CHAPTER 7

MILLING MACHINES AND MILLING OPERATIONS

CHAPTER LEARNING OBJECTIVES

Upon completing this chapter, you should be able to do the following:

- *Describe and explain the use of milling machines.*
- *Describe the major components of milling machines.*
- *Describe and explain the use of workholding devices.*
- *Describe and explain the use of milling machine attachments.*
- *Explain indexing.*
- *Explain the selection and use of milling cutters.*
- *Explain milling machine setup and operation.*
- *Explain the use of feeds, speeds. and coolants in milling operations.*

A milling machine removes metal with a revolving cutting tool called a milling cutter. With various attachments, you can use milling machines for boring, slotting, circular milling, dividing, and drilling; cutting keyways, racks, and gears; and fluting taps and reamers.

You must be able to set up the milling machine to machine flat, angular, and formed surfaces. These jobs include the keyways, hexagonal and square heads on nuts and bolts, T-slots and dovetails, and spur gear teeth. To set up the machine, you must compute feeds and speeds, and select and mount the proper holding device and the proper cutter to handle the job.

You must also know how to align and level the machine. Manufacturers provide these instructions for their machines; follow them carefully.

As with any shop equipment you must observe all posted safety precautions. Review your equipment operators manual for safety precautions and any chapters of *Navy Occupational Safety and Health (NAVOSH) Program Manual for Forces Afloat*, OPNAV Instruction 5100.19B, that pertain to the equipment you will be operating.

Most Navy machine shops have the knee and column type of milling machine. This machine has a fixed spindle and a vertically adjustable table. We will discuss the knee and column type of milling machine in this chapter, but keep in mind that most of the information we give you also applies to other types of milling machines such as a horizontal boring mill, which is a typical bed-type milling machine.

The Navy uses three types of knee and column milling machines; the universal, the plain, and the vertical spindle, which we will describe in the next paragraphs. Where only one type can be installed, the universal type is usually selected.

The UNIVERSAL MILLING MACHINE (fig. 7-1) has all the principal features of the other types of milling machines. It can handle nearly all classes of milling work. You can take vertical cuts by feeding the table up or down. You can move the table in two directions in the horizontal plane—either at a right angle to, or parallel to, the axis of the spindle. The principal advantage of the universal mill over the plain mill is that you can swivel the table on the saddle. Therefore, you can move the table in the horizontal plane at an angle to the axis of the spindle.

Deleted—No permission granted for electronic copy.

Figure 7-1.—Universal milling machine.

Deleted—No permission granted for electronic copy.

Figure 7-2.—Plain milling machine.

Deleted—No permission granted for electronic copy.

Figure 7-3.—Vertical spindle milling machine.

Deleted—No permission granted for electronic copy.

Figure 7-4.—Small vertical milling machine.

This machine is used to cut most types of gears, milling cutters, and twist drills and is used for various kinds of straight and taper work.

The PLAIN MILLING MACHINE (fig. 7-2) is the simplest milling machine because it has only a few of the features found on the other machines. You can move the table in three directions: longitudinally (at a right angle to the spindle), transversely (parallel to the spindle), and vertically (up and down). This machine's major advantage is its ability to take heavy cuts at fast speeds. The machine's rigid construction makes this possible.

The VERTICAL SPINDLE MILLING MACHINE (fig. 7-3) has the spindle in a vertical position and at a right angle to the surface of the table. The spindle has a vertical movement, and the table can be moved vertically, longitudinally, and transversely. You can control movement of both the spindle and the table manually or by power. You can use this machine for face milling, profiling, and die sinking and for various odd-shaped jobs. You also can use it used to advantage to bore holes. Various small vertical spindle milling machines (fig. 7-4) are also available for light, precision milling operations.

Deleted—No permission granted for electronic copy.

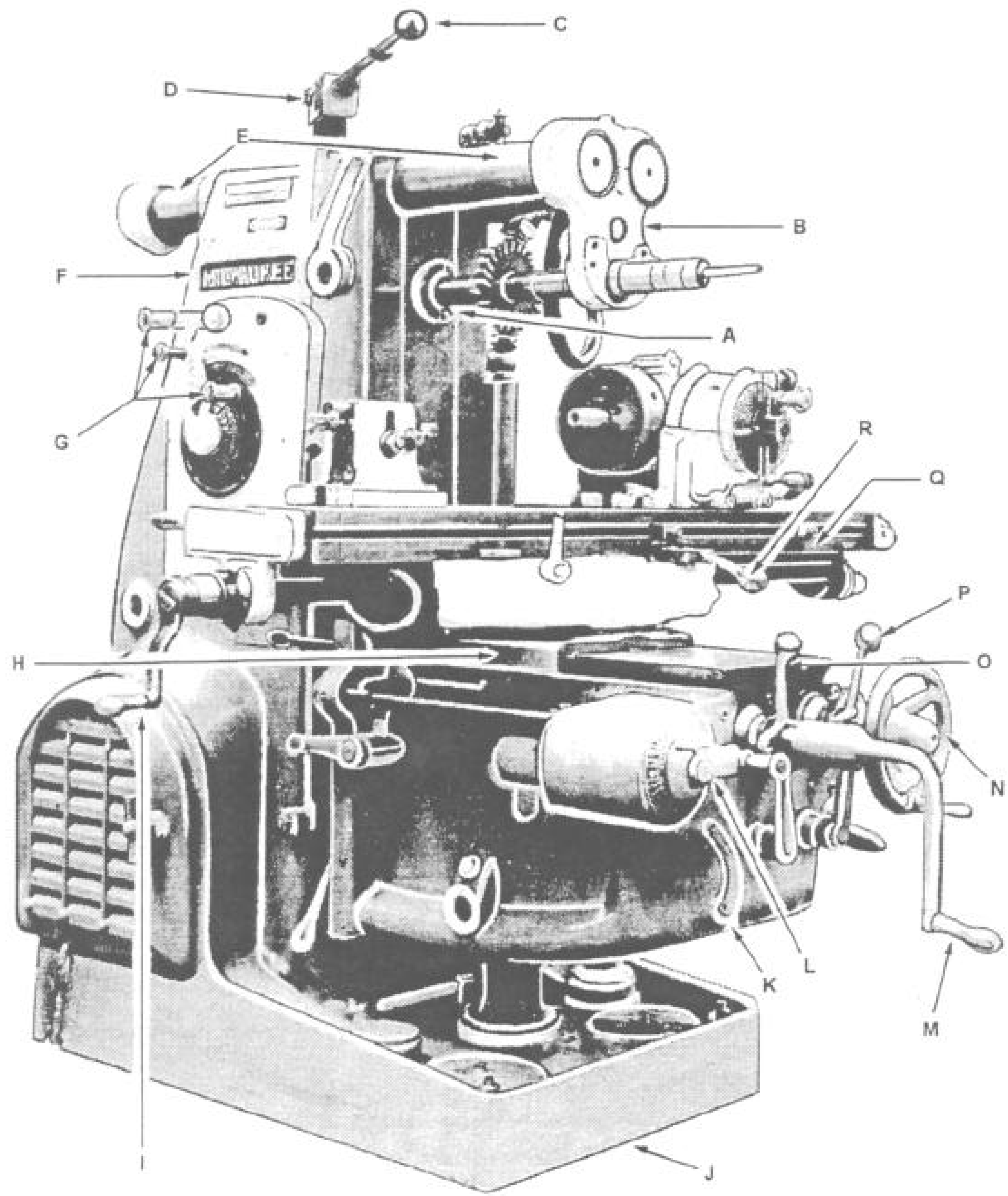
Figure 7-5.—Plain milling machine, showing operation controls.

MAJOR COMPONENTS

You must know the name and purpose of each of the main parts of a milling machine to understand the operations discussed later in this chapter. Keep in mind that although we are discussing a knee and column milling machine you can apply most of the information to the other types.

Figure 7-5 shows a plain knee and column milling machine, and figure 7-6 shows a universal knee and column milling machine. Look at these figures to help you identify the components described in the following paragraphs.

COLUMN: The column, including the base, is the main casting that supports all other parts of the machine. An oil reservoir and a pump in the column keep the spindle lubricated. The column rests on a base that contains a coolant reservoir and a pump that you can use when you perform any machining operation that requires a coolant.



A. Spindle	**G. Spindle speed selector levers**	**M. Knee elevating crank**
B. Arbor support	**H. Saddle and swivel**	**N. Transverse handwheel**
C. Spindle clutch lever	**I. Longitudinal handcrank**	**O. Vertical feed control**
D. Switch	**J. Base**	**P. Transverse feed control**
E. Overarm	**K. Knee**	**Q. Table feed trip dog**
F. Column	**L. Feed dial**	**R. Longitudinal feed control**

28.366

Figure 7-6.—Universal knee and column milling machine with horizontal spindle.

KNEE: The knee is the casting that supports the table and the saddle. The feed change gearing is enclosed within the knee. It is supported and is adjusted by turning the elevating screw. The knee is fastened to the column by dovetail ways. You can raise or lower the knee by either hand or power feed. You usually use hand feed to take the depth of cut or to position the work and power feed to move the work during the operation.

SADDLE and SWIVEL TABLE: The saddle slides on a horizontal dovetail (which is parallel to the axis of the spindle) on the knee. The swivel table (on universal machines only) is attached to the saddle and can be swiveled approximately 45° in either direction.

POWER FEED MECHANISM: The power feed mechanism is contained in the knee and controls the longitudinal, transverse (in and out), and vertical feeds. To set the rate of feed on machines, like the one in figure 7-5, position the feed selection levers as indicated on the feed selection plate. On machines like the one in figure 7-6, turn the speed selection handle until the desired rate of feed is indicated on the feed dial. Most milling machines have a rapid traverse lever that you can engage when you want to temporarily increase the speed of the longitudinal, transverse, or vertical feeds. For example, you would engage this lever to position or align the work.

NOTE: For safety reasons, you must use extreme caution whenever you use the rapid traverse controls.

TABLE: The table is the rectangular casting located on top of the saddle. It contains several T-slots in which you can fasten work or workholding devices. You can move the table by hand or by power. To move it by hand, engage and turn the longitudinal handcrank. To move it by power, engage the longitudinal directional feed control lever. You can position this lever to the left, to the right, or in the center. Place the end of the lever to the left to feed the table toward the left. Place it to the right to feed the table toward the right. Place it in the center to disengage the power feed or to feed the table by hand.

SPINDLE: The spindle holds and drives the various cutting tools. It is a shaft mounted on bearings supported by the column. The spindle is driven by an electric motor through a train of gears, all mounted within the column. The front end of the spindle, which is near the table, has an internal taper machined in it. The internal taper (3 1/2 inches per foot) permits you to mount tapered-shank cutter holders and cutter arbors. Two keys, located on the face of the spindle, provide a positive drive for the cutter holder, or arbor. You secure the holder, or arbor, in the spindle by a drawbolt and jamnut, as shown in figure 7-7. Large face mills are sometimes mounted directly to the spindle nose.

OVERARM: The overarm is the horizontal beam to which you fasten the arbor support. The overarm may be a single casting that slides in dovetail ways on the top of the column (fig. 7-5) or it may consist of one or two cylindrical bars that slide through holes in the column, as shown in figure 7-6. To position the overarm on some machines, first unclamp locknuts and then extend the overarm by turning a crank. On others, move the overarm by simply pushing on it. You should extend the overarm only far enough to position the arbor support to make the setup as rigid as possible. To place arbor supports on an overarm such as the one shown as B in figure 7-6, extend one of the bars approximately 1 inch farther than the other bar. Tighten the locknuts after you position the overarm. On some milling machines the coolant supply nozzle is fastened to the overarm. You can mount the nozzle

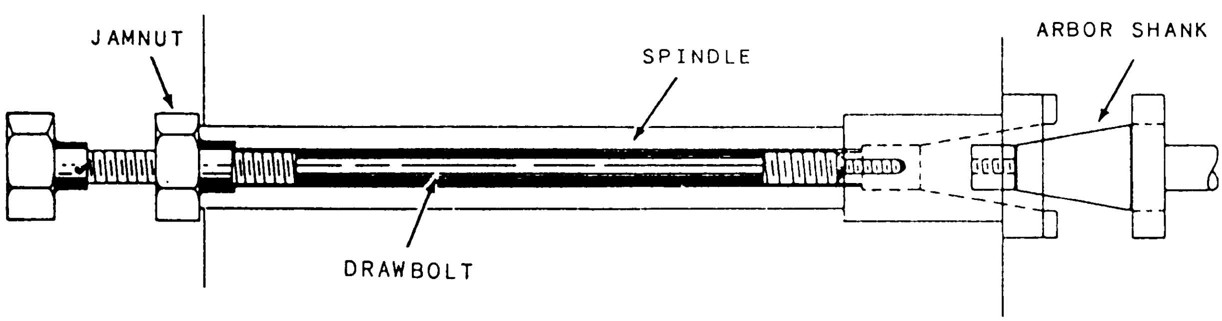


Figure 7-7.—Spindle drawbolt.

Deleted—No permission granted for electronic copy.

Figure 7-8.—Milling machine vises.

with a split clamp to the overarm after you have placed the arbor support in position.

ARBOR SUPPORT: The arbor support is a casting that contains a bearing that aligns the outer end of the arbor with the spindle. This helps to keep the arbor from springing during operations. Two types of arbor supports are commonly used. One has a small diameter bearing hole, usually 1-inch maximum diameter. The other has a large diameter bearing hole, usually up to 2 3/4 inches. An oil reservoir in the arbor support keeps the bearing surfaces lubricated. You can clamp an arbor support at any place you want on the overarm. Small arbor supports give additional clearance below the arbor supports when you are using small diameter cutters.

However, small arbor supports can provide support only at the extreme end of the arbor. For this reason they are not recommended for general use. Large arbor supports can provide support near the cutter, if necessary.

NOTE: Before loosening or tightening the arbor nut, you must install the arbor support. This will prevent bending or springing of the arbor.

MACHINE DESIGNATION: All milling machines are identified by four basic factors: size, horsepower, model, and type. The size of a milling machine is based on the longitudinal (from left to right) table travel in inches. Vertical, cross, and longitudinal travel are all closely related as far as overall capacity is concerned. For size designation, only the longitudinal travel is used. There are six sizes of knee-type milling machines, with each number representing the number of inches of travel.

Standard Size	Longitudinal Table Travel
No. 1	22 inches
No. 2	28 inches
No. 3	34 inches
No. 4	42 inches
No. 5	50 inches
No. 6	60 inches

If the milling machine in your shop is labeled No. 2HL, it has a table travel of 28 inches; if it is labeled No. 5LD, it has a travel of 50 inches. The model designation is determined by the manufacturer, and features vary with different brands. The type of milling machine is designated as plain or universal, horizontal or vertical, and knee and column or bed. In addition, machines may have other special type designations.

Standard equipment used with milling machines in Navy ships includes workholding devices, spindle attachments, cutters, arbors, and any special tools needed to set up the machines for milling. This equipment allows you to hold and cut the great variety of milling jobs you will find in Navy repair work.

WORKHOLDING DEVICES

The following workholding devices are the ones you will probably use most frequently.

VISES

The vises commonly used on milling machines are the flanged plain vise, the swivel vise, and the toolmaker's universal vise (fig. 7-8). The flanged vise provides the most support for a rigid workpiece. The swivel vise is similar to the flanged vise, but the setup is less rigid because the workpiece can be swiveled in a horizontal plane to any required angle. The toolmaker's universal vise provides the least rigid support because it is designed to set up the workpiece at a complex angle in relation to the axis of the spindle and to the surface of the table.

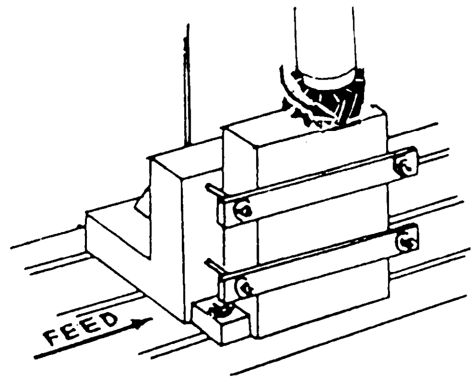


Figure 7-9.—Right-angle plate.

RIGHT-ANGLE PLATE

The right-angle plate (fig. 7-9) is attached to the table. The right-angle slot permits mounting the index head so the axis of the head is parallel to the milling machine spindle. With this attachment you can make work setups that are off center or at a right angle to the table T-slots. The standard size plate T-slots make it convenient to change from one setting to another to mill a surface at a right angle.

TOOLMAKER'S KNEE

The toolmaker's knee (fig. 7-10) is a simple but useful attachment used to set up angular work, not only for milling but also for shaper, drill press, and grinder operations. You mount a toolmaker's knee, which may have either a stationary or rotatable base,

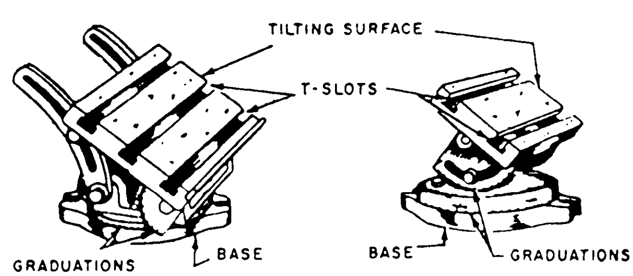


FIgure 7-10.—Toolmaker's knees.

to the table of the milling machine. The base of the rotatable type is graduated in degrees. This feature allows you to machine compound angles. The toolmaker's knee has a tilting surface with a built-in protractor head graduated in degrees to set the table or a vernier scale for more accurate settings.

CIRCULAR MILLING ATTACHMENT

The circular milling attachment, or rotary table (fig. 7-11), is used to set up work that must be rotated in a horizontal plane. The worktable is graduated (1/2° to 360°) around its circumference. You can turn the table by hand or by the table feed mechanism through a gear train, as shown in figure 7-11. An 80 to 1 worm and gear drive contained in the rotary table and index plate arrangement makes this device useful for accurate indexing of horizontal surfaces.

INDEXING EQUIPMENT

Indexing equipment (fig. 7-12) is used to hold and turn the workpiece so that a number of accurately spaced cuts can be made (gear teeth for example).

Deleted—No permission granted for electronic copy.

Figure 7-11.—Circular milling attachment with power feed mechanism.

Deleted—No permission granted for electronic copy.

Figure 7-12.—Indexing equipment.

The workpiece may be held in a chuck or a collet, attached to the dividing head spindle, or held between a live center in the dividing index head and a dead center in the footstock. The center of the footstock can be raised or lowered to set up tapered workpieces. The center rest can be used to support long slender work.

Figure 7-13 shows the internal components of the dividing head. The ratio between the worm and the gear is 40 to 1. By turning the worm one turn, you rotate the spindle 1/40 of a revolution. The index plate has a series of concentric circles of holes. You can use these holes to gauge partial turns of the worm shaft and to turn the spindle accurately in amounts smaller than 1/40 of a revolution. You can secure the index plate either to the dividing head housing or to a rotating shaft and you can adjust the crankpin radially for use in any circle of holes. You can also set the sector arms as a guide to span any number of holes in the index plate to provide a guide to rotate the index crank for partial turns. To rotate the workpiece, you can turn the dividing head spindle one of two ways: Do it directly by hand by disengaging the worm and drawing the plunger back, or by the index crank through the worm and worm gear.

The spindle is set in a swivel block so you can set the spindle at any angle from slightly below horizontal to slightly past vertical. We said earlier

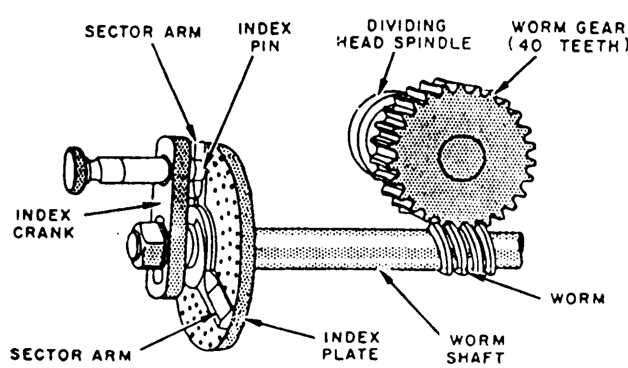


Figure 7-13.—Dividing head mechanism.

that most index heads have a 40 to 1 ratio. One well-known exception has a 5 to 1 ratio (see fig. 7-14). This ratio is made possible by a 5 to 1 gear ratio between the index crank and the dividing head spindle. The faster movement of the spindle with one turn of the index crank permits speedier production. It is also an advantage when you true work or test it for run out with a dial indicator. While this dividing head is made to a high standard of accuracy, it does not permit as wide a selection of divisions by simple indexing. Later in this chapter, we'll discuss differential indexing that you can do on the 5 to 1 ratio dividing head by using a differential indexing attachment.

The dividing head (also called an index head) may also be geared to the lead screw of the milling machine by a driving mechanism to turn the work-as required for helical and spiral milling. The index head may have one of several driving mechanisms. The most common of these is the ENCLOSED DRIVING MECHANISM, which is standard equipment on some makes of plain and universal knee and column milling machines. The enclosed driving mechanism has a lead range of 2 1/2 to 100 inches and is driven directly from the lead screw.

Figure 7-15 shows the gearing arrangement used on most milling machines. The gears are marked as follows:

Deleted—No permission granted for electronic copy.

Figure 7-14.—Universal spiral dividing head with a 5 to 1 ratio between the spindle and the index crank.

Deleted—No permission granted for electronic copy.

Figure 7-15.—Enclosed driving mechanism.

A = Gear on the worm shaft (driven)

B = First gear on the idler stud (driving)

C = Second gear on idler stud (driven)

D = Gear on lead screw (driving)

E and F = Idler gears

LOW LEAD DRIVE: For some models and makes of milling machines a low lead driving mechanism is available; however, additional parts must be built into the machine at the factory. This driving mechanism has a lead range of 0.125 to 100 inches.

