmission built into the column of the machine. This transmission provides the different spindle speeds that are available on the machine. The power for the three table feeds is taken off to the vertical shaft by means of a pair of bevel gears. The vertical shaft has a long spline upon which a gear slides. This gear actuates a gear train that provides all of the power table feeds. A selective change gear box located either in the column or in the knee provides for the different feed rates. In modern milling machines the feed rate is given in terms of *inches per minute* of table travel. In addition to the three directions of power feeds, a rapid traverse in three directions is usually available. The rapid traverse, used to rapidly position the table, spares much manual effort.

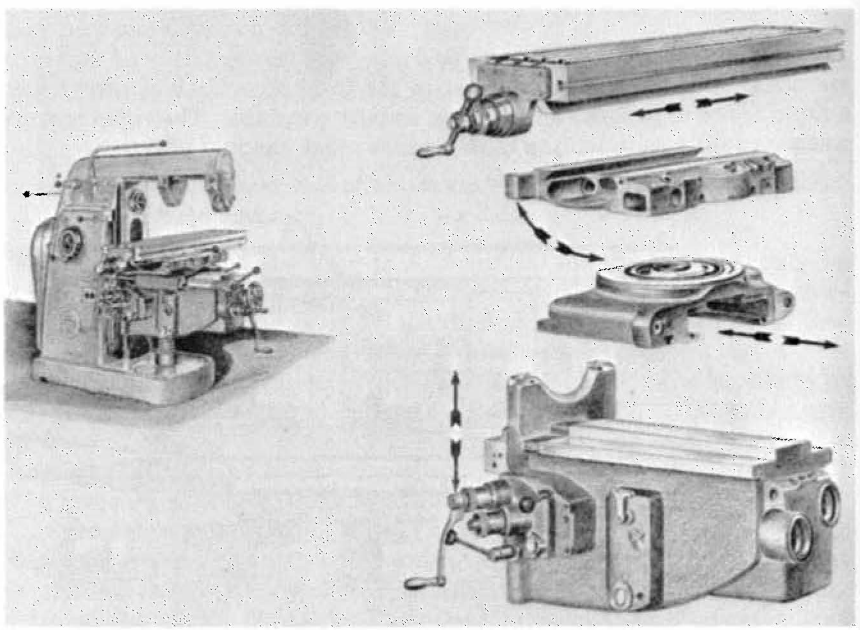


Courtesy of Cincinnati Milacron

Fig. 4-4. Section through a milling-machine spindle.

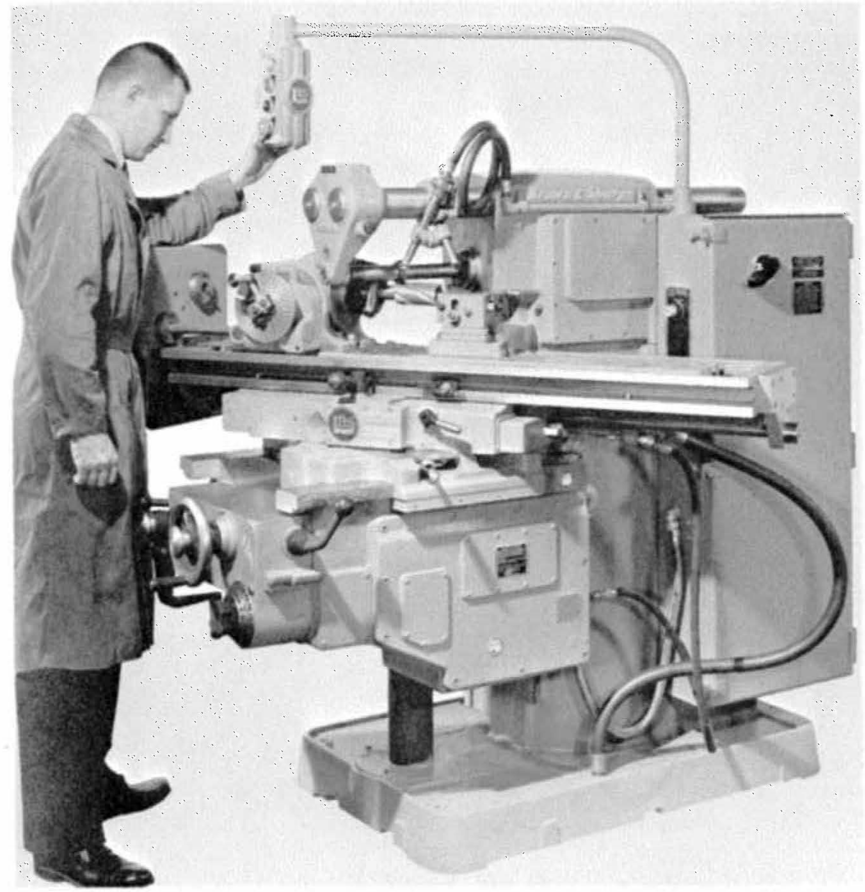
The spindle is mounted on precision machine-tool grade antifriction bearings which are housed in the column, as seen in Fig. 4-4. The spindle is hollow. A draw-in bar, shown being placed through the spindle, is used to hold milling cutters, arbors, and adaptors in the spindle. The inside of the spindle nose has an American Standard Milling Machine Spindle Nose Taper which is 3.500 inches per foot for all sizes. This is a self-releasing taper, used to locate the cutters, arbors, or adaptors. Two lugs or keys on the face of the nose act as drivers for driving the cutters. There are four standard sizes (No. 30, 40, 50, and 60) of milling-machine spindle nose tapers, the size used depending upon the size of the machine. The outside diameter of the spindle nose is ground to a very close tolerance. Large face milling cutters, which are mounted on the outside of the spindle nose, are held to the spindle by four socket-head cap screws which fit into threaded holes in the face of the spindle.

The overarm is mounted on the top of the column (Fig. 4-2) or inside two bored holes located on the top of the column (Fig. 4-6). The purpose of the overarm is to support and align the arbor supports and various other attachments. The overarm in Fig. 4-2 is a heavy rectangular casting with a built-in vibration dampener. Some milling machines can be equipped with a rectangular overarm with a built-in overhead spindle or vertical head as shown in Fig. 4-30. This permits the machine to be used as either a vertical or a horizontal milling machine. If desired, the horizontal and



Courtesy of Cincinnati Milacron

Fig. 4-5. Principal components of a universal knee-and-column-type milling machine.

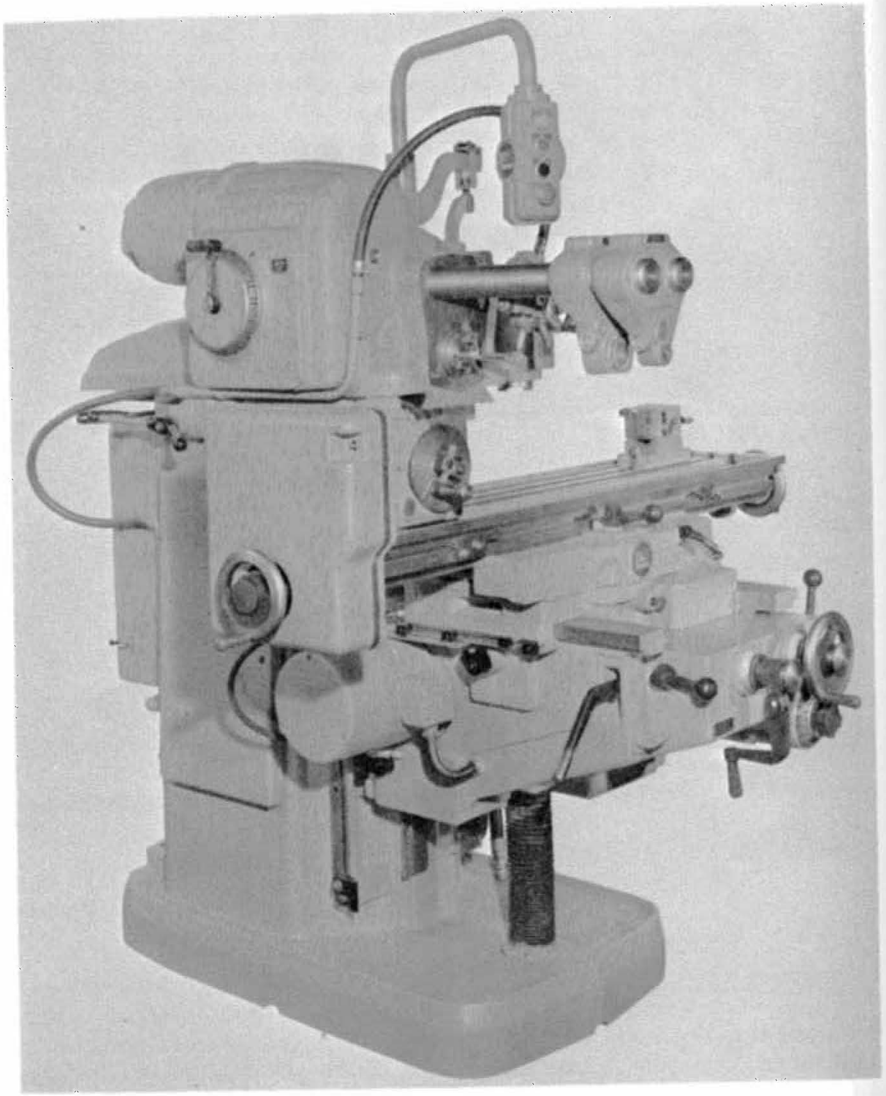


Courtesy of the Brown & Sharpe Manufacturing Company

Fig. 4-6. Universal column and knee milling machine milling helical flutes in a milling cutter blank.

overhead spindles could be used simultaneously. Another overarm design used on modern milling machines is the double overarm shown in Figs. 4-6 and 4-7. The two solid cylindrical bars that form the overarm are contained in two holes bored in the upper part of the column.

Arbor supports mounted on the lower side of the overarm are used to hold the outer ends of long arbors on which milling cutters are mounted. They can be seen mounted in position in Figs. 4-1 through 4-7. Two styles of arbor supports can be seen in Fig. 4-7. The one mounted on the inside has a larger bearing and must be used when a Style B arbor (Fig. 4-18) is used. This arbor support is also used on the large bearing collar of a Style A arbor. The support shown on the outside in Fig. 4-7 can only support the outer end of a Style A arbor. The inside arbor support in Fig. 4-7, having



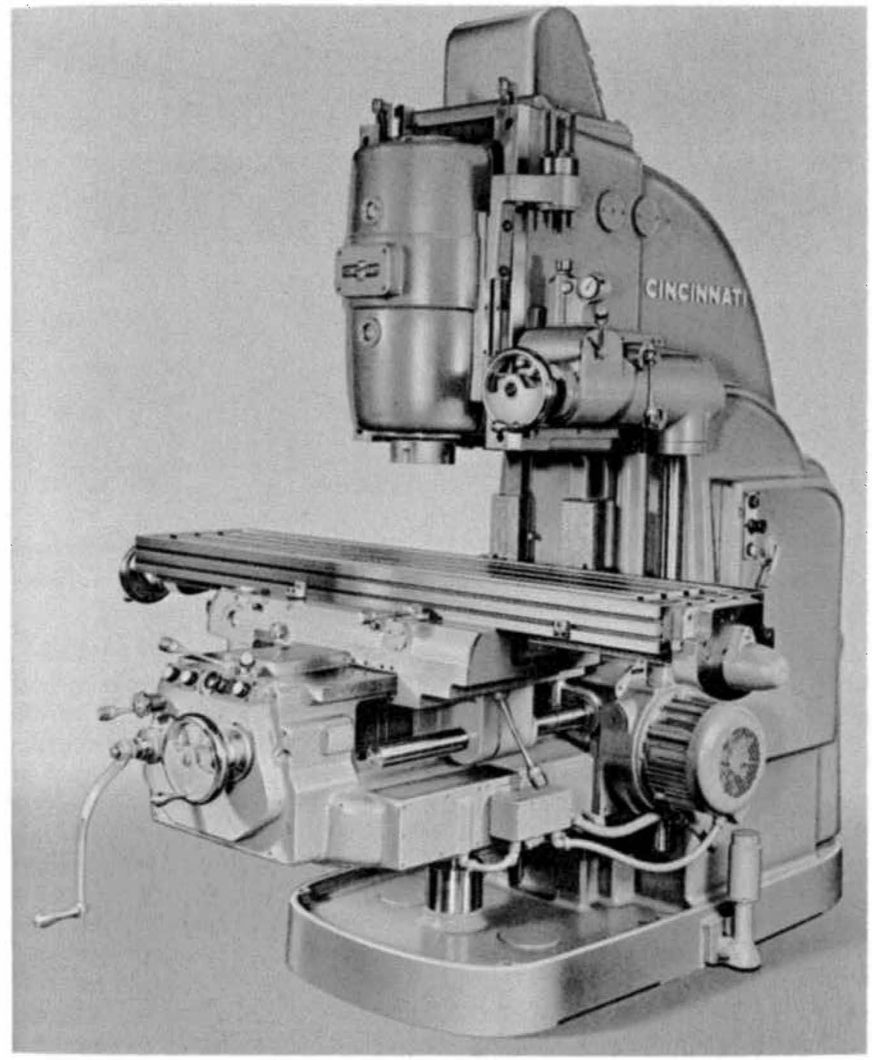
Courtesy of the Brown & Sharpe Manufacturing Company

Fig. 4-7. Sliding-head-type universal column-and-knee-type milling machine.

a larger bearing, can support a heavier cutting load and can be positioned anywhere along the length of the arbor. Thus, this arbor support can be placed as close to the cutter as possible to provide maximum support. The arbor support on the outside in this illustration can only support a Style A arbor at its extreme outer end, where the arbor is reduced in diameter. This type of arbor support, however, has the advantage of providing more

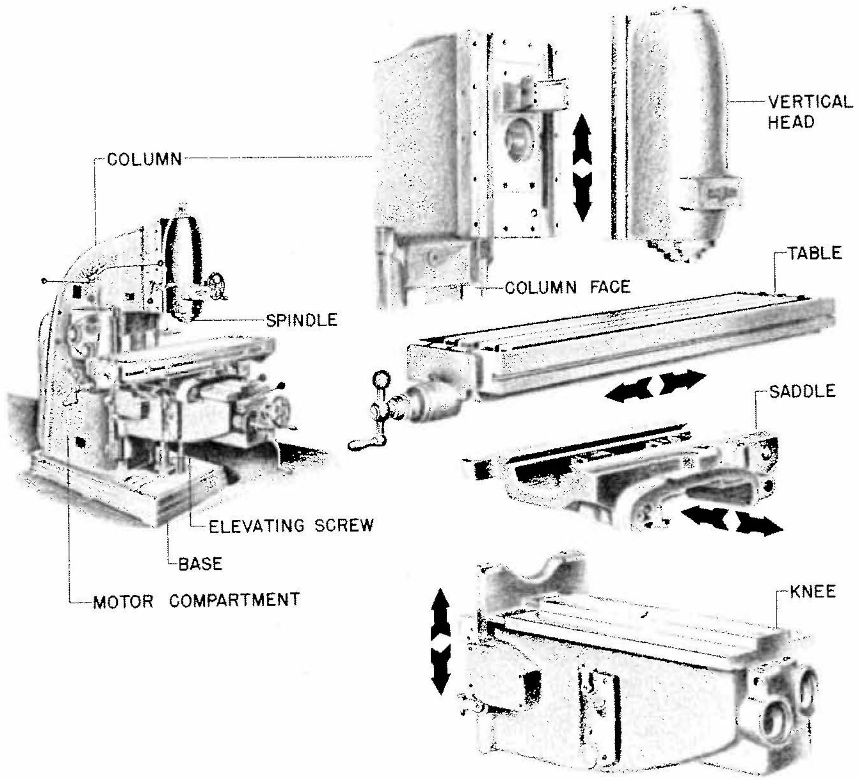
clearance below the arbor so that workpieces, vises, clamps, etc., can be brought closer to the arbor. This is often an advantage when parts are milled with a cutter having a small diameter.

Universal Knee-and-Column-Type Milling Machine. A universal knee-and-column-type milling machine, shown in Figs. 4-5 to 4-7, is very similar in construction to a plain knee-and-column-type milling machine. The principal difference is in the construction of the saddle and in the addition of swiveling table housing (Fig. 4-6), which allows the table to be



Courtesy of Cincinnati Milacron

Fig. 4-8. Vertical knee-and-column-type milling machine.

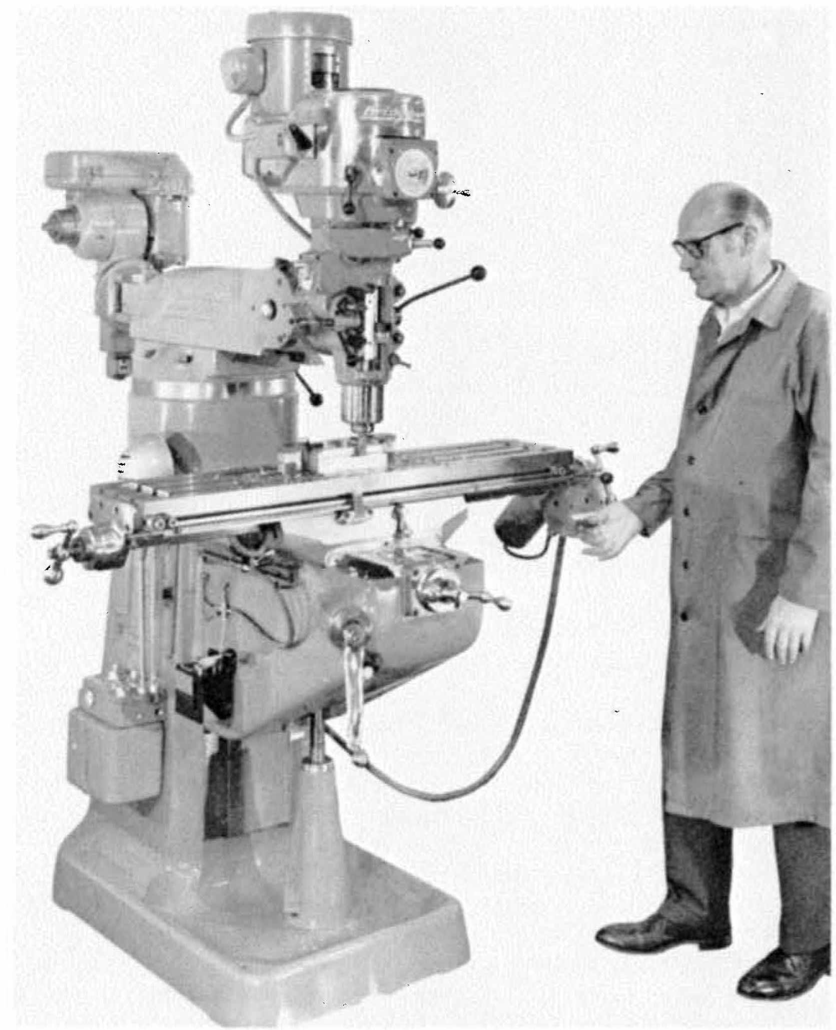


Courtesy of Cincinnati Milacron

Fig. 4-9. Principal components of a vertical knee-and-column-type milling machine.

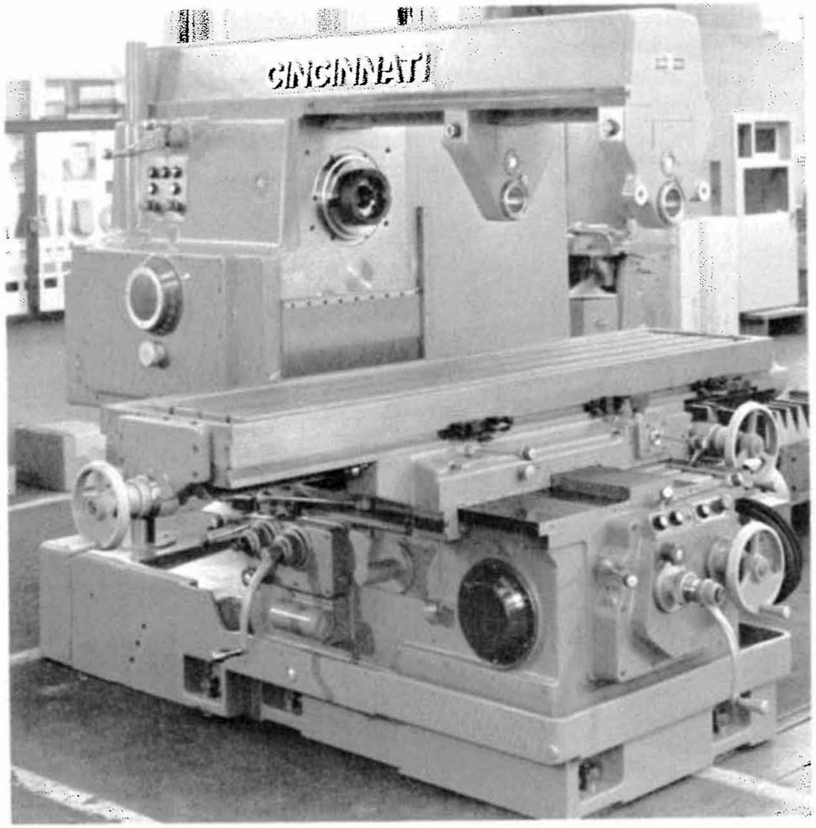
swiveled at an angle to the axis of the spindle. Generally the tables of universal milling machines will swivel approximately 45 degrees in both directions from the normal position perpendicular to the axis of the spindle. This feature extends the operating range of the milling machine by making it possible to cut helical grooves with cutters mounted on the milling-machine arbor. In Fig. 4-6, a universal column-and-knee-type milling machine is shown cutting a helical flute in a milling cutter blank. A universal spiral index head is used to rotate the milling cutter blank as it is fed into the cutter by the longitudinal table feed. The index head is also used to space the flutes around the circumference. Note that the table is swiveled to the helix angle of the flute. A sliding-head-type universal milling machine is shown in Fig. 4-7, where the spindle head is mounted on a dovetail slide on top of the column. Although the spindle is rigidly held in a fixed position inside the spindle head, it can be moved toward or away from the face of the column by moving the entire spindle head on the dovetail slide. In this way the nose of the spindle can be brought

as close to the workpiece as possible so that arbor-mounted cutters can be placed close to the nose. Thus the rigidity of the setup is improved. The sliding head can also reach over the table to allow spindle-mounted cutters such as end mills and face mills to reach surfaces on the workpiece that cannot be positioned close to the column. The electric drive motor and the entire speed change gear train are mounted on the sliding head. A



Courtesy of Bridgeport Machines, Inc.

Fig. 4-10. Toolroom-type vertical milling machine equipped with a milling head and a vertical shaping head.



Courtesy of Cincinnati Milacron

Fig. 4-11. Fixed-bed-type milling machine for general-purpose work.

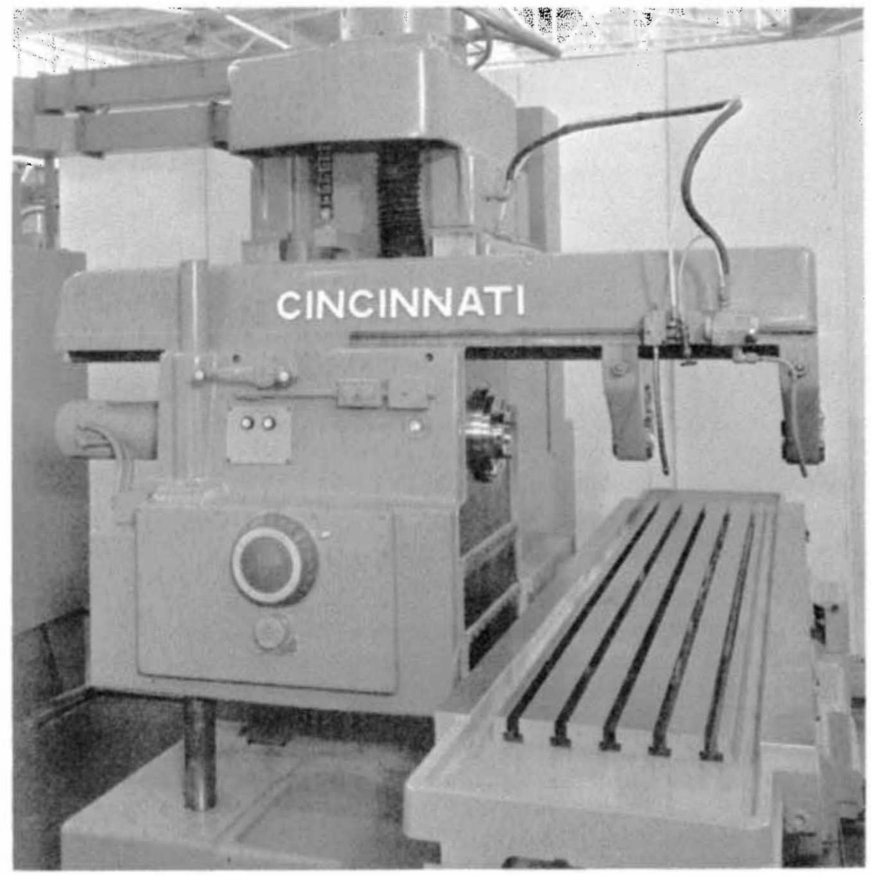
separate motor, which can be seen extending out from the left side of the knee, provides the drive for the power feeds.

Vertical Knee-and-Column-Type Milling Machine. A vertical-type knee and column milling machine is shown in Figs. 4-8 and 4-9. The spindle of the vertical milling machine is located vertically, parallel to the face of the column, and perpendicular to the top of the table. The vertical head can be moved up and down by hand or by power feed. This machine is especially suitable for performing operations which require the use of end mills and face milling cutters such as milling dies, cutting profiles, milling molds and for locating and boring holes in jigs and fixtures. The vertical milling machine shown in Fig. 4-10 is a light-duty milling machine for which there is a wide range of application in toolroom work as well as for performing other light-duty milling operations. Two heads are mounted on a ram which can be swiveled to bring either head into the operating position over the table. Several different heads are available. The machine

in Fig. 4-10 has a milling head and a vertical shaping head shown at the rear of the machine. Both heads can be set in angular positions. The spindle of the milling head is of the quill-type construction which can be moved up and down like a drill press spindle. The table is fed by hand, although a longitudinal power feed can be obtained as an attachment. The sensitivity and ease of handling of this machine make it especially adaptable for doing fine intricate work.

Fixed-Bed-Type Milling Machines

The distinguishing characteristic of all fixed-bed-type milling machines is the absence of the knee construction found in the knee-and-column-type milling machines. Some fixed-bed-type milling machines are designed for general purpose work while others are intended for high-production work. A general-purpose fixed-bed-type milling machine is shown in Fig. 4-11.



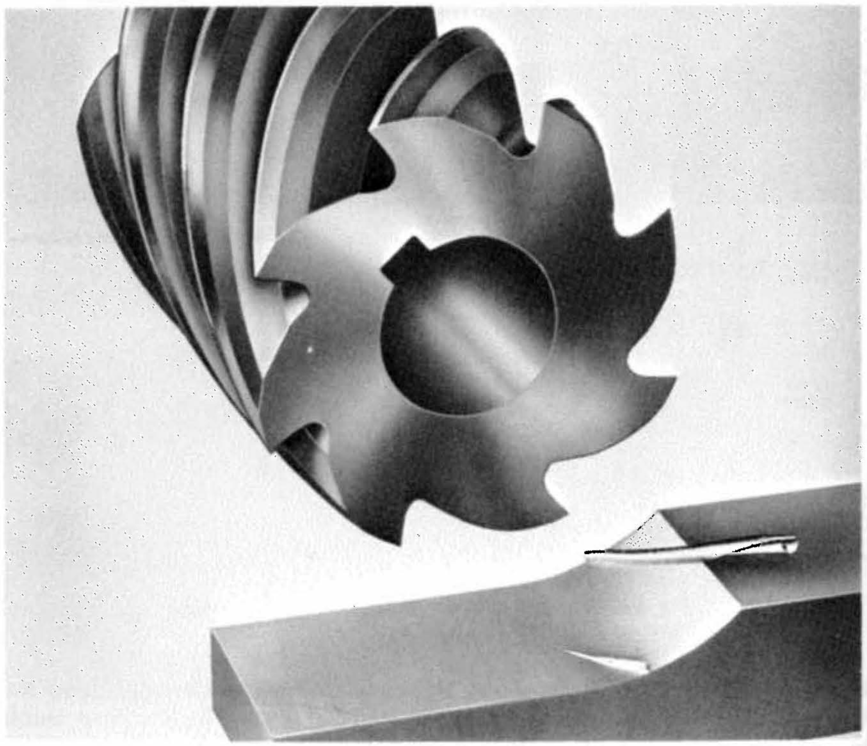
Courtesy of Cincinnati Milacron

Fig. 4-12. Spindle carrier of fixed-bed milling machine.

speed suddenly drop in rapid succession can set up the vibratory condition called chatter.

The tendency for chatter to occur can be substantially reduced by forming the cutting edge into a helix, as shown in Fig. 5-8. At the beginning of the cut, the length of the cutting edge penetrating into the workpiece is small. As the cutter continues to rotate, the length of the cutting edge in contact with the workpiece increases, gradually forming a widening chip until the maximum is reached. The depth of the cut and the helix angle of the cutter will determine the length of tooth travel during which the maximum tooth contact and chip width occur. The tooth contact and chip width then gradually decrease until zero is reached as the tooth leaves the work. Thus, this gradual building up and release of the cutting load will avoid the shock that accompanies a sudden release of the cutting load. In addition, in most instances two teeth are simultaneously cutting, as in Fig. 5-8, to further stabilize the cutting load.

Since the chip is formed at an angle when the teeth are on a helix angle, it is often assumed that a shearing action is taking place between the cutting edge and the chip. It must be emphasized that this does not occur, for there is no axial sliding motion between the cutting edge and the chip. The width of the chip is at all times equal to the length of the cutting edge that is in contact with the work, and the cutting action is the same for helical teeth as for straight teeth.



Courtesy of Cincinnati Milacron

Fig. 5-8. Heavy duty plain milling cutter with a 45-degree helix angle showing chip formation.

End Milling Cutters

End milling cutters constitute a large group of milling cutters made to a variety of shapes and sizes. The group of typical end milling cutters displayed in Fig. 5-9 are characterized by having cutting edges on the end face as well as on the periphery. Also, they are always held in the milling-machine spindle by a collet chuck or some kind of adaptor. Among the most versatile of cutting tools, end mills are used to mill plane surfaces, slots, profiles, and three-dimensional contours.

