are made with straight and helical flutes to ream holes with diameters from $\frac{1}{4}$ to $1\frac{5}{16}$ in.

**Carbide-tipped chucking reamers** (Figure H-118) are often used in production setups, particularly where abrasive materials or sand and scale (as in castings) are encountered. The right-hand helix chucking reamer (Figure H-119) is recommended for ductile or highly abrasive materials, or when machining blind holes. The carbide-tipped left-hand helix chucking reamer (Figure H-120) will produce good finishes on heat-treated steels up to RC 40. These reamers should be used only on through holes, because the chips push out through the hole ahead of the reamer. All expansion reamers (Figure H-121) after becoming worn can be expanded and resized by grinding. This feature offsets normal wear from abrasive materials and provides for a long tool life. These tools should not be adjusted for reaming size by loosening or tightening the expansion plug but only by grinding.

Reaming is intended to produce accurate and straight holes of uniform diameter. The required accuracy depends on a high degree of surface finish, tolerance on diameter, roundness, straightness, and absence of bellmouth at the ends of holes. To make an accurate hole it is necessary to use reamers with adequate support for the cutting edges; an adjustable reamer may not be adequate. Machine reamers are often made of either high-speed steel or cemented carbide. Reamer cutting action is controlled to a large extent by the cutting speed and feed used.

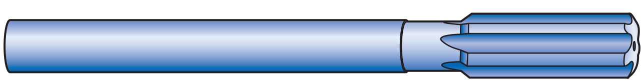

**Figure H-118** Carbide-tipped straight flute chucking reamer.

**Figure H-119** Carbide-tipped helical flute chucking reamer, right-hand helix.

**Figure H-120** Carbide-tipped helical flute chucking reamer, left-hand helix.

**Figure H-121** Carbide-tipped expansion reamer.

**SHOP TIPS–CRAFTSMANSHIP**

The accuracy of the hole diameter and the quality of the surface finish depend much on the condition of the starting hole. The roundness, straightness, and location of the drilled hole determines the quality of the hole produced. Reamers try to follow the centerline of the drilled hole, so it may be necessary to bore the hole before reaming to obtain the required locational accuracy.

## SPEEDS

The most efficient cutting speed for machine reaming depends on the type of material being reamed, the amount of stock to be removed, the tool material being used, the finish required, and the rigidity of the setup. A good starting point, when machine reaming, is to use one third to half of the cutting speed used for drilling the same materials. Table H-5 may be used as a guide.

## FEEDS

Feeds in reaming are usually two to three times greater than those used for drilling. The amount of feed may vary with different materials, but a good starting point would be between .0015 and .004 in. per revolution. Too low a feed may "glaze" the hole, which has the result of work hardening the material, causing occasional chatter and excessive wear on the reamer. Too high a feed tends to reduce the accuracy of the hole and the quality of the surface finish. Generally, it is best to use as high a feed as possible to produce the

**Table H-5** Reaming Speeds[a]

| | |
|---|---|
| Aluminum and its alloys | 130–200 |
| Brass | 130–200 |
| Bronze, high tensile | 50–70 |
| Cast iron | |
| Soft | 70–100 |
| Hard | 50–70 |
| Steel | |
| Low-carbon | 50–70 |
| Medium-carbon | 40–50 |
| High-carbon | 35–40 |
| Alloy | 35–40 |
| Stainless steel | |
| AISI 302 | 15–30 |
| AISI 403 | 20–50 |
| AISI 416 | 30–60 |
| AISI 430 | 30–50 |
| AISI 443 | 15–30 |

[a]Cutting speeds in surface feet per minute (fpm or sfm) for reaming with an HSS (high-speed steel) reamer. Where conditions permit the use of carbide reamers, the speeds may often be increased over those recommended for HSS reamers. The limiting factor is usually an absence of rigidity in the setup. Any chatter, which is often caused by too high a speed, is likely to chip the cutting edges of a carbide reamer. Always select a speed that is slow enough to eliminate chatter. Close tolerances and fine finishes often require the use of considerably lower speeds than those recommended in Table H-5.

required finish and accuracy. When a drill press that has only a hand feed is used to ream a hole, the feed rate should be estimated just as it would be for drilling. About twice the feed rate should be used for reaming as would be used for drilling in the same setup when hand feeding.