LONG and SHORT LEAD DRIVE: When an extremely long or short lead is required, you can use the long and short lead attachment (fig. 7-16). As with the low-lead driving mechanism, the milling machine must have certain parts built into the machine at the factory. In this attachment, an auxiliary shaft in the table drive mechanism supplies power through the gear train to the dividing head. It also supplies the power for the table lead screw, which is disengaged from the regular drive when the attachment is used. This attachment provides leads in the range between 0.010 and 1000 inches.

Deleted—No permission granted for electronic copy.

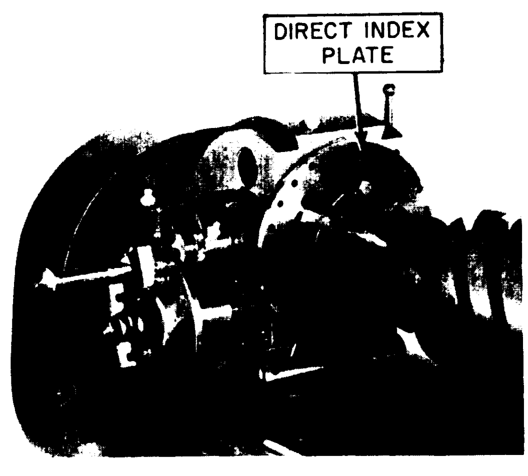
Figure 7-16.—The long and short lead attachment.

INDEXING THE WORK

Indexing is done by the direct, plain, compound, or differential method. The direct and plain methods are the most commonly used; the compound and differential methods are used only when the job cannot be done by plain or direct indexing.

DIRECT INDEXING

Direct indexing, sometimes referred to as rapid indexing, is the simplest method of indexing. Figure 7-17 shows the front index plate attached to the work spindle. The front index plate usually has 24 equally spaced holes. These holes can be engaged by the front index pin, which is spring-loaded and moved in and out by a small lever. Rapid indexing requires that the worm and the worm wheel be disengaged so that the spindle can be moved by hand. Numbers that can be divided into 24 can be indexed in this manner.



Courtesy of Brown and Sharpe Manufacturing Company

28.209

Figure 7-17.—Direct index plate.

Rapid indexing is used when a large number of duplicate parts are to be milled.

To find the number of holes to move the index plate, divide 24 by the number of divisions required.

Number of holes to move = 24/ N where N = required number of divisions

Example: Indexing for a hexagon head bolt: because a hexagon head has six flats,

$$\frac{24}{N} = \frac{24}{6} = 4 \text{ holes}$$

IN ANY INDEXING OPERATION, ALWAYS START COUNTING FROM THE HOLE ADJACENT TO THE CRANKPIN. During heavy cutting operations, clamp the spindle by the clamp screw to relieve strain on the index pin.

PLAIN INDEXING

Plain indexing, or simple indexing, is used when a circle must be divided into more parts than is possible by rapid indexing. Simple indexing requires that the spindle be moved by turning an index crank, which turns the worm that is meshed with the worm wheel. The ratio between worm and the worm wheel is 40 to 1. One turn of the index crank turns the index head spindle 1/40 of a complete turn. Therefore, 40 turns of the index crank are required to revolve the spindle chuck and the job 1 complete turn. To determine the number of turns or fractional parts of a turn of the index crank necessary to cut any required number of divisions, divide 40 by the number of divisions required.

Number of turns of the index crank = $\frac{40}{N}$

where N = number of divisions required

Example 1: Index for five divisions

$$\frac{40}{N} = \frac{40}{5} = 8 \text{ turns}$$

There are eight turns of the crank for each division.

Example 2: Index for eight divisions

$$\frac{40}{N} = \frac{40}{8} = 4 \text{ turns}$$

Example 3: Index for 10 divisions

$$\frac{40}{N} = \frac{40}{10} = 4 \text{ turns}$$

When the number of divisions required does not divide evenly into 40, the index crank must be moved a fractional part of a turn with index plates. A commonly used index head comes with three index plates. Each plate has six circles of holes, which we will use as an example.

Plate 1: 15-16-17-18-19-20

Plate 2: 21-23-27-29-31-33

Plate 3: 37-39-41-43-47-49

The previous examples of using the indexing formula 40/N gave results in complete turns of the index crank. This seldom happens on the typical indexing job. For example, indexing for 18 divisions

$$\frac{40}{N} = \frac{40}{18} = 2\frac{4}{18} \text{ turns}$$

The whole number indicates the complete turns of the index crank, the denominator of the fraction represents the index circle, and the numerator represents the number of holes to use on that circle. Because there is an 18-hole index circle, the mixed number 2 4/18 indicates that the index crank will be moved 2 full turns plus 4 holes on the 18-hole circle that you will find on index plate 1. The sector arms are positioned to include 4 holes and the hole in which the index crankpin is engaged. The number of holes (4) represents the movement of the index crank; the hole that engages the index crankpin is not included.

When the denominator of the indexing fraction is smaller or larger than the number of holes contained in any of the index circles, change it to a number representing one of the circles of holes. Do this by multiplying or dividing the numerator and the denominator by the same number. For example, to index for the machining of a hexagon (N = 6):

$$\frac{40}{N} = \frac{40}{6} \times \frac{3}{3} = \frac{120}{18} = 6\frac{12}{18} = 6\frac{2}{3} \text{ turns}$$

The denominator 3 will divide equally into the following circles of holes, so you can use any plate that contains one of the circles.

Plate 1: 15 and 18

Plate 2: 21 and 33

Plate 3: 39

To apply the fraction 2/3 to the circle you choose, convert the fraction to a fraction that has the number of holes in the circle as a denominator. For example, if you choose the 15-hole circle, the fraction 2/3 becomes 10/15. If plate 3 happens to be on the index head, multiply the denominator 3 by 13 to equal 39. In order not to change the value of the original indexing fraction, also multiply the numerator by 13.

$$\frac{2}{3} \times \frac{13}{13} = \frac{26}{39}$$

The original indexing rotation of 6 2/3 turns becomes 6 26/39 turns. Thus, to mill each side of a hexagon, you must move the index crank 6 full turns and 26 holes on the 39-hole circle.

When there are more than 40 divisions, you may divide both the numerator and the denominator of the fraction by a common divisor to obtain an index circle that is available. For example, if 160 divisions are required, $N = 160$; the fraction to be used is

$$\frac{40}{N} = \frac{40}{160}$$

Because there is no 160-hole circle, this fraction must be reduced. To use a 16-hole circle, divide the numerator and denominator by 10.

$$\frac{40/10}{160/10} = \frac{4}{16}$$

Turn 4 holes on the 16-hole circle.

It is usually more convenient to reduce the original fraction to its lowest terms and then multiply both terms of the fraction by a factor that will give a number representing a circle of holes.

$$\frac{40}{160} = \frac{1}{4}$$

$$\frac{1}{4} \times \frac{4}{4} = \frac{4}{16}$$

The following examples will further clarify the use of this formula:

Example 1: Index for 9 divisions.

$$\frac{40}{N} = \frac{40}{9} = 4\frac{4}{9}$$

If an 18-hole circle is used, the fraction becomes 4/9 × 2/2 = 8/18. For each division, turn the crank 4 turns and 8 holes on an 18-hole circle.

Example 2: Index for 136 divisions.

$$\frac{40}{N} = \frac{40}{136} = \frac{5}{17}$$

There is a 17-hole circle, so for each division turn the crank 5 holes on a 17-hole circle.

When setting the sector arms to space off the required number of holes on the index circle, DO NOT count the hole that the index crankpin is in.

Most manufacturers provide different plates for indexing. Later model Brown and Sharpe index heads use two plates with the following circle of holes:

Plate 1: 15, 16, 19, 23, 31, 37, 41, 43, 47

Plate 2: 17, 18, 20, 21, 27, 29, 33, 39, 47

The standard index plate supplied with the Cincinnati index head is provided with 11 different circles of holes on each side.

Side 1: 24-25-28-30-34-37-38-39-41-42-43

Side 2: 46-47-49-51-53-54-57-58-59-62-66

ANGULAR INDEXING

When you must divide work into degrees or fractions of a degree by plain indexing, remember that one turn of the index crank will rotate a point on the circumference of the work 1/40 of a revolution. Since there are 360° in a circle, one turn of the index crank will revolve the circumference of the work 1/40 of 360°, or 9°. Therefore, to use the index plate and fractional parts of a turn, 2 holes in an 18-hole circle equal 1° (1/9 turn × 9°/turn), 1 hole in a 27-hole circle equals 1/3° (1/27 turn × 9°/turn), 3 holes in a 54-hole circle equal 1/2° (1/18 turn × 9°/turn). To determine the number of turns and parts of a turn of the index crank for a desired number of degrees, divide the number of degrees by 9. The quotient will represent the number of complete turns and fractions of a turn that you should rotate the index crank. For example, use the following calculation to determine 15° when an index plate with a 54-hole circle is available.

$$\frac{15}{9} = 1\frac{6}{9} \times \frac{6}{6} = 1\frac{36}{54}$$

or one complete turn plus 36 holes on the 54-hole circle. The calculation to determine 13 1/2° when an index plate with an 18-hole circle is available, is as follows:

$$\frac{13.5}{9} = 1\frac{4.5}{9} \times \frac{2}{2} = 1\frac{9}{18}$$

or one complete turn plus 9 holes on the 18-hole circle.

When indexing angles are given in minutes, and approximate divisions are acceptable, you can determine the movement of the index crank and the proper index plate by the following calculations. To determine the number of minutes represented by one turn of the index crank, multiply the number of degrees covered in one turn of the index crank by 60:

$$9° \times 60' = 540$$

Therefore, one turn of the index crank will rotate the index head spindle 540 minutes.

The number of minutes (540) divided by the number of minutes in the division desired gives you the total number of holes there should be in the index plate used. (Moving the index crank one hole will rotate the index head spindle through the desired number of minutes of angle.) This method of indexing can be used only for approximate angles since ordinarily the quotient will come out in mixed numbers or in numbers for which there are no index plates available. However, when the quotient is nearly equal to the number of holes in an available index plate, the nearest number of holes can be used and the error will be very small. For example the calculation for 24 minutes would be

$$\frac{540}{24} = \frac{22.5}{1}$$

or 1 hole on the 22.5-hole circle. Since the index plate has no 22.5-hole circle, you should use a 23-hole circle plate.

If a quotient is not approximately equal to an available circle of holes, multiply by any trial number that will give a product equal to the number of holes in one of the available index circles. You can then move the crank the required number of holes to give the desired division. For example, use the following calculation to determine 54 minutes when an index plate that has a 20-hole circle is available.

$$\frac{54}{540} = \frac{1}{10} \times \frac{2}{2} = \frac{2}{20} \begin{matrix}\text{(no. of holes)} \\ \text{(20–hole circle)}\end{matrix}$$

or 2 holes on the 20-hole circle.

COMPOUND INDEXING

Compound indexing is a combination of two plain indexing procedures. You will index one number of divisions by using the standard plain indexing method, and another by turning the index plate (leaving the crankpin engaged in the hole as set in the first indexing operation) by a required amount. The difference between the amount indexed in the first and second operations results in the spindle turning the required amount for the number of divisions. Compound indexing is seldom used because (1) differential indexing is easier, (2) high-number index plates are usually available to provide any range of divisions normally required, and (3) the computation and actual operation are quite complicated, making it easy for errors to be introduced.

We will briefly describe compound indexing in the following example. To index 99 divisions proceed as follows:

1. Multiply the required number of divisions by the difference between the number of holes in two circles selected at random. Divide this product by 40 (ratio of spindle to crank) times the product of the two index hole circles. Assume you have selected the 27- and 33-hole circles. The resulting equation is

$$\frac{99 \times (33 - 27)}{40 \times 33 \times 27} \times \frac{99 \times 6}{40 \times 33 \times 27}$$

2. To make the solution easier, factor each term of the equation into its lowest prime factors and cancel where possible. For example:

$$\frac{(\not{3} \times \not{3} \times \not{11})\ (\not{3} \times \not{2})}{(\not{2} \times 2 \times 2 \times 5)\ (\not{11} \times \not{3})\ (\not{3} \times \not{3} \times 3)} = \frac{1}{60}$$

The result of this process must be in the form of a fraction as given (that is, 1 divided by some number). Always try to select the two circles that have factors that cancel out the factors in the numerator of the problem. When the numerator of the resulting fraction is greater than 1, divide it by the denominator and use the quotient (to the nearest whole number) instead of the denominator of the fraction.

3. The denominator of the resulting fraction derived in step 2 is the term used to find the number of turns and holes to index the spindle and index plate. To index for 99 divisions, turn the spindle by an amount equal to 60/33 or one complete turn plus 27 holes in the 33-hole circle; turn the index plate by an amount equal to 60/27, or two complete turns plus 6 holes in the 27-hole circle. If you turn the index crank clockwise, turn the index plate counterclockwise and vice versa.

DIFFERENTIAL INDEXING

Differential indexing is similar to compound indexing except that the index plate is turned during the indexing operation by gears connected to the dividing head spindle. Because the index plate movement is caused by the spindle movement, only one indexing procedure is required. The gear train between the dividing head spindle and the index plate provides the correct ratio of movement between the spindle and the index plate.

Figure 7-18 shows a dividing head set up for differential indexing. The index crank is turned as it is for plain indexing, thus turning the spindle gear and then the compound gear and the idler to drive the gear that turns the index plate. The manufacturer's technical manuals give specific procedures to install the gearing and arrange the index plate for differential indexing (and compound indexing).

To index 57 divisions, for example, take the following steps:

1. Select a number greater or lesser than the required number of divisions for which an available index plate can be used (60 for example).

2. The number of turns for plain indexing 60 divisions is 40/60 or 14/21, which will require 14 holes in a 21-hole circle in the index plate.

3. To find the required gear ratio, subtract the required number of divisions from the selected number or vice versa (depending on which is larger), and multiply the result by 40/60 (formula to index 60 divisions). Thus:

$$\text{Gear ratio} = (60 - 57) \times \frac{40}{60} = 3 \times \frac{40}{60} = \frac{2}{1}$$

The numerator indicates the spindle gear; the denominator indicates the driven gear.

Deleted—No permission granted for electronic copy.

Figure 7-18.—Differential Indexing.

4. Select two gears that have a 2 to 1 ratio (for example a 48-tooth gear and a 24-tooth gear).

5. If the selected number is greater than the actual number of divisions required, use one or three idlers in the simple gear train; if the selected number is smaller, use none or two idlers. The reverse is true for compound gear trains. Since the number is greater in this example, use one or three idlers.

6. Now turn the index crank 14 holes in the 21-hole circle of the index plate. As the crank turns the spindle, the gear train turns the index plate slightly faster than the index crank.

WIDE RANGE DIVIDER

In the majority of indexing operations, you can get the desired number of equally spaced divisions by using either direct or plain indexing. By using one or the other of these methods, you may index up to 2,640 divisions. To increase the range of divisions, use the high-number index plates in place of the standard index plate. These high-number plates have a greater number of circles of holes and a greater range of holes in the circles than the standard plates. This increases the range of possible divisions from 1,040 to 7,960.

In some instances, you may need to index beyond the range of any of these methods. To further increase the range, use a universal dividing head that has a wide range divider. This type of indexing equipment

Deleted—No permission granted for electronic copy.

Figure 7-19.—The wide range divider.

allows you to index divisions from 2 to 400,000. The wide range divider (fig. 7-19) consists of a large index plate and a small index plate, both with sector arms and a crank. The large index plate (A, fig. 7-19) has holes drilled on both sides and contains 11 circles of holes on each side of the plate. The number of holes in the circles on one side are 24, 28, 30, 34, 37, 38, 39, 41, 42, 43, and 100. The other side of the plate has circles containing 46, 47, 49, 51, 53, 54, 57, 58, 59, 62, and 66 holes. The small index plate has two circles of holes and is drilled on one side only. The outer circle has 100 holes and the inner circle has 54 holes.

The small index plate (B, fig. 7-19) is mounted on the housing of the planetary gearing (D, fig. 7-19), which is built into the index crank (F, fig. 7-19) of the large plate. As the index crank of the large plate is rotated, the planetary gearing assembly and the small index plate and crank rotate with it.

As with the standard dividing head, the large index crank rotates the spindle in the ratio of 40 to 1. Therefore, one complete turn of the large index crank rotates the dividing head spindle 1/40 of a turn, or 9°. By using the large index plate and the crank, you can index in the conventional manner. Machine operation is the same as it is with the standard dividing head.

When the small index crank (E, fig. 7-19) is rotated, the large index crank remains stationary, but the main shaft that drives the work revolves in the ratio of 1 to 100. This ratio, superimposed on the 40 to 1 ratio between the worm and worm wheel (fig. 7-20), causes the dividing head spindle to rotate in the ratio of 4,000 to 1. This means that one complete revolution of the spindle will require 4,000 turns of the small index crank. Turning the small crank one complete turn will rotate the dividing head spindle 5 minutes, 24 seconds of a degree. If one hole of the 100-hole circle on the small index plate were to

Deleted—No permission granted for electronic copy.

Figure 7-20.—Section through a dividing head showing the worm, worm wheel, and worm shaft.

be indexed, the dividing head spindle would make 1/400,000 of a turn, or 3.24 seconds of a degree.

You can get any whole number of divisions up to and including 60, and hundreds of others, by using only the large index plate and the crank. The dividing head manufacturer provides tables listing many of the settings for specific divisions that you may read directly from the table without further calculations. If the number of divisions required is not listed in the table or if there are no tables, use the manufacturer's manual or other reference for instructions on how to compute the required settings.

ADJUSTING THE SECTOR ARMS

To use the index head sector arms, turn the left-hand arm to the left of the index pin, which is inserted into the first hole in the circle of holes that is to be used. Then, loosen the setscrew (C, fig. 7-19) and adjust the right-hand arm of the sector so that the correct number of holes will be contained between the two arms (fig. 7-21). After making the adjustments, lock the setscrew to hold the arms in position. When setting the arms, count the required number of holes

Deleted—No permission granted for electronic copy.

Figure 7-21.—Principal parts of a late model Cincinnati universal spiral index head.

from the one in which the pin is inserted, considering this hole as zero. Then, use the index sector and you will not need to count the holes for each division. When using the index crank to revolve the spindle, you must unlock the spindle clamp screw. However, before you cut work held in or on the index head, lock the spindle again to relieve the strain on the index pin.

CUTTERS AND ARBORS

When you perform a milling operation, you move the work into a rotating cutter. On most milling machines, the cutter is mounted on an arbor that is driven by the spindle. However, the spindle may drive the cutter directly. We will discuss cutters in the first part of this section and arbors in the second part.

CUTTERS

There are many different milling machine cutters. Some can be used for several operations, while others can be used for only one. Some have straight teeth and others have helical teeth. Some have mounting shanks and others have mounting holes. You must decide which cutter to use. To do so, you must be familiar with the various milling cutters and their uses. The information in this section will help you to select the proper cutter for each of the various operations you will perform. In this section we will cover cutter types and cutter selection.

Standard milling cutters are made in many shapes and sizes for milling both regular and irregular shapes. Various cutters designed for specific purposes also are available; for example, a cutter for milling a particular kind of curve on some intermediate part of the workpiece.

Milling cutters generally take their names from the operation they perform. The most common cutters are (1) plain milling cutters of various widths and diameters, used principally for milling flat surfaces that are parallel to the axis of the cutter; (2) angular milling cutters used to mill V-grooves and the grooves in reamers, taps, and milling cutters; (3) face milling cutters used to mill flat surfaces at a right angle to the axis of the cutter; and (4) forming cutters used to produce surfaces with an irregular outline.

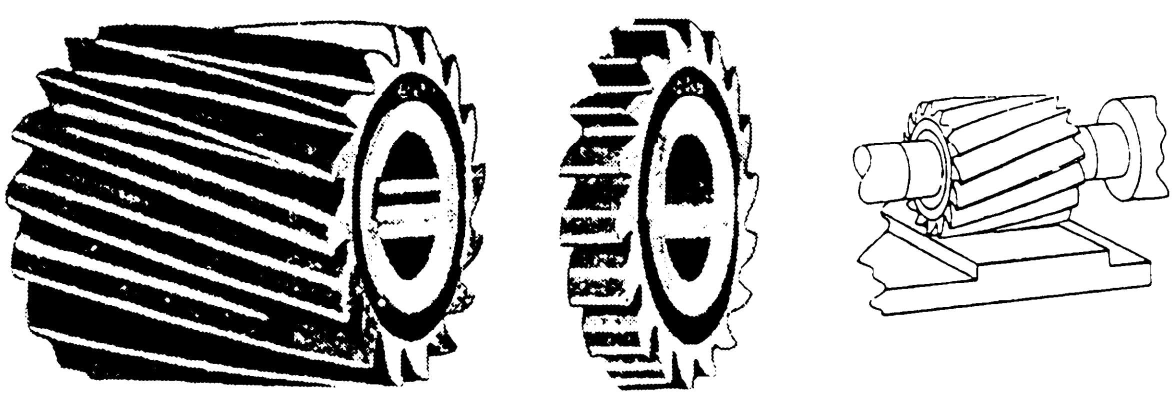
Milling cutters may also be classified as arbor-mounted, or shank-mounted. Arbor-mounted cutters are mounted on the straight shanks of arbors. The arbor is then inserted into the milling machine spindle. We'll discuss the methods of mounting arbors and cutters in greater detail later in this chapter.

Milling cutters may have straight, right-hand, left-hand, or staggered teeth. Straight teeth are parallel to the axis of the cutter. If the helix angle twists in a clockwise direction (viewed from either end), the cutter has right-hand teeth. If the helix angle twists in a counterclockwise direction, the cutter has left-hand teeth. The teeth on staggered-tooth cutters are alternately left-hand and right-hand.

Types and Uses

There are many different types of milling cutters. We will discuss these types and their uses in the following sections.