The elements of an end milling cutter are shown in Fig. 5-10. The radial rake angle is generally small, as it is limited by the necessity of keeping the end cutting edges approximately radial. The helical rake angle and

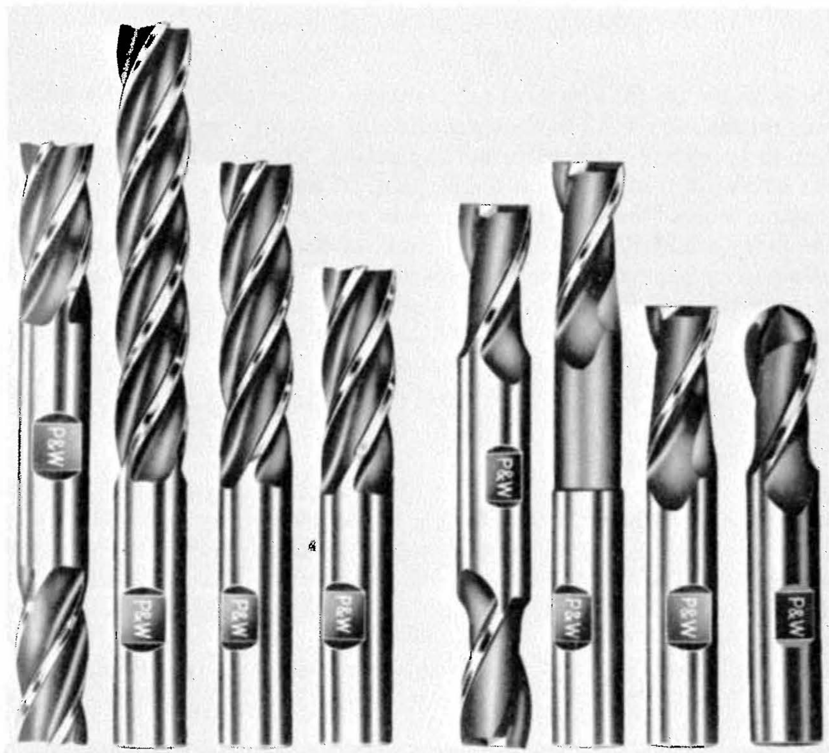


Fig. 5-9. A group of different end milling cutters.

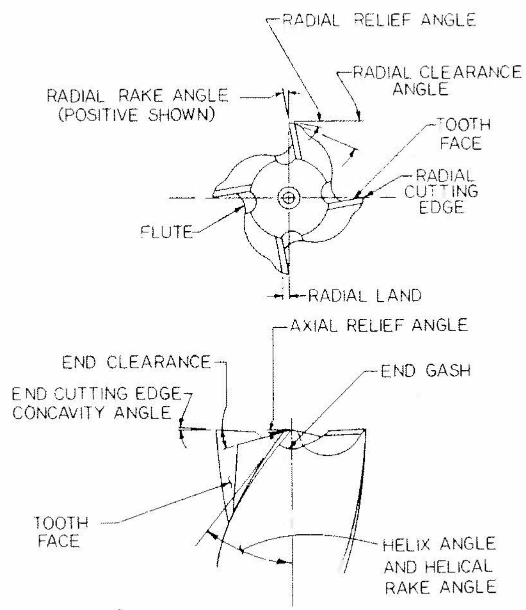


Fig. 5-10. Elements of end milling cutters.

the helix angle are for practical purposes the same angle. The helix of the peripheral cutting teeth has the same effect on end mills as on plain milling cutters in providing a smoother cutting action. There is, however, a limitation on the size of the helix angle. It must not be made so large that the cutting edge of the end cutting teeth is weakened.

The relief angle of the peripheral teeth, or the "radial relief angle," is determined by the size of the end mill and by the material to be cut. For most applications, including most steels and cast irons, the following radial relief angles (given in terms of the cutter diameter first followed by the radial relief angle) are recommended:

$\frac{1}{16}''$—22°, $\frac{1}{8}''$—17°, $\frac{3}{16}''$—14°, $\frac{1}{4}''$—12°, $\frac{3}{8}''$—12°, $\frac{1}{2}''$—11°, $\frac{3}{4}''$—10°, 1″—9°, $1\frac{1}{2}''$—8°, 2″—7°

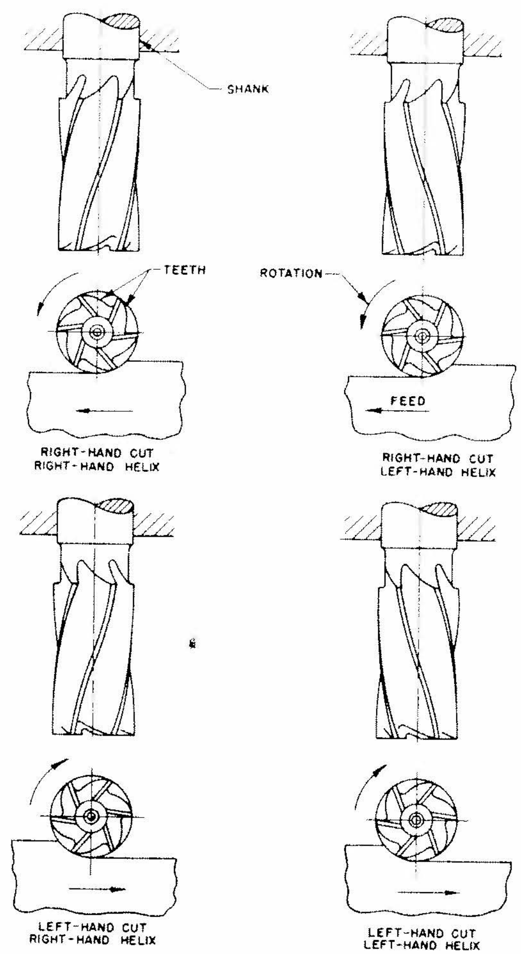
More detailed recommendations for the radial relief angle are provided in Chapter 14, Table 14-1. Depending on the method by which the radial relief angle is ground, three types of relief are used; namely, concave, flat, and eccentric. The three types of relief are shown in Fig. 14-5, Chapter 14.

On most end mills the relief angle on the end cutting teeth—the axial relief angle—should be about 4 degrees. Two fluted center cutting end mills should have an axial relief angle of about 7 degrees because they are often fed endwise, or plunged into solid stock, to be used as a twist drill.

The flutes in an end mill may have a right- or left-hand cut as shown in Fig. 5-11. The cut refers to the side of the flute on which the face of the

teeth are located. Furthermore, end mills with a right-hand cut are designed to cut while rotating counterclockwise when viewed from the end having the teeth. Left-hand-cut end mills are designed to cut while rotating clockwise when viewed from this end. Although the helix may be right- or left-handed for either cut, usually a right-hand-cut flute has a right-hand helix and a left-hand-cut flute has a left-hand helix. In this way the end-cutting edges have a positive rake angle.

Most standard end milling cutters are made to have two or four flutes in sizes up to approximately 1 inch. Larger end mills, up to 2 inches in diameter, are made with six or eight flutes. Three-fluted, center-cut-type end mills are made with diameters up to 3 inches. Increasing the number



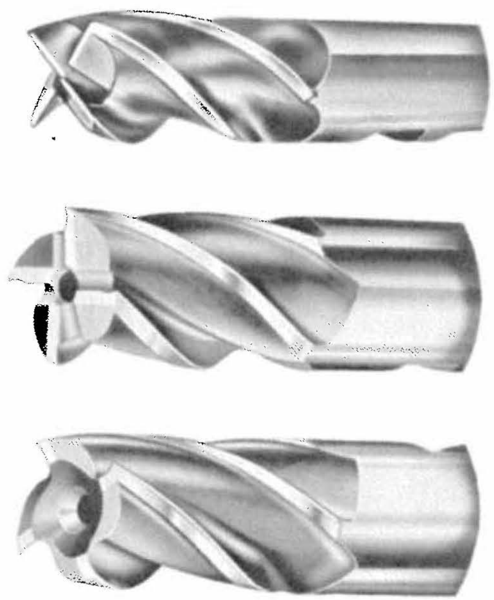
Courtesy of Cincinnati Milacron

Fig. 5-11. The four combinations of hand of helix and hand of cut for end mills.

of flutes on the end mill helps to stabilize the cutter when milling slots and allows a faster feed (inches per minute) to be used. The flutes must, however, be large enough to provide adequate space for the chips.

Figure 5-12 illustrates the three different types of construction which can be used for the end teeth of end milling cutters with four or more teeth. The end mill at the bottom has a cupped-type end on which the end teeth extend only to approximately the bottom of the flute. The conventional square end mill in the center has a notch in the end which permits the end teeth to extend to the center hole. The added length of the end teeth allows radii or forms to be ground on the end when required. The end mill at the top has two end teeth which join in the center. This type, called a *center-cutting end mill*, can plunge directly into a workpiece to create an opening or a hole in much the same manner as a twist drill. After it has been plunged to the required depth, it can be made to cut a slot such as a keyseat. The four peripheral teeth produce a good finish on the workpiece and allow a reasonable feed rate to be used.

The most common center-cutting-type end mill is the *two-fluted end mill*, which in some shops is called a *two-lipped end mill*. Four typical two-fluted end mills are shown at the right in Fig. 5-9. The conventional square end two-fluted end mill is the least expensive and the most easily sharpened of the center-cutting-type end mill. As it can easily be sunk directly into the workpiece like a drill, it is frequently used to cut keyseats



Courtesy of The Metal Cutting Tool Institute

Fig. 5-12. General-purpose four-fluted end mills showing three types of construction for the end cutting edges.

and other slots that do not extend to an open end or shoulder. Two-fluted-ball end mills, such as shown at the right in Fig. 5-9, are used to cut complex three-dimensional contours such as are encountered in dies and molds. The operation for which they are used is called *die-sinking*.

Although taper-shank end mills are still being made, the majority of the end mills have a straight shank. Some straight-shank end mills have cutting teeth on both ends, with the shank in the center of the cutter. The majority of the taper-shank end mills have a Brown & Sharpe taper.

Rough-Cutting End Mills. These end mills have been developed primarily to remove a large amount of stock, as shown in Fig. 5-13. Their teeth, shaped in the form of a radius which produces a small chip that does not load the flutes, are positioned around the cutter in the form of a left-hand helix like a left-hand thread. This makes the teeth overlap each other so that the surface produced is flat. The teeth are form-relieved and are sharpened by grinding only the face of the flutes.

Indexable Insert End Mills. Cemented-carbide, indexable-insert end milling cutters, Fig. 5-14, operate at a faster cutting speed and a faster table feed than conventional high-speed-steel end mills. Indexable insert end mills are available with one or two cemented-carbide inserts. Each

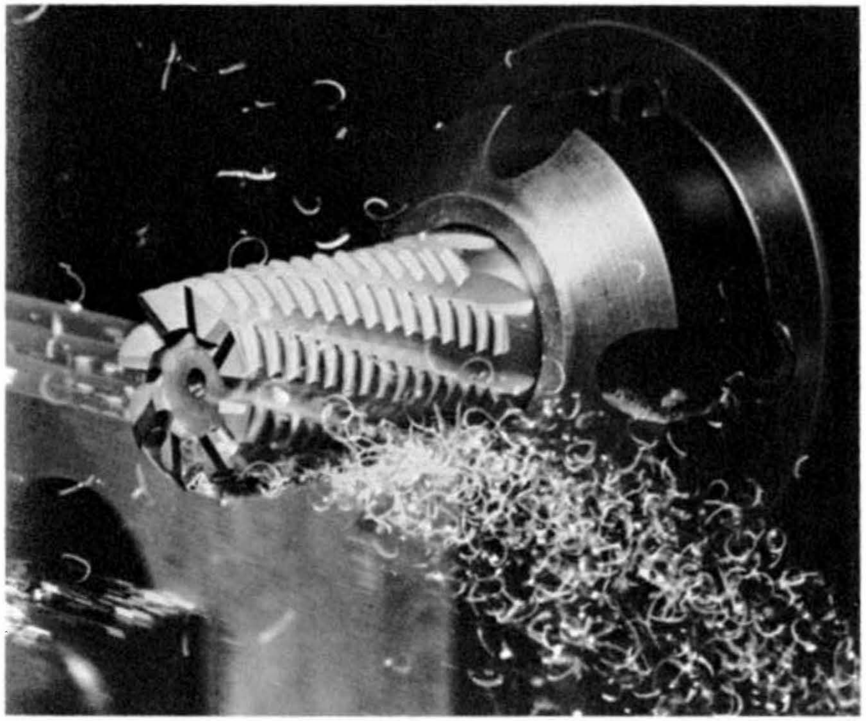
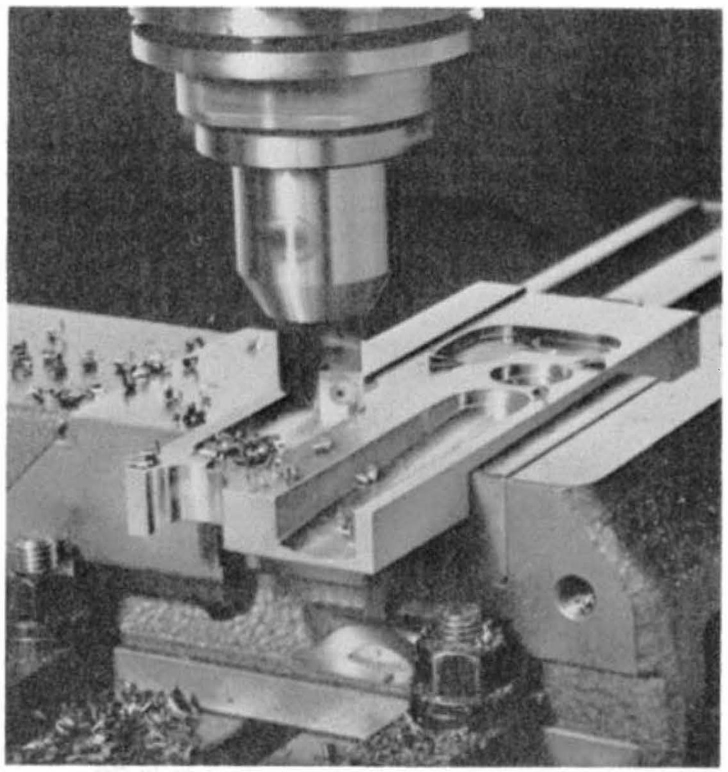


Fig. 5-13. Rough cutting end mill taking a typical cut in a workpiece.

insert has four cutting edges; when a cutting edge is worn the insert can rapidly be indexed to provide a sharp cutting edge until all the available cutting edges are used up. Both single and double insert end mills are designed to perform the operations shown in Fig. 5-14, which includes plunge cutting (partially or completely through the workpiece), slot milling, pocket milling, peripheral milling, and ramping. In addition, they can also be used to counterbore and to spot face.

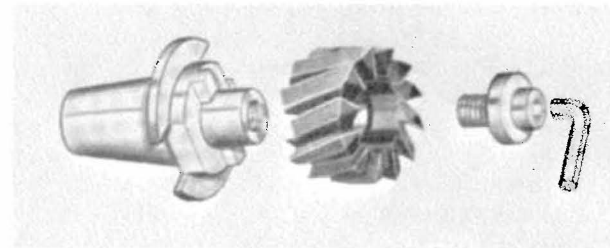
Shell End Milling Cutters

Standard shell end milling cutters are available in sizes from 1¼-inch diameter to 6-inch diameter. They are intended for taking surfacing cuts and corner cuts and are not generally used to cut slots. As shown in Fig. 5-15, shell end mills are mounted on shell end mill arbors, which are fitted directly into the spindle of the milling machine. One obvious advantage of the shell end mill is that when the body of the cutter is worn, the shank can be reused on another shell end milling cutter. Shell end milling cutters



Courtesy of the Valenite Div. of the Valeron Corp.

Fig. 5-14. Indexable insert end mill cutter showing the type of cuts that can be taken with this cutter.



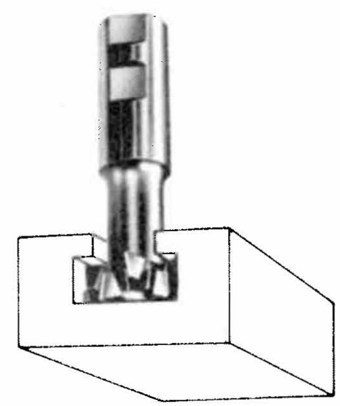
Courtesy of The Metal Cutting Tool Institute

Fig. 5-15. Shell end mill and shell end mill adaptor.

tend to bridge the gap between face milling cutters and end milling cutters.

The corner of the shell end milling cutter is the point where the cutting edges on the periphery meet the cutting edges on the end. The corner may be square, rounded, or chamfered. The most frequently used is the square corner, which is easiest to grind. However, this corner does tend to be a focal point of edge wear. The rounded corner, while working very well, is difficult to grind. The chamfered corner is best because it is easy to grind and wears well. The chamfer (or radius in the case of a rounded corner) should not exceed $1/16$ inch in length, since the small chamfer will not interfere with the main flow of the chip which is approximately perpendicular to the peripheral cutting edges. The chip flow from a larger chamfer will tend to interfere with the peripheral cutting-edge chip, resulting in an increase in the cutting temperature, accelerated tool wear, shorter cutter life, and quite often a deterioration in the resulting surface finish.

T-Slot Cutter. The milling cutter shown in Fig. 5-16 is used to cut T-slots in machine tool tables and accessories. A slot must first be cut in the work-



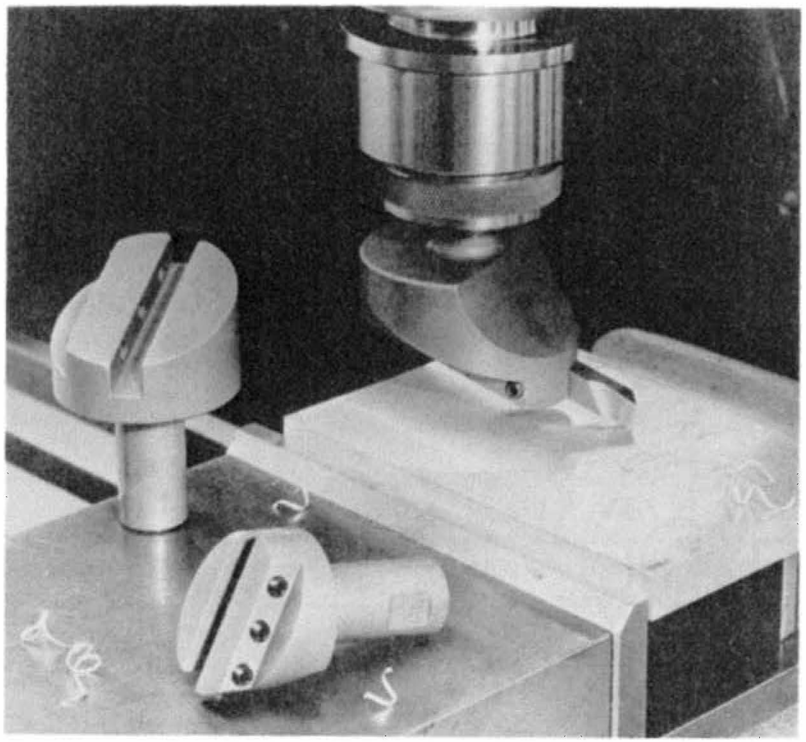
Courtesy of Niagara Cutter Inc.

Fig. 5-16. T-slot cutter.

Another type of fly cutter is shown in Fig. 5-32. It offers many advantages in performing face milling operations for which it is intended. The cutting tool is a standard single-point, high-speed-steel tool bit which can be sharpened by hand. When used with a fine table feed rate, this cutter will produce an excellent surface finish on most materials. Since only a single cutting edge engages the workpiece, the cutting force is light, enabling frail parts to be milled and setups to be used which are somewhat less secure than required when milling with a face milling cutter. On light, low powered milling machines, relatively large surfaces can be milled in a single pass that would otherwise require a series of passes with an end milling cutter.

Cutting Speed for Milling

The cutting speed for milling is the speed at the periphery of the cutter as it is rotating. In the customary inch system of units the cutting speed is given in terms of feet per minute, or fpm, which is sometimes called surface feet per minute, or sfpm. In the metric system the cutting speed



Courtesy of the City Tool Die & Mfg. Co., Inc.

Fig. 5-32. Fly cutter for performing face milling operations with a single point tool.

is in meters per minute, or m/min. The recommended cutting speeds feet per minute are given in Tables 5-3 through 5-7 for milling various materials with high-speed steel and cemented carbide cutters. To obtain meters per minute, multiply feet per minute by .3048. For each material a range of values is given to account for the shop variables encountered, which will be discussed in the following paragraph.

In addition to the cutting tool material, the cutting speed depends primarily on the work material and its hardness. The hardness range for which the listed cutting speed is valid in the case of each material is given in the tables. In general, an increase in the hardness of a material reduces the speed at which it can be cut. Since the hardness of a material is not always known in the shop, the material condition that is associated with a corresponding hardness in the table is given. The cutting speed is also influenced by the feed rate, and to a lesser extent by the depth of cut. Heavier cuts using a heavy feed require a slower cutting speed than do lighter cuts. Since the cost of replacing and sharpening a milling cutter is more than the cost of a single-point cutting tool, a longer tool life is more desirable for milling than for turning; therefore, the cutting speed for milling should be somewhat slower than for turning under the same tool and work material conditions. When using cemented carbide milling cutters the grade of carbide used has an influence on the cutting speed that can be used. The correct grade, as recommended by the carbide producer or cutter manufacturer, must be used. Where they can be used, coated carbides can often cut successfully using a cutting speed that is 20 to 40 per cent and sometimes up to 50 per cent higher than the values given in the cutting speed tables. In general, cemented carbide cutters having indexable inserts are operated at a somewhat faster cutting speed than those having brazed-on carbide tips, or blades to which the carbide is brazed. Other factors to consider in selecting the cutting speed are the design of the milling cutter and the rigidity of the workpiece, the setup, and the machine. When starting out to mill a new material, it is usually advisable to start at the lower end of the range of values given in the table; then, as experience is gained, the cutting speed may be increased.

Calculating the Cutting Speed

The formulas for calculating the speed of the milling machine spindle and the cutter are given below for inch and for metric units. Since the calculated speed may not be available on the machine, the closest available speed should be used. On some machines the range between speeds is large and it may be advisable to use the closest lower speed available.

$$N = \frac{12\,V}{\pi D} \qquad \text{(Inch units only)} \qquad (5\text{-}1)$$

$$N = \frac{1000\,V}{\pi D} \qquad \text{(Metric units only)} \qquad (5\text{-}2)$$

Table 5-3. Cutting Speed in Feet per Minute for Milling Plain Carbon and Alloy Steels

Material AISI and SAE Steels	Hardness, HB*	Material Condition*	Cutting Speed, fpm	
			HSS	Carbide
Free Machining Plain Carbon Steels (Resulphurized), 1212, 1213, 1215	100-150	HR, A	140	600
	150-200	CD	130	550
1108, 1109, 1115, 1117, 1118, 11120, 1126, 1211	100-150	HR, A	130	550
	150-200	CD	115	500
1132, 1137, 1139, 1140, 1144, 1146, 1151	175-225	HR, A, N, CD	115	450
	275-325	Q and T	70	290
	325-375	Q and T	45	200
	375-425	Q and T	35	170
Free Machining Plain Carbon Steels (Leaded), 11L17, 11L18, 12L13, 12L14	100-150	HR, A, N, CD	140	600
	150-200	HR, A, N, CD	130	625
	200-250	N, CD	110	400
Plain Carbon Steels, 1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514	100-125	HR, A, N, CD	110	425
	125-175	HR, A, N, CD	110	400
	175-225	HR, N, CD	90	350
	225-275	CD	65	250
1027, 1030, 1033, 1035, 1036, 1037, 1038, 1039, 1040, 1041, 1042, 1043, 1045, 1046, 1048, 1049, 1050, 1052, 1524, 1526, 1527, 1541	125-175	HR, A, N, CD	100	375
	175-225	HR, A, N, CD	85	325
	225-275	N, CD, Q and T	70	225
	275-325	Q and T	55	200
	325-375	Q and T	35	160
	375-425	Q and T	25	140
1055, 1060, 1064, 1065, 1070, 1074, 1078, 1080, 1084, 1086, 1090, 1095, 1548, 1551, 1552, 1561, 1566	125-175	HR, A, N, CD	90	350
	175-225	HR, A, N, CD	75	300
	225-275	N, CD, Q and T	60	200
	275-325	Q and T	45	160
	325-375	Q and T	30	145
	375-425	Q and T	15	125
Free Machining Alloy Steels (Resulphurized), 4140, 4150	175-200	HR, A, N, CD	100	400
	200-250	HR, N, CD	90	350
	250-300	Q and T	60	280
	300-375	Q and T	45	220
	375-425	Q and T	35	160
Free Machining Alloy Steels (Leaded), 41L30, 41L40, 41L47, 41L50, 43L47, 51L32, 52L100, 86L20, 86L40	150-200	HR, A, N, CD	115	425
	200-250	HR, N, CD	95	375
	250-300	Q and T	70	260
	300-375	Q and T	50	210
	375-425	Q and T	40	180
Alloy Steels, 4012, 4023, 4024, 4028, 4118, 4320, 4419, 4422, 4427, 4615, 4620, 4621, 4626, 4718, 4720, 4815, 4817, 4820, 5015, 5117, 5120, 6118, 8115, 8615, 8617, 8620, 8622, 8625, 8627, 8720, 8822, 94B17	125-175	HR, A, N, CD	100	400
	175-225	HR, N, CD	90	350
	225-275	CD, N, Q and T	60	250
	275-325	Q and T	50	200
	325-375	Q and T	40	175
	375-425	Q and T	25	150

* Abbreviations designate: HR, hot rolled; CD, cold drawn; A, annealed; N, normalized; Q and T, quenched and tempered; HB, Brinell hardness number; and HRC, Rockwell C scale hardness number.

Table 5-3 (*Cont.*) Cutting Speed in Feet per Minute for Milling Plain Carbon and Alloy Steels

Material AISI and SAE Steels	Hardness, HB*	Material Condition*	Cutting Speed, fpm	
			HSS	Carbide
Alloy Steels, 1330, 1335, 1340, 1345, 4032, 4037, 4042, 4047, 4130, 4135, 4137, 4140, 4142, 4145, 4147, 4150, 4161, 4337, 4340, 50B44, 50B46, 50B50, 50B60, 5130, 5132, 5140, 5145, 5147, 5150, 5160, 51B60, 6150, 81B45, 8630, 8635, 8637, 8640, 8642, 8645, 8650, 8655, 8660, 8740, 9254, 9255, 9260, 9262, 94B30	175-225	HR, A, N, CD	75	310
	225-275	N, CD, Q and T	60	260
	275-325	N, Q and T	50	210
	325-375	N, Q and T	35	180
	375-425	Q and T	20	140
Alloy Steels, E51100, E52100	175-225	HR, A, CD	65	300
	225-275	N, CD, Q and T	60	250
	275-325	N, Q and T	40	130
	325-375	N, Q and T	30	100
	375-425	Q and T	20	60
Ultra High Strength Steels (Not AISI) AMS 6421 (98B37 Mod.), AMS 6422 (98BV40), AMS 6424, AMS 6427, AMS 6428, AMS 6430, AMS 6432, AMS 6433, AMS 6434, AMS 6436, AMS 6442, 300M, D6 ac	220-300	A	60	250
	300-350	N	45	180
	350-400	N	20	130
	43-48 HRC	Q and T	..	100
	48-52 HRC	Q and T	..	60
Maraging Steels (Not AISI) 18% Ni Grade 200, 18% Ni Grade 250, 18% Ni Grade 300, 18% Ni Grade 350	250-325	A	50	250
	50-52 HRC	Maraged	..	60
Nitriding Steels (Not AISI) Nitralloy 125, Nitralloy 135, Nitralloy 135 (Mod.), Nitralloy 225, Nitralloy 230, Nitralloy N, Nitralloy EZ, Nitrex 1	200-250	A	60	280
	300-350	N, Q and T	25	200

* Abbreviations designate: HR, hot rolled; CD, cold drawn; A, annealed; N, normalized; Q and T, quenched and tempered; HB, Brinell hardness number; and HRC, Rockwell C scale hardness number.

Where: N = Spindle and milling cutter speed; rpm
V = Cutting speed; fpm, or m/min
D = Diameter of milling cutter; in. or mm
$\pi = 3.14$ (pi)

Example 5-2

A ½-inch (12.7 mm) diameter high-speed-steel end mill has four teeth and is to cut an O2 oil-hardening tool steel having a hardness of 200-220 HB. The cutting speed for this steel is 50 fpm ($50 \times .3048 = 15.2$ m/min). Calculate the spindle speed, using both inch and metric units.

Table 5-4. Cutting Speed in Feet per Minute for Milling Tool Steels

Material Tool Steels (AISI Types)	Hardness, HB*	Material Condition*	Cutting Speed, fpm HSS	Carbide
Water Hardening W1, W2, W5	150-200	A	85	250
Shock Resisting S1, S2, S5, S6, S7	175-225	A	55	215
Cold Work, Oil Hardening O1, O2, O6, O7	175-225	A	50	200
Cold Work, High Carbon High Chromium D2, D3, D4, D5, D7	200-250	A	40	150
Cold Work, Air Hardening				
A2, A3, A8, A9, A10	200-250	A	50	200
A4, A6	200-250	A	45	160
A7	225-275	A	40	140
Hot Work, Chromium Type H10, H11, H12, H13, H14, H19	150-200	A	60	250
	200-250	A	50	200
	325-375	Q and T	30	150
	48-50 HRC	Q and T	—	80
	50-52 HRC	Q and T	—	60
	52-54 HRC	Q and T	—	40
	54-56 HRC	Q and T	—	20
Hot Work, Tungsten Type H21, H22, H23, H24, H25, H26	150-200	A	55	200
	200-250	A	45	170
Hot Work, Molybdenum Type H41, H42, H43	150-200	A	55	180
	200-250	A	45	140
Special Purpose, Low Alloy L2, L3, L6	150-200	A	65	300
Mold				
P2, P3, P4, P5, P6	100-150	A	75	350
P20, P21	150-200	A	60	300
High-Speed Steel				
M1, M2, M6, M10, T1, T2, T6	200-250	A	50	175
M3-1, M4, M7, M30, M33, M34, M36, M41, M42, M43, M44, M46, M47, T5, T8	225-275	A	40	150
T15, M3-2	225-275	A	30	130

* Abbreviations designate: A, annealed; Q and T, quenched and tempered; and HB, Brinell hardness number.

$$N = \frac{12\,V}{\pi D} = \frac{12 \times 50}{\pi \times .5} = 382 \text{ rpm} \qquad N = \frac{1000\,V}{\pi D} = \frac{1000 \times 15.2}{\pi \times 12.7} = 381 \text{ rpm}$$

Milling Machine Table Feed Rate

The milling machine table feed rate should always be calculated in order to obtain the maximum production rate from the cutter and the machine. Using a machine table feed rate which has been determined by

Table 5-5. Cutting Speed in Feet per Minute for Milling Stainless Steels

Material Stainless Steels	Hardness, HB*	Material Condition*	Cutting Speed, fpm HSS	Carbide
Free Machining Stainless Steels				
(Ferritic), 430F, 430F Se	135-185	A	95	375
(Austenitic), 203EZ, 303, 303 Se, 303MA, 303Pb, 303Cu, 303 Plus X	135-185	A	90	325
	225-275	CD	75	300
(Martensitic), 416, 416 Se, 416 Plus X, 420F, 420F Se, 440F, 440F Se	135-185	A	95	375
	185-240	CD	80	325
	275-325	Q and T	50	225
	375-425	Q and T	20	100
Stainless Steels				
(Ferritic), 405, 409, 429, 430, 434, 436, 442, 446, 502	135-185	A	75	275
(Austenitic), 201, 202, 301, 302, 304, 304L, 305, 308, 321, 347, 348	135-185	A	60	200
	225-275	CD	50	180
(Austenitic), 302B, 309, 309S, 310, 310S, 314, 316, 316L, 317, 330	135-185	A	50	200
(Martensitic), 403, 410, 420, 501	135-175	A	75	325
	175-225	A	65	275
	275-325	Q and T	40	175
	375-425	Q and T	25	100
(Martensitic), 414, 431, Greek Ascoloy	225-275	A	55	225
	275-325	Q and T	45	180
	375-425	Q and T	25	100
(Martensitic), 440A, 440B, 440C	225-275	A	50	180
	275-325	Q and T	40	140
	375-425	Q and T	20	100
(Precipitation Hardening) 15-5PH, 17-4PH, 17-7PH, AF-71, 17-14Cu Mo, AFC-77, AM-350, AM-355, AM-362, Custom 455, HNM, PH13-8, PH14-8Mo, PH15-7Mo, Stainless W	150-200	A	60	200
	275-325	H	50	180
	325-375	H	40	110
	375-450	H	25	75

* Abbreviations designate: A, annealed; CD, cold drawn; Q and T, quenched and tempered; H, precipitation hardened; and HB, Brinell hardness number.

a guess can cause the milling cutter teeth to be overloaded or drastically underloaded, each of which will have an adverse effect on the cutter. The milling machine table feed rate is expressed in terms of inches per minute (in./min), or millimeters per minute (mm/min) on metric machines. To convert from one to the other: multiply in./min by 25.4 to obtain mm/min; divide mm/min by 25.4 to obtain in./min.