## STOCK ALLOWANCE

The stock removal allowance should be sufficient to assure a good cutting action of the reamer. Too small a stock allowance results in burnishing (a slipping or polishing action) or wedges the reamer in the hole causing excessive wear or breakage of the reamer. The condition of the hole before reaming also has an influence on the reaming allowance, because a rough hole will need a greater amount of stock removed than an equal-sized hole with a fairly smooth finish. See Table H-6 for commonly used stock allowances for reaming. When materials that work harden readily are reamed, it is especially important to have adequate material for reaming.

## CUTTING FLUIDS

To ream a hole to a high degree of surface finish, a cutting fluid is needed. A good cutting fluid will cool the workpiece and tool and will also act as a lubricant between the chip and the tool to reduce friction and heat buildup. Cutting fluids should be applied in sufficient volume to flush the chips away. Table H-7 lists some cutting fluids used for reaming different materials.

## REAMING PROBLEMS

Chatter is often caused by lack of rigidity in the machine, the workpiece, or the reamer itself. This condition may be corrected by reducing the speed, increasing the feed, putting a chamfer on the hole before reaming, using a reamer with a pilot (Figure H-122), or reducing the clearance angle on the cutting edge of the reamer. Carbide-tipped reamers especially cannot tolerate even a momentary chatter at the start of a hole, as such a vibration is likely to chip the cutting edges. Oversized holes can be caused by inadequate workpiece support, worn guide bushings, worn or loose spindle bearings, or bent reamer shanks. When reamers gradually start cutting larger holes, it may be because the work material is galling or forming a built-up edge on reamer cutting surfaces (Figure H-123). Mild steel and some aluminum alloys are particularly troublesome in this area. Changing to a different coolant may help. Reamers with highly polished flutes, margins, and relief angles or reamers that have special surface treatment may also improve the cutting action.

Bellmouthed holes are caused by misalignment of the reamer with the hole. The use of accurate bushings or pilots may correct bellmouth, but in many cases the only solution is the use of floating holders. A floating holder will allow movement in some directions while restricting it in others. A poor finish can be improved by decreasing the feed, but this will also increase the wear and shorten the life of the reamer. A worn reamer will never leave a good surface finish,

**Table H-6** Stock Allowance for Machine Reaming

| Reamer Size (in.) | Allowance (in.) |
|---|---|
| $\frac{1}{32}$ to $\frac{1}{8}$ | .003 to .006 |
| $\frac{1}{8}$ to $\frac{1}{4}$ | .005 to .009 |
| $\frac{1}{4}$ to $\frac{3}{8}$ | .007 to .012 |
| $\frac{3}{8}$ to $\frac{1}{2}$ | .010 to .015 |
| $\frac{1}{2}$ to $\frac{3}{4}$ | $\frac{1}{64}$ to $\frac{1}{32}$ |
| $\frac{3}{4}$ to 1 | $\frac{1}{32}$ |

**Table H-7** Cutting Fluids Used for Reaming

| Material | Dry | Soluble Oil | Kerosene | Sulfurized Oil | Mineral Oil |
|---|---|---|---|---|---|
| Aluminum | | X | X | | |
| Brass | X | X | | | |
| Bronze | X | X | | | X |
| Cast iron | X | | | | |
| Steels | | | | | |
| Low-carbon | | X | | X | |
| Alloy | | X | | X | |
| Stainless | | X | | X | |