PLAIN MILLING CUTTER.—You will use plain milling cutters to mill flat surfaces that are parallel to the cutter axis. As you can see in figure 7-22, a plain milling cutter is a cylinder with teeth cut on the circumference only. Plain milling cutters are made in a variety of diameters and widths.



28.372

Figure 7-22.—Plain milling cutters.

Note in figure 7-23, that the cutter teeth may be either straight or helical. When the width is more than 3/4 inch, the teeth are usually helical. The teeth of a straight cutter tool are parallel to axis of the cutter. This causes each tooth to cut along its entire width at the same time, causing a shock as the tooth starts to cut. Helical teeth eliminate this shock and produce a free cutting action. A helical tooth begins the cut at one end and continues across the work with a smooth shaving action. Plain milling cutters usually have radial teeth. On some coarse helical tooth cutters the tooth face is undercut to produce a smoother cutting action. Coarse teeth decrease the tendency of the arbor to spring and give the cutter greater strength.

A plain milling cutter has a standard size arbor hole for mounting on a standard size arbor. The size of the cutter is designated by the diameter and width of the cutter, and the diameter of the arbor hole in the cutter.

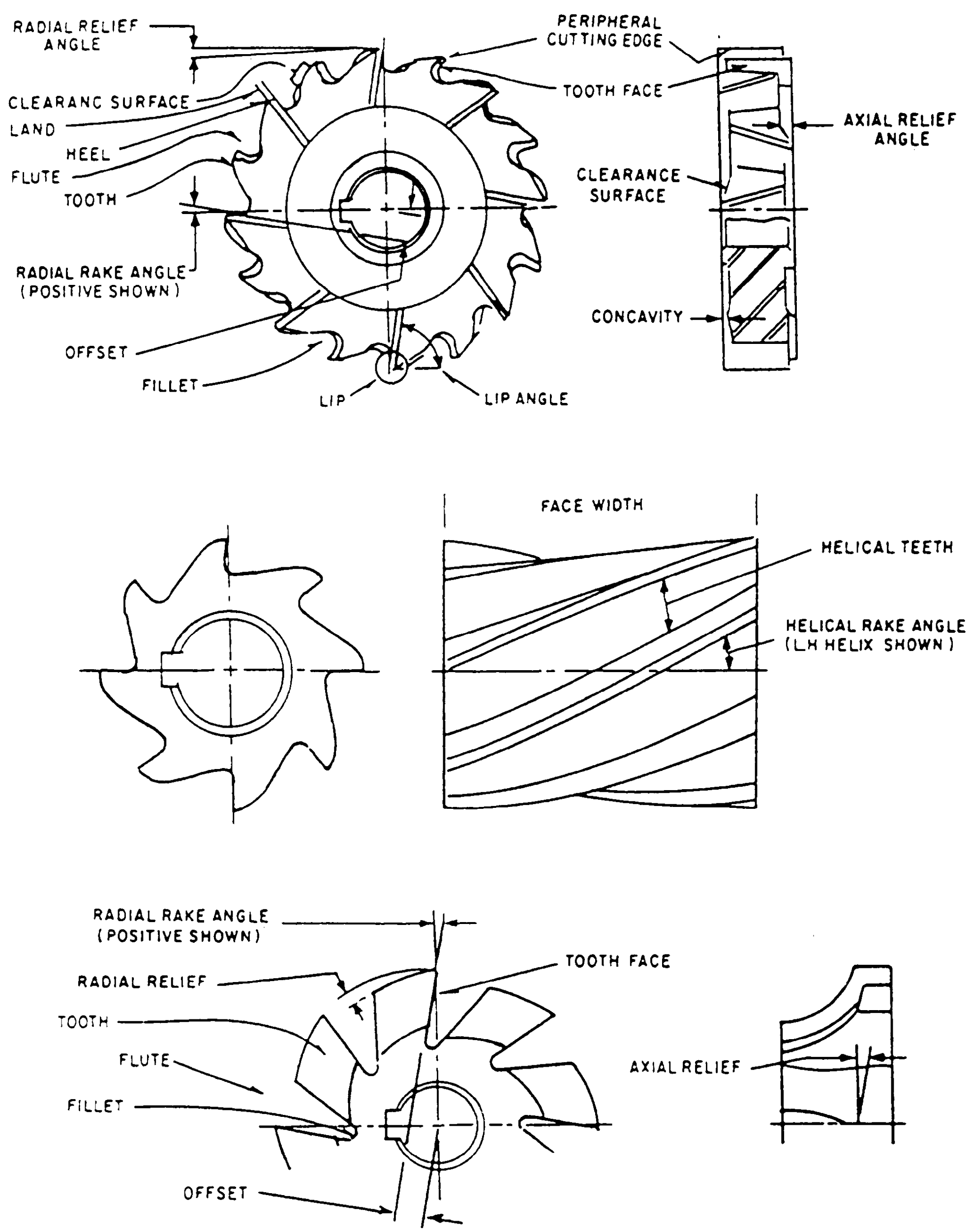


Figure 7-23.—Milling cutter terms.

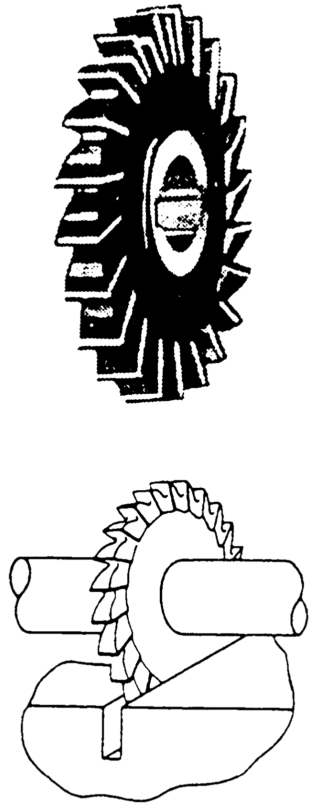


Figure 7-24.—Side milling cutter.

SIDE MILLING CUTTER.—The side milling cutter (fig. 7-24) is a plain milling cutter with teeth cut on both sides as well as on the periphery or circumference of the cutter. You can see that the portion of the cutter between the hub and the side of the teeth is thinner to give more chip clearance. These cutters are often used in pairs to mill parallel sides. This process is called straddle milling. Cutters more than 8 inches in diameter are usually made with inserted teeth. The size designation is the same as for plain milling cutters.

HALF-SIDE MILLING CUTTER.—Half-side milling cutters (fig. 7-25) are made particularly for jobs where only one side of the cutter is needed. These cutters have coarse, helical teeth on one side only so that heavy cuts can be made with ease.

SIDE MILLING CUTTER (INTERLOCKING).—Side milling cutters whose teeth interlock (fig. 7-26) can be used to mill standard size slots. The width is regulated by thin washers inserted between the cutters.

METAL SLITTING SAW.—You can use a metal slitting saw to cut off work or to mill narrow slots. A metal slitting saw is similar to a plain or side milling cutter, with a face width usually less than 3/16 inch. This type of cutter usually has more teeth for a given diameter than a plain cutter. It is thinner at the center than at the outer edge to give proper clearance for milling deep slots. Figure 7-27 shows a metal slitting saw with teeth cut in the circumference of the cutter only. Some saws, such as the one in figure 7-28, have side teeth that achieve better cutting action, break up chips, and prevent dragging when you cut deep slots. For heavy sawing in steel, there are metal slitting saws with staggered teeth, as shown in figure 7-29. These cutters are usually 3/16 inch to 3/8 inch thick

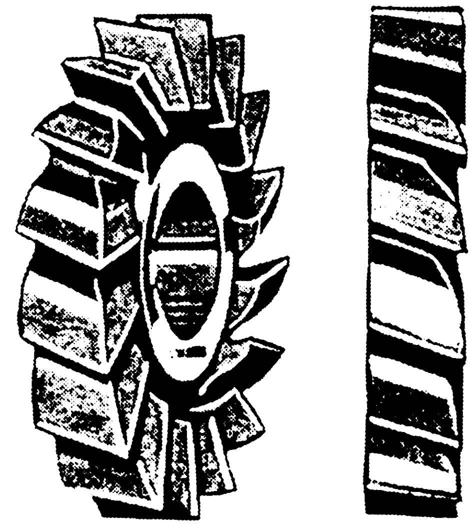


Figure 7-25.—Half-side milling cutter.

SCREW SLOTTING CUTTER.—The screw slotting cutter (fig. 7-30) is used to cut shallow slots, such as those in screwheads. This cutter has fine teeth cut on its circumference. It is made in various thicknesses to correspond to American Standard gauge wire numbers.

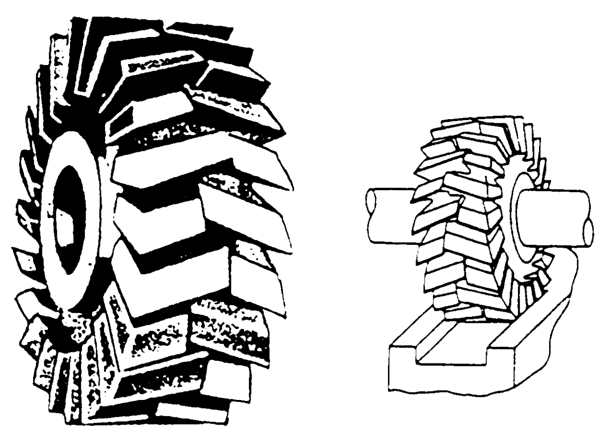


Figure 7-26.—Interlocking teeth side milling cutter.

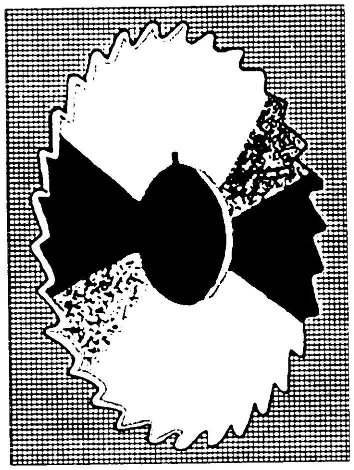


Figure 7-27.—Metal slitting saw.

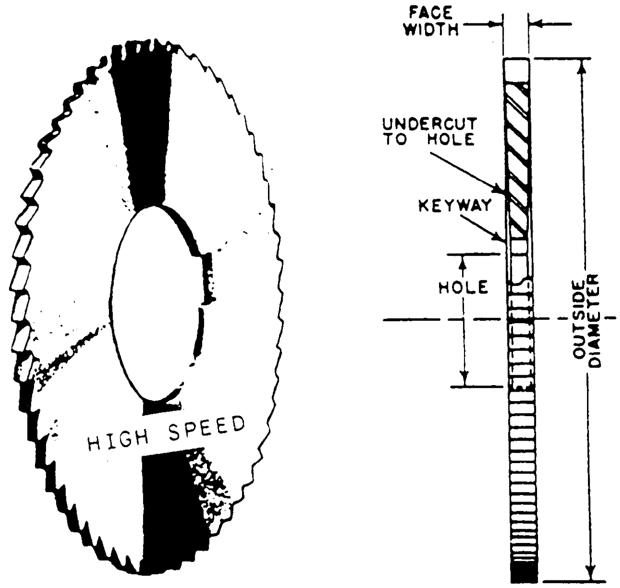


Figure 7-30.—Screw slotting cutter.

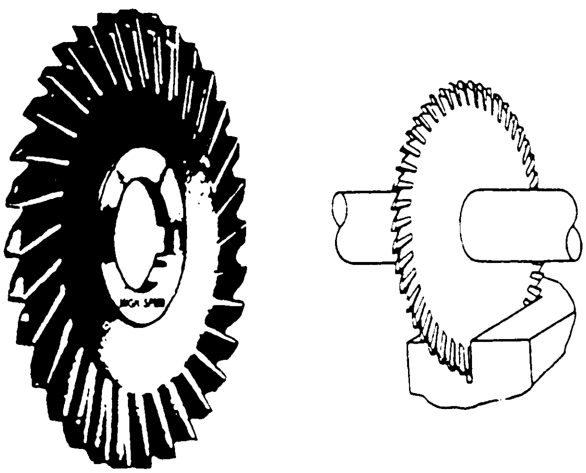


Figure 7-28.—Slitting saw with side teeth.

ANGLE CUTTER.—Angle cutters are used to mill surfaces that are not at a right angle to the cutter axis. You can use angle cutters for a variety of work, such as milling V-grooves and dovetail ways. On work such as dovetailing, where you cannot mount a cutter in the usual manner on an arbor, you can mount an angle cutter that has a threaded hole, or is constructed like a shell end mill, on the end of a stub or shell end mill arbor. When you select an angle cutter for a job, you should specify the type, hand, outside diameter, thickness, hole size, and angle.

There are two types of angle cutters-single and double. The single-angle cutter, shown in figure 7-31, has teeth cut at an oblique angle with one side at an

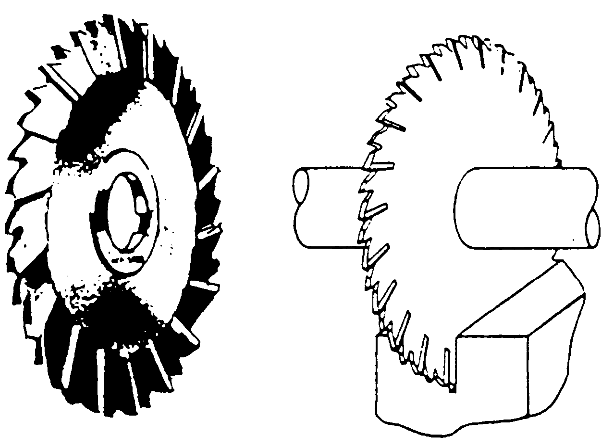


Figure 7-29.—Slitting saw with staggered teeth.

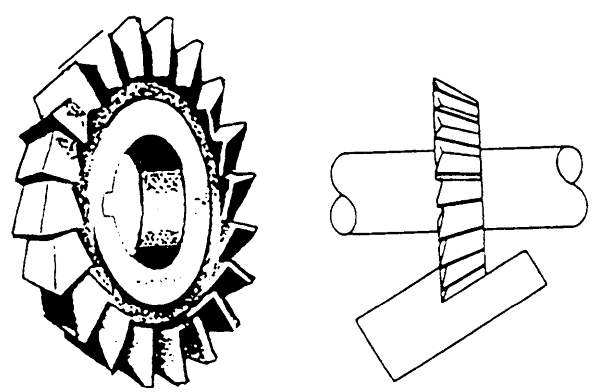


Figure 7-31.—Single-angle cutter.

angle of 90° to the cutter axis and the other usually at 45°, 50°, or 80°.

The double-angle cutter (fig. 7-32) has two cutting faces, which are at an angle to the cutter axis. When both faces are at the same angle to the axis, you obtain the cutter you want by specifying the included angle. When they are different angles, you specify the angle of each side with respect to the plane of intersection.

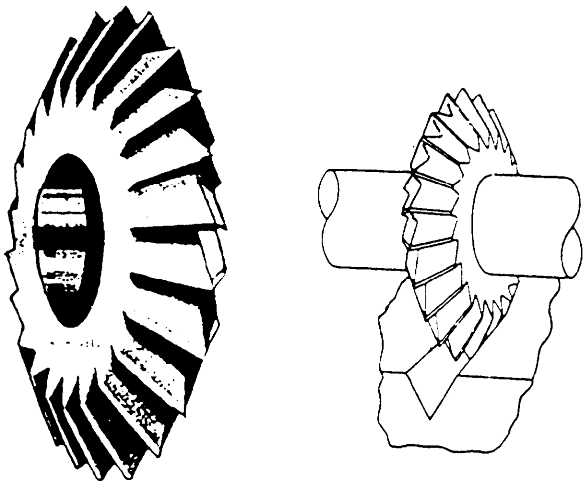
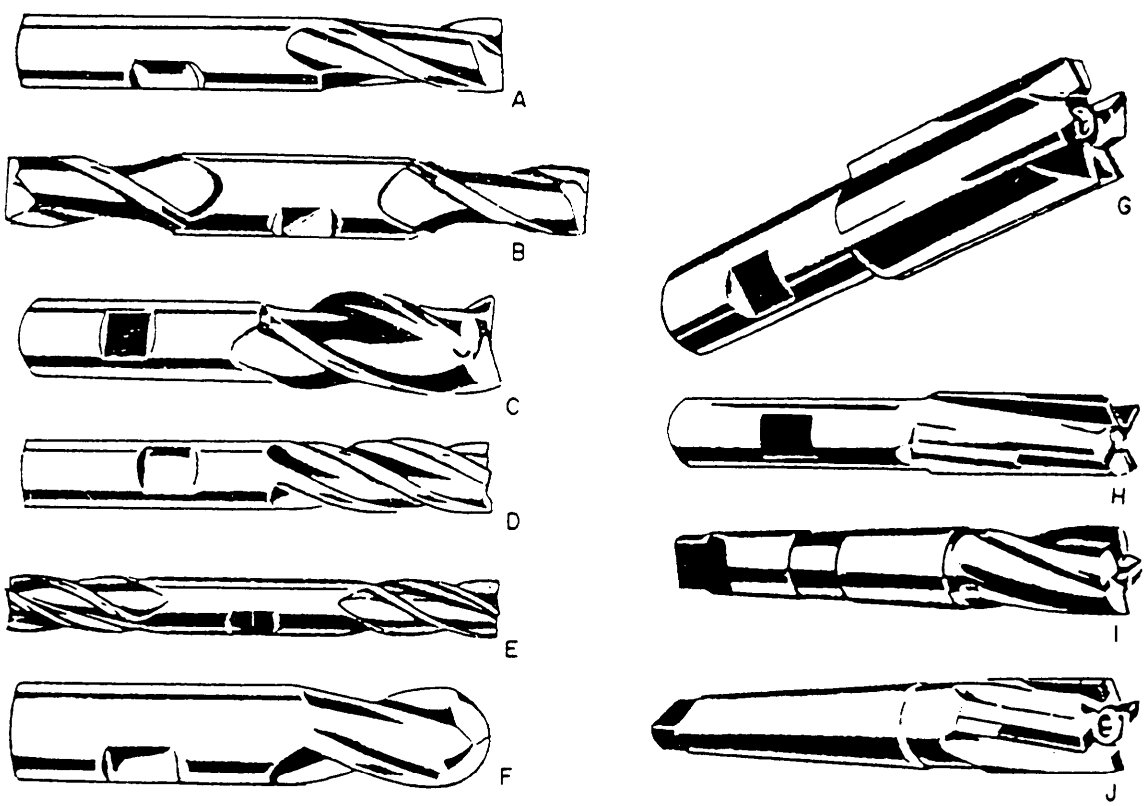


Figure 7-32.—Double-angle cutter.

END MILL CUTTERS.—End mill cutters may be the SOLID TYPE with the teeth and the shank as an integral part (fig. 7-33), or they may be the SHELL TYPE (fig. 7-34) in which the cutter body and the shank or arbor are separate. End mill cutters have teeth on the circumference and on the end. Those on the circumference may be either straight or helical (fig. 7-35).



A. Two-flute single-end
B. Two-flute double-end
C. Three-flute single-end
D. Multiple-flute singleend
E. Four-flute double-end
F. Two-flute ball-end
G. Carbide-tipped, straight flutes
H. Carbide-tipped, right-hand helical flutes
I. Multiple-flute with taper shank
J. Carbide-tipped with taper shank and helical flutes

Figure 7-33.—End mill cutters.

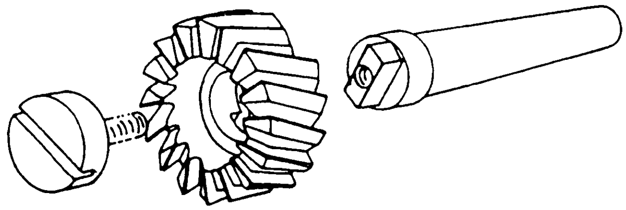


Figure 7-34.—Shell end mill.

Except for the shell type, all end mills have either a straight shank or a tapered shank is mounted into the spindle of the machine to drive the cutter. There are various types of adapters used to secure end mills to the machine spindle.