Chip Load per Tooth. The basic measure of the feed rate of milling cutters is the chip load per tooth, which for customary inch units is expressed in terms of inch per tooth (in./tooth); in SI metric units it is expressed in terms of millimeters per tooth (mm/tooth). They can be converted from one into the other as follows: multiply in./tooth by 25.4 to obtain mm/tooth; divide mm/tooth by 25.4 to obtain in./tooth. Recommended values of the basic feed rate are given in Tables 5-8 and 5-9 for different types of milling cutters and for different materials. Feed rates

Table 5-6. Cutting Speed in Feet per Minute for Milling Ferrous Cast Metals

Material Ferrous Cast Metals	Hardness, HB*	Material Condition*	Cutting Speed, fpm	
			HSS	Carbide
Gray Cast Iron				
ASTM Class 20	120-150	A	100	425
ASTM Class 25	160-200	AC	80	325
ASTM Class 30, 35, and 40	190-220	AC	70	250
ASTM Class 45 and 50	220-260	AC	50	190
ASTM Class 55 and 60	250-260	AC, HT	30	110
ASTM Type 1, 1b, 5 (Ni-Resist)	100-215	AC	50	200
ASTM Type 2, 3, 6 (Ni-Resist)	120-175	AC	40	190
ASTM Type 2b, 4 (Ni-Resist)	150-250	AC	30	180
Malleable Iron				
(Ferritic), 32510, 35018	110-160	MHT	110	475
(Pearlitic), 40010, 43010, 45006, 45008, 48005, 50005	160-200	MHT	80	375
	200-240	MHT	65	250
(Martensitic), 53004, 60003, 60004	200-255	MHT	55	225
(Martensitic), 70002, 70003	220-260	MHT	50	200
(Martensitic), 80002	240-280	MHT	45	130
(Martensitic), 90001	250-320	MHT	25	110
Nodular (Ductile) Iron				
(Ferritic), 60-40-18, 65-45-12	140-190	A	75	425
(Ferritic-Pearlitic), 80-55-06	190-225	AC	60	325
	225-260	AC	50	200
(Pearlitic-Martensitic), 100-70-03	240-300	HT	40	160
(Martensitic), 120-90-02	270-330	HT	25	90
	330-400	HT	—	30
Cast Steels				
(Low Carbon), 1010, 1020	100-150	AC, A, N	100	375
(Medium Carbon), 1030, 1040, 1050	125-175	AC, A, N	95	375
	175-225	AC, A, N	80	325
	225-300	AC, HT	60	250
(Low Carbon Alloy), 1320, 2315, 2320, 4110, 4120, 4320, 8020, 8620	150-200	AC, A, N	85	325
	200-250	AC, A, N	75	300
	250-300	AC, HT	50	225
(Medium Carbon Alloy), 1330, 1340, 2325, 2330, 4125, 4130, 4140, 4330, 4340, 8030, 80B30, 8040, 8430, 8440, 8630, 8640, 9525, 9530, 9535	175-225	AC, A, N	70	300
	225-250	AC, A, N	65	250
	250-300	AC, HT	50	200
	300-350	AC, HT	30	180
	350-400	HT	..	125

* Abbreviations designate: A, annealed; AC, as cast; N, normalized; HT, heat treated; MHT, malleablizing heat treatment; and HB, Brinell hardness number.

Table 5-7. Cutting Speed in Feet per Minute for Milling Light Metals and Copper Alloys.

Materials	Material Condition *	Cutting Speed, fpm	
		HSS	Carbide
Light Metals			
All Wrought Aluminum Alloys	CD	600	1200
	ST and A	500	1100
All Aluminum Sand and Permanent Mold Casting Alloys	AC	750	1400
	ST and A	600	1200
All Aluminum Die Casting Alloys †	AC	125	550
	ST and A	100	450
† except Alloys 390.0 and 392.0	AC	80	500
	ST and A	60	425
All Wrought Magnesium Alloys	A, CD, ST and A	800	2000
All Cast Magnesium Alloys	A, AC, ST and A	800	2000
Copper Alloys			
314 Leaded Commercial Bronze; 332 High Leaded Brass; 340 Medium Leaded Brass; 342 High Leaded Brass; 353 High Leaded Brass; 356 Extra-High Leaded Brass; 360 Free-Cutting Brass; 370 Free-Cutting Muntz Metal; 377 Forging Brass; 385 Architectural Bronze; 485 Leaded Naval Brass; 544 Free-Cutting Phosphor Bronze	A	300	650
	CD	350	600
226 Jewelry Bronze; 230 Red Brass; 240 Low Brass; 260 Cartridge Brass, 70%; 268 Yellow Brass; 280 Muntz Metal; 235 Low Leaded Brass; 365 Leaded Muntz Metal; 368 Leaded Muntz Metal; 443 Admiralty Brass (inhibited); 445 Admiralty Brass (inhibited); 651 Low Silicon Bronze; 655 High Silicon Bronze; 675 Manganese Bronze; 687 Aluminum Brass; 770 Nickel Silver; 796 Leaded Nickel Silver	A	200	500
	CD	250	550
102 Oxygen-Free Copper; 110 Electrolytic Tough Pitch Copper; 122 Phosphorus Deoxidized Copper; 170 Beryllium Copper; 172 Beryllium Copper; 175 Beryllium Copper; 210 Gilding, 95%; 220 Commercial Bronze; 502 Phosphor Bronze, 1.25%; 510 Phosphor Bronze, 5%; 521 Phosphor Bronze, 8%; 524 Phosphor Bronze, 10%; 614 Aluminum Bronze; 706 Copper Nickel, 10%; 715 Copper Nickel, 30%; 745 Nickel Silver; 752 Nickel Silver; 754 Nickel Silver; 757 Nickel Silver	A	100	200
	CD	110	225

* Abbreviations designate: A, annealed; AC, as cast; CD, cold drawn; and ST and A, solution treated and aged.

Table 5-8. Feed in Inches per Tooth (f_t) for Milling with High-Speed Steel Cutters

Material	Hardness, HB	End Mills							Plain or Slab Mills	Form Relieved Cutters	Face Mills and Shell End Mills	Slotting and Side Mills
		Depth of Cut, .250 in.			Depth of Cut, .050 in.							
		Cutter Diam., in.			Cutter Diam., in.							
		½	¾	1 and up	¼	½	¾	1 and up				
		Feed per Tooth, inch										
Cast Steel	100-180	.001	.003	.003	.001	.002	.003	.004	.003-.008	.004	.003-.012	.002-.008
	180-240	.001	.002	.003	.001	.002	.003	.003	.003-.008	.004	.003-.010	.002-.006
	240-300	.001	.002	.002	.0005	.002	.002	.002	.002-.006	.003	.003-.008	.002-.005
Zinc Alloys (Die Castings)	...	.002	.003	.004	.001	.003	.004	.006	.003-.010	.005	.004-.015	.002-.012
Copper Alloys (Brasses & Bronzes)	100-150	.002	.004	.005	.002	.003	.005	.006	.003-.015	.004	.004-.020	.002-.010
	150-250	.002	.003	.004	.001	.003	.004	.005	.003-.015	.004	.003-.012	.002-.008
Free Cutting Brasses & Bronzes	80-100	.002	.004	.005	.002	.003	.005	.006	.003-.015	.004	.004-.015	.002-.010
Cast Aluminum Alloys—As Cast	...	.003	.004	.005	.002	.004	.005	.006	.005-.016	.006	.005-.020	.004-.012
Cast Aluminum Alloys—Hardened	...	.003	.004	.005	.002	.003	.004	.005	.004-.012	.005	.005-.020	.004-.012
Wrought Aluminum Alloys—Cold Drawn	...	.003	.004	.005	.002	.003	.004	.005	.004-.014	.005	.005-.020	.004-.012
Wrought Aluminum Alloys—Hardened	...	.002	.003	.004	.001	.002	.003	.004	.003-.012	.004	.005-.020	.004-.012
Magnesium Alloys	...	.003	.004	.005	.003	.004	.005	.007	.005-.016	.006	.008-.020	.005-.012
Ferritic Stainless Steel	135-185	.001	.002	.003	.001	.002	.003	.003	.002-.006	.004	.004-.008	.002-.007
Austenitic Stainless Steel	135-185	.001	.002	.003	.001	.002	.003	.003	.003-.007	.004	.005-.008	.002-.007
	185-275	.001	.002	.003	.001	.002	.002	.002	.003-.006	.003	.004-.006	.002-.007
Martensitic Stainless Steel	135-185	.001	.002	.002	.001	.002	.003	.003	.003-.006	.004	.004-.010	.002-.007
	185-225	.001	.002	.002	.001	.002	.002	.003	.003-.006	.004	.003-.008	.002-.007
	225-300	.0005	.002	.002	.0005	.001	.002	.002	.002-.005	.003	.002-.006	.002-.005
Monel	100-160	.001	.003	.004	.001	.002	.003	.004	.002-.006	.004	.002-.008	.002-.006

Table 5-8 (*Cont.*) Feed in Inches per Tooth (f_t) for Milling with High-Speed Steel Cutters

Material	Hardness, HB	End Mills							Plain or Slab Mills	Form Relieved Cutters	Face Mills and Shell End Mills	Slotting and Side Mills
		Depth of Cut, .250 in.			Depth of Cut, .050 in.							
		Cutter Diam., in.			Cutter Diam., in.							
		½	¾	1 and up	¼	½	¾	1 and up				
		Feed per Tooth, inch										
Free Machining Plain Carbon Steels	100-185	.001	.003	.004	.001	.002	.003	.004	.003-.008	.005	.004-.012	.002-.008
Plain Carbon Steels, AISI 1006 to 1030; 1513 to 1522	100-150	.001	.003	.003	.001	.002	.003	.004	.003-.008	.004	.004-.012	.002-.008
	150-200	.001	.002	.003	.001	.002	.002	.003	.003-.008	.004	.003-.012	.002-.008
AISI 1033 to 1095; 1524 to 1566	120-180	.001	.003	.003	.001	.002	.003	.004	.003-.008	.004	.004-.012	.002-.008
	180-220	.001	.002	.003	.001	.002	.002	.003	.003-.008	.004	.003-.012	.002-.008
	220-300	.001	.002	.002	.001	.001	.002	.003	.002-.006	.003	.002-.008	.002-.006
Alloy Steels having less than 3% Carbon. Typical examples: AISI 4012, 4023, 4027, 4118, 4320, 4422, 4427, 4615, 4620, 4626, 4720, 4820, 5015, 5120, 6118, 8115, 8620, 8627, 8720, 8822, 9310, 93B17	125-175	.001	.003	.003	.001	.002	.003	.004	.003-.008	.004	.004-.012	.002-.008
	175-225	.001	.002	.003	.001	.002	.003	.003	.003-.008	.004	.003-.012	.002-.008
	225-275	.001	.002	.003	.001	.001	.002	.003	.002-.006	.003	.003-.008	.002-.006
	275-325	.001	.002	.002	.001	.001	.002	.002	.002-.005	.003	.002-.008	.002-.005
Alloy Steels have 3% Carbon or more. Typical examples: AISI 1330, 1340, 4032, 4037, 4130, 4140, 4150, 4340, 50B40, 50B60, 5130, 51B60, 6150, 81B45, 8630, 8640, 86B45, 8660, 8740, 94B30	175-225	.001	.002	.003	.001	.002	.003	.004	.003-.008	.004	.003-.012	.002-.008
	225-275	.001	.002	.003	.001	.001	.002	.003	.002-.006	.003	.003-.010	.002-.006
	275-325	.001	.002	.002	.001	.001	.002	.003	.002-.005	.003	.002-.008	.002-.005
	325-375	.001	.002	.002	.001	.001	.002	.002	.002-.004	.002	.002-.008	.002-.005
Tool Steel	150-200	.001	.002	.002	.001	.002	.003	.003	.003-.008	.004	.003-.010	.002-.006
	200-250	.001	.002	.002	.001	.002	.002	.003	.002-.006	.003	.003-.008	.002-.005
Gray Cast Iron	120-180	.001	.003	.004	.002	.003	.004	.004	.004-.012	.005	.005-.016	.002-.010
	180-225	.001	.002	.003	.001	.002	.003	.003	.003-.010	.004	.004-.012	.002-.008
	225-300	.001	.002	.002	.001	.001	.002	.002	.002-.006	.003	.002-.008	.002-.005
Ferritic Malleable Iron	110-160	.001	.003	.004	.002	.003	.004	.004	.003-.010	.005	.005-.016	.002-.010
Pearlitic-Martensitic Malleable Iron	160-200	.001	.003	.004	.001	.002	.003	.004	.003-.010	.004	.004-.012	.002-.008
	200-240	.001	.002	.003	.001	.002	.003	.003	.003-.007	.004	.003-.010	.002-.006
	240-300	.001	.002	.002	.001	.001	.002	.002	.002-.006	.003	.002-.008	.002-.005

Table 5-9. Feed in Inches per Tooth for Milling with Cemented Carbide Cutters

Material	Hardness, HB	Face Mills	Slotting and Side Mills
		Feed per Tooth, inch	
Free Machining Plain Carbon Steels	100-185	.008-.020	.003-.010
Plain Carbon Steels, AISI 1006 to 1030, 1513 to 1522	100-150	.008-.020	.003-.010
	150-200	.008-.020	.003-.010
Plain Carbon Steels, AISI 1033 to 1095, 1524 to 1566	120-180	.005-.020	.003-.010
	180-220	.005-.020	.003-.010
	220-300	.003-.012	.003-.008
Alloy Steels having less than .3% Carbon content. Typical examples: AISI 4012, 4023, 4027, 4118, 4320, 4422, 4427, 4615, 4620, 4626, 4720, 4820, 5015, 5120, 6118, 8115, 8620, 8627, 8720, 8822, 9310, 93B17	125-175	.006-.020	.003-.010
	175-225	.006-.020	.003-.010
	225-275	.006-.016	.003-.010
	275-325	.004-.012	.003-.008
	325-375	.003-.008	.003-.007
Alloy Steels having .3% Carbon content, or more. Typical examples: AISI 1330, 1340, 4032, 4037, 4130, 4140, 4150, 4340, 50B40, 50B60, 5130, 51B60, 6150, 81B45, 8630, 8640, 86B45, 8660, 8740, 94B30	175-225	.005-.020	.003-.010
	225-275	.004-.012	.003-.008
	275-325	.003-.010	.003-.008
	325-375	.003-.008	.003-.007
Tool Steels	200-275	.004-.012	.003-.007
	275-325	.003-.010	.003-.006
	36-45 HRC	.003-.006	.002-.005
	45-55 HRC	.003-.005	.002-.003
Ferritic Stainless Steels	110-160	.005-.015	.003-.010
Austenitic Stainless Steels	135-185	.005-.012	.003-.010
	185-275	.005-.010	.003-.008
Martensitic Stainless Steel	135-185	.005-.015	.003-.010
	185-225	.005-.010	.003-.008
	225-300	.004-.008	.003-.007
Precipitation Hardening Stainless Steels	Annealed	.004-.012	.003-.010
	275-350	.003-.008	.002-.005
	350-450	.002-.005	.002-.004
Cast Steel	100-180	.008-.020	.003-.010
	180-240	.005-.016	.003-.010
	240-300	.004-.012	.003-.008
Gray Cast Iron	140-185	.008-.020	.005-.012
	185-225	.008-.016	.005-.010
	225-300	.005-.012	.004-.008
Ferritic Malleable Iron	110-160	.005-.020	.004-.012
Pearlitic-Martensitic Malleable Iron	160-200	.005-.020	.003-.010
	200-240	.005-.016	.003-.010
	240-300	.004-.010	.003-.008
Nodular (Ductile) Iron	140-200	.008-.020	.003-.010
	200-275	.006-.014	.003-.008
	275-325	.005-.012	.003-.007
	325-400	.003-.008	.002-.004
Copper Alloys (Brasses and Bronzes)	100-150	.005-.020	.003-.012
	150-250	.004-.014	.003-.010

Table 5-9 (*Cont.*) Feed in Inches per Tooth for Milling with Cemented Carbide Cutters

Material	Hardness, HB	Face Mills	Slotting and Side Mills
		Feed per Tooth, inch	
Wrought and Cast Aluminum Alloys	....	.005-.020	.005-.020
Wrought and Cast Magnesium Alloys	....	.005-.020	.005-.020
Superalloys	....	.003-.010	.002-.006
Titanium Alloys	....	.003-.010	.002-.006
Nickel Alloys	....	.003-.010	.002-.006
Monel	....	.003-.010	.002-.006
Plastics, Hard Rubber, etc.	....	.003-.015	.003-.012

less than approximately .001 in./tooth (0.03 mm/tooth) should not be used except with small end mills and when milling certain very hard materials. At such low feed rates the teeth will tend to rub against the workpiece instead of penetrating to form a chip, resulting in excessive tool wear. With a dull cutter this may occur at even higher feed rates.

Table Feed Rate Formula. The formula for the table feed rate, given below, can be used either with customary inch or SI metric units, but not both at the same time.

$$f_m = f_t\, n_t\, N \tag{5-3}$$

Where: f_m = Feed rate of table; in./min, or mm/min
f_t = Feed rate of cutter; in./tooth, or mm/tooth
n_t = Number of teeth on cutter
N = Spindle speed, or cutter speed; rpm

Example 5-2

A ½-inch (12.7 mm) diameter high-speed steel end mill having four teeth is to be used to mill a .250-inch (6.35-mm) deep slop in D2 high carbon, high chrome, tool steel which has been annealed to a hardness of 200-210 HB. The spindle speed to be used is 300 rpm. Calculate the table feed rate by using the inch units and then by using the metric units.

The feed rate selected from Table 5-8 for a ½-inch end mill is .001 in./tooth, or 0.025 mm per/tooth.

Inch units: $f_m = f_t\, n_t\, N = .001 \times 4 \times 300 = 1.2$ in./min
Metric units: $f_m = f_t\, n_t\, N = 0.025 \times 4 \times 300 = 30$ mm/min (1.2 in./min)

Estimating Milling Power

The available power on any machine tool places a limit on the size of the cut it can take. When a large amount of metal must be removed from

$$f_m=\frac{Q_{max}}{w\ d}=\frac{3.18}{5\times .250}$$
$$=2.5 \text{ in./min} \quad \text{(rounded)}$$

$$N=\frac{f_m}{f_t\ n_t}=\frac{2.5}{.005\times 10}$$
$$=50 \text{ rpm}$$

$$V=\frac{\pi\ D\ N}{12}=\frac{\pi\times 8\times 50}{12}$$
$$=105 \text{ fpm}$$

This cutting speed is below that recommended for cemented carbide and very close to the cutting speed recommended for high-speed steel. On the 5 hp milling machine, it is evident that the maximum production rate can be obtained with a high-speed-steel cutter. The tool steel block may be machined by shell end milling or by slab milling; in this case slab milling is selected. A 4-inch diameter, 6-inch wide, heavy-duty plain milling cutter having 10 teeth will be used. The feed rate selected for this cutter is .005 in./tooth. From the tables, $W=1.10$ and $C=1.19$.

$$Q_{max}=\frac{P_m\ E}{K_p\ C\ W}=\frac{5\times .80}{.88\times 1.19\times 1.10}$$
$$=3.47 \text{ in.}^3\text{/min}$$

$$f_m=\frac{Q_{max}}{w\ d}=\frac{3.47}{5\times .250}$$
$$=2.75 \text{ in./min} \quad \text{(rounded)}$$

$$N=\frac{f_m}{f_t\ n_t}=\frac{2.75}{.005\times 10}$$
$$=55 \text{ rpm}$$

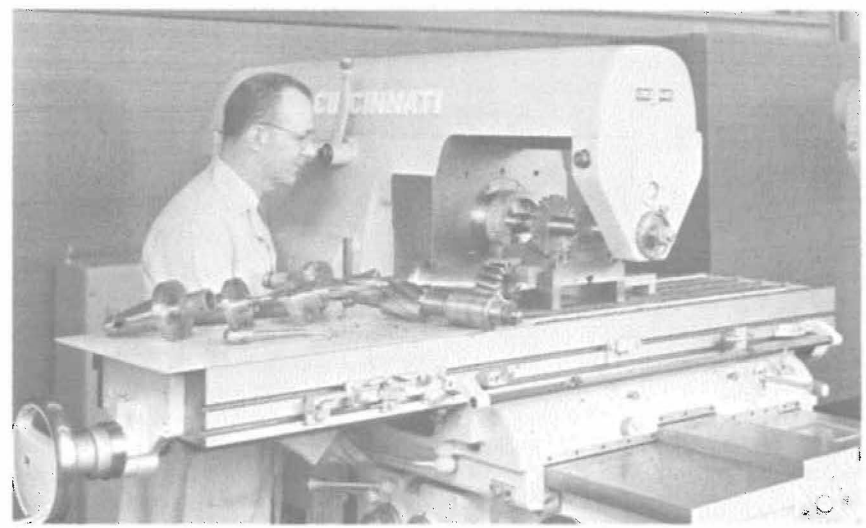
$$V=\frac{\pi\ D\ N}{12}=\frac{\pi\times 4\times 55}{12}$$
$$=58 \text{ fpm}$$

While this recommendation is perfectly valid for the conditions in the example that were examined, it is not a general recommendation. Each job is unique and must be examined on its own merits. Moreover, it is usually necessary to make small adjustments to suit the spindle speeds and the feed rates that are available on the machine. Generally, a set of cutting conditions can be found that will utilize the capacity of the machine when making such adjustments.

CHAPTER 6

Milling Machine Operations

Milling machines are used to perform a large variety of machining operations. In addition to those that can be classified as strictly milling operations using milling cutters, other operations, such as slotting, drilling, boring, reaming, etc., which do not utilize milling cutters and are performed on other machine tools, are often also performed on the milling machine. An example of the variety of operations that can be performed can be obtained by studying Fig. 6-1. The different kinds of cutting tools used to machine the part that is clamped to the table can be seen in the illustration. Although much of the work done on a milling machine involves the production of plane or contoured surfaces, large- and small-diameter holes are also frequently produced. Operations involving the use of the dividing head will be treated later in chapters exclusively devoted to this topic.



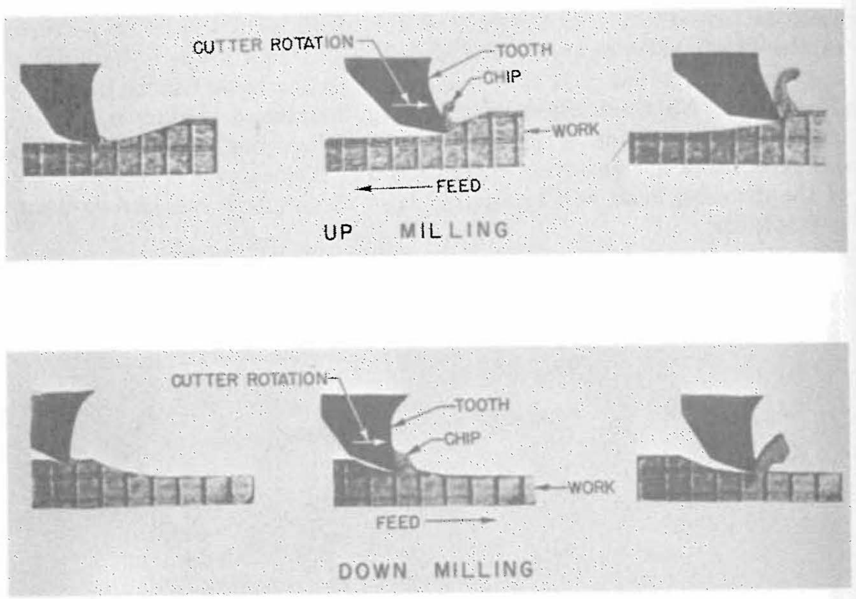
Courtesy of Cincinnati Milacron

Fig. 6-1. Straddle milling the inside face of a casting. Tools for other milling operations are placed on a board to protect the milling machine and the cutting edges from damage.

Many of the principles pertaining to the operation of other machine tools discussed in previous chapters are used in conjunction with work commonly done on the milling machine. These include the principles of drilling, reaming, boring, and precision hole location. The principles of clamping in making a setup on planers apply equally well to clamping workpieces directly to the milling-machine table. The principles involved in doing accurate work in a shaper vise should be reviewed, for they also apply to milling-machine work when the part is held in a vise. Although it would be repetitious to cover all of this material again in this chapter, these principles must be kept in mind when the workpiece is set up and cuts taken on the milling machine.

Conventional and Climb Milling

Conventional milling is also called *up milling*. As illustrated in the upper view of Fig. 6-2, the direction of motion of the milling cutter tooth as it engages the work is opposite from the direction of the movement of



Courtesy of Cincinnati Milacron

Fig. 6-2. Upper view—conventional up milling. Lower view—climb or down milling.

the work caused by the table feed. The cutting forces resulting from this method of milling will keep the feed screw nut against the same side of the feed screw thread as when feeding the table toward the cutter without taking a cut. Thus, the table and the workpiece will never have a tendency to pull toward the cutter because of lost motion between the nut and the table feed screw.

In conventional milling a very thin chip is formed at the beginning of the cut. The thickness of the chip increases as the tooth proceeds along its path until it reaches a maximum in the position where the tooth leaves the workpiece.

Climb milling has an advantage when certain materials, such as aluminum, are milled, because it produces a much better surface finish on the workpiece than can be obtained by conventional milling. Climb milling is also called *down milling*. As the lower view in Fig. 6-2 shows, the milling cutter tooth and the workpiece move in the same direction. The velocity of the milling cutter tooth is faster than the velocity of the table feed, which moves the work into the cutter and thereby forms the chip. The cutting force resulting from climb milling is in the same direction as the feed. This will cause the feed screw, which is attached to the table, to pull away from the side of the feed screw nut against which it was bearing as the work was approaching the cutter. In effect, since the workpiece will be pulled into the cutter by the action of the cutting forces, the workpiece, the cutter, and the milling-machine arbor can all be seriously damaged. Climb milling, therefore, must not be used in most instances, unless the milling machine is equipped with a backlash eliminator. Light profiling-type cuts can often be taken with end milling cutters using the climb milling method. The magnitude of the cutting forces is usually low, and the weight of the table is sufficient to prevent the workpiece from being pulled into the cutter. Sometimes clamping the table lightly will add an additional drag to the table so that the work will not be pulled into the cutter.

Figure 6-2 shows that in climb or down milling the maximum chip thickness occurs at a point close to the position where the tooth makes the initial contact with the workpiece. As the cut continues the chip thickness decreases, reaching a minimum where the tooth leaves the workpiece.

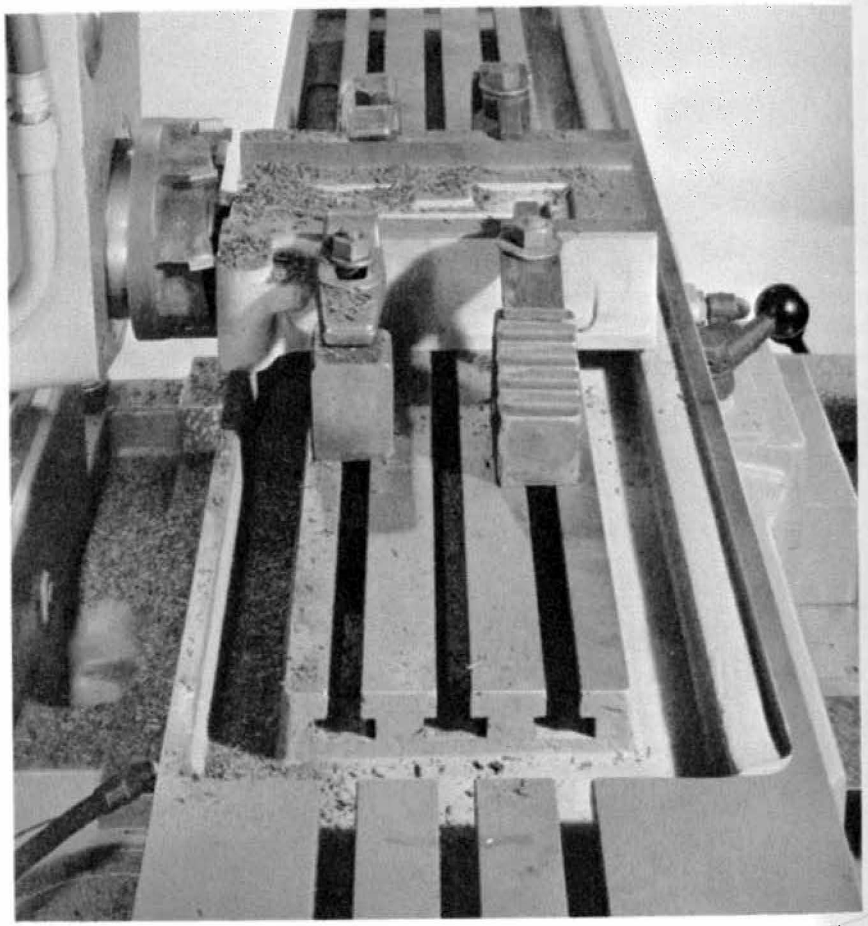
Setting Up the Workpiece

For most jobs done on a milling machine, setting up the workpiece is the most difficult and critical part of the work. The workpiece must not only be securely clamped, but also be held on the machine in such a position that each surface to be machined will, when finished, be accurately aligned with other surfaces on the part. Accuracy in making a setup is essential on most jobs; without it, close tolerance work cannot be done, unsatisfactory workpieces that have to be scrapped will result. Each setup must be planned in advance and then carried out with care and patience.