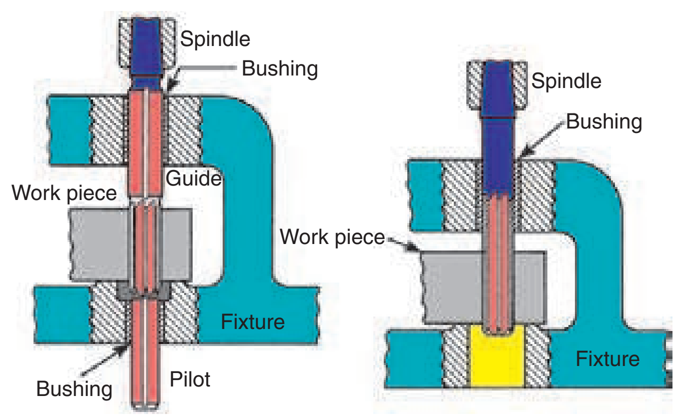


**Figure H-122** Use of pilots and guided bushings on reamers. Pilots are provided so that the reamer can be held in alignment and can be supported as close as possible while allowing for chip clearance (*Besley Cutting Tools, Inc*.).

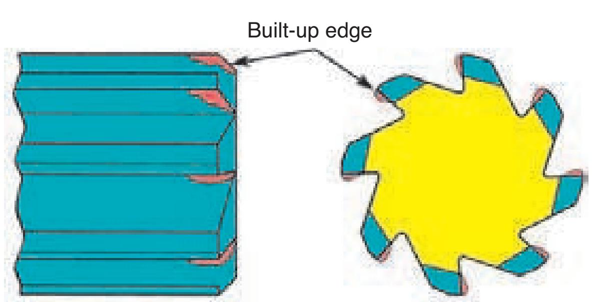


**Figure H-123** Reamer teeth having built-up edges.

# Vertical Milling Machine Operations

The vertical milling machine is one of the most versatile machine tools found in the machine shop. The purpose of this unit is to explore some of this versatility and give you an idea of the wide scope of machining capability on this machine.

**OBJECTIVES**

After completing this unit, you should be able to:

- Calculate cutting speeds and feeds for end milling operations.
- Identify and select vertical milling machine setups and operations for a variety of machining tasks.

## CLIMB AND CONVENTIONAL MILLING

In milling, the direction that the workpiece is being fed can be either the same as the direction of cutter rotation or opposed to the direction of cutter rotation. When the direction of feed is opposed to the direction of rotation, this is said to be *conventional* or *up milling* (Figure J-48). When the direction of feed is the same as the direction of cutter rotation, this is said to be *climb* or *down milling,* because the cutter is attempting to climb onto the workpiece as it is fed into the cutter. If there is any large amount of backlash in the table or saddle nuts, the workpiece can be pulled into the cutter during climb milling. This can result in a broken cutter, damaged workpiece, and possible injury from flying metal. Climb milling should be avoided in most cases. However, in certain situations it may be desirable to climb mill. For example, if the milling machine has ball nuts and screws, and backlash is virtually eliminated, climb milling is an acceptable technique. Even on conventional machines, climb milling with a light cut can result in a better surface finish, because chips are not swept back through the cut. During any milling operation, all table movements should be locked except the one that is moving. This will ensure the most rigid setup possible. Spiral fluted end mills may work their way out of a split collet when deep heavy cuts are made or when the end mill gets dull. As a precaution, to warn you that this is happening, you can make a mark with a felt-tip pen on the revolving end mill shank where it meets the collet face. Observing this mark during the cut will give you an early indication if the end mill is changing its position in the collet.

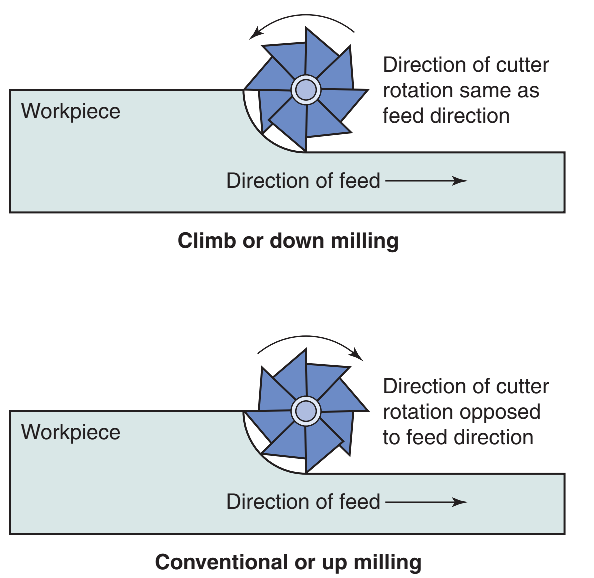


**Figure J-48** Climb and conventional milling.

## CUTTING PERFORMANCE FACTORS

Detailed information on machinability, cutting speeds, feeds, and cutting fluids is found in Section F.