End milling involves the machining of surfaces (horizontal, vertical, angular, or irregular) with end

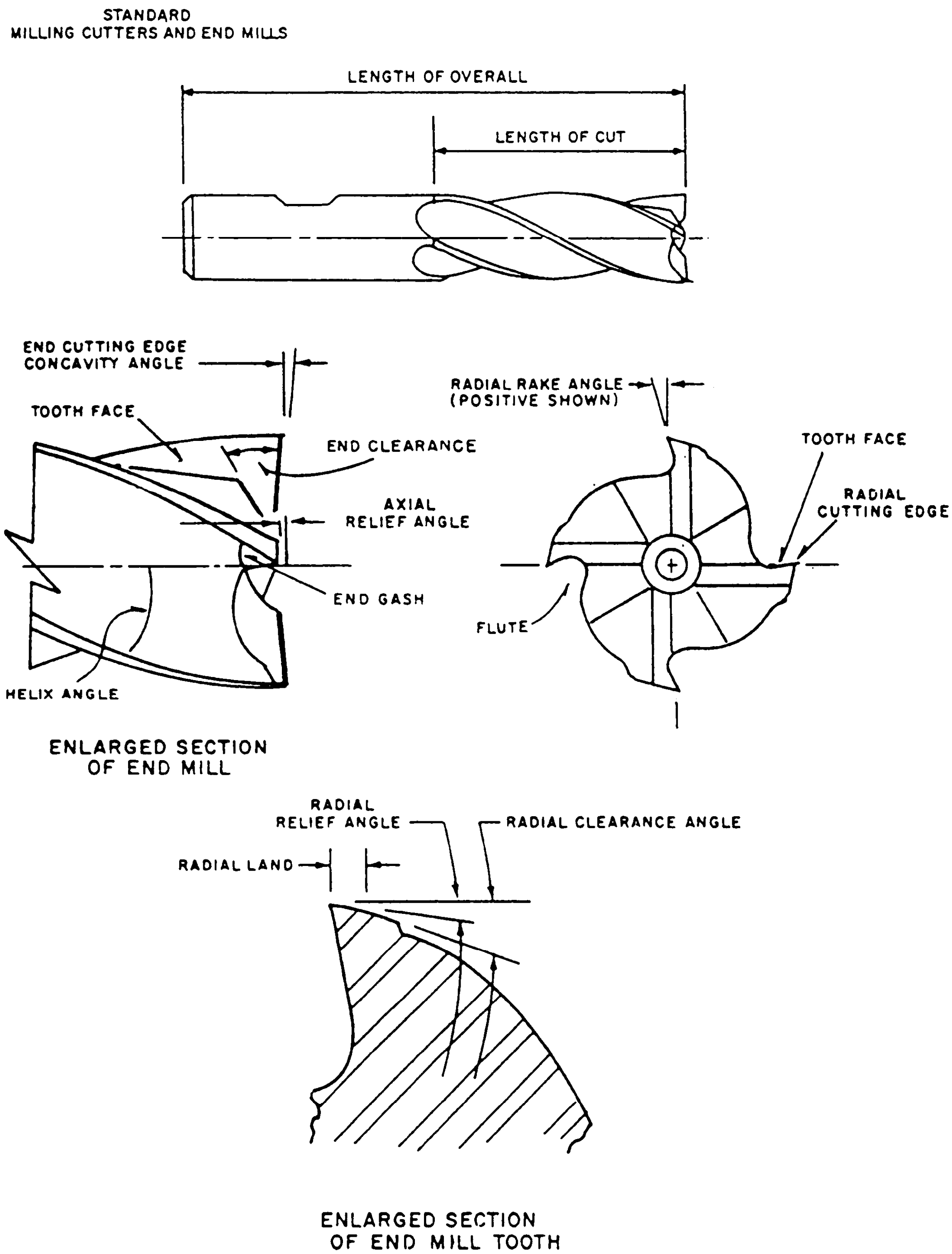


Figure 7-35.—End mill terms.

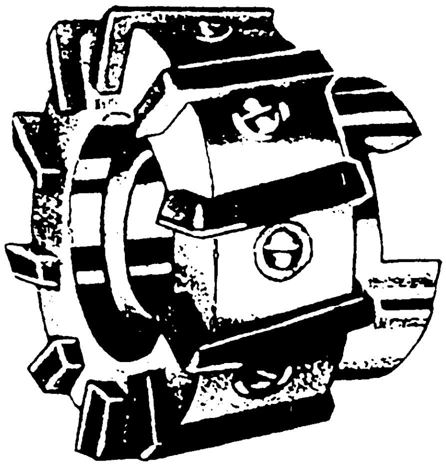


Figure 7-36.—Inserted tooth face milling cutter.

mill cutters. Common operations include the milling of slots, keyways, pockets, shoulders, and flat surfaces, and the profiling of narrow surfaces.

End mill cutters are used most often on vertical milling machines. However, they also are used frequently on machines with horizontal spindles. Many different types of end mill cutters are available in sizes ranging from 1/64 inch to 2 inches. They may be made of high-speed steel, have cemented carbide teeth, or be of the solid carbide type.

TWO-FLUTE END MILLS have only two teeth on their circumference. The end teeth can cut to the center. Hence, they may be fed into the work like a drill; they can then be fed lengthwise to form a slot.

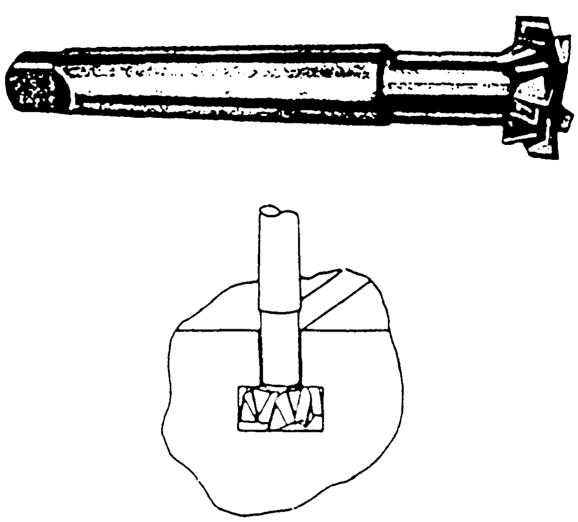


Figure 7-37.—T-slot cutter.

These mills may be either the single-end type with the cutter on one end only, or they may be the double-end type. (See fig. 7-33.)

MULTIPLE-FLUTE END MILLS have three, four, six, or eight flutes and normally are available in diameters up to 2 inches. They may be either the single-end or the double-end type (fig. 7-33).

BALL END MILLS (fig. 7-33) are used to mill fillets or slots with a radius bottom, to round pockets and the bottom of holes, and for all-around die sinking and die making work. Two-flute end mills with end cutting lips can be used to drill the initial hole as well as to feed longitudinally. Four-flute ball end mills with center cutting lips also are available. These work well for tracer milling, fillet milling, and die sinking.

SHELL END MILLS (fig. 7-34) have a hole used to mount the cutter on a short (stub) arbor. The center of the shell is recessed for the screw or nut that fastens the cutter to the arbor. These mills are made in larger sizes than solid end mills, normally in diameters from 1/4 to 6 inches. Cutters of this type are intended for slabbing or surfacing cuts, either face milling or end milling, and usually have helical teeth.

FACE MILLING CUTTER.—Inserted tooth face milling cutters (fig. 7-36) are similar to shell end mills in that they have teeth on the circumference and on the end. They are attached directly to the spindle nose and use inserted, replaceable teeth made of carbide or any alloy steel.

T-SLOT CUTTER.—The T-slot cutter (fig. 7-37) is a small plain milling cutter with a shank. It is

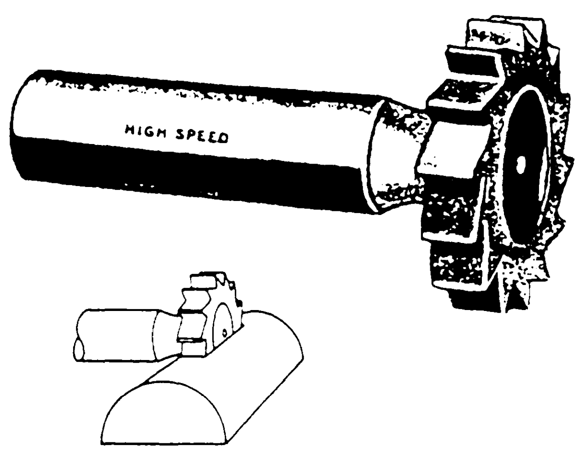


Figure 7-38.—Woodruff keyseat cutter.

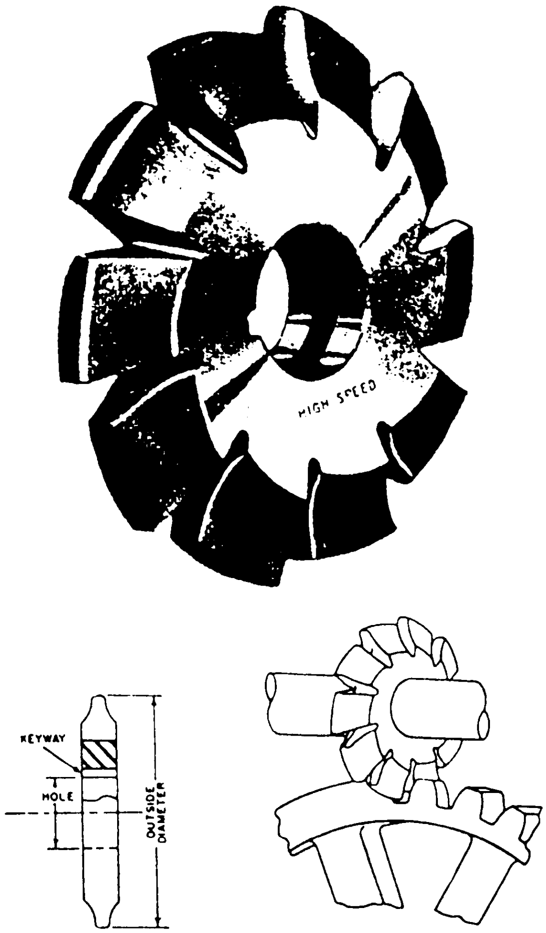


Figure 7-39.—Involute gear cutter.

designed especially to mill the "head space" of T-slots. T-slots are cut in two operations. First, you cut a slot with an end mill or a plain milling cutter, and then you make the cut at the bottom of the slot with a T-slot cutter.

Figure 7-40.—Concave cutter.

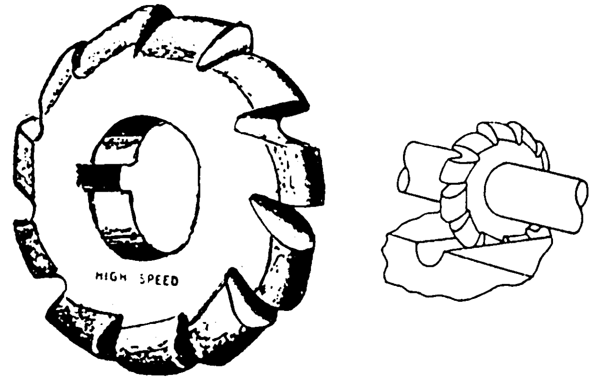


Figure 7-41.—Convex cutter.

WOODRUFF KEYSEAT CUTTER.—A Woodruff keyseat cutter (fig. 7-38) is used to cut curved keyseats. A cutter less than 1/2 inch in diameter has a shank. When the diameter is greater than 1/2 inch, the cutter is usually mounted on an arbor. The larger cutters have staggered teeth to improve the cutting action.

GEAR CUTTERS.—There are several types of gear cutters, such as bevel, spur, involute, and so on. Figure 7-39 shows an involute gear cutter. You must select the correct cutter for a particular type of gear.

CONCAVE AND CONVEX CUTTERS.—A concave cutter (fig. 7-40) is used to mill a convex surface, and a convex cutter (fig. 7-41) is used to mill a concave surface.

CORNER ROUNDING CUTTER.—Corner rounding cutters (fig. 7-42) are formed cutters that are used to round corners up to one-quarter of a circle.

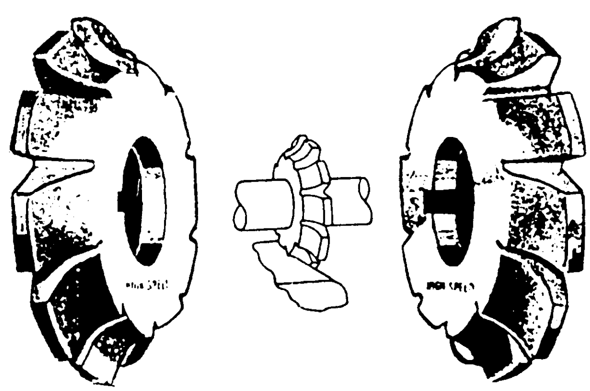


Figure 7-42.—Corner rounding cutter.

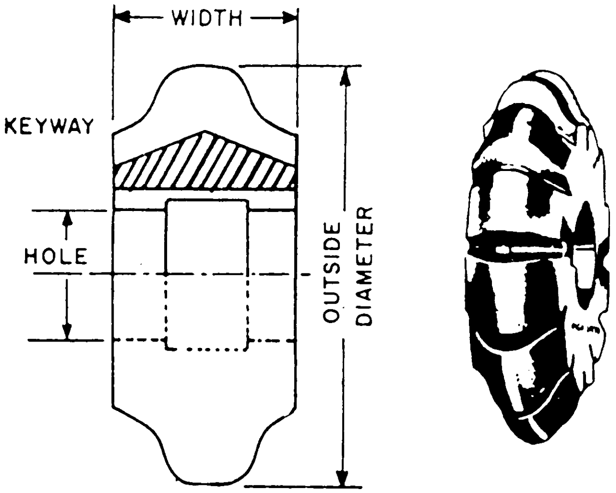


Figure 7-43.—Sprocket wheel cutter.

SPROCKET WHEEL CUTTER.—The sprocket wheel cutter (fig. 7-43) is a formed cutter that is used to mill teeth on sprocket wheels.

GEAR HOB.—The gear hob (fig. 7-44) is a formed milling cutter with teeth cut like threads on a screw.

FLY CUTTER.—The fly cutter (fig. 7-45) is often manufactured locally. It is a single-point cutting tool similar in shape to a lathe or shaper tool. It is held and rotated by a fly cutter arbor. There will be times when you need a special formed cutter for a very limited number of cutting or boring operations. This will probably be the type of cutter you will use since you can grind it to almost any form you need.

We have discussed a number of the more common types of milling machine cutters. For a more detailed discussion of these, other types, and their uses, consult the *Machinery's Handbook*, machinist publications, or the applicable technical manual. We will now discuss the selection of cutters.

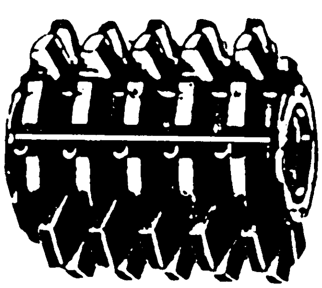


Figure 7-44.—Gear hob.

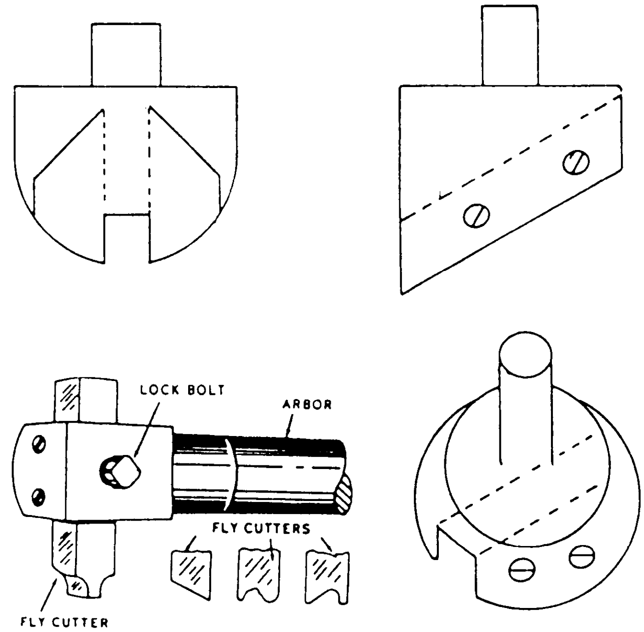


Figure 7-45.—Fly cutter arbor and fly cutters.

Selection

Each cutter can do one kind of job better than any other cutter in a given situation. A cutter may or may not be limited to a specific milling operation. To select the most suitable cutter for a particular operation, you must consider the kind of cut to be made, the material to be cut, the number of parts to be machined, and the type of milling machine available.

Another factor that affects a milling operation is the number of teeth in the cutter. If there are too many teeth, the space between them is so small that it prevents the free flow of chips. The chip space should also be smooth and free of sharp corners to prevent the chips from clogging the space. A coarse-tooth cutter is more satisfactory for milling material that produces a continuous and curled chip. The coarse teeth not only permit an easier flow of chips and coolant but also help to eliminate chatter. A fine-tooth cutter is more satisfactory for milling a thin material. It reduces cutter and workpiece vibration and the tendency for the cutter teeth to "straddle" the work and dig in.

Another factor you should consider in selecting a cutter is its diameter. Select the smallest diameter cutter that will allow the arbor to pass over the work without interference when you take the cut. Figure 7-46 shows that a small cutter takes a cut in less time than a larger cutter.

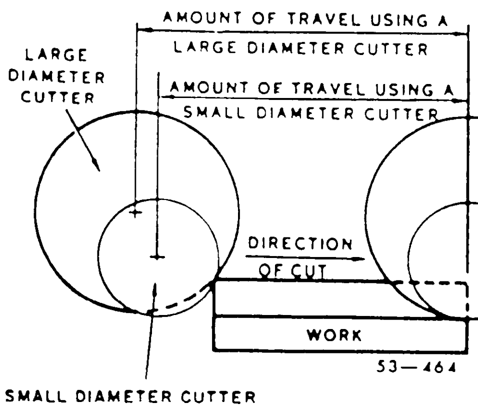


Figure 7-46.—Cutter diameter selection.

ARBORS

You can mount milling machine cutters on several types of holding devices. You must know the devices and the purpose of each of them to make the most suitable tooling setup for the operation. We will cover the various types of arbors and the mounting and dismounting of arbors in this section.

NOTE: Technically, an arbor is a shaft on which a cutter is mounted. For convenience, since there are so few types of cutter holders that are not arbors, we will refer to all types of cutter holding devices as arbors.

Standard Arbor

There are several types of milling machine arbors. You will use the common or standard types (fig. 7-47) to hold and drive cutters that have mounting holes. One end of the arbor usually has a standard milling machine spindle taper of 3 1/2 inches per foot. The largest diameter of the taper is identified by a number. For example, the large diameter of a No. 40 milling machine spindle taper is 1 3/4 inches. The following numbers represent common milling machine spindle tapers and their sizes:

Number	Large Diameter
10	5/8 inch
20	7/8 inch
30	1/4 inch
40	1 3/4 inches
50	2 3/4 inches
60	4 1/4 inches

Standard arbors are available in styles A and B, as shown in figure 7-47. Style A arbors have a pilot-type bearing usually 1 1/32 inch in diameter. Style B arbors have a sleeve-type outboard bearing. Numerals identify the outside diameter of the bearing sleeves, as follows:

Sleeve Number	Outside Diameter
3	1 7/8 inches
4	2 1/8 inches
5	2 3/4 inches

The inside diameter can be any one of several standard diameters that are used for the arbor shaft.

Style A arbors sometimes have a sleeve bearing that permits the arbor to be used as either a style A or a style B arbor. A code system, consisting of numerals and a letter, identifies the size and style of the arbor. The code number is stamped into the flange

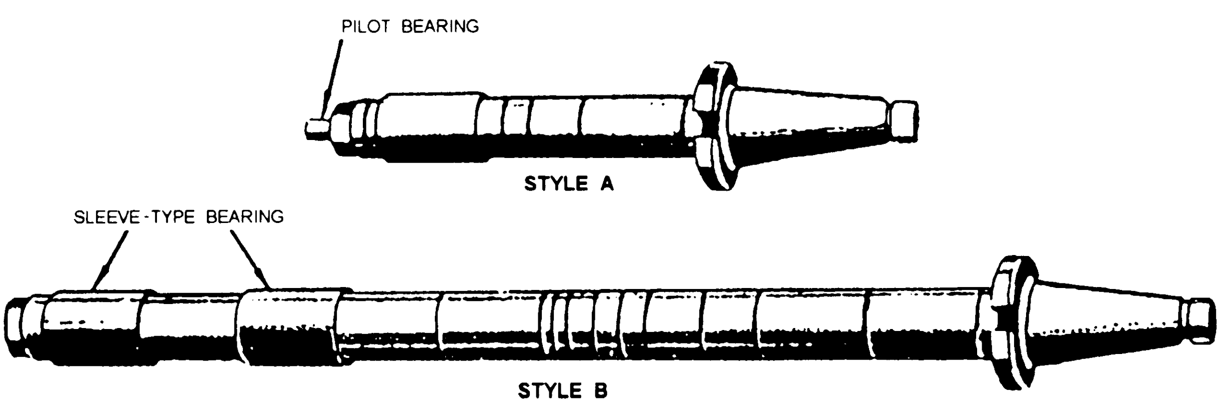


Figure 7-47.—Standard milling machine arbors.

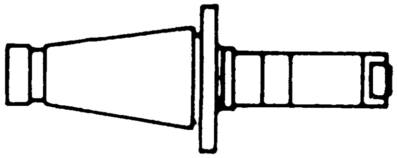


Figure 7-48.—Stub arbor.

or on the tapered portion of the arbor. The first number of the code identifies the diameter of the taper. The second (and if used, the third number) identifies the diameter of the arbor shaft. The letter identifies the type of bearing. The numbers following the letter identifies the usable length of the arbor shaft. Sometimes an additional number is used to identify the size of the sleeve-type bearings. The meaning of a typical code number 5-1¼-A-18-4 is as follows:

5 = taper number—50 (the 0 is omitted in the code)
1¼ = shaft diameter—1¼ inches
A = style A bearing—pilot type
18 = usable shaft length—18 inches
4 = bearing size—2 1/8 inches diameter

Stub Arbor

Arbors that have very short shafts, such as the one shown in figure 7-48, are called stub arbors. Use stub arbors when it is impractical to use a longer arbor.

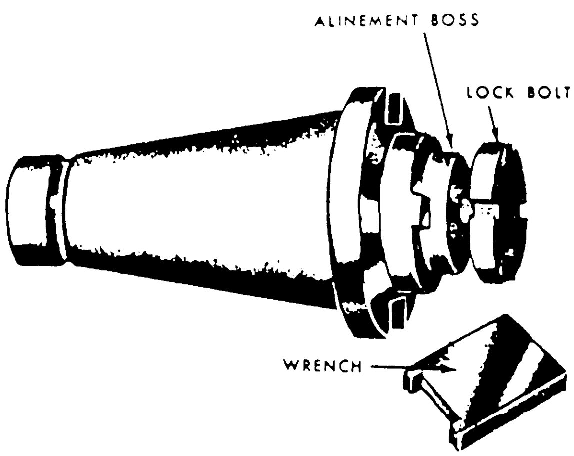


Figure 7-49.—Shell end mill arbor.

You will use arbor spacing collars of various lengths to position and secure the cutter on the arbor. Tighten the spacers against the cutter when you tighten the nut on the arbor. Remember, never tighten or loosen the arbor nut unless the arbor support is in place.