The first step, then, is to plan the setup. While each setup is unique, inasmuch as it is made to suit the part to be milled, there are several basic types of setups, which will now be described. Large workpieces are usually placed on the top of the milling machine table, and are clamped there by means of strap clamps and T-slot bolts, as shown in Figs. 6-4, 6-5, and 6-12. The principles of applying strap clamps have been treated in detail in previous chapters and will not be repeated here, except to say that the steel cutters. The larger cutter, having carbide teeth, will therefore not limit the spindle speed that can be used.

Face Milling

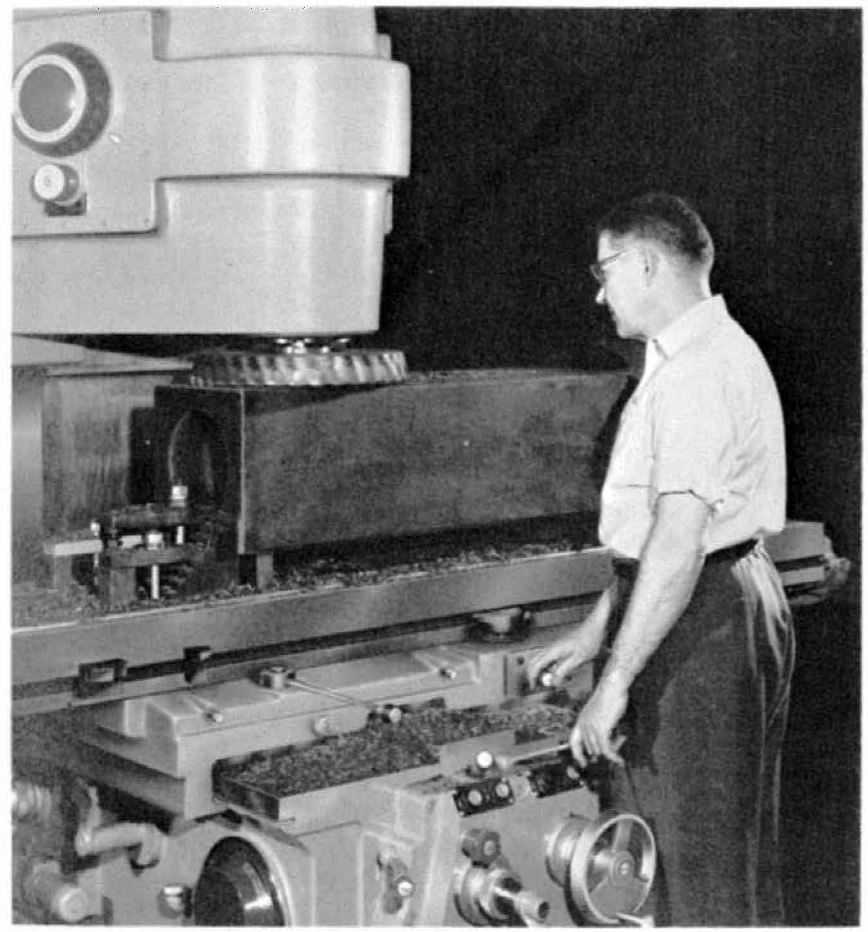
Face milling is an operation for producing plane or flat surfaces using a face milling cutter. Very fast metal-removal rates are possible with face milling, especially when cemented carbide face milling cutters are used as these can be operated at high cutting speeds. A typical cemented carbide face milling operation is shown in Fig. 6-15. The setup is designed so that the cutter will cut on the downward path of the teeth. There are two important reasons for doing this: 1. The cutting forces push the casting



Courtesy of the Brown & Sharpe Manufacturing Company

Fig. 6-15. Face milling with a carbide face milling cutter on a horizontal-spindle milling machine.

down against the table instead of against the strap clamps; and, 2. The flow of the chips removed is directed downward and away from the operator. When face milling, the cutting forces are often very large and possible movement of the workpiece must be prevented by directing these forces toward a solid object, such as the milling machine table, to solid stops firmly clamped to the table, or to the solid jaw of a vise. The flow of chips resulting from a high-speed cemented carbide face milling operation can be hazardous; they must be directed away from the operator and protection must also be provided for other persons in the vicinity. Figure 6-16 illustrates a heavy face milling operation on a vertical milling machine. Two step blocks are clamped to the table at each end of the casting to prevent it from moving. The cut is taken by feeding from the operator's



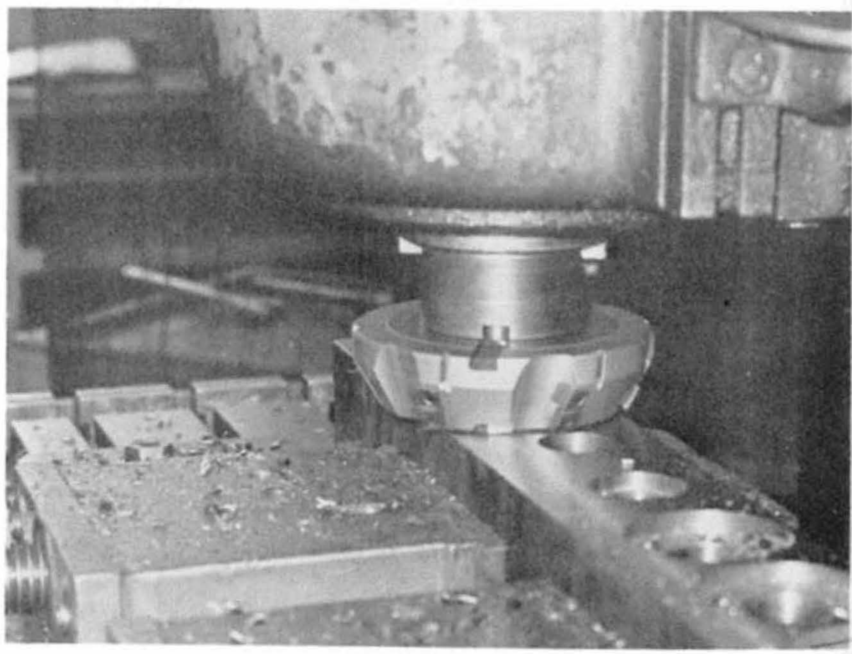
Courtesy of Cincinnati Milacron

Fig. 6-16. Face milling with a carbide face milling cutter on a vertical milling machine.

left to his right so that the chip flow is directed toward the back of the machine and away from the operator.

In face milling the automatic power feed should be kept engaged until the cutter is completely clear of the workpiece; otherwise the surface produced may not be perfectly flat. Although most of the metal is cut off by the primary cut taken by the peripheral teeth of the face milling cutter, the secondary cut taken at the "back" part of the turn as the cutter revolves will remove a small amount of metal. If the secondary cut is not taken completely across the workpiece, it is obvious that somewhat more metal will be removed from that portion of the workpiece on which the secondary cut has been taken. The secondary cut will usually leave telltale feed marks on the workpiece. On some milling machines (see Fig. 4-29) these feed marks can be eliminated by slightly tilting the spindle into the cut. When a very smooth surface finish is required, a face milling cutter having a wiper blade or finishing inserts mounted on the face of the cutter should be used. The cemented carbide face milling cutter in Fig. 6-17 has two finishing inserts mounted on its face that produce a very smooth finish on the milled surface.

Face milling cuts may be taken by straddling the cut as shown in view A, Fig. 6-18. However, it is usually better to cut on either one side or the



Courtesy of The Ingersoll Milling Machine Co., Cutting Tool Div.

Fig. 6-17. Taking a finishing cut on a surface with a face milling cutter having finishing teeth on face to produce a smooth surface.

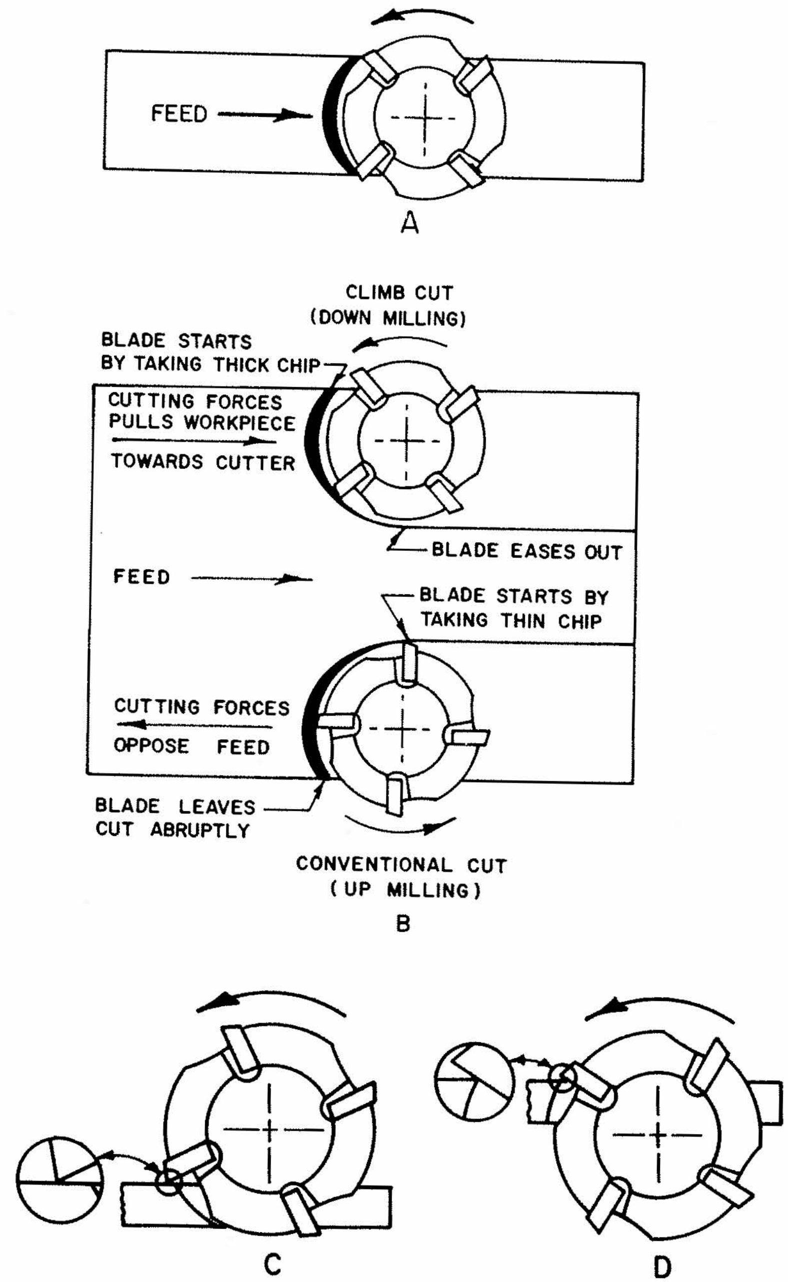


Fig. 6-18. The fundamental methods of face milling. A. Taking a straddle cut. B. Taking a climb cut and a conventional cut. C. Face milling with body outside of workpiece; blade makes initial contact at tip of edge. D. Face mill with body inside workpiece; blade makes contact away from tip of edge.

other of the face of the cutter as in view B. Whenever possible, the workpiece should be cut in one pass; the cutter diameter should be larger than the width of the cut by a ratio of 4 to 3, or 3 to 2. Very wide surfaces must be milled by a series of overlapping cuts. Overlapping face milling cuts should always be taken with the feed used to take each cut going in the same direction. The cutter is returned to the starting point for each new cut by moving the table so that the cutter passes around the outside of the surface being cut.

Face milling cuts can be taken by conventional cut (up milling) or by a climb cut (down milling), as shown in view B, Fig. 6-18. On a conventional cut the chip is initially very thin; it increases and then decreases slightly until it leaves the cut somewhat abruptly. When the face milling cutter has sharp, positive rake blades good results are obtained by this method. The cutting forces generated by the blades oppose the feed thus preventing the backlash between the feedscrew and nut from causing the workpiece to be pulled into the cutter. It is recommended that a conventional cut be used for face milling on older machines and on machines that do not have a blacklash eliminator (see Fig. 4-31). The disadvantage of the conventional cut is that dull cutters and negative-rake face milling cutters have difficulty in starting the cut in the very thin chip region. In this area dull, honed, and negative-rake cutting edges tend to rub against rather than penetrate the surface of the workpiece, thereby generating heat and causing the cutting edge to wear. Also, the rubbing action work-hardens the work surface which may, in some cases, be severe enough to make penetration by the next blade more difficult.

On a climb cut, the blade, or insert, of the cutter starts by taking a heavy cut and eases out of the cut by producing a thin chip. Since most cemented carbide face milling cutters have a negative radial-rake angle, the face milling cut taken with these cutters should be a climb cut whenever possible. The climb cut, however, tends to pull the work into the cutter; therefore, this method should not be used unless the machine is either equipped with a backlash eliminator or the workpiece is heavy enough to prevent it from being pulled by the cutter. Also, climb cuts are not recommended when the edge of the workpiece into which the initial penetration is made by the cutting edges of the cutter has a heavy abrasive scale.

A very important consideration when using cemented carbide face milling cutters is the entry angle of the cutter teeth. This is illustrated in views C and D, Fig. 6-18. Cemented carbides, although very hard, are also brittle. As a result, they do not withstand shock loads well, especially when they occur in a weak area, such as at the tip of the cutting edge. Shock loads at the tip of the cutting edge very often cause the cutting edge to fail by breaking. When taking a climb cut, the initial contact with the edge of the workpiece causes a shock load, and the position on the cutting edge where this will occur is determined by the entry angle. In view C, Fig. 6-18, the center of the face milling cutter body is outside the edge of the workpiece against which the cutting edges make their initial contact. When the cutter body is in this position, the initial load on the teeth is taken at the very tip of the cutting edge, where, as mentioned previously, it is weak and liable to break. When the center of the cutter body is inside the edge of the workpiece against which initial contact is made by the teeth, as in view D, the initial contact on the teeth is behind the tip of the cutting edge where the teeth have greater strength and are better able to withstand shock loads. Therefore, whenever possible, when using a cemented carbide face milling cutter, the center of the cutter body should be inside the edge of the workpiece against which the teeth make their initial contact, as in views B and D.

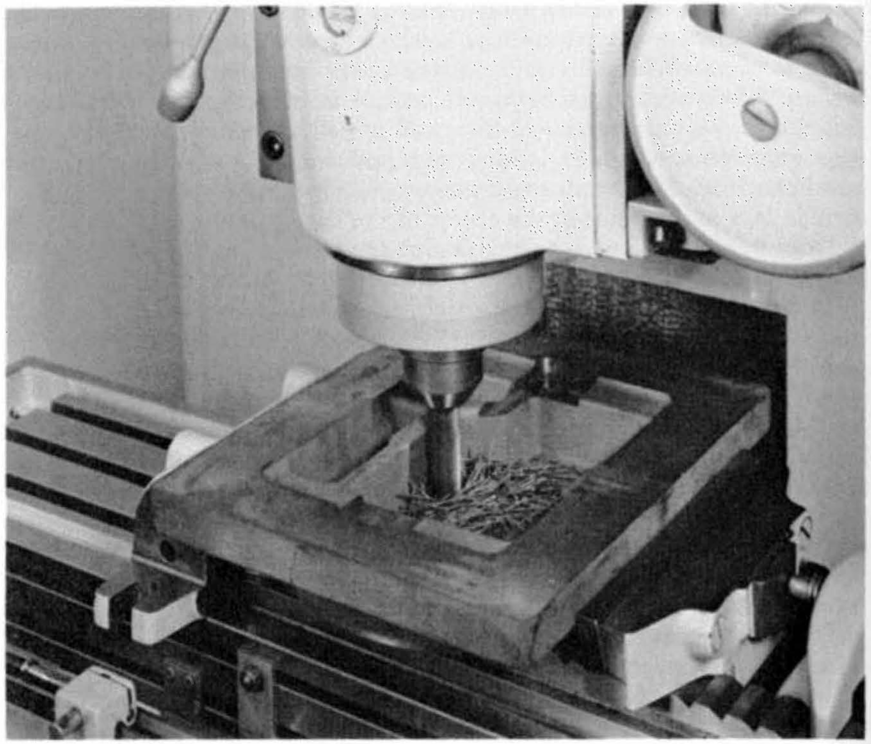
Another serious problem that sometimes occurs when face milling with cemented carbides is *chip sticking*. A chip may stick or be welded to the face of a carbide blade, or insert, as it leaves the cut and be carried around until the blade or insert starts into the next cut. When this blade or insert starts to penetrate the workpiece, the welded-on chip sets up a very high load which frequently causes the cutting edge to break. A prime cause of chip welding is using too light a feed per tooth; and here increasing the feed will often help. Other causes of chip sticking may be due to the grade of carbide used, choosing a speed that is too slow when cutting a soft material, too high when cutting a hard material, or taking a cut that is too wide. These causes suggest the steps to be taken to overcome this problem. Occasionally, changing the cut from a conventional to a climb cut will be helpful.

Face milling cutters, depending on their size, are capable of very high metal removal rates which require a large amount of power. The available power on the machine often places a limit on the size of cut and the type of face milling cutter that can be used. When planning to take a heavy face milling cut it is always advisable to determine in advance whether the machine has the power to take the cut, using the method described in Chapter 5 of this volume.

End Milling

End milling cutters are very versatile cutters that can perform a wide variety of operations. Their usefulness is extended by the many types of end milling cutters that are available. An end milling cutter is shown taking a facing cut on an inside surface of a casting in Fig. 6-19. The casting is clamped in an All-Steel vise, and the cut is taken by engaging the transverse automatic power feed. The facing cut shown is taken by using only the peripheral teeth or cutting edges of the end milling cutter, making the cut similar to slab milling. In Fig. 6-19 a pocket that has previously been milled out with the end mill can be seen just behind the cutter. When cutting the pocket the peripheral teeth of the end mill remove most of the metal, but the end teeth do take a light scraping cut which is similar to the action of the end teeth on a face milling cutter.

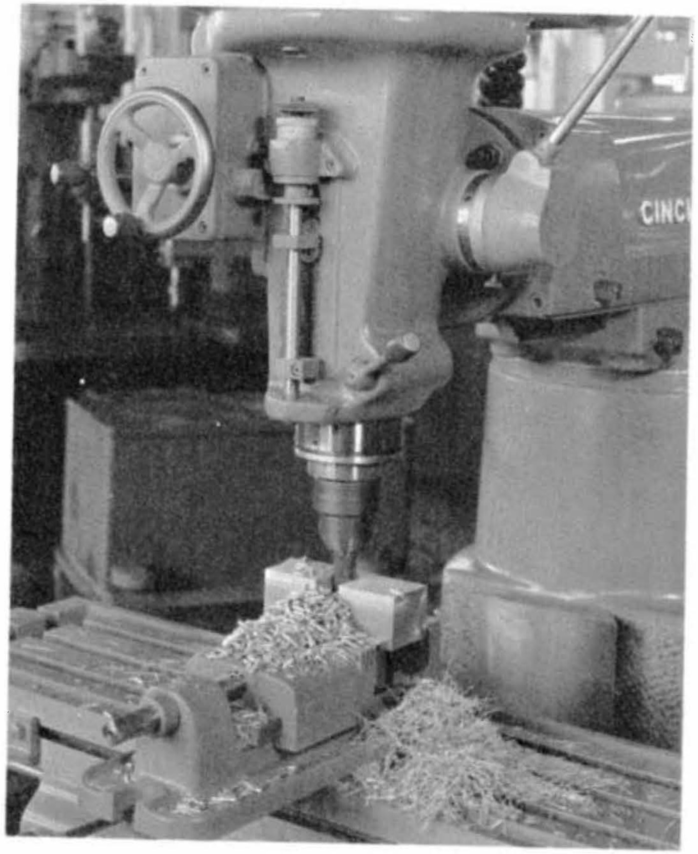
End mills are often used to cut slots, as shown in Fig. 6-20. One problem that can occur when slotting with an end milling cutter is the "par-



Courtesy of Cincinnati Milacron

Fig. 6-19. Facing a surface inside a casting with an end mill.

allelogram" or "wobble" slot, which is a slot with sides that are parallel to each other but not perpendicular to the bottom of the slot. This condition occurs most frequently when a two-fluted end mill is used with a large helix angle. It also occurs when an excessive flute length is projecting from the spindle in which the end mill is held. The principal cause of the parallelogram slot is the deflection of the end milling cutter brought about when one flute is cutting into the material while the other flute is not cutting and is unsupported by a side of the slot. This condition is prevented by increasing the spindle speed and decreasing the feed rate so that the chip load on each tooth is reduced. Decreasing the length of the end that is projecting from the end of the spindle will improve the rigidity of the setup and thereby reduce the tendency of the end mill to deflect and to produce a parallelogram slot. The parallelogram slot can also be prevented by using a four-fluted center-cutting type of end mill which receives better support from the sides of the slot. When the correct spindle speed and table feed rates are used, and when the setups of the work and the cutting tool are rigid, a straight slot can be cut with a two-fluted end mill.



Courtesy of Cincinnati Milacron

Fig. 6-20. Milling a slot with an end milling cutter.

Shell end milling cutters are usually used like face milling cutters to mill plane surfaces, and the principles of applying the shell end milling cutters are the same as previously described for face milling cutters. They are also frequently used to cut square corners and to mill wide slots. To mill plane surfaces they are sometimes used as a plain milling cutter by cutting with their peripheral teeth only. A typical shell end milling operation is shown in Fig. 6-21.

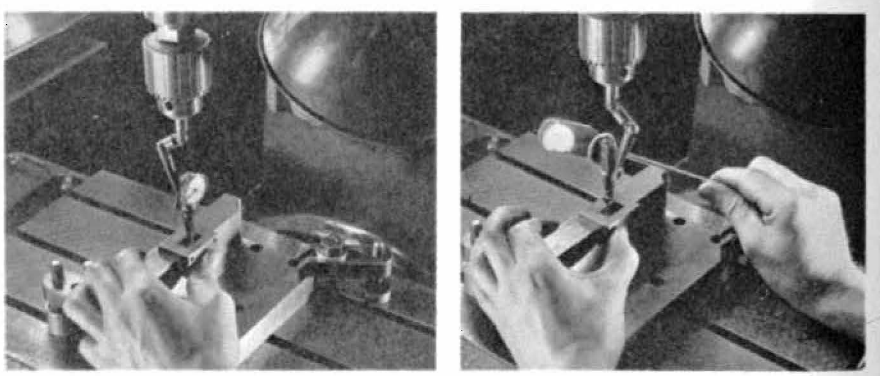
Milling Keyseats

Keyseats are slots that are cut lengthwise in shafts in which keys are held. The keys are used to transmit the driving torque that is conveyed to or from the shaft by pulleys, gears, or sprockets which are attached to the shaft. When required, keyseats are also used to align various machine elements which are sometimes attached to shafts. In addition, keys are

A faster method of picking up an edge is by using the edge finder developed by the Moore Special Tool Co., Inc. Shown in Fig. 15-22 by the kind permission of this company is a detailed drawing of this edge finder. All of the critical dimensions on this edge finder are made to gagemaker's tolerances, including the parallelism and perpendicularity of the gaging surfaces. The upper part of this edge finder has a .400 in. (10 mm in metric) slot. The inner surfaces of this slot are gaging surfaces against which indicator readings are made. The inside edge of the lower leg is also a gaging surface. It is located in the exact center of the slot and, when used, it is held against the edge on the workpiece that is to be picked up. Figure 15-23 illustrates the procedure for using the edge finder to pick up the edge. The lower inside face of the leg is held against the edge on the workpiece while at the same time the two inside faces of the slot are indicated with a dial test indicator held in a jig borer spindle. When indicating each face, the spindle is rotated by hand to obtain the largest indicator reading. The table is moved to the position where both maximum indicator readings are equal; in this position the axis of the spindle will be aligned with the edge of the workpiece.

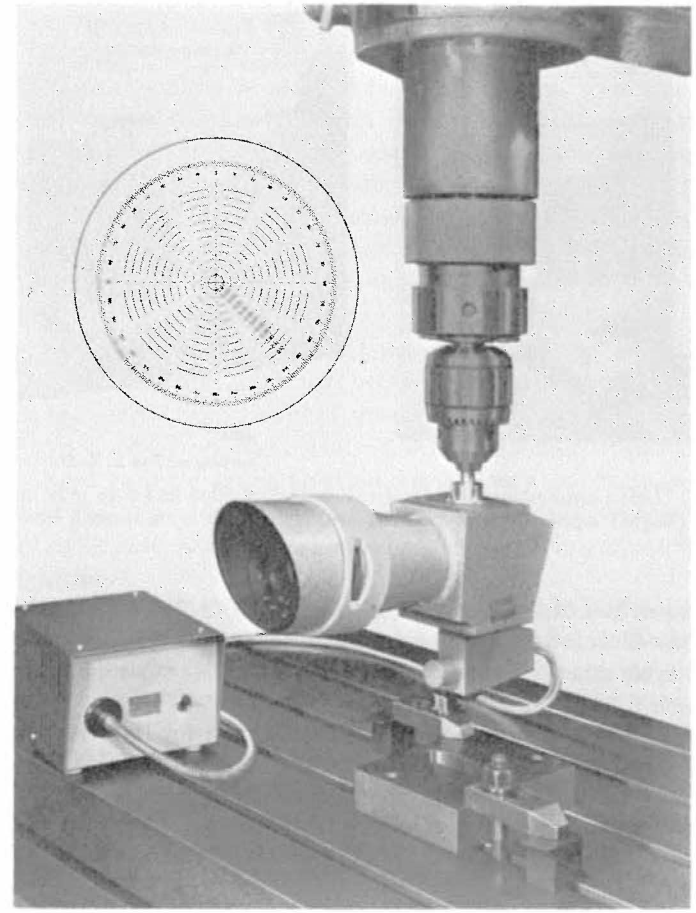
The centering projector, Fig. 15-24, is designed to be used as an edge finder. The eyepiece can be held stationary as the spindle is slowly turned. The magnification is such that "tenths" (.0001 inch) can be seen and the field of vision is large enough to show a fairly large area of the workpiece. Two crossed hairlines act as the center of the optical system. The center of the optical system should be checked, but when verified, it is only necessary to move the table until the edge of the workpiece coincides with one of the hairlines on the microscope in order to align the axis of the spindle on the edge of the workpiece.

A very popular edge finder is shown in Fig. 15-25. It can be obtained with a cone point at one end and a contact cylinder at the other end, or



Courtesy of The Moore Special Tool Company

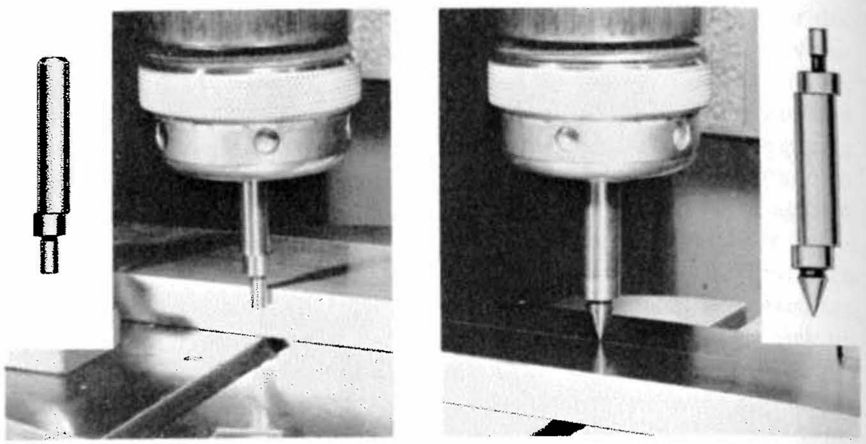
Fig. 15-23. Procedure for picking up the edge of a workpiece using a Moore Edge Finder.



Courtesy of Titan Tool Supply Co., Inc.

Fig. 15-24. Centering projector which can be used to pick up edges, small undercuts, circles, holes, radii, and outside diameters.

with only a contact cylinder. Only a very light pressure is required to slide the cone point and the contact cylinder across the end of the edge finder. The contact cylinder is always used to pick up an edge. The body of the edge finder is held in the machine spindle by a chuck. When the edge finder is used, the spindle must be rotating and the contact cylinder is intentionally offset so that there is a distinct wobble as it rotates. The edge of the workpiece is moved toward the edge finder by moving the table until they contact each other. As the workpiece continues to be moved toward the edge finder, the wobble of the contact cylinder decreases until it ceases altogether. When this occurs the table movement is stopped. At this position the distance between the edge and the axis of the spindle is



Courtesy of The L. S. Starrett Company

Fig. 15-25. (Left) Application of cylindrical contact of edge finder to pick up an edge. (Right) Application of pointed contact to pick up a scribed line.

equal to one-half of the diameter of the contact cylinder. Since the diameter of the contact cylinder on most edge finders is .200 inch, the distance that the table must be moved to align the axis of the spindle and the edge is .100 inch.

The cone-point shape contact is used to pick up lines scribed on the workpiece as shown in Fig. 15-25. In this case the cone point must be made to rotate without wobble. This can easily be accomplished by touching the rotating cone point lightly, with an object such as a pencil. The alignment is then determined visually by lowering the cone point over the line.

Locate the Hole. Precise coordinate table movements are used to position the workpiece so that the spindle axis is located exactly where the hole is to be machined. Essentially identical table position measuring systems are used on the saddle to measure the transverse table position and on the table to measure the longitudinal table position. On the Moore jig borer the coarse measuring system consists of a steel rule, called a scale, that is graduated in .1 inch increments with every inch marked. The fine measuring system is a micrometer dial attached to the very precise lead screw which is graduated to read .001 inch and an easily read vernier scale graduated to read .0001 inch. On the Moore metric jig borer the coarse scale is graduated in 1.0 mm increments, the micrometer dial in 0.01 mm increments, and the vernier scale in 1.0 μm, which is 0.001 mm. When the measuring system is set to read zero at the starting position, the coordinate readings at the hole location will be the same as the coordinate dimensions of the hole. The movement to each hole must be made in one direction only in order to eliminate the effect of backlash.

This should always be toward the left and toward the column of the machine. Although the holes can be machined in any sequence, it is easier and more efficient on the Moore jig borer to machine the hole that is closest to the starting point first, then the next closest, etc., until all of the holes have been machined. The same sequence is recommended for machines having an optical measurement system.