### Cutting Speeds

Cutting speed (CS) in milling is the rate at which a point on the cutter passes by a point on the workpiece in a given period of time. Cutting speed is an extremely important factor in all machining operations. For rotating milling cutters, CS is expressed as a function of cutter rpm by the formula

$$\text{rpm} = \frac{\text{CS} \times 4}{D}$$

where

rpm = revolutions per minute of the cutter

CS = cutting speed of the material being machined, in feet per minute

$D$ = diameter of the cutter, in inches

Cutting speed constants (Table J-1) are influenced by the cutting tool material, workpiece material, rigidity of the machine setup, and use of cutting fluids. As a rule, lower cutting speeds are used to machine hard or tough materials or when heavy cuts are taken and it is desirable to minimize tool wear and to extend tool life. Higher cutting speeds are used in machining softer materials to achieve better surface finishes. Higher speeds also apply when using small-diameter cutters for light cuts on frail workpieces and in delicate setups. Table J-1 gives starting values for common materials. These values may have to be varied up or down depending on the specific machining task. Always observe the cutting action carefully and make appropriate speed corrections as needed. Until you gain some experience in milling, use the lower values in the table when selecting cutting speeds. Ball-end end mills, corner rounding mills, and angle milling cutters should be operated at half to two thirds the speed of a comparable end mill.

## Advanced Tool Materials

The best machining results are achieved with rigid machines and setups using a minimum of tool overhang and machine spindle extension. Vibration-causing chatter will rapidly destroy tools. Many carbide grades are made for general-purpose machining within a range of different work materials. Others are designed for one specific application and will fail if used for anything else. When using carbide, cermet, ceramic, CBN, or diamond inserts, it is absolutely essential to refer to the cutting tool manufacturer's catalog for specific application recommendations for a specific insert. Do not use any of these tools unless you know the intended use and operating conditions. General recommendations are as follows:

1. **Ceramics** are used to machine all carbon, alloy, and stainless steels and superalloys.
2. **CBN** is used to machine hardened steel (RC 45 and above), chilled cast iron, and superalloys.
3. **Diamonds** are used to machine nonmetallic and nonferrous materials. Diamonds are not used to machine ferrous materials because these alloys chemically attack the diamonds and cause rapid tool wear. Diamonds are used most often in high-speed finishing and semi-finishing operations.

## Feed Rates

Another equally important factor in safe and efficient machining is the feed rate. Feed rate is the rate at which the material is advanced into the cutter or the cutter is advanced into the work material. Because each tooth of a multitooth milling cutter is cutting, a chip of a given thickness will be removed depending on the rate of feed. Chip thickness affects the life of the milling cutter. Excessive feed rates can cause a chipped cutting edge or a broken cutter. On the other hand, the highest practical feed rate per tooth will give the longest tool life between resharpenings. Feed rate in milling is measured in inches per minute, or ipm, and is calculated by the formula