Shell End Arbor

Shell end mill arbors (fig. 749) are used to hold and drive shell end mills. The shell end mill is fitted over the short boss on the arbor shaft. It is driven by two keys and is held against the face of the arbor by a bolt. Use a special wrench, shown in figure 7-48, to tighten and loosen the bolt. Shell end mill arbors are identified by a code similar to the standard arbor code. The letter *C* identifies a shell end mill arbor. A typical shell mill arbor code 4-1½-C-7/8 is identified as follows:

4 = taper code number—40
1½ = diameter of mounting hole in end mill—1½ inches
C = style C arbor—shell end mill
7/8 = length of shaft—7/8 inch

Fly Cutter Arbor

Fly cutter arbors are used to hold single-point cutters. These cutters (fig. 7-45) can be ground to any desired shape and held in the arbor by a locknut. Fly cutter arbor shanks may have a standard milling machine spindle taper, a Brown and Sharpe taper, or a Morse taper.

Screw Slotting Cutter Arbor

Screw slotting cutter arbors are used with screw slotting cutters. The flanges support the cutter and prevent it from flexing. The shanks on screw slotting cutter arbors may be straight or tapered, as shown in figure 7-50.

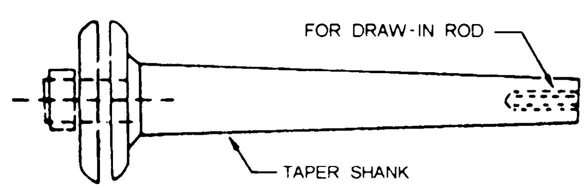


Figure 7-50.—Strew slotting cutter arbor.

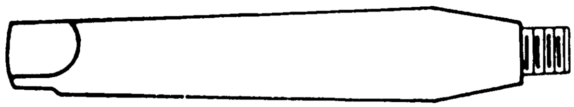


Figure 7-51.—Screw arbor.

Screw Arbor

Screw arbors (fig. 7-51) are used with cutters that have threaded mounting holes. The threads may be left- or right-hand.

Taper Adapter

Taper adapters are used to hold and drive taper-shanked tools, such as drills, drill chucks, reamers, and end mills. You insert the tool into the tapered hole in the adapter. The code for a taper adapter includes the number representing the standard milling machine spindle taper and the number and series of the internal taper. For example, the taper adapter code number 43M means:

4 = taper identification number—40
3M = internal taper—number 3 Morse

If a letter is not included in the code number, the taper is understood to be a Brown and Sharpe. For example, 57 means:

5 = taper number—50
7 = internal taper—number 7 B and S

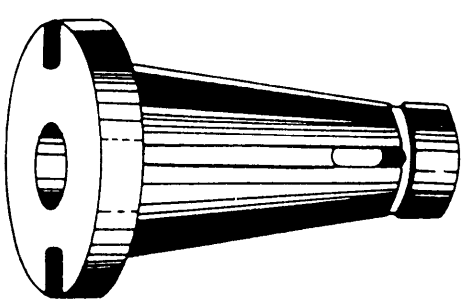


Figure 7-52.—Taper adapter.

and 50-10 means:

50 = taper identification number
10 = internal taper—number 10 B and S

Figure 7-52 shows a typical taper adapter. Some cutter adapters are designed to be used with tools that have taper shanks and a cam locking feature. The cam lock adapter code indicates the number of the external taper, number of the internal taper (which is usually a standard milling machine spindle taper), and the distance that the adapter extends from the spindle of the machine. For example, 50-20-3 5/8 inches means:

50 = taper identification number (external)
20 = taper identification number (internal)
3 5/8 = distance adapter extends from spindle is 3 5/8 inches

Cutter Adapter

Cutter adapters, such as the one shown in figure 7-53, are similar to taper adapters except they always have straight, rather than tapered, holes. They are used to hold straight shank drills, end mills, and so forth. The cutting tool is secured in the adapter by a setscrew. The code number indicates the number of the taper and the diameter of the hole. For example, 50-5/8 means the adapter has a number 50 taper and a 5/8-inch-diameter hole.

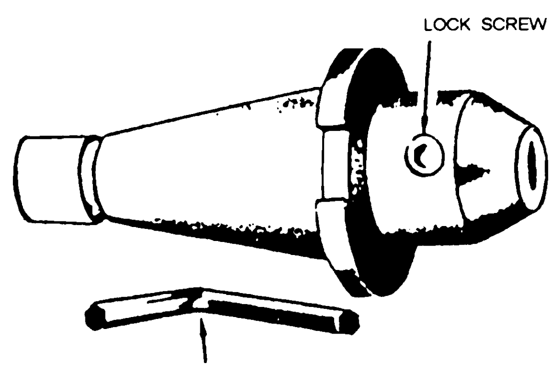


Figure 7-53.—Cutter adapter.

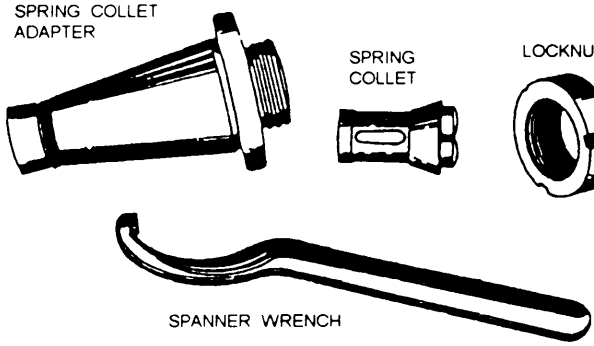


Figure 7-54.—Spring collet chuck adapter.

Spring Collet Chuck

Spring collet chucks (fig. 7-54) are used to hold and drive straight-shanked tools. The spring collet chuck consists of a collet adapter, spring collets, and a cup nut. Spring collets are similar to lathe collets. The cup forces the collet into the mating taper, causing the collet to close on the straight shank of the tool. The collets are available in several fractional sizes.

MILLING MACHINE OPERATIONS

The milling machine is one of the most versatile metalworking machines. It can be used for simple operations, such as milling a flat surface or drilling a hole, or more complex operations, such as milling helical gear teeth. It would be impractical to try to discuss all of its operations. Therefore, we'll limit our discussion to plain, face, and angular milling; milling flat surfaces on cylindrical work, slotting, parting, and milling keyseats and flutes; and drilling, reaming, and boring. Even though we will discuss only the more common operations, you will find that by using a combination of operations, you will be able to produce a variety of work projects.

PLAIN MILLING

Plain milling is the process of milling a flat surface in a plane parallel to the cutter axis. You get the work to its required size by individually milling each of the flat surfaces on the workpiece. You'll use plain milling cutters such as those shown in figure 7-22. If possible, select a cutter that is slightly wider than the width of the surface to be milled. Make the work setup before you mount the cutter; this may prevent cuts on your hands caused by striking the cutter. You can mount the work in a vise or fixture, or clamp it directly to the milling machine table. You can use the same methods that you used to hold work in a shaper. Clamp the work as closely as possible to the milling machine column so you can mount the cutter near the column. The closer you place the cutter and the work to the column, the more rigid the setup will be.

The following steps explain how to machine a rectangular work blank (for example, a spacer for an engine test stand):

1. Mount the vise on the table and position the vise jaws parallel to the table length.

NOTE: The graduations on the vise are accurate enough because we are concerned only with machining a surface in a horizontal plane.

2. Place the work in the vise, as shown in view A, figure 7-55.
3. Select the proper milling cutter and arbor.
4. Wipe off the tapered shank of the arbor and the tapered hole in the spindle with a clean cloth.
5. Mount the arbor in the spindle.
6. Clean and position the spacing collars and place them on the arbor so that the cutter is above the work.
7. Wipe off the milling cutter and any additional spacing collars that may be needed. Then, place the cutter, the spacers, and the arbor bearing on the arbor, with the cutter keyseat aligned over the key. Locate the bearing as

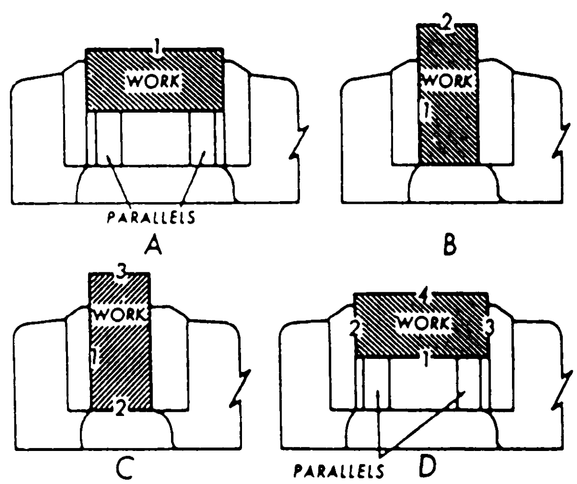


Figure 7-55.—Machining sequence to square a block.

closely as possible to the cutter. Make sure that the work and the vise will clear all parts of the machine.

8. Install the arbor nut and tighten it finger tight only.
9. Position the overarm and mount the arbor support.
10. After supporting the arbor, tighten the arbor nut with a wrench.
11. Set the spindle directional control lever to give the required direction of cutter rotation.
12. Determine the required speed and feed, and set the spindle speed and feed controls.
13. Set the feed trip dogs for the desired length of cut and center the work under the cutter.
14. Lock the saddle.
15. Engage the spindle clutch and pick up the cut.
16. Pick up the surface of the work by holding a long strip of paper between the rotating cutter and the work; very slowly move the work toward the cutter until the paper strip is pulled between the cutter and the work. Keep your fingers away from the cutter. A rotating milling cutter is very dangerous.
17. Move the work longitudinally away from the cutter and set the vertical feed graduated collar at ZERO.
18. Compute the depth of the roughing cut and raise the knee this distance.
19. Lock the knee, and direct the coolant flow on the work and on the outgoing side of the cutter.
20. Position the cutter to within 1/16 inch of the work, using hand table feed.
21. Engage the power feed.
22. After completing the cut, stop the spindle.
23. Return the work to its starting point on the other side of the cutter.
24. Raise the table the distance required for the finish cut.
25. Set the finishing speed and feed, and take the finish cut.
26. When you have completed the operation, stop the spindle and return the work to the opposite side of the cutter.
27. Deburr the work and remove it from the vise.

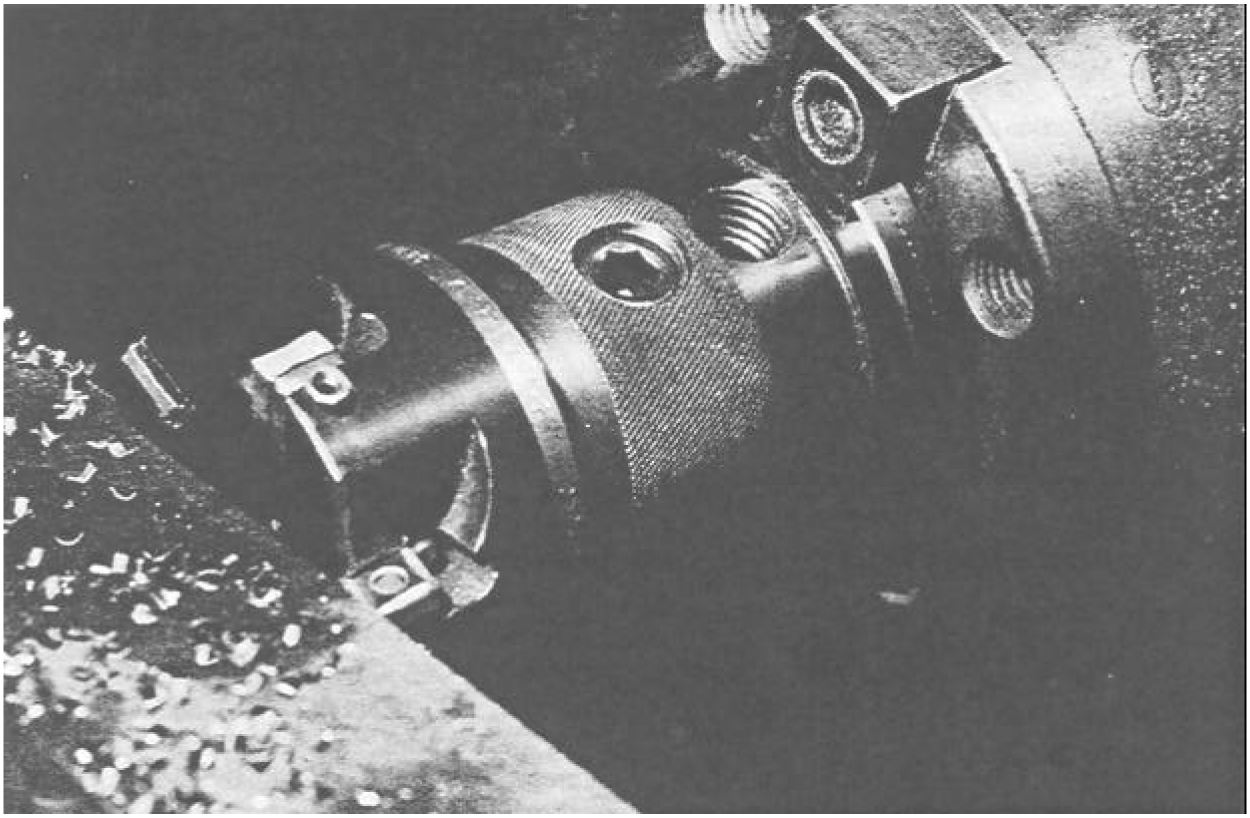
To machine the second side, place the work in the vise as shown in figure 7-55, view B. Rough and finish machine side 2, using the same procedures that you used for side 1. When you have completed side 2, deburr the surface and remove the work from the vise.

Place the work in the vise, as shown in figure 7-55, view C, with side 3 up. Then, rough machine side 3. Finish machine side 3 for a short distance, disengage the spindle and feed, and return the work to the starting point, clear of the cutter. Now you can safely measure the distance between sides 2 and 3. If this distance is correct, you can continue the cut with the same setting. If it is not, adjust the depth of cut as necessary. If the trial finishing cut is not deep enough, raise the work slightly and take another trial cut. If the trial cut is too deep, you will have to remove the backlash from the vertical feed before taking the new depth of cut. Use the following procedure to remove the backlash:

1. Lower the knee well past the original depth of the roughing cut.
2. Raise the knee the correct distance for the finishing cut.
3. Engage the feed and complete your cut.
4. Stop the spindle.
5. Return the work to the starting point on the other side of the cutter.
6. Deburr the work.
7. Remove the work from the vise.

Place side 4 in the vise, as shown in figure 7-55, view D, and machine the side, using the same procedure as for side 3. When you have completed side 4, remove the work from the vise and check it for accuracy.

This completes the machining of the four sides of the block. If the block is not too long, you can rough and finish mill the ends to size in the same manner in which you milled the sides. Do this by placing the block on end in the vise. You also may use face milling to machine the ends.



28.402

Figure 7-56.—Face milling.

FACE MILLING

Face milling is the milling of surfaces that are perpendicular to the cutter axis, as shown in figure 7-56. Use this method to produce flat surfaces and to machine work to the required length. In face milling, the feed can be either horizontal or vertical.

Cutter Setup

You can use straight-shank or taper-shank end mills, shell end mills, or face milling cutters for face milling. Select a cutter that is slightly larger in diameter than the thickness of the material you are machining. If the cutter is smaller in diameter than the thickness of the material, you will be forced to make a series of slightly overlapping cuts to machine the entire surface. Mount the arbor and the cutter before you make the work setup. Mount the cutter by any means suitable for the cutter you selected.

Work Setup

Use any suitable means to hold the work for face milling as long as the cutter clears the workholding device and the milling machine table. You can mount the work on parallels, if necessary, to provide clearance between the cutter and the table. Feed the work from the side of the cutter that will cause the cutter thrust to force the work down. If you hold the work in a vise, position the vise so the cutter thrust is toward the solid jaw. The ends of the work are usually machined square to the sides of the work; therefore, you'll have to align the work properly. If you use a vise to hold the work, you can align the stationary vise jaw with a dial indicator, as shown in figure 7-57. You can also use a machinist's square and a feeler gauge, as shown in figure 7-58.

Operation

Use the following procedure to face mill the ends of work:

1. Select and mount a suitable cutter.

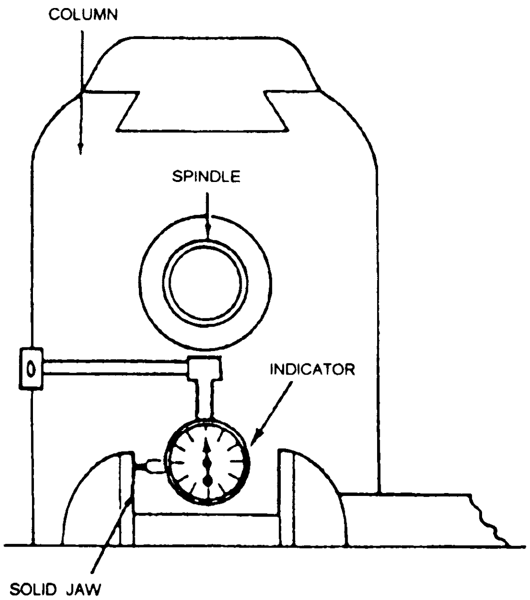


Figure 7-57.—Aligning vise jaws using an indicator.

2. Mount and position a vise on the milling machine table so the thrust of the cutter is toward the solid vise jaw.

3. Align the solid vise jaw square with the column of the machine, using a dial indicator for accuracy.

4. Mount the work in the vise, allowing the end of the work to extend slightly beyond the vise jaws.

5. Raise the knee until the center of the work is approximately even with the center of the cutter.

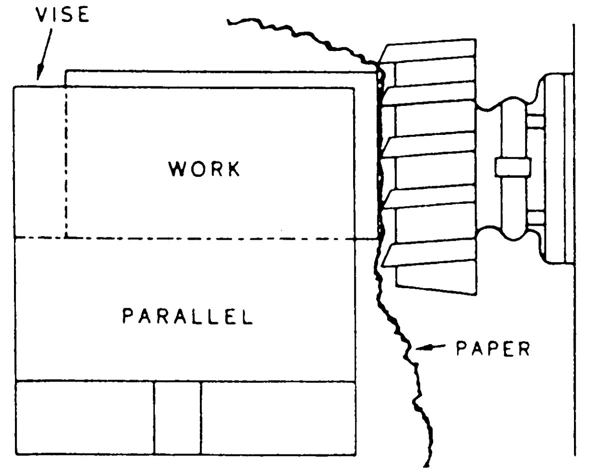


Figure 7-59.—Picking up the work surface.

6. Lock the knee in position.

7. Set the machine for the proper roughing speed, feed, and table travel.

8. Start the spindle and pick up the end surface of the work by hand feeding the work toward the cutter.

9. Place a strip of paper between the cutter and the work, as shown in figure 7-59, to help pick up the surface. When the cutter picks up the paper there is approximately 0.003-inch clearance between the cutter and the material being cut.

10. Once the surface is picked up, set the saddle feed graduated dial at ZERO.

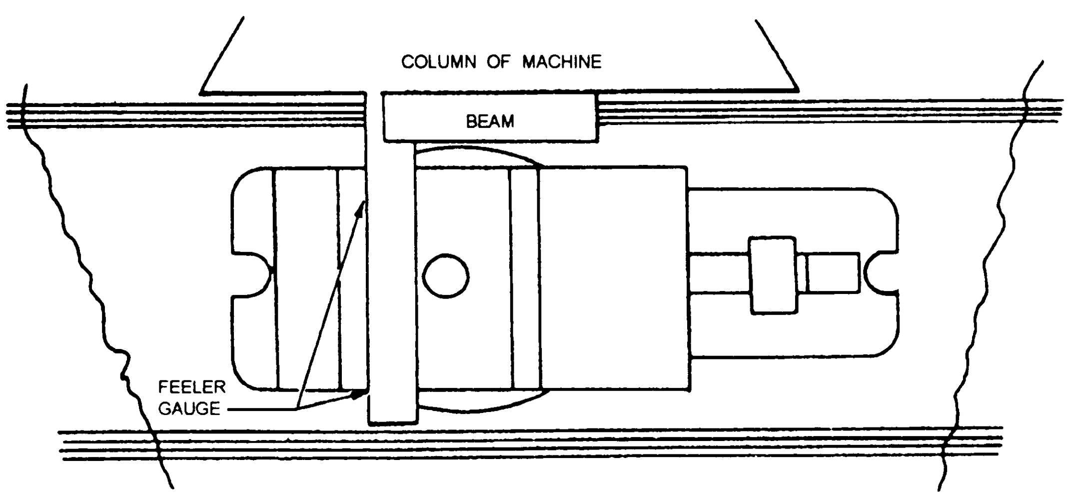


Figure 7-58.—Aligning vise jaws using a square.

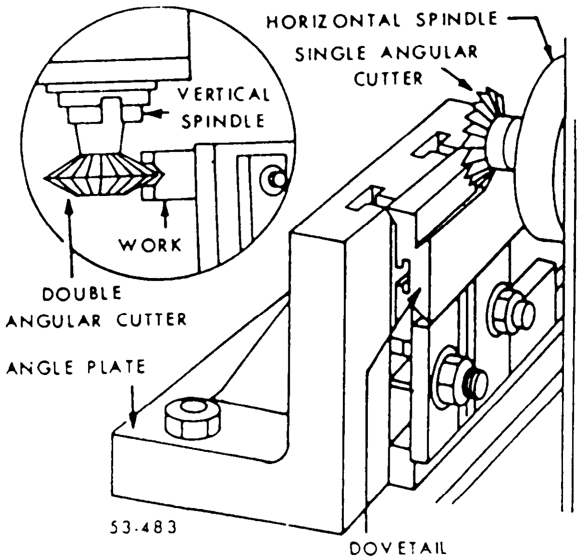


Figure 7-60.—Angular milling.