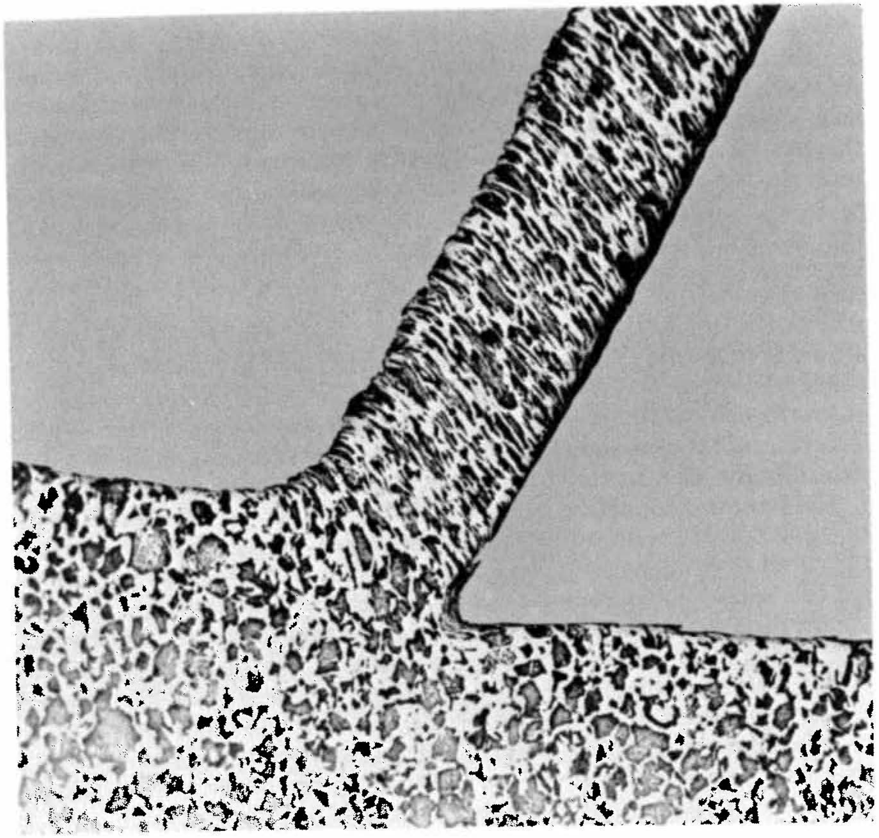
On jig borers having a table position measuring system utilizing end measuring rods, as shown in Fig. 15-16, this sequence is reversed. It is much more convenient and efficient on these machines to machine first the hole that is furthest from the starting position, then the next furthest hole, etc. The largest stack-up of end measuring rods in the troughs, in addition to a micrometer head, will occur when machining the hole furthest from the starting position. For machining the next furthest hole the length of this stack-up will decrease and will be shortest when machining the hole closest to the starting position. In this way end measuring rods are removed and replaced by shorter rods when positioning the table at each successive hole, and the table is moved progressively toward the starting position.

Drill and Bore the Hole. The sequence of operations used to produce a hole on the jig borer is based upon two basic principles: 1. A short, stiff drill, such as a center drill, that will not deflect when cutting should be used to start a hole whenever the work and the axis of the machine tool spindle are located at the exact position where the hole is wanted; 2. A single-point boring tool that is rotating about the axis of the machine tool spindle will generate or cut a surface that is concentric to the axis of rotation. Of course, a drill is required to open the hole so that the boring tool can enter and enlarge it.

The actual steps required to machine the hole are now listed in a numbered sequence.

1. Spot drill the hole with a center drill or a spotting drill. This operation is illustrated in Fig. 15-26. A conventional drill and countersink, or center drill, is recommended for this operation except when the holes to be machined are very small. In this case a special spotting drill, shown in Fig. 15-27, should be used. The spotting drills must be kept sharp because they are not intended to penetrate deeply into the workpiece. Sometimes all of the holes to be produced in a part are spot drilled before they are drilled and bored. Some jig borer operators will spot all of the holes lightly and then re-spot them to depth in order to insure against making errors.

2. Drill the hole. The initial hole should be drilled with a twist drill that will follow the countersink made by the center drill. Although larger drills are sometimes used, it is best not to start with a drill in excess of about one-half inch in diameter. If required, the hole can then be enlarged by drilling with successively larger drills.



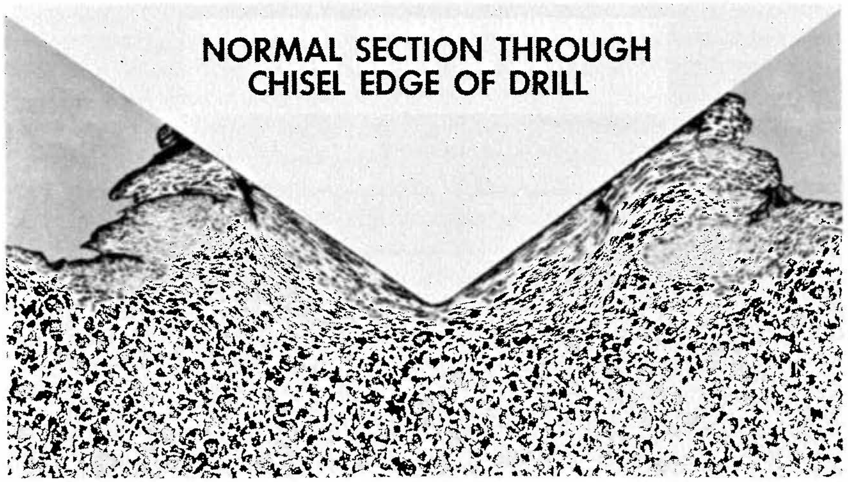
Courtesy of the National Twist Drill Company

Fig. 4-12. Photomicrograph of a metal chip formed by a cutting edge such as the lips of a twist drill.

can be readily seen in Fig. 4-14, which clearly shows the chip formed by the lip, as distinct from the chip formed by the chisel edge. In this respect, twist drills are unique among the metal-cutting tools.

Reamers

Figure 4-15 illustrates a chucking reamer and a hand reamer, and it defines their respective individual elements. Reamers are used to finish cut holes to a precise dimension and to provide a smooth surface finish on the walls of the hole. They are precision cutting tools that must always be handled with care to prevent chipping and other damage to the reamer teeth. *One cardinal principle in using a reamer is never to rotate it in the reverse direction.* Reamers should always be rotated in the forward, or cutting direction, at all times, regardless of whether they are entering or

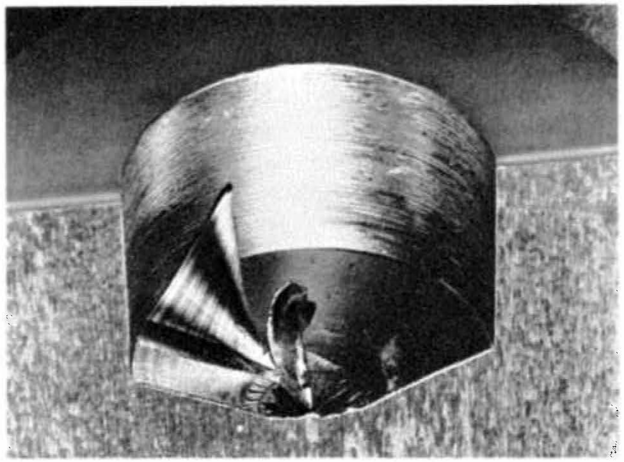


Courtesy of the National Twist Drill Company

Fig. 4-13. Photomicrograph through the axis of a drilled hole normal to the drill chisel-edge showing the complex cutting and extruding action by which the metal in this region is removed.

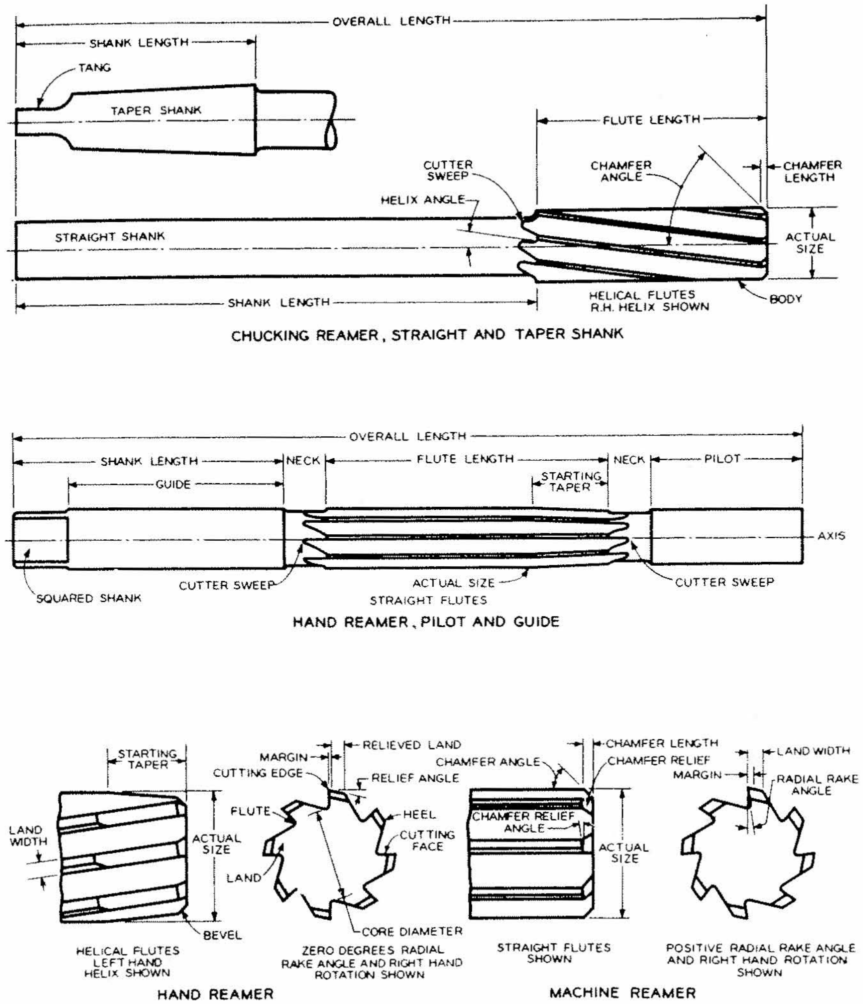
leaving the hole. Failure to do this will rapidly damage the margin and destroy the accuracy of the reamer. Reaming is a fast, efficient, and easily applied method of finishing holes.

There are two basic types of reamers: machine reamers and hand



Courtesy of the National Twist Drill Company

Fig. 4-14. Section of hole with drill removed to show separate chip formed by the lip and the chisel edge.



Courtesy of the Metal Cutting Tool Institute

Fig. 4-15. Reamer nomenclature.

reamers. Machine reamers may have a straight, or a taper, shank. They are used primarily in power-driven machine tools. Machine reamers have a chamfer ground on the ends of the lands. The chamfer angle is usually 45°, although other chamfer angles are sometimes used. The lands have a margin adjacent to the cutting edge, which is unrelieved. Relief is ground on the back of the chamfer, thereby making it into a sharp cutting edge. The chamfer edges do the cutting and remove the excess metal from the hole. The forces on the workpiece resulting from the cutting action of the reamer will cause the work to spring slightly away from the reamer, in the region of the chamfer edge. After the chamfer edge has passed a given portion of the hole, the work material will spring back and bear against

the margin of the lands of the reamer. The margin, being unrelieved, is a poor cutting edge. Primarily, it burnishes the sides of the hole to produce a good surface finish. It may also scrape a very small amount of metal out of the hole, thereby removing small irregularities. The margin also performs another important function in supporting and steadying the reamer as it passes through the hole. Sometimes, the margin is ground with a slight taper toward the chamfer edge. The length of this taper is from 1/8 to 3/16 inch and the angle, with respect to the axis of the reamer, is from 1° to 5°. This is sometimes called a lead. Actually, its function is to increase the burnishing action of the lands, thereby improving the surface finish of the hole.

Hand reamers are designed to be turned manually in reaming a hole. They are ground with a starting taper on the lands which is relieved to form a sharp cutting edge. (See Fig. 4-15.) A bevel edge is also ground on the end, however most of the cutting is done by the starting taper. Hand reamers are designed to remove less metal from a hole than machine reamers will. The margin of the hand reamer also burnishes the hole in the same manner as does the margin on the machine reamer.

The speed and feed through the hole of a hand reamer are controlled entirely by the workman. On machine reamers, the speed and feed are controlled by the machine setting. As a general rule, the cutting speed for reaming should be about one-half to two-thirds of the speed used for drilling the same material. Excessive cutting speeds will cause the reamer to chatter; consequently, the cutting speed must be kept low enough to prevent the occurrence of chatter. The feed for machine reaming is usually .0015 to .0040 inch per flute per revolution. Thus, the feed rate of a six-fluted reamer that is cutting at 100 rpm, using .004 inch per tooth feed, would be 6 × .004, or .024 inch per revolution. Expressed in different terms, the feed rate would be 100 × .024, or 2.4 inches per minute. When the feed used is too low the work may be glazed, the cutting edge will wear excessively, and, occasionally, chatter will occur. An excessively fast feed rate will reduce the accuracy of the hole and lower the quality of the surface finish.

It is very difficult to generalize on the amount of stock that should be removed by reaming as this is dependent upon the nature of the work material, the design of the reamer, the feed used, the finish required, and the depth of the hole. As a starting point for machine reaming, allow .008 to .012 inch on a 1/4-inch hole, .012 to .016 inch on a 1/2-inch hole, and .025 to .035 inch on holes up to 1 1/2-inches in diameter. For hand reaming, the stock allowance is much smaller. The allowance ranges from .001 to .010 inch.

Two other types of reamers that are sometimes used are rose reamers and taper reamers. Rose reamers are used in a machine. They are similar to machine reamers in construction; however, they are designed to remove much more metal from a hole. They do not ream a hole as accurately or with as good a surface finish as do the standard machine reamers. Taper reamers are used to produce taper holes. Larger size reamers are often made as shell reamers. The shank, or arbor, of the shell reamer is separate from the body of the reamer. The advantage of this design is that reamers of different size can be held on the same arbor and when a body is worn out the arbor does not require replacement.

Counterbores, Countersinks, and Spotfacers

Counterbores, countersinks, and spotfacers are frequently used in drill-press work in order to modify an existing hole. The counterbore, A, in Fig. 4-16, is used to enlarge the end of a hole by cutting a cylindrical surface that is concentric with the original hole. The concentricity is obtained by the pilot located on the end of the counterbore and which acts to guide the tool as it penetrates into the work. A countersink, B, in Fig. 4-16, is used to make a cone-shaped enlargement in the end of the hole. A spotfacing tool is shown at C. The purpose of the spotfacing operation

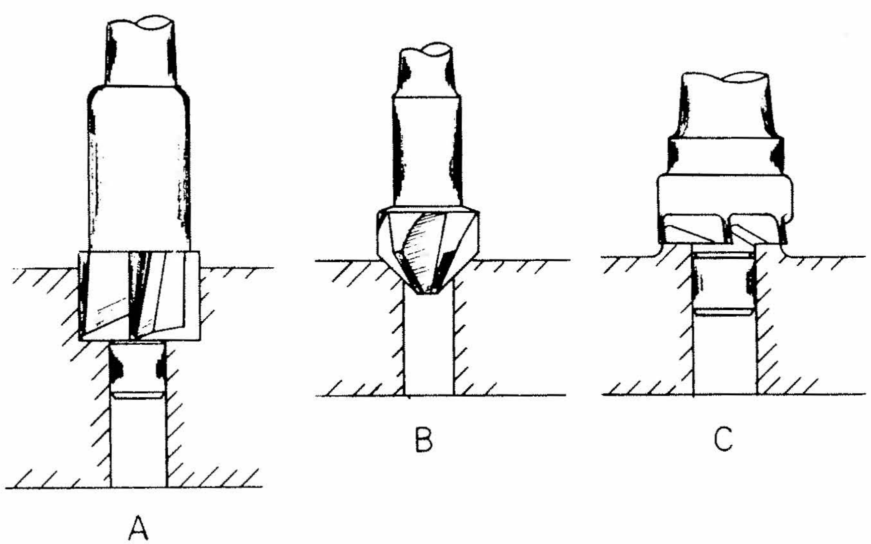
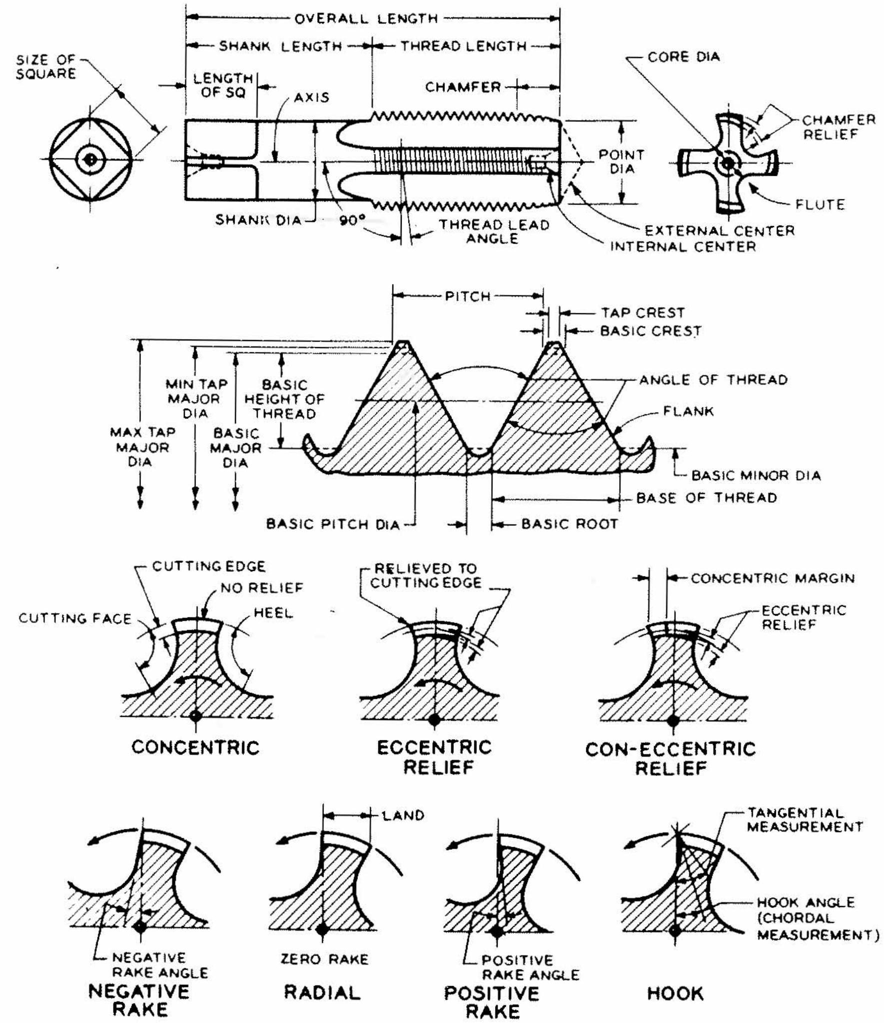


Fig. 4-16. A. Counterboring a hole. B. Countersinking a hole. C. Spot-facing a hole.

is to cut a smooth, flat surface which is perpendicular to the axis of the hole. Usually, this surface serves as a seat for the head of a cap screw or bolt; or as a seat for a nut.

Taps

The basic nomenclature of a tap is shown in Fig. 4-17. A tap is used to produce internal threads. It has thread-shaped teeth which cut a thread of similar form when it is screwed into a drilled hole. The first step in tapping an internal thread is to drill a hole with a tap drill. This is an ordinary drill that is appropriately sized so that it will leave just the right amount of metal in the hole for the tap to remove and finish



Courtesy of the Metal Cutting Tool Institute

Fig. 4-17. Basic nomenclature of a tap.

the thread forms to required size. The tap is then screwed into the workpiece by hand, with the aid of a tap wrench, or by power, as applied by a machine tool such as a lathe or a drilling machine. The feed of the tap must always be equal to the lead of the thread being cut. The lead of a thread is the distance that it advances in one revolution. A tap will generally feed itself, once it has started, as it will tend to follow that portion of the thread that already has been cut in starting.

The size of the tap drill to be used to tap the hole is dependent upon the diameter of the thread, the size of the thread tooth (or pitch of the thread), the material of which the thread is made, the length of engage-

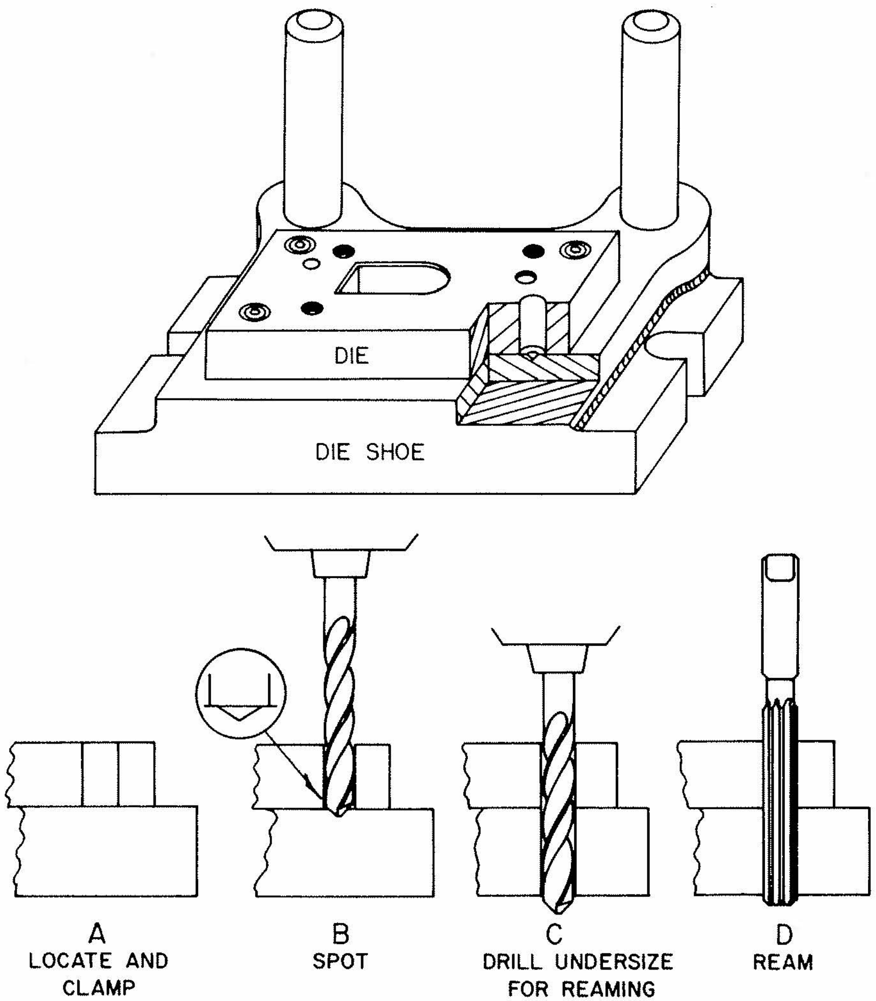


Fig. 5-17. Procedure for spotting a hole in a die shoe.

If less accuracy is satisfactory, a circle can be scribed with dividers, witness marks punched on the circle, and the hole drilled in the usual manner.

Transfer screws also have a sharp punch point on one end but their body is a screw thread designed to fit in a hole having a mating thread. They are used in a similar manner as transfer punches, except that it is usually necessary to tap one of the parts with a soft hammer instead of the transfer screw to form the punch mark.

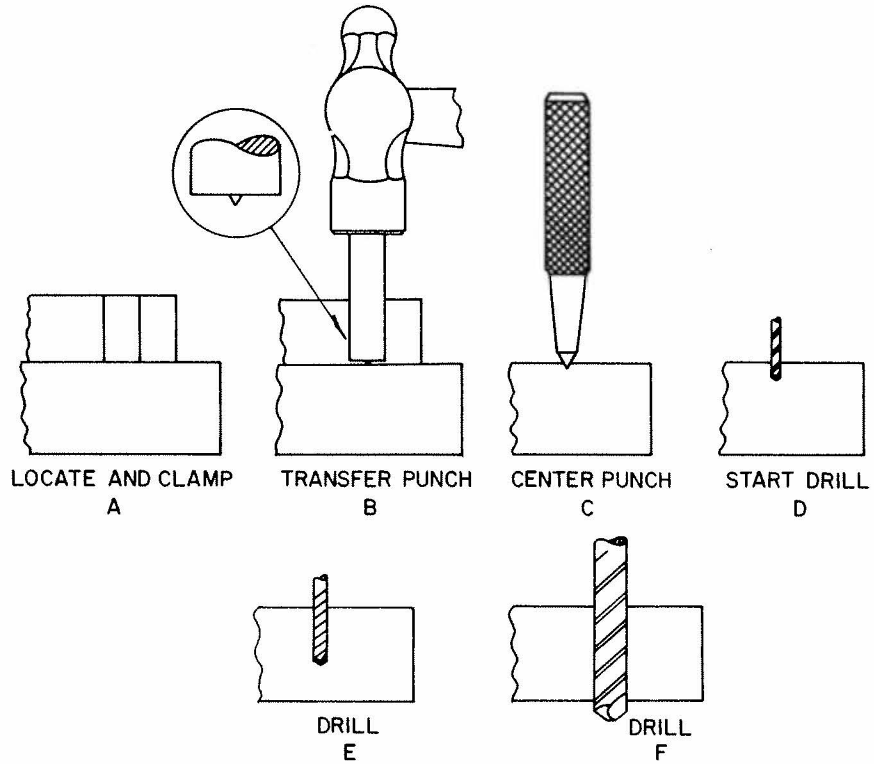


Fig. 5-18. Procedure for using a transfer punch.

Boring

In addition to the usual operations (drilling, reaming, counterboring etc.), drilling machines equipped with a power feed mechanism are sometimes used to perform boring operations. A single point tool is used in boring to enlarge an existing hole. Figure 15-3, in Chapter 15, illustrates a boring operation such as might be performed on a drilling machine, as well as on a jig boring machine. The single point boring tool is held in a boring bar that can be offset radially from its center of rotation by an offset boring head. This radial adjustment of the cutting tool enables it to be set to bore precise hole diameters.

Reaming

Reamers are used to finish drilled holes to size and to a smooth, surface finish. In order to produce accurate holes by reaming, the reamer must be sharp and correctly ground, the right amount of stock must be left in the hole for reaming, and the reamer must be accurately aligned with the axis of the drilled hole. The feed of the reamer through the hole must not be too fast (.0015 to .004 inch per land), otherwise the accuracy and the surface finish of the hole will be impaired. Sometimes a reamer will chatter. This chatter may exist only at the start of the hole as the reamer aligns itself with the drilled hole. Chatter is objectionable because it impairs the surface finish of the hole and it can damage the surface finish of the reamer. Cemented-carbide tipped reamers, in particular, are quite likely to chip if chatter occurs even at the start of the hole.

To eliminate chatter the following steps should be tried: reduce the cutting speed; vary the feed; reduce the relief angle on the reamer; use a piloted reamer; use a reamer having helical flutes; increase the rigidity of the setup; and, as a last resort, use a reamer having irregular spacing of the flutes. The use of a helical fluted reamer is much preferred to one having irregular spacing. The helix angle of helical fluted reamers should not be greater than necessary, as this will tend to increase the force required to feed the reamer through the hole. One exception to this are taper reamers; these will cut better with a very steep spiral. Often, the difficulties associated with chatter can be eliminated by exercising great care in aligning the reamer with the hole, at the start of the operation. This should never be done carelessly.

A good supply of the correct cutting fluid should be used for the reaming operation except when reaming cast iron, which should be reamed dry, with only a jet of compressed air to act as a cooling medium. Use of the correct cutting fluids often materially improves the surface finish obtained in a reamed hole. Generally, for reaming steel, a sulfurized mineral oil will work best although good results have also been obtained using soluble oil. Soluble oil and lard oil compounds can be used on aluminum. Brass may be reamed dry, although the application of a soluble oil or a lard base, oil compound is often advisable. Soluble oil, or a sulfurized mineral oil, is recommended for stainless steels. Many proprietary cutting fluids can be purchased which will do an excellent job.

Tapping

When tapping is to be done in a drilling machine there must be provision for rapidly reversing the spindle so that the tap can be backed out of the hole after the thread is cut to the required depth. Most radial drilling machines are designed with a spindle reverse mechanism that is actuated by the spindle control lever. Upright and bench drilling machines are usually not equipped with such a mechanism for reversing the rotation of the spindle thus a special attachment for this operation is required. An automatic reverse tapping attachment is shown in Fig. 5-19. This attachment has a taper shank which is inserted in the spindle of the machine. The body of the chuck and the gage *B*, which may be used to control the depth to which holes are tapped, are both prevented from rotating with the machine spindle by rod *A*. When tapping a hole the spindle is lowered by hand and the tap is fed to the required depth manually. When the lower end of the stop rod *B*, comes into contact with the face of the

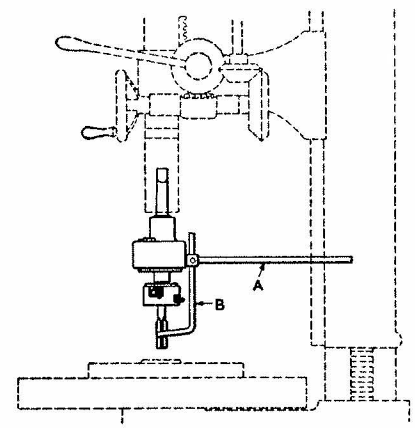


Fig. 5-19. Automatic tapping attachment applied to upright drilling machine.

work, the forward rotation of the tap is stopped. When the drill press spindle is raised, gears within the chuck body are engaged that cause the tap to back out of the hole rapidly. The tapping attachment may be equipped to have an adjustable friction drive that stops the rotation of the tap whenever the tapping torque exceeds a preset limit, thus safeguarding the taps against breakage.

There are many more variables that must be considered when selecting the cutting speed for tapping than for other machining operations; thus, it is not surprising that the best and most efficient cutting speeds for tapping cannot be tabulated with any assurance that the tap will then be operating at its highest efficiency. Among the factors that must be considered are:

1. Material to be tapped
2. Heat treatment and hardness of material to be tapped
3. Length of hole
4. Length of chamfer on top
5. Pitch of thread
6. Percentage of full thread to be cut
7. Cutting fluid to be used, and
8. Design of the drilling machine with respect to the sensitivity of the controls.

For example, speeds must be lowered as the length of the hole is increased because long holes accumulate chips, increase friction, and interfere with the flow of cutting fluids. Taps having long chamfers can be run faster than taps with short chamfers in short holes; however, in long holes the taps with the short chamfers can be run faster. Bottoming taps must be run slower than plug taps. A slower speed is required to tap full-depth thread than a 75-per cent-depth thread. When the tap diameter exceeds approximately ½ inch, coarse thread taps should be run more shell reamer is separate from the body of the reamer. The advantage of this design is that reamers of different size can be held on the same arbor and when a body is worn out the arbor does not require replacement.