$$\text{ipm} = F \times N \times \text{rpm}$$

**Table J-1** Cutting Speeds for Some Commonly Used Materials

| | Tool Material | | | | | | |
|---|---|---|---|---|---|---|---|
| **Work Material** | **High-Speed Steel** | **Uncoated Carbide** | **Coated Carbide** | **Cermet** | **Ceramic** | **CBN** | **Diamond** |
| Aluminum | | | | | | | |
| Low silicon | 300–800 | 700–1400 | | | | | 1000–5000 |
| High silicon | | | | | | | 500–2500 |
| Bronze | 65–130 | 500–700 | | | | | 1000–3000 |
| Gray cast iron | 50–80 | 250–450 | 350–500 | 400–1000 | 700–2000 | 700–1500 | |
| Chilled cast iron | | | | | 250–600 | 250–500 | |
| Low-carbon steel | 60–100 | 250–350 | 500–900 | 500–1300 | 1000–2500 | | |
| Alloy steel | 40–70 | | 350–600 | 100–300 | 500–1500 | 250–600 | |
| Tool steel | 40–70 | | 250–500 | | 500–1200 | 150–300 | |
| Stainless steel | | | | | | | |
| 200 and 300 series | 30–80 | 100–250 | 400–650 | | 300–1100 | | |
| 400 and 500 series | | | 250–350 | | 400–1200 | | |
| Nonmetallics | | 400–600 | | | | | 400–2000 |
| Superalloys | | 70–100 | 90–150 | | 500–1000 | 300–800 | |

Table J-2 Feeds for High-Speed Steel End Mills (feed per tooth in inches)

| Cutter Diameter (in.) | Aluminum | Brass | Bronze | Cast Iron | Low-Carbon Steel | High-Carbon Steel | Medium Alloy Steel | Stainless Steel |
|---|---|---|---|---|---|---|---|---|
| $\frac{1}{8}$ | .002 | .001 | .0005 | .0005 | .0005 | .0005 | .0005 | .0005 |
| $\frac{1}{4}$ | .002 | .002 | .001 | .001 | .001 | .001 | .0005 | .001 |
| $\frac{3}{8}$ | .003 | .003 | .002 | .002 | .002 | .002 | .001 | .002 |
| $\frac{1}{2}$ | .005 | .002 | .003 | .0025 | .002 | .002 | .001 | .002 |
| $\frac{3}{4}$ | .006 | .004 | .003 | .003 | .004 | .003 | .002 | .003 |
| 1 | .007 | .005 | .004 | .0035 | .005 | .003 | .003 | .004 |
| $1\frac{1}{2}$ | .008 | .005 | .005 | .004 | .006 | .004 | .003 | .004 |
| 2 | .009 | .006 | .005 | .005 | .007 | .004 | .003 | .005 |

where

ipm = feed rate, in inches per minute
$F$ = feed per tooth, in inches
$N$ = number of teeth on the cutter being used
rpm = revolutions per minute of the cutter

Table J-2 gives starting values for chip loads in common materials. Information on how the feed per tooth compares with the actual chip thickness per tooth is presented in Section M, Unit 3.

## Cutting Speed and Feed Rate Calculations

The first step is to calculate the correct rpm for the cutter. Refer to Table J-1 for the starting value of the cutting speed.

### EXAMPLE

Calculate the rpm for a $\frac{1}{2}$-in.-diameter HSS end mill machining aluminum.

$$\text{rpm} = \frac{\text{CS} \times 4}{D} = \frac{300 \times 4}{1/2} = \frac{1200}{.5} = 2400$$

The next step is to calculate the feed rate. Refer to Table J-2 for the starting values.

$$\begin{aligned}\text{ipm} &= F \times N \times \text{rpm}\\ &= .005\ (\text{feed per tooth, Table J-2})\\ &\quad \times N\,(\text{number of teeth}) \times \text{rpm}\\ &= .005 \times 2 \times 2400 = 24\ \text{ipm}\end{aligned}$$

Therefore, the cutter should revolve at 2400 rpm, and the feed rate should be 24 ipm.

## Depth of Cut

The third factor to consider in using end mills is the depth of cut. The depth of cut is limited by the amount of material that needs to be removed from the workpiece, by the power available at the machine spindle, and by the rigidity of the workpiece, tool, and setup. As a rule, the depth of cut for an end mill should not exceed half the diameter of the tool in steel. In softer metals, the depth of cut can be more. With roughing end mills, the depth of cut can be as much as one and a half times the cutter diameter and the width of cut half the cutter diameter. If deeper cuts needs to be made, the feed rate needs to be reduced to prevent tool breakage. The end mill must be sharp and should run concentric in the end mill holder. The end mill should be mounted with no more tool overhang than necessary to do the job.