11. Move the work away from the cutter with the table and direct the coolant flow onto the cutter.
12. Set the roughing depth of cut, using the graduated dial, and lock the saddle.
13. Position the work to about 1/16 inch from the cutter, and then engage the power feed.
14. After completing the cut, stop the spindle, and move the work back to the starting point before the next cut.
15. Set the speed and feed for the finishing cut, and then unlock the saddle.
16. Move the saddle in for the final depth of cut and relock it.
17. Engage the spindle and take the finish cut.
18. Stop the machine and return the work to the starting place.
19. Shut the machine off.
20. Remove the work from the vise. Handle it very carefully to keep from cutting yourself before you can deburr the work.
21. Next, mount the work in the vise so the other end is ready to be machined. Mill this end in the same manner as the first, but be sure to measure the length before you take the finishing cut. Before removing the work from the vise, check it for accuracy and remove the burrs from the newly finished end.

ANGULAR MILLING

Angular milling is the milling of a flat surface that is at an angle to the axis of the cutter. Normally, you will use an angular milling cutter, as shown in figure 7-60. However, you can perform angular milling with a plain, side, or face milling cutter by positioning the work at the required angle.

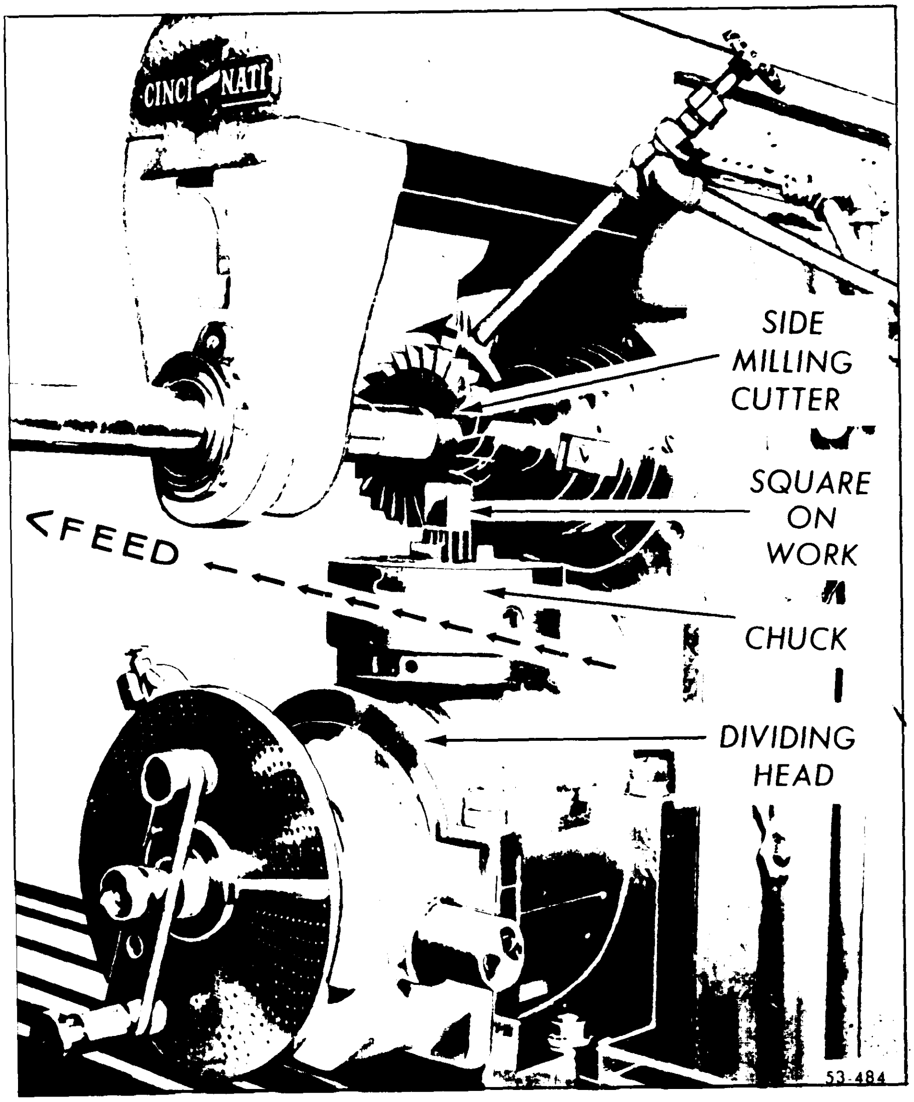
Many maintenance or repair tasks require you to machine flat surfaces on cylindrical work. They include milling squares and hexagons, and milling two flats in the same plane.

A square or hexagon is milled on an object to provide a positive drive, no slip area that can be grasped by various tools, such as wrenches and cranks. You will machine squares and hexagons frequently on the ends of bolts, taps, reamers, or other items that are turned by a wrench and on drive shafts and other items that require a positive drive. The following information will help you to understand the machining of squares and hexagons.

Cutter Setup

The two types of cutters you will use most often to machine squares or hexagons are side and end milling cutters. You can use side milling cutters to machine work that is held in a chuck and for heavy cutting. You can use end mills for work that is held in a chuck or between centers and for light cutting. If you use a side milling cutter, be sure the cutter diameter is large enough so you can machine the full length of the square or hexagon without interference from the arbor. If you use an end mill, be sure it is slightly larger in diameter than the length of the square or hexagon. The cutter thrust for both types should be up when the work is mounted vertically and down when it is mounted horizontally in order to use conventional (or up) milling.

The reason for what appears to be a contradiction in the direction of thrust is the difference in the direction of the feed. You can see this by comparing figures 7-61 and 7-62. The cutter shown in figure 7-61 rotates in a counterclockwise direction and the work is fed toward the left. The cutter shown in figure 7-62 rotates in a clockwise direction and the work is fed upward.



28.407

Figure 7-61.—Milling a square on work held vertically.

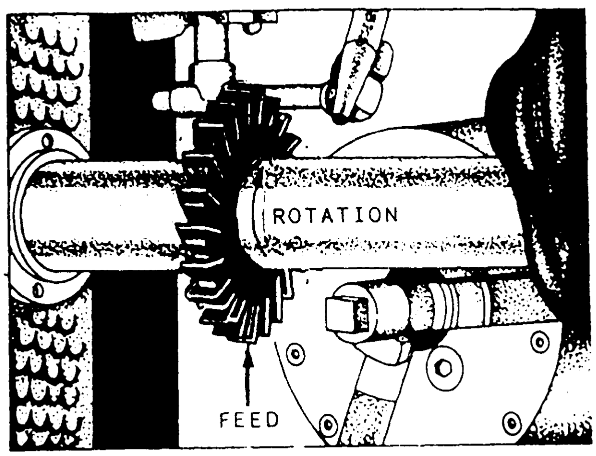
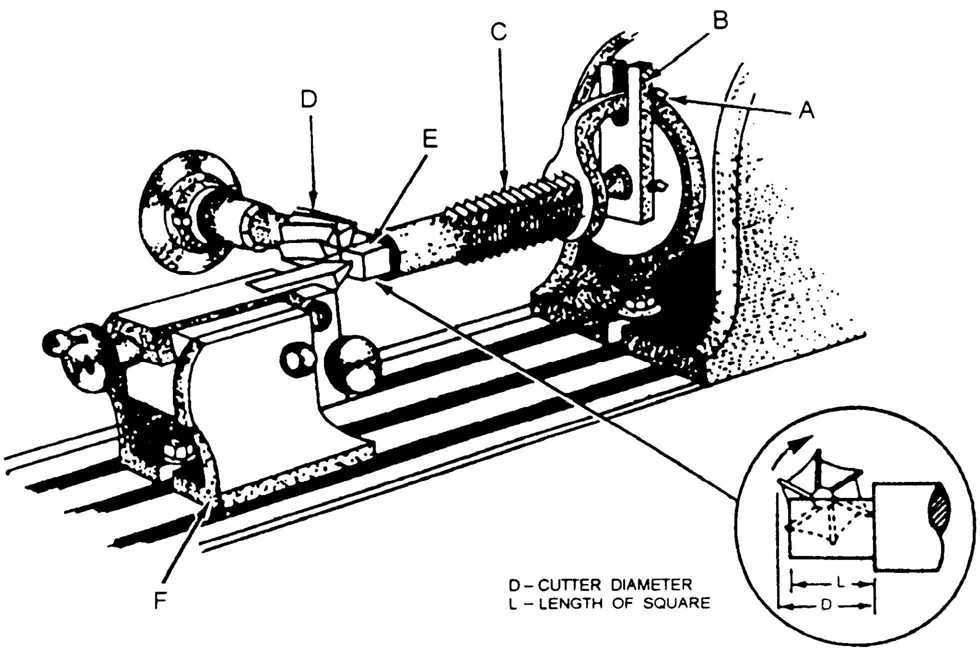


Figure 7-62.—Milling a square on work held horizontally.

Work Setup

We have already discussed the methods that you will usually use to mount the work. Regardless of the workholding method, you must align the index spindle in either the vertical or the horizontal plane. If you machine work between centers, you must also align the footstock center. If you use a screw-on chuck, consider the cutter rotary thrust applied to the work. Always cut on the side of the work that will tend to tighten the chuck on the index head spindle. When you mount work between centers, a dog rotates the work. The drive plate,



A. Lock screw for dog
B. Drive plate
C. Tap
D. End mill
E. Tap square
F. Footstock

Figure 7-63.—Milling a square using an end mill.

shown in figure 7-63, contains two lock screws. One lock screw clamps the drive plate to the index center and ensures that the drive plate moves with the index spindle. The other lock screw clamps the tail of the dog against the side of the drive plate slot, as shown in figure 7-63, A. This eliminates any movement of the work during the machining operation.

Calculations

The following information will help you determine the amount of material you must remove to produce a square or a hexagon. You must calculate the dimensions of the largest square or hexagon that you can machine from a piece of stock.

The size of a square (H in fig. 7-64) is measured across the flats. The largest square that you can cut from a given size of round stock equals the diameter of the stock in inches (which is also the diagonal of the square) times 0.707. This may be expressed as:

Opposite side = Side of a square

Hypotenuse = Diagonal of square

45° = 90° bisected

$$H = G \times 0.707 \text{ or } \frac{\textit{Opposite side}}{\textit{Hypotenuse}} = \text{sine } 45^\circ$$

The diagonal of a square equals the distance across the flats times 1.414. This is expressed as

$$G = H \times 1.414 \text{ or } \frac{\textit{Hypotenuse}}{\textit{Opposite side}} = \text{cosec } 45^\circ$$

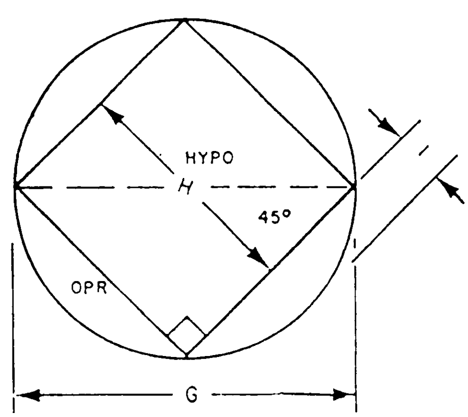


Figure 7-64.—Diagram of a square.

The amount of material that you must remove to machine each side of the square is equal to one-half the difference between the diameter of the stock and the distance across the flats.

$$I = \frac{G - H}{2}$$

You use the same formula

$$\left(I = \frac{G - H}{2}\right)$$

to determine the amount of material to remove when you machine a hexagon.

The size of the largest hexagon that you can machine from a given size of round stock (H in fig. 7-65) is equal to the diagonal (the diameter of the stock) of the hexagon times 0.866 or

Opposite side = Largest hexagon that can be machined

Hypotenuse = Diagonal or diameter of round stock

$$H = G \times 0.866 \text{ or } \frac{\textit{Opposite side}}{\textit{Hypotenuse}} = \text{sine } 60^\circ$$

The diagonal of a hexagon equals the distance across the flats times 1.155, or

$$G = H \times 1.155 \text{ or } \frac{\textit{Hypotenuse}}{\textit{Opposite side}} = \text{cosec } 60^\circ$$

The length of a flat is equal to one-half the length of the diagonal,

$$r = \frac{G}{2}$$

We will explain two methods used to machine a square or hexagon: work mounted in a chuck and work mounted between centers.

You can machine a square or hexagon on work mounted in a chuck by using either a side milling cutter or an end mill. We will discuss the side milling cutter first. Before placing the index head on the milling machine table, be sure the table and the bottom of the index head have been cleaned of all chips and other foreign matter. Spread a thin film of clean machine oil over the area of the table to which the index head will be attached to prevent corrosion.

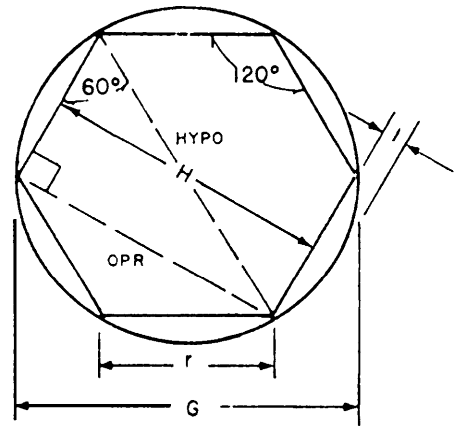


Figure 7-65.—Diagram of a hexagon.

NOTE: Because most index heads are quite heavy and awkward, you should get someone to help you place the head on the milling machine table.

After you have mounted the index head on the table, position the head spindle in the vertical position, as shown in figure 7-61. Use the degree graduations on the swivel block. This is accurate enough for most work requiring the use of the index head. The vertical position will allow you to feed the work horizontally.

Then, tighten the work in the chuck to keep it from turning due to the cutter's thrust. Install the arbor, cutter, and arbor support. The cutter should be as close as practical to the column. Remember, this is done so the setup will be more rigid. Set the machine for the correct roughing speed and feed.

1. With the cutter turning, pick up the cut on the end of the work.
2. Move the work sideways to clear the cutter.
3. Raise the knee a distance equal to the length of the flat surfaces to be cut.
4. Move the table toward the revolving cutter and pick up the side of the work. Use a piece of paper in the same manner as discussed earlier in this chapter and shown in figure 7-59.
5. Set the cross-feed graduated dial at ZERO.
6. Move the work clear of the cutter. Remember, the cutter should rotate so the cutting action takes place as in "up milling."
7. Feed the table in the required amount for a roughing cut.

8. Engage the power feed and the coolant flow.
9. When the cut is finished, stop the spindle and return the work to the starting point.
10. Loosen the index head spindle lock.
11. Rotate the work one-half revolution with the index crank.
12. Tighten the index head spindle lock.
13. Take another cut on the work.
14. When this cut is finished, stop the cutter and return the work to the starting point.
15. Measure the distance across the flats to determine whether the cutter is removing the same amount of metal from both sides of the work. If not, check your calculations and the setup for a possible mistake.
16. If the work measures as it should, loosen the index head spindle lock and rotate the work one-quarter revolution, tighten the lock, and take another cut.
17. Return the work to the starting point again.
18. Loosen the spindle lock.
19. Rotate the work one-half revolution.
20. Take the fourth cut.
21. Return the work again to the starting point and set the machine for finishing speed and feed.
22. Now, finish machine opposite sides (1 and 3), using the same procedures already mentioned.
23. Check the distance across these sides. If it is correct, finish machine the two remaining sides.
24. Deburr the work and check it for accuracy.

NOTE: You can also machine a square or hexagon with the index head spindle in the horizontal position, as shown in figures 7-62 and 7-63. If you use the horizontal setup, you must feed the work vertically.

Square or Hexagon Work Mounted Between Centers

Machining a square or hexagon on work mounted between centers is done in much the same manner as when the work is held in a chuck.

1. Mount the index head the same way, only with the spindle in a horizontal position. The feed will be in a vertical direction.
2. Insert a center into the spindle and align it with the footstock center.
3. Select and mount the desired end mill, preferably one whose diameter is slightly greater than the length of the flat you are to cut, as shown in figure 7-63.
4. Mount the work between centers. Make sure that the drive dog is holding the work securely.
5. Set the machine for roughing speed and feed.
6. Pick up the side of the work and set the graduated cross-feed dial at ZERO.
7. Lower the work until the cutter clears the footstock.
8. Move the work until the end of the work is clear of the cutter.
9. Align the cutter with the end of the work. Use a square head and rule, as shown in figure 7-66.

NOTE: Turn the machine off before you align the cutter by this method.

10. Move the table a distance equal to the length of the flat desired.
11. Move the saddle the distance required for the roughing depth of cut.
12. While feeding the work vertically, machine side 1. Lower the work to below the cutter when you have completed the cut.
13. Loosen the index head spindle lock and index the work one-half revolution to machine the flat opposite side 1.
14. Tighten the lock.
15. Engage the power feed. After completing the cut, again lower the work to below the cutter and stop the cutter.
16. Measure the distance across the two flats to check the accuracy of the cuts. If it is correct, index the work one-quarter revolution to machine another side. When you complete that side, lower the work, index one-half revolution, and machine the last

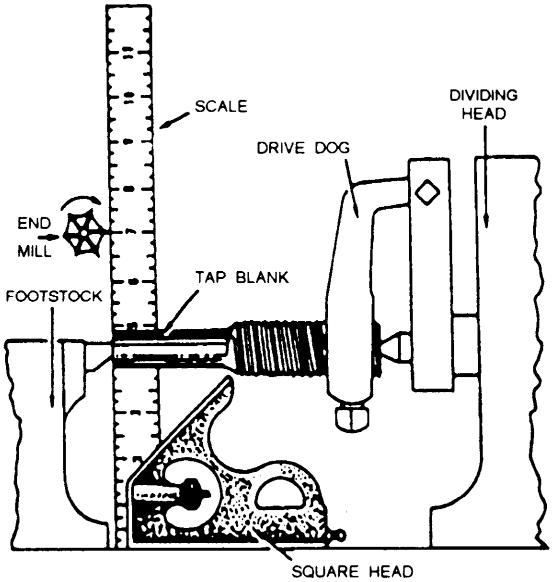


Figure 7-66.—Aligning the work and the cutter.

side. Remember to lower the work to below the cutter again.

17. Set the machine for finishing speed, feeds, and depth of cut, and finish machine all the sides.

18. Deburr the work and check it for accuracy.

Machining Two Flats in One Plane

You will often machine flats on shafts to serve as seats for setscrews. One flat is simple to machine. You can machine in any manner with a side or end mill, as long as you can mount the work properly. However, machining two flats in one plane, such as the flats on the ends of a mandrel, presents a problem because the flats must align with each other. A simple method is to mount the work in a vise or on V-blocks in such a manner that you can machine both ends without moving the work once it has been secured.

We will describe the method that is used when the size or shape of the work requires repositioning it to machine both flats.

1. Apply layout dye to both ends of the work.
2. Place the work on a pair of V-blocks, as shown in figure 7-67.
3. Set the scriber point of the surface gauge to the center height of the work. Scribe horizontal lines on both ends of the work, as illustrated in figure 7-67.
4. Mount the index head on the table with its spindle in the horizontal position.
5. Again, set the surface gauge scriber point, but to the center line of the index head spindle.
6. Insert the work in the index head chuck with the end of the work extended far enough to permit all required machining operations.
7. To align the surface gauge scriber point with the scribed horizontal line, rotate the index head spindle.
8. Lock the index head spindle in position.

You can mill these flats with either an end mill, a side mill, or a side milling cutter.

NOTE: Rotate the cutter in a direction that will cause the thrust to tighten the index head chuck on the spindle when you use a screw-on type of chuck.

9. Raise the knee with the surface gauge still set at center height until the cutter center line is aligned with the scriber point. This puts the center lines of the cutter and the work in alignment with each other.
10. Position the work so that a portion of the flat to be machined is located next to the cutter. Because of the shallow depth of cut, compute the speed and feed as if the cuts were finishing cuts.

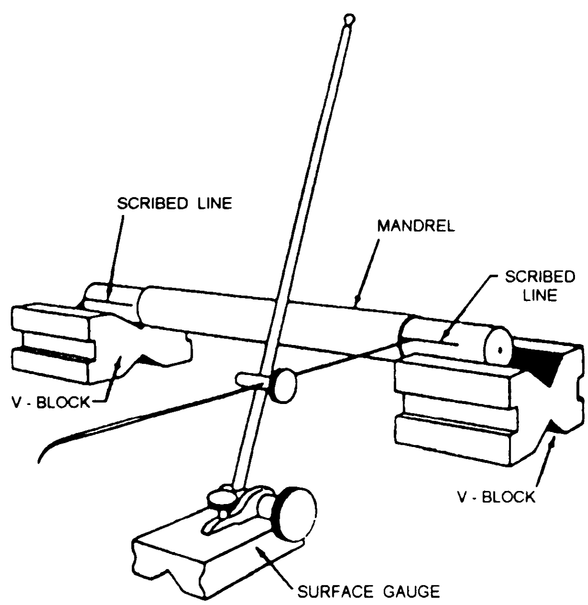


Figure 7-67.—Layout of the work.

11. After starting the machine, feed the work by hand so the cutter contacts the side of the work on which the line is scribed.

12. Move the work clear of the cutter and stop the spindle.

13. Check to see if the greater portion of the cutter mark is above or below the layout line. Depending on its location, rotate the index head spindle as required to center the mark on the layout line.

14. Once the mark is centered, take light "cut and try" depth of cuts until you reach the desired width of the flat.

15. Machine the flat to the required length.

16. When one end is completed, remove the work from the chuck. Turn the work end for end and reinsert it in the chuck.

17. Machine the second flat in the same manner as you did the first.

18. Deburr the work and check it for accuracy.

19. Check the flats to see if they are in the same plane by placing a matched pair of parallels on a surface plate and one flat on each of the parallels. If the flats are in the same plane, you will not be able to wobble the work.

SLOTTING, PARTING, AND MILLING KEYSEATS

Slotting, parting, and milling keyseats are all operations that require you to cut grooves in the work. These grooves are of various shapes, lengths, and depths, depending on the requirements of the job. They range from flutes in a reamer to a keyseat in a shaft, to the parting off of a piece of metal to a predetermined length.