Counterbores, Countersinks, and Spotfacers

Counterbores, countersinks, and spotfacers are frequently used in drill-press work in order to modify an existing hole. The counterbore, A, in Fig. 4-16, is used to enlarge the end of a hole by cutting a cylindrical surface that is concentric with the original hole. The concentricity is obtained by the pilot located on the end of the counterbore and which acts to guide the tool as it penetrates into the work. A countersink, B, in Fig. 4-16, is used to make a cone-shaped enlargement in the end of the hole. A spotfacing tool is shown at C. The purpose of the spotfacing operation

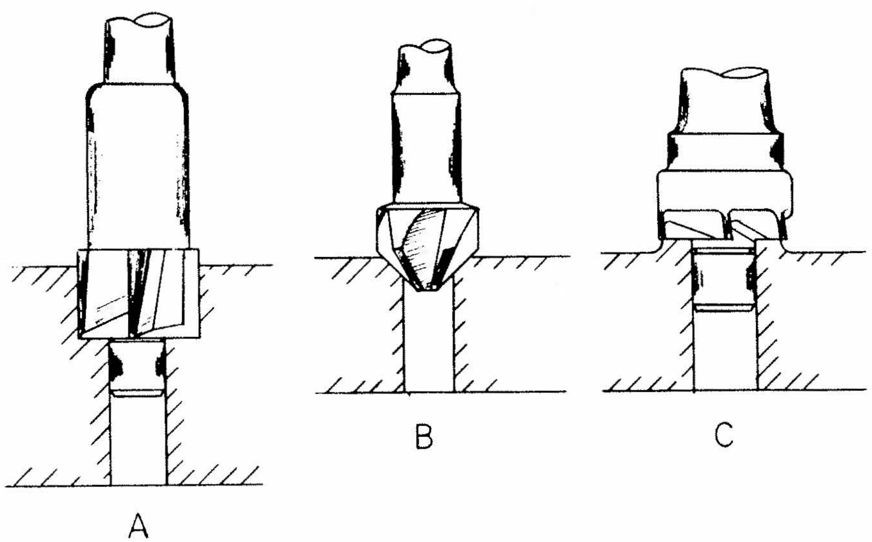
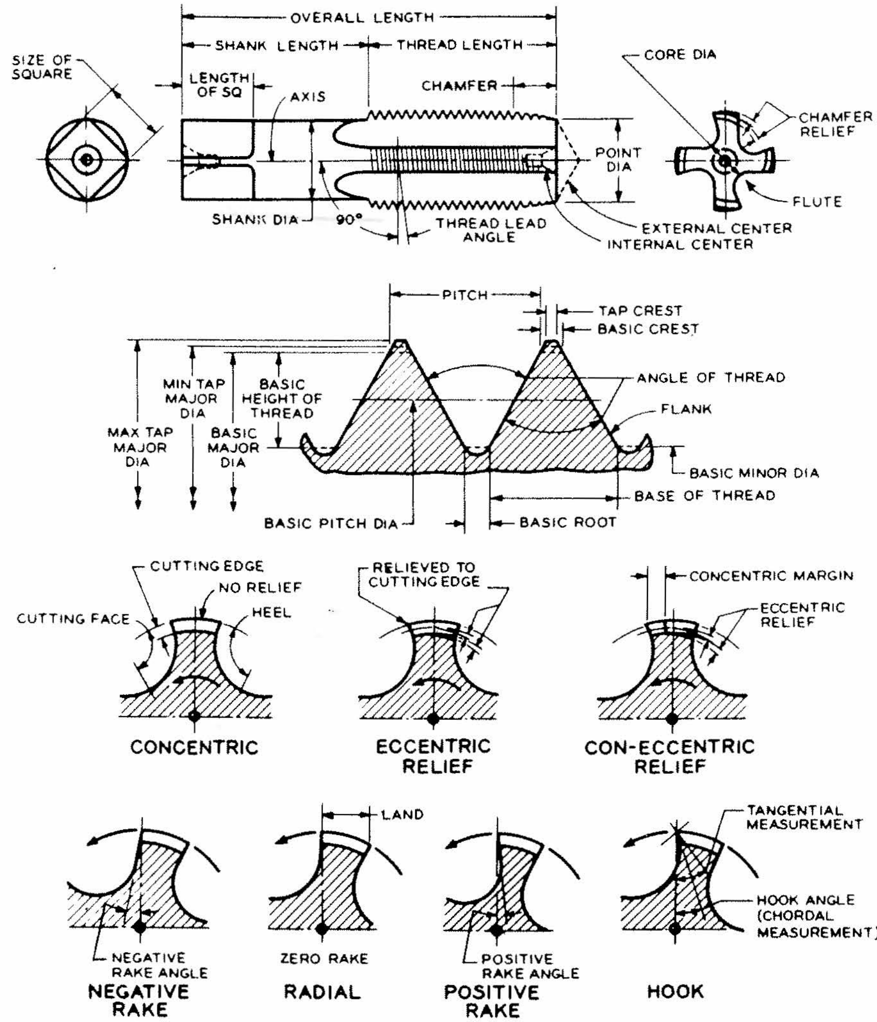


Fig. 4-16. A. Counterboring a hole. B. Countersinking a hole. C. Spot-facing a hole.

is to cut a smooth, flat surface which is perpendicular to the axis of the hole. Usually, this surface serves as a seat for the head of a cap screw or bolt; or as a seat for a nut.

Taps

The basic nomenclature of a tap is shown in Fig. 4-17. A tap is used to produce internal threads. It has thread-shaped teeth which cut a thread of similar form when it is screwed into a drilled hole. The first step in tapping an internal thread is to drill a hole with a tap drill. This is an ordinary drill that is appropriately sized so that it will leave just the right amount of metal in the hole for the tap to remove and finish



Courtesy of the Metal Cutting Tool Institute

Fig. 4-17. Basic nomenclature of a tap.

the thread forms to required size. The tap is then screwed into the workpiece by hand, with the aid of a tap wrench, or by power, as applied by a machine tool such as a lathe or a drilling machine. The feed of the tap must always be equal to the lead of the thread being cut. The lead of a thread is the distance that it advances in one revolution. A tap will generally feed itself, once it has started, as it will tend to follow that portion of the thread that already has been cut in starting.

The size of the tap drill to be used to tap the hole is dependent upon the diameter of the thread, the size of the thread tooth (or pitch of the thread), the material of which the thread is made, the length of engagement of the thread, and the available standard drill sizes. For ordinary manufacturing, the internal thread in a tapped hole need not be more than 75 or 80 per cent of the theoretical, standard thread depth. For some classes of work, not more than 50 per cent of the thread depth is required. The diameter of the tap drill should not be smaller than is necessary to give the required strength of thread, as every decrease in the diameter of the tap drill increases the power required for tapping, and, incidentally, the percentage of broken taps. Recommended tap drill sizes and hole sizes for tapping Unified Screw Threads can be found in various standard machine shop handbooks such as Machinery's Handbook.

Theoretical tap drill sizes for inch and metric threads can be calculated by the following formulas, in which the "Percent Full Thread" is expressed as a decimal; e.g., 75% is written .75 in the formula. The drill size nearest to the calculated hole size is used.

For American Unified Thread Form:

$$\text{Hole size} = \text{Basic major dia.} - \frac{1.08253 \times \text{percent full thread}}{\text{Number of threads per inch}} \tag{4-1}$$

For American National Thread form:

$$\text{Hole size} = \text{Basic major dia.} - \frac{1.299 \times \text{percent full thread}}{\text{Number of threads per inch}} \tag{4-2}$$

For ISO Metric Threads (all dimensions in millimeters):

$$\text{Hole size} = \text{Basic major dia.} - (1.08253 \times \text{pitch} \times \text{percent full thread}) \tag{4-3}$$

Example 4-1:

Find the tap drill diameter for a ½-13 Unified thread and for an M12-1.75 thread. The M12-1.75 thread designation means that it is metric (M), it has a diameter equal to 12 mm, and the pitch of the thread is 1.75 mm. In each case the tapped hole is to have a 60 percent full thread.

For the ½-13 Unified thread:

$$\text{Hole size} = .500 - \frac{1.08253 \times .60}{13} = .450 \text{ in.}$$

For the M12-1.75 metric thread:

$$\text{Hole size} = 12 - (1.08253 \times 1.75 \times .60) = 10.86 \text{ mm}$$

The nearest drill sizes are 29/64 in. and 11.0 mm, respectively.

Hand taps have a cylindrical shank and a driving square on the end to which a tap wrench can be attached. These taps are used for most machine tapping applications, as well as for hand tapping. They can be obtained as ground taps, which are finished by precision grinding, or as cut taps, which are finished by a cutting operation.

Hand taps are classified into the following types: taper, plug, and bottoming. Each type is distinguished by the amount of chamfer on the end. The angle formed between the chamfer (see Fig. 4-17) and the axis of the tap is called the chamfer angle. It determines the number of threads having their height reduced on the end of the tap. The chamfered threads are relieved behind the face and do all of the cutting to form the thread on the part. The number of flutes and the length of the chamfer determine the chip load, or the amount of material removed by each tooth.

Taper taps have a chamfer of 4½° which reduces the height of approximately 8 to 10 threads. Plug taps have a chamfer angle of 10½° with 3 to 5 threads reduced in height. Bottoming taps have a chamfer angle of 25° to 29°, which reduces approximately 1½ threads. The most useful tap for general tapping applications is the plug tap. Plug taps can be used to tap through holes and also blind holes where a limited number of imperfect teeth are allowed. Bottoming taps are used to finish the bottom of blind holes where only a few imperfect teeth are permitted. Taper taps are used primarily for hand tapping thin workpieces, for tapping through holes in some abrasive materials, and for tapping the larger thread sizes and thread forms, where the chip thickness per tooth must be kept low. When a particularly hard, tough material is encountered, it is sometimes helpful to use a taper and a plug tap alternately. The taper tap starts easily but may stick after penetrating deeper; it is then removed and a plug tap is inserted to relieve the long cut made by the taper tap. For some very difficult jobs, a series of succeedingly larger taps are used to tap the hole, such taps being made specially for each job. On the large majority of jobs, however, the plug tap can be used alone.

On a limited number of softer, ductile materials, threaded holes can be produced on a machine with a forming tap. This is usually associated with high volume production operations. Forming taps have no flutes of the conventional type. Instead of being round, the body of the tap is radially relieved to form a lobed construction, the number of lobes depending on the tap diameter. The nose end of the tap is tapered to allow it to enter the hole. Forming taps produce threads by plastic flow of the metal near the hole walls instead of by cutting. A chip is not formed. The thread form produced is not perfect, but usually there is no reduction in the strength of the thread.

A cutting fluid should always be used with all materials when tapping except on cast iron and plastics. When hand tapping a hole, the tap should be advanced approximately one turn and then rotated in the reverse direction until the chip is broken. By repeating this sequence until the hole is tapped, a smooth thread will be produced with minimum effort. Machine-driven taps are generally screwed into the hole while rotating in the forward direction and are reversed to remove the tap when the job is finished. On very difficult jobs the tap is sometimes reversed when cutting into the hole in a manner similar to hand tapping.

Spade Drills

Spade drills, Fig. 4-18, are frequently used to drill large diameter holes. They are made in sizes ranging from one to five inches. A spade drill consists of a holder made from heat-treated steel and a high-speed steel, or cemented-carbide, tipped blade. The blade is seated in a slot in the end of the holder which is clamped together by a screw to secure the blade in place. Each holder can hold a range of different blade sizes. The shank of the holder may be straight or have a standard Morse taper. Some spade drill holders have coolant holes drilled lengthwise which supply a stream of coolant close to the cutting edges. Notches or chip breakers are ground on the cutting edges to reduce the width of the chip. These notches are unevenly spaced so that they do not occur at the same radial distances on the two cutting edges. Instead of a single chip, the notches cause several smaller chips to form on each cutting edge that are easier to break. A chip curling groove (chip curler) is ground on the face of each cutting edge which tends to curl and break the chips for easy disposal.

Several different types of blades are shown in Fig. 4-18 to show the versatility of this method of drilling. Besides drilling holes through solid metal, stepped holes can be drilled, and both counterboring and spot-

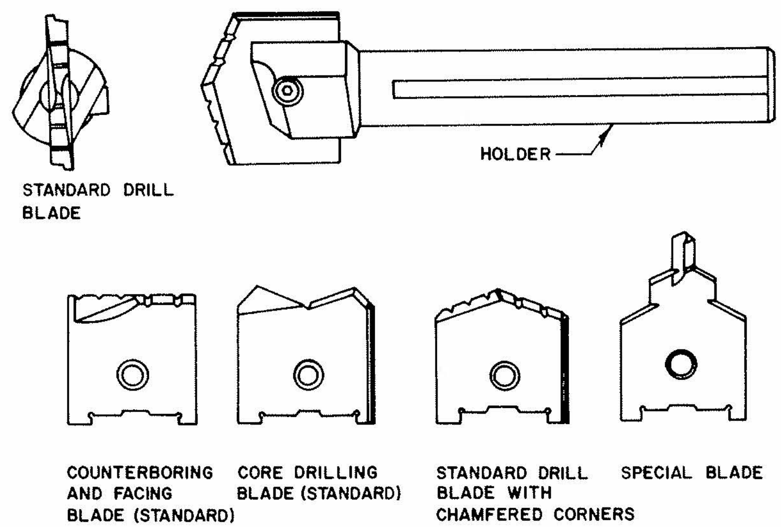


Fig. 4-18. Spade drill, including examples of standard and special blades.

facing can be done with spade drills. An important application of these drills is core drilling; i.e., enlarging an existing hole which may be a rough cored hole of a casting. The core drilling blade used for this operation has spurs ground on the end which bite into the metal and guide the drill, thereby drilling a hole that closely follows the path along which the drill is fed. Without the spurs, a core drill will be influenced by any eccentricity of the existing hole and may not drill a true hole. Core drilling operations may be performed on drilling machines having a power feed, on turret lathes, vertical turret lathes, on both horizontal and vertical boring mills, and on numerically controlled machines.

A power feed should be used when drilling with a spade drill. Only by applying extreme care and skill can spade drilling be accomplished with manual feed. The feed rate is a most important factor in successfully applying a spade drill and the constant feed rate supplied by a power feeding mechanism should be used whenever possible; for deep holes it should always be used. The feed rate should be adjusted so that the chips break into forms resembling a small "c" or number "9."

Spade drills are self starting if they are held in short stubby spade drill holders. Longer spade drills are started by first drilling a start hole or "cone" with a spade drill held in a stub holder. If the start drill is the same diameter as the second spade drill, the start drill should drill a hole that is to the full diameter of the spade drill. If the start drill is smaller than the second drill it should not drill to its full diameter, but should leave only a conical surface against which the second drill can start. Never use a center drill or drill a pilot hole with a twist drill to start a spade drill. When the spade drill contacts the sharp edge of the hole formed by these drills, it will tend to "hop around" which can cause the cutting edges of the spade drill to chip.

The size of the hole that can be drilled with a spade drill is limited only by the power and rigidity of the machine. When insufficient power is available to drill a large hole, the hole may be drilled in two passes using a smaller spade drill to drill the first hole. The power, torque, and thrust force required to feed the spade drill may be estimated by the procedures provided further on in this chapter.

Gun Drills

Gun drills are customarily used to drill deep holes where the length of the hole is five or six times its diameter. These are special drills and are usually used on special deep-hole drilling machines. They have a single flute and a single cutting edge which may be high-speed steel or a brazed-on cemented carbide tip. The cutting edge may be designed to cut to the center of the drill. Some gun drills have a cutting edge that does not cut to the center of the drill, thereby leaving a core of metal which can be easily removed if the hole is a through hole. This method of pro-

See that there is sufficient extra material for finishing on large castings to permit them to be "cleaned up," even though they warp. In such castings, make sure that the metal thickness will be sufficient after finishing, even though the castings do warp.

Make sure that sufficient sections are shown so that the pattern makers and molders will not be compelled to make assumptions about the form of any part of the casting. These details are particularly important when a number of sections of the casting are similar in form, while others differ slightly.

Checking Machined Parts: Study the sequences of operations in machining and see that all finish marks are indicated.

See that the finish marks are placed on the lines to which dimensions are given.

See that methods of machining are indicated where necessary.

Give all drill, reamer, tap, and rose bit sizes.

See that jig and gage numbers are indicated at the proper places.

See that all necessary bosses, lugs, and openings are provided for lifting, handling, clamping, and machining the piece.

See that adequate wrench room is provided for all nuts and bolt heads.

Avoid special tools, such as taps, drills, reamers, etc., unless such tools are specifically authorized.

Where parts are right- and left-hand, be sure that the hand is correctly designated. When possible, mark parts as symmetrical, so as to avoid having them right- and left-hand, but do not sacrifice correct design or satisfactory operation on this account.

When heat-treatment is required, the heat-treatment should be specified.

Check the title, size of machine, the scale, and the drawing number on both the drawing and the drawing record card.

ALLOWANCES AND TOLERANCES FOR FITS

Limits and Fits.—Fits between cylindrical parts, i.e., cylindrical fits, govern the proper assembly and performance of many mechanisms. Clearance fits permit relative freedom of motion between a shaft and a hole—axially, radially, or both. Interference fits secure a certain amount of tightness between parts, whether these are meant to remain permanently assembled or to be taken apart from time to time. Or again, two parts may be required to fit together snugly—without apparent tightness or looseness. The designer's problem is to specify these different types of fits in such a way that the shop can produce them. Establishing the specifications requires the adoption of two manufacturing limits for the hole and two for the shaft, and, hence, the adoption of a manufacturing tolerance on each part.

In selecting and specifying limits and fits for various applications, it is essential in the interests of interchangeable manufacturing that 1) standard definitions of terms relating to limits and fits be used; 2) preferred basic sizes be selected wherever possible to reduce material and tooling costs; 3) limits be based upon a series of preferred tolerances and allowances; and 4) a uniform system of applying tolerances (preferably unilateral) be used. These principles have been incorporated in both the American and British standards for limits and fits. Information about these standards is given beginning on page 627.

Basic Dimensions.—The basic size of a screw thread or machine part is the theoretical or nominal standard size from which variations are made. For example, a shaft may have a *basic* diameter of 2 inches, but a maximum variation of minus 0.010 inch may be permitted. The minimum hole should be of basic size wherever the use of standard tools represents the greatest economy. The maximum shaft should be of basic size wherever the use of standard purchased material, without further machining, represents the greatest economy, even though special tools are required to machine the mating part.

Tolerances.—Tolerance is the amount of variation permitted on dimensions or surfaces of machine parts. The tolerance is equal to the difference between the maximum and minimum limits of any specified dimension. For example, if the maximum limit for the diameter of a shaft is 2.000 inches and its minimum limit 1.990 inches, the tolerance for this diameter is 0.010 inch. The extent of these tolerances is established by determining the maximum and minimum clearances required on operating surfaces. As applied to the fitting of machine parts, the word tolerance means the amount that duplicate parts are allowed to vary in size in connection with manufacturing operations, owing to unavoidable imperfections of workmanship. Tolerance may also be defined as the amount that duplicate parts are permitted to vary in size to secure sufficient accuracy without unnecessary refinement. The terms "tolerance" and "allowance" are often used interchangeably, but, according to common usage, *allowance* is a difference in dimensions prescribed to secure various classes of fits between different parts.

Unilateral and Bilateral Tolerances.—The term "unilateral tolerance" means that the total tolerance, as related to a basic dimension, is in *one* direction only. For example, if the basic dimension were 1 inch and the tolerance were expressed as 1.000 – 0.002, or as 1.000 + 0.002, these would be unilateral tolerances because the total tolerance in each is in one direction. On the contrary, if the tolerance were divided, so as to be partly plus and partly minus, it would be classed as "bilateral."

Thus,

$$1.000\begin{smallmatrix}+0.001\\-0.001\end{smallmatrix}$$

is an example of bilateral tolerance, because the total tolerance of 0.002 is given in two directions—plus and minus.

When unilateral tolerances are used, one of the three following methods should be used to express them:

1) Specify, limiting dimensions only as
Diameter of hole: 2.250, 2.252
Diameter of shaft: 2.249, 2.247
2) One limiting size may be specified with its tolerances as
Diameter of hole: 2.250 + 0.002, −0.000
Diameter of shaft: 2.249 + 0.000, −0.002
3) The nominal size may be specified for both parts, with a notation showing both allowance and tolerance, as
Diameter of hole: 2¼ + 0.002, −0.000
Diameter of shaft: 2¼ − 0.001, −0.003

Bilateral tolerances should be specified as such, usually with plus and minus tolerances of equal amount. An example of the expression of bilateral tolerances is

$$2 \pm 0.001 \quad \text{or} \quad 2\begin{array}{l}+0.001\\-0.001\end{array}$$

Application of Tolerances.—According to common practice, tolerances are applied in such a way as to show the permissible amount of dimensional variation in the direction that is less dangerous. When a variation in either direction is equally dangerous, a bilateral tolerance should be given. When a variation in one direction is more dangerous than a variation in another, a unilateral tolerance should be given in the less dangerous direction.

For nonmating surfaces, or atmospheric fits, the tolerances may be bilateral, or unilateral, depending entirely upon the nature of the variations that develop in manufacture. On mating surfaces, with few exceptions, the tolerances should be unilateral.

Where tolerances are required on the distances between holes, usually they should be bilateral, as variation in either direction is normally equally dangerous. The variation in the distance between shafts carrying gears, however, should always be unilateral and plus; otherwise, the gears might run too tight. A slight increase in the backlash between gears is seldom of much importance.

One exception to the use of unilateral tolerances on mating surfaces occurs when tapers are involved; either bilateral or unilateral tolerances may then prove advisable, depending upon conditions. These tolerances should be determined in the same manner as the tolerances on the distances between holes. When a variation either in or out of the position of the mating taper surfaces is equally dangerous, the tolerances should be bilateral. When a variation in one direction is of less danger than a variation in the opposite direction, the tolerance should be unilateral and in the less dangerous direction.

Locating Tolerance Dimensions.—Only one dimension in the same straight line can be controlled within fixed limits. That dimension is the distance between the cutting surface of the tool and the locating or registering surface of the part being machined. Therefore, it is incorrect to locate any point or surface with tolerances from more than one point in the same straight line.

Every part of a mechanism must be located in each plane. Every operating part must be located with proper operating allowances. After such requirements of location are met, all other surfaces should have liberal clearances. Dimensions should be given between those points or surfaces that it is essential to hold in a specific relation to each other. This restriction applies particularly to those surfaces in each plane that control the location of other component parts. Many dimensions are relatively unimportant in this respect. It is good practice to establish a common locating point in each plane and give, as far as possible, all such dimensions from these common locating points. The locating points on the drawing, the locatingor registering points used for machining the surfaces and the locating points for measuring should all be identical.

The initial dimensions placed on component drawings should be the exact dimensions that would be used if it were possible to work without tolerances. Tolerances should be

given in that direction in which variations will cause the least harm or danger. When a variation in either direction is equally dangerous, the tolerances should be of equal amount in both directions, or bilateral. The initial clearance, or allowance, between operating parts should be as small as the operation of the mechanism will permit. The maximum clearance should be as great as the proper functioning of the mechanism will permit.

Direction of Tolerances on Gages.—The extreme sizes for all plain limit gages shall not exceed the extreme limits of the part to be gaged. All variations in the gages, whatever their cause or purpose, shall bring these gages within these extreme limits.

The data for gage tolerances on page 656 cover gages to inspect workpieces held to tolerances in the American National Standard ANSI B4.4M-1981.

Allowance for Forced Fits.—The allowance per inch of diameter usually ranges from 0.001 inch to 0.0025 inch, 0.0015 being a fair average. Ordinarily the allowance per inch decreases as the diameter increases; thus the total allowance for a diameter of 2 inches might be 0.004 inch, whereas for a diameter of 8 inches the total allowance might not be over 0.009 or 0.010 inch. The parts to be assembled by forced fits are usually made cylindrical, although sometimes they are slightly tapered. The advantages of the taper form are that the possibility of abrasion of the fitted surfaces is reduced; that less pressure is required in assembling; and that the parts are more readily separated when renewal is required. On the other hand, the taper fit is less reliable, because if it loosens, the entire fit is free with but little axial movement. Some lubricant, such as white lead and lard oil mixed to the consistency of paint, should be applied to the pin and bore before assembling, to reduce the tendency toward abrasion.

Pressure for Forced Fits.—The pressure required for assembling cylindrical parts depends not only upon the allowance for the fit, but also upon the area of the fitted surfaces, the pressure increasing in proportion to the distance that the inner member is forced in. The approximate ultimate pressure in tons can be determined by the use of the following formula in conjunction with the accompanying table of "Pressure Factors." Assuming that A = area of surface in contact in "fit"; a = total allowance in inches; P = ultimate pressure required, in tons; F = pressure factor based upon assumption that the diameter of the hub is twice the diameter of the bore, that the shaft is of machine steel, and that the hub is of cast iron:

$$P = \frac{A \times a \times F}{2}$$

Pressure Factors

Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor
1	500	3½	132	6	75	9	48.7	14	30.5
1¼	395	3¾	123	6¼	72	9½	46.0	14½	29.4
1½	325	4	115	6½	69	10	43.5	15	28.3
1¾	276	4¼	108	6¾	66	10½	41.3	15½	27.4
2	240	4½	101	7	64	11	39.3	16	26.5
2¼	212	4¾	96	7¼	61	11½	37.5	16½	25.6
2½	189	5	91	7½	59	12	35.9	17	24.8
2¾	171	5¼	86	7¾	57	12½	34.4	17½	24.1
3	156	5½	82	8	55	13	33.0	18	23.4
3¼	143	5¾	78	8½	52	13½	31.7	…	…

Allowance for Given Pressure.—By transposing the preceding formula, the approximate allowance for a required ultimate tonnage can be determined. Thus, $a = \frac{2P}{AF}$. The average ultimate pressure in tons commonly used ranges from 7 to 10 times the diameter in inches.

Expansion Fits.—In assembling certain classes of work requiring a very tight fit, the inner member is contracted by sub-zero cooling to permit insertion into the outer member and a tight fit is obtained as the temperature rises and the inner part expands. To obtain the sub-zero temperature, solid carbon dioxide or "dry ice" has been used but its temperature of about 109 degrees F. below zero will not contract some parts sufficiently to permit insertion in holes or recesses. Greater contraction may be obtained by using high purity liquid nitrogen which has a temperature of about 320 degrees F. below zero. During a temperature reduction from 75 degrees F. to −321 degrees F., the shrinkage per inch of diameter varies from about 0.002 to 0.003 inch for steel; 0.0042 inch for aluminum alloys; 0.0046 inch for magnesium alloys; 0.0033 inch for copper alloys; 0.0023 inch for monel metal; and 0.0017 inch for cast iron (not alloyed). The cooling equipment may vary from an insulated bucket to a special automatic unit, depending upon the kind and quantity of work. One type of unit is so arranged that parts are precooled by vapors from the liquid nitrogen before immersion. With another type, cooling is entirely by the vapor method.

Shrinkage Fits.—General practice seems to favor a smaller allowance for shrinkage fits than for forced fits, although in many shops the allowances are practically the same for each, and for some classes of work, shrinkage allowances exceed those for forced fits. The shrinkage allowance also varies to a great extent with the form and construction of the part that has to be shrunk into place. The thickness or amount of metal around the hole is the most important factor. The way in which the metal is distributed also has an influence on the results. Shrinkage allowances for locomotive driving wheel tires adopted by the American Railway Master Mechanics Association are as follows:

Center diameter, inches	38	44	50	56	62	66
Allowances, inches	0.040	0.047	0.053	0.060	0.066	0.070

Whether parts are to be assembled by forced or shrinkage fits depends upon conditions. For example, to press a tire over its wheel center, without heating, would ordinarily be a rather awkward and difficult job. On the other hand, pins, etc., are easily and quickly forced into place with a hydraulic press and there is the additional advantage of knowing the exact pressure required in assembling, whereas there is more or less uncertainty connected with a shrinkage fit, unless the stresses are calculated. Tests to determine the difference in the quality of shrinkage and forced fits showed that the resistance of a shrinkage fit to slippage for an axial pull was 3.66 times greater than that of a forced fit, and in rotation or torsion, 3.2 times greater. In each comparative test, the dimensions and allowances were the same.

Allowances for Shrinkage Fits.—The most important point to consider when calculating shrinkage fits is the stress in the hub at the bore, which depends chiefly upon the shrinkage allowance. If the allowance is excessive, the elastic limit of the material will be exceeded and permanent set will occur, or, in extreme conditions, the ultimate strength of the metal will be exceeded and the hub will burst. The intensity of the grip of the fit and the resistance to slippage depends mainly upon the thickness of the hub; the greater the thickness, the stronger the grip, and *vice versa.* Assuming the modulus of elasticity for steel to be 30,000,000, and for cast iron, 15,000,000, the shrinkage allowance per inch of nominal diameter can be determined by the following formula, in which A = allowance per inch of diameter; T = true tangential tensile stress at inner surface of outer member; C = factor taken from one of the accompanying tables, *Factors for Calculating Shrinkage Fit Allowances.*

For a cast-iron hub and steel shaft:

$$A = \frac{T(2 + C)}{30{,}000{,}000} \qquad (1)$$

When both hub and shaft are of steel:

$$A = \frac{T(1 + C)}{30{,}000{,}000} \qquad (2)$$

If the shaft is solid, the factor C is taken from Table 1; if it is hollow and the hub is of steel, factor C is taken from Table 2; if it is hollow and the hub is of cast iron, the factor is taken from Table 3.

Table 1. Factors for Calculating Shrinkage Fit Allowances

Ratio of Diameters $\frac{D_2}{D_1}$	Steel Hub	Cast-iron Hub	Ratio of Diameters $\frac{D_2}{D_1}$	Steel Hub	Cast-iron Hub
1.5	0.227	0.234	2.8	0.410	0.432
1.6	0.255	0.263	3.0	0.421	0.444
1.8	0.299	0.311	3.2	0.430	0.455
2.0	0.333	0.348	3.4	0.438	0.463
2.2	0.359	0.377	3.6	0.444	0.471
2.4	0.380	0.399	3.8	0.450	0.477
2.6	0.397	0.417	4.0	0.455	0.482

Values of factor C for solid steel shafts of nominal diameter D_1, and hubs of steel or cast iron of nominal external and internal diameters D_2 and D_1, respectively.

Example 1: A steel crank web 15 inches outside diameter is to be shrunk on a 10-inch solid steel shaft. Required the allowance per inch of shaft diameter to produce a maximum tensile stress in the crank of 25,000 pounds per square inch, assuming the stresses in the crank to be equivalent to those in a ring of the diameter given.

The ratio of the external to the internal diameters equals 15 ÷ 10 = 1.5; T = 25,000 pounds; from Table 1, C = 0.227. Substituting in Formula (2):

$$A = \frac{25{,}000 \times (1 + 0.227)}{30{,}000{,}000} = 0.001 \text{ inch}$$

Example 2: Find the allowance per inch of diameter for a 10-inch shaft having a 5-inch axial through hole, other conditions being the same as in Example 1.