Another factor to consider is the horsepower rating of the machine. For horsepower calculations, see Section K, Unit 5.

**SHOP TIP**

Occasionally, when an end mill is used to mill a groove or slot, the sides of this groove are not perpendicular to or parallel with the spindle axis. Grooves with leaning sides are caused by worn spindles or excessive tool projection from the spindle. Excessive feed rates or dull end mills will also cause this problem. The leaning slot is produced by an end mill that is deflected by high cutting forces (Figure J-49). To reduce the tendency of the tool to cut a leaning slot, if the spindle is in good condition, reduce the feed rate and use end mills with only a short projection from the spindle. Sharp end mills with straight flutes or low-helix-angle flutes will also give satisfactory results.

## Cutting Fluids

Milling, as well as other machining operations, is often performed using **cutting fluids.** Cutting fluids serve to dissipate heat generated by the friction of the cutter against the

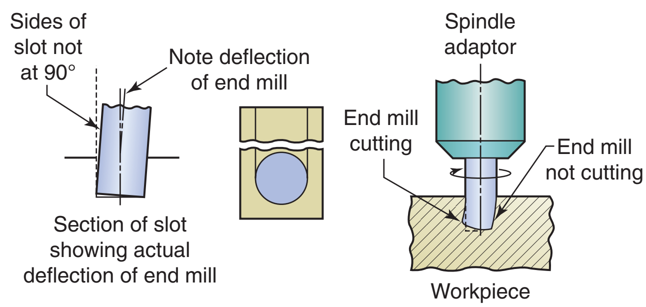


Figure J-49 The causes of a leaning slot in end milling.

workpiece. They also help lubricate the interface between the cutting edge and work and flush the chips away from the cutting area. Many machining operations are greatly improved by using cutting fluids. Cutting fluids increase productivity, extend tool life, and improve the surface finish of the workpiece. Cutting fluids are applied to the cut in a flood stream or by an air/cutting fluid mix mist. Cutting fluids generally reduce carbide tool life in milling cast iron and steel. The cutting fluid cannot reach the cutting edge while the insert is in the cut. After the extremely hot insert leaves the cut, the cutting fluid then has an extremely shocking cooling effect. These excessive temperature variations result in minute cracks along the cutting edge and premature tool failure. Cutting fluid recommendations of cutting tool manufacturers have to be followed to prevent tool failure caused by faulty coolant use.

Materials such as cast iron, brass, and plastics are often machined dry. A stream of compressed air can cool tools and keep the cutting area clear of chips. A safety shield and protective clothing should always be used when compressed air is applied, for protection from flying hot chips.

**SHOP TIP**

After selecting an end mill, check the condition of its cutting edges. If these cutting edges are dulled and show indication of wear, the end mill needs to be sharpened. If it is an indexable tool, you must rotate the insert to expose a new cutting edge, or change the end mill. End mills are sharpened on a tool and cutter grinder. It is important that the cutting edges be concentric with the tool shank and have the correct cutting edge angles. Dull tools create excessive cutting pressures that can break the end mill or leave an unsatisfactory surface finish. Worn end mills will also produce undersized grooves or slots.

## COMMON MILLING OPERATIONS

### Machining Steps and Squaring

Common milling operations on the vertical mill include machining steps (Figure J-50) and squaring or machining two surfaces perpendicular to each other (Figure J-51). The ends of the workpiece can be machined square and to a given length by using the peripheral teeth of an end mill. If a large amount of material has to be removed, it is best to use a roughing end mill first (Figure J-52), then finish to size with a regular end mill. On low-horsepower vertical mills, plunge cutting is an efficient method of removing material quickly (Figure J-53). In this operation, the end mill is plunged a predetermined width and depth of cut, retracted, then advanced and plunged again repeatedly. In plunging, the maximum cutting force is in the direction in which the machine is the strongest—in the axial direction.

**Figure J-50** Using an end mill to mill steps.

**Figure J-51** Using an end mill to square stock.

**Figure J-52** Roughing end mill used to remove a large volume of material.