Slotting

You can use slotting to cut internal contours, such as internal gears and splines and 6- or 12-point sockets. Most slotting is done with a milling machine attachment called a slotting attachment, as shown in figure 7-68. The slotting attachment is fastened to the milling machine column and driven by the spindle. This attachment changes the rotary motion of the spindle to a reciprocating motion much like that of a shaper. You can vary the length of the stroke within a specified range. A pointer on the slotting attachment slide shows the length of the stroke. You can pivot the head of the slotting attachment and position it at any desired angle. Graduations on the base of the slotting attachment show the angle at which the head is positioned. The number of strokes per minute is equal to the spindle rpm and is determined by the formula:

$$\text{Strokes per minute} = \frac{CFS \times 4}{\text{length of stroke}}$$

To make the cutting tools used with slotting attachments, grind them to any desired shape from high-speed steel tool blanks. Clamp the tool to the front of the slide or ram. You can use any suitable means to hold the work, but the most common method is to hold the work in an index head chuck If the slotted portion does not extend through the work, you will have to machine an internal recess in the work to provide clearance for the tool runout. When it is possible, position the slotting attachment and the work in the vertical position to provide the best possible view of the cutting action of the tool.

Parting

Use a metal slitting saw for sawing or parting operations and to mill deep slots in metals and in a variety of other materials. Efficient sawing depends to a large extent on the slitting saw you select. The work required of slitting saws varies greatly. It would not be efficient to use the same saw to cut very deep

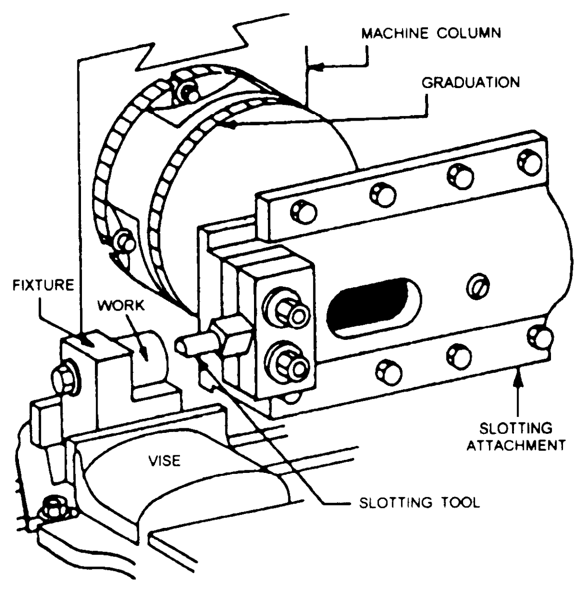


Figure 7-68.—Slotting attachment.

narrow slots, part thick stock, saw thin stock, or saw hard alloy steel. Soft metals, such as copper and babbitt, or nonmetallic materials, such as bakelite, fiber, or plastic, require their own style of slitting saw.

Parting with a slitting saw leaves pieces that are reasonably square and that require you to remove a minimum of stock to finish the surface. You can cut off a number of pieces of varying lengths and with less waste of material than you could saw by hand.

A coarse-tooth slitting saw is best to saw brass and to cut deep slots. A fine-tooth slitting saw is best to saw thin metal, and a staggered-tooth slitting saw is best to make heavy deep cuts in steel. You should use slower feeds and speeds to saw steels to prevent cutter breakage. Use conventional milling to saw thick material. To saw thin material, however, clamp the stock directly to the table and use down milling. Then, the slitting saw will tend to force the stock down on the table. Position the work so the slitting saw extends through the stock and into a table T-slot.

External Keyseat

It is less complicated to machine an external keyseat on a milling machine than on a shaper. In milling, it is no problem to start an external keyseat. Simply bring the work into contact with a rotating cutter and start cutting. You should be able to picture in your mind how you'll mill a straight external keyseat with a plain milling cutter or an end mill. If the specified length of the keyseat exceeds the length you can obtain by milling to the desired depth, you can move the work in the direction of the slot to get the desired length. It should be easier to picture in your mind how you'll mill a Woodruff keyseat. The secret is to select a cutter that has the same diameter and thickness as the key.

STRAIGHT EXTERNAL KEYSEATS.— Normally, you'll use a plain milling cutter to mill a straight external keyseat. You also can use a Woodruff cutter or a two-lipped end mill.

Before you can begin milling, align the axis of the work with the midpoint of the width of the cutter. Figure 7-69 shows one method of alignment.

Suppose you're going to cut a keyseat with a plain milling cutter. First, move the work until the side of the cutter is tangent to the circumference of the work. With the cutter turning very slowly and before contact is made, insert a piece of paper between the work and the side of the cutter. Continue moving the work toward the cutter until the paper begins to tear. When it does, lock the graduated dial at ZERO on the saddle feed screw. Then, lower the milling machine knee. Use the saddle feed dial as a guide, and move the work a distance equal to the radius of the work plus one-half the width of the cutter. This will center the cutter over the center line of the keyseat.

Use a similar method to align work with an end mill. Move the work toward the cutter while you hold a piece of paper between the rotating cutter and the work, as shown in figure 7-70. After the paper tears, lower the work to just below the bottom of the end

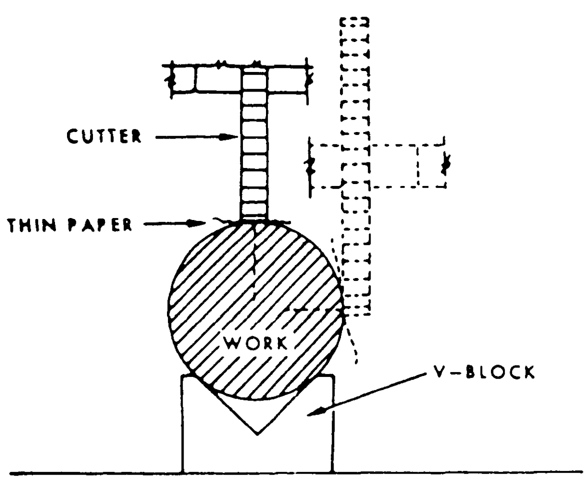


Figure 7-69.—Aligning the cutter using a paper strip.

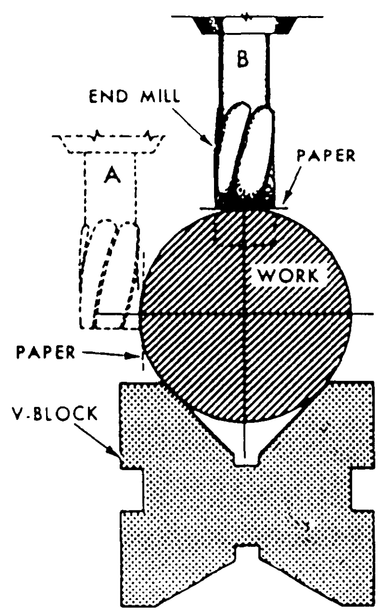


Figure 7-70.—Aligning an end mill with the work.

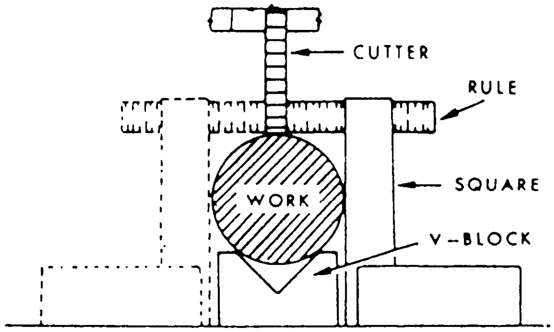


Figure 7-71.—Visual alignment of a cutter.

mill. Then, move the work a distance equal to the radius of the work plus the radius of the end mill. This will center the mill over the center line of the keyseat. Move the work up, using hand feed, until a piece of paper held between the work and the bottom of the end mill begins to tear, as shown in figure 7-70, B. Then, move the table and work away from the bottom of the end mill. Set and lock the graduated dial at ZERO on the vertical feed, and then feed up for the roughing cut. You can determine the cutter rpm and the longitudinal feed in the same manner as you do for conventional milling cutters. The higher speeds and feeds generate more heat, so flood the work and the cutter with coolant.

When extreme accuracy is not required, you can align the work with the cutter visually, as shown in figure 7-7 1. Position the work by eye as near as possible to the midpoint of the cutter. Make the final alignment by moving the work in or out a slight amount, as needed. The cutter should be at the exact center of the work diameter measurement of the steel rule. You can use this method with both plain milling cutters and end mills.

Before you begin to machine the keyseat, you should measure the width of the cut. You cannot be certain that the width will be the same as the thickness of the cutter. The cutter may not run exactly true on the arbor or the arbor may not run exactly true on the spindle. The recommended practice is to nick the end of the work with the cutter and then to measure the width of the cut.

Specifications for the depth of cut are usually Furnished. When they are not available, you can determine the total depth of cut for a square keyseat by using the following formula based on dimensions shown in figure 7-72.

$$\text{Total depth of cut } (T) = d + f$$

where

$$d = \frac{W}{2} = \text{depth of the keyseat}$$

$$f = R - \sqrt{R^2 - \left(\frac{W}{2}\right)^2} = \text{height of arc}$$

W = *width of the key*

R = *radius of the shaft*

The height of arc (f) for various sizes of shafts and keys is shown in table 7-1. Keyseat dimensions for rounded end and rectangular keys are contained in the *Machinery's Handbook.* Check the keyseats for accuracy with rules, outside and depth micrometers, vernier calipers, and go-no-go gauges. Use table 7-1 for both square and Woodruff keyseats, which will be explained next.

WOODRUFF KEYSEAT.—A Woodruff key is a small half-disk of metal. The rounded portion of the key fits in the slot in the shaft. The upper potion fits into a slot in a mating part, such as a pulley or gear. You align the work with the cutter and measure the width of the cut in exactly the same manner as you do to mill straight external keyseats.

A Woodruff keyseat cutter (fig. 7-73) has deep flutes cut across the cylindrical surface of the teeth. The cutter is slightly thicker at the crest of the teeth than it is at the center. This feature provides clearance between the sides of the slot and the cutter. Cutters

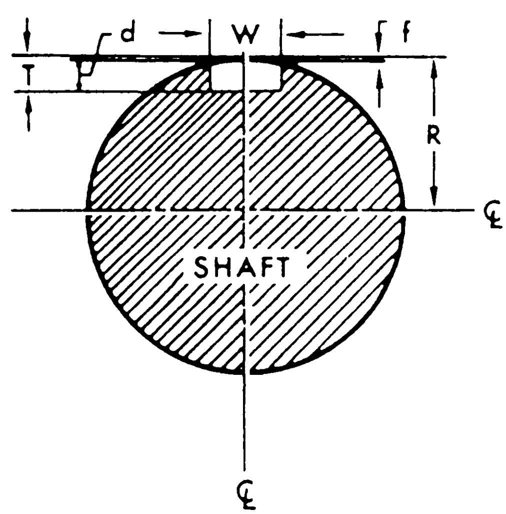


Figure 7-72.—Keyseat dimensions for a straight square key.

Table 7-1.—Values for Factor (f) for Various Sizes of Shafts

DIAMETER OF SHAFT (INCHES)	WIDTH OF KEY IN INCHES: 1/16	3/32	1/8	5/32	3/16	7/32	1/4	5/16
SHAFT SIZE	FACTOR (f)							
1/2	.002	.004	.008	.013	.018	.025	.033	---
5/8	.001	.003	.006	.010	.014	.019	.025	.042
3/4	.001	.003	.005	.008	.012	.016	.022	.034
7/8	.001	.002	.004	.007	.010	.014	.018	.028
1	.001	.002	.004	.006	.009	.012	.015	.024
1 1/8	----	.002	.003	.005	.008	.011	.014	.022
1 1/4	----	.002	.003	.005	.007	.010	.013	.019
1 1/2	----	.001	.002	.004	.006	.008	.011	.016
1 3/4	----	.001	.002	.003	.005	.007	.009	.014

with a 2-inch or larger diameter have a hole in the center to mount the arbor. On smaller cutters, the cutter and the shank are one piece. Note that the shank is "necked" in back of the cutting head to give additional clearance. Also, note that large cutters usually have staggered teeth to improve their cutting action.

We said earlier that to mill a Woodruff keyseat in a shaft, you should use a cutter that has the same diameter and thickness as the key. It is relatively simple to cut a Woodruff keyseat. You simply move the work up into the cutter until you get the desired keyseat depth. You may hold the work in a vise, chuck, between centers, or clamped to the milling

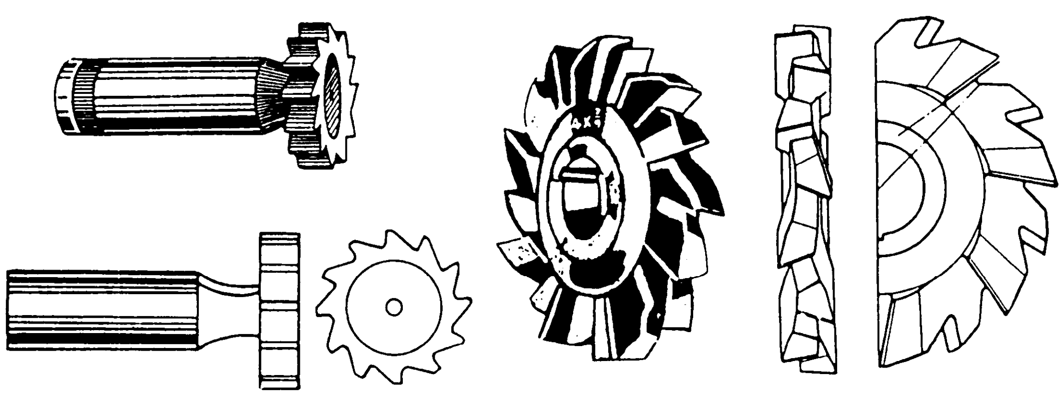


Figure 7-73.—Woodruff keyseat cutter.

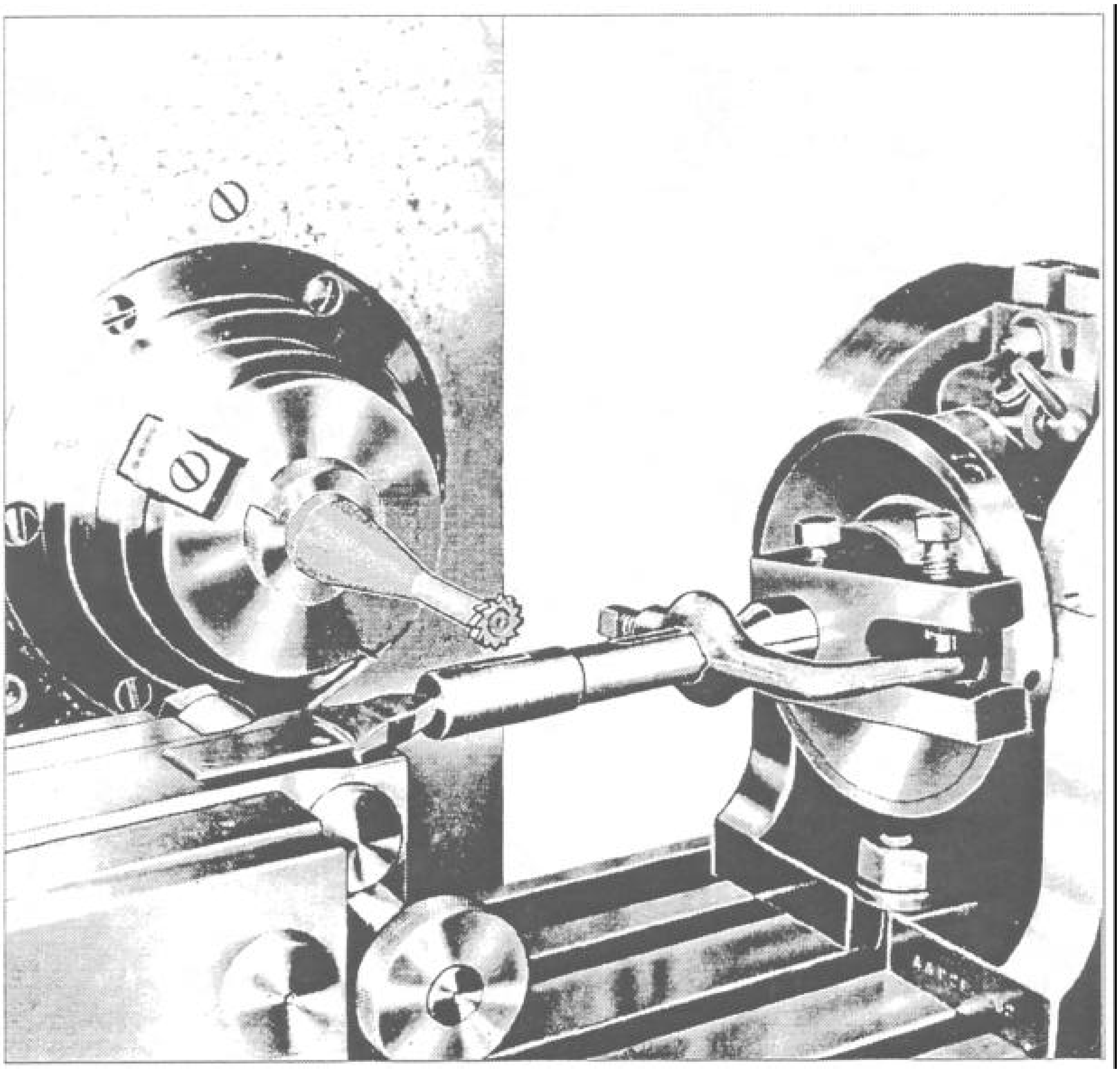


Figure 7-74.—Milling a Woodruff keyseat.

machine table. You will hold the cutter on an arbor, or in a spring collet or drill chuck that has been mounted in the spindle of the milling machine, as in figure 7-74.

In milling the keyseat, locate the cutter centrally over the position in which the keyseat will be cut and parallel with the axis of the work Raise the work by using the hand vertical feed until the revolving cutter tears a piece of paper held between the teeth of the cutter and the work. At this point, set the graduated dial on the vertical feed at ZERO and set the clamp on the table. With the graduated dial as a guide, raise the work by hand until the full depth of the keyseat is cut. If specifications for the total depth of cut are not available, use the following formula to determine the correct value:

$$\textit{Total depth } (T) = d + f$$

where

$$d \textit{ (depth of the keyseat)} = H - \frac{W}{2}$$

H = *total height of the key*

W = *width of the key*

The most accurate way to check the depth of a Woodruff keyseat is to insert a Woodruff key of the correct size in the keyseat. Measure over the key and the work with an outside micrometer to obtain the

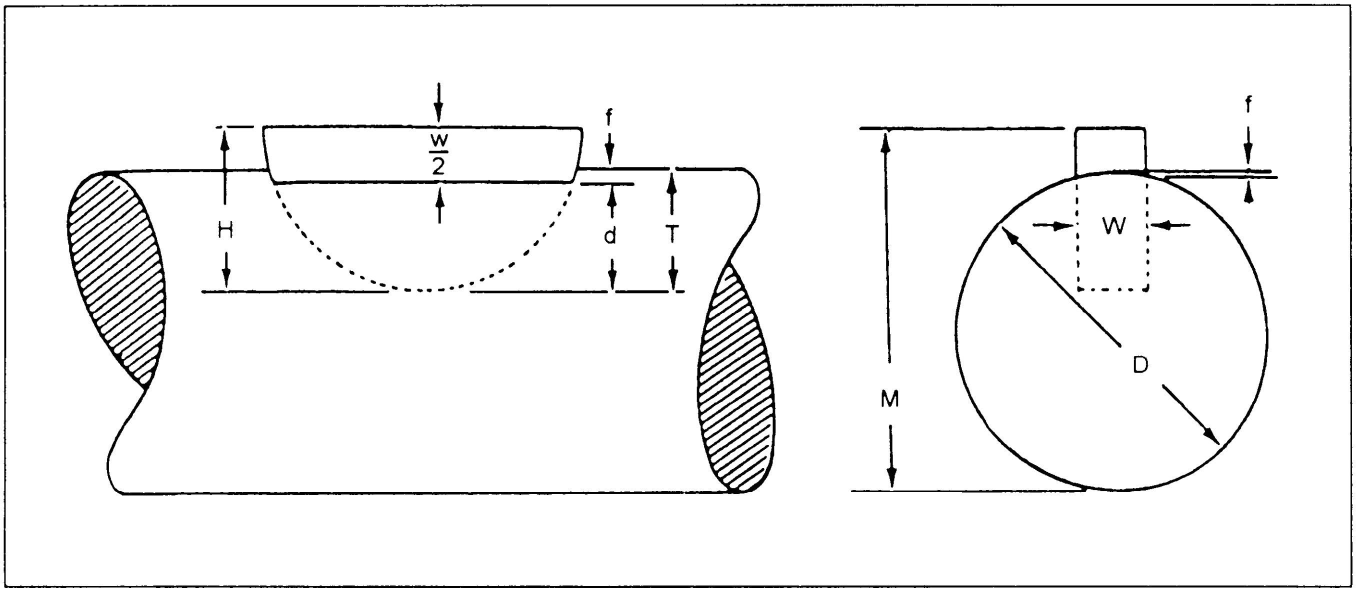


Figure 7-75—Dimensions for a Woodruff keyseat.

distance, M in figure 7-75. You can also determine distance M by using the formula:

$$M = D + \frac{(W)}{(2)} - f$$

where

M = *micrometer reading*

D = *diameter of the shaft*

f = *height of the arc between the top of the slot and the top of the shaft*

NOTE: Tables in some references may differ slightly from the above calculation for the value M, due to greater allowance for clearance at the top of the key.