The ratio of external to internal diameters of the hub equals 15 ÷ 10 = 1.5, as before, and the ratio of external to internal diameters of the shaft equals 10 ÷ 5 = 2. From Table 2, we find that factor C = 0.455; T = 25,000 pounds. Substituting these values in Formula (2):

$$A = \frac{25{,}000(1 + 0.455)}{30{,}000{,}000} = 0.0012 \text{ inch}$$

The allowance is increased, as compared with Example 1, because the hollow shaft is more compressible.

Table 2. Factors for Calculating Shrinkage Fit Allowances

$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C
1.5	2.0	0.468	2.4	2.0	0.798	3.4	2.0	0.926
	2.5	0.368		2.5	0.628		2.5	0.728
	3.0	0.322		3.0	0.549		3.0	0.637
	3.5	0.296		3.5	0.506		3.5	0.587
1.6	2.0	0.527	2.6	2.0	0.834	3.6	2.0	0.941
	2.5	0.414		2.5	0.656		2.5	0.740
	3.0	0.362		3.0	0.574		3.0	0.647
	3.5	0.333		3.5	0.528		3.5	0.596
1.8	2.0	0.621	2.8	2.0	0.864	3.8	2.0	0.953
	2.5	0.488		2.5	0.679		2.5	0.749
	3.0	0.427		3.0	0.594		3.0	0.656
	3.5	0.393		3.5	0.547		3.5	0.603
2.0	2.0	0.696	3.0	2.0	0.888	4.0	2.0	0.964
	2.5	0.547		2.5	0.698		2.5	0.758
	3.0	0.479		3.0	0.611		3.0	0.663
	3.5	0.441		3.5	0.562		3.5	0.610
2.2	2.0	0.753	3.2	2.0	0.909	…	…	…
	2.5	0.592		2.5	0.715		…	…
	3.0	0.518		3.0	0.625		…	…
	3.5	0.477		3.5	0.576		…	…

Values of factor C for hollow steel shafts and cast-iron hubs.
Notation as in Table 1.

Table 3. Factors for Calculating Shrinkage Fit Allowances

$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C
1.5	2.0	0.455	2.4	2.0	0.760	3.4	2.0	0.876
	2.5	0.357		2.5	0.597		2.5	0.689
	3.0	0.313		3.0	0.523		3.0	0.602
	3.5	0.288		3.5	0.481		3.5	0.555
1.6	2.0	0.509	2.6	2.0	0.793	3.6	2.0	0.888
	2.5	0.400		2.5	0.624		2.5	0.698
	3.0	0.350		3.0	0.546		3.0	0.611
	3.5	0.322		3.5	0.502		3.5	0.562
1.8	2.0	0.599	2.8	2.0	0.820	3.8	2.0	0.900
	2.5	0.471		2.5	0.645		2.5	0.707
	3.0	0.412		3.0	0.564		3.0	0.619
	3.5	0.379		3.5	0.519		3.5	0.570
2.0	2.0	0.667	3.0	2.0	0.842	4.0	2.0	0.909
	2.5	0.524		2.5	0.662		2.5	0.715
	3.0	0.459		3.0	0.580		3.0	0.625
	3.5	0.422		3.5	0.533		3.5	0.576
2.2	2.0	0.718	3.2	2.0	0.860	…	…	…
	2.5	0.565		2.5	0.676		…	…
	3.0	0.494		3.0	0.591		…	…
	3.5	0.455		3.5	0.544		…	…

Values of factor C for hollow steel shafts of external and internal diameters D_1 and D_0, respectively, and steel hubs of nominal external diameter D_2.

Example 3: If the crank web in Example 1 is of cast iron and 4000 pounds per square inch is the maximum tensile stress in the hub, what is the allowance per inch of diameter?

$$\frac{D_2}{D_1} = 1.5 \qquad T = 4000$$

In Table 1, we find that $C = 0.234$. Substituting in Formula (1), for cast-iron hubs, $A = 0.0003$ inch, which, owing to the lower tensile strength of cast iron, is endout one-third the shrinkage allowance in Example 1, although the stress is two-thirds of the elastic limit.

Temperatures for Shrinkage Fits.—The temperature to which the outer member in a shrinkage fit should be heated for clearance in assembling the parts depends on the total expansion required and on the coefficient α of linear expansion of the metal (i.e., the increase in length of any section of the metal in any direction for an increase in temperature of 1 degree F). The total expansion in diameter that is required consists of the total allowance for shrinkage and an added amount for clearance. The value of the coefficient α is, for nickel-steel, 0.000007; for steel in general, 0.0000065; for cast iron, 0.0000062. As an example, take an outer member of steel to be expanded 0.005 inch per inch of internal diameter, 0.001 being the shrinkage allowance and the remainder for clearance. Then

$$\alpha \times t^\circ = 0.005$$

$$t = \frac{0.005}{0.0000065} = 769 \text{ degrees F}$$

The value t is the number of degrees F that the temperature of the member must be raised above that of the room temperature.

ANSI Standard Limits and Fits (ANSI B4.1-1967 (R1994)).—This American National Standard for Preferred Limits and Fits for Cylindrical Parts presents definitions of terms applying to fits between plain (non threaded) cylindrical parts and makes recommendations on preferred sizes, allowances, tolerances, and fits for use wherever they are applicable. This standard is in accord with the recommendations of American-British-Canadian (ABC) conferences up to a diameter of 20 inches. Experimental work is being carried on with the objective of reaching agreement in the range above 20 inches. The recommendations in the standard are presented for guidance and for use where they might serve to improve and simplify products, practices, and facilities. They should have application for a wide range of products.

As revised in 1967, and reaffirmed in 1979, the definitions in ANSI B4.1 have been expanded and some of the limits in certain classes have been changed.

Factors Affecting Selection of Fits.—Many factors, such as length of engagement, bearing load, speed, lubrication, temperature, humidity, and materials must be taken into consideration in the selection of fits for a particular application, and modifications in the ANSI recommendations may be required to satisfy extreme conditions. Subsequent adjustments may also be found desirable as a result of experience in a particular application to suit critical functional requirements or to permit optimum manufacturing economy.

Definitions.—The following terms are defined in this standard:

Nominal Size: The nominal size is the designation used for the purpose of general identification.

Dimension: A dimension is a geometrical characteristic such as diameter, length, angle, or center distance.

Size: Size is a designation of magnitude. When a value is assigned to a dimension, it is referred to as the size of that dimension. (It is recognized that the words "dimension" and "size" are both used at times to convey the meaning of magnitude.)

Allowance: An allowance is a prescribed difference between the maximum material limits of mating parts. (See definition of *Fit*). It is a minimum clearance (positive allowance) or maximum interference (negative allowance) between such parts.

Tolerance: A tolerance is the total permissible variation of a size. The tolerance is the difference between the limits of size.

Basic Size: The basic size is that size from which the limits of size are derived by the application of allowances and tolerances.

Design Size: The design size is the basic size with allowance applied, from which the limits of size are derived by the application of tolerances. Where there is no allowance, the design size is the same as the basic size.

Actual Size: An actual size is a measured size.

Limits of Size: The limits of size are the applicable maximum and minimum sizes.

Maximum Material Limit: A maximum material limit is that limit of size that provides the maximum amount of material for the part. Normally it is the maximum limit of size of an external dimension or the minimum limit of size of an internal dimension.*

Minimum Material Limit: A minimum material limit is that limit of size that provides the minimum amount of material for the part. Normally it is the minimum limit of size of an external dimension or the maximum limit of size of an internal dimension.*

Tolerance Limit: A tolerance limit is the variation, positive or negative, by which a size is permitted to depart from the design size.

Unilateral Tolerance: A unilateral tolerance is a tolerance in which variation is permitted in only one direction from the design size.

Bilateral Tolerance: A bilateral tolerance is a tolerance in which variation is permitted in both directions from the design size.

Unilateral Tolerance System: A design plan that uses only unilateral tolerances is known as a Unilateral Tolerance System.

Bilateral Tolerance System: A design plan that uses only bilateral tolerances is known as a Bilateral Tolerance System.

Fit: Fit is the general term used to signify the range of tightness that may result from the application of a specific combination of allowances and tolerances in the design of mating parts.

Actual Fit: The actual fit between two mating parts is the relation existing between them with respect to the amount of clearance or interference that is present when they are assembled. (Fits are of three general types: clearance, transition, and interference.)

Clearance Fit: A clearance fit is one having limits of size so specified that a clearance always results when mating parts are assembled.

Interference Fit: An interference fit is one having limits of size so specified that an interference always results when mating parts are assembled.

Transition Fit: A transition fit is one having limits of size so specified that either a clearance or an interference may result when mating parts are assembled.

Basic Hole System: A basic hole system is a system of fits in which the design size of the hole is the basic size and the allowance, if any, is applied to the shaft.

Basic Shaft System: A basic shaft system is a system of fits in which the design size of the shaft is the basic size and the allowance, if any, is applied to the hole.

* An example of exceptions: an exterior corner radius where the maximum radius is the minimum material limit and the minimum radius is the maximum material limit.

Preferred Basic Sizes.—In specifying fits, the basic size of mating parts may be chosen from the decimal series or the fractional series in the following table.

Table 1. Preferred Basic Sizes

Decimal			Fractional					
0.010	2.00	8.50	1/64	0.015625	2 1/4	2.2500	9 1/2	9.5000
0.012	2.20	9.00	1/32	0.03125	2 1/2	2.5000	10	10.0000
0.016	2.40	9.50	1/16	0.0625	2 3/4	2.7500	10 1/2	10.5000
0.020	2.60	10.00	3/32	0.09375	3	3.0000	11	11.0000
0.025	2.80	10.50	1/8	0.1250	3 1/4	3.2500	11 1/2	11.5000
0.032	3.00	11.00	5/32	0.15625	3 1/2	3.5000	12	12.0000
0.040	3.20	11.50	3/16	0.1875	3 3/4	3.7500	12 1/2	12.5000
0.05	3.40	12.00	1/4	0.2500	4	4.0000	13	13.0000
0.06	3.60	12.50	5/16	0.3125	4 1/4	4.2500	13 1/2	13.5000
0.08	3.80	13.00	3/8	0.3750	4 1/2	4.5000	14	14.0000
0.10	4.00	13.50	7/16	0.4375	4 3/4	4.7500	14 1/2	14.5000
0.12	4.20	14.00	1/2	0.5000	5	5.0000	15	15.0000
0.16	4.40	14.50	9/16	0.5625	5 1/4	5.2500	15 1/2	15.5000
0.20	4.60	15.00	5/8	0.6250	5 1/2	5.5000	16	16.0000
0.24	4.80	15.50	11/16	0.6875	5 3/4	5.7500	16 1/2	16.5000
0.30	5.00	16.00	3/4	0.7500	6	6.0000	17	17.0000
0.40	5.20	16.50	7/8	0.8750	6 1/2	6.5000	17 1/2	17.5000
0.50	5.40	17.00	1	1.0000	7	7.0000	18	18.0000
0.60	5.60	17.50	1 1/4	1.2500	7 1/2	7.5000	18 1/2	18.5000
0.80	5.80	18.00	1 1/2	1.5000	8	8.0000	19	19.0000
1.00	6.00	18.50	1 3/4	1.7500	8 1/2	8.5000	19 1/2	19.5000
1.20	6.50	19.00	2	2.0000	9	9.0000	20	20.0000
1.40	7.00	19.50	…	…	…	…	…	…
1.60	7.50	20.00	…	…	…	…	…	…
1.80	8.00	…	…	…	…	…	…	…

All dimensions are in inches.

Preferred Series of Tolerances and Allowances (In thousandths of an inch)

0.1	1	10	100	0.3	3	30	…
…	1.2	12	125	…	3.5	35	…
0.15	1.4	14	…	0.4	4	40	…
…	1.6	16	160	…	4.5	45	…
…	1.8	18	…	0.5	5	50	…
0.2	2	20	200	0.6	6	60	…
…	2.2	22	…	0.7	7	70	…
0.25	2.5	25	250	0.8	8	80	…
…	2.8	28	…	0.9	9	…	…

Standard Tolerances.—The series of standard tolerances shown in Table 1 are so arranged that for any one grade they represent approximately similar production difficulties throughout the range of sizes. This table provides a suitable range from which appropriate tolerances for holes and shafts can be selected and enables standard gages to be used. The tolerances shown in Table 1 have been used in the succeeding tables for different classes of fits.

Table 1. ANSI Standard Tolerances *ANSI B4.1-1967 (R1987)*

Nominal Size, Inches		Grade									
		4	5	6	7	8	9	10	11	12	13
Over	To	Tolerances in thousandths of an inch[a]									
0	0.12	0.12	0.15	0.25	0.4	0.6	1.0	1.6	2.5	4	6
0.12	0.24	0.15	0.20	0.3	0.5	0.7	1.2	1.8	3.0	5	7
0.24	0.40	0.15	0.25	0.4	0.6	0.9	1.4	2.2	3.5	6	9
0.40	0.71	0.2	0.3	0.4	0.7	1.0	1.6	2.8	4.0	7	10
0.71	1.19	0.25	0.4	0.5	0.8	1.2	2.0	3.5	5.0	8	12
1.19	1.97	0.3	0.4	0.6	1.0	1.6	2.5	4.0	6	10	16
1.97	3.15	0.3	0.5	0.7	1.2	1.8	3.0	4.5	7	12	18
3.15	4.73	0.4	0.6	0.9	1.4	2.2	3.5	5	9	14	22
4.73	7.09	0.5	0.7	1.0	1.6	2.5	4.0	6	10	16	25
7.09	9.85	0.6	0.8	1.2	1.8	2.8	4.5	7	12	18	28
9.85	12.41	0.6	0.9	1.2	2.0	3.0	5.0	8	12	20	30
12.41	15.75	0.7	1.0	1.4	2.2	3.5	6	9	14	22	35
15.75	19.69	0.8	1.0	1.6	2.5	4	6	10	16	25	40
19.69	30.09	0.9	1.2	2.0	3	5	8	12	20	30	50
30.09	41.49	1.0	1.6	2.5	4	6	10	16	25	40	60
41.49	56.19	1.2	2.0	3	5	8	12	20	30	50	80
56.19	76.39	1.6	2.5	4	6	10	16	25	40	60	100
76.39	100.9	2.0	3	5	8	12	20	30	50	80	125
100.9	131.9	2.5	4	6	10	16	25	40	60	100	160
131.9	171.9	3	5	8	12	20	30	50	80	125	200
171.9	200	4	6	10	16	25	40	60	100	160	250

[a] All tolerances above heavy line are in accordance with American-British-Canadian (ABC) agreements.

Table 2. Relation of Machining Processes to Tolerance Grades

This chart may be used as a general guide to determine the machining processes that will under normal conditions, produce work withen the tolerance grades indicated.

(See also *Relation of Surface Roughness to Tolerances* starting on page 702.)

MACHINING OPERATION	TOLERANCE GRADES									
	4	5	6	7	8	9	10	11	12	13
Lapping & Honing	■	■								
Cylindrical Grinding		■	■	■						
Surface Grinding		■	■	■	■					
Diamond Turning		■	■	■						
Diamond Boring		■	■	■						
Broaching		■	■	■	■					
Reaming			■	■	■	■	■			
Turning				■	■	■	■	■	■	■
Boring					■	■	■	■	■	■
Milling							■	■	■	■
Planing & Shaping							■	■	■	■
Drilling							■	■	■	■

ANSI Standard Fits.—Tables 3 through 9 inclusive show a series of standard types and classes of fits on a unilateral hole basis, such that the fit produced by mating parts in any one class will produce approximately similar performance throughout the range of sizes. These tables prescribe the fit for any given size, or type of fit; they also prescribe the standard limits for the mating parts that will produce the fit. The fits listed in these tables contain all those that appear in the approved American-British-Canadian proposal.

Selection of Fits: In selecting limits of size for any application, the type of fit is determined first, based on the use or service required from the equipment being designed; then the limits of size of the mating parts are established, to insure that the desired fit will be produced.

Theoretically, an infinite number of fits could be chosen, but the number of standard fits shown in the accompanying tables should cover most applications.

Designation of Standard Fits: Standard fits are designated by means of the following symbols which, facilitate reference to classes of fit for educational purposes. The symbols are not intended to be shown on manufacturing drawings; instead, sizes should be specified on drawings.

The letter symbols used are as follows:

RC = Running or Sliding Clearance Fit

LC = Locational Clearance Fit

LT = Transition Clearance or Interference Fit

LN = Locational Interference Fit

FN = Force or Shrink Fit

These letter symbols are used in conjunction with numbers representing the class of fit; thus FN 4 represents a Class 4, force fit.

Each of these symbols (two letters and a number) represents a complete fit for which the minimum and maximum clearance or interference and the limits of size for the mating parts are given directly in the tables.

Description of Fits.—The classes of fits are arranged in three general groups: running and sliding fits, locational fits, and force fits.

Running and Sliding Fits (RC): Running and sliding fits, for which limits of clearance are given in Table 2, are intended to provide a similar running performance, with suitable lubrication allowance, throughout the range of sizes. The clearances for the first two classes, used chiefly as slide fits, increase more slowly with the diameter than for the other classes, so that accurate location is maintained even at the expense of free relative motion.

These fits may be described as follows:

RC 1 *Close sliding fits* are intended for the accurate location of parts that must assemble without perceptible play.

RC 2 *Sliding fits* are intended for accurate location, but with greater maximum clearance than class RC 1. Parts made to this fit move and turn easily but are not intended to run freely, and in the larger sizes may seize with small temperature changes.

RC 3 *Precision running fits* are about the closest fits that can be expected to run freely, and are intended for precision work at slow speeds and light journal pressures, but are not suitable where appreciable temperature differences are likely to be encountered.

RC 4 *Close running fits* are intended chiefly for running fits on accurate machinery with moderate surface speeds and journal pressures, where accurate location and minimum play are desired.

RC 5 and RC 6 *Medium running fits* are intended for higher running speeds, or heavy journal pressures, or both.

RC 7 *Free running fits* are intended for use where accuracy is not essential, or where large temperature variations are likely to be encountered, or under both these conditions.

RC 8 and RC 9 *Loose running fits* are intended for use where wide commercial tolerances may be necessary, together with an allowance, on the external member.

Locational Fits (LC, LT, and LN): Locational fits are fits intended to determine only the location of the mating parts; they may provide rigid or accurate location, as with interference fits, or provide some freedom of location, as with clearance fits. Accordingly, they are divided into three groups: clearance fits (LC), transition fits (LT), and interference fits (LN).

These are described as follows:

LC *Locational clearance fits* are intended for parts which are normally stationary, but that can be freely assembled or disassembled. They range from snug fits for parts requiring accuracy of location, through the medium clearance fits for parts such as spigots, to the looser fastener fits where freedom of assembly is of prime importance.

LT *Locational transition fits* are a compromise between clearance and interference fits, for applications where accuracy of location is important, but either a small amount of clearance or interference is permissible.

LN *Locational interference fits* are used where accuracy of location is of prime importance, and for parts requiring rigidity and alignment with no special requirements for bore pressure. Such fits are not intended for parts designed to transmit frictional loads from one part to another by virtue of the tightness of fit. These conditions are covered by force fits.

Force Fits: (FN): Force or shrink fits constitute a special type of interference fit, normally characterized by maintenance of constant bore pressures throughout the range of sizes. The interference therefore varies almost directly with diameter, and the difference between its minimum and maximum value is small, to maintain the resulting pressures within reasonable limits.

These fits are described as follows:

FN 1 *Light drive fits* are those requiring light assembly pressures, and produce more or less permanent assemblies. They are suitable for thin sections or long fits, or in cast-iron external members.

FN 2 *Medium drive fits* are suitable for ordinary steel parts, or for shrink fits on light sections. They are about the tightest fits that can be used with high-grade cast-iron external members.

FN 3 *Heavy drive fits* are suitable for heavier steel parts or for shrink fits in medium sections.

FN 4 and FN 5 *Force fits* are suitable for parts that can be highly stressed, or for shrink fits where the heavy pressing forces required are impractical.

Graphical Representation of Limits and Fits.—A visual comparison of the hole and shaft tolerances and the clearances or interferences provided by the various types and classes of fits can be obtained from the diagrams on page 633. These diagrams have been drawn to scale for a nominal diameter of 1 inch.

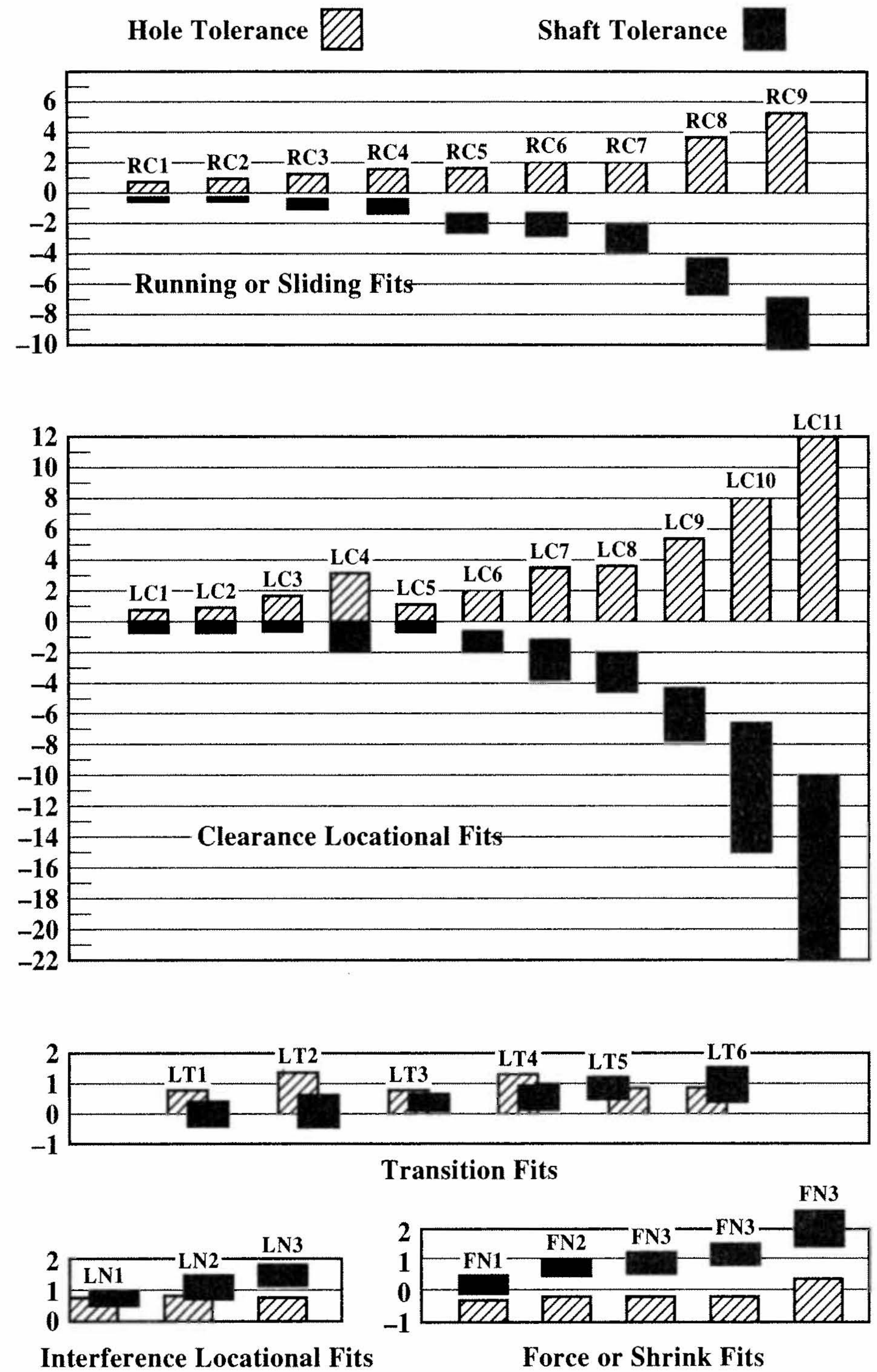
Use of Standard Fit Tables.—*Example 1:* A Class RC 1 fit is to be used in assembling a mating hole and shaft of 2-inch nominal diameter. This class of fit was selected because the application required accurate location of the parts with no perceptible play (see *Description of Fits*, RC 1 close sliding fits). From the data in Table 2, establish the limits of size and clearance of the hole and shaft.

Maximum hole = 2 + 0.0005 = 2.0005; minimum hole = 2 inches

Maximum shaft = 2 − 0.0004 = 1.9996; minimum shaft = 2 − 0.0007 = 1.9993 inches

Minimum clearance = 0.0004; maximum clearance = 0.0012 inch

Graphical Representation of ANSI Standard Limits and Fits



Diagrams show disposition of hole and shaft tolerances (in thousandths of an inch) with respect to basic size (0) for a diameter of 1 inch.

Table 3. American National Standard Running and Sliding Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches		Class RC 1			Class RC 2			Class RC 3			Class RC 4		
			Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits	
Over	To	Clearance[a]	Hole H5	Shaft g4	Clearance[a]	Hole H6	Shaft g5	Clearance[a]	Hole H7	Shaft f6	Clearance[a]	Hole H8	Shaft f7
		Values shown below are in thousandths of an inch											
0	0.12	0.1	+0.2	−0.1	0.1	+0.25	−0.1	0.3	+0.4	−0.3	0.3	+0.6	−0.3
		0.45	0	−0.25	0.55	0	−0.3	0.95	0	−0.55	1.3	0	−0.7
0.12	0.24	0.15	+0.2	−0.15	0.15	+0.3	−0.15	0.4	+0.5	−0.4	0.4	+0.7	−0.4
		0.5	0	−0.3	0.65	0	−0.35	1.12	0	−0.7	1.6	0	−0.9
0.24	0.40	0.2	+0.25	−0.2	0.2	+0.4	−0.2	0.5	+0.6	−0.5	0.5	+0.9	−0.5
		0.6	0	−0.35	0.85	0	−0.45	1.5	0	−0.9	2.0	0	−1.1
0.40	0.71	0.25	+0.3	−0.25	0.25	+0.4	−0.25	0.6	+0.7	−0.6	0.6	+1.0	−0.6
		0.75	0	−0.45	0.95	0	−0.55	1.7	0	−1.0	2.3	0	−1.3
0.71	1.19	0.3	+0.4	−0.3	0.3	+0.5	−0.3	0.8	+0.8	−0.8	0.8	+1.2	−0.8
		0.95	0	−0.55	1.2	0	−0.7	2.1	0	−1.3	2.8	0	−1.6
1.19	1.97	0.4	+0.4	−0.4	0.4	+0.6	−0.4	1.0	+1.0	−1.0	1.0	+1.6	−1.0
		1.1	0	−0.7	1.4	0	−0.8	2.6	0	−1.6	3.6	0	−2.0
1.97	3.15	0.4	+0.5	−0.4	0.4	+0.7	−0.4	1.2	+1.2	−1.2	1.2	+1.8	−1.2
		1.2	0	−0.7	1.6	0	−0.9	3.1	0	−1.9	4.2	0	−2.4
3.15	4.73	0.5	+0.6	−0.5	0.5	+0.9	−0.5	1.4	+1.4	−1.4	1.4	+2.2	−1.4
		1.5	0	−0.9	2.0	0	−1.1	3.7	0	−2.3	5.0	0	−2.8
4.73	7.09	0.6	+0.7	−0.6	0.6	+1.0	−0.6	1.6	+1.6	−1.6	1.6	+2.5	−1.6
		1.8	0	−1.1	2.3	0	−1.3	4.2	0	−2.6	5.7	0	−3.2
7.09	9.85	0.6	+0.8	−0.6	0.6	+1.2	−0.6	2.0	+1.8	−2.0	2.0	+2.8	−2.0
		2.0	0	−1.2	2.6	0	−1.4	5.0	0	−3.2	6.6	0	−3.8
9.85	12.41	0.8	+0.9	−0.8	0.8	+1.2	−0.8	2.5	+2.0	−2.5	2.5	+3.0	−2.5
		2.3	0	−1.4	2.9	0	−1.7	5.7	0	−3.7	7.5	0	−4.5
12.41	15.75	1.0	+1.0	−1.0	1.0	+1.4	−1.0	3.0	+2.2	−3.0	3.0	+3.5	−3.0
		2.7	0	−1.7	3.4	0	−2.0	6.6	0	−4.4	8.7	0	−5.2
15.75	19.69	1.2	+1.0	−1.2	1.2	+1.6	−1.2	4.0	+2.5	−4.0	4.0	+4.0	−4.0
		3.0	0	−2.0	3.8	0	−2.2	8.1	0	−5.6	10.5	0	−6.5

[a] Pairs of values shown represent minimum and maximum amounts of clearance resulting from application of standard tolerance limits.