FLY CUTTING

You will use a fly cutter when a formed cutter is required but not available. Fly cutters are high-speed steel tool blanks that have been ground to the required shape. Any shape can be ground on the tool if the cutting edges arc given enough clearance. Fly cutters are mounted in fly cutter arbors, such as the one shown in figure 7-45. Use a slow feed and a shallow depth of cut to prevent breaking the tool. It is a good idea to rough out as much excess material as possible with ordinary cutters and to use the fly cutter to finish shaping the surface.

DRILLING, REAMING, AND BORING

Drilling, reaming, and boring are operations that you can do very efficiently on a milling machine. The graduated feed screws make it possible to accurately locate the work in relation to the cutting tool. In each operation the cutting tool is held and rotated by the spindle, and the work is fed into the cutting tool.

Drilling and Reaming

Use the same drills and reamers that you use to drill and ream in the lathe and the drill press. Hold drills and reamers in the spindle by the same methods that you use to hold straight and taper-shanked end mills. You can hold the work in a vise, clamped to the table, held in fixtures or between centers, and in index head chucks, as you do in milling. Determine the speeds used to drill and ream in the same manner as you did those used to drill and ream in the lathe or the drill press. Feed the work into the drill or reamer by either hand or power feed. If you mount the cutting tool in a horizontal position, use the transverse or saddle feed. If you mount a drill or reamer in a vertical position, as in a vertical-type machine, use the vertical feed.

Boring

Of the three operations, boring is the only one that warrants special treatment. On a milling machine you

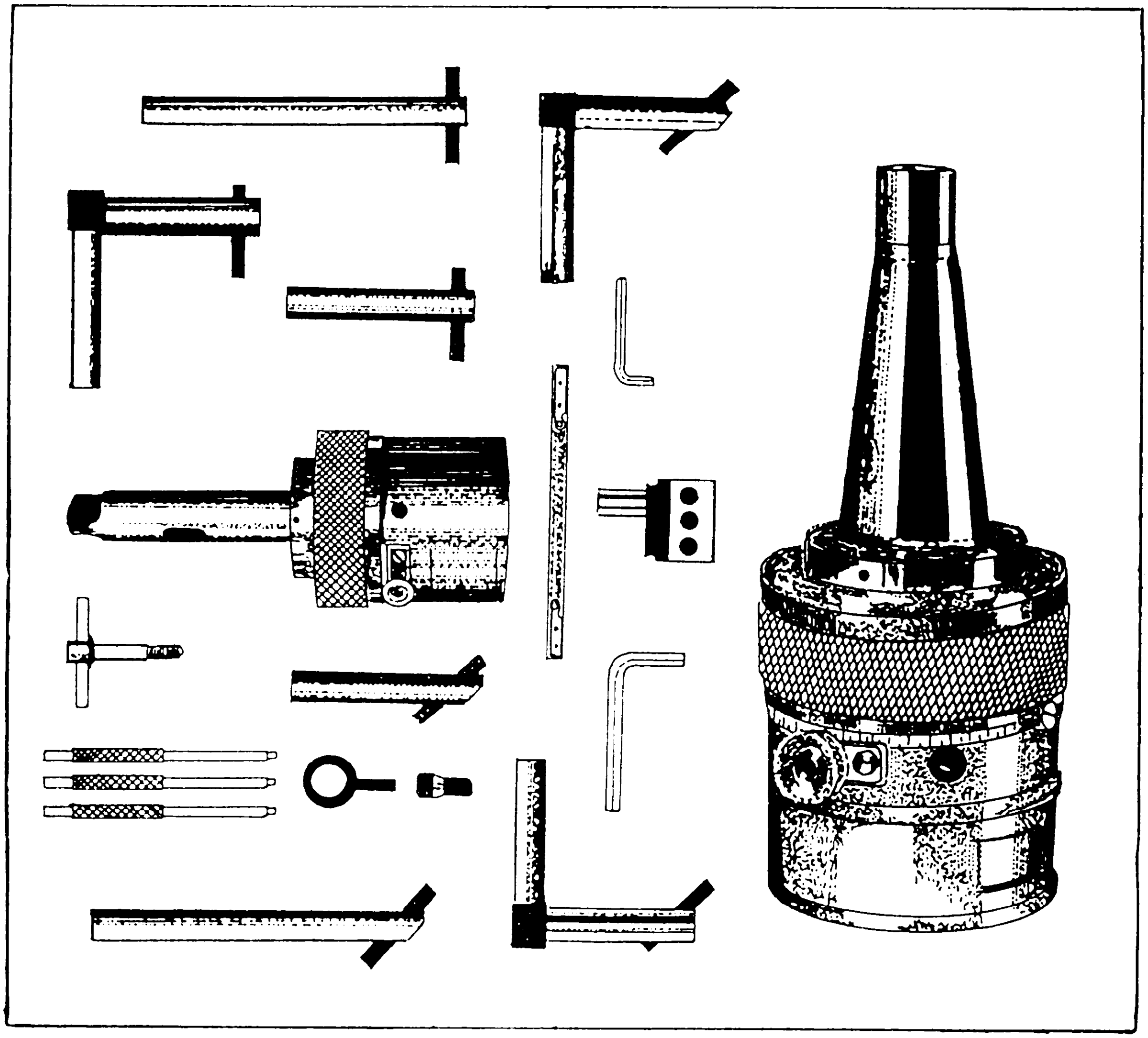


Figure 7-76.—Offset boring head and boring tools.

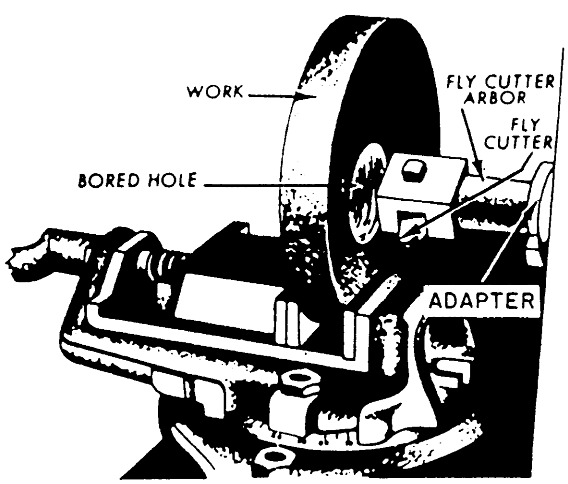


Figure 7-77.—Boring with a fly cutter.

usually bore holes with an offset boring head. Figure 7-76 shows several views of an offset boring head and several boring tools. Note that the chuck jaws, which grip the boring bar, can be adjusted at a right angle to the spindle axis. This feature lets you accurately position the boring cutter to bore holes of varying diameters. This adjustment is more convenient than adjusting the cutter in the boring bar holder or by changing boring bars.

Although the boring bars are the same on a milling machine as on a lathe or drill press, the manner in which they are held is different. Note in figure 7-77 that a boring bar holder is not used. The boring bar is inserted into an adapter and the adapter is fastened in the hole in the adjustable slide. Power to drive the boring bar is transmitted directly through the shank. The elimination of the boring bar holder

Deleted—No permission granted for electronic copy.

Figure 7-78.—Universal milling (head) attachment.

results in a more rigid boring operation, but the size of the hole that can be bored is more limited than those on a lathe or a drill press.

Fly cutters, which we discussed previously, can also be used for boring, as shown in figure 7-77. A fly cutter is especially useful to bore relatively shallow hales. The cutting tool must be adjusted for each depth of cut.

The speeds and feeds you should use to bore on a milling machine are comparable to those you would use to bore on a lathe or drill press. They also depend on the same factors: hardness of the metal, kind of metal in the cutting tool, and depth of cut. The boring bar is a single-point cutting tool; therefore, the diameter of the arc through which the tool moves is also a factor. For all of these reasons you must avoid too-great of speeds to prevent vibration.

MILLING MACHINE ATTACHMENTS

Many attachments have been developed that increase the number of jobs a milling machine can do, or that make such jobs easier to do.

UNIVERSAL MILLING ATTACHMENT

The universal milling (head) attachment, shown in figure 7-78, is clamped to the column of the milling machine. The cutter can be secured in the spindle of

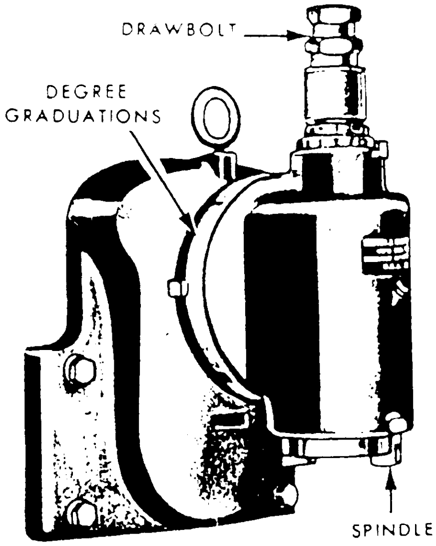


Figure 7-79.—Vertical milling attachment.

the attachment and then set by the two rotary swivels so the cutter will cut at any angle to the horizontal or the vertical plane. The spindle of the attachment is driven by gearing connected to the milling machine spindle.

VERTICAL MILLING ATTACHMENT

You can use a vertical milling attachment (fig. 7-79) to convert the horizontal spindle machine to a vertical spindle machine and swivel the cutter to any position in the vertical plane. You can use a universal milling attachment to swivel the cutter to any position in both the vertical and horizontal planes. These attachments will help simplify otherwise complex jobs.

HIGH-SPEED UNIVERSAL ATTACHMENT

You can use a high-speed universal attachment to perform milling operations at higher speeds than those for which the machine was designed. This attachment is clamped to the machine and driven by the milling machine spindle, as you can see in figure 7-80. You can swivel the attachment spindle head and cutter 360° in both planes. The attachment spindle is driven at a higher speed than the machine spindle. You must consider the ratio between the rpm of the two spindles when you calculate cutter speed. Drive small cutters, end mills, and drills at high rates of speed to maintain an efficient cutting action.

RACK MILLING ATTACHMENT

The rack milling attachment, shown in figure 7-81, is used primarily to cut teeth on racks, although it can be used for other operations. The cutter is mounted on a spindle that extends through the attachment parallel to the table T-slots. An indexing arrangement is used to space the rack teeth quickly and accurately.

FEEDS, SPEEDS, AND COOLANTS

Milling machines usually have a spindle speed range from 25 to 2,000 rpm and a feed range from 1/4 inch to 30 inches per minute (ipm). The feed is independent of the spindle speed; therefore, you can feed a workpiece at any rate available in the feed range regardless of the spindle speed. In the following paragraphs, we'll discuss some of the factors concerning the selection of appropriate milling feeds and speeds.

SPEEDS

Heat generated by friction between the cutter and the work may be regulated by the use of proper speed, feed, and cutting coolant. Regulation of this heat is

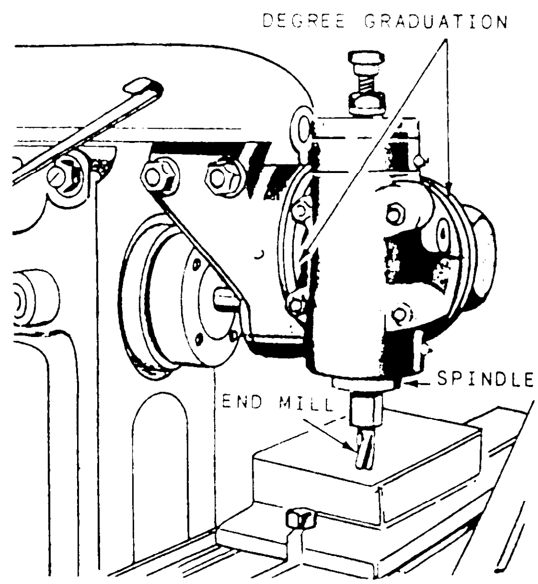


Figure 7-80.—High-speed universal milling attachment.

Deleted—No permission granted for electronic copy.

Figure 7-81.—Rack milling attachment.

very important because the cutter will be dulled or even made useless by overheating. It is almost impossible to provide any fixed rules that will govern cutting speeds because of varying conditions from job to job. Generally speaking, you should select a cutting speed that will give the best compromise between maximum production and longest life of the cutter. In any particular operation, consider the following factors to determine the proper cutting speed:

- Hardness of the material being cut: The harder and tougher the metal being cut, the slower should be the cutting speed.

- Depth of cut and desired finish: The amount of friction heat produced is directly proportional to the amount of material being removed. Therefore, you can often make finishing cuts at a speeds 40 to 80 percent higher than that used in roughing.

- Cutter material: You can operate high-speed steel cutters from 50 to 100 percent faster than carbon steel cutters because the high-speed cutters have better heat-resistant properties. Depending on the setup, you can operate carbide cutters at up to 4 times the speed of high-speed steel cutters.

- Type of cutter teeth: Cutters that have undercut teeth cut more freely than those that have a radial face. Therefore, you may run cutters with undercut teeth at higher speeds.

- Sharpness of the cutter: You can run a sharp cutter at a much higher speed than a dull cutter.

- Use of coolant: Sufficient coolant will usually cool the cutter so that it will not overheat even at relatively high speeds.

Table 7-2.—Surface Cutting Speeds

	HIGH-SPEED STEEL CUTTERS (FEET PER MINUTE)	
	ROUGH	FINISH
CAST IRON:		
Malleable	90	100
Hard castings	15	20
ANNEALED TOOL STEEL	40	50
LOW - CARBON STEEL	60	70
BRASS	110	150
ALUMINIUM	700	900

Use the approximate values in table 7-2 as a guide when you select the proper cutting speed for high-speed steel cutters. Refer to the manufacturer's recommendations if you are using carbide tooling. If you find you cannot suitably operate the machine, the cutter, or the work at the suggested speed, make an immediate readjustment.

Use table 7-3 to determine the cutter rpm for cutters varying in diameter from 1/4 inch to 5 inches. For example: You are cutting with a 7/16-inch cutter. If a surface speed of 160 fpm is required, the cutter rpm will be 1,398.

If the cutter diameter you are using is not shown in table 7-3, determine the proper rpm of the cutter by using the formula:

$$(a)\ rpm = \frac{Cutting\ speeed \times 12}{3.1416 \times Diameter}$$

$$or\ rpm = \frac{fpm}{0.2618 \times D}$$

where

rpm = revolutions per minute of the cutter

fpm = required surface speed in feet per minute

D = diameter of the cutter in inches

$$0.2618 = constant = \frac{\pi}{12}$$

Table 7-3.—Cutter Speeds in Revolutions Per Minute

Diameter of cutter (in.)	Surface speed (ft. per min.) 25	30	35	40	50	55	60	70	75	80	90	100	120	140	160	180	200
	Cutter revolutions per minute																
1/4	382	458	535	611	764	851	917	1,070	1,147	1,222	1,376	1,528	1,834	2,139	2,445	2,750	3,056
5/16	306	367	428	489	611	672	733	856	917	978	1,100	1,222	1,466	1,711	1,955	2,200	2,444
3/8	255	306	357	408	509	560	611	713	764	815	916	1,018	1,222	1,425	1,629	1,832	2,036
7/16	218	262	306	349	437	481	524	611	656	699	786	874	1,049	1,224	1,398	1,573	1,748
1/2	191	229	268	306	382	420	459	535	573	611	688	764	917	1,070	1,222	1,375	1,528
5/8	153	184	214	245	306	337	367	428	459	489	552	612	736	857	979	1,102	1,224
3/4	127	153	178	203	254	279	306	357	381	408	458	508	610	711	813	914	1,016
7/8	109	131	153	175	219	241	262	306	329	349	392	438	526	613	701	788	876
1	95.5	115	134	153	191	210	229	267	287	306	344	382	458	535	611	688	764
1 1/4	76.3	91.8	107	123	153	168	183	214	230	245	274	306	367	428	490	551	612
1 1/2	63.7	76.3	89.2	102	127	140	153	178	191	204	230	254	305	356	406	457	508
1 3/4	54.5	65.5	76.4	87.3	109	120	131	153	164	175	196	218	262	305	349	392	436
2	47.8	57.3	66.9	76.4	95.5	105	115	134	143	153	172	191	229	267	306	344	382
2 1/2	38.2	45.8	53.5	61.2	76.3	84.2	91.7	107	114	122	138	153	184	213	245	275	306
3	31.8	38.2	44.6	51	63.7	69.9	76.4	89.1	95.3	102	114	127	152	178	208	228	254
3 1/2	27.3	32.7	38.2	44.6	54.5	60	65.5	76.4	81.8	87.4	98.1	109	131	153	174	196	213
4	23.9	28.7	33.4	38.2	47.8	52.6	57.3	66.9	71.7	76.4	86	95.6	115	134	153	172	191
5	19.1	22.9	26.7	30.6	38.2	42	45.9	53.5	57.3	61.1	68.8	76.4	91.7	107	122	138	153

Example: What is the spindle speed for a 1/2-inch cutter running at 45 fpm?

$$rpm = \frac{45}{0.2618 \times 0.5}$$

$$rpm = 343.7$$

To determine cutting speed when you know the spindle speed and cutter diameter, use the following formula:

$$fpm \times 12 = rpm \times 3.1416 \times D$$

$$fpm = \frac{3.1416 \times Diameter \times rpm}{12}$$

$$fpm = 0.2618 \times D \times rpm$$

Example: What is the cutting speed of a 2 1/4-inch end mill running at 204 rpm?

$$fpm = 0.2618 \times D \times rpm$$

$$rpm = 0.2618 \times 2.25 \times 204$$

$$fpm = 120.1$$

FEEDS

The rate of feed is the rate of speed at which the workpiece travels past the cut. When selecting the feed, consider the following factors:

- Forces are exerted against the work, the cutter, and their holding devices during the cutting process. The force exerted varies directly with the amount of metal being removed and can be regulated by adjusting the feed and the depth of cut. The feed and depth of cut are therefore interrelated, and depend on the rigidity and power of the machine. Machines are limited by the power they can develop to turn the cutter and by the amount of vibration they can withstand during coarse feeds and deep cuts.

- The feed and depth of cut also depend on the type of cutter you are using. For example, do not attempt deep cuts or coarse feeds with a small diameter end mill; it will spring or break the cutter. You can feed coarse cutters with strong cutting teeth at a relatively high rate of feed because the chips will be washed out easily by the coolant.

- Do not use coarse feeds and deep cuts on a frail piece of work or on work mounted in such a way that the holding device will spring or bend.

- The desired degree of finish affects the amount of feed. A fast feed removes metal rapidly and the finish will not be very smooth. However, a slow feed and a high cutter speed will produce a finer finish. For roughing, it is advisable to use a comparatively low speed and a coarse feed. You will make more mistakes if you overspeed the cutter than if you overfeed the work. Overspeeding is indicated by a squeaking, scraping sound. If chattering occurs in the milling machine during the cutting process, reduce the speed and increase the feed. Other common causes of chattering are excessive cutter clearance, poorly supported work, or a badly worn machine gear.

One procedure used to select an appropriate feed for a milling operation is to consider the chip load of each cutter tooth. The chip load is the thickness of the chip that a single tooth removes from the work as it passes over the surface. For example, when a a cutter with 12 cutting teeth and a feed rate of 1 ipm turns at 60 rpm, the chip load of a single tooth of the cutter will be 0.0014 inch. An increase of cutter speed to 120 rpm reduces the chip load to 0.0007 inch; an increase of feed to 2 ipm increases chip load to 0.0028 inch. Use the following formula to calculate chip load:

$$Chip\ load = \frac{feed\ rate\ (ipm)}{cutter\ speed\ (rpm) \times number\ of\ teeth\ in\ the\ cutter}$$

Table 7-4.—Recommended Chip Loads

Material	Face Mills	Helical Mills	Slotting & Side Mills	End Mills	Form Relieved Cutters	Circular Saws
Plastic	.013	.010	.008	.007	.004	.003
Magnesium and alloys	.022	.018	.013	.011	.007	.005
Aluminum and alloys	.022	.018	.013	.011	.007	.005
Free cutting brasses & bronzes	.022	.018	.013	.011	.007	.005
Medium brasses & bronzes	.014	.011	.008	.007	.004	.003
Hard brasses & bronzes	.009	.007	.006	.005	.003	.002
Copper	.013	.010	.007	.006	.004	.003
Cast iron, soft (150-180 BH)*.	.016	.013	.009	.008	.005	.004
Cast iron, med. (180-220 BH)	.013	.010	.007	.007	.004	.003
Cast iron, hard (220-300 BH)	.011	.008	.006	.006	.003	.003
Malleable iron	.012	.010	.007	.006	.004	.003
Cast steel	.012	.010	.007	.006	.004	.003
Low carbon steel, free mach.	.012	.010	.007	.006	.004	.003
Low carbon steel . . .	.010	.008	.006	.005	.003	.003
Medium carbon steel	.010	.008	.006	.005	.003	.003
Alloy steel, annealed (180-220 BH)	.008	.007	.005	.004	.003	.002
Alloy steel, tough (220-300 BH)	.006	.005	.004	.003	.002	.002
Alloy steel, hard (300-400 BH)	.004	.003	.003	.002	.002	.001
Stainless steel, free mach.	.010	.008	.006	.005	.003	.002
Stainless steels	.006	.005	.004	.003	.002	.002
Monel metals.	.008	.007	.005	.004	.003	.002

* (BH: Brinell Hardness Number)

Table 7-4 shows recommended chip loads for milling various materials with various types of high-speed steel cutters.

COOLANTS

The purpose of a cutting coolant is to reduce frictional heat and thereby extend the life of the cutter's edge. Coolant also lubricates the cutter face and flushes away the chips, reducing the possibility of damage to the finish.

There are a number of synthetic coolants. Follow the manufacturer's recommendations when mixing them. If a synthetic coolant is not available, you can use soluble oil mixed at the rate of 40 parts water to 1 part oil.

When using a periphery milling cutter, apply the coolant to the point at which the tooth leaves the work. This will allow the tooth to cool before you begin the next cut. Allow the coolant to flow freely on the work and cutter.