Table 4. American National Standard Running and Sliding Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches		Class RC 5			Class RC 6			Class RC 7			Class RC 8			Class RC 9		
			Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits	
Over	To	Clearance[a]	Hole H8	Shaft e7	Clearance[a]	Hole H9	Shaft e8	Clearance[a]	Hole H9	Shaft d8	Clearance[a]	Hole H10	Shaft c9	Clearance[a]	Hole H11	Shaft
		Values shown below are in thousandths of an inch														
0	0.12	0.6	+0.6	− 0.6	0.6	+1.0	− 0.6	1.0	+1.0	− 1.0	2.5	+1.6	− 2.5	4.0	+2.5	− 4.0
		1.6	0	− 1.0	2.2	0	− 1.2	2.6	0	− 1.6	5.1	0	− 3.5	8.1	0	− 5.6
0.12	0.24	0.8	+0.7	− 0.8	0.8	+1.2	− 0.8	1.2	+1.2	− 1.2	2.8	+1.8	− 2.8	4.5	+3.0	− 4.5
		2.0	0	− 1.3	2.7	0	− 1.5	3.1	0	− 1.9	5.8	0	− 4.0	9.0	0	− 6.0
0.24	0.40	1.0	+0.9	− 1.0	1.0	+1.4	− 1.0	1.6	+1.4	− 1.6	3.0	+2.2	− 3.0	5.0	+3.5	− 5.0
		2.5	0	− 1.6	3.3	0	− 1.9	3.9	0	− 2.5	6.6	0	− 4.4	10.7	0	− 7.2
0.40	0.71	1.2	+1.0	− 1.2	1.2	+1.6	− 1.2	2.0	+1.6	− 2.0	3.5	+2.8	− 3.5	6.0	+4.0	− 6.0
		2.9	0	− 1.9	3.8	0	− 2.2	4.6	0	− 3.0	7.9	0	− 5.1	12.8	0	− 8.8
0.71	1.19	1.6	+1.2	− 1.6	1.6	+2.0	− 1.6	2.5	+2.0	− 2.5	4.5	+3.5	− 4.5	7.0	+5.0	− 7.0
		3.6	0	− 2.4	4.8	0	− 2.8	5.7	0	− 3.7	10.0	0	− 6.5	15.5	0	−10.5
1.19	1.97	2.0	+1.6	− 2.0	2.0	+2.5	− 2.0	3.0	+2.5	− 3.0	5.0	+4.0	− 5.0	8.0	+6.0	− 8.0
		4.6	0	− 3.0	6.1	0	− 3.6	7.1	0	− 4.6	11.5	0	− 7.5	18.0	0	−12.0
1.97	3.15	2.5	+1.8	− 2.5	2.5	+3.0	− 2.5	4.0	+3.0	− 4.0	6.0	+4.5	− 6.0	9.0	+7.0	− 9.0
		5.5	0	− 3.7	7.3	0	− 4.3	8.8	0	− 5.8	13.5	0	− 9.0	20.5	0	−13.5
3.15	4.73	3.0	+2.2	− 3.0	3.0	+3.5	− 3.0	5.0	+3.5	− 5.0	7.0	+5.0	− 7.0	10.0	+9.0	−10.0
		6.6	0	− 4.4	8.7	0	− 5.2	10.7	0	− 7.2	15.5	0	−10.5	24.0	0	−15.0
4.73	7.09	3.5	+2.5	− 3.5	3.5	+4.0	− 3.5	6.0	+4.0	− 6.0	8.0	+6.0	− 8.0	12.0	+10.0	−12.0
		7.6	0	− 5.1	10.0	0	− 6.0	12.5	0	− 8.5	18.0	0	−12.0	28.0	0	−18.0
7.09	9.85	4.0	+2.8	− 4.0	4.0	+4.5	− 4.0	7.0	+4.5	− 7.0	10.0	+7.0	−10.0	15.0	+12.0	−15.0
		8.6	0	− 5.8	11.3	0	− 6.8	14.3	0	− 9.8	21.5	0	−14.5	34.0	0	−22.0
9.85	12.41	5.0	+3.0	− 5.0	5.0	+5.0	− 5.0	8.0	+5.0	− 8.0	12.0	+8.0	−12.0	18.0	+12.0	−18.0
		10.0	0	− 7.0	13.0	0	− 8.0	16.0	0	−11.0	25.0	0	−17.0	38.0	0	−26.0
12.41	15.75	6.0	+3.5	− 6.0	6.0	+6.0	− 6.0	10.0	+6.0	−10.0	14.0	+9.0	−14.0	22.0	+14.0	−22.0
		11.7	0	− 8.2	15.5	0	− 9.5	19.5	0	−13.5	29.0	0	−20.0	45.0	0	−31.0
15.75	19.69	8.0	+4.0	− 8.0	8.0	+6.0	− 8.0	12.0	+6.0	−12.0	16.0	+10.0	−16.0	25.0	+16.0	−25.0
		14.5	0	−10.5	18.0	0	−12.0	22.0	0	−16.0	32.0	0	−22.0	51.0	0	−35.0

Tolerance limits given in body of table are added to or subtracted from basic size (as indicated by + or − sign) to obtain maximum and minimum sizes of mating parts.

All data above heavy lines are in accord with ABC agreements. Symbols H5, g4, etc. are hole and shaft designations in ABC system. Limits for sizes above 19.69 inches are also given in the ANSI Standard.

Table 5. American National Standard Clearance Locational Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class LC 1			Class LC 2			Class LC 3			Class LC 4			Class LC 5		
		Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits			Standard Tolerance Limits	
	Clearance[a]	Hole H6	Shaft h5	Clearance[a]	Hole H7	Shaft h6	Clearance[a]	Hole H8	Shaft h7	Clearance[a]	Hole H10	Shaft h9	Clearance[a]	Hole H7	Shaft g6
Over To	Values shown below are in thousandths of an inch														
0– 0.12	0 0.45	+0.25 0	0 –0.2	0 0.65	+0.4 0	0 –0.25	0 1	+0.6 0	0 –0.4	0 2.6	+1.6 0	0 –1.0	0.1 0.75	+0.4 0	–0.1 –0.35
0.12– 0.24	0 0.5	+0.3 0	0 –0.2	0 0.8	+0.5 0	0 –0.3	0 1.2	+0.7 0	0 –0.5	0 3.0	+1.8 0	0 –1.2	0.15 0.95	+0.5 0	–0.15 –0.45
0.24– 0.40	0 0.65	+0.4 0	0 –0.25	0 1.0	+0.6 0	0 –0.4	0 1.5	+0.9 0	0 –0.6	0 3.6	+2.2 0	0 –1.4	0.2 1.2	+0.6 0	–0.2 –0.6
0.40– 0.71	0 0.7	+0.4 0	0 –0.3	0 1.1	+0.7 0	0 –0.4	0 1.7	+1.0 0	0 –0.7	0 4.4	+2.8 0	0 –1.6	0.25 1.35	+0.7 0	–0.25 –0.65
0.71– 1.19	0 0.9	+0.5 0	0 –0.4	0 1.3	+0.8 0	0 –0.5	0 2	+1.2 0	0 –0.8	0 5.5	+3.5 0	0 –2.0	0.3 1.6	+0.8 0	–0.3 –0.8
1.19– 1.97	0 1.0	+0.6 0	0 –0.4	0 1.6	+1.0 0	0 –0.6	0 2.6	+1.6 0	0 –1	0 6.5	+4.0 0	0 –2.5	0.4 2.0	+1.0 0	–0.4 –1.0
1.97– 3.15	0 1.2	+0.7 0	0 –0.5	0 1.9	+1.2 0	0 –0.7	0 3	+1.8 0	0 –1.2	0 7.5	+4.5 0	0 –3	0.4 2.3	+1.2 0	–0.4 –1.1
3.15– 4.73	0 1.5	+0.9 0	0 –0.6	0 2.3	+1.4 0	0 –0.9	0 3.6	+2.2 0	0 –1.4	0 8.5	+5.0 0	0 –3.5	0.5 2.8	+1.4 0	–0.5 –1.4
4.73– 7.09	0 1.7	+1.0 0	0 –0.7	0 2.6	+1.6 0	0 –1.0	0 4.1	+2.5 0	0 –1.6	0 10.0	+6.0 0	0 –4	0.6 3.2	+1.6 0	–0.6 –1.6
7.09– 9.85	0 2.0	+1.2 0	0 –0.8	0 3.0	+1.8 0	0 –1.2	0 4.6	+2.8 0	0 –1.8	0 11.5	+7.0 0	0 –4.5	0.6 3.6	+1.8 0	–0.6 –1.8
9.85– 12.41	0 2.1	+1.2 0	0 –0.9	0 3.2	+2.0 0	0 –1.2	0 5	+3.0 0	0 –2.0	0 13.0	+8.0 0	0 –5	0.7 3.9	+2.0 0	–0.7 –1.9
12.41– 15.75	0 2.4	+1.4 0	0 –1.0	0 3.6	+2.2 0	0 –1.4	0 5.7	+3.5 0	0 –2.2	0 15.0	+9.0 0	0 –6	0.7 4.3	+2.2 0	–0.7 –2.1
15.75– 19.69	0 2.6	+1.6 0	0 –1.0	0 4.1	+2.5 0	0 –1.6	0 6.5	+4 0	0 –2.5	0 16.0	+10.0 0	0 –6	0.8 4.9	+2.5 0	–0.8 –2.4

[a] Pairs of values shown represent minimum and maximum amounts of interference resulting from application of standard tolerance limits.

Table 6. American National Standard Clearance Locational Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class LC 6			Class LC 7			Class LC 8			Class LC 9			Class LC 10			Class LC 11		
		Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits	
	Clearance[a]	Hole H9	Shaft f8	Clearance[a]	Hole H10	Shaft e9	Clearance[a]	Hole H10	Shaft d9	Clearance[a]	Hole H11	Shaft c10	Clearance[a]	Hole H12	Shaft	Clearance[a]	Hole H13	Shaft
Over To	Values shown below are in thousandths of an inch																	
0 – 0.12	0.3 1.9	+1.0 0	–0.3 –0.9	0.6 3.2	+1.6 0	– 0.6 – 1.6	1.0 2.0	+1.6 0	– 1.0 – 2.0	2.5 6.6	+2.5 0	– 2.5 – 4.1	4 12	+4 0	– 4 – 8	5 17	+6 0	– 5 – 11
0.12 – 0.24	0.4 2.3	+1.2 0	–0.4 –1.1	0.8 3.8	+1.8 0	– 0.8 – 2.0	1.2 4.2	+1.8 0	– 1.2 – 2.4	2.8 7.6	+3.0 0	– 2.8 – 4.6	4.5 14.5	+5 0	– 4.5 – 9.5	6 20	+7 0	– 6 –13
0.24 – 0.40	0.5 2.8	+1.4 0	–0.5 –1.4	1.0 4.6	+2.2 0	– 1.0 – 2.4	1.6 5.2	+2.2 0	– 1.6 – 3.0	3.0 8.7	+3.5 0	– 3.0 – 5.2	5 17	+6 0	– 5 –11	7 25	+9 0	– 7 –16
0.40 – 0.71	0.6 3.2	+1.6 0	–0.6 –1.6	1.2 5.6	+2.8 0	– 1.2 – 2.8	2.0 6.4	+2.8 0	– 2.0 – 3.6	3.5 10.3	+4.0 0	– 3.5 – 6.3	6 20	+7 0	– 6 –13	8 28	+10 0	– 8 –18
0.71 – 1.19	0.8 4.0	+2.0 0	–0.8 –2.0	1.6 7.1	+3.5 0	– 1.6 – 3.6	2.5 8.0	+3.5 0	– 2.5 – 4.5	4.5 13.0	+5.0 0	– 4.5 – 8.0	7 23	+8 0	– 7 –15	10 34	+12 0	–10 –22
1.19 – 1.97	1.0 5.1	+2.5 0	–1.0 –2.6	2.0 8.5	+4.0 0	– 2.0 – 4.5	3.6 9.5	+4.0 0	– 3.0 – 5.5	5.0 15.0	+6 0	– 5.0 – 9.0	8 28	+10 0	– 8 –18	12 44	+16 0	–12 –28
1.97 – 3.15	1.2 6.0	+3.0 0	–1.0 –3.0	2.5 10.0	+4.5 0	– 2.5 – 5.5	4.0 11.5	+4.5 0	– 4.0 – 7.0	6.0 17.5	+7 0	– 6.0 –10.5	10 34	+12 0	–10 –22	14 50	+18 0	–14 –32
3.15 – 4.73	1.4 7.1	+3.5 0	–1.4 –3.6	3.0 11.5	+5.0 0	– 3.0 – 6.5	5.0 13.5	+5.0 0	– 5.0 – 8.5	7 21	+9 0	– 7 –12	11 39	+14 0	–11 –25	16 60	+22 0	–16 –38
4.73 – 7.09	1.6 8.1	+4.0 0	–1.6 –4.1	3.5 13.5	+6.0 0	– 3.5 – 7.5	6 16	+6 0	– 6 –10	8 24	+10 0	– 8 –14	12 44	+16 0	–12 –28	18 68	+25 0	–18 –43
7.09 – 9.85	2.0 9.3	+4.5 0	–2.0 –4.8	4.0 15.5	+7.0 0	– 4.0 – 8.5	7 18.5	+7 0	– 7 –11.5	10 29	+12 0	–10 –17	16 52	+18 0	–16 –34	22 78	+28 0	–22 –50
9.85 – 12.41	2.2 10.2	+5.0 0	–2.2 –5.2	4.5 17.5	+8.0 0	– 4.5 – 9.5	7 20	+8 0	– 7 –12	12 32	+12 0	–12 –20	20 60	+20 0	–20 –40	28 88	+30 0	–28 –58
12.41 – 15.75	2.5 12.0	+6.0 0	–2.5 –6.0	5.0 20.0	+9.0 0	– 5 –11	8 23	+9 0	– 8 –14	14 37	+14 0	–14 –23	22 66	+22 0	–22 –44	30 100	+35 0	–30 –65
15.75 – 19.69	2.8 12.8	+6.0 0	–2.8 –6.8	5.0 21.0	+10.0 0	– 5 –11	9 25	+10 0	– 9 –15	16 42	+16 0	–16 –26	25 75	+25 0	–25 –50	35 115	+40 0	–35 –75

Tolerance limits given in body of table are added or subtracted to basic size (as indicated by + or – sign) to obtain maximum and minimum sizes of mating parts.

All data above heavy lines are in accordance with American-British-Canadian (ABC) agreements. Symbols H6, H7, s6, etc. are hole and shaft designations in ABC system. Limits for sizes above 19.69 inches are not covered by ABC agreements but are given in the ANSI Standard.

Table 7. ANSI Standard Transition Locational Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class LT 1			Class LT 2			Class LT 3			Class LT 4			Class LT 5			Class LT 6		
		Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits			Std. Tolerance Limits	
Over – To	Fit[a]	Hole H7	Shaft js6	Fit[a]	Hole H8	Shaft js7	Fit[a]	Hole H7	Shaft k6	Fit[a]	Hole H8	Shaft k7	Fit[a]	Hole H7	Shaft n6	Fit[a]	Hole H7	Shaft n7
	Values shown below are in thousandths of an inch																	
0 – 0.12	−0.12	+0.4	+0.12	−0.2	+0.6	+0.2							−0.5	+0.4	+0.5	−0.65	+0.4	+0.65
	+0.52	0	−0.12	+0.8	0	−0.2							+0.15	0	+0.25	+0.15	0	+0.25
0.12 – 0.24	−0.15	+0.5	+0.15	−0.25	+0.7	+0.25							−0.6	+0.5	+0.6	−0.8	+0.5	+0.8
	+0.65	0	−0.15	+0.95	0	−0.25							+0.2	0	+0.3	+0.2	0	+0.3
0.24 – 0.40	−0.2	+0.6	+0.2	−0.3	+0.9	+0.3	−0.5	+0.6	+0.5	−0.7	+0.9	+0.7	−0.8	+0.6	+0.8	−1.0	+0.6	+1.0
	+0.8	0	−0.2	+1.2	0	−0.3	+0.5	0	+0.1	+0.8	0	+0.1	+0.2	0	+0.4	+0.2	0	+0.4
0.40 – 0.71	−0.2	+0.7	+0.2	−0.35	+1.0	+0.35	−0.5	+0.7	+0.5	−0.8	+1.0	+0.8	−0.9	+0.7	+0.9	−1.2	+0.7	+1.2
	+0.9	0	−0.2	+1.35	0	−0.35	+0.6	0	+0.1	+0.9	0	+0.1	+0.2	0	+0.5	+0.2	0	+0.5
0.71 – 1.19	−0.25	+0.8	+0.25	−0.4	+1.2	+0.4	−0.6	+0.8	+0.6	−0.9	+1.2	+0.9	−1.1	+0.8	+1.1	−1.4	+0.8	+1.4
	+1.05	0	−0.25	+1.6	0	−0.4	+0.7	0	+0.1	+1.1	0	+0.1	+0.2	0	+0.6	+0.2	0	+0.6
1.19 – 1.97	−0.3	+1.0	+0.3	−0.5	+1.6	+0.5	−0.7	+1.0	+0.7	−1.1	+1.6	+1.1	−1.3	+1.0	+1.3	−1.7	+1.0	+1.7
	+1.3	0	−0.3	+2.1	0	−0.5	+0.9	0	+0.1	+1.5	0	+0.1	+0.3	0	+0.7	+0.3	0	+0.7
1.97 – 3.15	−0.3	+1.2	+0.3	−0.6	+1.8	+0.6	−0.8	+1.2	+0.8	−1.3	+1.8	+1.3	−1.5	+1.2	+1.5	−2.0	+1.2	+2.0
	+1.5	0	−0.3	+2.4	0	−0.6	+1.1	0	+0.1	+1.7	0	+0.1	+0.4	0	+0.8	+0.4	0	+0.8
3.15 – 4.73	−0.4	+1.4	+0.4	−0.7	+2.2	+0.7	−1.0	+1.4	+1.0	−1.5	+2.2	+1.5	−1.9	+1.4	+1.9	−2.4	+1.4	+2.4
	+1.8	0	−0.4	+2.9	0	−0.7	+1.3	0	+0.1	+2.1	0	+0.1	+0.4	0	+1.0	+0.4	0	+1.0
4.73 – 7.09	−0.5	+1.6	+0.5	−0.8	+2.5	+0.8	−1.1	+1.6	+1.1	−1.7	+2.5	+1.7	−2.2	+1.6	+2.2	−2.8	+1.6	+2.8
	+2.1	0	−0.5	+3.3	0	−0.8	+1.5	0	+0.1	+2.4	0	+0.1	+0.4	0	+1.2	+0.4	0	+1.2
7.09 – 9.85	−0.6	+1.8	+0.6	−0.9	+2.8	+0.9	−1.4	+1.8	+1.4	−2.0	+2.8	+2.0	−2.6	+1.8	+2.6	−3.2	+1.8	+3.2
	+2.4	0	−0.6	+3.7	0	−0.9	+1.6	0	+0.2	+2.6	0	+0.2	+0.4	0	+1.4	+0.4	0	+1.4
9.85 – 12.41	−0.6	+2.0	+0.6	−1.0	+3.0	+1.0	−1.4	+2.0	+1.4	−2.2	+3.0	+2.2	−2.6	+2.0	+2.6	−3.4	+2.0	+3.4
	+2.6	0	−6.6	+4.0	0	−1.0	+1.8	0	+0.2	+2.8	0	+0.2	+0.6	0	+1.4	+0.6	0	+1.4
12.41 – 15.75	−0.7	+2.2	+0.7	−1.0	+3.5	+1.0	−1.6	+2.2	+1.6	−2.4	+3.5	+2.4	−3.0	+2.2	+3.0	−3.8	+2.2	+3.8
	+2.9	0	−0.7	+4.5	0	−1.0	+2.0	0	+0.2	+3.3	0	+0.2	+0.6	0	+1.6	+0.6	0	+1.6
15.75 – 19.69	−0.8	+2.5	+0.8	−1.2	+4.0	+1.2	−1.8	+2.5	+1.8	−2.7	+4.0	+2.7	−3.4	+2.5	+3.4	−4.3	+2.5	+4.3
	+3.3	0	−0.8	+5.2	0	−1.2	+2.3	0	+0.2	+3.8	0	+0.2	+0.7	0	+1.8	+0.7	0	+1.8

[a] Pairs of values shown represent maximum amount of interference (–) and maximum amount of clearance (+) resulting from application of standard tolerance limits. All data above heavy lines are in accord with ABC agreements. Symbols H7, js6, etc., are hole and shaft designations in the ABC system.

Table 8. ANSI Standard Interference Location Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class LN 1			Class LN 2			Class LN 3		
		Standard Limits			Standard Limits			Standard Limits	
Over – To	Limits of Interference	Hole H6	Shaft n5	Limits of Interference	Hole H7	Shaft p6	Limits of Interference	Hole H7	Shaft r6
	Values shown below are given in thousandths of an inch								
0– 0.12	0	+0.25	+0.45	0	+0.4	+0.65	0.1	+0.4	+0.75
	0.45	0	+0.25	0.65	0	+0.4	0.75	0	+0.5
0.12– 0.24	0	+0.3	+0.5	0	+0.5	+0.8	0.1	+0.5	+0.9
	0.5	0	+0.3	0.8	0	+0.5	0.9	0	+0.6
0.24– 0.40	0	+0.4	+0.65	0	+0.6	+1.0	0.2	+0.6	+1.2
	0.65	0	+0.4	1.0	0	+0.6	1.2	0	+0.8
0.40– 0.71	0	+0.4	+0.8	0	+0.7	+1.1	0.3	+0.7	+1.4
	0.8	0	+0.4	1.1	0	+0.7	1.4	0	+1.0
0.71– 1.19	0	+0.5	+1.0	0	+0.8	+1.3	0.4	+0.8	+1.7
	1.0	0	+0.5	1.3	0	+0.8	1.7	0	+1.2
1.19– 1.97	0	+0.6	+1.1	0	+1.0	+1.6	0.4	+1.0	+2.0
	1.1	0	+0.6	1.6	0	+1.0	2.0	0	+1.4
1.97– 3.15	0.1	+0.7	+1.3	0.2	+1.2	+2.1	0.4	+1.2	+2.3
	1.3	0	+0.8	2.1	0	+1.4	2.3	0	+1.6
3.15– 4.73	0.1	+0.9	+1.6	0.2	+1.4	+2.5	0.6	+1.4	+2.9
	1.6	0	+1.0	2.5	0	+1.6	2.9	0	+2.0
4.73– 7.09	0.2	+1.0	+1.9	0.2	+1.6	+2.8	0.9	+1.6	+3.5
	1.9	0	+1.2	2.8	0	+1.8	3.5	0	+2.5
7.09– 9.85	0.2	+1.2	+2.2	0.2	+1.8	+3.2	1.2	+1.8	+4.2
	2.2	0	+1.4	3.2	0	+2.0	4.2	0	+3.0
9.85– 12.41	0.2	+1.2	+2.3	0.2	+2.0	+3.4	1.5	+2.0	+4.7
	2.3	0	+1.4	3.4	0	+2.2	4.7	0	+3.5
12.41– 15.75	0.2	+1.4	+2.6	0.3	+2.2	+3.9	2.3	+2.2	+5.9
	2.6	0	+1.6	3.9	0	+2.5	5.9	0	+4.5
15.75– 19.69	0.2	+1.6	+2.8	0.3	+2.5	+4.4	2.5	+2.5	+6.6
	2.8	0	+1.8	4.4	0	+2.8	6.6	0	+5.0

All data in this table are in accordance with American-British-Canadian (ABC) agreements.

Limits for sizes above 19.69 inches are not covered by ABC agreements but are given in the ANSI Standard.

Symbols H7, p6, etc., are hole and shaft designations in the ABC system.

Tolerance limits given in body of table are added or subtracted to basic size (as indicated by + or – sign) to obtain maximum and minimum sizes of mating parts.

Table 9. ANSI Standard Force and Shrink Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class FN 1			Class FN 2			Class FN 3			Class FN 4			Class FN 5		
	Interference[a]	Standard Tolerance Limits		Interference[a]	Standard Tolerance Limits		Interference[a]	Standard Tolerance Limits		Interference[a]	Standard Tolerance Limits		Interference[a]	Standard Tolerance Limits	
Over To		Hole H6	Shaft		Hole H7	Shaft s6		Hole H7	Shaft t6		Hole H7	Shaft u6		Hole H8	Shaft x7
	Values shown below are in thousandths of an inch														
0– 0.12	0.05	+0.25	+0.5	0.2	+0.4	+0.85				0.3	+0.4	+0.95	0.3	+0.6	+1.3
	0.5	0	+0.3	0.85	0	+0.6				0.95	0	+0.7	1.3	0	+0.9
0.12– 0.24	0.1	+0.3	+0.6	0.2	+0.5	+1.0				0.4	+0.5	+1.2	0.5	+0.7	+1.7
	0.6	0	+0.4	1.0	0	+0.7				1.2	0	+0.9	1.7	0	+1.2
0.24– 0.40	0.1	+0.4	+0.75	0.4	+0.6	+1.4				0.6	+0.6	+1.6	0.5	+0.9	+2.0
	0.75	0	+0.5	1.4	0	+1.0				1.6	0	+1.2	2.0	0	+1.4
0.40– 0.56	0.1	+0.4	+0.8	0.5	+0.7	+1.6				0.7	+0.7	+1.8	0.6	+1.0	+2.3
	0.8	0	+0.5	1.6	0	+1.2				1.8	0	+1.4	2.3	0	+1.6
0.56– 0.71	0.2	+0.4	+0.9	0.5	+0.7	+1.6				0.7	+0.7	+1.8	0.8	+1.0	+2.5
	0.9	0	+0.6	1.6	0	+1.2				1.8	0	+1.4	2.5	0	+1.8
0.71– 0.95	0.2	+0.5	+1.1	0.6	+0.8	+1.9				0.8	+0.8	+2.1	1.0	+1.2	+3.0
	1.1	0	+0.7	1.9	0	+1.4				2.1	0	+1.6	3.0	0	+2.2
0.95– 1.19	0.3	+0.5	+1.2	0.6	+0.8	+1.9	0.8	+0.8	+2.1	+1.0	+0.8	+2.3	1.3	+1.2	+3.3
	1.2	0	+0.8	1.9	0	+1.4	2.1	0	+1.6	2.3	0	+1.8	3.3	0	+2.5
1.19– 1.58	0.3	+0.6	+1.3	0.8	+1.0	+2.4	1.0	+1.0	+2.6	1.5	+1.0	+3.1	1.4	+1.6	+4.0
	1.3	0	+0.9	2.4	0	+1.8	2.6	0	+2.0	3.1	0	+2.5	4.0	0	+3.0
1.58– 1.97	0.4	+0.6	+1.4	0.8	+1.0	+2.4	1.2	+1.0	+2.8	1.8	+1.0	+3.4	2.4	+1.6	+5.0
	1.4	0	+1.0	2.4	0	+1.8	2.8	0	+2.2	3.4	0	+2.8	5.0	0	+4.0
1.97– 2.56	0.6	+0.7	+1.8	0.8	+1.2	+2.7	1.3	+1.2	+3.2	2.3	+1.2	+4.2	3.2	+1.8	+6.2
	1.8	0	+1.3	2.7	0	+2.0	3.2	0	+2.5	4.2	0	+3.5	6.2	0	+5.0
2.56– 3.15	0.7	+0.7	+1.9	1.0	+1.2	+2.9	1.8	+1.2	+3.7	2.8	+1.2	+4.7	4.2	+1.8	+7.2
	1.9	0	+1.4	2.9	0	+2.2	3.7	0	+3.0	4.7	0	+4.0	7.2	0	+6.0
3.15– 3.94	0.9	+0.9	+2.4	1.4	+1.4	+3.7	2.1	+1.4	+4.4	3.6	+1.4	+5.9	4.8	+2.2	+8.4
	2.4	0	+1.8	3.7	0	+2.8	4.4	0	+3.5	5.9	0	+5.0	8.4	0	+7.0
3.94– 4.73	1.1	+0.9	+2.6	1.6	+1.4	+3.9	2.6	+1.4	+4.9	4.6	+1.4	+6.9	5.8	+2.2	+9.4
	2.6	0	+2.0	3.9	0	+3.0	4.9	0	+4.0	6.9	0	+6.0	9.4	0	+8.0

Table 9. *(Continued)* ANSI Standard Force and Shrink Fits *ANSI B4.1-1967 (R1987)*

Nominal Size Range, Inches	Class FN 1			Class FN 2			Class FN 3			Class FN 4			Class FN 5		
	Interference[a]	Standard Tolerance Limits		Interference[a]	Standard Tolerance Limits		Interference[a]	Standard Tolerance Limits		Interference[a]	Standard Tolerance Limits		Interference[a]	Standard Tolerance Limits	
Over To		Hole H6	Shaft		Hole H7	Shaft s6		Hole H7	Shaft t6		Hole H7	Shaft u6		Hole H8	Shaft x7
	Values shown below are in thousandths of an inch														
4.73– 5.52	1.2	+1.0	+2.9	1.9	+1.6	+4.5	3.4	+1.6	+6.0	5.4	+1.6	+8.0	7.5	+2.5	+11.6
	2.9	0	+2.2	4.5	0	+3.5	6.0	0	+5.0	8.0	0	+7.0	11.6	0	+10.0
5.52– 6.30	1.5	+1.0	+3.2	2.4	+1.6	+5.0	3.4	+1.6	+6.0	5.4	+1.6	+8.0	9.5	+2.5	+13.6
	3.2	0	+2.5	5.0	0	+4.0	6.0	0	+5.0	8.0	0	+7.0	13.6	0	+12.0
6.30– 7.09	1.8	+1.0	+3.5	2.9	+1.6	+5.5	4.4	+1.6	+7.0	6.4	+1.6	+9.0	9.5	+2.5	+13.6
	3.5	0	+2.8	5.5	0	+4.5	7.0	0	+6.0	9.0	0	+8.0	13.6	0	+12.0
7.09– 7.88	1.8	+1.2	+3.8	3.2	+1.8	+6.2	5.2	+1.8	+8.2	7.2	+1.8	+10.2	11.2	+2.8	+15.8
	3.8	0	+3.0	6.2	0	+5.0	8.2	0	+7.0	10.2	0	+9.0	15.8	0	+14.0
7.88– 8.86	2.3	+1.2	+4.3	3.2	+1.8	+6.2	5.2	+1.8	+8.2	8.2	+1.8	+11.2	13.2	+2.8	+17.8
	4.3	0	+3.5	6.2	0	+5.0	8.2	0	+7.0	11.2	0	+10.0	17.8	0	+16.0
8.86– 9.85	2.3	+1.2	+4.3	4.2	+1.8	+7.2	6.2	+1.8	+9.2	10.2	+1.8	+13.2	13.2	+2.8	+17.8
	4.3	0	+3.5	7.2	0	+6.0	9.2	0	+8.0	13.2	0	+12.0	17.8	0	+16.0
9.85– 11.03	2.8	+1.2	+4.9	4.0	+2.0	+7.2	7.0	+2.0	+10.2	10.0	+2.0	+13.2	15.0	+3.0	+20.0
	4.9	0	+4.0	7.2	0	+6.0	10.2	0	+9.0	13.2	0	+12.0	20.0	0	+18.0
11.03– 12.41	2.8	+1.2	+4.9	5.0	+2.0	+8.2	7.0	+2.0	+10.2	12.0	+2.0	+15.2	17.0	+3.0	+22.0
	4.9	0	+4.0	8.2	0	+7.0	10.2	0	+9.0	15.2	0	+14.0	22.0	0	+20.0
12.41– 13.98	3.1	+1.4	+5.5	5.8	+2.2	+9.4	7.8	+2.2	+11.4	13.8	+2.2	+17.4	18.5	+3.5	+24.2
	5.5	0	+4.5	9.4	0	+8.0	11.4	0	+10.0	17.4	0	+16.0	24.2	0	+22.0
13.98– 15.75	3.6	+1.4	+6.1	5.8	+2.2	+9.4	9.8	+2.2	+13.4	15.8	+2.2	+19.4	21.5	+3.5	+27.2
	6.1	0	+5.0	9.4	0	+8.0	13.4	0	+12.0	19.4	0	+18.0	27.2	0	+25.0
15.75– 17.72	4.4	+1.6	+7.0	6.5	+2.5	+10.6	+9.5	+2.5	+13.6	17.5	+2.5	+21.6	24.0	+4.0	+30.5
	7.0	0	+6.0	10.6	0	+9.0	13.6	0	+12.0	21.6	0	+20.0	30.5	0	+28.0
17.72– 19.69	4.4	+1.6	+7.0	7.5	+2.5	+11.6	11.5	+2.5	+15.6	19.5	+2.5	+23.6	26.0	+4.0	+32.5
	7.0	0	+6.0	11.6	0	+10.0	15.6	0	+14.0	23.6	0	+22.0	32.5	0	+30.0

[a] Pairs of values shown represent minimum and maximum amounts of interference resulting from application of standard tolerance limits.

All data above heavy lines are in accordance with American-British-Canadian (ABC) agreements. Symbols H6, H7, s6, etc., are hole and shaft designations in the ABC system. Limits for sizes above 19.69 inches are not covered by ABC agreements but are given in the ANSI standard.